dency toward spurious oscillations in both viscous and inviscid problems. On the positive side, however, the Roe scheme is very accurate, the Liou-Steffen technique rivals that accuracy at a reduced cost, and the Van Leer algorithm can provide essentially nonoscillatory solutions for hypersonic flow conditions. The search for the optimal algorithm for compressible flows is probably still open.

Acknowledgments

This work was funded in part by the National Science Foundation, which supports the Engineering Research Center for Computational Field Simulation. The first author would like to acknowledge the partial support he obtained from the Polytechnic of Bari, Italy, while he was working at the Engineering Research Center.

References

[1]Roe, P. L., "Characteristic-Based Schemes for the Euler Equations," *Annual Review of Fluid Mechanics,* Vol. 18, 1986, pp. 337–365.

[2]Peery, K. M., and Imlay, S. T., "Blunt-Body Flow Simulations," AIAA Paper 88-2904, July 1988.

[3]Liou, M.-S., and Steffen, C. J., Jr., "A New Flux Splitting Scheme," NASA TM 104404, May 1991.

[4]Bergamini, L., and Cinnella, P., "A Comparison of 'New' and 'Old' Flux-Splitting Schemes for the Euler Equations," AIAA Paper 93-0876, Jan. 1993.

[5]Cinnella, P., and Grossman, B., "Flux-Split Algorithms for Hypersonic Flows," *Computational Methods in Hypersonic Aerodynamics,* edited by T. K. S. Murthy, Computational Mechanics Publications, Southampton, England, UK, 1991, Chap. 5, pp. 150–202.

[6]Whitfield, D. L., Janus, J. M., and Simpson, L. B., "Implicit Finite Volume High Resolution Wave-Split Scheme for Solving the Unsteady Three-Dimensional Euler and Navier-Stokes Equations on Stationary or Dynamic Grids," Mississippi State Univ., EIRS Report MSSU-EIRS-ASE-88-2, Mississippi State, MS, Feb. 1988.

[7]Hirsch, C., *Numerical Computation of Internal and External Flows,* Vol. 2, Wiley, Chichester, England, UK, 1990.

[8]Whitfield, D. L., and Janus, J. M., "Three-Dimensional Unsteady Euler Equations Solution Using Flux Vector Splitting," AIAA Paper 84-1552, June 1984.

Differencing of Density in Compressible Flow for a Pressure-Based Approach

G. P. Greyvenstein* and J. P. Meyer†
Potchefstroom University for CHE, Potchefstroom 2520, South Africa

Nomenclature

a = coefficient in the discretization equation
p = pressure
s = source term
u = x-direction velocity
x = coordinate
y = coordinate
z = nozzle length
α = weighing factor
Δt = time step
Δx = x-direction width of the control volume
ρ = density

Subscripts
E = neighbor in the positive x direction, i.e., on the eastern side
e = control-volume face between P and E
P = central grid point under consideration
W = neighbor in the negative y direction, i.e., on the western side
w = control-volume face between P and W
$*$ = based on the guessed pressure
$'$ = corrected value

Superscripts
o = old value
T = for energy equations
U = for momentum equation

Received Aug. 15, 1992; revision received June 14, 1993; accepted for publication July 26, 1993.

*Head, Department of Mechanical Engineering, Private Bag X6001.
†Professor, Department of Mechanical Engineering, Private Bag X6001. Member AIAA.

Introduction

BECAUSE of the difference in nature between compressible and incompressible flows, different computational schemes have been developed through the years to deal with these two types of flows. In the case of compressible flows, methods have been developed that use density as a primary variable. Such methods are known as density-based methods. Examples of density-based methods are the well-known methods of MacCormack[1] and Beam and Warming.[2] Unlike density-based methods, pressure-based methods use pressure as a primary variable. Examples of pressure-based methods are the well-known SIMPLE[3] and PISO[4] methods.

The purpose of this paper is to investigate the differencing scheme of density used for the convection terms in the momentum transport equations. To provide a clear understanding, we will discuss these concepts in the framework of a one-dimensional scheme. Fortunately, all of these concepts as developed within this one-dimensional framework are readily extended to two or three dimensions.

Discretized Equations

Different methods can be used to discretize the equations of motion. In this paper the control-volume-based discretization scheme described by Patankar[5] is applied. Discretization of continuity, momentum, and energy equations leads to the following algebraic equations:

$$y_P \Delta x \left(\frac{\rho_P - \rho_P^o}{\Delta t} \right) = (\rho_w u_W y_w - \rho_e u_P y_e) \tag{1}$$

$$a_P^u u_P = a_E^u u_E + a_W^u u_W - b_P^u (p_E - p_P) + s_P^u \tag{2}$$

$$a_P^T T_P = a_E^T T_E + a_W^T T_W + s_P^T \tag{3}$$

The solution of Eq. (2) using the guessed pressure field results in the following equation for tentative velocities:

$$a_P^u u_P^* = a_E^u u_E^* + a_W^u u_W^* - b_P^u (p_E^* - p_P^*) + s_P^u \tag{4}$$

If Eq. (4) is subtracted from Eq. (2), the following equation is obtained:

$$a_P^u u_P' = a_E^u u_E' + a_W^u u_W' - b_P^u (p_E' - p_P') \tag{5}$$

where u' is the velocity correction and p' is the pressure correction. The actual velocities and pressures are given by

$$u = u^* + u' \tag{6}$$

and

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

If the terms $a_E^u u_E'$ and $a_W^u u_W'$ in Eq. (5) are neglected, the following equation for velocity correction is obtained:

$$u_P' = D_P (p_P' - p_E') \tag{8}$$

where

$$D_P = \frac{b_P^u}{a_P^u} \tag{9}$$

The guessed pressure field is also used to calculate tentative densities using the equation of state. The result is

$$p^* = \frac{\rho^*}{RT} \tag{10}$$

The density correction due to the pressure correction is given by

$$\rho' = \frac{p'}{RT} \tag{11}$$

The true density is given by

$$\rho = \rho^* + \rho' \tag{12}$$

Consider the term ($\rho_e u_P$) in Eq. (1). If the density ρ_e is taken as the upstream value ρ_P, and if we use Eqs. (6) and (12), we can write

$$\rho_e u_P = \rho_P u_P = (\rho_P^* + \rho_P')(u_P^* + u_P') = \rho_P^* u_P^* + \rho_P^* u_P'$$
$$+ \rho_P' u_P^* + \rho_P' u_P' \approx \rho_P^* u_P^* + \rho_P^* u_P' + \rho_P' u_P^* \tag{13}$$

In a similar way the term ($\rho_w p_W$) is approximated by

$$\rho_w u_W = \rho_W u_W \approx \rho_W^* u_W^* + \rho_W^* u_W' + \rho_W' u_W^* \tag{14}$$

The reason why the densities are upstreamed is to avoid negative coefficients in the pressure correction equation. This equation is obtained by replacing Eqs. (8), (11), (13), and (14) in Eq. (1). After rearrangement we get

$$a_P^p p_P' = a_E^p p_E' + a_W^p p_W' + s_P^p \tag{15}$$

where

$$a_E^p = D_P \rho_P^* y_e \tag{16}$$

$$a_W^p = D_W \rho_W^* y_W + \frac{u_W^* y_W}{RT_W} \tag{17}$$

$$a_P^p = D_P \rho_P^* y_e + D_W \rho_W^* y_w + \frac{u_P^* y_e}{RT_P} + \frac{y_P \Delta x_P}{\Delta t R T_P} \tag{18}$$

$$s_P^p = -\left[y_P \frac{\Delta x_P}{\Delta t} (\rho_P^* - \rho_P^o) + \rho_P^* u_P^* y_e - \rho_W^* u_W^* y_w \right] \tag{19}$$

It was found that upwinding of the densities in Eq. (1) causes unnecessary smearing of shocks. A remedy for this problem is to upwind only the density corrections. If this practice is applied to the terms ($\rho_e u_P$) and ($\rho_w u_W$) in Eq. (1), the following result is obtained:

$$\rho_e u_P \approx \rho_e^* u_P^* + \rho_e^* u_P' + \rho_P' u_P^* \tag{20}$$

$$\rho_w u_W \approx \rho_w^* u_W + \rho_w^* u_W' + \rho_W' u_W^* \tag{21}$$

The values of densities on the control-volume boundaries in Eqs. (20) and (21), i.e., ρ_e^* and ρ_w^*, are calculated as the mean of the neighboring control-volume values. Thus

$$\rho_e^* = (\rho_E^* + \rho_P^*)/2 \tag{22}$$

$$\rho_w^* = (\rho_W^* + \rho_P^*)/2 \tag{23}$$

The expression of the mass fluxes across the control-volume boundaries according to Eqs. (20–23) does not cause negative coefficients in Eq. (15). It is also interesting to note that when a steady-state solution is obtained, the pressure corrections and thus also the density corrections become zero. In such a case, the mass flux terms given by Eqs. (20–23) reduce to

$$\rho_e u_P = (\rho_E + \rho_P) u_P/2$$

$$\rho_w u_W = (\rho_W + \rho_P) u_W/2$$

which is equivalent to a central-differencing scheme for the density derivative in the continuity equation. It can therefore be concluded that the form given by Eqs. (20–23) leads to a second order approximation of the continuity equation. The practice where only the density corrections are upwinded and where the tentative densities are expressed by Eqs. (22) and (23) will be referred to as the central-differencing scheme for density in the mass conservation (continuity) equation.

One-Dimensional Flow Through a Convergent-Divergent Nozzle

A nozzle with a parabolic profile is considered. The throat of the nozzle is exactly in the middle and the ratio of the inlet area to the throat area is 0.25. The only boundary values that are specified are the inlet pressure and the outlet pressure. The inlet pressure is taken as 1×10^5 Pa and the outlet pressure as 6.667×10^4 Pa. This implies a pressure ratio of 1.5 across the nozzle. With this specific choice of geometry and boundary conditions, a normal shock is present in the divergent part of the nozzle. The fact that only the pressures are specified on the boundaries insures that not only the ability of the methods to predict the strength and shape of the shocks is tested but also their ability to predict the position of the shocks correctly. An evenly spaced grid with 100 increments is used.

Figure 1 shows the results of the SIMPLE method for a nonstaggered grid with a mixed differencing scheme for the convection terms in the momentum and energy equations and a central-differencing scheme for the densities in the continuity equation. The effect of the weighing factor α is also shown. The closer to 1 the value of α becomes, the more smeared the shock becomes, whereas the closer to 0.5 the value of α becomes, the more oscillatory the results become. Best results are obtained with a value of α of 0.75. For this value the results of the SIMPLE method and the method of MacCormack are very close to each other.

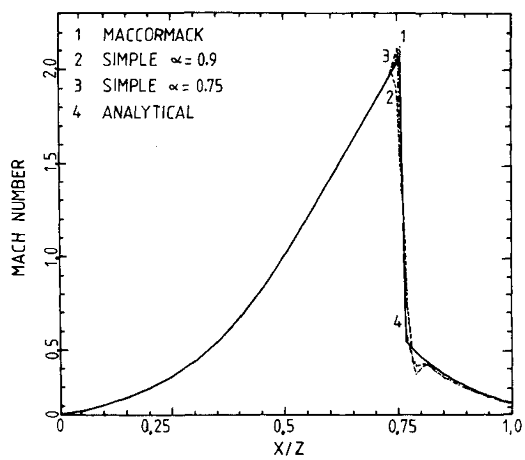

Fig. 1 Comparison between the SIMPLE method with a mixed-differencing scheme for the convection terms and a central-differencing scheme for densities in the continuity equation and the method of MacCormack.

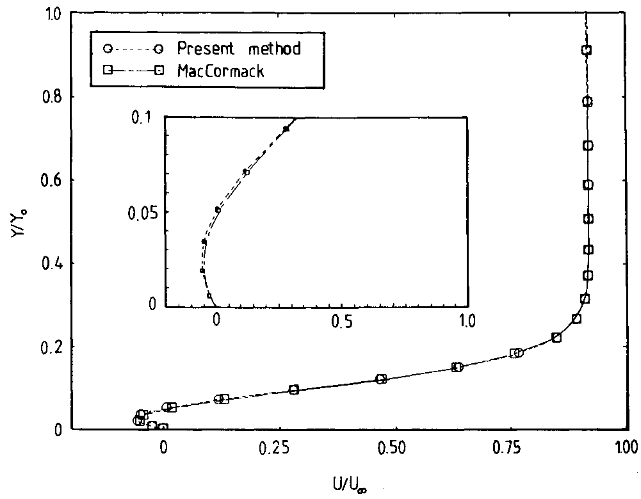

Fig. 2 Velocity profile in the separation region. The largest separation is shown; U_∞ is the velocity at the inflow boundary.

Two-Dimensional Flow

Although the SIMPLE algorithm has been described within the framework of a one-dimensional coordinate system, the basic method, together with the nonstandard features discussed in the previous section, can easily be extended to two or three dimensions. The equations of motion for two-dimensional viscous compressible flow and the extension of the original SIMPLE method to two dimensions are discussed by Van Doornmaal et al.[6] Because of the lack of space, the extension of the one-dimensional scheme to two dimensions will not be discussed in detail here. The results of a shock-boundary interaction are, however, given.

The geometry and flowfield representing the interaction are discussed by MacCormack.[1] An externally generated shock wave is incident upon the boundary layer of a flat plate. For a strong enough shock wave, the boundary layer will separate from the surface of the plate and reattach downstream. A region of rotating fluid exists between the separation and reattachment points that causes the boundary layer to thicken and generate a series of compression and expansion waves that eventually form the reflected shock wave. According to MacCormack[1] the separation region is fairly sensitive to calculation and therefore serves as a good test for a numerical method.

In Fig. 2 the calculated velocity distribution in the separation region with the present method, using a nonstaggered grid, is compared with the calculated results of the 1969 explicit method of MacCormack[1] for a shock angle of 32.585 deg and a Reynolds number of 2.9×10^5. The agreement between the two methods is good.

Concluding Remarks

For one-dimensional flow through a nozzle, it can be deduced that a mixed-differencing scheme for the convection terms and a central-differencing scheme for densities in the continuity equation considerably improve the accuracy of the SIMPLE method. For the shock-boundary interaction problem, the SIMPLE method for a nonstaggered grid with mixed differencing for the convection terms and central differencing for densities compares very well with the method of MacCormack.

References

[1]MacCormack, R. W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," AIAA Paper 81-0110, St. Louis, MO, 1981.

[2]Beam, R. M., and Warming, R. F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA Journal*, Vol. 16, No. 4, 1978, pp. 393–402.

[3]Patankar, S. V., and Spalding, D. B., "A Calculation Procedure for Heat Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," *International Journal for Heat and Mass Transfer*, Vol. 15, 1972, pp. 1787–1806.

[4]Issa, R. I., "Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting," *Journal of Computational Physics*, Vol. 61, 1985.

[5]Patankar, S. V., *Numerical Heat Transfer and Fluid Flow*, McGraw-Hill, New York, 1980.

[6]Van Doornmaal, J. P., Raithby, G. D., and McDonald, B. H., "The Segregated Approach to Predicting Viscous Compressible Fluid Flows," *Journal of Turbomachinery*, Vol. 109, 1987, pp. 268–277.

Calculation of Matched Pressure Properties of Low-Altitude Rocket Plumes

Terrence M. Cain*
New College, Oxford OX1 3BN, England, United Kingdom

Nomenclature

A = area, m
D = plume drag, N
F = actual rocket thrust, N
F_{max} = maximum possible rocket thrust, N
$\dot{m}$ = rocket exhaust mass flow, kg/s
p = pressure, Pa
R = gas constant, J/kg/K
r = plume boundary radius, m
s = distance along plume boundary, m
T = temperature, K
u = axial velocity, m/s
γ = ratio of specific heats
θ = angle of plume boundary relative to plume axis, rad

Subscripts

b = station at which the pressure within the plume is uniform and equal to the ambient pressure
c = rocket combustion chamber
e = nozzle exit plane
∞ = limiting state where p and T are absolute 0

Introduction

An accurate estimate of the temperature of the exhaust gas in the plume of a rocket at low altitude is required to predict the plume's infrared emission. Sukanek[1] proposed an approximate one-dimensional method of calculating the temperature based on the Jarvinen and Hill universal plume model. Pearce and Dash[2] corrected an inconsistency in Sukanek's approach but introduced two of their own. This Note draws attention to these errors and demonstrates that the universal plume model is inappropriate for this problem.

Discussion

Consider a control volume (cv) drawn along the boundary between the airstream and the exhaust plume of a rocket, Fig. 1. The exhaust gas enters the cv parallel with the nozzle axis and at uniform pressure p_e (the nozzle exit pressure). The gas leaves the cv far downstream in an axial direction at the pressure of the surrounding atmosphere p_b. Mixing with the airstream and acceleration of the rocket are neglected and the momentum balance is simply

$$\int_0^{S_b} 2\pi r \sin\theta (p - p_b)\, ds = \dot{m}(u_b - u_e) + (p_b - p_e) A_e = \dot{m} u_b - F \quad (1)$$

Received Nov. 20, 1992; revision received June 20, 1993; accepted for publication July 3, 1993.

*Junior Research Fellow. Member AIAA.