Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Drag reduction investigation for hypersonic lifting-body vehicles with aerospike and long penetration mode counterflowing jet

Fan Deng [a,b], Feng Xie [b,d], Ning Qin [b], Wei Huang [c], Lansong Wang [a,e], Hongyu Chu [a]

[a] *China Academy of Launch Vehicle Technology, Beijing 100076, China*
[b] *Department of Mechanical Engineering, University of Sheffield, Sheffield S1 3JD, UK*
[c] *Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha, Hunan 410073, China*
[d] *Hypervelocity Aerodynamics Institute, China Aerodynamics Research and Development Center, Mianyang, Sichuan 621000, China*
[e] *Department of Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK*

ARTICLE INFO

Article history:
Received 15 March 2017
Received in revised form 11 August 2017
Accepted 17 January 2018
Available online xxxx

Keywords:
Hypersonic
Lifting-body vehicle
Drag reduction
Aerospike
Counterflowing jet

ABSTRACT

This study describes numerical computations of aerospike and counterflowing jet as the drag reduction approach of a hypersonic lifting-body model for Mach 8 flow at 40 km altitude. The three-dimensional Reynolds-average Navier–Stokes equations and the laminar condition have been utilized to obtain the flow field properties. Both steady-state and time-accurate computations are performed for the models in order to investigate the drag reduction effect and the periodic oscillation characteristics of long penetration mode (LPM) jet. The obtained results show that both methods can significantly modify the external flowfields and strongly weaken or disperse the shock-waves of the vehicle, then achieve an obvious drag reduction effect in the range of small angles of attack. This value is 7.25% for the model with aerospike and 8.80% for the model with counterflowing jet at angle of attack of 6°. Compared with the aerospike, the counterflowing jet shows a better drag reduction effect in the gliding angle of attack, and this leads to a larger lift-to-drag ratio of 3.58. Meanwhile, the displacement of center of pressure of the whole vehicle is smaller in the flight phase. The oscillation frequency of the counterflowing jet at angle of attack of 6° is 444 Hz, and the oscillation characteristics of the drag force is due to the variation of the pressure distribution induced by the oscillation of the shock wave structure. The results may be of high practical significance and show the possibility of developing a feasible system using the counterflowing jet as an active flow control to reduce the drag force during the gliding phase with maximum lift-to-drag ratio.

1. Introduction

In the design of hypersonic vehicles with long time flying in near space, the lifting-body is one of the main aerodynamic configurations due to its good lift-to-drag ratio and internal loading space. The nose of the vehicle is subjected to severe aerodynamic heating and strong shock wave drag during high speed flight, contributing disproportionately to the vehicle drag and aerothermal loads, which translates into poor aerodynamic performance and stringent thermal protection system requirements, and other performance penalties including vehicle range, weight and payload. Thus, a variety of techniques have been implemented, and these techniques include aerospike [1–3], counterflowing jet [4–6] and energy deposition [7]. However, the power budget and the system complexity are highly prohibitive for using the energy deposition concept. In addition, the high temperature gas produced by local energy deposition probably imposes a heavier burden on the design of thermal protection systems for hypersonic vehicles. For this reason, at present, the main approaches for drag reduction of the hypersonic vehicle are aerospike and counterflowing jet [8,9], and Wang et al. [10] gave a detail review on the experimental investigation on drag and heat flux reduction in supersonic/hypersonic flows in 2016.

A hypersonic lifting-body vehicle yields a strong detached bow shock wave ahead of its nose [11]. This shock wave is responsible for the elevated pressure levels attained by the downstream flow. Therefore, it is believed that the excessive drag force can be reduced by altering the flow field pattern ahead of the vehicle nose so as to eliminate the strong shock wave or replace it with a weaker system of shock waves. Early research focused on flow mechanism around the spike, Mair [12] experimentally

---

E-mail address: dengfan8245@sina.cn (F. Deng).

https://doi.org/10.1016/j.ast.2018.01.039
1270-9638/

Please cite this article in press as: F. Deng et al., Drag reduction investigation for hypersonic lifting-body vehicles with aerospike and long penetration mode counterflowing jet, Aerosp. Sci. Technol. (2018), https://doi.org/10.1016/j.ast.2018.01.039

Nomenclature

Symbol	Description	Unit
D	vehicle nose diameter	mm
L	vehicle length	mm
L_1	first cone length	mm
L_2	second cone length	mm
Φ_1	first cone base diameter	mm
Φ_2	second cone base diameter	mm
S	span of the triangular wings	mm
χ	leading edge swept angle of the wing	°
P	static pressure	Pa
P_0	inflow total pressure	Pa
P_{0f}	freestream total pressure behind shock wave	Pa
P_{oj}	jet total pressure	Pa
P_j	jet static pressure	Pa
P_∞	inflow static pressure	Pa
Ma	Mach number	
Re	Reynolds number based on the vehicle length	
α	angle of attack	°
C_D	drag force coefficient	
C_L	lift force coefficient	
k	lift-to-drag ratio	
X_{cp}	pressure center coefficient	

examined the supersonic flow around spiked flat cylindrical and hemisphere cylindrical models, recorded a sign of flow instability around spiked bodies, and proposed the first explanation of this form of "flow oscillation" based on the pressure difference between the flow downstream of the reattachment shock wave and the flow inside the recirculation zone. Maull [13] refined the mechanism of boundary layer separation over the spike length based on the pressure equilibrium on both sides of the separation shock wave, and he also clarified the mechanism of flow stability based on the mass flow equilibrium between the flow scavenged by the shear layer and that reversed inside the dead air zone. The governing factor was argued to be the flow turning angle on the model's face at the reattachment point. Wood [14] used the term "dividing streamline" to refer to the streamline that links the separation and reattachment points of the shear layer at steady flow conditions. At the same time, Chapman et al. [15] argued that if the total pressure of the dividing streamline equals to the peak pressure downstream of reattachment shock, the dividing streamline stagnates on the forebody and a stable shear layer is attained. Ahmed and Qin [16–18] found that both spike length and aerodisk diameter determined the flow mode, and the flow around the spiked model can be stable if the main geometric parameters are reasonable prescribed. This provides the researchers with motivation to optimize the spike in order to achieve the best drag reduction effect in a stable flow field.

The geometric parameters of the spike, such as spike length, aerodisk geometry, forward body geometry and relative spike diameter, were investigated for drag reduction effect [19–21]. Some researchers [22–24] investigated the aerodynamic characteristics of conical, hemispherical, flat-faced aerospike, and hemispherical and flat-faced disk attached to the aerospike, and they found that the aerodisk spike has a superior drag reduction capability as compared to the other aerospikes. The influences of the spike length, shape, spike nose configuration and angle of attack on the drag reduction were experimentally studied by Kalimuthu et al. [25], and they found that the aerodisk with the spike length to cylinder diameter ratio being 2.0 is the most effective among the models tested. Yadav and Guven [26] proved numerically that the double-disk aerospike is superior to the single-disk aerospike with the same overall length and hemispherical cap size in the drag reduction of the main body. To study the aerodynamic characteristic of spike at a certain angle of attack, Schülein [27] introduced the concept of "pivoting spike" in which the spike is maintained to be aligned with the freestream direction while the whole body is at an incidence angle. He experimentally examined the pivoting spike at Mach 2, 3 and 5 with up to 30° incidence angle, and the experimental results presented show clearly the advantages of the aligned spikes over the conventional fixed spikes.

Another drag reduction approach is setting a counterflowing jet at the stagnation zone of the vehicle [28], and the large scale vortices develop gradually at a recirculation zone when the jet terminates through a Mach disk and reverses its orientation as a conical free shear layer. The recirculation zone ahead of the vehicle has a great impact on the reduction of the drag force [29].

Finley [30] performed a series of experiments in which a jet issues from an orifice at the nose of a body in the supersonic flow to oppose the mainstream, and an analytical model of the flow which suggests that the aerodynamic features of a steady flow depend primarily on a jet flow-force coefficient and the Mach number of the jet on its exit plane was developed. The transition between LPM (Long Penetration Mode) and SPM (Short Penetration Mode) was abrupt, and it was shown to occur at various PRs (jet total pressure ratio) depending on body size and the jet exit Mach number [31]. Josyula et al. [32] investigated the potential applications of a counterflowing drag reduction technique to assess performance improvements on aerospace vehicles. It was demonstrated that 30–50% drag reduction can be achieved by counterflowing blowing against a supersonic stream of Mach 4 or higher. Li et al. [33] investigated the drag reduction mechanism in a supersonic blunt body with different jet strategies, and they found that taking the drag reduction and heat protection into consideration together, the effect of square shape is the best in all considered strategies.

Bushnell and Huffman [34] studied long penetration jet interactions, and they observed that the transition from LPM to SPM occurs at a fixed PR for all engine sizes tested. Shang et al. [35,36] focused on the pressure ratio, which describes the counterflow jet phenomenon, and they discovered the shock bifurcation phenomenon and shock-wave interaction by experimental and computational studies. Fomichev et al. [37] experimental and numerical studied the impact of counterflowing plasma jets on integral and distributed aerodynamic characteristics of blunted bodies in hypersonic flows with a counterflowing plasma jet, and a decrease in the total-drag coefficient for a 60°-cone up to 25% at the LPM regime was obtained.

Kulkarni et al. [38] demonstrated reduction in the aerodynamic drag force for a blunt cone flying at hypersonic Mach number by heat addition into the shock layer in shock tunnel, and the experimental data have shown about 47% reduction in the aerodynamic drag force for a chromium plated 60° apex angle blunt cone in Mach 8 hypersonic flow. Aruna and Anjalidevi [39] investigated the influence of the counterflowing jet on the drag reduction around two blunt cone flare bodies in the hypersonic turbulent flow at Mach number of 6.5, and comparing with the values pertained to the case in the absence of jet, substantial reductions in total drag around 37.54%.

Most of the above studies mainly focus on the influence of the drag reduction technologies on nose with different shapes, rarely involving the whole vehicle, especially the drag reduction effect of

Please cite this article in press as: F. Deng et al., Drag reduction investigation for hypersonic lifting-body vehicles with aerospike and long penetration mode counterflowing jet, Aerosp. Sci. Technol. (2018), https://doi.org/10.1016/j.ast.2018.01.039