

文章编号:1000-8055(2012)01-0124-05

Numerical study of compression corner flowfield using Gao-Yong turbulence model

GAO Ge¹, ZHANG Chang-xian¹, YAN Wen-hui², WANG Yong¹(1. National Key Laboratory of Science and Technology on Aero-Engine Aero-thermodynamics,
School of Jet Propulsion, Beijing University of Aeronautics and Astronautics,

Beijing 100091, China;

2. Chinese Aeronautical Establishment, Beijing 100012, China)

Abstract: A numerical simulation of shock wave turbulent boundary layer interaction induced by a 24° compression corner based on Gao-Yong compressible turbulence model was presented. The convection terms and the diffusion terms were calculated using the second-order AUSM (advection upstream splitting method) scheme and the second-order central difference scheme, respectively. The Runge-Kutta time marching method was employed to solve the governing equations for steady state solutions. Significant flow separation-region which indicates highly non-isotropic turbulence structure has been found in the present work due to intensity interaction under the 24° compression corner. Comparisons between the calculated results and experimental data have been carried out, including surface pressure distribution, boundary-layer static pressure profiles and mean velocity profiles. The numerical results agree well with the experimental values, which indicate Gao-Yong compressible turbulence model is suitable for the prediction of shock wave turbulent boundary layer interaction in two-dimensional compression corner flows.

Key words: shock wave; turbulent boundary layer; interaction;

Gao-Yong compressible turbulence model; compression corner flow

CLC number: V211.3

Document code: A

Shock wave turbulent boundary layer interaction (SWTBLI) is a classical and fundamental problem of fluid mechanics. Numerical study of this issue based on Reynolds-averaged Navier-Stokes (RANS) equations has been extensively implemented^[1-5]. It has become more effective and played an important role in studying supersonic or hypersonic turbulent flows for designing aircraft configurations or controlling flows. It is well known that a shock wave turbulent boundary interaction of two-dimensional compression corner flow is a benchmark test for the evaluation of turbulence models in predicting flow compressibility and anisotropy. The compression corner configuration has been studied extensively by experiments which produce numerous

experimental data. Bookey P B et al.^[6] performed experiments on a 24° compression corner configuration. Settles G S et al.^[7] studied two-dimensional compression corner problems in detail. These results of this study serve as the foundation for the compilation of a database of experimental results at flow conditions accessible to numerical simulations. Hence the objective of present work is to apply the Gao-Yong compressible turbulence model which employs no empirical coefficients or wall functions to simulate 24° compression corner flow. Comparisons between the computations and experiments are presented in this paper, including surface pressure distribution, velocity profiles, and pressure profiles. The computed results obtained from this turbu-

Received:2011-08-16; **revision received:**2011-10-31

E-mail:buaa@sjp, buaa.edu.cn

lence model agree well with the experimental values, which illustrates its capability of predicting shock wave turbulent boundary layer interaction of two-dimensional compressible corners flow.

1 Governing equations

GAO Ge and YONG Yan^[8] proposed a statistical partial average scheme for incompressible turbulent flows in 2004, in which, the first order statistical moment of turbulent fluctuations is retained as a turbulent drift flow, which enables us to rationalize the 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 Gao-Yong compressible turbulence model (a complete set of equations of compressible turbulent flow) without employing any empirical coefficients or wall functions. Numerical simulations^[8-12] of various turbulent benchmark flow have been conducted using this turbulence model, including plane jet flow, round jet flow, laminar-turbulent transition, backward facing step flow, single fin flow, transonic airfoil flow and coherent structures in turbulent boundary layer and so on. The model performed well in these fields. These equations, normalized by the free stream density (ρ_∞), temperature (T_∞), and the characteristic length (L), are given in the following without derivation.

Continuity equation of the mean flow

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

Momentum equation of the mean flow

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j u_i + p \delta_{ij}) = \frac{\partial}{\partial x_j}(\sigma_{ij} + \hat{\sigma}_{ij}^{\text{turb}}) \quad (2)$$

Energy equation of the mean flow

$$\begin{aligned} \frac{\partial(\rho E)}{\partial t} + \frac{\partial}{\partial x_j}[(\rho E + p)u_j] = \\ \frac{\partial}{\partial x_j}[u_i(\sigma_{ij} + \hat{\sigma}_{ij}^{\text{turb}}) - (q_j + q_j^{\text{turb}})] \end{aligned} \quad (3)$$

Continuity equation of the drift flow

$$\frac{\partial}{\partial x_j}(\rho \hat{u}_j) = 0 \quad (4)$$

Momentum equation of the drift flow

$$\begin{aligned} \frac{\partial(\rho \hat{u}_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho \hat{u}_j u_i + \rho \hat{u}_j u_i + \hat{p} \delta_{ij}) = \\ \frac{\partial}{\partial x_j}(\hat{\sigma}_{ij} + \hat{\sigma}_{ij}^{\text{turb}}) - \frac{\partial}{\partial x_j}(\sigma_{ij} + \sigma_{ij}^{\text{turb}}) \end{aligned} \quad (5)$$

Mechanical energy equation (for the computation of turbulence characteristic length scale)

$$\begin{aligned} \sum_{n=1}^{\infty} \frac{\left(l_j \frac{\partial}{\partial x_j}\right)^n \rho u_i \hat{u}_i}{n!} = \\ l_i \left[-\frac{\partial}{\partial x_j}(\rho \hat{u}_j u_i + \hat{p} \delta_{ij}) + \frac{\partial}{\partial x_j}(\hat{\sigma}_{ij} + \hat{\sigma}_{ij}^{\text{turb}}) - \right. \\ \left. \frac{\partial}{\partial x_j}(\sigma_{ij} + \sigma_{ij}^{\text{turb}}) - \alpha \frac{D(\rho u_i)}{Dt} \right] \end{aligned} \quad (6)$$

Constitutive equations

$$\sigma_{ij} = 2\mu \epsilon_{ij} - \frac{2}{3}\mu \epsilon_{kk} \delta_{ij} \quad (7)$$

$$\hat{\sigma}_{ij} = 2\mu \hat{\epsilon}_{ij} - \frac{2}{3}\mu \hat{\epsilon}_{kk} \delta_{ij} \quad (8)$$

$$\sigma_{ij}^{\text{turb}} = 2\mu_{IJ} \epsilon_{ij} - \frac{2}{3}\mu_{KK} \epsilon_{kk} \delta_{ij} \quad (9)$$

$$\hat{\sigma}_{ij}^{\text{turb}} = 2\mu_{IJ} \hat{\epsilon}_{ij} - \frac{2}{3}\mu_{KK} \hat{\epsilon}_{kk} \delta_{ij} \quad (10)$$

Where I, J , and K taking the same values as i, j , and k , do not imply tensor summation, and the dynamic viscosity coefficient μ can be calculated by Sutherland's law in the form of

$$\mu = \mu_0 \left(\frac{T}{T_0}\right)^{1.5} \frac{T_0 + T_s}{T + T_s} \quad (11)$$

Turbulent viscosity coefficient μ_{IJ} is

$$\mu_{IJ} = |\xi_{ij}| + |\xi_{ji}| \quad (12)$$

where

$$\xi_{ij} = \rho l_j \hat{u}_i \quad (13)$$

The heat flux terms

$$q_j = \kappa \frac{\partial T}{\partial x_j} = \frac{c_p \mu}{Pr} \quad (14)$$

$$q_j^{\text{turb}} = \kappa^{\text{turb}} \frac{\partial T}{\partial x_j} = \frac{c_p \mu^{\text{turb}}}{Pr^{\text{turb}}} \quad (15)$$

The above equations are closed by the equation of state of a perfect gas

$$p = \frac{1}{\gamma} \rho T \quad (16)$$

2 Numerical simulations

In this numerical simulation, 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 AUSM (advection upstream splitting method) scheme, 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 , respectively. Acceleration techniques, such as residual averaging and local time stepping, are adopted to speed up the convergence of the numerical solution.

The computational domain extends from $x/\delta_0 = -4.0 \sim 7.3$ in the x direction and from $y/\delta_0 = 0 \sim 8.25$ in the y direction. The leading edge of the flat plate is placed at $x/\delta_0 = -4.0$ and the 24° corner at $x/\delta_0 = 0$. δ_0 is the turbulent boundary layer thickness equal to 2.3 cm. The boundary conditions are defined as follow: at the inlet, all of the flowfield variables are specified based on the experiment conducted by Settles G S et al^[13], such as free stream Mach number $Ma_\infty = 2.85$, stagnation pressure $p^* = 6.89 \times 10^5$ Pa, stagnation temperature $T^* = 262$ K and average free stream unit Reynolds number $Re_\infty = 7.3 \times 10^7 \text{ m}^{-1}$. On the solid wall (flat plate and compression ramp), the no-slip condition and the wall temperature of 276 K are employed. At the outflow boundary, the second derivative of pressure, and the first derivatives of velocity and temperature have been set equal to zero in the direction parallel to the ramp. At the upper boundary, the first derivatives of all flowfield variables in the y direction are assumed to be zero. A single structured mesh grid which has 100 nodes along the x direction and 60 nodes in the y direction normal to the solid wall is employed in present computational case. The mesh nodes at the y direction are refined toward the solid wall surface for resolving the velocity boundary layer.

Computational results of the two-dimensional compression corner obtained by Gao-Yong compressible turbulence model are presented as follows. Fig. 1 shows the distribution of the

streamlines and velocity vectors. The flow separation ahead of the corner and the reattachment downstream of the corner are obvious, which are induced by the shock wave turbulent boundary layer interaction in the corner region. It is known that various turbulent characteristic lengths exist in a boundary layer, especially for separation flow with highly non-isotropic turbulent viscosity. The abscissa and ordinate are normalized by the boundary layer thickness δ_0 .

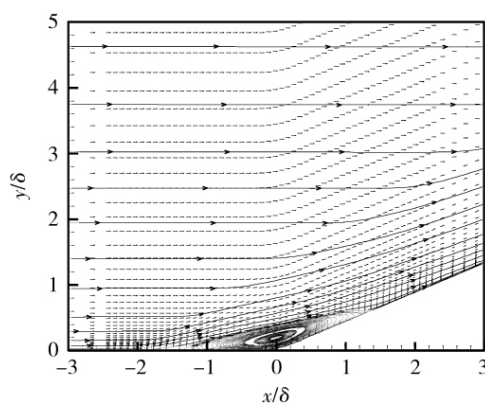


Fig. 1 Streamtrace patterns in the compression corner

Fig. 2 shows the comparison of the static pressure calculated using Gao-Yong turbulence equations with the experimental measurements of Settles G S^[13]. A large separation induced by shock wave turbulent boundary interaction takes place at $x/\delta_0 = -1.5$. It can be seen from Fig. 2 that the calculated separation point is in good agreement with the experimental data which separate at $x/\delta_0 = -1.46$. The surface static pressure distribution exhibits well-defined “kinks”

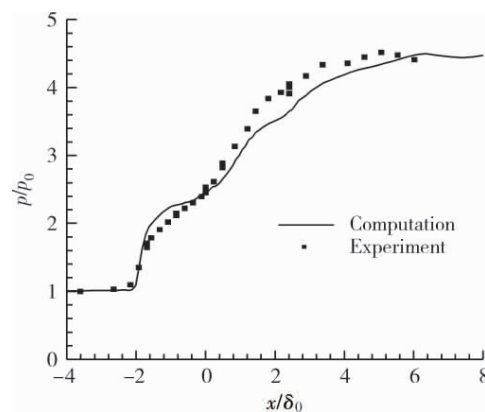


Fig. 2 Wall pressure distribution

zone^[14], which indicates the presence of flow separation. However, the numerical simulation tends to overpredict the separation. The surface pressure in the region between separation point and the origin point of the corner is overpredicted but the surface pressure in the region downstream of the corner is underpredicted. Overall the present simulation gives a reasonable result in this flow.

Five mean streamwise velocity profiles which are taken at different stations normal to solid surface are displayed in Fig. 3. The five locations are $x/\delta_0 = -3.012, -0.482, 0, 1.446, 4.819$, respectively. These plots serve to illustrate the changes of velocity profiles in boundary-layer which occur along the length of the interaction flowfield. The location of $x/\delta_0 = -3.012$ is

in front of the separation point, the locations of $x/\delta_0 = -0.482$ and 0 are in the separation-region, and the location of $x/\delta_0 = 1.446$ and 4.819 behind the reattachment point. At the first position, the manifestation of the velocity profile looks like the one in the usual two dimension boundary layer which is in the equilibrium status. At the locations of $x/\delta_0 = -0.482$ and 0 , which are in the separation zone, computational mean velocity profiles are over estimated compared with experimental results. The primary reason is that turbulent viscosity calculated by this turbulence model is smaller than that of real flow. Good agreements between the computational results and experimental values are obtained far-away downstream of reattachment point.

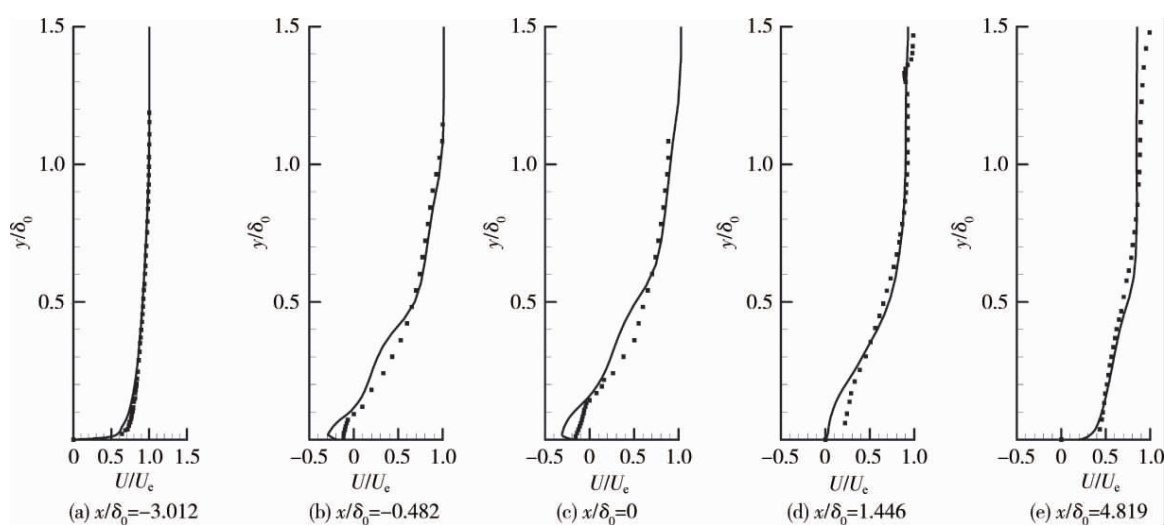


Fig. 3 Mean streamwise velocity profiles at different stations

Fig. 4 shows calculated boundary-layer static pressure profiles which are compared with experimental values. The locations of the boundary-layer static pressure profiles are same as the locations of velocity profiles in Fig. 3. It can be seen from this figure that the pressure gradient at the first position is zero, which just like in the plate boundary layer. After the separation point, the static pressure profiles reveal a large normal static pressure gradient at the position where the data line crosses through the shock-

wave. And the profile at $x/\delta_0 = 4.819$ displays a lingering normal static pressure gradient in the boundary layer, etc., in the region $y/\delta_0 < 1.0$, emerging from the interaction. It is believed that the downstream normal static pressure gradients are produced by waves which are remnants of the interaction at the compression corner^[15]. Based on the figures, it can be found that the agreements between the computational results and experimental values are well.

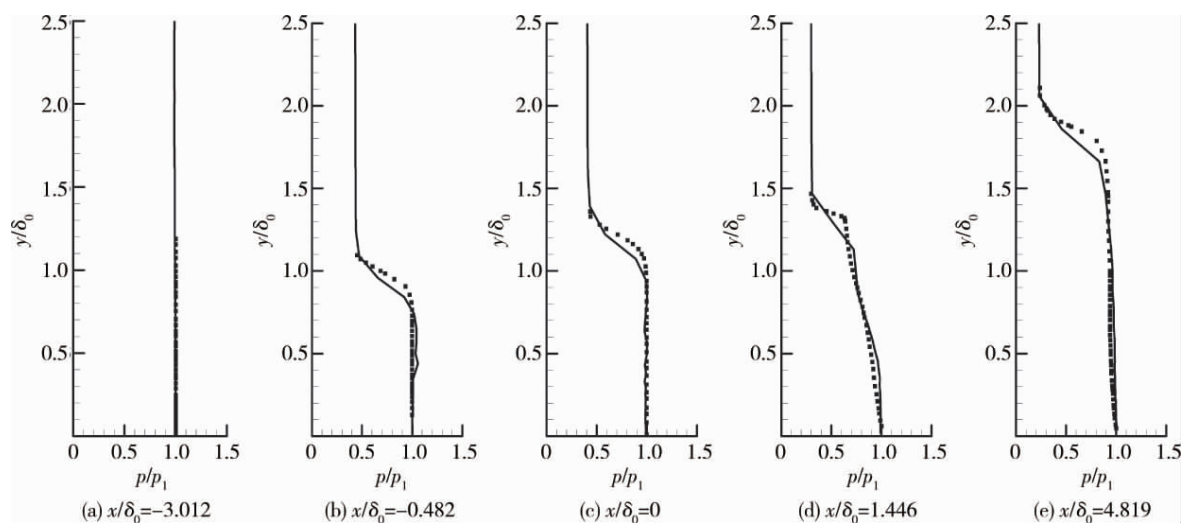


Fig. 4 Boundary layer static pressure profiles distribution

3 Conclusions

Shockwave-turbulent boundary layer interaction induced by a 24° compression corner has been simulated using Gao-Yong compressible turbulence model. The mean surface pressure and mean velocity profiles predicted show good agreement with experiment. The numerical results preliminarily prove that this set of turbulent equations can successfully predict two-dimensional compression corner flow. Non-isotropic eddy viscosity coefficients composed of a couple of turbulent characteristic velocity vector and turbulent characteristic length vector (Eq. 13) play an important role in the present computation. This fundamental simulation supports further development of Gao-Yong compressible turbulence model for complex three-dimensional turbulent flows.

References:

- [1] Adnan Q, Nadeem H, Sanjeev S. New scheme for the computation of compressible flows[J]. AIAA Journal, 2006, 44(5): 1025-1039.
- [2] Wu M, Martin M P. Direct numerical simulation of supersonic turbulent boundary layer over a compression ramp [J]. AIAA Journal, 2007, 45(4): 879-889.
- [3] Edwards J R, Choi J I, Boles J A. Large-eddy/Reynolds-averaged Navier-Stokes simulation of a Mach 5 compression-corner interaction[J]. AIAA Journal, 2009, 46(4): 373-385.
- [4] Martin M P, Taylor E M, Wu M, et al. Bandwidth-optimized WENO scheme for the effective direct numerical simulation of compressible turbulence[J]. Journal of Computational Physics, 2006, 220(1): 270-289.
- [5] Knight D, Yan H, Panaras A G, et al. Advances in CFD prediction of shock wave turbulent boundary layer interactions[J]. Progress in Aerospace Sciences, 2003, 39(2): 2283-2291.
- [6] Bookey P B, Wyckham C, Smits A J, et al. New experimental data of STBLI at DNS/LES accessible Reynolds numbers[R]. AIAA-2005-309, 2005.
- [7] Settles G S, Perkins J J, Bogdonoff S M. Investigation of three-dimensional shock/boundary-layer interactions at swept compression corners[J]. AIAA Journal, 1980, 18(7): 779-785.
- [8] GAO Ge, YONG Yan. On incompressible turbulent flow: partial average based theory and applications[J]. Journal of Hydraulic Research, 2005, 43(4): 399-407.
- [9] LI Zhiqiang, LI Zhengqiang, YAN Wenhui. Numerical simulation of shock wave/flat-plate turbulent boundary layer interaction[J]. Journal of Aerospace Power, 2008, 23(6): 1003-1007. (in Chinese)
- [10] YAN Wenhui, YAN Wei, GAO Ge. Simulation of transonic viscous flows around RAE2822 airfoil in Gao-Yong compressible turbulence model[J]. Chinese Journal of Computational Physics, 2008, 25(6): 694-700. (in Chinese)
- [11] YAN Wenhui, GAO Ge, YONG Yan. Numerical simulation of 3-D shock wave/turbulent boundary layer interaction using Gao-Yong turbulence model[J]. Journal of Aerospace Power, 2009, 24(10): 2193-2200. (in Chinese)
- [12] GAO Ge, YONG Yan. Partial-average-based equations of incompressible turbulent flow[J]. International Journal of Non-Linear Mechanics, 2004, 39(9): 1407-1419.
- [13] Settles G S, Vas I E, Bogdonoff S M. Shock wave turbulent boundary layer interaction at high Reynolds number, including separation and flowfield measurements[R]. AIAA-1976-164, 1976.
- [14] Kuehn D M. Experimental investigation of the pressure rise required for the incipient separation of turbulent boundary layers in two-dimensional supersonic flow[R]. NASA-Me-1-21-59A, 1959.
- [15] Settles G S, Fitzpatrick T J, Bogdonoff S M. Detailed study of attached and separated compression corner flowfields in high Reynolds number supersonic flow[J]. AIAA Journal, 1978, 17(6): 579-585.