Aerospace Science and Technology 36 (2014) 1–4

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Short communication

Coupled external and internal flow simulation of a Liquid Fuelled Ramjet vehicle

MSR Chandra Murty, Debasis Chakraborty *

Directorate of Computational Dynamics, Defence Research and Development Laboratory, Hyderabad, 500058, India

ARTICLE INFO

Article history:
Received 14 January 2014
Received in revised form 10 March 2014
Accepted 22 March 2014
Available online 28 March 2014

Keywords:
Liquid Fuelled Ramjet
Computational fluid dynamics
Air-intake

ABSTRACT

For finalization of mission parameters of LFRJ-Technology Demonstrator vehicle with a belly-mounted twin intake configuration, CFD has been used extensively to obtain aerodynamic characteristics of the cruise vehicle as well as CV with booster. As the flow field is highly complex due to the asymmetric intake-body configuration, conventional engineering methods are inadequate and detailed CFD based methods have been introduced at early stage of vehicle design cycle. A coupled non-reacting external and internal flow through intakes, combustor and nozzle are solved at different flight conditions (Mach 1.8 to 2.5 and altitude 0.7 to 5 km at AOA $= -5°$ to $5°$) to obtain aerodynamic parameters. 3D RANS equations are solved along with SST k-ω turbulence model. Effects of grid and turbulence models are also studied. Utilizing CFD data, the external and internal flow configuration have been progressively evolved. CFD based methods have helped in generating aerodynamic characteristics of the vehicle which were very useful for mission study.

1. Introduction

Liquid Fuelled RamJet (LFRJ) propulsion system is an attractive choice for long duration supersonic tactical applications over a wide operating regime because of its higher speed, performance and manoeuvrability through extended powered flight [7]. Ramjet vehicles are boosted to operational speed (usually by a solid rocket) at which time the ramjet ignites and sustains the required thrust for supersonic flight. Starting from 1945, several liquid fuelled ramjet missiles were designed, flight tested and operationalized in USA (Talos and Bomrac), erstwhile USSR (SA-4), Britain (Bloodhound) and France (CT-41), etc. India is pursuing the development of liquid fuelled ramjet vehicle and the schematic of a LFRJ technology demonstrator is shown in Fig. 1. It is a belly mounted twin-intake ramjet configuration and is to be boosted to supersonic Mach number by a solid rocket motor.

As the missile configuration is complex and asymmetric, conventional engineering methods are inadequate to bring out the aerodynamic characteristics. Higher performance requirements and tighter constraints on system size and weight have led to the use of high fidelity techniques for analyzing and designing such systems right from the conceptual stages. With the advent of powerful parallel computers, robust numerical algorithms, CFD has emerged as an efficient design tool for aerodynamic and propulsive characterization of such a system. Traditionally, CFD studies are applied separately for external and internal flows for the aerodynamic and propulsive design respectively. Although, this demarcation of external and internal flow can be justified for rocket (non-air breathing) propulsion system, the high speed air-breathing propulsion system requires a coupled simulation of external and internal flows as the flow fields are highly dependent on each other. Saha et al. [5] employed the coupled external and internal flow simulation to estimate the performance parameters of installed air intakes of a solid fuelled integral ram-rocket at different angles of attack up to 6° and obtained reasonable match with experimental results. In this paper, a coupled CFD analysis of external and internal flows is presented to estimate the aerodynamic parameters of LFRJ technology demonstrator. Air intake performance parameters are also evaluated from the simulation results.

2. CFD analysis

3D Reynolds averaged Navier–Stokes equations are solved using a commercial CFD package Fluent 12.1 [1]. A cell-centred finite volume method based on the linear reconstruction scheme that

* Corresponding author. Tel.: +91 40 245 83310; fax: +91 40 243 40037.
E-mail address: debasis_cfd@drdl.drdo.in (D. Chakraborty).

http://dx.doi.org/10.1016/j.ast.2014.03.011

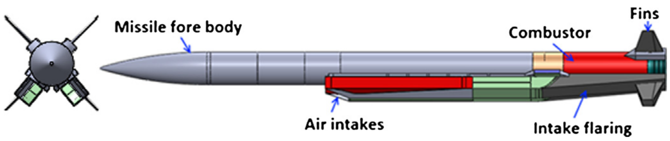


Fig. 1. Schematic configuration of LFRJ Technology demonstrator.

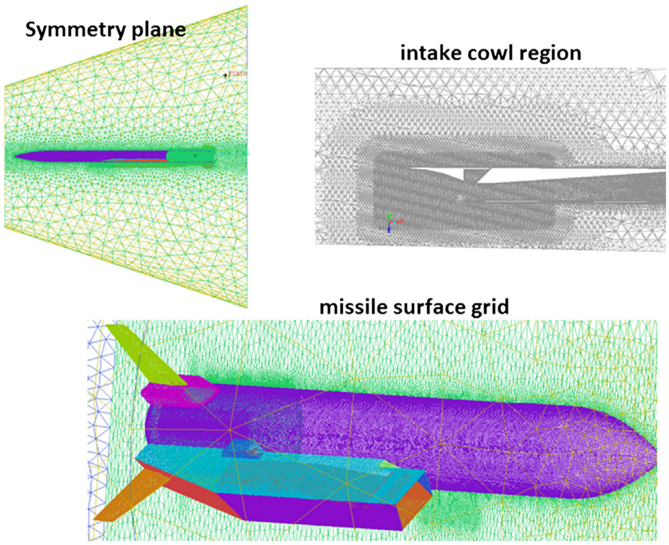


Fig. 2. Computational grid at different regions.

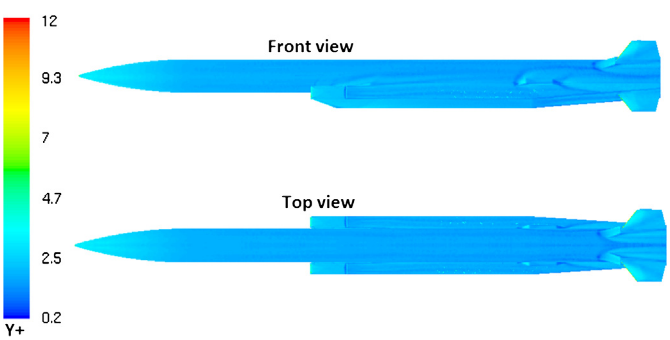


Fig. 3. Wall y^+ contour.

allows use of computational elements with arbitrary polyhedral topology, including quadrilateral, hexahedral, triangular, tetrahedral, pyramidal, prismatic and hybrid meshes is employed. Density based implicit coupled solver is chosen for solving the governing equations. Inviscid fluxes are discretized by 2nd order accurate Roe scheme [4] whereas 2nd order central differencing scheme is used to discretize the viscous fluxes. Turbulence is modelled by both Mentor's SST turbulence model [3] and Spalart and Allmaras [6] and the results are compared. High quality unstructured computational mesh has been generated for the full missile with all the geometric complexities using ICEM-CFD [2] as shown in Fig. 2. Very fine near wall mesh ($y^+ \sim 3$) is made to capture the boundary layer right up to the wall without using any wall functions, wall y^+ contours are shown in Fig. 3. The grid spacing near the wall is about 10 μm. Internal flow path includes supersonic diffuser (3 compression ramps), constant area throat, subsonic diffuser, combustor including nozzle at the end. All the present simulations have been performed without fuel injection (cold flow jet-on). External surface includes missile cylindrical body, nose region, intake outer walls, flange covers, actuator covers, tail fins, intake flares and missile base region. The inflow and the far field are given prescribed Mach

Table 1
Predicted aerodynamic parameters with different grids.

Grid	C_d external	C_d total	C_n external	C_n total
7.5 million (Grid-1)	0.502	0.786	−0.350	−0.273
4.5 million (Grid-2)	0.488	0.782	−0.352	−0.269

Table 2
Comparison of drag components for two different turbulence models.

Turb. model	Wave drag	Viscous drag	Total drag
SST	0.4578	0.1390	0.5968
SA	0.4625	0.1409	0.6034

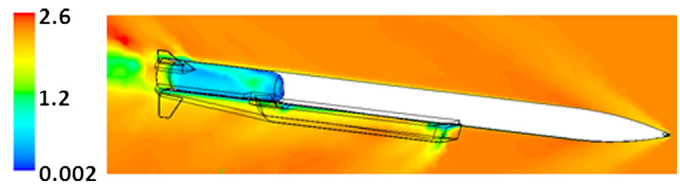


Fig. 4. Typical Mach number contours in symmetry plane for $M_\infty = 2.0$ and $\alpha = 0°$.

number, pressure and temperature and outlet is prescribed with supersonic outflow condition. Grid independence of the solution is demonstrated in Table 1 by comparing the aerodynamic parameters with two different grids of size 7.5 millions and 4.5 millions and the maximum difference is found to be less than 1.5%.

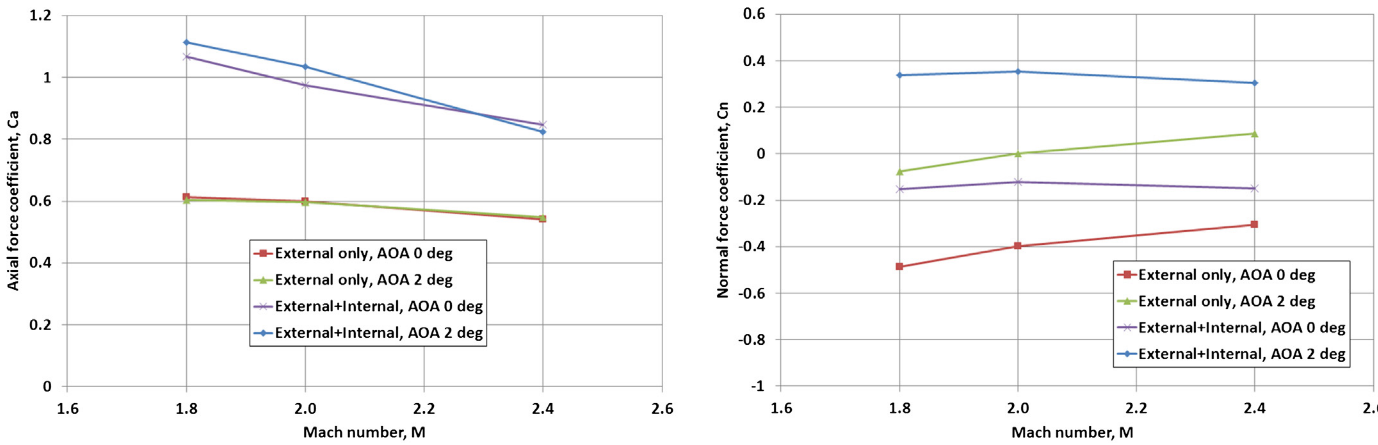
Effect of turbulence model on overall aero parameters is estimated by testing (a) Mentor's SST k-ω and (b) SA model for version-1 geometry with flow conditions of Mach 2, altitude 2 km and AOA 2°. Close agreement of overall aero parameters is seen between the two turbulence models as shown in Table 2. Further calculations have been done using SST k-ω turbulence model owing to its better performance in separated flows. In the present case, since we are analyzing the performance of the system for design condition, non-reacting simulations are performed in both external and internal flowpath.

3. Results and discussion

3.1. Analysis of baseline configuration

Baseline configuration (version-1) is simulated at different flow conditions ($M_\infty = 1.8$, 2.0 and 2.4 and $\alpha = 0$ and 2°). Typical Mach number contours for baseline configuration are shown in Fig. 4. Essential flow features are well captured like oblique shocks on missile fore body, intake diverter flow shocks and deceleration of flow inside intake and combustor path and plume expanding into the atmosphere. Mass capture ratio (MCR) for Mach nos. 1.8, 2 and 2.4 is predicted as 60%, 62% and 67%. It is to be noted that even at Mach 2.4, above the design Mach number (2.3), there is considerable spillage. The intake diverter (single wedge) is causing stagnation of flow and aiding in further spillage and reduction in intake stream tube size.

Computed external (integrated over external surfaces) and total (integrated over internal and external surfaces) force coefficients i.e., axial force coefficient (C_a), normal force coefficient (C_n) and pitching moment coefficient (C_m) w.r.t. nose tip are presented in Fig. 5. Missile length (L) and missile fuselage area are used as reference values for calculating the forces and moments. An average drag coefficient of $0.6 \pm 10\%$ for external surface alone configuration and $0.95 \pm 20\%$ for full vehicle configuration is predicted for AOA upto 2° and Mach number 1.8 to 2.4. Though a −ve value of C_n is observed at AOA = 0°, it move towards +ve value at higher



(a) (b)

Fig. 5. Variation of (a) axial force coefficient and (b) normal force coefficient variation with free stream Mach number for base line configuration.

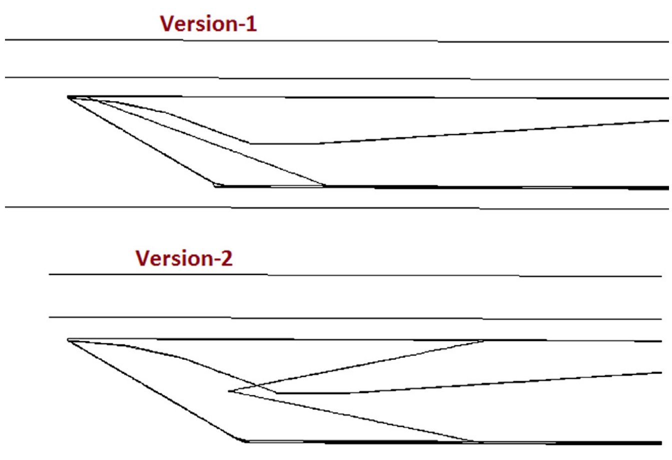


Fig. 6. Comparison of old and updated diverter shape.

AOA. Intake outer surfaces contribute majority of negative normal forces, basically due to its asymmetric placement about the pitch axis (X axis). It is observed that the total pressure close to the intake outer surface (top and bottom) is same, but the velocity is higher close to the bottom surface compared to top surface. This has resulted in lower pressure at bottom surface compared to top surface. At lower Mach numbers, intake flow spillage causes additional drag on the missile.

3.2. Analysis of improved design

From the above 3D detailed full missile simulations, it was observed that even with design Mach number, there was considerable intake spillage. Various design changes like reduction of ramp edge thickness of the intake, change of boundary layer bleed system, diverter shapes, increase of throat heights, change of actuator covers, tail fins are incorporated in the design and the simulations are carried out for the same Mach number and angle of attack conditions.

3.2.1. Effect of diverter shape

To avoid large region of stagnant flow ahead of diverter, the diverter shape is modified from wedge shape to double wedge shape as shown in Fig. 6. Simulation has been updated for flow conditions of ($M_\infty = 2.0$, $Alt = 2$ km and $\alpha = 2°$). The effect of this geometric update on flow physics can be observed from Table 3 and Mach contours, Fig. 7. Significant drop in drag numbers and clean flow ahead of intake is seen. Detailed CFD study helped in understanding complex flow physics.

Table 3
Comparison of force coefficients showing the effect of diverter.

	C_a, external	C_n, external	C_a, total	C_n, total
V1	0.6119	−0.4876	0.9753	−0.1220
V2	0.5323	−0.3635	0.8619	−0.1935
%diff	13	25.4	11.6	–

3.2.2. Effect of intake leading edge thickness

To understand intake leading edge thickness ($L_{e,t}$) on flow features, two 2D simulations with 0.5 mm and 2 mm $L_{e,t}$ have been simulated. Mach contours comparison for the two cases are shown in Fig. 8. With increase in $L_{e,t}$ from 0.5 mm to 2 mm, shock system has changes significantly, third oblique shock is disappeared, separation bubble size has increased, mass capture fell by 8%. This study indicates leading edge shaping should be done carefully while designing intake.

In the next version, many geometric updates have been done viz., tail fins, control surface actuation, intake internal flow path, intake bleed system and throat height, etc. Variation of C_a and C_n with AOA are shown in Fig. 9. C_a is minimum at zero AOA, for higher AOA it increases. C_n shows linear characteristics with AOA. Average C_a of $0.5 \pm 10\%$ for external configuration and $0.8 \pm 10\%$ for total configurations is predicted within this operating range. Compared to Mass Capture of 60–67% for baseline configuration, the MCR for the improved configuration has increased to 83–100% for various operating conditions.

4. Conclusion

Detailed CFD analysis has been performed to estimate aerodynamic parameters of two versions of Liquid Fuel RamJet Technology Demonstrator (LFRJ-TD). Due to complex aero configuration, CFD based methods are employed at very beginning of the design cycle. Two versions are studied and fine-tuned using CFD predictions. Various components are modified to achieve better aero performance like intake diverter, intake, actuator coverts, tail fins, etc. Because of belly mounted intake a −ve C_n is generated at AOA $= 0°$ but as the angle of attack increases, C_n move towards +ve value. This data indicates that for cruise, the vehicle should have +ve AOA. C_a of about $0.5 \pm 10\%$ for external configuration, $0.8 \pm 10\%$ for total cruise vehicle is predicted.

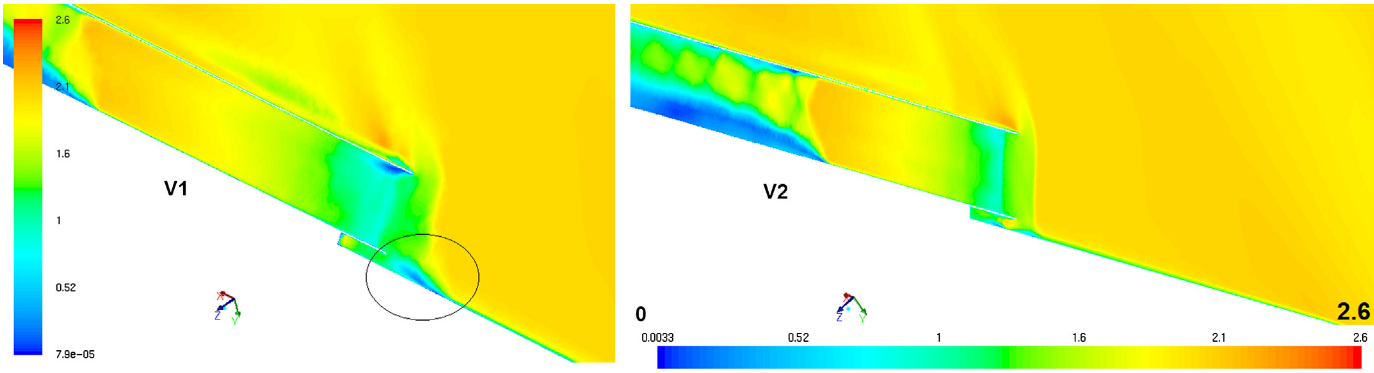


Fig. 7. Mach number contours in front of diverter for V1 and V2, showing effect of diverter shape.

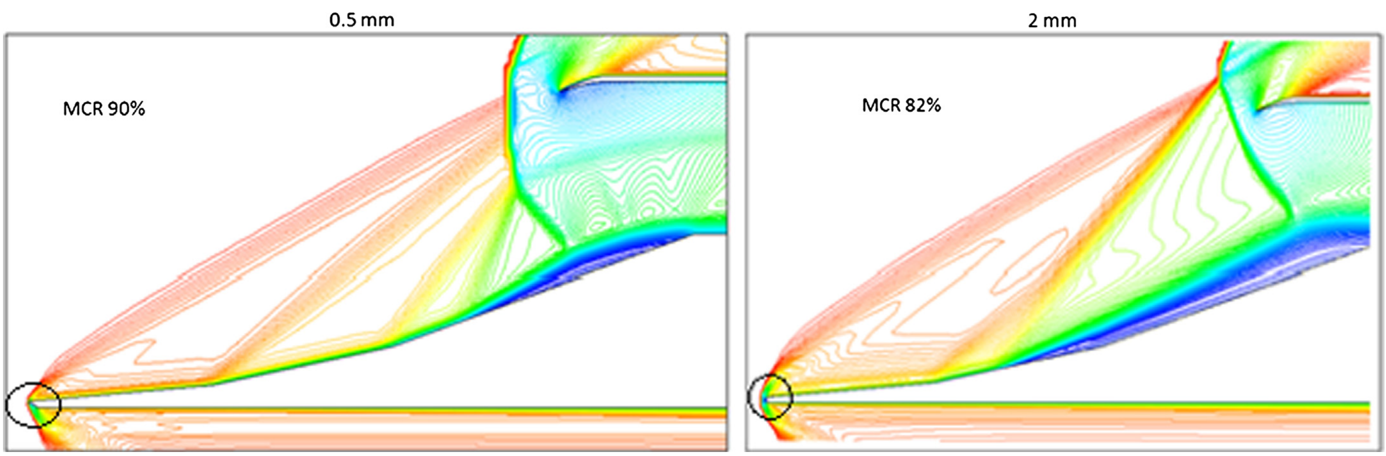


Fig. 8. Mach contours for different ramp edge thickness (a) 0.5 mm and (b) 2 mm.

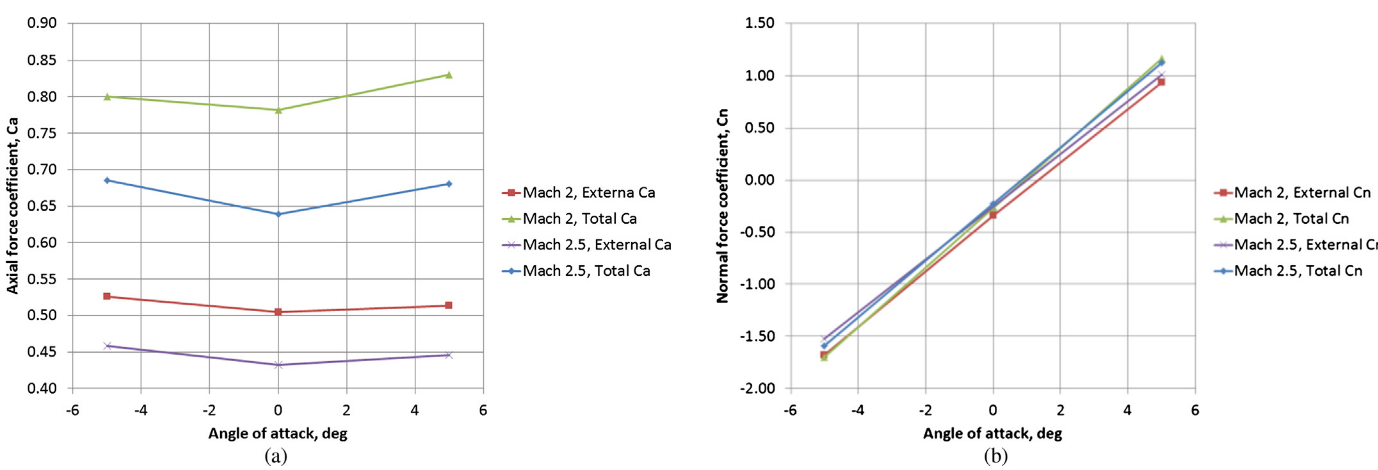


Fig. 9. Variation of (a) axial force coefficient and (b) normal force coefficient variation with free stream Mach number for improved configuration.

Conflict of interest statement

This research received no specific grant from any funding agency in the public, commercial or not-for-profit sectors.

References

[1] Fluent 12.1 Theory manual, Ansys Inc., 2009.
[2] ICEM CFD, 11.0 Modeling, Meshing Guide, Ansys Inc., 2007.
[3] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA J. 32 (8) (August 1994) 1598–1605.
[4] P.L. Roe, Characteristic based schemes for the Euler equations, Annu. Rev. Fluid Mech. 18 (1986) 337–365.
[5] Soumyajit Saha, P.K. Sinha, Debasis Chakraborty, CFD prediction of ramjet intake characteristics at angle of attack, J. Aerosp. Sci. Technol. 62 (3) (August 2010) 159–165.
[6] P. Spalart, S. Allmaras, A one-equation turbulence model for aerodynamic flows, Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics, 1992.
[7] F.C. Zarlingo, Overview of propulsion concepts for tactical missiles, AGARD-CP-493, 1990.