# The Space-Time CE/SE Method for the Navier-Stokes Equations in Three Spatial Dimensions

Zeng-Chan Zhang[1]
S.T. John Yu[2]
Mechanical Engineering
Wayne State University
Detroit, MI

Xiao-Yen Wang[3]
Aerospace Eng. & Mechanics
University of Minnesota
Minneapolis, MN

Sin-Chung Chang[4]
Philip C.E. Jorgenson[5]
NASA Glenn Research Center
Cleveland, OH

## Abstract

In this paper, we report an extension of the Space-Time CE/SE Method for the three dimensional Navier-Stokes equations. In the setting of space-time flux calculation by the CE/SE method, various methods for calculating the viscous flux terms were tried. One of the most robust methods is based on the midpoint rule, which is used in the numerical examples shown here. The newly developed three-dimensional Navier Stokes solver retains all favorable features of the original CE/SE method, i.e., high fidelity resolution of unsteady flows, easy implementation of non-reflective boundary conditions, simple computational logic, and efficient numerical operation count. Numerical results of a three-dimensional driven cavity flow and a swirling jet are reported. For such low-speed flows, no preconditioning is applied to the flow equations, and the new solver is robust for flows at all speeds.

## 1. Introduction

The Space-Time Conservation Element and Solution Element Method, or the CE/SE Method for short, originally developed by Chang and co-workers [1-10], is a novel numerical framework for conservation laws. The CE/SE method has many non-traditional features, including a unified treatment of space and time, separated definitions of conservation element (CE) and solution element (SE), the easiness of implementing the non-reflective boundary condition, and a shock capturing strategy without using a Riemann solver. In addition, the CE/SE method is a genuine multidimensional scheme because no dimensional splitting is applied to calculate fluxes, and thus the method is suitable for treating conservation laws with source terms. Moreover, the CE/SE method is based on triangles and tetrahedrons for solving flows in two and three dimensions, respectively, and it is suitable for unstructured meshes. To date, numerous solutions have been obtained, including traveling and interacting shocks, acoustic waves, shedding vortices, detonation waves, shock/acoustic waves interaction, shock/vortex interaction, shock/boundary layer interaction, and cavitating flows.

The objective of the present paper is to extend the CE/SE method for viscous flows in three spatial dimensions. In the CE/SE method, linear elements were used to discretize the flow equations in Solution Elements (SEs). Viscous terms, however, are second-order and special treatments were developed to calculate the viscous effects as an integral part of the space-time flux balance. In numerical calculations, we adopt a newly solid wall boundary treatment[9], in which the slip wall condition for the Euler equations is an asymptotic limit of the no-slip condition for viscous flows with viscosity approaching null.

To demonstrate the capabilities of the newly developed two- and three-dimensional Navier Stokes code, we have successfully calculated low-Mach-number compressible flows and incompressible flows, including the buoyancy-driven gas flows,

[1] Research associate, AIAA Member, Email: zz9@students.eng.wayne.edu
[2] Associate Professor, AIAA Member, Email: styu@me1.eng.wayne.edu
[3] Visiting Scientist, Email: wangxy@aem.umn.edu
[4] Senior Aerospace Engineer, Email: sin-chung,chang@lerc.nasa.gov
[5] Senior Aerospace Engineer, Email:aejorgen@lerc.nasa.gov

driven cavity flows, and flows over a circular cylinder[10]. Without applying preconditioning to the flow equations, the CE/SE method can be straightforwardly used for flows at all speeds[8].

The rest of this paper is organized as follows. In Section 2, the CE/SE method for three-dimensional Navier-Stokes equations will be presented. The midpoint rule is used to integrate the viscous fluxes as part of the space-time flux conservation. In Section 3, two viscous flows are calculated using the new CE/SE NS code: a three-dimensional driven cavity flow in a square box, and a swirling jet in a square cylinder. We then offer the concluding remarks and give cited references

## 2. The CE/SE Scheme for the 3D Navier-Stokes Equations

Consider the following three-dimensional Navier-Stokes Equations,

$$\frac{\partial U_m}{\partial t}+\frac{\partial E_{im}}{\partial x}+\frac{\partial F_{im}}{\partial y}+\frac{\partial G_{im}}{\partial z}-\frac{\partial E_{vm}}{\partial x}-\frac{\partial F_{vm}}{\partial y}-\frac{\partial G_{vm}}{\partial z}=0 \tag{2.1}$$

where $m = 1, 2, 3, 4, 5$ indicating the continuity, three momentum, and the energy equations. $U_m$=(ρ, ρu, ρv, ρw, $E_t$ ). $E_i$, $F_i$ and $G_i$ are the inviscid fluxes, and $E_v$, $F_v$ and $G_v$ are viscous fluxes. Let $x_1 = x$, $x_2 = y$, $x_3 = z$ and $x_4 = t$ be the coordinates of a four-dimensional Euclidean space $E_4$. The integral counterpart of Eq. (2.1) is

$$\oint_{S(V)} H_m \cdot ds = 0\ , \quad m = 1, 2, 3, 4, 5 \tag{2.2}$$

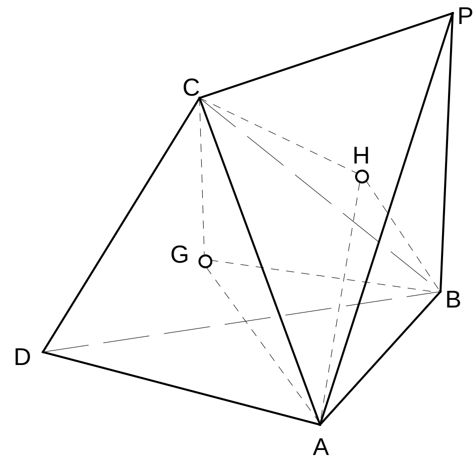
where $H_m$=($E_{im}$-$E_{vm}$, $F_{im}$-$F_{vm}$, $G_{im}$-$G_{vm}$, $U_m$) are the space-time current density vectors of mass, x, y, and z momentum, and energy, respectively. $S(V)$ is the boundary of a space-time region V in $E_4$. The flux vector $H_m$ can be decomposed into the inviscid and viscous parts:

$$H_m = H_{im} - H_{vm} \tag{2.3}$$

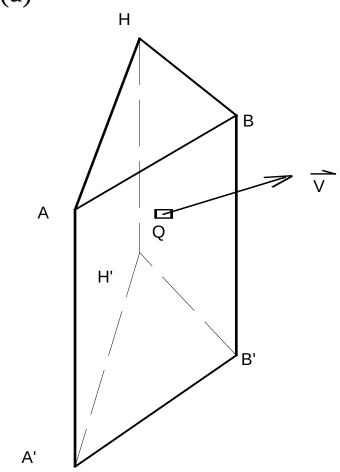
with

$$H_{im} = (E_{im}, F_{im}, G_{im}, U_m), \tag{2.4a}$$
$$H_{vm} = (E_{vm}, F_{vm}, G_{vm}, 0), \tag{2.4b}$$

In the CE/SE method, the flow variables $U_m$ and its spatial derivatives $U_{mx}$, $U_{my}$ and $U_{mz}$ are unknowns to be solved simultaneously. Thus four sets of conservation conditions are required at each mesh point for each conservation law. The four flux conservation conditions are enforced over four CEs associated with each mesh point. In the present method, tetrahedrons are used as the basic building blocks of the 3-D spatial mesh.




(a)




(b)

Fig. 2.1 (a) The definitions of the CEs and SE; (b) The calculation of the viscous flux

Consider the tetrahedron ABCD in Fig.2.1(a), in which one of its four neighboring tetrahedrons ABCP is also plotted. Points G and H are the centroids of ABCD and ABCP, respectively (let Ω denote the set of centroid G of each tetrahedron). The two tetrahedrons share the face ABC. Suppose that the points G, H, A, B, C and D are at time level $t=t^n$, points G′, H′, A′, B′, C′ and D′ are the corresponding points at $t=t^n-\Delta t/2$, and points G″, H″, A″, B″, C″ and D″ are those at $t=t^n+\Delta t/2$. Then the cylinder GABCHG′A′B′C′H′ is defined as one CE associated with the space-time mesh point G(j, n), with GABCH as its spatial base. In a similar fashion, three additional CEs associated with the mesh point G(j, n) can be constructed by considering in turn the three tetrahedrons that share with ABCD one of its other three surfaces. Assume points E, F and I are the centroids of the other three neighboring tetrahedrons sharing BCD, ABD and CDA, respectively. The points E′, F′ and I′ are the corresponding points at time level $t=t^n-\Delta t/2$, while points E″, F″ and I″ are those at $t=t^n+\Delta t/2$. Then the other three CEs are defined as the cylinder GBCDEG′B′C′D′E′, GABDFG′A′B′D′F′ and GCDAIG′C′D′A′I′, respectively, i.e., the polyhedrons GBCDE, GABDF and GCDAI are the spatial projections of the other three CEs associated with grid point G(j, n). The geometrical center of the summation of the four CEs is denoted by G* corresponding to point G. Numerical values of flow variables are stored at such point.

Similar to that in one and two spatial dimensions, there is only one solution element (SE) associated with each mesh point. Here the SE associated with point G is defined as the union of G′A′B′G″A″B″, G′B′C′G″B″C″, G′A′C′G″A″C″, G′D′C′G″D″C″, G′D′B′G″D″B″, G′A′D′G″A″D″, AHBECFDI and their immediate neighborhoods, Refer to Fig. 2.1(a).

In calculating the space-time flux conservation over the four CE′s, we use different treatments for inviscid and viscous terms. For the inviscid part, we use the same treatment as that for the Euler equations, i.e., inside each *SE(j, n)*, the flow variables are assumed continuous and can be described by using the first-order Taylor series expansions. Thus $U_m$(x, y, z, t) , $E_{im}$(x, y, z, t) , $F_{im}$(x, y, z, t) and $G_{im}$(x, y, z, t) can be approximated by

$$U^*_m(x,y,z,t;j,n) = (U_m)^n_j + (U_{mx})^n_j(x-x_j) + (U_{my})^n_j(y-y_j) + (U_{mz})^n_j(z-z_j) + (U_{mt})^n_j(t-t^n) \quad (2.5a)$$

$$E^*_{im}(x,y,z,t;j,n) = (E_{im})^n_j + (E_{imx})^n_j(x-x_j) + (E_{imy})^n_j(y-y_j) + (E_{imz})^n_j(z-z_j) + (E_{imt})^n_j(t-t^n) \quad (2.5b)$$

$$F^*_{im}(x,y,z,t;j,n) = (F_{im})^n_j + (F_{imx})^n_j(x-x_j) + (F_{imy})^n_j(y-y_j) + (F_{imz})^n_j(z-z_j) + (F_{imt})^n_j(t-t^n) \quad (2.5c)$$

$$G^*_{im}(x,y,z,t;j,n) = (G_{im})^n_j + (G_{imx})^n_j(x-x_j) + (G_{imy})^n_j(y-y_j) + (G_{imz})^n_j(z-z_j) + (G_{imt})^n_j(t-t^n) \quad (2.5d)$$

Accordingly,

$$H^*_{im}(x,y,z,t;j,n) = (E^*_{im}(x,y,z,t;j,n), F^*_{im}(x,y,z,t;j,n), G^*_{im}(x,y,t;j,n), U^*_m(x,y,t;j,n)) \quad (2.6)$$

As a result, Eq. (2.2) can be approximated by

$$\oint_{S(CE^{(l)}(j,n))} (H^*_{im} - H_{vm}) \cdot ds = 0 \ , \ \text{l}=1,2,3,4 \quad (2.7)$$

where $S(\mathrm{CE}^{(l)}(j, n))$ is the boundary of $\mathrm{CE}^{(l)}$ with l =1, 2, 3, and 4.

To calculate the viscose terms, the midpoint rule is used. Note that the forth component of $H_{vm}$ is a null. Refer to Eq. (2.4b). Thus, in calculating the viscous fluxes over a CE, we only need to calculate integrals over lateral volumes expanding in time in the space-time domain. For example, in calculating viscous flux over CE of the cylinder GABCHG′A′B′C′H′, we only need to calculate the integrals of viscous terms over six lateral volumes GABG′A′B′, GBCG′B′C′, GACG′A′C′ and HABH′A′B′, HBCH′B′C′ and HACH′A′C′. For example, the integral of $H_{vm}$ on the volume HABH′A′B′ is calculated by

$$\int_{HABH'A'B'} H_{vm} \cdot dS \approx V_x \cdot E_{vm}((U_m, U_{mx}, U_{my}, U_{mz})^{n-1/4}_Q) + V_y \cdot F_{vm}((U_m, U_{mx}, U_{my}, U_{mz})^{n-1/4}_Q) + V_z \cdot G_{vm}((U_m, U_{mx}, U_{my}, U_{mz})^{n-1/4}_Q) \quad (2.8)$$

where $\Delta\vec{V} = (V_x,\ V_y,\ V_z,\ V_t)$ is the volume vector, defined as the unit outward normal vector multiplied by its volume. And Q is the centroid of the volume HABH′A′B′. Because the volume HABH′A′B′ is a part of the SE of point H′, we can use the parameters at point H′ at $t = t^n - \Delta t/2$ to approximate $U_m$ at point Q, i.e.,

$$(U_m)_Q \approx (U_m)_{H'} + (U_{mx})_{H'}(x_Q - x_{H'}) + (U_{my})_{H'}(y_Q - y_{H'}) + (U_{mz})_{H'}(z_Q - z_{H'}) + \Delta t/4 \cdot (U_{mt})_{H'} \quad (2.9)$$

To calculate $U_{mx}$, $U_{my}$ and $U_{mz}$ at point Q, we assumed a linear distribution of $U$ in each SE, and then the following approximations can be employed:

$$(U_{mx})_Q \approx (U_{mx})_{H'}; \quad (U_{my})_Q \approx (U_{my})_{H'} \quad (U_{mz})_Q \approx (U_{mz})_{H'}; \quad (2.10)$$

Similar treatment is applied to other conservation elements.

To proceed, substitute Eqs. (2.5) and (2.6) into Eq. (2.7), and use the above treatment for viscous terms. We then get the following four discrete equations:

$$\left[\sum{}^+_{l1} U_m + \sum{}^+_{l2} U_{mx} + \sum{}^+_{l3} U_{my} + \sum{}^+_{l4} U_{mz}\right]^n_j = \left[\sum{}^-_{l1} U_m + \sum{}^-_{l2} U_{mx} + \sum{}^-_{l3} U_{my} + \sum{}^-_{l4} U_{mz}\right]^{n-1/2}_{j,l} \quad (2.11)$$

where $l = 1,2,3,4$ for flux conservation over the four CEs. $\Sigma_{lk}^{\pm}$ ($l$, $k$ = 1, 2, 3, and 4) are the coefficient matrices, which is similar as that tabulated in reference [4], but here they include the viscous terms.

By adding the four equations together, we obtain the following numerical solution for $U_m$.

$$(U_m)_j^n = \frac{1}{V_h}\sum_{l=1}^{4}\left[\sum{}_{l1}^{-} U_m + \sum{}_{l2}^{-} U_{mx} + \sum{}_{l3}^{-} U_{my} + \sum{}_{l4}^{-} U_{mz}\right]_{j,l}^{n-1/2} \qquad (2.12)$$

where $V_h$ is the volume of the polyhedron BHCEDFAI. By solving any three of Eqs. (2.11), we can obtain the numerical solutions for $(U_{mx})_j^n$, $(U_{my})_j^n$ and $(U_{mz})_j^n$, which we denote as $(U_{mx}^a)_j^n$, $(U_{my}^a)_j^n$ and $(U_{mz}^a)_j^n$.

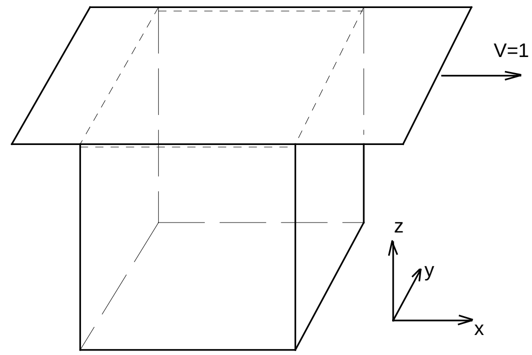
Equation (2.12) for $U_m$, in conjunction with three of the four equations in Eqs. (2.11) for $U_{mx}$, $U_{my}$ and $U_{mz}$, are the space-time CE/SE scheme for the three-dimensional Navier-Stokes equations. This is similar to the a-scheme for Euler equations [1-4]. Using the same method as that in [1-4], we can get the a-ε and the a-ε-α-β schemes for the Navier Stokes solver. Since the above scheme is based on tetrahedrons, it can be directly used in unstructured mesh.

## 3. Numerical Results

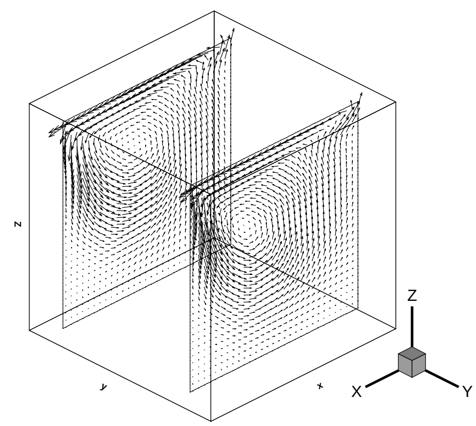
To demonstrate the capabilities of the present scheme, two three-dimensional flow problems are calculated. The first problem is the three-dimensional driven cavity flow, and the second one is the swirling jet in a square box.

### 3.1 Driven Cavity Flows

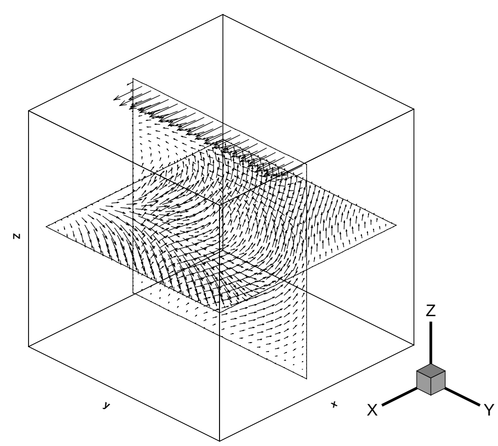
The first problem is a three-dimensional driven cavity flow. The top lid is moving in the x direction with a constant speed. Refer to Fig. 3.1(a). The Reynolds number of the flow is Re = 400. 72,000 tetrahedrons are used here. In Figure 3.1(b)-(f), we show the velocity vectors and pressure contours on some planes. A steady state solution is obtained after about 10,000 iterations. The CPU time is about 5 hours on a PC Pentium II 450. It can be seen that the velocity vector and pressure distributions on the mid x-z plane are similar to its two-dimensional counterpart. On the central line of the box with x = 0.5 and y = 0.5, the distribution of velocity component u agrees well with Ghia's data [11].



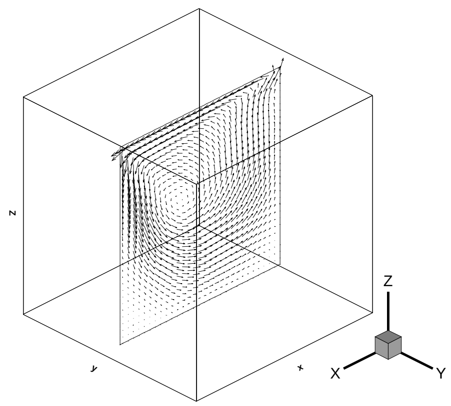

(a)



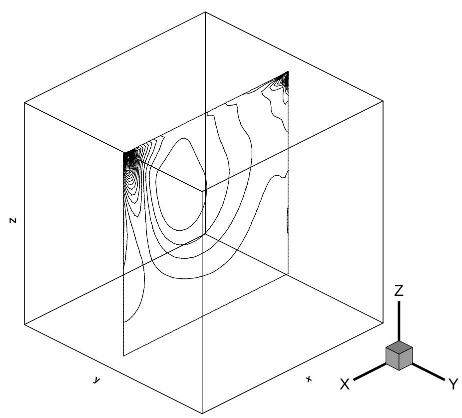

(b)



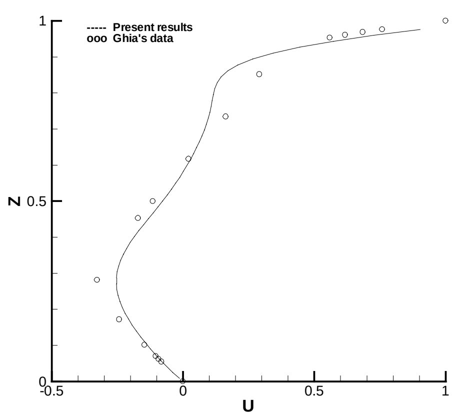

(c)




(d)




(e)

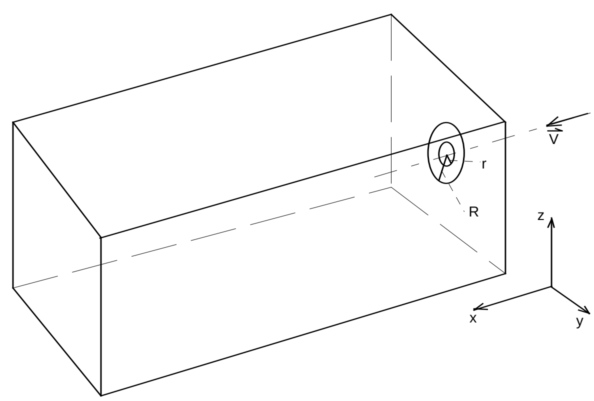



(f)

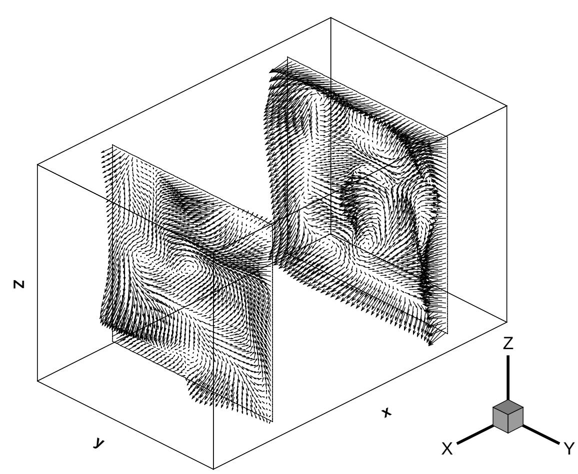
Fig. 3.1, (a) The schematic of the 3-D driven cavity flow; (b) Velocity vectors on two different x-z planes; (c) Velocity vectors on the mid x-y and y-z planes; (d) and (e) Velocity vectors and pressure contours on the mid x-z plane, respectively; (f) Velocity u distribution on the x=0.5 and y=0.5 central line.

### 3.1 Swirling Jet in a Square Box

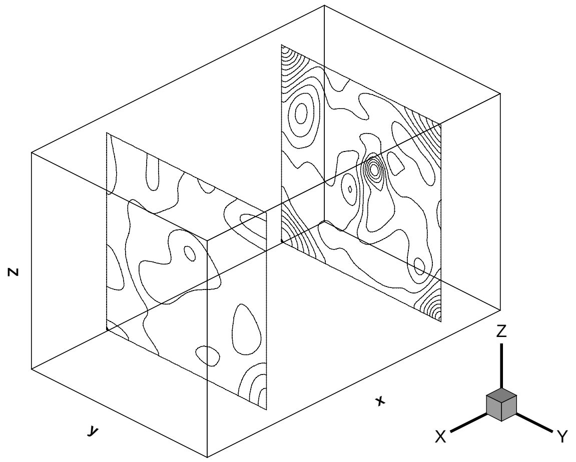
The Second problem is a three-dimensional swirling jet in a square box [12]. This is the first step for the simulation of the real chemical flow in a combustion chamber. Figure 3.2(a) is the schematic of this problem, and the flow is from left to right injected through a circular hole and a circular annulus around the hole. The airflow coming out from the inner circular hole is a straight jet. The flow from the annulus is swirling. The computational domain is 10.0×6.0×6.0 discretized by 160,000 tetrahedrons. The swirling number S=0.8, and the annulus ratio is R/r=2.0. Figure 3.1 (b), (c) and (d) show the velocity vectors, the pressure contours and the vorticity contours at two different y-z planes after 25,000 iterations. We can discern several vortices in addition to the main swirling jet. In future, solutions with fine mesh to resolve the complex flow physics will be presented.



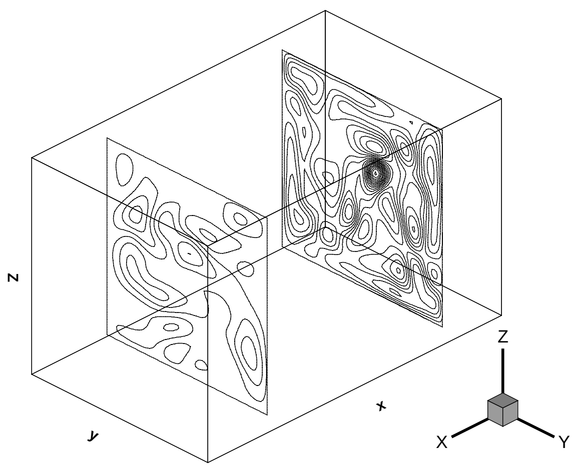

(a)




(b)




(c)




(d)

Fig. 3.2, (a) The schematic of the 3-D Swirling jet in a square cylinder; (b), (c) and (d) Velocity vectors, pressure contours and vorticity contours on two different y-z planes, respectively.

## 4. Concluding Remarks

In this paper, we report an extension of the space-time CE/SE method for solving three-dimensional Navier Stokes equations. This new Navier Stokes solver retains all favorable features of the original CE/SE method, including the unified treatment of space and time, accurate computation of space-time flux conservation, and high-fidelity resolution of unsteady flow field. Tetrahedrons are used as the fundamental spatial mesh element, and thus the present scheme is compatible with both structured and unstructured meshes. In calculating the flux balance, the integral of the viscous flux terms is based on the mid-point rule. Numerical results show that the present scheme is effective, robust and accurate. Moreover, the new code can be used for low-Mach number compressible flows as well as high-speed flows.

## References

1. Chang, S.C., “The Method of Space-Time Conservation Element and Solution Element – A New Approach for Solving the Navier Stokes and Euler Equations,” *J. Comp. Phys.*, vol. 119 (1995) pp. 295-324
2. Chang, S.C., Yu, S.T., Himansu, A., Wang, X.Y., Chow, C.Y. and Loh, C.Y., “The Method of Space-Time Conservation Element and Solution Element – A New Paradigm for Numerical Solution of Conservation Laws,” Computational Fluid Dynamics Review, vol. 1, edited by Hafez, M. and Oshima, K., World Scientific Publisher, (1998) pp. 206-240.
3. Chang, S.C., Wang, X.Y. and Chow, C.Y., “The Space-Time Conservation Element and Solution Element Method: A New High-Resolution and Genuinely Multidimensional Paradigm for Solving Conservation Laws,” *J. Comp. Phys.*, vol. 156 (1998) pp.89-136.
4. Wang, X.Y. and Chang, S.C., “A 3-D Non-Splitting Structured/Unstructured Euler Solver Based on the Space-Time Conservation Element and Solution Element Method,” AIAA Paper 99-3278, June 1999.
5. Zhang, Z.C. and Yu, S.T., “Shock Capturing without Riemann Solver---A Modified Space-Time CE/SE Method for Conservation Laws,” AIAA Paper 99-0904, Jan. 1999.
6. Zhang, Z.C., Yu, S.T., Chang, S.C., Himansu, A. and Jorgenson, P., “A Modified Space-Time CE/SE Method for Euler and Navier-Stokes Equations,” AIAA Paper 99-3277, June 1999.
7. Zhang, Z.C. and Yu, S.T., “A Non-Oscillatory Central Scheme for Conservation Laws,” AIAA Paper 99-3576, June 1999.
8. Zhang, Z.C., Yu, S.T., Wang, X.Y., Himansu, A., Chang, S.C. and Jorgenson, P., “The CE/SE Method for Navier-Stokes Equations Using Unstructured Meshes for

Flows at All Speeds," AIAA Paper 2000-0393, Jan. 2000.

9. Chang, S.C., Zhang, Z.C., Yu, S.T. and Jorgenson, P., "A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the Space-Time CE/SE Method," Proceeding of ICCFD, Kyoto, Japan 2000.

10. Zhang, Z.C., Yu, S.T., Chang, S.C. and Jorgenson, P., " Calculations of Low-Mach-Number Viscous Flows without Preconditioning by the Space-Time CE/SE Method," Proceeding of ICCFD, Kyoto, Japan 2000.

11. Guia, U., Guia, K.N. and Shin, C.T., "High Reynolds Number Solutions for Incompressible Flow Using the Navier-Stokes Equations and Multigrid Method," *J. Comp. Phys.*, vol. 48 (1982) pp. 387-411.

12. Akselvoll, K. and Moin, P., "Large-Eddy Simulation of Turbulent Confined Coannular Jets,' *J. Fluid Mech.*, vol. 315(1996), pp.387-411.