Conjugate filters with spectral-like resolution for 2D incompressible flows

Yun Gu a, Y.C. Zhou a, G.W. Wei a,b,*

a Department of Computational Science, National University of Singapore, Singapore 117543
b Department of Mathematics, Michigan State University, East Lansing, MI 48824, USA

Received 28 February 2002; received in revised form 9 July 2002; accepted 15 July 2002

Abstract

A conjugate filter oscillation reduction scheme originally developed for compressible flows and in general for hyperbolic conservation laws is applied to the solution of the incompressible Navier–Stokes equation with periodic boundary conditions. Conjugate low-pass and high-pass filters are constructed by using a local spectral method, the discrete singular convolution algorithm. A spectral-like resolution, i.e., near the machine precision obtained at a sampling rate close to the Nyquist limit (2 points per wavelength), is achieved in treating a smooth initial value problem which admits an exact solution. The spectral-like resolution is enhanced by the use of conjugate low-pass filters in treating the double shear layer and multi-shear layer problems, which exhibit extremely small flow features.

© 2002 Elsevier Science Ltd. All rights reserved.

Keywords: Incompressible flows; Conjugate filters; Discrete singular convolution; Local spectral method; Spectral resolution

1. Introduction

In this work, we develop a conjugated filter approach to solve the Navier–Stokes equation in its primitive variable, describing an incompressible fluid flow

\[
\frac{\partial \hat{u}}{\partial t} + u \cdot \nabla u = -\nabla p + \frac{1}{Re} \nabla^2 u \\
\nabla \cdot u = 0,
\]

(1)

*Corresponding author. Address: Department of Mathematics, Michigan State University, East Lansing, MI 48824, USA.

E-mail address: wei@math.msu.edu (G.W. Wei).
where \( \mathbf{u} \) is the velocity vector having its \( x \)- and \( y \)-components \( u(x,y,t) \) and \( v(x,y,t) \), respectively. Here, \( p \) is the pressure. The equation is defined in a 2D square domain \([0,2\pi] \times [0,2\pi]\), with periodic boundary conditions, i.e., the Taylor problem. However, our conjugated filter scheme can be applied to higher dimensional problems with complex boundary conditions.

Numerical simulation of fluid flows is a non-trivial task even with high-performance computers. For compressible flows, the possible presence of shock wave invokes great difficulties in the numerical solution of compressible Navier–Stokes equations. For example, many sophisticated shock-capturing schemes may encounter difficulties for the interaction of turbulent boundary layer and shock [1], and in freely decaying (or homogeneous) turbulence [2]. For incompressible fluid flows, complex geometries, non-smooth initial values and high Reynolds numbers can create severe challenges in solving the Navier–Stokes equation. For example, with a non-smooth initial value, the velocity fields of Eq. (1) can rapidly evolve into singularities during the time integration. The spatial scale of vorticity structures becomes smaller and smaller. As spectral methods can have the highest numerical resolution in their own spectral domains, it may appear that the use of spectral methods is the solution to this problem. However, this is not the case. Due to the non-linearity of the governing equation, the center of the power spectral distribution of the solution continuously shifts towards the high frequency end during the time evolution although the total energy is conserved. Gradually, it becomes very difficult to fully resolve the small-scale flow features on a given computational mesh. As a consequence, the error in high frequency responses will rapidly accumulate and lead to numerical instability in the time integration. Although one can in principle adapt a sufficiently refined grid to better resolve high frequency components by using spectral methods, a long time integration will eventually collapse with a finite grid due to the Nyquist limit.

To enhance the numerical resolution on a given grid, one has to effectively control the high frequency error accumulation during the time integration. The use of shock capturing schemes for high frequency error reduction in incompressible flows was pioneered by Bell et al. [3], and by E and Shu [4]. Bell et al. introduced a shock-capturing scheme for incompressible Euler flows [3]. It is an interesting application as shock-capturing schemes are designed for compressible flows, or in general, for hyperbolic conservation laws. Indeed, the second-order Godunov scheme used by Bell et al. [3] works very well for resolution enhancement in an incompressible flow. Subsequently, E and Shu investigated the resolution enhancement by using their high-order essentially non-oscillatory (ENO) scheme [4]. The projection to divergence-free velocity fields was achieved by a fourth-order central difference scheme and a mild high-order filtering. Their method was validated for a smooth initial value, the Taylor problem, for which the exact solution is available. In a dramatical case, a discontinuous initial data, they demonstrated that a stable solution up to 10 time units could be resolved. Recently, Liu and Shu [5] tested their high-order discontinuous Galerkin method for 2D incompressible flows in the vorticity stream-function formulation. Essentially, all successful shock-capturing schemes, including the non-linear filters proposed by Engquist et al. [6], are capable of controlling high frequency errors and thus, enhancing the numerical resolution when they applied to incompressible flows. Linear phase filters are generally desirable because they are free of dissipation errors in association with the first-order derivative and dispersion errors in association with the second-order derivative. Moreover, it is important to use higher-order schemes which have correct frequency responses for a wider frequency range. Therefore, high-order shock-capturing schemes should perform better in resolution enhancement for incompressible flows.
The purpose of the present paper is to study the performance of a recently developed conjugate filter oscillation reduction (CFOR) scheme [7] for incompressible flows. The essential idea of the CFOR scheme is to effectively eliminate the numerical errors of the high-pass filters at the high frequency region by using a conjugated low-pass filter. This set of high-pass and low-pass filters are conjugated in the sense that they are derived from one generating function and consequently have essentially the same degree of regularity, smoothness, time-frequency localization, effective support and bandwidth. The CFOR scheme was extensively validated [7–9] for a variety of hyperbolic conservation law systems, such as the wave equation and Burgers’ equation with Riemann type of initial values, the Sod and Lax problems, the shock entropy wave interaction, the flow past a forward facing step, the double Mach reflection and the 2D shock vortex interaction. In our work, all conjugated filters are constructed by using a local spectral method, the discrete singular convolution (DSC) algorithm [10], which was proposed as a potential approach for the computer realization of singular integrations. By appropriately choosing DSC kernels, the DSC approach exhibits global methods’ accuracy for integration and local methods’ flexibility in handling complex geometries and boundary conditions. The DSC algorithm was very successful in solving the incompressible Navier–Stokes equation [11–13]. As DSC kernels are either symmetric or antisymmetric, all CFOR filters are of linear phase. Consequently, they do not admit any dissipation error for approximating the advective flow motion and dispersion error for velocity diffusion (or pressure prediction). However, like the spectral method, the DSC algorithm alone encounters numerical instability when applied to incompressible flows with discontinuous initial values. Therefore, we resort to the CFOR scheme to control the numerical error accumulation and maintain the numerical stability.

The organization of the present paper is as follows. Section 2 is devoted to the spatial and temporal discretizations. The CFOR scheme and the DSC algorithm are briefly reviewed. Numerical validation and illustration of the CFOR scheme for incompressible flows are presented in Section 3. The spectral-like accuracy of the CFOR scheme is demonstrated by using the Taylor problem with high wavenumbers. Non-smooth initial values are constructed with double shears and four shears to examine the performance of the present scheme. Concluding remarks are summarized in Section 4.

2. Methodology

2.1. Discrete singular convolution filters

Singular convolutions appear in many science and engineering problems. DSC is a method for the computer realization of singular convolutions. In this work, we are interested in the numerical approximation of the singular kernels of the delta type

\[ T(x) = \delta^{(n)}(x), \quad (n = 0, 1, 2, \ldots). \]  

Here, kernel \( T(x) = \delta(x) \) is the delta distribution. Higher-order kernels, \( T(x) = \delta^{(n)}, (n = 1, 2, \ldots) \) are defined in terms of distribution derivatives. These kernels are singular and cannot be directly digitized in computers. Hence, it is crucial to construct sequences of approximations \( (T_a) \)
\[
\lim_{x \to x_0} T(x) = T(x), \quad (3)
\]

where \(x_0\) is a generalized limit. A typical example used in the DSC algorithm is the regularized Shannon’s delta kernel (RSK)

\[
\delta_{\sigma, \Delta}(x - x_k) = \frac{\sin \frac{\pi}{\Delta} (x - x_k)}{\frac{\pi}{\Delta} (x - x_k)} e^{-\frac{(x-x_k)^2}{2\sigma^2}}, \quad (4)
\]

where \(\Delta\) is the grid spacing. Since the truncation error is very small due to the use of the Gaussian regularizer, the expression given by Eq. (4) is practically a finite impulse response (FIR) low-pass filter and has an essentially compact support for numerical interpolation. The derivatives of \(\delta_{\sigma, \Delta}(x - x_k)\) can be carried out analytically as the RSK is \(C^\infty\)

\[
\delta_{\sigma, \Delta}^{(n)}(x - x_k) = \left( \frac{d}{dx} \right)^n \delta_{\sigma, \Delta}(x - x_k), \quad (n = 1, 2, \ldots). \quad (5)
\]

Expressions in Eq. (5) are high-pass filters as their filter responses vanish at zero frequency, see Fig. 1. Since both the low-pass filter and high-pass filters are constructed by using the DSC delta kernel, they have essentially the same degree of regularity, smoothness, time-frequency localization, effective support and bandwidth. Therefore, we refer this set of filters as conjugated filters.

In the DSC algorithm, a function and its \(n\)th order derivative are approximated as

\[
f^{(n)}(x) \approx \sum_{k=-M}^{M} \delta_{\sigma, \Delta}^{(n)}(x - x_k) f(x_k) \quad (n = 0, 1, 2, \ldots), \quad (6)
\]

where \(\delta_{\sigma, \Delta}(x - x_k)\) is a collective symbol for DSC kernels, and \(2M + 1\) is the computational bandwidth. Numerical solution of differential equations can be easily implemented in a collocation scheme by using Eq. (6).

Fig. 1. Frequency responses of the conjugate DSC filters (in the unit of \(\pi/\Delta\)). The maximum amplitude of the filters is normalized to the unit. Stars: conjugate low-pass filter; solid line: first-order high-pass filter; dash-dots: second-order high-pass filter; small dots: ideal filters.
2.2. Oscillation reduction scheme

Fig. 1 illustrates the frequency responses of the conjugate DSC low-pass filter, the first- and second-order high-pass filters at $\sigma = 3.2\Delta$. Indeed, all the conjugate filters have essentially the same effective bandwidth, which is about $0.7 (\pi/\Delta)$. The frequency responses of all the conjugate filters are very accurate for the frequency below $0.7 (\pi/\Delta)$. However, in the very high frequency region, frequency response of both the low-pass filter and the first-order high-pass filter is under estimating, whereas that of the second-order high-pass filter is over estimating. Obviously, the limited numerical resolution of the high-pass filters leads to significant high frequency errors of a function during solving a differential equation. Such high frequency errors will be accumulated and amplified in iterations and cause numerical instability. The proposed idea is to use the conjugated low-pass filter to intelligently eliminate the high frequency errors produced by the conjugate high-pass filters during a numerical computation

$$v^{n+1} = H(u^n),$$

$$u^{n+1} = \begin{cases} v^{n+1} & \delta \mathcal{W}^{n+1} < \eta \\ G(v^{n+1}) & \delta \mathcal{W}^{n+1} \geq \eta, \end{cases}$$

where $H$ refers to the treatment by DSC high-pass filters, $\delta^{(q)}_{\sigma,\pi/\Delta} (q = 1, 2)$, and $G$ represents the convolution with the DSC low-pass filter $\delta_{\sigma,\pi/\Delta}$. Here $\eta$ is a threshold value [7,8], and $\mathcal{W}$ is a high-pass measure, which is defined via a multi-scale wavelet transform of a set of discrete function values [7]. The implementation of DSC filters is very simple as explicitly shown in Eq. (6). As a consequence, the resulting numerical calculations are correct and reliable for the frequency below the effective bandwidth of conjugated filters. Since the DSC low-pass filters are interpolative, it is necessary to implement them via prediction $[u(x_k) \rightarrow u(x_k+(1/2))]$ and restoration $[u(x_k+(1/2)) \rightarrow u(x_k)]$. The use of such adaptive filters is computationally efficient because it does not need to judge local flow directions as a Riemann solver does. Numerical demonstration of the present scheme for incompressible flows is presented in the rest of this paper.

We solve the Navier–Stokes equations (1) in terms of velocity and pressure fields, to which the conjugated low-pass filter is applied. A third-order Runge–Kutta scheme is used for time advancement in the present work, although an implicit scheme may also be easily implemented. All the spatial variables of the governing equations are discretized by using the DSC algorithm. A fractional time step and potential function method [14,12,13] is adopted to overcome the difficulty in the treatment of the pressure field in incompressible flows. A standard conjugate gradient solver is utilized for solving the resulting Neumann–Poisson equation. It is straightforward to extend the conjugated filters to higher-order spatial dimensions by tensorial products.

3. Validation and numerical examples

In this section we consider a few examples to explore the performance of the CFOR scheme for the incompressible Navier–Stokes equations (1). Some of these examples have been considered by the previous researchers [3–5] to illustrate their methods.
Example 1. Let us consider the following initial values for the velocity fields

\[
\begin{align*}
    u(x, y, 0) &= -\cos(kx) \sin(ky), \\
    v(x, y, 0) &= \sin(kx) \cos(ky),
\end{align*}
\]

where \( k = 1, 2, \ldots \) is the wavenumber. This example is employed to check computational accuracy and numerical order of the CFOR scheme for smooth solutions. The corresponding exact solution is given by

\[
\begin{align*}
    u(x, y, t) &= -\cos(kx) \sin(ky) e^{-2k^2t/Re}, \\
    v(x, y, t) &= \sin(kx) \cos(ky) e^{-2k^2t/Re}, \\
    p(x, y, t) &= -\frac{1}{4}[\cos(2kx) + \cos(2ky)] e^{-4k^2t/Re}.
\end{align*}
\]

Obviously, on a given finite mesh \( N \times N \), the ratio of grid points per wavelength (PPW) decreases as the \( k \) value increases, see Fig. 2. Moreover, for a finite Reynolds number, \((Re \neq \infty)\), the Fourier spectrum distribution of the solution has a Lorentz lineshape. As the time increases, the width of the frequency peak becomes broader and reaches higher wavenumbers. Consequently, the problem becomes more difficult to solve for a large \( k \) and/or a long time, and constitutes a severe test for the numerical resolution of a potential scheme.

We first consider an easy case, \( k = 1 \). This case was used by E and Shu to test the accuracy of their third-order ENO scheme [4]. We choose \( \Delta x = \Delta y = 1/N \) with \( N = 16 \). The Navier–Stokes equations (1) with \( Re = 10^4 \) is integrated up to \( t = 2 \). In Table 1, we list the \( L_\infty \) errors. Obviously, the present CFOR scheme has reached the machine precision at a small mesh of \( 16^2 \). The accuracy
Table 1

$L_\infty$ error of the Taylor problem ($Re = 10^4$, $t = 2$). The PPW ratio is evaluated according the pressure wave, i.e., $N/2k$

<table>
<thead>
<tr>
<th>N</th>
<th>k</th>
<th>1</th>
<th>3</th>
<th>5</th>
<th>6</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>Error</td>
<td>2.29E–11</td>
<td>1.49E–4</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>PPW</td>
<td>8</td>
<td>2.67</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>32</td>
<td>Error</td>
<td>2.44E–15</td>
<td>5.33E–15</td>
<td>3.45E–9</td>
<td>4.54E–8</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>PPW</td>
<td>16</td>
<td>5.33</td>
<td>3.2</td>
<td>2.67</td>
<td>–</td>
</tr>
<tr>
<td>64</td>
<td>Error</td>
<td>2.50E–15</td>
<td>5.85E–15</td>
<td>3.79E–14</td>
<td>4.66E–12</td>
<td>1.70E–6</td>
</tr>
<tr>
<td></td>
<td>PPW</td>
<td>32</td>
<td>10.67</td>
<td>6.4</td>
<td>5.3</td>
<td>2.67</td>
</tr>
</tbody>
</table>

Fig. 3. The vorticity contour for (11) with $\rho = \pi/15$, $\Delta t = 0.002$, $N = 64$. 
of the CFOR scheme is apparently limited by the double precision algorithm used in the computation.

To test the capability of the CFOR scheme, we next consider a number of large \( k \) values with a variety of cell numbers. We still choose \( Re = 10^4 \). On a finer grid \( (N = 32) \), the present scheme is already accurate enough to support the numerical integration with \( k = 6 \) as shown Table 1. It is noted that the numerical order of the present scheme is as high as 34.7, indicated by the case of \( k = 3, N = 16, \) and 32. An alternative representation for the resolution of a numerical scheme is its Fourier analysis. The frequency response of the DSC scheme and the spectrum distribution of two typical pressure waves \( p(x) = -(1/4)(\cos(2kx) + 1.0)e^{-8k^2/Re} \) with \( k = 6, 12 \) are plotted in Fig. 2. It is seen that the spectrum distribution of the pressure wave is wide-banded and has its

![Image of vorticity and velocity contours](image_url)

**Fig. 4.** (a) The vorticity contour for (11) with \( \rho = \pi/15, \Delta t = 0.001, N = 128. \) (b) Contours of velocity \( u \). (c) Contours of velocity \( v \). (d) Contours of pressure \( p \).
peak values at wavenumbers 0 and $2k$. As the differentiation error grows towards the high wavenumber end, the accuracy of the numerical solution would degenerate significantly with the increasing of the wavenumber $k$, as shown in Table 1. We also employ the ratio of grid PPW, a concept commonly used in computational electromagnetics, to measure the resolution of present CFOR scheme. The PPW gives good measure of the numerical resolution of a computational method in case the involved frequency distribution of a problem is relatively focussed at a peak value. We compute the PPW value according to the wavenumber of the second pressure peak, i.e., $2k$. For $k = 6$ and $N = 32$, the PPW ratio is about 2.67. While limited by applicable boundary conditions, usually, the Fourier spectral method could provide the lowest PPW ratio for approximating sinusoidal waves, i.e., 2 PPW, the Nyquist rate. When the mesh of $64^2$ is used, the CFOR
The scheme is still accurate to 6 significant figures at 2.67 PPW. Such an accuracy is quite close to the spectral accuracy and is the best ever reported for this problem. Unlike the Fourier spectral method, the present approach can be used for complex boundary conditions and geometries [12,13].

**Example 2.** Having established our confidence on the use of the CFOR scheme for treating smooth initial values, we further consider a case which does not admit an analytical solution. The initial condition of the Euler equation ($Re = \infty$) consists of a double shear of form [3,4],

\[
u(x, y, 0) = \begin{cases} 
\tanh((y - \pi/2)/\rho) & y \leq \pi \\
\tanh((3\pi/2 - y)/\rho) & y > \pi,
\end{cases}
\]  

(11)
where $\delta = 0.05$ and $\rho = \pi/15$. The initial flow field consists of a horizontal shear layer of finite thickness described by the tanh function, perturbed by a small amplitude vertical velocity described by the sine function. Physically, each of the shear layers making up the boundaries of a jet flow evolves into a periodic array of large vortices, with the shear layer between the rolls being thinned by the large straining field. Eventually, these thinned layers wrap around the large rolls. The time evolution of the top and bottom layers are half-period shifted mirror images of one another as determined by the symmetry and boundary condition of the problem. The vorticity fields develop two large-scale singularities in the computational domain.
For this problem, the Fourier spectral method was found to produce wriggles in its vorticity fields obtained by using the mesh of $512^2$ points \cite{4}. In Figs. 3 and 4(a), we plot the vorticity contours obtained by using $64 \times 64$ and $128 \times 128$ meshes respectively. Before $t = 6$, the vorticity fields are well resolved on both grids. As time progresses, the shear layers between the vortices become thinner. The coarser grid of $64^2$ can still give smooth and stable results. However, on the finer grid $128^2$, the CFOR scheme is able to resolve great details of the solution even at $t = 10$. Fig. 4(b)–(d) show the contours of $u$, $v$ and $p$, respectively. Some discontinuities in the velocity fields can be visualized at late times. As the solution of an elliptic problem should be, the pressure field is much smoother compared to the corresponding velocity fields.

The kinetic energy of the Euler system, defined by

$$K = \frac{1}{2} \int (u^2 + v^2) \, dx \, dy$$

is an invariant quantity. Its variation in time for computed solutions is a gross measure of the resolution. In Fig. 5, we depict the time history of the kinetic energy. Apparently, the finer grid preserves the kinetic energy much better.

**Example 3.** As the CFOR scheme is able to provide very high resolution with a coarse grid, we explore its capability of handling a multi-shear problem of the form

\[
\begin{align*}
  u(x, y, 0) &= \begin{cases} 
    \tanh[(y - \pi/4)/\rho] & y \leq \frac{\pi}{2} \\
    \tanh[(3\pi/4 - y)/\rho] & \frac{\pi}{2} < y \leq \pi \\
    \tanh[(y - \pi/2)/\rho] & \pi < y \leq \frac{3\pi}{4} \\
    \tanh[(3\pi/2 - y)/\rho] & y > \frac{3\pi}{4}
  \end{cases} \\
  v(x, y, 0) &= \delta \sin(2x),
\end{align*}
\]

Fig. 5. The time history of the kinetic energy.
where $\delta = 0.05$ and $\rho = \pi/15$. Obviously, this initial value is more complex than the earlier case and having more shear boundary layers. It is very difficult for a low-order shock-capturing scheme to preserve the numerical resolution with a small mesh. We test our approach with a $128^2$ mesh and results are plotted in Fig. 6 for $t = 4, 6, 8$ and $10$. As the amplitude of the vertical velocity perturbation is relatively twice as large as that used in the earlier case, the system evolves much faster compared with the double shear case. It is more difficult to resolve the fine roll-up structures up to $t = 10$. Obviously, the CFOR scheme performed very well for this Euler problem.

**Example 4.** Our final set of test calculations are for the same initial value given by Eq. (11), except that the shear layers bounding the jet are infinitely thin, i.e., $\rho \to \infty$ in (11). Long-time integration of this case is quite challenging without an efficient numerical scheme. In addition to providing a
stringent test of the proposed scheme, this example is also motivated by the widespread practice of using finite difference algorithms for the Euler equation to calculate the dynamics of sharp shear layers for compressible flows. The justification usually given for the physical correctness of such calculations is that if the point at which the shear layer is created is computed correctly, then the numerical dissipation in the finite difference algorithm will mimic the physical dissipation mechanism, and lead to results which have the correct large-scale dynamics. In the present approach, instead of using the numerical dissipation, spurious oscillations are cleaned by a low-pass filter. The capability of the present CFOR scheme to compute Euler solutions for discontinuous data would validate our approach for being used in viscosity free physical models.

Fig. 7. (a) The vorticity contour for (11) with $\rho \to \infty$, $\Delta t = 0.001$, $N = 128$. (b) Contours of velocity $u$. (c) Contours of velocity $v$. (d) Contours of pressure $p$. 
In Fig. 7(a), we plot vorticity contours of solutions obtained by using the CFOR scheme. The results serve to validate our claim that we have no Reynolds number stability restrictions. In particular, our CFOR scheme remains stable, even with discontinuous initial data. Fig. 7(b)–(d) display the contours of $u$, $v$ and $p$, respectively. Obviously, singularities have developed in all flow fields.

4. Conclusion

In conclusion, a novel approach, the CFOR scheme [7,9] is introduced for solving the incompressible Navier–Stokes equation. The CFOR scheme was originally proposed for solving
Burgers’ equation with all possible Reynolds numbers and for treating hyperbolic conservation laws. The essence of the scheme is to adaptively implement a conjugated low-pass filter to effectively remove the accumulated numerical errors produced by a set of high-pass filters. As all conjugated filters are generated by using DSC kernels [10], they have essentially the same degree of regularity, smoothness, time-frequency localization, effective support and bandwidth.

The incompressible Navier–Stokes equations are solved in primitive variables by using the CFOR scheme. A third-order Runge–Kutta scheme is utilized for the time advancement. A modified marker and cell scheme is employed to result in the Neumann–Poisson equation for the pressure, whose iterations are achieved by the standard conjugate gradient method.
The computational accuracy and numerical resolution are examined by using a few benchmark examples. The first example is a smooth initial value problem which admits an exact solution, and thus provides an objective calibration of potential numerical schemes. In particular, it is difficult to solve the problem with a very high wavenumber in the initial wave. In the present study, we choose a variety of wavenumbers ranging from 1 to 12 to test the capability of the CFOR scheme. In some situations, the ratio of grid PPW is close to the Nyquist limit of 2. The performance of CFOR scheme near the Nyquist rate confirms its nature of spectral-like resolution for solving the incompressible Navier–Stokes equation. The second example is the standard double shear layer problem which was considered in the pioneer works of Bell et al. [3], and of E and Shu [4]. The present scheme works very well for this problem even with a small mesh. To further illustrate the

Fig. 7 (continued)
potential of the CFOR scheme, we construct a four-shear initial value, which is much more
difficult to handle with a coarse mesh. Excellent results are obtained for this problem. Finally, we
consider the problem of discontinuous initial double shear layers. The CFOR scheme remains
stable and reasonable solution is obtained up to time 10. Some fine details of circulating roll-ups
can be resolved.

Acknowledgement

This work was supported in part by the National University of Singapore.

References

Fluids 2002;31:245–68.
[4] E W, Shu C-W. A numerical resolution study of high order essentially non-oscillatory schemes applied to
37.
press.
1999;110:8930–42.
[12] Wan DC, Patnaik BSV, Wei GW. A new benchmark quality solution for the buoyancy driven cavity by discrete