GT2010-&&) ($

SOME LIMITATIONS OF TURBOMACHINERY CFD

John D Denton
Whittle Laboratory
Cambridge, UK

ABSTRACT

CFD is now an essential tool for the design of all types of turbomachinery. However, as engineers are exposed more and more to the results of CFD and less and less to experimental data there is a danger that they may not realise the limitations of CFD and so will view its predictions as more reliable than they really are and not question them sufficiently. This is particularly dangerous when CFD is use as part of an optimisation procedure. The objective of this paper is to try to expose some of the limitations of CFD as used for routine turbomachinery design.

CFD is not an exact science. Errors can arise from the following sources:
- Numerical errors due to finite difference approximations.
- Modeling errors, where the true physics is not known or is too complex to model – e.g. turbulence modeling.
- Unknown boundary conditions, such as inlet pressure or temperature profiles.
- Unknown geometry such as tip clearances or leading edge shapes.
- Assumption of steady flow.

Each of these sources of error is discussed and examples of the differences they can cause in the predictions are shown. Despite these limitations CFD remains an extremely valuable tool for turbomachinery design but it should be used on a comparative basis and not trusted to give quantitative predictions of performance.

1. INTRODUCTION

The exact definition of the term Computational Fluid Dynamics (CFD) as applied to turbomachinery is not clear. Throughflow calculations and inviscid 2D blade to blade calculations were developed and used in the 1960's, 3D inviscid Euler solutions started to appear and be used in the 1970's, 3D Navier-Stokes (N-S) solutions in the 1980's and 3D steady multistage and unsteady single stage calculations in the 1990's. In this paper CFD will be taken to mean to 3D N-S calculations so that throughflow calculations, 2D calculations and inviscid calculations are not considered.

Until the past decade CFD calculations were mainly performed by specialists, often those who had been involved with the development of the code, or by designers with many years experience of the type of machine to which it was being applied. However, nowadays CFD is widely used by young engineers who may have little experience of the underlying numerical method or of the performance to be expected from the machine they are designing. Hence there is a danger that they will not realise the limitations of the results that they obtain. This overconfidence in the results of CFD is fostered by the vendors of commercial packages, who are unlikely ever to admit that their code can give poor results, and by the authors of technical papers who naturally want to show their code or their company in a good light. When comparing the results of CFD with test data it is very common for authors to reject any results that do not agree well, to change the grid or some constant in the code and repeat the calculation until satisfactory agreement is obtained, then to publish the good result. The unsatisfactory results are seldom shown and so readers of the literature are left with a false impression of the accuracy of the CFD.

The author acts as a consultant to several gas and steam turbine companies and has seen very many comparisons of CFD predictions with test data and with different codes. The limitations then become all too apparent. He has often heard engineers say " I have redesigned machine x and obtained y% improvement in efficiency " , when they really mean that they have performed a CFD calculation that predicts the improvement, rather than tested the machine and measured the improvement. Unless the limitations in accuracy of CFD predictions are realised there is a danger that expensive manufacture and testing will be performed with only a limited chance of success.

Perhaps even more dangerous that the over-confidence of young engineers is the natural inclination of managers to seize on any opportunity to reduce the costs of developing

new designs. Often managers in charge of aerodynamic design are specialists in a different discipline, such as stress analysis, where the computer calculations are more reliable, and so they have a similar high level of confidence in CFD. They are then likely to use the argument that CFD can predict performance to cut down on the use of test rigs and to proceed straight from CFD to production. This can lead to very expensive remedial action.

In the past the accuracy of CFD was mainly limited by numerical errors due to the limited number of grid points that could be used. Nowadays computer performance and costs have improved so enormously that the effects of numerical errors are usually small. However, turbulence and transition modeling have improved little over the years are remain a major source of error, especially for separated and transitional flows. The assumption of steady flow in multistage calculations is a source of error whose magnitude is not fully understood. However, in most applications the main sources of error are likely to be unknown boundary conditions, such as inlet pressure and temperature profiles, and unknown geometry such as tip clearance or leading edge shape. It may be argued that the latter are not errors in the CFD itself but they certainly play a large part in determining the accuracy of the results and so will be considered here.

There have been a number of comparisons of different CFD codes on the same test data. Casey (1) gives a useful review of these together with a summary of the lessons learned from them. Casey's review also provides a detailed discussion of many of the points raised in the current paper. The rotor 37 blind test case, which was conducted by IGTI in 1993, Denton (2), is not representative of current capability but illustrated that at that time very large differences could be obtained by different users of the same code, as well as between different codes. It is significant that a major cause of error in most of the predictions was subsequently found to be the neglect of the effects of a small hub leakage upstream of the rotor, Shabbir et al (3). Dunham & Meauze (4) reported the findings of an AGARD working group on CFD in 1998, but this was only applied to single blade rows. A more recent comparison was performed by Woolatt et al (5) on a 3 stage high speed compressor, four anonymous codes were compared with the test data. The comparison with the overall compressor performance is reproduced in Fig. 1, the predicted efficiencies vary by 2.5%, the choking mass flow varies by at least 2% and the mass flow at stall varies by 7%.

The objective of this paper is to help CFD users understand the most likely sources of error in their predictions. Each major source of error is discussed and examples of the magnitude of its effects are given. It should be emphasised that the discussion is only about the type of CFD that is currently used for routine design. Specialised methods, such as LES or DNS, or special turbulence models, may be able to give much better results in some cases but are either too complex, too time consuming or too temperamental for routine use.

2. NUMERICAL ERRORS

Numerical errors arise from the finite difference approximations that are inevitable in any numerical method. They also arise from the artificial viscosity or smoothing that is necessary to stabilise many codes. The most common numerical approximation is that the flow properties vary linearly between two grid points and so the error is proportional to the (square of the grid spacing) x the second derivative of the flow property concerned. The error produced by second order artificial viscosity has exactly the same dependency. Hence it is clear that the largest errors will occur in regions where both the second derivatives and the grid spacing are large. Hence the grid spacing should be reduced in such regions. A highly loaded leading edge is a typical example of such a region.
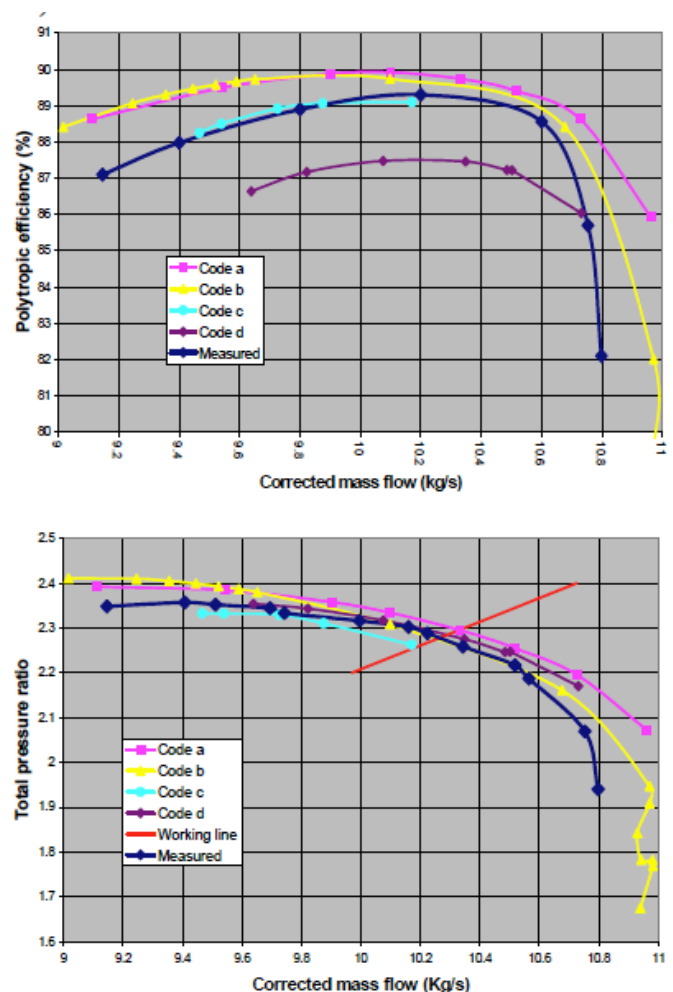

Fig.1 Performance of a 3 stage compressor. From Woolatt et al. GT2005-68793.

The effect of such numerical errors is to act like an additional viscous term in the equations and hence to cause spurious entropy to be created in the flow. Once created this numerical entropy is convected downstream from its source and influences the whole downstream flow. Hence the

effects of numerical errors are not localized and errors created at a leading edge can ruin the whole calculation. These errors can also cause loss of mass flow conservation and it is always best to check that the ratio of inlet to outlet mass flow rate from each blade row is close to unity, say within 0.1%.
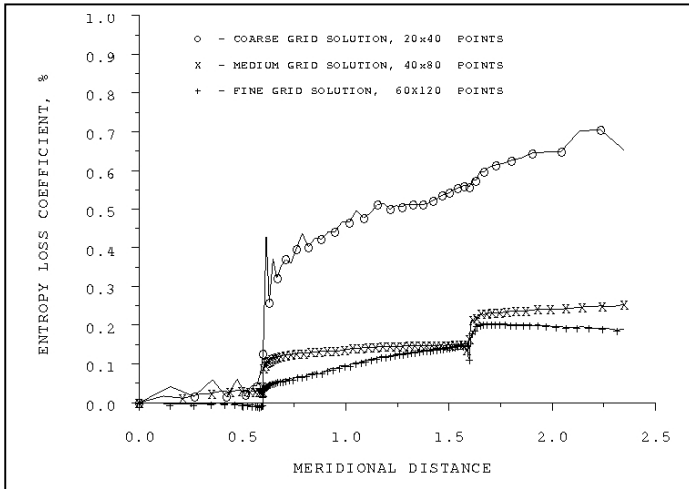

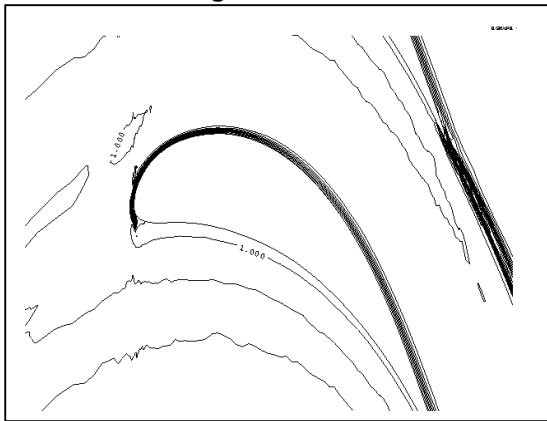**Fig. 2a. Growth of the entropy loss coefficient through a turbine cascade.**

Fig. 2b. Contours of entropy loss coefficient for the 40 x 80 grid. Contour interval = .01 .

The magnitude of numerical errors will always tend to decrease as the number of grid points is increased and the increase in computer power, and hence in the number of grid points used, has led to a considerable reduction in their importance. The best check on the magnitude of numerical errors is to look at the entropy created in an inviscid flow and to turn this into a loss coefficient. Fig. 2a shows the growth in entropy loss coefficient through a turbine cascade with varying number of grid points in the blade-to-blade passage, and using a simple H mesh. Fig. 2b shows a contour plot of entropy through the cascade for the 40x80 grid. The latter shows that most of the loss is created at the leading edge whilst the bulk of the flow is almost loss-free. The 20x40 point grid has a loss coefficient of 0.7% due to numerical losses, mainly arising at the leading edge, this would not be acceptable for performance

prediction since the profile loss coefficient is likely to be of order 3.0%. The leading edge loss would be greatly reduced if an O mesh rather than an H mesh were used. The 60x120 point grid has a numerical loss coefficient of 0.18%, which arises at both the leading edge, on the suction surface and at the trailing edge. Whilst this loss is still significant it would not grossly increase any viscous loss. It may be concluded that with an H mesh around 400,000 grid points per blade row are necessary to keep the numerical losses within acceptable limits.

It must be noted that some numerical errors or artificial viscosity are essential to capture shock waves accurately, if they are too small then there will be overshoots and undershoots in velocity and pressure before and after the shock, if they are too large the shock will be smeared. Adaptive second order artificial viscosity is usually used to facilitate this.

3. MODELLING ERRORS

These include turbulence and transition modeling and the use of mixing planes to allow steady solutions for blade rows in relative motion.

3.1 Turbulence modeling. The author is not an expert on turbulence modeling but has seen enough results from it to know that different models, or different constants in the same model, can give very different results. For example, in a recent study Pecnik et al (6) used 3 different turbulence models and several different levels of free stream turbulence in a modern CFD code. For a transonic turbine stator blade the predicted energy loss coefficient varied from 2.8% to 3.75% and none of the models gave the correct shock position.

For attached fully turbulent boundary layers all turbulence models should have been calibrated to give good
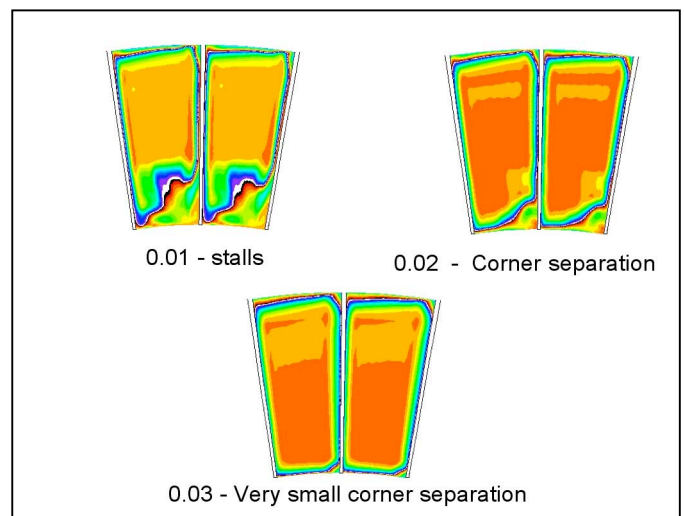

Fig. 3. Effects of different mixing length limits on a compressor separation.

predictions, given sufficient mesh points. The main differences occur in separating and fully separated flows. In a separated flow it is the effective turbulent viscosity at the edge of the separated region that prevents the separation growing uncontrollably. Separated flows at high Reynolds numbers are never steady and the flow at the edge of the separation is likely to be affected more by large length scale unsteadiness than by the small scale turbulence that is usually modeled. In this situation LES calculations might be expected to give better results than RANS calculations. The differences between models are likely to most severe for compressors near their stall point. To illustrate this Fig. 3 shows a the stator exit flow from a compressor stage computed with three difference values of the mixing length limit which is used in the author's model. This parameter controls the turbulent viscosity in separated regions.

3.2 Transition modeling The state of the boundary layer has a very large influence on the entropy creation in it and hence on the loss of a blade row. If the boundary layer is either fully laminar or fully turbulent its growth and loss can be reasonably well predicted by CFD. However, in many practical cases the boundary layer is transitional and the transition is influenced by many factors. These include: Reynolds number, turbulence level, pressure gradient, surface roughness, surface curvature and 3D effects. Methods of predicting transition were comprehensively reviewed by Mayle in his 1991 IGTI Scholar lecture (7) and correlations exist for the effects of Reynolds number, pressure gradient and turbulence level. At low Reynolds numbers, $Re < 5x10^5$, transition often occurs via a separation bubble and further correlations are needed to predict the behaviour of this. None of these correlations is of great accuracy but more importantly the free stream turbulence,
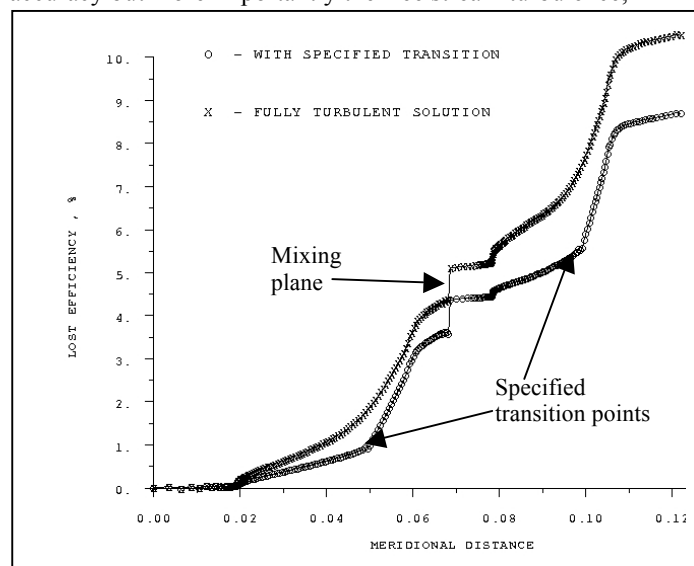

Fig. 4. **Effect of the transition point on the loss growth through a low pressure turbine stage**

which can have a dominant influence, is seldom known. The boundary layer behaviour under unsteady flow is even more complex and the interaction of a laminar boundary layer with the increased turbulence in a passing wake has a large influence on the separation bubble and loss, see Halstead et al (8), Coull et al (9).

Transition is even more complex in 3D flows such as those near the endwalls of a blade row. Here the 3D separations and strong pressure gradients may cause the endwall boundary layer to relaminarise, as found by Harrison (10) and by Holley (11) for turbine cascades. It is unlikely that any CFD method can yet predict this behaviour.

In most CFD calculations there is no attempt to predict transition as part of the calculation, instead the transition point is simply specified at a fixed point on the blade surface. In many cases this is simplified to an assumption of fully turbulent flow or to transition being specified at the location of peak suction. Fig. 4 illustrates the importance of the choice of transition point by comparing the cumulative efficiency loss through a low pressure turbine stage with either fully turbulent flow or with transition specified at around peak suction. The transition point was specified at the same axial position on both blade surfaces and endwalls so the sudden increase in the rate of entropy generation is more apparent. The difference in predicted stage efficiency was 1.8%, equivalent to about 20% difference in lost efficiency. Given the arbitrary choice of transition point it is clear that the accuracy of efficiency prediction is limited at these low Reynolds numbers.

3.3 Mixing Planes. Mixing planes are used to enable steady calculations of the flow through blade rows in relative motion. At a fixed axial location and for each spanwise grid point the flow upstream of the mixing plane is assumed to mix out at to a flow with pitchwise uniform enthalpy and entropy. This uniform flow is then the steady inlet flow to the next blade row. The mixing process should conserve mass, energy and momentum but, as with any real mixing process, the entropy will increase. This mixing loss can be seen as the jump in lost efficiency at the mixing plane in Fig. 4. The flow downstream of the mixing plane should have pitchwise uniform stagnation enthalpy and entropy but in general its static pressure and flow direction should be pitchwise non-uniform as determined by the downstream row. There are different methods of trying to satisfy these requirements as described by Denton (12) and Holmes (13). The unsteady interaction between blade rows is strongest when the flow is transonic with shock waves and the model becomes even more dubious in such cases. Fig. 5 compares a mixing plane calculation with an unsteady calculation on a turbine stage with blade exit Mach numbers close to unity. The large differences are all too apparent.

Adamcyzk's average passage method (14) avoids using a mixing plane by performing calculations on overlapping
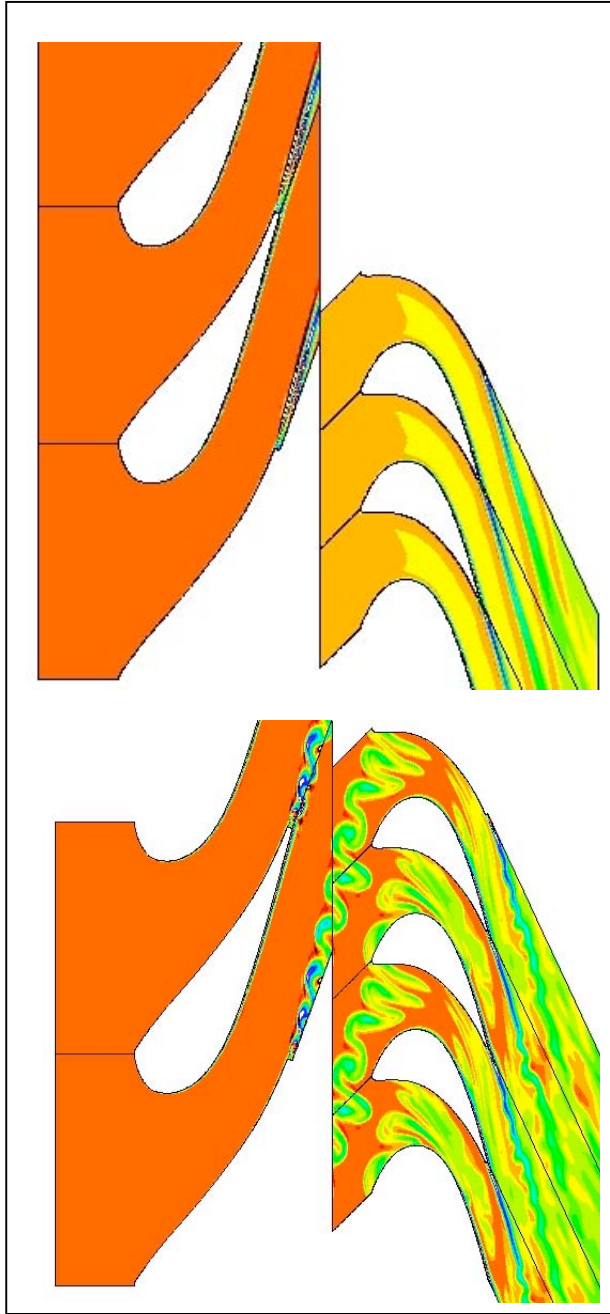
Fig. 5. Comparison of mixing plane and unsteady calculations. Entropy contours in a turbine stage.

grids and allowing the mixing loss to take place via deterministic stresses within the downstream row. Although this is more realistic than the mixing plane model the modeling is more complex and still requires many simplifying assumptions.

The central assumption of the mixing plane approach is that the mixing loss generated at the mixing plane is the same as the loss generated when the real flow mixes out in the unsteady environment within the downstream row. There is no a-priori theory the justify this assumption, in fact it is well known that

the mixing loss in a wake is reduced when the wake is stretched and increased when it is compressed. It is generally accepted that a wake is stretched in a compressor but the behaviour in a turbine is more complex and it is not obvious whether the loss is increased or decreased. Praisner et al (15) claim significant differences, up to 0.9% in efficiency, between steady and unsteady calculations on LP turbine blades, and attribute it to this effect. However, the author has never observed such large differences. Given that the mixing loss in a 2D wake is typically about 15% of the 2D loss and that most of the mixing takes place within a few trailing edge thicknesses of the trailing edge, it is clear that any large difference must be associated with mixing out of the large scale 3D features near the endwalls.

Unsteady calculations overcome this problem but are still too time consuming for use in routine design and so mixing planes are widely used. Given the uncertainty about the losses they generate this adds a further potential source of error to such calculations.

3.4 Trailing edge modeling. The flow downstream of a thick trailing edge (TE) is always unsteady with vortex shedding, as seen in Fig. 5. This generates a low base pressure acting on the TE and the drag associated with this leads to increased wake depth and entropy generation downstream. Steady calculations cannot model this TE loss without some special treatment. In fact a thick TE is an exception to the rule that a fine grid will always give more accurate results. If a fine grid is used around TE it is usually found that the flow does not separate early enough at the junction between the pressure surface and the trailing edge circle. It remains attached for some distance around the TE and the resulting extremely high streamline curvature generates a low pressure on the rear of the pressure surface as illustrated in Fig. 6. Very often this low pressure results in negative blade loading near the TE. An example is the
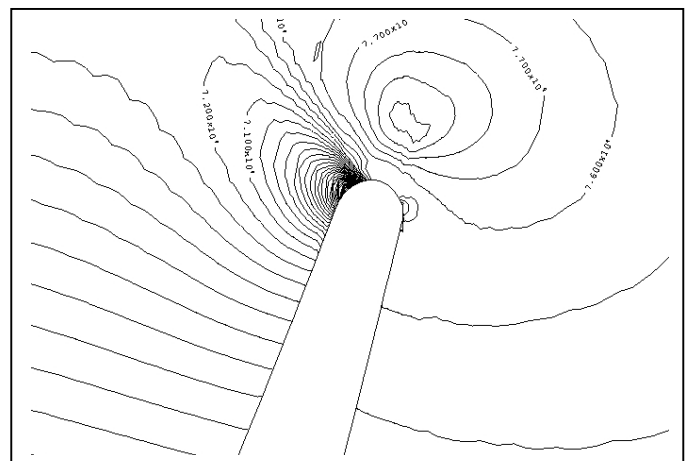

Fig. 6. Predicted static pressure contours round a turbine trailing edge.
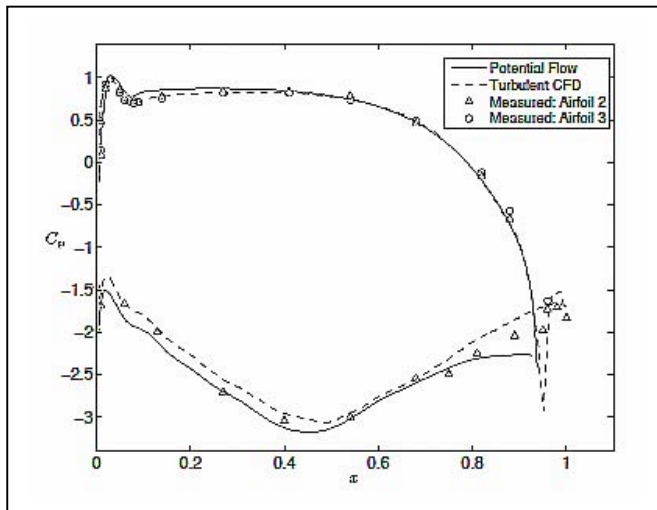
Fig.7. Calculated and measured pressure coefficient on Langston's turbine cascade. From Holley (2005).

calculation by Holley (11) on the well-known Langston turbine cascade, which has an exceptionally thick TE, this is reproduced in Fig. 7. Many calculations, such as that in Fig. 6, show a greater unloading than this.

This negative loading is never found in experiments. One of its effects is an underturning of the flow due to the reduced lift on the blade, in fact this causes a feedback situation in that the increased deviation causes more turning around the TE and hence an even lower pressure on the rear of the pressure surface. The other effect is a lower average pressure acting on the TE and the momentum loss associated with this will cause the CFD to generate more loss downstream. For a turbine blade with a thick TE the increase in loss coefficient can be of order $1 - 2\%$, which is very significant.

Because of this problem the author has always preferred to use a relatively coarse grid with a cusp at the trailing edge. This avoids a sudden change of flow direction around the TE, but the cusp must be carefully modeled to carry no load and if started upstream of the TE it may affect the throat area. An alternative approach is to use a fine grid but to force the flow to separate by means of an imposed body force. Neither of these methods is likely to give the correct base pressure acting on the TE and so will not predict the correct loss. Ning & He (16) show that the correct base pressure, and hence loss, can be predicted by using unsteady calculations to obtain a deterministic stress which is then input to a steady calculation.

a) Periodic tip

b) Pinched tip

c) Gridded tip, turbulent

d) Gridded tip laminar
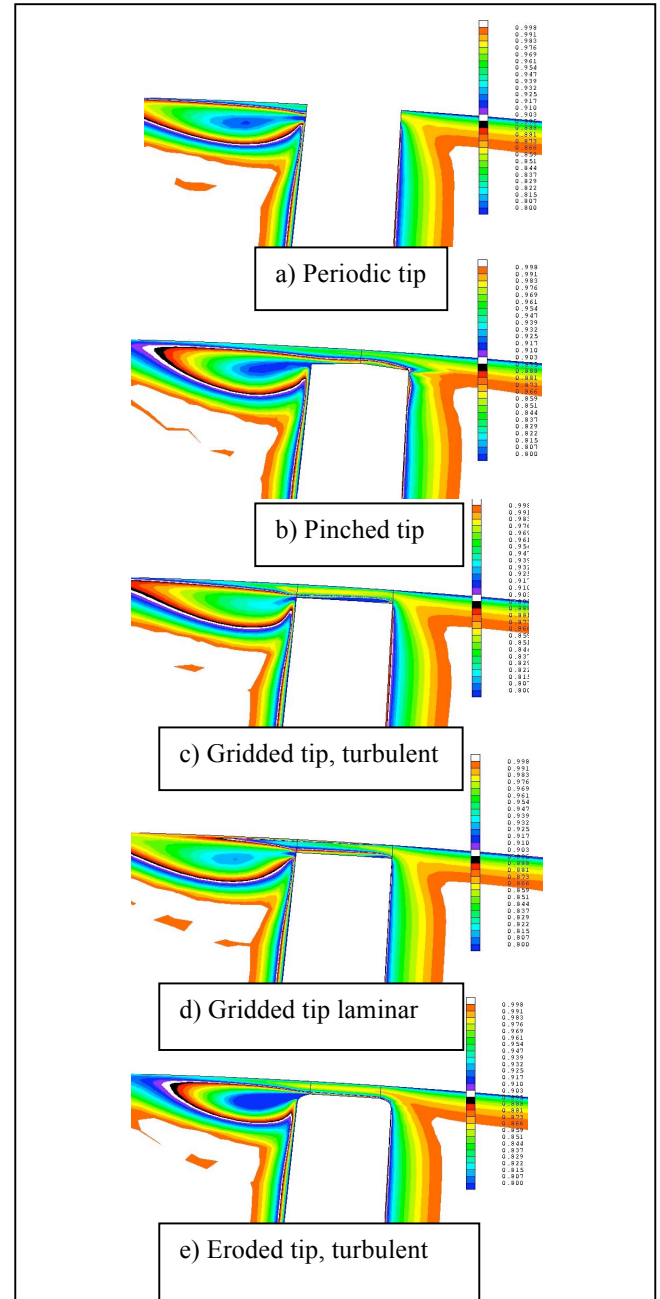
e) Eroded tip, turbulent

Fig. 8. Entropy contours in a compressor rotor for different tip grid models.

	Pinched tip	Periodic tip	Laminar gridded tip	Turbulent gridded tip	Eroded turbulent gridded tip
Percentage tip leakage flow	1.316	0.869	0.909	1.028	1.306

Table 1. Comparison of tip leakage flow rates for different tip models

6

3.5 Tip Leakage modeling. Tip leakage flow plays a very important part in all types of turbomachinery. It causes typically 1/3 of the total loss and for compressors it plays a large part in the stall inception. However, modeling of the tip leakage for plain tip clearances is very imprecise. This is illustrated in Fig. 8 for a compressor rotor with a tip gap 1% of span. It is quite common to use a pinched tip model as illustrated in Fig. 8b but this will not allow the leakage flow to contract to a jet and so will overestimate the amount of leakage. Some codes recommend that the tip clearance be reduced to about 60% of the true value to allow for this. An alternative is not to contract the blade thickness at the tip but simply to apply periodicity between the suction and pressure surfaces over the region of the tip gap as shown in Fig. 8a. This does not allow for chordwise transport of fluid within the tip gap and is seen by the flow as a sharp edge with zero thickness, hence the jet contraction is greater and the leakage flow less than other models. Use of a gridded tip gap as illustrated in Fig. 8c is more realistic but the validity of any turbulence model in such a complex flow is dubious. The flow accelerates into the gap dampening any turbulence and the Reynolds number based on tip clearance is low, typically of in the range 2000-5000, so it is arguable that the flow in the gap is transitional, possibly laminar up to the throat of the leakage jet followed by a turbulent reattachment. The way that this is modeled can have a significant effect on the leakage. Figs 8c and 8d compare the leakage pattern when the flow in the tip gap is fully laminar and fully turbulent. As expected the laminar flow has a larger separation, and hence a lower leakage flow, than the turbulent one. A comparison of the predicted leakage flow for all these cases is given in Table 1.

Also included in Fig. 8e is a calculation where the tip gap has been eroded to give slightly rounded corners, as is likely to occur in service. The leakage flow is significantly increased by this small change in geometry. The large differences in leakage flow will have a significant effect on the predicted efficiency of the stage where the leakage occurs and also on the inlet flow to the downstream stage.

4. UNKNOWN BOUNDARY CONDITIONS

The boundary conditions used for turbomachinery calculations are usually the distribution of stagnation pressure, stagnation temperature and flow angles at inlet to the machine and the spanwise variation of static pressure at exit. The turbulence quantities such as turbulent kinetic energy and its length scale may also be required at inlet. At the exit it is common to specify the pressure at one point and to apply simple radial equilibrium to obtain the spanwise static pressure variation. The specified pressure determines the operating point of the machine but if the radial equilibrium is applied well downstream of the last blade row it does not usually have much effect on the predicted performance. However, the inlet boundary conditions can have a very significant effect and in a real machine these are almost never known accurately.

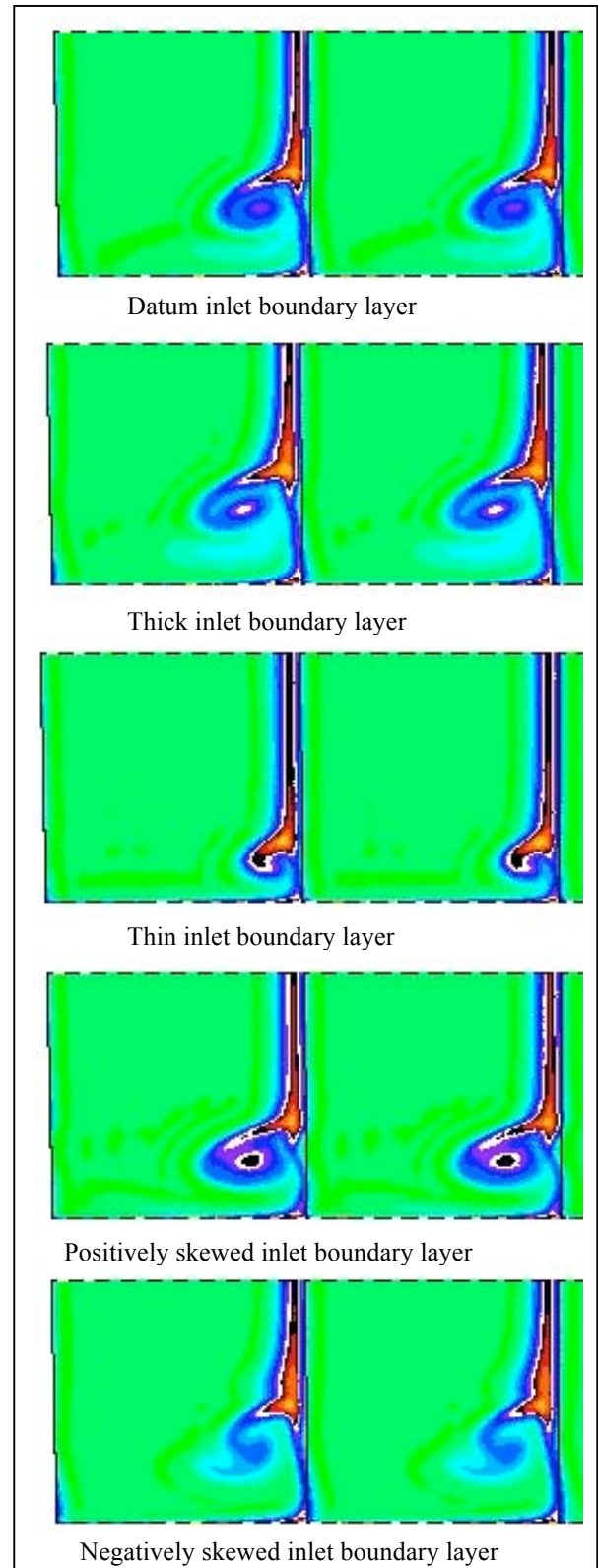Datum inlet boundary layer

Thick inlet boundary layer

Thin inlet boundary layer

Positively skewed inlet boundary layer

Negatively skewed inlet boundary layer

Fig. 9. Effect of different inlet boundary layers on secondary flow. Entropy contours.

4.1 Annulus boundary layers. The most important factor, and also the least likely to be known, are the flow details in the annulus (endwall) boundary layers. The endwall boundary layer thicknesses and velocity profiles have a large influence on the endwall losses in both turbines and compressors and on the stalling behaviour of compressors. These boundary layers are especially difficult to specify because they are usually formed by a combination of boundary layer growth, secondary flows and leakage flows. As a result they are not colateral and the resulting skew introduces streamwise vorticity, which has a large effect on the secondary flow in the downstream blade rows. As an example of this Fig. 9 shows predicted contours of entropy behind Harrison's turbine cascade (10) with different endwall boundary layers. The datum inlet boundary was that measured by Harrison and the skewed boundary layers have the same stagnation pressure profile as the datum but with the flow angle reduced by 30° (i.e. closer to axial) near the endwall for negative skew and increased by 30° near the wall for positive skew. Negative skew is representative of the boundary layer in a real turbine and positive skew in a compressor. Fig. 9 and Table 2 show that the positive skew is predicted to give the largest endwall loss, this is rather surprising as this type of skew introduces streamwise vorticity in a direction to oppose the secondary flow and negative skew is known to increase the endwall loss in a turbine cascade, as shown by Walsh et al (17).

	Datum inlet BL	Thick inlet BL	Thin inlet BL	Positively Skewed BL	Negatively skewed BL
Gross loss	7.0 %	7.9%	5.6%	7.7%	7.0%
Inlet BL loss	0.6%	1.2%	0.0%	0.4%	0.75%
Profile loss	3.8%	3.85%	3.6%	3.8%	3.5%
Net endwall loss	2.6%	2.85%	2.0%	3.5%	2.75%

Table 2. Stagnation pressure loss coefficients for different endwall boundary layers.

In addition to the effect on endwall loss the different inlet inlet boundary layers cause slightly different profile loss (Table 2) as a result of different values of stream surface contraction. They also cause very different secondary flows and hence change the inlet angle to the downstream blade row. It is re-emphasised that in a practical situation the state of the endwall boundary layer is almost certainly unknown.

4.2 Free stream turbulence level. Another inlet boundary condition, which may be important, is the free stream turbulence. Again this is very unlikely to be known in a practical situation. The free stream turbulence affects the
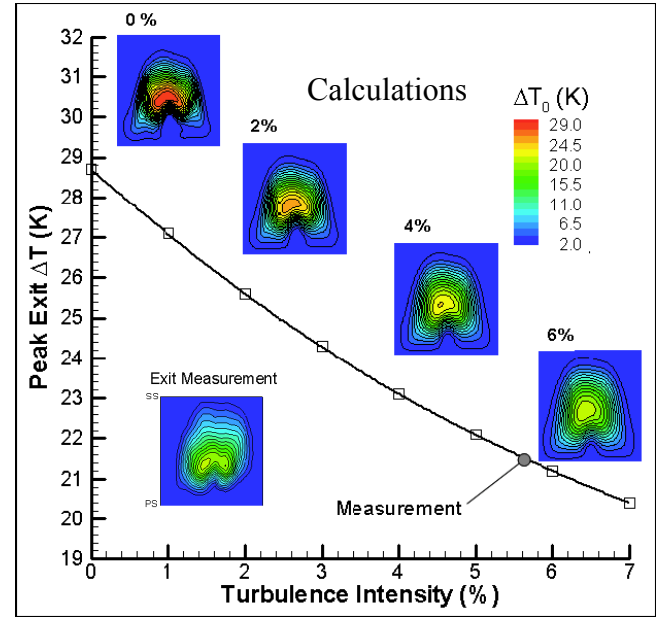
Fig. 10. Effect of varying turbulence level on the spread of a hot spot – from Ong et al (15).

mixing of enthalpy and entropy non-uniformities and, as previously mentioned, it has a large affect on boundary layer transition. The mixing of temperature non-uniformity is particularly important for the flow leaving a combustor and entering a high pressure stator blade. The turbulence level leaving the combustor is likely to be very high but the actual value can only be guessed. Fig. 10, from Ong et al (18), shows experiments and calculations on a simulated hot spot passing through a nozzle. Without free stream turbulence the CFD completely fails to predict the spread of the hot gas. By gradually changing the inlet turbulence in the calculation, and comparing with the experiment, good predictions are obtained with a turbulence level of about 6%.

A similar use of CFD results to calibrate the inlet turbulence is shown by Aalburg et al (19) who systematically varied the inlet turbulence length scale to obtain agreement with their experimental data.

5 UNKNOWN GEOMETRY

Unknown geometry may apply to either blade profiles or to seal clearances. It is true that this does not represent an error in the CFD method itself but it does have a large influence on the accuracy of its predictions. Blade profiles are usually known accurately when they are cold and stationary but the hot running profile may be significantly different. This problem is most serious at the tips of thin compressor blades, which have significant amounts of untwist due to gas and centrifugal loading. One degree of twist on a compressor tip section will change the local mass flux by about 3% and may completely change the matching

of the compressor. The running blade shapes should be estimated by stress analysis in such circumstances.

5.1 Seal and tip clearances. Seal and tip clearances represent an even larger source of error than blade profiles because they are seldom known accurately. In addition to its well known influence on efficiency, tip clearance has a major influence on compressor stall and this is discussed in section 6. Leakages from seals on the hub or casing can also have a large influence on the endwall boundary layers and hence on the endwall loss. Reid et al (20) show that the small leakage from a strip seal on one endwall of a turbine stator can reduce the stage efficiency by about 1.5%. Although they were able to predict this reasonably well using specialized CFD with the seal clearances known, such leakage flows are usually neglected during routine design.

Another source of unknown geometry is due to erosion in service. This can affect tip and seal clearances and the shape of critical areas such as a compressor blade leading edge. The significant effect of rounding the corners on a tip gap is illustrated in Fig. 8 and this is in addition to any changes in the clearance itself.

5.2 Leading edge shape. The flow around the leading edge (LE) of a compressor blade is critically dependent on the exact shape of the LE, which is possibly less than 0.5mm in thickness. Manufacturing errors are likely to have a significant effect on the LE shape as are changes due to erosion in service Unless the LE is very carefully designed and manufactured it is likely to have a spike of high velocity on the suction surface at the blend point between and LE curve and the blade suction surface. This is illustrated in Fig. 11.

A very fine grid around the LE is needed to capture details of this extremely rapid acceleration and deceleration, even for inviscid flow. A laminar boundary layer grows from the stagnation point but this separates after the peak suction point of the spike and the subsequent turbulent reattachment causes a turbulent boundary layer to exist over the remainder of the suction surface. The separation and reattachment is greatly influenced by unsteady interaction with wakes from upstream blade rows. The boundary layer that grows from the stagnation point to the point of peak suction in the spike is so thin that it is unlikely to be resolved by RANS type methods yet it has a critical influence on the downstream boundary layer growth and consequently on the overall blade loss. Goodhand & Miller (21) show that the difference between the profile loss of a blade with a small spike and one with a larger one (diffusion factor > 0.1) is about 30%, equivalent to over 1% change in compressor efficiency. It is extremely unlikely that this can be predicted by CFD although integral type boundary layer solvers may be able to approximate it. A more important limitation is that, in service, the actual leading edge shape is likely to be unknown.

5.3 Surface roughness. Surface roughness may be considered as one aspect of geometry, it can have a large influence on boundary layer growth and loss at high Reynolds

numbers and also influences boundary layer transition. A typical value of surface roughness for a newly machined blade is in the range 1-5 microns. If the roughness is known and the boundary layers are fully turbulent its effect on the skin friction can be reasonably well predicted by using modified wall functions, as described by Gbadebo et al (22). Fig. 12 illustrates some predictions of overall loss coefficient, i.e. profile + endwall loss, using this approach for a high pressure steam turbine stator blade, assuming fully turbulent boundary layers. Although the behaviour at lower Reynolds numbers, where transition is likely, may not be representative of the true variation, the trends at $Re > 10^6$ should be correct. This blade has a chord of 90mm and operates with a Reynolds number of order 5×10^7. When new its roughness height is likely to be of order 5 microns and so it can be seen from Fig. 12 that its performance is dominated by surface roughness, blades with a smaller chord will be even more affected. Although the roughness may be known when new it will increase considerably in service and the actual operating value is unlikely to be know. This is one of the main reasons why the efficiency of

Fig. 11.   Flow around a thin leading edge.

Copyright © 2010 by ASME

many turbomachines deteriorates with service.

In addition to its effects on the skin friction Gbadebo et al show large effects of roughness on 3D separations. They used a roughness height of 25 microns in their large low speed compressor, Re = 2.7 x 10^5, which they say is equivalent to a typical in-service value of 1.5-2.0 microns in an aero engine compressor.

Most CFD calculations assume smooth surfaces. Whilst this is reasonable for new blades at low Reynolds numbers, say Re < 10^6, it is certainly not representative of many in-service situations. In particular, high pressure stages of both steam and gas turbines are likely to have significantly lower efficiency than such calculations would suggest.
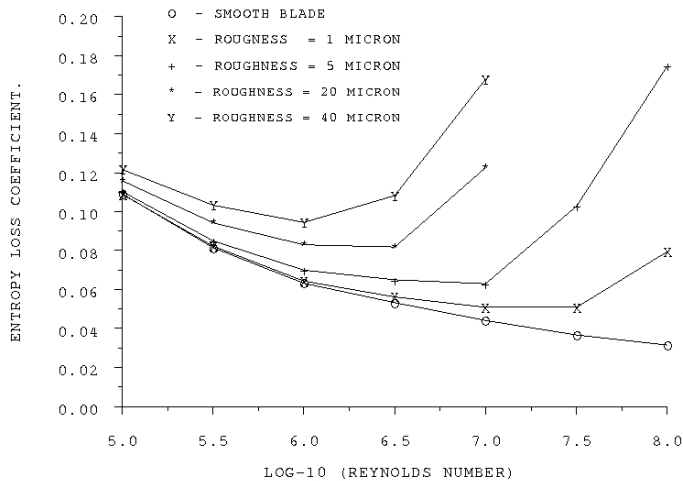

Fig. 12. Predicted effect of surface roughness on the loss of a turbine stator with chord = 90mm .

6.   COMPRESSOR STALL

Prediction of the stalling point of a compressor is perhaps the most challenging task to which CFD is applied. The stalling of a compressor is subject to most of the unknown and imprecise factors discussed in this paper. It is influenced by tip clearance, tip erosion, blade untwist, turbulence modeling, transition modeling, surface roughness, unsteady flow and by the upstream and downstream ducting. In particular stall is an inherently unsteady process involving many blade passages and even the whole annulus. Despite this it is usually predicted by steady calculations and is taken to be the point at which the calculation will no longer converge. The failure is usually due to the growth of a large separation and is therefore heavily dependent on the turbulence modeling. It is usually found that the CFD failure occurs at a higher mass flow and lower pressure ratio than is found experimentally. The author suspects that this is because the free stream turbulence levels are greater than those predicted by CFD and these help to restrict the growth of separations.

These difficulties make accurate prediction of the stall point of even a single stage compressor beyond the capability of most CFD. It will be necessary to perform whole annulus unsteady calculations with a fine grid, known geometry, known inlet conditions and excellent turbulence modeling to get accurate results. For a multistage compressor the problem is even more challenging since the errors are cumulative, any error in predicting the pressure ratio or blockage of one stage, say due to incorrect tip leakage flow, will alter the matching of the other stages and may either increase or reduce their loading. Hence stall, i.e. breakdown of the calculation, may be triggered in the wrong stage. Given these difficulties it is surprising that CFD does as well as it does in predicting multi-stage compressor behaviour.

To illustrate the difficulties of this task and to show how wrong CFD can be, Fig. 13 shows predictions for a single stage medium speed compressor by 2 different widely used CFD codes. The predictions were obtained by a respected organisation, which was not the source of either the data or the codes. Code A, if run in steady mode, gets the characteristic reasonably correct at high flow coefficients but fails well before stall. In unsteady mode it is able to get further up the characteristic with fairly good accuracy, but still fails before the experimental stall point. Code B gets the pressure rise at a given mass flow far too low and also fails to reach the measured stall point, this is most likely due to it producing too much blockage from either blade surface or endwall separations.
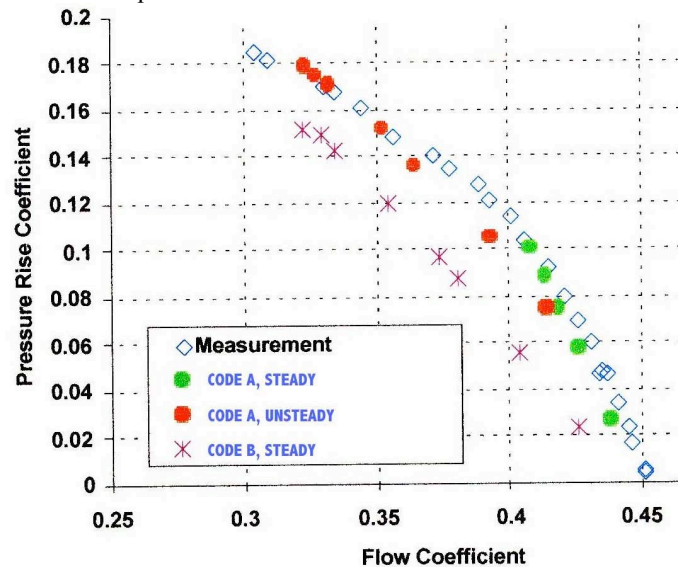

Fig. 13. Stall predictions for a single stage compressor.

CONCLUSIONS

This paper has tried to show that predictions of turbomachinery performance using CFD are subject to a good deal of uncertainty. Some of the errors are due to the limitations in modeling in the CFD itself and some are due to unknown geometry and unknown boundary conditions. Predictions of blade row loss and machine efficiency

together with compressor stall margin are the quantities most likely to be in error and any CFD used for to predict these should be calibrated by comparison with test data as frequently as possible. Hence the availability of CFD should not be used as an excuse for reduced experimental testing, rather the testing should be designed to calibrate the CFD rather than to model one particular design.

The modeling errors have reduced over the years as CFD has developed and no doubt they will continue to do so. However, it seems unlikely that a universal turbulence or transition model will be developed until DNS calculations become a design tool. Some geometrical errors, such as seal and tip clearances, may be reduced by better manufacturing and assembly tolerances but some, such as erosion and operating levels of roughness, will always remain unknown. If modeling and geometry errors can be reduced some boundary condition errors will also reduce because they can be more accurately predicted by CFD on the upstream stages. However, turbulence levels, especially that leaving a combustor, are always going to be subject to a great deal of uncertainty.

Despite these limitations CFD remains an extremely valuable tool for turbomachinery design and the author would certainly not suggest that less use should be made of it. However, it must be used on a comparative basis rather than as an absolute predictor of performance. For example it is possible to use CFD to compare two designs at the same tip clearance even if the operating clearance is not known and the CFD may then be used to examine the sensitivity to tip clearance. Perhaps the most important result of CFD is an improved ability for the user to understand the flow physics, this requires the solutions to be studied in detail rather than just looking at a few overall performance numbers. Given such understanding it is usually possible to improve undesirable flow features even when quantitative predictions of the improvement cannot be trusted.

REFERENCES

1. Casey, M.V. Third state of the art review for thematic area 6, CFD for turbomachinery internal flows. QNET-CFD. 2004.
2. Denton, J.D. Lessons from rotor 37. J. of Thermal Science. Vol 6, No.1, March 1997.
3. Shabbir, A. Celestina, M.L. & Strazisar, A.J. 1997. The effect of hub leakage on two high sped axial flow compressor rotors. ASME paper 97-GT-346
4. Dunham, J. & Meauze, G. 1998. An AGARD working group study of 3D Navier-Stokes codes applied to single turbomachinery blade rows. ASME paper 98-GT-50.
5. Woollatt, G. Lippett, D. Ivey, P.C. Timmis, P. & Charnley, B.A. 2005. The design, development and evaluation of 3D aerofoils for high speed axial compressors. Part 2: Simulation and comparison with experiment. ASME paper GT2005-68793.
6. Pecnik, P. Pieringer, P. & Sanz, W. 2005. Numerical investigation of the secondary flow of a transonic turbine stage using various turbulence closures. ASME paper GT2005-68754.
7. Mayle, R.E. 1991. The role of laminar-turbulent transition in gas turbine engines. ASME paper 91-GT-261.
8. Halstead, D.E. Wisler, D.C. Okiishi, T.H. Walker, G.J. Hodson, H.P. & Shin, H.W. 1997. Boundary layer development in axial compressors and turbines. Pts 1-4. ASME J Turbomachinery, vol 119.
9. Coull, J.D. Thomas, R.L. & Hodson, H.P. 2008. Velocity distributions for low pressure turbines. ASME paper GT2008-50589.
10. Harrison, S. 1990. The influence of blade lean on turbine losses. ASME paper 90-GT-55.
11. Holley, B.M. Becz, S. & Langston, L.S. 2005. Measurement and calculation of turbine endwall pressure and shear stress. ASME paper GT-2005-68256.
12. Denton, J.D. 1990. The calculation of three-dimensional viscous flow through multistage turbomachines. ASME paper 90-GT-19.
13. Holmes, D.G. 2008. Mixing planes revisited: A steady mixing plane approach designed to combine high levels of conservation and robustness. ASME paper GT-2008-51296.
14. Adamczyk, J. J. 2000. Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design, ASME, Journal of Turbomachinery, Vol. 122, April 2000.
15. Praisner, T.J. Clark, J.P. Nash, T.C. Rice, M.J. & Grover, E.A. 2006. Performance impacts due to wake mixing in axial flow turbomachinery. ASME paper GT2006-90666.
16. Ning, W. & He, L. 2001. Some modeling issues on trailing edge vortex shedding. AIAA Journal. Vol 39, No 5.
17. Walsh, J.A. & Gregory-Smith, D.G. 1989. Inlet skew and growth of secondary losses and vorticity in a turbine cascade. ASME paper 89-GT-65.
18. Ong, J. & Miller, R.J. 2008. Hot streak and vane coolant migration in a downstream rotor. ASME paper GT2008-50971.
19. Aalburg, C. Simpson, A. Schmitz, M.B. Michelassi,V. Evangelisti, S. Belardini, E. & Ballarini, V. 2008. Design and testing of multistage centrifugal compressors with small diffusion ratios. ASME paper GT2008-51263.
20. Reid, K. Denton, J.D. Pullan, G. Curtis, E.M. & Longley, J.P. 2005. The interaction of turbine inter platform leakage with the mainstream flow. ASME paper GT2005-68151.
21. Goodhand, M.N. & Miller, R.J. 2009. Compressor leading edge spikes: A new performance criterion. ASME paper GT2009-59205.
22. Gbadebo, S.A. Hynes, T.P. & Cumpsty, N.A. 2004. Influence of surface roughness on three-dimensional separation in axial compressors. ASME paper GT2004-53619.