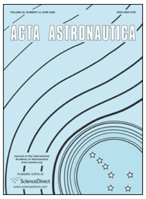
Acta Astronautica 99 (2014) 92–98

Contents lists available at ScienceDirect

Acta Astronautica

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



The drag reduction in spherical spiked blunt body

Kamyar Mansour*, Mahdi Khorsandi[1]

Aerospace Engineering Group, Amirkabir University of Technology, Tehran 15875-4413, Iran

ARTICLE INFO

Article history:
Received 8 December 2013
Received in revised form
9 February 2014
Accepted 12 February 2014
Available online 28 February 2014

Keywords:
Spike
Blunt body
Drag
Stand-off distance
Numerical solution

ABSTRACT

When flying at hypersonic speeds, it is a fundamental requirement to reduce the high drag resulting from a blunt nose cone. An aero spike can be attached on the front of the nose cone to obtain a high drag and heat load reduction.

In this paper hypersonic flow with Mach number 6 around a spike is numerically simulated. The standard k–ε model was used for solving corresponding Reynolds Average Navier–Stoke equations. The computed results show that the drag coefficient of the spiked blunt body is reduced respect with no spiked body. Our numerical calculations also agrees with corresponding experimental work. Moreover the computed shock stand-off distance is in reasonable agreement with other related works.

1. Introduction

Vehicles like space plane, reusable launch vehicles, missiles, interplanetary probes, which fly at supersonic and hypersonic speeds, usually employ blunt nosed bodies for deceleration and thermal management. A blunt nosed body creates a bow shock wave at high Mach number, which in turn produces very high pressure in the forward region of the fore body, leading to a high wave drag and aerodynamic heating during the atmospheric flight.

It is advantageous to have a vehicle with low drag coefficient in order to minimize the thrust required from propulsive system during the supersonic and hypersonic regime. There is a need for trade-off between thermal protection system mass and high wave drag associated with such high speed vehicles. A spike in front of the blunt nose offers an effective means of such drag reduction. Flow past the spike creates a conical shock wave and remains away from the body. Flow behind the conical shock wave separates on the spike and a conical shaped re-circulation zone forms in the vicinity of the stagnation region. Due to the recirculation, the pressure and wall heat flux decrease in the forward facing region of the blunt nose. However, the reattachment of the shear layer on the shoulder of the hemispherical body increases the local heat flux and pressure. Of course the flow structure was well studied in the past we just overview it.

At high-speed flow past a blunt body generates a bow shock (Fig. 1) which causes a rather high surface pressure and as a result the development of a high aerodynamic drag. The pressure on the body surface can be substantially reduced if, instead of a normal shock, an oblique shock is generated by an aero spike.

Many numeric and experimental studies on spiked bodies have been carried out to study the effects of aerodynamic heating and the surface pressure distributions at supersonic and hypersonic Mach numbers [1–9]. The conventional aero spike produces a region of recirculating separated flow [10] that shields the nose from the oncoming flow, and reduces aerodynamic drag [11]; it also reduces a wall heat flux and protects the surface with electronic embedded sensors. The separated flow may

---

* Corresponding author. Tel.: +98 21 64543216.
E-mail addresses: mansour@aut.ac.ir, kamy_mansour@yahoo.com (K. Mansour), m.khorsandi@aut.ac.ir (M. Khorsandi).

[1] Tel.: +98 21 22480075.

http://dx.doi.org/10.1016/j.actaastro.2014.02.009
0094-5765

become unstable under certain conditions pertaining to the spike geometry, diameter of the hemispherical cap and free stream Mach number.

Crawford [1] experimentally investigated the effects of the spike length on the nature of the flow field for a free stream Mach number 6.8 and Reynolds number 0.12×10^6 and 1.5×10^6 based on the cylinder diameter. Spike drastically influences the aerodynamic drag of the blunted body in the high-speeds. However, because of the reattachment of the shear layer on the shoulder of the hemispherical body, the pressure near that point becomes large.

The Experimental investigation of spiked-nose bodies at a Mach number of 14 and a Reynolds number of 0.365×10^6 [2] has shown the heat transfer to be less than half the blunt value. The flow over the separated region and body was believed to be entirely laminar.

Milichev et al. [4] have experimentally investigated the influence of four different types of spikes attached to a hemisphere-cylinder body at Mach number 1.89, Reynolds number 0.38×10^6 based on the cylinder diameter, and angle-of-attack 2 deg. They observed in the experimental studies that the reliable estimating of aerodynamic effects of the spike can be made in conjunction with flow visualization technique.

Yamauchi et al. [5] have numerically investigated 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 where L is the spike length and D is the blunt diameter.

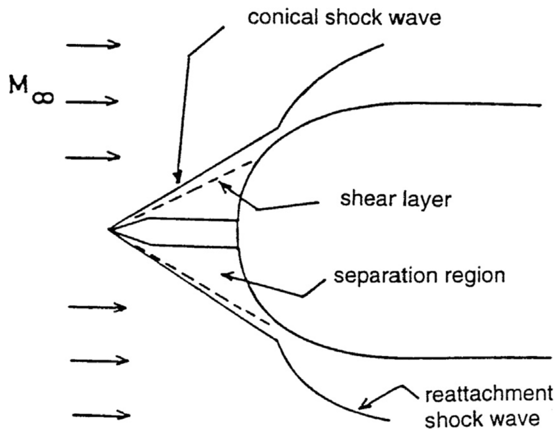


Fig. 1. Flow field over spiked blunt.

Mehta [6,8,9] and Asif [7] used different codes and software to solve the compressible viscous Navier–Stokes equations and investigated the effect of spike on aerodynamic and thermodynamic characteristic on blunt body of different spike, angle of attack and Mach number.

Kalimuthu [12] has experimentally studied at measuring 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. We add there has been continued interest in the application of active flow control concepts to modify or change the external flow fields of vehicles in order to reduce wave drag and aero-thermal loads [13] and of course spike is an alternative.

2. Geometric characteristics

Model is a hemisphere nose with cylinder body. The diameter of the cylinder, D, is 40 mm and length of the parental body is $1.25D$. Hemisphere aero spike configurations were used in this study. The dimensions of the hemisphere spike body considered in the present investigation are shown in Fig. 2. The spike consists of a hemispherical nose and a cylindrical part. The diameter of the cylindrical spike is $0.1D$. Two spikes of length (L) 1.5 and 2D were tested. Spikes were attached to the nose of the parental body using a screw.

3. Grid generation

The geometry is modeled using a structured grid. In this study, there is no side slip angle, hence only 180-deg grid is needed to simulate the flow conditions and axisymmetric condition is applied through the center of the geometries. Grid is densely packed towards the body to capture the boundary layer due to the viscous and turbulence effects. Fig. 3 shows a typical structured grid around spike in three views. To demonstrate that the results are grid-independent, we carried out numerical simulations for several grids with different sizes.

4. The equations and numerical method

The time-dependent axisymmetric compressible Navier–Stokes equations in general are written in integral form, the system of equations is augmented by the ideal gas law for

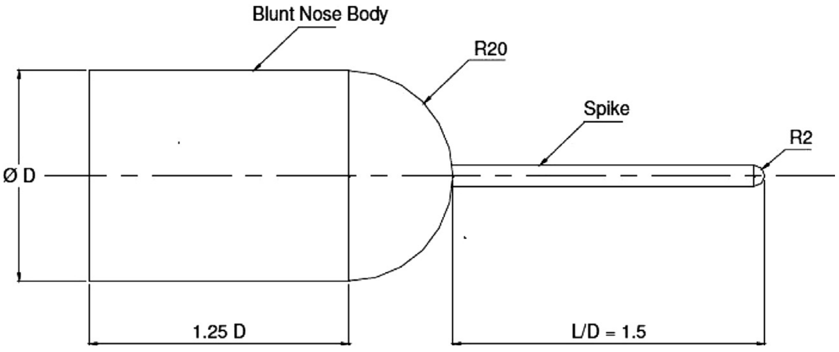


Fig. 2. Geometry of the model.