J. Indian Inst. Sci., July–Aug. 1995, 75, 373–387.

A comparative evaluation of upwind schemes for compressible Euler and Navier–Stokes equations

J. RICHARD AMALADAS AND HEMANT KAMATH
Department of Aerospace Engineering, Indian Institute of Science, Bangalore 560 012

Abstract

Issues involved in the comparative evaluation of upwind schemes for Euler and Navier–Stokes equations have been addressed. Apart from accuracy, these include robustness and computational efficiency attained in the presence of convergence acceleration devices. A survey of upwind schemes and convergence acceleration devices is presented. A selective study, based on a comparison of accuracy for Roe and Osher schemes, has been conducted in a unified framework. Computations in subsonic, transonic, supersonic and hypersonic regimes have been carried out and comparisons of lift, drag, skin-friction and heat-transfer coefficients with reference solutions have been reported.

Keywords: Upwind schemes, Roe, Osher, Euler and Navier–Stokes equations.

1. Introduction

This investigation is motivated by the necessity for a detailed evaluation of numerical schemes based on upwind algorithms for inviscid and viscous compressible flow. Significant developments in computational fluid dynamics (CFD) for Euler and Navier–Stokes equations have occurred during the past decade. The advances in high-performance computers coupled with the evolution of sophisticated algorithms continue to provide impetus for the computation of fluid flows with ever-increasing complexity, both geometrically and physically. In spite of all the progress in algorithm development, many impediments still need to be overcome before CFD techniques can be routinely used for the design of aerospace vehicles, which require robust, accurate and efficient computations of complex flows. The resolution of flow phenomena to an appropriate extent for a large set of design and off-design parameters involves computational time scales which are prohibitive even with the high-performance computers available at present. This problem gets compounded when the conservation equations for mass, momentum and energy have to be supplemented by additional conservation equations representing complex physical processes, which include compressible turbulence and chemical reactions, among others. These considerations necessitate the optimization of algorithms for Euler and Navier–Stokes equations to a high degree. In particular, the role of convergence acceleration becomes increasingly important at the cutting edge of CFD research.

Modern numerical schemes for compressible Euler and Navier–Stokes equations have been remarkably successful in predicting flow fields containing intricate patterns involving shocks, expansions and slip surfaces/vortex sheets. In particular, shock-capturing techniques have the inherent ability to predict the locations and strength of flow discontinuities and their interaction without prior knowledge of their presence. These schemes can be classified as central or upwind, based on the spatial discretization of convective terms appearing in the governing equations. The popularity of central difference schemes in the past could mainly be attributed to the ease with which their implementation could be carried out. Unfortunately, these schemes require artificial dissipation terms to ensure stability while computing flow fields containing moderate to strong discontinuities, which introduces some degree of empiricism, leading to loss of robustness. On the contrary, upwind schemes are naturally dissipative since their construction involves a more rigorous representation of the associated flow physics.

2. Survey of upwind schemes

Numerical schemes based on upwind algorithms[1-5] have convincingly demonstrated their superiority for the computation of inviscid and viscous compressible flow[6,7]. Upwind schemes can broadly be classified as flux-vector- and flux-difference-splitting schemes.

2.1. *Flux-vector-splitting schemes*

Flux vector splitting for Euler equations is a consequence of regarding the fluid as an ensemble of particles, some of them moving forward and the others moving backward. This forms the basis of splitting the fluxes of mass, momentum and energy into forward and backward components. Flux vector splitting can be interpreted as an approximate technique for integrating the collisionless Boltzmann equation[8] and the resulting numerical schemes are termed Boltzmann-type[9]. The most prominent upwind schemes, based on the flux-vector-splitting concept, are due to Steger–Warming[2] and Van Leer[3]. The Steger–Warming splitting exploits the homogeneous property of the inviscid flux vector to construct the split fluxes for a perfect gas. However, the lack of homogeneity prevents the straightforward extension of Steger–Warming splitting for real gases. The nondifferentiability of split fluxes at sonic and stagnation points even for a perfect gas constitutes a serious deficiency in the Steger–Warming splitting technique. Van Leer flux vector splitting, developed subsequently based on certain mathematical constraints, was designed to overcome the limitations and deficiencies of the Steger–Warming splitting technique. The continuous differentiability of split fluxes, due to its polynomial representation in terms of the Mach number, enhances the convergence characteristics of the Van Leer scheme, resulting in unrivalled superiority for Euler computations. In addition, the extension of Van Leer splitting for real gases is straightforward, with an intrinsic simplicity not found in many other upwind schemes. Flux-vector-splitting schemes have proved to be extremely successful in capturing steady discontinuities represented by nonlinear waves, which include shocks. On the contrary, these schemes are not effective in capturing steady discontinuities represented by linear waves which result in diffusion of contact discontinuities and slip surfaces. For these reasons, flux-vector-splitting schemes are primarily suited for Euler computations. Navier–Stokes computations with these schemes result in artificial thickening of shear layers[7] unless the computational grid is sufficiently fine.

2.2. *Flux-difference-splitting schemes*

Flux-differences-splitting schemes, by virtue of their superior ability to capture steady discontinuities corresponding to linear and nonlinear waves, have been favoured for Navier–Stokes computations in recent years. These schemes are based on noniterative solution procedure for the linearized analogue of the classical Riemann problem[10] and hence they are referred to as approximate Riemann solvers. The most popular upwind schemes, based on flux-difference-splitting concept, are due to Roe[4] and Osher[5]. Roe's scheme involves determining the solution of the linearized Riemann problem, obtained by replacing the exact Jacobian of the inviscid flux by a mean-valued analogue satisfying certain constraints. Osher's scheme represents the flux difference as a path integral in state space and then the Jacobian components corresponding to forward and backward waves are integrated along a path which is piecewise aligned with the corresponding eigenvectors of the Jacobian matrix. The real gas extensions of flux-difference-splitting schemes due to Roe and Osher are much more complicated than their perfect gas versions. This is primarily due to the necessity of introducing nonphysical auxiliary variables to ensure that the solution procedure for the Riemann problem remains noniterative even for real gases.

3. Convergence acceleration devices

With a variety of possibilities for the spatial discretization of convective terms based on upwind algorithms alone, the techniques for time integration are comparatively limited. Time integration techniques can be classified as time-accurate or time-inaccurate. Accuracy and stability are two important aspects of a time-integration scheme and for a time-accurate technique both are equally significant whereas for a time-inaccurate technique only the latter is relevant. Time-accurate techniques are necessary when the transient evolution of the flow field needs to be resolved by the numerical schemes. In many instances, however, only the steady-state solution of an initial-boundary-value problem is sought and time-inaccurate techniques are preferred for their computational efficiency. In such formulations, the steady-state solution can be obtained by marching the governing equations in time by starting from a suitable set of initial values while imposing the appropriate boundary conditions until all the transient terms become negligible. Time-inaccurate techniques degrade the time variable, which is primarily responsible for the physical evolution of the flow field, to an iteration parameter for convergence acceleration towards steady state.

3.1. *Implicit time integration techniques*

Time integration techniques could alternately be classified as explicit or implicit. Apart from minimal requirements of computer memory, explicit marching schemes are endowed with an appealing simplicity and readily lend themselves to vectorization and parallelization procedures. However, these schemes are constrained by a local time step limitation, which is of the order of the smallest time scale that occurs in a computational cell during the physical evolution of the flow field. Hence, explicit marching schemes are preferred for time-accurate computations, where it becomes necessary to constrain the time steps as dictated by the resolution requirements of the flow transients. However, explicit schemes do not possess the desired computational efficiency for obtaining steady-state solutions of Euler and Navier–Stokes equations. Implicit schemes, endowed with superior damping characteristics for solution transients, are preferred for time-inaccurate computations. They permit significantly larger time steps compared to explicit schemes and thus offer enhanced robustness for steady-state computations. However, implementation of implicit schemes requires computationally expensive Jacobian matrices, whose construction gets further complicated when it involves upwind discretization of convective terms. Upwind schemes with first-order representation of convective terms in the implicit operator enhance the diagonal dominance of the resulting block matrix system of equations, which can be efficiently solved using approximate factorization or relaxation techniques. A variety of these convergence acceleration strategies for upwind schemes have been incorporated in industrial computer codes such as CFL3D (NASA, Langley), GASP (NASA, Langley), USA (Rockwell International), CNS(NASA, Ames) among others, which are used for flow computations involving complex aerodynamic configurations.

As far as implicit upwind schemes are concerned, the Jacobian matrices arising from flux-vector-splitting algorithms of Steger–Warming and Van Leer are algebraically and computationally simpler than those resulting from flux-difference-splitting algorithms of Roe and Osher. However, the algorithms due to Van Leer and Osher yield continuously differentiable fluxes which enhance the convergence characteristics of implicit schemes. In recent years, hybrid algorithms have been proposed[11,12] which attempt to combine the computational efficiency of flux vector splitting with the accuracy of flux-difference-splitting schemes. In this category of schemes, AUSM (advection upstream splitting method) and HUS (hybrid upwind splitting) are prominent in having achieved remarkable success, overcoming some of the deficiencies of existing upwind schemes[11,12]. However, the development of hybrid schemes has occurred relatively recently and it would be premature to draw comprehensive conclusions based on limited computational results. Hybridization procedures involving flux-vector-splitting and flux-difference-splitting schemes have been explored in the past[13] in an attempt to improve upon the computational efficiency of implicit schemes. This is based on the rationale that for implicit time integration schemes, the steady-state solutions of Euler and Navier–Stokes equations are governed by the explicit operator, whereas the implicit operator is primarily responsible for convergence. It would be advantageous to invoke an inconsistent linearization strategy to select the implicit operator based on a scheme which yields continuously differentiable fluxes and then the choice of explicit operator would be dictated by the scheme which yields the desired steady-state solution. Several possible combinations of explicit and implicit operators, based on upwind schemes due to Steger–Warming, Van Leer and Roe, have been analysed by Liou and Van Leer[13] for computational performance involving Euler and Navier–Stokes equations.

3.2. *Generalized conjugate–gradient method*

Apart from implicit time integration techniques, a variety of convergence acceleration devices with differing complexities can be availed for improving the efficiency of numerical schemes for steady-state computations. They include generalized conjugate gradient, multigrid and preconditioning methods among others, that have been tried out for

Euler and Navier–Stokes equations. The viability of preconditioning techniques coupled with GMRES (Generalized Minimum RESidual) algorithm, which is a conjugate gradient-type iterative procedure, has been demonstrated by Venkatakrishnan[14] for compressible Navier–Stokes computations. A major difficulty with GMRES is that the computational work per iteration and the overall storage requirements grow linearly with the number of iterations involved in the minimization procedure. Consequently, it is highly impractical to execute the full version of the algorithm and it becomes necessary to resort to restart options, which often results in slow convergence for complex flow problems. One of the most efficient and universal techniques available for convergence acceleration is the multigrid method, which was originally developed for elliptic partial differential equations and has been extended to a much more general category of equations by Brandt[15].

3.3. *Multigrid method*

The multigrid method[16] involves the iterative solution of a system of discrete equations on a prescribed grid, by repetitive interactions with a hierarchy of coarser grids, which exploits the inherent relationships that exist among the discretized governing equations on a sequence of grids. Typically, each grid is involved in a two-way interaction process that improves accuracy on the next coarser grid by correcting its discretized governing equation and additionally it provides corrections to the approximate solution of the next finer grid. The principal advantage of multigrid over other acceleration techniques can be attributed to its convergence rate being nearly independent of the size of the system whose solution is sought. In contrast, the rate of convergence of other acceleration techniques degrades noticeably as the system size increases and this can be readily observed for computations involved in grid refinement studies. Furthermore, the additional storage requirements necessary for the implementation of multigrid acceleration is minimal if the data structure for the sequence of grids can be organized based on a compact representation. The application of multigrid acceleration for compressible Euler and Navier–Stokes computations based on upwind schemes is still a subject of considerable research.

3.4. *Preconditioning techniques*

In recent years, preconditioning algorithms have been developed for steady-state solutions of Euler and Navier–Stokes equations[17] to alleviate the stiffness caused by the disparity in characteristic wave speeds of the convective terms. This disparity is pronounced for stagnation and transonic flow conditions apart form low Mach number flows tending towards the incompressible limit. In such situations, a deterioration in convergence rates of numerical schemes can be observed even when acceleration devices such as implicit time integration and multigrid techniques are availed[17]. Furthermore, loss of accuracy of steady-state solutions has been reported for computations involving low Mach number incompressible flows based on algorithms developed for compressible flows. To overcome the degradation in accuracy and convergence, local preconditioning techniques have been proposed to reduce the disparity in characteristic wave speeds of the convective terms of Euler and Navier–Stokes equations. The time evolution is incorrectly described by the preconditioned governing equations, which may be acceptable provided the steady-state solution is unaltered.

The present research[18] is directed towards a comprehensive investigation involving the flux-vector-splitting algorithms due to Steger–Warming and Van Leer, apart from the flux-difference-splitting algorithms due to Osher and Roe. Two hybrid algorithms, AUSM[11] and HUS[12], have also been considered in this investigation[18]. The performance of these algorithms has been analysed in a unified framework based on two-dimensional Euler and Navier–Stokes computations in subsonic, transonic, supersonic and hypersonic flow regimes. Apart from accuracy, the upwind algorithms have been evaluated for their robustness and computational efficiency achieved relative to various convergence acceleration devices. These include implicit time integration involving a variety of relaxation strategies, multigrid and local preconditioning techniques among others, which have been systematically investigated on an individual as well as cumulative basis. The details of the investigation can be found in Amaladas[18] and the remaining part of the paper will be limited to a selective study from this source. In particular, a direct comparison of flux-difference-splitting schemes due to Roe and Osher has not been established earlier[13]. A comparison of accuracy for these two schemes based on Euler and Navier–Stokes computations will be presented here, whereas the issues pertaining to computational efficiency can be found in Amaladas[18]. A variety of inviscid and viscous compressible flows, which are well documented in literature, have been computed in a finite-volume framework with third-order accuracy attained by incorporating a MUSCL strategy[19]. A comparative evaluation of Roe and Osher schemes involving flow field computations in subsonic, transonic, supersonic and hypersonic regimes has been conducted, which includes comparisons of lift, drag, skin-friction and heat-transfer coefficients with extensively validated computational and experimental results.

4. Euler computations

A series of Euler computations were carried out that would clearly indicate the similarities and reveal the differences between the two flux-difference-splitting schemes that were selected for this comparative evaluation. A standard test case involving flow past a NACA 0012 airfoil in the subsonic, transonic and supersonic flow regimes has been chosen

Table I
Comparison of reference and computed lift and drag coefficients for inviscid flow past NACA 0012 airfoil

Test case parameters	*Reference values*	*Roe scheme*	*Osher scheme*
$M_\infty = 0.63$, $\alpha = 2.0°$	$C_L = 0.3335$ $C_D = 0.00003$	$C_L = 0.33373$ $C_D = 0.00017$	$C_L = 0.33372$ $C_D = 0.00017$
$M_\infty = 0.80$, $\alpha = 1.25°$	$C_L = 0.3632$ $C_D = 0.0230$	$C_L = 0.35552$ $C_D = 0.02258$	$C_L = 0.35495$ $C_D = 0.02255$
$M_\infty = 0.85$, $\alpha = 1.0°$	$C_L = 0.3793$ $C_D = 0.05760$	$C_L = 0.38065$ $C_D = 0.05467$	$C_L = 0.37992$ $C_D = 0.05464$
$M_\infty = 1.20$, $\alpha = 0.0°$	$C_L = 0.0000$ $C_D = 0.09600$	$C_L = 0.00000$ $C_D = 0.09592$	$C_L = 0.00000$ $C_D = 0.09590$

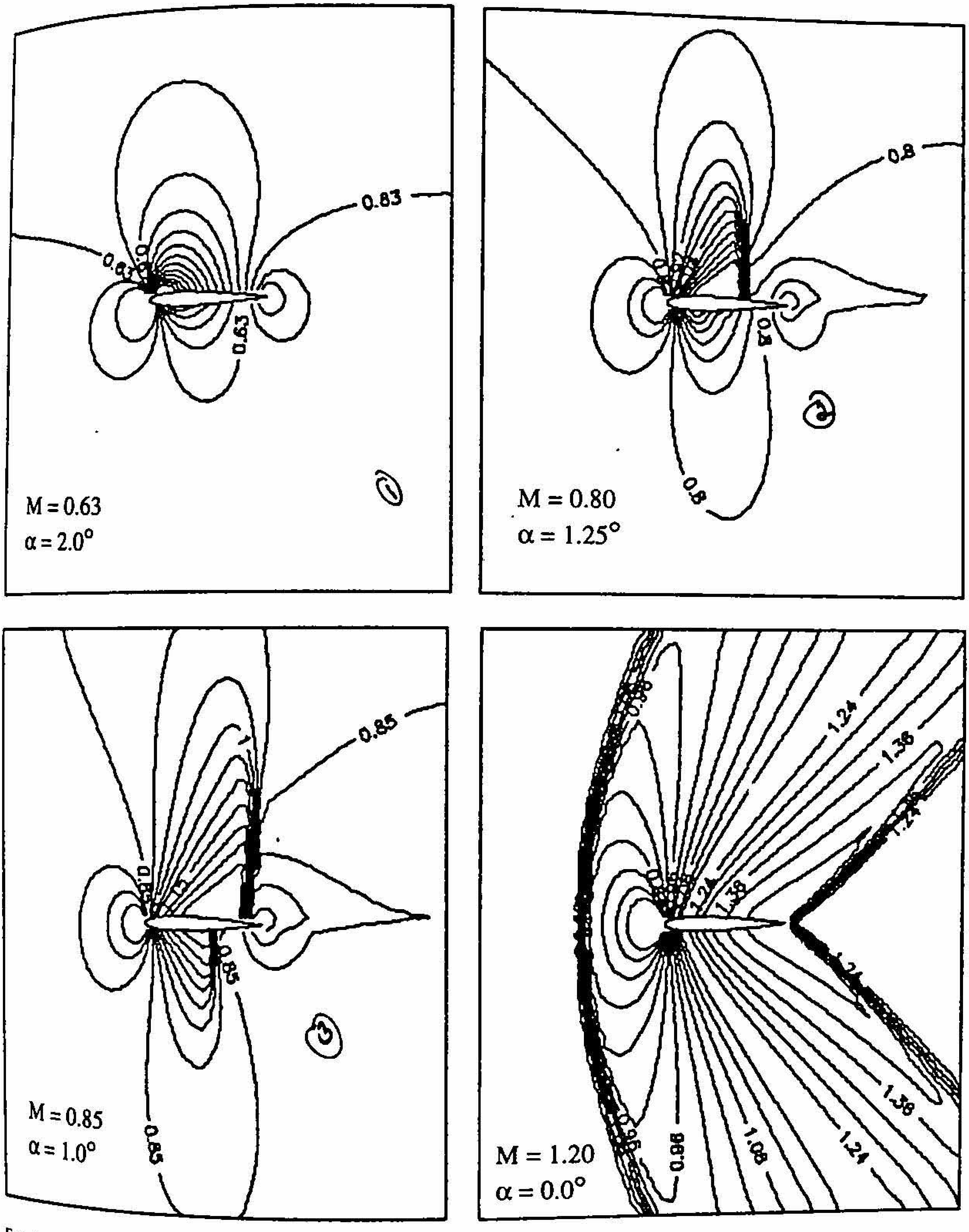


FIG. 1. Mach contours for inviscid flow over NACA 0012 airfoil based on Roe's scheme.

to evaluate the relative accuracy of Roe and Osher schemes compared to reference solutions reported in the literature[20, 21]. In particular, lift and drag coefficients obtained from the computed solutions form the basis of comparison. Table I contains the four sets of parameters for which computations were carried out on a 128×32 grid for both the

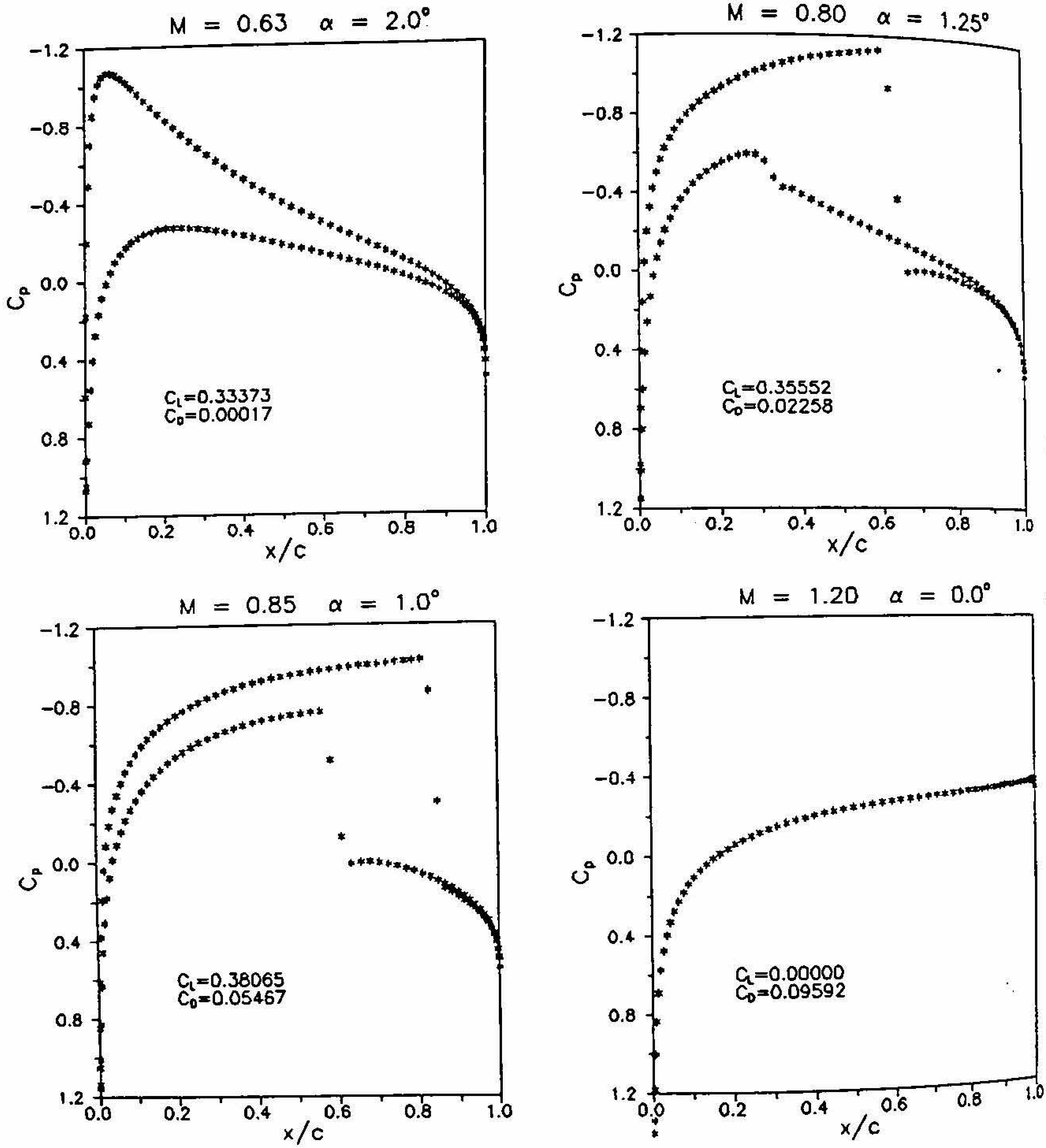


FIG. 2. Surface-pressure distributions for inviscid flow over NACA 0012 airfoil based on Roe's scheme.

schemes in a unified framework. Other entries in the table include the lift and drag coefficients obtained from the computed solutions in addition to the coefficients reported in the literature[20,21] for these cases.

The flow features corresponding to these four cases are depicted in Fig. 1 in the form of Mach contours obtained from the computed solutions based on Roe's scheme. The Mach contours from Osher's scheme are not shown here since they are virtually

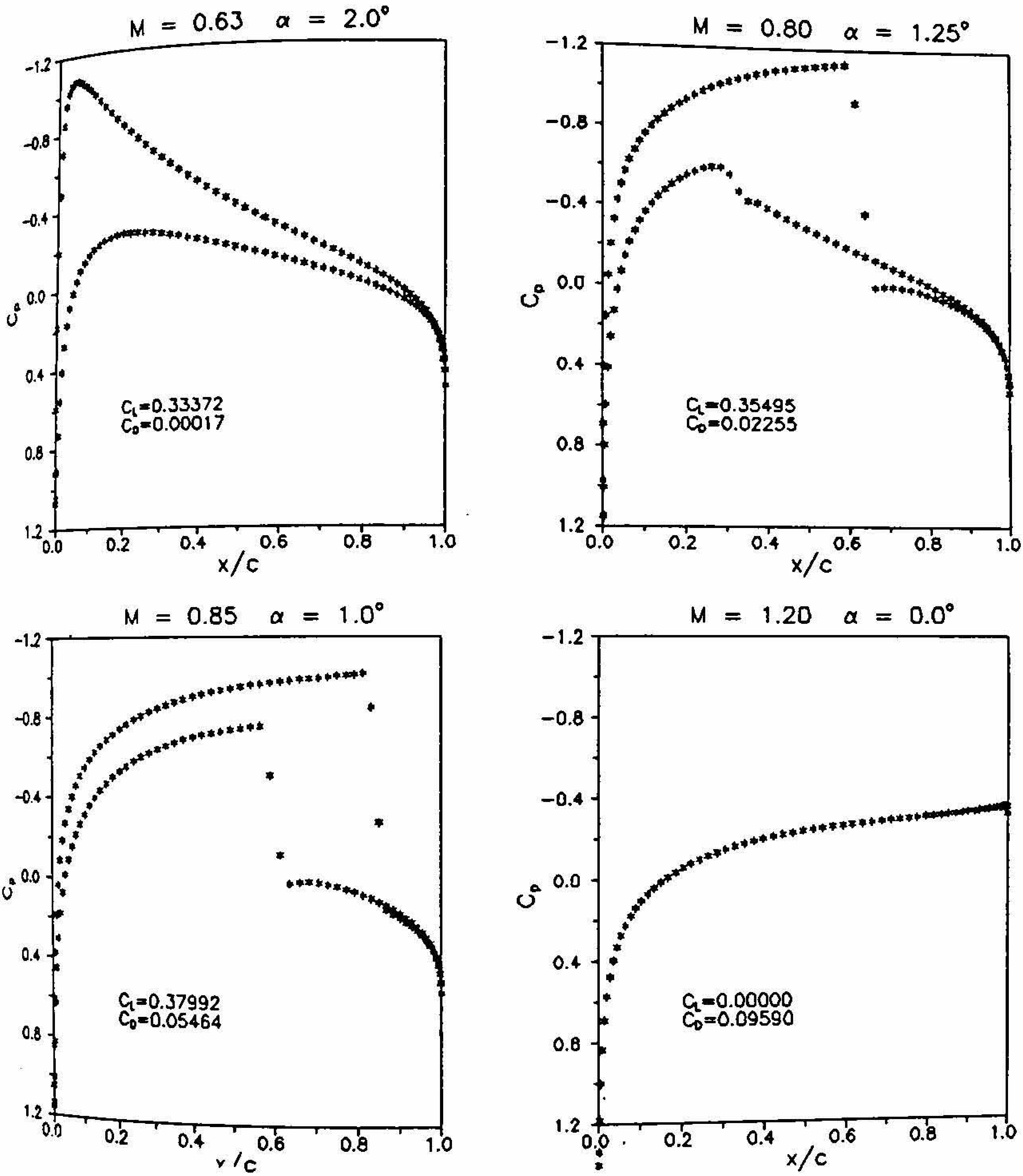


FIG. 3. Surface-pressure distributions for inviscid flow over NACA 0012 airfoil based on Osher's scheme.

indistinguishable from those in Fig 1. The coefficients of pressure distributions on the lower and upper surfaces of the airfoil in each of the four cases are shown in Figs 2 and 3 for the two flux-difference-splitting schemes. The first case corresponds to a shock less subsonic flow, where an exact solution[22] based on a hodograph method is also available for comparison. For a NACA 0012 airfoil which has been extended to a sharp trailing edge, the hodograph method yields a lift coefficient $C_L = 0.335$. In this case the drag coefficient $C_D \equiv 0$, which is in accordance with the so-called D'Alembert's zero-drag paradox. It may be noted that the lift and drag coefficients obtained from Roe and Osher schemes are nearly identical, comparing quite favourably with reference values listed in Table I. The reference drag coefficient is in better agreement with D'Alembert's zero-drag paradox since the reference solutions were computed on a much finer grid than that corresponding to the present study. The second and third cases correspond to transonic flows with shocks on upper and lower surfaces of the airfoil. In the second case, the shock present on the lower surface of the airfoil is extremely weak and is not clearly revealed by the Mach contours in Fig. 1. This weak shock is better represented in the coefficient of pressure distributions for the two schemes as shown in Figs 2 and 3. It can be observed from these figures that both these schemes are capable of resolving shocks with at most two interior zones for two-dimensional transonic flow cases. However, for one-dimensional flows, Roe's scheme can represent shocks with zero or one interior zone as compared to two for Osher's scheme[18]. Unlike Roe's scheme, the flux-difference-splitting scheme due to Osher yields continuously differentiable fluxes which contribute to the enhanced convergence of implicit schemes. However, this advantage is more than offset by the computationally expensive expression for the flux resulting from Osher's algorithm. Extensive comparisons of the computational efficiencies of implicit Roe and Osher schemes have been carried out by Amaladas[18]. As far as accuracy is concerned, these two schemes yield virtually identical results for the lift and drag coefficients as well as for the coefficient of pressure distributions for the four cases.

A prominent failing of Roe's scheme has been observed[23–25] for blunt-body computations, where it is possible to obtain a spurious solution in which a protuberance appears ahead of the bow shock along the stagnation line. This 'carbuncle phenomenon' is found to be more pronounced at higher Mach numbers for a grid which is closely aligned with

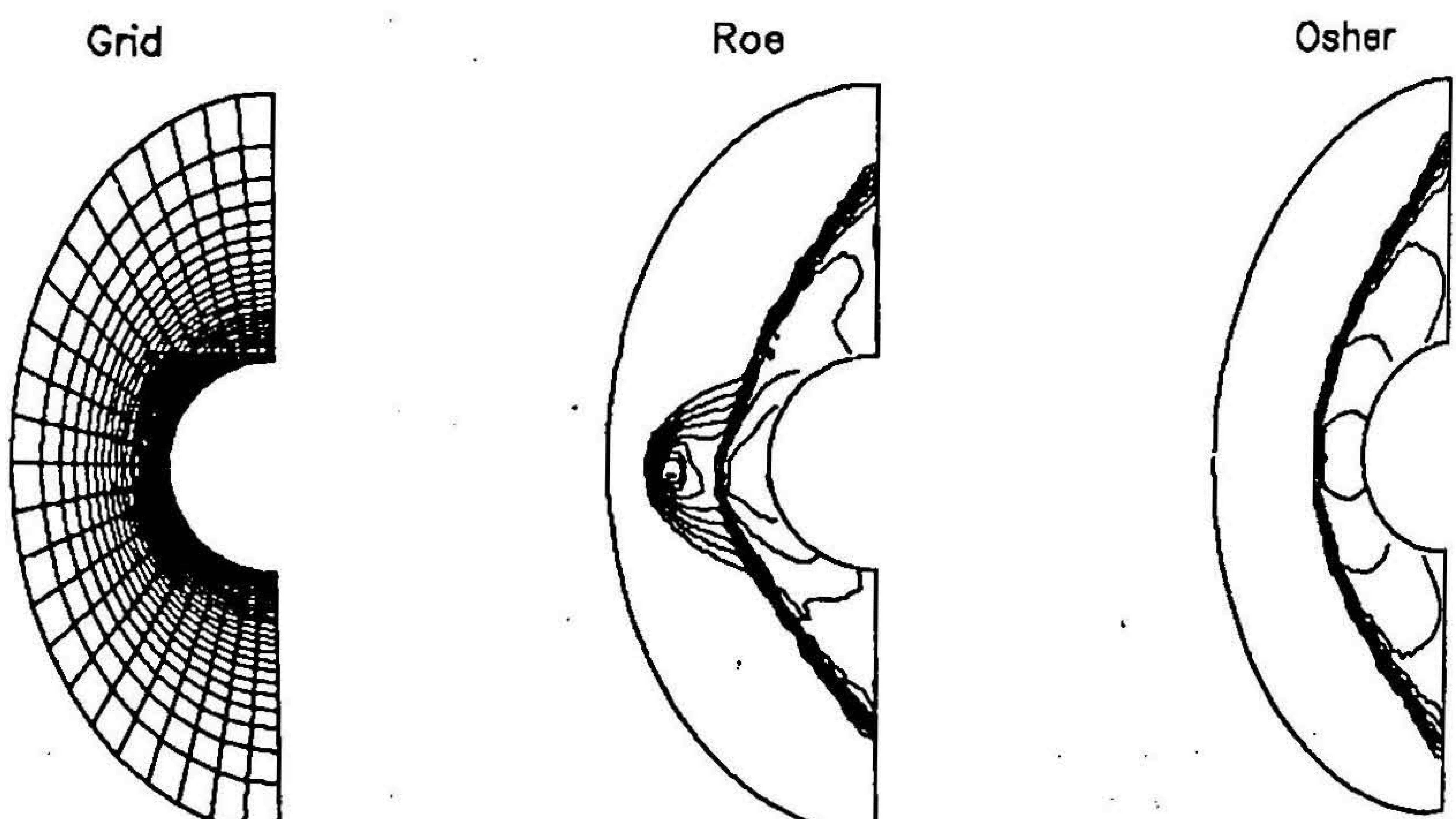


FIG. 4. Mach contours for inviscid flow over a blunt-body based on Roe and Osher schemes.

the bow shock. The failure of Roe's scheme in this case is attributed to the lack of dissipation provided by the linear waves, in a direction tangential to the shock, for counter acting the perturbations carried by the nonlinear waves[25]. It has been conjectured[24] that Osher's scheme would also be plagued by this carbuncle phenomenon for blunt-body computations. To characterize this phenomenon for the two flux-difference-splitting schemes, computations involving supersonic flow past a circular cylinder were carried out. With a free-stream Mach number of 6.0, the grid was adjusted until Roe's scheme yielded a pronounced carbuncle shock, which is displayed in Fig. 4 in the form of Mach contours. It was found that with the same grid as shown in the figure. Osher's scheme did not exhibit any visible evidence of a carbuncle shock. The Mach contours obtained from the computed solutions based on Osher's scheme are shown in Fig. 4. However, upon further examination it was found that the solution did develop a mild asymmetry but it was confined to a small region in the neighbourhood of the stagnation point. It was also observed that further adjustment of the grid to the bow shock could not produce any noticeable asymmetry in the solution obtained with Osher's scheme. Even though the lack of dissipation at the stagnation point does not have a significant influence on the accuracy of the solution in this case, the resulting degradation

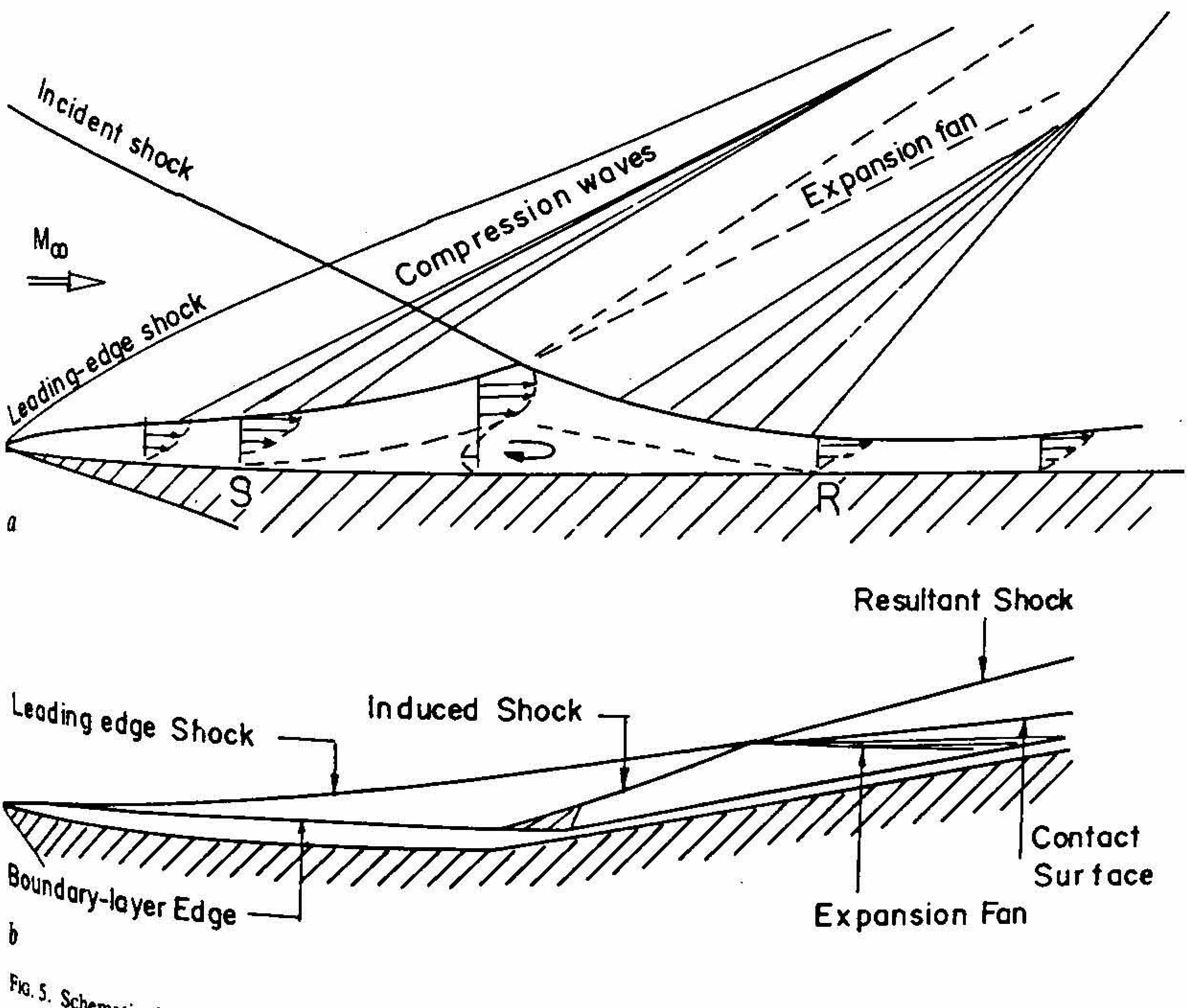
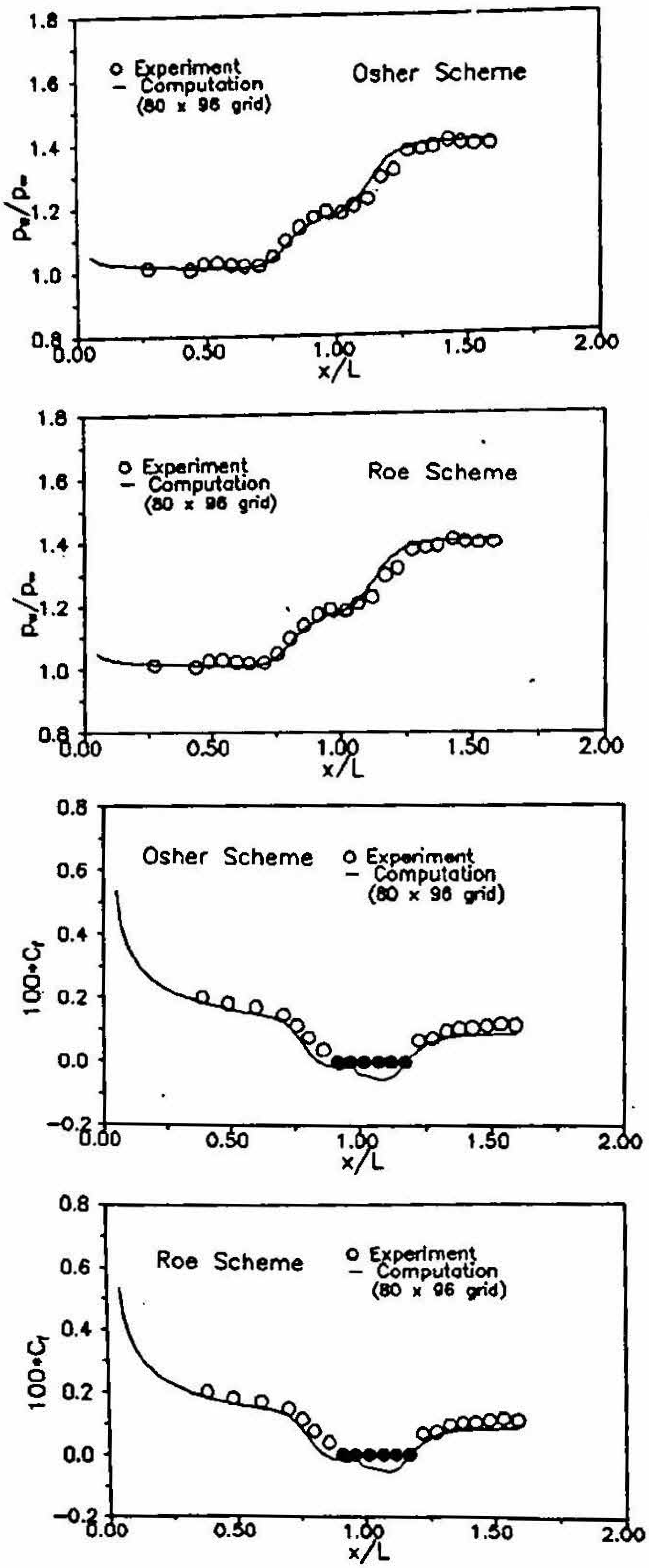


FIG. 5. Schematic diagrams of flow field for shock-wave-boundary-layer interaction test cases.



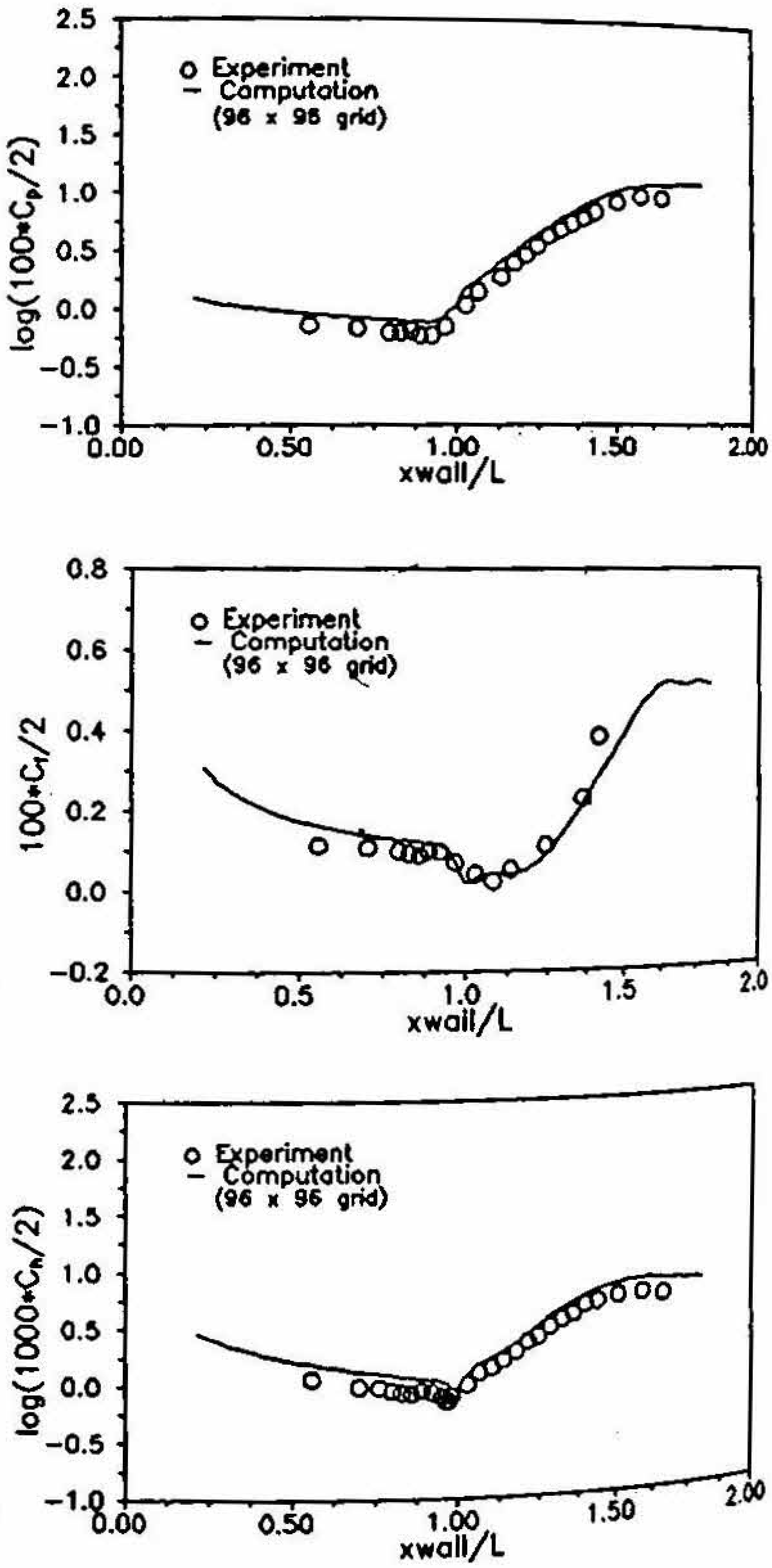


FIG. 6. Comparison of computed surface-pressure distribution and skin-friction coefficient based on Roe and Osher schemes with experimental data.

FIG. 7. Comparison of computed surface-pressure, skin-friction and heat-transfer coefficients based on Roe's scheme with experimental data.

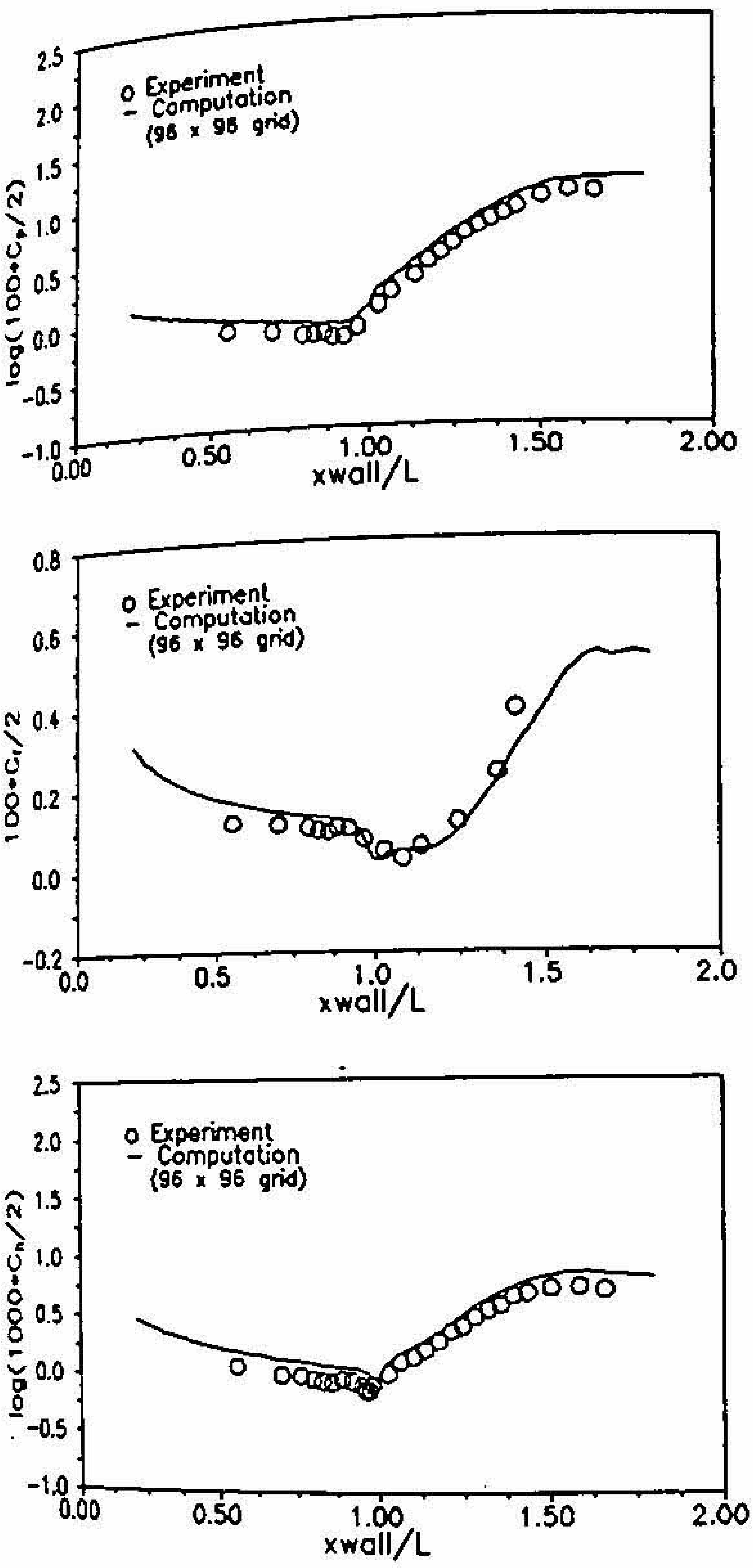


FIG. 8. Comparison of computed surface-pressure, skin-friction and heat-transfer coefficients based on Osher's scheme with experimental data.

in convergence is particularly severe. This has been comprehensively investigated by Amaladas[18].

5. Navier–Stokes computations

A series of Navier–Stokes computations were also carried out for this comparative evaluation based on Roe and Osher schemes. Two test cases involving shock-wave–boundary-layer interactions were chosen and the corresponding schematic diagrams of the flow field are shown in Fig. 5. The first test case is based on the experiments reported by Hakkinen[26], where an oblique shock wave interacts with a laminar boundary layer, causing it to separate and subsequently reattach, thus creating a separation bubble. The Mach number upstream of the oblique shock is 2.0 and the corresponding shock angle is 32.58°. The Reynolds number is 2.96×10^5 based on the shock impingement distance on the flat plate for an inviscid flow. The surface pressure distribution and the skin-friction coefficient obtained from the computed solutions based on Roe and Osher schemes are plotted in Fig. 6 along with the experimental data from Hakkinen[26]. The filled symbols, representing the experimental data for negative values of the skin-friction coefficient in the separated flow region, indicate that their magnitudes could not be measured and have been set to zero for convenience. It is observed from Fig. 6 that Roe and Osher schemes yield results that are nearly indistinguishable from each other and compare quite well with the experimental data for surface-pressure distribution as well as skin-friction coefficient.

The second test case is based on the experiments reported by Holden and Moselle[27] involving hypersonic flow past a corner formed by a flat plate with a ramp of 15° compression angle. The upstream Mach number is 14.1 and the static temperature of 160 R is low enough for real-gas effects to be important. The Reynolds number is 7.2×10^4/ft and the flow remains fully laminar. The wall temperature is fixed at 535 R. The surface-pressure, skin-friction and heat-transfer coefficients obtained from the computed solutions are plotted in Figs 7 and 8 along with the corrected experimental data provided by Dave Rudy of NASA Langley Research Center. The computational and experimental data compare very well with each other. It can be observed that the shock formed by the ramp of 15° compression angle does not quite separate the flow since negative values of skin-friction coefficient are not encountered. However, for 18° compression angle, a small region of separated flow occurs and the corresponding comparisons between computational and experimental data can be found in Amaladas[18].

6. Conclusions

A systematic evaluation of the flux-difference-splitting schemes due to Roe and Osher has been carried out based on Euler and Navier–Stokes computations that were designed to compare and contrast the two schemes in a unified framework. Extensively validated computational and experimental data reported in literature form the basis for comparison of the relative accuracy of the two schemes. Numerical experiments seem to indicate that the lack of dissipation at stagnation conditions for blunt-body flows does not result in catastrophic consequences for Osher's scheme as it does in Roe's scheme. For other flow computations that were carried out, these two schemes yield virtually identical results.

References

1. Roe, P. L. Characteristic-based schemes for the Euler equations, *A. Rev. Fluid Mech.* 1986, **18**, 337–365.

2. Steger, J. L. and Warming, R. F. Flux-vector splitting of the inviscid gas dynamics equations with applications to finite difference methods, *J. Comput. Phys.*, 1981, **40**, 263–293.

3. Van Leer, B. Flux-vector splitting for the Euler equations, *Lecture Notes in Physics*, Vol. 170, 1982, Springer–Verlag.

4. Roe, P. L. Approximate Riemann solvers, parameter vectors, and difference schemes, *J. Comput. Phys.*, 1981, **43**, 357–372.

5. Osher, S. and Solomon, F. Upwind difference schemes for hyperbolic systems of conservation laws, *Math. Comput.*, 1982, **38**, 339–374.

6. ANDERSON, W. K., THOMAS, J. L. AND VAN LEER, B. A comparison of finite volume flux-vector splittings for the Euler equations, *AIAA J.*, 1986, **24**, 1453–1460.

7. VAN LEER, B., THOMAS, J. L., ROE, P. L. AND NEWSOME, R. W. *A comparison of numerical flux formulas for the Euler and Navier–Stokes equations*, AIAA paper 87-1104, 1987.

8. PERTHAME, B. Boltzman type schemes for gasdynamics and the entropy property, *SIAM J. Numer. Anal.*, 1990, **27**, 1405–1421.

9. HARTEN, A., LAX, P. D. AND VAN LEER, B. On upstream differencing and Godunov-type schemes for hyperbolic conservation laws, *SIAM Rev.*, 1983, **25**, 35–61.

10. SMOLLER, J. *Shock waves and reaction–diffusion equations*, 1983, Springer-Verlag.

11. LIOU, M. S. AND STEFFEN, C. J. *A new flux splitting scheme*, NASA TM 104404, 1991.

12. COQUEL, F. AND LIOU, M. S. Stable and low diffusive hybrid upwind splitting methods. In *Computational Fluid Dynamics*,'92, Vol. I, (Ch. Hirsch *et al* (eds)), Elsevier Science Publishers B.V.

13. LIOU, M. S. AND VAN LEER, B. *Choice of implicit and explicit operators for the upwind differencing method*, AIAA Paper 88-0624, 1988.

14. VENKATAKRISHNAN, V Preconditioned conjugate gradient methods for the compressible Navier–Stokes equations, *AIAA J.*, 1991, **29**, 1092–1100.

15. BRANDT, A. Multi-level adaptive solution to boundary value problems, *Math. Comput.*, 1977, **31**, 333–390.

16. BRIGGS, W. L. *A multigrid tutorial*, 1987, SIAM.

17. TURKEL, E. Review of preconditiong methods for fluid dynamics, *Appl. Numer. Math.*, 1993, **12**, 257–284.

18. RICHARD AMALADAS, J. Ph.D. Thesis (in preparation), Department of Aerospace Engineering, Indian Institute of Science, Bangalore, 1995.

19. VAN LEER, B. Towards the ultimate conservative difference scheme V. A second order sequel to Godunov's method, *J. Comput. Phys.*, 1979, **32**, 101–136.

20. VIVIAND, H. Numerical solutions of two-dimensional reference test cases, AGARD AR 211 1985, see also Pulliam, T. H. and Barton, J. T. *Euler computations of AGARD working group 07 Airfoil test cases*, AIAA paper 85-0018, 1985.

21. DERVIEUX, A., VAN LEER, B., PERIAUX, J. AND RIZZI, A. (EDS) *Numerical simulation of compressible Euler flows. Notes on numerical fluid dynamics*, 26, Vieweg, Braunschweig 1989, see also *Proc. GAMM Workshop on the Numerical Simulation of Compressible Euler flows*, INRIA, Rocquencourt, France, 1986.

22. LOCK, R. C. *Test cases for numerical methods in two-dimensional transonic flows*, AGARD Report 575, 1970.

23. PEERY, K. M. AND IMLAY, S. T. *Blunt-body flow simulations*, AIAA Paper 88-2904, 1988.

24. LIN, H. C. *Dissipation additions to flux–difference splitting*, AIAA Paper 91-1544, 1991.

25. QUIRK, J. J. *A contribution to the great Riemann solver debate*, NASA CR 191409, 1992.

26. HAKKINEN, R. J., GREBER, I., TRILLING, L. AND ABARBANEL, S. S. The interaction of an oblique shock wave with a laminar boundary layer, NASA Memo 2-28-59W, 1959.

27. HOLDEN, M. S. AND MOSELLE, J. R. *Theoretical and experimental studies of the shock wave–boundary layer interaction on compression surfaces in hypersonic flow*, 1970, ARL 70-0002, Aerospace Research Laboratories, Wright-Patterson, AFB, Oh.