FLOW FIELD INVESTIGATION OF A RECTANGULAR SUPersonic AIR-INTAKE WITH COWL BENDING

S. Das* and J.K. Prasad+

Abstract

Experimental and computational studies are made on a rectangular mixed compression air-intake designed for Mach 2.2 to study the effect of cowl bending. Flow field details inside the intake have been captured using schlieren flow visualisation and measurement of static and total pressures. Three dimensional simulations have been made with RANS solver adopting k-ω turbulence model using FLUENT. Studies show that the configuration indicate unstart without any cowl bending whereas startup is observed with adoption of small cowl bend. In addition, improvement in quality of flow inside the intake has been observed. In general, agreements between experimental and computational results are good. Studies have been also made at different back pressures. Provision of a small cowl bending could lead to better performance of intake.

Nomenclature

A = area
AR = aspect ratio (B/Hc)
B = width of intake
H = height of intake
L = overall length of intake
m = mass flow rate
P = static pressure
Po = total pressure
PB = back pressure ratio (Pb/Pi)
X = distance along the length of intake
Y = distance along the height of intake
Z = distance along the width of intake

Subscripts
b = back
c = capture
e = diffuser exit location
i, inf = freestream condition
t = throat location
max = maximum distance

Introduction

Study of air-breathing engine flow field has been the topic of active research, during the past few decades due to its practical applications. The requirement of air-breathing engine having better performance has necessitated the development of an efficient air-induction system. The incoming air at supersonic and hypersonic speeds needs to be compressed and delivered to the combustor with possible minimum losses and efficient diffusion. This could be achieved through a suitably designed air-intake. The performance of such air-intakes is governed by many factors which will depend on external and internal compression, contraction ratio, throat area, throat location, divergence of subsonic diffuser etc. Mixed external/internal compression air-intakes are generally being adopted at higher Mach numbers. Presence of leading edge shock systems, terminal normal shock inside the intake, boundary layer interactions and the growing boundary layer in the subsonic diffuser are some of the critical features of an air-intake flow field. Fig.1 shows the schematic of flow field for a typical mixed compression intake at supersonic speed. At the design condition, the external shock system generated by the ramp is expected to get reflected at the cowl tip and will lead to further internal compression with the formation of terminal normal shock near the intake throat, followed by a trailing shock train. Interaction of reflected shock wave with the boundary layer on the ramp surface might lead to flow separation, which in turn may reduce the overall performance of the intake. Intake having flow separations in the internal duct is likely to be more susceptible to unstart or presence of buzz phenomena due

* Senior Lecturer              +Professor
Department of Space Engineering and Rocketry, Birla Institute of Technology, Mesra, Ranchi-835215, India
Email : sudipdas@bitmesra.ac.in; jkprasad.1@gmail.com
Manuscript received on 25 Aug 2008; Paper reviewed, revised and accepted on 11 Mar 2009
to possible flow induced shock oscillations. In addition, the presence of sideplate induces three-dimensional effects, which will further complicate the flow field. Studies are being made to avoid these problems, by adopting various methods like bleeding, variable geometry, mass spillage, isolators, diffuser shapes, vortex generators, etc., to improve the performance. Each of these methods has its own advantage and disadvantage due to operating range and associated complexities and incorporation of additional sub systems.

Extensive studies are made by Murakami et al. [1], Smart et al. [2] and Dawei et al. [3] to characterise air-intakes on a fixed geometry. Adoption of variable geometry and boundary layer bleed to improve intake performance is reported by Watanabe et al. [4]. Studies of air-intake on geometry at design Mach number by Anderson et al. [5], and Neale et al. [6] indicated the unstart of intake. Adoption of different techniques like moving geometry, bleeding, etc., indicated the start of intake at Mach numbers higher than design Mach number.

The internal shock characteristics of a rectangular intake has been captured through experiments by Hirschen et al. [7]. It is reported that the shape of diffuser is responsible for strong shock-boundary layer interaction and the separation of flow. Hermann et al. [8] and Reinartz et al. [9] have reported the effect of isolator lengths on flow field in a hypersonic inlet through numerical and experimental studies which improved the flow quality. Simulations of a mixed compression three-dimensional supersonic inlet carried out by Mizukami et al. [10] reports the existence of complex flow field near the junction of cowl tip and sideplate. Unsteadiness in flow field inside intake ducts, at supercritical condition has been captured adopting computations by Liu et. al. [11], Hsieh et. al. [12], Biedron et. al. [13] and Newsome [14], etc.

Valorani et al. [15] has reported the necessity of adoption of bent cowl for mixed compression intake. Studies with bent cowl or suitably curved cowls leads to improvement in performance as reported by Yanagi et al. [16], Kubota et al. [17], Hermann et al.[18], Trapier et al. [19], etc. Computational studies made on two-dimensional intake with cowl bending is reported by Das and Prasad [20].

The present study is intended to obtain the overall flow field details around a fixed geometry rectangular mixed compression air-intake designed for a Mach number of 2.2 with adoption of a cowl, bent at an angle for possible improvement in performance. Three dimensional computations using Fluent and experiments are made to capture the overall flow field inside the intake. Studies are also made to characterise the flow field at different back pressures.

### Geometrical Details of Intake

A rectangular mixed compression air-intake adopted in the present study is similar to the geometry reported by Neale [6], which has design Mach number of 2.2. The dimensional detail of the intake is presented in Fig.2. The external compression is achieved by two ramps having angles of $\theta_1 = 7$ and $\theta_2 = 14$ degrees. The internal surface of the cowl was either parallel to the freestream direction similar to Ref. [6] or had a bending of $\theta_5 = 2$ degrees, such that it is deflected away from the centerline flow upto the minimum area of the duct (intake throat). The upper wall of diffuser is maintained parallel to the freestream direction, whereas the lower wall of diffuser has the divergence angles of $\theta_3 = 2.5$ degrees and $\theta_4 = 6$ degrees. Further downstream it has a constant area duct. The capture height ($H_c$) of intake was 15mm and width was 15mm corresponding to AR = 1.0.

### Experimental Details

The experiments were performed using the Supersonic Wind Tunnel at Birla Institute of Technology, Mesra, Ranchi. It is a blowdown type wind tunnel having test section size of 50mm x 100mm and Mach number ranging from 1.5 to 3.0. Fig.3 shows the photograph of supersonic wind tunnel.

A model, as per the dimensions given in Fig.2 was fabricated using EDM wire cutting machine which en-
sured the dimensional accuracy better than 0.01mm. The overall length of model was 119mm and width of 15mm. The model was made in modular form for ease in handling. Different models were used for flow visualisation and static pressure measurement. A typical photograph of model presented in Fig.4 without one side plate, which shows the inside details of air-intake. Schlieren flow visualisation technique was adopted to capture the overall flow field inside the intake, by adopting transparent side plates fabricated from Plexiglass and hence it had limited transparency. The schlieren photographs were captured using a Digital SLR camera (Model: Sony DSLR A100K).

Static pressures at different locations on the ramp surface were measured by providing pressure ports of 0.8mm diameter and connected to multi-channel mercury manometer. Measurement of pitot pressure at the exit of intake was made using a pitot rake with 5 numbers of probe. This probe assembly could be placed either in horizontal or vertical directions. In order to make studies at different back pressures, arrangements were made to throttle the exit area by traversing a blunt cone plug from the exit. The blocked area could be estimated using limited measurements depending upon the axial locations of the plug. Four numbers of pitot probes placed circumferentially on the plug was used to measure the pitot pressures at exit while throttling the intake exit. All the tests were made with a freestream total pressure of 3.5x10^5 N/m², which was measured using a Sensym made ASCX150DN pressure sensor. Present series of tests were made at a fixed Mach number of 2.2 and Reynolds number of 3.8x10^7 per meter.

**Computations**

Three-dimensional numerical simulations were made to capture the flow field around air-intake using commercial software FLUENT. Geometry of intake and Mach number was similar to the one adopted in the experiments (Fig.2). Computations are performed using finite-volume technique to solve compressible Reynolds Averaged Navier Stokes equations. Explicit Coupled solver was adopted with upwind discretisation scheme for convective terms in flow and transport equations, i.e 2nd order upwind for flow and 1st order upwind for transport equations. Diffusion and source terms are discretised using 2nd order central differencing scheme. Simulations were made adopting "k-ω" turbulence model, which is recommended for use in complex boundary layer interaction flows as reported in Ref. [9]. The computational domain used for the present computations is presented in Fig.5. This has been arrived at after due consideration of effect of external boundaries and as well to minimise the computation time.

The boundary conditions adopted for the computations are also shown in the same figure. The outlet boundaries outside the intake is arrived at after making several iterations to ensure that reduction in the effective computational area do not lead to any adverse effect. This process was very effective in reducing the computation time.
Block structured grids were adopted to compute the external and internal flow features of the intake. Grids were made with uniformly distributed hexahedral cells with variation of spacing in y and z-direction near the walls. A typical surface grid distribution and grids at different planes showing the grid resolutions in the internal duct is presented in Fig. 6. In addition, two-dimensional inviscid simulation was also made adopting uniformly distributed quadrilateral cells prior to three-dimensional simulations, to confirm the theoretical flow field features of the intake design.

**Boundary Conditions**

Inlet boundary was defined by specifying stagnation and static pressures corresponding to Mach 2.2. A small freestream turbulent intensity and a calculated viscosity ratio were specified at inlet for turbulent computations. Pressure outlet boundary condition was adopted for all the outlet boundaries. Symmetry boundary condition was adopted as only one half of the model was considered. No-slip boundary conditions were enforced at all the solid walls. Computations are made for free exit flow and also with pressurised exit condition. Free exit flow indicates supersonic flow at the exit where the flow variables were extrapolated from interior grids. In case of computation with back pressure, a pre-defined back pressure was enforced by specifying flow condition corresponding to subsonic flow.

**Grid independence Tests**

For three-dimensional computations, grids were refined at different levels inside the intake duct. Results for three different grids, Fine (60x35x150), Medium (40x25x114) and Coarse (30x20x88), were obtained. Fig. 7 shows the computed pressure distribution on the ramp surface of the intake for these three grids. This indicates that results are within 5%. Based on this comparison and as well considering computation time, it was decided to adopt the medium grid (40x25x114). All the computations were performed on a workstation and CPU time of approximately eight hours was required to achieve a converged solution.

**Convergence**

Computation was initiated by specifying initial value to the entire domain as specified for inlet boundary condition. During computations, the residuals of continuity and turbulent kinetic energy were monitored. In addition, mass flux between inflow and outflow boundaries and the \( y^+ \) value on the ramp surface was also monitored to confirm the solution convergence. A 4-stage multigrid was adopted for faster convergence. Fig. 8 shows the typical monitor for the converged RMS residuals.
Validation

Experimental results reported in Ref. [8-9] for a typical hypersonic inlet geometry at supersonic speed has been used to validate the present three dimensional computations. Simulations were made on similar geometry [Ref.9] for free flow exit condition. Comparison of the pressure distribution obtained from the present computation with experimental results on the ramp surface is presented in Fig.9.

It shows fairly good agreement and indicates the sufficiency of grid distribution, turbulence modelling, boundary conditions etc., being adopted for the present computations. Based on reasonably good comparison achieved, further computations are made for the present intake geometry with similar grids, boundary conditions and turbulence model.

Results and Discussion

Numerical simulations and experiments have been made on a rectangular mixed compression air-intake designed for Mach 2.2 having aspect ratio of 1. Experiments were made initially with free exit flow (corresponds to no throttling in the experiments and no enforcement of back pressures for computations at the diffuser exit). Under the free flow exit definition, the flow is supersonic at the exit. In order to study the flow field with different back pressures, experiments are made by restricting the exit area using a plug at the exit, whereas a back pressure was enforced at the exit for computations. The results obtained from free flow as well as pressurised exit conditions are presented and discussed.

Free Flow Exit

Computations were made on two-dimensional intake geometry with zero cowl bending for inviscid case to ascertain the design condition. Numerical schlieren obtained from the inviscid simulation is presented in figure 10, which shows the oblique shocks converging on cowl lip and the shock reflections and expansions in the diffuser duct. Three-dimensional inviscid simulations were also made on this geometry which indicated start of intake. Simulations were made adopting k-ω turbulent model on similar geometry having aspect ratio of 1. Results indicated the unstart of intake, which could be seen from numerical schlieren presented in Fig.11. The unstart of the intake could be characterised by the internal shock reflections in the duct. Strong interaction between the reflected shock from the cowl tip and the boundary layer on ramp is anticipated which could cause flow separation near the throat of intake. This separation might lead to increase in the boundary layer thickness and hence decrease in throat area leading to formation of a near normal shock just upstream of the throat.

Experiments are made to capture the flow inside the intake adopting transparent side plate. The observed flow field through schlieren system is presented in Fig.12, which clearly indicates the presence of normal shock just upstream of throat. Comparison with corresponding computed results indicate similar behaviour. Results from computation and experiments indicated unstart of the intake, hence attempts were made to reduce the strength of

![Fig.10 Numerical Schlieren for Inviscid Case without Cowl Bending](image1)

![Fig.11 Numerical Schlieren without Cowl Bending](image2)
reflected cowl shock by providing a small bending to cowl. Computations were made for the intake having cowl bent angle ($\theta_5$) of 2 degrees. This angle has been arrived at after making computations at different angles ranging from 0.5 to 4 degrees in steps of 0.5 and indicated maximum improvement in performance at 2 degrees [Ref.20]. The corresponding numerical schlieren at $\theta_5 = 2^\circ$ is presented in Fig.13. Comparison with results on intake without bending (Fig.11), clearly indicates that the provision of a small cowl bending has led to disappearance of normal shock at the cowl tip. Experiments were made with cowl bending of 2 degrees and the shadowgraph showing the flow inside the intake is presented in Fig.14. Comparison with Fig.12 also indicates the start of intake due to presence of cowl bending and there is good correspondence between computed and experimental result.

Figure 15 shows the comparison of measured and computed static pressures on the centre line of ramp surface with and without cowl bending. A steep pressure rise...
at the end of 2nd ramp observed for the intake without cowl bending indicates the presence of normal shock, which is also seen from schlieren presented in Fig.11 and 12, whereas for the bent cowl, supersonic diffusion is observed from the schlieren presented in Fig.13 and 14. Series of compression and expansion waves are captured in the duct. These results indicate a reasonably good comparison between computations and experiments. Mach number at the diffuser exit along the width of intake estimated through measurement of pitot pressures and static pressure near the exit is presented in Fig.16, for the case of with and without cowl bending. The bending of cowl leads to increase in Mach number.

Figures 13 and 14 indicated that the external oblique shocks do not coalesce at the cowl lip, which may be due to the presence of sideplates. Numerical schlieren as well as the shadowgraph show the presence of small spillage near the cowl tip. Velocity vectors on the sideplate presented in Fig.17 for 2 degree cowl bending, indicates the presence of a small separation zone near the junction of cowl tip and sideplate. It may be due to interaction of shock and boundary layer on the sideplate as reported in Ref. [10]. The enlarged view of velocity vectors at different spanwise stations i.e $Z/Z_{max} = 0.97$, 0.78 and 0.63, emphasizing the flow field near cowl tip junction is presented in Fig.18. The bending of flow towards the ramp surface seems to decrease with increase in distance from sidewall. The leading edge angle of sideplate provided could also be responsible for separation seen near the cowl tip / sideplate edge and induced spillage. Computed pressures on ramp and cowl surfaces at different spanwise locations indicated small difference, indicating that the effects on the overall flow field is likely to be small. However for off-design operation of intake, the effect could be appreciable.
Measurement of pitot pressures at the diffuser exit were made at 6 locations in $y$-direction and 5 locations in $z$-direction. Comparison of pressure recovery obtained through experiments and computations is shown in Fig. 19. Pressure recovery of the intake, which is defined as ratio of averaged total pressure at the exit and freestream total pressure needs to be obtained. As the distribution has some deformation near the surface, averaging of the values have been made by excluding the values obtained up to a distance of 20% of width and height from the sidewall, cowl and ramp surfaces. Computed pressure recovery for the intake with bent cowl is about 93.6% which is about 8% higher than without cowl bending.

Pressurised Exit

Studies have been made to obtain the flow field with pressurised exit, which represents the conditions of the engine inlet. In computation, the back pressure ($PB$) could be enforced at the exit of intake, where $PB$ is defined as ratio of exit pressure and freestream pressure. In experiments, back pressure was changed by blocking the exit area with the help of a conical plug (Fig. 4) which could be traversed. This is represented as ratio of area at the exit to throat area ($A_e/A_t$). These studies have been made with cowl bending of 2 degrees only.

Figure 20 show the numerical schlieren along the centreline of intake having cowl bending of 2 degrees at $PB = 6$. Formation of normal shock and separation of flow in the diffuser section could be captured. Comparison with the numerical schlieren (Fig. 13) observed for free exit flow ($PB=1.3$), the change in flow field due to back pressure could be seen. Separated zone on the cowl and ramp surface has different behaviour indicating that flow at the exit is not likely to be uniform. Centreline pressure distribution presented in Fig. 21 shows most of the features observed in numerical schlieren. Fig. 22 and 23 shows the schlieren and measured pressure at $A_e/A_t = 1.15$. There is no direct way to correlate $A_e/A_t$ and $PB$. Based on the similarity in flow feature, it seems that $A_e/A_t = 1.15$.
corresponds to PB=6, however this needs further investigation.

Effect of back pressure (PB) on the overall flow field in the intake could be seen through the numerical schlieren presented in Fig.24. The location of normal shock moves upstream with increase in PB and almost approaches to throat location at a value of PB=7.2. Computation with PB larger than this value indicated the presence of instability and limits the critical operating range of intake.

Computed centreline pressure distribution on ramp surface at different back pressures is presented in Fig.25. Movement of normal shock with increase in PB is also captured through jump in pressure. Downstream of the normal shock, gradual increase in pressure represents the presence of a stable shock system in the duct. It was observed that a small increase in PB beyond 7.2, leads to oscillations in the normal shock and further increase in PB will lead to subcritical operation of intake.

Measured centreline pressure distribution on ramp surface for different $A_c/A_t$, is presented in Fig.26. The behaviour is almost similar to Fig.25. With decrease in $A_c/A_t$,
the normal shock moves upstream. A stable normal shock in diffuser duct is observed upto $A_e/A_0=1.15$. A small reduction in $A_e/A_0$ leads to oscillation of shock in the region of second ramp and throat. Further reductions in $A_e/A_0$ lead to expulsion of shock system indicating sub-critical operation of intake.

Figure 27 shows the computed normalised total pressure distribution across the diffuser exit height at the centre of intake ($Z/Z_{\text{max}}=0.5$). The behaviour is highly non-uniform in comparison to free exit flow (Fig.19a). The location of maximum total pressure move towards ramp surface with increase in PB, which could also be seen through the numerical schlieren presented in Fig.24.

![Fig.27 Computed Total Pressure Distribution at Diffuser Exit with Different back Pressures](image)

![Fig.25 Computed Centerline Pressure Distribution at Different Back Pressures](image)

![Fig.26 Measured Centerline Pressure Distribution for Different Throttled Exit](image)

![Fig.28 Computed Total Pressure Distribution at Diffuser Exit (PB = 6.0)](image)
The total pressure distribution at various Y/Y\text{max} and Z/Z\text{max} at the diffuser exit for a typical pressurised exit condition of PB=6.0 is shown in Fig.28. The variation of total pressures along the width and height at different stations is presented in Fig.28, which shows the variation in total pressure indicating non-uniformity of flow. Contours of computed total pressure at different streamwise locations for supercritical and critical operation of the intake is presented in Fig.29, which also shows increase in non-uniformity both in longitudinal and lateral directions.

Measurements of pitot pressures were made using a rake having four pitot pressure tubes only. The value of maximum total pressure for different A∞/A∞ presented in Fig.30 indicates the maximum pressure recovery of about 83.4% at A∞/A∞ = 1.15, indicating critical operation. As the flow is highly three dimensional, attempt has been made to compute overall pressure recovery by averaging the pressures at the exit excluding highly non-uniform zone near the solid surfaces of intake as done in the case of free exit flow. This has been done to arrive at a nominal pressure recovery. The computed values are presented in Fig.31 for the case of with and without cowl bending. This indicates a definite gain in performance of intake with cowl bending indicating the possibility of adopting cowl bending as one of the means to improve the performance.

Conclusions

Experimental and computational studies have been made to obtain the flow field in a rectangular mixed compression supersonic intake designed for Mach 2.2. Inviscid computations without cowl bending indicated intake start whereas computations with turbulence and experiments shows unstart, which could be due to a flow induced separation on the ramp surface. Presence of a small bending of cowl, leads to start of intake and as well improvement in flow quality inside the intake. Studies are also made with bent cowl for free exit flow and pressurised exit flow condition. Comparison between experimental and numerical simulation indicates reasonably good agreement. Effect due to sideplates on the internal flow of intake seems to be small. Use of either throttling or application of back pressure at the exit, leads to movement of terminal shock in upstream direction. Results show the
presence of boundary layer separations on the ramp and cowl surfaces. It is observed that provision of a small cowl bending lead to better performance of intake. Therefore, adoption of bent cowl could be considered as one of the means to alleviate starting problems of intake and as well to improve performance of air-intakes.

Acknowledgement

The authors sincerely thank Dr. Vinay Sharma and Mr. Sorai Mahato of Department of Production Engineering, BIT Mesra, for model fabrication using EDM wire cut machine.

References


7. Hirschen, C., Herrmann, D. and Gulhan, A., "Experimental Investigations of the Performance and Un-


