Direct numerical simulation of a supersonic compression corner

Zhengyuan Wu, Zhenxun Gao*, Chongwen Jiang, Chunhian Lee

National Laboratory for Computation Fluid Dynamics, School of Aeronautic Science & Engineering, Beihang University, Beijing, 100191, China

*Email: gaozhenxun@buaa.edu.cn

Abstract. Direct numerical simulation (DNS) of a supersonic compression corner has been performed using a turbulent-library based inflow generation method, which means adding turbulence fluctuations from preexisted turbulence library files on the current inflow boundary. Illustration of application of such method is given. Physically turbulence boundary layer (TBL) can be obtained after a short distance ‘transition’ process. Qualitative results show that the current numerical simulation is able to obtain the fully developed turbulent boundary layer in the downstream and capture the shock structure well. The accuracy and reliability of current method is proved by the comparison of quantified results between DNS and experimental data. The interaction between the shock wave and the turbulent boundary layer near the corner has been analysed. Turbulent intensity is enhanced passed through the shock and the mean velocity profiles in the separation zone have suffered significant changes.

1. Introduction

With the rapid development of computer hardware capabilities and the continuous numerical simulation research of turbulent, high-fidelity turbulent simulation method has attracted more and more attention. High-fidelity turbulent simulation methods, such as direct numerical simulation (DNS) and large eddy simulation (LES), can obtain more accurate turbulent flow field information and finer turbulence structures, but on the other hand, the difficulty of its application increases as well. A significant problem which DNS or LES confronted is the dependence on the accurate initial turbulent field or the inflow turbulent boundary conditions [1-2]. Due to the space-time highly coupled nature of turbulent, the construction of initial turbulent field or inflow turbulent boundary is a challenge. For uniform shear turbulence, such as isotropic decaying turbulence flow, the turbulent initial field can be constructed based on the relevant turbulent theory. However, for wall-bounded turbulence flow such as turbulent boundary layer flow, it is much more difficult to accurately specify the inflow turbulent boundary. In order to carry out high-fidelity numerical simulation research of the turbulent boundary layer, several turbulent inflow generation methods have been proposed, including recycling-rescaling-based method, turbulence library-based method, transition-inducing approaches and synthetic turbulence generators and so on [3-4]. These methods have been widely used in DNS or LES research [5]. Despite considerable success it gained, each method has its own drawbacks [4]. Therefore, related researches are still in progress.

A turbulence library file-based inflow generation method is used in current research, which means to extract the turbulent fluctuation information from pre-existed files. And then the extracted fluctuation...
information will be added on the current inflow boundary. The overall process of such method is illustrated in Figure 1. The pre-existing files used in current work which contains realistic turbulent fluctuations are obtained from a delayed detached eddy simulation (DDES) study of supersonic cavity flow [6]. The above DDES study is able to generate real turbulent boundary layer on the downstream area while turbulent inflow information is not needed on the upstream inflow boundary.

The inflow fluctuations added by this method, which come from the real turbulent flow field, match the actual turbulence inflow boundary closer than the white noise disturbance. Thus, this method is more likely to trigger out truly fully developed turbulent boundary layer downstream. At the same time, this method is easy to apply and suitable for parallel computing. The feasibility and accuracy of this method have also been verified by DNS applications [7].

![Figure 1. Illustration of the library based turbulent inflow generation method.](image)

As shown in Figure 1, the above inflow turbulence generation method is applied to the DNS study of a typical supersonic compression corner flow. Generally, the compression corner flow consists of three main flow areas: the flat plate turbulent boundary layer upstream of the corner, the shock/turbulent boundary layer interaction (STBLI) area near the corner and the reattached turbulent boundary layer downstream of the corner. The first purpose of this paper is to verify the accuracy of the introduced turbulent inflow generation method, mainly by focusing on its performance on the upstream flat plate turbulent boundary layer. Another motivation is to enhance our understanding of the shock/turbulent boundary layer interaction phenomena by the DNS approach.

The rest of this paper is organized as follows: chapter 2 gives an introduction of the governing equations and the numerical methods used; chapter 3 introduces the basic concepts and details of the turbulent library file-based generation method; chapter 4 presents the analysis and discussion of the DNS study supersonic compression corner, and summary is shown in chapter 5.

2. Numerical method

This chapter introduces the flow governing equations and corresponding numerical methods used. Meanwhile, a brief introduction of the logarithmic-law theory of turbulent boundary layer is presented.

2.1. Governing equation

For calorically perfect gas, the instantaneous compressible N-S equations in the Cartesian coordinate system can be expressed as:
\[ \begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} &= 0 \\
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j + \delta_{ij} p)}{\partial x_j} &= \frac{\partial \tau_{ij}}{\partial x_j} \quad (i=1,2,3)
\end{align*} \] (1)

Variables \( \rho, u_i, p \) and \( T \) represent the density, velocity, pressure and static temperature respectively; \( e \) and \( h \) stand for the internal energy and enthalpy per unit mass. \( \tau_{ij} \) is the molecular stress tensor, written as

\[ \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_j} \delta_{ij} \] (2)

The viscosity coefficient \( \mu \) is calculated by the Sutherland formula. \( q_i \) is the convective heat which can be computed using Fourier heat conduction law, and the heat transfer coefficient \( \lambda \) is obtained according to the Prandtl constant \( Pr \), that is, \( \lambda = \frac{c \mu}{Pr} \). \( \rho, \rho \) and \( T \) satisfy the ideal gas state equation \( p = \rho RT \).

2.2. Log-Law theory of turbulent boundary layer

The so-called log-law theory is one of the classic theories of the turbulent boundary layer. To better illustrate the theory, we first introduce the definition of relevant quantities. The wall friction velocity is defined as \( u_r = (\tau_0/\rho_w)^{1/2} \). Based on \( u_r \), other dimensionless quantities can be defined such as \( U^* = U/u_r \), and the dimensionless distance \( y^* = y/u_r \). The log-law theory indicates that there is a region in the turbulent boundary layer where \( U^* = 1/\ln(y^*) + C \). \( \kappa \) is the Karman constant and \( C \) is the constant coefficient. This relationship is one of the criteria for verifying the accuracy of DNS studies.

3. Turbulence library file-based inflow generation method

Generally, the inflow boundary conditions for a DNS study include two parts: mean profile information and turbulent fluctuation information, wherein the mean profile information can be obtained by RANS simulation. The key is to get realistic inflow turbulent information. As discussed before, the turbulence library file-based method is adopted to generate inflow turbulent data in this work. Bilinear interpolation of grid point coordinates between the library file and the current case’s inflow boundary is applied to get the corresponding turbulent fluctuation value \( Q' \). Further treatments of inflow velocity fluctuations are carried out based on equation (3), whose details can be found in the work of Martin [7]. After the inflow velocity fluctuations \( u', v', w' \) have been obtained, the corresponding \( T' \) and \( \rho' \) are then calculated according to the strong Reynolds analogy (Morkovin 1962) according to equation (4).

\[ \sqrt{\frac{\rho(y^*)}{\rho_w}} \frac{V(x^*, y^*, z^*)}{u_r} \bigg|_{\text{Mod}1} = \sqrt{\frac{\rho(y^*)}{\rho_w}} \frac{V(x^*, y^*, z^*)}{u_r} \bigg|_{\text{Mod}2} \] (3)

\[ T' = -b(\gamma - 1)M^2 \frac{u'}{u} \frac{T}{\rho} \] (4)

Now the inflow boundary condition at each time step reads

\[ Q = \tilde{Q} + \tilde{Q}', \tilde{Q} = u, v, w, \rho, T \] (5)
4. Results and Discussions

4.1. Flow condition and numerical setup

The compression corner configuration of Ringuette et al. [8] is studied. The freestream flow parameters are $M_\infty=2.92$, $P_\infty=2410\text{Pa}$, $T_\infty=109.25\text{K}$, $\rho_\infty=0.0768\text{kg/m}^3$, $Re/\text{m}=6.15\times10^6$. Adiabatic wall boundary condition is assumed. Multi-block structure mesh is generated. The two-dimensional computation mesh is shown in Figure 2. Computation grids is distributed uniformly in the spanwise (Z) direction with $Az = 0.105\text{mm}$ ($Az^+=6$) and the lengths of spanwise domain is $2\delta$. The grid spacing in X direction in the upstream area is set to be $Ax=0.125\text{mm}$ ($Ax^+=7$), which decrease to smaller value near the corner. Cells are clustered in the near wall region using a hyperbolic tangent function and an initial grid spacing $\Delta y$ of roughly $3.5\times10^6\text{m}$, in which case a maximum $y^+$ of the initial grid of approximately 0.2 is obtained. The total amount of grid points is 30 million.

The simulation is performed using ACANS, a finite difference code developed by Gao et al. [9]. No extra variables will be defined for the current DNS research. Global time step of $5\times10^3\text{s}$ is used for time advance. As for spatial discretization, Steger-Warming flux splitting is employed for the advective fluxes, combined with a seventh-order WENO interpolation approach to increase the spatial accuracy. Viscous terms were discretized by forth-order centered schemes. The characteristic time of the flow is defined as $T_{ref}=L/c$. The statistics process begins when the flow has reached dynamic balance in a statistical sense, which is after calculation of $5T_{ref}$ physical time, and ends when the statistical mean and root mean square (RMS) results only change slightly as advance time increase.

![Figure 2. Spanwise centerplane of the computation mesh, not all grid point is shown.](image)

4.2. Basic results

The instantaneous temperature contour in XY plane is presented in Figure 3. After a ‘transition’ process, the flow developed to a realistic TBL. This conclusion can also be inferred from the skin friction coefficient ($C_f$) distribution along the streamwise direction as shown in Figure 4. $C_f$ tends to decrease first and then increase steadily to the peak value, recovering to the actual level of a balanced TBL eventually. The flow separated near the corner due to the anti-pressure gradient resulted from the oblique shock and a separation bubble formed. Variation of the mean and fluctuating properties throughout the separation bubble is studied. The interaction between shock and TBL and the role separation bubble plays will be analysed in the section 4.4.

In order to better identify the vortex structures, the Q criterion of vortex identification is used. Instantaneous iso-surface of Q criteria with a value of $1.6\times10^8\text{s}^{-2}$ is shown in Figure 5, and the iso-surface has been re-rendered based on the local pressure values. Three-dimensional vortex structures in different regions can be observed. Small scale vortex structures are distributed in recirculation zone. Large eddies formed in the reattachment boundary and detached larger vortex structure can be found downstream.

![Figure 3.](image)

![Figure 4.](image)

![Figure 5.](image)
Figure 3. Instantaneous temperature contour in X-Y plane (with a constant Z value).

Figure 4. $C_f$ distribution along the streamwise direction.

Figure 5. Instantaneous iso-surface contour of Q criteria.

Figure 6 shows the instantaneous velocity contour in the wall-parallel X-Z plane (with a constant value of $y^+$ equals to 20), from which a typical high-low-speed near-wall streaks can be clearly observed. The high-low-speed near-wall streaks refer to the staggered distribution of striped low-speed region and striped high-speed region in the spanwise direction. This is also one of the typical features of the turbulent boundary layer. Further downstream, after flow separation occurs, the typical near-wall streaks disappear in the separation zone.
4.3. Discussion of the upstream turbulent boundary layer

The discrepancies of boundary layers thick $\delta$ and other parameters between DNS and experimental in the upstream TBL are small. Mean streamwise velocity profile along boundary layer matches the experimental result well as shown in Figure 7 (a). Figure 7 (b) plots the relation between Van Driest transformed velocity $U^+\text{-VDT}$ and $y^+$. The log-law region can be well described by $(1/\kappa) \log y^+ + C$, with $\kappa=0.41$ and $C=5.5$. Besides, $U^+\text{-VDT}$ result in the sub-viscous layer stratifies the theory of $u^+=y^+$ very well. As for the mean temperature–velocity relationship, the Crocco’s relation of equation (6) is verified in Figure 8.

$$\frac{T}{T_\theta} = \frac{T_w}{T_\theta} + \frac{T_r - T_w}{T_\theta} \left( \frac{u}{u_\delta} \right) + \frac{T_r - T_w}{T_\theta} \left( \frac{u}{u_\delta} \right)^2$$

*Figure 6. Instantaneous velocity contour in the wall-parallel X-Z plane.*

*Figure 7. Mean streamwise velocity results in the upstream TBL.*

*Figure 8. Crocco’s relation of mean temperature-velocity.*

Flow parameters such as the boundary layer thickness $\delta$, the Reynolds number $Re_\theta=\rho u_\delta \theta / \mu_\delta$ and the friction coefficient $C_f$ and so on at $x/\delta=6$ are compared with the experimental results as shown in Table 1. It can be seen that the results of the current DNS calculation match the experimental data well.
Table 1. Comparison of flow properties at a specific station of the boundary layer

|   | M  | Reθ | δ, mm | δ*, mm | θ, mm | H=δ*/θ | Cf×1000 | uτ |
|---|----|-----|-------|--------|--------|--------|----------|-----|
| Experiment[8] | 2.9 | 2400 | 6.7 | 2.36   | 0.43   | 5.49   | 2.17     | 32.9|
| DNS-Current   | 2.9 | 2450 | 6.7 | 2.6    | 0.48   | 5.41   | 2.13     | 30.0|
| DNS[10]       | 2.9 | 2300 | 6.4 | 1.8    | 0.38   | 4.74   | 2.17     | 34.0|

The variations of budget terms of the turbulent kinetic transport equation along wall-normal direction in the boundary layer at x/δ=-6 is given in Figure 9. Obviously, viscous dissipation term, the viscous diffusion term and the production term dominate the budget process, seeing from Figure 9. Both the viscous dissipation term and the viscous diffusion term get its maximum value at the wall and the decrease away from the wall. The production term reaches its peak value at y+~20, and then decrease gradually towards the out edge of boundary layer. The value of the turbulent transport term is relatively small. The values of the pressure dilatation term and the pressure diffusion term near the wall are not negligible, but their values decay rapidly as y+ increases, and the effects of these two terms can be offside near the wall. The value of the pressure work term is small and is not shown in the figure.

Figure 9. Budget terms of the turbulent kinetic transport equation.

4.4. Analysis of the shock/turbulent boundary layer interaction

Seeing from the mean pressure contour given in Figure 10, in the downstream region, the presence of the shock wave produces significant inverse pressure gradient along the flow direction, thereby gradually reducing the flow velocity, causing flow separation ultimately, producing a closed separation bubble near the corner. Variations of the velocity profiles along the wall-normal direction at different streamwise directions include station upstream of the separation zone, station in the separation zone and station downstream of the separation bubble are presented in Figure 11, which shows the some tendency that the velocity decreases gradually until reaches the separation critical point due to the inverse pressure gradient, and then recovers to the normal profiles in the reattached boundary layer. After the shock wave, the turbulence intensity in the separation zone is significantly enhanced as shown in Figure 12. Besides, Figure 13 indicates that the u+~y+ distribution deviates from the classical log-law theory in the separation zone.
Figure 10. Mean pressure contour.

Figure 11. Mean velocity profiles at different streamwise stations.

Figure 12. Turbulent kinetic energy contour.

Figure 13. $u^+$ distribution of different streamwise stations.
To further demonstrate the ability of current DNS study to identify fine turbulence structures, Figure 14 shows the instantaneous density gradient contour in the X-Y plane. The small shocklet structures in the upstream turbulent boundary layer can be clearly observed. Meanwhile, the $\lambda$ shape shock structure formed due to the STBLI is well captured as marked in Figure 14. Besides, it can also be found that the density gradient is amplified through the shock. The wall-pressure signals at different locations are presented in Figure 15, which shows that the wall pressure fluctuation is significantly enhanced in the separation zone.

![Figure 14. Instantaneous density gradient magnitude contour in X-Y plane.](image)

![Figure 15. Wall-pressure signals at different streamwise locations for the DNS.](image)

5. Conclusions
In this paper, DNS study of supersonic compression corner has been performed using a turbulence library file-based inflow turbulence generation method. Qualitative results such as temperature contour, Q criterion and near-wall high-low speed streaks show that the current numerical simulation is able to obtain the fully developed turbulent boundary layer in the downstream and capture the shock structure well. Then, comparisons of surface friction coefficient $C_f$, boundary layer thickness and wall friction velocity and so on between DNS and experimental data are given, shows that corresponding DNS technique applied to the supersonic turbulent boundary layer is accurate and reliable. Finally, the interaction between the shock wave and the turbulent boundary layer near the corner area is analysed. First of all, it can be seen that the resolution of computational mesh and the numerical scheme adopted is sufficient to capture fine vortex structures in the turbulent boundary layer and the accurate shock structure. Secondly, the inverse pressure gradient along the streamwise direction caused by STBLI is strong enough to trigger outflow separation, producing a closed separation bubble. The mean velocity
profiles in the separation zone have suffered significant changes compared to those in the upstream boundary layer. Consequently, the $u^{+}$-$y^{+}$ distribution curves in the separation zone deviate from the logarithm-law theory. Meanwhile, it is found that the turbulent intensity is enhanced passed through the shock, and the wall pressure fluctuation experience rapid increase after the shock. The above conclusions can enhance our understanding of STBLI phenomenon.

Acknowledgments
This research is supported by the National Natural Science Foundation of China (NSFC, Grant Nos. 91641123, 11872094) and the Foundation for Innovative Research Groups of the National Natural Science Foundation of China through grant 11721202.

References
[1] Wu X H 2017 Inflow Turbulence Generation Methods Annu. Rev. Fluid Mech 49 2349
[2] Tabor G and Baba-Ahmadi M 2010 Inlet Conditions for Large Eddy Simulation: A Review Comput. Fluids. 39 553567
[3] Lund T S, Wu X, and Squires K D 1998 Generation of Turbulent Inflow Data for Spatially- Developing Boundary Layer Simulations J. Comput. Phys. 140 233258
[4] Dhamankar N S and Blaisdell G A2017 Overview of Turbulent Inflow Boundary Conditions for Large-Eddy Simulations AIAA J. 56 1317-1334
[5] Lee S, Lele S K and Moin P 1992 Simulation of spatially evolving turbulence and the applicability of Taylor’s hypothesis in compressible flow Phys. Fluids 4 15211530
[6] Wu Z Y, Gao Z X, Jiang C W and Lee C H 2019 An in-depth numerical investigation of a supersonic cavity-ramp flow with DDES method Aerosp. Sci. Technol. 89 253263
[7] Martin M P 2007 Direct numerical simulation of hypersonic turbulent boundary layers. Part 1. Initialization and comparison with experiments J. Fluid Mech. 570 347364
[8] Ringuette M J, Bookey P, Wyckham C and Smits A J 2009 Experimental Study of a Mach 3 Compression Ramp Interaction at $Re_{θ}=2400$ AIAA J. 47 373385
[9] Gao Z X, Jiang C W, Pan X W and Lee C H 2015 Combustion Heat-Release Effects on Supersonic Compressible Turbulent Boundary Layers AIAA J. 53 19491968
[10] Wu M W and Martin M P 2007 Direct Numerical Simulation of Supersonic Turbulent Boundary Layer over a Compression Ramp AIAA J. 45 879889