Finite-Element Solutions of the Compressible Navier-Stokes Equations

W. G. Habashi*
Concordia University, Montreal, Quebec, Canada
M. F. Peeters†
Pratt & Whitney Canada, Mississauga, Ontario, Canada
G. Guevremont‡
Concordia University, Montreal, Quebec, Canada
and
M. M. Hafez§
University of California, Davis, California

The compressible Navier-Stokes equations are solved by a new formulation using a coupled stream function/vorticity finite-element approach. By taking advantage of the natural boundary conditions of the finite-element formulation, there is no need for an explicit wall vorticity formula. Numerical experiments demonstrate that the upwinding required by other schemes for the stabilization of the iterative procedure, at moderate and high Reynolds numbers, is unnecessary in the present formulation. Finally, the pressure is also determined through a scheme allowing the use of natural boundary conditions. Results are presented for laminar and turbulent external and internal flows and indicate that the method has high accuracy and stability.

Nomenclature

A	= domain area
C	= meridional velocity component, $\sqrt{u^2+v^2}$
H	= enthalpy
$L-2$	= residual norm $=\Sigma R^2$
M	= Mach number
N	= shape function
n	= distance normal to boundary of domain
p	= pressure
q	= speed
R	= residual of governing equation
Re	= Reynolds number
s	= distance along boundary of domain
u,v	= velocity components
W	= weight function
x,r	= axisymmetric coordinates
x,y	= Cartesian coordinates
W	= weight function
∂	= partial derivative
γ	= isentropic exponent
ρ	= density
Ψ	= stream function
ω	= vorticity
Ω	$=\omega/r$
μ	= viscosity
δ	= variation
$\Delta(\ )$	= change in quantity ( )

Subscripts

i	= nodal index
n	= derivative normal to boundary
r	= derivative with respect to r
s	= derivative along boundary
x	= derivative with respect to x
y	= derivative with respect to y
∞	= freestream value

Received July 29, 1986; revision received Nov. 12, 1986.

*Professor and Consultant, Pratt & Whitney Canada.
†Senior Aerodynamicist.
‡Research Associate.
§Professor and Consultant, Pratt & Whitney Canada.

Introduction

THE formulation of the Navier-Stokes equations, by means of the stream-function/vorticity approach, is considered to have many drawbacks, among them: 1) the need to specify vorticities at solid walls, which only can be done approximately and iteratively; 2) the need for upwinding in order to stabilize the iteration algorithm; 3) the problem of nonuniqueness of mass flow vs Mach number for stream-function formulations in transonic flows; and 4) the extension of stream-function formulations to three-dimensional flows. In fact, the formulation has received scant attention for even two-dimensional subsonic flows. Yet, the stream-function/vorticity formulation has the advantage of guaranteeing mass conservation. It is also considered as the natural extension of inviscid stream-function codes and especially of turbomachinery inviscid through-flow codes, which have become an important design tool in the industry.

In this paper, a finite-element scheme is presented that deals with the first two drawbacks, with a view to tackle the transonic and three-dimensional problems in the future. These transonic and three-dimensional extensions of stream-function solvers are also no longer considered difficult and several schemes have recently appeared in the literature.[1,2]

The new formulation is based on the simultaneous solution of the two equations. It can be considered as an extension of the ideas of Campion-Renson and Crochet[3] and of Dhatt et al.,[4] but with fundamental improvements. While Dhatt et al.[4] have demonstrated stable solutions at high Reynolds numbers without using upwinding, their scheme would be difficult and inaccurate in approximating the no-slip condition in complex geometries, which are the natural terrain of finite-element solutions. On the other hand, Campion-Renson and Crochet[3] presented a formulation in which the Dirichlet boundary condition on Ψ replaces the vorticity transport equation at solid-wall boundaries, while the no-slip condition is satisfied naturally by the stream-function equation. Their results indicate that stability problems exist in their solution, even at low Reynolds numbers.

In addition to generalizing Refs. 3 and 4, this work addresses the difficulties of accurately obtaining the pressure. For incompressible flows, the pressure equation is traditionally solved with a Poisson equation as a postprocessing exercise once the velocity field has been determined. The boundary conditions for this equation are difficult and may lead to inaccuracies. This work presents a Poisson equation for pressure whose natural boundary conditions automatically satisfy the momentum equations. This is particularly useful for compressible flows, where the pressure equation is repeatedly solved.

Governing Equations and Problem Formulation

Method of Solution for Stokes Flow ($Re = 0$)

The scheme will be outlined for Stokes flow to gain insight into the proper imposition of the boundary conditions. The governing equations for two-dimensional planar incompressible flow are

$$\nabla^2\Psi + \omega = 0; \quad \Psi_y = u, \quad \Psi_x = -v \tag{1}$$

$$\nabla^2\omega = 0; \quad \omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \tag{2}$$

Multiplying Eq. (1) by ω and Eq. (2) by Ψ, summing, and integrating by parts leads to the following variational principle:

$$I = \iint[-(\nabla\omega)\cdot(\nabla\Psi) + \tfrac{1}{2}\omega^2]\,\mathrm{d}A \tag{3}$$

A similar conclusion has been drawn by Gunzburger and Peterson.[5] This result is not surprising since the Stokes problem can be described by a biharmonic equation in terms of Ψ only, which is the Euler-Lagrange equation of $(\nabla^2\Psi)^2$. The $\Psi - \omega$ system is a special case, provided a proper boundary condition for ω is used. The problem, however, is that there is no explicit boundary condition on ω and the Ψ equation is overdetermined.

Taking the variation of Eq. (3) and using Green's theorem, the following equations can be obtained:

$$\iint\delta\omega[\nabla^2\Psi + \omega]\,\mathrm{d}A = \iint[-(\Psi_x\delta\omega_x + \Psi_y\delta\omega_y) + \omega\delta\omega]\,\mathrm{d}A + \int\delta\omega\Psi_n\,\mathrm{d}s \tag{4a}$$

$$\iint\delta\Psi[\nabla^2\omega]\,\mathrm{d}A = \iint[-(\omega_x\delta\Psi_x + \omega_y\delta\Psi_y)]\,\mathrm{d}A + \int\delta\Psi\omega_n\,\mathrm{d}s \tag{4b}$$

The variations $\delta\Psi$ and $\delta\omega$ are arbitrary, except that $\delta\Psi = 0$ when Ψ is specified and $\delta\omega = 0$ when ω is specified. At solid walls, Ψ is specified and thus Eq. (4b) becomes trivial since $\delta\Psi = 0$ and is replaced by the Dirichlet condition $\Psi = \Psi(s)$. The second boundary condition $[\Psi_n = \Psi_n(s)$ or $0]$ is satisfied naturally by the contour integral of Eq. (4a). In the present work, $(\Psi - \omega)$ are coupled and there is no need to identify which boundary condition applies to which equation.

For inviscid flows ($Re = \infty$), the Dirichlet condition on Ψ replaces the stream-function equation at solid walls, while no boundary condition is imposed on ω.

Method of Solution for High *Re* Compressible Turbulent Flow

The ideas outlined above can be easily extended to the high *Re* regime. The governing equations for axisymmetric, steady, compressible turbulent flow are

$$\frac{\partial}{\partial x}\left(\frac{1}{\rho r}\Psi_x\right) + \frac{\partial}{\partial r}\left(\frac{1}{\rho r}\Psi_r\right) = -\omega \tag{5}$$

$$\frac{\partial}{\partial x}[r^3(\mu\Omega)_x] + \frac{\partial}{\partial r}[r^3(\mu\Omega)_r] - Re\cdot r^2\left[\Omega_x\Psi_r - \Omega_r\Psi_x - \left(\frac{C^2}{2}\right)_x\rho_r + \left(\frac{C^2}{2}\right)_r\rho_x\right] = 0 \tag{6}$$

where $\Omega = \omega/r$. The weak form of the solution is obtained by writing the weighted residual form of the equation. The Galerkin weighted residual scheme is selected here and can be written as

$$\iint W_1\left[\frac{\partial}{\partial x}\left(\frac{1}{\rho r}\Psi_x\right) + \frac{\partial}{\partial r}\left(\frac{1}{\rho r}\Psi_r\right) + \omega\right]\mathrm{d}A = 0 \tag{7}$$

$$\iint W_2\left\{\frac{\partial}{\partial x}[r^3(\mu\Omega)_x] + \frac{\partial}{\partial r}[r^3(\mu\Omega)_r] - Re\cdot r^2\left[\Omega_x\Psi_r - \Omega_r\Psi_x - \left(\frac{C^2}{2}\right)_x\rho_r + \left(\frac{C^2}{2}\right)_r\rho_x\right]\right\}\mathrm{d}A = 0 \tag{8}$$

where W_1 and W_2 are weight functions.

$$\iint\left[\frac{W_{1x}}{\rho r}\Psi_x + \frac{W_{1r}}{\rho r}\Psi_r - W_1\omega\right]\mathrm{d}A - \int\frac{W_1}{\rho r}\Psi_n\,\mathrm{d}s = 0 \tag{9a}$$

$$\iint\left[W_{2_x}r^3(\mu\Omega)_x + W_{2_r}r^3(\mu\Omega)_r + W_2 Re\cdot r^2\left\{\Omega_x\Psi_r - \Omega_r\Psi_x + \left(\frac{C^2}{2}\right)_r\rho_x - \left(\frac{C^2}{2}\right)_x\rho_r\right\}\right]\mathrm{d}A - \int W_2 r^3(\mu\Omega)_n\,\mathrm{d}s = 0 \tag{9b}$$

It is clear from the variational analysis of the previous section that, at solid walls, Eq. (9b) must be ignored and thus W_2 should be chosen to be zero at these boundaries, while there are no restrictions on W_1. Equation (9b) is then replaced by the condition $\Psi = \Psi(s)$, while the dropping out of the contour integral of Eq. (9a) satisfies the no-slip condition implicitly.

Using the Newton-Raphson method, the linearized equations for the change in the variables Ψ and Ω are rewritten as

$$\iint\left[\frac{W_{1x}}{\rho r}(\Delta\Psi)_x + \frac{W_{1r}}{\rho r}(\Delta\Psi)_r - W_1\Delta\omega\right]\mathrm{d}A = -R_1 \tag{10a}$$

$$\iint\left[W_{2_x}r^3(\mu\Delta\Omega)_x + W_{2_r}r^3(\mu\Delta\Omega)_r + W_2 Re\cdot r^2\left\{\Psi_r(\Delta\Omega)_x + \Omega_x(\Delta\Psi)_r - \Psi_x(\Delta\Omega)_r - \Omega_r(\Delta\Psi)_x\right\}\right]\mathrm{d}A = -R_2 \tag{10b}$$

where R_1 and R_2 are the residuals of Eqs. (9a) and (9b), respectively.

Equations (10a) and (10b) are solved with the unknowns Ψ and Ω approximated as

$$\Psi = \sum_1^8 N_i^\Psi\Psi_i \qquad \text{thus} \quad \Delta\Psi = \sum_1^8 N_i^\Psi\Delta\Psi_i \tag{11a}$$

$$\Omega = \sum_1^4 N_i^\Omega\Omega_i \qquad \text{thus} \quad \Delta\Omega = \sum_1^4 N_i^\Omega\Delta\Omega_i \tag{11b}$$

where N^Ψ and N^Ω are finite-element shape functions. In Ref. 6, numerical experiments were carried out with several combinations of weight functions and elements and it was concluded that bilinear shape functions (i.e., four nodes) should preferably be used for N^Ω, while N^Ψ should be chosen as biquadratic (i.e., eight nodes). The weight function W_1 is chosen as N^Ψ, while W_2 is chosen as N^Ω:

$$W_1 = N^\Psi \qquad W_2 = N^\Omega \tag{12a}$$

The geometry, however, is always represented with 8-node elements, hence,

$$x = \sum_1^8 N_i^\Psi x_i \qquad y = \sum_1^8 N_i^\Psi y_i \tag{12b}$$

Updating the Pressure

To obtain a Poisson equation for pressure whose natural boundary conditions satisfy the momentum equations is straightforward and has been demonstrated in Ref. 6. By taking the divergence of the momentum equations,

$$p_x = f \equiv \text{remainder of } x \text{ momentum equation} \tag{13}$$

$$p_r = g \equiv \text{remainder of } r \text{ momentum equation} \tag{14}$$

yields

$$\nabla^2 p = f_x + g_r \tag{15}$$

The weighted residual equation for Eq. (15) is

$$\iint W[(p_{xx} - f_x) + (p_{rr} - g_r)]\,\mathrm{d}x\mathrm{d}r \tag{16}$$

Integrating the complete equation by parts yields

$$\begin{aligned}&\iint [W_x(p_x - f) + W_r(p_r - g)]\,\mathrm{d}x\mathrm{d}r\\ &= \int W(p_x - f)\mathrm{d}r + \int W(p_r - g)\mathrm{d}x\end{aligned} \tag{17}$$

The contour integrals of Eq. (17) are the momentum equations (13) and (14) themselves and thus must vanish. These natural boundary conditions for pressure are an obvious advantage of the present method compared to primitive variable solvers. The pressure level is set by specifying the pressure at one point of the flow.

Updating the Density

Under the assumption of constant total enthalpy,

$$\frac{\gamma}{\gamma - 1}\frac{p}{\rho} + \frac{1}{2}(u^2 + v^2) = H_\infty \tag{18}$$

The density is obtained from this equation once the pressure has been determined. Equation (18), while exact for inviscid flows, is a reasonable approximation for the viscous energy equation in the absence of heat exchange. This should not be conceived of as a limitation to the present method, since the complete energy equation can be solved without difficulty, if needed.

Turbulence Modeling

To include the effects of turbulence, the Baldwin-Lomax model is used. This model does not require the specification of a boundary-layer thickness and hence presents particular advantages for internal flows. Details of the model will not be given here as they are outlined in Baldwin and Lomax.[7] However, it should be noted that the distance from the wall, which often appears as a parameter of turbulence models, is measured in the present work by moving away from the wall along equipotential lines obtained from the solution of

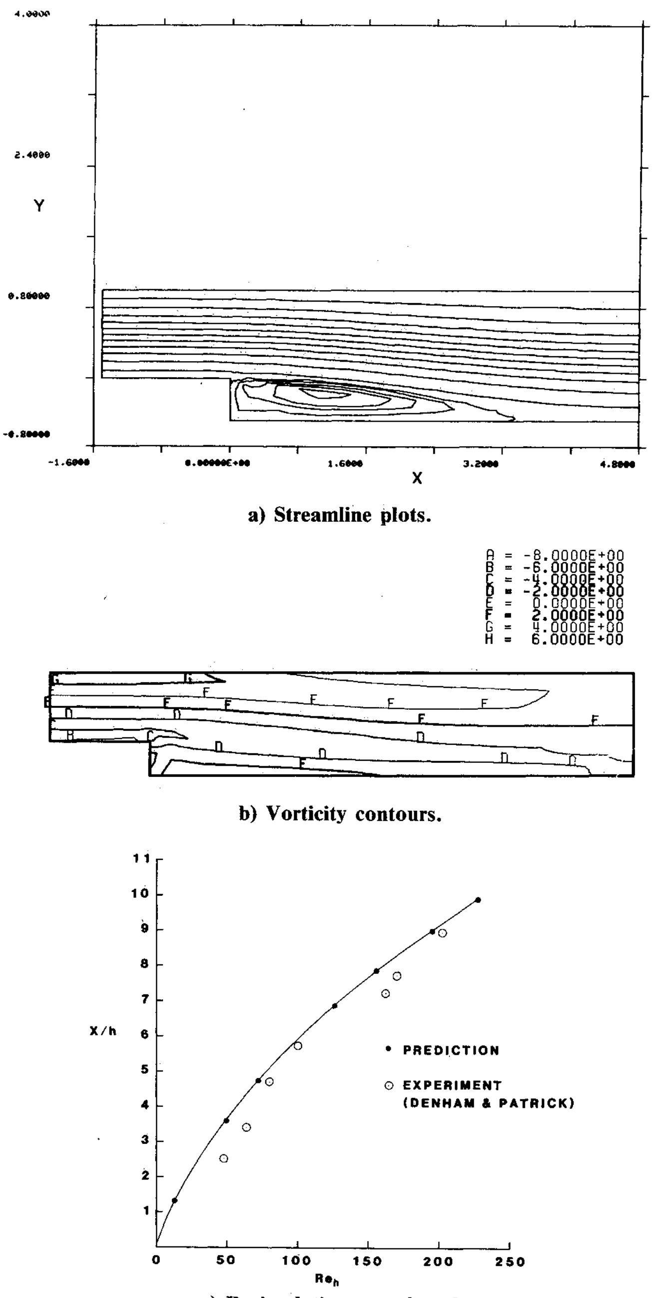
$$\nabla^2 \Phi = 0 \tag{19}$$

Solution Strategy

At every iteration, the coupled $\Psi - \Omega$ system is solved and the turbulent viscosity updated. At every second iteration, the pressure equation is solved and the density updated, assuming constant total enthalpy throughout the flowfield. The problem is started at $Re = 0$ and marched in Re until the target Re is reached. At the intermediate Re's a relaxed convergence criterion is used, while at the desired Re the $L-2$ residual is reduced to 10^{-8}.

Results

Results are presented for external as well as internal flows. For internal flows, the cases presented are: laminar incom-



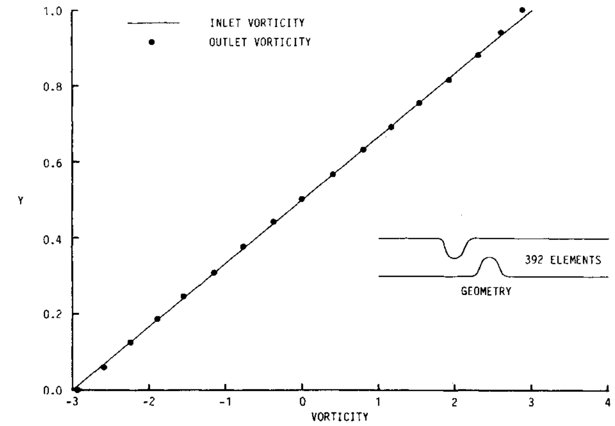
a) Streamline plots.

b) Vorticity contours.

c) Recirculation zone length.

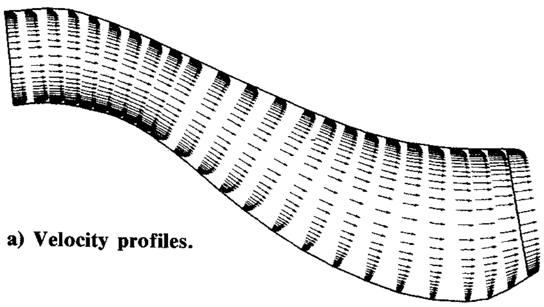
Fig. 1 Flow over a backward-facing step, $Re_h = 125$.

a) Geometry.

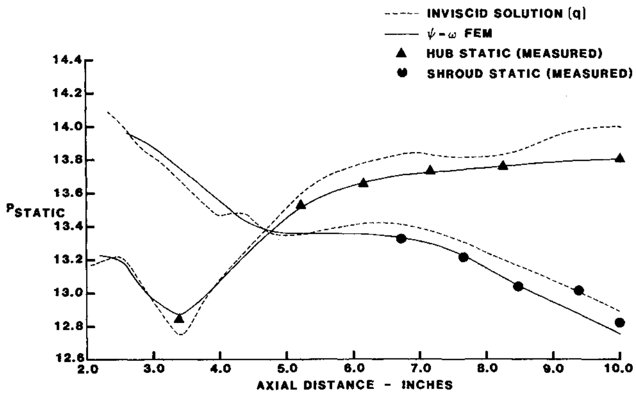


b) Exit vs inlet vorticity.

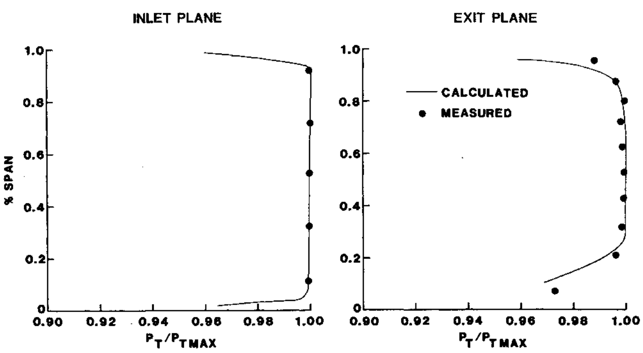
Fig. 2 Inviscid test case, $Re = \infty$.



a) Velocity profiles.



b) Static pressure profiles.



c) Total pressure profiles.

Fig. 3 Intercompressor duct.

pressible flow over a backward-facing step, an inviscid ($Re = \infty$) incompressible case in a highly turning duct, and a compressible flow in a modern intercompressor duct. For external flows, laminar flow over a NACA 0012 airfoil at two Mach numbers is presented.

Internal Flows

The laminar problem over a backward facing step is analyzed. Streamline plots as well as vorticity contours at $Re = 125$ are given in Figs. 1a and 1b. The reattachment length vs Re is plotted in Fig. 1c with the experimental results of Denham and Patrick[8] included for comparison. The predicted values compare quite well with the experimental ones.

Figure 2a presents the grid and geometry for the inviscid ($Re = \infty$) test case. The total number of elements is 392, yielding 1724 degrees of freedom. The presence of the two bumps would likely lead to a viscous-like behavior, i.e., the flow would tend to separate, in any scheme that introduces artificial viscosity through upwinding or otherwise. In the absence of artificial viscosity, the inlet vorticity profile should be preserved at the exit. As can be seen in Fig. 2b, the exit vorticity predicted by the present scheme is virtually identical to the inlet vorticity, demonstrating conclusively the absence of artificial viscosity.

Figure 3a presents velocity vectors in a modern intercompressor duct at $Re = 130{,}000$. Figure 3b plots predicted and measured static pressure distributions along the hub and shroud and also contrasts the results with an inviscid transonic flow solver,[9] of which the present scheme is an extension. Figure 3c plots predicted and measured total pressure distributions at inlet and exit. The present results compare well with the experimental ones.[10] Convergence is achieved in 23 iterations and the code executes in 7.5 min on an IBM 3081.

External Flows

To illustrate the application of the method to external aerodynamics the flow over a symmetric NACA 0012 airfoil is presented at $Re = 10{,}000$ and freestream Mach numbers of 0.2 and 0.5. The C-grid of Fig. 4 is generated by a method similar to the one outlined in Rizzi.[11] For both cases, 1610 elements were used, yielding 6685 degrees of freedom for the half-problem. The solution is obtained by solving the equations at intermediate Reynolds numbers of 0, 1000, and 5000. An underrelaxation of 0.6 is used at the Reynolds numbers of 5000 and 10,000. After the target Re solution is converged to a residual of 1, no underrelaxation is utilized. Convergence at the final Re occurs in 14 iterations and is shown in Fig. 5. The pressure coefficient distributions for both Mach numbers are shown in Fig. 6. The results for $M_\infty = 0.2$ compare well to those by Steger.[12] The streamlines

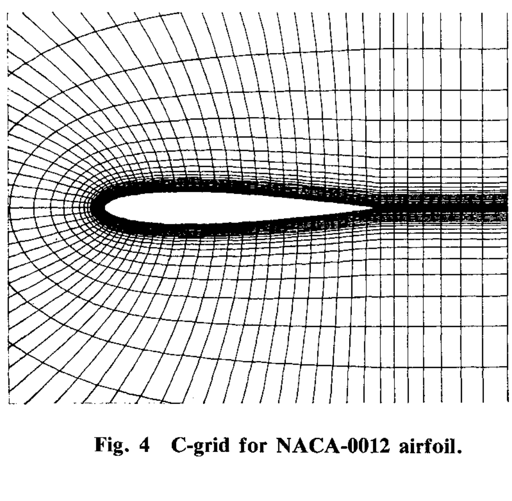


Fig. 4 C-grid for NACA-0012 airfoil.

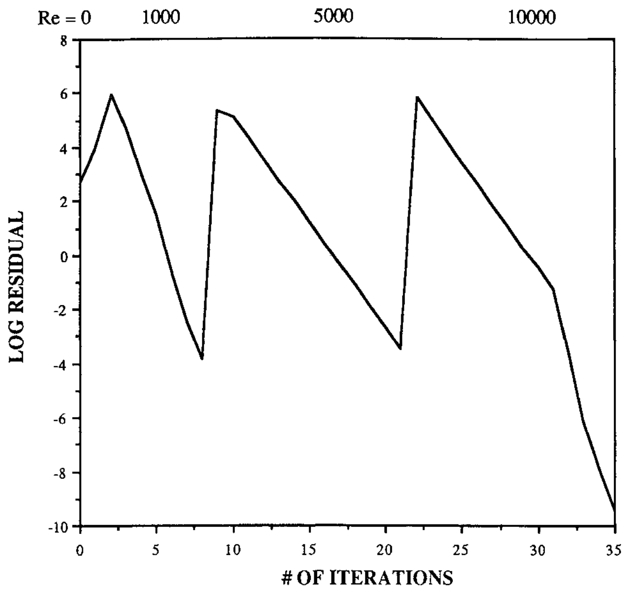


Fig. 5 Convergence history for NACA-0012 airfoil, $Re = 10,000$, $M = 0.5$.

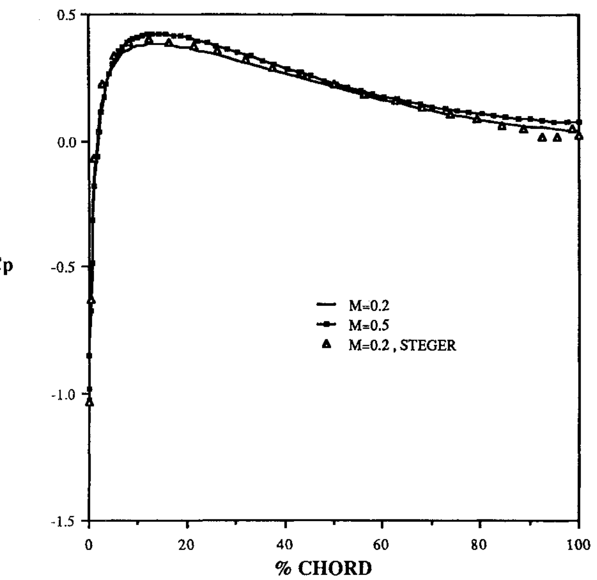


Fig. 6 C_p distribution for NACA-0012 airfoil, $Re = 10,000$, $M = 0.5$.

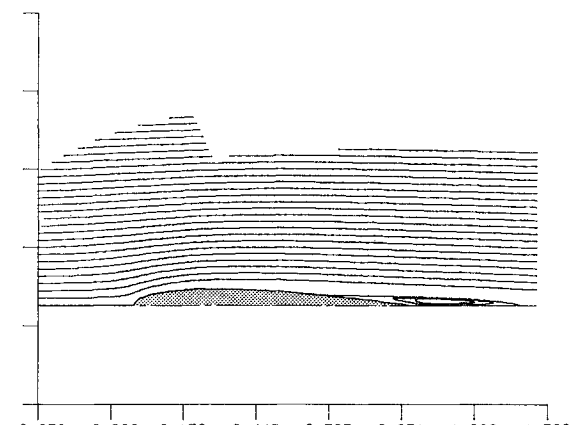


Fig. 7 Flow and separation zone for NACA-0012 airfoil, $Re = 10,000$, $M = 0.5$.

and recirculation zones for the $M_\infty = 0.5$ case are shown in Fig. 7.

Solution times are accelerated by a factor of 20 when using the vectorized matrix solver of the Cyber-205 compared to using the VAX 8600.

Conclusions

The finite-element method has been shown to be ideally suited to the formulation of the stream function/vorticity problem. Stable solutions can be obtained with no upwinding. Moreover, explicit wall vorticity formulas are dispensed with. A finite-element formulation of the pressure equation also proves very appropriate. The accuracy of imposition of boundary conditions is improved through the geometric accuracy of quadratic elements in approximating boundaries in practical applications.

Acknowledgments

The authors would like to acknowledge the gracious donation of computer time from Supercomputing Services of the University of Calgary on the Calgary Cyber-205 and from the Centre de Recherche en Informatique de Montréal (CRIM) on their VAX-8600. The present work was supported by Pratt & Whitney Canada and by PRAI Grant P-8219 and Strategic Grant G-1613 of the Natural Sciences and Engineering Research Council of Canada.

References

[1]Habashi, W. G. and Hafez, M. M., "Finite Element Stream Function Solutions of Transonic Rotational Internal and External Flows," *Journal of Numerical Methods for Partial Differential Equations*, Vol. 1, 1985, pp. 127–144.

[2]Sherif, A. and Hafez, M. M., "Computation of Three-Dimensional Flows Using Two Stream Functions," AIAA Paper 83-1948, 1983.

[3]Campion-Renson, A. and Crochet, M. J., "On the Stream Function Vorticity Finite Element Solutions of Navier-Stokes Equations," *International Journal for Numerical Methods in Engineering*, Vol. 12, 1978, pp. 1809–1818.

[4]Dhatt, G., Fomo, B. K., and Bourque, C. A., "A $\Psi-\omega$ Finite Element Formulation for the Navier-Stokes Equations," *International Journal for Numerical Methods in Engineering*, Vol. 17, 1981, pp. 199–212.

[5]Gunzburger, M. D. and Peterson, J. S., "On the Finite Element Approximation of the Streamfunction-Vorticity Equations," *Computer Methods for PDEs* V, IMACS Conference, 1984, pp. 47–56.

[6]Peeters, M. F., Habashi, W. G., and Dueck, E. G., "Finite Element Solutions of the Navier-Stokes Equations," *Proceedings of the 4th International Conference on Laminar and Turbulent Flow*, Pineridge Press, 1985, pp. 433–444; also *International Journal for Numerical Methods in Fluids*, Vol. 7, Jan. 1987, pp. 17–27.

[7]Baldwin, B. F. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, 1978.

[8]Denham, M. K. and Patrick, M. A., "Laminar Flow Over a Downstream Facing Step in a Two-Dimensional Channel," *Transactions of the Institution of Chemical Engineers*, Vol. 52, 1974, pp. 364–367.

[9]Habashi, W. G. and Youngson, G. G., "A Transonic Quasi-3D Analysis for Gas Turbine Engines Including Split-Flow Capability for Turbofans," *International Journal for Numerical Methods in Fluids*, Vol. 3, 1983, pp. 1–21.

[10]Wood, L., "Intercompressor Duct Research Testing," Pratt & Whitney Canada Inc., Montreal, Internal Rept., 1985.

[11]Rizzi, A., "Computational Mesh for Transonic Airfoils," *Numerical Methods for the Computation of Inviscid Transonic Flows*, edited by A. Rizzi and H. Viviand, Vieweg and Sohn, Braunschweig, FRG, 1981.

[12]Steger, J. L., "Implicit Finite-Difference Simulation of Flow about Arbitrary Two-Dimensional Geometries," *AIAA Journal*, Vol. 16, July 1978, pp. 679–686.