Aerospace Science and Technology 77 (2018) 696–703

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Drag and heat reduction efficiency research on opposing jet in supersonic flows

BinXian Shen [*], WeiQiang Liu, Liang Yin

Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha 410073, China

ARTICLE INFO

Article history:
Received 9 January 2018
Received in revised form 8 March 2018
Accepted 29 March 2018
Available online 5 April 2018

Keywords:
Supersonic vehicle
Opposing jet
Drag force
Heat reduction efficiency

ABSTRACT

Opposing jet has been proven as a feasible strategy to achieve drag and heat reduction in hypersonic vehicles. In actual flight, the reduction efficiency of the drag and heat flux of opposing jet should be maximally enhanced to reduce the size of the coolant supply system, which occupies considerable space and weight. The current study investigates the drag reduction efficiency (E_f) and heat flux reduction efficiency (E_h) of unit mass opposing jet. An opposing jet flow around a blunt body is solved by using a Navier–Stokes equation with the SST k–ω turbulence model. Flow fields, drag force, and heat flux have been obtained, and typical results have been validated with experiments performed in the literature. Detailed results show that a high jet temperature is beneficial for strengthening E_f but exerts minimal influence on drag reduction. E_h is influenced by jet temperature and pressure. A high jet temperature weakens heat flux reduction in opposing jet but a reasonable jet temperature can promote the E_h though it is higher than normal temperature. The reduction efficiency of drag and heat flux can be improved with optimized jet temperature and pressure in the preconditioning requirement of drag and heat flux reduction, which benefits from saving the occupied space, reducing system weight, and promoting vehicle loading.

1. Introduction

In various engineering applications, heat transfer control is a pivotal factor which determined the effectiveness of applications [1,2]. Especially in aerospace engineering, not only the drag force but also the heat loading are essential to their design [3]. To our knowledge, the high drag force induced by shock waves and aerodynamic heating in hypersonic flows are the most serious problems in the design of hypersonic vehicles [4]. The wave drag force plays a key role in the total drag force in hypersonic vehicles, and it increases sharply with an increasing free-stream Mach number [5]. Simultaneously, maximal heat flux typically occurs at the stagnation point behind a shock wave and exhibits a strong relationship with shock wave intensity. Therefore, drag force and heat flux can be reduced by improving shock wave structures.

Numerous techniques, such as a forward-facing cavity [6], retractable aero-spikes [7] and opposing jets [8], have been proposed to improve the flow field in front of a vehicle nose. Different strategies exhibit varying characteristics in a hypersonic flow field. Basic theory indicates that opposing jet is an accessible means to achieve the drag and heat flux reduction function in hypersonic vehicles. Opposing jet from a stagnation point can push shock waves away, thereby forming a recirculation region on each side. Drag force decreases considerably when shock waves separate from the nose wall, and serious aerodynamic heating is isolated by the recirculation stream. Therefore, the opposing jet feature significantly influences drag and heat flux reduction [9].

The excellent performance in drag and heat flux reduction of opposing jet in hypersonic vehicles has been demonstrated [10,11]. The influences of operating conditions and physical dimensions, such as jet properties [12], angle attacks [13], and injector configuration [14,15], on drag and heat flux reduction have been discussed numerically or experimentally. However, the coolant supply of opposing jet has been a challenge to vehicle designers given the limited thrust and space of vehicles. This problem can be overcome by improving the gas supply system. Thus, a gas generator is considered to generate fuel gas using solid fuel on the nose tip of hypersonic vehicles. Solid fuel gas generators feature many advantages over conventional high-pressure gas holders in terms of saving space and reducing weight [16].

Another means to save space and reduce weight is to promote the efficiency of drag and heat flux reduction with unit mass opposing jet. An optimized injector configuration can promote the heat flux reduction efficiency, for example, in same work condi-

---

[*] Corresponding author.
E-mail address: shenbinxian_1603@163.com (B.X. Shen).

https://doi.org/10.1016/j.ast.2018.03.051

tions, an array of micro-jets can protect the vehicles from heat loading as opposed to a typical single jet [17]. Then, the combinatorial strategies between the opposing jet and other techniques can also improve the efficiency, such as combination of opposing jet and forward-facing cavity [18], combination of opposing jet and spike [19], combination of opposing jet and aerodisk [20].

The efficiency of unit mass opposing jet also can be promoted by improving the jet properties. On the basis of the perfect gas equation, mass flow rate is determined by the pressure and temperature of opposing jet. The aforementioned literature indicates that drag force and heat flux are reduced by increasing *PR*. However, increasing pressure causes increasing mass flow rate according to the perfect gas equation. Consequently, determining whether the efficiency of unit mass flow rate increases as increasing pressure is difficult. Jet temperature is also considered in an opposing jet thermal protection system [21,22]. The increasing jet temperature leads to decreasing heat flux reduction, but the mass flow rate also decreases with the increasing jet temperature, the relationship between the efficiency of unit mass flow rate and the jet temperature remains unclear. In brief, it is a valuable job to investigate the relationship between the efficiency of unit opposing jet and jet properties.

In the present work, we have studied the effects of opposing jet properties on the drag force and heat flux reduction. Furthermore, the influence of the jet temperature and pressure on the reduction efficiency of the drag and heat flux is investigated. Two new parameters, namely, E_f and E_h, are introduced to evaluate the efficiency of drag and heat flux reduction with opposing jet, and how operating conditions influence E_f and E_h is discussed.

2. Physical model and numerical approach

2.1. Physical model

The structures of a blunt body are shown in Fig. 1, and the diameter of the body is 50 mm. The free-stream condition is obtained according to the flying condition at a height of 25 km. The total pressure and total temperature of the free stream are 4.02 MPa and 1812 K, respectively. The flying Mach number is 6. The diameter of the jet orifice is 4 mm, which is set as the pressure inlet boundary. Jet pressure is related to free stream. The jet pressure ratio (*PR*), which has been proven helpful in establishing the relationship between opposing jet and free stream, is defined as follows:

$$PR = \frac{P_{0j}}{P_{0\infty}} \tag{1}$$

where P_{0j} and $P_{0\infty}$ refer to the total pressure of the jet and free stream, respectively. The *PR* is set to 0.075–0.25 to compare the drag and heat flux reduction efficiency of mass flow rate under different *PR* values, and the selected values can guarantee that the flow around the blunt is in stable short jet penetration mode (SPM). Jet temperature is set to 300–1500 K to investigate the influence of jet temperature on drag and heat flux reduction efficiency. The wall is assumed to be isothermal and no-slip at a temperature of 295 K. The flow is hypersonic at the outlet. Thus, the physical information of the outlet is extrapolated from the internal flow field. The detailed information of the structures and boundary is provided in Figs. 1–2 and Table 1. The opposing jet gas is set to be fuel gas. Fuel gas, which includes nitrogen, water vapor, and carbon dioxide, is shown in Table 2 and referred from a previous work [16].

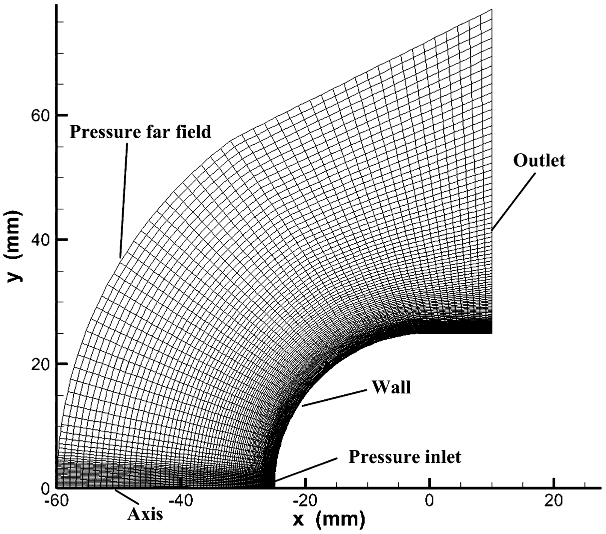

Fig. 1. Calculation region and boundary conditions.

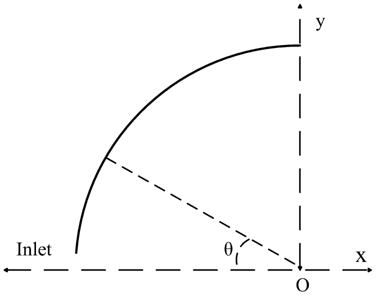

Fig. 2. Schematic for radius angle.

Table 1
Boundary conditions.

Far-field pressure	Pressure inlet	Wall
Air	Fuel gas	$T_w = 295$ K
$Ma_\infty = 6$	$Ma_j = 1$	No slip
$P_{0\infty} = 4.02$ MPa	$PR = 0.075$–0.25	
$T_{0\infty} = 1812$ K	$T_{0j} = 300$–1500 K	

Table 2
Mole fraction of the fuel gas.

Fuel gas	CO_2	N_2	H_2O (g)
Mole fraction	0.2571	0.3142	0.4287

2.2. Numerical approach

The ANSYS Fluent 16.0 working in a Dell workstation at Science and technology on Scramjet Laboratory is used to obtain the computational data. It can provide a parallel computing environment for flow solutions. For this study, the 2D axisymmetric Reynolds-averaged Navier–Stokes (RANS) equations are applied as governing equations. The equation is solved with a density-based (coupled) double precision solver. The advection upstream splitting method with a spatial second-order upwind scheme is adopted. The Courant–Friedrichs–Levy number is initially set to 0.25 to ensure stability and then increased to 4 to accelerate convergence speed.

The two-equation shear transport (SST k–ω) model of Menter is used. The SST k–ω model combines the advantages of k–ω and k–ε model, which is insensitive to the specification of free-stream turbulence level. Meanwhile, it is highly suitable in adverse pressure