HiSST: International Conference on
High-Speed Vehicle Science Technology
26–29 November 2018, Moscow, Russia

Drag Reduction of Hypersonic Spiked Blunt Body with Sideward Jets

Guilai Han[1], *Zonglin Jiang*[2]

Abstract

Drag reduction for blunt body flying at hypersonic speed were numerically investigated in this paper, with spikes and sideward jets applied to re-configurated the flow field. The spikes and the sideward jets led to the formation of the conical shock, the reattached shock, and the circumfluence, instead of a bow shock in front of the blunt body. The flow field reconfiguration played a dominant role in drag reduction. This paper proposed a parameter for the design of drag reduction of the spiked bodies, by combing the locations of the reattached shock and its interaction with the conical shock.

Keywords: *hypersonic, drag reduction, spike, sideward jet, numerical simulation*

Nomenclature (Tahoma 11 pt, bold)

Ma – Mach number of the incoming flow
L – The length of the spike
D – The diameter of the blunt body (sphere cylinder)
y_1 – The location of the reattached shock
y2 – The location of the interaction between the reattached shock and the conical shock

1. Introduction

The concept of a spiked blunt-body was first proposed by Bogdonoff et al. [1]. Investigations showed that spikes can construct a separation zone over blunt bodies to significantly lower the aerodynamic heat rate and pressure distribution, which was valuable for thermal protection and drag reduction. Han set up a new non-ablation concept utilizing adaptive drag reduction and thermal protection system (NADTPS) for hypersonic vehicles basing on a spiked blunt body with sideward jet nearby the spike nose [2]. This paper investigated the spiked blunt bodies with and without sideward jets flying at hypersonic speed, using numerical simulations. The incoming flow Mach number changed from 5 to 10, and the spike length ratio over the blunt body diameter changed from 0.5 to 1.0. The flow structure reconfiguration, especially the shock wave structure, was analyzed by flow field visualization. Aerodynamic force characteristics were calculated and compared with those of blunt bodies to analyze drag reduction effects and the corresponding mechanisms.

2. Governing Equations and Numerical Methods

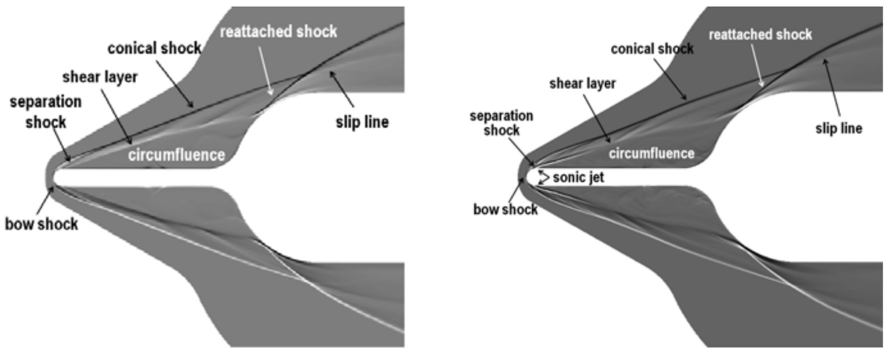
The two-dimensional axisymmetric compressible Navier-Stokes equation set was applied as the governing equations, with the thermally perfect gas model and ideal gas state equation. The 2nd order DCD scheme [3] was adopted to discrete the convective terms, and the Steger-Warming method [4] was applied for flux vector splitting. The 2nd order central difference scheme was used for viscous term discretization. The local time step method [5] was introduced to accelerate the simulation convergence process using different time steps at grid points.

3. Result and Discussion

[1] *State Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China, Email: hanguilai@imech.ac.cn*
[2] *State Key Laboratory of High Temperature Gas Dynamics, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China, Email: zljiang@imech.ac.cn*

The typical shock wave structures of pure spiked blunt bodies with defined labels are shown in Fig. 1(a) and Fig.1(b). Density gradient in the y direction was used to describe a schlieren photograph and re-configurated flow field. The spike nose became the new stagnation with a little bow shock in front and conical shock around. The reattached shock was formed on the blunt-body shoulder, resulting in a zone with high pressure. The inverse pressure gradient caused some of the gas flow to the spike nose, in the opposite direction of incoming flow. Thus, a circumfluence was generated around the spike and the blunt-body nose.

The tendency of the 3rd order polynomial cure shows that there might exist a limited value of y_1/D, which maintained the reattached shock position as the spike length increased. Based on the numerical results, the limit value was about 0.43, located at the angle about 60-degree to the central line, as shown in Fig.2(a). Different from the reattached shock, the interaction between the conical shock and the reattached shock moved away from the central line as the spike length increased, so does the slip line resulted by the interaction. The drag coefficients in the 72 cases of the pure spiked bodies and the spiked bodies with sonic sideward jets were plotted together, as shown in Fig.2(b). With y_1^2L/D^3 taken as the horizontal coordinate, the distribution of data points was significantly changed and more concentrated in the two drawn lines. Thus, the parameter y_1^2L/D^3 might be a simple and suitable description for the two main factors during flow field reconfiguration. In addition, drag coefficient tendency changes and becomes more gently when y_1^2L/D^3 is greater than 0.14. The reattached shock and the circumfluence criterion for drag reduction could be used by determining the critical value. Therefore, $y_1^2L/D^3 \approx 0.14$ can be considered as a criterion for the design optimization regarding the spike length and the strength of sideward jets.



(a) (b)

Fig 1. Typical flow field wave structure around the spiked blunt-body (*Ma*=6.0, *L*/*D*=1.0): (a) without sideward jet; (b) with sideward sonic jet nearby the spike nose.

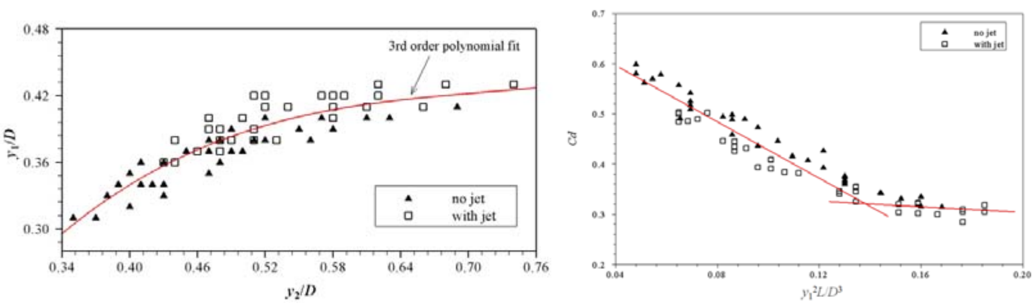


Fig 2. The combination of the parameter for the design of drag reduction system: (a) A third-order polynomial fit y_1/D and y_2/D; (b)Drag coefficients vary with the combined parameter y_1^2L/D_3.

References

1. S.M. Bogdonoff and I.E. Vas, "Preliminary investigations of spiked bodies at hypersonic speeds," Journal of the Aerospace Sciences, vol. 26, no. 2, pp. 65-74, 1959.

2. G.L. Han, Investigations on New Concept of Non-ablation and Adaptive Drag Reduction and Thermal Protection System for Hypersonic Vehicles [D], Beijing: Institute of Mechanics, Chinese Academy of Sciences, 2010.
3. Z.L. Jiang, "On the Dispersion-controlled Principles for Non-oscillatory Shock Capturing Schemes," Acta Mechanica Sinica, vol. 20, no. 1, pp. 1-15, 2004.
4. J.F. Steger and R.F. Warming, "Flux Vector Splitting of the Inviscid Gasdynamic Equations with Applications to Finite Difference Methods," Journal of Computational Physics, vol. 40, no. 2, pp.263-293, 1981.
5. D.X. Fu and Y.W. Ma, Computational Fluid Dynamics, Higher Education Press, Beijing, 2002.