The First Chinese
International Conference on CFD
Mianyang, China, July 15-16, 2015

CICCFD1-2015-xxxx

DNS of Transitional and Turbulent Flow Simulations

Yufeng Yao[1,2] and Yao Zheng[1]

[1]School of Aeronautics and Astronautics, Zhejiang University, Zhejiang 310027, China
[2]Faculty of Environment and Technique, University of the West of England, Bristol BS16 1GQ, UK

Corresponding author: Yufeng.Yao@uwe.ac.uk

Abstract: An in-house direct numerical simulation (DNS) code SBLI has been developed and used extensively for high-performance computation (HPC) of transitional and turbulent flows, most related to shock/boundary-layer interaction problems. The code has also proved to be very adaptable with its variants being used for a wide range of turbulent flows, including transonic cavity flows, turbulent spot interactions and separation bubbles on an airfoil at incidence. Recently the code has been re-engineered to include modern software engineering techniques that allows it to achieve better parallel scalability on various HPC platforms.

Keywords: Direct Numerical Simulation, Numerical Algorithms, Turbulent Flow.

1 Introduction

This paper presents a recent advancement on developing an in-house direct numerical simulation (DNS) code (denoted as SBLI thereafter). The code was primarily designed for simulation of shock wave and turbulent boundary layer interactions [1, 2], and later the code has also proved to be very adaptable with its variants being used for a wide range of turbulent flows, including transonic cavity flows, turbulent spot interactions and separation bubbles on an airfoil at incidence.

2 Code Description and Case Study

The SBLI code solves the three-dimensional compressible Navier-Stokes equations for the Cartesian velocity components on curvilinear grids. High-order central differencing schemes (4th-order and 6th-order) are adopted for the spatial discretizations [3] and a compact low-storage three-step Runge-Kutta algorithm (3rd-order) for the time advancement [4]. A numerical treatment based on the entropy splitting developed by Yee, Sandham and co-workers [3, 5] has been used to improve the computational stability of previous attempts at combining high accuracy codes for direct numerical simulation with those codes capable of shock capturing. When the flow contains discontinuities such as shock waves, the TVD scheme with the artificial compression method (ACM) of Yee et al. [3] is applied. The code has been parallelized using the MPI library. It is widely understood that numerical simulation of shock-wave interaction with turbulence has additional difficulties as discussed by Yee et al. [3] previously. The fundamental problem is that the best numerical methods for turbulence are extremely inefficient for shock flows, while the best methods for shock waves are much too dissipative for accurate resolution of turbulence. The technique of using TVD limiter with artificial compression method of Yee et al. [3] has been proved to be one of best solutions and has been implemented into the SBLI code. To demonstrate the code, a configuration of an oblique shock impinging onto a spatially developing boundary-layer along flat-plate (see Figure 1a) has been studied (also known as Kazter case [6]), with a focus on the Mach number effects on unstable disturbances developing in the vicinity of a separation bubble [7]. The study included the early stages of breakdown to turbulence after the separation bubble. Figure 1b gives the skin friction coefficient and wall pressure distributions, where the dashed line is from Wasistho [8] and the square symbols are the experimental data from Hakkinen [9].

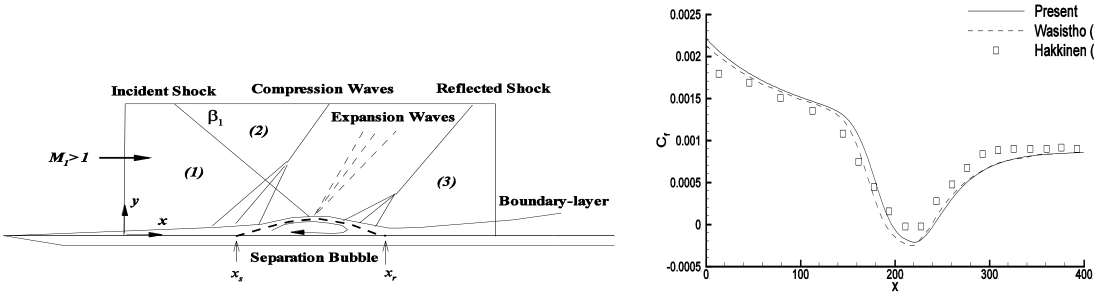

Figure 1: Schematic view of an oblique shock wave impinging onto a spatially-developing boundary layer on a flat plate.

3 Conclusion and Future Work

The development of an in-house DNS code (SBLI) for high-performance computation of turbulent flows has been described. Recent re-engineering work has brought some newly developed numerical features and treatments from several variants of the code strand back together in a unique version. The code has gone through stringent and systematic verification and validation processes. The code has successfully applied to variety test cases from low-Re channel to moderate to high-Re transitional and turbulent flows including airfoil stall, transonic bump, shock impinging, etc. A full-3D curvilinear version of the code is also developed that can be used for complex flow problems.

References

[1] Y.F. Yao and N.D. Sandham. Direct numerical simulation of turbulent flow over a bump with shock/boundary-layer interactions. In Rodi, W. and Furyo, N. (eds.), Proceedings of Engineering Turbulence Modelling and Measurements, Elsevier, pp. 677-686, 2002.

[2] N.D. Sandham, Y.F. Yao and A.A. Lawal. Large-eddy simulation of transonic turbulent flow over a bump. Int. J. Heat and Fluid Flow, 24(4): 584-595, 2003.

[3] H.C. Yee, N.D. Sandham and M.J. Djomehri. Low-dissipative high-order shock-capturing methods using characteristic-based filters. J. Comp. Phys., 150(1): 199-238, 1999.

[4] A.A. Wray. Very low storage time-advancement schemes. Internal Report, NASA Ames Research Centre, Moffett Field, California, 1986.

[5] N.D. Sandham, Q. Li and H.C. Yee. Entropy splitting for high-order numerical simulation of compressible turbulence. J. Comp. Phys., 178(2): 307-322, 2002.

[6] Katzer, E., On the lengthscales of laminar shock/boundary-layer interaction. J. Fluid Mech., 206: 477-496, 1989.

[7] Y.F. Yao, K.L. Narasimhan, N.D. Sandham and G.T. Roberts. The effect of Mach number on unstable disturbance in shock/boundary layer interactions. Physics of Fluids, 19(5), 054104, 2007

[8] B. Wasistho. Spatial direct numerical simulation of compressible boundary layer flow. Ph.D. Dissertation, University of Twente, Netherlands, 1998.

[9] R.J. Hakkinen, I. Greber, L. Trilling and S.S. Abarbanel. The interaction of an oblique shock wave with a laminar boundary layer. NASA Memo 2-18-59W, 1959