Computational fluid dynamics for turbomachinery design

J D Denton* and **W N Dawes**
Whittle Laboratory, Department of Engineering, University of Cambridge, UK

Abstract: Computational fluid dynamics (CFD) probably plays a greater part in the aerodynamic design of turbomachinery than it does in any other engineering application. For many years the design of a modern turbine or compressor has been unthinkable without the help of CFD and this dependence has increased as more of the flow becomes amenable to numerical prediction. The benefits of CFD range from shorter design cycles to better performance and reduced costs and weight. This paper presents a review of the main CFD methods in use, discusses their advantages and limitations and points out where further developments are required. The paper is concerned with the application of CFD and does not describe the numerical methods or turbulence modelling in any detail.

Keywords: computational fluid dynamics, turbomachinery, turbines, compressors, three-dimensional flow

NOTATION

p static pressure
u flow velocity
S deterministic stress

ρ fluid density
θ circumferential distance

Subscripts and superscripts

1   before mixing
2   after mixing
$^{-}$ area average
$^{\sim}$ mixed out value

1 INTRODUCTION

The application of numerical methods to turbomachinery dates back to the 1940s, in fact methods were even formulated before the advent of the digital computers that were necessary to implement them. The definition of blade-to-blade (S1) and hub-to-tip (S2) stream surfaces was introduced by Wu [1] and this viewpoint domi-

nated the subject until the early 1980s when fully three-dimensional (3D) methods first became available. Wu's S1/S2 approach was far ahead of his time in that he saw it as a method of solution for fully 3D flow. Although such an approach is perfectly valid it is seldom implemented, even today, because of its complexity. In practice, Wu's ideas were considerably simplified by assuming that the S1 stream surfaces were surfaces of revolution (i.e. untwisted) while the S2 stream surfaces were reduced to a single mean stream surface that could be treated as an axisymmetric flow (Fig. 1). It must be recognized that this is a model of real 3D flow; the accuracy of the model (as distinct from that of the numerical method) can only be established by experience and comparison with test data.

The axisymmetric hub-to-tip (S2) calculation is often called the 'throughflow calculation' and has become the backbone of turbomachinery design, while the 'blade-to-blade' (S1) calculation remains the basis for defining the detailed blade shape. The numerical methods used and the practicalities of implementing calculations on both of these families of stream surfaces will be discussed in Sections 2 and 3.

Fully 3D methods replace the stream surface calculations by a single calculation for the whole blade row. This removes the modelling assumptions of the quasi-three-dimensional (Q3D) approach but requires far greater computer power and so was not usable as a routine design tool until the late 1980s. For similar reasons, early methods had to use coarser grids that introduced larger numerical errors than in the Q3D approach. The
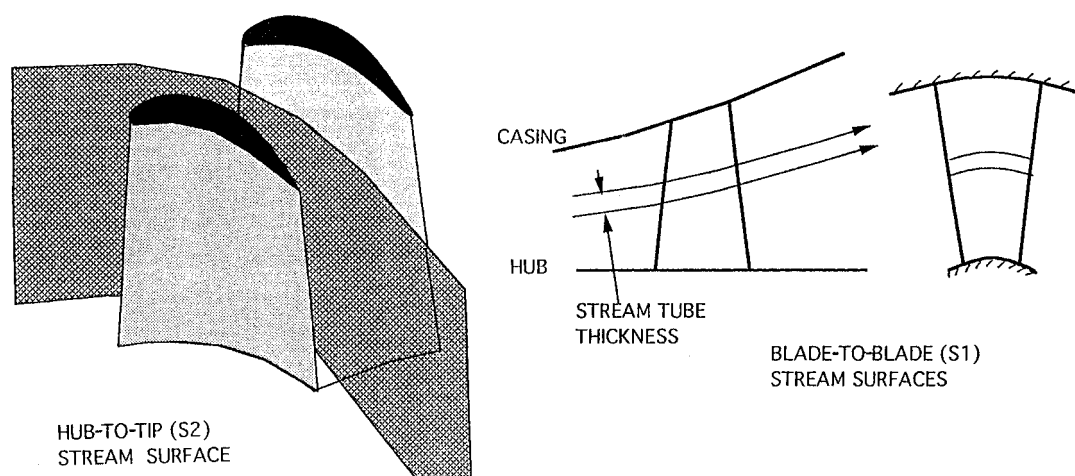
Fig. 1 Hub-to-tip and blade-to-blade stream surfaces

methods used for 3D calculations and their limitations are discussed in Section 4.

The main modelling limitation for 3D calculations on single blade rows arises from the boundary conditions applied at inlet and exit which have to be obtained from a throughflow calculation. This limitation has recently been overcome by the advent of multistage 3D calculations which solve for a whole stage or for the whole machine with boundary conditions only being applied at inlet and outlet, exactly as in a throughflow calculation. However, because the real interaction between blade rows is unsteady (owing to the relative rotation) while most current calculations are steady, some modelling of the unsteady interaction is necessary. This should not be thought of as a disadvantage of multistage 3D calculations; it is merely that it is now possible to contemplate some approximate modelling of the unsteady interaction which was not even thought about in previous methods. Methods of doing this are the subject of active development and are discussed in Section 5.

All modelling limitations (except turbulence) are removed if the 3D unsteady flow through multiple blade rows can be calculated. This is just becoming possible but requires too much computer power to be usable for routine design work. Two-dimensional unsteady calculations or 3D calculations for a limited number of blade passages are more affordable and are mainly used for estimating the unsteady forces on blades. In the foreseeable future it will be possible to calculate the 3D unsteady flow through the complete annulus of a whole machine with many blade rows. The accuracy of such calculations will, however, be limited by the limitations of turbulence and transition modelling for many years to come, if not for ever.

As the flow through the primary flow path becomes more and more predictable, that in secondary flow paths, such as tip leakage flows, cavity flows, cooling flows and bleed flows, becomes increasingly important. Calculation of such flows and their interaction with the primary flow

requires the ability to handle very complex geometries and is best suited to unstructured grids. This type of method is discussed in Section 6.

2 THROUGHFLOW CALCULATIONS

Throughflow calculations remain the most important tool of the turbomachinery designer. At the very outset of the design process, after the annulus shape and mean blade angles have been determined by a one-dimensional calculation, the throughflow calculation will be used to obtain the spanwise variations in flow angle at inlet and outlet to the blade rows. It may be used in the design (or inverse) mode, in which the spanwise variation in blade work is specified and the resulting velocity distribution and blade inlet and exit angles are predicted, or in the direct (or analysis) mode, when the blade exit angles are specified and the velocity distribution and blade inlet flow angles are predicted. The analysis mode can also be used for off-design performance prediction, in which case all blade angles are assumed to be known.

A review of throughflow calculation methods, covering both theory and application, was carried out by Hirsch and Denton in 1981 [2]. Although this review is 16 years old, little has changed in the meanwhile. The streamline curvature method remains the dominant numerical scheme because of its simplicity and ability to cope with mixed subsonic–supersonic flow. The only alternative numerical method commonly used is the stream function method [3]. This effectively solves the same equations as the streamline curvature method but uses an axisymmetric stream function rather than solving directly for the primary variables. In principle this simplifies the numerics because the continuity equation is satisfied via the boundary conditions of the specified stream function on the hub and casing rather than by an iteration. However, the stream function method encounters severe problems when the flow becomes tran-

sonic because there are then two possible velocity distributions for any one stream function distribution and it is not obvious whether to take the subsonic or the supersonic solution. Because of this the stream function method is not widely used for compressible flow machines.

The natural formulation of the throughflow problem is to obtain the solution with a specified mass flowrate. This is clearly not possible in the analysis mode if the machine is choked because there is then only one possible mass flowrate which is not known a priori. Even when unchoked, the extremely rapid variation in exit pressure with mass flow can cause instabilities in high pressure ratio machines. This can be overcome by working in the inverse mode, where the blade flow angles and hence throat areas are allowed to vary during the calculation, but this is not possible for off-design calculations. An extension of the streamline curvature method that enables it to be used in the analysis mode with a specified pressure ratio rather than a specified mass flow, and also to calculate multiple choked blade rows, is described by Denton [4]. This extension is necessary for many turbine applications.

Time-dependent solutions of the Euler equations on an S2 stream surface, as pioneered by Spurr [5], have not been widely adopted. Although the approach has

attractions in working with a specified pressure ratio rather than a specified mass flow and in automatically predicting choking and shock waves, it has limitations in that all shock waves are calculated as normal shocks and so the associated loss may be overpredicted.

In many applications, throughflow calculations are little more than vehicles for inclusion of empiricism in the form of loss, deviation and blockage correlations and their accuracy is determined by the accuracy of the correlations rather than that of the numerics. This is especially true of the deviation and blockage in compressors and of the secondary flow deviation in turbines. If experimental values of loss and deviation are input to the calculation it is usually found that the spanwise variation in velocity is predicted almost exactly. However, if empirical correlations are used the results can be badly wrong. Figure 2 compares predictions and experiment for a turbine when both correlations and experimental data are used to predict the exit flow angles. The calculation with experimental blade exit angles is almost exact even though correlations were still used to predict the loss. The authors' experience is that correlations for the spanwise variation in loss and deviation are only applicable to a very limited range of machines and so should be treated with great suspicion.

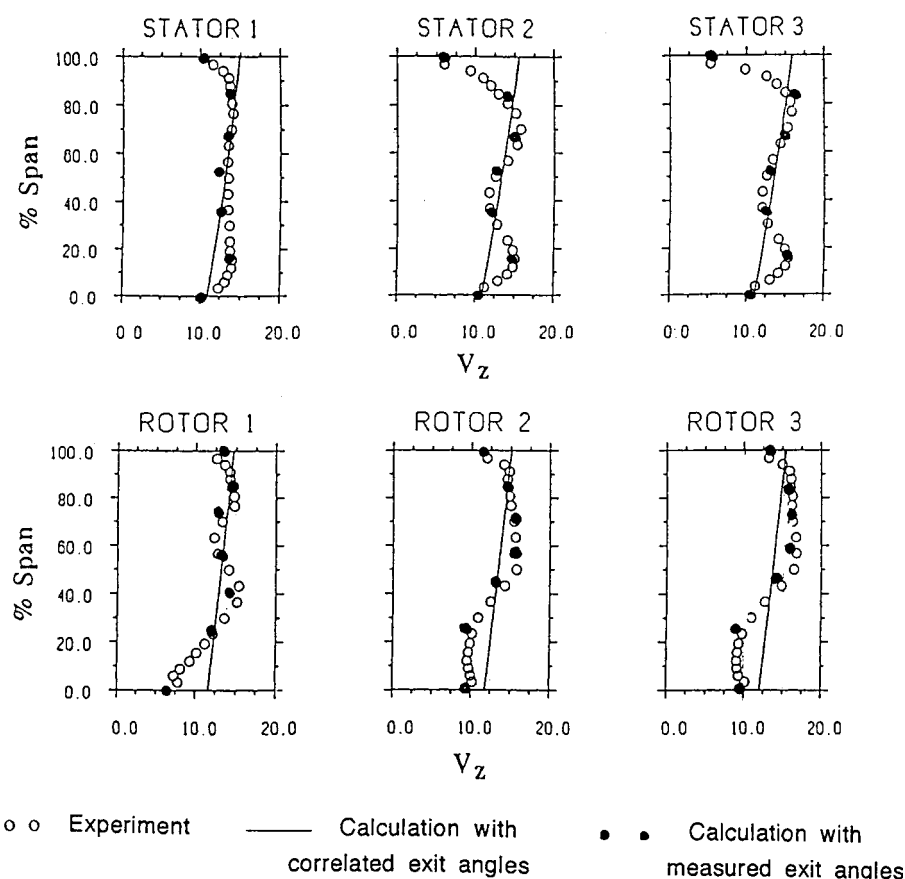The use of blockage factors in compressor calculations

Fig. 2 Development of the axial velocity profile through a three-stage turbine

is only necessary because most throughflow methods do not predict the development of annulus boundary layers. These occur because of skin friction on the annulus walls and increased losses near the blade ends. If these are modelled in the calculation then the increased work done on the low-velocity fluid in the boundary layers causes its temperature to increase unrealistically. This must be prevented by modelling the mixing that occurs between adjacent streamlines in the real machine. Figure 3 shows the improved predictions that can be obtained when mixing is included. Mixing occurs because of both turbulent diffusion and secondary flows. It was originally modelled by Adkins and Smith [6] using secondary flow theory and subsequently by Gallimore and Cumpsty [7] using turbulent diffusion. More recently Leylek and Wisler [8] have shown that both are important in compressors; Lewis [9] has developed a similar model for turbines. The authors' feeling is that any theoretical modelling of mixing is so oversimplified that it is preferable and simpler to include an empirical rate of exchange of mass, enthalpy, angular momentum and entropy between streamlines.

Perhaps the most sophisticated development of the throughflow approach to date is the 'viscous throughflow' developed by Gallimore [10]. In this the entropy generation on the endwalls is modelled together with the losses, secondary deviations and tip leakage flows near the ends of the blades. Mixing between streamlines is used to prevent accumulation of entropy and enthalpy near the endwalls. The modelling relies entirely on empiricism and so is only applicable to compressors similar to those for which it was developed. It does, however, predict the endwall effects and overall performance of such machines very well.

One weakness of most throughflow calculations is their inability to predict the recirculating flows that occur well off-design, i.e. in regions where the meridional velocity is negative. For the streamline curvature method this limitation is inherent because the calculation marches through the machine from inlet to outlet and assumes that enthalpy and entropy convect along the streamlines in the direction of marching. The stream function method also assumes that enthalpy and entropy convect but because it does not march from inlet to outlet the direction of convection is not important and the method can be made to predict reverse flows. An example of such a method is given by Petrovic and Riess [11]. However, it is clear that no method can predict reverse flow at the downstream boundary unless extra boundary conditions are applied.

It is nowadays common to try to include some measure of the effects of 3D flow in throughflow calculations. If the 3D flow is due to spanwise components of the blade force then it can in principle be modelled in a throughflow calculation. An example is the effect of blade lean on the mean stream surfaces. Figure 4 shows a comparison of a full 3D solution and a throughflow solution for a turbine stage with a strongly leaned stator. To obtain reasonable agreement between the methods it was necessary to use three calculating stations within the stator row and two within the rotor in the throughflow calculation. If, however, the 3D flow is due to stream surface twist, then it cannot be modelled by a throughflow calculation with only a single mean hub-to-tip stream surface. An example of this type of flow is that due to blade sweep. In principle this may be calculated by a full S1/S2 iterative solution with multiple S2 stream surfaces, but this is seldom done.
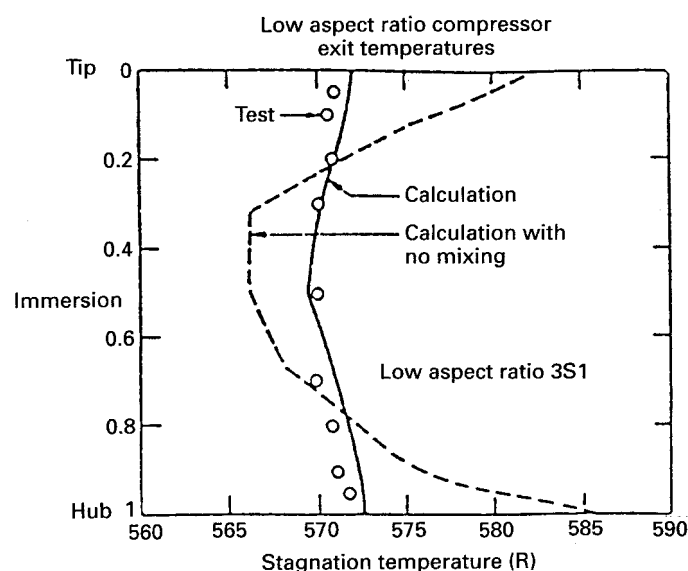

Fig. 3   Throughflow predictions for an axial compressor with and without spanwise mixing. (From Adkins and Smith [6])
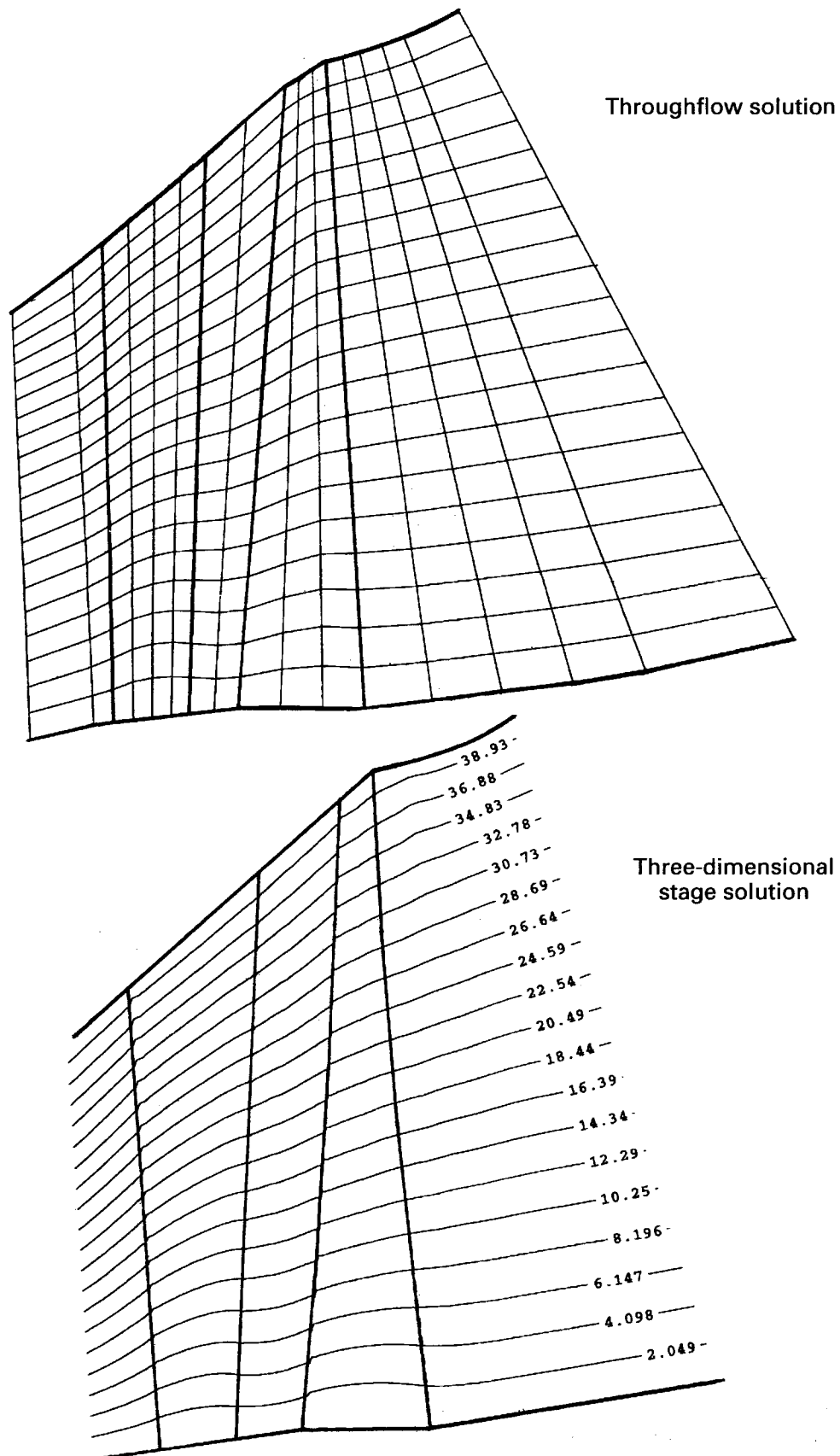
Throughflow solution

Three-dimensional stage solution

38.93
36.88
34.83
32.78
30.73
28.69
26.64
24.59
22.54
20.49
18.44
16.39
14.34
12.29
10.25
8.196
6.147
4.098
2.049

Fig. 4 Computed streamline pattern through a steam turbine last stage with a leaned stator

3   QUASI-3D BLADE-TO-BLADE FLOW

These methods calculate the flow on a blade-to-blade (S1) stream surface given the stream surface shape, with the objective of designing the detailed blade profile. The stream surface is best thought of as a stream tube (Fig. 1) with an associated stream surface thickness and radius which are obtained from the throughflow calculation. Accurate specification of the radius and thickness variation is essential as they can have a dominant effect on the blade surface pressure distribution. The variation in stream surface thickness also significantly affects the boundary layer growth [12] and must be included in any boundary layer calculation. As with throughflow methods the calculation may be in either direct (or analysis) mode, when the blade shape is prescribed and its surface pressure distribution calculated, or in inverse mode, where the required blade surface pressure distribution is prescribed and a blade shape is sought. In the former case blade design proceeds iteratively by adjusting the blade shape until the required pressure distribution is obtained, the changes on each iteration being chosen by the designer. In the latter case the changes in blade shape are produced by the computer but constraints must be applied, e.g. on blade thickness, to ensure that a mechanically acceptable blade is produced. In both cases the required blade surface pressure distribution must be chosen by the designer.

Many different numerical methods have been developed for this task. Initially streamline curvature and stream function methods were popular, but both have difficulty coping with transonic flow and they have now largely been abandoned. Velocity potential methods can be made to calculate transonic flows with weak shock waves but they have seen limited use in turbomachinery. An exception is the finite element velocity potential method of Whitehead [13] which has been widely used in both the direct and inverse mode. Direct solutions of the Euler equations were initially introduced to calculate transonic flows, where their shock-capturing ability compensated for their increased computational cost and limited accuracy. However, as methods and computers have improved, they are now widely used for all types of flow. A hybrid streamline curvature Euler solver developed by Giles and Drela [14] is now proving very popular owing to its ability to work with a coupled boundary layer calculation and to work in both direct and inverse mode.

The numerical methods described above are inviscid and need to be coupled to a boundary layer calculation if they are to be used to predict blade loss. For compressor blades the boundary layer blockage must be included in the inviscid calculation as it significantly affects the blade surface pressure distribution [15]. For most turbine blades the boundary layer is so thin that it may be calculated separately after obtaining the surface pressure distribution from an inviscid calculation.

A recent alternative to coupled inviscid/boundary layer calculations is the direct solution of the Navier–Stokes (N–S) equations which predict the boundary layer growth as part of the main calculation. These demand a much finer grid near to the blade surfaces than do inviscid calculations and so are considerably more 'expensive'. Nevertheless the N–S equations for blade-to-blade flow are now routinely solved as part of the design process, requiring only a few minutes CPU time on a modern workstation. There remains controversy about the best turbulence and transition models to use and about how many mesh points are necessary within the boundary layers. The Baldwin–Lomax turbulence mixing length model is by far the most popular and even the k–ε model, widely used in other applications, has been relatively little used in turbomachinery. The authors believe that no one model is applicable to all types of flow and so at present there is little incentive to use any but the simplest models with empirical tuning where necessary. The number of mesh points needed may be considerably reduced by using wall functions to calculate the skin friction. With these, remarkably good solutions may be obtained with only about six grid points in the boundary layer.

There is much discussion of the optimum type of grid which should be used for blade-to-blade Euler or N–S calculations [16]. The more orthogonal the grid the smaller will be the numerical errors due to the discretization and so the fewer grid points should be needed. However, no one type of grid is ideal for all types of blade row and the generation of the more sophisticated grids may require considerable human intervention. The authors' experience is that it is nowadays easily possible to use sufficient grid points (say 5000 per blade passage) to ensure negligible numerical errors even when using the simplest H grid. Hence, there is now little incentive to use the more complex grids which may take longer to generate (especially more man time) than it takes to solve the problem with a simple grid. Figure 5 shows results for the flow around a turbine blade leading edge (the most difficult region to predict accurately) obtained with the simple H mesh illustrated. About 10 grid points are located on the leading edge circle and the stagnation point is clearly visible. Although there is no analytical solution for such flows, the excellent stagnation pressure conservation shows that numerical errors are very small. For this type of flow there is little reason to use the unstructured grids that are useful for more complex geometries since an unstructured solver will inevitably be slower than a structured one.

Whatever type of grid is used there remains a potential problem at the blade trailing edge where the Kutta condition must be applied. The real flow at a blade trailing edge is always unsteady with regular vortex shedding [17] and a low average pressure acting on the trailing edge. This base drag contributes a significant part of the loss. It is found that with a coarse grid the Kutta condition does not need to be applied explicitly and the
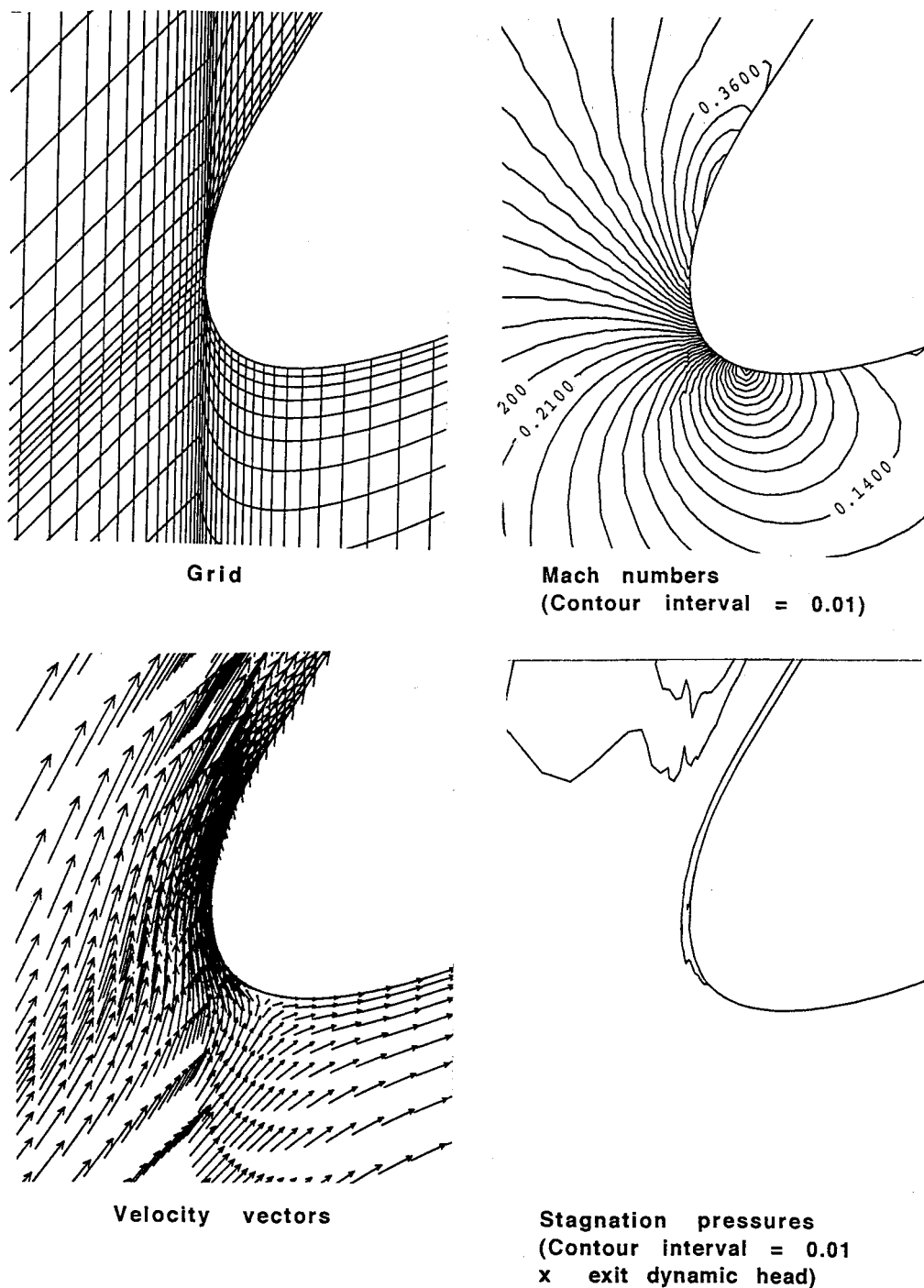
Grid

Mach numbers
(Contour interval = 0.01)

Velocity vectors

Stagnation pressures
(Contour interval = 0.01
x exit dynamic head)

Fig. 5 Flow around the leading edge of a highly loaded turbine blade

numerical viscosity in the solution will automatically make the flow leave the trailing edge smoothly. However, with a fine mesh around the trailing edge the flow tries to approach a stagnation point on the trailing edge circle and the point where the Kutta condition should be applied is not well defined. It is then usually found that the predicted flow accelerates to high velocities at the start of the trailing edge circle on both blade surfaces and the trailing edge loading is highly negative. This occurs even with steady N–S solvers which should allow the flow to separate before accelerating but in practice seldom seem to do so. This type of trailing edge flow, with velocity spikes and negative loading, is never found in practice and the numerical solution may be misleading with the wrong exit flow angle and high trailing edge loss. Both authors strongly recommend that, unless it is hoped to capture the unsteady vortex shedding, a cusp should be placed at the trailing edge of the blade so that the flow leaves the blade smoothly. The cusp should be unloaded, so that it introduces blockage but exerts no
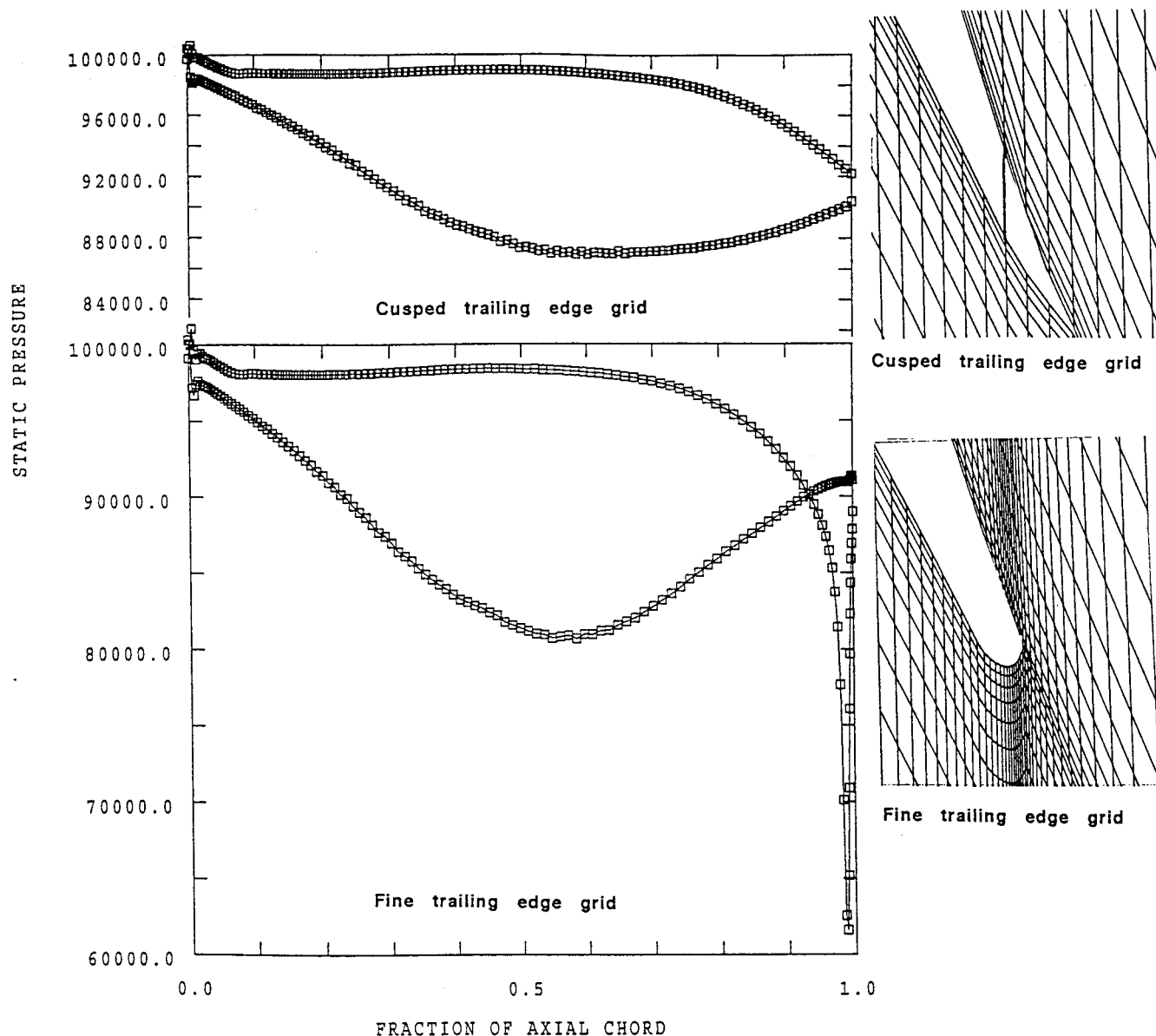
Fig. 6 Effect of a cusped trailing edge and a refined trailing edge grid on the solution for a turbine blade

tangential force on the flow. Figure 6 shows N–S solutions for a turbine blade with a fine grid around the trailing edge circle and with a relatively coarse grid with a cusp. The predicted blade exit angles differed by 5° (63.6 and 68.8°) and the loss coefficient for the fine grid solution (6.3 per cent) was almost double that with the cusped trailing edge (3.4 per cent). There is no doubt that the latter solution is the more realistic.

It is often asked how accurately loss can be predicted by N–S solutions. This depends greatly on the type of blade and the Reynolds number. At high Reynolds numbers, where boundary layers are fully turbulent, and with a thin trailing edge, the loss is largely determined by the skin friction and so should be accurate, say, to ±10 per cent. However, boundary layer transition is more difficult to predict and at transitional Reynolds numbers the loss may be in error by 50 per cent. Similarly, in many

turbine blades one-third of the total loss arises from the low base pressure acting on the trailing edge [18]. This low pressure arises from the vortex shedding and cannot be predicted by a steady calculation, giving perhaps a 20 per cent error in loss prediction. Despite their inability to predict the loss with high accuracy, N–S solvers should be able to predict the trends in loss variation with design changes and so can point the way to more efficient designs.

4 THREE-DIMENSIONAL SINGLE BLADE ROW CALCULATIONS

Many flow features in turbomachinery are fully three-dimensional and cannot be predicted by the Q3D approach; examples are the effects of blade lean and

sweep, of tip leakage and secondary flows. Fully 3D solutions are essential to predict these. Such methods have evolved over the past 20 years and are now routinely used in design. They are most commonly used as a final check on a design produced by the Q3D model but in some cases, such as transonic fans and last stage steam turbine blades, where the flow is dominated by 3D effects, they are now the primary design tool. Denton [19] strongly advocates that they should be used in this way for all types of blading because, in addition to its improved accuracy, the approach avoids the tedious iteration between S1 and S2 stream surfaces. It is much more difficult to apply inverse methods in 3D because the blade loading is influenced by both the blade profile and the blade stacking. A few inverse 3D methods have been reported [20] but they are not yet widely used in industry.

Only a limited number of numerical schemes have been applied to 3D turbomachinery flow. By far the most common are time marching solutions of the Euler or N–S equations [21, 22]. Many different solution algorithms have been used, but nowadays the most common is the four-step Runge–Kutta scheme with variable time steps and a multigrid used to speed up the convergence. Implicit solutions of the time dependent equations have not yet proved competitive with these simpler explicit methods. Pressure correction methods have been less frequently used with the exception of the methods by Moore and Moore [23], Hah [24] and more recently the commercial code TASCflow.

The same numerical schemes tend to be used for both inviscid and viscous flows but many more mesh points and high levels of grid refinement are necessary for viscous calculations. In the absence of complicating features such as shock waves, secondary flows or tip leakage, an accurate inviscid solution for a single blade row requires about 40 000 grid points. Such a solution takes about half an hour on a modern workstation and so can be used routinely as part of the design process. A viscous calculation with shock waves and tip leakage requires about 300 000 points to achieve a grid independent solution, although useful comparisons can be made with only 100 000 points. The use of wall functions, rather than a zero slip condition, to reduce the number of grid points needed is even more advantageous in 3D than in Q3D calculations. The body force model developed by Denton [25] enables N–S solutions to be obtained with negligible increase in CPU time per grid point compared with Euler solutions. Freezing the viscous terms and only recalculating them every 10 steps has the same effect. A useful study of grid dependence and solution accuracy is contained in the results for the ASME CFD test case of rotor 37, a transonic fan [26].

The choice of grid is more limited in 3D than in Q3D flow as it is difficult to patch together the more complex grids on different spanwise grid surfaces, particularly for highly twisted blades. In practice the simple H mesh is by far the most common one used for 3D calculations. Boundary conditions for 3D calculations must usually be obtained from a throughflow calculation which provides the spanwise stagnation pressure, stagnation temperature and flow angle variations at inlet and the static pressure variation at exit. It is also common to specify the exit pressure at only one spanwise position and obtain its spanwise variation from simple radial equilibrium; however, this assumes that there is no spanwise pressure gradient owing to meridional streamline curvature at the downstream boundary.

A major objective of 3D Euler and N–S solvers is to predict secondary flows due to inlet endwall boundary layers. For highly turning blades it is found that these can be well predicted, even by inviscid calculations, provided the shape and thickness of the inlet boundary layer are known. In most practical applications these are unknown, which is a severe limitation on the ability of single blade row calculations to predict the secondary flow in real machines. Perhaps the best that can be done in a multistage machine is to feed the predicted exit flow distribution back into the inlet boundary conditions and so simulate the repeating stage condition.

However, the 3D effects of blade twist, lean and sweep can be readily predicted and used as part of the design [27]. Tip leakage flows are surprisingly easy to calculate because the leakage flow itself is primarily inviscid and only its rate of mixing with the mainstream flow is dependent on turbulence modelling. Perhaps the most challenging task of 3D viscous solvers is to predict the blade surface and endwall corner separations that frequently occur in compressors. Predictions of these are very dependent on turbulence modelling and on the inlet endwall boundary layers and it is unlikely that current methods can provide accurate results without considerable empiricism. Figure 7 shows results from 12 different blind calculations on rotor 37, illustrating significant differences between the methods. These differences are largely due to differences in predicting the blade deviation, especially near to the hub [28].

5 MULTISTAGE 3D CALCULATIONS

Whenever two adjacent blade rows are in relative rotation there is an unsteady interaction between them. Since unsteady 3D calculations are still too expensive to contemplate as design tools, some method must be developed for predicting the time average of the unsteady flow from a steady calculation. Hence some way of averaging out the unsteady interactions must be applied. Early methods of doing this [29] simply circumferentially averaged the flow leaving one blade row before feeding it into the next row. This averaging is a type of mixing process and it assumes that the mixing out of the non-uniform flow occurs instantaneously at a 'mixing plane' rather than
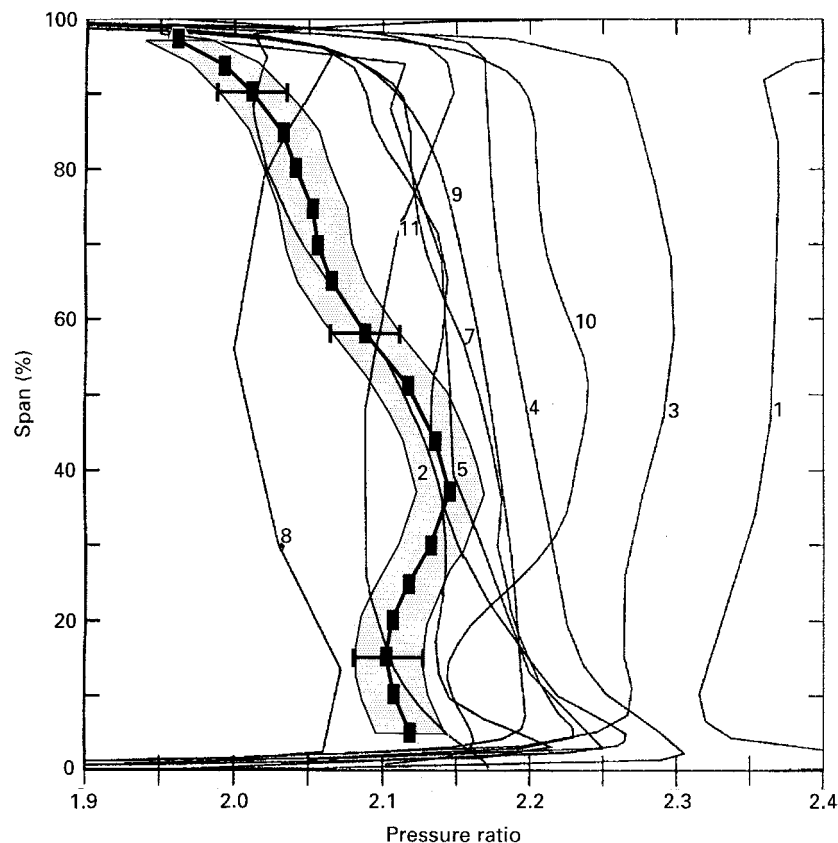
Fig. 7   Radial variation in total pressure downstream of rotor 37: thin lines are different predictions; shaded area is experimental uncertainty

gradually through the downstream blade rows. The mixing generates loss (entropy) and there is no guarantee that the loss generated by instantaneous mixing at a mixing plane will be the same as that generated by the gradual mixing that occurs in practice. The only known study of this approximation is by Fritsch and Giles [**30**] who performed both steady and unsteady calculations on a two-dimensional turbine blade. Figure 8 shows results from their calculations, showing that there was significantly less mixing loss, corresponding to about 10 per cent change in total loss, when the mixing occurred gradually within the rotor.

A simple averaging process also generates errors due to the imposition of a circumferentially uniform flow too close to the trailing and leading edges of the blade rows. These may give incorrect blade loading and false entropy generation. However, more recently, improved mixing plane treatments have been developed [**31**] that do not assume circumferentially uniform flow at the mixing plane and allow the latter to be located very close to the leading or trailing edges of the blade rows. With this model of the flow, calculations for multiple blade rows and even for whole multistage compressors and turbines are now possible. Typically about 70 000 grid points would be used per blade row, enabling up to 12 blade

rows to be calculated on a modern workstation in run times of the order of 24 h. Figure 9 shows the results from such a calculation on a four-stage turbine.

With these developments, modern 3D multistage calculations can be seen as an extension of throughflow calculations with most of the empiricism removed and with the great advantage that the blade-to-blade calculations are obtained simultaneously. With the inclusion of tip leakage they can in principle be used to predict the overall machine performance. It cannot be expected that the predictions of boundary layer growth and secondary flows will be very accurate, given the limited number of grid points, the neglect of unsteady interactions and the limitations of turbulence and transition modelling. However, the authors' experience is that the efficiency of many machines is predicted surprisingly well. The implication is that many loss sources in machines are not dependent on small details of the flow, but rather on the mixing out of relatively large scale non-uniformities. These methods are starting to be used for design and their use can be expected to increase in future.

A more rigorous approach to the mixing problem was taken by Adamcyzk [**32**] who introduced the concept of 'deterministic stresses'. His model attempts to deal
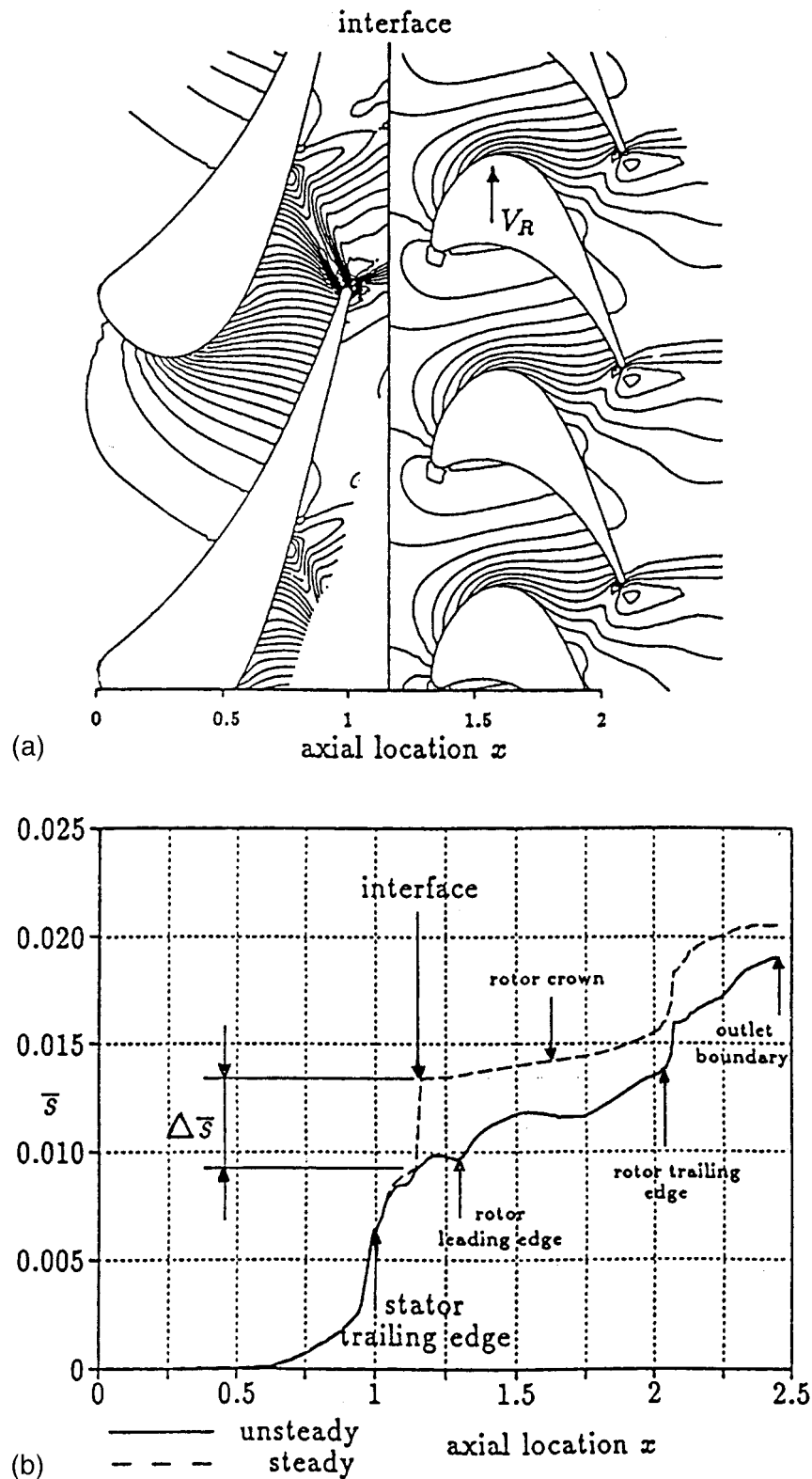
Fig. 8 Computed flow in a transonic turbine stage: (a) static pressure contours of the time averaged flow and (b) axial development of the entropy flux. (From Fritsch and Giles [**30**])

systematically with the averaging of the unsteady flow without giving up the advantage of performing steady flow simulations. To understand the basic principles of his method, consider the two simple examples illustrated

in Fig. 10. The first example (taken from Rhie *et al.* [**33**]) concerns the mixing of a 2D wake, say at mid-span. The classical mixing plane approach performs a mixing calculation at constant area, leading to the following
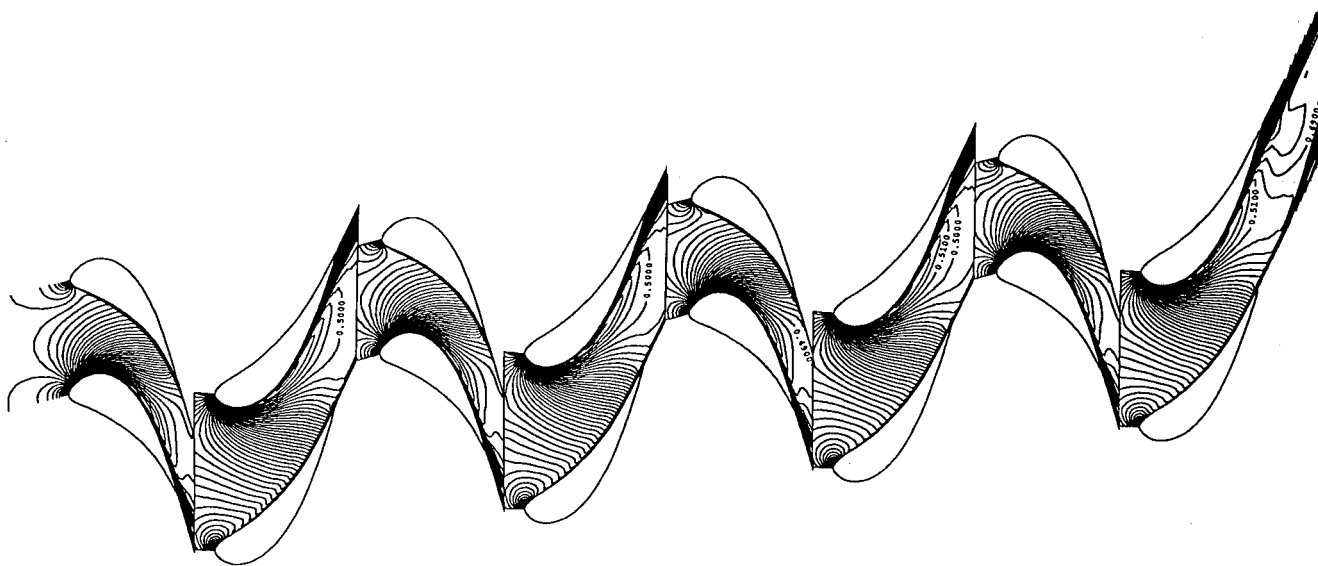
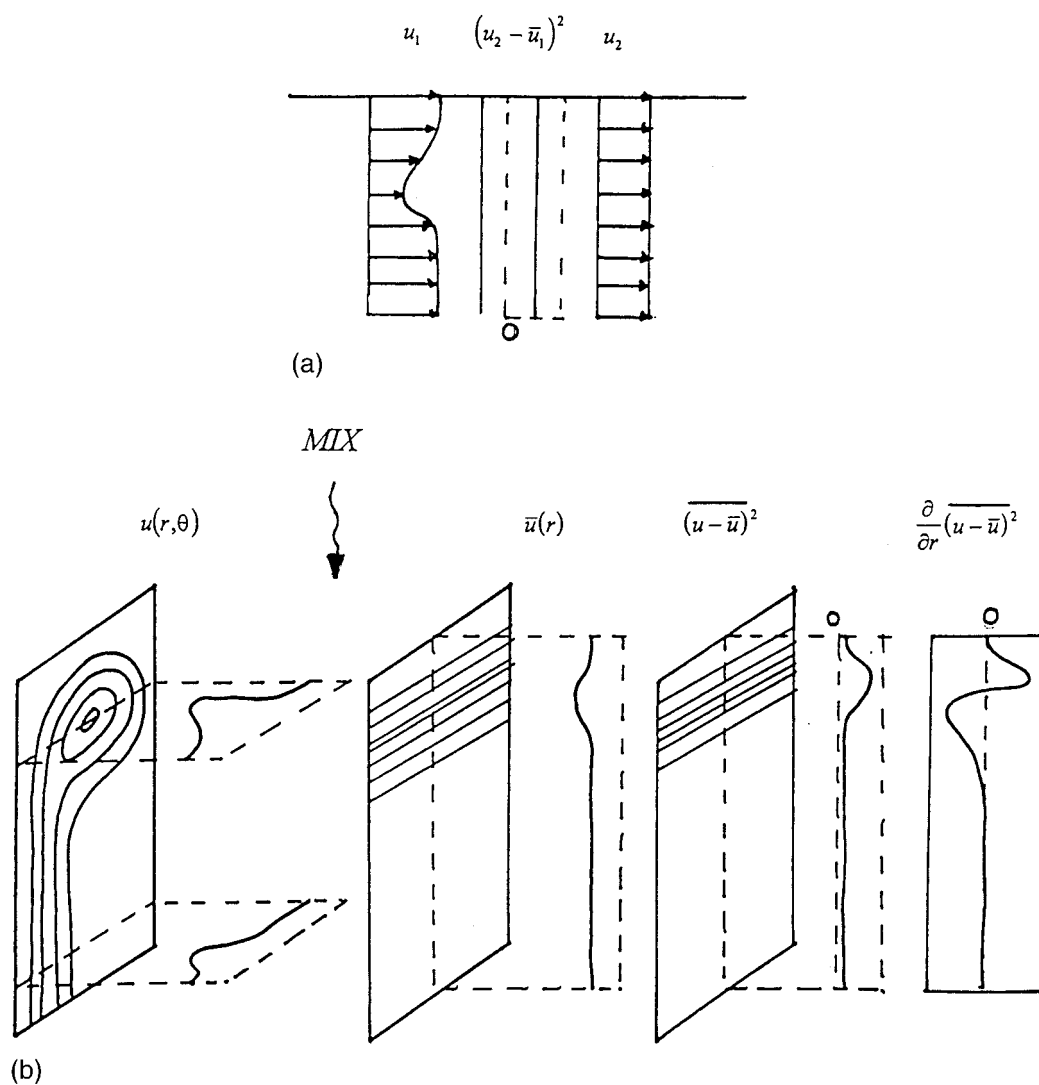**Fig. 9**　Mach number contours from a 3D solution for a four-stage turbine

Fig. 10　(a) Two-dimensional and (b) three-dimensional mixing problems

equations (for incompressible flow):

continuity $\qquad \int \rho u_1 \, \mathrm{d}\theta = \rho \tilde{u}_2$

momentum $\qquad \int (\rho u_1^2 + p_1) \, \mathrm{d}\theta = \rho \tilde{u}_2^2 + \tilde{p}_2$

where the pitch has been taken as unity to ease the algebra, u is the axial velocity and $\sim$ represents the 'mixed-out' average. An area average (which would be a density average for compressible flow) is defined in the conventional way, e.g.

$$\bar{p}_1 = \int p_1 \, \mathrm{d}\theta$$

Thus, the downstream pressure is

$$\tilde{p}_2 = \bar{p}_1 + \int \rho u_1^2 \, \mathrm{d}\theta - \rho \left(\int u_1 \, \mathrm{d}\theta \right)^2$$

Hence, if there is a significant difference between the 'mean square' and the 'square mean' blade exit flow, then there will be a jump in static pressure which may be thought of as being due to the lost wake blockage.

The deterministic stress approach handles the mixing plane via the following averages:

continuity $\qquad \int \rho u_1 \, \mathrm{d}\theta = \rho \bar{u}_2$

momentum $\qquad \int (\rho u_1^2 + p_1) \, \mathrm{d}\theta = \rho \bar{u}_2^2 + \bar{p}_2 + S$

where S is the 'deterministic stress' defined by

$$S = \int \rho u_1^2 \, \mathrm{d}\theta - \rho \left(\int u_1 \, \mathrm{d}\theta \right)^2$$

Now, all of the quantities are area averaged (density averaged for compressible flow) across the mixing plane so that

$$\bar{u}_2 = \bar{u}_1$$

as before, but now the deterministic stress has been chosen so that

$$\bar{p}_2 = \bar{p}_1$$

This does not introduce any false blockage at the mixing plane and preserves the velocity triangles across the mixing plane.

Stress S is equal to the jump in static pressure obtained in the simple approach. In Adamcyzk's model, S is evaluated whenever the flow is non-uniform and is included in the flow equations that are solved. In particular, in the simple example of Fig. 10a, S would decay downstream of the mixing plane, thus allowing mixing to occur gradually. Clearly some model of how S decays in reality is needed, but even a crude model is likely to be much better than instantaneous mixing. Adamcyzk

adopted a rather complex model for calculating the decay of S, involving iteration between solutions on overlapping grids. The present authors do not feel that this level of complexity is justified by the assumptions inherent in the modelling and suggest that much simpler modelling, such as that used by Bolger [34], is all that is justified.

In 3D flow, as illustrated in Fig. 10b, the deterministic stresses can influence spanwise mixing which is of great significance in multistage compressors. To make the point more clearly, write u' as the difference between the local flow and the circumferential average:

$$u' = u(\theta) - \int u \, \mathrm{d}\theta = u - \bar{u}$$

The deterministic stresses can be re-cast as

$$S = \int (u - \bar{u})^2 \, \mathrm{d}\theta = \int (u')^2 \, \mathrm{d}\theta = \overline{u'^2}$$

which is just like a Reynolds stress. The 3D example illustrated in Fig. 10b shows an upstream flow containing axial velocity deficits due to both a wake and a strong vortex (perhaps an overtip leakage vortex or corner stall). The forms of $\bar{u}$ and u' are then indicated together with the resulting distribution over the whole mixing plane of the deterministic stress, $S = \overline{u'^2}$. The blockage contributions are of the form of a background shift and enter via modelling $\partial S / \partial X$. The radial derivatives, $\partial S / \partial r$, are likely to be much more significant. These represent spanwise body forces producing spanwise velocities and mixing. Again, in the absence of unsteady simulation, the axial decay of these $\partial S / \partial r$ terms must be obtained by modelling. Seen from this perspective, the 'deterministic stresses' recall very strongly the classic work of Adkins and Smith [6] who introduced spanwise mixing into throughflow simulations using similar correlated body forces which were obtained from auxiliary secondary flow estimations (Fig. 3 is an example of their work). The work to make axisymmetric throughflow simulations consistent with 3D N–S calculations via so-called 'perturbation terms' (see, for example, Jennions and Stow [35]) is also addressing the same problem via similar techniques.

Rhie *et al.* [33] show that inclusion of these deterministic stresses produces significantly different results to a crude mixing plane in a multistage compressor and that the 'deterministic stress' approach seems closer to experiment. However, it is not clear how much of the improvement could have been obtained by use of a modern mixing plane alone without the deterministic stresses. There is a need for some well post-processed unsteady simulations to study the magnitude of the deterministic stresses and produce models for their decay.
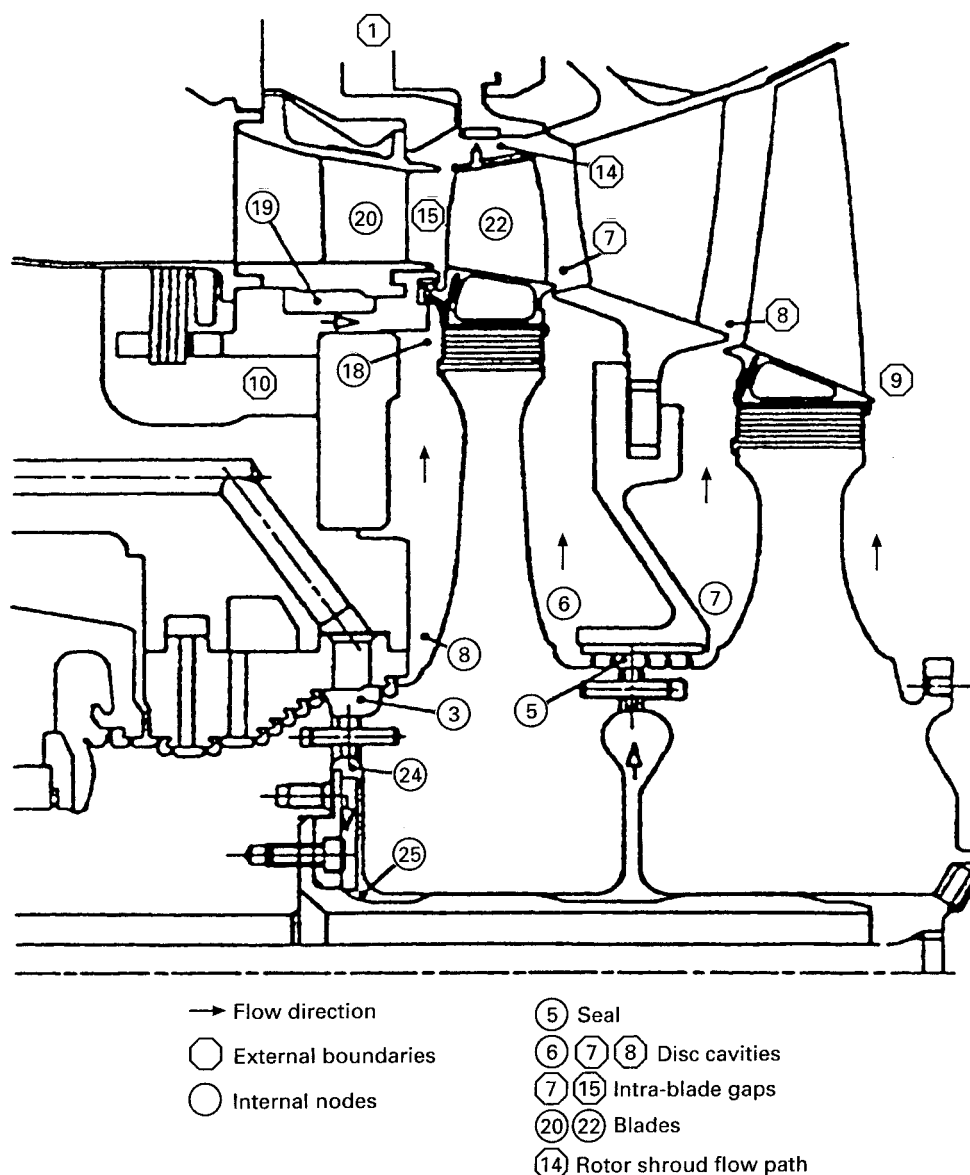
> → Flow direction
> ◯ External boundaries
> ◯ Internal nodes
>
> ⑤ Seal
> ⑥⑦⑧ Disc cavities
> ⑦⑮ Intra-blade gaps
> ⑳㉒ Blades
> ⑭ Rotor shroud flow path

Fig. 11   Internal air flow system for a turbine. (From Hannis *et al.* [**36**])

5.1   Secondary/primary flow path interaction

While the secondary gas path (disc cavity flows, shroud leakage flows, 'hot gas ingress', etc.) has always received attention, it is increasingly being recognized that the interaction between the secondary and primary flows can have a surprising impact on overall aerodynamic performance. Figure 11, adapted from Hannis *et al.* [**36**], shows a typical range of concerns. Traditional work on the turbine secondary air system has treated the system as being essentially decoupled from the primary flow. However, there is now a trend for the two flows to be treated together (e.g. reference [**37**]). The essence of many interactions is that the circumferential gaps in the hub platforms and casing–shroud gaps cannot support the strong blade-to-blade pressure gradients associated with the primary flow. This tends to modify the blade loadings and pressure distributions near the endwalls. Within these gaps, circumferential flow is driven by pressure differences comparable with blade loading levels, producing velocity levels comparable with blade speed. Fluid 'surfs' around the annulus in and out of the gap with a net leakage flow of, say, 1–2 per cent but velocity levels of around 100 per cent of free stream and in doing so generates significant amounts of loss. This is especially severe for compressor blading which typically has much more loading near the leading edge than a turbine blade and so quite dramatic changes in blade loading and loss can take place compared with the design intent [**28**].

Any serious multistage compressor design system must take into account the interaction of the secondary gas path with the primary flow. Figure 12 taken from Le Jambre *et al.* [**38**] shows an attempt to model a bleed
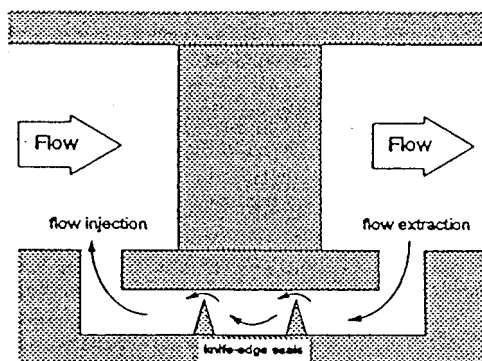
Schematic of cavity model
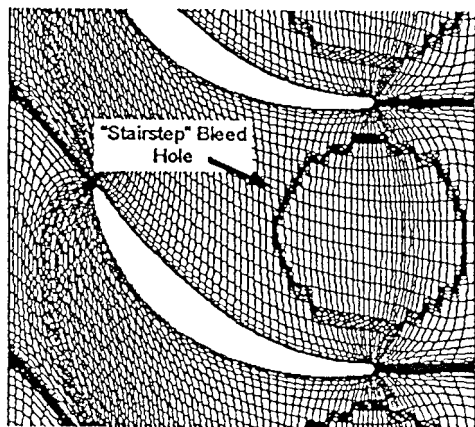
Grid detail showing model of bleed hole

Fig. 12 Modelling of compressor under-hub leakage using a structured mesh. (From Le Jambre *et al.* [38])

hole in a compressor stator row. The inability of such a hole to support axial or circumferential pressure gradients can severely modify the pressure distribution of the blades.

To gain meaningful access to these problems, CFD must be able to simulate both the primary and secondary gas paths simultaneously. This implies that unstructured meshes, or at least multiblock structured meshes, must be deployed and is currently an area of very active research.

The challenge of this area for CFD is more fundamental than turbulence modelling; simply representing the complex geometry and building a mesh now require the ability to import and interpret CAD files. There are a number of areas to resolve here: geometric integrity; automatic assimilation of mesh topology; speed (in wall clock hours) of data transfer from CAD to CFD; speed with which changes to the geometry can be made and fed back to the CAD model to impact on other disciplines like stress analysis. The two meshing approaches that have been successfully utilized to date are unstructured meshes of tetrahedrons or multiblock meshes with

sets of structured meshes. An example of the unstructured approach, taken from Dawes [39], is shown in Fig. 13. This shows a turbine stator together with a model disc cavity system. The secondary air flow and, in particular, the circumferential relief of the blade-to-blade pressure field near the hub contribute to a stronger than expected endwall secondary flow.

5.2 Design optimization

There is continued commercial pressure to produce components of the highest possible quality in the shortest possible time. The increasing sophistication of CFD analysis methods can only be justified if it contributes to relieving this pressure. The ability to perform a CFD simulation does not in itself improve design and performing many costly simulations may actually delay the discovery of an acceptable design!

The classical approach to design is to rely on creative designers who have developed intuition and physical understanding over years of focused activity. CFD is especially good at helping develop this understanding since many more blade designs, ranges of parameters, etc. can be studied than would ever be feasible experimentally—but this still takes time.

The recent response has been to pursue inverse design strategies in which the aerodynamics are specified somehow, perhaps as a blade surface pressure distribution, and the geometry that would deliver this is computed. There have been some notable successes here, especially in the areas of supercritical airfoil design, but only in two dimensions. The key problem, above all in 3D, is to know what to desire as a flow field and the key difficulty is to decide if it is possible to realize the desired flow.

The use of optimization techniques in design seems to represent a practical way forward. Optimization simply seeks to produce a better design than a datum subject to design constraints. This process has been increasingly adopted by the airframe industry. Examples of its use for turbomachinery are the work of Shelton *et al.* [40] for blade-to-blade flow and of Wakely [41] for throughflow.

There is sure to be much more interest in design optimization in turbomachinery in the future, not least because, as design cycle time is driven down by economic pressure, the designer must spend more time understanding the physics of his flows and not waste time simply exercising codes.

6 CONCLUDING REMARKS

The range of CFD methods applied to turbomachinery flows has been described with the emphasis on the practicality and limitations of the techniques. It is very important that a designer using CFD appreciates the underlying
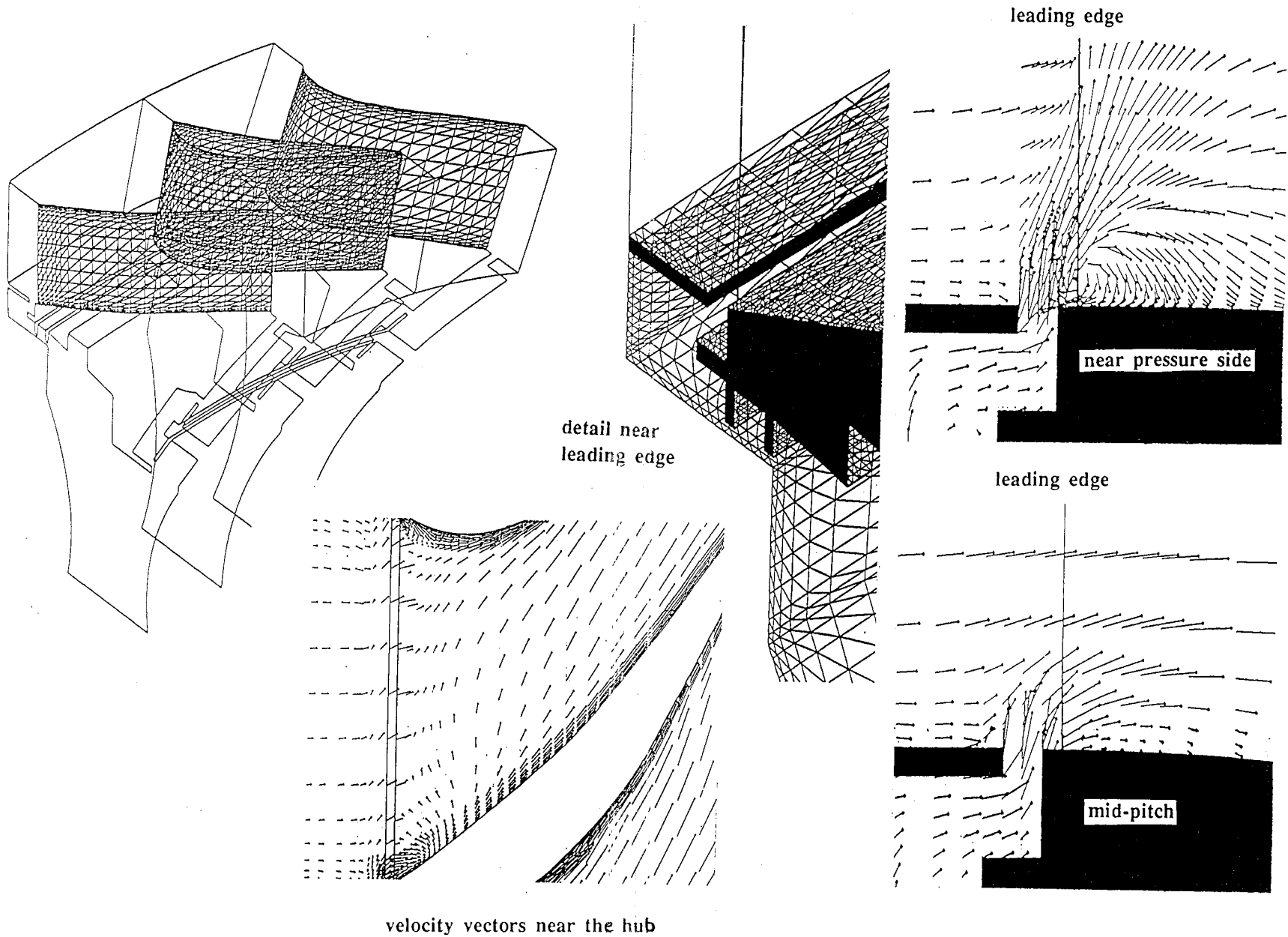
leading edge

near pressure side

leading edge

mid-pitch

detail near
leading edge

velocity vectors near the hub

Fig. 13 Coupled turbine stator–disc cavity flow simulation

assumptions and limitations of the codes he is using and hopefully this paper will help to ensure this.

Many of the techniques are now mature and little further improvement can be expected in future. Others are limited by the accuracy of transition and turbulence modelling and these limitations will not be overcome in the short term.

The following areas offer promise for future work:

(a) blade row interaction,
(b) component interaction,
(c) secondary–primary path interaction,
(d) design optimization,
(e) CAD/CFD design integration.

REFERENCES

 1 **Wu, C. H.** A general through flow theory of fluid flow with subsonic or supersonic velocities in turbomachines of arbitrary hub and casing shapes. NACA paper TN2302, 1951.
 2 **Hirsch, Ch.** and **Denton, J. D.** Throughflow calculations in axial turbomachines. AGARD AR 175, 1981.
 3 **Marsh, H.** A digital computer program for the through flow fluid mechanics in an arbitrary turbomachine using a matrix method. ARC R&M 3509, 1968.
 4 **Denton, J. D.** Throughflow calculations for transonic axial flow turbines. *Trans. ASME, J. Engng for Power*, April 1978, **100**.
 5 **Spurr, A.** The prediction of 3D transonic flow in turbomachinery using a combined throughflow and blade to blade time marching method. *Int. J. Heat and Fluid Flow*, 1980, **2**(4).
 6 **Adkins, G. G.** and **Smith, L. H.** Spanwise mixing in axial flow turbomachines. ASME paper 81-GT-57, 1981.
 7 **Gallimore, S. J.** and **Cumpsty, N. A.** Spanwise mixing in multistage axial flow compressors, Part 1. *Trans. ASME, J. Turbomachinery*, 1986, **108**.
 8 **Leylek, J. H.** and **Wisler, D. C.** Mixing in axial flow compressors. Conclusions drawn from 3D Navier Stokes analysis and experiments. ASME paper 90-GT-352, 1990.
 9 **Lewis, K. L.** Spanwise transport in axial flow turbines, Parts 1 and 2. ASME papers 93-GT-289 and 290, 1993.
10 **Gallimore, S. J.** Viscous throughflow modelling of axial compressor blade rows using a tangential blade force hypothesis. ASME paper 97-GT-415, 1997.
11 **Petrovic, M. V.** and **Riess, R.** Off-design flow analysis and performance prediction of axial turbines. ASME paper 97-GT-55, 1997.
12 **Dunham, J.** The effect of stream surface convergence on turbomachinery blade boundary layers. *Aeronautical J.*, 1974, **178**(758/759).
13 **Whitehead, D. S.** and **Newton, S. G.** Finite element method for the solution of 2D transonic flow in cascades. *Int. J. Numerical Methods in Fluid*, 1985, **5**, 115–132.
14 **Giles, M. B.** and **Drela, M.** Two dimensional transonic aerodynamic design method. *AIAA J.*, 1987, **25**(9).
15 **Calvert, W. J.** and **Ginder, R. B.** Quasi-3D calculation system for the flow within transonic compressor blade rows. ASME paper 85-GT-22, 1985.
16 **Li, H., Chen, S.** and **Martin, H. F.** Two dimensional viscous blade flow code using shifted periodic grids. ASME paper 97-GT-516, 1997.
17 **Cicatelli, G.** and **Sieverding, C. H.** The effect of vortex shedding on the unsteady pressure distribution around the trailing edge of a turbine blade. ASME paper 96-GT-359, 1996.
18 **Roberts, Q.** and **Denton, J. D.** Loss production in the wake of a simulated turbine blade. ASME paper 96-GT-421, 1996.
19 **Denton, J. D.** Designing in three dimensions. AGARD Lecture Series 195, 1994.
20 **Demeulenaere, A.** and **Van den Braembussche, R.** Three dimensional inverse design method for turbomachinery blading design. ASME paper 96-GT-39, 1996.
21 **Denton, J. D.** Calculation of three dimensional flow through any type of turbomachine. AGARD Lecture Series 140, 1985.
22 **Dawes, W. N.** A numerical analysis of three-dimensional viscous flow in a transonic compressor rotor and comparison with experimental data. ASME paper 86-GT-16, 1986.
23 **Moore, J.** and **Moore, J. G.** A calculation procedure for 3D viscous compressible duct flow. Parts 1 and 2. *Trans. ASME, J. Fluid Engng*, December 1979, **101**.
24 **Hah, C.** A Navier–Stokes analysis of 3D turbulent flows inside turbine blade rows. *Trans. ASME, J. Engng for Power*, 1984, **106**, 421–429.
25 **Denton, J. D.** Use of a distributed body force to simulate viscous effects in turbomachinery flow calculations. ASME paper 86-GT-144, 1986.
26 **Denton, J. D.** Lessons from rotor 37. *J. Thermal Sci.*, 1977, **6**(1).
27 **Denton, J. D.** and **Xu, L.** The exploitation of three-dimensional flow in turbomachinery design. *Proc. Instn Mech. Engrs, Part C, Journal of Mechanical Engineering Science*, 1999, **213**(C2), 125–137.
28 **Shabbir, A., Celestina, M. L., Adamczyk, J.J.** and **Strazisar, A. J.** The effect of hub leakage on two high speed axial flow compressor rotors. ASME paper 97-GT-346, 1997.
29 **Denton, J. D.** Extension of the finite volume time marching method to three dimensions. VKI Lecture Series 1979-7, 1979.
30 **Fritsch, G.** and **Giles, M. B.** An asymptotic analysis of mixing loss. ASME paper 93-GT-345, 1993.
31 **Denton, J. D.** Calculation of 3D viscous flow through multistage turbomachines. *Trans. ASME, J. Turbomachinery*, 1992, **114**(1).
32 **Adamczyk, J. J.** Model equation for simulating flows in multistage turbomachinery. ASME paper 85-GT-226, 1985.
33 **Rhie, C. M., Gleixner, A. J., Spear, D., Fischberg, C. J.** and **Zacharias, R. M.** Development and application of a multistage Navier–Stokes solver. Part I: multistage modelling using body forces and deterministic stresses. ASME paper 95-GT-342, 1995.
34 **Bolger, J.** PhD thesis, Cambridge University, 1998 (to be submitted).
35 **Jennions, I. K.** and **Stow, P.** Quasi-three dimensional turbomachinery blade design system—Part I: throughflow analysis. ASME paper 84-GT26, 1984.
36 **Hannis, J. M., Maltson, J. D., Corry, R. J.** and **Johnson, N.** Analysis and validation of turbine disc cooling. ASME paper 96-GT-97, 1996.
37 **Chew, J. W., Green, T.** and **Turner, A. B.** Rim sealing of rotor–stator wheelspaces in the presence of external flows. ASME paper 94-GT-26, 1994.

38 Le Jambre, C. C., Zacharias, R. M., Biederman, B. P., Gleixner, A. J. and **Yetka, C. J.** Development and application of a multistage Navier–Stokes solver. Part II: application to a high pressure compressor design. ASME paper 95-GT-343, 1995.

39 Dawes, W. N. Unsteady flow and loss production in centrifugal and axial compressor stages. AGARD-CP-571, *Loss Mechanisms and Unsteady Flows in Turbomachines*, Derby, 1995.

40 Shelton, M. L., Gregory, B. A., Lawson, S. H., Moses, H. L., Doughty, R. L. and **Kiss, T.** Optimisation of a transonic turbine airfoil using artificial intelligence, CFD and cascade testing. ASME paper 93-GT-161, 1993.

41 Wakely, G. and **Grant, J.** The application of formal optimisation methods to the design of steam turbine reaction bladepaths. IMechE paper S461/008/96, 1996.