Aerospace Science and Technology 52 (2016) 256–268

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

The effects of cross section variations on the pressure distribution around a long axisymmetric body

Mohammad Reza Heidari

Aerospace Engineering Faculty, Parand Branch, Islamic Azad University, Tehran, Iran

ARTICLE INFO

Article history:
Received 24 October 2015
Received in revised form 9 February 2016
Accepted 4 March 2016
Available online 9 March 2016

Keywords:
Supersonic flow
Pressure distribution
Boundary layer tripping
Cross section variation
Long axisymmetric body
Wind Tunnel Test

ABSTRACT

In the present study, the impacts of embedding belts are experimentally studied for a supersonic flow field around a long axisymmetric body. Also, the longitudinal and circumferential pressure coefficients along with the boundary layer profile are investigated for −2 to 6 degrees angles of attack. To this end, two conical–cylindrical belts were installed at the middle and end parts of ogival cylinder model. To ensure the turbulent flow around the model, a trip strip causing artificial disturbances is utilized and the acquired results are compared with the acquired results from the model without a trip strip. To study the effects of the cross section variations on the pressure distribution as well as the boundary layer profiles, three different belts with various leading edge angles were installed at different locations along the cylindrical section of the model. These belts significantly affected both the surface pressure distribution and the boundary layer profile. Passing of flow over the belt leads to sudden variations in pressure coefficient distribution on the belt that is due to sudden cross section variations and consequent development of reverse flow or flow separation regions and production of vortices along the flow path. Also, the presence of belt leads to the development of oblique shock wave on the model, which in turn reduces the Mach number downstream the belt. Studying the circumferential pressure distribution reveals that the presence of belt leads to more asymmetric flow downstream the model, which is intensified as the angle of attack increases.

1. Introduction

The required space for the apparatus, systems, etc. of long-body aeronautical vehicles is among the major parameters affecting length and diameter of the vehicle. Increasing this space causes an increase in both the body length and the fineness ratio (L/d) [1]. For these types of bodies, the problem of flow separation and boundary layer growth at various flight conditions is worth significant consideration. If the flow over the control surfaces is separated, their performances are significantly affected, especially for those located close to the end of the body. Boundary layer growth and its separation, affect the aerodynamic characteristics, particularly drag force and stability criterion, both of which have important roles in the vehicle performance and its mission implementations [2].

Considering the lack of direct solutions in turbulence numerical modeling and regarding this fact that experimental studies are not able to show all details of flow field, it can be concluded that a combination of experimental and numerical methods should be used to study the complex nature of flow field around a long-body aeronautical vehicle. Heidari et al. [3] conducted a computational investigation as well as an experimental one to study the pressure distribution around complex bodies of revolution. They used a Multi-Block numerical method for a supersonic flow over a body with zero angle of attack to solve the thin layer Navier–Stokes (TLNS) equations. They used implicit Beam and Warming central differencing scheme [4] for discretization of equations while the Baldwin–Lomax was used to solve the turbulence phenomenon. They compared their numerical results with experimental data in terms of lift and drag coefficients and found good agreement between them.

Another investigation carried out by Heidari et al. [5] contains a computational simulation of turbulent subsonic/supersonic flow over a long body of revolution, including the base, in which the multi-Block method was engaged for solving downstream flow while the upstream flow was solved by a Multi-Zone approach. The details of this study have been addressed thoroughly in the referenced manuscript. Different Mach and Reynolds numbers were

E-mail address: heidarimr@piau.ac.ir.

http://dx.doi.org/10.1016/j.ast.2016.03.002

Nomenclature

Cp	pressure coefficient
d	body diameter
L/d	fineness ratio
M	Mach number
x	longitudinal position from the nose
z	perpendicular distance from the top of the body surface
α	angle of attack (degree)
θ	circumferential angle (degree)

Subscripts

∞	free stream
ts	trip strip

studied and at the end, it was concluded that the selected computational method is able to solve the flow at the interface region with high accuracy and the acquired solution yields the flow physics, such as vertical structures, clearly. Also, it was found that using combined multi-zone/multi-block approach takes less CPU time and computer memory (compared to using multi-block alone).

Artificial boundary layer tripping by a surface roughness element will generally cause earlier transition due to additional disturbances it induces within the boundary layer. Two different geometries exist: two-dimensional roughness (a cylindrical stretch across the flow) and three-dimensional roughness (a sphere or spike or single grain of sand). These are single roughness. There is also the possibility of distributed roughness, such as sandpaper or rows of cylinder or multiple rivets. Two- or three-dimensional roughness applies quite different effects. If the wire height is much smaller than the local displacement thickness (δ^*), little effect and transition occur far away from the wire [6]. Trip strip reduces Re_{tr} and moves the location of transition toward the wire and therefore one can control the transition location. There are several investigations on the effects of diameter and place of trip wire on transition location for flow over flat plate, especially in an incompressible flow. The effect of roughness on the laminar-turbulent transition in supersonic flow is much less than that in the incompressible flow. The critical roughness height for supersonic flow is about three to seven times greater than that for the incompressible flow. Experiments at a Mach number of $M = 5.8$ shows that even a trip wire may introduce no turbulence at all [7].

Heidari et al. [8] investigated the boundary layer distribution and the way the shock is developed around a long cylinder, both numerically and experimentally. An extensive experimental and numerical study on a long axisymmetric tapered body at supersonic speed was performed to investigate the pressure distributions and boundary layer profiles at various angles of attack. Schlieren technique was used to visualize the flow and shock formation at varies conditions.

Artificial boundary layer tripping utilizing the single cylindrical wire has more effect on the boundary layer thickness and its shape and less effect on the surface pressure distribution, especially for distances far away from the trip strip location. Furthermore, it is shown that the location of the trip strip installation as well as the value of its diameter applies significant effects on total and static pressure distributions. Trip strip increases the thickness of the boundary layer for the test model. Decreasing the diameter of the trip strip and varying its installation point toward the end of the body could not remove the oblique shock forming ahead of and behind the wire. The shape of the boundary layer near the wall as well as its thickness varied significantly due to the presence of the trip strip.

For some rockets and missiles, the after body cross-section varies longitudinally. These design methods are particularly found in space mission vehicles. Furthermore, due to the lack of sufficient space for arranging the system, i.e., actuator of controlled fins, avionics, etc., it is necessary to increase the body cross-section in the vicinity of those systems. Lack of space usually appears when controlled fins are installed on the motor surface. Hence, the body cross-section needs to be increased [9]. However, it is not an easy task to calculate the flow parameters as well as their variations for non-zero angles of attack in tapered. It takes considerable memory and CPU time to calculate flow over these bodies. In addition, as angle of attack increases, the flow over a portion of the body may separate making the flow more complicated. Moreover, experimental data for flow properties along tapered bodies used for validation of CFD codes are rare [10,11].

McDaniel and Evans studied the induced rolling moment of Canard-Controlled missiles [12]. As their conclusion, they presented a semi-experimental model for predicting the induced rolling moment of this type of vehicles, the acquired results of which are in good agreement with the experimental data for a wide range of subsonic and supersonic Mach numbers as well as various angles of canard and attack.

Lesieutre and Quijano numerically investigated the induced vortices caused by missile fins for different angles of attack [13]. In this research, the effects of canard as well as the middle fins on the pressure distribution at the missile's end fins are studied by calculating the pressure distribution for different angles of attack and the acquired results showed acceptable accuracy comparing with the experimental ones.

Ridluan studied the vortices around a missile as well as their interaction with the fins under subsonic and supersonic regimes by means of numerical simulation [14]. The acquired results (which are in the forms of force and moment coefficients) were verified by being compared with the available engineering software (DATCOM). The acquired results from this study show that a bow shock is formed at the middle section, in front of the fins, and the nose.

Here, extensive wind tunnel tests on a long axisymmetric body were performed to investigate pressure distributions, boundary layer profiles and flow characteristics at various angles of attack and at a constant supersonic Mach number of 1.6. Because of low maneuverability of high fineness ratio missiles, the range of angle of attack for the present study was taken to be moderate. Also, the effects of varying cross sectional areas on surface static pressure distribution and boundary layer profile are thoroughly investigated. To this end, two belts having different cut-off angles are installed on the cylindrical portion of the model. One of these belts was installed at the beginning of the after body part, while the other one was located near the end of the body. Different bodies were generated by varying the belt angles, and the effects of the variable body cross section were studied on pressure signature and boundary layer profiles.

2. Experimental equipments and tests

All tests were conducted in the trisonic wind tunnel of Qadr Research Center, QRC. The equipments used for this investigation included: Schlieren visualization system, A/D board, traversing mechanism, rake, vacuum pump, manometer, pressure transducer,