14^{th} Australasian Fluid Mechanics conference
Adelaide University, Adelaide, Australia
10-14 December 2001

A New NVD Scheme in Pressure-Based Finite –Volume Methods

M.H Djavareshkian

Mechanical Engineering Department, Faculty
of Engineering, Tabriz University, Tabriz 51664, Iran

Abstract

The paper presents an implementation of a new NVD scheme into an implicit finite volume procedure, which uses pressure as a working variable. This scheme with the minimum number of adjustable parameters is robust and does not create convergence problems on the wide range of test cases. The method is applied to the computation of steady transonic and supersonic flows over bump in-channel geometry as well as to the transient shock-tube problem. The results are compared with other computations published in the literature.

Introduction

Many NVD schemes now exist for the resolution of steep gradients (such as shocks) arising in fluid flow problems e.g. the SMART scheme of Gaskell and Lau [2], the SOUCUP scheme of Zhu & Rodi [8], and the STOIC scheme of Darwish [1]. Most of those schemes use different differencing schemes through the solution domain. This procedure includes some kind of switching between the differencing schemes. Switching introduces additional non-linearity and instability into the computation. The worst case is that instead of a single solution for steady state problem, the differencing scheme creates two or more unconverged solution with the cyclic switching between them. In that case it is impossible to obtain a converged solution and the convergence stalls at some level. This is a very unsatisfactory feature of a differencing scheme and it should be avoided.

This paper presents a new scheme with the minimum number of adjustable parameters, into an implicit non-uniform finite-volume procedure, which uses the pressure as a working variable. This scheme is robust and does not create convergence problems on the wide range of test cases. One advantage of this scheme in comparison with all other differencing schemes is some kind of switching. Only two differencing schemes, central differencing and blending between upwind and central differencing are included where the blending factor is determined automatically. The scheme is considered to be a kind of smooth switch.

Finite Volume Discretization

The basic equations, which describe conservation of mass, momentum and scalar quantities, can be expressed in the following vector form, which is independent of coordinate system used.

$$\frac{\partial \rho}{\partial t} + div(\rho \vec{V}) = S_m \qquad (1)$$

$$\frac{\partial (\rho \vec{V})}{\partial t} + div(\rho \vec{V} \otimes \vec{V} - \vec{T}) = \vec{S}_v \qquad (2)$$

$$\frac{\partial (\rho \phi)}{\partial t} + div(\rho \vec{V} \phi - \vec{q}) = \vec{S}_\Phi \qquad (3)$$

Where ρ, $\vec{V}$ and ϕ are respectively the density, velocity vector and scalar quantity, $\vec{T}$ is the stress tensor and $\vec{q}$ is the scalar flux vector. The latter two are usually expressed in terms of basic dependent variables. The stress tensor for a Newtonian fluid is:

$$\vec{T} = -(P + \frac{2}{3}\mu \, div \vec{V})\vec{I} + 2\mu \vec{D} \qquad (4)$$

And the Fouries-type law usually gives the scalar flux vector:

$$\vec{q} = \Gamma_\phi \, grad \Phi \qquad (5)$$

For the purpose of illustration eqn (3) may be expressed in 2D Cartesian coordinates as:

$$\overbrace{\frac{\partial (\rho \phi)}{\partial t}}^{transient\ term} + \overbrace{\frac{\partial (\rho u \phi)}{\partial x} + \frac{\partial (\rho v \phi)}{\partial y}}^{convection\ term}$$
$$\overbrace{-\frac{\partial}{\partial x}(\Gamma_\phi \frac{\partial \phi}{\partial x}) - \frac{\partial}{\partial y}(\Gamma_\phi \frac{\partial \phi}{\partial y})}^{diffusion\ term} = \overbrace{S_\phi}^{source} \qquad (6)$$

Integration of eqn (6) over a finite volume (see e.g. Fig.1) and application of the Gauss divergence Theorem yield a balance involving the rate of change in ϕ, face fluxes and volume-integrated net source. The transient term is approximated by the Euler implicit scheme for the purpose of this work, although other temporal schemes are also possible.

The diffusion flux is approximated by central differences and can be written for cell-face east (e) of the control volume in Fig. (1) as an example as:

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

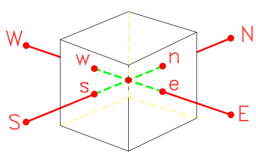


Fig.1: Finite volume and storage arrangement

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 is:

$$I_f^c = (\rho.v.A)_f \phi_f = F_f \phi_f \tag{8}$$

The value of the dependent variable ϕ_f is not known and should be estimated using an interpolation procedure, from the values at neighbouring grid points. The details of how the interpolation is made is dealt with later, it suffices to say that the discretised equations resulting from each approximations take the form:

$$A_P.\phi_P = \sum_{m=E,W,N,S} A_m.\phi_m + S_\phi^{©} \tag{9}$$

Where the $A's$ are coefficients the expressions for which are given later.

Normalized Variable Diagram

Leonard [6] proposes a new way of describing the boundedness criterion for a differencing scheme used for pure advection problems, Called the Normalized Variable diagram (NVD). For any cell face, considering the direction of the convection, three nodes are selected: two nodes adjacent to the face (C, D) and the far upstream node (U), as shown in Fig.2.

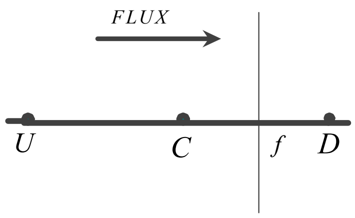


Fig. 2: Node values in the normalized variable approach

The locally normalized variable is defined as:

$$\tilde{\phi} = \frac{\phi - \phi_u}{\phi_d - \phi_u} \tag{10}$$

If the locally normalized variable is written for the value at the central node

$$\tilde{\phi}_c = \frac{\phi_c - \phi_u}{\phi_d - \phi_u} \tag{11}$$

All the differencing schemes could be written in the form:

$$\tilde{\phi}_f = f(\tilde{\phi}_c)$$

Gaskell and Lau [2] show that the boundedness criterion for these schemes could be defined in the NVD diagram, showing $\tilde{\phi}_f$ as a function of $\tilde{\phi}_c$ as a shaded region on Fig. 3 or with the following conditions:

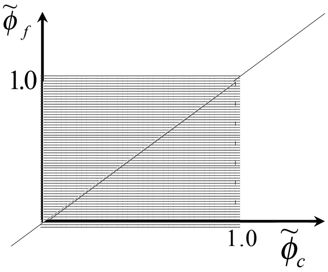


Fig. 3:Boundedness criterion in the NVD diagram

1) For $0 \langle \tilde{\phi}_c \langle 1$, $\tilde{\phi}_f$ is bounded by the function $\tilde{\phi}_f = \tilde{\phi}_c$ and above by unity, and passes through the points (0,0) and (1,1).

2) For $\tilde{\phi}_c \langle 0$ or $\tilde{\phi}_f \rangle 1$, $\tilde{\phi}_f$ is equal to $\tilde{\phi}_c$

The Development of the SBIC Scheme

Here we present the Second and Blending Interpolation Combined (SBIC) scheme. Some parts of the new scheme have been shown in Fig. 4. For $\tilde{\phi}_c$ outside the interval [0,1] upwind differencing should be used. For the interval [k, 1] central differencing should be used. The parameter k (0< k < ½) should be set to a fixed value, depending on how much of central differencing is used. For the interval [0,k], a smooth function is used, in fact the value of $\tilde{\phi}_f$ is bounded by the values obtained from upwind and central differencing. It would be therefore ideal to use some kind of blending between the two to obtain a good evaluation of the face value. The other conclusion that can be drawn are that the blending should be used over the whole interval 0< $\tilde{\phi}_c$ < k and that the values of the blending factor should be determined on the basis of $\tilde{\phi}_c$, $\tilde{x}_c$ and $\tilde{x}_f$.

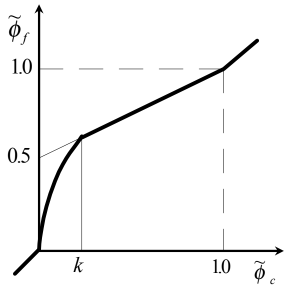


Fig. 4:SBIC scheme in the NVD diagram

The limits on the select on of each value of k can be determined in the following way. Obviously the lower limit is to keep k=0 , which would represent switching between upwind and central differencing. This should not be favored because, it is essential to avoid the abrupt switching between the schemes in order to achieve the converged solution. The upper limit of k is 0.5, since it represents the constant gradient and there is no need to use anything else than central differencing in that case. The value of k should be kept as low as possible in order to achieve the maximum resolution of the scheme provided the selected value does not interfere with the convergence of the solution.

Convective Fluxes

The expression for momentum and energy fluxes in eqn (8) are determined by the NVD scheme used for interpolation from nodes at the neighbouring points. For new scheme, the value of the dependent variable ϕ_f or $\tilde{\phi}_f$ is calculated according to table 1

A	$\tilde{\phi}_f = \tilde{\phi}_c$	$\tilde{\phi}_c \notin [0,1]$
B	$\tilde{\phi}_f = -\frac{\tilde{x}_c - \tilde{x}_f}{\tilde{x}_c - 1}\tilde{\phi}_c^2 + (1 + \frac{\tilde{x}_c - \tilde{x}_f}{k(\tilde{x}_c - 1)})\tilde{\phi}_c$	$\tilde{\phi}_c \in [0,k]$
C	$\tilde{\phi}_f = \frac{\tilde{x}_c - \tilde{x}_f}{\tilde{x}_c - 1} + \frac{\tilde{x}_f - 1}{\tilde{x}_c - 1}\tilde{\phi}_c$	$\tilde{\phi}_c \in [k,1]$

Table 1: The calculation of the $\tilde{\phi}_f$ with SBIC scheme

When all other fluxes at the various cell faces are calculated according to SBIC scheme, then they are introduced into the discretised equations (eqn. 9).

Solution Algorithm

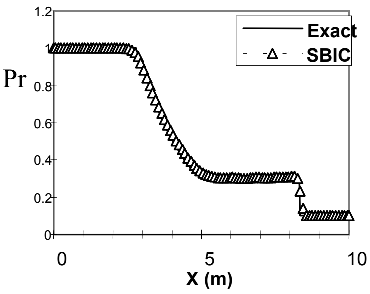
Most contemporary pressure-based methods employ a sequential iteration technique in which the different conservation equations are solved one after another. The common approach taken in enforcing continuity is by combining the equation for continuity with those of momentum to derive an equation for pressure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

The present work employs the PISO technique (Issa [3]) in which the implicitly discretised equations are solved at each time step by a sequence of predictor and corrector steps. This scheme is especially efficient for unsteady flows, as it does not involve expensive iteration. For steady flows, time marching is effected until the steady state is reached.

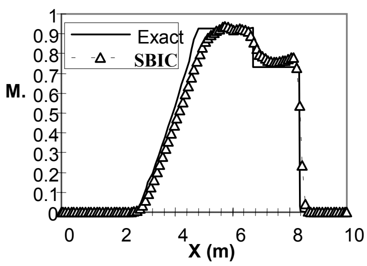
Results

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 solution when 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.

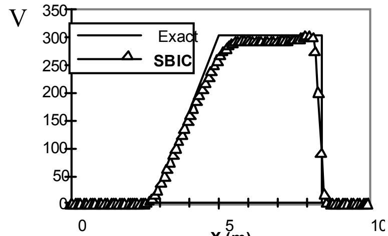
Fig. 5 shows the spatial distribution of pressure ratio, Mach number, density and velocity along the shock tube at a given instant in time in a shock-tube for an initial pressure of 10. The results of computation on a mesh of 100 nodes are compared with the analytic solution. It can be seen that the shock is sharply captured, the contact discontinuity is better resolved and oscillation is not relatively produced for the new NVD scheme.



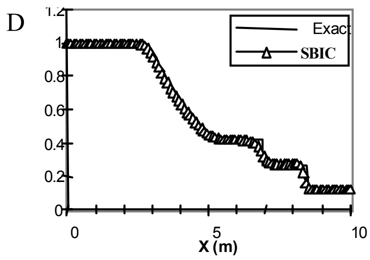
(One) Pressure ratio distribution



(b) Mach No. Distribution



(c) Velocity (m/s) distribution



(d) Density (kg/m3) distribution

Figure 5:Shock-tube results for an initial pressure ratio

$$\frac{P_H}{P_L} = 10 \text{ at time } t_0 = 6.0$$

Figure 6 shows the geometry of a 10% thick bump on a channel wall with the mesh (66x18) to compute this steady two-dimensional inviscid flow case.

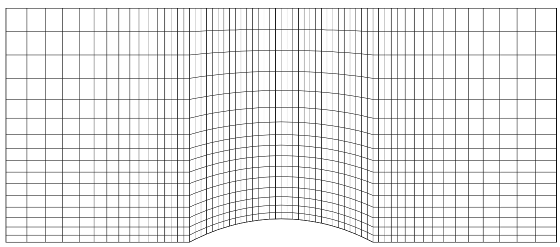
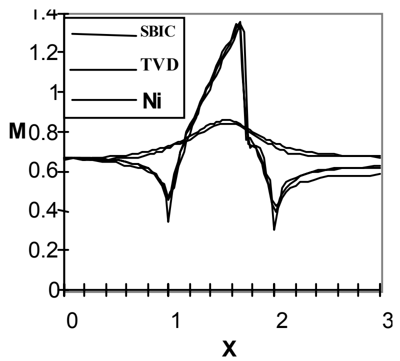


Figure 6: Geometry

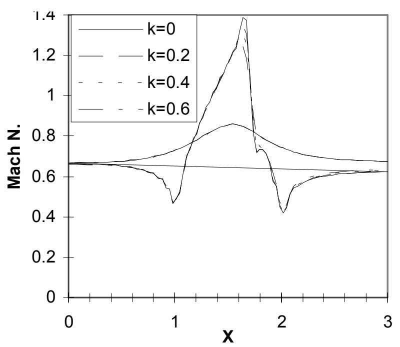
Two cases were considered for this geometry, one with Mach number of 0.675 leading to transonic flow over the bump and the second case with inlet Mach number of 1.6 resulting in a supersonic flow.

Figures 7(a) and 7(b) present the result of transonic flow for the same geometry. The results of this case is compared with those of Issa and Javareshkian [4,5], which were carried out with a TVD scheme and the Ni [7] scheme is constructed by combining the multiple-grid technique with the second order accurate finite volume integration method for the Euler equation. It can be observed that the shock gradient is the same for all schemes. Fig. 7(b) shows the results of the SBIC scheme for different k. It is observed that when k=0 is used, the shock is predicted with slight overshoot and for k = 0.6, the shock is slightly smeared. Fig. 8 shows the results of the supersonic case. It is seen that TVD and SBIC schemes capture sharp shocks at the leading edge but the SBIC scheme is slightly smeares the shock at the trailing edge.

The agreement between the two solutions is remarkable, thus once again verifying the validity of the present NVD scheme.



(a) Mach No. distribution



(b) Mach No. distribution for difference k

Figure 7: Inviscid transonic flow over 10% thick bump in channel

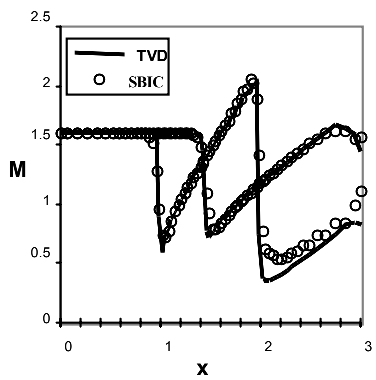


Figure 8: Inviscid supersonic flow over 10% thick bump in channel

Conclusion

A pressure-based implicit procedure has been described. It incorporates a new NVD scheme. The SBIC scheme is applied to both transient and steady state flows and the results, compare very well with another schemes.

References

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

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

[3] Issa R.I. (1986), “solution of implicit discretised fluid flow equations by operator splitting”, J. Comput. Phys. Vol. 62, pp. 40

[4] Issa R.I. and Javareshkian M.H. (1996), “ Application of TVD Scheme in Pressure-Based Finite-Volume Methods”, ASME Conference, Fluids Engineering Division, Volume 3, pp.159-164

[5] Issa R.I. and Javareshkian M.H. (1988), “Compressible Inviscid and Turbulent Flow Calculation using a Pressure-Based Method with a TVD Scheme”, AIAA Journal, Vol. 36, No.9, pp.1652-1657

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

[7] Ni R.H. (1983), “A Multiple grid scheme for solving the Euler equations”, AIAA J., Vol.20, pp.375

[8] Zhu J. & Rodi W. (1991), “A Low Dispersion and Bounded Convection Scheme”, Computer Methods in Applied Mechanics and Engineering, Vol. 92, pp.87-96