Acta Astronautica 143 (2018) 92–104

Contents lists available at ScienceDirect

Acta Astronautica

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

Heat transfer reduction using combination of spike and counterflow jet on blunt body at high Mach number flow

Z. Eghlima, K. Mansour *, K. Fardipour

Aerospace Engineering Department, Amirkabir University of Technology, Tehran, Iran

ARTICLE INFO

Keywords:
Hypersonic
Spike
Counterflow jet
Heat transfer reduction

ABSTRACT

Heat transfer reduction around blunt bodies is one of the important issues in the field of high speed aerodynamics. Using of spikes and counterflow jets each of them separately for reducing of drag force and heat transfer is well known. The present work is description of flow field around a hemispherical nose cylinder with a combination of spike and counterflow jet at free stream of Mach number of 6. The air gas was injected through the nozzle at the nose of the hemispherical model at sonic speed. In this numerical analysis, axisymmetric Reynolds-averaged Navier-Stokes equations was solved by k-ω (SST) turbulence model. The grid study was done and the results are validated with experimental results for spiked body without jet condition. Then the results presented for different lengths of spike and different pressures of counterflow jets. The results show a significant reduction in the peak heat transfer about 60%–78% for different models compared to the spherical cylinder model without any jet and spike. Furthermore, also our results indicate that the heat transfer reduction is increased even more with increasing of the length of the spike.

1. Introduction

A hypersonic flow over a blunt body leads to a bow shock wave in front of it, which causes a rather high pressure drag and consequently high surface heat transfer. The reduction of drag force and heat transfer in aerodynamic applications by a spike or a jet spike on a blunt-nosed body at supersonic and hypersonic flows is well known and was studied past century [1–7]. The wave drag reduction by using a spiked or opposing jet is derived from both the splitting of a single strong shock into multiple shock waves and effectively replacing the blunt body by a slender displacement. Even if the accumulative pressure rise across the multiple and sequential shock wave is identical to that of a single shock, the entropy jump across the multiple wave system is much less. This difference is because of proportionality of the cubic power of the pressure jump due to the entropy increment across each shock wave. The blunt body with injection will, thus, produce a lower wave drag. There are also disadvantages of these drag reduction devices in that they can induce unsteady motion with large-amplitude oscillations through free-shear-layer instability.

Flow behind the conical shock wave separates on the spike and creates a conical shaped recirculation zone appears in the vicinity of the stagnation region. Due to the formation of the flow recirculation, the pressure and wall heat flux are reduced in the forward facing region of the blunt body. However, the reattachment of the shear layer on the shoulder of the hemispherical body increases the local heat flux and pressure. The reattachment shock is moved downstream as depicted in Fig. 1, which is a function of the geometrical parameter of the spike and the shape of the spiked nose body.

Fig. 2 shows a typical jet issuing from a body against a supersonic airstream. The bow shock stands away from the body surface, and takes a form appropriate to a new body consisting of the original body with a protrusion due to the jet flow. The boundary of this protrusion is defined by the interface, the stream surface between the jet flow and the mainstream flow.

1.1. Previous studies about counterflow jet

The opposing jet in supersonic flows has been considered a lot because of its wide applications on drag and heating reduction at supersonic and hypersonic flows. The experiments on a jet from a blunt body opposing supersonic flows mainly investigated the mean flow quantities, such as the pressure distribution on the body surface, the bow shock stand-off mean position and the shock structures complexity [1,7–9]. These studies revealed that the total pressure ratio of the jet to the free stream is a key parameter affecting the aerodynamic features. In addition, complex sustained motions of the flow field were observed

* Corresponding author. Tel.: +98-2164543216.
E-mail address: Mansour@aut.ac.ir (K. Mansour).

https://doi.org/10.1016/j.actaastro.2017.11.012
Received 18 October 2017; Accepted 13 November 2017
Available online 14 November 2017

Nomenclature

Cd	Drag coefficient
C_P	Pressure coefficient
d_j	Diameter of the jet section
D	Diameter of the main sphere
e	Total energy per unit mass
k	Turbulent kinetic energy
H	Enthalpy
h	Heat transfer coefficient
L	Length of the spikes
M	Mach number
p	Pressure
q	Heat flux
PR	Ratio of jet to free stream total pressures
R	Radius of base of model
S	Source term
T	Temperature
u	Velocity in x-dir
v	Velocity in y-dir
w	Velocity in z-dir
Y	Dissipation term
γ	Specific heat ratio
ε_m	Turbulent kinetic energy
θ	inclination of the dividing streamline
μ	Dynamic viscosity
ρ	Density
σ	Turbulent Prandtl number
τ	Stress tensor
Ω	Mean vorticity
ω	Specific dissipation rate

Subscripts

0	Due to base model (without any jet and spike)
f	Local fluid properties
j	Jet
O	Total or stagnation value
rad	Due to radiation
w	Wall condition
∞	Free stream

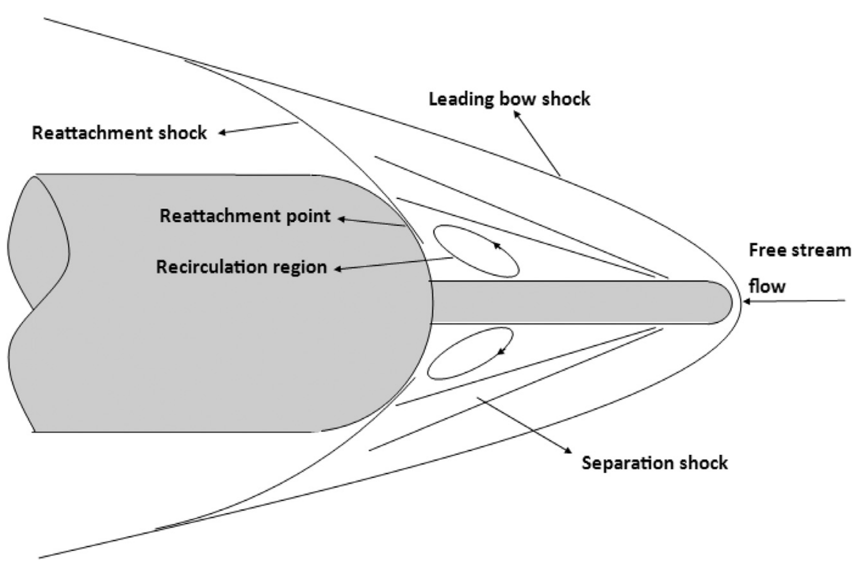

Fig. 1. Principal features of the high speed flow around a schematic spiked spherical nose cylinder.

experimentally in some jet conditions.

Recently, with the development of the aeronautics and astronautics, the advantage of opposing jet is more appealing to the researcher. In this century, some scholars kept doing research on this method. Hayashi et al. [10,11] did the numerical and experiment studies of thermal protection system by opposing jet and obtained some valuable conclusions. This paper showed that as the pressure ratio was increased, the heat flux was decreased at each point and the remarkable reduction of aerodynamic heating was also observed in nose region. The high precise simulation of Navier-Stokes equations was used by Tian [12] to study the detailed influences of the free Mach number, jet Mach number, attack angle on the heat flux reduction and the mechanism was discussed.

1.2. Previous studies about aerospike

Flow fields around a spiked body were experimentally investigated in the 1950s. For example, flow fields around a spiked blunt body at Mach number 6.8 were experimentally investigated by Crawford in 1959 [2]. Researches continued in the last century about aerospikes [3,5,6]. Yamauchi et al. [13] in 1995 have numerically studied the flow field around a spiked blunt body at free stream Mach numbers of 2.01, 4.14 and 6.80 for different ratio of L/D (Length of spike to base diameter). Mehta et al., in 2000 [14] calculated the flow field around a forward facing spike attached to a hemisphere-cylinder nose tip at a free stream Mach number of 6.8 for different spike lengths. They observed that the shear layer created by the spike passes through the reattach-ment shock wave giving the peak wall pressure and heat flux on the blunt body, which is influenced by the conical–shock reattachment interaction. Asif and Zahir in 2004 [15] studied supersonic flow ($M_\infty = 1.8$) and hypersonic flow ($M_\infty = 5$, 6.8, 8) around a blunt nose body with the attachments of 4 forward facing spikes and estimated aerodynamic forces using CFD tool, PAK-3D. In this paper, four different geometries of spikes and two different lengths have been examined to study the forebody flow and its effects on static aerodynamics coefficients. In 2010, Kalimuthu and