Aerospace Science and Technology 50 (2016) 266–271

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Aerothermal exploration of reaction control jet in supersonic crossflow at high altitude

M.S.R. Chandra Murty, Anand V. Bhandarkar, Debasis Chakraborty *

Directorate of Computational Dynamics, Defence Research and Development Laboratory, P.O. – Kanchanbagh, Hyderabad-58, India

ARTICLE INFO

Article history:
Received 25 March 2015
Received in revised form 7 January 2016
Accepted 10 January 2016
Available online 14 January 2016

Keywords:
Multijet
RANS
Aerothermal

ABSTRACT

Multiple reaction control jets injected normal to free-stream is used to manoeuvre aerospace vehicle at high altitudes. Detailed aerothermal analysis of a high speed aerospace vehicle with multiple lateral jets is carried out in its full trajectory covering wide range of Mach numbers and altitudes. Three dimensional RANS simulations with laminar-turbulent transition models are performed at several instances in the flight trajectory using commercial CFD solver. Numerical simulations captured all complex flow phenomena of free stream & multi-jet interaction at high altitudes and its influence on vehicle airframe temperature. Heat flux data base obtained from CFD analysis is used for transient thermal analysis of flight vehicle. High temperature local hot spots in jet wake regions and detailed thermal analysis of total vehicle provided important inputs to the system design.

1. Introduction

Due to operational reasons, aerospace vehicles need to be manoeuvred at high altitudes and conventional control elements like fins and wings etc. become less effective due to decreased air density/dynamic pressure. Side control jets located at various positions of the vehicle are generally employed [1,2] to control the vehicle at high altitudes by injecting high pressure jets transverse to main stream. The side control jet motors are smaller in thrust levels compared to main rocket motor and employ liquid propulsion system in on/off mode depending on flight requirement. In its passive state, side jet control produces no additional drag as none of its components intrudes in the flow path and it has quick response time. Thus lateral jet altitude control has been a preferred choice for aerospace vehicle control at high altitudes.

Detailed review of transverse jet exhausted into supersonic free stream is presented by Champigny and Lacau [1] which describe the complex structure of the flow field consisting of a bow shock, separation region ahead of the jet, barrel shock and counter rotating vortex pair in the wake of the jet. The flow structure of transverse jet in supersonic cross stream is shown schematically by Ben-Yakar et al. [3] and is reproduced in Fig. 1. The operating altitude, free stream Mach number, pressure ratios of the jet and free stream, diameter and shape of the side jet nozzle etc have significant effect on the jet shape and its penetration into the supersonic free stream. Cassel [2] proposed a combination of CFD, wind tunnel and flight testing to understand the complex flow characteristics of jet interaction problem. Recent advances of CFD have enabled direct solution of Jet Interaction (JI) flow field under many circumstances of application interest. Fric et al. [4] categorised vortical flow structure into various groups through experimental observations as a) horseshoe vortex, b) jet shear layer vortices, c) wake regions and d) counter rotating vortex pair (CVP). Effect of freestream boundary layer thickness and momentum ratio (J) on surface pressure field were experimentally investigated by Hojaji et al. [5]. It was found that increased boundary layer thickness ahead of jet causes lesser surface pressure ahead of jet and therefore can affect heat transfer rates. Guelhan et al. [6] measured surface heat transfer rates due to jets injected in hypersonic cross flow. Review of several experimental works on 'Jets in crossflow' can be found in Ref. [7]. Aswin and Chakraborty [8] numerically studied the side jet interaction for missile type configuration experiments performed by Stahl et al. [9]. RANS predictions show reasonable agreement with measured wall pressures, it was found that pitching moment is linearly varying with jet momentum ration. RANS predictions of Sriram et al. [10] could be able to capture important flow features like CVP etc. however they could not explain the large unsteady vortical structures. Though DES, LES and DNS predictions provide better insight of flow features, these methods are prohibitively expensive for high Reynolds number flows in increasing order. Few cases of DES/LES predictions of similar problems are found in Refs. [11–16].

Studies related to multiple jet interactions with free stream at high altitude are rather limited in open literature. When pitch, roll

* Corresponding author. Tel.: +91 40 24583310.
E-mail address: debasis_cfd@drdl.drdo.in (D. Chakraborty).

http://dx.doi.org/10.1016/j.ast.2016.01.004
1270-9638/

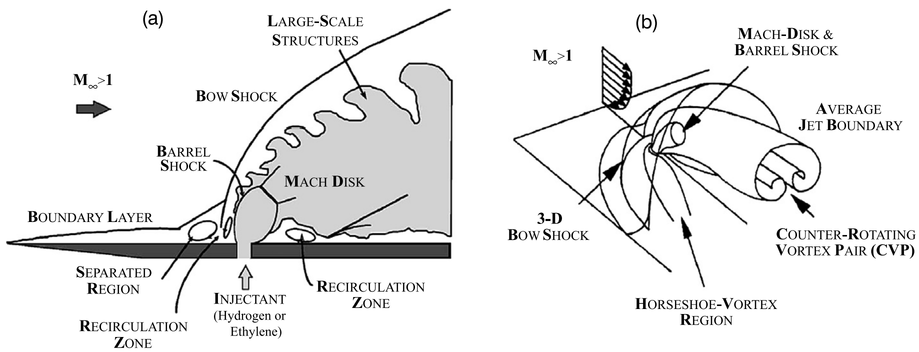

Fig. 1. Schematic of jet injection in high speed cross flow, a) jet structure in axial plane and b) three dimensional features of the jet near field. Figure taken from Ref. [3].

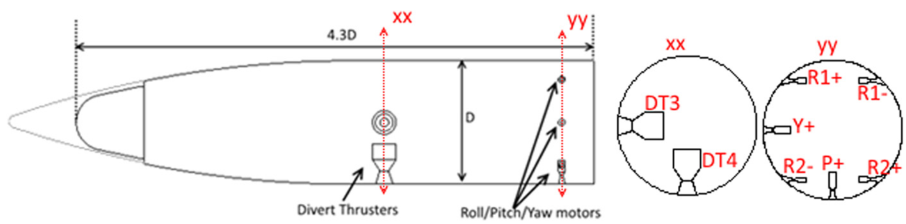

Fig. 2. Schematic of terminal stage of vehicle with Divert and R/P/Y motors.

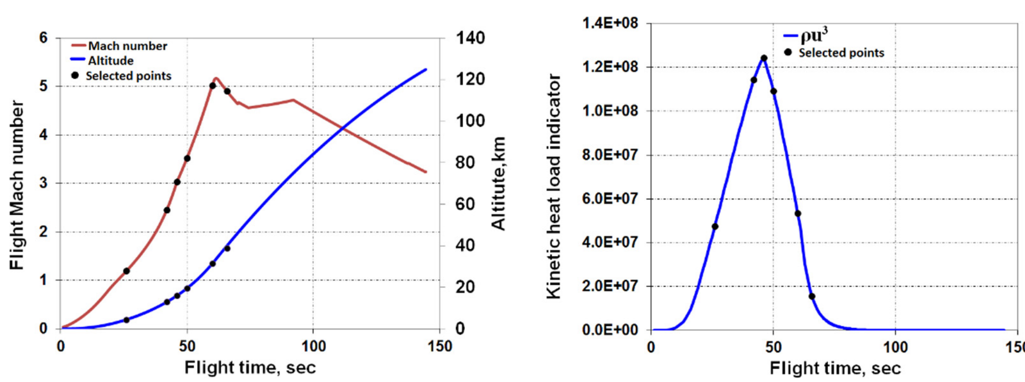

Fig. 3. Flight trajectory and kinetic heat load indicator (ρu^3).

and yaw controls are required simultaneously, a number of hot jets are employed and the motor plumes interact with each other as well as with the free stream. This creates a complex flow pattern around the vehicle body causing some hot gas gazed over the vehicle surface and creates local hotspots which need to be given consideration for thermal safety of the airframe. Saha et al. [17] presented CFD studies giving qualitative features of the multijet–free stream interaction for two discrete altitudes.

In this work, a detailed numerical aerothermal analysis with 3D RANS equations with transition model is presented of an aerospace vehicle in its full trajectory considering different forebody shapes (caused due to heat shield ejection) and multiple lateral jets for simultaneous pitch, roll and yaw control. At select points in the trajectory, steady CFD analysis is carried out and flow field is used for thermal analysis of total vehicle. Flow parameters and skin temperature distributions caused due to aerodynamic heating as well as multijet–freestream interactions are also analysed.

2. The geometry and flight trajectory

The schematic of the vehicle configuration is shown in Fig. 2. After crossing severe aerodynamic heating in the ascent phase, the heat shield (nose cap) is ejected at 39 km altitude leaving the seeker radome exposed to atmosphere and subsequent target locking. A forward facing step is formed at radome and vehicle body joining location. All the reaction control jets are operated after heat shield separation. The divert thrusters are located at CG of the vehicle and Roll/Pitch/Yaw (RPY) thrusters are placed in the rearward portion of the vehicle. After heat shield separation, the vehicle is controlled by reaction control jets i.e., two divert thrusters, four roll thrusters, two pitch and two yaw thrusters. Typical flight trajectory and kinetic heat load indicator (ρu^3) are shown in Fig. 3. Peak Mach number ($\sim$5) is achieved at about 25 km altitude.

3. The code and computational details

Reynolds averaged Navier–Stokes equations are solved in a finite volume framework using commercial CFD code ‘Ansys Fluent’ [18]. Fluid flow equations are solved by employing a cell-centred finite volume method based on the linear reconstruction scheme that allows use of computational elements with arbitrary polyhedral topology, including quadrilateral, hexahedral, triangular, tetrahedral, pyramidal, prismatic and hybrid meshes. Structured computational meshes were generated using ICEMCFD 14.5 [19]. Necessary care is taken in placing the first grid point near the wall boundaries by maintaining proper wall y^+ values and to capture the boundary layer right up to the wall without using any wall functions. Also, very fine grid is provided around jets to resolve high pressure ratios of the jets. Computational domain differs for transonic and supersonic test cases. The inflow boundary, out-