REVIEW

Modeling turbulent recirculating flows by finite-volume methods—current status and future directions

M. A. Leschziner
University of Manchester Institute of Science and Technology, Manchester, UK

Three-dimensional and recirculating flows of direct industrial relevance and realistic geometry are now well within the scope of CFD capabilities. However, none of the fundamental modeling issues, be it numerical or physical, can be said to have reached a degree of maturity for any particular practice to be perceived or accepted as a standard, and research continues at an unabated level along all fronts. This paper aims to outline the current status of some of the influential issues contributing towards any computational procedure based on the finite-volume approach and intended for modeling complex separated turbulent flow. The main areas covered are: discretization of convection, including bounding schemes, solution algorithms—specifically aspects concerned with maintaining spatial and intervariate coupling, geometric flexibility and turbulence modeling. The survey, however fleeting and superficial, indicates that current developments point towards the use of bounded quadratic approximations, multilevel (or multigrid) acceleration (for steady problems) and second-moment (Reynolds stress/flux) closures within the framework of nonorthogonal grids in which a Cartesian velocity decomposition is adopted.

Keywords: finite-volume methods; recirculating flows; turbulence; discretization; numerical solutions

Introduction

There are few practically relevant flows which are free from recirculation zones or which are not profoundly marked by memory features reflecting the existence of such zones in some upstream region. Recirculation is often very extensive (Figure 1a) encompassing a major proportion of the domain and dictating the operational characteristics of the flow-containing devices; flows in cavities, in plenum chambers, between groynes, in IC-engine cylinders, jet-engine combustors and across heated or cooled rod bundles are but a few examples in which virtually no flow portion can be viewed as being divorced from the separation process. In other cases, recirculation is a more localized, less dominant feature (Figure 1b); examples here include the vortex behind a weak sudden expansion or constriction in a pipe, separation from the suction side of an airfoil at moderate incidence, recirculation in a moderately curved duct or pipe, the wake of a car body, and recirculation behind a low-velocity jet injected into a cross-flowing stream.

However localized separation and recirculation might be, their effect on the overall behavior of the flow will, in general, be disproportionately important, as may be illustrated by the following examples, supplementing Figure 1c:

(1) the separation behind a small rearward-facing step will provoke a significant enhancement in the turbulence level downstream of reattachment and hence strongly elevate the rate of mixing and wall heat-transfer level;
(2) a small separation region on the suction side of an airfoil will strongly reduce its lift;
(3) a modest amount of stall in a compressor cascade will seriously reduce the compressor's performance;
(4) separation in the small recess region between the two elements of a "high-lift" airfoil arrangement will materially alter the structure of the unseparated boundary layer on the suction side of the rear element;
(5) separation in a diffuser will drastically reduce, if not entirely nullify, its pressure-recovery performance;
(6) the recirculation zone provoked by a baffle or swirl in a combustion chamber will dictate the efficiency of the combustion process—indeed, its detailed structure may well define the dividing line between satisfactory performance and catastrophic blowout;
(7) local separation in a curved duct will significantly modify the cross-duct profiles of streamwise velocity at the duct's exit.

The above list should suffice to justify the assertion that any computational approach to describing complex flows which contain regions of separation must yield an accurate representation of the structure of these regions, be they extensive or localized. In some cases, the primary influence of separation on other parts of the flow could be described by capturing merely the location and size of the recirculation zone. However, it is unlikely that a procedure unable to model the structure of the zone adequately would yield its global characteristics. It is partly this requirement which makes the computation of turbulent recirculating flow a particularly challenging task.

Although first numerical calculations of laminar separated flow hark back to the "stone age" of the CFD era,[1–4] meaningful simulations of complex, high-Reynolds-number

The paper is adapted from an invited contribution to the Proceedings of the 3rd Int. Symp. on Refined Flow Modelling and Turbulence Meaurements, held in Tokyo, July 26–28, 1988. The complete volume of proceedings may be obtained from Universal Academy Press, Inc CPO Box 235, Tokyo 100-91, Japan.

Address reprint requests to Dr. Leschziner at the University of Manchester Institute of Science and Technology, Manchester M60 1QD, UK.

Received 22 January 1989; accepted 12 May 1989

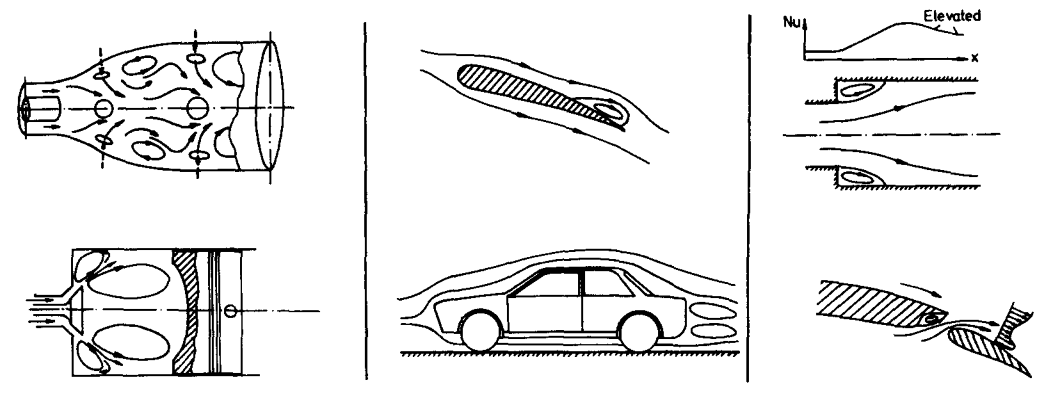


Figure 1 Examples of separated flow: (a) extensive recirculation; (b) localized recirculation; (c) influence of recirculation on remote structure

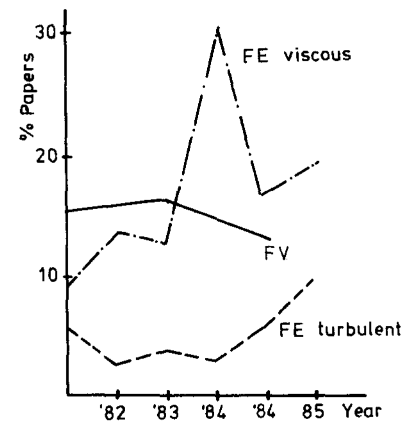


Figure 2 Proportion of conference papers focusing on FE and FV methods[19]

flows emerged only towards the end of the 1960's,[5–8] mainly as a result of computer hardware developments, the emergence of finite-volume approaches incorporating velocity/pressure staggering and stable upwind differencing for convection, and the formulation of practically applicable transport closures of turbulence based on the eddy-viscosity concept.[9,10] The results of these early efforts are still very much in evidence, for even a cursory search in recent journal issues and conference proceedings will readily yield a good crop of papers reporting k-ε-model solutions for complex recirculating and swirling flows generated with the upwind or hybrid schemes[11,12] embedded in codes of the *TEACH*-type[13] or derivatives thereof. Moreover, a number of commercial CFD codes, whilst offering impressive geometric flexibility are—in terms of numerics and turbulence modeling—often little removed from the original 1970–75 techniques. Yet, this traditional route no longer typifies the current status, for the fruits of research over the past two decades directed towards formulating novel approaches as well as towards improving existing methodologies—in terms of numerical accuracy, geometric generality, solution efficiency and turbulence modeling—are being increasingly exploited in the practical environment; it is, in fact, the objective of this account to provide an overview, albeit a sketchy and incomplete one, of some recent developments in these areas.

The emergence of finite-element (FE) algorithms and hybrid finite-element/finite volume (FE/FV) schemes for fluid flow (the latter for unseparated duct flow[14,15]) and their often "painful" extension to include transport models of the k-ε-type[16–23] is perhaps the most noteworthy novel feature in the CFD scene of the 1980's. Hutton, *et al.*[19] have chosen to convey the rate of progress of FE methods by means of Figure 2 which contrasts the number of papers dealing with FV and FE techniques and presented in two series of major biennial CFD conferences in the period 1980–1985. While this figure must be viewed with some caution, in that it ignores the type and complexity of flows tackled, it does nevertheless show a significant increase in the number of successful applications to turbulent flows, following the emergence of techniques, such as streamwise upwinding and Petrov–Galerkin discretization, which overcome the numerical instabilities provoked by the inclusion of k-ε-type models within earlier purely Galerkin schemes.

With the increasing importance of FE approaches having been pointed out, the remaining part of this paper restricts attention to the FV framework which still maintains a clearly dominant position in terms of the level of complexity of the physical processes routinely resolved. Efforts in this area have centered, principally, on nine main topics or issues:

(1) the improvement of accuracy, mainly that associated with the approximation of convection by means of Eulerian and Lagrangian (time–space characteristics) approaches;
(2) and related to (1), the introduction of boundedness into inherently oscillatory (usually higher-order) convection schemes;
(3) a stable implementation of nonstaggered ("collocated") volume arrangements within primitive-variable and velocity/vorticity formulations;
(4) the broadening of geometric flexibility by use of general orthogonal and nonorthogonal, structured meshes and zonal domain-splitting techniques;
(5) grid generation including multi-block schemes and flow-adaptive meshes;
(6) the increase in solution economy by use of coupled solvers, implicit time-marching techniques and multilevel/multigrid schemes;
(7) the improvement of physical realism by use of second-moment closures in conjunction with improved wall laws and low-Reynolds-number near-wall models;
(8) the development of procedures for shock- and geometry-induced separation in transonic and high-Mach-number subsonic conditions, incorporating turbulence-transport models;
(9) the extension and application of compressible-flow solvers (such as those based on the Beam and Warming and the McCormack schemes) to incompressible or nearly incompressible conditions, in conjunction with the artificial compressibility concept.

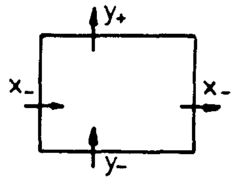


Figure 3 Basic finite volume

Within the constraints of the present coarse-grained overview, only some of the issues appearing in the above list can be addressed, and then only in a rather superficial manner. An inkling of just how superficial the coverage is bound to be, may be conveyed by picking out the topic of grid generation as an example and noting that Thompson *et al.*,[24] when reviewing the subject in 1982, cite no fewer than 340 references; the current number must surely exceed double that number. What is attempted, therefore, in the following sections, always in the context of the finite-volume approach, is to convey a flavor of some recent developments and to speculate on future directions in four areas, namely: the approximation of convection, including bounding; handling of intervariate and spatial coupling, the latter principally by use of multilevel convergence acceleration; geometric flexibility and zonal domain-splitting; and the use of second-moment-closure methodologies.

Approximation of convection

Overview

With attention focused, for simplicity, on the Cartesian finite volume shown in Figure 3, convection of any scalar flow property Φ manifests itself through the integral cell-face fluxes on the right-hand side of the following cell-integrated balance equation

$$\int_{x-}^{x+}\int_{y-}^{y+}[(\rho\Phi)^{n+1}-(\rho\Phi)^{n}]\,\mathrm{d}x\mathrm{d}y$$
$$=-\int_{t^n}^{t^{n+1}}\int_{y-}^{y+}[(\rho U\Phi)_{x_+}-(\rho U\Phi)_{x_-}]\,\mathrm{d}y\mathrm{d}t$$
$$-\int_{t^n}^{t^{n+1}}\int_{x-}^{x+}[(\rho V\Phi)_{y_+}-(\rho V\Phi)_{y_-}]\,\mathrm{d}x\mathrm{d}t$$
$$+\text{Diffusion}+\text{Sources} \qquad (1)$$

If $\bar{\Phi}$ denotes the average value of Φ over the related cell face, Equation (1) simplifies to,

$$\int_{x-}^{x+}\int_{y-}^{y+}[(\rho\Phi)^{n+1}-(\rho\Phi)^{n}]\,\mathrm{d}x\mathrm{d}y$$
$$=-\int_{t^n}^{t^{n+1}}\{[(\rho U\bar{\Phi})_{x_+}-(\rho U\bar{\Phi})_{x_-}]\,\Delta y$$
$$+[(\rho V\bar{\Phi})_{y_+}-(\rho V\bar{\Phi})_{y_-}]\,\Delta x\}\,\mathrm{d}t+\cdots \qquad (2)$$

The task of a convection scheme is to provide approximations for the cell-face values in terms of neighboring nodal values at time levels $n+1$, n, $n-1$, etc. To this end, a system of nodes must be defined and located relative to the system of cells, and possible alternative arrangements are shown in Figure 4. Of these, the cell-centered form, (*a*), is by far the most popular, being extensively used for incompressible recirculating flow and being also applied to turbulent flow in which separation arises as a consequence of shock/boundary-layer interaction[25–27]. The cell-vertex arrangement, (*b*),[28–30] originally devised for inviscid transonic flow, has recently been adapted to shock-induced turbulent separation by Dimitriadis and Leschziner.[31,32] Finally, arrangement (*c*) may be referred to, loosely, as the "upwinded control volume" method, and this has been proposed, as well as applied to multidimensional flows, by Moore and Moore.[33,34]

Although convection is represented by simple first-order derivatives (or flux differences) of the transported flow property, this simplicity is highly deceptive, and the approximation of convective transport remains one of the central, as yet unresolved issues of CFD. The problem is one of reconciling accuracy, stability, boundedness and algebraic simplicity. A somewhat oversimplified, yet essentially valid view of the conflict presenting itself is provided by the observation that stability and boundedness rely on some kind of diffusive "smoothing" mechanism, while accuracy relies on precisely the opposite, namely the absence of numerical smoothing. Numerical diffusion may be an inherent feature of the approximation, being the result of leading even-order truncation errors. The most prominent class of approximations which "exploits" this mechanism to the extent of achieving unconditional boundedness and stability (in a linear sense) is that based on first-order upwinding, applied in any multi-dimensional situation as a superposition of one-dimensional, mesh-line-directed forms, rather than in a streamline-directed manner. This class includes the *Exponential Scheme*,[11] the *Power-Law Differencing Scheme*[12] and the *Hybrid-Differencing Scheme (HDS)*,[11] the last incorporating the pure *Upwind Scheme*. The very mechanism responsible for stability and boundedness in the above approximations also results in second-order artificial cross-flow diffusion, which can be highly damaging to solution accuracy when high shear rates combine with low grid-line density and a significant degree of flow-to-grid skewness.

The role of *grid-line density* is an important one to highlight in the above context, for artificial diffusion can be depressed to an insignificant level, thus becoming an irrelevant issue, provided the internal distances (and with these, the cell-Reynolds number) are made sufficiently small; indeed, in such circumstances, second-order central differencing can be used without fear of instability and significant "wiggles," whether in a transient or steady-state framework. The obvious obstacle to this route is expense, not only in terms of storage but also in terms of solution (CPU) time which tends to rise as N^{α}, where N is the number of nodes and α is of order 2 to 3. When all that is required is the steady-state solution, this obstacle can be lowered, if not removed, by use of the multilevel technique, and this will be the subject of a separate section below. However, when transient features are to be resolved accurately, time-stepping at a spatially invariant interval corresponding to a maximum Courant number not far in excess of unity is unavoidable, in which case the use of highly dense grids is an extremely costly proposition. Here, parallel processing, by means of transputer arrays, for example, seems to offer an increasingly promising relief route, and applications to quite complex flows are beginning to emerge.[35,36]

The multilevel (or multigrid) technique, while holding great promise in store, is still very much in its infancy in the context of recirculating-flow computations, particularly for three-dimensional configurations where storage restrictions prevent the use of the high grid node densities at which CPU savings significantly exceed the overheads associated with

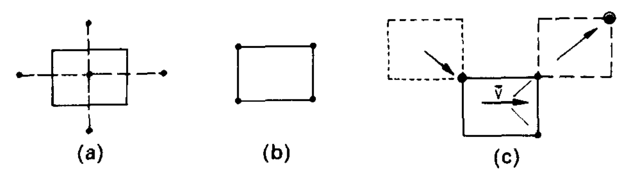


Figure 4 Alternative nodal grids

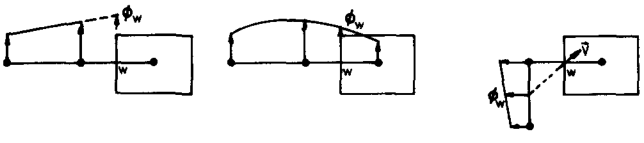


Figure 5 Interpolated value at "west"-face of cell: (a) HOUS; (b) QUICK; (c) SUDS

inter-grid-transfer operations. The principal route thus followed over the past two decades towards achieving an acceptable compromise between accuracy and stability at economically tolerable grid densities has been via a wide variety of (nearly) nondiffusive approximation schemes. The approaches adopted fall, essentially, into one of the following six categories:

(1) higher-order upstream-weighted collocation or skew-upwind schemes for steady-state problems without bounding;
(2) schemes under (1) in conjunction with oscillation-damping/bounding algorithms;
(3) locally analytic schemes for steady-state problems;
(4) higher-order collocation and osculating schemes combined with Lagrangian, time–space characteristics approaches;
(5) higher-order osculating (compact) schemes for steady-state problems;
(6) second-order centered schemes combined with fractional time-step methods, second-order/fourth-order smoothing, flux limitation[37] or "Total Variation Diminishing" (TVD) schemes.[38,39]

The order adopted above is intended to broadly reflect the frequency of use in general recirculatory conditions and, more specifically, in turbulent flows. It should be pointed out, however, that many applications have not been made within the finite-volume context, and these do not, therefore, justify more than a cursory mention in the present account; indeed, schemes in the last two categories will not be considered herein.

Unbounded higher-order and skew schemes

With conventional first-order upstream-weighted formulations left aside, schemes in this group have formed the basis for the majority of the more recent steady recirculating-flow computations reported in the literature. Approximations figuring most prominently are the second-order upwind scheme (HOUS),[40,41] the quadratic upstream-weighted interpolation scheme (QUICK),[42] and the skew-upwind differencing scheme (SUDS).[43] The manner in which the three schemes approximate any cell-face value is indicated graphically in Figure 5.

A large number of studies have been reported, particularly since 1980, in which the characteristics of the above schemes have been systematically investigated and compared to simpler first-order approximations.[44–57] Most studies confine themselves to linear steady convection/diffusion problems and laminar flows, the latter being mainly lid- and thermally-driven cavity flows and, in one case, an impinging-jet flow.[53] Leschziner and Rodi[45] have, however, extended their considerations of QUICK and SUDS to two-dimensional recirculating jet flows computed with the k-ε eddy-viscosity model, while Han *et al.*[46] focus on QUICK's performance in a turbulent cavity flow. A further noteworthy study is that of Demuren,[58] in which QUICK has been compared to the first-order hybrid scheme in the context of computations for 3-D turbulent jets injected into a cross flow.

While no categorically clear-cut verdict can be extracted from the wide variety of cases examined, there is a fair degree of consensus that QUICK gives, overall, the best performance, and it is this scheme which appears to be increasingly used for complex two- and three-dimensional turbulent-flow calculations.[59–82] Some contradictory conclusions emerge from studies of Vanka,[55] Shyy[50] and Castro and Jones[56] on the second-order upwind scheme, with Vanka observing a performance not greatly superior to that of the first-order variant and Shyy and Castro reporting much more favorable performance characteristics. Shyy and Braaten[83] have consequently adopted the second-order scheme in their later three-dimensional turbulent-flow calculation. All schemes are found to yield unbounded solutions with the skew scheme tending to produce the largest oscillations and observed to perform badly in cavity flow.[53] Different versions of the skew scheme have been applied to turbulent flows by Boysan *et al.*,[84] El Tahry[85] and Benodekar *et al.*,[86] the last two adopting bounded versions.

Some final brief comments are appropriate in this section in relation to the computation of transient flow. Calculations for turbulent recirculating flows combining collocation schemes of the type considered here (i.e., other than those of the first-order upwind variety) with statistical models of turbulence are rare (Refs. 85 and 67), both relating to IC–engine cylinder flow. A larger number of applications can be found, however, in situations in which instabilities, in the form of periodic vortex shedding and large turbulent eddies, were to be resolved at moderately high Reynolds numbers with the aid of high-order schemes. Recent calculations of vortex shedding behind square obstacles using QUICK in conjunction with a Lagrangian or an Euler-implicit approximation in time have been reported by Davis and Moore,[87] Davis *et al.*,[88] Durao and Pereira[89] and Franke and Schönung,[90] the last study extending to circular cylinders. Takemoto and Nakamura[91] used a QUICK/Adams–Bashforth scheme to compute three-dimensional transient features in circular bends, resolving Taylor–Görtler-type vortices. Applications of third-order upstream-weighted approximations somewhat akin to QUICK within LES schemes have been reported by Kawamura *et al.*,[92] Kawamura and Kuwahara[93] and Kuwahara and Shirayama,[94] although here a more usual approach is to adopt second-order central differences in conjunction with multi-time-step schemes, such as the Adams–Bashforth method.

In summary, if recent trends are taken to indicate future developments then the above considerations point to an increasing dominance of upstream-weighted collocation schemes of formal order 3 (in uniform mesh). The tendency of these schemes to produce unphysical oscillations at high cell Peclet numbers is generally perceived as being the schemes' most serious limitation and disadvantage. However, a number of alternative approaches to limiting or removing oscillations, with no significant penalty to accuracy, have been formulated recently—a topic to which attention is directed next.

Bounding schemes for collocation approximations

The observation that the collocation schemes considered in the previous section give rise to unphysical oscillations has led to efforts aimed towards constructing composite schemes which achieve an acceptable compromise between accuracy and boundedness. It is of some importance to point out here that the oscillations in question are not simply undesirable from a fundamental or cosmetic point of view, but can also prevent convergence (towards the exact numerical solution) when the schemes are applied to turbulence-transport equations, for these cannot generally tolerate the negative values arising for turbulence energy and dissipation as a consequence of oscillations at the lower edges of high-gradient regions.

Most approaches to bounding are based either on switching from one scheme to another or truncating the range of

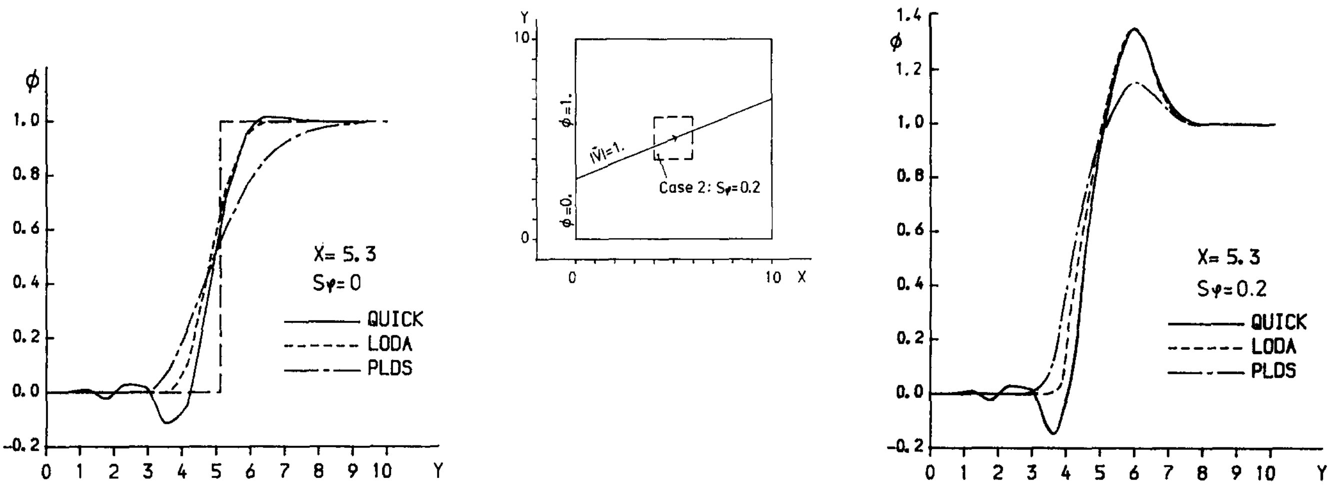


Figure 6 Pure steady convection of scalar discontinuity with and without source; bounded QUICK scheme "LODA"[104]

interpolation, subject to certain criteria, or on a selective blending of the oscillatory scheme with a diffusive first-order scheme.* Other practices, specifically used in supersonic- and transonic-flow, Euler solvers, involve the explicit introduction of second-order and fourth-order diffusion to counteract shock-induced oscillations in the unbounded scheme[97,98] or the construction of special upwind schemes based on the "Total-Variation-Diminishing" concept;[38,39,98] these practices will not be pursued here, therefore.

Composite schemes of the first type have been proposed by Gaskell and Lau,[72] Leonard[99,100] and Hassan *et al.*[101] Leonard's method switches between QUICK, an upwinded exponential scheme (involving a fit of an exponential function to three nodes, one of which being shifted in the upstream direction) and the first-order upwind scheme, depending upon the ratio $(\Phi_U-\Phi_{UU})/(\Phi_D-\Phi_{UU})$, where U, UU and D identify upstream, remote-upstream and downstream nodes, respectively, relative to the cell face being considered. Gaskell and Lau,[72] in contrast, switch between QUICK, the second-order upwind and the first-order upwind approximations. Both composite schemes are shown to perform well, though, at the time of writing, Leonard's scheme appears to have been tested only for a one-dimensional Euler solution and two-dimensional convection of a scalar discontinuity.

Blending schemes have been proposed by Chapman,[102] Lai and Gosman[103] (see also Refs. 85, 86, 104). All schemes essentially mix the unbounded approximation with a proportion of the first-order upwind scheme according to:

$$\Phi_i=\gamma_i\Phi_i^{(P)}+(1-\gamma_i)\Phi_i^{(U)}$$
$$i=\text{cell faces};\ P=\text{"parent"};\ U=\text{"upwind"} \tag{3}$$

The methods vary in the manner in which the weighting factors γ_i are determined. Zhu and Leschziner,[104] for example, combine QUICK and the upwind scheme, requiring positiveness of the primary coefficients linking any one node to its four immediate neighbors. An important overriding constraint is, however, that no mixing is introduced when the solution varies monotonically, regardless of the sign of the primary coefficients. This condition is satisfied by a continual examination of the local solution at all nodes and a comparison of nodal values with their surrounding neighbors. Mixing is implemented only if a local oscillation is detected. That this simple practice is effective is demonstrated in Figure 6, which shows test calculations for scalar convection, with and without a source. A very similar performance is observed in nonlinear conditions, and the method has also been applied successfully to turbulent flows.

The generally favorable behavior displayed by some of the above blending schemes, combined with their simplicity, gives rise to the expectation that turbulent-flow algorithms, particularly those employing advanced multiequation turbulence-closure models within complex three-dimensional domains, are likely to increasingly opt for some bounded version of the QUICK scheme.

Locally analytic methods

The concept of this class of methods rests on the observation that an accurate, unconditionally bounded scheme for a one-dimensional convection/diffusion problem can be obtained by solving analytically the equation,

$$\rho U\frac{d\Phi}{dx}=\Gamma_\Phi\frac{d^2\Phi}{dx^2} \tag{4}$$

between any two nodes, with the (local) boundary conditions being the nodal values. It is this approach which gives rise to the well-known *Exponential Scheme*.[11] The advantages of this locally exact scheme's accuracy are lost in two- and three-dimensional conditions, however, when it is applied to each coordinate direction separately, with the resulting scheme arising as an additive summation of the componential contributions. In such a case, the scheme deteriorates to a level comparable to that of the upwind approximation at high cell Peclet numbers.

A proper generalization of the above approach to multi-dimensional flows is possible, and has been proposed by Stubly *et al.*[105] and Chen and Li.[106] In both cases, an approximation scheme is constructed by performing an analytic integration of the two- or three-dimensional linear (or linearized) transport equation over a small region with local boundary conditions around the area prescribed with the aid of functional fits to nodes lying on the area's (volume's) circumference. In two-dimensional cases, the resulting approximation scheme is a 9-point weighted-average formula with the weighting factors (referred to as "influence coefficients") being somewhat complicated functions of the nodal values entering the local boundary conditions. Recent applications of this method to recirculating

* It is interesting to observe here that some flux-correction schemes[37,95] adopt the reverse path, namely that of first producing a diffusive solution and then introducing a carefully measured "antidiffusive" flux. A somewhat related method by Dukowicz and Ramshaw[96] introduces cross-flow antidiffusion as part of the approximation scheme itself.

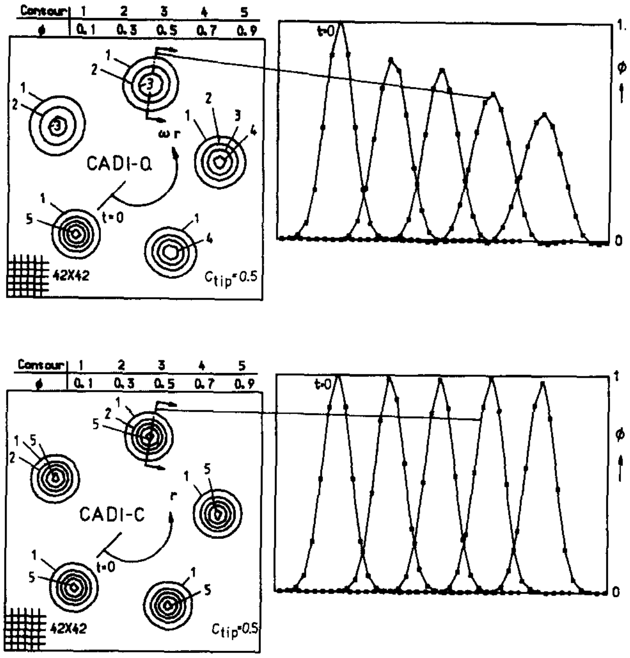


Figure 7 Convective rotation of scalar cone; cubic and quadratic splines with ADI characteristics scheme (CADI-C & CADI-Q respectively)[120]

flow, both laminar and turbulent, have been reported by Chen *et al.*,[107] Chen and Chen,[108] Chen and Yoon,[109] Chen *et al.*,[110] Choi and Chen[111] and Piquet and Visonneau.[112] The accuracy of the scheme is generally impressive, but not uniformly so, as shown by test calculations contributed to a performance comparison conducted by Smith and Hutton.[47] Moreover, the algorithmic complexity of the method has severely impeded its widespread use, and extensions to three-dimensional conditions are very rare.[112]

An interesting approach attempting to combine the simplicity of the one-dimensional analytic solution with the accuracy of the much more complex multi-dimensional generalization has been proposed by Wong and Raithby.[113] As before, the starting point is the two-dimensional linear transport equation, but written here as an iterative two-level approximation,

$$\left\{\rho U\frac{\partial\Phi}{\partial x}-\Gamma_\Phi\frac{\partial^2\Phi}{\partial x^2}\right\}^k=\left\{-\rho V\frac{\partial\Phi}{\partial y}+\Gamma_\Phi\frac{\partial^2\Phi}{\partial y^2}\right\}^{k-1} \qquad (5)$$

This then is a quasi one-dimensional equation with the right-hand side evaluated from the previous iteration. In the next iteration, the x- and y-directed terms are interchanged.

The performance of this scheme has been examined by Prakash[114] for linear cases and by Huang *et al.*[53] for both linear and nonlinear conditions. For linear tests, the scheme was found to yield impressively accurate solutions, but for nonlinear conditions Huang *et al.* encountered serious numerical stability problems preventing full convergence.

Lagrangian techniques

In cases where transient features are to be resolved accurately and economically, higher-order spatial approximations are of little use unless accompanied by accurate temporal formulations. For example, any attempt to combine the QUICK scheme with an Euler-implicit approximation would not be much superior to first-order upwinding in resolving transients, because the second-order temporal truncation error arising from the Euler scheme is, essentially, equivalent to a spatial diffusion term.

Approximate factorization schemes such as ADI offer some advantages here, but schemes based on the use of time–space characteristics are much more attractive from a fundamental point of view, for they mimic transient convection precisely if the spatial variation is represented without error. Schemes of this type have been formulated and applied by Glass and Rodi,[115] Leonard,[116] Davis and Moore,[87] Casulli,[117] Viollet *et al.*,[118] Dewagenaere *et al.*[119] and Nasser and Leschziner.[120] Glass and Rodi's scheme is an explicit, nonconservative, scalar-transport approximation based on an earlier proposal by Holley and Preissmann.[121] Leonard's proposal is essentially a combination of QUICK and the explicit characteristics scheme, while Davis and Moore's method is its two-dimensional extension. The method of Nasser and Leschziner[120] is perhaps the most advanced, in that it employs cubic splines and upstream- weighted osculating polynomials in conjunction with an approximate factorization (ADI-type) scheme incorporating time–space characteristics which are constructed from velocity information on both the forward and backward time levels. This scheme has been applied to scalar convection, steady and transient cavity flows and vortex-shedding phenomena. Its performance is demonstrated for pure scalar convection in Figure 7 which shows the transport of a scalar Gaussian cone by a rotational velocity field. As can be seen, there is virtually no attenuation or spread of the initially prescribed field.

The future prospects of Lagrangian schemes for general flows are uncertain. Their main drawbacks are complexity, unboundedness (arising from the spatial interpolation) and the fact that they presume strong convective dominance—a condition which is often not satisfied in highly turbulent conditions, particularly within slow-moving recirculation zones. It is the writer's view that preference will be given to traditional methods combining bounded higher-order collocation schemes with approximate factorization methods.

Convergence acceleration

Convergence, used here in the sense of the approach towards the exact numerical solution of the discretized equation set, is a meaningful concept principally in the context of steady-state solutions.* Such solutions may be obtained either with time-marching schemes (often with spatially varying time intervals) or with iterative algorithms. In the former group, convergence acceleration is synonymous with an increase in the permitted forward step size, leading to fewer time steps, while in the latter, acceleration means a decrease in the number of iterations.

The rate of convergence is essentially dictated by the degree of intervariate and spatial coupling maintained by the solution algorithm in question. Complete coupling can only be achieved for linear equation sets, in which case the direct solution would lead to the desired result in one sweep. Invariably, the set to be solved is nonlinear, however. Following its linearization (e.g., via a generalized Newton–Raphson approach), full coupling would involve, in a two-dimensional framework, the repeated inversion of an $N \times N$ block matrix with each block being $K \times K$ in size, where K is the number of variables. Such an approach is too costly in practice, and a partially or fully uncoupled, segregated methodology is almost invariably resorted to (one notable exception is the approach of Vanka[122–124]).

Except for a few velocity/vorticity formulations,[125–127] vorticity/vector-potential techniques[128,129] and artificial-com-

* The concept can also apply to transient cases, however, if an implicit (unfactorized) scheme is applied, in which case an iterative solution sequence must strictly be performed within any one time interval. Also, in incompressible flows, such in-step iteration is required to satisfy the mass-continuity constraint.

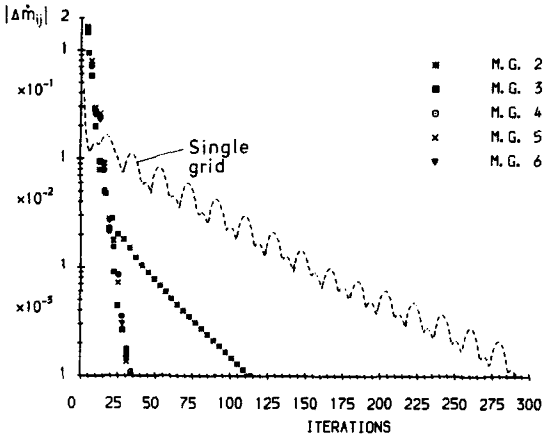


Figure 8 Convergence acceleration, in terms of sum of absolute mass residuals, with multilevel technique; driven cavity, Re = 100, 64 × 64 grid[166]

pressibility approaches,[130–132] the large majority of recent applications involve the use of velocity/pressure or velocity/pressure-correction algorithms (the once popular vorticity/streamfunction method is rarely used now). In some cases—almost invariably laminar—a partially coupled solution is adopted with the momentum and pressure (or continuity) equations solved simultaneously in a point-wise or line-implicit (block ADI-type) manner.[122–124,133–137] An interesting variant in which compact groups of four neighboring nodes are treated implicitly, while intergroup coupling is handled iteratively, is presented by Satofuka.[138]

Most recirculating-flow schemes—and virtually all those applied to turbulent flow—employ uncoupled schemes, however, involving a sequential solution for velocity components and pressure. Such an approach naturally tends to slow down convergence (although an implicit solution is not always superior in terms of resource requirements[133]), and a significant number of pressure-coupling schemes have been devised in an effort to enhance convergence. Prominent examples are SIMPLE,[139] SIMPLER,[140], SIMPLEST,[141] SIMPLEC,[142] PISO,[143,144] SNIP[145] and PUMPIN.[146] Performance comparisons involving subsets of these algorithms have been reported by Latimer and Pollard,[147] Jang *et al.*[148] and Huang,[60] but no clear consensus can be claimed to have emerged on the algorithms' order of efficiency. PISO appears to perform best in laminar conditions, both steady and transient, but experience in turbulent-flow conditions does not, generally, suggest dramatic advantages over the simplest scheme, SIMPLE.

The degree of spatial coupling between nodal values is the second important issue affecting economy of solution. A high degree of coupling can, in principle, be achieved by use of ADI-type schemes, Stone's strongly implicit scheme (or variants thereof)[149,150] and a variety of preconditioned conjugate-gradient methods.[151–153] In practice, particularly when complex systems of five and more coupled, nonlinear, partial-differential equations are to be solved, none of the methods is spectacularly more efficient than others. The application of Stone's method and preconditioned conjugate-gradient schemes specifically to the pressure or pressure-correction equation does tend to pay some useful dividends, but the effectiveness of both approaches declines seriously when these are applied to the Navier–Stokes equations. The conjugate-gradient method suffers from sensitivity to preconditioning which is problem-dependent, while Stone's method is adversely affected by its sensitivity to the values of iteration parameters which must be chosen with little guidance from theoretical considerations.

A technique which is beginning to emerge as a generally powerful and highly economical convergence accelerator in recirculating flow is the multilevel method originated by Brandt.[154]* The method is based on the observation that the short wave-length Fourier components of the solution-error vector, present in any iterative solution of a linear set of algebraic equations, decay much faster than the long waves. Since the shortest waves are two mesh intervals long, this observation suggests that a high rate of error decay would be achieved if the iterative relaxation of the residuals were to be carried out on successively coarser meshes, followed by a reverse transfer towards the finest mesh to achieve the desired accuracy. The method is well established for single-variable systems and has also been fairly extensively used to accelerate time-marching Euler solutions for compressible flow.[29,30] The efficient application of the method, in its nonlinear form as a "full approximation scheme" (FAS), to recirculating flows is still in its infancy. Applications within the staggered, primitive-variable approach have been reported by Sivaloganathan and Shaw,[156,157] Gaskell *et al.*,[158] Fuchs,[159] Phillips and Schmidt,[160] Phillips *et al.*,[161] Miller and Schmidt,[162] Vanka[136] and Thompson and Ferziger,[163] while nonstaggered cell-centered arrangements have been considered by Arakawa *et al.*,[164] Barcus *et al.*[165] and Beeri and Leschziner.[166] In most cases, typical acceleration factors for lid-driven cavity flows at Re = 100 computed with grid sizes of order 64 × 64, is of order 20, as is demonstrated by Figure 8, though this factor decreases with increasing Reynolds number. Very recent, as yet unpublished efforts indicate that the method's effectiveness carries over to highly skewed nonorthogonal, nonstaggered arrangements.[167] Further applications within streamfunction/vorticity schemes have been presented by Ghia *et al.*[168,169] and Schröder and Hänel.[170] The use of point-coupled multilevel schemes, employing block Gauss–Seidel relaxation, appears to yield a further significant convergence acceleration (factor 2 to 3), as demonstrated by Vanka[136] and Arakawa *et al.*[164] The obvious next step up the ladder of implicitness is the implementation of the multilevel method as a block-ADI or block-line-Gauss–Seidel solver, and first efforts in this direction have already been reported by Hutchinson *et al.*[171] and Napolitano.[172]

The crucial question which has yet to be answered with any degree of confidence is whether the dramatic gain in efficiency observed for laminar flows carries over to turbulent conditions. Recent results by Phillips *et al.*[161] are somewhat disappointing, but as yet unreported work by Scheuerer *et al.*[173] indicates that a proper implementation of the Full Approximation Scheme in conjunction with the k-ε model yields substantial savings in resources here too. This conclusion is supported by recent work of Dimitriadis and Leschziner[174] who have applied multigrid acceleration to shock-induced separation in a turbulent flow computed with a cell-vertex scheme and three variants of the k-ε model. As an example, Figure 9 demonstrates the effect of increasing the number of grid levels from one to four on the rate of decline of the average, normalized density residual in a turbulent transonic flow over a bump.

Geometric flexibility

Lack of geometric flexibility and adaptability is frequently claimed by FE protagonists to be a decisive disadvantage of the FV approach. There can be no argument, of course, about the very high level of flexibility offered by the unstructured

* A technique developed by Hutchinson and Raithby[155]—termed the Additive Correction Strategy—is closely related to the multigrid method (see also Hutchinson *et al.*[171]).

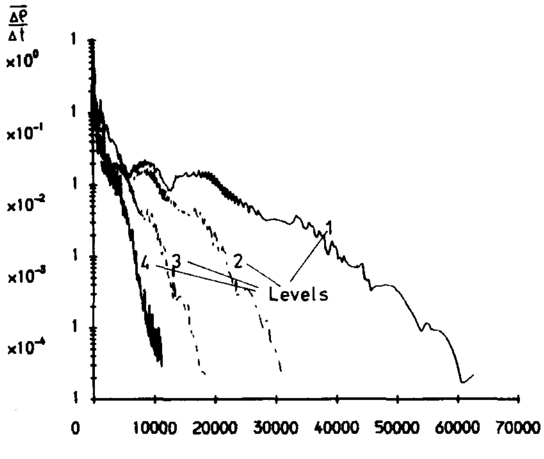


Figure 9 Convergence acceleration, in terms of average density residual, with multilevel scheme; turbulent transonic flow over bump, M = 1.5, 193 × 49 grid[174]

nature of FE discretization, particularly when use is made of higher-order elements. However, great strides have been made over the past decade towards narrowing the gap between FV and FE strategies, and in many complex geometric applications structured FV techniques easily stand their ground when compared to FE schemes in terms of geometric adaptability alone. Moreover, efforts are in progress to construct unstructured FV strategies offering the same level of flexibility as the FE method. Leaving these ongoing, as yet largely unpublished efforts aside, developments have progressed along three main fronts:

(1) algebraic and differential grid generation, including flow-adaptive control;
(2) discretization and solution of transport equations within general-orthogonal and nonorthogonal grids, using both staggered and unstaggered grid systems;
(3) domain-decomposition (multi-block) and coupling techniques.

The subject of grid generation is huge and defies an even superficial overview within this account (for wide-ranging reviews see Refs. 24, 175, 176). This proliferation is perhaps partly rooted in the fact that generating a complex grid involves the application of well-defined mathematical techniques, while solving the flow-governing equations is often a more difficult task, particularly if the set solved includes turbulence–transport equations.

Structured grids, suitable for FV computations, may be generated by algebraic or differential transformation equations. The latter technique is more widespread as it enables the automatic generation of smooth orthogonal or nonorthogonal grids. Such grids arise from solving, numerically, pairs (or triples in 3-D) of Poisson-type equations which govern the variation of the physical (metric) coordinates (x, y) in terms of transformed coordinates (ξ, η) which form a rectangular grid. The nonhomogeneous parts of the equations can be used to exercise a significant level of grid control, in terms of disposition and local density, and the nature of boundary conditions applied (Dirichlet or Neumann) dictates whether the grid is orthogonal or not. The generation process described above becomes difficult in the case of multiply connected domains or when complex "finger-like" multi-zone domains are to be covered, particularly when these are three-dimensional. In such circumstances, block or zonal grids may be generated separately and coupled in an iterative manner while the generation within the blocks is in progress. Examples of this technique in complex two-dimensional cases are presented by Kadja[59] (orthogonal meshes) and Häuser *et al.*,[177] while grids composed of as many as 37 separate blocks in three-dimensional geometries are reported by Thompson.[178]

However sophisticated grid-generation capabilities might be, their usefulness clearly rests on the level of their utilization in fluid-flow solvers. Here again, rapid progress has been made over the past few years. Recent computations of two-dimensional turbulent recirculating flows with curved-orthogonal grids and Reynolds-stress closures have been reported by Kadja,[59] Leschziner *et al.*[179] (using composite, zonal meshes), and Jones and Manners;[180] many more two-dimensional, orthogonal-grid applications making use of eddy-viscosity models are sprinkled in the literature and will not be mentioned here. Calculations with two-dimensional nonorthogonal grids and the k-ε model have been presented by Shyy,[52] Demirdzic *et al.*,[144] Peric,[181] Vu and Shyy,[182] Rodi *et al.*[183] and Agouzoul *et al.*[184]—the last two making use of a nonstaggered cell arrangement. Much progress has also been made to broaden substantially the geometric scope of FV techniques for three-dimensional applications, with general recirculating-flow procedures being formulated by Burns *et al.*,[74] Shyy and Braaten,[83] Takemoto and Nakemura,[91] Dewagenaere *et al.*,[119] Peric,[181] Maliska and Raithby,[185] Kwak *et al.*,[186] Reggio *et al.*,[187] Reggio and Camarero,[188] Demirdzic,[189] Rodi *et al.*,[190] Leschziner and Dimitriadis,[191] Hoholis and Leschziner,[67] and Coupland and Priddin.[192] Some schemes, such as the last two, employ curved-orthogonal grids in two dimensions with the third coordinate being cylindrical-polar. This obviously restricts their geometric capabilities, but the level of physical modeling embedded in them is especially high (in respect of turbulence, mass transfer and cold chemical reaction or combustion); indeed, the scheme of Hoholis and Leschziner has now been extended by Lin and Leschziner[81] to include a second-moment turbulence-transport closure, and has been applied, with QUICK approximating convection, to swirling combustor flows. Most other procedures employ nonorthogonal mesh systems, though in the scheme of Leschziner and Dimitriadis nonorthogonality is only permitted in one plane. Here again, however, this particular scheme possesses a special feature not found elsewhere, namely a domain-decomposition capability, as indicated in Figure 10. The figure provides a simple illustration of the zonal approach (in terms of flow modeling, not simply grid generation) which is set to substantially broaden further the geometric scope of FV-calculation methodologies. The area is still in its infancy, but impressive progress is reported by Glowinski *et al.*[193] (in the FE context), Meakin and Street[194] and Shaw *et al.*[195] The last reference suggests, in fact, that multi-block calculations of flows around entire aircraft using dozens, if not hundreds, of blocks are about to become possible.

Turbulence closure

At present, the large majority of industrial flow computations make use of the eddy–viscosity concept to relate the turbulent stresses and fluxes to the mean flow. In most cases, the eddy viscosity is obtained from the turbulence energy k and its rate of dissipation ε, which are, in turn, extracted from transport equations containing convective, diffusive, generative and dissipative contributions. Such an approach is attractive on numerical grounds: a viscosity formulation—and the second derivatives of the diffused property that go with it—offers the opportunity to construct, within an implicit approach, a "composite" discretization scheme in which diffusion is coupled to con-

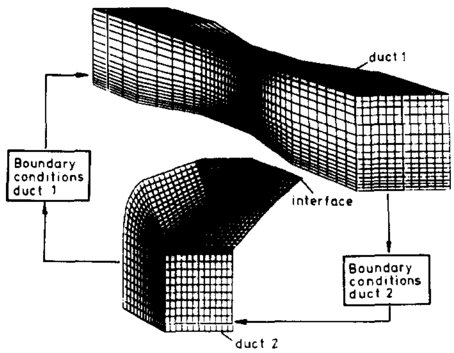
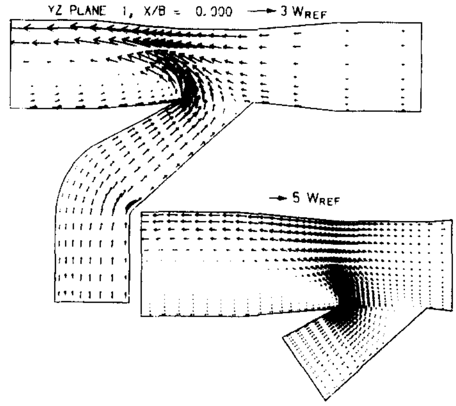


Figure 10 Domain decomposition in 3-D junction flow[191]



vection, thereby strongly promoting stability. Moreover, the equations contributing to the viscosity model are relatively easy to code and solve. Viscosity formulations are far less attractive on physical grounds, however, for they do not perform well in flows in which body forces—arising from strong curvature, recirculation, swirl and buoyancy—play an important role. Such body forces are known to interact selectively with different normal and shear stresses—principally via anisotropy-promoting stress generations and opposing isotropization processes—and this selective, or rather discriminatory influence cannot be captured by use of a model which relates all stresses to the mean field by a single isotropic parameter. The fact that some essential elements of the interaction between curvature and turbulence can only be explained by reference to the individual stress-generation terms[62] leads to the conclusion that any turbulence model expected to yield a high degree of generality must be based on equations describing the processes affecting the balance of each Reynolds stress (and, if appropriate, flux) separately; it is this approach which is steadily gaining momentum and seems set to progressively erode the predominance of simpler eddy–viscosity approaches.

Exact forms of such equations—the Reynolds-stress and flux-transport equations—can be derived by taking velocity-weighted moments of the Navier–Stokes equations and combining these with the Reynolds equations; similar manipulations applied to the scalar-transport equation lead to the flux equations.[196,197] Adopting, for clarity, a simple descriptive representation, one may write the stress and flux equations in the following form:

$$\text{Convection}\,(\overline{u_iu_j}) = \text{Diffusion}\,(\overline{u_iu_j}) + \text{Production}\,(\overline{u_iu_j}) + \text{Pressure-strain}\,(\overline{u_iu_j}) - \text{Dissipation}\,(\overline{u_iu_j})$$

$$\text{Convection}\,(\overline{u_ic}) = \text{Diffusion}\,(\overline{u_ic}) + \text{Production}\,(\overline{u_ic}) + \text{Pressure-scrambling}\,(\overline{u_ic}) - \text{Dissipation}\,(\overline{u_ic})$$

In the above equations, when written in their full mathematically correct form, convection and, most importantly, production need not be modeled, for both only contain mean-flow quantities and the stresses (or fluxes) themselves. The remaining terms however, contain higher-order moments (for example, triple correlations of the form $\overline{u^2v}$ and $\overline{p\,\partial u/\partial x}$) or indeterminable correlations such as the product of strain fluctuations. It is this which necessitates approximations to be postulated if the stress and flux equations are to be closed at second-moment level. Of course, these approximations are certain to introduce errors into the equations, thereby diminishing their capabilities. Yet, the expectation is that the retention of the exact production terms for each stress, coupled with reasonably good modeling proposals for diffusion, pressure-strain and dissipation, would ensure a high level of generality.

Applications of stress models to relatively simple curved and buoyant boundary-layer-type flows[198,199] have, indeed, shown that the models return the correct response to the anisotropy-promoting agents. Evidence which has emerged from studies on much more complex recirculating flows is not always conclusive or consistent, partly because of significant differences in geometry and boundary conditions, and partly as a consequence of different model variants being used. In a number of cases,[200,201] the response of the solution to the turbulence representation has been completely masked by numerical errors provoked by the use of the first-order upwind approximation within a hybrid central/upwind-differencing scheme for convection. These errors are particularly damaging in the context of stress closures which do not naturally yield diffusivity-containing second-order terms enabling the central-differencing part of the hybrid scheme to operate without loss of iterative stability. The absence of such numerically stabilizing terms appears also to have seriously hindered the use of stress closures in combination with accurate, numerically nondiffusive discretization schemes. However, a variety of stability promoting measures[85,202] and the use of time-marching has enabled the recent application of stress closures to a fair range of two-dimensional recirculating flows, including some with very strong swirl, density variations and combustion.[63–65] The use of stress closure in three-dimensional recirculating flows is very much in the initial stages.[80,81]

Recent stress/flux-closure calculations for 2-D flows have been reported by Kadja,[59] Huang,[60] Leschziner,[61,62] Hogg and Leschziner,[63–65] Fu *et al.*[66] Jones and Marquis,[68] McGuirk *et al.*,[69] McGuirk and Papadiminitriou,[70] Prudhomme and Elghobashi,[71] Gaskell and Lau,[72] Weber *et al.*,[76] Kim and Chung,[77] Boysan *et al.*,[84] El Tahry,[85] Leschziner *et al.*,[179] Jones and Manners,[180] Sindir,[203] Fu *et al.*,[204] Yap,[205] Amano,[206] Amano and Goal, [207,208] Sloan *et al.*,[209] and Truelove and Mahmud.[210] First applications of stress closures to shock-induced separation have been presented by Benay *et al.*[25] and Vandromme and HaMinh.[26]

A lengthy discussion would be required at this juncture to point out common features and differences arising from the various studies referred to above, and to identify probable roots for defects. Moreover, any differences would need to be carefully put in relation to the particular closure variant and near-wall treatment adopted, as well as to a host of numerical issues, and this for every group of flows having similar geometric and flow characteristics. Clearly, such an endeavor would go beyond the framework of the present brief survey. A general conclusion emerging, however, is that flows which are dominated by large

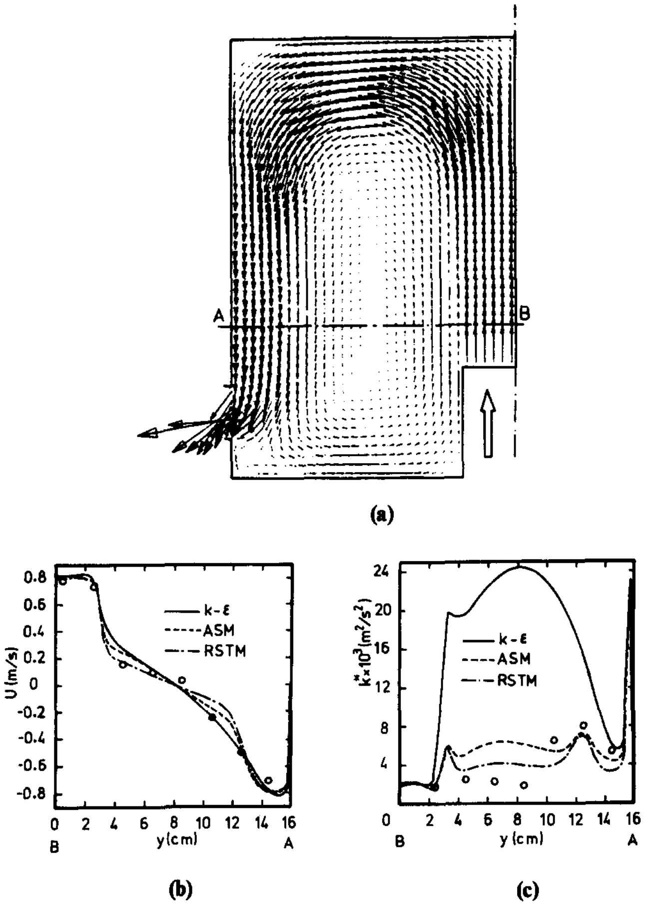


Figure 11 Response of plenum-flow solution to switch from k-ε to Reynolds-stress models:[202] (a) velocity field, RSTM; (b) velocity profile along A-B; (c) profile of $k^* = 0.5(\bar{u}^2 + \bar{v}^2)$ along A-B

recirculation zones and/or are subjected to strong swirl tend to derive the greatest benefits from stress closures.

In order to give a flavor of the possible level of response to a switch from the k-ε to a stress closure, results are shown below for three geometries:

(1) a plenum chamber into which a jet is injected centrally, creating a large recirculation zone which occupies a major proportion of the solution domain;[211]

(2) an expanding annular passage bounded by an axially movable stepped and shaped center-body which is suspended in a diffuser, imparting a severe adverse pressure gradient on the flow;[212]

(3) a strongly swirling annular flow, injected together with a nonswirling central jet into a circular pipe.[213] Comparisons of calculated solutions with experimental data are shown in Figures 11–13 and have been taken from Huang and Leschziner,[202] Leschziner *et al.*,[179] and Hogg and Leschziner,[63] respectively, where RSTM denotes stress-transport closure and ASM identifies an algebraic approximation thereof. All show strong sensitivity to the turbulence model, suggesting a strong interaction between turbulence and streamline curvature which results in a marked attenuation of turbulence transport. The plenum flow is characterized by a large central recirculation zone which is driven by a strongly curved wall jet. The orientation of curvature in the shear layer bordering the central region, relative to the primary shear strain normal to the streamlines is such that turbulence activity is attenuated. As seen from Figure 11, the consequences are a steepening of velocity gradient and a dramatic reduction in turbulence-energy level. A similar process is responsible for the significant enlargement of the recirculation zone in Figure 12, with the predicted reattachment point being moved close to that observed experimentally at $X/H = 10$. The fact that agreement between ASM-computed and measured pressure recovery is good in this case may be taken as an indication that the shape of the recirculation zone is correctly represented. The intense interaction between swirl and turbulence is illustrated in Figure 13 which relates to a strongly-swirling "subcritical" vortex-tube flow. Here, swirl is so intense that the shear-stress field essentially collapses, leading to near-inviscid conditions (although a significant level of turbulence energy persists). The eddy-viscosity model, being unable to account for curvature, returns, in contrast to the stress closure, a solid-body-type swirl-velocity field indicative of an excessive level of diffusive transport.

A final example, shown in Figure 14, is intended to give a qualitative indication of the current status of stress modeling in complex geometries and flow conditions. The geometry is part of a combustor model, examined experimentally by Koutmos,[214] into which a swirling flow is introduced through the left inlet plane. Dilution jets are injected radially, entraining swirling fluid and leading to a rapidly rotating vortex at the center-line whose structure and axial variation strongly affect the axial center-line velocity. Figure 14c contrasts velocity variations obtained by Lin and Leschziner[81] with two approximations of convection, two grids and two variants of stress-transport closure, within a curved-orthogonal finite-volume framework. The variant identified by IPCM is one in which the pressure-strain component in the stress model—that responsible for isotropization by an appropriate redistribution of turbulence energy among the three normal-stress components—is related not only to stress production but also stress convection.[204] The result is a model which, in contrast to established versions, is invariant to coordinate rotation—a property which is of particular importance in swirling flows. This new model was found by Fu *et al.*[204] to be superior to the version denoted by RSTM in strongly swirling two-dimensional flows, and its superiority is also reflected by the three-dimensional application considered in Figure 14.

While stress closures are observed to bring about notable improvements in predictive accuracy, it must be said that differences between calculations and measurements are seldom reduced to insignificant levels, indicating that model defects remain. The models for the isotropizing pressure-strain interaction and the (isotropic) dissipation are known to be major contributors to observed errors. Both model components are subjects of current turbulence-modeling research at UMIST,[204,215,216] and new proposals have shown promise when applied to simple free and near-wall flows; their use in recirculating flow can be expected to follow in the not too distant future.

Concluding remarks

The paper cannot claim to have provided more than a bird's eye view of some of the influential issues contributing to the calculation of complex turbulent flows by the finite-volume technique. Notwithstanding, a number of general trends can be detected which are likely to set the scene of industrially-related CFD for the next few years.

- There is clearly an increased awareness that geometric complexities must be addressed, and this appears to be increasingly done within the framework of structured non-orthogonal grid strategies and a Cartesian (rather than mesh-line-oriented) velocity-vector decomposition. There is

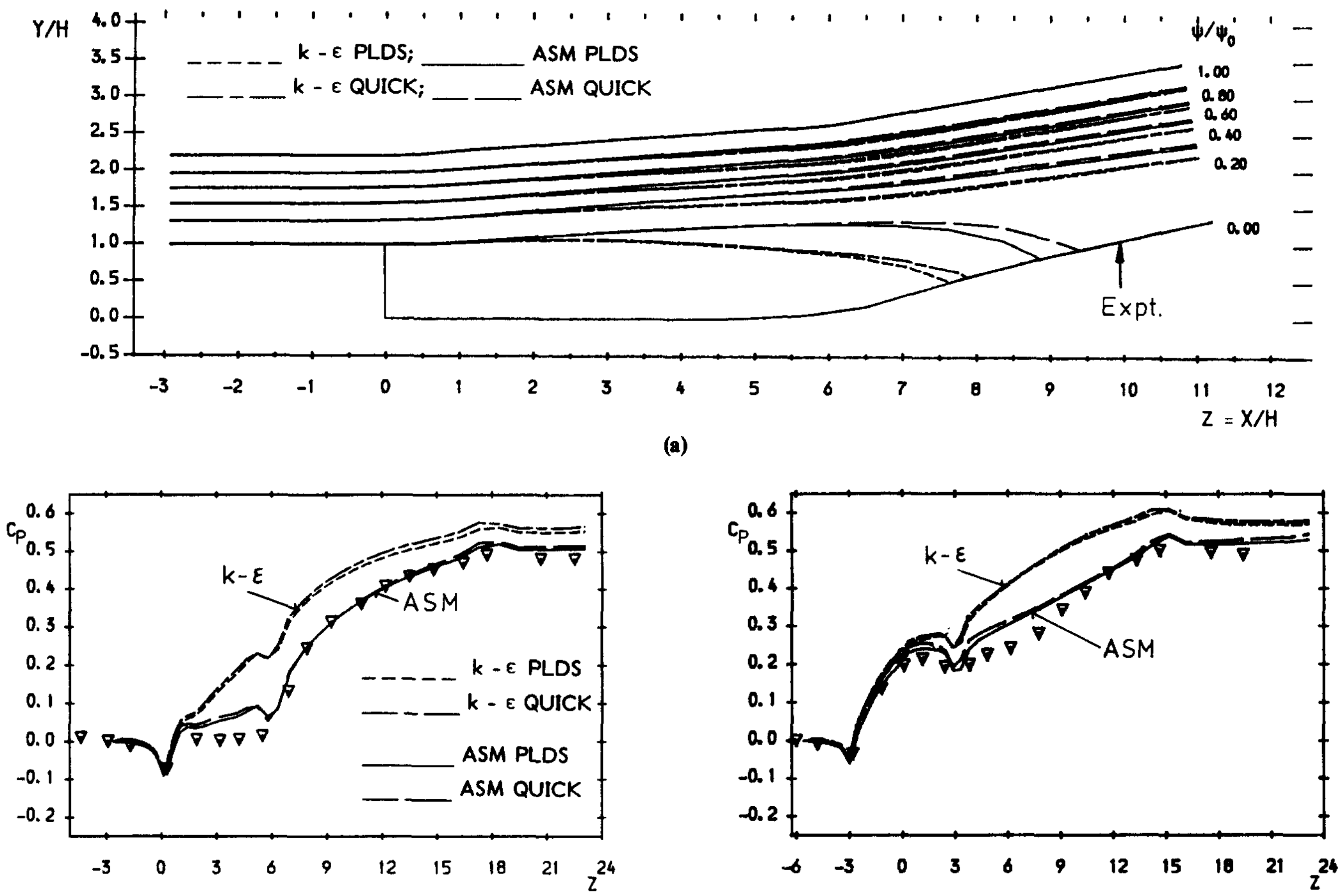


Figure 12 Response of solution of separated annular flow to switch from k-ε to algebraic Reynolds stress model:[179] (a) streamlines; (b) pressure recovery along diffuser wall for two centerbody positions

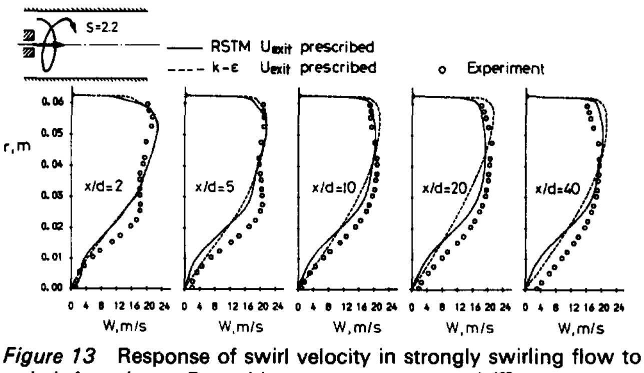
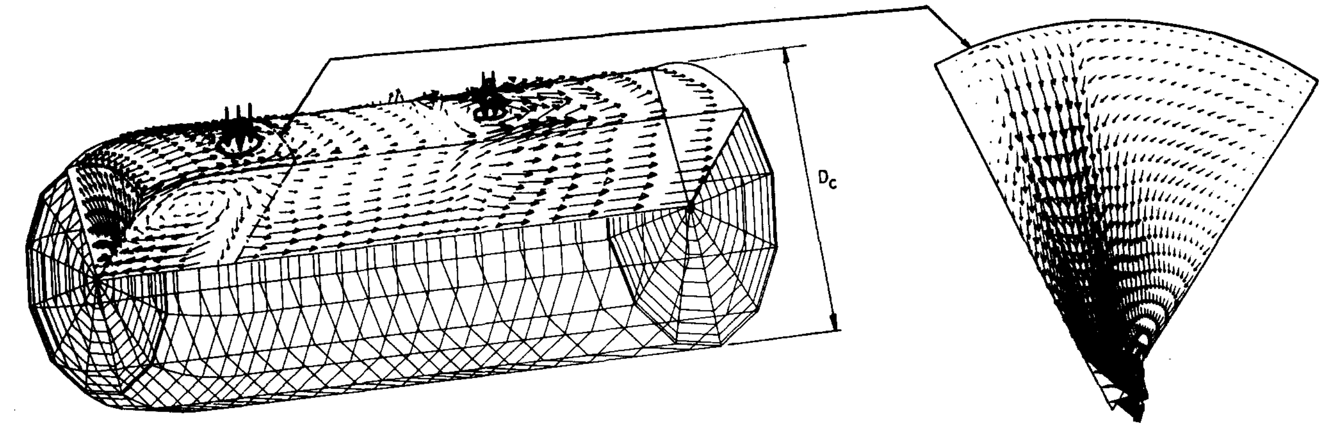


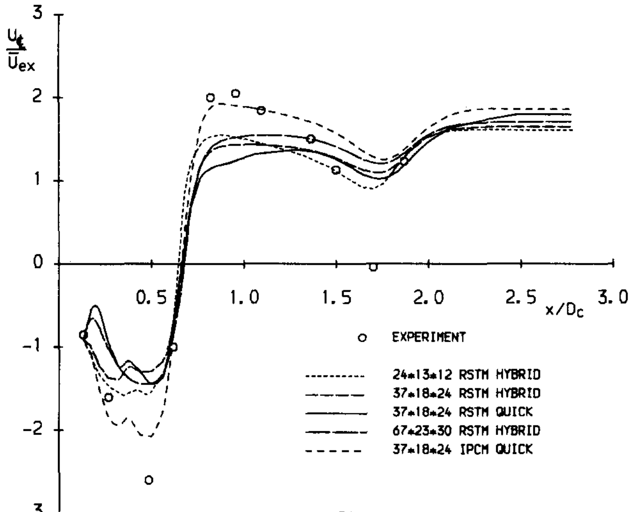
Figure 13 Response of swirl velocity in strongly swirling flow to switch from k-ε to Reynolds-stress-transport model[63]

also a trend towards domain-decomposition techniques for covering very complex domains which cannot be effectively handled by a single structured grid. While emerging unstructured strategies will undoubtedly find their way into industrial applications, it is unlikely that they will oust structured approaches altogether, if only because the latter offer simplicity and ease of solution.

- Discretization of convection in physically complex conditions appears to be increasingly based on QUICK-type schemes, with bounded versions becoming more widespread. This trend is likely to continue.
- Multilevel convergence acceleration looks very promising indeed and is being investigated intensively. Very recent results for turbulent flows are encouraging, and it can be expected that this technique will become a standard constituent of fluid-flow codes, provided skewness and complex turbulence models are not found to decisively erode the method's efficiency. The utility of the technique for three-dimensional flows is, at present, limited by lack of memory resources, hindering the use of dense meshes in all three coordinate directions.
- There is a clear trend towards the use of second-moment (Reynolds-stress) closures, particularly in flows dominated by recirculation zones and/or subjected to swirl. Such models are very complex and resource-intensive, yet none is a panacea. Developments in this area, including such related to the interaction between turbulence, combustion and multiphase mixtures, are likely to be slow and to retard the progress of CFD as a truly predictive technique for industrial applications.
- Parallel computing by means of large arrays of processors may yet turn out to be the joker in the pack. A possible scenario, perhaps around 1995, is that highly efficient parallel-FORTRAN compilers would replace the transputer-oriented language OCCAM, enabling large codes using fine grids to be executed cheaply with little attention needed to be paid to solver efficiency or order of accuracy.



(a)



(b)

Figure 14 Side injection into swirling flow in combustor model:[81] (a) overall view; (b) variation of center-line velocity; sensitivity to grid, convection scheme and turbulence model

References

1 Thom, A. The flow past circular cylinders at low speeds. *Proc. Royal Society of London*, 1933, **A141**, 651–666

2 Shortley, G. H. and Weller, R. The numerical solution of Laplace's equation. *J. Appl. Physics*, 1938, **9**, 334–348

3 Southwell, R. W. *Relaxation Methods in Theoretical Physics.* Oxford University Press, 1946

4 Allen, D. M. and Southwell, R. V. Relaxation methods applied to determine the motion in two dimensions, of a viscous fluid past a fixed cylinder. *Quart. J. of Mech. and Appl. Maths.*, 1955, **8**, 129–145

5 Harlow, F. H. and Welch, J. E. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *Phys. of Fluids*, 1965, **8**, 2182–2189

6 Thoman, D. C. and Szewczyk, A. A. Numerical solution of time-dependent two-dimensional flow of a viscous, incompressible fluid over stationary and rotating cylinders. *Tech. Rep. 66–14*, Heat Transfer and Fluid Mechanics Lab, Dept. of Mech. Engg., University of Notre Dame, Indiana, 1966

7 Gentry, R. A., Martin, R. E. and Daly, B. J. An Eulerian differencing method for unsteady compressible flow problems. *J. Comp. Phys*, 1966, **1**, 93–99

8 Gosman, A. D., Pun, W. M., Runchal, A. K., Spalding, D. B., and Wolfshtein, M. *Heat and Mass Transfer in Recirculating Flows.* Academic Press, London and New York, 1969

9 Jones, W. P. and Launder, B. E. The prediction of laminarization with a two equation model of turbulence. *Int. J. Heat and Mass Transfer*, 1972, **15**, 301–304

10 Launder, B. E. and Spalding, D. B. The numerical computation of turbulent-flow. *Comp. Meths. Appl. Mech. and Engng.*, 1974, **3**, 269–289

11 Spalding, D. B. A novel finite-difference formulation for differential expressions involving both first and second derivatives. *Int. J. for Num. Meths. Engng.*, 1972, **4**, 551–559

12 Patankar, S. V. *Numerical Heat Transfer and Fluid Flow.* Hemisphere, McGraw-Hill, New York, 1980

13 Gosman, A. D. and Ideriah, F. J. K. TEACH-2E: A general computer program for two-dimensional, turbulent, recirculating flows. Internal Report, Dept. of Mech. Eng., Imperial College, London, 1976 (Revised by Arnal, M. P. Report FM-83-2, Dept. of Mech. Engng., University of California, Berkeley, 1983)

14 Baliga, B. R. and Patankar, S. V. A control-volume finite-element method for two-dimensional fluid flow and heat transfer. *Num. Heat Transfer*, 1983, **6**, 245–261

15 Prakash, C. and Patankar, S. V. A control-volume finite-element method for predicting flow and heat transfer—Part 1: Description of the method. *Num. Heat Transfer*, 1987, **12**, 389–412

16 Betts, P. L. and Haroutunian, V. Finite-element calculation of transient dense gas dispersion. *Stably Stratified Flow and Dense Gas Dispersion*, Proc. 1988 IMA Conf., Chester, Series 15 (J. S. Pattock, Ed.) Oxford University Press, 1986, 349–384

17 Betts, P. L. and Haroutunian, V. k-ε modelling of turbulent flow over a backward-facing step by a finite-element method, comparison with finite-volume solutions and experiments. *Proc. 4th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1985, 574–585

18 Hutton, A. G. Current progress in the simulation of turbulent incompressible flow by the finite-element method. *Proc. 4th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1985, 289–305

19 Hutton, A. G., Smith, R. M., and Hickmott, S. The computation of turbulent flows of industrial complexity by the finite-element method—progress and prospects. *Int. J. for Num. Meths. in Fluids*, 1987, **7**, 1277–1298
20 Baker, A. J., Kim, J. W., Freels, J. D., and Orzechowski, J. A. On a finite-element CFD algorithm for compressible, viscous and turbulent aerodynamic flows. *Int. J. for Num. Meths. in Fluids*, 1987, **7**, 1235–1259
21 Goussebaile, J., Jacomy, A., Hauguel, A., and Gregoire, J. P. A finite-element algorithm for turbulent flow processing: a k-ε model. *Proc. 4th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1985, 330–344
22 Gregoire, J. P., Benque, J. P., Lasbleiz, and Goussebaile, J. 3-D Industrial flows calculations by finite element method. *Lecture Notes in Physics*, Vol. 218, Springer, 1985, 245–249.
23 Autret, A., Grandotto, M., and Dekeyser, I. Finite-element computation of a turbulent flow over a two-dimensional backward-facing step. *Int. J. for Num. Meths. in Fluids*, 1987, **7**, 89–102
24 Thompson, J. F., Zahir, U., Warsi, A., Wayne, C., and Mastin, J. Boundary-fitted coordinate systems for numerical solution of partial differential equations—a review. *J. Comp. Phys.*, 1982, **47**, 1–108
25 Benay, R., Coet, M. C., and Delery, J. A study of turbulence modelling in transonic shock-wave/boundary-layer interactions. *Proc. 6th Symp. on Turbulent Shear Flows*, Toulouse, 1987, 8.2.1–8.2.6
26 Vandromme, P. and Ha Minh, H. Physical analysis for turbulent boundary-layer/shock-wave interactions using second-order closure predictions. *Turbulent Shear-Layer/Shock-Wave Interactions*, Proc. IUTAM Symposium, Palaiseau, France, Springer-Verlag, 1985, 127–136
27 Peace, A. J. A Navier-Stokes method for calculation of axisymmetric afterbody/nozzle flowfields. *Internal Report No. 73*, Aircraft Research Association Ltd., 1988
28 Ni, R. H. A multiple grid scheme for solving the Euler equations, *J. AIAA*, 1982, **20**, 1565–1571
29 Hall, M. G. A cell vertex multigrid scheme for solution of the Euler equations. *Num. Meth. for Fluid Dynamics* (K. W. Morton and M. J. Bains, Eds.), Clarendon Press, 1982, 303–345
30 Jameson, A. A vertex-based multigrid algorithm for three-dimensional compressible flow calculations. *Numerical Methods for Compressible Flows—Finite Difference, Element and Volume Techniques* (T. E. Tesduyar and T. J. R. Hughes, Eds.), ASME/AMD, Vol. 78, 1986
31 Dimitriadis, K. P. and Leschziner, M. A. Computation of shock/turbulent-boundary-layer interaction with cell-vertex method and two-equation turbulence model. *Proc. Royal Aeronautical Society Conf. on the Prediction and Exploitation of Separated Flow*, London, 1989, 10.1–10.15
32 Dimitriadis, K. P. and Leschziner, M. A. Cell-vertex algorithm for turbulent transonic flow. *Proc. 6th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1989 (to be published)
33 Moore, J. G. and Moore, J. Calculation of horseshoe vortex flow without numerical mixing. ASME paper, 85-GT-241, 1985
34 Moore, J. G. Calculation of 3-D flow without numerical mixing. *AGARD Lecture Series*, No. 140, 3-D Computation Techniques Applied to internal Flows in Propulsion Systems, 1985, 8.1–8.15
35 Linden, Y., Steckel, B., Stüben, K. Parallel multigrid solution of the Navier-Stokes equations on general two-dimensional domain, Arbeitspapier der GMD 294, Gesellschaft für Mathematik und Datenverarbeitung, St. Augustin, FRG, 1988
36 Clarke, N. R. and Tutty, O. R. Two-dimensional flow simulations using vortex methods implemented on transputer arrays. *Proc. Royal Aeronautical Soc. Conf. on the Prediction and Exploitation of Separated Flow*, London, 1989, 3.1–3.9
37 Boris, I. P. and Book, D. L. Flux-corrected transport I—SHASTA, a fluid transport algorithm that works. *J. Comp. Phys.*, 1973, **11**, 38–69
38 Harten, A., Engquist, B., Osher, S., and Chakravarthy, S. R. Uniformly high order accurate essentially nonoscillatory schemes, III. *J. Comp. Phys.*, 1988, **71**, 231–303
39 Chakravarthy, S. R. and Osher, S. High resolution applications of the Osher upwind scheme for the Euler equations. AIAA Paper 83-1943, 1983
40 Price, H. S., Varga, R. S., and Warren, J. E. Application of oscillation matrices to diffusion-convection equation. *J. Mathematics and Physics.*, 1966, **45**, 301–311
41 Atias, M., Wolfshtein, M., and Israeli, M. Efficiency of Navier-Stokes solvers. *J. AIAA*, 1977, **15**, 263–266
42 Leonard, B. P. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Comp. Meths. Appl. Mech. and Engng.*, 1979, **19**, 59–98
43 Raithby, G. D. Skew upstream differencing schemes for problems involving fluid flow. *Comp. Meths. Appl. Mech. and Engng.*, 1976, **9**, 153–164
44 Leschziner, M. A. Practical evaluation of three finite-difference schemes for the computation of steady-state recirculating flows. *Comp. Meths. in Appl. Mech. and Engng.*, 1980, **23**, 293–312
45 Leschziner, M. A. and Rodi, W. Calculation of annular and twin parallel jets using various discretization and turbulence model variations. *ASME J. Fluid. Engng.*, 1981, **103**, 352–360
46 Han, T., Humphrey, J. A. C., and Launder, B. E. A comparison of hybrid and quadratic-upstream differencing in high-Reynolds-number elliptic flows. *Comp. Meths. in Appl. Mech. and Engng.*, 1981, **29**, 81–95
47 Smith, R. M. and Hutton, A. G. The numerical treatment of advection: A performance comparison of current methods. *Num. Heat Transfer*, 1982, **5**, 439–461
48 Pollard, A. and Siu, A. L. W. The calculation of some laminar flows using various discretisation schemes. *Comp. Meths. in Appl. Mech. and Engng.*, 1982, **35**, 293–313
49 Wilkes, N. S. and Thompson, C. P. An evaluation of higher order upwind differencing for elliptic flow problems. *Proc. 3rd Int. Conf. on Num. Meths. in Laminar and Turbulent Flow*, Seattle, 1983, 248–257
50 Shyy, W. A study of finite-difference approximations to steady-state convection-dominated flow problems. *J. Comp. Phys.*, 1985, **57**, 415–438
51 Shyy, W. A numerical study of annular dump diffuser flows. *Comp. Meths. in Appl. Mech. and Engng.*, 1985, **53**, 47–65
52 Shyy, W. and Correa, S. M. A systematic comparison of several numerical schemes for complex flow calculations. AIAA Paper 85-0440, 1985
53 Huang, P. G., Launder, B. E., and Leschziner, M. A. Discretization of nonlinear convection processes: A broad-range comparison of four schemes. *Comp. Meths. in Appl. Mech. and Engng.*, 1985, **48**, 1–24
54 Patel, M. K. and Markatos, N. C. An evaluation of eight discretization schemes for two-dimensional convection-diffusion equations. *Int. J. for Num. Meths. in Fluids*, 1986, **6**, 129–154
55 Vanka, S. P. Second-order upwind differencing in a recirculating flow. *J. AIAA*, 1987, **25**, 1435–1441
56 Castro, I. P. and Jones, J. M. Studies in numerical computations of recirculating flows. *Int. J. for Num. Meths. in Fluids*, 1987, **7**, 793–823
57 Vu, T. C. and Shyy, W. Navier-Stokes computation of radial inflow turbine distributor. ASME, *J. Fluids Engng.*, 1988, **13**, 29–32
58 Demuren, A. O. False diffusion in three-dimensional flow calculations. *Computers and Fluids*, 1985, **13**(4), 411–419
59 Kadja, M. Computation of recirculating flow in complex geometries with algebraic second-moment-closures. Ph.D. Thesis, Faculty of Technology, University of Manchester, 1987
60 Huang, P. G. The computation of elliptic turbulent flows with second moment closure models. Ph.D. Thesis, Faculty of Technology, University of Manchester, 1986
61 Leschziner, M. A. Finite-volume computation of recirculating flow with Reynolds-stress closures. *Proc. 3rd Int. Conf. on Num. Meths. for Nonlinear Problems*, Dubrovnik, 1986, 847–868
62 Leschziner, M. A. Numerical implementation and performance of Reynolds-stress closure in finite-volume computations of recirculating and strongly swirling flows. *Introduction to the Modelling of Turbulence*, VKI lecture notes. Brussels, 1987
63 Hogg, S. and Leschziner, M. A. Computation of highly swirling confined flow with a Reynolds-stress turbulence model. *J. AIAA*, 1989, **27**, 57–63

64 Hogg, S. and Leschziner, M. A. Second-moment-closure calculation of strongly-swirling confined flow with large density gradients. *Int. J. Heat and Fluid Flow*, 1989, **10**, 16–27
65 Hogg, S. and Leschziner, M. A. Second-moment computation of strongly-swirling reacting flow in a model combustor. *Proc. 3rd Int. Conf. on Computational Combustion*, Antibes, France, 1989 (to be published)
66 Fu, S., Huang, P. G., Launder, B. E., and Leschziner, M. A. A comparison of algebraic and differential second-moment closures with and without swirl. *ASME J. Fluids Engng*, 1988, **110**, 216–221
67 Hoholis, E.G. and Leschziner, M. A. Numerical simulation of jet injection into a quiescent combustion-chamber model. *Proc. 1985 ASME Winter Meeting*, Int. Symp. on Flows in Internal Combustion Engines—III, FED, 1985, **28**, 87–96
68 Jones, W. P. and Marquis, A. J. A method for computing three-dimensional flows using nonorthogonal boundary-fitted coordinates. *Proc. 5th Symp. on Turbulent Shear Flows*, Cornell University, Ithaca, NY, 1985, 20.1–20.6
69 McGuirk, J. J., Papadimitriou, C., and Taylor, A. M. K. P. Reynolds-stress model calculations of two-dimensional plane and axisymmetric recirculating flows. Presented at *5th Symp. on Turbulent Shear Flows*, Cornell University, 1985 (not included in Proceedings)
70 McGuirk, J. J. and Papadimitriou, C. Stably stratified free surface shear layers with internal hydraulic jumps. *Stably Stratified Flow and dense gas dispersion*, Proc. 1986 IMA Conf., Chester (J. S. Pattock, Ed.) Series 15, Oxford University Press, Oxford, 1988, 385–407
71 Prud'homme, M. and Elghobashi, S. Turbulent heat transfer near the reattachment of flow downstream of a sudden pipe expansion. *Num. Heat Transfer*, 1986, **10**, 349–368
72 Gaskell, P. H. and Lau, A. K. C. An assessment of direct stress modelling for elliptic turbulent flows with the aid of a nondiffusive, boundedness-preserving, discretization scheme. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 351–362
73 Durst, F., Founti, M., and Obi, S. Experimental and computational investigation of the two-dimensional channel flow over two fences in tandom. *ASME J. Fluids Engng.*, 1988, **110**, 48–54
74 Burns, A. P., Jones, I. P., Knightley, J. R., and Wilkes, N. S. The implementation of a finite difference method for predicting incompressible flows in complex geometries. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 339–350
75 Boysan, F., Zhou, T., Vasquez-Malebran, and Swithenbank, J. Calculation of turbulent swirling flows with a second order Reynolds-stress-closure. Internal Report, Dept. of Chemical Engng., University of Sheffield, Sheffield, 1985
76 Weber, R., Boysan, F., Swithenbank, J., and Roberts, P. A. Computations of near-field aerodynamics of swirling expanding flows. *Proc. 21st Combustion Symposium*, IFRF Document No. K-20/A/216, The Combustion Institute, 1986
77 Kim, K. Y. and Chung, M. K. Calculation of a strongly swirling turbulent round jet with recirculation by an algebraic stress model. *Int. J. Heat and Fluid Flow*, 1988, **9**, 62–68
78 Iacovides, H., Launder, B. E., and Loizou, P. A. Numerical computation of turbulent flow through a square-sectioned 90° bend. *Int. J. Heat and Fluid Flow*, 1987, **8**, 320–325
79 Azzola, J., Humphrey, J. A. C., Iacovides, H. and Launder, B. E. Developing turbulent flow in a U-bend of circular cross-section: Measurement and computation. *ASME J. Fluid Engng.*, 1986, **108**, 214–221
80 Manners, A. P. The calculation of the flows in gas turbine combustion systems. Ph.D. Thesis, University of London, 1988
81 Lin, C. A. and Leschziner, M. A. Computation of three-dimensional injection into swirling flow with second-moment closure. *Proc. 6th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1989 (to be published)
82 Murakami, S. and Kato, S. Numerical and experimental study on room airflow—3-D predictions using the k-ε turbulence model. *J. Building and Environment*, 1989, **24**, 85–97
83 Shyy, W. and Braaten, M. E. Three-dimensional analysis of the flow in a curved hydraulic turbine draft tube. *Int. J. for Num. Meths. in Fluids*, 1986, **6**, 861–882
84 Boysan, F., Weber, R., Swithenbank, J., and Lawn, C. J. Modelling coal-fired cyclone combustors. *Combustion and Flame*, 1986, **63**, 73–86
85 El Tahry, S. Application of a Reynolds stress model to engine-like flow calculations. *ASME J. Fluids Engng.*, 1985 **107**, 444–450
86 Benodekar, R. W., Goddard, A. J. H., Gosman, A. D., and Issa, R. I. Numerical prediction of turbulent flow over surface-mounted ribs. *J. AIAA*, 1985, **23**, 359–366
87 Davis, R. W. and Moore, E. F. A numerical study of vortex shedding from rectangles. *J. Fluid Mech.*, 1982, **116**, 475–506
88 Davis, R. W., Moore, E. F., and Purtell, L. P. A numerical-experimental study of confined flow around rectangular cylinders. *Phys. Fluids*, 1984, **27**, 46–59
89 Durao, D. F. G. and Pereira, J. C. F. A numerical-experimental study of confined unsteady laminar flow around a square obstacle. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 261–272
90 Franke, R. and Schönung, B. Die numerische Simulation der Laminaren Wirbelablösung an Zylindern mit quadratischen oder kreisförmigen querschnitten. Report SFB 210/T/39, Sonderforschungsbereich 210, University of Karlsruhe, 1988
91 Takemoto, Y. and Nakamura, Y. Solutions of circular-sectioned pipe flows using a three-dimensional generalized quick scheme. *Proc. 1987 Int. Symp. on Computational Fluid Mechanics*, Sydney, Elsevier Publishers, Amsterdam, 1988
92 Kawamura, T., Takami, H., and Kuwahara, K. New higher-order upwind scheme for incompressible Navier-Stokes equations. *Lecture Notes in Physics*, 1984, **218**, 291–295
93 Kawamura, T. and Kuwahara, K. Computation of high-Reynolds-number flow around circular cylinders with surface roughness. Paper AIAA 84-0340, 22nd Aerospace Science Meeting, Reno, Nevada, 1984
94 Kuwahara, K. and Shirayama, S. *Direct and Large Eddy Simulation of Turbulence* (U. Schumann and R. Friedrich, Eds.), Notes on Numerical Fluid Mechanics, Vol. 15, Vieweg Verlag, 1986, 227
95 Huh, K. Y., Golay, M. W. and Manno, V. P. A method for reduction of numerical diffusion in the donor cell treatment of convection. *J. Comp. Phys.*, 1986, **63**, 201–221
96 Dukowicz, J. K. and Ramshaw, J. D. Tensor viscosity method for convection in numerical fluid dynamics. *J. Comp. Phys.*, 1979, **32**, 71–79
97 Jameson, A., Schmidt, W., and Turkel, E. Numerical solution of the Euler equations by finite volume methods using Runge-Kutta time-stepping schemes. AIAA paper 81-1259, 1981
98 Roe, P. L. Characteristic-based schemes for the Euler equations. *Annual Review of Fluid Mechanics*, 1986, **18**, 337–365
99 Leonard, B. P. Locally modified quick scheme for highly convective 2-D and 3-D flows. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 35–47
100 Leonard, B. P. Sharp simulation of discontinuities in highly convective steady flows. NASA Tech. Memorandum 100240, ICOMP-87-9, 1987
101 Hassan, Y. A., Rice, J. G., and Kim, J. H. A stable mass-flow-weighted two-dimensional skew upwind scheme. *Num. Heat Transfer*, 1983, **6**, 395–408
102 Chapman, M. FRAM—Nonlinear damping algorithms for the continuity equation. *J. Comp. Phys.*, 1981, **44**, 84–103
103 Lai, K. Y. M. and Gosman, A. D. Finite-difference and other approximations for the transport and Navier-Stokes equations. Report FS/82/16, Mech. Engng. Dept., Imperial College, London, 1982
104 Zhu, J. and Leschziner, M. A. A local oscillation-damping algorithm for higher-order convection schemes. *Comp. Meths. in Appl. Mech. and Engng.*, 1988, **67**, 355–366
105 Stubley, G. D., Raithby, G. D., Strong, A. B., and Woolner, K. A. Simulation of convection and diffusion processes by standard finite difference schemes and by influence schemes. *Comp. Meths. in Appl. Mech. and Engng.*, 1982, **35**, 153–168
106 Chen, C. J. and Li, P. Finite-differential method in heat conduction—application of analytic solution technique. ASME paper 79-WA/HT-50, 1979
107 Chen, C. J., Nasseri-Neshat, and Ho, K. S. Finite-analytic numerical solution of heat transfer in two-dimensional cavity

flow. *Num. Heat Transfer*, 1981, **4**, 179–197
108 Chen, C. J. and Chen, H. C. Finite analytic numerical method for unsteady two-dimensional Navier-Stokes equations. *J. Comp. Phys.*, 1984, **53**(2), 209–226
109 Chen, C. J. and Yoon, Y. H. Prediction of turbulent heat transfer in flows past a cylindrical cavity. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 1542–1553
110 Chen, J. C., Yu, C. H., and Chadran, K. B. Finite-analytic numerical method and its application to flow modelling. *Proc. 3rd Int. IAHR Symp. on Refined Flow Modelling and Turbulence Measurements*, Tokyo, 1988, I.45–I.54
111 Choi, S. K. and Chen, C. J. Finite analytic numerical solution of turbulent flow past axisymmetric bodies. *Proc. 3rd Int. IAHR Symp. on Refined Flow Modelling and Turbulence Measurements*, Tokyo, 1988, 121–125
112 Piquet, J. and Visonneau, M. Computation of the three-dimensional nominal wake of a shiplike body. *Proc. 3rd Int. IAHR Symp. on Refined Flow Modelling and Turbulence Measurements*, Tokyo, 1988, 249–256
113 Wong, H. H. and Raithby, G. D. Improved finite-difference methods based on a critical evaluation of the approximation errors. *Num. Heat Transfer*, 1979, **2**, 139–163
114 Prakash, C. Application of the locally analytic differencing scheme to some test problems for the convection–diffusion equation. *Num. Heat Transfer*, 1984, **7**, 165–182
115 Glass J., Rodi, W. A higher order numerical scheme for scalar transport. *Comp. Meths. in Appl. Mech. and Engng.*, 1982, **33**, 337–358
116 Leonard, B. P. Adjusting quadratic upstream algorithms for transient incompressible convection. *Proc. 4th AIAA Computational Fluid Dynamics Conference*, Williamsburg, Virginia, 1979, 292–299
117 Casulli, V. Numerical solution of the Navier-Stokes equations at high Reynolds number. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 286–299
118 Viollet, P. L., Benque, J., and Goussbaile, J. Two-dimensional numerical modelling of nonisothermal flows for unsteady thermal-hydraulic analysis. *Nuclear Science and Engng.*, 1983, **84**, 350–372
119 Dewagenaere, P., Esposito, P., Lana, F., and Viollet, P. L. Three-dimensional computations of nonisothermal wall bounded complex flows. *Lecture Notes in Physics*, Springer, 1985, **218**, 191–197
120 Nasser, A. G. F. and Leschziner, M. A. Computation of transient recirculating flow using spline approximation and time–space characteristics. *Proc. 4th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Swansea, 1985, 480–491
121 Holley, F. M. and Preissman, A. Accurate calculation of transport in two dimensions. *ASCE J. Hydraulics Div.*, 1977, **103**, HY 11, 1259–1277
122 Vanka, S. P. and Leaf, G. K. An efficient finite-difference calculation procedure for multi-dimensional fluid flows. *Proc. AIAA/SAE/ASME 20th Joint Propulsion Conf.*, Cininnati, 1984, Paper AIAA-84-1244
123 Vanka, S. P. Block-implicit calculation of steady turbulent recirculating flows. *Int. J. Heat and Mass Transfer*, 1985, **28**(11), 2093–2103
124 Vanka, S. P. Block-implicit coupled calculation of internal fluid flows. *Proc. 5th Symp. on Turbulent Shear Flow*, Cornell University, 1985, 20.27–20.32
125 Toumi, A. and Ta-Phuoc Loc Numerical study of three dimensional viscous incompressible flow by vorticity and velocity formulation. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 595–606
126 Guj, G. and Stella, F. Numerical solutions of high-Re recirculating flows in vorticity–velocity form. *Int. J. for Num. Meths. in Fluids*, 1988, **8**, 405–416
127 Orlandi, P. Vorticity–velocity formulation for high Re flows. *Computers and Fluids*, 1987, **15**, 137–150
128 Lin, A., de Vahl Davis, G., Leonardi, E., Reizes, J. A. Numerical study of the three-dimensional incompressible flow between closed rotating cylinders. *Lecture Notes in Physics*, 1985, **218**, 380–387
129 Wong, A. K. and Reizes, J. A. An effective velocity-vector potential formulation for the numerical solution of three-dimensional duct flow problems. *J. Comp. Phys.*, 1984, **55**, 98–114
130 Valentine, D. T. and Hyde, G. W. Axisymmetric laminar flow over an annular backstep: a numerical study. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 607–618
131 Rizzi, A. and Eriksson, L. E. Computation of inviscid incompressible flow with rotation. *J. Fluid Mech.*, 1985, **153**, 275–372
132 Choi, D. and Merkle, C. L. Application of time-iterative schemes to incompressible flow. *J. AIAA*, 1985, **23**, 1518–1524
133 Zedan, M. and Schneider, G. E. Investigation of simultaneous variable solution for velocity and pressure in incompressible fluid flow problems. *Proc. AIAA 18th Thermophysics Conf.*, Montreal, 1983, Paper AIAA-83-1519
134 Galpin, P. F., Van Doormaal, J. P., and Raithby, G. D. Solution of the incompressible mass and momentum equations by application of a coupled equation line solver. *Int. J. for Num. Meths. in Fluids*, 1985, **5**, 615–625
135 Mansour, M. L. and Hamed, A. Implicit solution of unsteady incompressible Navier-Stokes equations in primitive variables. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 300–311
136 Vanka, S. P. Block-implicit multigrid solution of Navier-Stokes equations in primitive variables. *J. Comp. Phys.*, 1986, **65**, 138–158
137 Wu, M. and Leschziner, M. A. Block-implicit solution of pressure-linked equations. *Proc. 2nd UMIST Colloquium on Computational Fluid Dynamics*, Dept. of Mech. Engng., UMIST, Manchester, 1986
138 Satofuka, N. Group explicit methods for the solution of fluid dynamics equations. *Proc. Int. Conf. on Computational Fluid Dynamics*, Sydney, 1988, Elsevier Publishers, Amsterdam, 117–134
139 Patankar, S. V. and Spalding, D. B. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *Int. J. Heat Mass Transfer*, 1972, **15**, 1787–1806
140 Patankar, S. V. A calculation procedure for two-dimensional elliptic situations. *Num. Heat Transfer*, 1981, **4**, 409–425
141 Spalding, D. B. Mathematical modelling of fluid mechanics, heat transfer and chemical reaction processes: a lecture course. Report HTS/80/1, Dept. Mech. Engng., Imperial College, London, 1980
142 Van Doormaal, J. P. and Raithby, G. D. Enhancement of the simple method for predicting incompressible fluid flows. *Num. Heat Transfer*, 1984, **7**, 147–163
143 Issa, R. Solution of the implicitly discretised fluid flow equations by operator-splitting. *J. Comp. Phys.*, 1986, **62**, 40–65
144 Demirdzic, I., Gosman, A. D., Issa, R. I., and Peric, M. A calculation procedure for turbulent flow in complex geometries. *Computers and Fluids*, 1987, **15**, 251–274
145 Pun, M. M. and Spalding, D. B. A general computer program for two-dimensional elliptic flows. Report HTS/76/2, Dept. Mech. Engng., Imperial College, London, 1976
146 Raithby, G. D. and Schneider, G. E. Numerical solution of problems in incompressible fluid flow: treatment of the velocity–pressure coupling. *Num. Heat Transfer*, 1979, **2**, 417–440
147 Latimer, B. R. and Pollard, A. Comparison of pressure–velocity coupling solution algorithms. *Num. Heat Transfer*, 1985, **8**, 635–652
148 Jang, D. S., Jetli, R., and Acharya, S. Comparison of the PISO and SIMPLEC algorithms for the treatment of the pressure–velocity coupling in steady flow problems. *Num. Heat Transfer*, 1986, **10**, 209–228
149 Stone, H. L. Iterative solution of implicit approximations of multidimensional partial differential equations. *SIAM J. of Num. Analysis*, 1968, **5**, 530–558
150 Schneider, G. E. and Zedan, M. A modified strongly implicit procedure for the numerical solution of field problems. *Num. Heat Transfer*, 1981, **4**, 1–19
151 Kershaw, D. S. The incomplete Cholesky-conjugate gradient method for the iterative solution of systems of linear equations. *J. Comp. Physics*, 1978, **26**, 43–65

152 Gresho, P. M. Time integration and conjugate gradient methods for the incompressible Navier-Stokes equations. *Proc. VI Int. Conf. on Finite Elements in Water Resources*, Lisbon, Portugal, 1986

153 Meijerink, J. A. and Van der Vorst, H. A. Guidelines for the usage of decompositions in solving sets of linear equations as they occur in practical problems. *J. of Comp. Physics*, 1981, **44**, 134–155

154 Brandt, A. A multi-level adaptive solution to boundary-value problems. *Math. of Comp.*, 1977, **31**(138), 330–390

155 Hutchinson, B. R. and Raithby, G. D. A multigrid method based on the additive correction strategy. *Num. Heat Transfer*, 1986, **9**, 511–537

156 Sivaloganathan, S. and Shaw, G. J. An efficient nonlinear multigrid procedure for the incompressible Navier-Stokes equations. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 221–233

157 Sivaloganathan, S. and Shaw, G. J. A multigrid method for recirculating flows. *Int. J. for Num. Meths. in Fluids*, 1988, **8**, 417–440

158 Gaskell, P. H., Lau, A. K. C., and Wright, N. G. Two efficient solution strategies for use with high order discretisation schemes in the simulation of fluid flow problems. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 210–220

159 Fuchs, L. A local mesh-refinement technique for incompressible flows. *Computers and Fluids*, 1986, **14**, 69–81

160 Phillips, R. E. and Schmidt, F. W. A multilevel-multigrid technique for recirculating flows. *Num. Heat Transfer*, 1985, **8**, 573–594

161 Phillips, R. E., Miller, T. F., and Schmidt, F. W. A multilevel-multigrid algorithm for axisymmetric recirculating flows. *Proc. 5th Symp. on Turbulent Shear Flows*, Cornell University, Ithaca, 1985, 20.21–20.25

162 Miller, T. F. and Schmidt, F. W. Evaluation of a multilevel technique applied to the Poisson and Navier-Stokes equations. *Num. Heat Transfer*, 1988, **13**, 1–26

163 Thompson, M. C. and Ferziger, J. H. An adaptive multigrid solution technique for the steady state incompressible Navier-Stokes equations. *Proc. 1987 Int. Conf. on Computational Fluid Dynamics*, Sydney, 1988, Elsevier Publishers, Amsterdam, 715–724

164 Arakawa, C., Demuren, A. Q., Schönung, B., and Rodi, W. Application of multigrid methods for the coupled and decoupled solution of the incompressible Navier-Stokes equations. *Proc. 7th GAMM Conf. on Num. Meths. in Fluid Mechanics*, Louvain-La-Neuve, Belgium, *Notes on Numerical Fluid Mechanics*, 1987, **20**, Vieweg Verlag

165 Barcus, M., Peric, M., and Scheuerer, G. A control-volume based full multigrid procedure for the prediction of two-dimensional laminar incompressible flows. *Proc. 7th GAMM Conf. on Num. Meths. in Fluid Mechanics*, Louvain-La-Neuve, Belgium 1987

166 Beeri, Z. and Leschziner, M. A. Multi-level convergence acceleration in elliptic-flow computations with a cell-centered scheme. *Proc. 3rd UMIST Colloquium on Computational Fluid Dynamics*, Dept. of Mech. Engng., UMIST, Manchester, 1988, 3.4

167 Lien, F. S. *Private Communication*, UMIST, Dept. Mech. Engng., 1989

168 Ghia, U., Ghia, K. N., and Shin, C. T. High-Re solutions for incompressible flow using the Navier-Stokes equations and multigrid method. *J. Comp. Phys.*, 1982, **48**(3), 387–411

169 Ghia, K. N., Ghia, U., and Shin, C. T. Study of fully developed incompressible flow in curved ducts, using a multi-grid technique. *ASME J. Fluids Engng.*, 1987, **109**, 226–236

170 Schröder, W. and Hänel, D. Multigrid solution of the Navier-Stokes equations for the flow in a rapidly rotating cylinder. *Lecture Notes in Physics*, Springer-Verlag, 1985, **218**, 487–491

171 Hutchinson, B. R., Galpin, P. F., and Raithby, G. D. Application of the additive correction multigrid strategy to the coupled fluid-flow equations. *Num. Heat Transfer*, 1988, **13**, 133–147

172 Napolitano, N. Efficient solution of two-dimensional steady separated flows. *Proc. 1987 Int. Conf. on Num. Fluid Dynamics*, Sydney, 1988, Elsevier Publishers, 89–102

173 Scheuerer, G. Private Communication, LSTM, University of Erlangen, 1969

174 Dimitriadis, K. P. and Leschziner, M. A. Multilevel convergence acceleration for viscous and turbulent transonic flows computed with cell-vertex method. *Proc. 4th Copper Mountain Conference on Multigrid Methods*, Copper Mountain, Colorado, 1989, **5**

175 Thompson, J. F. Grid generation techniques in computational fluid dynamics. *J. AIAA*, 1984, **22**, 1505–1523

176 Eiseman, P. R. Grid generation for fluid mechanics computations. *Annual Review of Fluid Mechanics*, 1985, **17**, 487–522

177 Häuser, J., Paap, H. G., Eppel, D., and Sengupta, S. Boundary conformed coordinate systems for selected two-dimensional fluid flow problems. Part II: Application of the BFG method. *Int. J. Num. Meths. in Fluids*, 1986, **6**, 529–540

178 Thompson, J. F. Composite grid generation with the eagle code. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flow*, Montreal, 1987, 1059–1071

179 Leschziner, M. A., Kadja, M., and Lea, C. A combined experimental and computational study of a separated flow in an expanding annular passage. *Proc. 3rd Int. Symp. on Refined Flow Modelling and Turbulence Measurements*, Tokyo, 1988, 83–91

180 Jones, W. P. and Manners, A. The calculation of the flow through a two-dimensional faired diffuser. *Proc. 6th Symp. on Turbulent Shear Flows*, Toulouse, 1988, 17.7.1–17.7.5

181 Peric, M. A finite volume method for the prediction of three-dimensional fluid flow in complex ducts. Ph.D. Thesis, University of London, 1985

182 Vu, T. C. and Shyy, W. Navier-Stokes computation of radial inflow turbine distributor. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flows*, Montreal, 1987, 1601–1612

183 Rodi, W., Majumdar, S., and Schönung, B. Finite-volume method for two-dimensional incompressible flow with complex boundaries. *Proc. 8th Int. Conf. on Computing Methods in Applied Sciences and Engineering*, Versailles, France, 1987, 183–209

184 Agouzoul, M., Reggio, M., and Camarero, R. A numerical study of turbulent flows using a nonstaggered grid. *Proc. Int. Conf. on Computational Fluid Dynamics*, Sydney, 1988, Elsevier Publishers, 193–204

185 Maliska, C. R. and Raithby, G. D. A method for computing three-dimensional flows using nonorthogonal boundary-fitted coordinates. *Int. J. Num. Meths. in Fluids*, 1984, **4**, 519–537

186 Kwak, D., Chang, J. L. C., and Shanks, S. P. A solution procedure for three-dimensional incompressible Navier-Stokes equation and its application. *Lecture Notes in Physics*, 1985, **218**, 346–350

187 Reggio, M., Agouzoul, M., and Camarero, R. Computation of incompressible turbulent flows by an oppose-differencing scheme. *Num. Heat Transfer*, 1987, **12**, 307–320

188 Reggio, M. and Camarero, R. Numerical solution procedure for viscous incompressible flows. *Num. Heat Transfer*, 1986, **10**, 131–146

189 Demirdzic, I. A finite-volume method for the computation of fluid flow in complex geometries. Ph.D. Thesis, University of London, 1982

190 Rodi, W., Majumdar, S., and Schönung, B. Calculation procedure for incompressible three-dimensional flows with complex boundaries. *Finite Approximations in Fluid Mechanics II*, Notes on Numerical Fluid Mechanics, Vieweg Verlag, 1989

191 Leschziner, M. A. and Dimitriadis, K. P. Computation of three-dimensional turbulent flow in nonorthogonal junctions by a branch-coupling method. *Computers and Fluids*, 1989, **17**, 371–396

192 Coupland, C. and Priddin, C. H. Modelling the flow and combustion in a production gas turbine combustor. *Proc. 5th Symp. on Turbulent Shear Flows*, Cornell University, 1985, 10.1–10.6

193 Glowinski, R., Dinh, Q. V., and Periaux, J. Domain decomposition methods for nonlinear problems in fluid dynamics. *Comp. Meths. in Appl. Mech. and Engng.*, 1983, **40**, 27–109

194 Meakin, R. L. and Street, R. L. Domain-splitting methods for geometrically complex flows. *Proc. 5th Int. Conf. on Num. Meths. in Laminar and Turbulent Flow*, Montreal, 1987, 69–80

195 Shaw, J. A., Forsey, C. R., Weatherill, N. P. and Rose, K. E. A block structured mesh generation technique for aerodynamic geometries, *Proc. 1st Int. Conf. on Numerical Grid Generation in Computational Fluid Dynamics*, Hauser, J. and Taylor, C. (Eds.), Pineridge Press, Swansea, 1986

196 Launder, B. E., Reece, G. J., and Rodi, W. Progress in the development of a Reynolds-stress turbulence closure. *J. Fluid Mech.*, 1975, **68**, 537–566

197 Gibson, M. M. and Launder, B. E. Ground effects on the pressure fluctuations in the atmospheric boundary layer. *J. Fluid Mech.*, 1978, **86**, 491–511

198 Rodi, W. and Scheuerer, G. Calculation of curved shear layers with two-equation turbulence models. *Phys. of Fluids*, 1982, **26**, 1422–1436

199 Hossain, M. S. and Rodi, W. A turbulence model for buoyant flow and its application to vertical buoyant jets. *Turbulent Buoyant Jets and Plumes* (W. Rodi, Ed.), HMT Series, Pergamon Press, Elmsford, 1982

200 Pope, S. B. and Whitelaw, J. H. The calculation of near-wake flows. *J. Fluid Mech.*, 1976, **73**, 9–32

201 Naot, D. Iterative procedure for simultaneous calculation of stress transport turbulence model in two-dimensional elliptic fields. *Computational Techniques in Transient and Turbulent Flow* (C. Taylor and K. Morgan, Eds.), 1982, 192–221

202 Huang, P. G. and Leschziner, M. A. Stabilization of recirculating flow computations performed with second-moment closure and third-order discretization. *Proc. 5th Symp. on Turbulent Shear Flows*, Cornell University, Ithaca, 1985, 20.7–21.2

203 Sindir, M. M. S. Numerical study of turbulent flows in backward-facing step geometries. Ph.D. Thesis, Mech. Engng. Dept., University of California, Davis, 1982

204 Fu, S., Launder, B. E., and Leschziner, M. A. Modelling strongly swirling recirculating jet flow with Reynolds-stress transport closure. *Proc. 6th Symp. on Turbulent Shear Flows*, Toulouse, 1987, 17.6.1–17.6.6

205 Yap, C. R. Turbulent heat and momentum transfer in recirculating and impinging flows. Ph.D. Thesis, Faculty of Technology, University of Manchester, Manchester, 1987

206 Amano, R. S. Comparison of pressure–strain correlation models for the flow behind a disk. *J. AIAA*, 1986, **24**, 1870–1872

207 Amano, R. S. and Goal, P. A numerical study of a separating and reattaching flow by using Reynolds-stress turbulence closure. *Num. Heat Transfer*, 1984, **7**, 343—357

208 Amano, R. A. and Goal, P. Investigation of third-order closure model of turbulence for the computation of incompressible flows in a channel with a backward-facing step. *ASME J. Fluids Engng.*, 1987, **109**, 424–428

209 Sloan, D. G., Smith, P. J., and Smoot, L. D. Modelling of swirl in turbulent flow systems. *Prog. Energy and Comb. Sci.*, 1986, **12**, 163–250

210 Truelove, J. S. and Mahmud, T. Calculation of strongly swirling jet flow. *Proc. 9th Australasian Fluid Mechanics Conf.*, University of Auckland, Auckland, 1986, 429

211 Boyle, D. R. and Golay, M. W. Measurement of a recirculating two-dimensional flow and comparison to turbulence model prediction I: Steady-state case. *ASME J. Fluids Engng.*, 1983, **105**, 446–454

212 Lea, C. J. Hot-wire measurements of turbulent flow over a sudden centrebody contraction placed in a diffuser. M.Sc. Thesis, Faculty of Technology, University of Manchester, Manchester, 1987

213 So, R. M. C., Ahmed, S. A., and Mongia, H. C. An experimental investigation of gas jets in confined swirling air flow. NASA, CR-3832, 1984

214 Koutmos, P. An isothermal study of gas turbine flows. Ph.D. Thesis, University of London, London, 1985

215 Fu, S., Launder, B. E., and Tselepidakis, D. P. Accommodating the effects of high strain rates in modelling the pressure-strain correlations. *Report TFD/87/5*, Dept. of Mech. Engng., UMIST, Manchester, 1987

216 *Proc. 3rd UMIST Colloquium on Computational Fluid Dynamics*, Dept. of Mech. Engng., UMIST, Manchester, 1988