INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 9, 921–941 (1989)

# THE DEVELOPMENT OF ADVANCED COMPUTATIONAL METHODS FOR TURBOMACHINERY BLADE DESIGN

P. STOW

*Theoretical Science, Rolls-Royce plc, Derby, U.K.*

## SUMMARY

The paper describes the basic components of a turbomachinery blade design system in use within Rolls-Royce. A number of modelling aspects of the advanced computational methods in use and under development are reviewed together with areas for future research and development.

A quasi-3D blade design system which is used for both compressors and turbines is described covering through-flow and blade-to-blade analysis. Various features of blade-to-blade analysis are discussed including the use of compatible design and analysis modes and coupled boundary layer analysis capable of handling attached and separated flow; examples are included to show capabilities. Advances being made in the development and application of Reynolds-averaged Navier–Stokes models are covered showing capabilities with regard to loss and heat transfer prediction.

A fully coupled quasi-3D through-flow and blade-to-blade analysis system is described and results presented to show basic capabilities.

The need for 3D flow analysis is discussed and the elements of a 3D blade design system presented showing how this links to the traditional quasi-3D system. Examples are given showing basic capabilities of the methods available and under development.

Finally areas for future development are presented indicating the mathematical and numerical modelling problems to be addressed.

KEY WORDS Blade design Compressors Turbines Navier–Stokes Quasi-3D and 3D systems Through-flow

## 1. INTRODUCTION

The development and application of computational fluid dynamics (CFD) in Rolls-Royce is part of a major initiative on computer-aided engineering and manufacture (CAEM) which is seen as a key element in ensuring future competitiveness in the design and development of turbomachinery.

At each stage in the design process of a particular component, sophisticated mathematical models of the physical phenomena are employed (see Figure 1). These models are incorporated into computing systems aimed at ensuring efficient usage and rapid assimilation of results. Individual systems link together through databases to enable rapid interaction of the various aspects of the design process, e.g. aerodynamics, stressing, etc.

CAEM is aimed at reducing both design and development times and costs through more exact modelling at each stage of the design process and by easier, faster interaction between the various stages. CFD is the core of computing systems for all aerodynamic and thermodynamic design and analysis. The emphasis in this paper is on a subset of this activity, the development and application of advanced CFD methods for turbomachinery blade design. The methods are applied in Rolls-Royce within the framework of a blade design system which is described below.

*Received 14 March 1988*

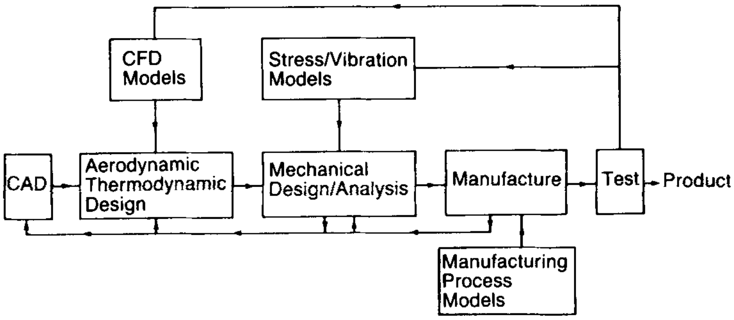


Figure 1. Computer-aided engineering

Traditional turbomachinery blade design systems are based heavily on the work Wu[1] using quasi-3D through-flow and blade-to-blade programs (see Figure 2). In this approach sections of a blade are designed on isolated axisymmetric stream-surfaces using a blade-to-blade program, with information from a through-flow analysis being used to define the geometry of the stream surfaces to be used and the stream tube height variation as well as to provide the inlet and outlet flow conditions to be achieved. The different types of blade-to-blade methods adopted, e.g. design analysis, etc., are discussed in the next section together with the development of models for loss analysis. Once the blade sections have been designed they are stacked radially and circumferentially to produce the three-dimensional blade geometry taking into account aerodynamic, mechanical or blade-cooling constraints.

With such a design system and the models adopted there are a number of phenomena that cannot be accounted for, e.g. stream surface twisting, the three-dimensional nature of the blade surface boundary layers, blade–annulus boundary layer interactions, blade over-tip leakage flows, etc. These features can only be analysed using fully three-dimensional methods; how such methods

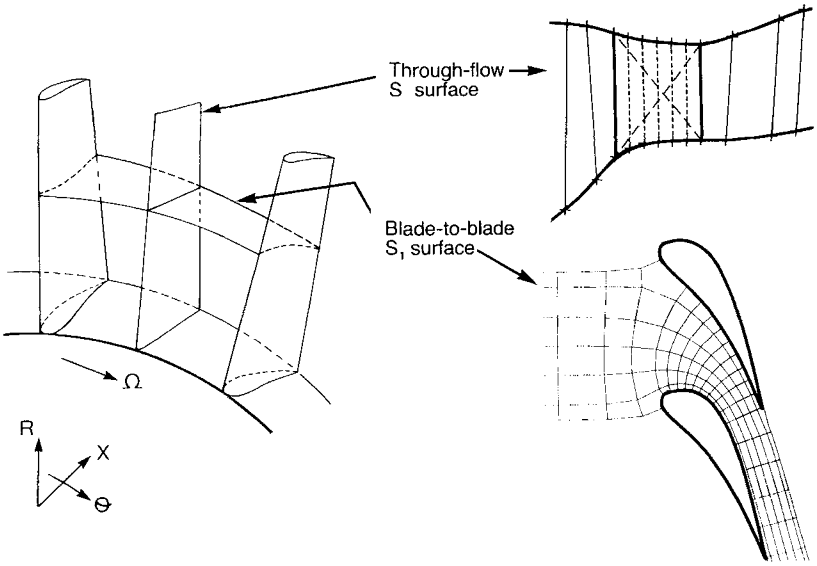


Figure 2. Quasi-3D geometries

are linked to the traditional design system will be discussed later. In some cases, however, valuable information about the effects of features such as the annulus geometry and blade lean can be assessed early in the design process using a linked quasi-3D system prior to full 3D analysis; this will be discussed later.

## 2. QUASI-3D BLADE-TO-BLADE ANALYSIS

In the design process a through-flow analysis gives the necessary inlet and outlet flow conditions for each section of each blade row in order to achieve parameters such as stage or blade pressure ratio or work output. This means that the desired lift of each blade section is known. The design freedom lies with the lift distribution from blade leading to trailing edges, which in turn determines the characteristics of the blade surface boundary layers and the efficiency of the blade. It is important therefore that a designer has methods that enable him to determine the boundary layer characteristics and that allow him to analyse quickly the effects of changes to his design variables.

### *2.1. Inviscid analysis*

Inviscid blade-to-blade methods generally fall into two categories, design and analysis. With the former the desired blade surface velocity is prescribed and the method produces the blade geometry. With the latter the blade geometry is prescribed and the method produces the blade surface velocity distribution. A design method in many ways offers advantages from a pure aerodynamics point of view, where desirable boundary layer development can be reflected in the velocity prescribed, whereas an analysis method is often needed in order to satisfy mechanical or blade-cooling constraints. A method with compatible mixed design and analysis modes combines the best features of both and offers considerable advantages for practical design. Such a method based on a finite element full velocity potential analysis is described by Cedar and Stow.[2] In this the geometry can be prescribed over part of the blade, in which case the method produces the surface velocity, and the velocity can be prescribed over the remainder, the method producing the geometry. In the basic analysis mode the system of non-linear equations arising from the finite element method is solved using a Newton–Raphson procedure which ensures rapid convergence. Simple three-node triangular elements are adopted, which means that the area integrals involved and the Jacobian matrix in the Newton–Raphson procedure can be calculated analytically.

In the design mode, changes to the blade shape are modelled using a surface transpiration technique which avoids mesh reconstruction each time the geometry changes in the iterative design procedure. The transpiration mass flux is related to changes in the blade geometry in the same manner that a transpiration boundary layer is modelled. This is easily included in the finite element method through the boundary conditions. The surface transpiration mass flux is in turn related to desired changes in the blade surface velocity using what is called an influence matrix; this is determined efficiently using a slight adaptation of the Newton–Raphson procedure. The iterative solution is fast to converge, giving rise to a very versatile interactive design tool.

The way of using such a method is to start with an analysis of an initial blade geometry. Loss-producing features can be identified using a coupled boundary layer analysis. A velocity distribution can then be prescribed over part of the blade, aimed at improving boundary layer development, and the blade geometry determined. Figure 3 from Reference 2 shows how the method has been used to remove a shock wave identified from an analysis of a blade whilst maintaining the same lift. Figure 3(a) gives the surface Mach number distribution for the original blade; it also shows the desired design distribution which has been changed only over part of the suction surface. Figure 3(b) shows the original and resulting blade shapes. It can be imagined that

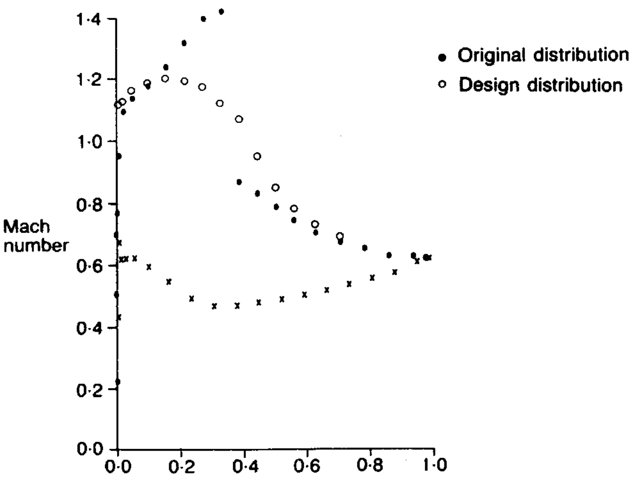


Figure 3(a). Supercritical compressor blade with shock

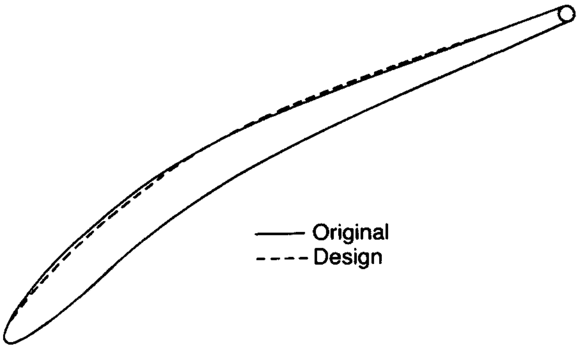


Figure 3(b). Change in blade shape to remove shock

to achieve this result using only an analysis type of approach would have been difficult and time-consuming.

A further application is shown in Figure 4 to remove a Mach number over-speed around the leading edge of a blade in order to improve the boundary layer behaviour at design and off-design conditions.

Similar mixed design and analysis modes have been added to other methods available in the blade design computer system.

### *2.2. Coupled boundary layer analysis*

Boundary layer analysis is an essential element in the design procedure.

The standard approach adopted is to couple the inviscid blade-to-blade analysis with a boundary layer analysis, either finite difference or integral approach; the latter is often preferred because the speed of the analysis enables interactive capabilities of an inviscid method to be maintained.

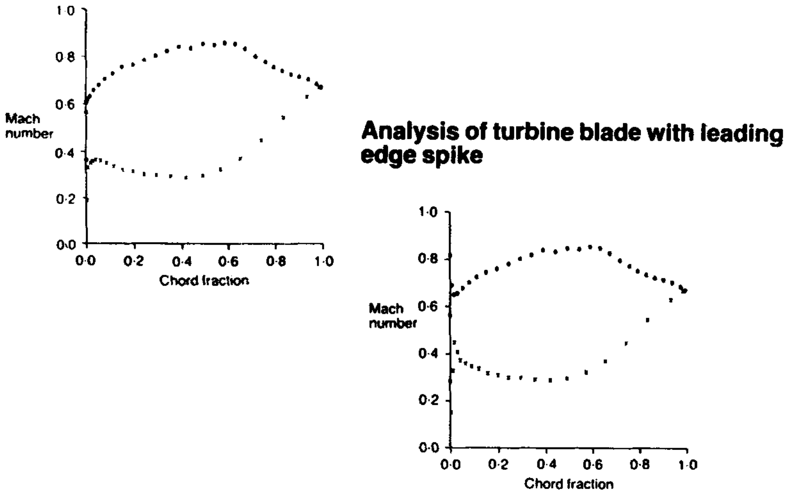


Figure 4(a). Turbine blade with leading edge spike

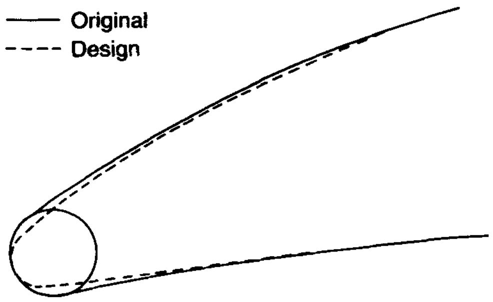


Figure 4(b). Change in blade shape to remove leading edge spike

The effects of the boundary layer on the inviscid mainstream can be modelled using either a displacement model, in which the blade geometry is altered by the boundary layer thickness, or by a surface transpiration model, in which mass, momentum and energy are ejected at the blade surface. The type of model adopted is governed to a large extent by properties of the inviscid method.

In general the boundary layer and inviscid calculations must be iteratively coupled, the type of coupling necessary being determined by the magnitude of the boundary layer effects. In cases where the boundary layer remains attached, direct mode coupling can be adopted in which the surface velocity distribution from the inviscid calculation is used as input to the boundary layer calculations to determine the displacement thickness. This is then used in the inviscid calculation and the procedure repeated; usually under-relaxation is needed to ensure convergence. This procedure has been adopted with all blade-to-blade methods available within the design system, e.g. streamline curvature, velocity potential and Euler methods.

Stow and Newman[3] give details of the approach for the finite element method described in Reference 2 using an integral boundary layer method and surface transpiration model. The integral method adopted handles laminar and turbulent flow, with transition correlations based on the work of Abu Ghannam and Shaw[4] being used to predict the start and end transition and the starting conditions for turbulent flow. Laminar separation bubbles are handled using correlations due to Roberts.[5] The integral method is continued downstream of the trailing edge to calculate the wake development. A near-wake 'jump' model, due to Newman,[6] is adopted in order to calculate starting conditions for the wake calculation from those at the trailing edge. With this model the conservation equations are written in a jump form, enabling the effects of the trailing edge base pressure to be incorporated.

Figure 5 shows predictions for a low-pressure turbine blade tested in cascade by Hodson.[7] Figure 5(a) shows the mesh used, with Figure 5(b) indicating the good agreement of the predicted blade surface Mach number with experiment. Figure 5(c) shows a comparison of the predicted and measured suction surface boundary layer momentum and displacement thickness; it can be seen that good agreement is found. In this example the leading edge velocity over-speed creates a laminar separation bubble, with almost immediate relaminarization after reattachment and later natural transition towards the blade trailing edge.

It is found that in many cases the trailing edge base pressure can have an important influence on the predicted loss and should be included in any calculation. Figure 6 shows the effect for typical turbine and compressor blades; the effect is much larger for the turbine blade because of the larger

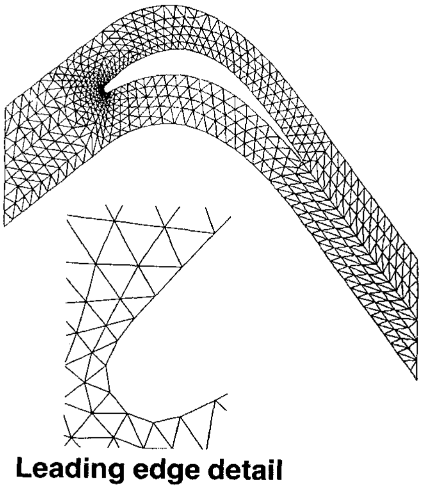


Figure 5(a) LP turbine mesh

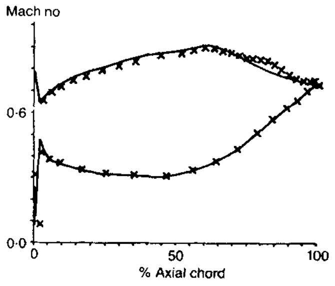


Figure 5(b). Surface Mach number

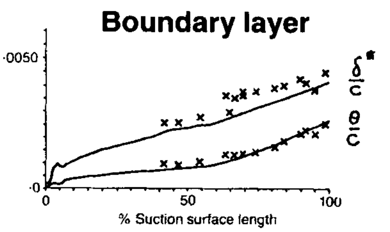


Figure 5(c). Suction surface boundary layer

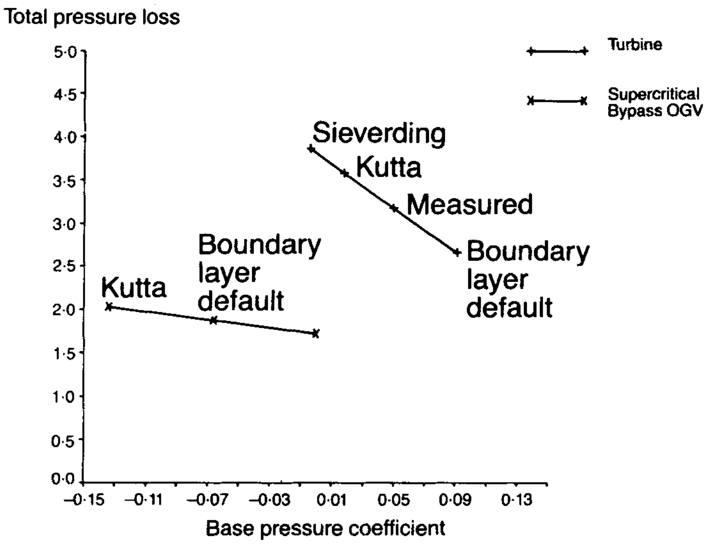


Figure 6. Effects of base pressure on loss

ratio of blade trailing edge thickness to boundary layer momentum thickness. Currently the base pressure must be supplied by a designer or a correlation used.

In cases where the boundary layer approaches separation, it is found that direct mode coupling suffers convergence problems unless heavy under-relaxation is used or an upper limit is placed on the form factor. To handle such cases, an alternative form of coupling has been developed in which the boundary layer equations are used in an inverse form; in this approach the boundary layer displacement thickness is prescribed and the corresponding blade surface velocity derived from the boundary layer equations. A semi-inverse mode of coupling has been developed for the finite element method discussed earlier and is presented in Newman and Stow.[8]

This uses the direct mode of the inviscid calculation together with an inverse mode of the integral boundary layer in regions where separation is expected, the direct mode being adopted elsewhere.

Figure 7 shows the coupling procedure adopted and indicates that a correction technique must be applied to the displacement thickness to ensure that the velocity distributions from the inviscid and boundary layer calculations agree. The inviscid influence matrix, mentioned earlier in connection with the design mode, is used in this procedure. A similar boundary layer influence matrix can be determined numerically by perturbing the boundary layer equations and relating changes in velocity to those in displacement thickness. With the semi-inverse approach described, rapid convergence is found and the resulting program can be used in a completely interactive manner. Figure 8 shows results for a compressor blade with a large suction surface diffusion; it can be seen that the predictions of the suction surface boundary layer parameters are in good agreement with experiment. The main limitations with such an approach lie not with problems in the coupling procedure but with limitations of the boundary layer method in handling flows with large turbulent separations.

A similar approach to that discussed above but adopting a full inverse coupling technique in regions of turbulent separation has been developed by Calvert[9] and is available in the blade design system. This is based on the same boundary layer method but adopts Denton's[10] time-marching Euler method.

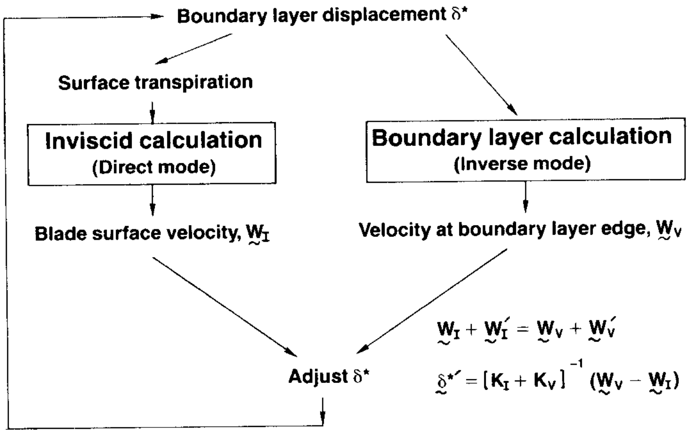


Figure 7. Semi-inverse boundary layer coupling

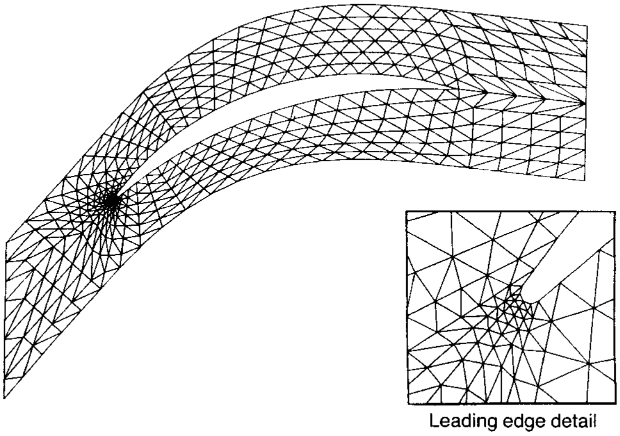


Figure 8(a). V2 compressor cascade

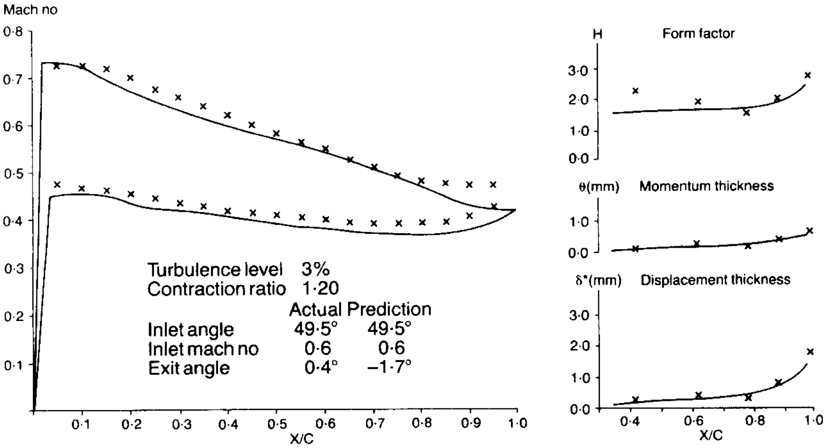


Figure 8(b). Mach number and boundary layer details

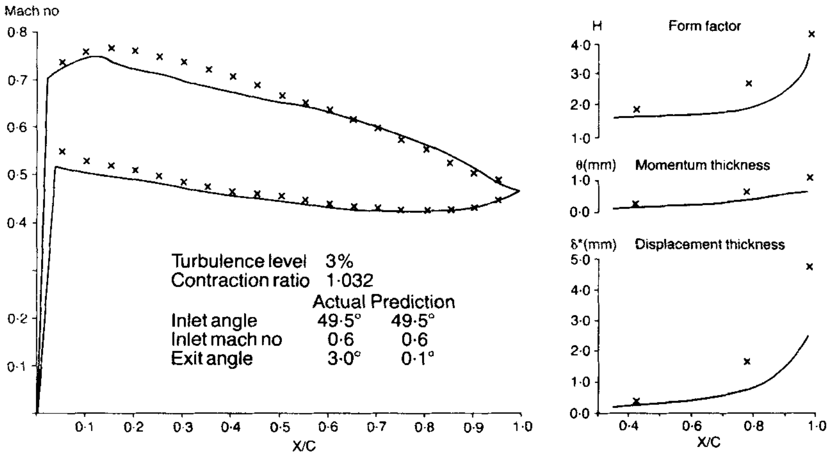


Figure 8(c). Mach number and boundary layer details

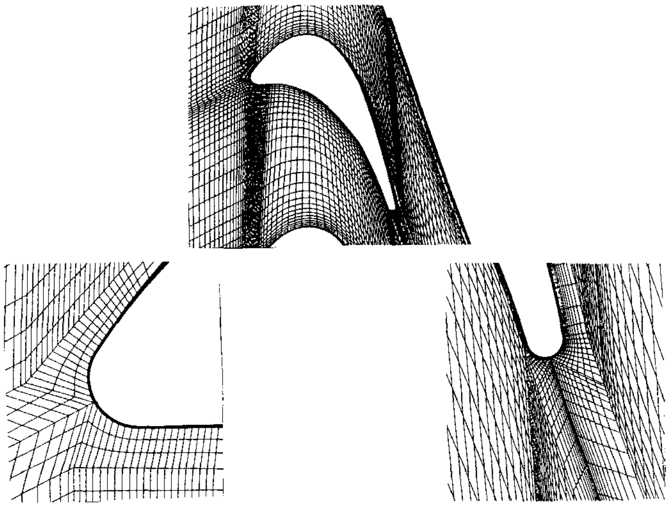
### 2.3. *Reynolds-averaged Navier–Stokes methods*

As was mentioned above, the main limitations with the coupled inviscid–boundary layer approach lie in limitations of the boundary layer method. The areas requiring improvement at the current time are concerned with separation and transition, e.g. modelling laminar separation bubbles, transitionally separated flow and turbulent separation. It is believed that improvements of both integral and finite difference techniques are possible and developments are in progress to improve these aspects of modelling. The other approach being pursued is the development of methods for solving the Reynolds-averaged Navier–Stokes equations. The main interest lies with loss and surface heat transfer prediction at both design and off-design conditions; consequently turbulence and transition modelling are important.

Two main techniques are being developed, pressure correction[11] and time-marching. With regard to the latter, both cell-centred schemes, attributed to Jameson *et al.*,[12] and cell-node-based schemes, attributed to Ni,[13] are under development; see Norton *et al.*[14] and Carrahar and Kingston[15] for more details. In addition, various time-marching strategies for advancing the solution in time are available and being developed, e.g. a fully coupled implicit scheme in two dimensions and explicit Runge–Kutta and two-step Ni-type Lax–Wendroff in two and three dimensions. Grid systems with the various codes range from a simple H-grid to a more elaborate embedded O–C–H grid; Figure 9 shows the latter grid for a high-pressure turbine.

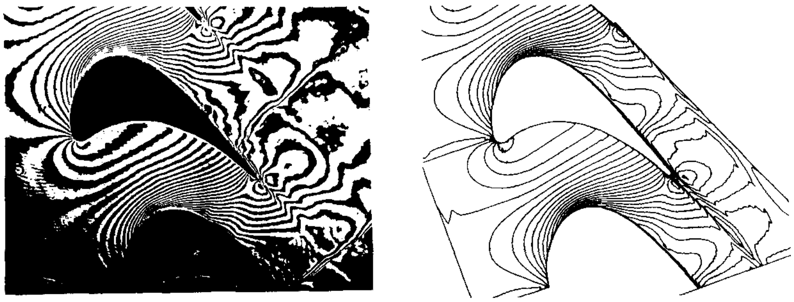
Figures 10 and 11 show typical results taken from Connell[16] using the fully implicit cell-centred time-marching code described by Norton *et al.* About 7000 points are used; it can be seen that good overall agreement exists with the holographic experimental data, although some smearing of the trailing edge shock is evident in the case with the higher exit Mach number.

Currently only relatively simple turbulence models are adopted, e.g. a Baldwin–Lomax mixing-length model or one-equation kinetic energy mixing-length model.[17] Figure 12 shows results from Connell[18] using an implicit time-marching method with the mixing-length model; similar results are found using the kinetic energy model. It can be seen that good agreement exists between predicted isentropic 'surface' Mach number and that measured by Nicholson *et al.*;[19] it should be mentioned that a similar level of agreement is found using current inviscid methods with a Kutta model, indicating that the boundary layer has little effect on the surface Mach number. Figure 13 shows the good agreement found in the surface heat transfer. It should be pointed out that the



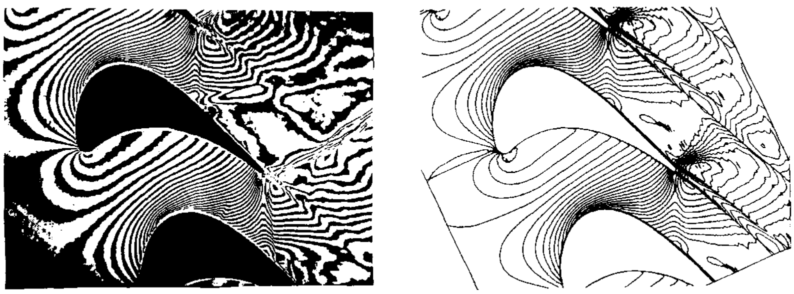
Leading edge Trailing edge

Figure 9. Turbine grid system



Experiment Theory

Figure 10. Comparison of experimental and computational iso-density contours at exit Mach number 0.989



Experiment Theory

Figure 11. Comparison of experimental and computational iso-density contours at exit Mach number 1.191

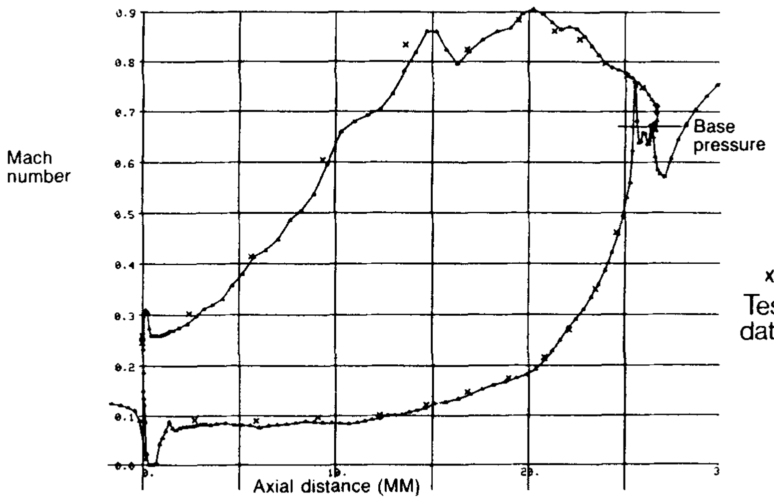


Figure 12. Blade surface Mach number

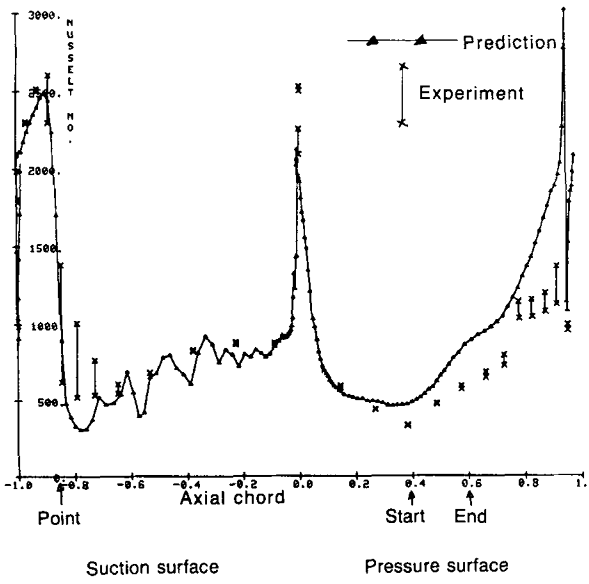


Figure 13. Surface heat transfer results

spatial variations on the suction surface of this blade can be directly correlated with the curvature of the blade and in no way are a consequence of any spatial instability. It should also be mentioned that in the predictions the start and end of transition were specified as indicated in the figure.

Transition is modelled using an intermittency function with various options for determining the region over which it is applied. As was indicated above, one option is to specify the start and end of transition. An alternative is to predict the start and end using either a correlation, as discussed earlier in the integral boundary layer approach, or the kinetic energy model developed by Birch.[17] Figure 14 shows results from Birch for the high-pressure turbine blade shown in Figure 12 but at a

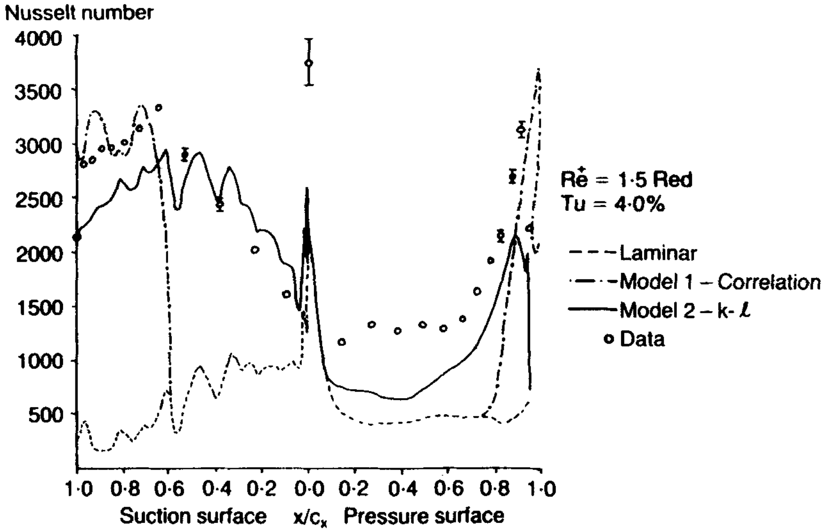


Figure 14. Surface heat transfer results

higher Reynolds number than in Figure 13. It can be seen that fairly good agreement exists for the kinetic energy model on the suction surface, where the correlation model predicts the start of transition to be too late. The pressure surface results indicate that although transition starts at about 60% axial chord, there is some effect of freestream turbulence on the laminar boundary layer. This is accounted for to some extent in the kinetic energy model but totally neglected in the correlation model. At the low freestream turbulence level tested it is found that the correlation model performs better than indicated in Figure 14 and the kinetic energy model less well, indicating that although encouraging results are being produced, further development of the models is necessary.

Transition and turbulence modelling are important in determining the quality of profile loss prediction, i.e. loss up to the blade trailing edge. Modelling of the trailing edge base flow region is important in determining the quality of the overall loss including the base pressure and mixing effects. In many cases it is found that the flow in the trailing edge region is unsteady. Using a constant time step in the implicit method discussed earlier, it has been found that very realistic results can be produced in terms of the unsteady periodic flow structure around and downstream of the trailing edge and that the shedding frequency agrees closely with that expected. However, this is a costly procedure to adopt, especially in three-dimensional flow and especially with conditionally stable explicit time-marching methods. In addition, it is the mean flow effects, loss, etc. that are of main interest, at least at this stage. As a consequence, mechanisms for producing the mean flow effects are being considered. It has been found that the unsteady nature of the trailing edge flow can be suppressed by adding smoothing or numerical viscous effects in that region. Care must be taken, however, so as not to generate significant additional, spurious loss using such a technique. Figure 15, taken from Connell,[16] shows predictions using this technique for the high-pressure turbine blade shown in Figure 10 and 11; it can be seen that fairly good agreement in the overall mixed out loss is found with experiment. Also shown in the figure is the predicted profile loss, indicating the importance of the trailing edge region, at least near sonic conditions, in determining the overall loss characteristics of this profile. It should be mentioned that a similar level of agreement is found using the coupled inviscid–boundary layer technique if the measured

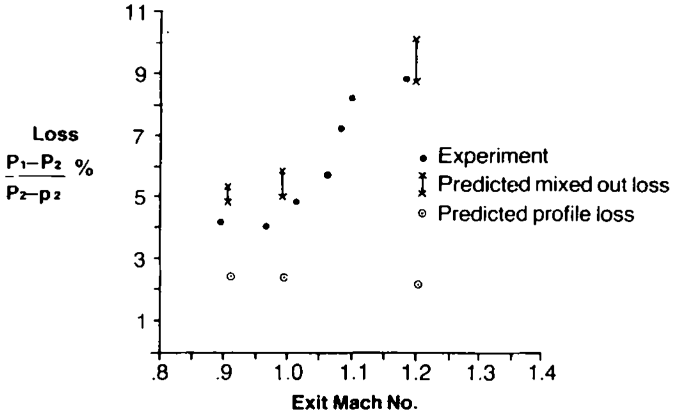


Figure 15. Loss prediction against exit Mach number

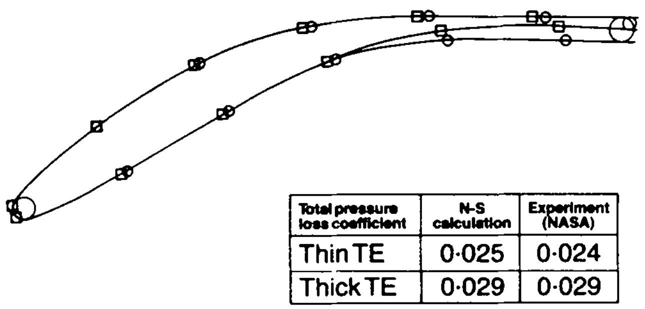
base pressure is input into the mixing–loss calculation. The difference with the Navier–Stokes approach is that no such input data are required.

Further studies have been completed on the effects of trailing edge thickness on the overall loss indicating that the approach adopted agrees very well with the experimental findings in both magnitude and trends, see Figure 16, from Stafford and Birch.[20]

It is clear from the developments and applications being undertaken that Reynolds-averaged Navier–Stokes methods offer great promise in terms of loss and heat transfer prediction, understanding of flow phenomena and modelling opportunities. The methods are, however, much slower than current design methods adopting coupled techniques and consequently tend to used more as research tools, at least in two dimensions, the situation being different in three-dimensional analysis. It is clear, however, that developments in solution algorithms, convergence techniques and the application of supercomputers will reduce elapsed time sufficiently to ensure more widespread application within design systems.

## 3. THROUGH-FLOW ANALYSIS

Through-flow analysis is a crucial element in the overall blade design procedure by virtue of it being used to determine the inlet and exit conditions to be achieved by each blade row. It is also important in the quasi-3D approach to blade design in that it is used to determine the details of the



| Total pressure loss coefficient | N-S calculation | Experiment (NASA) |
|---|---|---|
| Thin TE | 0·025 | 0·024 |
| Thick TE | 0·029 | 0·029 |

Figure 16. Compressor blade with thick/thin trailing edge

stream surfaces and stream tube heights to be used in designing sections of a blade. The stream surface radius and especially the stream tube height variation have an important effect in the blade-to-blade analysis and consequently in the performance of a blade section geometry. In turn these quantities are affected by the blade, e.g. the thickness, axial and circumferential stacking of the sections, and by the annulus geometry. As a consequence a linked through-flow blade-to-blade analysis is often necessary as part of the design procedure. It can also provide valuable information and understanding of the effects of variables such as annulus profiling, blade stack, etc. early in the design process prior to a full three-dimensional analysis.

Such a coupled through-flow analysis has been presented by Jennions and Stow.[21] This uses a passage-averaging technique in which the through-flow equations, within the blade row, are integrated from one blade to the next to give equations in terms of passage-averaged quantities. The resulting through-flow equations include the effects of the blade turning, blockage, axial and circumferential blade leans, blade profile losses and circumferential non-uniformities across the passage.

Figure 17 (a) shows results from Jennions and Stow[22] for the high-pressure turbine nozzle guide vane shown in Figure 17 (b). It can be seen that good agreement exists with the experimental data. The linked analysis performed was inviscid with no account of loss in the through-flow and adopting a streamline curvature blade-to-blade program; convergence of the procedure was obtained in seven iterations. Further results from the system are shown in Jennions and Stow[22] for a different stack of the blade sections in Figure 17(b). Again good agreement with experiment was found. It should be mentioned that the secondary flow deviations, i.e. the flow overturning and under turning associated with passage vortices, are not predicted with such a coupled system unless some specific and separate account is taken of the secondary flow, e.g. inclusion of secondary losses.

The coupled analysis discussed has been extended to include the various blade-to-blade programs available within the design system so that analysis of fans and subsonic and transonic compressors and turbines is possible. The fan analysis system, involving the coupling of the blade-

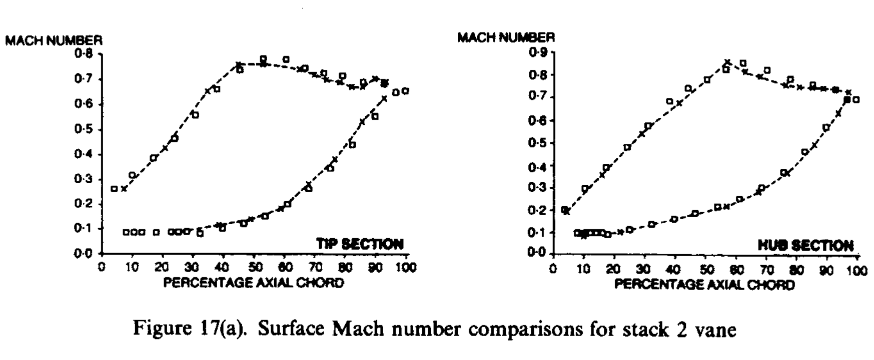
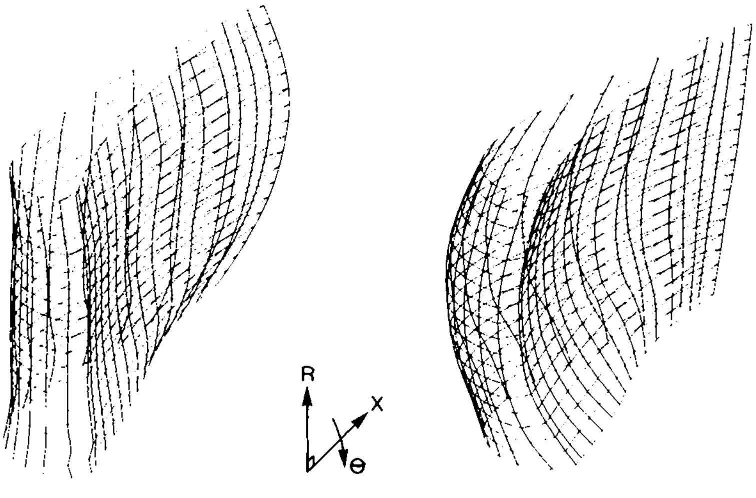


Figure 17(a). Surface Mach number comparisons for stack 2 vane



**Stack 1** **Stack 2**

Figure 17(b). Stack 2 vane

to-blade program of Reference 9, follows closely that developed by Calvert and Ginder[23] in which blade profile, mixing and secondary losses are included.

## 4. THREE-DIMENSIONAL ANALYSIS

As already discussed above, in the quasi-3D design system blade sections are designed on isolated axisymmetrical stream surfaces. In reality the stream surfaces will twist under the influence of streamwise vorticity, from upstream blade rows or created within the blade row, and the blade force, etc; this means that there will be flow through the surfaces used for designing. In addition, in the blade-to-blade boundary layer analysis adopted the effects of cross-flows normal to the stream surface are assumed small and ignored; in some cases, however, they may be significant, e.g. near the tips of rotors. A further complication comes from the blade–annulus interaction, where the annulus boundary layer separates ahead of the blade to form a horseshoe vortex which affects the development of the blade boundary layers through the blade passage.

In order to account for these effects, a three-dimensional analysis system is needed. Figure 18 shows the elements of such a system and how it links to the conventional quasi-3D design system. Also indicated are the routes back into the conventional system to allow changes to the blade sections or to the blade stack after a three-dimensional analysis.

In general in a three-dimensional analysis one is interested in studying the effects of changes in design parameters, e.g. blade stack, annulus curvature, on the secondary flows and secondary losses of a blade row. However, very useful information can often be obtained using an inviscid analysis in which the effects of the annulus boundary layers are accounted for only at the inlet to the blade row. In such an approach the inlet boundary layer is modelled by adopting an approximate inlet total pressure profile which gives rise to low-momentum fluid but still has some slip velocity at the wall. The low-momentum fluid is then acted on by the blade force within the blade row, giving rise to secondary flows at the exit. The effects of numerical viscosity are to accelerate the inlet low-momentum fluid and consequently these must be controlled by the use of a

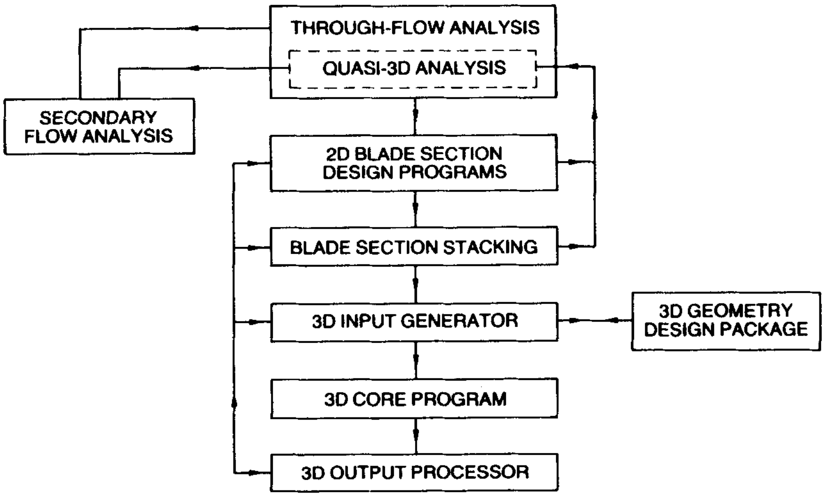


Figure 18. 3D aerodynamic design system

refined grid near the end-wall and avoidance of excessive smoothing. It is found that good predictions of secondary flow angles can result from the procedure, at least for isolated blade rows.

Care is needed, however, in applying such a technique to stage calculations, where the effect of each blade row is to produce a skewed inlet end-wall boundary layer to the following row. Birch[24] has shown that the effects of viscosity are very important in determining the development of the skewed inlet boundary layer. In a compressor the blade force and viscosity tend to have opposite effects on the skew in the boundary layer and large errors can arise in the prediction of the secondary flow using a purely inviscid analysis due to neglecting the action of viscosity on the skew. In turbines the blade force and viscosity tend to act in the same manner on the skew in the boundary layer and the effect of ignoring viscous action is less dramatic than in a compressor. Although some compensation for the effects of viscosity can be made in either case, the situation is unsatisfactory and indicates that a viscous analysis should be adopted.

The situation with regard to the development of methods for the solution of the Reynolds-averaged Navier–Stokes equations in 3D is basically as described earlier in 2D, with pressure correction and time-marching techniques being developed, the 2D methods and developments leading naturally into 3D.

With the time-marching technique, as was discussed earlier, a cell-centre-based scheme and cell-node-based scheme are under development. Figures 19(a) and 19(b), from Connell,[25] give results from the fully explicit cell-centred scheme with an embedded O–C–H grid system for a high-pressure turbine blade tested in cascade by Camus *et al.*; [26] about 40000 points have been used. The figures show velocity vectors near to the tip end-wall and the suction surface of the blade. These are in good qualitative agreement with flow visualization pictures and show the formation of the leading edge horseshoe vortex and the subsequent effect on the development of the blade boundary layers.

Results for this blade using the cell-node-based scheme are given in Carrahăr and Kingston[15] and show good agreement with experiment.

With the pressure correction technique two basic approaches are available: a semi-elliptic space-marching technique, described by Moore and Moore,[27] and a fully elliptic technique, also described by Moore and Moore.[11] The space-marching technique is ideally suited to flows where

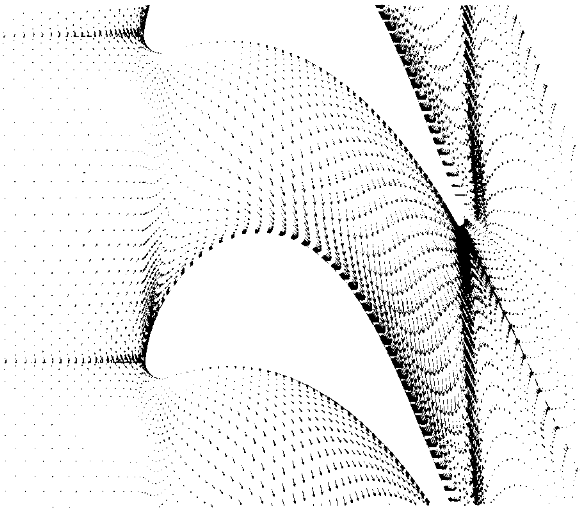


Figure 19(a). Velocity vectors near tip end-wall

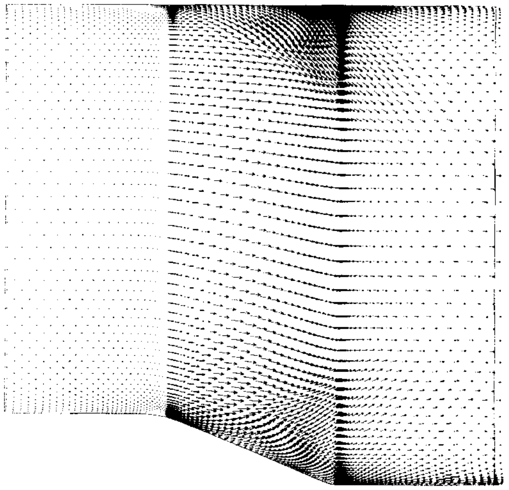


Figure 19(b). Velocity vectors near suction surface plane

separation does not occur in the streamwise marching direction; separations in planes normal to this direction can be handled. This means that secondary flows found in ducts, exhausts, etc. are well modelled with the technique. When applied to blade rows, the effects of separations near the leading edge, due to the end-wall horseshoe vortex, or reversed flows near blade trailing edges are handled only in an approximate manner; even so, valuable information on secondary flows is still produced. In order to remove these limitations, a fully elliptic procedure has been developed that is capable of handling the flow separations found at the leading and trailing edges of blades and over the tip of rotating blades. Results from this method can be seen in Moore and Moore[11] where it was shown that good loss predictions were found for the low-speed turbine tested. Further results are given in Northall *et al.*[28] for a low-speed annular turbine tested by Sieverding *et al.*[29] Figure 20 gives the end-wall velocity vectors for this example and shows that the main features are modelled even with the relatively modest grid of about 20 000 points used. Figure 21 shows a comparison of the mass-averaged whirl angles against radial height at two traverse planes. It can be seen that in general good agreement exists, showing that the secondary flow is well modelled. This is reinforced by the results given in Figure 22 which show that the predicted losses agree well with those measured. Transition in this example was specified from knowledge of the experiment, with the end-wall regions being taken as fully turbulent and the position of point-wise transition on the suction and pressure surfaces being specified. Reference 28 briefly discusses the sensitivity of the results to the assumptions made.

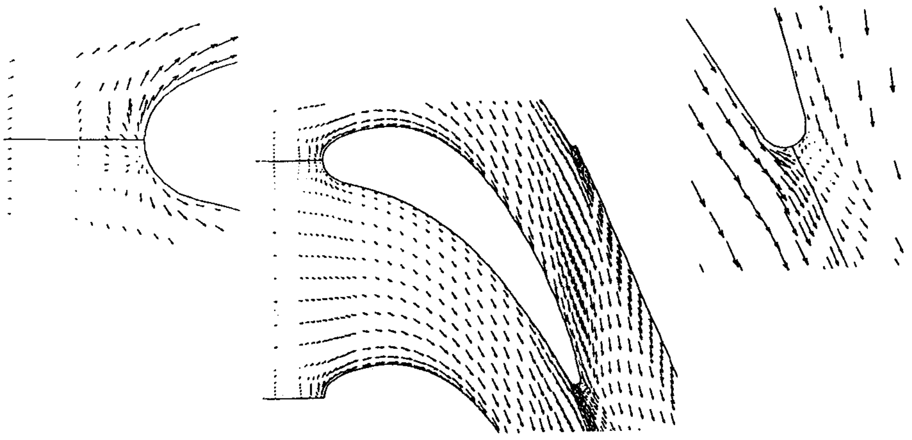
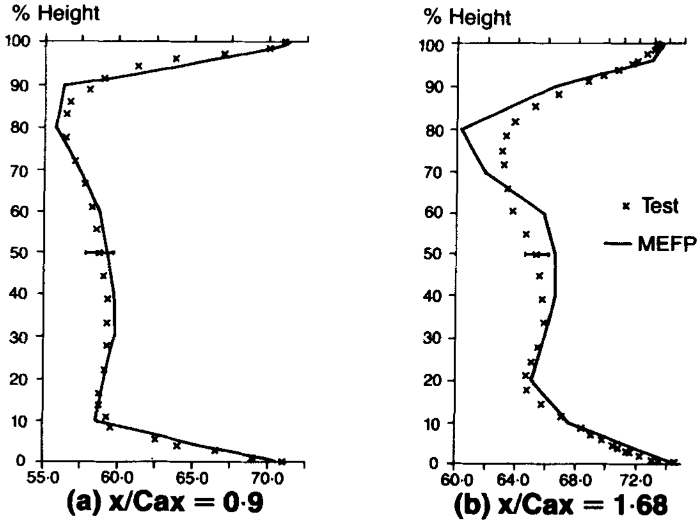


Figure 20. Hub end-wall velocities



**(a) x/Cax = 0·9**

**(b) x/Cax = 1·68**

Figure 21. Mass-averaged whirl angles versus radial height

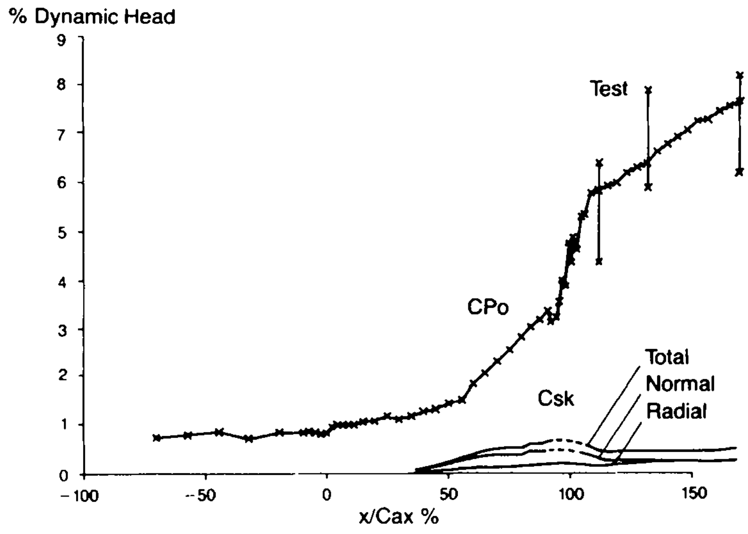


Figure 22. Total pressure loss and secondary kinetic energy

## 5. FUTURE DEVELOPMENTS

The work described above has tended to concentrate on areas where capabilities have been developed and evaluated. At this stage it would be useful to discuss briefly areas where further development is underway or is needed in the future and to indicate the mathematical and numerical modelling problems to be addressed.

In 2D and 3D steady blade-to-blade flow analysis extensions to both turbulence and transition models are required in order to handle both attached and separated flows associated with design and off-design behaviour. For example, any transition model needs to be able to handle natural transition, laminar separation-promoted transition and transitional separation. It is contemplated that in addition to extending existing simple models, higher-order turbulence models, i.e. algebraic and Reynolds stress models, will be adopted; it will be important, however, to assess at each stage what advances are being made in relation to the additional computational costs of the higher-order methods.

The work discussed so far has concentrated on steady flow analysis, although unsteady analysis in connection with flutter and forced response, i.e. incoming wake excitation, is undertaken routinely; for example, the full potential finite element program mentioned in Section 2 adopts a linearized unsteady analysis for flutter. Development of other methods and techniques described earlier is already underway and is aimed at both linearized and full non-linear analysis of the unsteady flow associated with flutter and blade row interaction. Single blade row analysis will be performed initially, with the inclusion of the wake from the upstream blade row, but eventually full stage interaction is envisaged. In considering unstalled flutter and forced response analysis, inviscid models are adequate; whereas for stalled flutter and blade row interaction effects on blade loss and heat transfer characteristics, viscous models are needed. With regard to the latter, turbulence and transition models are needed to describe the major phenomena. Initially quasi-steady models are envisaged, but later models capable of describing the unsteady features of the interaction will be needed.

In addition to the development of the mathematical models associated with the phenomena of interest, there is the need to develop various aspects of the numerical models; for example, grid systems for the analysis of multiple blade rows, accurate numerical schemes in order to minimize the number of nodes required, techniques for ensuring rapid convergence of the iterative procedures.

## 6. CONCLUSIONS

The basic elements of a quasi-3D turbomachinery blade design system in use within Rolls-Royce have been presented and examples given to illustrate capabilities. Coupled inviscid–boundary layer techniques can produce versatile tools for the design and loss optimization of blade sections. The main limitations of the approach lie with the boundary layer modelling of transitional and separated flows at design and off-design conditions; it is believed that extensions are possible within the framework of current boundary layer models to extend capabilities. Reynolds-averaged Navier–Stokes methods have been found to offer great promise with regard to loss and heat transfer prediction and in generating understanding to improve simpler models. Whereas the speed of these calculations currently limits the range of application, at least in two dimensions, advances being made in both algorithms and computers will mean that these will see more widespread application in the future.

Traditional blade design systems treat through-flow and blade-to-blade analyses separately. However, by adopting a coupled solution procedure, useful information can be produced about the effects of blade stack, annulus profiling, etc. prior to a full three-dimensional analysis.

For many blade rows, because of the phenomena involved and the aspect ratio of the blades, a full 3D analysis is needed. Inviscid calculations can produce useful results, but their impact is limited when considering the effects of inlet boundary layers from an upstream blade row; a viscous model should be adopted in this case. The use of Reynolds-averaged Navier–Stokes methods in 3D blade design systems is well established; these methods offer the opportunity of understanding the effects of design variables on secondary flows and secondary losses and of eventually being used for loss optimization.

Further advances in blade design are expected to come from studying the unsteady effects of blade row interaction. It is clear that as the numerical techniques are developed further, attention will have to je given to the development of turbulence and transition models in order to explain the phenomena and fully exploit the methods as design tools.

## REFERENCES

1. C. H. Wu, 'A general theory of three-dimensional flow in subsonic and supersonic turbomachines of axial, radial and mixed flow types', NACA TN 2604 (1952).
2. R. D. Cedar and P. Stow, 'A compatible mixed design and analysis. Finite element method for the design of turbomachinery blades', *Int. j. numer. methods fluids*, **5**, 331–345 (1985).
3. P. Stow and S. P. Newman, 'Coupled inviscid–boundary layer methods for turbomachinery blading design', *Joint IMA/SMAI Conf. on Computational Methods in Aeronautical Fluid Dynamics*, University of Reading, April 1987.
4. B. J. Abu-Ghannam and R. Shaw, 'Natural transition of boundary layers—the effect of turbulence, pressure gradient and flow history', *J. Mech. Eng. Sci.*, **22**(5), 213 (1980).
5. W. B. Roberts, 'Calculation of laminar separation bubbles and their effect on airfoil performance', *AIAA J.* **18**(1), (1980).
6. S. P. Newman, 'Addition and evaluation of an integral wake calculation in FINSUP', Rolls-Royce (private communication).
7. H. P. Hodson, 'Boundary layer transition and separation at the leading edge of a high speed turbine blade', *ASME Paper 84-GT-241*, 1984.
8. S. P. Newman and P. Stow 'Semi-inverse mode boundary layer coupling', *IMA Conf. on Numerical Methods for Fluid Dynamics*, University of Reading, 1985.

9. W. J. Calvert, 'An inviscid-viscous interaction treatment to predict the blade-to-blade performance of axial compressors with leading edge normal shock, waves', *ASME Paper 82-GT-135*, 1982.
10. J. D. Denton, 'An improved time marching method for turbomachinery flow calculation', *ASME Paper 82-GT-239*, 1982.
11. J. Moore and J. G. Moore, 'Performance evaluation of a linear turbine cascade using a 3D viscous calculation', *Trans. ASME, J. Eng. Gas Turbines Power*, **107**, 969–975 (1985).
12. A. Jameson, W. Schmidt and E. Turkel, 'Numerical solution of the Euler equations by finite volume methods using Runge–Kutta time stepping schemes', *AIAA Paper 81-1259*, 1981.
13. R. H. Ni, 'A multiple-grid scheme for solving the Euler equations', *AIAA J.* **20**, 1565–1571 (1982).
14. R. J. G. Norton, W. T. Thompkins and R. Haimes, 'Implicit finite difference schemes with non-simply connected grids. A novel approach', AIAA *22nd Aerospace Sciences Meeting*, Reno, January 1984.
15. D. Carrahar and T. R. Kingston, 'Some turbomachinery blade passage analysis methods—retrospect and prospect', *Transonic and supersonic Phenomena in* Turbomachines, AGARD, Munich, 1986.
16. S. D. Connell, 'An evaluation of FANSI-2 implicit Navier–Stokes code over a range of exit Mach numbers', Rolls-Royce (private communication).
17. N. T. Birch, 'Navier–Stokes predictions of transition, loss and heat transfer in a turbine cascade', *ASME Paper GT-22* (1987).
18. S. D. Connell, 'Turbine evaluation of FANSI-2 implicit Navier–Stokes code', Rolls-Royce (private communication).
19. J. H. Nicholson, A. E. Forest, M. L. G. Oldfield and D. L. Shultz, 'Heat transfer optimised turbine rotor blades—an experimental study using transient techniques', *ASME Paper 82-GT-304*, 1982.
20. R. J. Stafford and N. T. Birch, 'An investigation of the effects of thick and thin trailing edges in compressor cascades', Rolls-Royce (private communication).
21. I. K. Jennions and P. Stow, 'A quasi three-dimensional turbomachinery blade design system. Part I—Through-flow analysis', *ASME Paper 84-GT-26*, 1984; 'Part II—Computerised system', *ASME Paper 84-Gt-27*, 1984.
22. I. K. Jennions and P. Stow, 'The importance of circumferential non-uniformities in a passage-averaged quasi-three-dimensional turbomachinery design system', *ASME Paper 85-IGT-63*, Beijing, 1985.
23. W. J. Calvert and R. B. Ginder, 'A quasi-three-dimensional calculation system for the flow within transonic compressor blade rows', *ASME Paper 85-GT-22*, 1985.
24. N. T. Birch, 'The effects of viscosity on inlet skew in axial flow turbomachinery calculations', *Institute of Mechanical Engineers Conf. on Computational Methods in Turbomachinery*, University of Birmingham, 1984.
25. S. D. Connell, 'Three-dimensional viscous time-marching results', Rolls-Royce (private communication).
26. J. J. Camus, J. D. Denton, J. V. Soulis and C. T. J. Scrivener, 'An experimental and computational study of transonic three-dimensional flow in a turbine cascade', *ASME Paper 83-GT-12*, 1983.
27. J. Moore and J. G. Moore, 'Calculations of three-dimensional viscous flow and wake development in a centrifugal impeller', *Trans. ASME, J. Eng. Power*, **103**, 367–372 (1981).
28. J. D. Northall, J. G. Moore and J. Moore, 'Three-dimensional viscous flow calculations for loss prediction in turbine blade Rows', *Int. Conf. on Turbomachinery*, Institute of Mechanical Engineers, Cambridge, 1987.
29. C. H. Sieverding, W. Van Hove and E. Boletis, 'Experimental study of the three-dimensional flow field in an annulus turbine nozzle guide vane', *ASME Paper 83-GT-120*, 1983.