Acta Astronautica 131 (2017) 204–225

Contents lists available at ScienceDirect

Acta Astronautica

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

A study of performance parameters on drag and heat flux reduction efficiency of combinational novel cavity and opposing jet concept in hypersonic flows

Xi-wan Sun, Zhen-yun Guo, Wei Huang*, Shi-bin Li, Li Yan

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

ARTICLE INFO

Keywords:
Hypersonic flow
Drag and heat reduction
Forward-facing cavity
Opposing jet
Jet operating conditions
Freestream angle of attack
Physical dimensions

ABSTRACT

The drag reduction and thermal protection system applied to hypersonic re-entry vehicles have attracted an increasing attention, and several novel concepts have been proposed by researchers. In the current study, the influences of performance parameters on drag and heat reduction efficiency of combinational novel cavity and opposing jet concept has been investigated numerically. The Reynolds-average Navier-Stokes (RANS) equations coupled with the SST k-ω turbulence model have been employed to calculate its surrounding flowfields, and the first-order spatially accurate upwind scheme appears to be more suitable for three-dimensional flowfields after grid independent analysis. Different cases of performance parameters, namely jet operating conditions, freestream angle of attack and physical dimensions, are simulated based on the verification of numerical method, and the effects on shock stand-off distance, drag force coefficient, surface pressure and heat flux distributions have been analyzed. This is the basic study for drag reduction and thermal protection by multi-objective optimization of the combinational novel cavity and opposing jet concept in hypersonic flows in the future.

1. Introduction

The drag reduction and thermal protection system applied to hypersonic vehicles have attracted an increasing attention because the high pressure and aerodynamic heating will cause the damage of the aircraft surface and electronic devices. It attaches profound importance to conduct a survey on drag and heat reduction schemes of hypersonic re-entry vehicle [1,2]. The research progress of experimental investigations have been reviewed in Ref. [3]. The combinational opposing jet and aerospike concept [4–9], the combinational opposing jet and forward-facing cavity concept [10–12] and the combinational forward-facing cavity and energy deposition concept [13] have made up for some defects such as the ablation of aerospike and the unsteady flows in forward-facing cavity, and improved the drag and heat reduction efficiency greatly.

In our previous study [14], the research progress of combinational forward-facing cavity and opposing jet concept was investigated in detail, and a parabolic cavity configuration was proposed to substitute the conventional one, as shown in Fig. 1. After comparing the drag and heat reduction efficiency of the two configurations numerically, we come to the conclusion that the multi-objective optimization [15,16] is an essential process since each scheme has both advantages and disadvantages. The parametric investigation on the performance should be conducted based on the numerical method as a preliminary research for optimization [17,18]. The jet operating conditions, freestream angle of attack (AoA) and physical dimensions may be worth selecting as the parametric variables to minimize both the drag force and heat transfer rate [19–22]. In the current study, the effects of these parameters on shock stand-off distance [23], drag force coefficient, surface pressure and heat flux distribution are investigated numerically by the commercial software FLUENT. However, the axisymmetric assumption employed in Ref. [14]. is abandoned, and three-dimensional flowfields [24] are simulated in this study. Research progress of relative topics will be reviewed in corresponding sections.

In the following study, the drag and heat reduction mechanism induced by a combinational novel cavity and opposing jet concept will be investigated numerically. In Section 2, the three-dimensional physical model is described, as well as the boundary conditions. In Section 3, the numerical method is provided, as well as the selection of spatially accurate model and grid independence analysis. The effects of

---

* Corresponding author.
E-mail address: gladrain2001@163.com (W. Huang).

http://dx.doi.org/10.1016/j.actaastro.2016.11.044
Received 19 October 2016; Accepted 28 November 2016

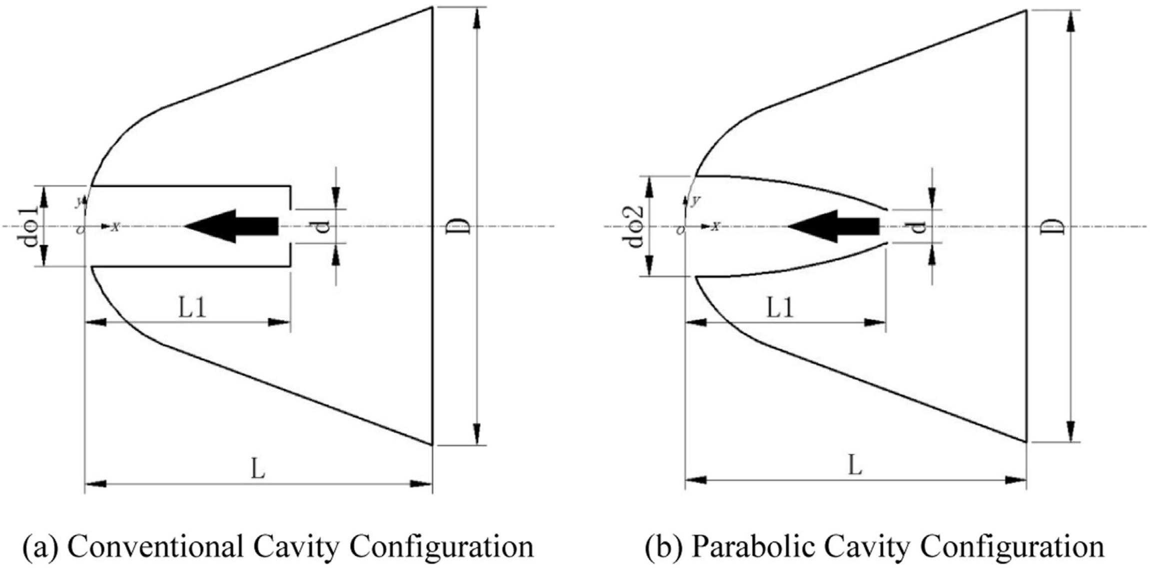

Fig. 1. Sketch of the geometrical models employed in the current study [14].

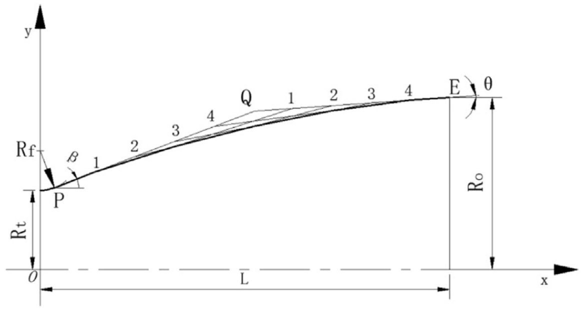

Table 1
Boundary conditions.

Contents		Unit	
Freestream Mach number	Ma_∞	–	7.96
Freestream total pressure	$P_{0\infty}$	Pa	1,939,211
Freestream total temperature	$T_{0\infty}$	K	1955
Opposing jet Mach number	Ma_{opp}	–	0.5, 1, 2
Jet total pressure ratio	PR	–	0.07, 0.14, 0.28
Jet static pressure		Pa	72,374, 144,749, 289,498
Opposing jet species	–	–	H_2, CH_4, N_2, Air, CO_2
Opposing jet total temperature	T_{0j}	K	300
Wall temperature	T_w	K	300

jet operating conditions, freestream angles of attack and physical dimensions are studied in Sections 4, 5 and 6 respectively. In Section 7, some valuable conclusions are summarized.

2. Physical model

2.1. Geometric model

The geometric models employed have been introduced in Ref. [14], and the generation principle by graphing method has been sketched in Fig. 2. The graphing method has the merits of simpleness and perceptual intuition, but lacks of precision. Moreover, it would be inconvenient for batch production of geometric models when conducting optimization. In the current study, the analytical solution is presented by computing the undetermined coefficients of the parabolic curve PE equation in Fig. 2. A general parabolic equation can be expressed as

$$\begin{cases} y^2 + 2bxy + cx^2 + 2dx + 2ey + f = 0 \\ b^2 = c \end{cases} \tag{1}$$

The coefficients are solved by the coordinate values of the points P and E, as well as the tangent slope. The vertical coordinate and the slope at the point P are denoted as m and q respectively. The coordinates of the point E are denoted as $(n,\ p)$, and p equals the radius of the cavity exit, R_o. The equation is solved by the software MATLAB, and the solution of the straight line and invalid solution should be both abandoned. The coefficients in Eq. (1) are as follows

$$\begin{cases} b = -\dfrac{mq - pq}{2m - 2p + nq} \\ c = b^2 \\ d = \dfrac{2m^2pq - nm^2q^2 - 4mp^2q + 3nmpq^2 + 2p^3q - 2np^2q^2}{4m^2 + 4mnq - 8mp + n^2q^2 - 4npq + 4p^2} \\ e = -\dfrac{2m^3 + 2m^2nq - 2m^2p + mn^2q^2 - 2mp^2 - 2np^2q + 2p^3}{4m^2 + 4mnq - 8mp + n^2q^2 - 4npq + 4p^2} \\ f = \dfrac{4m^3p + m^2n^2q^2 + 4m^2npq - 8m^2p^2 - 4mnp^2q + 4mp^3}{4m^2 + 4mnq - 8mp + n^2q^2 - 4npq + 4p^2} \end{cases} \tag{2}$$

It should be noted that a geometric constrain conditions should be satisfied, namely the tangent slope at the point P should be greater than the slope of the line segment PE.

$$\frac{p - R_t - R_f(1 - cos\beta)}{L - (R - \sqrt{R^2 - p^2}) - R_f \sin\beta} < \tan\beta \tag{3}$$

Herein, the radius of the front ball is denoted as R.

2.2. Boundary conditions

Similar to our previous study [14], the boundary conditions used in this study is shown in Table 1, and this is also referred to Saravanan et al.'s experiment [25] and Lu et al.'s study [26]. The outlet of the flow field is kept as a hypersonic one so that the outlet boundary condition is extrapolated [26]. The air is assumed to be calorically perfect gas. The jet exit orifice is set to be pressure inlet, and the jet pressure ratio (PR) is defined as follows [8,27] :

$$PR = \frac{P_{0j}}{P_{0\infty}} \tag{4}$$

Herein, P_{0j} and $P_{0\infty}$ are total pressures of the jet and freestream respectively. The jet PRs chosen in Table 1 can ensure steady-state flowfields around the hypersonic blunt body [28]. The stability of