HiSST: International Conference on
High-Speed Vehicle Science Technology
26–29 November 2018, Moscow, Russia

Computation of Hypersonic Flows on Compression Corner using modified k-ω model

Amjad A. Pasha[1], Khalid A. Juhany[2]

Abstract

A hypersonic vehicle during its complete flight regime encounters a wide range of conditions. These flight conditions may vary from low to high Mach numbers at variety of angles of attack. The surface wall temperatures associated with high Reynolds flows lead to heat transfer and wall shear stress issues. The resulting flow separation, often associated with shock wave/boundary layer interactions, generally leads to increased energy losses in the system and degrades the performance of such aerodynamic control surfaces such as the fins installed on the vehicle. Reynolds-averaged Navier-Stokes equations using standard turbulence models result in incorrect separation bubble size for large separated flows. This results in inaccurate aerodynamic loads such as the wall pressure, local skin friction distribution and heat transfer rate. In the former studies, the shock-unsteadiness correction was applied to the standard two-equation k-ω model which improved the separation bubble size leading to more accurate pressure predictions and the shock definition. In this work, a shock unsteadiness modification to k-ω model is applied to the hypersonic flows, based on a parameter which is dependent upon the local strength of the shock wave subjected to upstream turbulent boundary layer fluctuations. Compression corner flows with different deflection angles θ ranging from 15^o to 38^o at Mach 9.22 are simulated first. This is followed by further simulations where free stream Mach numbers are varied from M_∞ = 5 to 9. A separate study is conducted to assess the effect of cool and hot temperatures T_w under isothermal surface conditions. Reynolds number effects will also be investigated where the boundary layer based Re_δ is varied from $1x10^5$ to $4x10^5$.The ramifications of θ, M_∞, T_w and Re_δ upon surface pressure, skin friction and heat transfer rates, particularly in the interaction region, forms the basic theme of this research.

Keywords: high-speed flows, shock wave, turbulent boundary-layer, separation bubble, turbulence modeling

1. Introduction

The shock-wave turbulent boundary-layer interactions (SWTBLI) occur commonly in hypersonic cruising vehicles. The movement of fins or control surfaces to maneuver launch vehicles and missiles could result in SWTBLI at high Mach flows. The adverse pressure gradient across the shock wave causes boundary-layer to separate. The flow reattaches in the downstream and gives rise to local peak wall pressure, skin friction and heat transfer rates, which can be far in excess as compared to the attached boundary layer [1]. A hypersonic vehicle during its flight covers a wide range of conditions. The vehicle can fly at different angles of attack from low to high Mach numbers at varying Reynolds numbers and wall temperatures. Therefore, the effect of parameters like deflection angle, Mach number and wall temperature on flow separation is beneficial for improving the design of the vehicle under consideration [2-5].

The experimental results coupled with numerical validations have shed considerable light on the flow dynamics of SWTBLI flows [4]. The direct numerical simulations (DNS) and large eddy simulations (LES) approaches are more accurate but have less fidelity in computing high Reynolds number SWTBLI flows. In contrary to these approaches, Reynolds-averaged Navier-Stokes (RANS)

[1] *Assistant Professor, Department of Aeronautical Engineering, King Abdul Aziz University,Jeddah- 21589, Saudi Arabia, aapasha@kau.edu.sa*
[2] *Chairman, Department of Aeronautical Engineering, King Abdul Aziz University,Jeddah- 21589, Saudi Arabia, khalid@kau.edu.sa*

approach, though, lag accuracy to a small extent but have high fidelity in computing these flows. Numerical simulations were performed for supersonic and hypersonic compression corner SWTBLI flows using standard Spalart-Allamras, k-ω and k-ε turbulence models [3-11]. It is observed that the standard turbulence models predict initial pressure rise location far downstream and local wall peak pressure upstream, hence result in a smaller separation bubble as compared to the experiments. These models produce wall heat flux in a similar way corresponding to the variation of wall pressure. This is because the standard one- and two equation turbulence models were designed incompressible flows and therefore exclude the average effect of shock-unsteadiness caused by upstream turbulent boundary-layer fluctuations [6]. Therefore, these models produce unrealistic high values of turbulent kinetic energy across the shock wave [6]. The shock-unsteadiness in SWTBLI flows is caused by the upstream turbulent eddies interacting with a shock-wave and its average effect on the mean flow was first modeled by Sinha et al. [6]. Later, the shock-unsteadiness modification of Sinha et al. [7] was implemented to one- and two-equation turbulence models like Spalart-Allmaras, k-ε and k-ω to supersonic and hypersonic SWTBLI flows [7-11]. The shock-unsteadiness correction to standard turbulence models improved the prediction of surface properties, the separation bubble size and wall heat transfer rate and matched the experimental data better than the original models. Therefore, a reasonable accuracy can be obtained with an appropriate selection of the turbulence models to compute high-speed flows, which are difficult to simulate as compared to incompressible flows.

In this article, we apply shock-unsteadiness modified k-ω model [7,11] to compute experimental test cases of Elfstrom [12] and Coleman et al. [13] for two-dimensional compression corner flows with different: deflection angles, wall temperatures, Mach numbers and Reynolds number. These configurations possess high Mach number and high deflection angles and result in high compression regions and make the numerical simulations stiff, thus, making a challenging task to simulate in contrary to the supersonic SWTBLI flows. In earlier computations [7-10], the shock-unsteadiness modification was applied based on the average values of the shock-unsteadiness parameter which in turn was calculated based on the Mach number normal to the shock wave. Also, the turbulent Prandtl number was assumed to be constant. In a recent work, the modification is implemented based on the local values of the strength of the shock wave with variable Prandtl number across the shock. This new version of shock-unsteadiness modified k-ω model [11] improved the heat transfer rate as compared to the former modified k-ω model [7]. Our focus of work is to simulate numerically hypersonic SWTBLI compression corner flows by implementing the new version of shock unsteadiness modified k-ω model [11] which accounts the variation in turbulent Prandtl number to predict the flow-field, wall pressure and heat transfer rate.

2. Analysis

2.1 Experimental data

The experiments were conducted in the gun tunnel with nitrogen gas as the working fluid to investigate hypersonic SWTBLI cases of two-dimensional compression corner flows. The model comprised of a leading sharp flat plate of length = 43 cm and a trailing ramp of length = 25.4 cm, instrumented for pressure and heat transfer measurements with negligible end effects. These experimental data were reported in terms of non-dimensional pressure p_w/p_∞, and wall heat flux q_w/q_∞, for 15^o attached case, 30^o incipient separation case and fully separated 34^o and 38^o cases. Different Mach numbers ranging from 7 to 9.22 were tested with different plate lengths to generate a unit Reynolds number of $Re_{1\infty}$, = 4.5 x 10^7 m^{-1}. A free stream temperature of T_∞ = 64.5 K corresponds to a reservoir temperature of 1070 K, M_∞ = 9.22 and specific heat ratio of 1.367 was assumed.

2.2 Shock-unsteadiness k-ω model

Sinha et al. [6] studied the homogeneous turbulence interacting with a normal shock wave and observed that the unsteady shock motion dampens the amplification of turbulent kinetic energy, k across a shock. Based on the linear analysis theory, they propose the shock-unsteadiness by modifying the production of turbulent kinetic energy term in the standard k-ω turbulence model [7]. This effect is implemented by multiplying the eddy viscosity, $\mu_T = \bar{\rho}\, k/\omega$ by the factor,

$$c_\mu' = 1 - f_s[1 + b_1'/(\sqrt{3}\xi)] \quad (1)$$

$$b'_1 = \max[0, 0.4(1 - e^{1-M1n})] \quad (2)$$

Here, μ_T is eddy viscosity, $\bar{\rho}$ is mean density, ω is dissipation rate. The parameter b'_1 represents the damping effect caused by the coupling between the unsteady shock motion and the upstream velocity fluctuations, $\xi = S/\omega$ is dimensionless mean strain rate and the parameter $S = [2\, S_{ij}\, S_{ij} - 2/3\, S_{ii}^2]^{1/2}$. The empirical function $f_s = 0.5 - 0.5\, \tanh[5(\delta_0 S_{ii}/U_\infty) + 3]$ locates the region of shock-wave in terms of the ratio $\delta_0 S_{ii}/U_\infty$. Here, δ_0 is boundary-layer thickness upstream of SWTBLI and U_∞ is free stream velocity. In the recent work, more robust shock-unsteadiness is implemented by evaluating b'_1 based on the local value of the density ratio r, instead of the shock-normal Mach number M_{1n} dependence given by Eq. 2 in earlier studies. The b'_1 is curve fitted and is given by [11].

$$b'_1 = max\{0,\, 0.4\, [(r - 1)/5]^{0.3}\} \tag{3}$$

Also, a variable turbulent Prandtl Pr_T model is developed for canonical normal shock/homogeneous turbulence interactions [11].

$$Pr_T = 0.75\, \zeta[1 + b'_1(r - 1)] \tag{4}$$

Here, a parameter ζ is introduced to make the formulation consistent with the conventionally accepted Pr_T value of 0.89 in the boundary layers with no shock waves.

$$\zeta \;\; = [1 + (0.89/0.75 - 1)\, e^{\chi(1-r)}] \tag{5}$$

In the current numerical computations, we will be evaluating b'_1 given by Eq.3 and will implement the variation of turbulent Prandtl number given by Eq. 4.

2.3 Preliminary numerical results

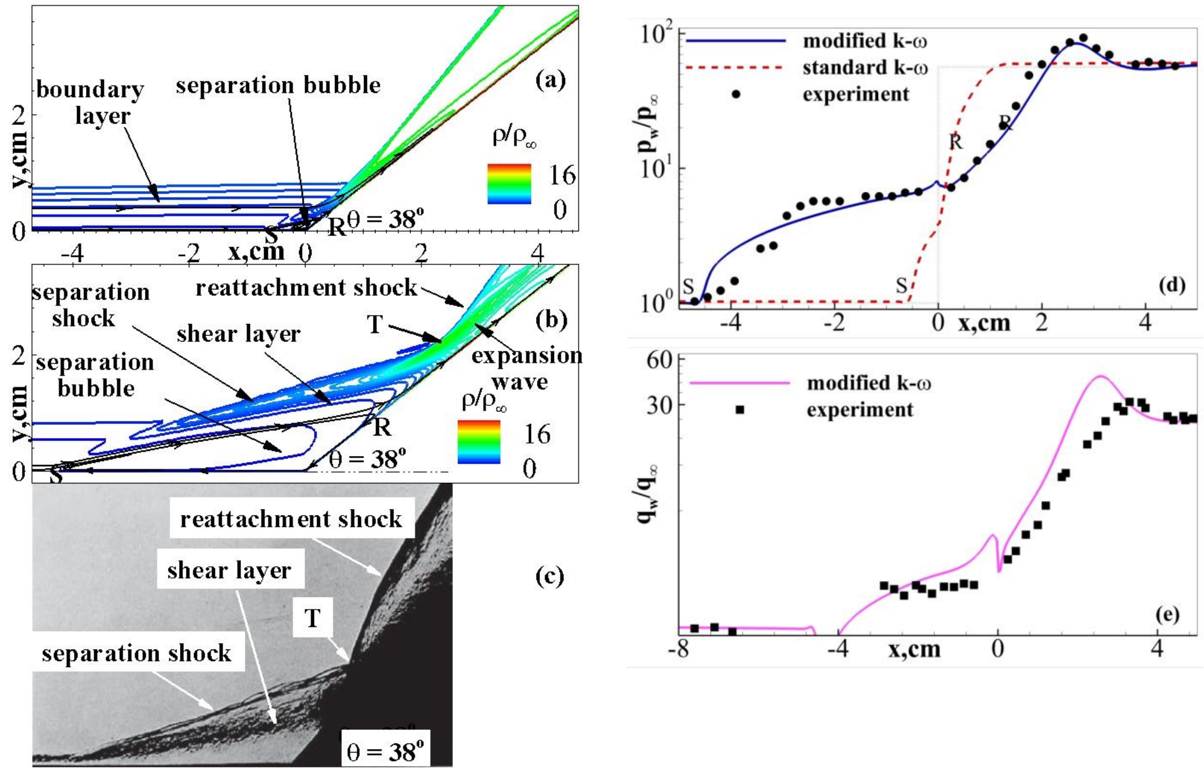

Fig 1. Computed density contours obtained using (a) standard k-ω [15] and (b) shock-unsteadiness modified k-ω model [11] compared with (c) experimental shadowgraph [1,12]; Comparison of computed (d) surface pressure and (e) heat transfer rate with experimental data [12] for 38° compression flow at Mach 9.22, $Re_\delta = 4 \times 10^5$ and $T_w/T_r = 0.3$.

The two-dimensional RANS equations are solved for the mean flow, based on the finite volume technique [14]. The two-equation turbulence models: standard k-ω of Wilcox [15] and shock-unsteadiness modified k-ω model [7] are used in the computations. Based on the grid convergence study, a grid size of 500 x 400 is used with the first grid at a distance of 1×10^{-6} m from the wall. A wall unit of $y^2+ < 1.8$ is obtained in the whole domain for this grid. CFL numbers up to 40 are used in our computations. Figure 1 shows a comparison of the computed density contours with the experiments. The high Mach number = 9.22 and high ramp angle = 38° makes it a difficult test case for simulations and to predict it accurately, due to its involvement of a very strong compression at corner. The higher shock strength results in the large size of the SWTBLI region. The computed results using standard k-ω model in Fig. 1a shows that the upstream turbulent boundary-layer on the plate interacts with the corner shock to predict a small flow separation in comparison to the experimental shadowgraph depicted in Fig. 1c. The standard k-ω over-amplifies the turbulent kinetic energy across shock wave, hence pushes the separation point location (S) far downstream as compared to the experiments. This effect is shown in wall pressure plot in Fig. 1d. A smaller separation size of 1.2 cm, results in wrong shock structure as compared to the experiments. The shock-shock interaction region is predicted upstream and causes the peak value to under-predict in the reattachment region (see Fig. 1d). In contrary to it, the computations using shock-unsteadiness modified k-ω model in Fig. 1b resembles closely to experiments in terms of the separation shock length, shear-layer, separation bubble size, and triple point (T) location. The shock-unsteadiness correction dampens the turbulent kinetic energy across the shock and causes the flow to separate earlier, to match the experimental initial pressure location in Fig. 1d. It results in a separation bubble size of 6 cm. The pressure plateau is also well predicted by the modified k-ω model. The separation shock and reattachment shock interaction forms an expansion fan at the triple point (T) as shown in Fig. 1b. This shock-shock interaction at (T) causes peak pressure rise at x = 3 cm and is accurately predicted close to the experiments in Fig. 1d. The drop in pressure at x = 3 cm is because of expansion fan-generated at (T). The heat transfer computations with constant Prandtl number of 0.89 is presented in Fig. 1e using modified k-ω model. The variation of Prandtl number as given by Eq.4 still have to be worked out carefully and will be presented in the full manuscript. The future work is directed to simulate the hypersonic SWTBLI flows using the shock-unsteadiness modified k-ω model [11] as described above to study the variation of the deflection angles from 15^o to 38^o, Mach numbers from 5 to 9.22, wall temperatures from and Reynolds numbers based on the boundary layer thickness Re_δ from 1 to 4 on the SWTBLI interaction region. The flow fields, surface pressure and heat transfer rate will be compared with the experimental results [12,13].

References

1. Babinsky, H., Harvey, J. K.: Shock Wave-Boundary Layer Interactions. Cambridge University Press (2011)
2. Holden, M. S., Wadhams, T. P.: A database of aerothermal measurements in hypersonic flow in "building block" experiments for CFD validation. AIAA-2003-1137 (2003)
3. Roy, C. J., Blottner, F. G.: Review and assessment of turbulence models for hypersonic flows. Prog. Aero. Sci. 42(7), 469-530, (2006)
4. Marvin, J. G., Brown, J. L., Gnoffo, P. A.: Experimental database with baseline CFD solutions: 2-D and axisymmetric hypersonic shock-wave/turbulent-boundary-layer interactions. NASA Ames Research Center, NASA TM-2013-216604, (2013)
5. Gaitonde, D. V.: Progress in shock wave/boundary-layer interactions. Prog. Aero. Sci. 72, 80-99, (2015)
6. Sinha, K., Mahesh, K., Candler, G. V.: Modeling shock-unsteadiness in shock/turbulence interaction. Phy. of Fluids, 15(8), 2290-2297, (2003)
7. Sinha, K., Mahesh, K., Candler, G. V.: Modeling the effect of shock-unsteadiness in shock turbulent boundary-layer interactions. AIAA Journal, 43(3), 586-594, (2005)
8. Pasha, A. A., Sinha, K.: Shock-unsteadiness model applied to oblique shock wave/turbulent boundary-layer interaction. Int. J. Com. Fluid Dyn., 22(8), 569-582, (2008)
9. Pasha, A. A., Sinha, K.: Shock unsteadiness model applied to hypersonic shock wave turbulent boundary-layer interactions", J. Prop. Power, 28(1), 46-60, (2012)
10. Roy, S., Pathak, U., Sinha, K.: Variable turbulent Prandtl number model for shock/boundary-layer interaction. AIAA J., 56(1), 342-355, (2018)
11. Pasha, A. A., Juhany, K. A., Khalid, M.: Numerical prediction of shock/boundary-layer interactions at high Mach numbers using a modified Spalart–Allmaras model, Engg. App. Comp. Fluid Mech., 12:1, 459-472, (2018)
12. Elfstrom, G. M.: Turbulent hypersonic flow at a wedge-compression corner. J. Fluid Mech. 53(1), 113-127, (1972)
13. Coleman, G. T., Stollery, J. L.: Heat Transfer from hypersonic turbulent flow at a wedge compression corner. J. Fluid Mech., 56(4), 741-752, (1972)
14. Sinha, K., Candler G. V.: Convergence improvement of two-equation turbulence model calculations. 29th AIAA Fluid Dyn. Conf., AIAA-Paper-1998-2649, (1998)
15. Wilcox, D. C.: Formulation of the k-ω turbulence model revisited. AIAA J. 46(11), 2823-2838, (2008)