

A finite-volume algorithm for all speed flows

F. Moukalled and M. Darwish

*American University of Beirut, Faculty of Engineering & Architecture,
Mechanical Engineering Department, P.O.Box 11-0236, Beirut, Lebanon.*

ABSTRACT. A new collocated finite volume-based solution procedure for predicting viscous compressible and incompressible flows is presented. The technique is equally applicable in the subsonic, transonic, and supersonic regimes. Pressure is selected as a dependent variable in preference to density because changes in pressure are significant at all speeds as opposed to variations in density which become very small at low Mach numbers. The newly developed algorithm has two new features; (i) the use of the Normalized Variable and Space Formulation methodology to bound the convective fluxes; and (ii) the use of a High-Resolution scheme in calculating interface density values to enhance the shock capturing property of the algorithm.

Keywords: Pressure-based method, All speed flows, High-Resolution algorithm.

1. Introduction

In Computational Fluid Dynamics (CFD) a great research effort has been devoted to the development of accurate and efficient numerical algorithms suitable for solving flows in the various Reynolds and Mach number regimes. The type of convection scheme to be used in a given application depends on the value of Reynolds number. On the other hand, the Mach number value dictates the type of algorithm to be utilized in the solution procedure. These algorithms can be classified into two groups: density-based methods and pressure-based methods, with the former used for high Mach number flows, and the latter for low Mach number flows.

The ultimate goal however, is to develop a unified algorithm capable of solving flow problems in the various Reynolds and Mach number regimes. To understand the difficulty associated with the design of such an algorithm, it is important to understand the role of pressure in compressible flow [KAR 86]. In the low Mach number limit where density becomes constant, the role of pressure is to act on velocity through continuity so that conservation of mass is satisfied. Obviously, for low speed flows, the pressure gradient needed to drive the velocities through momentum conservation is of such magnitude that the density is not affected significantly and the flow can be considered nearly incompressible. Hence, density and pressure are very weakly related. As a result, the continuity equation is decoupled from the momentum equations and can no longer be considered as the equation for density. Rather, it acts as a constraint on the velocity field. Thus, for a sequential solution of the equations, it is necessary to devise a mechanism to couple the continuity and momentum equations through the pressure field. In the

hypersonic limit where variations in velocity become relatively small as compared to the velocity itself, the changes in pressure do significantly affect density. In this limit, the pressure can be viewed to act on density alone through the equation of state so that mass conservation is satisfied [KAR 86] and the continuity equation can be viewed as the equation for density.

The above discussion reveals that for any numerical method to be capable of predicting both incompressible and compressible fluid flows the pressure should always be allowed to play its dual role and to act on both velocity and density to satisfy continuity. Several researchers [KAR 86, RHI 86, MAR 94, DEM 93, LIE 93] have worked on extending the range of pressure-based methods to high Mach numbers. In most of the published work the first order upwind scheme is used to interpolate for density, exception being in the work presented in [DEM 93] where a central difference scheme blended with the upwind scheme is used. The blending relies on a factor varying between 0 and 1, which is problem dependent and has to be adjusted to eliminate oscillation or to promote convergence. In the work presented in [LIE 93] the retarded density concept is utilized in calculating the density at the control volume faces. This concept is based on factors that are also problem dependent and requires the addition of some artificial dissipation to stabilize the algorithm (second-order terms were introduced), which complicate its use.

To this end, the objective of this paper is to present a newly developed collocated pressure-based solution procedure that is equally valid at all Reynolds and Mach number values. The algorithm will have two new features. The first one is the use of the Normalized Variable Formulation (NVF) [LEO 87] and/or the Normalized Variable and Space Formulation (NVSF) [DAR 94] methodology in the discretization of the convective terms. The second one, is the use of High-Resolution (HR) schemes in the interpolation of density in the source of the pressure correction equation and the convective fluxes in order to enhance the shock capturing capability of the method.

The increase in accuracy with the use of HR schemes for density is demonstrated by comparing predictions, for the flow over a bump, obtained using the third-order SMART scheme for all variables except density (for which the Upwind scheme is used) against another set of results obtained using the SMART scheme for all variables including density.

2. Finite volume discretization of the transport equations

The conservation equations governing two-dimensional compressible flow problems may be expressed in the following general form:

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla \cdot (\rho\mathbf{v}\phi) = \nabla \cdot (\Gamma^\phi \nabla \phi) + Q^\phi \quad [1]$$

where ϕ is any dependent variable, \mathbf{v} is the velocity vector, and ρ , Γ^ϕ , and Q^ϕ are the density ($=P/RT$), diffusivity, and source terms, respectively. Integrating the above equation over a control volume (Fig. 1) and applying the divergence theorem, the following discretized equation is obtained:

$$\frac{\partial}{\partial t} [(\rho\phi)_p] V + (J_e + J_w + J_n + J_s) = Q^\phi V \quad [2]$$

where J_f represents the total flux of ϕ across face 'f' and is given by

$$J_f = (\rho \mathbf{v} \phi - \Gamma^\phi \nabla \phi) \cdot \mathbf{S}_f \quad [3]$$

Each of the surface fluxes J_f contains a convective contribution, J_f^C , and a diffusive contribution, J_f^D , hence:

$$J_f = J_f^C + J_f^D \quad [4]$$

where

$$J_f^C = (\rho \mathbf{v} \phi)_f \cdot \mathbf{S}_f \quad J_f^D = (-\Gamma^\phi \nabla \phi)_f \cdot \mathbf{S}_f \quad [5]$$

The diffusive flux at the control volume face 'f' is discretized using a linear symmetric interpolation profile so as to write the gradient as a function of the neighboring grid points. The convective flux across face f can be written as:

$$J_f^C = C_f \phi_f \quad [6]$$

where C_f is the convective flux coefficient at cell face 'f'. As can be seen from [6] the accuracy of the control volume solution for the convective scalar flux depends on the proper estimation of the face value ϕ_f as a function of the neighboring ϕ nodes values. Using some assumed interpolation profile, ϕ_f can be explicitly formulated in terms of its node values by a functional relationship of the form:

$$\phi_f = f(\phi_{nb}) \quad [7]$$

where ϕ_{nb} denotes the neighboring node ϕ values. After substituting [7] into [6] for each cell face and using the resulting equation along with the discretized form of the diffusive flux, [2] is transformed after some algebraic manipulations into the following discretized equation:

$$a_P^\phi \phi_P = \sum_{NB(P)} a_{NB}^\phi \phi_{NB} + b_P^\phi \quad [8]$$

where the coefficients a_P^ϕ and a_{NB}^ϕ depend on the selected scheme and b_P^ϕ is the source term of the discretized equation.

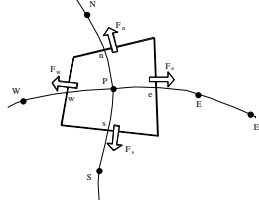


Figure 1. Control volume.

3. The NVSF methodology for constructing HR schemes

As mentioned earlier, the discretization of the convection flux is not straightforward and requires additional attention. Since the intention is to develop a high-resolution algorithm, the highly diffusive first order UPWIND scheme [PAT 81] is excluded. As such, a high order interpolation profile is sought. The difficulties associated with the use of such profiles stem from the conflicting requirements of accuracy, stability, and boundedness. Solutions predicted with high order profiles tend to provoke oscillations in the solution. To suppress these

oscillations, the composite flux limiter method [LEO 88] is adopted here. The formulation of high-resolution flux limiter schemes on uniform grid has recently been generalized in [LEO 88] through the Normalized Variable Formulation (NVF) methodology and on non-uniform grid in [DAR 94] through the Normalized Variable and Space Formulation (NVSF) methodology. To the authors' knowledge, the NVSF formulation has never been used to bound the convection flux in compressible flows. It is an objective of this work to extend the applicability of this technique to compressible flows. For more details the reader is referred to [DAR 94].

4. High resolution algorithm

The need for a solution algorithm arises in the simulation of flow problems because a scalar equation does not exist for pressure. Hence, if a segregated approach is to be adopted, coupling between the u , v , ρ , and P primitive variables in the continuity and momentum equations will be required.

The segregated algorithm adopted in this work is the SIMPLE algorithm [PAT 81], which involves a **predictor** and a **corrector** step. In the **predictor** step, the velocity field is calculated based on a guessed or estimated pressure field. In the **corrector** step, a pressure (or a pressure-correction) equation is derived and solved. Then, the variation in the pressure field is accounted for within the momentum equations by corrections to the velocity and density fields. Thus, the velocity, density, and pressure fields are driven, iteratively, to better satisfying the momentum and continuity equations simultaneously and convergence is achieved by repeatedly applying the above-described procedure.

The key step in deriving the pressure-correction equation is to notice that in the predictor stage a guessed or estimated pressure field from the previous iteration, denoted by $P^{(n)}$, is substituted into the momentum equations, the resulting velocity field, denoted by \mathbf{v}^* , which now satisfies the momentum equations, will, in general, not satisfy the continuity equation. Thus, a correction is needed in order to obtain a velocity and pressure fields that satisfy both equations. Denoting the pressure, velocity, and density corrections by P' , $\mathbf{v}'(u', v')$, and ρ' , respectively, the corrected fields are given by:

$$P = P^{(n)} + P', \quad \mathbf{v} = \mathbf{v}^* + \mathbf{v}', \quad \text{and} \quad \rho = \rho^{(n)} + \rho' \quad [9]$$

Combining momentum and continuity and substituting P , \mathbf{v} , and ρ using [9], the final form of the pressure-correction equation is:

$$a_P^{P'} P'_P = a_E^{P'} P'_E + a_W^{P'} P'_W + a_N^{P'} P'_N + a_S^{P'} P'_S + b_P^{P'} \quad [10]$$

where

$$b_P^{P'} = - \frac{(\rho_P - \rho_P^0)}{\delta t} V - \left[\rho_e^* U_e^* + \rho_w^* U_w^* + \rho_n^* U_n^* + \rho_s^* U_s^* \right] \quad [11]$$

From [11] it is clear that the starred continuity equation appears as a source term in the pressure correction equation. Moreover, in a pressure-based algorithm, the pressure-correction equation is the most important equation that gives the pressure, upon which all other variables are dependent. Therefore, the solution accuracy depends on the proper estimation of pressure from this equation. Definitely, the

more accurate the interpolated starred density (ρ^*) values at the control volume faces are, the more accurate the predicted pressure values will be. The use of a central difference scheme for the interpolation of ρ^* leads to instability at Mach numbers near or above 1 [KAR 86, DEM 93]. On the other hand the use of a first order upwind scheme lead to excess diffusion [KAR 86]. The obvious solution would be to interpolate for values of ρ^* at the control volume faces the same way interpolation for other dependent variables is carried. That is to employ the bounded HR family of schemes for which no problem-dependent factors are required. Adopting this strategy, the discretized form of the starred continuity equation becomes:

$$b_p^{p'} = -\frac{(\rho_p - \rho_p^o)}{\delta t} V - \left[(\rho_e^*)^{HR} U_e^* + (\rho_w^*)^{HR} U_w^* + (\rho_n^*)^{HR} U_n^* + (\rho_s^*)^{HR} U_s^* \right] \quad [12]$$

The same procedure is also adopted for calculating the density when computing the mass flow rate at a control volume face in the general conservation equation.

5. Results and discussion

The validity of the above described solution procedure is demonstrated in this section by presenting solutions to the inviscid flow over a bump. The physical situation consists of a channel of width equal to the length of the circular arc bump and of total length equal to three lengths of the bump. Results are presented for three different types of flow (subsonic, transonic, and supersonic). For subsonic and transonic calculations, the thickness-to-chord ratio is 10% and for supersonic flow calculations it is 4%. In all flow regimes, predictions obtained over a relatively coarse grid using the SMART scheme for all variables including density are compared against results obtained over the same grid using the SMART scheme for all variables except density, for which the UPWIND scheme is used. Due to the unavailability of an exact solution to the problem, a solution using a dense grid is generated and treated as the most accurate solution against which coarse grid results are compared.

5.1 Subsonic flow over a circular arc bump

With an inlet Mach number of 0.5, the inviscid flow in the channel is fully subsonic and symmetric across the middle of the bump. Isobars displayed in Fig. 2(a) reveal that the coarse grid solution obtained with the SMART scheme for all variables falls on top of the dense grid solution. The use of the upwind scheme for density however, lowers the overall solution accuracy. The same conclusion can be drawn when comparing the Mach number distribution along the lower and upper walls of the channel. As seen in Fig. 2(b), the coarse grid profile obtained using the SMART scheme for density is closer to the dense grid profile than the one predicted employing the upwind scheme for density. The difference in results between the coarse grid solutions is not large for this test case. This is expected since the flow is subsonic and variations in density are relatively small. Larger differences are anticipated in the transonic and supersonic regimes.

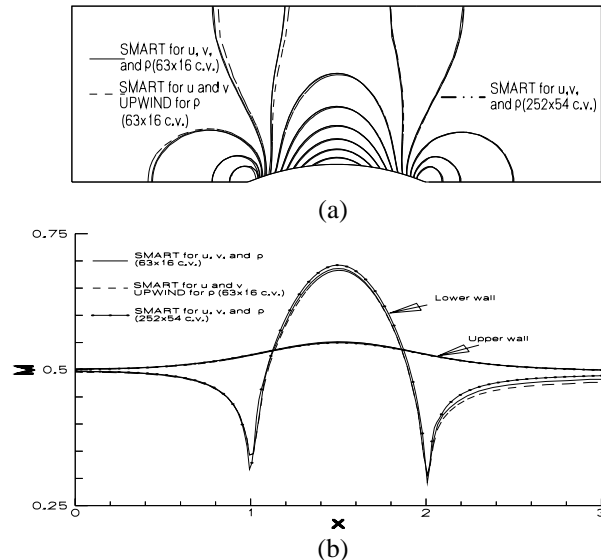


Figure 2. Subsonic flow over a 10% circular bump; (a) isobars and (b) profiles along the walls.

5.2 Transonic flow over a circular arc bump

Results for an inlet Mach number of 0.675 are displayed in Fig. 3. Figure 3(a) presents a comparison between the coarse grid and dense grid results. As shown, the use of the HR SMART scheme for density greatly improves the predictions. Isobars generated over a coarse grid (63x16 c.v.) using the SMART scheme for all variables are very close to the ones obtained with a dense grid (252x54 c.v.). This is in difference with coarse grid results obtained using the upwind scheme for density and the SMART scheme for all other variables, which noticeably deviate from the dense grid solution. This is further apparent in Fig. 3(b) where Mach number profiles along the lower and upper walls are compared. As shown, the most accurate coarse grid results are those obtained with the SMART scheme for all variables and the worst ones are achieved with the upwind scheme for all variables. The maximum Mach number along the lower wall ($\cong 1.41$), predicted with a dense grid, is in excellent agreement with published values [DEM 93]. By comparing course grid profiles along the lower wall, the all-SMART solution is about 11% more accurate than the solution obtained using SMART for all variables and upwind for density and 21% more accurate than the highly diffusive all-upwind solution.

5.3 Supersonic flow over a circular arc bump

Computations are presented for an inlet Mach number value of 1.4. Mach number contours are compared in Fig. 4(a). As before, the course grid all-SMART results (58x18 c.v.), being closer to the dense grid results (158x78 c.v.), are more accurate than those obtained when using the upwind scheme for density. The Mach profiles along the lower and upper walls, depicted in Fig. 4(b), are in excellent

agreement with published results [NI 82] and reveal good enhancement in accuracy when using the SMART scheme for evaluating interface density values. The use of the upwind scheme to compute density deteriorates the solution accuracy even though a HR scheme is used for other variables. The all-upwind results are highly diffusive.

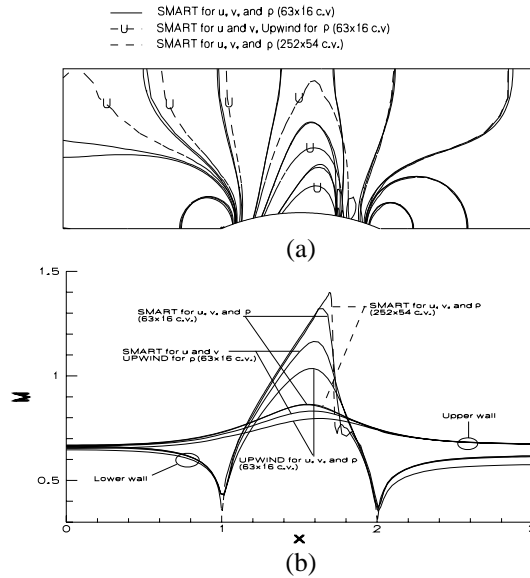


Figure 3. Transonic flow over a 10% circular bump; (a) isobars using various schemes, and (b) profiles along the walls.

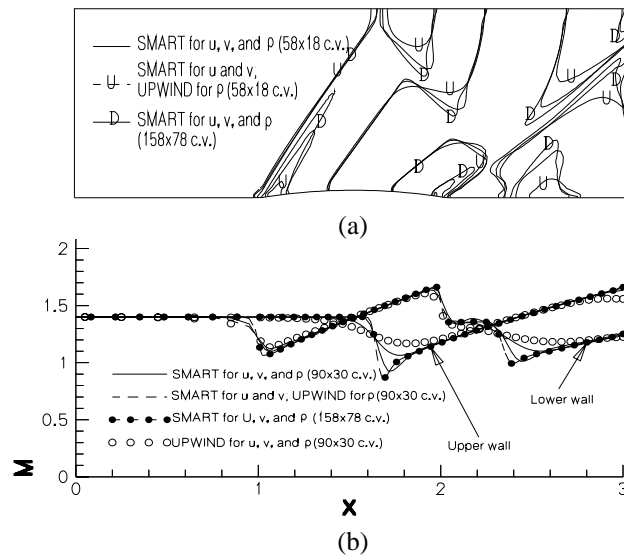


Figure 4. Supersonic flow over a 4% circular bump ($M_{in}=1.4$); (a) Mach number contours using various schemes, (b) profiles along the walls.

6. Concluding Remarks

A new high-resolution pressure-based algorithm for the solution of fluid flow at all speeds was formulated. The new features in the algorithm are the use of a HR scheme in calculating the density values at the control volume faces and the use of the NVSF methodology for bounding the convection fluxes. Results obtained were very promising and revealed good enhancement in accuracy at high Mach number values when calculating interface density values using a High-Resolution scheme.

7. Acknowledgments

The financial support provided by the European Office of Aerospace Research and Development (EOARD) (SPC-99-4003) is gratefully acknowledged.

8. Bibliography

- [DAR 94] Darwish, M.s. and Moukalled, F., "Normalized Variable and Space Formulation Methodology For High-Resolution Schemes," Numerical Heat Transfer, Part B, vol. 26, pp. 79-96, 1994.
- [DEM 93] Demirdzic, I., Lilek, Z., and Peric, M., "A Collocated Finite Volume Method For Predicting Flows at All Speeds," International Journal for Numerical Methods in Fluids, vol. 16, pp. 1029-1050, 1993.
- [KAR 86] Karki, K.C., "A Calculation Procedure for Viscous Flows at All Speeds in Complex Geometries," Ph.D. Thesis, University of Minnesota, June 1986.
- [LEO 87] Leonard, B.P., "Locally Modified Quick Scheme for Highly Convective 2-D and 3-D Flows," Taylor, C. and Morgan, K. (eds.), Numerical Methods in Laminar and Turbulent Flows, Pineridge Press, Swansea, U.K., vol. 15, pp. 35-47, 1987.
- [LEO 88] Leonard, B.P., "Simple High-Accuracy Resolution Program for Convective Modelling of Discontinuities," International Journal for Numerical Methods in Engineering, vol. 8, pp. 1291-1318, 1988.
- [LIE 93] Lien, F.S. and Leschziner, M.A., "A Pressure-Velocity Solution Strategy for Compressible Flow and Its Application to Shock/Boundary-Layer Interaction Using Second-Moment Turbulence Closure," Journal of Fluids Engineering, vol. 115, pp. 717-725, 1993.
- [MAR 94] Marchi, C.H. and Maliska, C.R., "A Non-orthogonal Finite-Volume Methods for the Solution of All Speed Flows Using Co-Located Variables," Numerical Heat Transfer, Part B, vol. 26, pp. 293-311, 1994.
- [NI 82] Ni, R.H., "A Multiple Grid Scheme for Solving the Euler Equation," AIAA Journal, vol. 20, pp. 1565-1571, 1982.
- [PAT 81] Patankar, S.V., Numerical Heat Transfer and Fluid Flow, Hemisphere, N.Y., 1981.
- [RHI 86] Rhie, C.M., "A Pressure Based Navier-Stokes Solver Using the Multigrid Method," AIAA paper 86-0207, 1986.