Two upwinding schemes for nonlinear problems in fluid dynamics

Giseli A. B. Lima, Laís Corrêa, Valdemir G. Ferreira

Instituto de Ciências Matemáticas e de Computação - USP: Av. Trabalhador sãocarlense 400, Zip code: 13560-970, São Carlos, SP, Brazil
E-mail: giabl, lacorrea, pvgf@icmc.usp.br

Abstract. The correct modeling for processes involving convection, without introducing excessive artificial damping while retaining high accuracy, stability, boundedness and simplicity of implementation continues being nowadays a challenging task for CFD practitioners. The objective of this study is to present and evaluate the performance of two new upwinding schemes, namely SDPUS-C1 and EPUS, for nonlinear convection term discretization. Both SDPUS-C1 and EPUS schemes satisfy the TVD principle of Harten and are based on the NVD formulation of Leonard. Firstly, a description of the schemes is presented and then the numerical results are provided for one- and two-dimensional hyperbolic conservation laws. Finally, as an application, the SDPUS-C1 and EPUS schemes are employed for the simulation of two-dimensional incompressible fluid flows involving moving free surfaces. The numerical experiments show that the proposed upwinding schemes perform very well.

1. Introduction
The numerical solution of complex fluid dynamics problems has emerged as a viable alternative to both experimental and analytical studies. To make the simulations of these problems more acceptable and reliable, there is an increasing demand for development, analysis and implementation of a upwinding convective scheme (in general nonlinear) that offers simplicity, accuracy, robustness and versatility. Such a scheme is particularly important when one wants to solve full transient balance laws that model incompressible viscous flows involving moving free surfaces.

In the published literature there is a variety of techniques to approximate convective terms. For instance, first order upwind schemes, such as FOU [6], are unconditionally stable, but they have a diffusive character that, in general, smooths the solution. On the order hand, classical high resolution schemes, such as central differences and QUICK [11] scheme (and its related QUICK with estimated stream terms - QUICKEST) can often produce unphysical oscillations which, in the most of the time, can lead to numerical instability. Indeed, all of the linear schemes with accuracy above first order are oscillatory, according to Godunov’s theorem.

To overcome these defects produced by the aforementioned schemes, a number of monotonic high-order upwind schemes have appeared such as, for example, the sharp and monotonic algorithm for realistic transport (SMART) [8], the variable-order non-oscillatory scheme (VONOS) [20] and an adaptative bounded version of the QUICKEST (ADBQUICKEST) [7]. In addition to this list, one may cite the monotone upstream scheme for conservation laws (MUSCL) originally pioneered by van Leer [19] and the associated limiters developed for the past twenty
years, for example the van Albada [18] (and its variants). Thus, in this work, we presented the development of two new monotonic techniques, namely Six-Degree Polynomial Upwind Scheme of $C^1$ Class (SDPUS-C1) and Eight-degree Polynomial Upwind Scheme (EPUS), for the discretization of the convection flux terms, yielding stable and accurate upwinding schemes. These schemes are nonlinear and based on the NVD formulation of Leonard [12] and satisfy the TVD [9] principle. In the development of the schemes, we have also included the ideas of Lin and Chieng [15], which say if a scheme is of $C^1$ class then it provides good results on coarse meshes. In this context, the main focus of this study is to evaluate the performance of the SDPUS-C1 (a scheme of $C^1$ class) and EPUS (a scheme of $C^2$ class) for the numerical solution of hyperbolic system of conservation laws and related fluid mechanics equations. The primary motivation for the development of simple, stable, and accurate upwinding schemes, for hyperbolic conservation equations and related fluid dynamics problems, is that there are few flux-based upwinding schemes that are relatively simple to understand and easy to implement into a new or existent computer code.

The structure of the article is as follows. In section 2, the theoretical foundations are described. Subsequently, in section 3, a description of the SDPUS-C1 and EPUS schemes is presented. In section 4, numerical results are provided for inviscid Euler and viscous Navier-Stokes equations. Finally, conclusions and future works are presented in section 5.

2. Theoretical foundations

We now concentrate on the theoretical foundations of high resolution total variation diminishing (TVD) techniques for homogeneous scalar conservation laws. In these laws, the numerical fluxes are calculated by using the modified flux approach of Harten [9], a well established technology to construct high resolution shock capturing upwinding schemes. This is accomplished by interpolating a numerical flux, say $\phi_f$, at a boundary face $f$ between two control volumes, using three neighboring grid points, namely $D$ (Downstream), $U$ (Upstream) and $R$ (Remote-upstream), labeled according to the local wind direction (upwinding). For multidimensional systems, this strategy is applied in the same fashion, with each convective derivative approximated along of the relevant variable (direction-by-direction). In summary, the numerical flux is calculated according to $\hat{\phi}_f = \hat{\phi}_f(\phi_D, \phi_U, \phi_R)$. To simplify this functional relationship, the original variables are transformed in NV as

$$\hat{\phi}_f = \hat{\phi}_f(\hat{\phi}_U).$$  \hspace{1cm} (2)

The advantage of this new formulation is that the interface value $\hat{\phi}_f$ depends of $\hat{\phi}_U$ only, since $\hat{\phi}_D = 1$ and $\hat{\phi}_R = 0$. Thus, the functional relationship can be rewritten as

$$\hat{\phi}_f = \hat{\phi}_f(\hat{\phi}_U).$$  \hspace{1cm} (3)

The normalized variable diagram (NVD), proposed by Leonard [12], is based on the definition (1) and functional relationship (2). According to Leonard [12], for $0 \leq \hat{\phi}_U \leq 1$, it is possible to derive a nonlinear monotonic third order NV scheme by imposing the following conditions: $\hat{\phi}_f(1) = 1$ and $\hat{\phi}_f(0) = 0$ (necessary conditions); $\hat{\phi}_f(1/2) = 3/4$ (a necessary and sufficient condition to reach second order of accuracy); and $\hat{\phi}_f'(1/2) = 3/4$ (a necessary and sufficient condition to reach third order of accuracy). In addition, Leonard recommends that, for values of $\hat{\phi}_U < 0$ or $\hat{\phi}_U > 1$, the scheme must be extended in a continuous manner using the FOU [6] scheme, namely

$$\hat{\phi}_f = \hat{\phi}_U.$$  \hspace{1cm} (3)
Another important convective stability criterion (a form of nonlinear stability) is the TVD constraint of Harten [9], which is a purely scalar property and ensures that spurious oscillations are removed from the numerical solution. In summary, by considering a sequence of discrete approximations $\phi(t) = \phi_i(t)$ to a scalar, the total variation (TV) at time $t$ of this sequence is defined by $TV(\phi(t)) = \sum_{i \in \mathbb{Z}} |\phi_{i+1}(t) - \phi_i(t)|$. Then, an upwinding scheme is TVD if the following condition is satisfied

$$TV(\phi^{n+1}) \leq TV(\phi^n), \quad \forall n \in \mathbb{N}.$$

After one has developed a NVD/TVD-based upwinding scheme, one may derive the associated flux-limiter by rewriting the original scheme in the following way (see, for example, Sweby [17] and Waterson and Deconinck [21]):

$$\hat{\phi}_f = \hat{\phi}_U + \frac{1}{2} \psi(r)(1 - \hat{\phi}_U),$$

where $\psi(r)$ is the flux-limiter that determines the level of antidifusividade, $r$ being a local shock sensor given by ratio of consecutive gradients as $r = \frac{\partial \phi}{\partial x}_g / \frac{\partial \phi}{\partial x}_f$. On uniform meshes, it is given by $r = (\phi_U - \phi_R) / (\phi_D - \phi_U)$ and in NV by $r = (\hat{\phi}_U) / (1 - \hat{\phi}_U)$. With the concepts introduced by Harten [9], Sweby [17] proposed, in the NVD context, restrictions for a TVD scheme. These define the TVD regions shown in figures 1 and 2. So a scheme is TVD, if the functional relationship $\hat{\phi}_f(\hat{\phi}_U)$ and flux limiter $\psi(r)$ are contained in the regions 1 and 2, respectively.

![Figure 1. TVD region for $\hat{\phi}_f(\hat{\phi}_U)$.][1]

![Figure 2. TVD region for $\psi(r)$.][2]

### 3. The SDPUS-C1 and EPUS schemes

The SDPUS-C1 and EPUS schemes are derived, in the context of NV of Leonard [12] and TVD constraint of Harten [9], by following the procedure described above. The numerical solutions obtained with these schemes can lead to accurate results (up to 3th order) away from discontinuities, as well as well defined, very steep, monotone profiles at those locations where discontinuities (or large gradients) occur in the solution. The SDPUS-C1 scheme is developed by assuming that the NV at the cell interface $f$, $\hat{\phi}_f$, are related to $\hat{\phi}_U$ as a six-degree polynomial function of the form

$$\hat{\phi}_f = \sum_{k=0}^{6} b_k \hat{\phi}_U^k,$$

for $0 \leq \hat{\phi}_U \leq 1$, and a linear function (the FOU scheme), given by (3), for $\hat{\phi}_U < 0$ or $\hat{\phi}_U > 1$. We set a free parameter (say $b_2 = \gamma$) in equation (6) and determine the other coefficients by imposing...
the four conditions of Leonard outlined above, plus the continuous differentiability condition for $\dot{\phi}_f = \dot{\phi}_f(\hat{\phi}_U)$; in another words, the equations (6) and (3) are linked on the points (0, 0) and (1, 1) with the same values to the first derivatives. So, the SDUPS-C1 scheme becomes a continuously differentiable function. It is important to observe here that, according to Lin and Chieng [15], if this property is not satisfied then convergence problems may be found in unsteady calculations when large time steps are employed. Similarly, the EPUS scheme is derived by considering a differentiable function. It is important to observe here that, according to Lin and Chieng [15], if this property is not satisfied then convergence problems may be found in unsteady calculations when large time steps are employed. Similarly, the EPUS scheme is derived by considering a continuous differentiability property for $\hat{\phi}_U = \hat{\phi}_f(\hat{\phi}_U)$, plus the condition $\phi''(0) = \phi''(1) = 0$ for obtaining $C^2$ class.

In summary, the SDPUS-C1 and EPUS upwinding schemes in NV are given by

- **SDPUS-C1:**
  \[
  \dot{\phi}_f = \begin{cases} 
  -24 + 4\gamma \hat{\phi}_U^8 + (68 - 12\gamma)\hat{\phi}_U^5 + (-64 + 13\gamma)\hat{\phi}_U^4 + \gamma \hat{\phi}_U^3 + \hat{\phi}_U, & \hat{\phi}_U \in [0, 1], \\
  20 - 6\gamma \hat{\phi}_U^4 + \hat{\phi}_U, & \hat{\phi}_U \not\in [0, 1],
  \end{cases}
  \]  
  \tag{8}

- **EPUS:**
  \[
  \dot{\phi}_f = \begin{cases} 
  -4(\lambda - 24)\hat{\phi}_U^8 + 16(\lambda - 23)\hat{\phi}_U^7 + 528 - 25\lambda \hat{\phi}_U^6, & \hat{\phi}_U \in [0, 1], \\
  + (19\lambda - 336)\hat{\phi}_U^5 + (80 - 7\lambda)\hat{\phi}_U^4 + \hat{\phi}_U, & \hat{\phi}_U \not\in [0, 1],
  \end{cases}
  \]  
  \tag{9}

where $\gamma$ and $\lambda$ are free parameters.

In the same way as was done by Ferreira et al. [7], the corresponding flux-limiter function for SDPUS-C1 is derived by combining equations (4), (5) and (8) to obtain

\[
\psi(r) = \max \left\{ 0, \frac{0.5(|r| + r)(4r^2 + 12r)}{(1 + |r|)^4} \right\}.
\]  
\tag{10}

Analogously, by combining equations (4), (5) and (9) one derives the flux-limiter for EPUS scheme, namely

\[
\psi(r) = \max \left\{ 0, \frac{0.5(|r| + r)(2\lambda - 32)r^4 + (160 - 4\lambda)r^3 + 2\lambda r^2)}{(1 + |r|)^7} \right\}.
\]  
\tag{11}

It can be seen that both SDPUS-C1 (a scheme of $C^1$ class) and EPUS (a scheme of $C^2$ class) are nonlinear and combine boundedness with second/third order of accuracy. In particular, it can be shown that SDPUS-C1 and EPUS schemes are TVD if $\gamma \in [4, 12]$ and $\lambda \in [16, 95]$, respectively. In figures 3 and 4, it is depicted the flux-limiter functions for the cases $\gamma = 12$ and $\lambda = 95$. Lima [14] and Corrêa [5] demonstrate, from numerical experimentation, that these values have provided good results for problems possessing discontinuous solutions.

4. **Numerical results**

In order to demonstrate the behavior, validity, flexibility and robustness of the SDPUS-C1 and EPUS schemes, some numerical tests will be carried out. Comparisons with reference solutions are initially assessed. The SDPUS-C1 and EPUS schemes are then used to solve incompressible Navier-Stokes equations involving arbitrarily moving free surfaces.
4.1. Hyperbolic conservation laws

Many problems in science and engineering involve quantities that are preserved and that lead to certain types of nonlinear PDEs called hyperbolic conservation laws. In the two-dimensional (2D) case, they are defined by

\[
\frac{\partial \phi}{\partial t} + \frac{\partial F(\phi)}{\partial x} + \frac{\partial G(\phi)}{\partial y} = 0, \tag{12}
\]

where \( \phi = \phi(x, y, t) : \mathbb{R}^2 \times \mathbb{R} \rightarrow \mathbb{R}^m \) is the m-dimensional vector of the conserved variables. \( F(\phi) = F(\phi(x, y, t)) \) and \( G(\phi) = G(\phi(x, y, t)) : \mathbb{R}^m \rightarrow \mathbb{R}^m \) are convective flux functions. In this study, we consider two particular cases namely, 1D and 2D Euler equations. In order to resolve these hyperbolic systems, we have been used the well established CLAWPACK (Conservation LAW PACKage) code of LeVeque [13]. This code employs the finite volume methodology and has been equipped with the SDPUS-C1 and EPUS flux-limiters.

4.1.1. Test case 1 (1D Euler equations): We consider an inviscid compressible gas modeled by 1D Euler equations (12) with \( \phi = [\rho, \rho u, E]^T \), being \( \rho \), \( u \), \( \rho u \), \( E \) and \( p \) the density, velocity, momentum, total energy and the pressure, respectively. In order to close this system, the ideal gas equation of state \( p = (\gamma - 1)(E - \frac{1}{2}\rho u^2) \) is assumed, where \( \gamma = 1.4 \) is the ratio of specific heats. Here, we solve the problem suggested in [23], which involves multiple interactions of strong shocks. The initial condition is

\[
[p_0, \quad v_0, \quad p_0]^T = \begin{cases} 
[1, \quad 0, \quad 1000]^T, & \text{if } 0 \leq x \leq 0.1, \\
[1, \quad 0, \quad 0.01]^T, & \text{if } 0.1 < x \leq 0.9, \\
[1, \quad 0, \quad 100]^T, & \text{if } 0.9 < x \leq 1.0.
\end{cases} \tag{13}
\]

The implemented boundary condition is the zero-order extrapolation (see [13]).

The reference solution was calculated on a mesh size of 2000 computational cells using the Godunov method with term correction with monotinized centered (MC) flux-limiter (see [13]). The numerical solutions were obtained on a mesh size of 1000 computational cells using the SDPUS-C1 and EPUS flux-limiters. Figure 5 depicts the reference and numerical solutions for \( \rho \) at time \( t = 0.038 \) and CFL=0.5. It can be seen that the shock is well resolved with both SDPUS-C1 and EPUS upwinding schemes, without apparition of spurious oscillations. It is also observed from this figure that the EPUS scheme provides more accurate results.
4.1.2. Test case 2 (2D Euler equations): This case considers the 2D Euler equations (12), defined on $[0,1] \times [0,1]$, with $\phi = [\rho, \rho u, \rho v, E]^T$, $F(\phi) = [\rho, \rho u^2 + p, \rho uv, (E + p)u]^T$ and $G(\phi) = [\rho v, \rho uv, \rho v^2 + p, (E + p)v]^T$. The variables have been previously defined. In order to close the system, the ideal gas equation of state $p = (\gamma - 1)(E - \frac{1}{2}\rho(u^2 + v^2))$ with $\gamma = 1.4$, was considered. In particular, we considered the shock-shock interaction problem, which consists of the interaction of two oblique shocks (states $\circ\circ$ and $\circ\circ$) with two normal shocks (states $\circ\circ$ and $\circ\circ$). The initial conditions are

$$[\rho_0, u_0, v_0, p_0]^T = \begin{cases} [1.5, 0, 0, 1.5]^T & \text{state } \circ\circ, \\ [0.13799, 1.2060454, 1.2060454, 0.0290323]^T & \text{state } \circ\circ, \\ [0.5322581, 1.2060454, 0, 0.3]^T & \text{state } \circ\circ, \\ [0.5322581, 0, 1.2060454, 0.3]^T & \text{state } \circ\circ. \end{cases}$$

Zero-order extrapolation on the boundary was used.

The reference solution was calculated using the Godunov method with a correction term containing the MC flux-limiter [13]. Both the numerical and reference solutions were calculated using the CLAWPACK code on a mesh size of 200 $\times$ 200 computational cells at CFL=0.8. Figure 6 depicts the solutions for contours of the density profile at time $t = 0.8$. It is also report in figure 7 the distribution of the density along of the diagonal ($y = x$). It is seen that the SDPUS-C1 and EPUS schemes solved well the complex structure in the solution, with the EPUS scheme providing the best results.
Figure 7. Reference solution and numerical results for Euler equations in terms of $\rho$ on the line $y = x$.

4.2. Navier-Stokes equations

The governing PDEs considered for the simulation of incompressible laminar flows involving moving free surfaces are the Navier-Stokes and mass conservation equations. In the conservative form, these equations are given by

$$\frac{\partial u}{\partial t} + \frac{1}{r^\alpha} \frac{\partial (ruu)}{\partial r} + \frac{\partial (uv)}{\partial z} = \frac{\partial p}{\partial r} + \frac{1}{Re} \frac{\partial}{\partial z} \left( \frac{\partial u}{\partial z} - \frac{\partial v}{\partial r} \right) + \frac{g_r}{Fr^2}, \tag{15}$$

$$\frac{\partial v}{\partial t} + \frac{1}{r^\alpha} \frac{\partial (r^\alpha vu)}{\partial r} + \frac{\partial (vv)}{\partial z} = \frac{\partial p}{\partial z} + \frac{1}{Re} \frac{\partial}{\partial r} \left( r^\alpha \left( \frac{\partial u}{\partial z} - \frac{\partial v}{\partial r} \right) \right) + \frac{g_z}{Fr^2}, \tag{16}$$

$$\frac{1}{r^\alpha} \frac{\partial (r^\alpha u)}{\partial r} + \frac{\partial v}{\partial z} = 0, \tag{17}$$

where $u = u(r, z, t)$ and $v = v(r, z, t)$ are, respectively, the components of velocity vector in $r$-direction and $z$-direction; $g = (g_r, g_z)^T$ is gravitational acceleration vector and $p$ is pressure. The dimensionless parameters $Re = U_0L_0/\nu$ and $Fr = U_0/\sqrt{L_0/g}$ are, respectively, the Reynolds and Froude numbers, where $\nu$ is coefficient of molecular kinematic viscosity. $U_0$ and $L_0$ are velocity and length scales, respectively. The parameter $\alpha$ in equations (15)–(16) defines the coordinate system. When $\alpha = 0$, 2D Cartesian coordinates are considered with $r$ interpreted as $x$ and $z$ interpreted as $y$, and when $\alpha = 1$ cylindrical coordinates is assumed.

4.2.1. Test case 3 (Broken dam problem): Results are presented now for the collapse of a column of water onto a horizontal wall for 2D case. Equations (15)-(17) are solved with $\alpha = 0$. By using the SDPUS-C1 scheme implemented into the 2D version of the Freeflow code of Castelo et al. [3] (a finite difference methodology on a structured grid), we performed a simulation of this unsteady moving free surface flow. The geometry used is a fluid column ($a = b = 0.057 m$) in hydrostatic equilibrium and confined between walls. Initially, a wall is instantaneously removed and the fluid is subject to gravity and free to flow out along a rigid horizontal wall. The simulation was performed by using a mesh size of $1000 \times 200$ computational cells, and with the following input data: domain of $0.5 m \times 0.1 m$; $g = 9.81 m/s^2$; $L_0 = a = 0.057 m$; $U_0 = \sqrt{g \cdot L_0} = 0.74778 m/s$; $\nu = 10^{-6} m^2/s$; and $Fr = 1$. Figure 8 shows the numerical results and the experimental data for the position of the fluid front $x_{\text{max}}$ versus non-dimensional time. In order to provide a stiffer test for the performance of SDPUS-C1, we compared the calculated surge front position (as a function of non-dimensional time) against other sophisticated techniques presented by Colagrossi and Landrini [4], namely the smoothed
particle hydrodynamics (SPH), boundary element method (BEM), level set, and an approach by Ritter. As shown in figure 8, the calculations with SDPUS-C1 agree fairly well with the data, giving us confidence in the numerical solution.

Figure 8. Computation with SDPUS-C1 scheme and experimental data for the fluid front position $x_{\text{max}}$ versus non dimensional time.

4.2.2. Test case 4 (Circular hydraulic jump problem): The equations (15)-(17) are solved with $\alpha = 1$. When an vertical free jet of liquid impinges perpendicularly onto a flat (horizontal) plate it can, for certain moderate values of the Reynolds number, create a circular hydraulic jump. The phenomenon occurs at a critical radius, where there is a sudden transition from shallow rapidly flowing fluid to deep, much slower flowing fluid. A better understanding of this phenomenon and the instabilities when it is turbulent is of commercial interest, since jet impingement is often used in cooling systems and the flow of the fluid beyond the jump can degrade the efficiency of the system.

The axisymmetric version of the Freeflow code [3] equipped with the EPUS scheme was run on this problem using three meshes, namely $200 \times 126 (\delta_x = \delta_y = 0.00025 \text{ m})$; $400 \times 252 (\delta_x = \delta_y = 0.000125 \text{ m})$ and $800 \times 504 (\delta_x = \delta_y = 0.000625 \text{ m})$ computational cells (known hereafter as Mesh I, Mesh II and Mesh III, respectively). The radius of the inlet $a = 0.008 \text{ m}$ and the velocity of the fluid at this boundary $U_0 = 3.75 \times 10^{-1} \text{ ms}^{-1}$ have been used as the scaling parameters. The jet flow rate $Q = \pi U_0 a^2 = \nu Re a = 0.75 \times 10^{-5} \text{ m}^3\text{s}^{-1}$, producing a Reynolds number of 250, and a constant inflow-to-plate distance of $H = 0.03 \text{ m}$ were employed in the simulations. We begin by verifying that the EPUS scheme provides good estimates for the position of the jump (the results with the SDPUS-C1 scheme have been similar - not shown). For this, the scaling relations for the radius of the jump $r_{\text{jump}} = (\frac{(27g^{-1/4})/(2^{-1/4}35\pi)}{2^{3/2}}Q^{2/3}H^{-1/6}\nu^{-1/3}$ of Brechet and Néda [2], and $r_{\text{jump}} = Q^{5/8}\nu^{-3/8}g^{-1/8}$ of Bohr et al. [1] were used for comparison. Table 1 shows the jump radius obtained from the simulation results and the theoretical scaling laws. One can see that the calculated estimates for the jump with the EPUS scheme on the three meshes, particularly the one on the fine mesh (Mesh III), are in reasonable agreement with the theoretical scaling law of Brechet and Néda [2].

| Scaling relations  | EPUS estimate |
|-------------------|----------------|
|                   | Mesh I | Mesh II | Mesh III |
| Brechet and Néda [2] | 1.3 e-2 | 1.8 e-2 | 1.4 e-2 |
| Bohr et al. [1]  | 5.9 e-2 | 1.6 e-2 | 1.4 e-2 |

A comparison was then performed between the total thickness of the fluid layer $h$, obtained from the numerical results and the viscous solution of Watson [22]; this is displayed in figure
9. The numerical solutions were calculated by using Meshes I, II and III at a time step of $1.3 \times 10^{-4}$s. We restricted the analysis to the region $0.2 < (r/a)Re^{-\frac{4}{3}} < 0.8$ because Watson’s analysis is only valid under the restriction $r >> a$ and the presence of the outflow-boundary (see [22]). One can clearly see from figure 9 that the numerical solutions do not show oscillatory behavior, and as the mesh size is decreased, the solution converges, indicating the convergence of the method for this complex nonlinear free surface flow problem. Watson’s approximate solution is only valid over a restricted range of $r$ and the results presented in figure 9 are extremely good over that range.

Finally, in order to check the effect of the time step on the numerical solution we compute, on the Mesh I ($200 \times 126$), the fluid layer $h$ using different time steps (from $10^{-4}$s to $10^{-6}$s). In figure 10, the numerical results of three simulations using the time steps $1.3 \times 10^{-4}$s, $6.5 \times 10^{-5}$s and $2.7 \times 10^{-6}$s are presented. It can be seen that no significant effect was detected in the numerical solutions by reducing or increasing the time step.

5. Conclusions and future work

We have presented two new TVD-based upwinding schemes (SDPUS-C1 and EPUS) for the numerical solution of nonlinear hyperbolic conservation laws and related incompressible moving free surface flows. The results of these computations clearly show that these upwinding schemes can be used to simulate hyperbolic systems and complex fluid flow problems without difficulty.

The results also clearly demonstrate that the main advantage of these schemes are their abilities to simultaneously handle calculations of slowly varying flows as well as rapidly varying flows containing shocks or discontinuities such as those commonly occurring in hydraulic jumps or breaking dams. In particular, the spurious overshoot in the vicinity of the shock may be damped by adding diffusion, i.e., simply decreasing the Courant parameter $CFL$.

It is important to be aware that there does exist another very successful class of high resolution shock-capturing schemes, namely the essentially non-oscillatory schemes (ENO) [10] (and its related weighted ENO). However, ENO schemes are generally thought of as applying to hyperbolic systems. The main point of this paper was to demonstrate that the both SDPUS-C1 and EPUS schemes can be employed to solve a wide range of difficult problems.

Encouraged by the results in this paper, the authors are planning for the future to apply SDPUS-C1 and EPUS schemes to the solution of 3D hyperbolic conservation laws and even more challenging 3D turbulent and viscoelastic flows involving moving free surfaces.
Acknowledgements
We thank the financial support given by FAPESP (Grants 2008/07367-9 and 2009/16954-8) and CNPq (Grants 300479/2008-5). This work was also carried out in the framework of the Instituto Nacional de Ciência e Tecnologia em Medicina Assistida por Computação Científica (CNPq, Brazil).

References
[1] Bohr T, Dimon P and Putkaradze V 1993 Shallow-water approach to the circular hydraulic jumps Journal Fluid Mechanics 254 635-648
[2] Brechet Y and Z Nédé 1999 On the circular hydraulic jump Am. Journal Physics 67 (8) 723-731
[3] Castelo A F, Mangiavacchi, Tomé M F, Cuminato J A, Ferreira V G and Mckee S 2001 Surface tension implementation for GENSMAC 2D Journal of the Brazilian Society of Mechanical Sciences XXIII 532-532
[4] Colagrossi A and Landrini 2003 Numerical simulation of inter facial flows by smoothed particle hydrodynamics Journal Computational Physics 191 448-475
[5] Corrêa L 2010 Solving fluid dynamics problems using a new polynomial upward convection scheme Dincon’10 9th Brazilian Conference on Dynamics Control and their Applications
[6] Courant R, Isaacson E and Rees M 1952 On the solution of nonlinear hyperbolic differential equations by finite differences Community Pure Applied Mathematics 5 243-255
[7] Ferreira V G, Kurokawa F A, Queiroz R A B, Kaibara M K, Oishi C M, Cuminato J A, Castelo A, Tom M F and Mckee S 2009 Assessment of a high-order finite difference upwind scheme for the simulation of convection-diffusion problems International Journal for Numerical Methods in Fluids 60 1-26
[8] Gaskell P H and Lau A K 1988 Curvature-compensated convective transport: SMART, a new boundedness-preserving transport algorithm International Journal for Numerical Methods in Fluids 8 617-641
[9] Harten A 1983 High resolution schemes for hyperbolic conservation laws Journal of Computational Physics 49 357-393
[10] Jiang G S and Shu C W 1996 Efficient implementation of weighted ENO scheme Journal of Computational Physics 126 202-228
[11] Leonard B P 1979 The QUICK algorithm: a uniformly third-order finite difference method for highly convective flows computational Methods Applicate Mechanics Engineering 19 59-98
[12] Leonard B P 1988 Simple high-accuracy resolution program for convective modeling of discontinuities International Journal for Numerical Methods in Fluids 8 1291-1318
[13] LeVeque R J 2004 Finite volumes methods for hyperbolic problems Press Syndicate of the University of Cambridge
[14] Lima G A B 2010 Desenvolvimento de estratégia de captura de descontinuidades para leis de conservação e problemas relacionados em dinâmica dos fluidos Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação master thesis
[15] Lin H and Chiang C C 1991 Characteristic-based flux limiters of an essentially third-order flux-splitting method for hyperbolic conservation laws International Journal for Numerical Methods in Fluids 13 287-307
[16] Rai A , Dandapat B S and Poria S 2008 Circular hydraulic jump in generalized-Newtonian fluids arXiv:0809.2231v3 [physics.flu-dyn]
[17] Sweby K O 1984 High resolution schemes using flux-limiters for hyperbolic conservation laws SIAM Journal on Numerical Analysis 21 995-1011
[18] van Albada G D, van Leer B and Roberts W W A comparative study of computational methods in cosmic gas dynamics Astronomy and Astrophysics 108 76-84
[19] van Leer B 1977 Towards the ultimate conservative difference scheme IV a new approach to numerical convection Journal of Computation Physics 23 263-275
[20] Varonos A and Bergeles G 1988 Development and assessment of a variable-order non-oscillatory scheme for convection term discretization International Journal for Numerical Methods in Fluids 26 1-16
[21] Waterson N P and Deconinck H 2007 Design principles for bounded higher-order convection schemes - a unified approach Journal of Computation Physics 224 182-207
[22] Watson E J 1964 The radial spread of a liquid jet over a horizontal plane Journal of Fluid Mechanics 20 481-499
[23] Woodward P and Collela P 2002 The numerical simulation of two-dimensional fluid flow with strong shocks Journal Computational Physics 149 439-456