Acta Astronautica 133 (2017) 50–62

Contents lists available at ScienceDirect

Acta Astronautica

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

Analysis of flowfield characteristics for equal polygon opposing jet on different freeflow conditions

Shi-bin Li*, Zhen-guo Wang, Wei Huang*, Li Yan

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

ARTICLE INFO

Keywords:
Equal polygon
Opposing jet
Drag reduction
Aero-heating

ABSTRACT

Drag and heat flux reduction efficiency induced by the equal polygon opposing jet could be higher than the one issuing from conventional circular orifice. The two-equation SST *k-w* turbulence model has been utilized to study the detailed information of flow field, and the numerical method has been validated against the available experimental data in the open literature. At last, the effects of angle of attack (α), angle of sideslip (β) and angle of jet flow (δ) on the aerodynamic characteristics have been investigated numerically in the current study. The obtained results show that the drag force increases with a higher absolute value of α and the Stanton number (St) reaches its maximum when α=−10°. Moreover, the peak St appears when α=5°. The influence of β on drag reduction shows similarity with that of α, and the maximum C_c appears when $|\beta|$=10°. More severe aero-heating occurs on the upper surface when $|\alpha|$ or $|\beta|$ changes from 0° to 15°. The δ can reduce the maximum heat flux ($Q_{\max}$) in the whole flow field and improve the thermal protection characteristics. The $Q_{\max}$ can reduce 22.5% and 9.3% at most when δ is −5° and 5° respectively.

1. Introduction

When hypersonic vehicles cruise in the region of near-space, they will have to withstand severe aero-heating environment, as well as the large drag force induced by the shock wave. This will bring about some challenges for the aircraft design. In the past decades, the wall heat flux and pressure distributions had been investigated by many researchers [1–12], and many strategies have been proposed for the drag and heat flux reduction [13–24]. Recently, some novel concepts [25–30] have been put forward due to the disadvantages of traditional schemes, such as the combinational opposing jet and aerospike concept, the combinational opposing jet and forward-facing cavity concept and the combinational forward-facing cavity and energy deposition concept [21]. Although there are many novel strategies to achieve the drag reduction and thermal protection in supersonic flows, the opposing jet is still popular because of its practicality.

The conventional blow downtype supersonic wind tunnel was used by Hayashi [31] to study the influence of total pressure ratio on aero-heating reduction induced by opposing jet. At the same time, the influence of opposing jet on thermal protection performance was also investigated in his research paper [24]. Venukumar [32] conducted an experimental study on the drag reduction characteristics of opposing jet in supersonic flow, and the maximum drag reduction of 45% could be realized. The influence of nozzle diameter ratio on aero-heating reduction had been investigated by Chen [33]. Li [30] had studied the effect of the injector configuration of opposing jet on the drag and heat flux reduction, and concluded that the pentacle shape owns the best drag reduction performance, as well as a better aero-heating characteristics than the traditional circular configuration. At the same time, Li [34] also proposed the design method for equal polygon injector and studied the influence of the jet angle number (N) on the drag and flux reduction in supersonic flow. The research shows that the performance of drag and heat flux reduction efficiency is the highest when N is 7, namely the drag and peak heat flux can decrease 23.6% and 60.6% respectively in the whole flow field.

And, there are many parameters for the opposing jet [35–37] to affect the flow field characteristics, such as the working and structure parameters, which may affect the drag reduction and thermal protection greatly. Therefore, the following two objectives will be pursed in the current study, namely (1) the visualization of the shock wave structure around blunt cone at the different free flow conditions, (2) the influence of flow parameters on the drag reduction and aero-heating of equal polygon, for example angle of attack (α), angle of sideslip (β) and angle of jet flow (δ). The details of this study are discussed in the subsequent sections, with some conclusions presented at last.

* Corresponding authors.
E-mail addresses: lishibin104@163.com (S.-b. Li), gladrain2001@163.com (W. Huang).

http://dx.doi.org/10.1016/j.actaastro.2017.01.011
Received 15 December 2016; Accepted 8 January 2017
Available online 12 January 2017

Nomenclature

α	Angle of attack
β	Angle of sideslip
δ	Angle of jet flow
N	Number of jet angle
R	Radius of the farthest position for equal polygon jet
r	Radius of the vertex position for inner-corners
θ	Cross angle for opposing jet angle
s	Area of opposing jet
χ	Ratio of r and R
Pr	Prandtl number
St	Stanton number
D	Drag force
D_O	Drag force when α and β are 0°
E	Drag reduction ratio
Q	Heat flux
P	Static pressure
T	Static temperature
ρ	Density
u	Velocity
C_P	Pressure coefficient
C_L	Lift coefficient
C_D	Drag coefficient
C_C	Cross-stream force coefficient
μ	Viscosity coefficient
γ	Specific heat ratio
L/D	Lift-to-drag ratio
Ma	Mach number

Subscripts

∞	Free-stream conditions
w	Wall conditions
aw	Adiabatic wall conditions
max	Maximum

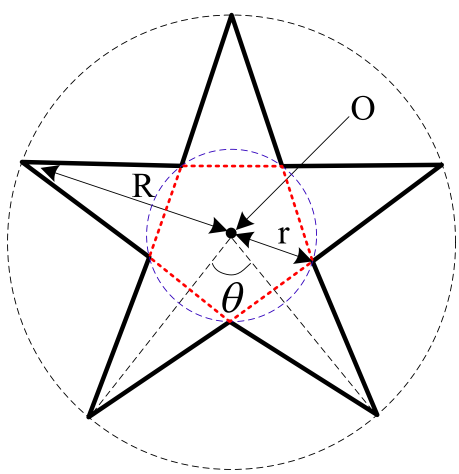

Fig. 1. Equal polygon jet project.

2. Physical model and numerical approach

2.1. Physical model

According to the rule of equal jet area [30], many different opposing jet projects have been designed. The influential region of the opposing jet on three-dimensional flow field will change by varying the shape of polygon [34], which is related with the jet angle number (N). So, the influential region of flow field will change according to our designs, and this will affect the performance of hypersonic vehicle. It can be referred in Ref. [30] for the physical mechanism of the drag and heat flux reduction when the equal polygon scheme is applied in opposing jet.

The equation of equal polygon jet area can be expressed as

$$S = R \times r \times N \times \sin\left(\frac{\pi}{N}\right) \tag{1}$$

The details are shown in Fig. 1. From Eq. (1), it can be concluded that R is inverse proportion with r when S and N are fixed. Then, we can get the values for R and r if we know the value of $\chi(\chi=r / R)$. Based on the previous research results [34,38], the study chooses the shape of N being 7 as the opposing jet scheme, and gets the geometrical model, see Fig. 2. Fig. 2 also contains the coordinate system and the given curves for the following discussion. Where, the detailed information about the physical model is listed in the Table 1.

2.2. Numerical approach

In the study, the 3D RANS Equations and shear-stress transport (SST) k-ω turbulence model [40,41] have been employed to numerically study the flow field caused by the opposing jet. The equations are solved along with the density based (coupled) single precision solver of FLUENT [40], and the wall Prandtl number for the turbulence model is set to be 0.85 in order to test its effect on the numerical results. The advection upstream splitting method (AUSM) flux vector splitting is employed to quicken the convergence speed, and the Courant–Friedrichs–Levy (CFL) number is set at 0.2, and then changed automatically with the process of convergence by using Solution Steering in FLUENT solver. The wall is assumed as the no-slip and isothermal boundary conditions and its temperature is set to be 295 K. At the outflow, all the physical variables are extrapolated from the internal cells. The solutions can be considered as converged when the residuals reach their minimum values after falling for more than six orders of magnitude, and the difference between the computed inflow and outflow mass flux is required to drop below 0.001 kg s^{-1}.

At the same time, the computational domain is structured by the commercial software ICEM CFD 14.0 [42] from University of Liverpool, and the grid is multi-blocked and highly concentrated close to the wall surfaces, and the scale for the first layer mesh is 10^{-7} m in order to ensure the accuracy of the numerical simulation that will result in a suitable value of y^+ for all the flow fields, namely its maximum value is less than 1. As for the boundary conditions, the inflow boundary is considered as the pressure-far-field [34], and the opposing jet is considered as the pressure inlet. The specific parameters are shown in the Table 2.

According to the above conditions, the Reynolds Number is 2.0×10^6