Numerical simulation of separation flows induced by shockwave using an anisotropic turbulence model

C J Liu 1, X C Wang 1, T F Peng 1 and W H Yan 1,2

1 School of Mechanical and Materials Engineering, North China University of Technology, Beijing, 100144, China
2 Corresponding author: Wenhui Yan, School of Mechanical and Materials Engineering, North China University of Technology, Beijing, 100144, China

Email: abuuaa@163.com

Abstract. This paper studies the performance of a nonlinear anisotropic turbulence model developed in the basis of a statistical partial average scheme. The first order velocity of turbulent fluctuations, retained by a novel average scheme, and turbulent length scale can be determined from the momentum equations and mechanical energy equation of the fluctuation flow, respectively. The two physical quantities are readily to construct the nonlinear anisotropic eddy viscosity tensor and to significantly improve computational results. Two typical shock-wave-boundary-layer-interaction (SWBLI) separation flows are deeply studied using CFD method. One is an incident oblique shock wave impinging on a flat plate with a turbulent boundary layer, and the other one is a transonic turbulent separation flow in a converging-diverging transonic diffuser. Comparisons between the computational results and experimental data are carried out for velocity profiles, density contours, pressure distribution, skin friction coefficient, Reynolds stress as well as streamline vectors distribution. Without using any empirical coefficients and wall functions, the numerical results are in good agreement with the available experimental data, which further confirms that the nonlinear anisotropic eddy viscosity tensor is of the decisive factor for the success of the computational results.

1. Introduction
Shock wave turbulent boundary layer interaction(SWTBLI) has been extensively studied [1-6] since Ferri A. had made the first observation during the test of an airfoil in high speed tunnel in 1939. Along with the development of the computer science, numerical simulation has become more effectively and played an important role in studying supersonic or hypersonic turbulent flows for designing aircraft configurations or controlling flows. It is well known that interaction of incident shock wave with the turbulent boundary layer and converging-diverging transonic diffuser flow are the benchmark tests for evaluating the validation of turbulence models and computational methods in predicting flow separation and recovery phenomena [7]. Lots of numerical simulations have been conducted for this problem. Knight et al. [8] reported that direct numerical simulation (DNS) and large-eddy simulation (LES) provides good results for low-Reynolds-number natural transition [9] and transitions in laminar separation bubbles [10]. An approach of Reynolds-averaged Navier-Stokes (RANS) equation coupled with a turbulence model is more extensively applied in the study of SWTBLI because of its stability.
and adaptation for wide range of Reynolds numbers. The final results of interaction significantly depend on the flow regime in the boundary layer and shock-wave intensity and angle of incident [11]. The objective of present work is to apply a new turbulent model which include none empirical coefficients or wall functions to simulate incident oblique shock wave turbulence boundary layer interaction on a flat plate. For different shock intensities, events with no flow separation, incipient separation region, and large separation bubble are considered. Another simulation flow is a transonic viscous separation flow in a converging-diverging diffuser. Flow relaxation effect in the turbulent separation flow is very important. It was investigated in the transonic viscous flows by many researchers [12-14]. Flow relaxation effect means the changing rate of Reynolds stress lags behind the changing rate of strain significantly. Further explaining in the respect of numerical simulation of mean quantity, the calculated velocity profiles have larger defect than the experimental data. We studied a transonic viscous flow in the Sajben transonic diffuser to evaluate the flow relaxation performance using this turbulence model. This transonic diffuser problem is a representative internal-flow case. We are going to focus on Reynolds stress-strain constitutive relationship and eddy viscosity coefficients to study the flow relaxation in this present work. Comparisons between the computations and experiment are presented in this present paper, including surface pressure distribution, velocity profiles, and the skin friction coefficient distribution. The computed results obtained from this turbulent model agree well with the experimental values.

2. Governing Equations
A statistical partial average scheme [15] is applied in the study of fluctuation variables of turbulent flows. As a result, the first order statistical moment of turbulent fluctuations is retained as a turbulent drift flow, which enables us to rationalize modeling of correlation terms by a momentum transfer chain and the concept of non-isotropic turbulent viscosity. With the aid of Favre-average, we can derive this compressible turbulent model (a complete set of equations of compressible turbulent flow) without employing any empirical coefficients or wall functions. Many benchmarks of turbulent flow have been completed using this turbulent model, which include plane jet, round jet, laminar-turbulent transition, backward facing step flow and coherent structures in turbulent boundary layer so on. These equations, normalized by the free stream density, \( \rho_{\infty} \), temperature, \( T_{\infty} \), and the characteristic length, \( L \), are given in the following without derivation.

Continuity equation of the mean flow:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_j \right) = 0
\]  

(1)

Momentum equation of the mean flow:

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho u_i u_j + p \delta_{ij} \right) = \frac{\partial}{\partial x_j} \left( \sigma_{ij} + \rho \nu_{ij} \right)
\]  

(2)

Energy equation of the mean flow:

\[
\frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_j} \left( (\rho E + p) u_j \right) = \frac{\partial}{\partial x_j} \left[ \mu_j \left( \sigma_{ij} + \rho \nu_{ij} \right) - (q_j + q_{ij}^{\text{turb}}) \right]
\]  

(3)

Continuity equation of the drift flow:

\[
\frac{\partial}{\partial x_j} (\rho u_j) = 0
\]  

(4)

Momentum equation of the drift flow:
\[ \frac{\partial (\rho \hat{u}_j)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \hat{u}_i u_j + \rho \hat{u}_j u_i + \hat{p} \delta_{ij}) = \frac{\partial}{\partial x_j} (\sigma_{ij} + \sigma_{ij}^{\text{turb}}) - \frac{\partial}{\partial x_j} (\sigma_{ij} + \sigma_{ij}^{\text{turb}}) \]  \hspace{1cm} (5)

Mechanical energy equation (for the computation of turbulence characteristic length scale):
\[
\sum_{n=1}^{\infty} \left( \begin{array}{c} \ell_j \\ \frac{\partial}{\partial x_j} \end{array} \right)^n \rho u_i \hat{u}_j = \ell_i \left( \begin{array}{c} -\frac{\partial}{\partial x_j} \left( \rho \hat{u}_j u_i + \hat{p} \delta_{ij} \right) + \frac{\partial}{\partial x_j} \left( \sigma_{ij} + \sigma_{ij}^{\text{turb}} \right) \\ -\frac{\partial}{\partial x_j} \left( \sigma_{ij} + \sigma_{ij}^{\text{turb}} \right) - \alpha \frac{D(\rho u_i)}{Dt} \end{array} \right) \]  \hspace{1cm} (6)

 Constitutive Equations:
\[ \sigma_{ij} = 2 \mu e_{ij} - \frac{2}{3} \mu e_{ik} \delta_{ij} \]  \hspace{1cm} (7)
\[ \dot{\sigma}_{ij} = 2 \dot{\mu} e_{ij} - \frac{2}{3} \dot{\mu} e_{ik} \delta_{ij} \]  \hspace{1cm} (8)
\[ \sigma_{ij}^{\text{turb}} = 2 \mu_{ij} e_{ij} - \frac{2}{3} \mu_{KK} e_{ik} \delta_{ij} \]  \hspace{1cm} (9)
\[ \dot{\sigma}_{ij}^{\text{turb}} = 2 \mu_{ij} e_{ij} - \frac{2}{3} \mu_{KK} \dot{e}_{ik} \delta_{ij} \]  \hspace{1cm} (10)

Where, \( I, J, \) and \( K, \) taking the same values as \( i, j, \) and \( k, \) do not imply tensor summation, and the dynamic viscosity coefficient \( \mu \) can be calculated by Sutherland’s law in the form
\[ \mu = \mu_0 \left( \frac{T}{T_0} \right)^{1.5} \frac{T_d + T_s}{T + T_s} \]  \hspace{1cm} (11)

Turbulent viscosity coefficient \( \mu_{ij} \) is
\[ \mu_{ij} = |\xi_{ij}| * |\xi_{ij}| \]  \hspace{1cm} (12)
where,
\[ \xi_{ij} = \rho \dot{e}_{ij} \hat{u}_i \]  \hspace{1cm} (13)

The heat flux terms:
\[ q_j = \kappa \frac{\partial T}{\partial x_j} = \frac{c_p \mu}{Pr} \]  \hspace{1cm} (14)
\[ q_{ij}^{\text{turb}} = \kappa_{ij}^{\text{turb}} \frac{\partial T}{\partial x_j} = \frac{c_p H_{ij}^{\text{turb}}}{Pr_{ij}^{\text{turb}}} \]  \hspace{1cm} (15)

The above equations are closed by the equation of state of a perfect gas:
\[ p = \frac{1}{\gamma} \rho T \]  \hspace{1cm} (16)
3. Numerical Simulations
In our numerical computation, we discretize the governing equations using the cell central finite volume method. The computational algorithm is the explicit, upwind-difference with the inviscid fluxes obtained on the surface of the control volume by Roe’s flux-difference-splitting technique, and diffusion terms evaluated by the second-order central-difference, respectively. We determine the steady state solution of the differential equations for each computational cell by using five-stage Runge-Kutta algorithm, in which the Runge-Kutta coefficients are set equal to $1/4$, $1/6$, $3/8$, $1/2$, and $1/1$, respectively. ‘M.U.S.C.L.’ (Monotonic Upstream Scheme for Conversation Law [16,17]) extrapolation technique and Koren flux limiter [18] are employed to maintain accuracy, monotonicity and robustness in cases of shockwave discontinuities. Acceleration techniques, such as residual averaging and local time stepping, are adopted to speed up the convergence of the numerical solution.

3.1. An incident oblique shock wave impinging on a flat plate
In the present computation, we choose the rectangle of $0 \leq x \leq 0.32$ and $0 \leq y \leq 0.1215$ as the computational domain bounded by the upper, inlet, outlet and wall of flat plate. The velocity, density, and temperature profiles in the inlet section are determined by boundary layer computations under the condition of matching experimental and numerical values of integral boundary layer characteristics and skin friction. The boundary conditions on the flat plate are isothermal, no-slip and zero pressure gradients. Neumann type boundary conditions are employed on the upper and outlet boundaries. A rectangular regular grid which has 80 nodes along the plate and 50 nodes in the direction normal to the flat plate refined toward the plate surface and shock wave location is used in the computational cases.

Computational results of the flat plate shock wave turbulence boundary layer interaction obtained by using this compressible turbulent model are presented for different impinging shock angles and Mach numbers at a specified Reynolds number. The first computation case is conducted at a Reynolds number of $2.84 \times 10^5$, an impinging angle of 31.2471 degrees and the free stream Mach number of 2.0. Corresponding experiment was conducted by MacCormak R. W. and Baldwin B. S. [19], Xu Wanwu, Wang Zhenghua and al. [20] computed this case using WENO scheme and Runge-Kutta algorithm. Because of the weak shock intensity, there is no boundary layer separation in allover the flow fields. A comparison of pressure distribution and skin friction between the computed results and experimental data are presented in the figure 1 and figure 2, respectively. All positive skin friction shown in the figure 2 reconfirms there is no separation flow on the flat plate. It can be found the fact form the figures that the agreements between the computations and experiment are very good. This is important ability which good turbulence model must qualify for numerical simulation of attachment flows.

![Figure 1](image1.png)  
**Figure 1.** Comparison of computed and experimental of pressure distribution.  
(Re = $2.84 \times 10^5$, $M = 2.0$, $\beta = 31.2471$)

![Figure 2](image2.png)  
**Figure 2.** Distribution of skin friction coefficient on the wall of flat plate.  
(Re = $2.84 \times 10^5$, $M = 2.0$, $\beta = 31.2471$)
The second case of interaction is carried out at a Reynolds number of 2.96×10^5, an impinging angle of 32.585 degrees and the free stream Mach number of 2.0. MacCormack [19,21] and others made many numerical simulations on flat plate shock wave boundary layer interaction at the same parameters. In this case, the higher incident shock intensity induces a separation region with a primary vortex on the plate, which is displayed in figure 3. It is can be found in figure 4 which shows distribution of pressure contours of the flow field that computational results including impinging shock, shock induced by front edge of flat plate, reflected shock and expansion region are in accordance with phenomenon of experiment [19,22]. The comparison of skin friction between computed results and experimental data is displayed in the figure 5. It can be seen form the figure 3 and figure 5 that the separation point is about at 0.15 and the reattachment point is at 0.22 around, which agrees with the experiment pretty well. Moreover, the primary vortex is obviously presented on the flat plate. The correspondence of skin friction between the computation and experiment before separation point is good, but the discrepancy between them becomes obvious behind the reattachment point. The most important reason of this discrepancy is that three-dimensional effect is significantly enhanced behind the reattachment point, and two-dimensional numerical simulation can’t address reality flows exactly. This reason was found in many other studies [3,20,23]. Figure 6 shows comparison of pressure distribution between computation results and experimental values. Three mean velocity profiles which are taken in different stations normal to the flat plate are displayed in the figure 7. The three locations are \( x = 0.10(a), x = 0.18(b), x = 0.30(c) \), respectively. The location of (a) is in front of the separation point, the location of (b) is in the separation-region, and the location of (c) is behind the reattachment point. The third and fourth cases are preformed at the free stream Mach number of 2.1 and 2.2 respectively, with a specified Reynolds number of 2.96×10^5 and an impinging angle of 32.585 degrees. The purpose is that we want to figure out the effect of slightly higher Mach number for separation flow induced by shock wave boundary layer interaction. With the increasing incident shock wave intensity, interaction becomes more intense and gives rise to more complex flow patterns.

![Figure 3](image3.png)  
**Figure 3.** Streamlines around separated region.  
(Re = 2.96×10^5, M = 2.0, \( \beta = 32.585^\circ \))

![Figure 4](image4.png)  
**Figure 4.** Distribution of pressure contours.  
(Re = 2.96×10^5, M = 2.0, \( \beta = 32.585^\circ \))

![Figure 5](image5.png)  
**Figure 5.** Distribution of skin friction on the wall of plate.  
(Re = 2.96×10^5, M = 2.0, \( \beta = 32.585^\circ \))

![Figure 6](image6.png)  
**Figure 6.** Comparison of computed and experimental of pressure distribution.  
(Re = 2.96×10^5, M = 2.0, \( \beta = 32.585^\circ \))
Figure 7. Velocity profiles on the wall of the flat plate. — computation experiment. 
(Re = 2.96 × 10^5, M = 2.0, β = 32.585°)

Figure 8 shows a new little secondary vortex under the primary vortex with the parameters of M = 2.1, Re = 2.96 × 10^5, β = 32.585°. Figure 9 displays a larger secondary vortex and the bigger length of separation region by increasing the Mach number from 2.1 to 2.2. Therefore, we reach a conclusion that the secondary vortices will be induced in the primary vortex, when Mach number is greater than 2.0 for specified Re = 2.96 × 10^5 and β = 32.585°. Yiqing Shen et al. [24] calculated a series of shock wave boundary layer interaction using second-order-accurate total variation diminishing (TVD) schemes. They drew a similar conclusion that the secondary vortices will be induced in the primary vortex, when β ≥ 34.047° for M = 2.0 and Re = 2.96 × 10^5 or M > 2.0 for the specified Re = 2.96 × 10^5 and β = 32.585°.

Figure 8. Streamlines around separation-region. 
(Re = 2.96 × 10^5, M = 2.1, β = 32.585°)

Figure 9. Streamlines around separation-region. 
(Re = 2.96 × 10^5, M = 2.2, β = 32.585°)

3.2. Separation flow in a converging-diverging transonic diffuser
A transonic viscous flow in the converging-diverging diffuser flow was chosen as a representative internal-flow case which was conducted experiment study by Sajben. We choose the ratio of exit static pressure to inflow total pressure 0.82 and 0.72. The bottom wall of diffuser is straight, and the top wall is a curve forming a converging-diverging channel with the bottom wall. The heights of entrance, exit and throat are 62mm, 66mm and 44mm, respectively. This diffuser has an entrance to throat ratio of 1.4 and an exit to throat ratio of 1.5. the grid used in present work is 129 × 81, which is sufficiently clustered in the vertical direction so that the first point next to the wall resides inside of the laminar sublayer and y⁺ < 1. The boundary conditions are set as following: stagnation temperature T* = 292 K and stagnation pressure P* = 135 kPa are assigned on the inlet boundary. The flow enters into the duct at M = 0.46.
The ratio of 0.82 based on exit static pressure to inflow total pressure is set up on the outlet boundary. The top wall and bottom wall are considered to be isothermal and non-slipping. The Reynolds number based on incoming flow conditions Re = $0.56 \times 10^6$ m$^{-1}$. The grid used in the numerical simulation is shown in figure 10.

![Figure 10. The grid used in the numerical simulation.](image)

(a) bottom wall pressure distributions.  
(b) top wall pressure distributions.

![Figure 11. Pressure distributions along diffuser walls in the weak interaction.](image)

The pressure distributions along the bottom and top walls of the transonic diffuser are shown in figure 11(a) and (b). The ratio of exit static pressure to inflow total pressure is 0.82. The calculated results are in good agreement with the experimental values because of the weak shock wave boundary layer interaction and there is no separation region. When the intensity of shockwave is increased to the ratio of 0.72. The pressure distributions along the bottom and top walls of the wall displayed in figure 12(a) and (b). The simulated results are in good agreement with experimental data at downstream of the shock. The shock location is well matched with the experiment. Shock intensity, however, is slightly weaker than the experimental result. The density in the transonic diffuser flow field is shown in the contour plot in figure 13.

![Figure 12. Pressure distribution on diffuser walls in the strong interaction.](image)
4. Conclusions

Turbulence models should be able to describe turbulence characteristics of non-linearity, anisotropy and non-equilibrium, especially in the complex separation flows. It is essential to obtain the effective turbulence fluctuation information. For the purpose of retaining the mean behavior of turbulent fluctuations, the Partial-average statistical scheme was introduced to deal with the turbulent fluctuation information. Interactions between oblique shock wave and turbulence boundary layer on a flat plate and in a converging-diverging transonic diffuser have been simulated using anisotropic turbulent model. The Roe scheme with ‘M.U.S.C.L.’ extrapolation technique adopted in the present work has guaranteed the accuracy and stability of the simulation. The numerical results have preliminary proved that this set of turbulent equations can successfully predict shock wave turbulence boundary layer interaction. Non-isotropic eddy viscosity coefficients which are obtained by dyad of turbulent characteristic velocity vector and turbulent characteristic length vector play an important role in present computation. This fundamental simulation supports this set of equations further development for complex three-dimensional turbulent flows.

References

[1] Ferri A 1939 Eng. Trans 17 1940
[2] Chaudhuri A and Hadjadj A 2016 Aerospace Science and Technology 53 10-21
[3] Junxiang S, Tiancheng X and Steven R 2015 Journal of Aerospace Engineering 28 1-11
[4] Tong F, Yu C and Tang Z 2017 Comput Fluids 2017 149 56–69
[5] Vaidheeswaran A and Hibiki T 2017 Int J Heat Mass Transf 115 741–52
[6] Fedorova N N, Fedorchenko I A and Sch’ulein E 2001 Computational Fluid Dynamics Journal 10 376–81
[7] Sch’ulein E 2006 AIAA Journal 44 1732-41
[8] Knight D, Yan H and Panaras A G 2003 Progress in Aerospace Sciences 39 2283-91
[9] Bake S, Meyer D, G W 2002 Journal of Fluid Mechanics 4 217-43
[10] Alam M and Sandham N D 2000 Journal of Fluid Mechanics 410 1-28
[11] Fedorova N N and Fedorchenko I A 2004 Journal of Applied Mechanics and Technical Physics 45 358-66
[12] Yangwei L, Jianuo W and Lipeng L 2016 Mod Phys Lett B 30 573-81
[13] Hadidoolabi M. and Ansarian H 2015 Proc Inst Mech Eng G J Aerosp Eng 230 1334-47
[14] Soltani M R, Younsi S J and Daliri A 2014 Proc Inst Mech Eng G J Aerosp Eng 229 1495-509
[15] Ge G and Yan Y 2005 J Hydraulic Research 43 399-407
[16] Van Leer B 1979 Journal of Computational Physics 32 101-36
[17] Anderson W K and Thomas J L 1986 AIAA Journal 1986 24 1453-60
[18] Juntasaro V, Marquis A 2004 International Journal of Computational Fluid Dynamics 18 569-76
[19] MacCormack R W and Baldwin B S 1975 13th AIAA Aerospace Sciences Meeting 20-22
[20] Xu Wan-wu, Wang Zheng-hua and Hou Zhong-xi 2000 Journal of national university of defense technology 22 20-24
[21] MacCormack R W 1971 Numerical solution of the interaction of a shock wave with a laminar boundary layer (Berlin: Springer) 8 153-63
[22] Hakkinen R J, Greber I, Trilling L and Abarbanel S S The Interaction of an Oblique Shock

Figure 13. Density in the converging-diverging transonic diffuser.
Wave with a Laminar Boundary Layer *NASA* no. 19980231989

[23] David S Dolling 2001 50 Years Shock Wave/boundary Layer Interaction Research - What Next? *AIAA Journal* **39** 1517-31

[24] Yiqing Shen, Ruquan Wang and Hongzhi Liao 2000 *International Journal for Numerical Methods in Fluids* **33** 23-34