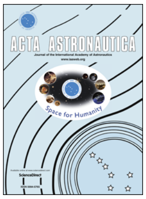
Acta Astronautica 121 (2016) 164–171

Contents lists available at ScienceDirect

Acta Astronautica

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



Heat flux reduction mechanism induced by a combinational opposing jet and cavity concept in supersonic flows

Wei Huang*, Yan-ping Jiang, Li Yan, Jun Liu

Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha, Hunan 410073, People's Republic of China

ARTICLE INFO

Article history:
Received 11 September 2015
Received in revised form
18 December 2015
Accepted 6 January 2016
Available online 15 January 2016

Keywords:
Hypersonic vehicle
Heat flux reduction
Opposing jet
Cavity
Supersonic flow

ABSTRACT

The thermal protection on the surface of hypersonic vehicles attracts an increasing attention worldwide, especially when the vehicle enters the atmosphere at high speed. In the current study, the Reynolds-averaged Navier-Stokes (RANS) equations coupled with the Menter's shear stress transport (SST) model have been employed to investigate the heat flux reduction mechanism induced by the variations of the cavity configuration, the jet pressure ratio and the injectant molecular weight in the combinational opposing jet and cavity concept. The length of the cavity is set to be 6 mm, 8 mm and 10 mm in order to make sure that the cavity configuration is the "open" cavity, and the jet pressure ratio is set to be 0.4, 0.6 and 0.8 in order to make sure that the flow field is steady. The injectant is set to be nitrogen and helium. The obtained results show that the aft angle of the cavity only has a slight impact on the heat flux reduction, and the heat flux peak decreases with the decrease of the length of the cavity. The design of the thermal protection system for the hypersonic blunt body is a multi-objective design exploration problem, and the heat flux distribution depends on the jet pressure ratio, the aft wall of the cavity and the injectant molecular weight. The heat flux peak decreases with the increase of the jet pressure ratio when the aft angle of the cavity is large enough, and this value is 45°.

1. Introduction

The drag reduction and thermal protection on the surface of a hypersonic vehicle becomes especially important when the vehicle enters the atmosphere at high speed [1–3], and damage on the nose of the vehicle induced by high temperature and high pressure must be prevented.

Many techniques, for example, concentrated energy deposition along the stagnation streamline [4], a retractable aerospike ahead of the blunt body [5], and a counterflowing jet in the stagnation zone of the blunt body [6–8], have been proposed to reduce the drag and heat release, and they can be categorized into passive and active approaches according as to whether the method can be controlled or not. The passive approach is the retractable aerospike installed ahead of the blunt body employed to reduce the intensity of the shock wave, and the active approaches are energy deposition along the stagnation streamline and a counterflowing jet in the stagnation zone of the blunt body. Huang et al. [9] combined the retractable aerospike and the opposing jet to reduce the drag force of the hypersonic blunt body, and it is found that the drag reduction coefficient increases with the increase of the length-to-diameter ratio of aerospike and the jet pressure ratio.

Recently, the combinational opposing jet and cavity concept has been proposed to reduce the drag force and the heat flux [10,11], and the cooling effect has been produced by the oscillation of the bow shock wave [12]. When the jet pressure ratio is 0.2, the cavity would waste the

---

* Corresponding author.
Tel.: +86 731 84576447; fax: +86 731 84512301.
E-mail address: gladrain2001@163.com (W. Huang).

http://dx.doi.org/10.1016/j.actaastro.2016.01.008
0094-5765/

energy of the counterflowing jet, and this condition may be an unstable condition. Thus, it must be avoid in the current study. However, when the jet pressure ratio is 0.4, the cavity would act just like a speed-up nozzle for the counterflowing jet, and this would be beneficial for the drag and heat flux reduction [13].

However, in the opinion of the authors, the influences of the cavity configuration, the jet pressure ratio and the injectant molecular weight on the heat flux reduction of the combinational opposing jet and cavity concept have been rarely investigated, especially the cavity configuration with the aft angle. This configuration has been widely employed as the flameholder in the flowpath of the scramjet engine [14], and it would reduce the heat flux distribution on the aft wall of the cavity, as well as its drag force. Thus, this information needs to be explored further.

In the current study, the heat flux reduction mechanism induced by the variation of the operational and structural parameters of the combinational opposing jet and cavity concept has been investigated numerically. In Section 2, the physical model and numerical methods have been described in detail. In Section 3, the code validation and grid independency analysis have been provided, and the predicted results have been compared with the available experimental data in the open literature. In Section 4, the effects of cavity configuration, jet pressure ratio and injectant molecular weight on the heat flux reduction mechanism have been analyzed, and some results have been stated. In Section 5, some conclusions have been provided.

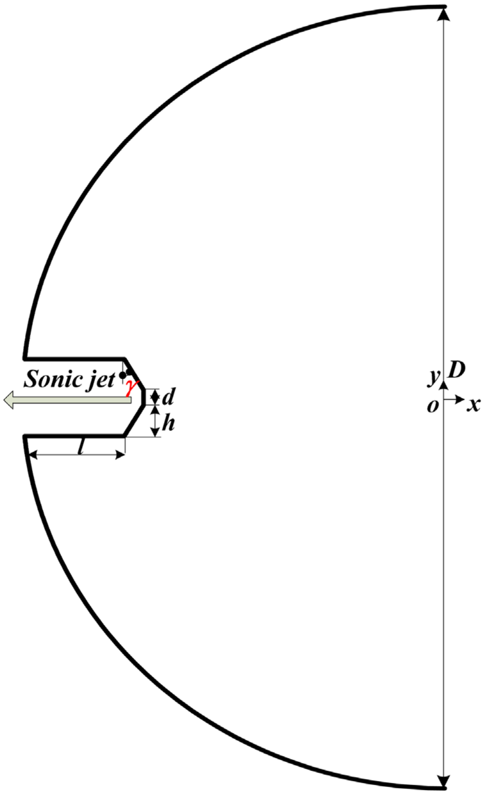


Fig. 1. Sketch of the geometrical model employed in the current study.

2. Physical model and numerical approach

2.1. Physical model

The diameters of the hypersonic blunt body (D) and the jet orifice (d) are set to be 50 mm and 1 mm, respectively, see Fig. 1, and the center of the hypersonic blunt body is set as the origin of the coordinate system. Fig. 1 depicts the sketch of the geometrical model employed in this article. The height of the cavity (h) maintains constant, and its value is 2 mm, and the length (l) and aft angle (γ) of the cavity are varied in order to investigate the influence of the cavity configuration on the heat flux reduction mechanism of the combinational opposing jet and cavity concept.

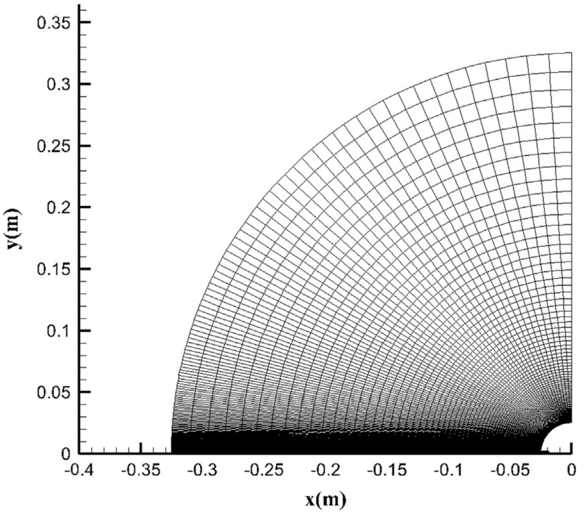


Fig. 2. Sketch diagram of the grid system.

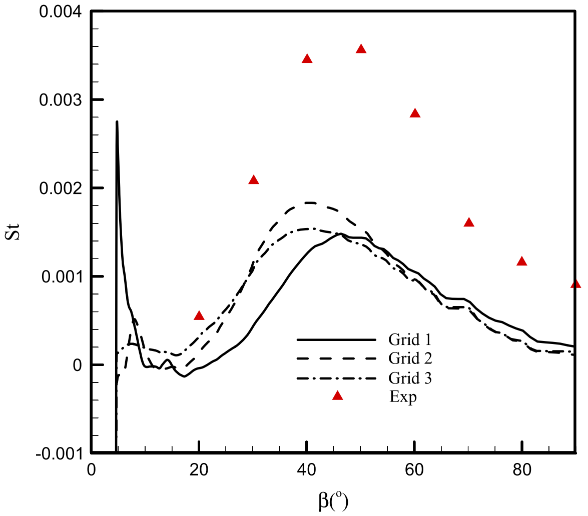


Fig. 3. Comparison of the Stanton number distributions along the blunt body surface with different numbers of grid cell (Grid 1 with 45,892 cells, Grid 2 with 69,252 cells and Grid 3 with 84,972 cells), and the experimental data was obtained by Hayashi et al. [15].