Proceedings of the 6th International and 43rd National Conference on Fluid Mechanics and Fluid Power
December 15-17, 2016, MNNITA, Allahabad, U.P., India

FMFP2016PAPER NO.327

Computational Analysis of Side Jet Interaction With a Super-sonic Cross-flow

Apurva Bhagat
Research Scholar
Indian Institute of Technology,
Hyderabad, 502285
Email: me13m1026@iith.ac.in

Harshal Gijare
Research Scholar
Indian Institute of Technology,
Hyderabad, 502285
Email: me13m1028@iith.ac.in

Nishanth Dongari
Assistant Professor
Indian Institute of Technology,
Hyderabad, 502285
Email: nishanth@iith.ac.in

Abstract
We have numerically investigated the interaction of a side jet positioned on the small rocket, with the supersonic cross-flow. An open source CFD tool, OpenFOAM is used to model the complex flow of a jet-atmosphere interaction. The flow fields are computed by the steady 3-dimensional Navier-Stokes solver with k- ω SST turbulence model. Our solver is validated with the experimental pressure data available on the rocket wall and a systematic study is done by varying parameters like jet pressure ratio. Aerothermodynamic coefficients for various flow conditions are reported, and pitching moments and normal forces are found to vary linearly with the jet pressure ratios. Possible contamination of the on-board sensor located on the rocket wall due to impinging plumes is also examined. This study helps in designing effective missile control by selection of the location of jet and pressure ratios.

Keywords—CFD; OpenFOAM; rhoCentralFoam; side jet control; supersonic cross-flow.

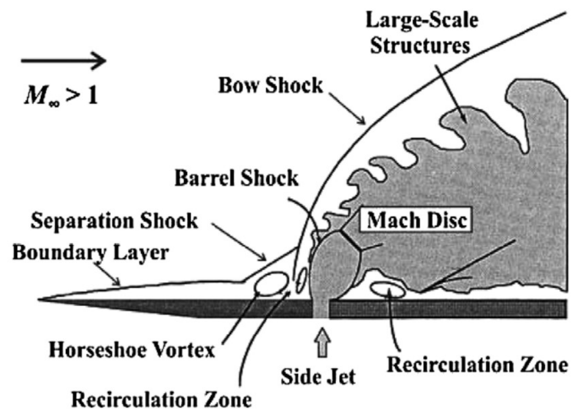

Fig. 1: **Schematic of flow structure of jet interaction with supersonic cross-flow.**

I. INTRODUCTION

Side jet control plays an important role in controlling manoeuvrability and agility of the missile. Advantage of side jet over the aerodynamic control surface is the precise control of forces and quick response time. Side jet control is effective at both, the higher stagnation pressures conditions and the rarefied conditions (altitude $>$ 70 km). Figure 1 demonstrates the complex flow structure developed due to the interaction of jet and supersonic cross-flow. Freestream hits the jet flow, which acts as a blunt obstacle and a bow shock is developed. Interaction of the jet flow with the boundary layer around missile produces a separation shock upstream of the jet and recirculation zone in the downstream region.

Stahl et al. [1] carried out extensive wall pressure measurements on missile models to investigate the interference of a side jet with the cross flow. Aswin et al. [2] further carried out numerical analysis for the same case, and a very good agreement between experimental and numerical results was obtained. Similar problems are analysed using DES/LES by a few researchers.

Major objective of this work is to develop an open source CFD solver for the application of complex side jet and freestream interaction, and parametric study to design an effective missile control. The solver can tackle multi-species, high temperature transport modelling (T 6000 K) and work on parallel computing architecture.

II. PROBLEM SETUP

Figure 2 demonstrates the schematic of the missile model which consists of a cone, cylindrical fuselage and flare. Outer diameter (D) of the cylindrical fuselage is 40 mm and exit diameter (d) of cylindrical nozzle is 4 mm. Nozzle is located at distance 4.3 D and radiance sensor at 4.6 D from the origin.

To study the interference of side jet flow in supersonic cross-flow atmosphere, simulations are carried out step-wise: for external flow only (without jet flow), for jet flow only (without external flow), and with both external and jet flow. Validation study is carried out for the case, where freestream Mach number (Ma_∞) = 3, Reynolds number Re_D = 1.9 $\times$ 10^6 and pressure ratio of jet pressure to ambient pressure (PR) p_{oj}/p_∞ = 200. This case is referred as the baseline case of the paper and all other cases will be compared against this data. Simulations are conducted for PR = 55, 110 and 200. Behavior of bow shock and recirculation zones are investigated by varying parameters like PR. Lift coefficient, drag coefficient, pitching moment and the heat load values on the radiance sensor are reported.

III. COMPUTATIONAL METHODOLOGY

OpenFOAM (Open Field Operation and Manipulation) is an open source CFD software, which is parallel friendly and handles both the structured and unstructured meshes for complex geometries. It is based on C++ library tools and

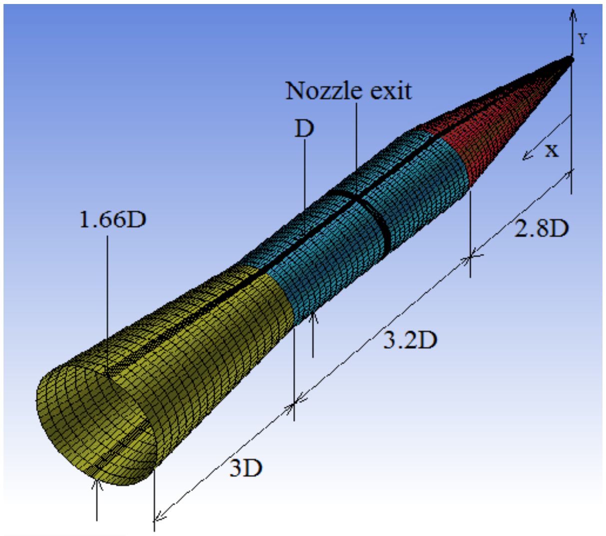

Fig. 2: **Schematic of missile model under consideration (D = 40 mm). Structured mesh on the missile wall (very fine near nozzle) is demonstrated.**

a collection of various applications (created using these libraries). The *rhoCentralFoam* is a density-based compressible flow solver based on central upwind schemes of Kurganov and Tadmor [4].

The *rhoCentralFoam* solver has been validated by Greenshields et al. [5] for supersonic jet experiment by Ladenburg et al. [6] and various standard compressible flow cases. k - ω SST turbulence model is implemented in simulations which is the mix of k - ω and k - ϵ models. The k - ω SST model is merited for its good behavior in adverse pressure gradients and separating flow.

3D structured mesh is generated which consists of 43 million hexahedral elements and fig.2 demonstrates mesh on the wall of the missile. Average Y+ value on the fuselage of missile is 22.31 and cell-size near the wall is 0.01 mm (which can capture the desired Y+ value and viscous effects) and spline law is applied for gradual growth in mesh. Mesh independence study is carried out for 3 different meshes by varying no. of cells on the missile wall, and solution is found to be independent of mesh refinement.

Boundary conditions are given as follows: 1. Missile - isothermal wall (no-slip for velocity, no-jump for temperature). 2. Nozzle - pressure inlet. 3. Outlet - pressure far-field. 4. Inlet - freestream conditions. In current simulations, a calorically perfect ideal gas, air is used for both the freestream flow and nozzle flow. Each test case is simulated in parallel on 32 Intel Haswell cores on the HPC facility at IIT Hyderabad.

IV. RESULTS AND DISCUSSION

Figure 3 demonstrates the comparison of C_{pdif} data (case - PR = 200) of *rhoCentralFoam* solver against experimental data [1], where C_p is the pressure coefficient defined as, $C_p = (p - p_\infty)/p_\infty k Ma_\infty^2$ and $C_{pdif} = C_{pwithsideJet} - C_{pwithoutsideJet}$ C_p values are high at the upstream of nozzle location (x/D = 4.3) due to bow shock wave and reduces in the downstream wake region. The *rhoCentralFoam* values are in decent agreement with the experimental data except in the downstream of jet.

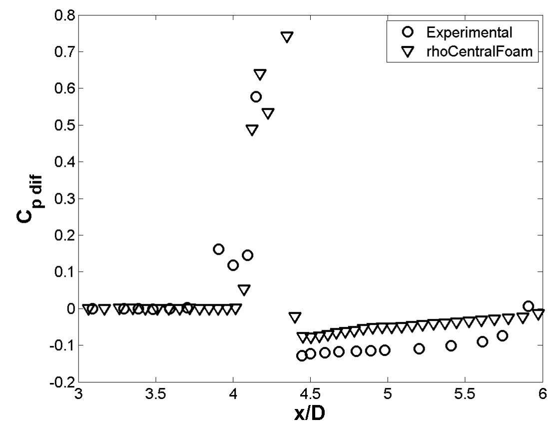

Fig. 3: **Distribution of pressure coefficient difference on cylindrical fuselage.**

Effect of change in PR - To investigate the effect of change in jet pressure ratio, simulations are carried out for PR = 55, 110 and 200. Freestream Ma is 3 and exit Ma of jet flow is 1 for this study. Fig.4 demonstrates density distribution and flow features for the baseline case in XY plane (refer fig.2).

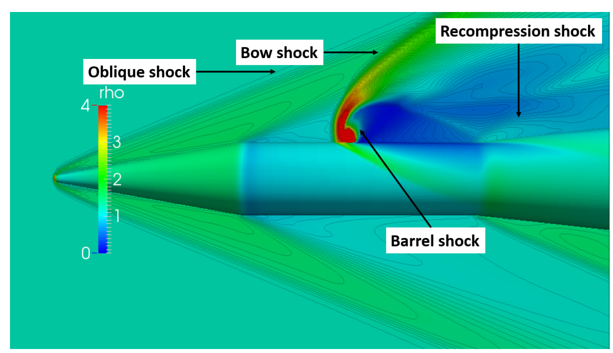

Fig. 4: **Density contours and flow features (PR = 200) in X-Y plane.**

Fig.5 provides pressure distribution on the missile surface for various PRs. It can be observed that bow shock shifts upstream of the jet as PR increases and low pressure region in the downstream region is more pronounced at higher PR. However, change in PR does not affect the flow on the cone as well as flare region of the missile. Higher pressure zone in the upstream of jet and low pressure zone downstream to the injection point are the major contributors of the moment of missile. Desired control forces can be obtained by choosing a suitable location of jet and an optimum range of PR.

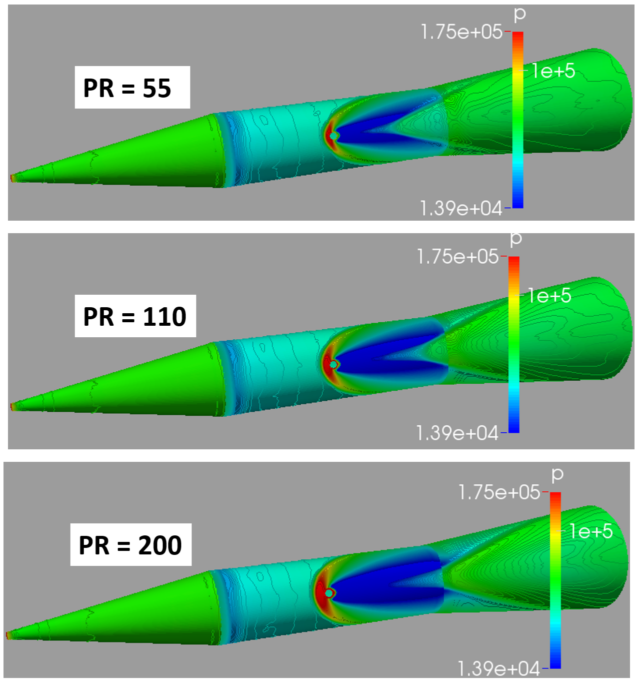

Fig. 5: **Pressure distribution for various jet pressure ratios on the missile wall.**

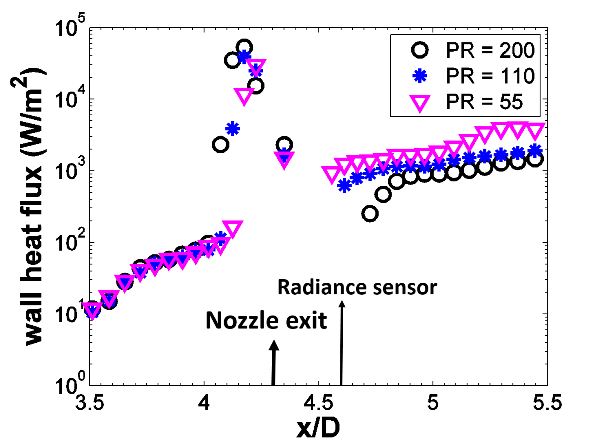

Fig. 6: **Wall heat flux distribution for various jet pressure ratios on the missile wall.**

Table 1 reports values of aero-thermodynamic coefficients for 3 jet pressure ratio cases. Here, C_m (moment coefficient) and C_L (lift coefficient) linearly increase with increase in PR, while C_D (drag coefficient) inversely varies with the same. This can be explained by the fact that higher PR case acts as bigger obstruction to free-stream flow, while lower PR cases leading to increase in drag in the forward direction.

Heat load values on the radiance sensor (which is located at x/D = 4.6) are reported. Fig.6 demonstrates wall heat flux distribution on fuselage missile body. It is observed that heat load on the radiance sensor reduces with increase in PR whereas heat flux values increase in bow shock region. This is because recompression/barrel shock is stronger for higher PR

TABLE 1: **Aero-thermodynamic coefficients and Heat load.**

	PR = 200	PR = 110	PR = 55
C_m	0.1352	0.1080	0.0760
C_D	0.6173	0.6210	0.6252
C_L	0.1478	0.1214	0.0876
Wall Heat Load (W/m^2)	-723.33	628.61	1246.9

leading to intense freestream-jet flow interaction and kinetic energy of freestream flow is converted to thermal energy. Jet flow expands rapidly in the downstream region where radiance sensor is located, and a low pressure and temperature zone is formed downstream of the nozzle. Temperature of jet flow for PR = 200 case drops below wall temperature and cooling of wall takes place, which causes the negative heat load value at the sensor. Therefore, thermal protection system for the missile surface should be designed considering the location of the sensor and PR value.

V. CONCLUSIONS

Numerical simulations are carried out to demonstrate the interference effects of a side jet in the supersonic cross-flow atmosphere. OpenFOAM open source CFD tool is used to model a realistic 3-dimensional missile geometry in the continuum regime. Solver is validated with the experimental surface pressure data on the missile wall. Parametric studies are carried out to study the effect of change in jet pressure ratio on the aerodynamic coefficients and heat load on the sensor. Heat load on the fuselage increases with PR in the upstream region of jet, as freestream and plume interaction is intense at high PR. Heat load reduces with increase in PR, in the downstream region of jet, due to the formation of a low pressure wake. By careful selection of the location of jet and pressure ratios, it is possible to design an effective missile control. An efficient thermal protection system can be designed based on predicted heat load values.

REFERENCES

[1] Stahl, Bernhard, Helmut Esch, and Ali Glhan. "Experimental investigation of side jet interaction with a supersonic cross flow." Aerospace Science and Technology 12.4 (2008): 269-275.

[2] Aswin, G., and Debasis Chakraborty. "Numerical simulation of transverse side jet interaction with supersonic free stream." Aerospace Science and Technology 14.5 (2010): 295-301.

[3] Gimelshein, S. F., D. A. Levin, and A. A. Alexeenko. "Modeling of chemically reacting flows from a side jet at high altitudes." Journal of spacecraft and rockets 41.4 (2004): 582-591.

[4] Kurganov, Alexander, Sebastian Noelle, and Guergana Petrova. "Semidiscrete central-upwind schemes for hyperbolic conservation laws and Hamilton–Jacobi equations." SIAM Journal on Scientific Computing 23.3 (2001): 707-740.

[5] Greenshields, Christopher J., et al."Implementation of semidiscrete, nonstaggered central schemes in a colocated, polyhedral, finite volume framework, for highspeed viscous flows." International journal for numerical methods in fluids 63.1 (2010): 1-21.

[6] Ladenburg, R., C. C. Van Voorhis, and J. Winckler."Interferometric studies of faster than sound phenomena. Part II. Analysis of supersonic air jets." Physical Review 76.5 (1949): 662.

Acknowledgements

The research is supported by DST:SERB/F/2684/2014-15 and MHRD fellowship. We are also thankful to Dr. V. K. Sarswat for valuable suggestions.