

Shock/turbulence interaction: turbulence modeling and scramjet application

Krishnendu Sinha*

Vighnesh Pawar†

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

Abstract

Shock/turbulent boundary layer interaction is a focus of active research because of its relevance to practical applications like scramjet inlets. Flow separation due to shock waves in the inlet duct can significantly deteriorate the performance. Computational fluid dynamic prediction of shock/turbulent boundary layer interactions are often limited by the accuracy of the turbulence models. Traditional models like $k - \epsilon$, $k - \omega$ and Spalart-Allmaras cannot predict the correct level of turbulent kinetic energy k downstream of a shock wave. In-depth study of the equations governing k -amplification at a shock reveals new physical mechanisms caused by the coupling of shock motion with the turbulent velocity fluctuations. Incorporating this additional term in the turbulence models results in significant improvement in their predictions. The new models are validated against experimental data available for canonical shock/turbulent boundary layer interactions. In-house CFD codes with advanced turbulence models are then used to investigate realistic scramjet inlet flowfields. Geometric details and three-dimensionality of a real-life configuration result in highly complex shock/turbulent boundary layer interactions. To understand such flow fields and predict them accurately is the focus of on-going research.

Nomenclature

ρ	Density.
τ_{ij}	Reynolds stress tensor.
μ_T	Turbulent eddy viscosity.
S_{ij}	Mean strain rate tensor.
k	Turbulent kinetic energy.
ϵ	Turbulent dissipation rate.
P_k	Production of turbulent kinetic energy.

1 Introduction

Interaction of shock waves generated by different parts of a vehicle with the turbulent boundary layer on the walls is common in many high-speed applications. These interactions result in a complex flow pattern involving local flow separation and reattachment, free shear layers, and expansion fans. The turbulent fluctuations in a flow can be drastically amplified on passing through a shock. The level of turbulent fluctuations downstream of the shock has a dominant effect on the mean flow field. Traditional turbulence models used in Reynolds-averaged Navier Stokes method are often inaccurate in shock-turbulence interaction, limiting the potential of RANS predictions in high-speed flow applications.

Shock/turbulence interaction is a complex process involving several different physical mechanisms that affect the amplification of turbulent kinetic energy across the shock. An in-depth study of the flow physics is therefore necessary to understand the limitations of existing turbulence models and make improvements to

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

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

overcome these limitations. The interaction of homogeneous isotropic turbulence with a normal shock is an ideal model problem to study the flow physics. The amenability of the problem to theoretical analysis and availability of direct numerical simulation data allows a detailed study of the underlying physics. The models thus developed are validated against experimental data in shock/turbulent boundary layer interactions in canonical configurations.

Shock/turbulent boundary layer interactions are common in scramjet inlets. The resulting flowfield has significant effect on the design and performance of the inlet. The complexity of the shock-turbulence interaction is greatly enhanced by the geometric details of a practical configuration. In addition, the three-dimensionality of the geometry makes it difficult to analyze the flowfield in detail. Understanding the flow pattern, involving three-dimensional flow separation, multiple shock waves and their interactions is a challenging task, and is a topic of active research.

2 Simulation methodology

We solve the Reynolds Averaged Navier-Stokes equations for the mean flow, as presented in Ref. [1]. The $k - \epsilon$, $k - \omega$ and Spalart-Allmaras (SA) models are used for calculating the eddy viscosity. In-house code of Sinha *et al.* [2] 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 [3]. The method is second order accurate both in stream-wise and wall normal directions. The implicit method of Wright *et al.* [4] is used to integrate in time and to reach steady-state solution. The code is validated in several high-speed flow application. Ref. [5]

3 Turbulence Modelling

Majority of turbulence models used in practice are based on the Boussinesq approximation, which models Reynolds stresses in terms of eddy viscosity μ_T .

$$\bar{\rho}\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\mu_T S_{kk}\delta_{ij} - \frac{2}{3}\rho k\delta_{ij} \quad (1)$$

μ_T is computed from turbulent quantities like the turbulent kinetic energy k and the turbulent dissipation rate ϵ . The transport equation for k in the $k - \epsilon$ turbulence model has source terms corresponding to production and dissipation mechanisms. The production term given by

$$P_k = \mu_T \left(2S_{ij}S_{ji} - \frac{2}{3}S_{ii}^2 \right) - \frac{2}{3}\rho k S_{ii} \quad (2)$$

is active in a shock wave and assumes very high values. For example, in case of a normal shock,

$$P_k = \mu_T \left(\frac{\partial u}{\partial x} \right)^2 \quad (3)$$

This results in unrealistically high values of k downstream of the shock (see Fig. 1). The amplification of k increases rapidly as the grid is refined to get a thinner shock, leading to problems in grid convergence. It is argued in Ref. [6] that this unphysical effect is caused by the fact that the eddy viscosity assumption breaks down in highly non-equilibrium flows like shock/turbulence interaction. Suppressing the eddy viscosity, for example, in a realizable $k - \epsilon$ turbulence model yields lower values of k -amplification (see Fig. 1).

The fundamental interaction of homogeneous isotropic turbulence with a normal shock is studied in detail in Ref. [6]. It is found that eddy viscosity corrections of the form used in realizable models even in the limiting case of $\mu_T = 0$, are not enough to match DNS data. A detailed study of the transport equation for k at the shock revealed a new physical mechanism due to unsteady shock motion that could explain the discrepancy between model and DNS. The turbulent fluctuations cause local distortion in an otherwise steady shock wave. The unsteady shock motion gets coupled to the incoming velocity fluctuations, resulting in a negative source term in the k -equation. This damping mechanism was modeled using theoretical analysis. The production term in the new model has the form

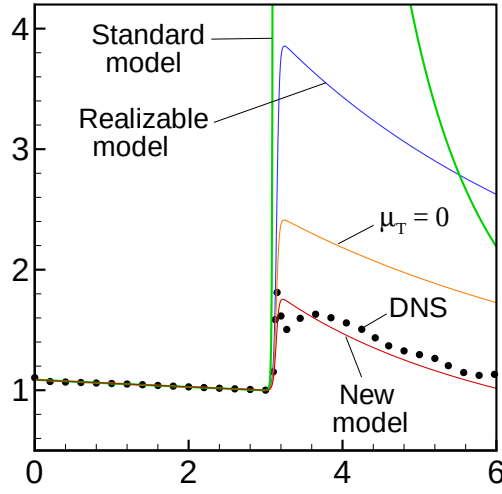


Figure 1: Evolution of k in the interaction of homogenous isotropic turbulence with a normal shock (located at $M = 3.0$). Different variations of the $k - \epsilon$ model are compared with DNS data. The standard $k - \epsilon$ model predicts very high amplification of turbulent kinetic energy. The amplification is reduced by using realizable $k - \epsilon$ model and further by using $\mu_T = 0$. The new model including shock-unsteadiness effect reproduces the correct level of turbulent kinetic energy downstream of the shock. Figure reproduced from Ref. [6]

$$P_k = -\frac{2}{3}\bar{\rho}kS_{ii}(1 - b'_1) \quad (4)$$

where $b'_1 > 0$ is the shock-unsteadiness modeling parameter and is a function of the upstream Mach number normal to the shock. The new model was found to match DNS data well (see Fig. 1). Further details are given in Ref. [6] and [7].

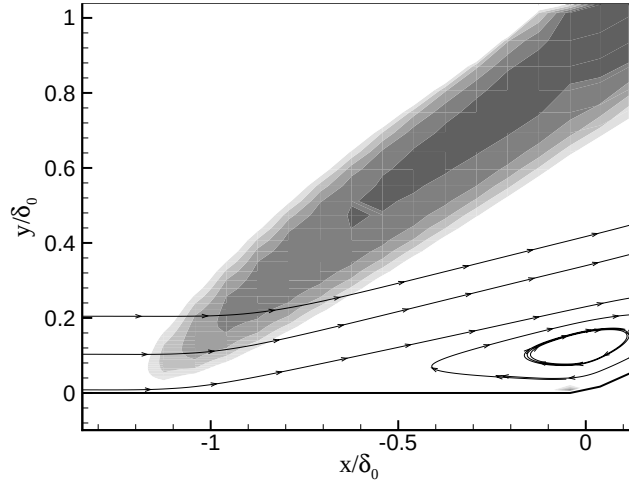


Figure 2: Magnified view of compression corner flow at Mach 2.84 and 24 deg ramp. The separation bubble is identified in terms of reversed flow streamlines and the separation shock corresponds to the grey region where S_{ii} is negative. Figure reproduced from Ref. [8]

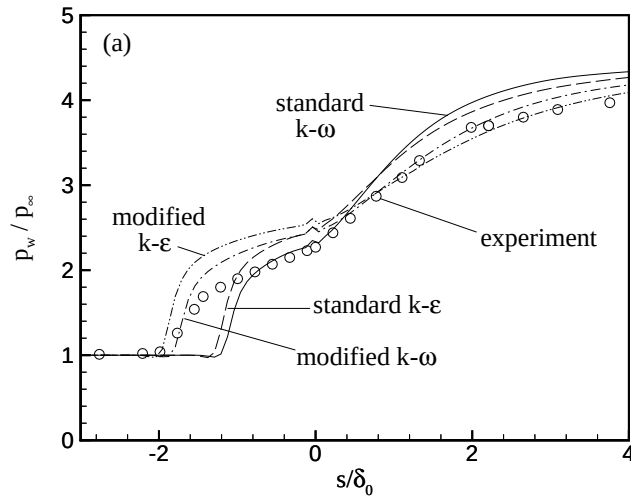


Figure 3: Variation of normalised surface pressure along a 24-deg compression corner obtained using different $k - \epsilon$ and $k - \omega$ models. The models including shock-unsteadiness correction move the separation point upstream as compared to the standard models. The location of the corresponding pressure rise (at $\delta_0 = -2$) matches better with experimental data. Figure reproduced from Ref. [8]

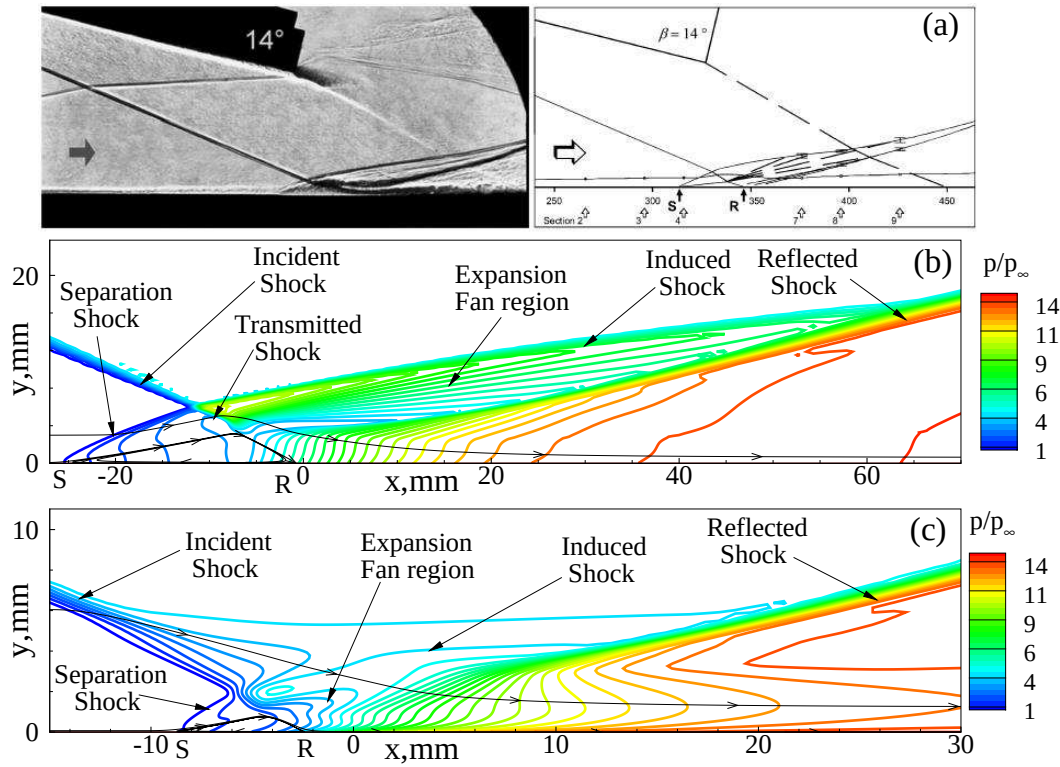


Figure 4: Oblique shock wave generated by a 14 deg wedge at Mach 5 impinging on a turbulent boundary layer. (a) Experimental shadowgraph and schematic compared with (b) computed pressure contours using shock unsteadiness modified $k - \omega$ model and (c) standard $k - \omega$ model. The shock unsteadiness modification increases the size of separation bubble greatly, such that the shape and location of different shock waves match experimental results better than that of the standard model. Figure reproduced from Ref. [9]

The shock unsteadiness correction is incorporated in standard $k - \epsilon$, $k - \omega$ and Spalart Allmaras models, and applied to canonical flows like compression corner in Ref. [8]. The test cases are selected such that detailed experimental data is available for turbulence model validation. The shock unsteadiness correction is implemented in such a way that it is effective only in high compression regions of the flow, and the original model is recovered outside of shock waves. A negative value of S_{ii} is used to identify the shock waves in the flow (see Fig. 2). The modification results in an improved prediction of the separation bubble size and the resulting surface pressure distribution (see Fig. 3). The location and structure of shock waves are also greatly improved by shock-unsteadiness correction, as observed in the oblique shock impinging on a turbulent boundary layer (Fig. 4) .

4 Scramjet Applications

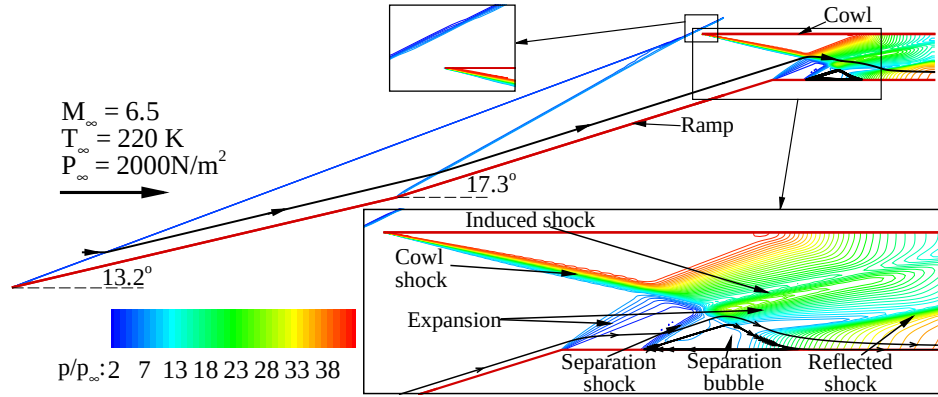


Figure 5: Shock structure in a simulated two-dimensional scramjet inlet geometry. The oblique shocks generated by the forebody ramps intersect slightly away from the cowl lip (inset). The cowl shock reflects from the opposite wall causing local flow separation and reattachment (inset).

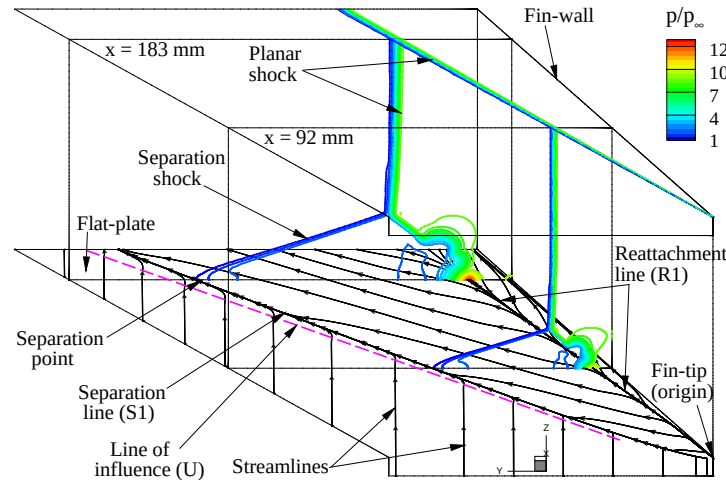


Figure 6: Shock structure in a simulated three-dimensional shock/turbulent boundary layer interaction. The oblique/planar shock generated by the fin splits into a lambda structure (shown by pressure contours in $x=92$ and 183 sections). Surface streamlines are plotted on the adjacent wall to show the location of separation and reattachment lines, and the resulting helical flow pattern. Figure reproduced from Ref. [10].

The flowfield generated on the forebody and intake duct of a scramjet engine (see Fig. 5) are characterized by shock/turbulent interactions of the kind discussed above. For example, reflection of the cowl shock on

the opposite wall often results in flow separation that can be detrimental to the performance of the inlet. Although the basic nature of the interaction is identical to that presented in Fig. 4, geometric complexity make the overall flowfield more complex. Presence of an expansion corner upstream of the cowl shock impingement location makes the shock wave stronger and alters the boundary layer on the bottom wall. Experimental data for such configurations are often unavailable, and extensive CFD validation of the kind presented in Refs. [8] and [9] are helpful in predicting these flowfields reliably.

Three-dimensionality of a real inlet geometry results in additional complexity in the flow pattern. The cowl shock interacts with the boundary layer on the side wall. This configuration is studied in literature as the single fin geometry, where the fin acts as a shock generator and the boundary layer on the adjacent wall separates to form a conical region. The footprint of this helical flow in terms of the surface streamlines is shown in Fig. 6, with the separation and reattachment lines identified. The inviscid shock bifurcates into a lambda structure near the side wall. The triple point formed at the junction of shock waves generates a jet like flow. Impingement of this supersonic jet on the wall leads to localized high pressure and heat transfer on the surface.

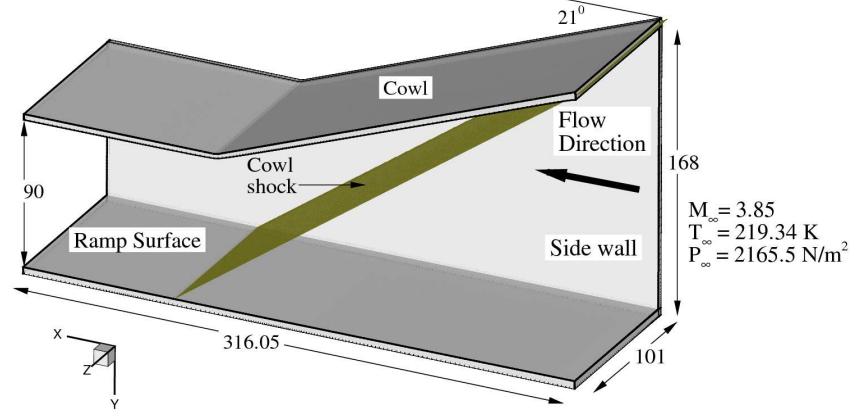


Figure 7: Simulated three-dimensional geometry of scramjet inlet duct to study the interaction of cowl shock with the side wall and the ramp surface. The cowl is modeled as a 21 deg shock generator and the ramp surface is taken to be flat to avoid additional complexity due to the presence of an expansion corner.

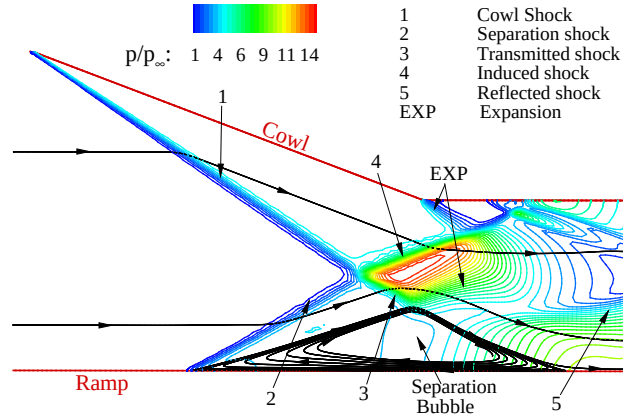


Figure 8: Pressure contours drawn in a streamwise plane away from the side-wall show a flow pattern similar to that in Fig. 5. Impingement of the cowl shock on the ramp surface results in a large separation bubble, and several other shock waves and expansion fans.

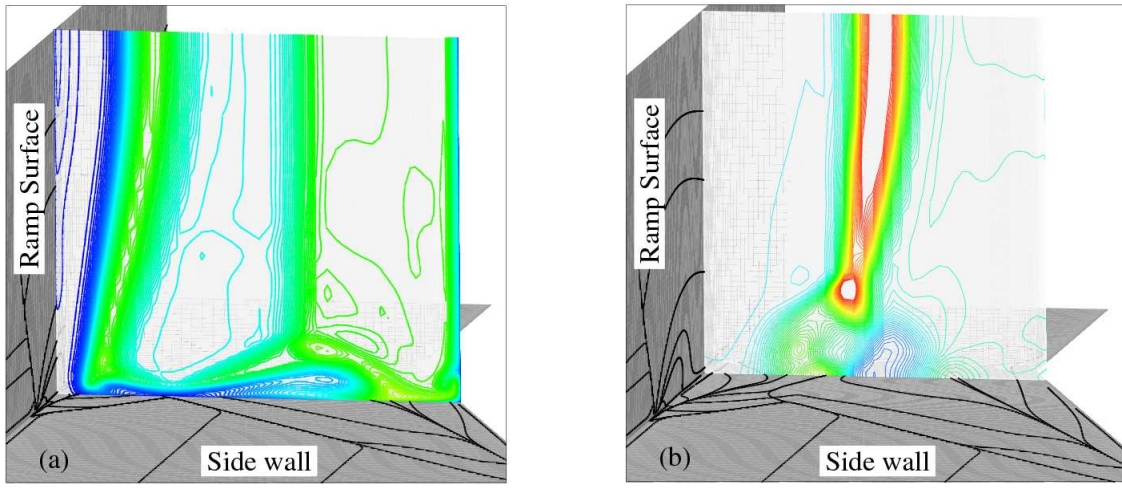


Figure 9: Shock structure in the vicinity of the side wall as shown by pitot pressure contours plotted in two transverse planes. The surface streamlines on the sidewall are also shown for reference.

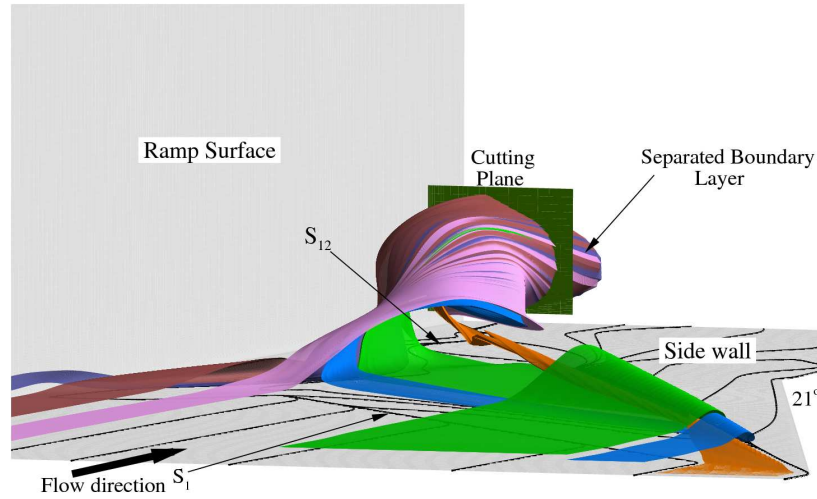


Figure 10: Three-dimensional separation of the sidewall boundary layer due to interaction with the cowl shock, as shown in terms of different stream surfaces originating in the incoming boundary layer.

The actual flow in an inlet duct is a combination of the single fin and oblique shock impingement interactions discussed above. The geometry shown in Fig. 7 is used to study the flowfield. The shock pattern away from the side wall (Fig. 8) is similar to the 2D result presented in Fig. 5. However, the shock waves get significantly distorted as we approach the side wall, with the shock-shock interaction typical of single fin flow clearly visible in Fig. 9(a). Merging of different shock waves can be seen in Fig. 9(b) plotted further downstream. The surface stream lines plotted on the side wall show the conical single fin pattern, which gets altered by the flow separation on the opposite wall. Boundary layer flow separating from the side wall rolls up into a streamwise vortex (see Fig. 10) that covers an appreciable portion of the inlet duct. The resulting flow distortion can have significant effect on the flow quality entering the combustor.

5 Summary

Flow prediction in scramjet inlets pose significant challenges for CFD because of the geometric complexity and shock/turbulent boundary layer interactions. Systematic study of a fundamental shock/turbulence

interaction has led to the identification of new physical mechanisms that play an important role in turbulence amplification through a shock wave. Models for these physical processes are developed using theoretical analysis. The new models are implemented as simple modifications in existing CFD codes, and validated against experimental data available for shock/boundary layer interaction in simplified geometries. These in-house tools are used in understanding and predicting the complex three-dimensional flowfields in realistic scramjet inlets.

References

- [1] Wilcox, D.C., 2006, *Turbulence Modeling for CFD*, 3rd ed., DCW Industries, Inc.
- [2] Sinha, K., Candler, G.V., 1998, "Convergence Improvement of Two-Equation Turbulence Model Calculations," AIAA Paper, **98-2649**.
- [3] 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.
- [4] 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.
- [5] Reddy, D.S.K., Sinha, K., 2009, "Hypersonic Turbulent Flow Simulation of FIRE II Reentry Vehicle Afterbody," *Journal of Spacecraft and Rockets*, **46**(4).
- [6] Sinha, K., Mahesh, K., Candler, G.V., 2003, "Modeling Shock-Unsteadiness in Shock/Turbulence Interaction," *Physics of fluids*, **15**(8), pp. 2290–2297.
- [7] Veera, V.K., Sinha, K., 2009, "Modelling the effect of upstream temperature fluctuations on shock/homogeneous turbulence interaction," *Physics of fluids*, **21**(2).
- [8] 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.
- [9] Pasha, A.A., Sinha, K., 2008, "Shock-unsteadiness model applied to oblique shock wave/turbulent boundary-layer interaction," *International Journal of Computational Fluid Dynamics*, **22**(8), pp. 569–582.
- [10] Pasha, A.A., Sinha, K., 2009, "Simulation of a three-dimensional shock/boundary layer interaction in a single-fin configuration," 11th Annual CFD Symposium, Bangalore, India.