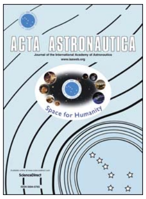
Acta Astronautica 132 (2017) 230–242

Contents lists available at ScienceDirect

Acta Astronautica

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



Fluid-thermal analysis of aerodynamic heating over spiked blunt body configurations

Qihao Qin, Jinglei Xu*, Shuai Guo

Jiangsu Province Key Laboratory of Aerospace Power System, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China

ARTICLE INFO

Keywords:
Spiked blunt body
Aerodynamic heating
Fluid-thermal analysis
MpCCI

ABSTRACT

When flying at hypersonic speeds, the spiked blunt body is constantly subjected to severe aerodynamic heating. To illustrate the thermal response of different configurations and the relevant flow field variation, a loosely-coupled fluid-thermal analysis is performed in this paper. The Mesh-based parallel Code Coupling Interface (MpCCI) is adopted to implement the data exchange between the fluid solver and the thermal solver. The results indicate that increases in spike diameter and length will result in a sharp decline of the wall temperature along the spike, and the overall heat flux is remarkably reduced to less than 300 W/cm^2 with the aerodome mounted at the spike tip. Moreover, the presence and evolution of small vortices within the recirculation zone are observed and proved to be induced by the stagnation effect of reattachment points on the spike. In addition, the drag coefficient of the configuration with a doubled spike length presents a maximum drop of 4.59% due to the elevated wall temperature. And the growing difference of the drag coefficient is further increased during the accelerating process.

1. Introduction

In hypersonic regimes, the blunt body is widely used on re-entry vehicles and long-range missiles for its lower aerodynamic heating levels ($q_w \sim 1/D^{0.5}$, where q_w denotes the heat flux in the wall and D is the diameter of the blunt body) and larger available space compared with the pointed body [1-3]. But the rise of pressure drag is inevitable due to the strong detached bow shock generated ahead of the blunt body, which increases the fuel usage and reduces the payload. Hence, it is a tough task to minimize the drag and aerodynamic heating level at the same time. In order to reduce the drag and heat flux further, a lot of techniques have been proposed [4], including the forward-facing cavity, the forward-facing spike, the opposing jet, etc. Among of them, the spiked blunt body has been considered as the most promising for its simpleness and efficiency, and has been focused for a long time.

Earlier researches of the spiked blunt body are mainly based on the comparison of experimental results and theoretical calculations. Dem'ianov and Shmanenkov [5] and Belov [6] theoretically pointed out the significance of recirculation zones in front of the blunt body in reducing the drag. Kenworthy [14] systematically studied the axisymmetric unsteady separation in front of the blunt body. Moreover, Zapryagaev and Kavun [7] illustrated the influence of length and spike cone angle on the flow structure and defined the character of the pulsation flow mode. The first aerodynamic heating study of the spiked blunt body can be traced back to 1954. Stalder and Nielsen [8] experimentally examined the heat transfer and pressure distribution of the spiked hemisphere-cylinder. Heat transfer to the spiked model was doubled compared with the unspiked one regardless of the spike length and tip geometry. The best configuration with spike was found to be able to reduce the pressure drag approximately 45%. Then, Crawford [9] conducted an extensive investigation of the drag and aerodynamic heating on the spiked hemisphere-cylinder of variable length and Reynolds number at Mach 6.8. The results indicated that the heat transfer was greatly influenced by the flow type over the separated boundary and the ratio of drag to heat transfer was reduced due to the existence of the spike regardless of the spike length and transition location. Holden [10] also conducted rich experimental investigations to study the heat transfer to the spiked blunt body. He found that the aerodynamic heating level was determined by the reattachment angle and larger angle yielded smaller peak value of heat flux.

Numerical investigations of the spiked blunt body began in the late 1980s and attracted more interests in recent years. Yamauchi et al. [11] examined the effects of spike length, Mach number and angle of attack

---

Abbreviations: APIs, Application Programming Interfaces; MpCCI, Mesh-based parallel Code Coupling Interface; RANS, Reynolds-Averaged Navier-Stokes; SST, Shear Stress Transport

* Corresponding author.

E-mail addresses: qinqh@nuaa.edu.cn (Q. Qin), xujl@nuaa.edu.cn (J. Xu), shuaiguo0916@nuaa.edu.cn (S. Guo).

http://dx.doi.org/10.1016/j.actaastro.2016.12.037
Received 19 October 2016; Accepted 29 December 2016
Available online 30 December 2016
0094-5765/ © 2016 IAA. Published by Elsevier Ltd. All rights reserved.

Nomenclature

A_{ref}	Reference area
A_1, B_1, C_1	Separation point
A_2, B_2, C_2	Reattachment point
C_d	Drag coefficient
C_p	Pressure coefficient
D	Blunt body diameter
E	Total energy [J/kg]
$\vec{F_c}$	Vector of convective flux
$\vec{F_v}$	Vector of viscous flux
H	Total enthalpy [J/kg]
L	Spike length
Ma	Mach number
Pr	Prandlt number
Q	Heat source
R	Gas constant
S	Control surface
T	Temperature [K]
$\vec{U}$	Vector of convective flux
$\widetilde{U}$	Contravariant velocity
V	Control volume
X, Y, Z	Temperature peak point
X_d	Aerodynamic drag
c	Specific heat capacity [J/(kg K)]
d	Spike diameter
$\vec{n}$	Unit normal vector
p	Pressure [Pa]
q	Heat flux [W/cm^2]
t	Time
Δt	Time step size
u	Velocity
y^+	Nondimensional wall distance

Greek scripts

γ	Specific heat ratio
δ	Semi-cone angle
θ	Included angle
λ	Thermal conductivity [W/(m K)]
μ	Viscosity [N s/m^2]
ρ	Density [kg/m^3]

Subscripts

r	Radial direction
s	Structure
w	Wall
x	Axial direction
∞	Freestream condition

on the spiked blunt body by solving the three-dimensional thin-layer compressible Navier-Stokes equations. And a good agreement was achieved between the computational results and Crawford's experimental data [9]. Feszty et al. [12,13] numerically analyzed the driving mechanism of two unsteady flow modes rising over axisymmetric spiked bodies, which was an exact reproduction of the experimental investigation of Kenworthy [14]. Gauer and Paull [15] compared the drag and heat transfer reduction of a spike with variable length and shape with the unspiked nose cone at three chosen mission points, using the flow solver CFD-FASTRAN. Gerdroodbary and Hosseinalipour [16] simulated the effectiveness of cut spike, sharp spike, flat aerodisk spike and hemispherical disk spike at various angles of attack. Mansour and Khorsandi [17] modeled the hypersonic flow around the spike by solving the Reynolds-Averaged Navier-Stokes (RANS) equations and a drag reduction of 40% was observed compared with the unspiked body. Isaev et al. [18–20] conducted several simulations to investigate the drag reduction characteristics of the spiked blunt body. They found that the disk-cylinder-disk arrangement brought about 12% decrease of base drag compared to the variant without a rear disk. And a small-sized disk placed in front of the Soyuz-type missile helped to decrease the drag more than 30%. In addition, for cylindrical bodies with protruding rod-supported disk, the increase in rod length also helped to reduce the drag. Ahmed and Qin [21–26] conducted extensive researches on the spiked blunt body. They simulated the flow field over conical, disk and flat spiked bodies and investigated the flow asymmetry around axisymmetric spiked blunt bodies at hypersonic speeds. An explanation for the mechanism of drag reduction and the cause of flow instability was given through the concept of “effective body”. In addition, they conducted surrogate-based multi-objective optimization for the spiked blunt body to minimize the drag and aerodynamic heating effect. The results showed that the aerodynamic heating level was mainly determined by the forebody shape whereas the spike length controlled the drag response.

In most of previous numerical simulations, the fluid-thermal interaction between the spiked blunt body and the external flow field was neglected. The surfaces of the model were commonly assumed to be adiabatic or isothermal [27], which omitted the structural thermal response to the aerodynamic heating of freestream and thus failed to provide accurate thermal boundary conditions for the flow field. According to the theoretical results of Wood [28], however, an increasing spike temperature could result in the rise of the recirculation zone temperature, which led to an outward movement of the reattachment point. It means that the external flow features can be affected by the temperature distribution along the spike and thus the dynamic performance of the spiked blunt body changes. Therefore, it is of great importance to conduct fluid-thermal analysis in the study of spiked blunt body at hypersonic speeds.

The objectives of this paper are firstly, to obtain the instantaneous thermal response of the spiked blunt body under both steady and unsteady freestream conditions. Secondly, to analyze the impact of spike shapes on the distribution of the heat flux and temperature along the coupling surfaces. Thirdly, to make an explanation to the presence and movement of the peak values in the heat flux and temperature evolution. Finally, to illustrate the variation of pressure distribution and drag coefficient due to the elevated wall temperature. So the paper is structured as follows. Section 1 is a brief introduction to the previous researches on the spiked blunt body. Section 2 introduces the methodology and modeling. The results and findings are presented in Section 3. The paper finalizes with the main conclusions in Section 4.

2. Methodology and modeling

2.1. Governing equations

To perform fluid-thermal analysis, the partitioned approach is adopted due to its flexibility and efficiency for most aerodynamic heating problems, compared with the monolithic approach [29,30]. In the partitioned approach, the fluid and structure are solved individually and the simulation of heat transfer process is implemented through the exchange of convective heat flux and wall temperature across the fluid-structural interfaces.

The unsteady viscous compressible flow is modeled by solving RANS equations. Because of the axial symmetry of the spiked blunt body configurations and the corresponding flow field at zero angle of attack, the flow is assumed to be axisymmetric with circumferential swirl and rotation omitted. The integral form of the governing equation

Download English Version:

https://daneshyari.com/en/article/5472571

Download Persian Version:

https://daneshyari.com/article/5472571

Daneshyari.com