Acta Astronautica 133 (2017) 103–110

Contents lists available at ScienceDirect

Acta Astronautica

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

Research paper

Drag reduction for the combination of spike and counterflow jet on blunt body at high Mach number flow

Z. Eghlima, K. Mansour*

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

A R T I C L E I N F O

Keywords:
Hypersonic
Spike
Counterflow jet
Drag reduction

A B S T R A C T

Drag reduction at high speed flows around blunt bodies is one of the major challenges in the field of aerodynamics. Using of spikes and counterflow jets each of them separately for reducing of drag force is well known. The present work is description of flow field around a hemispherical nose cylinder with a new 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 results were 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 drag coefficient about 86–90% compared to the spherical cylinder model without jet and spike for practical models (L/D=1.5 and 2). Furthermore also our results indicate that the drag reduction is increased even more with increasing of the length of the spike.

1. Introduction

The drag reduction in aerodynamic applications by a spike or a jet spike on a blunt-nosed body at supersonic and hypersonic flows is well known and were studied years ago [1–6]. 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.

Fig. 1 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 [1,7,8]. 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 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 [8–10] did the numerical and experiment studies of thermal protection system by opposing jet and obtained some valuable conclusions. The high precise simulation of Navier-Stokes equations was used by Tian [11] 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.

---

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

http://dx.doi.org/10.1016/j.actaastro.2017.01.008
Received 16 November 2016; Accepted 6 January 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	Total enthalpy
h	Enthalpy
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)
j	Jet
O	Total or stagnation value
w	Wall condition
∞	Free stream

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 Crawford1 in 1959 [2]. Yamauchi et al. [12] 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 in 2000 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 [13]. Asif and Zahir in 2004 studied supersonic flow (M_∞=1.8) and hypersonic flow (M_∞=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 [14]. 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 2009, Mehta again studied numerically the effect of the various types of aerospike configurations on the reduction of aerodynamic drag and wall heat flux, this time at a length to diameter ratio of 0.5, at Mach 6 and at a zero angle of incidence [15]. In 2010, Kalimuthu and Mehta [16] experimentally studied the pressure variation on the blunt nose body and the aerodynamic coefficients such as drag, lift and pitching moment over the forward facing hemisphere aero spike at Mach number 6. In 2014, Mansour and Khorsandi [17] investigated numerically the surface pressure distribution and drag coefficient of a blunt nose cone with an aerospike in hypersonic free stream Mach number of 6. The geometric model they studied was the same model in ref.[16] and they used k-ε turbulence model.

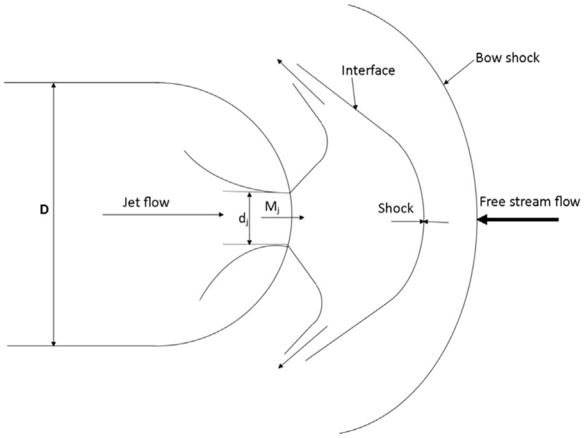

Fig. 1. - Principal features of the counterflow jet around a schematic spherical nose cylinder.

1.3. Previous studies about combination of the jet and spike

The concept of combination of these two methods is new; counterflow jet and aerospike. Jiang at 2009 [18] conducted Experiments in a hypersonic wind tunnel at a nominal Mach number of 6. It is shown previously that the shock/shock interaction on the blunt body is avoided due to injection and consequently the peak pressure at the reattachment point is reduced by 70% under a 4° attack angle. Wei Huang et al. at 2015 [19] investigated the influences of length-to-diameter ratio of aerospike and jet pressure ratio on the drag reduction by combinational opposing jet and aerospike concept at supersonic Mach number of 2.5. The rejecting gas was nitrogen. The maximum drag reduction coefficient is 65.02% compared to the without spike conditions (with jet), and it occurs when the jet pressure ratio is 0.4. The peak pressure location moves nearly from 40 to 55 deg, and it is nearly the same irrespective of the variation of the jet pressure ratio. Anyway, this method of combination has not been applied to the hypersonic flows previously.

Present study has calculated flow field around a hemispherical nose cylinder model in the free stream of Mach number of 6 without and with a counterflow jet at sonic speed. The material injecting to the jet is air and it is injected from the top of the spike's nose. The results for without jet conditions are validated with experimental and numerical studies in the given refs. [16] and [17]. The results show a significant reduction in pressure distribution on the surface and drag coefficient.

In this numerical analysis, axisymmetric Reynolds-averaged Navier-Stokes equations were solved by using of k-ω (SST) turbulence model. The results are presented for spiked blunt body without and with a counterflow jet with different length to base spike ratios and jet to free stream pressure ratios.

The purpose of this study is to show how much drag reduction produces by counterflow jet adding to the spiked blunt body at free stream Mach number of 6. Actually in addition to drag reduction due to the spike attached with a blunt body, what is the effect of counterflow jet on the pressure coefficient over main sphere at different pressures of jet and lengths of spike?