Tenth Fluid Dynamics, Nov. 2006
Yazd University, Mech. Engg. Dept.

UNO SCHEME FOR STEADY AND TRANSIENT COMPRESSIBLE FLOW IN PRESSURE-BASED METHOD

M.H. Djavareshkian
Assistant Professor

V. Enjilela
M. Sc. Student

Faculty of Mechanical Engineering Tabriz University, Iran
E-mail: djavaresh@Tabrizu.ac.ir

ABSTRACT

In this paper, a scheme based on Uniformally Nonoscillating (UNO) has been developed into an implicit finite volume procedure, which uses pressure as a working variable. The boundedness criterion is determined from UNO schemes. The numerical process is used for solution of Euler equations on a nonorthogonal mesh with collocated finite volume formulation. The developed scheme is applied to the computation of steady supersonic flow over a bump-in-channel geometry as well as to the transient shock-tube problem. The results of the UNO scheme are compared with analytical and other computations published in literature.

Keywords: Finite Volume, UNO, Transient, Pressure-base

1. INTRODUCTION

In computational fluid dynamics (CFD), great research efforts have been devoted to the development of accurate and efficient numerical algorithms suitable for solving flow in the various Reynolds and Mach number regimes. Several attempts have been made by incompressible fluid flow numerical researchers, towards the unification of numerical methods developed for incompressible and compressible flows. The main goal consists in the development of methods for computation of flows at all Mach numbers by extending the pressure-correction formulation to ensure shock-capturing properties. Leonard [1] has generalized the formulation of the high-resolution flux limiter schemes using what is called the normalized variable formulation (NVF). Many schemes based on the NVF has been developed in pressure-based method, for example SMART scheme[2], SFCD scheme[3], SOUCUP scheme[4], STOIC scheme[5], SBIC scheme base on variable and flux limiter[6,7]. Issa and Javareshkian [8] implemented a high resolution TVD scheme with characteristic-variables-based flux limiters into a pressure-based finite volume method. Batten et al.[9] utilized the TVD approach and adopted a time marching technique. The TVD and NVD schemes do not have oscillation at discontinuities because they are switched to first order scheme. The ENO scheme is presented for the first time by Harten *et. al* [10]. The ENO scheme do not have a Gibbs-like phenomenon O(1) at discontinuities, yet they may occasionally produce small spurious oscillations on the level O(h^r) of the truncation error. Kobayashi and Pereira [11] introduced an ENO scheme into pressure-correction solution procedures for the flux calculation, which they incorporated into a steady-state solution method.

The objection of this paper is to extend an Uniformally Nonoscillating (UNO) scheme in pressure-based method. The developed scheme is applied to the computation of steady supersonic flow over a bump-in-channel geometry as well as to the transient shock-tube problem. The results of the UNO scheme are compared with other computations published in literatures.

2. GOVERNING EQUATION

The basic equations, which describe conservation of mass, momentum, and scalar quantities, can be expressed in Cartesian tensor form as:

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

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j - T_{ij})}{\partial x_j}=S_i^u \qquad (2)$$

$$\frac{\partial(\rho \phi)}{\partial t}+\frac{\partial(\rho u_j \phi - q_j)}{\partial x_j}=S^\phi \qquad (3)$$

The stress tensor and scalar flux vector are usually expressed in terms of basic dependent variables. The stress tensor for a Newtonian fluid is:

$$T_{ij} = -p\delta_{ij} - \frac{2}{3}\mu \frac{\partial u_k}{\partial x_k}\delta_{ij} + \mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \qquad (4)$$

The scalar flux vector is usually given by the Fourier-type law:

$$q_j = \Gamma_\phi \left(\frac{\partial \phi}{\partial x_j}\right) \qquad (5)$$

3. DISCRETIZATION

The discretizations of the above differential equations are carried out using a finite-volume approach. First, the solution domain is divided into a finite number of discrete volumes or “cells “, where all variables are stored at their geometric centers (see e.g. Fig.1). The equations are then integrated over all the control volumes by using the Gaussian theorem. The development of the discrete expressions to be presented is affected with reference to only one face of the control volume, namely, e, for the sake of brevity. For any variable ϕ (which may now also stand for the velocity components), the result of the integration yields:

$$(6)\quad \frac{\delta \upsilon}{\delta t}[(\rho\phi)_p^{n+1}-(\rho\phi)_p^n]+I_e-I_w+I_n-I_s=S_\phi\,\delta\upsilon$$

Where I is the combined cell-face convection I^c and diffusion I^D fluxes. The diffusion flux is approximated by central differences and can be written for cell-face *e* of the control volume in Fig.(1) as:

$$I_e^D = D_e(\phi_p - \phi_E) - S_e^\phi \qquad (7)$$

Where S_e^ϕ stands for cross derivative arising from mesh non-orthogonality. The discretization of the convective flux, however, requires special attention and is the subject of the various schemes developed. A representation of the convective flux for cell-face ‘e’ is:

$$I_e^c = (\rho V.A)_e\phi_e = F_e\phi_e \qquad (8)$$

The expression for the I_e^c by the UNO scheme is dealt with later. The discretized equations resulting from each approximation take the form:

$$a_p.\phi_p = \sum_{m=E,W,N,S} a_m.\phi_m + S'_\phi \qquad (9)$$

Where *a*’s are the convection-diffusion coefficients.

4. CONVECTIVE FLUXES

The expression for the mass, momentum and energy fluxes is determined by the UNO scheme used for interpolation from nodes at the neighbouring points. The expression can be written for cell face "e" as:

$$I_e^c = \frac{1}{2}[I_E^c + I_P^c + R_e \phi_e] \tag{10}$$

Where $R_e \phi_e$ is a dissipation term, based on the characteristic field decomposition of the flux difference. The quantity R_e stands for the right eigenvector matrix, while ϕ_e is a vector containing the components of the anti-diffusive flux terms. According to Yee et al. (1985), a spatially second-order upwind formula for the components of ϕ_e is given by:

$$\phi_e^l = \frac{1}{2}\psi(a_{i+1/2}^l)[g_{i+1}^l + g_i^l] - \psi(a_{i+1/2}^l + \gamma_{i+1/2}^l)\alpha_{i+1/2}^l \tag{11}$$

The eiganvalues of the Jacobian matrix are denoted by a. The spatial increments of the characteristic variables α are obtained by:

$$\alpha_e^l = R_e^l (u_E^l - u_P^l) \tag{12}$$

For γ one can take (Yee et al.,1985):

$$\gamma_e^l = \frac{1}{2}\psi(a_e^l)\begin{cases} \dfrac{g_E^l - g_P^l}{\alpha_e^l} & \alpha_e^l \neq 0 \\ 0 & \alpha_e^l = 0 \end{cases} \tag{13}$$

The function ψ is required to prevent non-physical solutions such as expansion shocks and introduces a small amount of viscosity. Following the suggestion of Harten and Hyman (1983), it is taken as:

$$\begin{aligned} \psi(z) &= \frac{z^2}{4\varepsilon} + \varepsilon \quad for \quad |z| < 2\varepsilon \\ &= |z| \quad for \quad |z| \geq 2\varepsilon \end{aligned} \tag{14}$$

Where ε is an arbitrarily small number. The most important factor in Eq.(11) is the flux-limiter , g , which determines the accuracy and TVD-property of the scheme. This factor may be defied in any way chosen. For the present work, the $MINMOD$ limiter due to Harten (1983) is used; thus,

$$\begin{aligned} g_P &= MINMOD(s_P^+, s_P^-) \\ s_P^+ &= \alpha_e - \tfrac{1}{2}\beta_e \\ s_P^- &= \alpha_w + \tfrac{1}{2}\beta_w \\ \beta_e &= MINMOD(\beta_P, \beta_E) \\ \beta_w &= MINMOD(\beta_W, \beta_P) \\ \beta_W &= \alpha_w - \alpha_{ww} \\ \beta_P &= \alpha_e - \alpha_w \\ \beta_E &= \alpha_{ee} - \alpha_e \end{aligned} \tag{15}$$

where

$$MINMOD(x, y) = sign(x).Max(0, \min[|x|, y\, sign(x)] \tag{16}$$

5. SOLUTION ALGORITHM

Most contemporary pressure-based methods employ a sequential iteration technique in which the different conservation equations are solved one after another. The common approach taken in enforcing continuity is by combining the equation for continuity with those of momentum to derive an equation for pressure or pressure-correction. The PISO algorithm is used in this work.

6. RESULTS

Both two-dimensional steady and one-dimensional transient flows are computed and the results are compared either with existing numerical solutions obtained by others or with the analytic solutions when they are available. The test cases chosen are the normal benchmarks to which methods such as the one presented here are applied. The first case is that of the classical shock tube problem and the second is the bump-in-channel case.

Fig. 2 shows the spatial distribution of velocity, density, Mach number and pressure ratio, along the shock tube at a given instant in time in a shock-tube for an initial pressure of 10. The results of computation on a mesh of 100 nodes are compared with the analytic solution. It can be seen that the shock is sharply captured, and the contact discontinuity is better resolved and oscillation is not relatively produced for the UNO scheme.

The second case is supersonic flow over 4% thick bumps on a channel wall. For this test, at the inlet of the domain all flow variables are specified. At the outlet, all the flow variables are given by extrapolation for supersonic velocity. Slip boundary conditions are used on the upper and lower walls. A non-uniform grid of 90×30 in which the grid lines are closely packed in and near the bump region is shown in Fig. 3.

The results of supersonic flow with inlet Mach number equal to 1.4 over a 4% thick bump are shown in Fig. 4. The Mach number and pressure ratio distribution on the upper and lower surfaces for present scheme are compared with the TVD [8] prediction. The agreement between the two solutions is remarkable, thus once again verifying the validity of the UNO scheme in pressure-based algorithm.

7. CONCLUSION

A pressure-based implicit procedure is described. It incorporates an UNO scheme. The developed scheme is applied to both transient and steady state flows and the results are in well agreements with other schemes.

8. REFERENCES

[1] Leonard B.P., “Simple High-Accuracy Resolution Program for convective modeling of discontinuities”, *International Journal for Numerical Methods in Fluids*, Vol.8, pp.1291-1318, (1988)

[2]. Gaskell P.H. & Lau A.K.C., “Curvature-Compensated convective Transport: SMART, a new boundedness-preserving transport algorithm”, *International Journal for Numerical Methods in Fluid,* Vol.8, pp.617-641, (1988)

[3] H. Ziman, A computer Prediction of Chemically Reacting Flows in Strirred Tanks, *PhD thesis*, University of London, (1990)

[4] Zhu J. & Rodi W., Low Dispersion and Bounded Convection Scheme, *Computer Methods in Applied Mechanics and Engineering*, Vol. 92, pp.87-96, 1991

[5] Darwish M.S., A New High-Resolution Scheme based on the normalize variable Formulation, *Numerical Heat Transfer,* Part B, Vol.24, pp.353-371, (1993)

[6] Djavareshkian, M. H. A New NVD Scheme in Pressure-Based Finite-Volume Methods, *14th Australasian Fluid Mechanics conference, Adelaide University, Adelaide*, 10-14 December 2001, pp.339-342, Australia

[7] Widermann A. and Iwamoto J., “A Multigrid TVD Type schems for computing inviscid and viscous flows”, *Computers Fluids* Vol.23,No.5,pp.711-735, (1994)

[8] Issa R.I. and Javareshkian M.H., “Pressure-Based Compressible Calculation Method Utilizing Total Variation Diminishing Schemes”, *AIAA Journal*, Vol.36, No.9, pp.16521657, (1998)

[9] Batten P., Lien, F. S. and Leschziner M. A., “A Positivity-Preserving Pressure-Correction Method”, *Proceedings of 15th International Conf. On Numerical Methods in Fluid Dynamics*, Monterey, CA, 1996

[10] Harten, A., Engquist, B., Osher,S., Chakravarthy, S.R., “Uniformly High Order Accurate Essentially Non-oscillatory schemes,III” ,*J. Computational Physics*, Vol.71, No.2, pp.231-303, (1987)

[11] Kobyashi M. H., and Pereira J. C. F., “Characteristic-Based Pressure Correction at all Speed”, *AIAA Journal*, Vol.34, No.2, pp.272-280, (1996)

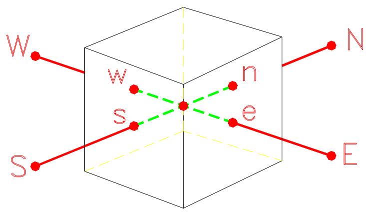
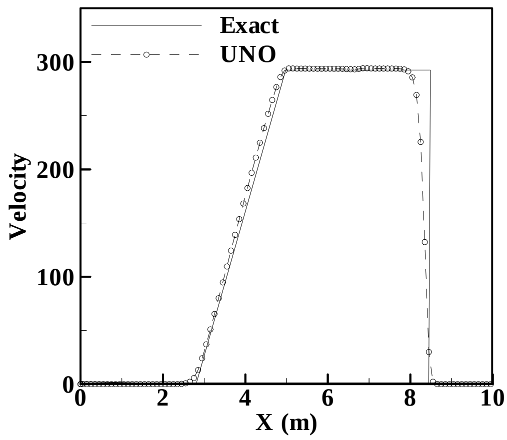
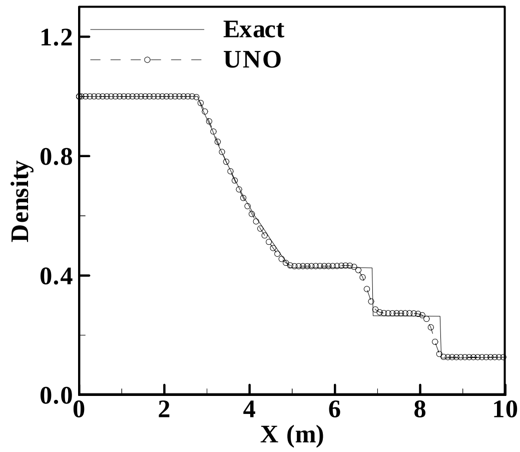


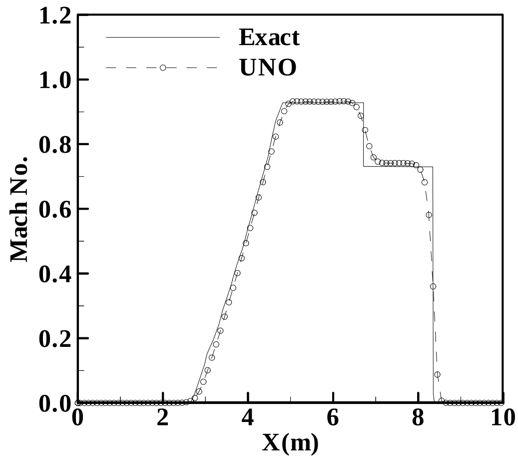
Fig.1: Finite volume and storage arrangement



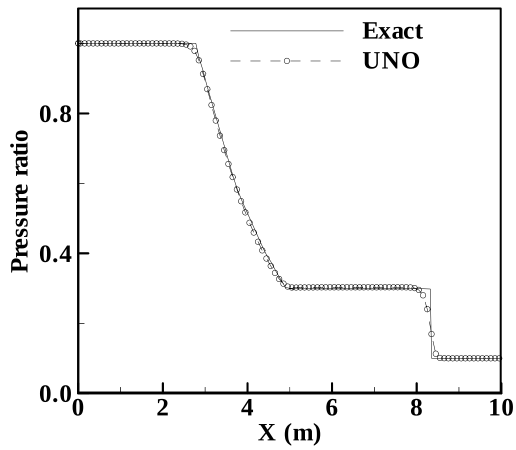
a) Velocity



b) Density



c) Mach number



d) Pressure ratio distribution

Figure 2: Shock-tube results for an initial pressure ratio $\frac{P_H}{P_L} = 10$ at time $t_0 = 6.0$

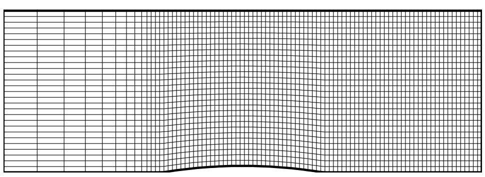


Figure 3: Geometry

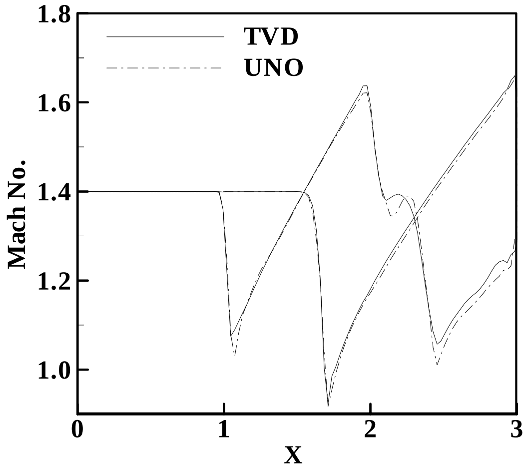


Fig 4(a) Mach number distribution

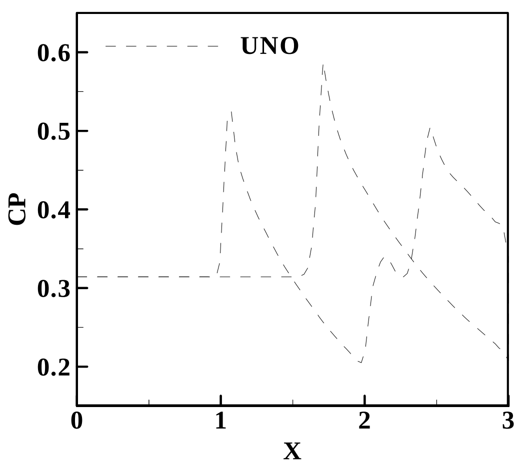


Fig 4 (b) Pressure ratio on lower wall

Fig. 4. Supersonic flow over 4% thick bump, inlet M_∞ =1.4