Acta Astronautica 144 (2018) 90–96

Contents lists available at ScienceDirect

Acta Astronautica

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

Thermal protection performance of opposing jet generating with solid fuel

Binxian Shen *, Weiqiang Liu

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

ARTICLE INFO

Keywords:
Supersonic vehicle
Thermal protection
Opposing jet
Fuel gas

ABSTRACT

A light and small gas supply device, which uses fuel gas generating with solid fuel as coolant gas, is introduced for opposing jet thermal protection in hypersonic vehicles. A numerical study on heat flux reduction in hypersonic flow with opposing jet is conducted to investigate the cooling efficiency of fuel gas. Flow field and cooling efficiency at different jet temperatures, as well as the effect of fuel gas, are determined. Detailed results show that shock stand-off distance changes with an increase in jet pressure ratio and remains constant with an increase in jet temperature. Cooling efficiency weakens with an increase in jet temperature and can be strengthened by enhancing jet pressure. Lastly, a remarkable heat flux reduction is observed with fuel gas injection with respect to no fuel gas injection when jet temperature reaches 900 K, thereby proving the positive cooling efficiency of fuel gas.

1. Introduction

Aerodynamic heating on the nose of hypersonic vehicles is the main limitation to the development of long-time and reusable hypersonic vehicles. Therefore, the project design of an appropriate thermal protection system for the nose of a hypersonic vehicle is important. Many techniques, for example, structural spike [1,2], opposing jet [3,4], and magnetohydrodynamics [5,6], have been proposed to improve aerodynamic heating environment in front of the vehicle nose. Different techniques exhibit varying characteristics in hypersonic flow field. Basic theory indicates that opposing jet is an accessible mean to protect the surfaces around the nose against aerodynamic heating.

The improvement of opposing jet efficiency has always been one of main topics of researchers. Opposing jet has many influencing parameters, including its operating conditions and physical dimensions. Hayashi et al. [7–9] undertook lots of studies numerically and experimentally on opposing jet in supersonic flow. Considerable heat flux reduction was observed, along with the influence of the total pressure ratio on the cooling efficiency performance of opposing jet. LU [10] numerically investigated the thermal performance of a spherical body which affected by attack angle. The influences of jet nozzle size and jet pressure on aerodynamic heating in opposing jet flow were discussed by Chen [11]. Tamada [12] demonstrated that a lower mass flow rate was required for an ogive body than for a hemispherical nose cylinder. Li [13,14] investigated the influence of nozzle shape, such as circle, square, or pentacle, on heat flux reduction in opposing jet flow. This previous study showed that the pentacle shape exhibited the best performance in heat reduction. In addition, the use of plasma jet to realize drag and heat flux reduction in hypersonic flows has been researched by scholars [15,16], and remarkable reduction has been proven by experimental and numerical results.

The heat flux reduction mechanism of opposing jet in hypersonic vehicles has been explored by many researchers, and remarkable cooling efficiency has been demonstrated. However, supplying coolant gas to opposing jet, which plays an important role in the application of opposing jet to real vehicles, is rarely studied. The weight and size of the supply system, are the key factors in real vehicles. In the current study, a gas generator similar to a solid rocket engine is considered to generate fuel gas using solid fuel on the nose tip of hypersonic vehicles. The solid fuel gas generator has many advantages over the conventional high-pressure gas holder, which is heavy and bulky. This generator is extensively used in certain diminutive devices, such as airbags in automobiles and solid micro-thrusters in microsatellites [17], because it can provide sufficient gas through a small and light device.

The fuel gas generating with solid fuel is considered the coolant gas of opposing jet. But the temperature of the jet is significantly higher than room temperature. The aerodynamic drag reduction caused by hot fuel gas was experimentally and numerically investigated by Ganiev et al. [18]. However, the heat flux reduction mechanism of hot fuel gas has not yet been explored. In the present study, the effect of fuel gas on the cooling efficiency of opposing jet is determined via numerical simulation. Remarkable aerodynamic heating reduction is realized due to fuel gas.

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

https://doi.org/10.1016/j.actaastro.2017.12.024
Received 28 June 2017; Received in revised form 17 November 2017; Accepted 15 December 2017
Available online 19 December 2017
0094-5765/

2. Fuel gas system for opposing jet

The structure of the fuel gas supply system is shown in Fig. 1. The system mainly comprises a gas generator, an igniter, and a nozzle. The gas generator and the igniter are built into the blunt nose, and solid fuel is stored in the gas generator. The exit of the gas generator is directly connected to the contraction nozzle to reduce occupied space. When the system works, the igniter first ignites the solid fuel. Then, the solid fuel burns rapidly and generates a considerable amount of fuel gas. Fuel gas is injected through the nozzle at a required pressure and speed to push away shock waves. Compared with coolant supply systems in ground experiments that comprise a high-pressure gas holder, a solenoid, a pressure reducing and other assembly units, the units of the fuel gas supply system are integrated into the gas generator, which better economizes space and weight. Moreover, the gas generator has advantages in terms of response speed over conventional means.

The temperature of fuel gas should be as low as possible to achieve effective cooling efficiency. However, fuel gas with room temperature is difficult to obtain. Several low-temperature solid fuels are listed in Table 1 [18–21], which shows that combustion temperature is approximately 700 K–1300 K. In this study, ADC/BCN/CuO is considered solid fuel, and the temperature of fuel gas is assumed to be 900 K.

3. Physical model and numerical approach

3.1. Physical model

The sphere-conical body which refers to Saravanan's experiment is used as the geometric models in the current study [22]. The diameters of the sphere and jet orifice are 30 mm and 2 mm, respectively. The sizes of the other structures are the same as those in the experiment of Saravanan.

3.2. Boundary conditions

Fig. 2 shows the locations of the boundary and Table 2 shows the boundary information. The parameters of free stream are also obtained from the study of Saravanan. The wall is assumed to be isothermal and no-slip at a temperature of 300 K. The flow at outlet is hypersonic, so the physical information of the outlet are extrapolated from the internal flow-field. Fuel gas, including water vapor, is unsuitable for simulating conditions when jet temperature is at room temperature. Air and fuel gas are used as opposing jet in different cases and are assumed to be thermally and calorically perfect. To describe the jet pressure of opposing jet, the jet PR is defined as follows:

$$PR = \frac{p_{0j}}{p_{0\infty}}, \quad (1)$$

where P_{0j} and $P_{0\infty}$ refers to the total pressure of the jet and the free stream, respectively. The jet PR is set to be 0.07, 0.14, which guarantees a steady flow around the sphere–conical body [23]. The jet temperature is set to 300, 600, and 900 K to evaluate the effect of temperature on the heat flux reduction mechanism. The assumed temperature of fuel gas selected in this study is 900 K.

3.3. Numerical approach

The two-dimensional axisymemetric Reynolds-averaged Navier–-Stokes (RANS) equations are applied as governing equation. The equation are solved with the density-based (coupled) double precision solver that uses Fluent. Advection Upstream Splitting Method (AUSM) with spatially second order upwind scheme is adopted. The Courant–Friedrichs–Levynumber (CFL) is initially set as 0.25 to ensure stability and

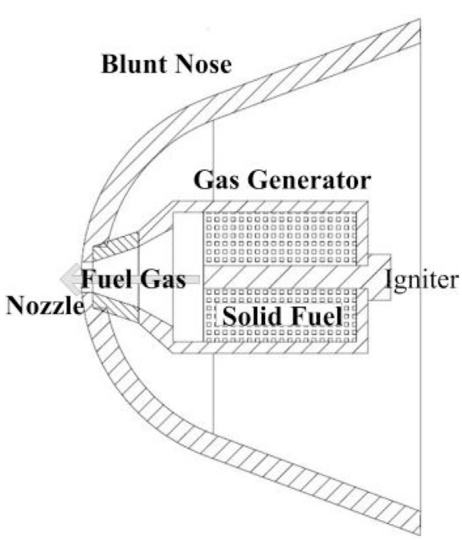

Fig. 1. Schematic of fuel gas opposing jet thermal protection system.

Table 1
List of solid fuels.

Solid fuel	Combustion temperature (K)	Combustion pressure (MPa)
GN/BCN/$NaHCO_3$/fluoride rubber	955	10
GN/BCN/GZT/Fluoride rubber	1276	25
ADC/BCN/CuO	972	8.73
5-AT/CuO	970	
NaN_3/LiF/Na_2Si0_3	700	5
PAK/KNO3/CuO	1310	

GN: guanidine nitrate; BCN: basic cupric nitrate; GZT: guanidinium azotetrazolate; ADC: azodicarbonamide; 5-AT: 5-aminotetrazole.

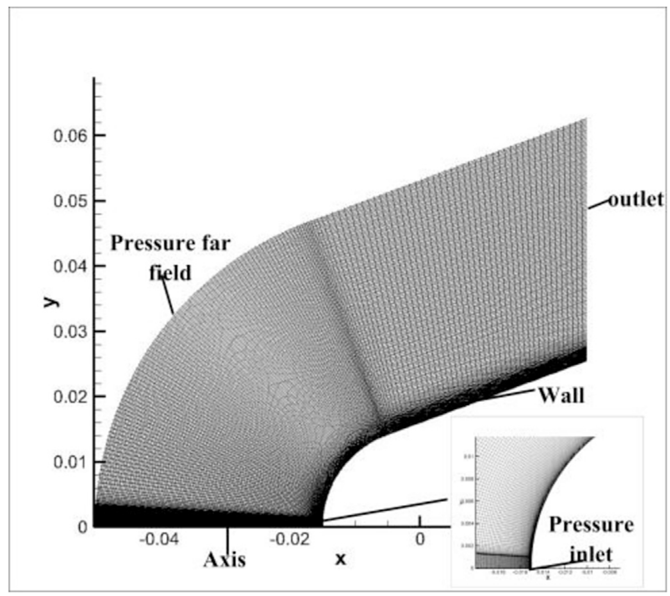

Fig. 2. Geometry and boundary conditions.

Table 2
Boundary conditions.

Far-field pressure	Pressure inlet	Wall
Air	Air/fuel gas	$T_w = 300$ K
$Ma_\infty = 7.96$	$Ma_j = 1$	No slip
$p_{0\infty} = 1939211$ Pa	$PR = 0.07, 0.14$	
$T_{0\infty} = 1955$ K	$T_{0j} = 300$–900 K	