The space–time upwind cell-vertex scheme for conservation laws: a Riemann solver-free Approach

Shuangzhang Tu*, Shahrouz Aliabadi

Clark Atlanta University, Atlanta, GA 30314, USA

Abstract

This paper presents a novel space–time upwind cell-vertex scheme (STU-CVS) for conservation laws. The new scheme is inspired by the revolutional space–time conservation element/solution element (CE/SE) method that utilizes a staggered space–time mesh to enforce the space–time flux conservation. It will be shown that the new scheme is independent of the underlying mesh, and the implementation of the new scheme becomes easier and more clear compared with the original CE/SE method. We also explore the inherent upwind nature of the new scheme as well as the CE/SE method. The inherent upwind nature eliminates the need of using Riemann solvers for advection-dominant problems. The numerical dissipation of the new scheme as well as the CE/SE method is small. Finally, we provide some preliminary numerical results to demonstrate the performance of the new scheme.

Keywords: Space-time; Upwind; Conservation law; Riemann solver-free and Euler equations

1. Introduction

About a decade ago, the space–time conservation element/solution element (CE/SE) method for conservation laws was invented by Chang and To [1]. The CE/SE method is revolutional, not evolutional, in many senses. The space–time CE/SE method has many non-traditional features. It introduces the concepts of conservation elements (CEs) and solution elements (SEs). The space–time domain is filled by nonoverlapping CEs. The boundary of each CE is divided into several parts, with each belonging to a unique SE. Linear variations of the solution are assumed within each SE. The final formulation is a result of enforcing the space–time flux conservation on a staggered space–time mesh. The CE/SE method is a Riemann solver-free approach. Even though it is a second-order scheme in both space and time, its numerical dissipation is extraordinarily small, making it capable of accurately handling both strong shocks and small disturbances simultaneously. The CE/SE method is able to simulate flows with very low Mach number without using any type of preconditioning technique.

Compared with other traditional numerical schemes using (approximate) Riemann solvers, the algorithm using the CE/SE method is extremely simple and highly accurate. We have many reasons to believe that the CE/SE method will be recognized as a new milestone on the evolution path of computational fluid dynamics (CFD). Surprisingly, despite the excellent performance of the CE/SE method, it seems that this scheme has not been well known in the CFD community. Most publications related to the CE/SE method are authored or coauthored by Chang, the inventor of the CE/SE method. See Chang and Wang [2] for an abundant list of those publications.

The original CE/SE method defines different CEs and SEs for rectangular and triangular meshes. In this paper, we will give new definitions of the CE and SE that are independent of the underlying mesh. The resultant new scheme is thus suitable for arbitrarily unstructured meshes. The new scheme in this paper will be named as a space–time upwind cell-vertex scheme (STU-CVS). We also explain why Riemann solvers are not necessary in STU-CVS as well as the CE/SE method by exploring both schemes' inherent upwind nature. Finally, we will provide some preliminary results to demonstrate the performance of the new scheme.

* Corresponding author. Tel.: +1 404 880 8898; E-mail: shuangzhang.tu@jsums.edu

Computational Fluid and Solid Mechanics 2005
K.J. Bathe (Editor)

2. The space–time upwind cell-vertex scheme

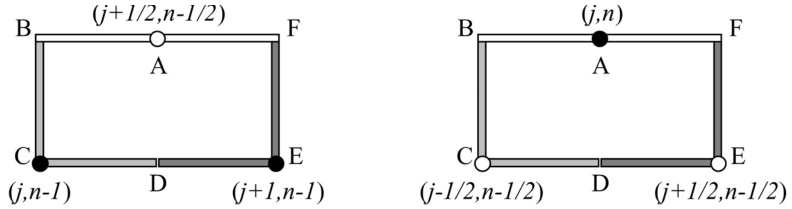
The present STU-CVS inherits the core idea of the CE/SE method that uses the staggered space–time mesh to enforce the space–time flux conservation. However, the staggered space–time mesh is realized through the alternate solution updating at cell centroids and vertices within a time step, which is distinct from the original CE/SE method.

In STU-CVS, we divide a single time step into two half time steps. Thus, $n = 1, 2, 3 \ldots$ represents each new time level and $n = \frac{1}{2}, \frac{3}{2}, \frac{5}{2} \ldots$ represents the half time level in each time step. We store unknowns on both vertices and cell centroids. However, the solutions at vertices and cell centroids are updated on different time levels. Suppose the initial conditions are given on each vertex of the mesh. We can march the solution in time in an alternating way. At each half time level (*cell level*), the solutions at cell centroids are updated using the known vertex solutions at previous old time level. At each new time level (*vertex level*), the solutions at vertices are updated using the known cell-centroid solutions at previous half time level.

Fig. 1 illustrates the conservation elements for one-dimensional (1D) mesh. The empty circles represent the cell centroids and the filled circles represent the vertices. The boundary of the CE is composed of several parts, with each part belonging to a unique SE. For example, the CE associated with $(j+\frac{1}{2}, n-\frac{1}{2})$ is a rectangular region connecting *ABCDEF* (see Fig. 1a). On the boundary of this CE, $\overline{BC}$ and $\overline{CD}$ belong to the SE $(j, n-1)$, $\overline{DE}$ and $\overline{EF}$ belong to the SE $(j+1, n-1)$, and $\overline{FB}$ belongs to the SE $(j+\frac{1}{2}, n-\frac{1}{2})$. Within each SE, the solution is assumed to be distributed linearly, so the one-point integration rule is sufficient to evaluate the flux across the boundary. Since the solution and its derivative at $(j, n-1)$ and $(j+1, n-1)$ are known, we can update the solution at $(j+\frac{1}{2}, n-\frac{1}{2})$ by enforcing the space–time flux conservation across the boundary of its CE. Fig. 2 illustrates the conservation elements for two-dimensional (2D) triangular mesh. The same definition can be applied to the quadrilateral mesh and even hybrid meshes without any ambiguity. We can readily realize that this definition of CEs is independent of the underlying mesh and thus is suitable for any type of conforming mesh. Note that for structured rectangular meshes, our new definition of the CE is similar to the second kind of CE proposed in Zhang et al. [3].

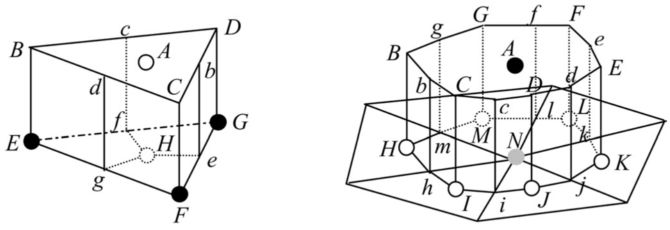
The implementation of the new STU-CVS becomes extremely simple and clear. The pseudo-code below illustrates the marching procedure.

```
call initialize_solution_at_vertices()
call initialize_gradient_at_vertices()
do its = 1, nts
  call update_solution_at_cell_
  centroids()
  call update_gradient_at_cell_
  centroids()
  call update_solution_at_vertices()
  call update_gradient_at_vertices()
enddo
```



(a) Cell level. (b) Vertex level.

Fig. 1. Illustration of the CEs of 1D meshes.



(a) Cell level. (b) Vertex level.

Fig. 2. Illustration of CEs of triangular meshes.

3. Inherent upwind nature of the new scheme

As is well known, a numerical scheme must be upwinded somehow for advection dominant problems to ensure numerical stability. In this section, we will show that the new scheme as well as the CE/SE method possess this upwind feature in an inherent way, which makes it distinct from and outstanding among many other numerical schemes where the upwinding idea is realized explicitly.

To illustrate the inherent upwind nature of the new scheme as well as the CE/SE method, we consider the simple 1D initial-value, linear advection problem

$$\frac{\partial u(x,t)}{\partial t} + a\frac{\partial u(x,t)}{\partial x} = 0 \tag{1}$$

where a is the advection speed, which is a constant. The space–time flux is $\mathbf{h} = (u, au)$.

Suppose we want to update the solution at vertex j at time level n. By enforcing the space–time flux conservation across the boundary of the CE of node (j, n) (see Fig. 1b), we can obtain the following formulation for u at (j, n):

$$u_j^n = \frac{1}{2}\left\{(1+\nu)u_{j-1/2}^{n-1/2} + (1-\nu)u_{j+1/2}^{n-1/2} + \frac{(1-\nu^2)\Delta x}{4}\left[(u_x)_{j-1/2}^{n-1/2} - (u_x)_{j+1/2}^{n-1/2}\right]\right\} \tag{2}$$

We look at the situation when the Courant number $\nu = \pm 1$, which is the stability limit of current explicit scheme; the sign of ν depends on the direction of the advection speed. From Eq. (2), we have

$$u_j^n = u_{j-1/2}^{n-1/2} \tag{3}$$

when $\nu = 1$. Eq. (3) clearly means that the solution u at time level n is simply the solution at time level $n - \frac{1}{2}$ at location $\frac{\Delta x}{2}$ upstream of the current location, which is exactly the demonstration of the theory of characteristics. A similar conclusion can be drawn when $\nu = -1$, in which the upwind direction will take the opposite direction. Therefore, the space–time formulation will automatically pick up the upwind direction. When $\nu = 0$, which never happens in reality since Δt cannot be 0 in a real simulation, then u in Eq. (2) will become the simple arithmetic average of two ingredients. For $0 < \nu < 1$, the CE/SE scheme is clearly an upwind-biased scheme. At this point, we can summarize the conclusions we have reached:

- When $\nu = \pm 1$, the CE/SE method is exactly the demonstration of the theory of characteristics. Indeed, we can expect that the a scheme tends to yield an exact solution for both u and u_x when $|\nu|$ becomes increasingly close to unity.
- When $0 < \nu < 1$, the CE/SE method is an upwind-biased scheme.

The inherent upwind nature of the new scheme as well as the CE/SE method explains why the Riemann solver is unnecessary. Therefore, many annoying numerical problems, such as the carbuncle phenomenon associated with the Riemann solver on multidimensional meshes, can be avoided. Another favorable outcome due to this inherent upwind nature is that the numerical dissipation is small in the new scheme as well as the CE/SE method and can be controlled in a more flexible way.

It is interesting to note the following: We did not intentionally enforce the upwind idea in deriving the final formulation. The upwind idea simply comes naturally out of this staggered space–time formulation. This, on the other hand, proves that the upwind idea is naturally needed for handling this type of problem.

4. Preliminary numerical results

4.1. One-dimensional linear sinusoidal wave advection problem

The first example is the 1D linear sinusoidal wave advection problem on the computational domain $[0, 2\pi]$ with the initial condition $u(x, 0) = \sin x$ and the periodic boundary conditions. The exact solution is the initial condition convected with the speed a, i.e. $u(x, t) = \sin(x - at)$. In this example, $a = 1$. We use this example to verify that the nondissipative a scheme [2] tends to yield the exact solution when the Courant number $|\nu|$ is increasingly close to unity. Table 1 shows the L_2 norm of the numerical error of u and u_x after ten time steps with different values of ν. As can be seen, the closer the value of ν is to 1, the smaller the errors for both u and u_x. Indeed, when $\nu = 1$, the solution is expected to be exact according to the theory of characteristics.

4.2. Two-dimensional isentropic vortex advection problem

The second example is the isentropic vortex advection problem on a 2D computation domain $[0, 10] \times [0, 10]$ [4]. The initial conditions are given as an isentropic vortex with the center at (5, 5), i.e.

$$\begin{aligned}
u(x,y,0) &= 1 - \frac{\epsilon}{2\pi}e^{0.5(1-r^2)}(y-5) \\
v(x,y,0) &= 1 + \frac{\epsilon}{2\pi}e^{0.5(1-r^2)}(x-5) \\
T(x,y,0) &= 1 - \frac{(\gamma-1)\epsilon^2}{8\gamma\pi^2}e^{1-r^2} \\
S(x,y,0) &= 1
\end{aligned}$$

Table 1
The scalar advection equation $a = 1$, $\Delta x = 2\pi/20$. L_2 error after ten time steps with different value of ν. Nondissipative a scheme is used

Δt	ν	$\|u^h - u\|_2$	$\|u_x^h - u_x\|_2$
0.3	0.9549296586	0.194183E–02	0.583557E–02
0.31	0.9867606472	0.599893E–03	0.180919E–02
0.314	0.9994930426	0.234305E–04	0.706811E–04
0.3141	0.9998113525	0.872320E–05	0.263146E–04
0.31415	0.9999705074	0.136409E–05	0.411494E–05
0.314159	0.9999991553	0.390692E–07	0.117857E–06
0.3141592	0.9999997920	0.962291E–08	0.290286E–07
0.31415926	0.9999999829	0.789006E–09	0.238014E–08
0.314159265	0.9999999989	0.528471E–10	0.159430E–09

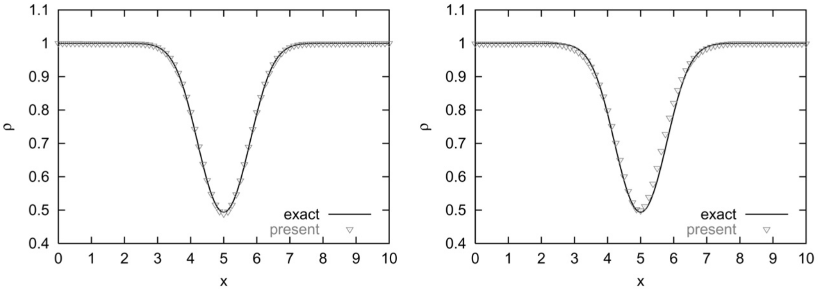
where $\epsilon = 5$ represents the vortex strength and $r^2 = (x-5)^2 + (y-5)^2$. The periodic boundary conditions on both directions are assumed. The density ρ and the pressure p can be obtained via

$$\rho(x,y,0) = \left(\frac{T(x,y,0)}{S(x,y,0)}\right)^{1/(\gamma-1)},$$
$$p(x,y,0) = \rho(x,y,0)T(x,y,0)$$

It can be verified that the Euler equations with the above initial and boundary conditions allows an exact solution that is the initial solution advected with the speed (1, 1) in the diagonal direction. Fig. 3 shows the comparison between the computed density distribution and the exact solution at the horizontal cut $y = 5$ for $T = 10$ and $T = 100$ on an 80×80 quadrilateral mesh. Equally good solutions, though not shown, can be obtained on the triangular mesh with comparable mesh resolution. If we compare the current solution with that obtained using a second-order total variation diminishing (TVD) scheme [4], we can come to the conclusion that the second-order space–time CE/SE method is superior to the second-order TVD scheme in terms of the solution resolution. The solution at $T = 100$ obtained using the second-order TVD scheme is completely unacceptable, since the peak of the vortex has been severely smeared.

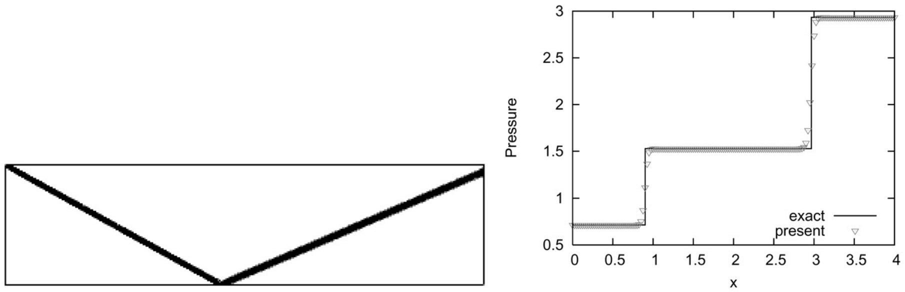
4.3. Two-dimensional shock reflection problem

This steady problem has been tested extensively using the CE/SE method [5,6]. An incident oblique shock with shock angle of 29° hit the horizontal wall and a reflected shock will turn the flow behind it back to the original freestream direction. The freestream flow is a supersonic flow of Mach 2.9. The computational domain is a 4.0×1.0 rectangle containing 160×60 uniform rectangles. The inflow conditions are $(\rho, u, v, p) = (1.0, 2.9, 0.0, 0.71428)$. The flow conditions at the top boundary are also given such that the specified incident oblique shock relations are satisfied. Fig. 4 shows the computed pressure contours and the pressure distribution at horizontal cut $y = 0.5$ compared with the exact solution. Both the incident shock and the reflected shock are captured sharply.



(a) $T = 10$. (b) $T = 100$.

Fig. 3. Vortex advection. Density at horizontal line $y = 5$. 80×80 quadrilateral mesh.



(a) Pressure contours. (b) Pressure at cut $y = 0.5$.

Fig. 4. Shock reflection. Pressure on 60×160 quadrilateral mesh.

5. Conclusions

This paper presents a novel STU-CVS for conservation laws. The STU-CVS is an extension of the revolutional space–time CE/SE method with new definitions of the CEs. The extension makes the implementation of STU-CVS easier and more clear for arbitrarily unstructured meshes, compared with the original CE/SE method. The inherent upwind feature of STU-CVS as well as the CE/SE method makes them successful for advection problems without resorting to the traditional (approximate) Riemann solvers. The preliminary results using the new scheme demonstrate that the new scheme is very promising in simulating advection-dominant problems, such as Euler equations. Applications for Navier–Stokes equations and extensions to three-dimensional cases are our ongoing work.

Acknowledgments

This work is funded in part by the Army High Performance Computing Research Center (AHPCRC) under the auspices of the Department of the Army, Army Research Laboratory contract numbers DAAD19–01–2–0014 and DAAD19–03–D–0001. Partial support for this publication made possible through support provided by NASA-Ames Research Station (NASA–AMES) and National Science Museum (NSF).

References

[1] Chang S-C, To WM. A New Numerical Framework for Solving Conservation Laws: The Method of Space–time Conservation Element and Solution Element. NASA TM 1991–104495, 1991.

[2] Chang S-C, Wang XY. Multi-dimensional courant number insensitive Ce/Se Euler solvers for applications involving highly nonuniform meshes. AIAA paper 2003–5285, 2003.

[3] Zhang Z-C, Yu ST, Chang S-C, Himansu A, Jorgenson PCE. A modified space–time conservation element and solution element method for Euler and Navier–Stokes equations. AIAA paper 1999–3277, 1999.

[4] Shu C-W. High order finite difference and finite volume WENO schemes and discontinuous galerkin methods for CFD. NASA/CR-2001–210865, May 2001.

[5] Chang SC, Wang, XY, Chow CY. The space–time conservation element and solution element method: a new high-resolution and genuinely multidimensional paradigm for solving conservation laws. J Computat Phys 1999;156:89.

[6] Zhang Z-C, Yu ST, Chang S-C. A space-time conservation element and solution element method for solving the two- and three-dimensional unsteady Euler equations using quadrilateral and hexahedral meshes. J Computat Phys 2002;175:168–199.