

Effect of geometric variations on three-dimensional flow separation in a practical scramjet inlet

Nilesh Rane* Vighnesh Pawar† Krishnendu Sinha‡
Department of Aerospace Engineering
Indian Institute of Technology Bombay
Mumbai-400076, India

Abstract

Three-dimensional simulations of a practical scramjet inlet is performed to study the shock-boundary layer interaction in the inlet duct. The cowl shock interacts with the boundary layers on the side wall and the opposite wall. Numerical simulations provide a detailed understanding of the complex three-dimensional flow separation and shock structure. Further, effect of the location and strength of expansion fans upstream of the shock impingement point is investigated. It is found that a careful placement of the expansion corners can limit the extent of the separation region significantly.

Keywords: hypersonic flow, inlet duct, shock wave, flow separation

1 Introduction

In a hypersonic inlet, the incoming flow is compressed through a series of oblique shock waves. These shock waves interact with the boundary layer that forms on the inlet wall. The adverse pressure gradient of the shock can be strong enough to cause local flow separation and reattachment. Separation bubble act as blockage in the inlet duct and can cause problems in starting, high localized heat transfer rate and skin friction, and degrade the quality of the flow entering the combustion chamber. In three-dimensional configuration, the shock boundary-layer interaction and resulting flow separation can be quite complex. It is important to understand the flow physics in detail, so as to explore ways of minimizing or eliminating flow separation and its adverse effect. Complex geometries of practical hypersonic inlets are therefore broken into fundamental pieces and studied in detail to understand the flow phenomenon thoroughly. These pieces can then be put together to understand the whole picture.

An extensive amount of research has been done on three-dimensional shock boundary layer interaction. The most fundamental interaction is generated by a single fin attached on a plate. The planar shock generated by the fin interacts with the boundary layer on the adjacent plate. The flow field generated by such a single fin configuration is studied in detail by Panaras [1], Settles [2] and Delery [3]. The crossing shock wave generated by two fins results in a more complex flow separation pattern on the adjacent plate. This double fin configuration is studied experimentally by Zheltovodov *et al.* [4] and computationally by Gaitonde *et al.* [5, 6].

The three-dimensional shock boundary-layer interaction in an inlet duct can be characterized in terms of the canonical flows discussed above. The shock wave generated by the cowl results in a single fin interaction on the side wall of the inlet. Interaction of the cowl shock on the opposite wall often results in a flow field similar to that seen in a double fin crossing shock wave boundary-layer interaction. In an earlier work [7], shock boundary-layer interaction in a rectangular duct was studied using Computational Fluid Dynamics. The geometry represents a simplified scramjet inlet, and the flow field shows marked similarities with that observed in a double fin configuration. The complex three-dimensional shock structure formed in the duct was found to exhibit many of the features observed by Gaitonde *et al* [5]. The three-dimensional separation of

*Post graduate student; e-mail:nileshjrane@aero.iitb.ac.in

†Research Assistant; e-mail:vighnesh@aero.iitb.ac.in

‡Corresponding author: Address: Assistant Professor, Department of Aerospace Engineering, Indian Institute of Technology Bombay, Mumbai-400076, India; e-mail:krish@aero.iitb.ac.in

the boundary-layer and the resulting longitudinal vortex formed in the duct are also similar to that observed in a double fin geometry.

In this paper we present three-dimensional simulations of a practical scramjet inlet geometry. Compared to the rectangular duct case [7], the baseline geometry described in the following section consists of multiple expansion corners and multiple cowl shocks, which may alter the shock-boundary-layer interaction in the inlet duct. The objective is to understand the three-dimensional shock structure and flow separation in a practical inlet geometry. Further, the effect of certain geometric variations on the inlet flow field is investigated. In particular, the number and location of the expansion corners upstream of the cowl shock impingement point are varied systematically. The resulting effect on the size of the separation bubble is discussed in detail.

2 Base line geometry and test matrix

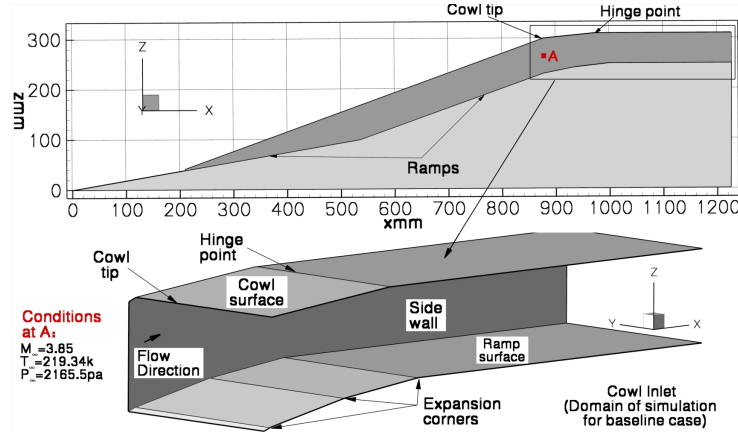


Figure 1: Three-dimensional scramjet inlet configuration, for baseline case.

A baseline scramjet inlet geometry is shown in Fig. 1. The total length of entire geometry is 1220 mm. The cowl-bent angle at the top is 4° and the cowl length from cowl tip to hinge location is 150 mm. Total three expansion corners are placed on the ramp wall. Height of the intake duct is 60 mm. A separate two-dimensional simulation for the symmetric plane of entire geometry was done with free stream conditions $M_\infty = 6.5$, $T_\infty = 220$ K and $P_\infty = 2000.3$ Pa. Figure 2 shows the normalized pressure contours on symmetric plane for two-dimensional simulation of the geometry. The part of the geometry before the first expansion corner on the ramp is regarded as fore-body and the remaining part of the geometry is adopted as a cowl inlet geometry for the baseline case-1. The exit velocity profile is extracted at the end of fore-body and was given to baseline case as inlet (see inset Fig. 2). The conditions noted at the cowl inlet, in inviscid region (point 'A' in Fig. 1), are approximately $M = 3.85$, $T = 525.5$ K and $P = 26$ kPa. The total length of the inlet duct from first ramp expansion corner to the end is 350 mm.

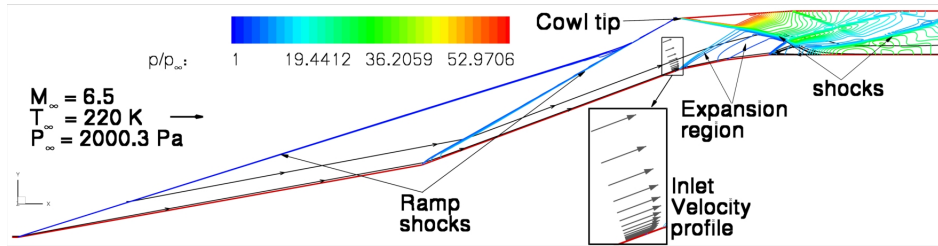


Figure 2: Normalized pressure contours on symmetric plane, obtained from two-dimensional simulation of scramjet inlet.

Table 1: Geometric details of three-dimensional inlet baseline case and variations with respect to it.

Case no.	Cowl bent angle, deg	Expansion corners	Position of last expansion corner form 1 st corner, mm	Change from baseline
1	4	3	120	Base-line case
3	4	3	130	Last expansion corner moved aft ward
4	10	curved	-	Curved expansion instead of 3 corners
5	10	3	150	Last expansion corner moved aft ward
6	10	2	150	Central expansion corner removed

3 Simulation methodology

We solve the Reynolds Averaged Navier-Stokes equations for the mean flow, as presented in Ref. [8]. The Spalart-Allmaras (SA) model is used for calculating the eddy viscosity. In-house code of Sinha *et al.* [9] is used in the simulations and is run on parallel machines using MPI. The governing equations are discretized in a finite-volume formulation where the inviscid fluxes are computed using a modified (low-dissipation) form of the Steger-Warming flux-splitting approach [10]. The method is second order accurate both in stream-wise and wall normal directions. The implicit method of Wright *et al.* [11] is used to integrate in time and to reach steady-state solution. The code is validated in several high-speed flow application [12, 13].

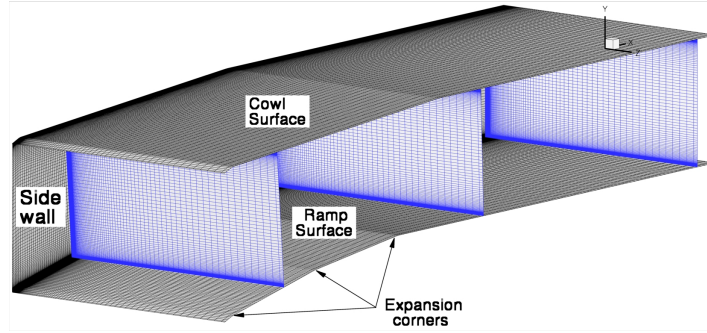


Figure 3: Grid used in simulations, for base line case.

A structured body-fitted grid has been used for simulating this geometry as shown in Fig. 3. Since the geometry is symmetric about the centre plane, only half of it is simulated. Based on the grid refinement study, the grid used for the simulation is sufficient to capture overall flow physics throughout the domain. The single block grid used for the simulations has about 200x200x200 grid cells. The grid is made fine near the walls and the cowl tip and expansion corners to capture the boundary layers, shocks and their interactions properly. The maximum $y2^+$ value calculated throughout the ramp surface is less than 1.7. The simulations were run parallelly on sixteen 2.66 GHz Intel processors of a Linux cluster. A total of 24,000 time-steps were computed at a maximum CFL number of 2000 to reach a steady-state solution. A typical simulation takes about 60 hrs to obtain a fully converged solution. The topological characteristics of grids used for all other cases is same with reference to baseline case.

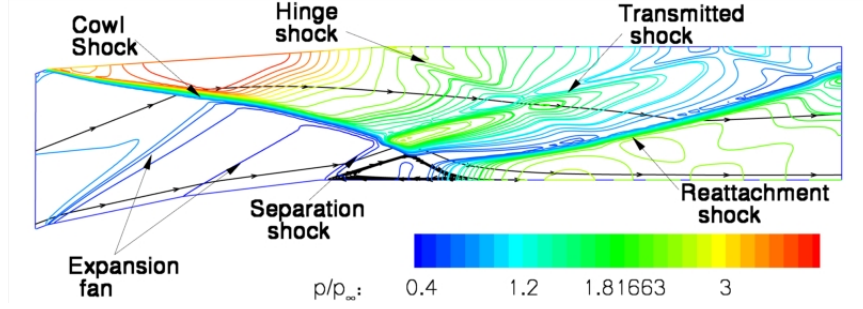


Figure 4: Normalized pressure contours in symmetry plane.

4 Base line results

The shock structure on the symmetry plane is shown in Fig. 4. Cowl shock hits the ramp wall and interacts with boundary layer on the ramp surface and forms a separated flow region. The cowl hinge shock is weak, and multiple expansion fans are visible clearly. The cowl shock bends when it interacts with expansion fans and there is an increase in its strength. Because of this the whole conical separation region on side wall is curved, as seen from the streamlines on sidewall (see Fig. 5). The conical separation zone is similar in structure to that in a single fin interaction. Hinge shock being weaker, there is no separation because of it. Also the separation bubble on the ramp wall shows same patterns as seen in the rectangular duct [7]. The features like foci, saddle points, and nodes and primary separation and reattachment lines are clearly shown in Fig. 5.

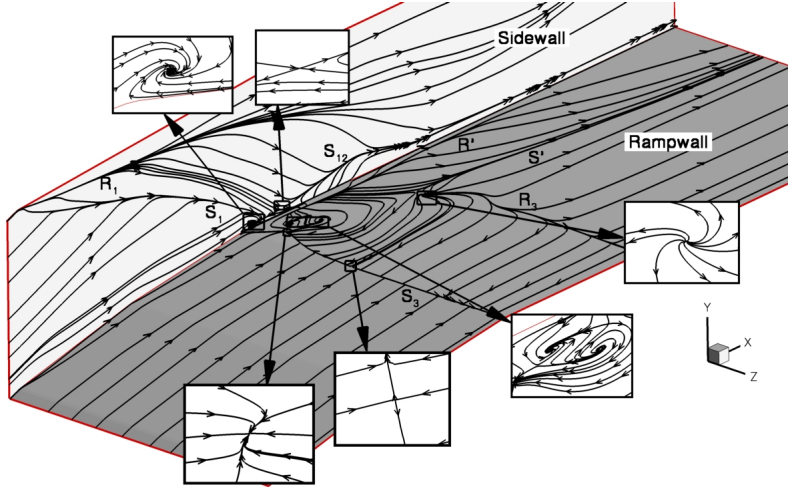


Figure 5: Surface streamline pattern on side and ramp wall, for base line case.

Figure 6 shows flow pattern in the core of the separation bubble. The fluid in the inner core comes from the sidewall and lifts off from the focus located on ramp wall, near the sidewall. Some of the flow from bottom-most part of separated boundary layer on ramp wall mingles with the separation bubble flow and forms the outer core. The fluid layers above this just envelop the separated region. Just as some fluid goes into the separated region, some from separation bubble escapes with this enveloping fluid layers. A lambda shock structure presented in [7] is formed due to the interaction between cowl shock and sidewall boundary layer (see Fig. 7).

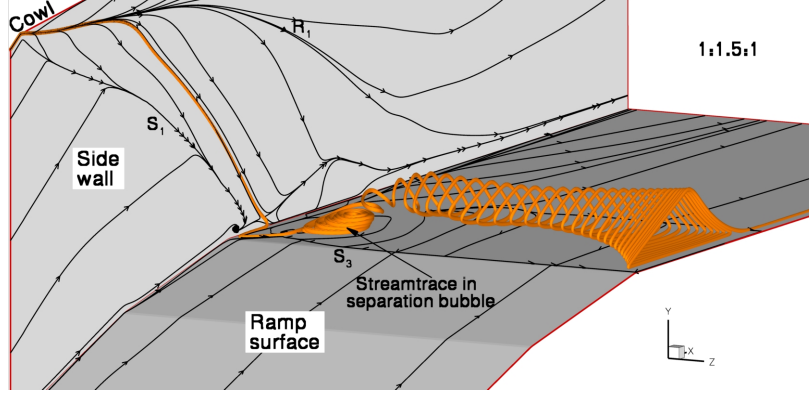


Figure 6: Flow pattern inside the separation bubble on Ramp surface for baseline case.

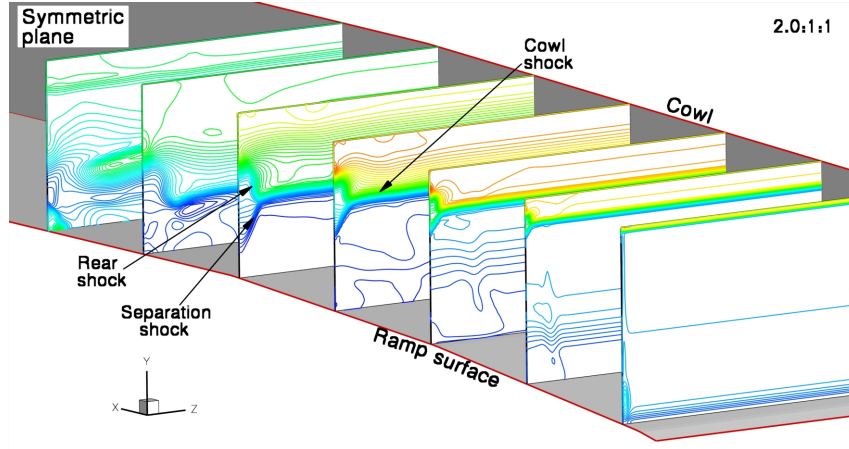


Figure 7: Normalized pressure depicting three-dimensional lambda shock structure on planes in flow normal direction.

5 Effect of expansion corner

To study the effect of expansion fans, four variations were tried. The baseline case-1 has three expansion corners. The last corner stands at 120 mm from the 1st corner, which is fixed in all cases. The case-4 has curved expansion surface instead of these three corners. In case-3, the last expansion corner is moved to 130 mm and in case-5 it is moved further downstream to 150 mm. Case-6 has same position of last corner as case- 5, but the middle expansion corner is eliminated. This increases the slope of the ramp upstream of the last corner. The effect of these changes on the flow field is studied.

From pressure contours on the symmetry plane shown in Fig. 8, no major change in the overall shock structure is observed. Case-4 has one expansion fan spread over a span instead of three distinct ones. The only noticeable change is the width of separation region marked by black streamlines in the Fig. 9. The separation is the least in case-6 and maximum in case-4 (see Table 2).

On the ramp wall (see Fig. 9), it can be seen that the position of reattachment line is almost constant (see Table 2). As reattachment line always lie just downstream of inviscid shock impingement location, it is clear that the impingement of shock is not getting affected much by change in the configuration of expansion fans. Only a small shift in the impingement point in aft ward direction is seen as the expansion corner is shifted downstream. This is obviously because of the lesser bending of cowl shock by the expansion fan as it is moved further downstream. The last expansion corner seems to be restricting the separation, as the separation line recedes back with the expansion corner in case-3. But the separation bubble slightly overshoots the expansion corner. The corner is shifted further downstream in case-5, the separation zone

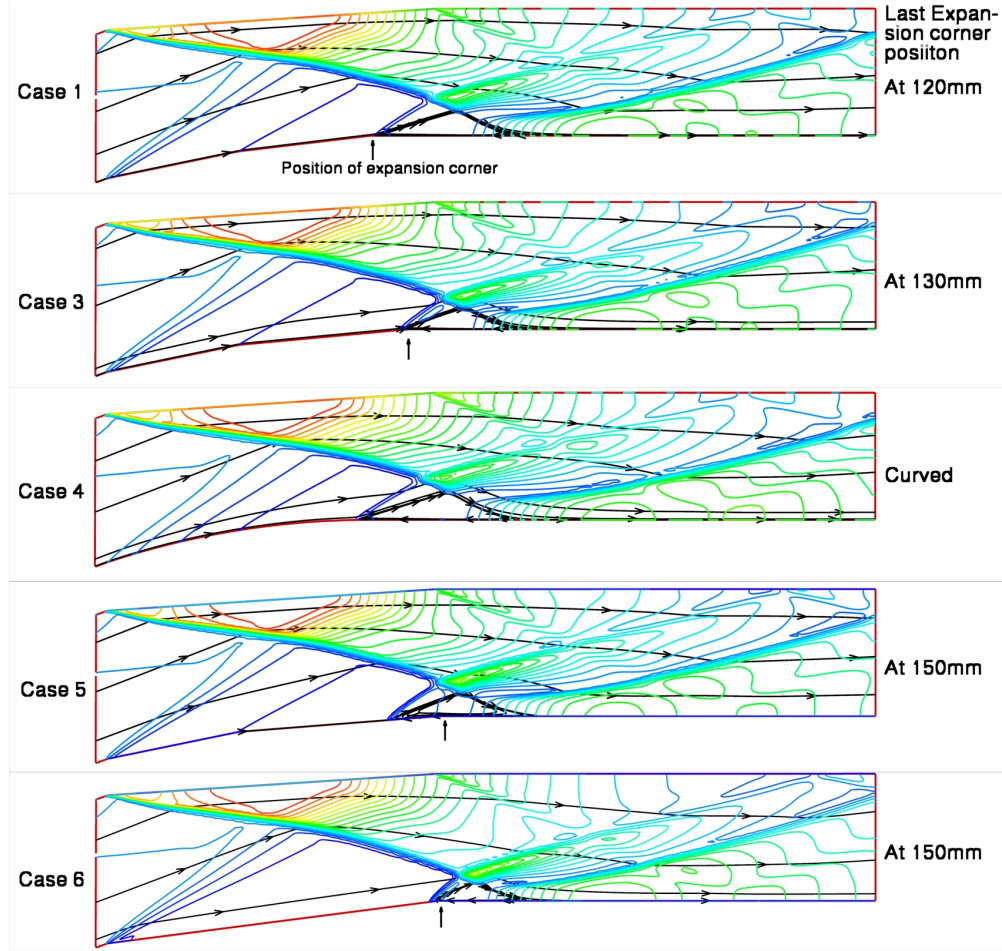


Figure 8: Normalized pressure contours on symmetric plane for case-1,3,4,5 and 6.

becomes as big as the baseline case. This clearly shows that, the expansion corner cannot restrain the separation bubble beyond a certain limit, after which we don't get any significant reduction in separation as compared to the baseline configuration.

Table 2: Effect of geometric variations of expansion corners on size of separation bubble.

Case No.	Position of last expansion corner from fixed 1 st expansion corner, mm	Ramp angle before last expansion corner, deg	Separation line X_1 , mm	Reattachment line X_2 , mm	Size of separation bubble $X_2 - X_1$, mm	% change from baseline case
1	120	6.9	121	194	73	0
3	130	5.9	132	194	62	-15
4	-	0.0	111	195	84	+15
5	150	4.6	148	199	71	-3
6	150	7.6	128	199	51	-30

The curved expansion corner is a circular arc, and thus has very small slope near the horizontal part of the ramp wall. This near-flat portion allows the separation bubble to spread in upstream direction. It seems that the slope of the ramp also has some role to play in restraining the separation line. To check this, case-6 was simulated, in which central corner was eliminated. This increases the slope of the ramp upstream of last expansion corner. From Fig. 9 it is clear that this increased slope helps in restricting the separation line

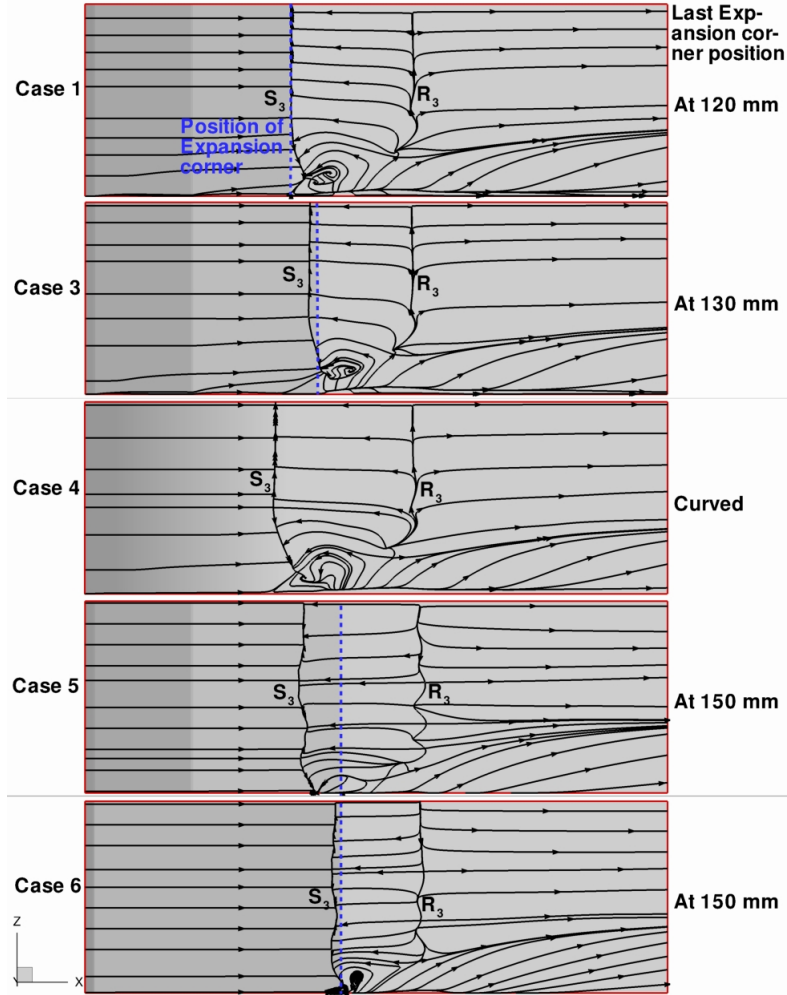


Figure 9: Surface streamlines on Ramp surface for case-1,3,4,5 and 6.

near the corner itself. It is clear from the analysis that curved expansion surface, at least with a circular arc, does not help in reducing the separation bubble size. In fact it causes increase in separation bubble size. Shifting the expansion corner restricts the separation. Indeed if the expansion is further extended and made to coincide the inviscid impingement location of cowl shock, the inviscid theory predicts that the shock and expansion fan will negate each other (not necessarily completely) and no separation will be there. In practice, there will be a separation bubble, as the separation bubble can overshoot the expansion corner.

In essence, shifting the expansion corner and increasing the slope of the ramp in the region upstream of the expansion corner together can reduce separation significantly by 30% in case-6. Optimization of these two parameters can be done to achieve least separation. Also it might be better to use sharp expansion corner rather than a curved one. It is effective in reducing the separation bubble significantly. The reason why an expansion corner could constraint the separation bubble is not clear, nor the extent to which it can be effective is clear. Further study is needed to investigate this matter in details. Note, that even in the baseline case-1, the separation bubble is constrained, which otherwise would have been much bigger. This is one of the reasons why the separation bubble is smaller in case-1 as compared to results in [7].

6 Conclusion

A practical scramjet inlet geometry is studied using three-dimensional simulations, extending the analysis done for a rectangular duct case in [7]. It is found that the analysis for the simple canonical geometry is directly extendable to more complicated realistic geometries, thus underlining the importance of studying shock wave/turbulent boundary-layer interaction using simplified geometries first. This scramjet inlet is used as a control case for further studies. Also effect of various geometric changes is studied through a set of cases. It is found out that the expansion corner limits the separation bubble to a great extent and shifting an expansion corner aft wards and keeping slope of ramp just before the expansion corner higher reduces the separation significantly.

Acknowledgement

We would like to thank Indian Space Research Organization (ISRO) for supporting this research under the RESPOND program.

References

- [1] Panaras, A., 1996, "Review of the Physics of Swept-Shock/Boundary Layer Interactions," Progress in Aerospace Sciences, **32**.
- [2] Settles, G., May 1993, "Shock-Wave/Boundary Layer Interactions in Supersonic and Hypersonic Flows," AGARD Report, Chap. 1, (792).
- [3] Delery, J. and Panras, A., 1996, "Shock Wave/Boundary Layer Interactions in High-Mach-Number-Flows," AGARD Report, Chap. 2, **2**(319).
- [4] Zheltovodov, A. A., Maksimov, A. I., Shevchenko, A. M. and Knight, D. D., 1998, "Topology of Three-Dimensional Separation under the conditions of Asymmetrical Interaction of Crossing Shock and Expansion waves with Turbulent Boundary Layer," Thermophysics and Aeromechanics, **5**(4).
- [5] Gaitonde, D. V., Visbal, M. R. and Shang, J. S., May-June 2001, "Sidewall Interaction in an Asymmetric Simulated Inlet Configuration," Journal of Propulsion and Power, **17**(3).
- [6] Gaitonde, D. V., Shang, J. S., Garrison, T. J., Zheltovodov, A. A. and Maksimov, A. I., December 1999, "Three-Dimensional Turbulent Interactions Caused by Asymmetric Crossing-Shock Configurations," AIAA, **37**(12).
- [7] Pawar, V., Rane, N. and Sinha, K., 2009, "Three-dimensional shock / turbulent boundary layer interaction in a simulated scramjet inlet," Symposium on Applied Aerodynamics and Design of Aerospace Vehicle (SAROD 2009) December 10-12, Bangalore, India.
- [8] Wilcox, D.C., 2006, *Turbulence Modeling for CFD*, 3rd ed., DCW Industries, Inc.
- [9] Sinha, K., Candler, G.V., 1998, "Convergence Improvement of Two-Equation Turbulence Model Calculations," AIAA Paper, **98-2649**.
- [10] MacCormack, R.W., Candler, G.V., 1989, "The Solution of Navier-Stokes Equations Using Gauss-Siedal Line Relaxation," Computers and Fluids, **17**(1), pp. 135–150.
- [11] Wright, M.J., Candler, G.V., Prampolini, M., 1996, "Data-Parallel Lower-Upper Relaxation Method for the Navier-Stokes Equations," AIAA Journal, **34**(7), pp. 1371–1377.
- [12] Pasha, A. A. and Sinha, K., September 2008, "Shock-unsteadiness model applied to oblique shock wave/turbulent boundary-layer interaction," International Journal of Computational Fluid Dynamics, **22**(8), pp. 569–582.
- [13] Pasha, A. A. and Sinha, K., August 2009, "Simulation of three-dimensional shock/boundary-layer interaction in a single-fin configuration," 11th Annual AeSI CFD Symposium 2009, IISc., Bangalore, India.