INT. J. NUM. METH. HEAT FLUID FLOW, VOL. 5, 877–888 (1995)

ACCURACY ANALYSIS OF CENTRAL SCHEMES WITH ARTIFICIAL DISSIPATION AND UPWIND FLUX-DIFFERENCE TVD SCHEMES FOR NAVIER-STOKES EQUATIONS

E. DICK AND J. STEELANT

Department of Mechanical and Thermal Engineering, Universiteit Gent, Sint-Pietersnieuwstraat 41, B-9000 Gent, Belgium

ABSTRACT

A comparison of the accuracy of the central discretization scheme with artificial dissipation and the upwind flux-difference TVD scheme has been made for the compressible Navier-Stokes equations for high Reynolds number flows.

First, a comparison is made on two one-dimensional model problems. Then the schemes are compared on flat plate boundary layer flow. It is shown that a central scheme basically has poor accuracy due to the isotropic nature of the artificial dissipation. An upwind scheme decomposes the flow into different components and adapts the dissipation to the velocity of the components. The associated ansitropic dissipation results in a good accuracy. It is further discussed how a central discretization scheme with artificial dissipation can be improved at the expense of the same complexity of an upwind scheme.

KEY WORDS Navier-Stokes equations Central schemes Upwind schemes Artificial dissipation

INTRODUCTION

The central type finite volume discretization method using artificial dissipation, originated by Jameson *et al.*[1], is nowadays a very popular method. Many researchers employ it in several variants. The method was originally developed for Euler equations and is still predominantly used for inviscid flow calculations. The artificial dissipation is a blend of a second order difference term and a fourth order difference term. The second order term is used to prevent oscillations at shock waves, while the fourth order term is meant for stability. The coefficients of both terms are determined by a pressure sensor. In the vicinity of a shock the fourth order term is switched off. In smooth regions of the flow the contribution of the second order term is negligible and essentially the fourth order term comes in to prevent the odd-even decoupling which would occur for a pure central scheme.

In its basic version, the method is found to be very inaccurate for high Reynolds number viscous flows. Apparently, the artificial dissipation, although of fourth order near solid walls, interacts with the physical viscous terms. The role of the artificial dissipation was analysed by Allmares[2]. He showed that for a boundary layer flow the contribution of the artificial dissipation terms in the flux-balance of the x-momentum equation is comparable to the contribution of the physical viscous terms. Allmares undertook the same analysis for an upwind flux-difference TVD scheme and found an almost negligible false diffusion. In both analyses he took the central difference approximation of the inviscid fluxes as the true physical fluxes.

0961–5539/95/1000877–12$2.00

Received July 1994
Revised November 1994

The deficiency of the central scheme with artificial dissipation is generally recognized. Most researchers using the method bring in means to reduce the artificial dissipation in the normal direction close to walls. Among the many existing examples, we cite here a method by Dawes[3] who puts the corresponding coefficient of the fourth order dissipation term equal to zero, a method by Arnone and Swanson[4] who scale the coefficient by a function of the grid aspect ratio and a method by Kunz and Lakshminarayana[5] who also scale the coefficient with grid aspect ratio and reduce it proportional to the square of the velocity. Near to walls, usually all artificial dissipation terms in the normal direction can be set equal to zero. Typical grids for practical calculations with Reynolds averaged Navier-Stokes equations in turbulent flows, like those used in the cited cases, have such a small mesh spacing in the direction normal to the wall so that the physical viscous terms are largely sufficient to prevent the odd-even decoupling. In a practical calculation the stretching of the grid in the direction normal to the wall is rather large. This has a consequence that, somewhat further away from the wall, the artificial dissipation terms become essential to guarantee the smoothness of the solution. This makes it very delicate to devise ad hoc means to steer the artificial dissipation. Allmares[2] mentions several attempts with disappointing results. The present authors share the same experience. In the cited examples[3,4,5], it is very difficult to judge on the accuracy. Calculated and experimental pressure plots compare well with each other. There is, however, no experimental information on skin friction. Even if there were, it would not be possible to evaluate the accuracy since results depend, critically, upon the turbulence model. To verify the accuracy, only laminar flat plate boundary layer flow can be used, as in the paper by Allmares.

Motivated by the observation that upwind TVD schemes seem to perform very well with respect to accuracy, Swanson and Turkel[6] developed a modification of the basic central scheme with artificial dissipation to bring it closer to upwind schemes. They replaced the scalar dissipation terms by a matrix dissipation term. Their matrix dissipation can be seen as a simplified version of the difference between an upwind TVD discretization and a central discretization without artificial dissipation. The purpose of the matrix form of the numerical dissipation is to apply the appropriate scaling of the dissipation in each flow equation. Like the upwind method, their method uses limiters. One of their simplifications consists of employing only limiters based on pressure differences. If they would use limiters on the appropriate variables of each flow equation, their method would have the same complexity as the upwind TVD method. Swanson and Turkel compare results obtained by their matrix dissipation method to results obtained by the scalar dissipation method and the upwind TVD method. Their conclusion is that there is significant improvement in the accuracy of the matrix dissipation method over the accuracy of the scalar dissipation method, but that the accuracy of the upwind TVD method is not obtained.

The aim of the present paper is, first, to give an explanation of the observed good accuracy obtainable with the upwind TVD method. The analysis is undertaken on one-dimensional model equations. Numerical illustrations are given for flow over a flat plate. The second objective is to explain why a central method with artificial dissipation can be used to obtain good accuracy when the artificial dissipation is properly scaled. Again, this is partly done by a one-dimensional model equation and by numerical illustrations for flow over a flat plate. Laminar flow over a flat plate is considered, but the Reynolds number is sufficiently high and the mesh spacing near the plate is sufficiently small, so that the test case is representative for practical applications.

THE NUMERICAL METHODS

The two-dimensional Navier-Stokes equations for unsteady compressible flow may be written in conservation form as,

$$\frac{\partial U}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}=\frac{\partial R}{\partial x}+\frac{\partial S}{\partial y} \tag{1}$$

where

$$U=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho E\end{Bmatrix},\quad F=\begin{Bmatrix}\rho u\\ \rho uu+p\\ \rho uv\\ \rho Hu\end{Bmatrix},\quad G=\begin{Bmatrix}\rho v\\ \rho uv\\ \rho vv+p\\ \rho Hv\end{Bmatrix}$$

and

$$R=\begin{Bmatrix}0\\ \tau_{xx}\\ \tau_{xy}\\ u\tau_{xx}+v\tau_{xy}+k\dfrac{\partial T}{\partial x}\end{Bmatrix},\quad S=\begin{Bmatrix}0\\ \tau_{yx}\\ \tau_{yy}\\ u\tau_{yx}+v\tau_{yy}+k\dfrac{\partial T}{\partial y}\end{Bmatrix}$$

with

$$\tau_{xx}=2\mu\frac{\partial u}{\partial x}-\lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)$$

$$\tau_{xy}=\tau_{yx}=\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)$$

$$\tau_{yy}=2\mu\frac{\partial v}{\partial y}-\lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)$$

where ρ is the fluid density; u, v velocity components in the x- and y-direction, respectively; p is pressure; T static temperature and E the total energy. Total enthalpy is defined by $H=E+p/\rho$. Fluid properties are: μ the viscosity; $\lambda=\frac{2}{3}\mu$ the second viscosity coefficient and k the heat transfer coefficient. The equation of state is given by $p=\rho RT$.

By use of Gauss' divergence theorem, the governing equations can be written in integral form which are the basis of the finite volume spatial discretization. The equations are integrated in time by the standard four-stage simplified Runge-Kutta scheme.

In the central scheme, a cell-vertex definition of inviscid and viscous fluxes is used, see *Figure 1*.

The cells are chosen as control volumes. Fluxes are obtained by piecewise linear integration. For the viscous fluxes, derivatives of variables at the vertices are needed. These are again

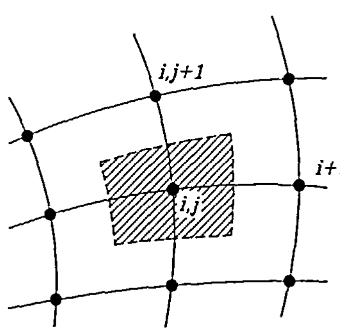

Figure 1 Finite volume discretization with cells and dual control volume (dashed)

determined by a cell-vertex approach. Mean values of derivatives are first calculated in the cells by the use of Gauss' theorem. A derivative at a vertex is defined by taking the mean over the neighbouring cells. To update the variables, a dual control volume around a vertex is employed. The flux balance for this control volume is defined by taking an average of the flux balances of the surrounding cells. The fourth order Jameson dissipation is constructed as per usual as a double pseudo-Laplacian. A pseudo-Laplacian is a 5-point difference molecule with value -4 at the central node and $+1$ at the other nodes. A double pseudo-Laplacian is the pseudo-Laplacian of the pseudo-Laplacian. Only the fourth order dissipation is used here since we only consider smooth flow fields. At boundaries, to define the Laplacian, exterior points are needed. The variables at the exterior points can be obtained by constant extrapolation, linear extrapolation or by mirroring. We illustrate these possibilities in the next paragraph. The exterior points do not need to have a precise geometrical location since the pseudo-Laplacian does not use distances.

A vertex-centered scheme is used for the upwind formulation of the inviscid fluxes. Determination of derivatives and the viscous fluxes is treated in the same way as in the central scheme. The control volume for the convective fluxes is the dual control volume shown in *Figure 1*. The discretization of the convective terms is effected by the polynomial flux-difference splitting[7]. The second order flux is defined by the flux extrapolation technique involving a limiter. The minmod limiter is used. Full details on the inviscid flux definitions are given in Reference 8. The treatment of the viscous terms is explained in Reference 9.

THE ONE-DIMENSIONAL MODEL PROBLEMS

The mass and momentum-x equation for a constant density and constant pressure flow under boundary layer conditions are,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\nu\frac{\partial^2 u}{\partial y^2}$$

Combining both equations, the momentum equation becomes,

$$v\frac{\partial u}{\partial y}=u\frac{\partial v}{\partial y}+\nu\frac{\partial^2 u}{\partial y^2}$$

This is a convection-diffusion equation with a source term $u(\partial v/\partial y)$ which is positive in flat plate boundary layer flow. A one-dimensional model equation that mimics this behaviour is,

$$v\frac{\mathrm{d}u}{\mathrm{d}y}=2\frac{v^2}{\nu}(C-u)+\nu\frac{\mathrm{d}^2 u}{\mathrm{d}y^2} \qquad (2)$$

for C, v and ν constant and positive. This equation has the solution,

$$u=C\left[1-\exp\left(-\frac{yv}{\nu}\right)\right]$$

The solution has boundary layer behaviour at $y=0$. The constant C can be used to normalize the solution at $u=1$ for $y=L$, giving,

$$C=\left[1-\exp\left(-\frac{Lv}{\nu}\right)\right]^{-1}$$

The model equation (2) differs from the equation that is usually employed to study boundary layer behaviour:

$$v\frac{du}{dy}=\nu\frac{d^2u}{dy^2} \tag{3}$$

This equation has the solution,

$$u=C\left[1-\exp\left(-\frac{(L-y)v}{\nu}\right)\right]$$

This solution has boundary layer behaviour at $y=L$. The constant C can be used to normalize the solution at $u=1$ for $y=0$, giving the same expression as for problem (2). The model problem (3) is not physically correct in the sense that the velocity v is directed towards the wall. We use both (2) and (3) to illustrate the behaviour of central and upwind discretization schemes.

In order to construct the solution, we use the time dependent form of the equations (2) and (3). For equation (2), this is,

$$\frac{\partial u}{\partial t}+v\frac{\partial u}{\partial y}=\frac{2v^2}{\nu}(C-u)+\nu\frac{\partial^2 u}{\partial y^2} \tag{4}$$

A central discretization of (4), without artificial viscosity gives,

$$\frac{\Delta u_i}{\Delta t}+v\frac{u_{i+1}-u_{i-1}}{2h}=\frac{2v^2}{\nu}\left(C-\frac{1}{4}u_{i+1}-\frac{1}{2}u_i-\frac{1}{4}u_{i-1}\right)+\frac{\nu}{h^2}(u_{i+1}-2u_i+u_{i-1})$$

where h is the constant mesh spacing and Δu_i the increment in time. The source term is averaged over the interval $(i-\frac{1}{2}, i+\frac{1}{2})$ in order to obtain second order accuracy. With the mesh spacing Reynolds number $Re_h=vh/\nu$, this equation is,

$$\Delta u_i+v\frac{\Delta t}{h}\left[\frac{u_{i+1}-u_{i-1}}{2}-2Re_h\left(C-\frac{1}{4}u_{i+1}-\frac{1}{2}u_i-\frac{1}{4}u_{i-1}\right)-\frac{1}{Re_h}(u_{i+1}-2u_i+u_{i-1})\right]=0 \tag{5}$$

The central difference discretization of (3) is similarly obtained. In the sequel, we construct the solution of (5) through a four stage Runge-Kutta type time stepping like we use it for the full Navier-Stokes equations. The boundary conditions are $u_0=0$, $u_N=1$ for (2) and $u_0=1$, $u_N=0$ for (3), where N is the number of discretization intervals. In the sequel, we always take $N=100$.

A central discretization with fourth order artificial dissipation has the form,

$$\Delta u_i+v\frac{\Delta t}{h}\left[\frac{u_{i+1}-u_{i-1}}{2}-2Re_h\left(C-\frac{1}{4}u_{i+1}-\frac{1}{2}u_i-\frac{1}{4}u_{i-1}\right)-\frac{1}{Re_h}(u_{i+1}-2u_i+u_{i-1})\right]$$
$$+\varepsilon(u_{i+2}-4u_{i+1}+6u_i-4u_{i-1}+u_{i-2})=0 \tag{6}$$

By bringing the artificial dissipation term into the fluxbalance, an associated mesh spacing Reynolds number is formed as,

$$Re_\varepsilon=\frac{v\Delta t}{\varepsilon h}$$

In an actual Navier-Stokes calculation, the time step is obtained from a CFL-number based on a maximum transport velocity, i.e. the sum of a convective velocity and the speed of sound (c). So, the CFL number is approximately given by $c\Delta t/h$. This CFL-number has the order of unity. So, the Reynolds number Re_ε has the order of $v/\varepsilon c$. With $\varepsilon\approx 0.01$ this number is of the order of 0.1 to 1 and is independent of mesh size. In the calculations to follow, we used $Re_\varepsilon=0.1$ and $Re_\varepsilon=1$. The mesh spacing Reynolds number based on the physical viscosity can be much smaller, depending on mesh size. In the calculations we put $Re_h=0.1$. This is not an unfavourable situation for the artificial dissipation. In an actual computation, the physical viscous terms can be less significant. This is illustrated later.

A first order upwind discretization has the form,

$$\Delta u_i + v\frac{\Delta t}{h}\left[(u_i - u_{i-1}) - 2Re_h\left(C - \frac{1}{4}u_{i+1} - \frac{1}{2}u_i - \frac{1}{4}u_{i-1}\right) - \frac{1}{Re_h}(u_{i+1} - 2u_i + u_{i-1})\right] = 0 \qquad (7)$$

A TVD-second order upwind discretization differs from the previous formula by replacement of the convective term by the non-linear combination,

$$u_i + \frac{1}{2}Lim[u_i - u_{i-1}, u_{i+1} - u_i] - u_{i-1} - \frac{1}{2}Lim[u_{i-1} - u_{i-2}, u_i - u_{i-1}]$$

where *Lim* is a limiter. We take here the MinMod limiter, i.e. the result is the argument with minimum modulus if both arguments have the same sign and is zero if the signs differ.

RESULTS FOR THE MODEL PROBLEMS

In *Table 1*, results are shown for the central discretization with artificial dissipation given by (6). In order to calculate the solution in the points 1 and $N-1$, the boundary values in the points 0 and N are used. In the determination of the artificial viscosity term, a point outside the domain $(0, L)$ is needed. This point can be defined in several ways. We used constant extrapolation (CTE) i.e. $u_{-1} = u_0$, linear extrapolation (LNE), i.e. $u_{-1} = 2u_0 - u_1$ or mirroring (MIR), i.e. $u_{-1} = u_1 - 2u_0$ and compared with the exact solution (EXA). Calculations were done both for $Re_\varepsilon = 0.1$ and $Re_\varepsilon = 1$. Linear extrapolation gives the best results and will therefore be used in the sequel. This is conform with the usual practice.

Tables 2 and *3* show the results for the different schemes with $Re_\varepsilon = 0.1$: the central discretization without artificial dissipation (CEN), the central discretization with artificial dissipation (ART), the first order upwind discretization (UP1) and the second order TVD upwind discretization (TVD). For the TVD-scheme also, linear extrapolation at boundaries is used.

For the model problem with source term, the TVD result is identical to the central result. In this case, the limiter chooses the central correction since the slope of the solution diminishes with the distance to the wall. The result is almost identical to the exact value. This model problem mimics best the physical boundary layer. For the model problem without source term, the TVD result differs slightly from the central result. The limiter chooses the second order upwind correction. So, for the physical boundary layer, the TVD-scheme generates very accurate results.

Table 1 Different extrapolation strategies in the central discretization with artificial viscosity. Model problem with source term, $Re_h = 0.1$

		$Re_\varepsilon = 1$			$Re_\varepsilon = 0.1$		
No.	EXA	CTE	LNE	MIR	CTE	LNE	MIR
0	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	0.0952	0.0876	0.0945	0.0817	0.0604	0.0905	0.0453
2	0.1813	0.1739	0.1807	0.1681	0.1372	0.1756	0.1180
3	0.2592	0.2526	0.2587	0.2472	0.2149	0.2537	0.1955
4	0.3297	0.3238	0.3294	0.3190	0.2882	0.3247	0.2699
5	0.3935	0.3882	0.3933	0.3838	0.3555	0.3891	0.3387
6	0.4512	0.4465	0.4511	0.4425	0.4169	0.4475	0.4015
7	0.5034	0.4992	0.5034	0.4957	0.4725	0.5003	0.4586
8	0.5507	0.5469	0.5507	0.5437	0.5229	0.5480	0.5103
9	0.5935	0.5901	0.5935	0.5872	0.5685	0.5912	0.5571

Table 2 Comparison of different schemes for the model with source term, $Re_t = 0.1$, $Re_h = 0.1$

No.	EXA	CEN	ART	UP1	TVD
0	0.0000	0.0000	0.0000	0.0000	0.0000
1	0.0952	0.0952	0.0905	0.0936	0.0952
2	0.1813	0.1814	0.1756	0.1785	0.1814
3	0.2592	0.2594	0.2537	0.2554	0.2594
4	0.3297	0.3299	0.3247	0.3251	0.3299
5	0.3935	0.3937	0.3891	0.3883	0.3937
6	0.4512	0.4515	0.4475	0.4456	0.4515
7	0.5034	0.5037	0.5003	0.4975	0.5037
8	0.5507	0.5510	0.5480	0.5445	0.5510
9	0.5935	0.5938	0.5912	0.5872	0.5938

Table 3 Comparison of different schemes for the model without source term $Re_t = 0.1$, $Re_h = 0.1$

No.	EXA	CEN	ART	UP1	TVD
91	0.5935	0.5938	0.5928	0.5760	0.5920
92	0.5507	0.5510	0.5495	0.5336	0.5493
93	0.5034	0.5037	0.5016	0.4869	0.5021
94	0.4512	0.4515	0.4485	0.4356	0.4499
95	0.3935	0.3937	0.3899	0.3791	0.3923
96	0.3297	0.3299	0.3252	0.3170	0.3287
97	0.2592	0.2594	0.2538	0.2487	0.2583
98	0.1813	0.1814	0.1754	0.1736	0.1806
99	0.0952	0.0952	0.0903	0.0909	0.0948
100	0.0000	0.0000	0.0000	0.0000	0.0000

The results obtained with the central discretization without artificial viscosity are always the best for the model problems. The high quality of these results is mainly due to the low value of the mesh spacing Reynolds number (0.1). In an actual Navier-Stokes computation, the mesh spacing Reynolds number is only very small near the wall and can have very high values far away from the wall. For values higher than 2, as is well known, the central discretization result becomes oscillatory. The TVD-method guarantees non-oscillatory results in regions of high mesh spacing Reynolds number but, as shown here, also guarantees high accuracy results in regions of low mesh spacing Reynolds number, i.e. near the walls.

In *Tables 2* and *3* the results of the first order upwind method are given to show that the TVD modification of the upwind method is really necessary to obtain good accuracy. Further, it is clear that the artificial dissipation method can only reach the quality of the central method if the artificial dissipation is completely switched off. This means that the corresponding coefficient is to be steered in function of the obtained solution. The coefficient must be zero there where it is allowed by the mesh spacing and the associated Reynolds number of the physical viscous terms. The cited methods[3,4,5] all try to achieve this. None of these methods, however, give an indication of the magnitude of the coefficient in the function of velocity. They all use essentially only geometrical information like distance to walls, orthogonality of the considered direction to the wall and cell aspect ratio. As already discussed, the matrix dissipation method of Swanson and Turkel has the potential to steer the coefficient of the artificial dissipation correctly. This would necessitate limiters based on velocity differences and would make the method completely equivalent to an upwind TVD method both in philosophy and complexity.

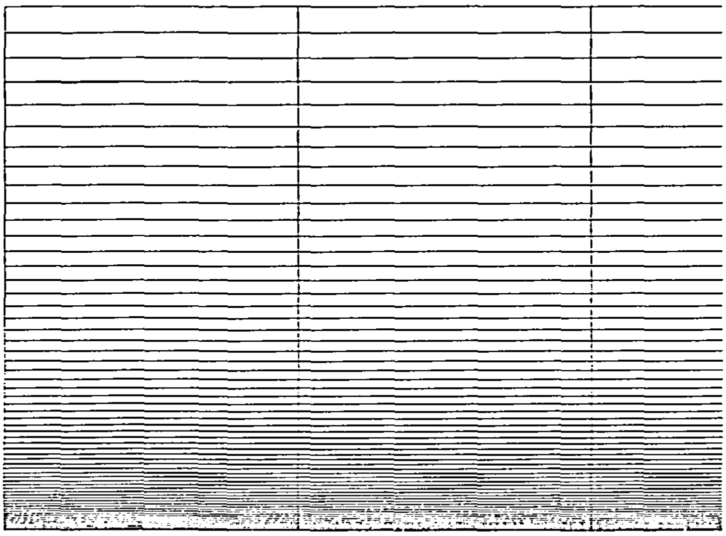

Figure 2 Detail of the grid near the wall

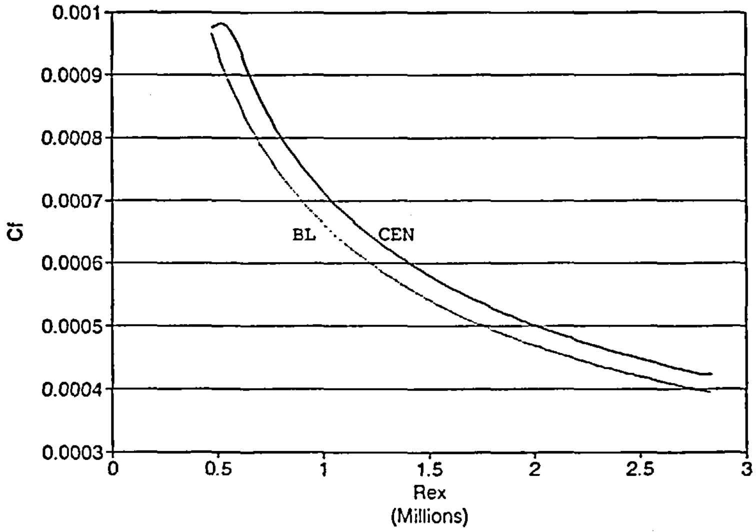

Figure 3 Skin friction for the central scheme with full artificial dissipation (CEN). Comparison with Blasius-solution (BL)

FLAT PLATE FLOW

The set of equations (1) has been used to calculate flow over a flat plate. The free stream has a velocity $U_\infty = 30\,m/s$ and zero pressure gradient, with a Reynolds number of $2 \cdot 10^6/m$. The problem has been solved both by the central method with artificial dissipation and by the TVD second order upwind scheme, on a 101×61 grid with approximately 30 points in the boundary layer. *Figure 2* shows a close-up of the grid near the wall. The mesh spacing in the longitudinal direction is constant. The length of the domain is 1 m and the mesh length is 0.01 m. In the

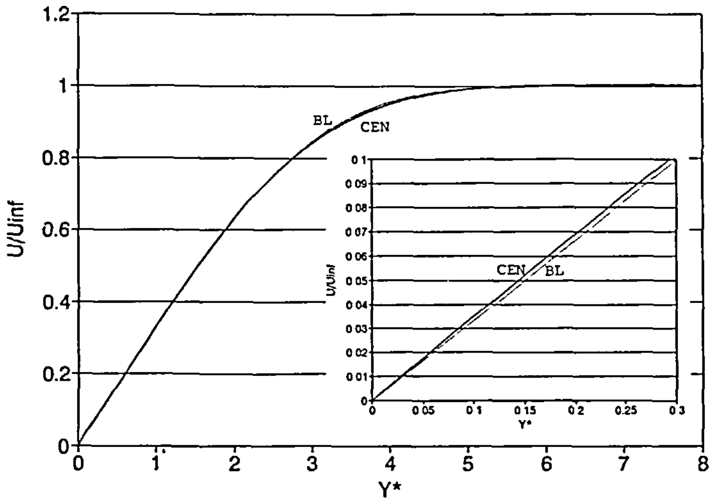

Figure 4 u-Profile for the central scheme with full artificial dissipation ($y^* = y\sqrt{U_{inf}/(\nu x)}$); insert: detail at the wall. Comparison with Blasius-solution

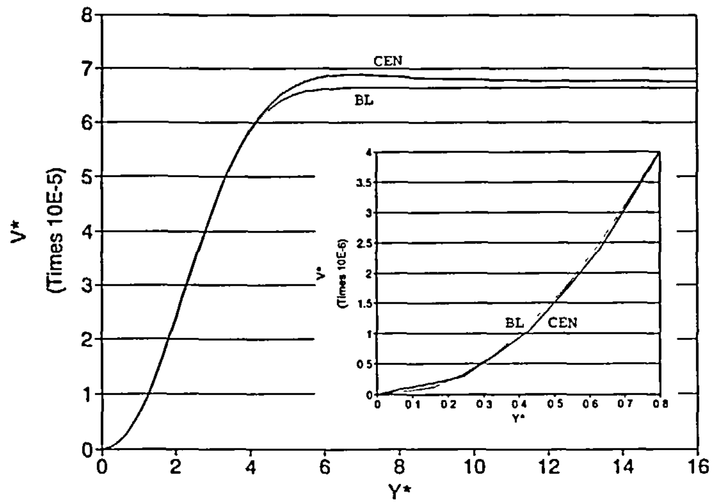

Figure 5 v-Profile for the central scheme with full artificial dissipation ($v^* = v\sqrt{x/(\nu U_{inf})}$); insert: detail at the wall. Comparison with Blasius-solution

transversal direction, the stretching factor is equal to 1.05. The mesh height of the first cell is 5×10^{-5} m. This results in an aspect ratio at the wall of 200. The height of the calculation domain is 0.017679 m.

At the inlet boundary, velocity components are prescribed according to the Blasius profile, a corresponding total temperature profile is given, whereas pressure is extrapolated from the interior. The leading edge of the plate lies at 0.2 m upstream of the inlet. The total temperature profile is calculated with a recovery factor of 0.84. So we use here classical boundary layer results[10]. At the outflow boundaries, i.e. the upper and right boundaries, pressure is imposed. At the flat plate zero velocity and zero heat flux are imposed.

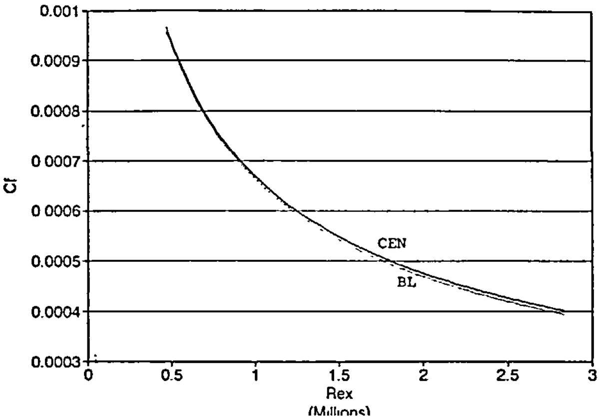

Figure 6 Skin friction for the central scheme with modified artificial dissipation.

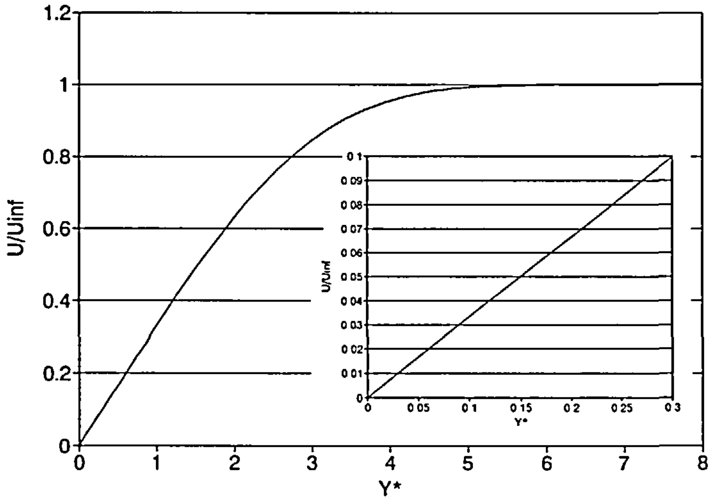

Figure 7 u-Profile for the central scheme with modified artificial dissipation; insert: detail at the wall

Figure 3 shows the skin friction coefficient in function of Re_x for the central scheme with linear extrapolation to determine the artificial dissipation. The artificial viscosity coefficient necessary to obtain stability was $\varepsilon = 0.04$. The dotted line represents the Blasius solution. The u-profile is compared with the exact Blasius profile in *Figure 4* for $Re_x = 2 \cdot 10^6$. The global correspondance is very good, but near the wall a difference between exact (dotted line) and calculated profiles is present which lays at the origin of the bad C_f-distribution in *Figure 3*. To be complete, *Figure 5* gives the v-profile which has also a deviation near the wall. The ε value to reach a monotone solution is rather high. This is due to the large mesh spacing Reynolds number in x-direction. The maximum value of the Reynolds number is $u\Delta x/\nu = 20{,}000$. This makes it again clear that an anisotropic form of artificial dissipation is necessary. The mesh spacing Reynolds number in y-direction at the middle of the plate ($Re_x = 1.5 \cdot 10^6$) and at the tenth grid point away from the wall is $v\Delta y/\nu = 0.4$. Close to the wall, this Reynolds number approaches zero.

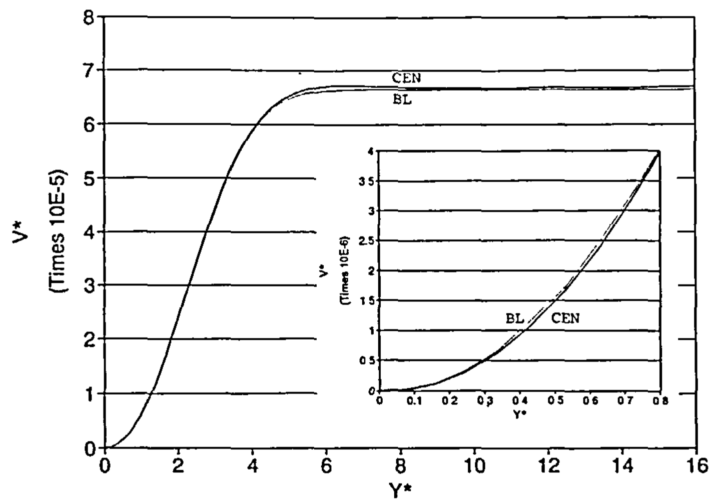

Figure 8 v-Profile for the central scheme with modified artificial dissipation; insert: detail at the wall

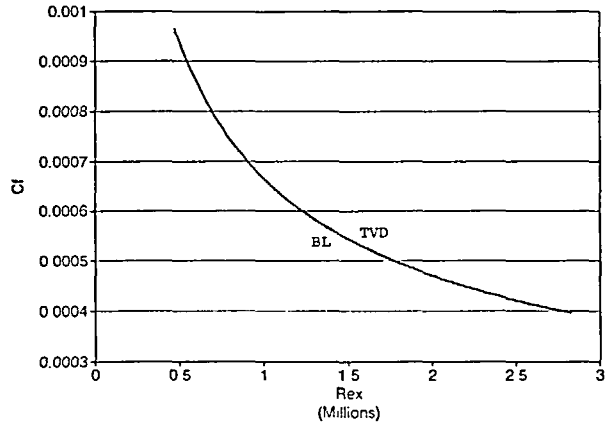

Figure 9 Skin friction for the TVD upwind scheme

When the magnitude of the dissipation is lowered in regions where viscous contributions are significant, a better solution is obtained. To illustrate this, the dissipation term was premultiplied by the factor,

$$exp\left[-\left|\frac{\tau_{xy}}{\tau_{xymax}}\right|\right]$$

where τ_{xy} is the local stress in a point and $\tau_{xy_{max}}$ is the maximum stress all over the field. Both C_f and velocity profile are closer to the analytical solution as shown in *Figures 6, 7* and *8*.

Results of the upwind TVD-scheme are shown in *Figure 9*. One clearly sees the excellent performance of a correctly scaled matrix dissipation. Also the velocity profiles are well represented and are almost identical to the correct profiles shown in *Figures 7* and *8*.

The flat plate results show that the central discretization scheme with standard artificial dissipation terms is very inaccurate. The accuracy can be much improved by lowering the artificial dissipation in regions where the physical viscosity terms are sufficient to guarantee stability. For the flat plate flow this is rather easy to implement. This is certainly not so evident for more complex flow problems. The TVD-methodology can be seen as an automatic means to adjust the artificial dissipation to the minimum necessary for stability.

CONCLUSIONS

It was shown that a central scheme with artificial dissipation basically results in poor accuracy due to excessive artificial dissipation in viscosity dominated regions of the flow field. It is necessary to steer the artificial dissipation in function of the local flow field. This is automatically realized in an upwind flux-difference TVD method resulting in very good accuracy.

ACKNOWLEDGEMENT

The research reported here was granted under contract 9.0001.91 by the Belgian National Science Foundation (N.F.W.O.) and under contract IUAP/17 as part of the Belgian National Programme on Interuniversity Poles of Attraction, initiated by the Belgian State, Prime Minister's Office, Science Policy Programming.

REFERENCES

1 Jameson, A., Schmidt, W. and Turkel, E. Numerical solutions of the Euler equations by finite volume methods using Runge-Kutta time-stepping schemes, *AIAA Paper 81-1259* (1981)
2 Allmares, S. Contamination of laminar boundary layers by artificial dissipation in Navier-Stokes solutions, *Num. Meth. for Fluid Dynamics*, 4 (Eds. M. J. Baines and K. W. Morton), Clarendon Press, Oxford, 443–449 (1993)
3 Dawes, W. N. The application of multigrid to Navier-Stokes simulation of 3D flow in axial and radial flow turbomachinery, *Int. J. Num. Meth. Fluids*, **8**, 1217–1227 (1988)
4 Arnone, A. and Swanson, R. C. A Navier-Stokes solver for turbomachinery applications, *J. of Turbomachinery*, **115**, 305–313 (1993)
5 Kunz, R. F. and Lakshminarayana, B. Three-dimensional Navier-Stokes computation of turbomachinery flows using an explicit numerical procedure and a coupled k-ε turbulence model, *J. of Turbomachinery*, **114**, 627–642 (1992)
6 Swanson, R. C. and Turkel, E. On central-difference and upwind schemes, *J. Comp. Phys.* **101**, 292–306 (1992)
7 Dick, E. Multigrid methods for steady Euler and Navier-Stokes equations based on polynomial flux-difference splitting, *Multigrid Meth. III, Int. Series on Num. Math.*, **98**, Birkhäuser Verlag, 1–20 (1991)
8 Dick, E. Multigrid solution of steady Euler equations based on polynomial flux-difference splitting, *Int. J. Num. Meth Heat & Fluid Flow*, **1**, 51–62 (1991)
9 Steelant, J. and Dick, E. A multigrid method for the compressible Navier-Stokes equations coupled to the k–ε turbulence equations, *Int. J. Num. Meth Heat Fluid Flow*, **4**, 99–113 (1994)
10 Schlichting, H. Boundary layer theory. McGraw-Hill (1979)