Defence Science Journal, Vol. 65, No. 3, May 2015, pp. 189-195, DOI : 10.14429/dsj.65.8254

Numerical Simulation of a Hypersonic Air Intake

Soumyajit Saha and Debasis Chakraborty*

Defence Research and Development Laboratory, Hyderabad – 500 058, India
**E-mail: debasis_cfd@drdl.drdo.in*

ABSTRACT

Numerical simulations were carried out to study the unsteady flow in an intake of hypersonic air-breathing vehicle. Unsteady RANS simulations were performed to examine started flow of the intake when cowl surface is parallel to the ramp surface. Though started, the flow was unsteady due to flow separation bubbles inside intake. Intake with larger cowl opening at which intake unstarted was also simulated. Simulations indicated unstarted flow, with large pressure oscillations. The numerically simulation results match reasonably well with experimental data. Calculated unstarting Mach number was found to be 3.0-3.2 in comparison of wind tunnel data of 3.6 for the same cowl opening angle.

Keywords: Numerical simulation, hypersonic flow, computational fluid dynamics, air intake, intake unstarting

1. INTRODUCTION

Intake performance is a critical point in the design of air breathing mission. The intake of an air breathing vehicle is required to capture and efficiently compress requisite quantity of air for engine operation. The design criteria of air intake are well documented in the literature[1]. In summary,

- Intake should provide adequate mass flow of air as demanded by the combustor.
- Intake should compress the flow as efficiently as possible, minimizing the viscous and shock losses. Intake contribution to overall vehicle drag should be kept at minimum.
- Intake performance should not be significantly reduced by operation at incidence.
- Intake must be able to tolerate the back pressure caused by heat addition.
- The velocity profile at intake exit should be as uniform as possible.

The overall vehicle performance depends greatly on the energy level and flow quality of the incoming air. Small loss in intake efficiency translates to a substantial penalty in engine thrust. Therefore the detailed analysis and assessment of flow behaviour through the intake and its interaction with external flow play an important role in the design evaluation and the optimization of the system performance.

Hypersonic intakes are designed as mixed compression intakes which are a combination of internal and external compressions. The flow pattern in mixed compression intake is shown schematically in Fig. 1. The bow shock of the vehicle forebody compresses the air followed by the number of compression at the central body which coalesces at the cowl lip at design Mach number. The flow is turned inward to the axial direction in the internal compression zone by the lip geometry. The interaction of the centre body and cowl shocks with forebody boundary layer may cause flow separation. The extent of separation depends on the strength of shocks and the nature of the forebody boundary layer. Despite its simple geometry, the intake is very sensitive to the interaction with the upstream external flow and downstream combustion process, and hence, exhibits complex flow phenomena over the expected range of operations.

Hypersonic intakes with mixed compression often encounter ‘unstart problem’. For scramjet operation, the freestream Mach number would be reduced by a factor of about 3 before it reaches the combustor. Such a highly convergent duct, in any given hypersonic Mach number flow, can support two distinctly different flow configurations: (i) A bow shock in front of the intake that diverts some flow overboard and the intake flow is subsonic (‘unstarted’) (ii) no bow shock, no overboard spillage, and the flow is supersonic throughout (‘started flow’). The unstart of the intake could occur due to several reasons, e.g. over contraction, variation of flight conditions, increase of back pressure in combustor, etc., or due to a combined effect of these factors. A theoretical maximum permissible ratio of area at entry to that at the throat (A/A*) is defined[2] for the starting of an internal compression intake based on the theory of oblique shocks. This however does not hold for hypersonic intakes where the boundary layer develops significantly and whose interaction with internal shocks are believed to be the prime cause of flow separation, leading to a complex oscillatory flow structure and expulsion of the shock and the unstart of the intake. ‘Started’ flow condition is required for scramjet operation. Under steady flow freestream conditions, high contraction ratio intakes do not start spontaneously. The normal shock, in front of the unstarted intake need to be swallowed for the establishment of stable hypersonic/supersonic flow in the intake. Various method namely (i) variable intake geometry, (ii) bleed bypass (iii)

Received 10 December 2014, revised 13 May 2015, online published 29 May 2015

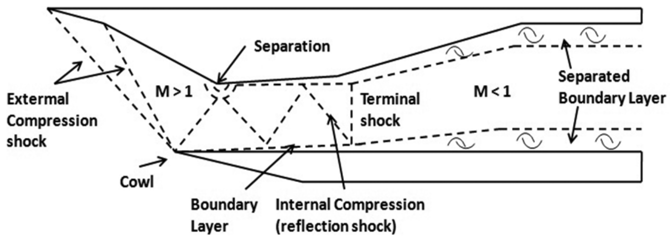


Figure 1. Schematic of the flow field in mixed compression intake.

overboard spillage, and (iv) starting using unsteady effects, etc. are proposed in the literature[3] to start supersonic intakes at any flight condition. But in a hypersonic flow situation which contains high enthalpy flow with high total temperature (~1800 K), any complex mechanical control system may cause severe structural and cooling problems. The predictions of intake unstart and the mitigation plan to reduce its occurrence and effect is essential for hypersonic intake design.

Experimental and numerical research is in progress to understand the causes of hypersonic intake unstart and means to avoid it. Starting of two fixed-geometry hypersonic intakes was studied experimentally by Schmitz and Bissinger[4]. Intake was reported to start at M = 4.3 and operation was found to be stable up to M = 6. Experimental and numerical studies of starting and throttling behaviour of a supersonic intake system of nine different configurations with geometric variation at different inlet Mach number was reported by Schneider and Koschel[5]. It was shown that geometry selection plays a major role for separation bubble size at ramp surface and bubble could be minimised without boundary layer bleed and high intake performance could be achieved. The effect of different bleed dimensions and internal contraction ratios on starting characteristics of 2-D and 3-D scramjet inlets at hypersonic Mach numbers was studied experimentally by Haberle and Gulhan[6,7]. Das and Prasad[8] have shown through their experimental and numerical investigations of mixed-compression intake flow-field at Mach no. 2.2 with different cowl deflections that small angle at the cowl lip leads to start of the intake and improve its performance. Reinhold[9], *et al.* studied a multiple strut-based 2-D hypersonic ramjet inlet flow-field using a parabolised Navier Stokes code and presented good comparison of simulation results against experiment. The interaction of forebody shock and cowl lip shock was studied by Lind[10], *et al.* and predicted very high pressure and temperature region around the cowl lip resulting in flow instability in the intake. Brenneis[11], *et al.* observed drastic change in flow-field behaviour of 2-D inlet at M = 7.4 with adiabatic wall compared to fixed temperature wall. Saha and Chakraborty[12] also found pronounced effect of adiabatic/ isothermal wall boundary conditions on starting Mach number for hypersonic intake with side fencing. Heated boundary layer for adiabatic condition is seen to cause large separation bubble at the intake entrance causing flow unstarting; while flow separation bubble is not observed for isothermal condition for the same free stream Mach number. Barber[13], *et al.* have shown that prediction of starting characteristics of hypersonic intakes is strongly dependent on viscous flow effects and proper choice of turbulence model is required to predict shock wave boundary layer interaction (SWBLI). Donde[14], *et al.* carried out numerical simulation of starting problem in a variable geometry hypersonic intake with a movable cowl. Dynamic meshes are used for depicting motion of the cowl. It was shown that the cowl needs to be rotated through 15.7° and then be brought back to the original position for restarting of the intake after an 'unstart'. It is clear from the above discussion that the starting problem of hypersonic intake has not been fully understood and unsteady flow-field inside the air intake needs to be investigated accurately to tackle this undesirable phenomenon. In this work, unsteady flow simulation is carried out of a hypersonic intake of a proposed hydrocarbon-fueled hypersonic flight configuration[15]. The vehicle configuration with the scramjet engine flow path is reproduced from[15] and is shown in Fig. 2. Thiagarajan and Satyanarayana[16] investigated experimentally the effect of geometrical variation on the starting characteristics of the isolated intake at free stream Mach number 3.6. Unsteady CFD simulation of the intake geometry at test condition has been carried out in the present work. Computed results have been compared with experimental values and insights have been obtained about the complex starting process through analyses of various flow variables.

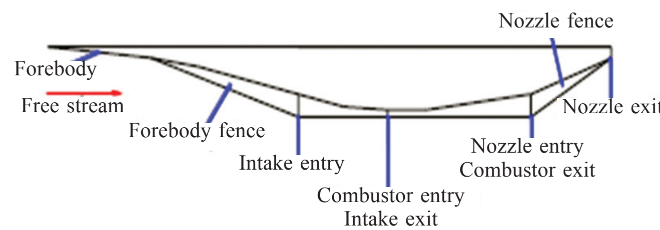


Figure 2. Schematic of engine flow path.

2. INTAKE GEOMETRY AND TEST CONDITIONS

Details of the model geometry and the test conditions are presented Thiagarajan & Stayanarayana[16]. The 1:8 scale intake model was tested in 340 mm diameter axisymmetric supersonic wind tunnel at DRDL to simulate the engine flow path from second ramp of the integrated scramjet vehicle up to the combustor exit. The free stream Mach number of the test was 3.6 and test Reynolds number was 8 x10^6 based on the length of the second ramp. The schematic of the test model is shown in Fig 3. The top and bottom plates represent the engine cowl and vehicle core body, respectively. Downstream engine configuration was also simulated in the model by putting 8 fuel injection struts along with a middle wall similar to the flight configuration. A 48 port motorized scanivalve was used to measure the internal wall pressure on the model. To start the intake at Mach no. 3.6, the cowl was set at 0° position. Gradually the cowl was opened up to ingest more mass flow of air. During the test it was observed that as the cowl angle was increased from 7°49′ to 8°49′, the flow through the intake gets unstarted, resulting in high spillage.

3. CODE AND COMPUTATIONAL DETAILS

Details of the computational methodology adopted in the study are provided by Saha[12], *et al*. Three-dimensional

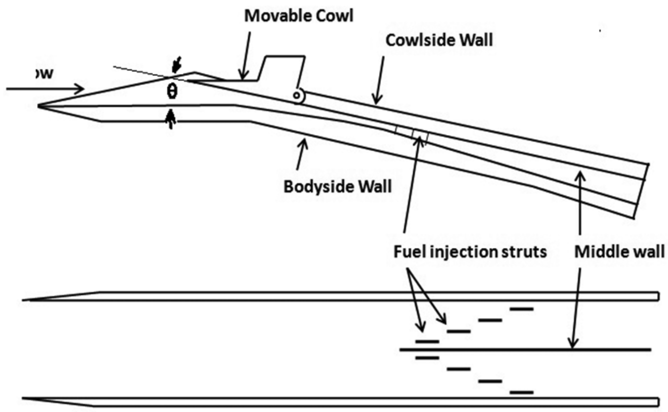


Figure 3. Test model of HSTDV intake.

unsteady Reynolds averaged navier Stokes (URANS) equations were solved using a commercial CFD software Fluent[17]. A 2nd order spatially accurate density-based solver with Roe-Flux difference splitting scheme[18] was used for spatial discretisation and 2nd order implicit Euler Scheme for temporal discretisation was used in the present solution. SST k- ω model of Mentor[19] which blends k-ε model and k-ω model (to take advantage of free stream independence of the boundary layer of the former and near-wall characteristics of the later) was used to model the turbulence.

Two different computational domains corresponding to two cowl opening angles 0° and 8°49′ have been considered in the simulation to study the starting/unstarting characteristics of the intake. The computational domain was extended in both transverse directions to accommodate for spillage of flow from an unstarted intake. The computational domain for cowl opening 0° is shown in Fig. 4. Gambit grid generator was used to generate high quality unstructured mesh involving 0.4 million grid in the computational domain. The computational mesh in symmetry plane for 0° cowl opening is shown in Fig. 5. Very fine mesh was taken near the solid wall to resolve the wall boundary layer. Typical y^+ near the wall is < 1. No formal grid-independence study was carried out for the present problem. Based on detailed grid-independence studies of our earlier high speed intake simulation[12], it is conjectured that the present grid is adequate to capture all pertinent flow features and the results are grid-independent. Fuel injection struts and middle wall in

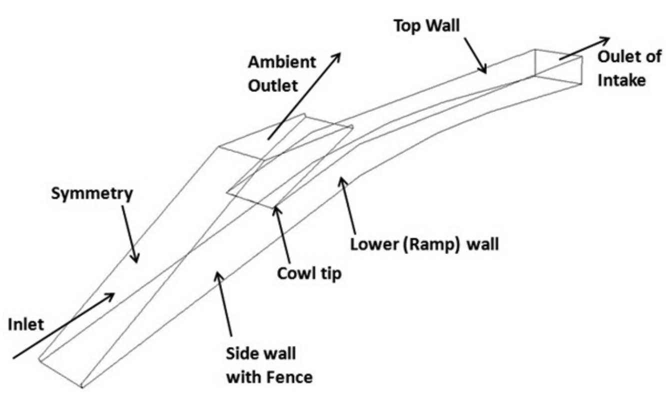


Figure 4. Computational domain for intake with 0° cowl opening.

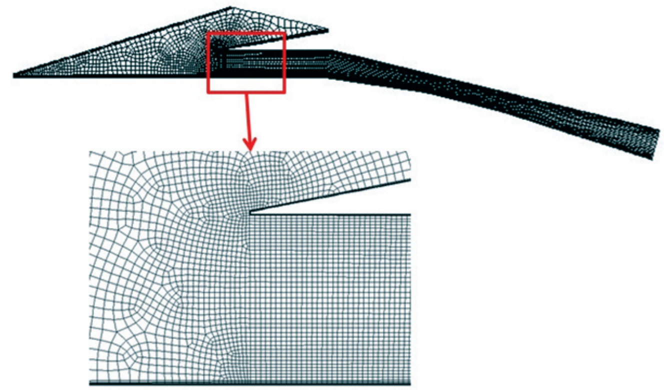


Figure 5. Computational grid for intake with 0° cowl opening with zoomed view near the cowl.

the combustor was not modelled in the simulation. Since these obstructions are placed much downstream and the blockage in the flow path is less than 5 per cent, these are not likely to affect the intake flow-field significantly. End-to-end simulation of hypersonic air breathing cruise vehicle involving non-reacting flow-field in the vehicle forebody, intake and other flow path, and reacting flow-field in the scramjet combustor has been carried out by Dharavath[20], *et al.* where in the fuel injection struts and middle wall is simulated. It was observed that the flow perturbation due to the reaction and the presence of struts are contained within the combustor and have not affected the flow-field in the intake.

All the simulations were carried out for free stream Mach number 3.6. Similar to the test condition, total pressure (P_0) of 14 bar and total temperature (T_0) of 300 K was imposed at the inflow. At ambient outlet 0.185 bar static pressure has been prescribed at the outlet of computational domain. Adiabatic and no-slip wall conditions are imposed for top, bottom, side, and cowl walls. Time step of 10 μs is considered in the simulation.

4. RESULTS AND DISCUSSION

To study the effect of cowl opening angle on starting characteristics of isolated intake, two different simulations corresponding to cowl opening angle of 0° and 8°49′ were carried out. The results of simulations are presented.

4.1 Case 1: 0° Cowl Opening Angle

Unsteady flow-field corresponding to 0°cowl opening angle (cowl is parallel to ramp surface), for a free stream flow of Mach number 3.6 is shown in the Mach number plot in the symmetry plane at three different time instants (time t0, t0+1.5ms, and t0+2.9ms) in Fig. 6. No shock structure ahead of intake cowl or spillage could be observed in the flow-field which indicates that the intake is in the started condition. Large separation regions inside the intake on both top and lower (ramp) walls are clearly observed. The separation bubble on the top wall was formed because the cowl shoulder acting as a compression corner in supersonic flow, and on the lower (ramp) wall, the bubble forms due to interaction of the shock from cowl shoulder and the boundary layer. Further downstream, Mach number increases due to expansion of

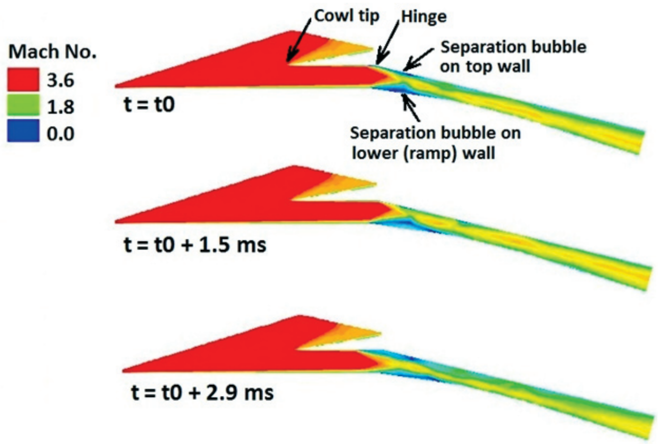


Figure 6. Mach number contour at various instants of time showing unsteadiness of flow in a started intake.

flow. The oscillating separation bubbles (changing position with time) are due to shock boundary layer interaction indicate unsteadiness of the flow in the intake. The ramp wall pressures at four different time instants are compared with test data[16] in Fig. 7 it is to be noted that in the experiment, no unsteady pressure measurements were carried out. The same time averaged experimental pressure distribution on the ramp wall was compared with simulation results at different instants of time. It is clear in the figure that the ramp surface pressure is fluctuating in time. But, in the absence of any unsteady pressure measurement, only the qualitative trends can be compared and which shows a reasonable match between experiment and computation. Figure 8 plots the time history of pressure fluctuations at a location on bottom ramp surface at a location x=0.257 m from start of ramp surface, i.e., where the bubble is located. The unsteady pressure fluctuation of amplitude of 0.2 bar over an average pressure of 0.5 bar, with a time period of 5 ms was computed.

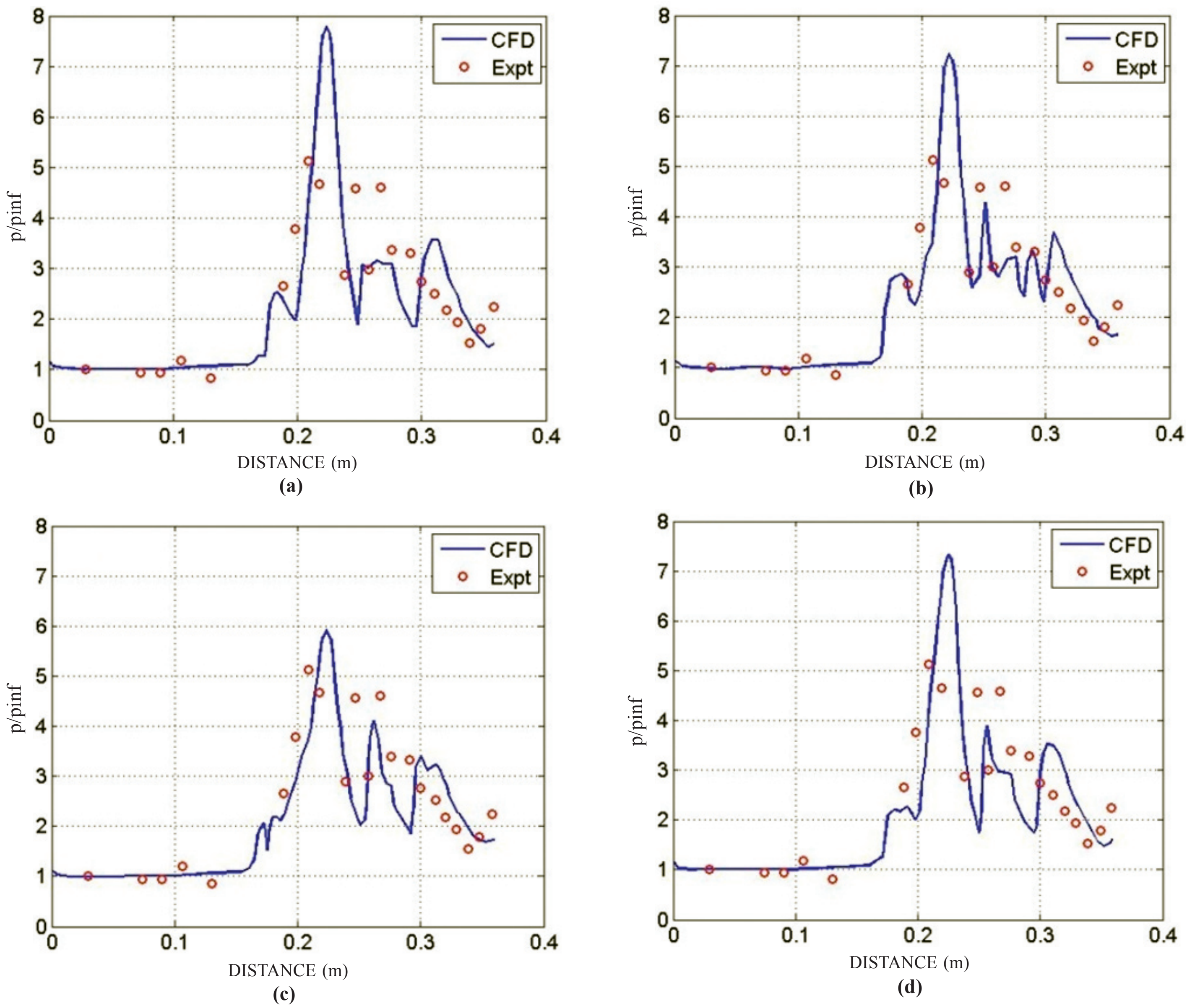


Figure 7. Ramp wall pressure comparison at various time instants for 0° cowl opening (a) *t* = t0 (b) *t* = t0 + 1.9 ms (c) *t* = t0 + 2.5 ms (d) *t* = t0 + 4.5 ms.

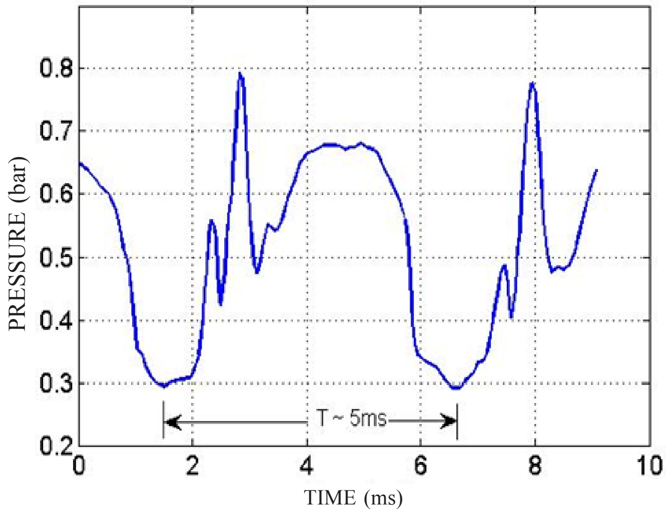


Figure 8. Pressure fluctuation at a point on ramp surface 0.257 mm from start of ramp.

4.2 Case 2 : 8°49′ Cowl Opening Angle

Initial guess corresponding to unstarted intake at lower free stream Mach number is provided for intake with 8°49′ cowl opening angle. The Mach number plots in the symmetry plane presented in Fig. 9 show a large bubble of separated flow ahead of intake entrance. The spillage of incident flow indicates that the intake is in unstarted condition. The changing positions of the separation, bubbles, both at cowl and core body surfaces, depict the unsteadiness of the flow. The comparison of computed wall pressure distribution with experimental data at various time instants on bottom ramp surface are presented in Fig. 10 show the wall pressure distribution on at different instants of time plotted against experimental data. Since the experimental data is not an unsteady data, the CFD prediction matches with the data only at certain instant of time.

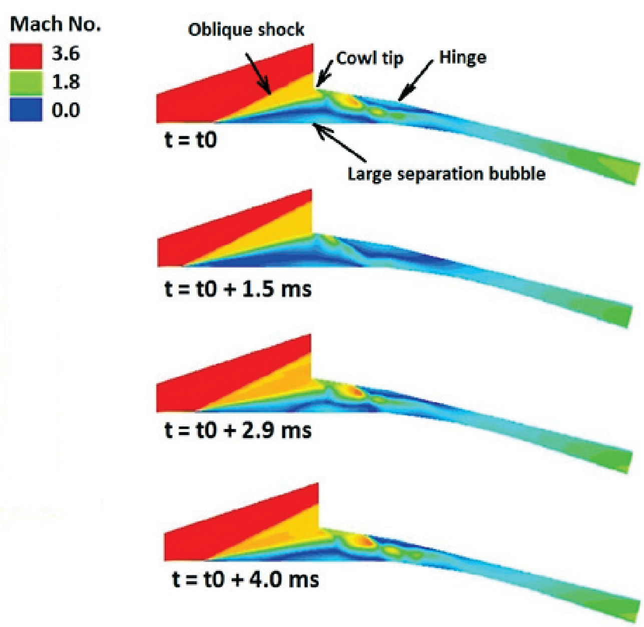
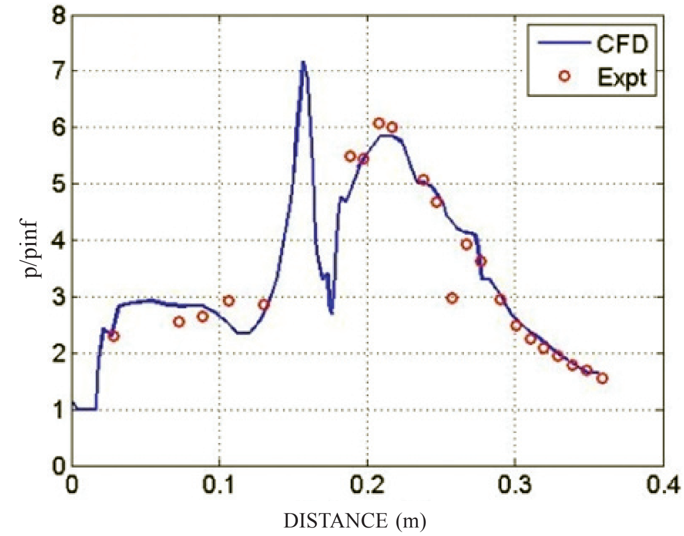
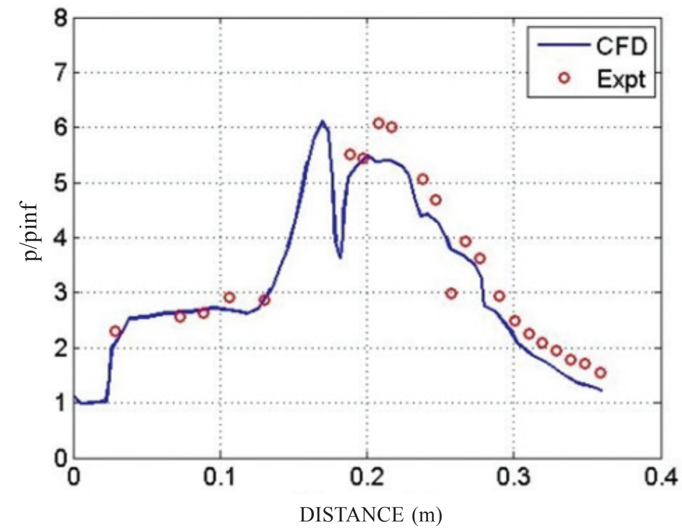


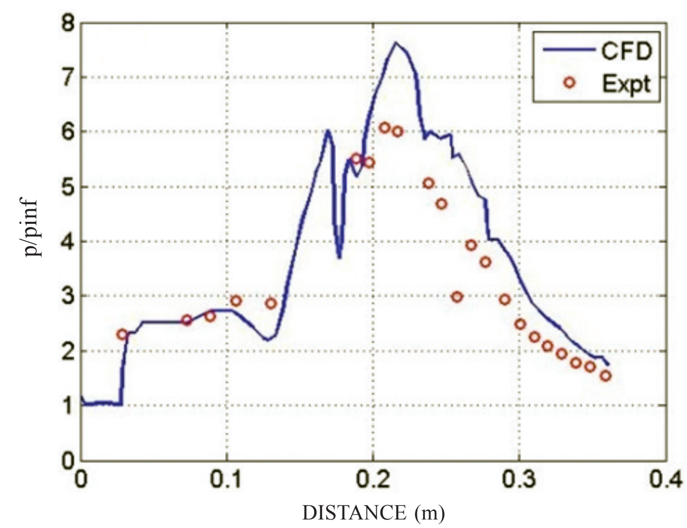
Figure 9. Mach number contour at various instants of time showing unsteadiness of flow in an unstarted intake.



(a)



(b)



(c)

Figure 10. Ramp wall pressure at different instants of time showing flow unsteadiness in an unstarted intake: (a) t = t0 (b) t = t0 + 0.75 ms, and (c) t = t0 + 2.5 ms.

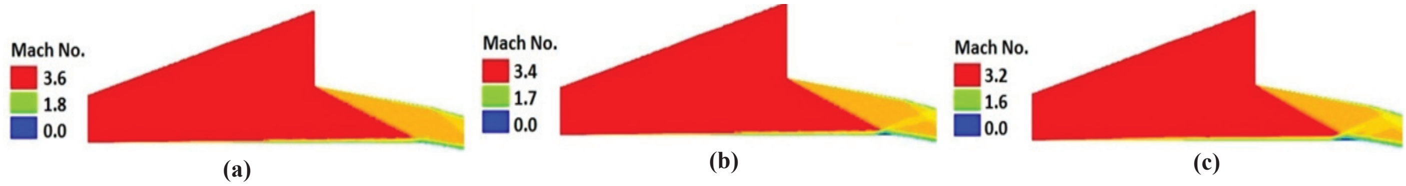


Figure 11. Mach number contour showing started intake at free stream Mach number: (a) 3.6, (b) 3.4, and (c) 3.2.

4.3 CFD Prediction of Unstarting Mach Number for 8°49′ Cowl Opening

Wind tunnel experiment[16] at a fixed Mach number of 3.6 indicates intake unstarts from a started condition when the cowl opening is increased from 7°49′ to 8°49′. Simulations were carried out to evaluate computationally the unstart Mach number for intake with 8°49′ cowl opening. Started solutions were obtained for stream Mach number of 3.6 with initial free stream guess condition. Solutions were then obtained for free stream Mach number of 3.4, and 3.2 starting from a converged started solution at a higher Mach number. Intake Mach number distribution for free stream Mach numbers 3.2, 3.4, and 3.6 so obtained presented in Figs. 11(a) - 11(c) indicate started intake. For free stream Mach number 3.0 (with converged started solution at Mach 3.2), the bubble formed due to cowl-reflected shock and ramp boundary layer becomes larger, and ultimately the intake unstarts. The development of flow as the intake gets unstarted is shown in Fig. 12. Thus computationally, the unstarting Mach number of the intake for 8°49′ cowl opening angle is in between 3.0-3.2, in comparison to the experimental value of Mach 3.6. This difference is attributed to the CFD modelling limitations as well as due to small geometrical mis-match between the test geometry and CAD model used in CFD simulation.

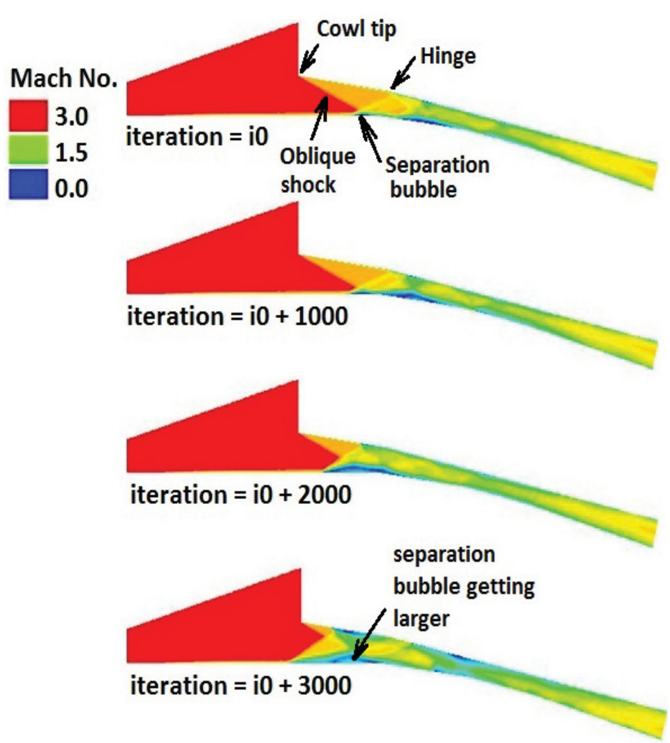


Figure 12. Mach number contours showing unstarting of intake at inlet M = 3.0.

5. CONCLUSIONS

Numerical simulations were carried out for analysing flow in 1:8 scale truncated intake of hypersonic air breathing vehicle, for which steady state experimental data was available. At zero degree cowl opening angle (when cowl is parallel to ramp surface), the intake is in started condition, but flow is unsteady inside. The computed pressure on ramp surface fluctuates with time and compare fairly well with test data. At a location where flow separation occurred due to interaction of cowl lip reflection shock and ramp surface boundary layer, pressure fluctuation was computed to have amplitude of 0.2 bar over an average of 0.5 bar with a time period of about 5 ms. Simulations with 8°49′ cowl opening demonstrated an unstarted intake and the computed wall pressure matched very well with test data at different instant of time. Computations also predicted unstarting Mach number between 3.0-3.2 for intake with 8°49′ cowl opening in comparison to Mach number 3.6 in test.

REFERENCES

1. Seddon, J. & Goldsmith, E.L. Intake aerodynamics. AIAA Education Series, 1985.
2. Timofeev, E.V.; Tahir, R.B. & Sannu, M. On recent developments related to flow starting in hypersonic air intakes. AIAA Paper, 2008-2512, 2008. doi: 10.2514/6.2008-2512
3. Kantrowitz, A. & Donaldson, C. Preliminary investigation of supersonic diffusers. NACA WR L-713, 1945.
4. Schmitz, D.M. & Bissinger, N.C. Design and testing of 2-D fixed –geometry hypersonic intakes. AIAA Paper No. 98-1529, 1998. doi: 10.2514/6.1998-1529
5. Schneider, A. & Koschel, W.W. Detailed analysis of a mixed compression hypersonic Intake. ISABE Paper No. 99-7036, 1999.
6. Haberle, J. & Gulhan, A. Experimental investigation of a two-dimensional and three dimensional scramjet inlet at Mach 7. *J. Propul. Power,* 2008, **24**(5), 1023-1034. doi: 10.2514/1.33546
7. Haberle, J. & Gulhan, A. Internal flow field investigation of hypersonic inlet at Mach 6 With Bleed. *J. Propul. Power,* 2007, **23**(5), 1007-1017. doi: 10.2514/1.29669
8. Das, S. & Prasad J. K. Starting characteristics of a rectangular supersonic air intake with cowl deflection. *Aeronaut. J.,* 2010, **114**(3), 177 -189.
9. Gerbsch, Reinhold A., & Agarwal, Ramesh K. Computation of hypersonic ramjet-inlet flowfields using an upwind parabolized Navier-Stokes Code. AIAA Paper No. 88-2828, 1988. doi: 10.2514/6.1988-2828
10. Lind, C.A. & Lewis, M. J. The effect of shock/shock interactions on the design of hypersonic inlets. AIAA Paper No. 1990-2217, 1990. doi: 10.2514/6.1990-2217

11. Brenneis, A. & Wanie, K.M. Navier-Stokes results for hypersonic inlet flows. AlAA Paper No. 1991-2472, 1991. doi: 10.2514/6.1991-2472
12. Saha, Soumyajit & Chakraborty, Debasis. Hypersonic intake starting characteristics – A CFD Validation study. *Def. Sci. J.,* 2012, **62**(3), 147-152. doi: 10.14429/dsj.62.1340
13. Barber, T.J.; Hiett, D. & Fastenberg, S. CFD Modelling of the hypersonic inlet starting problem. AIAA Paper No. 2006-0123, 2006. doi: 10.2514/6.2006-123
14. Donde, P.; Marathe, A.G. & Sudhakar, K. Starting in hypersonic intakes. AIAA Paper No. 2006-4510, 2006. doi: 10.2514/6.2006-4510
15. Pannerselvam, S.; Thiagarajan, V.; Ganesh Anavardham, T.K.; Geetha, J.J.; Ramanuchari,V. & Prahlada. Airframe integrated scramjet design and performance analysis. ISABE Paper 2005-1280, 2005.
16. Thiagarajan, V. & Stayanarayana. Effect of variations of geometry parameters on air-intake starting characteristics. *In* 8th Symposium on Applied Aerodynamics and Design of Aerospace Vehicles (SAROD 2013), Hyderabad, India, November 21-23, 2013, pp. 1-7.
17. Fluent 6.3 User's guide, 2006, Fluent Inc, USA.
18. Roe, P.L. Characteristic based schemes for the Euler equations. *Annu. Rev. Fluid Mech.*, 1986, **18,** 337-365. doi: 10.1146/annurev.fl.18.010186.002005
19. Menter, F.R. Performance of popular turbulence models for attached and separated adverse pressure gradient flows. *AIAA J.,* 1992, **30**(8), 2066-2072. doi: 10.2514/3.11180
20. Dharavath, Malsur; Manna, P. & Chakraborty, Debasis. Computational fluid dynamics simulation of tip-to-tail for hypersonic test vehicle. *J. Propul. Power,* (In press). doi: 10.2514/1.B35686

CONTRIBUTORS

Mr Soumyajit Saha has done simulation planning, grid generation, Unsteady simulation, Post processing of the results and preparation of the figures.

Dr Debasis Chakraborty has done Overall planning and guidance of the work, review of results and preparation of manuscript.