Acta Astronautica 142 (2018) 300–313

Contents lists available at ScienceDirect

Acta Astronautica

journal homepage: www.elsevier.com/locate/actaastro

Investigation of drag and heat reduction induced by a novel combinational lateral jet and spike concept in supersonic flows based on conjugate heat transfer approach

Liang Zhu [a], Xiong Chen [a,*], Yingkun Li [a], Omer Musa [a,b], Changsheng Zhou [a]

[a] School of Mechanical Engineering, Nanjing University of Science and Technology, Nanjing, 210094, PR China
[b] Mechanical Engineering Department, Omdurman Islamic University, Omdurman, Sudan

ARTICLE INFO

Keywords:
Lateral jet
Spike
Conjugate heat transfer
Drag reduction
Supersonic flow

ABSTRACT

When flying at supersonic or hypersonic speeds through the air, the drag and severe heating have a great impact on the vehicles, thus the drag reduction and thermal protection studies have attracted worldwide attention. In the current study, the Reynolds-averaged Navier-Stokes (RANS) equations coupled with the shear stress transport (SST) $k-\omega$ turbulence model have been employed to investigate the flow behavior induced by a novel combinational lateral jet and spike concept in supersonic flows. A coupling conjugate heat transfer (CHT) approach has been applied to investigate the thermal protection, which takes the heat transfer of structure into consideration. After the code was validated by the available experimental results and the gird independency analysis was carried out, the influences of the spike length ratio, lateral jet pressure ratio and lateral jet location on the drag and heat reduction performance are analyzed comprehensively. The obtained results show that a remarkable reduction in the drag and heat flux is achieved when a lateral jet is added to the spike. This implies that the combinational lateral jet and spike concept in supersonic flows have a great benefit to the drag and heat reduction. Both the drag and heat reduction decrease with the increase of the lateral jet pressure ratio, and the heat flux is more sensitive to the lateral jet pressure ratio. The lateral jet should not be located in the bottom of the spike in order to realize better drag and heat reduction performance. The drag and heat flux could be reduced by about 45% by reasonable lateral jet location. The drag decreases with the increase of the spike length ratio whereas the heat flux is affected by the spike length ratio just in a certain range.

1. Introduction

In recent years, supersonic vehicles have got wide attention in the world and lots of progress have been achieved. However, there still exist many challenges in engineering applications, such as severe aerodynamic heating and huge aerodynamic drag, which have significant impact on the overall performance of supersonic vehicles [1–3]. As well known, even two thirds of the total drag is induced by shock wave drag when the vehicle cruises at high speed through the air, and thermal protection system must be well designed to protect the vehicle structure from high temperature. Effective drag and heat reduction will improve the performance of the vehicle to a great extent, which means longer range, lighter structure and lower fuel consumption.

Generally, flow control technology can achieve the purpose of drag and heat reduction in supersonic flow and various techniques have been proposed, such as spike [4–8], opposing jet [9–11], a combination of spike and opposing jet [12–17], and energy deposition [12,18].

Spike technology, which can greatly reduce the drag by changing the position and shape of the bow shock in front of the blunt body, was experimentally investigated in the 1950s and numerous scholars have completed lots of research in recent years [4]. In 1995, Yamauchi [5] investigated the flow characteristics around a spiked blunt body at different free stream Mach numbers and studied the influence of the ratio of spike length to the base diameter on flow structure. Mehta [19] numerically studied the flow field around a spiked blunt body at a free stream Mach number of 6.8 for different spike lengths and conducted an investigation on the effects of the spike configurations in 2000. Kalimuthu and Mehta [20] studied the effects of the spike geometrical parameters on the drag, lift, pitching moment and pressure measurements experimentally in 2010. They found that the surface pressure over the

* Corresponding author.
E-mail address: chenxiong@njust.edu.cn (X. Chen).

https://doi.org/10.1016/j.actaastro.2017.11.001
Received 15 August 2017; Received in revised form 15 September 2017; Accepted 2 November 2017
Available online 21 November 2017
0094-5765/

blunt-body in the presence of spike gives the maximum pressure at about 50 deg. In 2014, Mansour and Khorsandi [6] investigated numerically the flow field with Mach number being 6 around a spike. The standard $k-\varepsilon$ turbulence model was used for solving Reynolds-averaged Navier-Stokes equations. The pressure coefficient on the blunt body was compared with experimental results and agreed well, and the drag coefficient was reduced by 40% respect with no spiked blunt body. However, spike technique has good performance on drag reduction under the condition of zero attack angle, while the shock/shock interaction will take place on the shoulder of the blunt body which results in extremely high pressure and temperature.

The opposing jet had been proposed for drag reduction and thermal protection in the early 1960s [21]. Compared with the spike, opposing jet presents an effective way in both drag reduction and thermal protection. Hayashi et al. [10,22] have conducted numerous studies on opposing jet by both experimental and numerical approaches. The results indicated that the opposing jet pushes the bow shock away the blunt body and forms a low temperature recirculation zone, which plays an important role in reducing the heat flux. What is more, they found out that the flow mode is defined by the ratio of the stagnation pressure of the jet to that of the free stream. In 2010, Chen [7] investigated the jet from a blunt body with a free stream Mach number 2.5 systematically by large-eddy simulation method for two total pressure ratios. The obtained results provide physical insight into the understanding of the mechanisms relevant to the opposing/supersonic flow reaction. Barzegar Gerdroodbary et al. [11] conducted a study on the transient flow behavior of a sonic counterflow from a blunt body. The classical single-component supersonic axisymmetric jet flow had not been employed and the coolant gas (Carbon Dioxide and Helium) was chosen to inject into the hypersonic air flow. It focused on the initial transient period of jet development, where a bow shock wave forms and travels downstream, and some valuable conclusions were obtained.

To be more efficient and effective, several novel combinatorial strategies, which combine the opposing jet and other techniques, have been proposed recently. The forward-facing cavity and opposing jet combinatorial thermal protection system was investigated numerically by Lu et al. [23] in 2009. The aerodynamic heating is reduced remarkably by the combinatorial thermal protection system, and the aerodynamic drag is also decreased. The opposing jet stagnation pressure is found to be a key parameter to the cooling efficiency of the combinatorial system. The larger the opposing jet stagnation pressure, the better the cooling efficiency. In 2014, Barzegar Gerdroodbary et al. [15] conducted a numerical investigation on a novel combinational technique which consists of a aerodisk and forward injection. They used different jets (helium and carbon dioxide) and studied the influences of the pressure ratio and the spike length. The results indicate that both the spike length and pressure ratio significantly influence the heat reduction. Moreover, the lighter gas proved to be more effective in reducing heat flux. In 2015, Huang et al. [24] numerically investigated the performance of combinational opposing jet and spike and discussed the influences of jet pressure ratio and spike length to the base diameter ratio at a free stream Mach number of 2.5. The results indicate that the maximum drag reduction coefficient reaches 65.02% and the peak pressure location moves nearly from 40 to 55 deg. In the same year, Huang et al. [25] studied another combinational opposing jet and acoustic cavity concept to get the drag and heat reduction mechanism by the numerical approach. The influences of the cavity location, the length-to-depth ratio of the cavity and the molecular weight of the jet had been analyzed. Liu and Jiang et al. [26,27] conducted experiments to study a novel combinational lateral jet and spike in the hypersonic wind tunnel at a free stream Mach number of 6 in 2009. The experimental data indicated that this strategy is of great benefit to drag reduction and thermal protection even under nonzero attack angle.

In most of previous studies, the fluid-thermal interaction between the flow field and the blunt body was neglected and the blunt body surface was commonly defined to be isothermal or adiabatic condition, which omitted the influence of the interface temperature on the flow field and thus failed to evaluate the heat flux accurately [28]. As well known, the large recirculation zone between the spike and shear layer has great impact on the drag and heat reduction [4,10]. And the wall temperature would affect the recirculation zone flow features [29], which means accurate thermal boundary prediction is of essential to investigate the thermal protection of the blunt body. On the other hand, setting the blunt body surface being isothermal or adiabatic condition is inconsistent with physical truth. Thus, taking the fluid-thermal interaction into consideration is of great importance to the aerodynamic drag and heating analysis of the spiked blunt body with a lateral jet at supersonic speeds.

This study aims to conduct a numerical investigation on the drag and heat reduction induced by a novel combinational lateral jet and spike concept in supersonic flows. A coupling conjugate heat transfer (CHT) approach has been applied in this study to deal with the fluid-thermal interaction between the flow field and the blunt body. Furthermore, the influences of the spike length ratio, lateral jet pressure ratio and lateral jet location on flow field are investigated numerically. This paper is organized as follows. In section 2, the physical model and numerical methodology have been described briefly. The validation of the code and the grid independency analysis have been carried out in section 3. Section 4 provides the visualization of the flow features around the spiked blunt body with a lateral jet, and the influences of the spike length ratios, lateral jet pressure ratios and lateral jet locations on the drag and heat reduction are analyzed. Finally, some conclusion remarks have been drawn in section 5.

2. Physical model and numerical approach

2.1. Physical model

The model is shown in Fig. 1 and the spiked blunt body configurations studied in the current study referenced the geometry in Refs. [24,25]. The diameter of the blunt body (D) is 50 mm and length of the parental body is 0.2D. The diameter of the spike is 4 mm and the ratio of L/D varies from 0.5 to 1.5. The lateral jet location is set on the spike and the width of lateral jet exit (r) equals 1 mm. A new parameter was introduced to define the lateral jet location.

$$LR = L_{jet}/L \tag{1}$$

where L is the length of the spike and L_{jet} is the distance from the lateral jet exit center to the spike bottom.

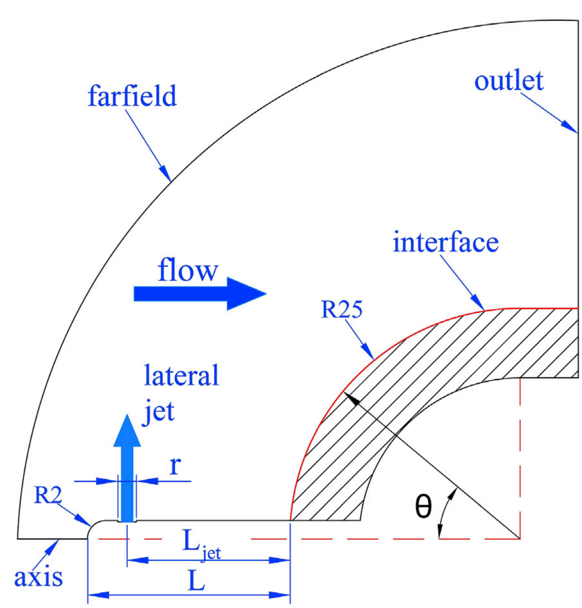

Fig. 1. Geometry and boundary condition types in the current study.