5. CONCLUDING REMARKS

The well-established CVFDM-OG's [1] are adequate for the solution of problems with simple domains in which orthogonal grids may be easily fitted. These methods are not well-suited for the solution of problems with complex domain shapes.

Most of the available CVFEM-NOG's are suitable only for mildly complex domains in which fairly orthogonal and smooth grids can be generated. As the non-orthogonality of the grid increases, the accuarcy and convergence characteristics of these methods deteriorate. This difficulty is caused by the increasing magnitude of the secondary fluxes in the convection-diffusion equations, the secondary mass fluxes in the continuity equations, and the curvature terms in the momentum equations.

For problems with truly complex domains, CVFEM's appear to hold the greatest promise. This is especially true of CVFEM's which employ unstructured grids. It should be noted, however, that most of the available CVFEM's have been applied only to incompressible, two-dimensional, laminar flow problems. Applications to turbulent flows [4], and extensions to three-dimensional [4] and compressible flow problems, are beginning to appear in the published literature, but these efforts are far from complete. Further improvements are also required with regard to interpolations functions, treatment of inflow and outflow boundaries, and equal-order velocity-pressure formulations.

In conclusion, it is hoped that all control-volume-based numerical methods for fluid flow and heat transfer will continue to receive open-minded attention of researchers.

6. REFERENCES

1. Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Hemisphere, New York, 1980.
2. Karki, K.C., A Calculation Procedure for Viscous Flow at all Speeds in Complex Geometries, Ph.D. Thesis, University of Minnesota, 1986.
3. Davidson, L. and Hedberg, P., Mathematical Derivation of a Finite-Volume Formulation for Laminar Flow in Complex Geometries, to appear, Intl. J. Num. Methods in Fluids.
4. Baliga, B.R. and Patankar, S.V., Elliptic Systems: Finite Element Method II, Chapter 11, Handbook of Numerical Heat Transfer (Eds. Minkowycz, W.Y. et al.), John Wiley & Sons, New York, 1988.
5. Prakash, C. and Patankar, S.V., Numerical Heat Transfer, Vol. 8, pp. 259-280, 1985.
6. Prakash, C., Numerical Heat Transfer, Vol. 9, pp. 253-276, 1986.
7. Prakash, C., Numerical Heat Transfer, Vol. 11, pp. 401-416, 1987.
8. Schneider, G.E. and Raw, M.J., Numerical Heat Transfer, Vol. 9, pp. 1-26, 1986.
9. Hookey, N.A., Baliga, B.R., and Prakash, C., Numerical Heat Transfer, Vol. 14, pp. 255-272, 1988.
10. Hookey, N.A. and Baliga, B.R., Numerical Heat Transfer, Vol. 14, pp. 273-293, 1988.
11. Raithby, G.D., Comp. Methods Appl. Mech. Eng., Vol. 9, pp. 153-164, 1976.

A BOUNDARY-FITTED FINITE VOLUME METHOD FOR THE SOLUTION OF COMPRESSIBLE AND/OR INCOMPRESSIBLE FLUID FLOWS USING BOTH VELOCITY AND DENSITY CORRECTION

C.R. Maliska and A.F.C. Silva
Department of Mechanical Engineering
Federal University of Santa Catarina
P.O.Box, 476 - 88049 Florianópolis - SC - Brazil

ABSTRACT

The present paper describes a numerical method for the solution of compressible and/or incompressible fluid flow problems, designed in the framework of a generalized coordinate system, which renders to the method the desired geometric flexibility. The coordinate transformation adopted is suitable for the solution of two-dimensional planar and axisymmetric flows. The possibility of handling all speed flows are accomplished by writing correction equations for velocity and density, making these variables to be both active in the mass conservation equation. Solutions are reported for the supersonic flow over a cylinder, a NACA 0012 aerofoil and the SCOUT launch vehicle. Whenever possible comparisons are made with the available experimental or numerical results.

1. INTRODUCTION

The prediction of the fluid flow motion and the heat transfer rates involved in many important engineering devices requires the solution of a coupled set of nonlinear partial differential equations representing conservation of mass, momentum and energy. General methods for the solution of this equation system is still a challenging task for the fluid dynamics theoreticians. Among the several key questions that need to be addressed, in order to obtain general and computationally efficient methods, is the extension of the existing numerical techniques to solve low Mach number as well as supersonic flows. An examination of the specialized literature demonstrates that the methods have been designed to solve, or compressible, or incompressible flow problems, and relatively few of them explore the possibility of solving flows in the incompressible limite as well as truly compressible flows. Another evidence is that the methods for compressible flows have their origin mainly among the researchers at aeroespace engineering, employing high order finite difference schemes, while the methods for incompressible flows experienced a strong development among analysts employing control volume methods. The efficient methods in the former class follow, in its majority, the approach of solving the conservation equations simultaneously, using ρ , ρu, ρv, ρw and e, as the dependent variable vector. Pressure is found through the equation of state. As formulated, the method does not work for incompressible flows, unless artificial compressibility is introduced.

By its turn, the existing segregated methods to handle incompressible flows are designed to take into account only the coupling between pressure and velocity. In compressible flows, where density is strongly dependent on pressure, this coupling is not important, making the procedure not to work for this class of problems.

The idea introduced in [1], as an extension of the methods for incompressible fluid flows, and in [2], in the context of the continuos

Eulerian approach, and further explored in [3,4], is to force both density and velocity to be active in the mass conservation equation. This procedure promotes the adequate coupling between pressure and velocity and pressure and density. This is accomplished by a special linearization of the mass conservation relation. A review of these methods and the establishment of a general structure to fit them is realized in [4], where it is also reported numerical results for two-dimensional compressible flows in the Cartesian framework.

In the present paper it is reported a numerical method for the solution of compressible and/or incompressible flows in arbitrary geometries, using the above cited mass linearization concept. The method is applied for the computation of two-dimensional planar and axisymmetric supersonic flows over arbitrary bodies. The relevant features of the method are now addressed. Further details can be found in [5,6].

2. MODEL DESCRIPTION

GOVERNING EQUATIONS. The conservation equations written in the Cartesian coordinate or in the cylindrical coordinate system are transformed to the new ξ, η coordinate system retaining its conservative form as

$$\frac{1}{J}\frac{\partial}{\partial t}(\rho\phi) + \frac{1}{x_2^j}\frac{\partial}{\partial\xi}(\rho x_2^j U\phi) + \frac{1}{x_2^j}\frac{\partial}{\partial\eta}(\rho x_2^j V\phi) = \frac{1}{x_2^j}\frac{\partial}{\partial\xi}\left[x_2^j(-\Gamma^\phi J\alpha)\frac{\partial\phi}{\partial\xi} + x_2^j(-\Gamma^\phi J\beta)\frac{\partial\phi}{\partial\eta}\right] + \frac{1}{x_2^j}\frac{\partial}{\partial\eta}\left[x_2^j(\Gamma^\phi J\gamma)\frac{\partial\phi}{\partial\eta} + x_2^j(-\Gamma^\phi J\beta)\frac{\partial\phi}{\partial\xi}\right] - \hat{P}^\phi + \hat{S}^\phi \tag{1}$$

where

$$U = u(x_2)_\eta - v(x_1)_\eta \qquad V = v(x_1)_\xi - u(x_2)_\xi$$
$$\alpha = (x_1)_\eta^2 + (x_2)_\eta^2 \qquad \beta = (x_1)_\xi(x_1)_\eta - (x_2)_\xi(x_2)_\eta \tag{2}$$
$$\gamma = (x_1)_\xi^2 + (x_2)_\xi^2 \qquad J = \left\{(x_1)_\xi(x_2)_\eta - (x_1)_\eta(x_2)_\xi\right\}^{-1}$$

and the expressions for $\hat{P}^\phi$ and $\hat{S}^\phi$ are given in Table I. The Cartesian velocity components are kept as dependent variables but there is no overlapping either of u or v momentum control volumes.

Table I - Expressions for $\hat{P}^\phi$ and $\hat{S}^\phi$

ϕ	$\hat{P}^\phi$	$\hat{S}^\phi$
1	0	0
u	$\frac{\partial P}{\partial\xi}(x_2)_\eta - \frac{\partial P}{\partial\eta}(x_2)_\xi$	$\frac{\mu}{3}\left\{(x_2)_\eta\frac{\partial}{\partial\xi}(\nabla.\vec{V}) - (x_2)_\xi\frac{\partial}{\partial\eta}(\nabla.\vec{V})\right\}$
v	$\frac{\partial P}{\partial\eta}(x_1)_\xi - \frac{\partial P}{\partial\xi}(x_1)_\eta$	$\frac{\mu}{3}\left\{(x_1)_\xi\frac{\partial}{\partial\eta}(\nabla.\vec{V}) - (x_1)_\eta\frac{\partial}{\partial\xi}(\nabla.\vec{V})\right\} - \frac{j\mu v}{Jx_2^2}$
T	0	$\left\{\frac{\partial P}{\partial t} + \nabla.(P\vec{V}) - P\nabla.\vec{V}\right\}/(Jc_p)$

In Eqs.(1) and (2) (x_1,x_2) are the coordinates (x,y) and (z,r), depending wether the transformation is from the Cartesian or from the cylindrical coordinate system. The two situations are obtained with j=0 and j=1, respectively. For ϕ equal to 1, u, v and T, plus the state equation in the form

$$\rho = \rho(P,T) \tag{3}$$

one recovers the equation set to be solved, in a segregated manner, using a finite volume method.

ALGEBRAIC EQUATIONS. The algebraic equations are obtained through the integration of Eq.(1) over time and over the control volume centered at P shown in Fig. 1. To illustrate, the integration of the second term in the left hand side gives

$$\int_t^{t+\Delta t}\int_{\xi_w}^{\xi_e}\int_{\eta_s}^{\eta_p}\frac{1}{x_2^j}\frac{\partial}{\partial\xi}(\rho x_2^j U\phi)\, x_2^j d\eta d\xi dt = \left\{(\dot{M}\phi)_e - (\dot{M}\phi)_w\right\}\Delta t \tag{4}$$

where the superscript "o" indicates the previous time level. For ϕ = 1 the integration of Eq.(1) results in

$$(M_P - M_P^o)/\Delta t + \dot{M}_e - \dot{M}_w + \dot{M}_n - \dot{M}_s = 0 \tag{5}$$

and for ϕ equal to u, v or T, the integration gives

$$a_P\phi_P = \Sigma a_{nb}\phi_{nb} + (M_P\phi_P)^o/\Delta t - L[\hat{P}^\phi]\,\Delta\xi\Delta\eta + L[\hat{S}^\phi]\Delta\xi\Delta\eta \tag{6}$$

where the notation L[ ] means the numerical approximation of the term inside the brackets.

LINEARIZATION OF THE MASS FLUXES. As discussed previously, there is the need of having both velocity and density active in the mass conservation equation to provide the method with the capability of solving incompressible and compressible fluid flow problems. This is done through the following linearization [4]. Let U^* and ρ^* estimates of the velocity and density to be corrected through

$$U = U^* + U' \tag{7}$$

$$\rho = \rho^* + \rho' \tag{8}$$

during the iteration procedure. Consider now the mass flow at the east face of a continuity control volume given by

$$\dot{M}_e = (\rho U x_2^j)_e\Delta\eta \tag{9}$$

Substituting Eqs.(7) and (8) in (9) and neglecting the product of the prime variables such that the algebraic equation results linear, one gets

$$\dot{M}_e = (\rho^* x_2^j U + \rho x_2^j U^* - \rho^* x_2^j U^*)_e\Delta\eta \tag{10}$$

The $\dot{M}_w$, $\dot{M}_n$ and $\dot{M}_s$ terms are linearized in the same manner.

The final step now is to obtain an equation for pressure. To this end ρ and U needs to be related to the pressure correction. This issue is now addressed.

PRESSURE EQUATION. Usingan estimate of the pressure and density fields in a linearized form of the state equation one obtains

$$\rho^* = C^\rho P^* + b^\rho \tag{11}$$

where the C^ρ coefficient is a function of temperature, temporarely frozen. Analogous, let ρ and P be the correct density and pressure fields such that

$$\rho = C^\rho P + b^\rho \tag{12}$$

From Eqs. (11), and (12) one obtains

$$\rho = \rho^* + C^\rho P' \tag{13}$$

where $P' = P - P^*$. To find a relation between the velocity field and the pressure correction field the momentum equations are used. For the Cartesian velocity component u, stored at the east face of the continuity control volume one can write[6]

$$u_P = u_P^* - d_P^u L[\hat{P}'^u]\Delta\xi \tag{14}$$

For a pseudo-velocity, v , that would be stored togheter with u, one has

$$v_P = v_P^* - d_P^u L[\hat{P}'^v]\Delta\xi \tag{15}$$

Using the relation between the contravariant and the Cartesian velocity components one gets

$$U_P = U_P^* - d_P^u\left\{\alpha(P'_E - P'_P) - \beta(P'_N + P'_{NE} - P'_S - P'_{SE})\Delta\xi/(4\Delta\eta)\right\} \tag{16}$$

Remember that U is a contravariant velocity component stored at the east face of a continuity control volume and, as so, it is the required velocity to enter Eq. (5). Similar expressions can be found for U_w, V_p and V_s. Substituting Eq. (16) and their analogous and Eq. (12) and their analogous into the mass conservation equation, an equation for the pressure correction P' is found, as

$$a_P P'_P = \Sigma\, a_{nb} P'_{nb} + b^{P'} \tag{17}$$

The P' field obtained is introduced in Eq.(16) and (12) to obtain the new velocity and density fields which satisfy mass conservation.

3.SOLUTION PROCEDURE

The following steps summarizes the procedure adopted for the solution of the equation set.

1. The domain is discretized using a structured boundary-fitted grid.
2. An estimate field is assumed for all variables. The contravariant U and V velocity components are calculated. Recall that the u and v velocities are not available at the same location and so, an averaging process is necessary.
3. Calculate source terms and coefficients for the momentum equations. Solve the linear systems to obtain u^* and v^*. Compute U^* and V^*.
4. Solve Eq.(17) for P'. Boundary conditions for pressure are automatically incorporated into the equations through a mass balance performed for the boundary control volumes[7]. Using P' calculate U, V and ρ such that mass is conserved. Compute U and V in the faces where the contravariant velocity components do not satisfy mass. Decode the u and v Cartesian velocity components.
5. Solve for T. Compute the new ρ field through the state equation.
6. Cycling back to step 2 is necessary to account for nonlinearities and interequation coupling.

To deal with the pressure-velocity coupling the SIMPLEC[8] method is used. The linear systems were solved using the MSI[9] procedure.

4.NUMERICAL RESULTS

To check the model several tests were realized and some of them will be reported here. No efforts were made in order to obtain the grid independent solution. Fig. 2 shows the 20x26 grid, the constant Mach lines and the velocity vector plot for the supersonic flow around a cylinder with a free-stream Mach number of 4.0. In the same figure it is identified the point were the sonic line crosses the cylinder surface. The location of this point is in good agreement with the numerical results reported in [10]. It is seen that the shock is smearead over few grid lines, since this is a shock-capturing and not a shock-fitting technique. In this case it is necessary to refine the grid near the shock for better capturing the shock. Despite the coarse grid employed the velocity vector plot clearly shows the location of the shock.

As a second test problem the supersonic flow over the NACA 0012 aerofoil with a Mach number equal to 2.0 is solved. Fig. 3 shows the 80x30 grid employed and Fig. 4 shows the constant density lines. It can be seen in Fig. 5 that the results agree well with the numerical[11] results up to half of the aerofoil chord. Beyond this point the grid is too coarse close to aerofoil surface and need to be refined.

To finalize the tests, the results for the axisymmetric flow over part of the SCOUT launch vehicle is presented. A free-stream Mach number equal to 2.16 was employed with the same ambient conditions used in the wind tunel tests [12]. The grid used is that of Fig. 6 with few more points close to the surface. The pressure coefficient is shown in Fig. 7. The agreement between the numerical and experimental results is excellent. The quality of the results deteriorates considerably if the grid concentration is not used.

5.CONCLUDING REMARKS

This paper has presented a numerical model for the solution of all speed flows employing a mass flux linearization which makes both density and velocity to be active in the mass conservation equation. The model is suitable for the solutions of fluid flow problems defined in arbitrary geometries and was applied for predicting the supersonic flow over bodies of arbitrary shape. The transformation employed permits the methodology to be applied for planar and axisymmetric flows. The results demonstrated that the model performs well. Besides the generality of the model and the easiness in applying boundary conditions, the robustness is, perhaps, the most important characteristic that should be pointed out. It was not experienced any convergence problem or stability difficulties during the course of the numerica experiments, as usually observed when methodologies specially designed to handle compressible fluid flow problems are employed.

6. REFERENCES

[1]. Patankar, S.V.,"Calculation of Unsteady Compressible Flows Involving Shocks", Mech. Eng. Dept., Imperial College, London Rept. UF/TN/A/4,1971.

[2]. Harlow, F.H. and Amsden, A.A.,"A Numerical Fluid Dynamics Calculation Method for All Flow Speeds", J. Comp. Phys., Vol 8, pp.197-213, 1971.

[3]. Hah, C.," A Navier-Stokes Analysis of Three-Dimensional Turbulent Flows Inside Turbine Blade Rows at Design and Off-Design Conditions", J. Eng. Gas Turbines Power, Vol 100, pp.421-429, 1984.

[4]. Van Doormaal, J.P.,"Numerical Methods for the Solution of Compressible and Incompressible Fluid Flows", Ph.D. Thesis, Dept. of Mech. Eng., University of Waterloo , Waterloo, Ont., Canada, 1985.

[5]. Silva, A.F.C. and Maliska, C.R.,"Uma Formulação Segregada em Volumes Finitos para Escoamentos Compressíveis e/ou Incompressíveis em Coordenadas Generalizadas", 2º Encontro Nacional de Ciências Térmicas, Aguas de Lindoia, SP. Dez. 1988.

[6]. Maliska, C.R. and Silva, A.F.C.," Desenvolvimento de Códigos Computacionais para Solução de Escoamentos de Alta Velocidade - Parte II", Report prepared for the Institute of Space Activities - CTA , Dez. 1987.

[7]. Maliska, C.R., "A Solution Method for Three-Dimensional Parabolic Fluid Flow Problems in Nonorthogonal Coordinates", Ph.D. Thesis, Dept. of Mech. Eng., University of Waterloo, Waterloo, Ont., Canada, 1981.

[8]. Van Doormaal, J.P., Raithby, G.D., and McDonald, B.H., "The Segragated Approach to Predicting Viscous Compressible Flows", ASME paper no. 86-GT-196, presented at the Intl. Gas Turbine Conference and Exhbit, Dusseldorf, West Germany, June 8-12, 1986.

[9]. Schneider, G.E., and Zedan, M.,"A Modified Strongly Implicit Procedure for the Numerical Solutions of Field Problems", Num. Heat Transfer, Vol 4, pp. 1-19, 1981.

[10].Hayes, W.D.,and Probstein, R.F.," Hypersonic Flow Theory - Inviscid Flows", Academic Press, New York, 1966.

[11].Peyret, R. and Taylor, T.D.," Computational Methods for Fluid Flow", Springer-Verlag New York Inc., 1983.

[12].Jernell, L.S.,"Aerodynamic Loading Characteristics of a 1/10-Scale Model of the Three-Stage Scout Vehicle at Mach Numbers from 1.57 to 4.65, NASA Technical Note D-1930, 1963.

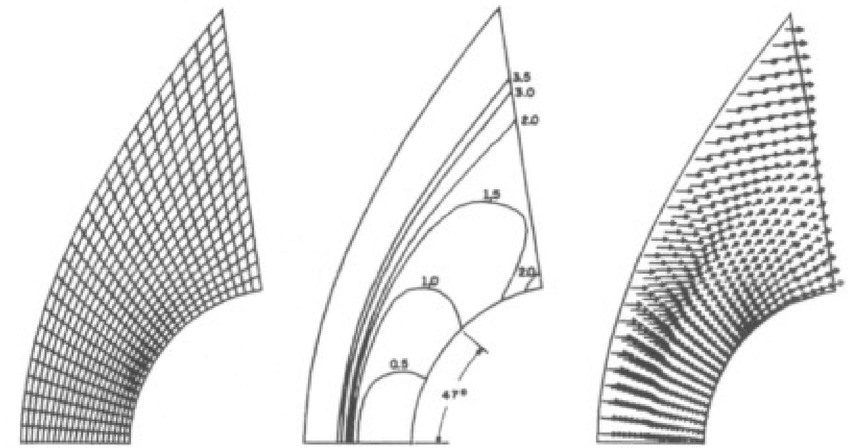

Fig. 2 - Grid, constant Mach lines and velocity vector plot for the cylinder in supersonic flow. Free stream Mach number equal to 4.0

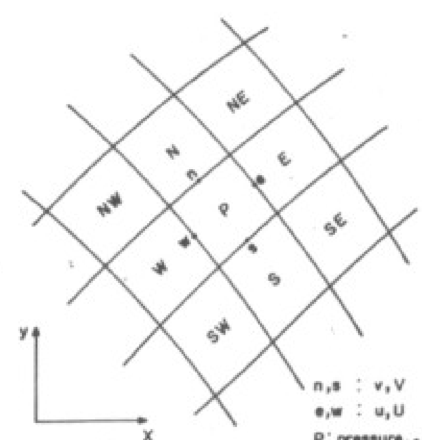

Fig. 1 - Elemental control volume

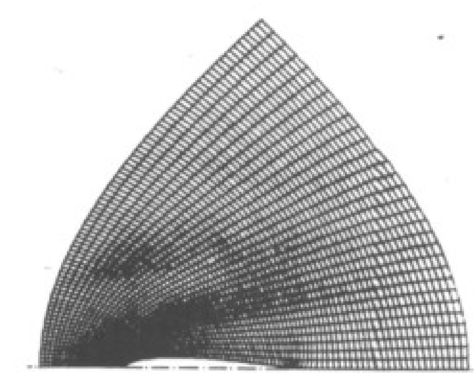

Fig. 3 - Grid employed for the NACA 0012 problem

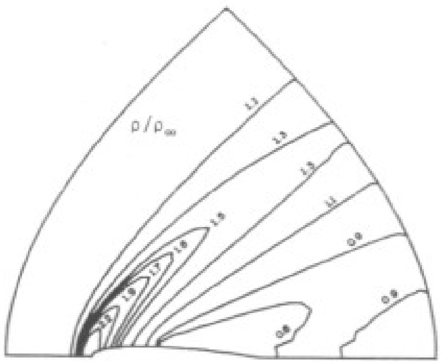

Fig. 4 - Constant density lines for the NACA 0012 problem

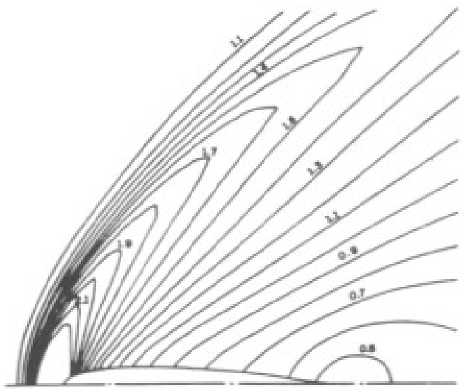

Fig. 5 - Constant density lines for the NACA 0012 problem. Numerical results from [11]

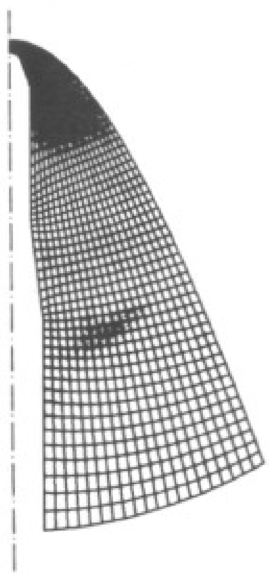

Fig. 6 - Grid employed for the SCOUT launch vehicle problem

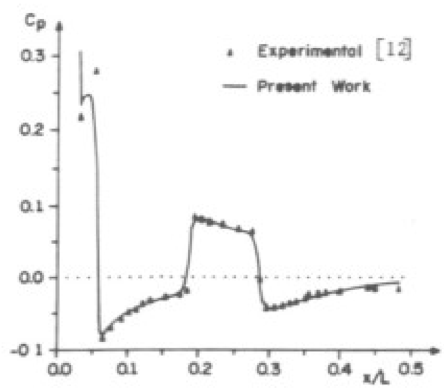

Fig. 7 - Pressure coefficients for the SCOUT problem. Comparison with experiments

A CONTROL VOLUME FINITE-ELEMENT METHOD FOR VISCOUS COMPRESSIBLE FLOWS

N.A. Hookey and B.R. Baliga
Department of Mechanical Engineering, McGill University
817 Sherbrooke Street West
Montreal, Quebec, Canada, H3A 2K6

ABSTRACT

This paper presents the formulation of a control volume finite-element method (CVFEM) for the solution of steady, two-dimensional, viscous compressible internal flow problems over a wide range of Mach numbers. In this method, the velocity components, temperature, and pressure, rather than density, are used as the dependent variables. The proposed formulation uses adaptive grid techniques to discretize calculation domains with triangular elements and to facilitate the calculation of accurate flow details in regions of high gradients. Within each element, the scalar dependent variables are interpolated with flow-oriented upwind-type interpolation functions, which include terms to account explicitly for the effect of a source on the distribution of the scalar variable in an element. These interpolation functions are used to derive algebraic approximations to integral conservation equations for polygonal control volumes constructed around each node in the calculation domain. The resulting set of nonlinear coupled algebraic discretization equations are solved by an iterative method in which a coupled equation line solution procedure is used to solve the continuity and momentum equations simultaneously along grid lines in the calculation domain.

1. INTRODUCTION

Compressible fluid flows frequently occur in the aerospace industry. Two examples of confined compressible flow are the flows through engine inlet ducts and through passageways between the blades of a gas turbine. Many of the available numerical methods for compressible flows are for inviscid fluids. These methods are suitable for many external flows, however, for internal flows, where boundary layers can grow and merge, the effects of viscosity are important and must be accounted for in the formulation of the numerical technique. Several examples of finite-difference methods for compressible flows are described in [1], recent examples of work in finite-volume methods are [2,3,4], and examples of Galerkin FEM's are [5,6].

In this paper, the formulation of a CVFEM for steady, two-dimensional, viscous compressible flows is presented. This method was developed from previous equal-order CVFEM's for incompressible fluid flows [7,8,9], and it combines their advantageous features with a new coupled equation line solver [10], and an adaptive grid generation technique [11,12]. The remainder of this paper will give a brief outline of the method and present some results.

2. PROPOSED METHOD

2.1 Two-Dimensional Formulation

Governing equations. With the choice of u, v, p, and T as the dependent variables, the equations governing steady, two-dimensional, viscous compressible fluid flows are the x and y momentum, continuity and energy equations.

$$\frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho vu) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right) + S^u + \frac{\mu}{3}\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) \qquad (1)$$