

Simulation of three-dimensional shock/boundary-layer interaction in a single-fin configuration

Amjad Ali Pasha*

Krishnendu Sinha[†]

Department of Aerospace Engineering
Indian Institute of Technology Bombay
Mumbai-400076, India

Abstract

The three-dimensional single-fin configuration finds application in an intake geometry where the cowl-shock wave interacts with the side-wall boundary-layer. Reynolds-averaged Navier-Stokes computations using standard Spalart-Allmaras model is carried out for three-dimensional single-fin configuration with well documented experimental data. The flow-physics is studied in the complex region of swept-shock-wave turbulent boundary-layer interaction and is correlated to the wall pressure and skin-friction. Computed wall pressure and skin-friction is compared to the measurements.

keywords : high speed flows; single-fin; shock wave; turbulent boundary-layer; separation bubble; turbulence modelling

Nomenclature

C_f	skin-friction coefficient
p	pressure
M	Mach number
V	total velocity
β	fin deflection angle
μ_T	eddy viscosity
μ	molecular viscosity
<i>subscripts</i>	
w	wall condition
∞	freestream condition
0	stagnation condition

1 Introduction

The single-fin configuration consists of a flat plate with a sharp fin mounted perpendicular to it. The oblique shock wave generated by the fin interacts with the turbulent boundary-layer on the plate and results in flow separation. The three-dimensional vortical flow thus generated alters the inviscid flow pattern. Additional shock waves, expansion regions and free shear layers are generated that result in a complex flow in the interaction region. Practical applications of single-fin configuration include scramjet inlets, where the oblique shock generated by the cowl interacts with the side wall boundary-layer.

The single-fin shock boundary-layer interaction flows are characterized by localized regions of increased pressure and heat-transfer rate. Prediction of these surface properties is important in the design of scramjet inlets. In addition, the flow distortion caused by the interaction can degrade the performance of these

*Doctoral student, e-mail:amjad@aero.iitb.ac.in

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

inlets significantly. Computational fluid dynamic approach is a useful tool to understand the complex three-dimensional flow pattern in these shock-wave/boundary-layer interactions and to predict its influence on the wall data. Reynolds-averaged Navier-Stokes method (RANS) is applied along with a turbulence model to compute these flowfields.

Experiments and computations were carried by many authors [1, 2, 3, 4, 5, 6] for single-fin geometry. Panaras [5] computed using RANS code for deflection angle of $\beta = 20^\circ$ and $M_\infty = 3.0$. The surface data was improved using modified Baldwin-Lomax model compared to the standard Baldwin-Lomax model. Edwards *et al.* [3] studied the performance of four different one-equation turbulence models for $M_\infty = 8$ and $\beta = 15^\circ$. Amongst them the standard Spalart-Allmaras model has shown to predict the surface properties close to the experiments. Thivet [6] computed three different single-fin configuration cases with $M_\infty = 3$, $\beta = 15^\circ$, $M_\infty = 4$, $\beta = 20^\circ$ and $M_\infty = 4$, $\beta = 30.6^\circ$, using the standard and modified $k-\omega$ models. The prediction of secondary vortex region was improved using modified $k-\omega$ model, hence improving the wall data in this region. The review articles [7, 8] discusses different single-fin configurations.

In this article we compute the flow of three-dimensional single-fin configuration in detail and study the flow features in the shock-wave/turbulent boundary-layer interaction region, which includes different shock-waves, expansion waves, vortex region and shear-layer. We present the RANS simulations using the standard Spalart-Allmaras model [9] for strongest shock-wave strength test case of Schulein [10], with $M_\infty = 5.0$ and fin deflection angle of $\beta = 23^\circ$. First we describe the test case followed by simulation methodology. In the next section we describe the flow physics of shock-wave/turbulent boundary-layer interaction region. In the last section we correlate flow physics with surface properties and compare the computed wall pressure and skin-friction with the experiments [10].

2 Test case

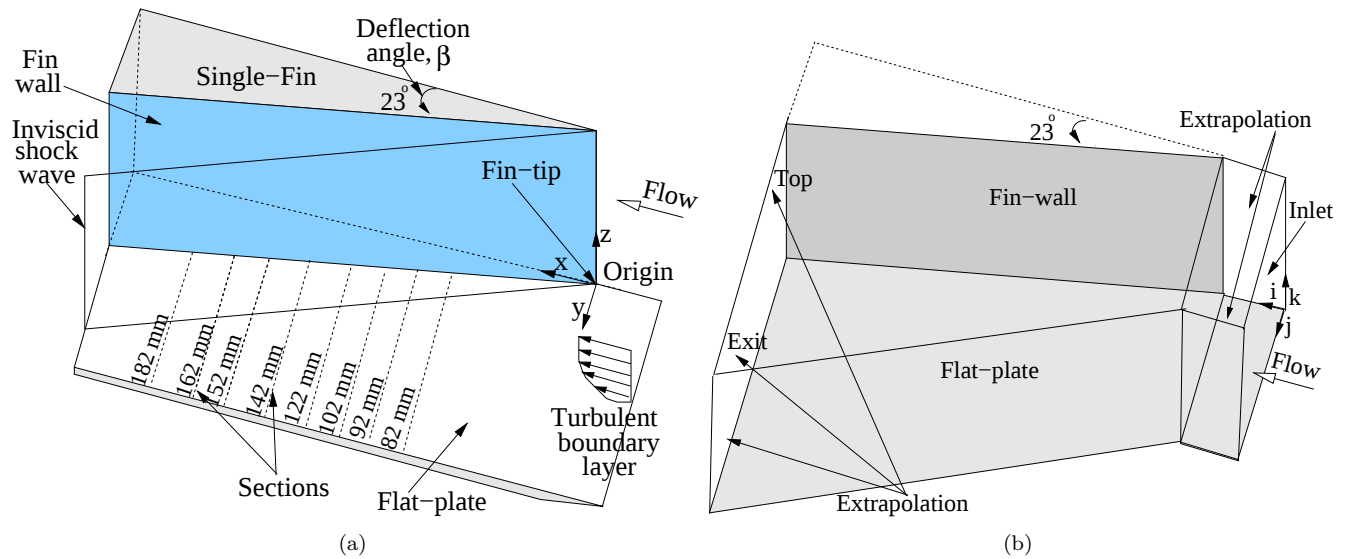


Figure 1: (a) Three-dimensional single-fin configuration with fin mounted on the flat-plate. The surface measurements [10] were taken along the dashed lines, each of them located from the fin-tip in the streamwise direction. (b) Computational domain with appropriate boundary conditions.

Schulein [10] experimental three-dimensional single-fin configuration is shown in Fig. 1a. It consists of a flat-plate and three-dimensional sharp-fin mounted on it. The fin is inclined with deflection angle of β , to the flow direction. Different deflection angles in the range of 8° to 23° were taken with stagnation temperature, $T_0 = 410$ K and stagnation pressure, $P_0 = 2.12$ MPa for $M_\infty = 5$. The fin height = 100 mm is taken, normal to the flat-plate and the fin-tip starts at a distance of 286 mm downstream of the flat-plate edge. Both the flat-plate and fin-wall are maintained under isothermal conditions of 300 K. The flow is turbulent in nature,

upstream of the interaction region with a unit Reynolds number of $Re_{1\infty} = 37 \times 10^6 \text{ m}^{-1}$. The wall data like pressure, skin-friction and heat-transfer rates were measured at different x-sections as shown in Fig. 1a. The undisturbed turbulent boundary-layer properties like δ , δ^* , θ and C_f were measured [10] at different locations on the flat-plate from separate experiments, under the same freestream conditions as of single-fin configuration.

3 Simulation methodology

We solve the three-dimensional RANS equations for the mean flow, as presented in Ref. [11]. The standard Spalart-Allmaras (SA) model [9] without any compressibility corrections is used for calculating the eddy viscosity. In-house code of Sinha *et al.* [12] 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 [13]. The method is second order accurate both in stream-wise and wall normal directions. The implicit method of Wright *et al.* [14] is used to integrate in time and to reach steady-state solution.

The calculated freestream conditions of $T_\infty = 68.3 \text{ K}$ and $p_\infty = 4008.5 \text{ N/m}^2$ are taken in our simulations. The computational domain and boundary conditions are identified in Fig. 1b. At the fin-wall and flat-plate an isothermal wall temperature $T_w = 300 \text{ K}$, no-slip and zero normal pressure gradient are applied. An extrapolation boundary condition is assigned at top and exit planes. For the turbulence quantities, the constant ratio of $\mu_{T_\infty}/\mu_\infty = 0.1$ is taken as freestream conditions [15] at the inlet. The wall boundary condition of $\mu_{T_w} = 0$ is applied at fin-wall and flat-plate. Inlet profiles for the computations are obtained from separate two-dimensional flat-plate simulation at freestream and wall boundary conditions identical to those listed above. The value of the momentum thickness reported in the experiments [10] is matched to obtain the mean flow and turbulence profiles at the inlet boundary of the computational domain. A structured Cartesian mesh is taken and based on the grid converged study a grid size of $140 \times 160 \times 160$ is obtained. The wall units of $y_2^+ < 0.05$ is obtained in the undisturbed boundary-layer region and values of $y_2^+ < 0.06$ are observed in the whole domain. CFL numbers up to 5000 is used in our computations and the solution converges in 30,000 iterations.

4 Flow physics

The isometric view of the flow solution of the three-dimensional swept shock-wave/turbulent boundary-layer interaction for single-fin geometry is shown in Fig. 2. The normalized pressure contours are taken at two x-sections of 92 and 183 mm to identify the shock structure. The flow pattern is depicted by streamlines taken at the cell adjacent to the flat-plate. Very near to the wall these streamlines behave similar to the skin-friction lines. When the flow interacts with the fin-wall at a deflection angle of $\beta = 23^\circ$, it results in the formation of a planar shock-wave as shown in the top plane. When the shock-wave interacts with the turbulent boundary-layer on the plate, it separates the boundary-layer and results in the formation of separation shock. The flow separates at primary separation line S1 and attaches at primary reattachment line R1 near the fin-wall. The separation shock-wave influences the upstream flow ahead of the separation line and this region is represented by line of influence U. The streamlines converge at separation line S1 and the fluid moves upwards normal to the plate and then turns in counter clockwise direction to form sheet of a helical flow as shown in Fig. 3. Two streamline surfaces at $z/\delta_0 = 0.8$ and $z/\delta_0 = 0.25$ are shown, where z is the normal distance from the plate and δ_0 is the undisturbed boundary-layer thickness. These streamline surfaces flowing from upstream of fin-tip, forms a vortical structure in the separated region. High turbulent intensities are observed in the region of vortex core. At the reattachment line the fluid impinges on the plate from top of it making the streamlines diverge in either direction and moves in the downstream direction. The inviscid shock-wave in Fig. 3 bifurcates into a lambda structure and encloses the vortex region. The helical tip of the vortex sheet starts near the fin-tip and forms a conical surface in the downstream direction.

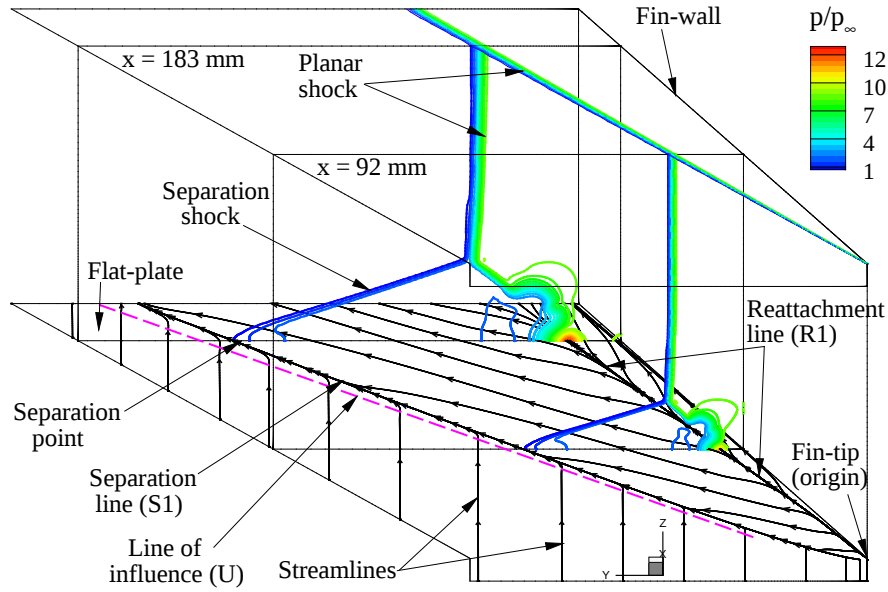


Figure 2: Computed surface streamlines on the flat-plate. The shock structure is shown by normalized pressure contours at x-sections = 92 mm and 183 mm in a single-fin configuration.

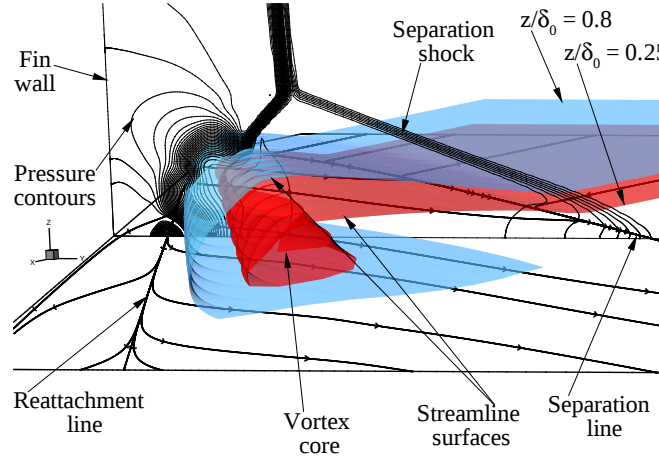


Figure 3: Computed streamline-surfaces starting at $z/\delta_0 = 0.8$ and $z/\delta_0 = 0.25$ showing vortex region, and the normalized pressure contours at x-section = 83 mm, as viewed opposite to the flow direction.

In Fig. 4a, the inviscid shock-wave interacts with the separation shock-wave to form the rear shock. This is a type IV shock-shock interaction [16]. The rear shock-wave emanating from triple point terminates on top of primary vortex. The expansion fan is generated when the rear shock-wave interacts with the shear layer and is shown in Fig. 4a. The flow remains supersonic in the separated region. Small subsonic pockets are observed near the corner region of fin-wall and plate. The computed contours of local stagnation pressure P_0 is normalized with the freestream stagnation pressure $P_{0\infty}$ and is shown in Fig. 5a. The shear-layer is observed over the vortex region, which emanates from the separation point and interacts with the rear shock-wave and then rolls up and turns back to form a tongue. The difference in the velocities across the rear shock-wave and inviscid shock-wave generates a shear layer in the form of the jet. The computed flow features in Fig. 5a match qualitatively with flow-field diagram of experiments [2] in Fig. 5b. The supersonic jet (shear-layer) emanating from the triple point of shock-shock interaction impinges on the flat-plate. High values of skin-friction, wall pressure and heat transfer rates are observed in the impingement region. A small

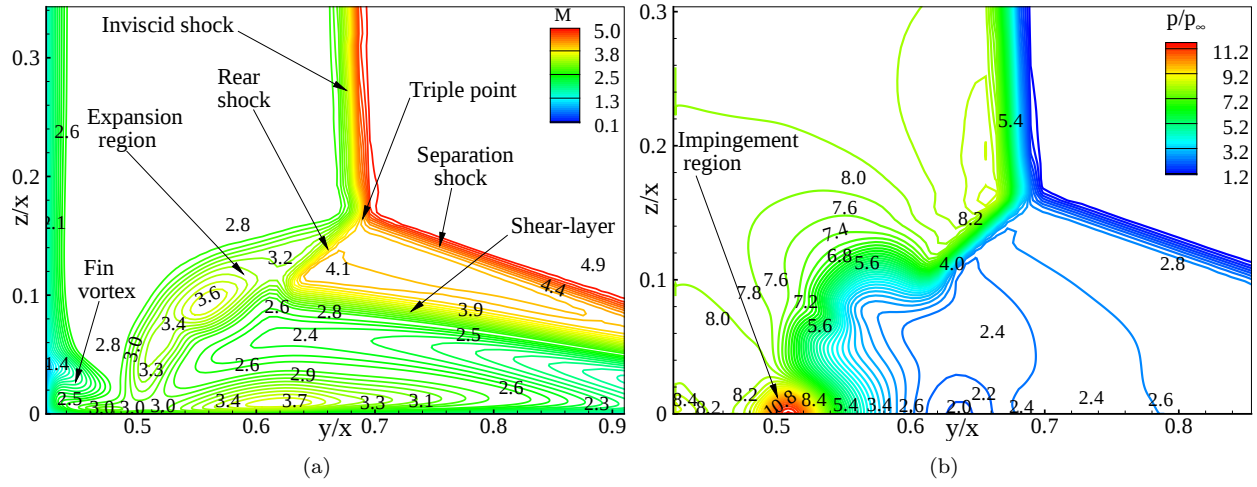


Figure 4: Enlarged view of computed (a) Mach contours and (b) normalized pressure contours, at x-section = 123 mm.

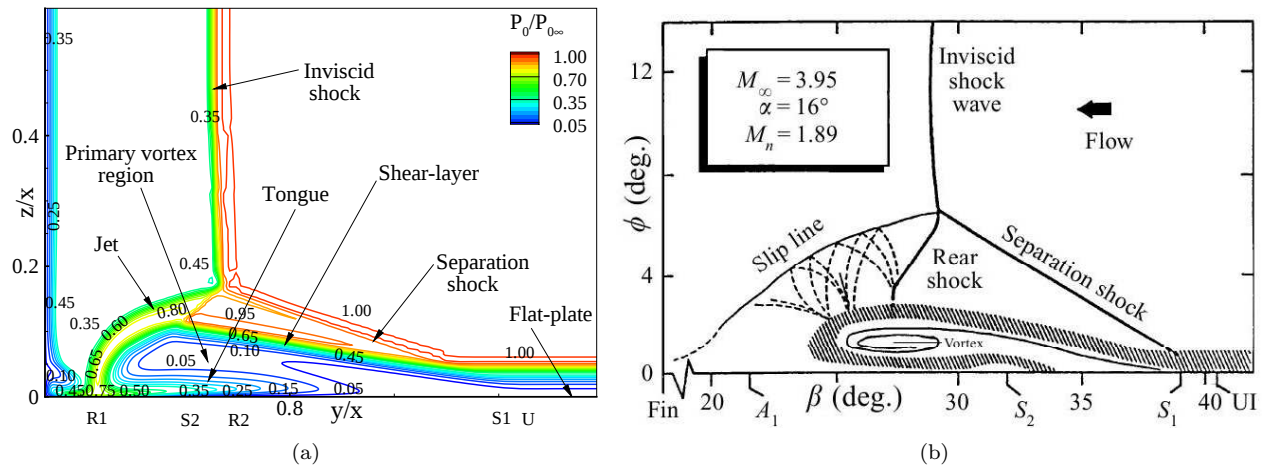


Figure 5: (a) Computed normalized stagnation pressure contours at x-section = 123 mm, for $\beta = 23^\circ$ and $M_\infty = 5$. (b) Experimental [2] pictorial diagram for deflection angle $\alpha = 16^\circ$ and $M_\infty = 3.95$.

fin-vortex is formed when the fluid coming from the inviscid region interacts with fin wall and turns in the clockwise direction, as viewed opposite to the flow direction and this region is depicted in Fig. 4a. This corner-vortex was observed in the vapour screen images of the experiments [1]. A secondary flow separation region was observed in the experiments of Schulein [10]. In the present computation secondary flow is not predicted, but its effect is observed in the region beneath the tongue, marked by S2 and R2.

5 Comparison of computed wall data with experiments

In this section we correlate the flow-physics with wall-data as depicted in Fig. 6. The computed contour plots of normalized local static pressure and velocity are overlapped with the normalized wall pressure and skin-friction on the flat-plate at x-section of 123 mm. In Fig. 6a, the wall pressure remains constant in the undisturbed boundary-layer before the interaction region. The separation shock effects the upstream flow at point of influence U and the wall pressure rises across the separation shock at primary separation point S1. It remains constant in the separated vortex flow and rises to peak values at primary reattachment R1. The wall pressure then decreases away from the reattachment region and rises to small values near the corner

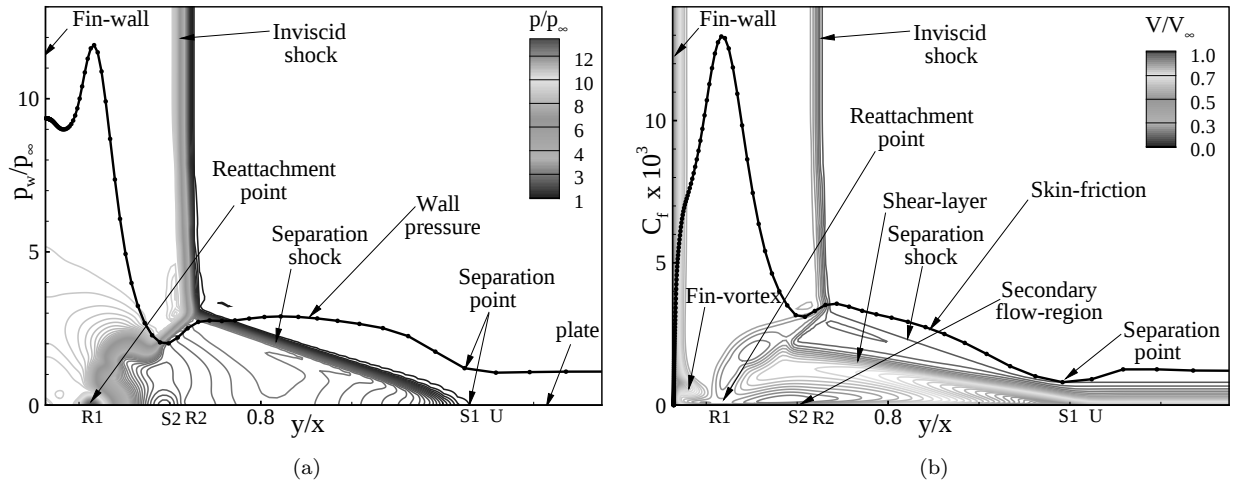


Figure 6: Computed (a) normalized pressure contours overlapped with normalized wall pressure and (b) normalized total velocity contours overlapped with skin-friction, at x-section = 123 mm.

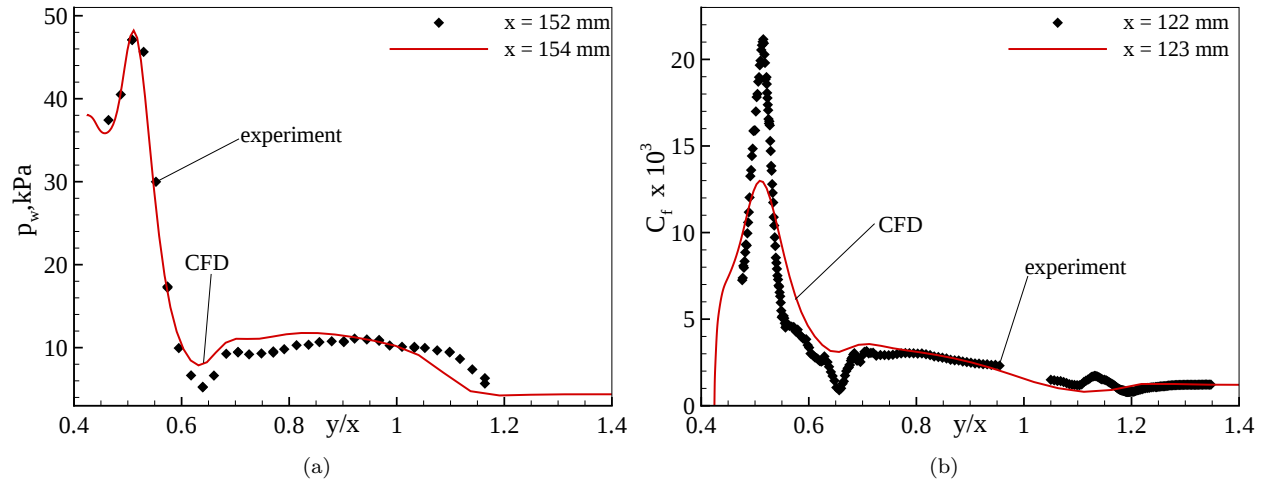


Figure 7: Computed (a) wall pressure at x-section = 154 mm and (b) skin-friction at x-section = 123 mm, compared with experiments [10].

region of plate/fin-wall. In Fig. 6b, the C_f does not vary significantly in the unperturbed boundary-layer before the region of influence U. Across the separation shock-wave the boundary-layer is compressed, hence increases the velocity gradients and increases C_f across it. The C_f rises between S1 and R2. It also rises between S2 and R2. At the reattachment point R1 high values of velocity gradient cause the C_f to reach peak values.

The CFD results of wall pressure and skin-friction variation is compared with experiments [10] in Fig. 7. The distance along y-axis (see Fig. 1b), is normalized with the corresponding x-section distance measured from the fin-tip. The computed initial pressure rise S1 is predicted downstream compared to experiments. The S1 experimental value is extrapolated in Table 1. The dip in the region between S2 and R2 shows higher values. The computed streamlines do not show distinct coalescence and divergence at S2 and R2 but predict dip in this region similar to that of experiments. The pressure distribution is very well predicted in the reattachment region R1 and matches close in the corner region of flat-plate/fin-wall. In Fig. 7b, the S1 is predicted downstream and R1 location is predicted accurately. The C_f is under-predicted by 42% at R1 and is over-predicted between S2 and R2. The measured primary vortex size between S1 and R1 is 82 mm.

The computed y/x values of initial pressure rise at S1, peak pressure rise at R1, values of S2 and R2

x-section, mm	S1		R1		S2		R2	
	experiment	CFD	experiment	CFD	experiment	CFD	experiment	CFD
82	1.3	1.24	-	0.5	-	0.65	0.73	0.71
92	1.27	1.22	-	0.51	0.66	0.65	0.72	0.7
122	1.23	1.16	-	0.51	0.64	0.64	0.72	0.72
152	1.22	1.14	0.52	0.52	0.65	0.65	0.69	0.7
162	-	1.26	-	0.51	-	0.64	-	0.7
182	-	1.12	-	0.52	0.64	0.65	0.68	0.7

Table 1: Computed y/x values of pressure rise at, primary separation point S1, reattachment point R1, secondary separation point S2 and reattachment point R2, compared with the experiments [10] at x-sections. The locations of these points are measured from the region of unperturbed boundary-layer to the fin-wall.

is compared with the experiments [10] and is listed in Table 1 for different sections. At all the x-sections, the computation predicts S1 downstream and underpredicts the size of the primary flow separation between S1 and R1. The y/x values of primary separation point S1, primary reattachment point R1 secondary separation point S2 and secondary reattachment point R2 can be obtained from computed C_f plots similar to that listed in Table 1.

6 Conclusions

In this article, we study three-dimensional shock-wave/boundary-layer interaction in a single-fin configuration with deflection angle of 23° and $M_\infty = 5$. The standard Spalart-Allmaras model is applied and the surface properties are compared with the experiments. The flow physics in shock-wave/boundary-layer interaction region is studied in detail and different features are identified. These include inviscid shock-wave, separation shock-wave, rear shock-wave, expansion waves, jet, shear-layer, fin-vortex, primary separated region of vortex, primary separation line and primary reattachment line. The influence of these flow features on surface properties is explained. The computed pressure distribution in the primary vortex region and the location, and the peak values of wall pressure at reattachment point are predicted accurately. No distinct secondary flow separation region is identified in the computations. The skin-friction distribution is predicted accurately in the undisturbed boundary-layer and in the primary vortex region. The computed skin-friction values are underpredicted in the reattachment region whereas its location is predicted accurately.

Acknowledgment

The authors thank Indian Space Research Organization (ISRO) for supporting this research under the RESPOND programme.

References

- [1] Kubota, H., Stollery, J., 1982, "An Experimental Study of the Interaction between a Glancing Shock Wave and a Turbulent Boundary Layer," *Journal of Fluid Mechanics*, **116**, pp. 431–58.
- [2] Alvi, F.S., Settles, G., 1992, "Physical Model of the Swept Shock Wave/Boundary-Layer Interaction Flowfield," *AIAA Journal*, **30**(9), pp. 2252–2258.
- [3] Edwards, J.R., Chandra, S., 1996, "Comparison of Eddy Viscosity-Transport Turbulence Models for Three-Dimensional Shock-Separated Flowfields," *AIAA Journal*, **34**(4), pp. 756–763.
- [4] Panaras, A.G., 1992, "The Effect of the Structure of Swept-Shock-Wave/Turbulent-Boundary-Layer Interactions on Turbulence Modelling," *Journal of Fluid Mechanics*, **236**, pp. 607–33.

- [5] Panaras, A.G., 2004, “Calculation of Flows Characterized by Extensive Crossflow Separation,” *AIAA Journal*, **42**(12), pp. 2474–2475.
- [6] Thivet, F., Jan. 2002, “Lessons Learned from RANS Simulations of Shock Wave/Boundary Layer Interactions,” *AIAA Paper*, **2002-0583**.
- [7] Panaras, A.G., 1996, “Review of the Physics of Swept-Shock/Boundary Layer Interactions,” *Progress in Aerospace Sciences*, **32**, pp. 173–244.
- [8] Knight, D., Yan, H., Panaras, A.G., Zheltovodov, A., 2003, “Advances in CFD prediction of Shock Wave Turbulent Boundary Layer Interaction,” *Progress in Aerospace Sciences*, **39**, pp. 121–184.
- [9] Spalart, P.R., Allmaras, S.R., Reno, NV, 1992, “A One-Equation Turbulence Model for Aerodynamic Flows,” *AIAA Paper*, (92-0439).
- [10] Schulein, E., 2006, “Optical Skin Friction Measurements in Short-duration Facilities,” *AIAA Journal*, **44**(8), pp. 1732–1742.
- [11] Wilcox, D.C., 2006, *Turbulence Modeling for CFD*, 3rd ed., DCW Industries, Inc.
- [12] Sinha, K., Candler, G.V., June 1998, “Convergence Improvement of Two-Equation Turbulence Model Calculations,” *AIAA Paper*, **98-2649**.
- [13] 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.
- [14] Wright, M., Candler, G., and Prampolini, M., 1996, “Data-Parallel Lower-Upper Relaxation Method for the Navier-Stokes Equations,” *AIAA Journal*, **34**(7), pp. 1371–1377.
- [15] Sinha, K., Mahesh, K., Candler, G.V., 2005, “Modeling the Effect of Shock-Unsteadiness in Shock/Turbulent Boundary-Layer Interactions,” *AIAA Journal*, **43**(3), pp. 586–594.
- [16] Edney, B., 1968, “Anomalous Heat Transfer and Pressure Distributions on Blunt Bodies at Hypersonic Speeds in the Presence of an Impinging Shock,” Report 115, Aeronautical Research Institute of Sweden, Stockholm.