Modelling of turbulent flow in gas-turbine blading: achievements and prospects

B. E. Launder*

Some of the problems associated with applying currently available viscous flow calculation schemes to turbulent flow in gas-turbine blading and passages are reviewed. These flows pose severe difficulties in both numerics and turbulence modelling, although the main emphasis here is on the latter aspect. Since complex strain fields and strong body forces are an intrinsic part of flow in turbomachinery, it is preferable that the turbulence modelling of these flows be based on an approximation of the Reynolds stress transport equations themselves. Some current views on closure approximations for these equations are discussed. Applications considered include the effects of free stream turbulence and streamline curvature, the mixing of blade wakes, and the three-dimensional flows that arise in a 90° bend and in the corner boundary layer near a blade root

Keywords: *turbines, fluid mechanics, turbulent flow*

The title of this paper is one of those cosy academic lies that are unquestioningly accepted as 'true enoughs' in the seminar rooms of our technological universities. This paper aims to show the extent to which flow computation schemes based on discretised forms of the averaged equations of motion succeed in mimicking the behaviour of certain *laboratory created* turbulent flows which, in at least some respects, resemble those found in gas-turbine blading and passages.

By comparison, say, with the shear flow developing over an aerofoil in an external flow, flows in gas turbines present many difficulties. Strain fields are more complex, especially those caused by injection through coolant passages in the combustion chamber and turbine blades and by the three dimensional shearing over the compressor and turbine blades. Because of the relatively small dimensions, chord Reynolds numbers are fairly low and 'inviscid' regions are either absent or strongly interacting with shear-flow regions. The large density variations in the combustion chamber between burned and unburnt gases produce problems in flow mixing that have only recently received serious consideration. Additional, substantial density variations well into the turbine section arise from blade cooling. This only adds to the problems of very severe favourable pressure gradient, streamline curvature and high external-stream turbulence level all of which have a marked effect on the boundary layer development over the blades. Finally, while nearly all turbulent flow aspects of turbine aerodynamics are currently treated as stationary phenomena, certain features, such as the successive impact of wakes from one blade row on the adjacent downstream row or the problem of rotating stall, on which Alan Stenning himself did some of his best work[75–78], are intrinsically periodic and need to be analysed as such.

When we speak of 'modelling' a turbulent flow, whether in a gas turbine or elsewhere, it is well to remind ourselves that the model will tend to reproduce only as much detail as is required for, or encompassed within, the originator's purpose. In the case of turbulent flow we do have, at one extreme, a highly detailed and accurate model available, the Navier–Stokes equations for a three-dimensional, time-dependent laminar motion. Turbulence is nothing but an extemely complex manifestation of such a flow. Although several groups, perhaps most notably that of Orzag at MIT, have developed numerical methods of solving the Navier–Stokes equations especially designed for the direct simulation of turbulence, even the proponents of these approaches see them as principally throwing light on the physics of turbulence rather than a vehicle for computing shear flows of practical interest. The problem is essentially one of scales: the smallest eddies present in a turbulent flow have a typical Reynolds number (based on their dimensions and associated fluctuating velocity) around unity, for it is only in such motions that substantial turbulent kinetic energy can be converted to heat by viscous action. The largest scale eddies, however, have a Reynolds number of order $k^{1/2}\delta/\nu$ (and may range from a few hundred to tens of thousands) where k is the turbulence energy and δ is the flow width. To resolve, for an inhomogeneous shear flow, both the fine and the large scales of motion by numerical simulation with

* Department of Mechanical Engineering, University of Manchester, Institute of Science and Technology, PO Box 88, Manchester, UK, M60 1QD.
The paper is based on the 3rd Alan Stenning Memorial Lecture delivered at the University of Glasgow, 28th January 1982.
Accepted for publication on 16th September 1982.

time steps sufficiently small to resolve the fine scale fluctuations seems to be a task that, for the foreseeable future, will not be routinely possible.

At the practical level, interest in turbulence resides in the larger scales of motion for they not only contain most of the Reynolds stress but also, through large-scale, vortex-stretching processes, appear to control the rate at which fluctuating energy is broken down into successively smaller eddies. Thus, indirectly, the large scale motions control even the viscous dissipation rates in the fine scale. It is this fact of turbulence life that has led to a great deal of interest in what have become known as 'large eddy simulations' (les). Although for an les a 3-dimensional time-dependent solving scheme is still employed, no attempt is made to resolve the finest scales of motion. Instead the effects of eddies smaller than the mesh are accounted for by a 'sub-grid-scale' turbulence model. Most simulations have adopted the Smagorinsky model in which the effective sub-grid-scale viscosity is given by a formula analogous to that of the Prandtl–Taylor mixing length hypothesis, save that the internode distance takes the role of the mixing length. A number of interesting large-eddy-simulations have been generated over the last five years, particularly by the group of Schumann and Grötzbach in Karlsruhe[1-3] and the team led by Reynolds and Ferziger at Stanford[4-6]. From the latter group Kim and Moin[6] have made a most impressive simulation of low Reynolds number flow in a plane channel. Using 500 000 grid nodes, they have successfully reproduced not only the mean and Reynolds stress profiles with good accuracy but also such qualitative features of the flow as the streakiness of the viscous sublayer region.

The detail is impressive, but it must be said that this computation[6] required about 40 hours on the Illiac IV computer. Although the development of better sub-grid-scale models would allow a reduction of the mesh density with no untoward deterioriation in accuracy, the sheer magnitude of an les computation suggests that such approaches will have little direct effect in the 1980's on the computation of flow in turbomachinery. It is for this reason that attention here is limited to approaches in which averaging is applied to the equations over a sufficient time period or over a sufficient number of realisations to remove all the non-periodic fluctuations. In place of instantaneous velocities, the subjects of the momentum equations become mean velocity components which exhibit, by comparison with the instantaneous quantities, only a gradual variation in space and time.

The essential problem of turbulence is the nonlinearity of the process of convective transport: the mean product of the mass flux and velocity vectors is not equal to the product of their means. In symbols, with upper and lower case u's denoting mean and fluctuating velocities, tildes and overbars representing instantaneous and averaged quantities, the problem for a uniform density fluid is expressed:

$$\rho\overline{\tilde{U}_j\tilde{U}_i} = \rho U_j U_i + \rho\overline{u_i u_j}$$

where $\tilde{U}_j \equiv U_j + u_j$, etc. The averaged products of fluctuating velocities $\rho\overline{u_i u_j}$ (the so-called Reynolds stresses) appear as unknowns in the averaged equations and the task of the turbulence model is that of providing a path for their determination.

A later section gives a short review of the most popular or promising approaches to modelling tur-

Notation

c_1, c_2, c_μ	Coefficients in turbulence model	v	Fluctuating velocity in y direction
c_f	Skin friction coefficient	x	Cartesian coordinate (non-tensor form) in flow direction
c_{f_0}	Skin friction coefficient with non turbulent external stream	x_j	Cartesian coordinate (usual notation: x_1—stream direction; x_2—direction normal to wall)
D_{ij}	Diffusive transport of stress	y	Cross stream coordinate
l_m	Mixing length	z	Spanwise coordinate
L^u	Length scale associated with streamwise fluctuations	δ	A typical shear flow width
k	Turbulent kinetic energy	δ_{995}	Boundary layer thickness (denoted by position where mean streamwise velocity is 99.5% of free-stream value)
r	Radius	δ_{ij}	Kronecker delta (equal to 1 for $i=j$, equal to zero for $i \neq j$)
R	Radius of curvature of bend	ε	Dissipation rate of turbulent kinetic energy
P_{ij}	Rate of stress generation by mean strain	ε_{ij}	Dissipation rate of Reynolds stress by viscous action
t	Fluctuating temperature	ϕ_{ij}	Pressure containing correlations in Eq (3) (suffices 1 and 2 denote 'turbulence' and 'mean-strain' parts of the process)
T	Mean temperature	ν	Kinematic viscosity
U, u	(without subscript) Mean and fluctuating velocity respectively in x direction	ρ	Density of fluid
$\tilde{U}_j$	Instantaneous velocity in direction x_j	μ_T	Turbulent viscosity
U_j	Mean velocity in direction x_j	η	Film cooling effectiveness
$\bar{U}$	Bulk mean streamwise velocity over duct cross section		
u_j	Fluctuating (turbulent) velocity in direction x_j		
$\overline{u_i u_j}$	Reynolds stress		
$\overline{u_i t}$	Turbulent 'heat' flux (strictly an enthalpy flux per unit specific heat)		

bulence in turbine-related shear flows. Before embarking on this, however, a few comments are offered on progress over the last two decades, in developing numerical procedures for handling the flow equations into which the turbulence model must fit, the vehicle for the engine so to speak. The selection is weighted heavily towards generally conceived treatments, for only these approaches have much potential for application to industrial problems. There is also an acknowledged bias towards the numerical solution methods evolved at Imperial College in the 60's and 70's by Professor Spalding and co-workers which provided a framework for much of the writer's own research in that period. The period of twenty years is chosen partly because it is a round number which is appropriately divisible by two and partly because 1962 can be said to have marked the start of the computer era. It was then that reliable, production-line digital computers such as the IBM 7090 started to appear in universities. Their coming had an immediate effect on methods being developed for turbulent boundary layers though initially this was evolutionary in form. The integral methods in use in 1961 initially gave way to more refined methods of the same type; for example the 1964 papers of Moses[7] and Spalding[8] were based on a more elaborate velocity profile family, the wall-plus-wake description popularised by Coles[9]. By the mid 1960's, however, it was realised by some that a numerical discretisation of the boundary layer equations offered advantages of flexibility and stability over multi-parameter integral approaches.

Nevertheless, in the procedure of Patankar and Spalding[10], which must be judged as the outstandingly successful boundary layer solving scheme of the period, the authors' connection with integral methods was clear to see. The set of difference equations was obtained not by the conventional approach of decomposing differentials into algebraic approximations but by integrating the equations of motion over a control volume surrounding the node. This methodology has become increasingly widespread and schemes formulated in this way nowadays are sometimes referred to as 'finite volume' methods. At the 1968 Stanford Conference on (2-dimensional) Turbulent Boundary Layers[11] 9 finite difference treatments and 20 integral methods tackled the various test cases. Four years later at the NASA Langley Conference on Free Shear Flows[12] that distribution had become 13:1, with finite difference schemes firmly in the ascendency.

Soon after the arrival of its boundary layer procedure, Spalding's research group produced[13–15] a computational procedure for the analysis of steady two-dimensional recirculating flows. Perhaps at this point, to avoid any impression that numerical fluid mechanics only began life with the digital computer, the pioneering work of Professor Thom and his successors should be mentioned[16–18]. With the assistance only of desk calculators, numerical solutions to the flow about cylinders for Reynolds numbers up to about 50 had been obtained from finite-difference discretisations of the stream function and vorticity equations. Attempts to extend the range of Reynolds numbers had, however, been thwarted by the failure to secure convergence, a problem that turned out to be due to a central-difference approximation for convective transport. The scheme developed in Ref 15, while also based on the stream function and vorticity, employed an upwind approximation for convection. This device largely removed, 'at a stroke' one is tempted to add, the stability problems that had dogged earlier work. Only years later were the side effects of this move to be fully appreciated.

While reducing the number of dependent variables, the use of the stream functions and vorticity had several drawbacks. It is not straightforward to extrapolate this approach to three-dimensional flow and, if the pressure field were of direct interest, it had to be recovered. This treatment, therefore, was fairly quickly superseded by recirculating flow methods based on 'primitive' variables, that is to say the velocity components and the static pressure (see for example Refs 19–21)†. Three-dimensional elliptic treatments emerged virtually in parallel with the two-dimensional schemes although their practical usefulness was severely limited by the capacity of the computers then available; indeed, this is still a limiting factor. Special methods, less expensive in storage and execution time, were evolved, however, to cope with three-dimensional boundary layers (for example Refs 22 and 23).

Thus, with the completion of the first decade of computer-based numerical fluid mechanics, considerable progress had been made in providing general calculation methods for viscous flows of many different types. Progress had also been made in the development of models for the turbulent transport rates. The succeeding decade, bringing us up to the present, should, one might suppose, have been the one that saw the wholesale adoption of these flow calculation methods throughout the advanced technology industries. But it was not. Indeed, the impact of numerical fluid mechanics on the turbomachinery industry in the period 1972–1982 is better described as a species of trench warfare than a computer revolution. Some reasons for this relatively slow introduction, at the practical level, of numerical methods are easy enough to see. A major reason must surely be that the real problems are harder to solve than the idealised ones that code developers habitually consider. Computer programs developed for research are not easily applied to industrial problems even when due care is taken to provide as general a formulation as possible. For example, although the recirculating flow procedure given by Gosman *et al*[15] allowed the use of arbitrary orthogonal coordinates, it contained no mesh-generation scheme to facilitate applications in the complex geometries arising in practical flows. Admittedly, much progress has been made from the mid 1970's in providing methods for body-fitting coordinates, stimulated especially by the work of Thompson and his colleagues[24,25]. The

† There is still widespread use of stream function and vorticity by numerical analysts, see for example the buoyant cavity test case considered at the Conference on Numerical Methods in Thermal Problems[71], Venice, 1981

fruits of that research and equally, that of the several powerful groups developing finite-element methods for turbulent flows, are by no means fully harvested, however. Other factors, perhaps most importantly the scarcity of trained personnel, have also contributed to the rather slow take-up by industry. Rather than lament a decade of half-missed opportunities, however, it is better to try and ensure that, at least, the next ten years will be the ones where numerical methods for computing turbulent shear flows do begin to make a real contribution.

Choice of turbulence model

At least two monographs, several extensive review articles, and countless papers have been written about turbulence models of the kind needed in numerical solutions of the averaged Navier–Stokes equations. Given such comprehensive accounts[26–32], no attempt is made to re-review the extensive literature here. Instead the main strategies are summarised and some personal views expressed on the various questions of choice confronting the user.

The advent of numerical methods for solving, in discretised form, the differential form of the flow equations provided the greatest possible stimulus to research on models for calculating the local turbulent stresses. Here the implied contrast is with the integral methods which generally required global models, such as the factors governing the fluid entrainment rates into the boundary layer. Practical finite difference schemes for turbulent flow employ what are known as single-point closures; these models involve correlations between two or more fluctuating quantities all of which are evaluated at the same point. Within this still wide range of closure options three types of model are seen as appropriate for use in practical calculations:

Boussinesq viscosity models (BVM)
Reynolds stress transport models (RSTM)
Algebraic stress models (ASM)

The ASM is a simplified form of RSTM and some, no doubt, would not accord it a separate classification.

Boussinesq viscosity models

BVM's are based on the idea that turbulence is described by the same type of stress-strain relation as a laminar Newtonian flow:

$$-\rho\left(\overline{u_i u_j}-\frac{1}{3}\delta_{ij}\overline{u_k u_k}\right)=\mu_T\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \qquad (1)$$

where the turbulent viscosity μ_T is to be determined and δ_{ij} is the Kronecker delta. Although this concept is associated with the name of Boussinesq, he made clear[33] that the idea went back at least as far as St Venant, roughly 150 years ago. One might comment that any idea that can survive that long can't be all bad. The first recognisable BVM, one that provided a strategy for finding the turbulent viscosity, was the Prandtl–Taylor mixing length hypothesis which, for the two dimensional thin shear flows that the originators had exclusively in mind, amounted to:

$$\mu_T=\rho l_m^2\left|\frac{\partial U}{\partial y}\right|$$

where $\partial U/\partial y$ is the mean strain and l_m the user-supplied mixing length which was related in a general way to the scale of the turbulent mixing. Current ideas on the BVM approach would hold that $(\partial U/\partial y)$ (which has the dimension $(\text{time})^{-1}$) really stood for a typical time scale of the stress-containing turbulent motion. In practice the mean field only provides an accurate guide to the time scale when stress generation and destruction rates are nearly in balance. On the axis of a jet the mixing length hypothesis would give an infinite turbulent time scale and thus zero turbulent diffusivity. Yet, if by some means we coloured the fluid on one side of the axis we would in fact observe that a rapid mixing of fluid occurred from one side of the jet to the other.

This is one of the fundamental weaknesses of the mixing length hypothesis. Another is that the turbulent length scale is not in general prescribable by algebraic formulae; like the time scale it depends on transport effects and both need to be found from transport equations. Indeed, contemporary BVM treatments are often referred to as 'two-equation models'. The two scalar properties that are the subject of the transport equations could in principle be the turbulent length and time scales but in practice a different pair has always been chosen. The turbulent kinetic energy k is nearly always adopted as one and, while a wide variety of dependent variables has been proposed as the subject of the second equation, the most popular has proved to be the dissipation rate of turbulence energy, ε. The turbulent viscosity is then obtained as:

$$\mu_T=c_\mu \rho k^2/\varepsilon \qquad (2)$$

where c_μ is usually taken as a constant approximately equal to 0.1. The turbulent transport equations, it must be underlined, are not exact. While only the diffusion process requires approximation in the turbulent kinetic energy equation, the ε equation (or one of the alternatives) owes more to intuition and computer optimisation than exact analysis. The main practical advantage of the $k\sim\varepsilon$ model over the other two-equation alternatives is that in thin shear flows it gives roughly the correct level of μ_T in both free shear flows and in flows along walls without the need for any wall-proximity corrections in the latter. This relative generality, alas, falls well short of universality for it is now known that in a turbulent boundary layer near separation or in the impingement region of a jet the ε equation leads to substantially too large levels of viscosity.

In favour of BVM's, it may be said that they are based on a simple-to-understand notion broadly in agreement with observed behaviour; momentum nearly always does get transferred down the velocity gradient. Moreover, from the point of view of the computational fluid mechanicist, turbulence models of this type are rather easily incorporated into his solving procedures and their form is generally conductive to stability. Finally, it is a closure level at which it is fairly easy to include the direct effects of molecular viscosity on the turbulence, an influence that becomes essential to include in the immediate vicinity of a wall. Against these undoubted virtues must be set a list of shortcomings. BVM's will give the correct level of shear stress only in simple strain fields. Moreover they always get the stress components with only small amounts of direct generation badly wrong. Because the turbulent viscosity is taken as a scalar, BVM's do not respond correctly to the effects of force fields which, by their nature, exhibit directional preference. Even in a simple shear in the absence of body forces the effective diffusivity in the boundary layer is still highly non-isotropic as one of the examples will later display.

These shortcomings have led several groups around the world to conclude that if numerical computations are to make a significant contribution to solving industrial problems of turbulent flow, BVM models, with all their attractions, need to be replaced by a more general representation. Although there have been one or two suggestions about developing a model for a tensorial turbulent viscosity, it is now generally agreed that the modelling level best suited to the task is the second moment closure or, as we here refer to it, the Reynolds stress transport model.

Stress transport and algebraic stress models

An exact transport equation for the rate of increase of Reynolds stress can readily be obtained by taking appropriate velocity-weighted averages of the Navier–Stokes equations. In symbolic form it may be written:

$$\frac{D\overline{u_i u_j}}{Dt} = P_{ij} + \phi_{ij} - \varepsilon_{ij} + D_{ij} \tag{3}$$

The symbols on the right of Eq (3) are shorthand notation for turbulence correlations which exert the following physical effects:

- P_{ij} stress generation through mean strain
- ϕ_{ij} randomising actions of pressure-containing correlations
- ε_{ij} viscous dissipation
- D_{ij} diffusive transport of stress by velocity and pressure fluctuations

The last three processes are not exactly determinable in a second-moment closure and they are nominally approximated term by term. The first can be written in full as:

$$P_{ij} \equiv -\left(\overline{u_i u_k}\frac{\partial U_j}{\partial x_k} + \overline{u_j u_k}\frac{\partial U_i}{\partial x_k}\right)$$

The summation convention is adopted whereby the repeated suffix k is to be summed over the three Cartesian directions. Evidently P_{ij} contains only mean velocity gradients and Reynolds stresses and thus requires no further approximation. It is a major term in nearly all turbulent flows and it is the fact that it can be handled exactly that offers the hope of being able to achieve a reasonable width of applicability for turbulence models devised at this level. The form of the term suggests far more subtle and intricate connections between the mean strain and stress fields than arise from the notion of an isotropic turbulent viscosity.

For turbomachine applications the most important of the unknown processes to approximate is the pressure-containing correlations, ϕ_{ij}. If we regard P_{ij} as the 'income' to the Reynolds stresses ϕ_{ij} represents the taxes. It is a fairly sophisticated tax structure (though possibly not as complex as that of our own society) with Robin Hood like 'rob the rich, reward the poor' elements in addition to the draining off of funds with no visible return with which we are also familiar. It is now generally recognised that any approximation to ϕ_{ij} should comprise two elements: one containing purely turbulence correlations (ϕ_{ij1}) and a second introducing an additional influence of mean strain (ϕ_{ij2}). Quite elaborate models have been devised for each of these processes (see for example Lumley[32]) though their use has largely been limited to highly contrived homogeneous flows. The more elaborate the model the more difficult the task of optimising the empirical coefficients and, on more that one occasion, such 'advanced' treatments have given entirely erroneous predictions when used to calculated a more complex shear flow (see for example Ref 39). It is partly for this reason that most applications to flows of the type found in gas turbines have so far been made with the following forms which, while undoubtedly giving only a grossly simplified account of the action of the pressure-containing correlations, nevertheless do capture the 'first-order' tendencies:

$$\phi_{ij} = \underbrace{-c_1\frac{\varepsilon}{k}\left(\overline{u_i u_j} - \frac{1}{3}\delta_{ij}\overline{u_k u_k}\right)}_{\phi_{ij1}} \underbrace{- c_2\left(P_{ij} - \frac{1}{3}P_{kk}\delta_{ij}\right)}_{\phi_{ij2}} \tag{4}$$

The proposal for ϕ_{ij1} is due to a very early contribution by Rotta[36], that for ϕ_{ij2} to Naot, Shavit and Wolfshtein[37] though it must be said that they recommended it as a replacement for ϕ_{ij1}. The first use of both the terms of Eq (4) was probably by Rodi[38]. To continue our fiscal metaphor, ϕ_{ij1} represents a wealth tax (though, for $i=j$ it will tend to increase the level of those components with less than the average wealth) while ϕ_{ij2} is an income tax (which again provides a refund for those normal stress elements with a below average income). The best choice of constant coefficients for c_1 and c_2 is approximately 1.8 and 0.55. Several attempts have been made to let c_1 and c_2 depend upon dimensionless turbulence parameters (eg Lumley[32], Chung and Adrian[40], Sindir[41], Cler[42]) but there is no clear evidence that, for a shear flow, this produces any overall improvement.

This is not to suggest that no further improvement in the model for ϕ_{ij} is possible. It probably does indicate, however, that rather than introduce empirical dependencies into the coefficients of Eq (4) it will be better to start with a somewhat more general mathematical framework within which to optimise the model. As we have noted above, several such schemes are currently under development by different groups around the world. Despite the difficulties of optimisation, we can expect to see their introduction to gas turbine flows in the next few years.

The vital role of the pressure interaction terms in both sustaining turbulence yet limiting its level may be seen from considering the direct generation and dissipation rates of the individual stress components in a simple shear flow ($U_1 = U_1(x_2)$, $U_2 = U_3 = 0$). Here we interject that viscous dissipation in a turbulent flow is generally held to take place in fine scale motions that have no sense of what is happening to the mean motion, or indeed, to the larger-scale turbulent eddies. In short, the dissipating eddies are regarded as isotropic which allows the tensorial dissipation rate ε_{ij} to be expressed in terms of the turbulence energy dissipation rate (a scalar) and the Kronecker delta:

$$\varepsilon_{ij} = \tfrac{2}{3}\delta_{ij}\varepsilon \tag{5}$$

The resultant production and dissipation rates for the individual components are given in Table 1. There is no direct generation of $\overline{u_2^2}$, the mean square level of turbulent velocity fluctuations in the direction of the mean velocity gradient; any turbulence activity in that direction is thus attributable to the intervention of pressure-containing correlations. Their action in transferring turbulence energy from the mean flow direction to the $x_2 - x_3$ plane completes what might be called turbulence's eternal triangle suggested in Fig 1: velocity fluctuations down the mean velocity gradient tend to augment shear stress $\overline{u_1u_2}$ which in turn leads to a generation of $\overline{u_1^2}$; some of this energy is diverted via pressure fluctuations into u_2, etc. Eq (5) indicates that there is no direct dissipation of shear stress and it is again ϕ_{ij1} which prevents the unlimited growth of this component. The reader may readily verify that the postulated form[4] of ϕ_{ij} does, qualitatively at least, provide a correct mixture of energy redistribution and shear stress destruction.

In most wall-bounded shear flows the processes, P_{ij}, ϕ_{ij1} and ϕ_{ij2} are collectively the most influential factors. To determine the relative levels of the stresses, though not their absolute levels, it is often sufficient to neglect transport effects entirely. If one makes this approximation of local-equilibrium, combination of Eqs (3), (4) and (5) gives the

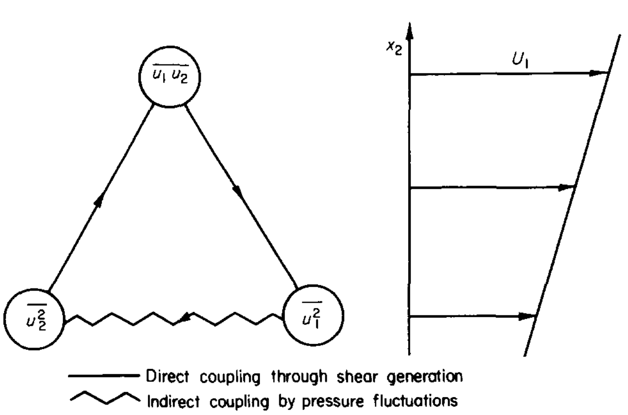

Fig 1 The eternal triangle of turbulent shear flows

Table 1 Production and dissipation rates

Component	P_{ij}	ε_{ij}
$\overline{u_1^2}$	$-2\overline{u_1u_2}\dfrac{dU_1}{dx_2}$	$\tfrac{2}{3}\varepsilon$
$\overline{u_2^2}$	0	$\tfrac{2}{3}\varepsilon$
$\overline{u_1u_2}$	$-\overline{u_2^2}\dfrac{dU_1}{dx_2}$	0

following algebraic expression for the Reynolds stresses:

$$(\overline{u_iu_j} - \tfrac{2}{3}\delta_{ij}k) = \frac{(1-c_2)}{c_1}\frac{k}{\varepsilon}(P_{ij} - \tfrac{1}{3}\delta_{ij}P_{kk}) \tag{6}$$

Eq (6) offers a constitutive relation between stress and strain which may be used in place of the Boussinesq formula (Eq (2)). It represents the simplest form of ASM. The scalars k and ε remain as unknowns and are conventionally obtained by solving the same pair of approximate transport equations as with the BVM 2-equation model. While the presence of the shear production tensor in Eq (6) leads in general to quite different stress strain relations than Eq (2), for the particular case of a simple shear, the formulae for the shear stress from these two approaches happen to be equivalent. This comparison helps show why, for the considered example, the idea of an effective turbulent viscosity 'works', yet underlines how dangerous it is to extend its use to complex strain fields. Most developers of ASM treatments feel able to do better than merely omitting transport effects on the relative stress levels. The transport of the stresses is approximated in terms of the corresponding transport of turbulence energy, the most widely used proposal being due to Rodi[43]. When this transport approximation is incorporated, Eq (6) is again recovered, save that the constant coefficient $(1-c_2)/c_1$ is replaced by a quantity dependent on the ratio of the rates of turbulence energy generation and dissipation.

Eq (6) is not suitable as it stands for use in boundary layer computations because, close to a rigid boundary, it gives too large levels of turbulent fluctuations normal to the wall (and it is these fluctuations which, as Fig 1 has indicated, are responsible for shear-stress generation). Turbulent pressure fluctuations reflect from the surface producing an 'echo' which impedes the flow of energy from streamwise to the normal direction. Models for the wall-reflection processes have been suggested[31,44,45] which seem to work well enough for plane or nearly plane surfaces. They fall well short of being theories, however, and their extension to more complex cases where, say, two walls intersect at right angles is not yet convincingly demonstrated.

The rationale of representing stress transport in terms of turbulence energy transport derives from the desire for comparative numerical simplicity rather than physical realism. Reynolds stress transport models (RSTM's) retain stress convection exactly and adopt less restrictive models of diffusive transport (D_{ij}) than is possible within an ASM framework. One must thus solve a system of (discretised) differential equations for each non-zero stress component rather than just one for the turbulence energy. Whether the extra computational effort is justified or not has not been established in any general way. Indeed, no general determination is possible since the particular flow considered and the accuracy required will always be important ingredients in deciding the issue. For two-dimensional thin shear flows, the additional storage and execution time for an RSTM is modest but, in the cases where comparisons can be drawn, the improvement in accuracy over that of an ASM treatment is usually rather slight. In three-dimensional and recirculating flows the additional core required for an RSTM treatment has proved to be something of a deterrent but, undoubtedly, as larger computer storage becomes available models of this type will be tested.

The above discussion has considered purely the question of calculating the Reynolds stresses. For flows in hot-section components of a gas turbine, turbulent heat fluxes will also be of interest; indeed, for some problems the principal interest. While the adoption of a uniform, isotropic turbulent Prandtl number is common in thermal boundary layer analysis this concept has similar weaknesses to that of the turbulent viscosity idea: it will at best be useful when turbulent heat fluxes are important in only one direction, as in a two-dimensional thermal boundary layer. For more general circumstances, models based on closure of transport-equations for the turbulent heat-fluxes are more likely to be successful. The simplest formula that can claim such an ancestry is readily obtained. Mean temperature gradients ($\partial T/\partial x_j$) present in a fluctuating turbulent field produce turbulent heat fluxes $\overline{u_i t}$ at a rate: $-\overline{u_i u_j}\,\partial T/\partial x_j$. Thus, by strict analogy with Eq (6), one is tempted to suggest the heat fluxes be approximated by:

$$\overline{u_i t} \propto -\frac{k}{\varepsilon}\overline{u_i u_j}\frac{\partial T}{\partial x_j} \tag{7}$$

Eq (7) has, indeed, been used by several workers[46,47] with a constant of proportionality of about 0.3. It is not as general as Eq (6) since it neglects the fact that mean velocity gradients also produce a heat flux, but nevertheless correctly accounts for phenomena that cannot be represented with an isotropic Prandtl number. Suppose, for example, one wishes to calculate how rapidly a hot streak in a boundary layer on a turbine blade will die out. Eq (7) tells us that, if the velocity field is two dimensional, the effective diffusivities in the direction normal to the blade surface (x_2) and that in the spanwise (root-to-tip) direction (x_3) are in the ratio $\overline{u_2^2}:\overline{u_3^2}$. The level of $\overline{u_3^2}$ near the surface is several times that of $\overline{u_2^2}$ implying that the diffusion of heat will proceed highly non-isotropically.

Some turbulent flow computations

The flow computations discussed in this section have been chosen for their intrinsic relevance to gas turbine flow. Broadly they proceed from the simple to the more complex and, so far as possible, complexities are added one at a time. Except where details of the numerical solution are provided it may be assumed that the computor used a sufficient density of nodes to render any numerical error insignificant. Rather more than half the examples are taken from computations submitted to the Stanford-AFOSR Conference on Complex Shear Flows held in September 1981. The selection aims to give a balanced view of what can be done and what cannot.

The first example, one of the Conference test cases, relates to the effect of high free stream turbulence on the skin friction coefficient in a turbulent boundary layer. Bradshaw[48] (see also Hancock[49]) proposed that the combined effects of turbulent length scale and turbulence intensity could be fairly well correlated by the chosen abscissa of Fig. 2. The symbols represent his proposal, not the experimental data themselves which, or course, show scatter about this position. The lines represent a mean locus through many sets of computations obtained by the UMIST group[50] for the BVM and ASM models, both based on solution of the standard k and ε equations. Several groups around the world tackled this case with similar results to that shown in Fig 2. The extra effect of free-stream turbulence on c_f is underestimated in the middle range of turbulence intensities, though apparently less so at the higher intensity end. It might be added that one of the most complete sets of experimental data by Charnay[51] displays a variation in much closer agreement with the computations. I conclude that the influence of free stream turbulence is probably acceptably accounted for.

As is well known, a much more dramatic effect of free stream turbulence is associated with laminar-turbulent transition. The turbulence models discussed in this paper are not able to predict transition phenomena in any general way. For the specific case found in gas turbines, where the transition process is dominated by diffusion to the wall of turbulence energy from outside the boundary layer, however,

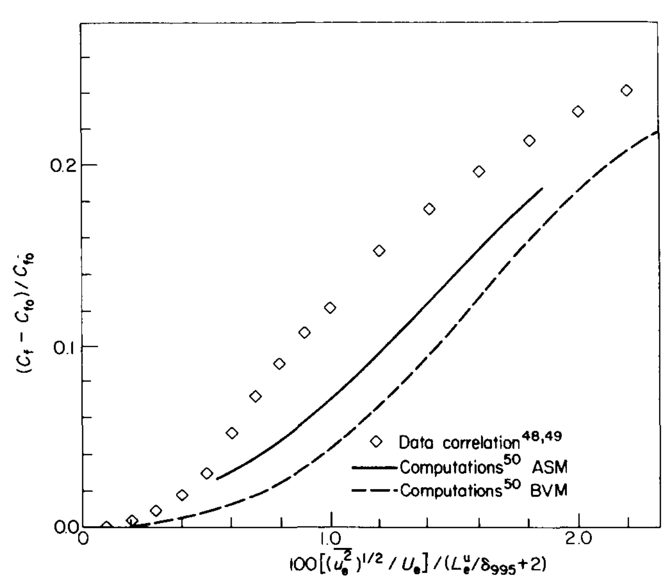

Fig 2 Effects of free-stream turbulence on surface friction

there are firmer grounds for optimism. Fig 3 shows predictions by Priddin[52] of the variation of heat transfer coefficient around the suction surface of a blade of a turbine cascade, the data being those of Turner[53]. The turbulence model employed was the $k-\varepsilon$ BVM incorporating low Reynolds number effects[54]. Three turbulence levels are shown, the lowest corresponding to the background turbulence level of the wind tunnel. In this latter case the flow remains laminar over the blade. At 2.2 and 5.9% turbulence intensity (upstream of the blade) significantly higher levels of heat transfer coefficient result, though at different chord postions the percentage augmentations are quite different. The computations mimic these effects very closely and, since they provide detailed predictions of the velocity field across the boundary layer, allow an explanation of the curiously non-uniform behaviour. At 2.2% intensity the external turbulence level is insufficient to provoke complete transition to turbulent flow given the strongly favourable pressure gradient. By contrast, at 5.9% virtually complete transition occurs at about 15% chord. The excellent measure of agreement obtained in this case probably gives a too favourable view of the ability of two-equation BVM's to account for diffusion-induced transition. Even allowing for a measure of good fortune, however, the results must still be seen as highly encouraging.

A final example concerning effects of a highly turbulent free stream is provided by Fig 4, again drawn from the work of the UMIST group for the Stanford Conference[50]. The development of skin friction in a 4° conical diffuser is shown for two different sets of entry conditions: in one case the entry flow is smooth and in the other the flow is delivered through a small diameter pipe which undergoes a sudden enlargement some fifteen diameters before the diffuser entry. The expansion provokes a large confined separated region leading to high entering turbulence levels. The distributions of wall friction are quite different for the two cases indicating that experiments conducted in 'clean' laboratory conditions are of little value for estimating performance of the same hardware in the highly turbulent environment of a gas turbine. The computations however manage to cope fairly well with the effects of the different initial conditions. The too high level of friction with the BVM for the case of low core intensity in fact arises from a tendency of the ε equation to produce too large length scales (and thus too large viscosities) as the boundary layer approaches separation.

Attention is now shifted to the effects of streamline curvature. The flow considered in Fig 5 is a rectangular sectioned duct of large aspect ratio[55]. One of the major sides of the test section provides, in its central portion, a convex surface of circular arc and in the initial and final sections a plane surface. The distance between the test surface and the opposite wall is carefully tailored to give only a small pressure gradient along the test plate itself. The behaviour of the skin friction coefficient is shown in Fig 5. The sudden dip in measured c_f corresponds with the entry to the curved section ($x_s = 0$) which extends as far as $x_s = 0.8$. The computations of this flow, abstracted from the UMIST submission to the Stanford Conference, broadly indicate that the ASM model shows a reaction to streamline bending similar to that found in the experiment while the BVM calculations exhibit hardly any response. The reasons for this lie in the different stress-strain connections adopted in the two models (Eqs (6) and (2)). With the Boussinesq model the imposition of a small

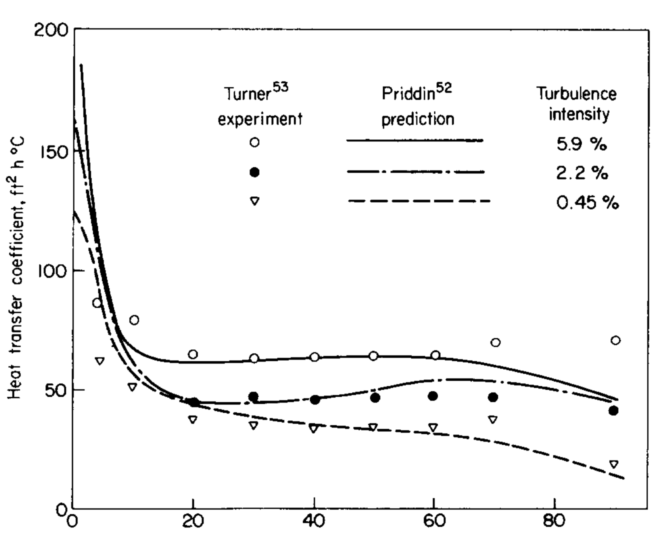

Fig 3 Diffusion-induced transition on turbine blade

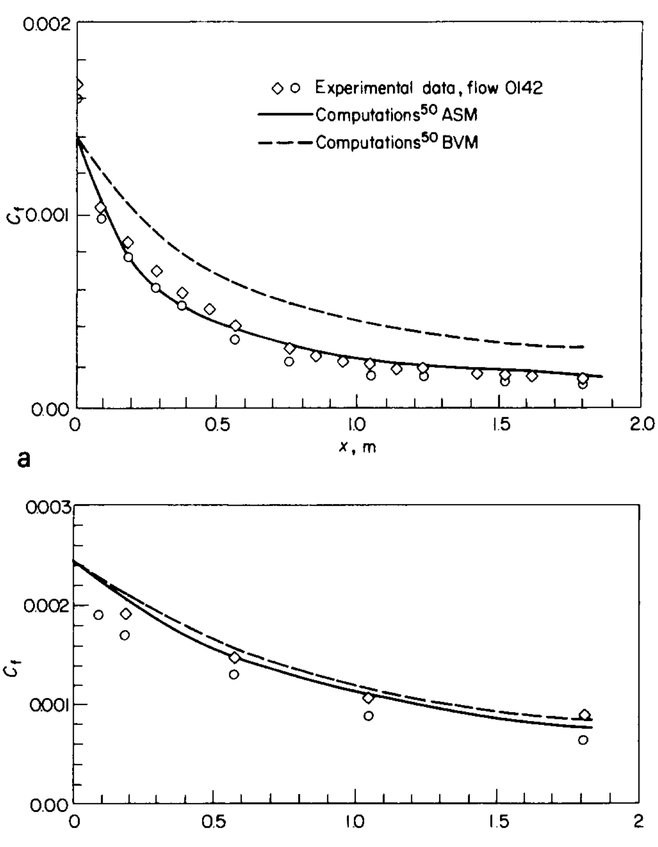

Fig 4 Skin friction coefficient in 40 conical diffuser (a) low core turbulence (b) 'high' core turbulence

secondary strain associated with curvature produces the same percentage change in stress level. In the ASM model, however, the various strain components are multiplied by different stress components; there is also cross-talk between the individual stress components. Collectively the effect is to render the system of equations highly sensitive to small secondary strains; indeed, almost as sensitive as real turbulence.

These examples have considered how boundary layer development is modified by high levels of free stream turbulence, pressure gradient and wall curvature. All these features are present in flow over blades. There are other factors as well, however, perhaps the most important of which is that real blade flows are significantly 3-dimensional. One of the Stanford test cases was the idealised wing-body junction of Shabaka[56] illustrated in Fig 6. Vortex lines in the wall boundary layer upstream of the 'wing' are bent around in the familiar horseshoe vortex giving rise to strong streamwise vorticity. The Conference did not require participants to start their computations ahead of the junction for the organisers suspected that the task of properly capturing the horseshoe vortex lay beyond the capabilities of present viscous flow computation schemes‡. Instead primary and secondary velocity profiles were supplied a short distance downstream of the leading edge. The computor's task was thus to calculate the three-dimensional boundary layer development downstream of that point. Fig 7 shows the computed and measured distributions of skin friction coefficient along the 'wing' (y) and 'body' (z) surfaces at the most downstream station. Away from the vicinity of the corner c_f falls with increasing z along the 'body' but shows a tendency to increase along the 'wing'. This contrasting behaviour arises from the decaying counter-clockwise flow rotation (viewed from upstream) produced by the horseshoe vortex which lifts fluid away from the wall along the 'body' but brings high velocity fluid towards the wall on the 'wing'. The computations are those submitted by Rodi's group[57] using the $k-\varepsilon$ BVM treatment. The agreement is, on the whole, impressive though the calculated behaviour misses the overshoot on c_f on the wing surface near the root. This feature, which appears to be due to a weak secondary streamwise vortex, might have been resolved had an ASM treatment been applied.

Three dimensionality of flow over blading is also important in the blade wakes. Hah and Lakshminarayana[58] have undertaken an impressive computational study of rotor wakes employing both BVM and ASM treatments. In these flows strong Coriolis force fields are present. A fully 3-dimensional discretisation of the Reynolds equations was made with 40 nodes in the streamwise and azimuthal (blade-to blade) directions and 10 radial nodes. The authors do not establish the insignificance of numerical diffusion but there is reason to suppose it is not a *major* factor since the flow is of thin shear flow type and the authors report that their computations were highly sensitive to the choice of turbulence model (numerical diffusion would dull this sensitivity). Mean velocity profiles are reproduced from their paper in Fig 8, where U, V and W denote streamwise azimuthal and radial components respectively, n is the azimuthal distance, S is the circumferential dis-

‡ They did, however, leave open the possibility of so doing in case anyone wished to demonstrate the wide applicability of his computational method. No-one did.

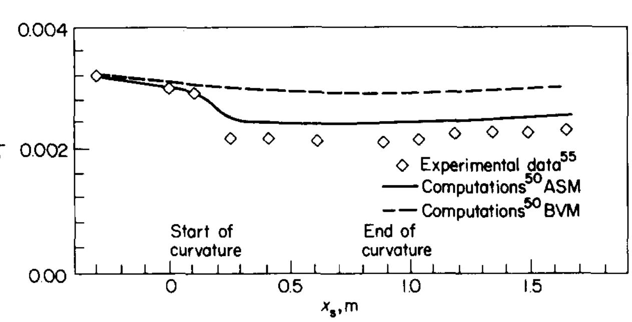

Fig 5 Development of skin friction on a convex surface

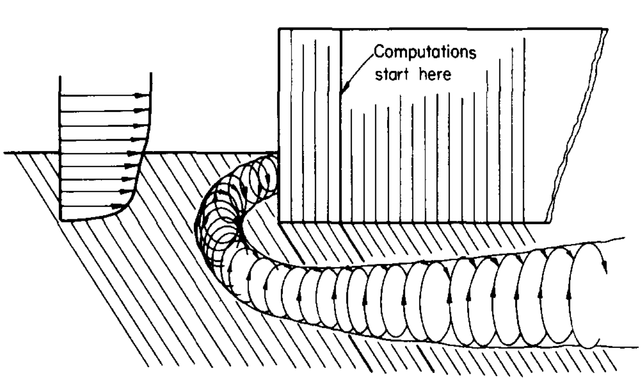

Fig 6 Idealised wing-body junction

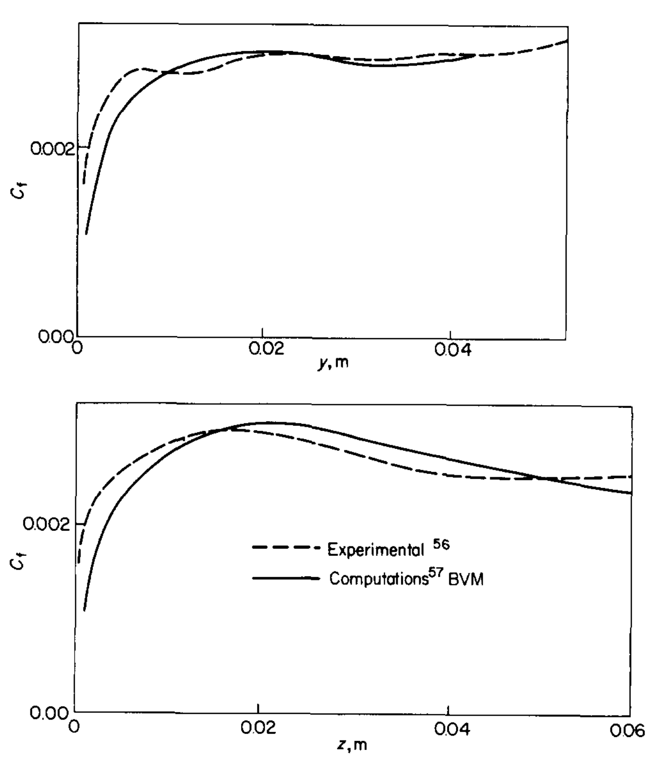

Fig 7 Computation of skin friction in flow over an idealised wing-body junction (Rodi et al[57])

tance between the blades at mid radius, and s is the distance downstream of the trailing edge. Comparisons with the data of Reynolds *et al*[72] are made at a ratio of local-to-tip radius of 0.8, the computations being those obtained with the ASM treatment. Agreement between the two is, as the authors themselves comment, remarkably good.

On turbine blades additional flow three-dimensionality may be introduced by the injection of coolant through holes in the blade. In a series of papers, Bergeles *et al*[59–61] have considered various aspects of this problem. The initial study, indicated in Fig 9, was the injection from a single isolated hole into a two-dimensional boundary layer in zero pressure gradient. An expanding 3-dimensional mesh covered the hole region as indicated. To save storage a 'semi-elliptic' rather than a fully 3-dimensional computation was undertaken which meant that only the pressure field was held in a 3-dimensional array. All other dependent variables were stored on just two adjacent planes at right angles to the primary flow with information successively passed from the upstream to the downstream plane. This device allowed the use of a finer mesh than would otherwise have been available (up to 80 nodes in the streamwise direction) but limited the study to low injection rates as the appearance of negative values of streamwise velocity could not be accommodated. (Beyond 5 diameters behind the hole where pressure variations were small computations switched to a boundary layer treatment in which even the pressure was stored only two dimensionally.) The focus of the computations was on the dilution and lateral spread of the cool fluid, particularly regarding the composition of the mixture at the wall. The film-cooling effectiveness η shown in Fig 10 gives the fraction of the cool fluid present at the surface for a mean injection velocity equal to 10% of the free stream value. The variation of η with downstream distance is shown for a line directly behind the injection hole and at 0.5*D*, 1.0*D* and 1.5*D* offset (*D* being the hole diameter). Agreement with the experimental data of Ramsey and Goldstein[62] is virtually complete. The turbulence model used was the usual $k\sim\varepsilon$ BVM except that the diffusivity in the lateral direction was enhanced relative to that normal to the surface in approximately the ratio $\overline{u_3^2}/\overline{u_2^2}$, in line with the indications of the ASM approach (the ratio $\overline{u_3^2}/\overline{u_2^2}$ was prescribed as an algebraic function of position in the boundary layer). The importance of this adaptation is seen in Fig 10(b) which includes predictions (shown by the broken line) with the standard $k\sim\varepsilon$ BVM. These computations were started at $x_1/D=6$, the origin of x_1 being the ***upstream*** lip of the hole. By $x_1/D=20$ a serious difference has developed in the η decay pattern, the levels on the centreline ($x_3/D=0$) being some 40% too high and that along $x_3/D=1$ a similar amount too low.

The same authors extended their studies to single and double rows of holes and to higher injection rates[60,61]. As the flow became more complex the quality of agreement gradually deteriorated. For holes located near the stagnation region of a turbine blade, external velocities parallel to the blade are low and a 'fully elliptic' numerical treatment is then needed. White[63] has reported successful application of computations of this type to stagnation region film cooling. At present computer times for these types

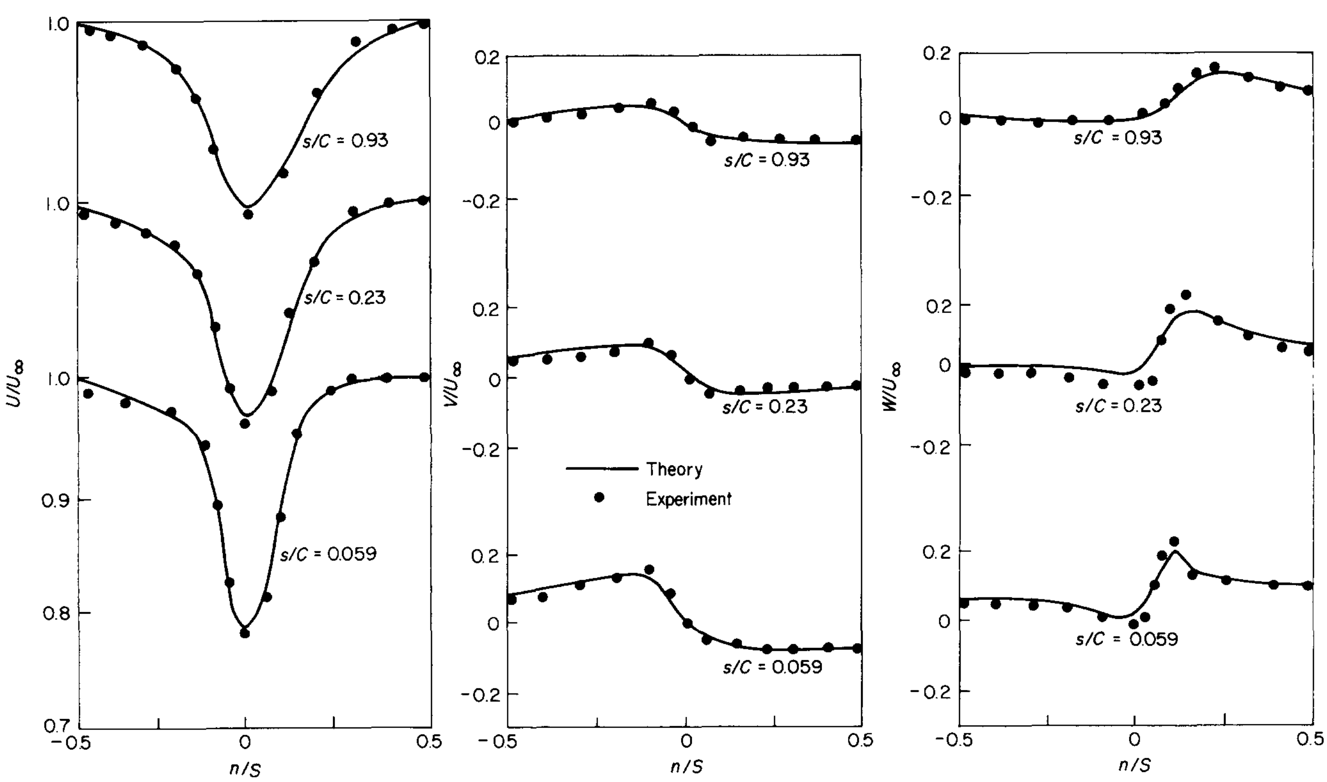

Fig 8 Development of three dimensional wakes from turbomachinery blades (Hah and Lakshminarayana[58]) (Reproduced by kind permission of the American Society of Mechanical Engineers)

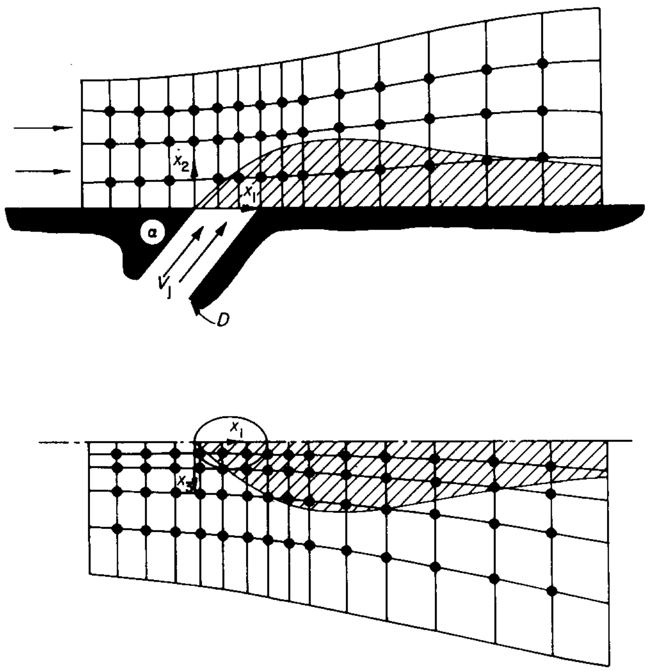

Fig 9 Computational mesh for numerical study of discrete-hole injection (Bergeles et al[59]) (Reproduced by kind permission of Numerical Heat Transfer)

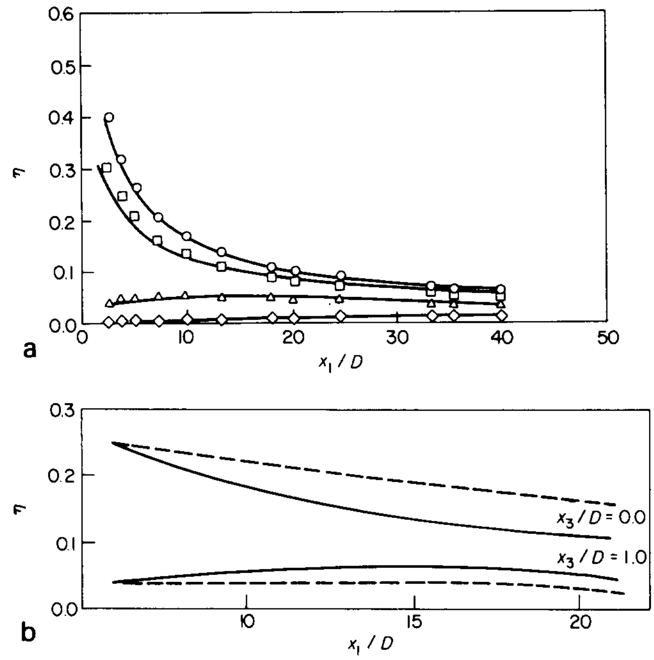

Fig 10 Film cooling effectiveness for flow downstream of a single 30° hole[59] (a) Comparison with experiments of Ramsey and Goldstein[62]; ○—$x_3/D = 0$; □—$x_3/D = 0.5$; △—$x_3/D = 1.0$; ◇—$x_3/D = 1.5$. (b) Comparison of predicted results from non-isotropic (solid line), and isotropic (----) diffusion models. (Reproduced by kind permission of Numerical Heat Transfer)

of calculation are too long, typically about one half hour on a Cyber 205, for routine pre-design type explorations, particularly as the accuracy of the prediction is uncertain. Simpler, laterally-averaged two-dimensional treatments such as developed by Crawford *et al*[70] require only about one hundredth of the computing effort and are clearly very useful in giving a guide to mean cooling effectiveness. Nevertheless for 'trouble shooting', such as in tracking down the cause of local hotspots giving recurrent blade failures, the additional detail provided by the 3-dimensional programs is invaluable.

Current trends are to make the coolant do as much cooling as possible before discharge to the outer surface of the blade. Two recent computational studies have examined different aspects of internal blade cooling. Howard *et al*[64] have considered heat and momentum transfer in long rotating passages where Coriolis forces exert strong effects on the turbulence structure. (The authors in fact regard their study as especially relevant to flow in the impeller passage of a centrifugal compressor.) The Coriolis forces directly affect the Reynolds stress production tensor. Howard *et al*[64], while not incorporating a full ASM treatment, have shown that the inclusion of the Coriolis terms as indicated by P_{ij} leads to fairly satisfactory accounting of the measured flow behaviour. The second study is the impinging jet exploration of Amano[65] which is closely related to splash-cooling problems. In this type of flow the level of wall stress or heat transfer coefficient is highly sensitive to the physical model of the immediate near-wall region where turbulent fluctuations are damped and significant transfer of momentum and heat occurs through molecular interactions. Amano's computations, using the standard $k-\varepsilon$ BVM and a simple two-layer model of the near-wall zone, achieve generally satisfactory agreement with available experiments not only of skin friction coefficient, of which an example appears in Fig 11, but also of the turbulence energy distributions near the surface.

The final example considered is perhaps the most difficult: the flow around a 90° bend in a duct of square cross section (Fig 12(a)). While not especially mimicking the flow in any gas turbine component, it has most of the main ingredients that make turbulent flows in turbines so challenging to predict. The computations shown in Fig 12(b) were contributed to the Stanford Conference by Chang *et al*[66]; they are problably the most successful set from

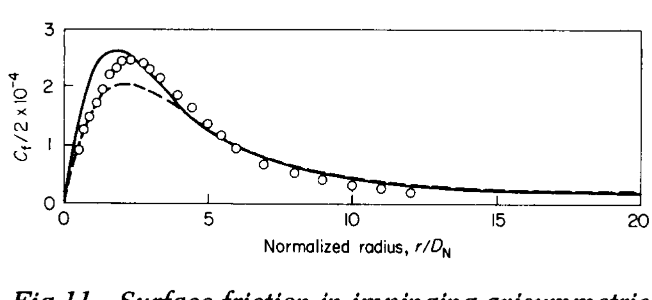

Fig 11 Surface friction in impinging axisymmetric turbulent jet. Computations; Amano[65], with BVM model. Experiment (○) Bradshaw and Love[73] ---- computations with uniform velocity at nozzle exit; —— computations with velocity varying with 1/5.5 power of distance from nozzle wall.

the three groups submitting computations for this flow. A 3-dimensional semi-elliptic code was employed with a 14×15 grid to map the half section (the flow being symmetric about the horizontal midplane of the duct) with a total of 94 streamwise sections divided roughly equally between the straight region of duct approaching the bend, the bend itself, and the downstream tangent. Fig 12(b) compares measured and calculated streamwise isovels at 71° from the bend inlet. The increase of pressure on the outside of the bend leads to a substantial secondary flow as sketched in Fig 12(a). This in turn distorts the streamwise isovels. Fig 12(b) in fact contains two sets of computations, based on different numerical approximations of convective transport. One of these, 'HYBRID', is a minor variant on the first-order upwind differencing which became the 'miracle' of the late 1960's but is now recognised to create so much numerical dispersion that, for all its stability, it seems of little value for 3-dimensional computations (where grids are necessarily fairly coarse) unless the grid can be adapted to follow the flow. Humphrey's[66] group's computations with the quadratic upstream differencing (QUICK) scheme of Leonard[67] (formally of 3rd-order accuracy) are the first reported with this discretisation for a 3-dimensional turbulent flow. In various tests of accuracy in simpler flows[68] it has performed encouragingly well, so there was especial interest in seeing what predictions it generated for the 90° bend. Fig 12(b) allows certain conclusions to be drawn, but leaves many more questions to be answered. For example, the fact that no contour for $U/\bar{U} = 1.28$ exists for the HYBRID computations is in line with the comments above on the serious numerical diffusion associated with this scheme. The QUICK results are better in this respect, indeed they are in significantly closer agreement with the experimental behaviour than those obtained with HYBRID. Yet, there are still substantial differences between experiment and computation. Are these due to weaknesses in the turbulence model (the standard $k-\varepsilon$ BVM was adopted), to a grid that was insufficiently fine even though QUICK was used, or to some other cause? The writer's hunch is that while errors arising from the first two sources may well be significant, a more important source of the discrepancy may be in handling the boundary condition for the secondary velocity component. Intensive computational work on this flow is in progress among several of the major groups in computational fluid mechanics. It is thus likely that conjecture on these topics can be turned to conclusion in the near future.

Concluding remarks

The final example, with its inconclusive result was, perhaps, the best example on which to end if a balanced view of where we stand is to emerge. The comparisons have shown that when flow complexities are added one at a time in the established style of academic research, the calculational schemes, comprising a turbulence model and a procedure for solving the equations, do quite well, particularly when ASM treatments for the turbulent stresses or fluxes are included. Successful prediction is, however, much less certain when all the complexities are present simultaneously and where severe storage or time limitations prevent as fine a grid being used as one would otherwise desire.

Recently a working party on turbulence modelling was set up to enquire whether uncertainties in turbulence modelling were responsible for the comparative failure of computational fluid mechanics to make an impact in the UK aeronautics industry, including the gas turbine industry. The working party's report[69] acknowledged the limited applicability of current turbulence models but

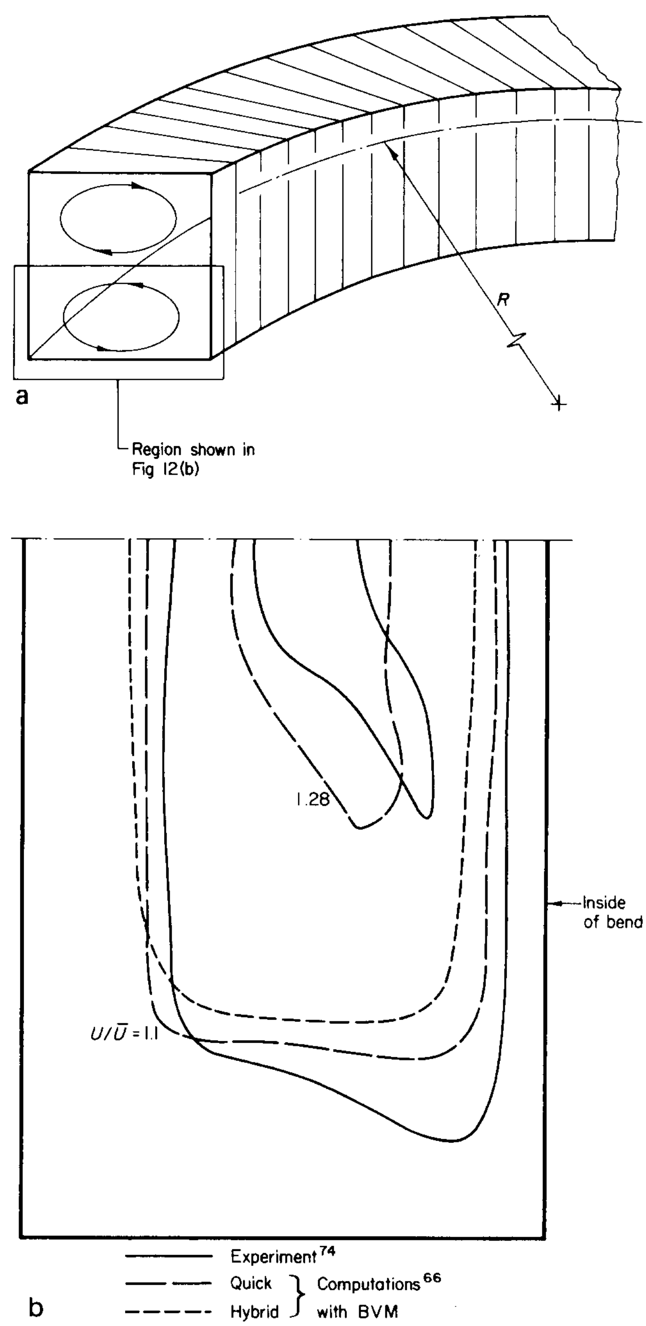

Fig 12 Computation of flow in square sectioned duct around a 90° bend. (a) Flow configuration (b) Computed and measured axial velocity contours at 71° into the bend.

identified the main problems to be software, ie the lack of sufficiently adaptable computer programs for industrial type use, and acute shortage of manpower with the requisite skills. Certainly, in comparison with current activity in France and its development over the last decade, the UK programme is risible. Predictably, the USA is mounting a strong effort through the various NASA laboratories and the myriad, small, spin-off companies that compete vigorously for the considerable monies made available by government agencies for computational studies in turbulent shear flows. Perhaps the most interesting aspect of the US scene, however, is that even there, where continual change is the norm and where innovation is traditionally expected to arrive under its own steam, major government funded programmes in computational fluid mechanics have been launched at selected universities to increase greatly the supply of young people with specialised training. If there is a need there, where the value of computational fluid mechanics is already widely accepted, how much more urgent is the need for stimulus elsewhere, not just in the universities but also in the aircraft and gas turbine industries.

References

1. **Schumann U.** *J. Comp. Phys., 1975,* **18**, *376–404*
2. **Grötzbach G. and Schumann U.** Turbulent Shear Flows—1. *Springer-Verlag, 1979, 370–385*
3. **Grötzbach G.** Direct numerical simulation of turbulent momentum and heat transfer in an internally heated fluid layer. *Proc. 7th Int. Heat Transfer Conference, Munchen, September 1982*
4. **Mansour N. N., Ferziger J. H. and Reynolds W. C.** Large-eddy simulation of a turbulent mixing layer. *Rep TF-11 Thermo-sicences, Div, Mech. Eng. Dept., Stanford University, April 1978*
5. **Feiereisen W. J., Shirani E., Reynolds W. C. and Ferziger J. H.** Direct simulation of homogeneous, turbulent shear flows on the ILLIAC-IV Computer: Applications to Compressible and Incompressible Modelling. *Proc. 3rd Turbulent Shear Flow Symposium p. 19. 31, Univ. California Davis, Sept, 1981*
6. **Moin P. and Kim J.** Numerical investigation of turbulent channel flow. *NASA Ames Research Center, July 1981*
7. **Moses H. L.** The behaviour of turbulent boundary layers in adverse pressure gradients *MIT Gas Turbine Lab. Rep 73, 1964*
8. **Spalding D. B.** A unified theory of friction, heat transfer and mass transfer in the turbulent boundary layer and wall jet *ARC CP 829, 1964*
9. **Coles D. E.** *J. Fluid Mech., 1956,* **1**, *191–226*
10. **Patankar S. V. and Spalding D. B.** *Int. J. Heat Mass Transfer, 1967,* **10**, *1389*
11. **Kline S. J., Morkovin M. V., Sovran G. and Cockrell D. J.** *(eds) Proc. Computation of Turbulent Boundary Layers–1968 Vol. 1, Stanford University, 1968*
12. Free Turbulent Shear Flows–Vol. 1 Conference Proceedings *NASA Langley Research Center NASA SP321, 1973*
13. **Wolfshtein M.** Convection processes in turbulent impinging jets *PhD Thesis Faculty of Engineering/University of London 1967*
14. **Runchal A. K. and Wolfshtein M.** *J. Mech. Eng. Sci., 1969,* **11**, *445*
15. **Gosman A. D., Pun W. P., Runchal A. K., Spalding D. B. and Wolfshtein M.** Heat and Mass Transfer in Recirculating Flows *Academic, London, 1969*
16. **Thom A.** *Proc. Roy Soc. A, 1933,* **141**, *651*
17. **Apelt C. J.** *ARC. R & M 3175, 1961*
18. **Kawaguti M.** *J. Phys. Soc. Japan, 1953,* **8**, *747*
19. **Harlow F. H. and Welch J. E.** *Phys. Fluids, 1965,* **8**, *2182*
20. **Thommen H. U.** *ZAMP, 1966,* **17**, *369*
21. **Chorin A. J.** *Math. Computation, 1968,* **22**, *745*
22. **Patankar S. V. and Spalding D. B.** *Int. J. Heat & Mass Transfer, 1972,* **15**, *1787*
23. **Briley W. R.** The computation of 3-dimensional viscous internal flows *Proc. 3rd Int. Conf. on Numerical Methods, Paris, Vol. II, 1972*
24. **Thompson J. F., Thames F. C. and Matsun C. W.** *J. Comp. Phys., 1974,* **15**, *299*
25. **Thomson J. F., Shanks S. P. and Wu J. C.** *AIAA J. 1974,* **12**, *787*
26. **Launder B. E. and Spalding D. B.** Mathematical Models of Turbulence. *Academic, London, 1972*
27. **Reynolds W. C.** *Adv. Chem. Energy, 1974,* **9**, *193*
28. **Reynolds W. C. and Cebeci T.** Calculation of Turbulence Flows. *Ch. 5 in Turbulence (ed. P. Bradshaw) Springer-Verlag, 1976*
29. **Launder B. E. and Spalding D. B.** *Comp. Meth. in Appl. Mech. Eng., 1974,* **3**, *269*
30. **Rodi, W.** Turbulence Models and their application in hydraulics. *IAHR, 1980*
31. **Launder B. E., Reece G. and Rodi W.** *J. Fluid Mech., 1975,* **68**, *537*
32. **Lumley J. L.** *Adv. Appl. Mech., 1978,* **18**, *123*
33. Boussinesq J. Essai sur la théorie des eaux courantes. *Mém. Acad. Sci., 1877,* **23**, *1*
34. **Prandtl L.** *ZAMM, 1925,* **5**, *136*
35. **Taylor G. I.** *Phil Trans. Roy. Soc. CCXV, 1915,* 1
36. **Rotta J. C.** *Zeitsch. für Physik, 1951,* **129**, *547*
37. **Naot D., Shavit A. and Wolfshtein M.** *Israel J. Technology, 1970,* **8**, *259*
38. **Rodi, W.** The Prediction of free boundary layers by use of a two-equation model of turbulence. *PhD Thesis University of London, Faculty of Engineering, 1972*
39. **Launder B. E. and Morse A. P.** Numerical prediction of axisymmetric free shear flows with a Reynolds stress closure. *Turbulent Shear Flows-1 (eds F. J. Durst, B. E. Launder, F. W. Schmidt, J. H. Whitelaw) 279–294, Springer-Verlag, 1979*
40. **Chung M. and Adrian R.** *Proc. 2nd Turbulent Shear Flow Symposium, London, p. 10–43, 1979*
41. **Sindir M.** Numerical study of turbulent flows in backward step geometries: comparison of four models of turbulence. *PhD Thesis, College of Engineering, Univ. Calif. Davis, 1982*
42. **Cler A.** Modélisation de la turbulence dans le cadre des fermetures en un point. Calcul d'échelles de temps et de longueur. *Thèse Docteur Ingenieur, Ecole Nationale Superieure de l'Aeronautique et de l'Espace, Toulouse, 1982*
43. **Rodi W.** *ZAMM, 1976,* **56**, *219*
44. **Shir C. C.** *J. Atmos Sci., 1973,* **30**, *1327*
45. **Gibson M. M. and Launder B. E.** *J. Fluid Mech., 1978,* **86**, *491*
46. **Daly B. J.** *J. Fluid Mech., 1974,* **64**, *129*
47. **Launder B. E.** Heat and Mass Transport. *Ch. 6 in Turbulence (ed. P. Bradshaw) Springer-Verlag, 1976*
48. **Bradshaw P.** *Proc. Stanford-AFOSR-HTTM Symposium on Complex Turbulent Flows Vol. I, p. 86 (ed. Kline S. J., Cantwell B. and Lilley G.) 1981*

49. **Hancock P. E.** Effect of free-stream turbulence on turbulent boundary layers. *PhD Thesis, Faculty of Engineering, University of London, 1980*

50. **Launder B. E., Leschziner M. A. and Sindir M.** The UMIST-UCD Computations for the AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows. *Ref TFD/81/1 Thermodynamics and Fluid Mechanics Div. UMIST, 1982*

51. **Charnay G., Comte-Bellot G. and Mathieu J. C.** Development of a turbulent boundary layer on a flat plate in an external turbulent flow. *AGARD Conf. Proc. 1971,* 93

52. **Priddin C. H.** Private Communication, 1972. See also: The behaviour of the turbulent boundary layer on curved porous walls. *PhD Thesis, Faculty of Engineering, University of London, 1975*

53. **Turner A. B.** *J. Mech. Eng. Sci., 1971,* 13

54. **Jones W. P. and Launder B. E.** *Int. J. Heat & Mass Transfer, 1973,* **15**, *301*

55. **Gillis J. C. and Johnston J. P.** Turbulent boundary layer on a convex curved surface. *Rep HMT-31. Mech. Eng. Dept. Stanford University, 1980*

56. **Shabaka I. M. M. A.** Turbulent flow in an idealised wing-body junction. *PhD Thesis, Faculty of Engineering, University of London, 1979*

57. **Rodi W., Celik I., Demuren A. O., Scheurer G., Shirani E., Leschziner M. A. and Rastogi A. K.** Calculations for the 1980–81 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows. *SFB 80 Rep T/199, University of Karlsruhe, 1981*

58. **Hah C. and Lakshminarayana B.** *J. Fluids Eng., 1980,* **102**, *462*

59. **Bergeles G., Gosman A. D. and Launder B. E.** *Numerical Heat Transfer, 1978,* **1**, *217*

60. **Bergeles G., Gosman A. D. and Launder B. E.** *ASME. J. Eng. Power, 1980,* **102**, *498*

61. **Bergles G., Gosman A. D. and Launder B. E.** *ASME. J. Heat Transfer, 1981,* **103**, *141*

62. **Ramsey J. W. and Goldstein R. J.** Interaction of a heated jet with a deflecting stream. *NASA-CR 72613, 1970*

63. **White A.** The prediction of the flow and heat transfer in the vicinity of a jet in cross-flow. *ASME Winter Annual Meeting Nov. 1980*

64. **Howard, J. H. G., Patankar S. V. and Bordynuik R. M.** *J. Fluids Eng., 1980,* **102**, *456*

65. **Amano R.** *PhD Thesis, College of Engineering, Univ. California Davis, 1981*

66. **Chang C. S. M., Han T. and Humphrey J. A. C.** Prediction of Case 512 for the 1981 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows. *Mech. Eng. Dept. Univ. California, Berkeley, 1982*

67. **Leonard B. P.** *Comp. Meth. Appl. Mech. Engng., 1979,* **19**, *59*

68. **Han T., Humphrey J. A. C. and Launder B. E.** *Comp. Meth. Appl. Mech. Engng., 1981,* **29**, *81*

69. Report of Working Party on Turbulence Modelling (B. Haines, Chairman). *Aeronautical Journal, Aug./Sept. 1982, 269–277*

70. **Crawford M. E., Kays W. M. and Moffat R. J.** *J. Eng. Power, 1980,* **102**, *1006*

71. **de Vahl Davis G. and Jones I. P.** Natural Convection in a square cavity—a comparison exercise. *Numerical Methods in Thermal Problems–II (eds. R. W. Lewis, et al.), 552, Pineridge Press, Swansea, 1981*

72. **Reynolds B., Lakshminarayana B. and Ravindranath A.** *AIAAJ, 1979,* **17**, *959*

73. **Bradshaw P. and Love E. M.** *Aero. Res. Counc. R & M 3205, 1959*

74. **Humphrey J. A. C.** Flows in ducts with curvature and roughness. *PhD Thesis, Faculty of Engineering, University of London, 1977*

75. **Stenning A. H.** Stall Propagation in a Cascade of Airfoils. *Journal of the Aeronautical Sciences, 1954*

76. **Stenning A H., Kriebel A. R. and Montgomery S.** Stall Propagation in Axial Compressers. *NACA TN-3580, 1956*

77. **Stenning A. H. and Kreibel A. R.** Stall Propagation in Cascades of Airfoils. *ASME Transactions, 1958, pp. 779–790*

78. **Stenning A. H., Seidel B. S. and Senoo Y.** The Effect of Cascade Parameters on Rotating Stall. *NASA TM 3-16-59W, 1959*

Alan Hugh Stenning

Alan Stenning graduated with first class honours from Glasgow University in 1950, after a brilliant undergraduate career. He then worked for Rolls-Royce at Derby for a short time before going on to Massachusetts Institute of Technology with a fellowship in 1950. MIT was to have a great influence on his subsequent research on small turbines in the Dynamic Analysis and Control Laboratory. Later he worked in the Gas Turbine Laboratory and a remarkable series of analyses and experiments on rotating stall established him in the front rank of the young mechanical engineers working in the U.S. in the late fifties.

He was then attracted to developments in nuclear engineering and moved to the Nuclear Engineering Department at MIT, with a brief spell in industry. In 1960 he joined Northern Research and Engineering as technical director, using his knowledge of turbomachinery design in a wide variety of consultancy projects.

During subsequent appointments at the University of Miami and Lehigh University, Stenning returned to his major interest of unsteady flow in heat transfer and in turbomachinery. He brought together all his work on inlet distortion and rotating stall effects in compressors in two superb lectures given to the ASME Turbomachinery course at Iowa State in 1973 shortly before he died. This work has now been published by the ASME, in whose journals he placed much of his work.

Stenning's great strength was in his ability to bring a brilliant mind to bear on a complex problem, no matter what new field it involved, and to solve that problem with analyses of simplicity and beauty. In Alan Stenning, Glasgow University and MIT together produced a distinguished mechanical engineer, one of the best of his generation.

J. H. Horlock

Stenning's contribution to turbomachinery fluid mechanics were summarised in the First Stenning Memorial Lecture delivered in April 1975 and published in December 1976:

Horlock J. H. Alan Hugh Stenning, 1928–73, Distinguished Engineer. Trans. ASME, Series I (J. Fluids Engineering) **98, 4**, p 607