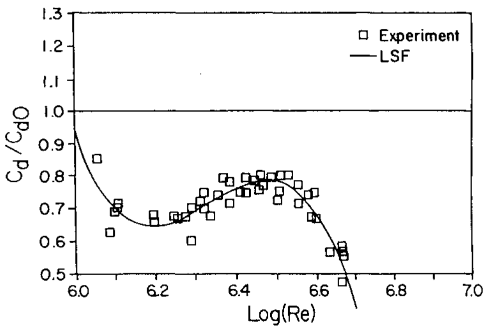


**Fig. 5** $C_d/C_{d0}$ **vs** $Re$ **indicating drag reduction.**

this trend of reduction continues at even higher $V$ values. In addition, it is the experience of the first author that this surface pattern for drag reduction is still effective for compressible cascade flow at $M = 0.9$. Studies with decelerating external flow are in progress and results will be presented when they become available.

## References


[1]Corke, T. C., Nagib, H. M., and Guezennec, Y., "A New View on Origin, Role and Manipulation of Large Scales in Turbulent Boundary Layers," NASA CR 165861, Nov. 1981.

[2]Walsh, M. J., "Riblets as a Viscous Drag Reduction Technique," *AIAA Journal*, Vol. 21, No. 4, 1983, pp. 485, 486.

[3]Bushnell, D. M., Hefner, J. N., and Ash, R. L., "Effect of Compliant Wall Motion on Turbulent Boundary Layers," *Physics of Fluids*, Vol. 20, 1977, pp. 531–548.

[4]Cary, A. M., Jr., Weinstein, L. M., and Bushnell, D. M., "Drag Reduction Characteristics of Small Amplitude Rigid Surface Waves," *Viscous Flow Drag Reduction*, edited by G. R. Hough, Vol. 72, Progress in Aeronautics and Astronautics, AIAA, New York, 1980, pp. 144–167.

[5]Gao, G., and Chow, W. L., "A New Theory of Turbulence Based on the Concept of Fractal Analysis," *Proceedings of the 17th Congress of the International Council of the Aeronautical Sciences*, Royal Aeronautical Society, London 1990, pp. 2254–2263.


# Outflow Boundary Conditions Using Duhamel's Equation


David Nixon*

*Nielsen Engineering & Research, Inc., Mountain View, California 94043*


## Introduction

MOST computational fluid dynamics (CFD) problems are solved assuming that the solution domain boundaries are either in a uniform stream or at a solid surface. For boundaries in a uniform stream that is at a known velocity and angle of attack, boundary conditions derived from one-dimensional characteristic theory can be applied in a relatively straightforward manner. However, this is not true for boundary conditions in locations where there are flow gradients present or where the flow is unsteady.

Nonreflecting boundary conditions have been developed as a promising means of applying general boundary conditions at an outflow. These boundary conditions prevent the reflections of outgoing waves back into the computational domain and also eliminate incoming waves, details of which are unknown since they originate from outside the domain. However, nonreflecting boundary conditions are only accurate for a limited subclass of flowfields. For instance, both Hedstrom[1] and Thompson[2] noted that these boundary conditions will be inaccurate for problems where the flow should be influenced by incoming waves. The error may be increased for viscous, subsonic, or incompressible flows, as the boundary conditions are derived from the hyperbolic Euler equations, and the loss of information about incoming waves at the boundary will have a greater effect on the interior flowfield.

Although some errors may be expected, nonreflecting boundary conditions provide one of the few options currently available for treating the outflow of unsteady nonuniform flows. In Ref. 3, Thompson's[2] multidimensional boundary condition formulation is extended to curvilinear coordinates and applied to subsonic flows with a wake structure that passes through the outflow boundary. It is concluded that nonreflecting boundary conditions are not adequate for flow in which the upstream influence of the flow outside the computational domain is significant, for example, in vortex shedding from a cylinder.

In the present Note, an alternative to nonreflecting boundary conditions is proposed and is applied to unsteady transonic potential flow about an airfoil. This approach is based on a time linearized version of the equations, but a representation of all other nonlinearities is retained in the model. In this study, it is found that adequate results could be obtained with a downstream boundary at 2% of the extent necessary for conventional nonreflecting boundary conditions. The ideas described here may be extended to treat the Navier-Stokes equations. If such an extension proves as accurate as the case presented here, then the computational domain for complex problems could be reduced significantly, with a corresponding increase in efficiency of the computation, either through reduced computer time or improved resolution.

## Analysis

In the following discussion, the interior and exterior domains refer to the regions in the interior of the computational domain, where the body resides, and the region exterior to the computational domain, respectively.

If a particular flow is started from zero, then the flow in the exterior domain is exact until waves from the disturbance in the interior domain cross the boundary between the domains. If true nonreflecting boundary conditions could be devised, then the outflow boundary condition might be correct if the computation were time accurate. An error or an ambiguity in the boundary conditions, due to approximations in the formulation, can give rise to a nonphysical flow in the exterior domain and, ultimately, in the interior domain because of incoming waves. A principal difficulty is in finding an analytic relationship that will give the correct boundary conditions for the governing equations. The progress[3] in nonreflecting boundary conditions shows that finding a true two-dimensional boundary condition that can represent, accurately, incoming waves is beyond the present state-of-the-art. The present work is concerned with an alternative formulation of outflow boundary conditions and is based on the following facts:

1) If the solution is started from a zero disturbance state, with a zero disturbance boundary condition on the far-field

Presented as Paper 90-3014 at the AIAA 8th Applied Aerodynamics Conference, Portland, OR, Aug. 20–22, 1990; received Oct. 15, 1990; revision received March 11, 1991; accepted for publication March 25, 1991. 

*President; currently, Shorts Professor of Aerospace Engineering, Queens Univ. of Belfast, Belfast BT9 5AG, Northern Ireland, United Kingdom.

boundary, then the outflow boundary condition is correct until the disturbance waves pass the boundary.

2) The flow in the exterior domain is determined solely by its inflow boundary conditions, which are identical with the outflow boundary conditions for the interior domain.

Consider the interior and exterior domains illustrated in Fig. 1. The domain to the right of AB is the exterior domain and that to the left of CD is the interior domain. There is an overlap of width one grid cell in the $x$ direction.

The object is to find the boundary condition on CD based on information on AB. Note that the flow in the exterior domain is defined solely by the correct data on AB.

If the flow in the exterior domain can be represented by a time linearized equation with a general boundary condition at $y_j$, $U_j(t)$ on AB, then the value of $U_k(t)$ on CD at the $k$th grid cell, $\tilde{U}_k(t)$, is given by Duhamel's equation

$$\tilde{U}_k(t) = \sum_{j=1}^{N} \left\{ U_{jk}^{(\epsilon)}(t)\epsilon_j(o) + \int_o^t U_{jk}^{(\epsilon)}(t-\tau)\frac{\mathrm{d}\epsilon_j(\tau)}{\mathrm{d}\tau}\,\mathrm{d}\tau \right\} \quad (1)$$

where $U_{jk}^{(\epsilon)}(t)$ is the indicial response of $U$ on CD at $y_k$, that is, the transient at $y_k$ due to a unit step change in the boundary conditions on AB at $y_j$, $U_j(t)$. $\epsilon_j(t)$ is the actual value of $U_j(t)$ obtained from a computation in the interior domain. The indicial response is obtained by running the CFD code over a large domain with a step change in $U_j$ at time zero at some $y$ station $y_j$ with

$$U_i(t) = 0 \qquad i \neq j \quad (2)$$

where $U_i(t)$ denotes a value at $y_i$. The time transient on the grid points on CD, $y_k$, is stored and, when divided by the magnitude of the step, becomes the indicial response $U_{jk}^{(\epsilon)}(t)$. In principle, this should be computed for all of the $j$ stations on AB and constitutes a formidable mass of information. However, certain simplifications can be made. First, since there is no origin on the infinite boundary AB, all that matters is the location of the $y_k$ point relative to the location of the step change; that is, the difference $y_k - y_j$ and, hence, the indicial response for the $j$th point on AB is the same as for any other cell provided the difference $y_k - y_j$ and the cell width is held constant. The cell used to impose the step change may be thought of as the base cell. This removes the amount of stored data by a factor of $N$, the number of $y$ grid points. If a variable grid is used, then the effects of varying grid size on $U_{jk}^{(\epsilon)}$ can be constructed by a sum of the indicial responses of the base cell. Second, the time variation can be approximated by curve fits that work quite well.[4] This removes the necessity to store every time step; only the coefficients of the approximating series need be stored. It is important to note that the indicial response needs to be computed only once for each flow parameter, for example, Mach number or Reynolds number, for a given set of governing equations. It is not dependent in any way on the geometry of the body in the interior domain.

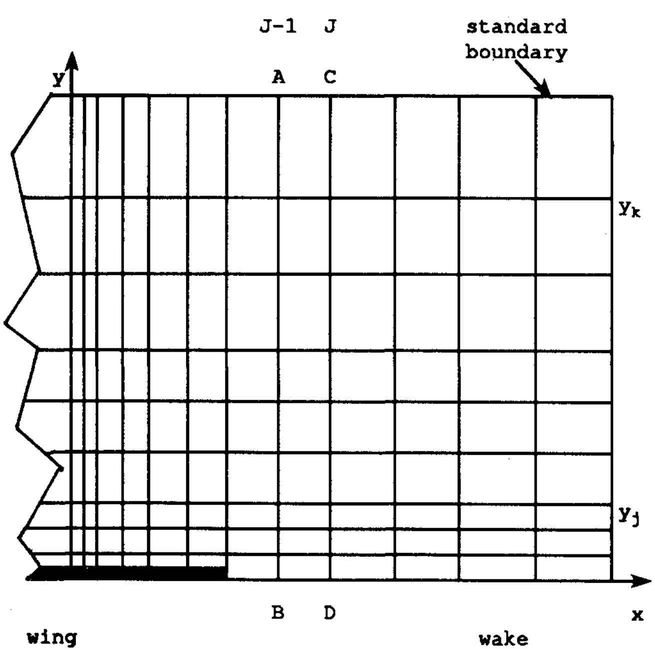


Fig. 1 Computational domain.

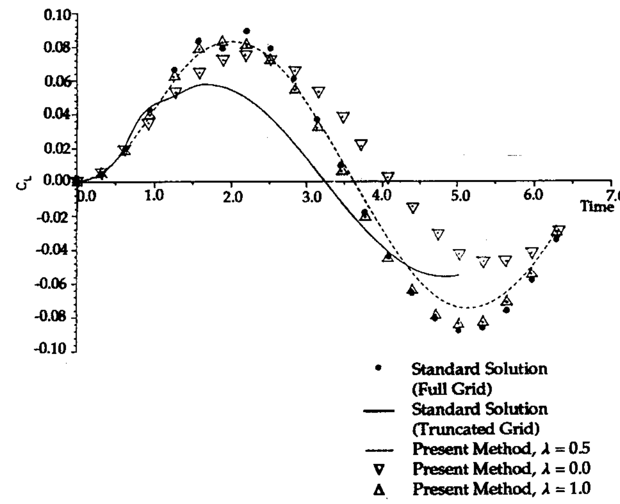


Fig. 2 NACA 64A010; $M_\infty = 0.8$, $\nu = 0.1$.

The concept is tested using the unsteady transonic small disturbance equation for a perturbation velocity potential $\phi$. Conventional solutions of this equation use a nonreflecting outflow boundary condition, although a Neumann or a Dirichlet boundary condition is also valid.

Consider now the value of $\phi$ at the boundary of a truncated grid, denoted by the subscript $J$ and sketched in Fig. 1. From Eq. (1), the velocity potential on the boundary at the $k$th grid point is

$$\phi_{Jk}(t) = \sum_{j=1}^{N} \left\{ U_{jk}^{(1)}(t)\phi_{J-1,j}(0) + \int_o^t U_{jk}^{(1)}(t-\tau)\frac{\mathrm{d}\phi_{J-1,j}(\tau)}{\mathrm{d}\tau} \right\} + U_k^{(2)}(t)\Delta\phi_{J-1}(0) + \int_o^t U_k^{(2)}(t-\tau)\frac{\mathrm{d}[\Delta\phi_{J-1}(\tau)]}{\mathrm{d}\tau}\,\mathrm{d}\tau \quad (3)$$

where $U_{jk}^{(1)}(t)$ is the indicial response at the $k$th point due to a step in $\phi_{J-1,j}$ at the $j$th point. $U_k^{(2)}(t)$ is the indicial response due to a step in $\Delta\phi_{J-1}$; this second term is due to the imposition of the Kutta condition at the trailing edge and a zero load condition at the wake. The boundary condition of the truncated grid is simply

$$\phi(x,y_k,t)\,|_{\text{boundary}} = \phi_{Jk}(t) \quad (4)$$

and $\phi_{Jk}(t)$ is given by Eq. (3). This boundary condition is only applicable if the calculation is started from a freestream value. If an accurate steady-state solution is available, then the boundary condition for an unsteady perturbed flow can be written as

$$\phi(x,y,t)\,|_{\text{boundary}} = \phi_s(x,y)\,|_{\text{boundary}} + \tilde{\phi}_{Jk}(t) \quad (5)$$

where $\phi_s(x,y)$ is the steady-state boundary condition and $\tilde{\phi}_{Jk}(t)$ is obtained from Eq. (3) by replacing $\phi_{J-1,j}(t)$ on the right side by $[\phi_{J-1,j}(t) - \phi_{sJ-1,j}(x,y)]$.

The integrals in Eq. (3) can be discretized using the trapezoidal rule to give

$$\phi_{J,k}(n\Delta t) = \frac{\lambda}{2}\,[U_{kk}^{(1)}(\Delta t) + U_{kk}^{(1)}(0)]\phi_{J-1,k}(n\Delta t) + G \qquad (6)$$

where $G$ is a term that is composed of values of $\phi_{J-1,j}$ at previous time steps. There are $n$ time steps of $\Delta t$. The factor $\lambda$ is introduced to allow for different kinds of boundary conditions. If $\lambda = 0$, then Eq. (6) represents an explicit boundary condition, because it is composed only of values at previous time steps. If $\lambda = 1$, then the boundary condition includes terms at the current time step. A fractional value of $\lambda$ represents an intermediate situation, for example, if the boundary condition is applied after the first sweep of an alternating-direction implicit algorithm.

## Results

The example chosen is a NACA 64A010 airfoil, oscillating in pitch about zero angle of attack, at a reduced frequency $\nu$ of 0.1, an amplitude of 1 deg, and at a freestream Mach number of 0.8.

In Fig. 2, results of the present method applied at $x = 1.12$, 1.35, and 1.75 are shown with the expected result that the larger the computation domain the more accurate the result. These results are compared with the standard result, that is, the result computed using XTRAN2L[5] in its standard mode. Also shown is the result using the standard method on the truncated grid; in this case and the present method, the grid is truncated at $x = 1.35$ compared with the standard grid boundary at $x = 21$. The airfoil trailing edge is at $x = 1$. The standard method for the truncated grid diverges at 150 time steps; in this case the shock wave is located at the trailing edge compared with the correct location at about 50% chord. It is important to note that even the largest domain shown here extends behind the airfoil only approximately 4% of the extent of the standard grid.

## Concluding Remarks

A novel method of computing boundary conditions for unsteady flow computations has been derived and tested for the unsteady transonic small disturbance equations. The results indicate that computational domains could be reduced considerably if this technique were used. There is no difficulty in principle in applying the present ideas to more complex equations, such as the Euler or Navier-Stokes equations.

## Acknowledgment

This work was supported by the Air Force Office of Scientific Research under Contract F49620-88-C-0006.

## References

[1]Hedstrom, G. W., "Nonreflecting Boundary Conditions for Nonlinear Hyperbolic Systems," *Journal of Computational Physics,* Vol. 30, No. 2, 1979, pp. 222–237.

[2]Thompson, K. W., "Time Dependent Boundary Conditions for Hyperbolic Systems," *Journal of Computational Physics,* Vol. 68, No. 1, 1987, pp. 1–24.

[3]Rodman, L. C., "The Application of Non-Reflecting Boundary Conditions to 2-D Unsteady Computations on Curvilinear Grids," AIAA Paper 90-1587, June 1990.

[4]Nixon, D., and Tzuoo, K.-L., "Prediction of Gust Loading and Alleviation at Transonic Speeds," *Journal of Aircraft,* Vol. 24, No. 10, 1987, pp. 703–709.

[5]Whitlow, W., "XTRAN2L—A Program for Solving the General Frequency Unsteady Transonic Small Disturbance Equation," NASA TM 85223, 1983.

# Alternating Direction Implicit Methods for the Navier-Stokes Equations

T. L. Tysinger* and D. A. Caughey†
*Cornell University, Ithaca, New York 14853*

## I. Introduction

In the numerical simulation of viscous flows at high Reynolds numbers, it is necessary to resolve the thin shear layers that develop near solid boundaries. Such thin shear regions require the use of grids with cells of very high aspect ratio, which are known to hinder convergence for steady problems when using explicit schemes. To overcome these difficulties, Caughey has developed a diagonal alternating direction implicit (ADI) algorithm for the solution of the Euler equations of inviscid, compressible flow.[1] Rapid convergence is achieved with the use of the implicit scheme within a multigrid framework.

Here, the method is extended to solve the Navier-Stokes equations. Attention is focused on methods of adding the viscous contributions in a way that does not disturb the overall stability and efficiency of the implicit scheme. No attempt is made here to incorporate a turbulence model, so the discussion will be limited to laminar flows.

## II. Governing Equations

Compressible viscous flows are governed by the Navier-Stokes equations. These equations require that both streamwise and normal viscous diffusion be computed. The nondimensionalized form of the full Navier-Stokes (FNS) equations is written in curvilinear coordinates as

$$\frac{\partial \underline{W}}{\partial t} + \frac{\partial \underline{F}_C}{\partial \xi} + \frac{\partial \underline{G}_C}{\partial \eta} = \frac{\partial \underline{F}_V}{\partial \xi} + \frac{\partial \underline{G}_V}{\partial \eta} \qquad (1)$$

where $\underline{W}$ is the transformed dependent variable, $\underline{F}_C(\underline{W})$ and $\underline{G}_C(\underline{W})$ are the convective flux vectors, and $\underline{F}_V(\underline{W}, \underline{W}_\xi, \underline{W}_\eta)$ and $\underline{G}_V(\underline{W}, \underline{W}_\xi, \underline{W}_\eta)$ are the viscous flux vectors. An equation of state is used to relate the pressure to the total energy.

Under conditions in which the flow has a predominant direction and is without massive separation, it is possible to neglect viscous diffusion in the streamwise direction without adversely affecting the quality of the solution. This results in the so-called thin layer approximation (TLA).

The viscous flux vectors can be decoupled into components that depend only on the vector of dependent variables and its derivative in either the $\xi$ or $\eta$ direction:

$$\underline{F}_V = \underline{F}_V(\underline{W}, \underline{W}_\xi, \underline{W}_\eta) = \underline{\tilde{F}}_V(\underline{W}, \underline{W}_\xi) + \underline{\hat{F}}_V(\underline{W}, \underline{W}_\eta) \qquad (2)$$

$$\underline{G}_V = \underline{G}_V(\underline{W}, \underline{W}_\xi, \underline{W}_\eta) = \underline{\tilde{G}}_V(\underline{W}, \underline{W}_\xi) + \underline{\hat{G}}_V(\underline{W}, \underline{W}_\eta) \qquad (3)$$

The TLA entails retaining only the surface normal or $\eta$ derivatives in the viscous terms of the Navier-Stokes equations [Eq.

Presented as Paper 91-0242 at the AIAA 29th Aerospace Sciences Meeting, Reno, NV, Jan. 7–10, 1991; received March 11, 1991; revision received Dec. 12, 1991; accepted for publication Dec. 13, 1991. 

*Graduate Research Assistant; currently, Research Scientist, Fluent, Inc., Lebanon, NH. Member AIAA.

†Professor, Sibley School of Mechanical and Aerospace Engineering. Associate Fellow AIAA.