



A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the CE/SE Method

Sin-Chung Chang
Glenn Research Center, Cleveland, Ohio

Zeng-Chen Zhang
Wayne State University, Detroit, Michigan

S.T. John Yu
Wayne State University, Detroit, Michigan

Philip C.E. Jorgenson
Glenn Research Center, Cleveland, Ohio

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at 301-621-0134
- Telephone the NASA Access Help Desk at 301-621-0390
- Write to:
NASA Access Help Desk
NASA Center for AeroSpace Information
7121 Standard Drive
Hanover, MD 21076



A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the CE/SE Method

Sin-Chung Chang
Glenn Research Center, Cleveland, Ohio

Zeng-Chen Zhang
Wayne State University, Detroit, Michigan

S.T. John Yu
Wayne State University, Detroit, Michigan

Philip C.E. Jorgenson
Glenn Research Center, Cleveland, Ohio

Prepared for the
First International Conference on Computational Fluid Dynamics (CFD)
sponsored by Japan CFD Society
Kyoto, Japan, July 10–14, 2000

National Aeronautics and
Space Administration

Glenn Research Center

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076
Price Code: A03

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100
Price Code: A03

Available electronically at <http://gltrs.grc.nasa.gov/GLTRS>

A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the CE/SE Method

Sin-Chung Chang and Philip C.E. Jorgenson
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Zeng-Chen Zhang and S.T. John Yu
Mechanical Engineering Department
Wayne State University
Detroit, Michigan 48202

Abstract. In the setting of the CE/SE method, a new and unified wall boundary treatment for the Navier-Stokes and Euler Equations is proposed. In essence, the shear stress exerted on the fluid by a wall is modeled as a source term as a part of local space-time flux conservation in the vicinity of a wall boundary. When the fluid is inviscid, the source term vanishes and the boundary condition reduces to the usual “slip” condition. On the other hand, when the fluid is viscous, the source-term effect is consistent with the traditional no-slip condition. Numerical results show that the new treatment is robust, efficient and accurate for viscous and inviscid flows.

1 Introduction

In conventional CFD, the slip condition is applied along the wall for inviscid flows, while no-slip condition is applied for viscous flows. In this paper, a unified wall boundary treatment for inviscid and viscous flows is proposed in the setting of the CE/SE method [1-2]. In essence, the shear stress exerted on the fluid by a wall is modeled as a source term as a part of local space-time flux conservation in the vicinity of a wall boundary. When the fluid is inviscid, the source term vanishes and the boundary condition reduces to the usual “slip” condition. On the other hand, when the fluid is viscous, the source-term effect is consistent with the traditional no-slip condition. The accuracy, efficiency and robustness of the new boundary treatment is demonstrated using numerical examples in which reflection of an oblique shock on a wall is calculated for both viscous and inviscid flows.

2 The CE/SE Scheme for 2D Viscous Flows

The 2D Navier-Stokes Equations in conservation form can be written as

$$\frac{\partial u_m}{\partial t} + \frac{\partial f_{im}}{\partial x} + \frac{\partial g_{im}}{\partial y} - \frac{\partial f_{vm}}{\partial x} - \frac{\partial g_{vm}}{\partial y} = 0, \quad m = 1, 2, 3, 4 \quad (1)$$

The column matrices formed by u_m , f_{im} , g_{im} , f_{vm} , g_{vm} , $m = 1, 2, 3, 4$, respectively, are:

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E_t \end{bmatrix} \quad F_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (E_t + p)u \end{bmatrix} \quad G_i = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (E_t + p)v \end{bmatrix}$$

$$F_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xx}u + \tau_{xy}v - q_x \end{bmatrix} \quad G_v = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \tau_{xy}u + \tau_{yy}v - q_y \end{bmatrix} \quad (2)$$

The inviscid fluxes f_{im} , g_{im} are functions of u_m , while the viscous fluxes f_{vm} and g_{vm} are functions of u_m , $\partial u_m / \partial x$, and $\partial u_m / \partial y$. Let $x_1 = x$, $x_2 = y$ and $x_3 = t$ be the coordinates of a Euclidean space E_3 . Then the integral form of Eq. (1) is

$$\oint_{S(V)} \mathbf{h}_m \cdot d\mathbf{s} = 0, \quad m = 1, 2, 3, 4 \quad (3)$$

where (i) $\mathbf{h}_m = (f_{im} - f_{vm}, g_{im} - g_{vm}, u_m)$, (ii) $S(V)$ is the boundary of a region V in E_3 , and (iii) $d\mathbf{s} = d\sigma \mathbf{n}$ with $d\sigma$ and \mathbf{n} , respectively, being the area and outward unit normal of a surface element on $S(V)$. Note that \mathbf{h}_m can be decomposed into the inviscid and viscous parts, i.e.,

$$\mathbf{h}_m = \mathbf{h}_{im} - \mathbf{h}_{vm} \quad (4)$$

where $\mathbf{h}_{im} = (f_{im}, g_{im}, u_m)$, and $\mathbf{h}_{vm} = (f_{vm}, g_{vm}, 0)$.

In the 2D CE/SE method [2], the space-time mesh used (Fig.1(a)) is constructed from a spatial triangle mesh (Fig.1(b)). The unknowns are stored at the space-time mesh points (marked by squares and circles in Fig.1(a)) with their spatial projections being the centroids of the triangles depicted in Fig.1(b). Each space-time mesh point is associated with three conservation elements (CEs) and one solution element (SE). As an example, at point G' , the associated CEs are cylinders $E'F'G'D'EFGD$ ($CE^{(1)}$), $A'B'G'F'ABGF$ ($CE^{(2)}$) and $C'D'G'B'CDGB$ ($CE^{(3)}$). The SE is the union of four planes $A'B'C'D'E'F'$, $GG''B''B$, $GG''D''D$ and $GG''F''F$, and their immediate neighborhood. As explained in [2], inside each SE (such as that associated with point G'), u_m , f_{im} , g_{im} are approximated by first-order Taylor's expansions in x , y and t with the expansion coefficients being the functions of the independent unknowns $(u_m)_{G'}$, $(u_{mx})_{G'}$ and $(u_{my})_{G'}$. By imposing numerical analogues of Eq. (3) over CEs and using other conditions, a family of CE/SE Euler solvers, referred to as the Euler α - ϵ - α - β schemes, is constructed in [2]. Since the current Navier-Stokes solver is an extension of the Euler α - ϵ - α - β scheme with additional treatments of the viscous terms, only these additional treatments will be discussed in this paper.

To proceed, note that the third (time) component of each vanishes. Thus, in calculating the total viscous flux leaving a CE (such as $CE^{(2)}$) through its boundary, we only need to evaluate the viscous fluxes leaving the CE through its four lateral surfaces $ABA'B'$, $BGB'G'$, $GFG'F'$ and $FAF'A'$. To explain how they are evaluated, as an example, we consider the surface $FAF'A'$ (Fig.1(c)). Let $\Delta \mathbf{S} = (S_x, S_y, 0)$ be the unit outward normal vector of $FAF'A'$ multiplied by its area. Then

$$\int_{FAF'A'} \mathbf{h}_{vm} \cdot d\mathbf{s} \approx S_x f_{vm} \left((u_m)_Q, (u_{mx})_Q, (u_{my})_Q \right) +$$

$$S_y g_{vm} \left((u_m)_Q, (u_{mx})_Q, (u_{my})_Q \right) \quad (5)$$

where Q denotes the centroid of $FAF'A'$. Because $FAF'A'$ belongs to the SE of point A , we assume that

$$(u_m)_Q \approx (u_m)_A + (u_{mx})_A(x_Q - x_A) + (u_{my})_A(y_Q - y_A) + (u_{mt})_A \Delta t / 4 \quad (6)$$

Note that, as explained in [2], $(u_{mt})_A$ is a function of the independent unknowns $(u_m)_A$, $(u_{mx})_A$ and $(u_{my})_A$. Furthermore, it is assumed that

$$(u_{mx})_Q \approx (u_{mx})_A, \quad (u_{my})_Q \approx (u_{my})_A \quad (7)$$

Since the above scheme is constructed using a triangular mesh, it is compatible with unstructured meshes. In addition, the above scheme has been extended to become 3D NS solvers, which however is out of the scope of this paper.

3 Unified Wall Boundary Treatment

For simplicity, consider the horizontal wall depicted in Fig. 2(a). No grid point is placed on the wall. Instead, a ghost point E , which is the mirror image of point G with respect to the wall, is introduced. The flow variables u_m and their spatial derivatives, u_{mx} and u_{my} ($m = 1, 2, 3, 4$), at point E are obtained from those of point G by assuming that, at any time t , the flow fields below and above \overline{DF} are the mirror images of each other, i.e., we have

$$\begin{cases} (u_m)_E = (u_m)_G, & m = 1, 2, 4 \\ (u_m)_E = -(u_m)_G, & m = 3 \end{cases} \quad \begin{cases} (u_{mx})_E = (u_{mx})_G, & m = 1, 2, 4 \\ (u_{mx})_E = -(u_{mx})_G, & m = 3 \end{cases}$$

$$\begin{cases} (u_{my})_E = -(u_{my})_G, & m = 1, 2, 4 \\ (u_{my})_E = (u_{my})_G, & m = 3 \end{cases} \quad (8)$$

Using the above conditions and the concepts of the dual space-time mesh discussed in [2], one can calculate the flow variables and their spatial derivatives at G for the next time step. The calculation involves an integration over three conservation elements $CE^{(i)}$, $i = 1, 2, 3$ (Fig. 2(b)). However, a special treatment for the calculation involving $CE^{(1)}$ is needed due to the wall boundary lying across $CE^{(1)}$.

Let (i) the viscosity μ be a constant; (ii) the wall be an insulated wall; (iii) S be the area of the rectangle $DFD'D'$; (iv) Q_+ and Q_- , respectively, be the points immediately above and below the centroid Q of $DFD'D'$. Then, because (i) $u = v = 0$ at the wall; and (ii) the numerical solution is linear in x , y , and t within a SE, the mass, momentum and energy fluxes entering into the fluid in the triangular cylinder $GDFG'D'F'$ from the wall form the row matrix

$$g_{w+} = S \left(0, -\frac{1}{Re_L} \frac{\partial u}{\partial y}, p - \frac{4}{3Re_L} \frac{\partial v}{\partial y}, 0 \right)_{Q_+} \quad (9)$$

On the other hand, the same four fluxes entering into the fluid in the triangular cylinder $EFDE'F'D'$ from the wall form the row matrix

$$g_{w-} = -S \left(0, -\frac{1}{Re_L} \frac{\partial u}{\partial y}, p - \frac{4}{3Re_L} \frac{\partial v}{\partial y}, 0 \right)_{Q_-} \quad (10)$$

Using the mirror image conditions, we have

$$\left(\frac{\partial u}{\partial y} \right)_{Q_-} = - \left(\frac{\partial u}{\partial y} \right)_{Q_+}, \quad \left(p - \frac{4}{3Re_L} \frac{\partial v}{\partial y} \right)_{Q_-} = \left(p - \frac{4}{3Re_L} \frac{\partial v}{\partial y} \right)_{Q_+} \quad (11)$$

By using Eqs. (9-11), it is concluded that the mass, momentum, and energy fluxes entering into the fluid in $GDEFG'D'E'F'$, i.e., $CE^{(1)}$, from the wall form the row matrix

$$g_w = g_{w+} + g_{w-} = -2S \cdot \left(0, \frac{1}{Re_L} \frac{\partial u}{\partial y}, 0, 0 \right)_{Q_+} \quad (12)$$

In the current treatment, the surviving x-momentum flux in Eq. (12) is treated as a source term in flux balance calculation involving conservation element $CE^{(1)}$. To calculate the flux g_w , we need to calculate $\partial u / \partial y$. For simple laminar flows with enough mesh resolution of the boundary layer, because $u = 0$ at point Q_+ , and $u \cong (u_G + u_{G'})/2$ at the midpoint M of $\overline{GG'}$, we assume

$$\left(\frac{\partial u}{\partial y} \right)_{Q_+} \approx \frac{(u_G + u_{G'})/2 - 0}{y_M - y_{Q_+}} \quad (13)$$

Note that, $g_w = 0$ for inviscid flows, and the current treatment becomes the usual “slip” condition. As such, the present boundary treatment is suitable for inviscid as well as viscous flows.

4 Numerical Results

Figure 3 is a schematic of a standard test problem for Navier Stokes solvers. When the shock is strong and the incident shock angle is large enough, boundary layer separation occurs at the shock impinging point. It is assumed that $M_\infty = 2.0$, $Re = 2.96 \times 10^5$ and $\beta = 32.6^\circ$. The computational domain is $[-1, 1.7] \times [0, 1.2]$. Clustered cells near the solid wall are employed to resolve the boundary layer. Here 28800 isosceles triangles are used.

The new solid boundary treatment is applied along the wall. In the left and upper boundaries, the flow conditions are fixed according to the incoming flows. Non-reflective boundary condition is used in the outlet boundary. The numerical results are shown in Fig. 4 for pressure contours, pressure distribution and skin friction distribution along the wall. The negative skin friction coefficient represents the recirculating flow region. The numerical results, including the separation location and length, compare favorably with the experimental data [3].

The second example is an inviscid shock reflection problem [4]. This problem has exact Euler solution. We apply the same wall boundary condition treatment with $1/R_{eL} \rightarrow 0$. A uniform mesh of 9600 conservation elements is used. The numerical results are shown in Fig.5. These results agree very well with the analytical solutions.

5 Concluding Remarks

A unified wall boundary treatment for viscous and inviscid flows has been described. In this treatment, the wall boundary condition for inviscid flows becomes a special case of that for viscous flows. Numerical results show that this new boundary treatment is robust, efficient, and very accurate for viscous as well as inviscid flows. This new treatment could also be easily extended to model complex flow physics along wall, e.g., flow permeation through a porous wall.

References

- [1] S.C. Chang: "The Method of Space-Time Conservation Element and Solution Element - A New Approach for Solving the Navier Stokes and Euler Equations," J. Comp. Phys., **119**, pp. 295-324 (1995)
- [2] S.C. Chang, X.Y. Wang and C.Y. Chow: "The Space-Time Conservation Element and Solution Element Method: A New High-Resolution and Genuinely Multidimensional Paradigm for Solving Conservation Laws," J. Comp. Phys., **156**, pp.89-136 (1999)
- [3] R.J. Hakkinen, I. Greber, L. Trilling and S.S. Abarbanel: "The Interaction of an Oblique Shock Wave with a Laminar Boundary Layer," NASA Memo 2-18-59W (1959)
- [4] Yee, H.C., Warming, R.F. and Harten, A., "Implicit Total Variation Diminishing(TVD) Schemes for Steady-State Calculations," AIAA paper 83-1902 (1983)

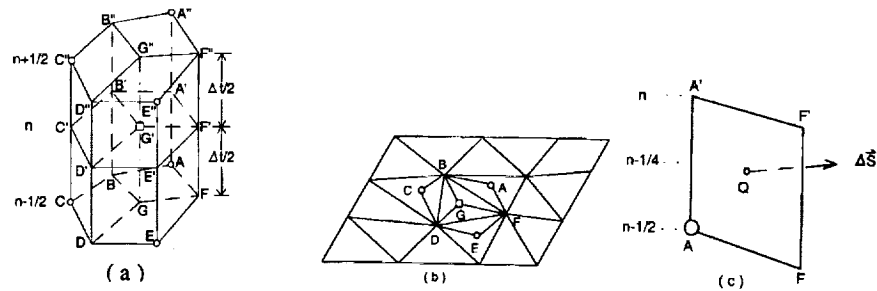


Fig. 1. A schematic of the CE/SE scheme: (a) a space-time mesh; (b) a triangular spatial mesh; (c) calculation of viscous fluxes.

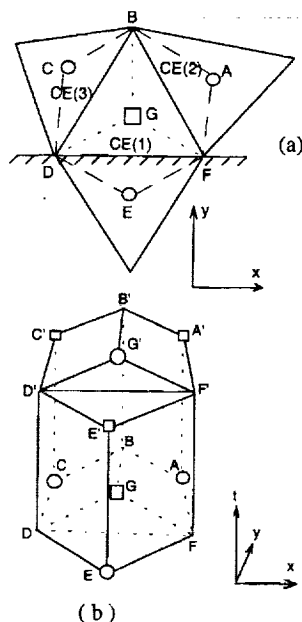


Fig. 2. A schematic for wall boundary treatment: (a) a spatial mesh and (b) CEs near the wall boundary.

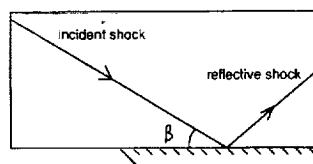
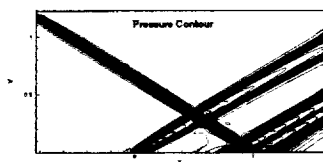
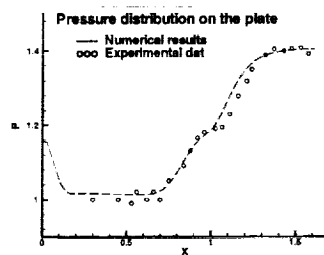


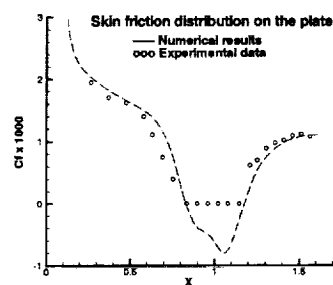
Fig. 3. Shock/boundary layer interaction problem.



(a) pressure contours.



(b) pressure distribution along the wall.

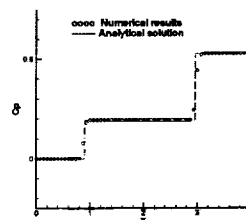


(c) skin friction along the wall.

Fig. 4. Numerical and experimental results of the shock/boundary layer interaction.



(a) pressure contours.



(b) pressure coefficients at $y = 0.5$.

Fig. 5. Euler solutions of the oblique shock reflection problem.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE October 2000		3. REPORT TYPE AND DATES COVERED Technical Memorandum
4. TITLE AND SUBTITLE A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the CE/SE Method			5. FUNDING NUMBERS WU-523-26-33-00	
6. AUTHOR(S) Sin-Chung Chang, Zeng-Chen Zhang, S.T. John Yu, and Philip C.E. Jorgenson				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191			8. PERFORMING ORGANIZATION REPORT NUMBER E-12483	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-2000-210517	
11. SUPPLEMENTARY NOTES Prepared for the First International Conference on Computational Fluid Dynamics (CFD) sponsored by the Japan CFD Society, Kyoto, Japan, July 10-14, 2000. Sin-Chung Chang and Philip C.E. Jorgenson, NASA Glenn Research Center; Zeng-Chen Zhang and S.T. John Yu, Wayne State University, Mechanical Engineering Department, Detroit, Michigan 48202-3940. Responsible person, Sin-Chung Chang, organization 5880, 216-433-5874.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Categories: 34 and 64 Available electronically at http://gltrs.grc.nasa.gov/GLTRS This publication is available from the NASA Center for AeroSpace Information, 301-621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) In the setting of the CE/SE method, a new and unified wall boundary treatment for the Navier-Stokes and Euler Equations is proposed. In essence, the shear stress exerted on the fluid by a wall is modeled as a source term as a part of local space-time flux conservation in the vicinity of a wall boundary. When the fluid is inviscid, the source term vanishes and the boundary condition reduces to the usual "slip" condition. On the other hand, when the fluid is viscous, the source-term effect is consistent with the traditional no-slip condition. Numerical results show that the new treatment is robust, efficient and accurate for viscous and inviscid flows.				
14. SUBJECT TERMS CE/SE method; Wall boundary treatment			15. NUMBER OF PAGES 12	
			16. PRICE CODE A03	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	