

Numerical study of transonic separation flow using an anisotropic turbulence model

YAN Wenhui¹, PENG Tengfei¹, YONG Yan²,
QIN Ruixin¹, MA Ruige¹

(1. School of Mechanical and Materials Engineering,
North China University of Technology, Beijing 100144, China;
2. Department of Civil, Environmental and Geomatics Engineering,
Florida Atlantic University, Boca Raton, Florida 33431, U. S. A.)

Abstract: A transonic turbulent separation flow in a converging-diverging transonic diffuser was studied, when there existed a separation bubble on the top wall of the diffuser triggered by strong shock-wave-boundary-layer-interaction (SWBLI). To capture the essential behavior of this complex flow, the current study utilized an anisotropic turbulence model developed on the basis of a statistical partial average scheme. The first order moment of turbulent fluctuations, retained by a novel average scheme, and the turbulent length scale, can be determined from the momentum equations and mechanical energy equation of the fluctuation flow, respectively. The two physical quantities were readily used to construct the nonlinear anisotropic eddy viscosity tensor and to significantly improve the computational results. Comparisons between the computational results and experimental data were carried out for velocity profiles, pressure distribution, skin friction coefficient, Reynolds stress as well as streamline vectors distribution. Without using any empirical coefficients and wall functions, the numerical results were in good agreement with the available experimental data, further confirming that the nonlinear anisotropic eddy viscosity tensor is the decisive factor for the success of the computational results.

Key words: transonic separation flow; eddy viscosity tensor; shock wave boundary layer interaction; anisotropic turbulence model; statistical partial average scheme

CLC number: V231.3

Document code: A

Turbulent separation flow is very common in aerospace engineering applications as it significantly affects the aircraft performance. Many interesting phenomena can be observed in the turbulent separation flow, such as flow detachment in the boundary layer induced by shock-wave, laminar-turbulent transition, and flow relaxation behind the flow separations and reattachments^[1-3]. Existing in the reattachment flow after separation, the flow relaxation takes place

when the rate of change in Reynolds stress significantly lags behind the rate of change in strain. As a result, the calculated mean velocity profiles have large defects compared to the experimental data. Due to its important applications in aerospace engineering, the turbulent separation flow has been a hot topic for researchers who tried to reveal the mechanism behind these complex phenomena. A transonic interaction between a steady shock wave and a turbu-

Received: 2019-06-14

Fundation: Aerospace Power Foundation (6141B09050397);

Foudamental Reasearch Foundation of North China University of Technology (110052971921/042)

Citation: YAN Wenhui, PENG Tengfei, YONG Yan, et al. Numerical study of transonic separation flow using an anisotropic turbulence model[J]. Journal of Aerospace Power, 2020, 35(1): 114-125.

lent boundary layer in a Mach number 1.4 channel flow was studied through experiments by Fulvio, et al.^[4]. They measured the mean velocity fields in the entire interaction region with an attention on the recirculation bubble at the downstream of the detachment and deliberated in details on the turbulent production term and the return to isotropy mechanism. Bogar, et al.^[5] chose a converging-diverging transonic diffuser as a representative internal-flow case to conduct a series of experiments for the study of flow compressibility and stability. They reached a conclusion that the single characteristic frequency observed does not follow acoustic predictions in the shock-induced separation. Szumowski, et al.^[6] conducted an experimental investigation of Laval nozzle flow related to exit pressure ratios, which corresponds to shock wave positions close to the nozzle throat. Also discussed in details were oscillatory instabilities on shock pattern, separation bubble and flow rate. Bruce, et al.^[7] did an experiment of the flow structure around a single three-dimensional transonic bump mounted on a flat plate in a wind tunnel. They found that span-wise pressure gradients across the bump ramp affect the magnitude and uniformity of flow-turning generated by the bump. Soltani, et al.^[8] investigated the performance of a supersonic mixed compression air intake. Their results showed that the application of the boundary layer suction at upstream of the intake throat can considerably improve the intake performance at its design and off-design conditions. In numerical simulations, Reynolds-averaged Navier-Stokes equations (RANS) in conjunction with different turbulence models are still of a popular method since the scale-resolving simulation of turbulent flows requires high computation resource, which is impractical for engineering applications. Thus, the RANS approach is an acceptable compromise of accuracy and efficiency. Vlahostergios, et al.^[9] simulated the flow in a transonic diffuser with models of one Reynolds-stress and two eddy viscosities. The compressibility effect was investigated, and detailed veloc-

ity and pressure coefficient distributions were compared with the available experimental data. Liu, et al.^[10] employed eight turbulence models, including Spalart-Allmaras (S-A) model, $k-\epsilon$ model, shear stress transport (SST) $k-\omega$ model and others to investigate transonic flows in the Sajben transonic diffuser. They compared performance of different models in terms of velocity profiles, pressure distribution, separation location as well as reattachment location. Hadidoolabi, et al.^[11] simulated compressible flows over a 60° sweep delta wing based on the $k-\omega$ SST turbulence model and a dual-time implicit time integration model. Also investigated were variations of flow patterns, pressure coefficient on the wall surface, and hysteresis loops associated with the vortex core location were investigated. Mazaheri, et al.^[12] mainly studied shock-wave-boundary-layer-interaction (SWBLI) as the key factor influencing the aerodynamic performance of transonic bumped airfoils. They gave detailed interpretations of the decay of mean velocity profiles at downstream of the separation flow.

When a turbulence model assumes an isotropic eddy viscosity in the modeling of a Reynolds stress tensor, it often yields inaccurate predictions of the normal Reynolds stresses and certain important characteristics of secondary flows in noncircular ducts and separated flows where the anisotropic normal Reynolds stresses play an important role^[13]. The interactive effects of shear, strong inverse pressure gradient and recirculation-related curvature close to a solid surface create a complex, highly anisotropic turbulence flow structure^[14]. The similar behavior can be observed in the transonic turbulent separation flow in a converging-diverging diffuser presented in the current work.

In the current study, the separation flow induced by SWBLI with highly anisotropic turbulence structure is simulated using the nonlinear anisotropic turbulence model^[15-16]. Reynolds stress-strain constitutive relationship and anisotropic eddy viscosity tensor used in this model

have improved the simulation accuracy. This non-linear anisotropic turbulence model was recently developed using the statistical partial average scheme for the purpose of retaining the mean behavior of turbulent fluctuations. Unlike the conventional statistical average schemes, the first order statistical moments of turbulent fluctuations provide a vital tool to uncover the relationship between mean flow and fluctuation flow through additional momentum equations of the mean fluctuation flow. The turbulent fluctuation flow is also contributes to the modeling of anisotropic eddy viscosity tensor. Comparisons between the computational results and available experimental data, including velocity profiles, pressure distribution, skin friction coefficient distribution, Reynolds stress as well as streamline vectors are also presented in this paper.

1 Governing equations

It is known that the highly irregular turbulent flows are governed by the Navier-Stokes (N-S) equations, and the average values of turbulent fluctuation variables are equal to zero. The statistical characteristics of the fluctuations, such as the fluctuation distribution versus time and spatial coordinates, are completely out of reach even though they do exist in reality. To recover their true characteristics, we consider φ_k being the k -th solution of a physical variable of the N-S equation. The corresponding fluctuation component is written as

$$\varphi'_k = \varphi_k - \bar{\varphi} \quad (1)$$

where $\bar{\varphi}$ is an ensemble average, or Reynolds average

$$\bar{\varphi} = \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{k=1}^N \varphi_k \quad (2)$$

And N is the total number of all solutions of the N-S equations. Based on Eq. (2), the ensemble average of all fluctuations vanishes, meaning

$$\bar{\varphi}' = 0 \quad (3)$$

The above Eq. (3) only reveals the fact that turbulent fluctuations are equal to zero.

In a compressible turbulent flow, the mass density ρ_k is still a function of space and time just

like other physical variables. The mass density is often explicitly coupled with other physical variables in the N-S equations which increase more difficulty to average turbulent fluctuations. This coupling effect is often conveniently treated in the following mass weighted ensemble average, or Favre average

$$\bar{\varphi} = \frac{1}{\bar{\rho}} \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{k=1}^N \rho_k \varphi_k \quad (4)$$

where $\bar{\rho}$ is the ensemble average of fluid mass density. Eq. (4) can also be written as

$$\bar{\varphi} = \frac{1}{\bar{\rho}} \overline{\rho \varphi} \quad (5)$$

which is an exchangeable form of the Favre average and the Reynolds average. The corresponding fluctuation component, therefore, is

$$\varphi''_k = \varphi_k - \bar{\varphi} \quad (6)$$

As a consequence of the Favre average,

$$\bar{\varphi}'' = 0 \quad (7)$$

It is found from the Eq. (7) that the fluctuation variable φ'' is statistically symmetric about zero too. This symmetry leads to an impossibility to directly observe fluctuation physical behavior and creates the well-known closure problem. Nevertheless, a portion of the fluctuation field is observable if it could be separated from the remaining fluctuations. Based on this idea, a statistical partial average scheme is created to observe and handle a portion of the fluctuation variables. In particular, the fluctuation variables are divided into two different groups based on a certain criterion, namely drift flow 1 and drift flow 2 respectively. The Favre average of each group of fluctuation variables is obtained by

$$\bar{\varphi}_1 = \frac{1}{\bar{\rho}} \lim_{N_1 \rightarrow \infty} \frac{1}{N_1} \sum_{k=1}^{N_1} \rho_k (\varphi''_k)_1 \quad (8)$$

and

$$\bar{\varphi}_2 = \frac{1}{\bar{\rho}} \lim_{N_2 \rightarrow \infty} \frac{1}{N_2} \sum_{k=1}^{N_2} \rho_k (\varphi''_k)_2 \quad (9)$$

where N_1 and N_2 are numbers of the solutions of the first and second groups of the fluctuation variables, respectively. The Favre average fluctuation variables Eq. (7) can be rewritten as

$$\frac{N_1}{N \bar{\rho}} \lim_{N_1 \rightarrow \infty} \frac{1}{N_1} \sum_{k=1}^{N_1} \rho_k (\varphi''_k)_1 +$$

$$\frac{N_2}{N\bar{\rho}} \lim_{N_2 \rightarrow \infty} \frac{1}{N_2} \sum_{k=1}^{N_2} \rho_k (\varphi_k'')_2 = 0 \quad (10)$$

or

$$M_1 \overline{\varphi_1} + M_2 \overline{\varphi_2} = 0 \quad (11)$$

in which $M_1 = \frac{N_1}{N}$ and $M_2 = \frac{N_2}{N}$. With definitions of

$$\hat{\varphi}_1 = M_1 \overline{\varphi_1} \quad (12)$$

and

$$\hat{\varphi}_2 = M_2 \overline{\varphi_2} \quad (13)$$

$$\hat{\varphi}_1 + \hat{\varphi}_2 = 0 \quad (14)$$

which shows the statistical symmetry property of the two groups of turbulent fluctuation variables about zero.

To have a better illustration of statistical partial average scheme of fluctuation variables, taking fluctuation velocity as an example, a fictitious non-dimensional fluctuation velocity shown in Fig. 1 could be used to explain its physical representations under the average of a certain time span. As an example, the division criterion of the fluctuation velocity is based on signs which means positive fluctuation velocities are in group 1 and negative fluctuation velocities in group 2. The time span for the positive fluctuation velocities is $N_1 = 12$ s, while that for the negative fluctuation velocities $N_2 = 24$ s. The average over the entire time span of $N = 36$ s is $[(6) \times (3) \times (4) + (-2) \times (2) \times (6) + (-4) \times (2) \times (6)]/36 = 0$. When taking averages of

fluctuation velocities in group 1 and fluctuation velocities in group 2 separately, it will be found that the average of the positive fluctuation velocities over the time span of N_1 is $\bar{u}_1 = (6) \times (3) \times (4)/12 = 6$. The average of the negative fluctuation velocities over the time span of N_2 is $\bar{u}_2 = [(-2) \times (2) \times (6) + (-4) \times (2) \times (6)]/24 = -3$. Although two drift flows seem to carry different information, they are statistically symmetric in the sense of weighted averages, $\hat{u}_1 = M_1 \bar{u}_1 = (12/36) \times (6) = 2$ and $\hat{u}_2 = M_2 \bar{u}_2 = (24/36) \times (-3) = -2$. The statistical symmetry Eq. (14) indicates that the fluctuation information can be represented by one of weighted drift flows.

With the first order statistical information of turbulent fluctuations in hand, we are ready to uncover the physical mechanism of nonlinear and anisotropy in complex turbulent flows. The governing equations are given in Cartesian coordinates as the following forms with the aid of statistical partial average scheme^[15-16]. These equations normalized by the free stream density, the free stream sound, the free stream static temperature and the characteristic length of the throat height H_{throat} of the transonic diffuser. Superscript “turb” means turbulence. The key equations used in this study are given here without detailed derivations.

Continuity equation of the mean flow:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \quad (15)$$

Momentum equation of the mean flow:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j u_i + p \delta_{ij}) = \frac{\partial}{\partial x_j} (\tau_{ij} + \tau_{ij}^{\text{turb}}) \quad (16)$$

Energy equation of the mean flow:

$$\begin{aligned} \frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_j} [(\rho E + p) u_j] = \\ \frac{\partial}{\partial x_j} [u_i (\tau_{ij} + \tau_{ij}^{\text{turb}}) - (q_j + q_j^{\text{turb}})] \end{aligned} \quad (17)$$

Continuity equation of the drift flow:

$$\frac{\partial}{\partial x_j} (\rho \hat{u}_j) = 0 \quad (18)$$

Momentum equation of the drift flow:

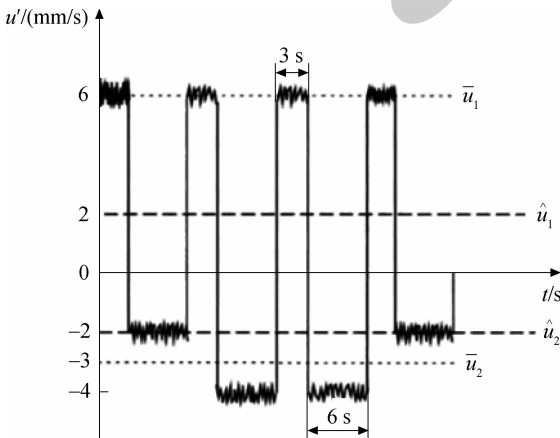


Fig. 1 Schematic fluctuation velocities under time average

$$\frac{\partial(\rho\hat{\mathbf{u}}_i)}{\partial t} + \frac{\partial}{\partial \mathbf{x}_j}(\rho\hat{\mathbf{u}}_i\mathbf{u}_j + \rho\hat{\mathbf{u}}_j\mathbf{u}_i + \hat{p}\delta_{ij}) =$$

$$\frac{\partial}{\partial \mathbf{x}_j}(\hat{\boldsymbol{\tau}}_{ij} + \hat{\boldsymbol{\tau}}_{ij}^{\text{turb}}) - \frac{\partial}{\partial \mathbf{x}_j}(\boldsymbol{\tau}_{ij} + \boldsymbol{\tau}_{ij}^{\text{turb}}) \quad (19)$$

Mechanical energy equation for the computation of turbulence characteristic length scale:

$$\sum_{n=1}^{\infty} \frac{\left(l_j \frac{\partial}{\partial \mathbf{x}_j}\right)^n \rho \mathbf{u}_i \cdot \hat{\mathbf{u}}_i}{n!} = l_i \left[-\frac{\partial}{\partial \mathbf{x}_j}(\rho\hat{\mathbf{u}}_j\mathbf{u}_i + \hat{p}\delta_{ij}) + \right.$$

$$\left. \frac{\partial}{\partial \mathbf{x}_j}(\hat{\boldsymbol{\tau}}_{ij} + \hat{\boldsymbol{\tau}}_{ij}^{\text{turb}}) - \frac{\partial}{\partial \mathbf{x}_j}(\boldsymbol{\tau}_{ij} + \boldsymbol{\tau}_{ij}^{\text{turb}}) \right] \quad (20)$$

Where i and j are subscripts of Einstein's summation convention, $\mathbf{x}$ represents the coordinates in Cartesian coordinate system, $\mathbf{u}$ is mean flow velocity, $\hat{\mathbf{u}}$ is drift flow velocity, p is pressure of mean flow, $\hat{p}$ is pressure of drift flow, ρ is mass density, E is total energy, $\mathbf{q}_j$ is laminar heat flux, $\mathbf{q}_j^{\text{turb}}$ is turbulent heat flux, $\mathbf{l}$ is turbulent characteristic length vector. δ_{ij} is Kronecker symbol. $n!$ means summation of series.

The laminar stress terms for mean flow $\boldsymbol{\tau}_{ij}$ and drift flow $\hat{\boldsymbol{\tau}}_{ij}$ can be described as follows

$$\boldsymbol{\tau}_{ij} = 2\mu\epsilon_{ij} - \frac{2}{3}\mu\epsilon_{kk}\delta_{ij} \quad (21)$$

$$\hat{\boldsymbol{\tau}}_{ij} = 2\mu\hat{\epsilon}_{ij} - \frac{2}{3}\mu\hat{\epsilon}_{kk}\delta_{ij} \quad (22)$$

Where ϵ is strain rate tensor, the laminar dynamic viscosity coefficient μ can be calculated by Sutherland's law in the form of

$$\mu = \mu_0 \left(\frac{T}{T_0}\right)^{1.5} \frac{T_0 + T_s}{T + T_s} \quad (23)$$

Where T is local temperature, T_0 is reference temperature of free-stream, T_s is a Sutherland's constant. The following constitutive relationship between stresses and strains needs to be observed in order to determine the turbulence stress tensor:

$$\begin{bmatrix} \tau_{xx} \\ \tau_{yy} \\ \tau_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \times \begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \epsilon_{xy} \\ \epsilon_{yz} \\ \epsilon_{zx} \end{bmatrix} \quad (24)$$

Most of the turbulent flow is anisotropic and three-dimensional in practice. The 36 constitutive coefficients C_{ij} in Eq. (24) are independent in this type of flow. These coefficients depend on coordinate systems. As the drift flows are statistically symmetric and strongly affect the turbulent mean flow, it is reasonable to assume that the drift flow variables constitute an orthotropic environment for the mean flow. If the coordinate system coincides with three principal material axes, n_1 , n_2 , and n_3 , where n_1 is in the mean flow streamline direction, n_2 and n_3 are orthogonal coordinates on the plane normal to the mean flow streamline, the constitutive relationship between the stresses and strains can be rewritten as

$$\begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{21} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{31} & C_{32} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{bmatrix} \times \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \epsilon_{12} \\ \epsilon_{23} \\ \epsilon_{31} \end{bmatrix} \quad (25)$$

Based on the theory of eddy viscosity turbulence model, the turbulent characteristic length vector $\mathbf{l}$ and drift flow velocity can be used to define eddy viscosity tensor as

$$\xi_{ij} = \rho l_j \hat{\mathbf{u}}_i \quad (26)$$

The constitutive equations with symmetric property are written as

$$\boldsymbol{\tau}_{ij}^{\text{turb}} = 2\mu_{IJ}\epsilon_{ij} - \frac{2}{3}\mu_{KK}\epsilon_{kk}\delta_{ij} \quad (27)$$

$$\hat{\boldsymbol{\tau}}_{ij}^{\text{turb}} = 2\mu_{IJ}\hat{\epsilon}_{ij} - \frac{2}{3}\mu_{KK}\hat{\epsilon}_{kk}\delta_{ij} \quad (28)$$

where $\boldsymbol{\tau}_{ij}^{\text{turb}}$ is turbulent stress tensor of mean flow and $\hat{\boldsymbol{\tau}}_{ij}^{\text{turb}}$ is turbulent stress tensor of drift flow, in which I , J , and K take the same values of i , j , and k , but do not imply tensor summation, μ_{IJ} is turbulent viscosity in orthotropic coordinate system and

$$\mu_{IJ} = |\xi_{ij}| + |\xi_{ji}| \quad (29)$$

Equation (25) can thus be written as

$$\begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{bmatrix} = 2 \begin{bmatrix} \frac{4}{3} |\xi_{11}| & -\frac{2}{3} |\xi_{22}| & -\frac{2}{3} |\xi_{33}| & 0 & 0 & 0 \\ -\frac{2}{3} |\xi_{11}| & \frac{4}{3} |\xi_{22}| & -\frac{2}{3} |\xi_{33}| & 0 & 0 & 0 \\ -\frac{2}{3} |\xi_{11}| & -\frac{2}{3} |\xi_{22}| & \frac{4}{3} |\xi_{33}| & 0 & 0 & 0 \\ 0 & 0 & 0 & |\xi_{12}| + |\xi_{21}| & 0 & 0 \\ 0 & 0 & 0 & 0 & |\xi_{23}| + |\xi_{32}| & 0 \\ 0 & 0 & 0 & 0 & 0 & |\xi_{31}| + |\xi_{13}| \end{bmatrix} \times \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \epsilon_{12} \\ \epsilon_{23} \\ \epsilon_{31} \end{bmatrix} \quad (30)$$

It is found in the Eq. (30) that there are only 6 independent constitutive coefficients for an orthotropic turbulent flow. The strain-stress relationship in the xyz -coordinates can be written as

$$\begin{bmatrix} \tau_{xx} \\ \tau_{yy} \\ \tau_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = 2\psi^{-1} \begin{bmatrix} \frac{4}{3} |\xi_{11}| & -\frac{2}{3} |\xi_{22}| & -\frac{2}{3} |\xi_{33}| & 0 & 0 & 0 \\ -\frac{2}{3} |\xi_{11}| & \frac{4}{3} |\xi_{22}| & -\frac{2}{3} |\xi_{33}| & 0 & 0 & 0 \\ -\frac{2}{3} |\xi_{11}| & -\frac{2}{3} |\xi_{22}| & \frac{4}{3} |\xi_{33}| & 0 & 0 & 0 \\ 0 & 0 & 0 & |\xi_{12}| + |\xi_{21}| & 0 & 0 \\ 0 & 0 & 0 & 0 & |\xi_{23}| + |\xi_{32}| & 0 \\ 0 & 0 & 0 & 0 & 0 & |\xi_{31}| + |\xi_{13}| \end{bmatrix} \psi \times \begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \epsilon_{xy} \\ \epsilon_{yz} \\ \epsilon_{zx} \end{bmatrix} \quad (31)$$

where ψ is a coordinate transformation matrix between the xyz and principal material coordinates n_1 , n_2 and n_3 . The Eq. (31) resembles Eq. (24), which has 36 non-zero coefficients of viscosity.

The above equations are closed by the ideal gas state equation, γ is ratio of specific heats:

$$p = \frac{1}{\gamma} \rho T \quad (32)$$

2 Numerical method and boundary conditions

The computer program code used in the present study has been validated through simulating slow-speed flow around NACA4412 airfoil^[16], shock wave turbulent boundary layer interaction^[17] in previous work. Details of the code are summarized as follows. A semi-discrete fi-

nite-volume approach was utilized for the spatial discretization of the time-dependent governing equations. The computational algorithm is the explicit, upwind-difference scheme with the convection terms discretized by Roe's flux-difference-splitting technique^[18] and diffusion terms evaluated by the second-order central-difference scheme. The monotonic upstream scheme for conservation law (MUSCL)^[19-20] extrapolation technique and Koren flux limiter^[21] were used to maintain accuracy, monotonicity and robustness for shockwave discontinuities. The steady state solution of the differential equations for each computational cell was determined using explicit five-stage Runge-Kutta^[22-23] marching algorithm, in which the Runge-Kutta coefficients were set to 1/4, 1/6, 3/8, 1/2, and 1. Acceleration techniques, such as residual averaging and local

time stepping, were adopted to accelerate the convergence of the numerical solution.

In the present work, the transonic viscous flow in the converging-diverging diffuser flow was chosen as a representative internal-flow case which was conducted experiment study by Bogar, et al^[5]. There are two different flow patterns in this converging-diverging diffuser depending on the ratio of exit static pressure to inflow total pressure. One is the weak-shock case with the ratio of 0.82, and the other is the strong-shock case with the ratio of 0.72. The strong-shock case is chosen because it reflects the typical internal transonic viscous turbulent separation flow which can be evaluated for the performance of the nonlinear Reynolds stress-strain relationship and the anisotropic eddy viscosity tensor. Fig. 2 is a schematic of the two-dimensional transonic diffuser geometry. The bottom wall 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 62 mm, 66 mm and 44 mm, respectively. This configuration has an entrance to throat ratio of 1.4, an exit to throat ratio of 1.5.

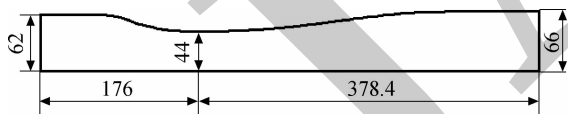


Fig. 2 Schematic of transonic diffuser (unit:mm)

Furthermore, the streamwise computational domain x/H_{throat} is between -4.00 and 8.60 , where x is the horizontal ordinate and H_{throat} the height of throat of the transonic diffuser. The original point is arranged at the throat location in order to compare with the existing experimental data. Fig. 3 shows a computational mesh generated using an algebraic method. In order to guarantee the grid fitting property and orthogonality to the geometric shape of the transonic diffuser, a hyperbolic sine function method is implemented. It was found that the grid is sufficiently clustered in the vertical direction so that the first point next to the wall resides inside of the lami-

nar sublayer and $y^+ < 1$. In addition, the grid was clustered in the region of shock wave to capture its details. In order to evaluate the grid effects in this numerical simulation, four sets of grids, namely 95×55 , 101×65 , 129×65 and 129×81 , are carried out respectively. It was found that the results from these four sets of grid are basically the same. The calculation results given later in this article were obtained through the grid of 101×65 .

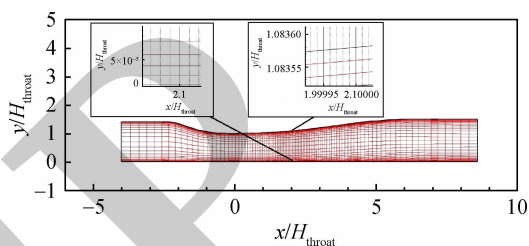


Fig. 3 Computational mesh for the transonic diffuser

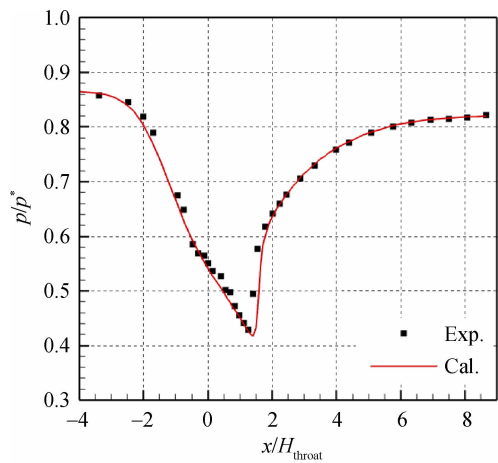
Transonic diffuser flow belongs to a typical internal-flow bounded by the bottom wall, top wall, inlet and outlet in the computational domain. The boundary conditions are set as following: stagnation temperature $T^* = 292$ K and stagnation pressure $p^* = 135$ kPa are assigned on the inlet boundary. The flow enters into the duct at $M = 0.46$. The ratio of 0.72 and 0.82 based on exit static pressure to inflow total pressure are 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 and H_{throat} $Re = 0.56 \times 10^6 \text{ m}^{-1}$. Mean displacement vector $\mathbf{l}$ and drift velocity vector $\hat{\mathbf{u}}$ are set to zero on the inlet boundary and walls, and extrapolated on the exit respectively.

3 Simulation results

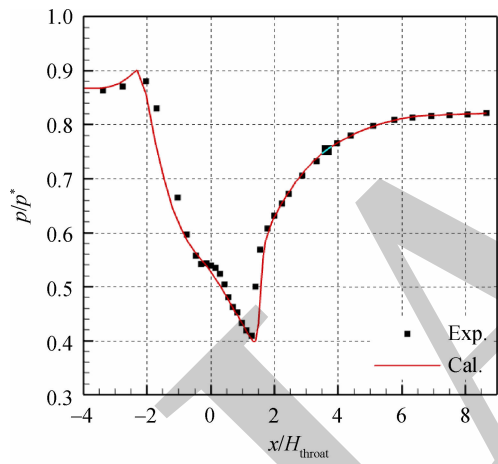
The pressure distributions along the bottom and top walls of the transonic diffuser are shown in Fig. 4(a) and Fig. 4(b). The ratio of exit static pressure to inflow total pressure is 0.82 in this situation. The ordinate in the Fig. 4 is the ratio of wall static pressure to total inlet pressure (p/p^*). 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.

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.



(a) Bottom wall pressure distributionsc



(b) Top wall pressure distributions

Fig. 4 Pressure distributions along diffuser walls in the weak interaction

In order to further study the separated flow caused by SWBLI, intensity of shockwave is increased to the ratio of 0.72 based on exit static pressure to inflow total pressure. The static pressure in the transonic diffuser flow field is shown in the contour plot in Fig. 5. It gradually decreases from the high pressure of the inlet to the low pressure at front of the normal shock, and then rises gradually until it is equal to the value of the exit. The pressure distributions along the bottom and top walls of the transonic diffuser displayed in Fig. 6(a) and Fig. 6(b) are compared with the available experimental data^[5]. It was found that the simulated results are

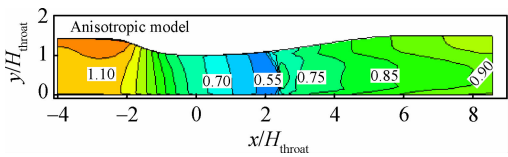
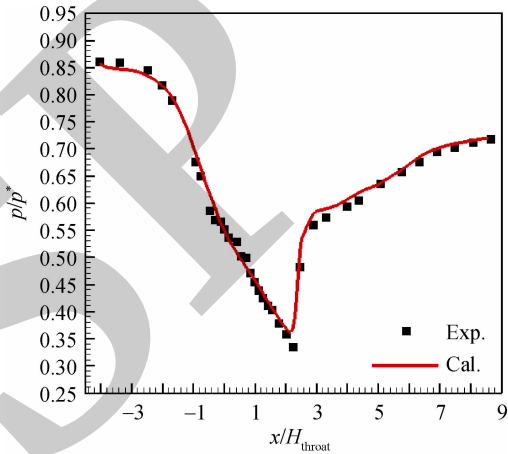
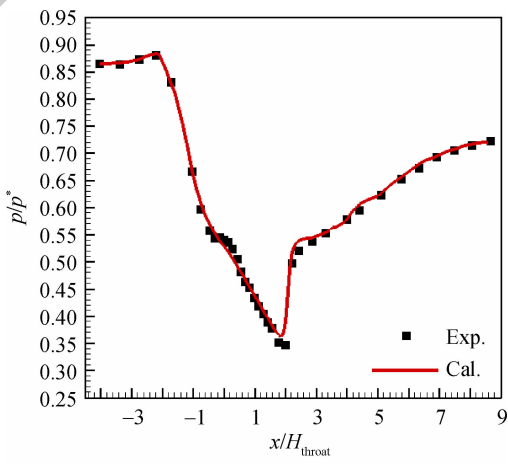


Fig. 5 Pressure contours in the flow field



(a) Bottom wall



(b) Top wall

Fig. 6 Pressure distribution on diffuser walls

Figure 7 shows the horizontal velocity profiles calculated at four axial locations at downstream of the shockwave at $x/H_{throat} = 2.88, 4.61, 6.34$ and 7.49 , respectively. H_{local} indicates the local diffuser height at different positions such as $x/H_{throat}, 2.88, 4.61$, etc. The corresponding experimental values are also given

in. The first two points, $x/H_{\text{throat}} = 2.88$ and 4.61, are located in the flow separation region and the other two are located behind the reattachment point according to the experiment. It is a challenge to simulate correct velocity profiles near the upper wall of the diffuser because of the highly intensified anisotropic turbulence in the

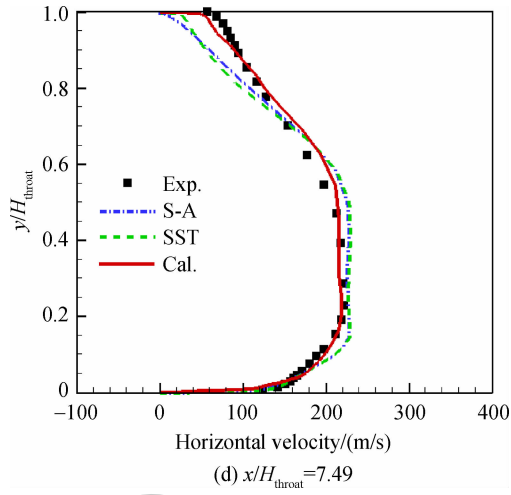
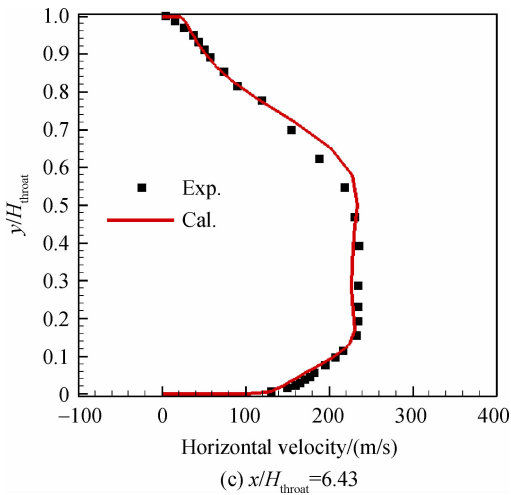
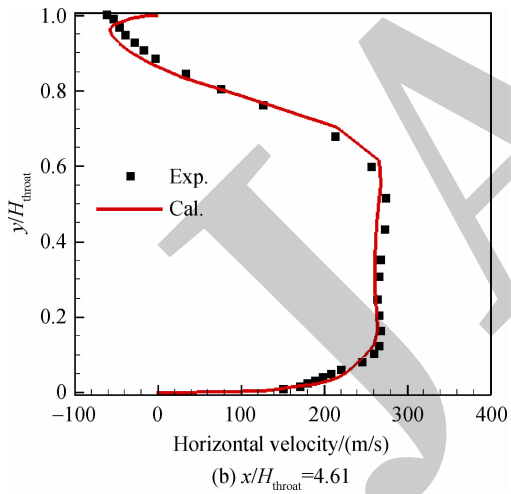
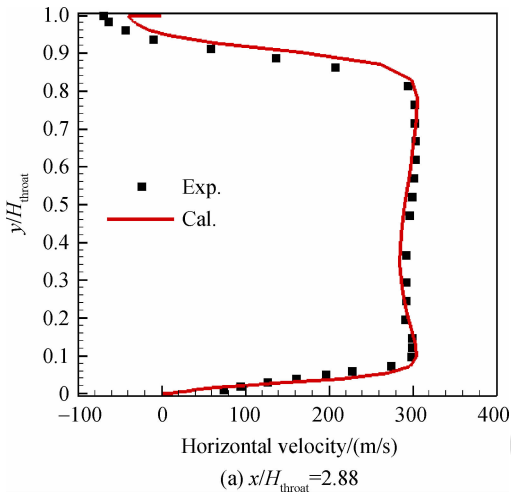


Fig. 7 Comparison of velocity profiles with experiment data at different locations

separation region. Calculated results by SST and S-A models are shown in the Fig. 7(d). The distribution of skin friction coefficients C_f on the top and bottom walls were also simulated. As shown in Fig. 8, two separation zones are evident due to a very small separation bubble on the bottom wall and a larger separation region on the top wall of the transonic diffuser. More specific, negative skin friction represents separation bubbles. The separation point and reattachment point also can be found in Fig. 8 through the zero values of skin friction.

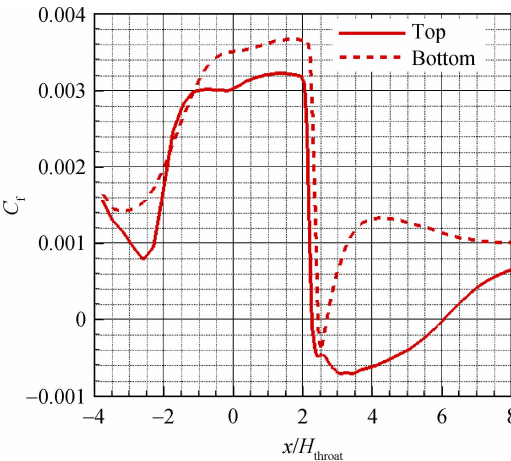


Fig. 8 Distribution of C_f calculated by anisotropic turbulence model

The separation location, reattachment location along the top wall and theseparation length Δx are listed in Table 1 which shows that the

separation and reattachment locations along the upper surface shift a little bit to downstream compared with the experimental data. Generally speaking, it is good agreement between the computation and experiment.

Table 1 Separation and reattachment locations		
Separation	Reattachment	Separationlength
x/H_{throat}	x/H_{throat}	$\Delta x/H_{\text{throat}}$
2.28	6.15	3.87
1.98	6.00	4.02

As shown in Fig. 9, simulated streamline vectors near the top wall of the transonic diffuser further confirms the separation region is around $x/H_{\text{throat}}=2.0$ and 6.0. It is reasonably agreed with the experimental data. In order to further investigate the performance of the anisotropic turbulence model in the reattachment region, the normal and shear Reynolds stress distributions are calculated as shown in Fig. 10. Abscissa represents Reynolds stress, $u'u'/u^2$ and $v'v'/u^2$ indicate normal Reynolds stress, $u'v'/u^2$ indicates shear Reynolds stress. As agreed with the experimental data, the flow is fully reattached at $x/H_{\text{throat}}=7.49$ after separation. Because of the flow separation, it is reasonable that the Reynolds stress in the upper reattachment region is much greater than that at the lower wall of diffuser.

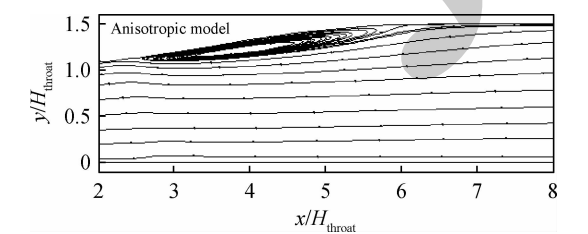


Fig. 9 Streamlines vector near the separation region behind shockwave

In the transonic or supersonic flows, the presence of a shockwave in a boundary layer may result in severe drawbacks because of the rapid retardation of the boundary layer flow and the propagation of shock in a multilayered structure^[24]. In this case, there is a sudden change of

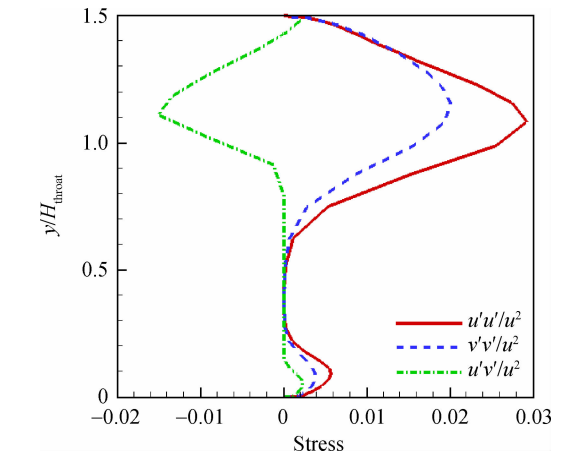


Fig. 10 Distribution of Reynolds stress at $x/H_{\text{throat}}=7.49$

velocity ahead and behind the shock wave, and the hypothesis of $\partial u/\partial x \ll \partial u/\partial y$ is no longer valid. The velocity gradient in x direction is not much smaller than that in the y direction. In other words, the magnitudes of $\partial u/\partial x$ and $\partial v/\partial y$ cannot be ignored^[25-26]. It is known that most turbulent flows in nature are anisotropic. The numerical study presented here shows that the drift flow defined in the anisotropic turbulence model can effectively construct the turbulence characteristic scales that are vital in modeling nonlinear anisotropic eddy viscosity tensor. This turbulence model is able to catch the essential behavior of the turbulent separation flow, especially in the separation region caused by SWBLI. The effect of Reynolds shear stress in 2-D flow on the nonlinear anisotropic eddy viscosity tensor can be analyzed by expanding the strain-stress relationship of Eq. (31), and Reynolds shear stress tensor can be written as

$$\tau_{xy}^{\text{turb}} = D_{31} \frac{\partial u}{\partial x} + D_{32} \frac{\partial v}{\partial y} + D_{33} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \tag{33}$$

which normal strain rate terms and the shear strain rate terms clearly demonstrate the interaction of the tension, extrusion and shearing in the turbulent flow. Furthermore, the nonlinear anisotropic eddy viscosity coefficients, such as D_{31} , D_{32} and D_{33} depend on turbulent fluctuation velocity vector $\hat{\mathbf{u}}$ obtained from Eq. (19) and turbulent fluctuation displacement vector $\mathbf{l}$ obtained from Eq. (20). The nonlinearity and anisotropy

of eddy viscosity tensor are of the decisive factor for the success of the computational results presented in this paper.

4 Conclusions

The transonic turbulent separation flow generated by strong shockwave-boundary layer interaction in the diffuser is highly anisotropic in nature because of the simultaneous presence of shear, strong inverse pressure gradient and curvature close to the upper solid surface. The anisotropic eddy viscosity tensor developed in the anisotropic turbulence model has the ability to describe the complex nature of the transonic turbulent separation flow. With the assumption that the maximum stress of the mean flow coincides with the direction of the turbulence characteristic length vector, the original 32 constitutive coefficients are reduced to 6 which greatly reduces the complexity of the modeling of eddy viscosity tensor. Thanks to the partial statistical average scheme developed in the anisotropic turbulence model, it is the first time that the characteristics of the fluctuation flow, including its length vector, can be quantitatively extracted from the momentum equations of the drift flow as well as the mechanical energy equation of the drift flow. The nonlinear anisotropic eddy viscosity can then be constructed using the conventional form shown in equation show in Eq. (26). As the interactions of tension, extrusion and shearing are considered in the Reynolds stress-strain constitutive relationship, the computational results, including pressure distribution and velocity profiles, are in good agreement with available experimental data.

Acknowledgements:

The authors would like to express their sincere appreciation to the Department of Civil Engineering at Florida Atlantic University and Key laboratory of aerodynamics and thermodynamics at Beijing University of Aeronautics and Astronautics.

References:

- [1] XI Chenchao, CHEN Weifang, GUO Zhongzhou, et al. Numerical simulation of implicit fully coupled SST and TNT turbulence models for high speed flows[J]. Journal of Aerospace Power, 2015, 30(4): 936-943. (in Chinese)
- [2] MOSTAFIN H, HOSSEIN A. Computational investigation of the flow structure over a pitching delta wing at supersonic speeds[J]. Journal of Aerospace Engineering, 2015, 230(7): 1334-1347.
- [3] WU Jianuo, LU Lipeng, LIU Yangwei. Performance evaluation of turbulence models for complex flows in sajbent diffuser[J]. Journal of Aerospace Power, 2017, 32(5): 1216-1223. (in Chinese)
- [4] FULVIO S, GILLES L, REYNALD B. PIV study on a shock-induced separation in a transonic flow[J]. Experiments in Fluids, 2012, 53(3): 815-827.
- [5] BOGAR T J, SAJBEN M, KROUTIL J C. Characteristic frequencies of transonic diffuser flow oscillations [J]. AIAA Journal, 1983, 21(9): 1232-1240.
- [6] SZUMOWSKI A P, OBERMEIRE F, MEIER G E A. Oscillation modes of laval nozzle flow [J]. Experiments in Fluids, 1995, 18(3): 145-152.
- [7] BRUCE P J K, BABINSKY H. Experimental study into the flow physics of three-dimensional shock control bumps [J]. Journal of Aircraft, 2012, 49(5): 1222-1233.
- [8] SOLTANI M R, YOUNSI S J, DALIRI A. Performance investigation of a supersonic air intake in the presence of the boundary layer suction[J]. Journal of Aerospace Engineering, 2014, 229(8): 1459-1509.
- [9] VLAHOSTERGIOS Z, YAKINTHOS K. Modelling the flow in a transonic diffuser with one reynolds-stress and two eddy-viscosity models[J]. Flow, Turbulence and Combustion, 2015, 94(3): 619-642.
- [10] LIU Yangwei, WU Jianuo, LU Lipeng. Performance of turbulence models for transonic flows in a diffuser[J]. Modern Physics Letters B, 2016, 30(25): 573-581.
- [11] HADIDOOABI M, ANSARIAN H. Computational investigation of the flow structure over a pitching delta wing at supersonic speeds[J]. Journal of Aerospace Engineering, 2015, 230(7): 1334-1347.
- [12] MAZAHARI K, KIANI K C, NEJATI A, et al. Optimization and analysis of shock wave/boundary layer interaction for drag reduction by shock control bump[J]. Aerospace Science and Technology, 2015, 42: 196-208
- [13] HYON K M, NOBUHIDE K. Prediction of anisotropy of the near-wall turbulence with an anisotropic low-Reynolds-number $k-\epsilon$ turbulence model[J]. Journal of Fluids Engineering, 1990, 112(4): 521-524.
- [14] LIEN F S, KALIZIN G. Computations of transonic flow with the v^2-f turbulence model[J]. International Journal of Heat and Fluid Flow, 2001, 22(1): 53-61.
- [15] GAO Ge, YONG Yan. On incompressible turbulent flow: partial average based theory and applications[J]. Journal of Hydraulic Research, 2005, 43(4): 399-407.
- [16] YAN Wenhui, XUE Ranran. Comparison of turbulence

models for numerical simulation of low-speed flow around NACA4412 airfoil[J]. *Acta Aeronautica et Astronautica Sinica*,2017,38(Suppl. 1):32-39. (in Chinese)

[17] YAN Wenhui, GAO Ge. Numerical study of turbulence models in sajben diffuser transonic separation flow[J]. *Journal of Propulsion Technology*,2016,37(9):1631-1637. (in Chinese)

[18] ROE P L. Approximate riemann solvers, parameter vectors,and difference schemes[J]. *Journal of Computational Physics*,1997,135(2):250-258.

[19] LEER B V. Towards the ultimate conservative difference scheme. v. a second-order sequel to godunov's method[J]. *Journal of Computational Physics*,1979,32(1):101-136.

[20] ANDERSON W K, THOMAS J L, LEER B V. Comparison of finite volume flux vector splittings for the Euler equations[J]. *AIAA Journal*,1986,24(9):1453-1460.

[21] JUNTASARO V, MARQUIS A J. Comparative study of flux-limiters based on must differencing scheme[J]. *International Journal of Computational Fluid Dynamics*,2004,18(7):569-576.

[22] JAMESON A. Numerical simulation of the euler equations by finite volume method using Runge-Kutta time stepping schemes[R]. *AIAA 81-1259*,1981.

[23] HIRSCH C. Numerical computation of internal and external flows;the fundamentals of computational fluid dynamics[M]. 2nd. ed. Burlington, US;Oxford Press,2007.

[24] JEAN D,DUSSAUGE J P. Some physical aspects of shock wave/boundary layer interactions[J]. *Shock Waves*,2009,19(6):453-468.

[25] DAVIDSON P A. Turbulence; an introduction for scientists and engineers[M]. New York;Oxford University Press,2004.

[26] POPE S B. Turbulent flows[M]. Cambridge, UK; Cambridge University Press,2000.

(编辑:张 雪)