Some Lessons Learned

N. A. Cumpsty
Imperial College,
London, UK SW7 2AZ

This paper looks at some familiar results, concepts, and ideas in a different way, drawing out some common unifying threads. The topics addressed are one-, two-, and three-dimensions; axisymmetry and its breakdown; gaps and leakage paths; and computation fluid dynamics (CFD) and the limitation of perfection. Certain overlapping issues come up in each of the topic areas. [DOI: 10.1115/1.4001222]

Preface

This paper is based on an invited lecture for a session organized to honor of Dr. Leroy Smith during the 2009 ASME Gas Turbine Technical Congress and Exposition in Orlando Florida. I had felt privileged to be able to contribute in this way and because I had learned so much over my career from Dr. Smith, I chose to devote the lecture to some of the lessons I had learned. The lessons are not restricted to things learned from him, but from others as well. The content then is not wholly new, but what is original is the way material is drawn together.

1 Introduction

The topics of the various sections will overlap somewhat, but the groupings are as follows:

- One, two, and three-dimensions
- Axisymmetry and its breakdown
- Gaps and leakage paths
- Computational fluid dynamics (CFD) and the limitation of perfection

The vehicle for discussing these topics will generally be a compressor or aircraft engine fan.

2 One-, Two-, and Three-Dimensions

2.1 Two-Dimensions. When one is introduced to turbomachinery as a student it is presented in two-dimensions and the early pioneers, such as Parsons or Stodola, presented two-dimensional drawings in what one would now call the blade-to-blade surface. Later it was realized that radial effects were important and another two-dimensional surface was introduced, the meridional plane, see Fig. 1. This approach of two perpendicular surfaces was known to be merely an approximation and the work of Wu [1] set out to remedy this, but at an unacceptable level of complexity.

The two-dimensional blade-to-blade surface remains an indispensiable part of the thinking of people working in turbomachinery. Through the velocity triangles it is the basis for estimating the work input or output and thence the pressure rise or fall of the blade row and stage. It lends itself to setting limits on the turning and pressure change capacity of the stage, with the diffusion factor being common for compressors and Zweifel coefficients being used for turbines. These continue to be useful and widely used, working well over much of the span of many blades. Although inviscid in concept, the viscous effects are usually added with correlations for loss and for flow angle deviation, though sometimes boundary layer calculations are carried out too.

The two-dimensional meridional plane also remains a key tool in the design and analysis of turbomachines. It is a way of linking the behavior of blades at different spanwise positions, allowing for both the variation in the flow with span including the variation in the blade inlet and outlet flow directions. The calculations are usually referred to as throughflow calculations and the methods have many refinements; they have become key vehicles for recording and then for implementing empirical and semiempirical experience of turbomachine aerodynamics. In many design and analysis systems they are the way that the blade rows and the annulus are considered together, with the spanwise variation incorporated and the shape and curvature of the annulus included. The approach to solving them, usually called streamline curvature, also sometimes gives its name to the meridional throughflow calculation. The treatment is essentially inviscid and, as Adkins and Smith [2] pointed out, this leads to serious errors near the endwalls (where losses are high) in multistage machines. They realized that a process of radial redistribution of loss (entropy) and momentum must take place, and they modeled this radial mixing based on secondary flow in the passages, giving rise to deterministic stresses. Gallimore and Cumpsty [3] recognized the need for radial transport, but, based on experiment, asserted that it was largely turbulent. Wisler et al. [4] used very detailed experiments in the GE low-speed four-stage compressor to show that both[1] effects were present.

Problems with the blade-to-blade treatment arise when there are marked variations in conditions along the span, more specifically when the rate of change in flow properties in the spanwise direction is comparable with or greater than the rate of change in the chordwise direction. Rapid change in the spanwise direction normally occurs near the endwalls. Whereas a compressor blade will normally be designed with an incidence not far from zero, close to the endwall the incidence rises and can greatly exceed values at which *two-dimensional* blades stall, as in a two-dimensional cascade. One of the first attempts to consider blade flows close to endwalls was by Wisler [5] in his notes on compressors. He described ways to modify the blades to allow for the endwall boundary layers, but this was a long time ago and the thinking was still two-dimensional in character. Wadia and Beacher [6] recognized that the endwall regions did not stall in the manner of a two-dimensional aerofoil, and sought to explain the behavior in terms of radial velocities relieving the flow. This was nearer the truth, but was not sufficiently three-dimensional, since it did not recognize that the flow sets itself up differently in three-dimensions and does not need relieving, a point addressed in the journal discussion to Wadia and Beacher [6].

The behavior of the blades near the endwall is not like a two-dimensional stall because the flows in 2D and 3D are quite different. In particular the pressure field near the endwall is established by the bulk of the flow, not just the flow experiencing high inci-

Contributed by the International Gas Turbine Institute (IGTI) of ASME for publication in the JOURNAL OF TURBOMACHINERY. Manuscript received July 26, 2009; final manuscript received July 27, 2009; published online May 11, 2010. Editor: David Wisler.

[1]Radial mixing is given some emphasis here because it was an exceptional period of intellectual competition carried out with great openness and courtesy. Those involved recall it as a stimulating and enjoyable episode a quarter century later.

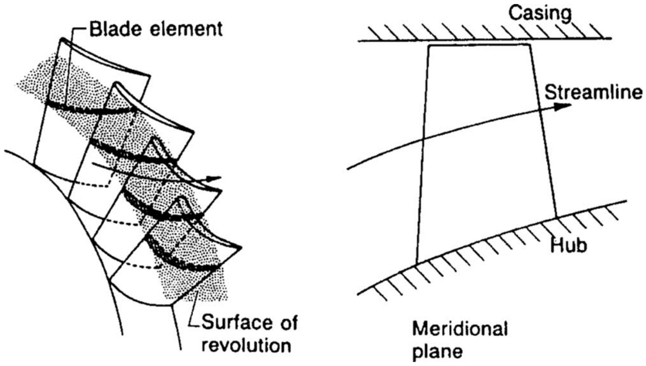

Fig. 1 Two-dimensional representation, blade-to-blade surface of revolution on the left, and meridional (throughflow) plane on the right

dence near the endwall; as a result the pressure distribution is not normally enough to cause a major separation of the two-dimensional kind.

2.2 Three-Dimensions. Although in the endwall region there is not a two-dimensional stall of a blade, some separation in the endwall-blade corner seems inescapable, even for the best designed and lightest loaded blades. To predict this, the flow must be studied in three-dimensions. Figure 2, taken from Gbadebo et al. [7] shows the surface flow patterns on the surface of a cascade, both from flow visualization and from CFD. Perhaps most surprising is the good predictions of CFD, given that the turbulence modeling was simple and a wall function was used. This is explicable because in the limit, as the wall is approached, the flow is laminar and the pattern is a result of the balance between laminar viscous stress and pressure stress, both of which are well predicted by CFD. It is very clear that although there is some separation, it is not the stall expected of a two-dimensional flow. This blade row was operating well and efficiently, with loading well within the normal constraints of the diffusion factor and de Haller number, and the integrated loss was in line with expectations for a good cascade.

A further observation can be made of the flow in Fig. 2: it is very complicated. There are three sorts of singularity in the surface flow (nodes, saddle points, and foci), whereas in 2D there is just one type, flow attachment and separation. In 2D separation can be understood in terms of an adverse pressure gradient producing a greater deceleration of the slower flow close to the wall, but in 3D there is no comparably simple explanation. In 2D separation is frequently a serious disturbance to the flow, often given the name of stall. In 3D separation is generally unavoidable, as in the corner where a blade meets the endwall, but does not represent a catastrophic breakdown or stall.

As shown in Fig. 2, it is possible to predict three-dimensional behavior even close to the wall with 3D CFD, but simple notions of cause and effect, such as those used effectively in 2D, are largely missing. It is essentially impossible to claim to understand

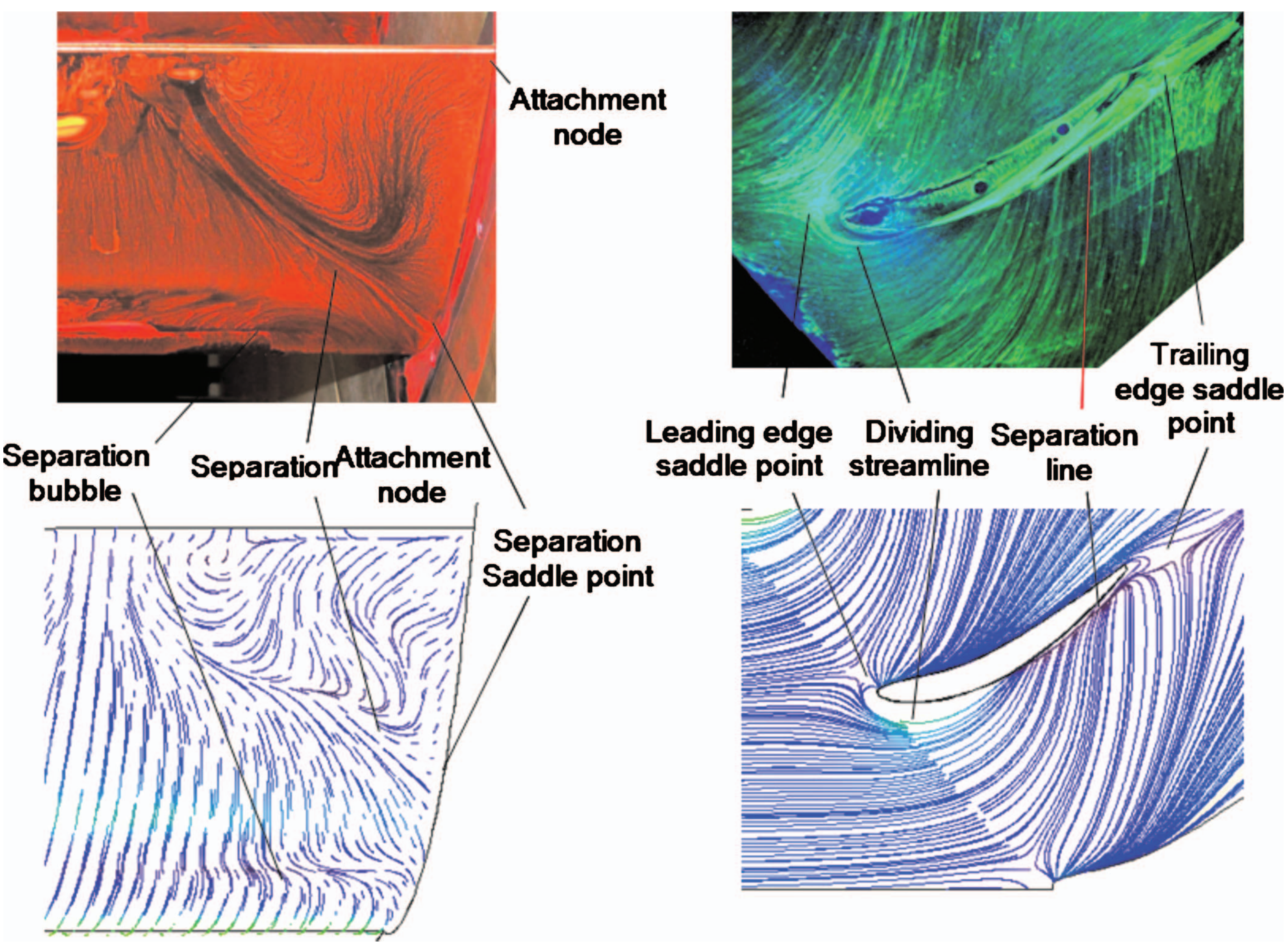

Fig. 2 Surface flow visualization (upper two pictures), and CFD (lower two pictures) for a compressor blade (left) and stationary endwall (right) at zero incidence [7]

3D flow in the way that one can in 2D. An example to support this assertion is the removal of the large corner separation by the presence of a small clearance gap [8]. Another example is the alteration of the corner separation by the presence of roughness on the blade [9]. The three-dimensional separations play an important part in determining the deleterious consequences of leakage through gaps in the endwall into the flow path, discussed below. Small amounts of leakage can have an effect apparently out of all proportion to the quantity of flow because they interact with the the 3D separations'.

Although the flow complexity is most striking for the surface flow pattern, even pressure distributions in 3D are not easily envisaged. For example, in 2D the consequence for the pressure field of the increasing blade camber is generally understood. In three-dimensions, the effect of a camber change over a small part of the span could not be anticipated with confidence, except by a three-dimensional calculation. Modern calculation methods do more than give the pressure distribution, but indicate where the losses were produced and where they were concentrated.

Not long ago the computation of the three-dimensional steady flow in a one blade row, including viscous effects, would have been challenging, but now full three-dimensional calculations of multistage machines are possible, often including unsteadiness. So the problem has shifted—almost anything can be calculated, given the commitment of computer resources, but understanding of the three-dimensional behavior remains a challenge. First, the two-dimensional "rules" for pressure gradient and separation do not apply in three-dimensions. Second, related to this, the three-dimensional flows seem too complicated and diverse for simple causal 3D rules. Acquiring an appropriate understanding in 3D is what aerodynamicists need to strive for with the new tools at their disposal.

There is a related problem: how to turn knowledge about 3D flow into useful guidelines for assessing aerodynamic behavior. This was addressed by Lei et al. [10], who found a parameter capable of post-dicting the change from a relatively benign 3D corner separation, present under all circumstances, to higher-loss, higher-blockage conditions, which are appropriate to describe as corner stall.

2.3 One-Dimension. At first sight it is paradoxical to consider one-dimension after two and three but there are reasons for this, most notably that it tends to get forgotten. Two- and three-dimensional flows in turbomachines are now, in some senses, satisfactorily predictable and serious performance shortfalls are not very likely to arise nowadays from two- or three-dimensional effects. The same is not true of one-dimensional effects. If a compressor or multistage turbine is to have a really substantial shortfall in performance it is likely to be the result of the errors in one-dimensional matching. Because of the growth in blockage in compressors, and the dependence of blockage on the pressure rise, the multistage compressor is the most common component to suffer from this.

The problem of matching a high-speed compressor was addressed in a paper by Adamczyk et al. [11] for a high-speed two-stage compressors. The performance of the compressor was below expectation because of errors in the matching of the stages.

To understand the one-dimensional flow the appropriate variable to use is the corrected mass flow or a nondimensional equivalent $\bar{m}=\dot{m}\sqrt{c_pT_0}/AP_0$, where $\dot{m}$ is the mass flow rate, the pressure and temperature are the stagnation values, and A is the flow effective area (the area after allowing for blockage by the boundary layers). For turbomachines the variation in stagnation temperature T_0 is commonly much less important that the variation in stagnation pressure P_0 and an adequate understanding can be found by considering only pressure. The term $\bar{m}$ is a function of the axial Mach number of the flow and the upper limit on $\bar{m}$ corresponds to the flow choking.

Consider a multistage compressor with N stages. The *outlet* flow from stage n is the *inlet* flow to stage $n+1$. In selecting the annulus height and the blade angles for stage $n+1$, the pressure and temperature rises in stages 1, though N will have been considered to arrive at the intended corrected mass flow into stage $n+1$. If, for example, the flow from stage n has a blockage greater than the value assumed in the design, then the pressure rise it produces will be lower than the design and the corrected mass flow into stage $n+1$ will be higher than the design. As a result stage $n+1$ will produce a lower pressure rise than intended in the design, and, in extreme cases, may choke. Likewise, if the pressure rise in the stages up to stage n exit is higher than the design, then the corrected mass flow into stage $n+1$ will be lower than the design and the stage may stall. For machines operating at comparatively low speeds (so that the flow into each stage is subsonic) there is usually a considerable range in $\bar{m}$ between choke and stall of each stage, but as the speed of the machine is increased the range narrows.

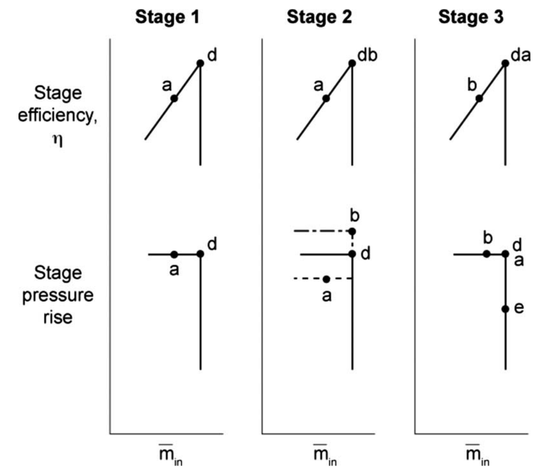

Fig. 3 Idealized stage characteristics for a three-stage, high-speed compressor

Figure 3 shows a highly idealized cartoon of the pressure ratio and the efficiency plotted against the corrected mass flow for a high-speed three-stage machine. In the design intent each stage has the same nondimensional performance when plotted against corrected mass flow into that stage. For this type of high-speed stage, maximum efficiency occurs just about at the "corner" where each speed line turns over at the top. The well matched machine needs to get all the stages close to the corner so that efficiency for each stage is near this maximum, points *d* in Fig. 3. For the third stage the operating point is determined by the outlet throttle, or other component downstream, and a point at lower pressure ratio but the same inlet corrected flow to the stage is shown as point *e*; point *e* can, in other words, move as a result of the throttle along the vertical speedline. The outlet throttle has no effect on the first and second stages, and if these stages are not operating correctly they cannot be corrected by opening or closing the downstream throttle.

Suppose that stage 2 produces a higher pressure ratio than the design intent, but the same inlet corrected mass flow at choke, the dot-dash chain line in Fig. 3. The higher pressure ratio leads to a reduced corrected mass flow at the exit from stage 2, and hence, the operating point for stage 3 moves to point *b*. In this idealized case the efficiency of stage 1 and 2 is not compromised, but the efficiency of stage 3 falls, even though the error is not in that stage but in stage 2; again note that the downstream throttle does not allow this to be corrected.

Suppose now that the peak pressure ratio for the stage is below the design intent, the dotted line in Fig. 3. The corrected mass flow out of stage 2 cannot exceed the choking inlet value for stage 3, and, since the pressure out of stage 2 is below the design intent, the mass flow through stage 2 must be reduced to point a. Since stage 2 is accepting less mass flow than the design intent, stage 1 also has a reduced mass flow. As a result the efficiency of stages 1 and 2 is reduced.

These idealized errors in performance relative to the design intent have been posed only for the pressure ratio in one stage, stage 2, but similar arguments can be used for the pressure ratio, efficiency, and mass flow capacity of all stages. Although not considered here, a drop in efficiency has the effect of raising the temperature out of the stage, which increases corrected mass flow into the next stage downstream.

There was a fashion not long ago to propose, or even to design, commercial engines with a small numbers of highly loaded compressor stages in the expectation of taking out cost (actually, it is not obvious that cost would be removed since each of the stages, notably the disks, would be more expensive). There are a number of reasons why these machines gave trouble, but at least one is the difficulty of matching the stages. One way of looking at this is to think of the effect on the variation in effective area A resulting from the leakage flow and tip clearance variation; errors in A translate into errors in $\bar{m}$, which then cause alterations of the stage pressure ratio and hence into mismatching of the whole machine. With considerable difficulty, matching is achieved for high-performance multistage military fans (say three stages and an overall pressure ratio of 5), but the problems increase as the number of stages and the overall pressure ratio increase (most core compressors for two-spool commercial engines would have pressure ratios in excess of 15). Even if the matching could be achieved in a new compressor on a test bed, the consequences of in-service damage, such as increased tip clearance, are likely to undo the stage matching. As noted above, the most effective way of reducing the efficiency of a compressor by a large amount is to get the stage matching wrong.

The one-dimensional consideration is based on corrected mass flow. As noted above, this includes the effective flow area A. The effective flow area is reduced as the flow blockage is increased, so the three-dimensional effects, such as corner separation, have an a marked influence on performance through this.

3 Axisymmetry and Its Breakdown

Just as one's thinking is determined by the two-dimensional way one learns about turbomachines, experience conditions one to think of the flow as axisymmetric. (The term axisymmetric is taken here to ignore the circumferential variation across the blade pitches). In many cases, assuming axisymmetry is perfectly adequate and it provides a very good working basis for most designs. Below are some examples where axisymmetry is not valid.

3.1 Geometric Asymmetry. Sometimes the annulus into which the engine or machine is placed is inherently nonaxisymmetric and this is true of the fan on the front of most civil engines. First, the intake is drooped and may not be circumferentially uniform. Second the bypass duct has to accommodate one or more pylons, and these create a circumferentially nonuniform pressure field: being a first order nonuniformity (i.e., one per circumference), its effect is felt well upstream so as to affect the fan rotor. Although such nonaxisymmetry is a complication, it is now readily accommodated by three-dimensional CFD encompassing the full annulus.

3.2 Rotating Stall. Of greater familiarity is the way in which compressor flows break down to form the rotating stall. Only rarely do compressors stall in a way that is axisymmetric (a sort of ring stall), and most commonly the flow divides circumferentially into regions of low flow (the stall cells) and unstalled regions in which the flow rate is substantially larger than that at which the stall is initiated. To balance the pressure rise from front to back the pattern of stall cells has to rotate making the flow unsteady in the blade passages. This has been an active area of research, which is unnecessary to go over here—the key point here is the preference of the flow to adopt a nonaxisymmetric pattern. More precisely, at low flow rates the axisymmetric flow is unstable but a nonaxisymmetric flow can be highly stable.

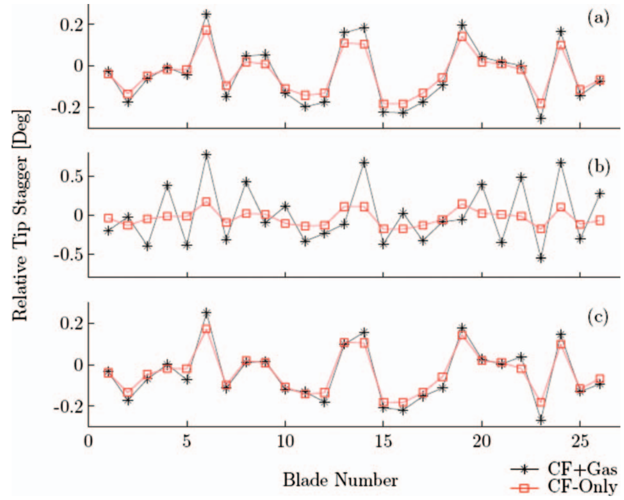

Fig. 4 Blade tip stagger for random variation in stagger without gas loads: (*a*) for flow unstarted, (*b*) for an intermediate condition, and (*c*) for flow started. Note different scale for (*b*) [13]

3.3 Supersonic Rotors. In high-speed compressor stages, those with supersonic relative flow into the rotor, the supersonic blade ideally operates in two ways, corresponding to the different lines in the schematic idealization in Fig. 3. When choked at the maximum corrected mass flow into the stage, the blades are referred to as started and the leading edge shock is attached to the leading edge of the blade. When the corrected mass flow is below the choked value the bow shock stands forward of the leading edge and the flow is referred to as unstarted. The pressure rise and efficiency are predicted to be at their peaks when the flow is on the point of changing from started to unstarted, the "corners" shown in Fig. 3. This location of peak efficiency can be predicted with CFD and was found analytically in 2D by Freeman and Cumpsty [12].

Analysis and CFD normally considers a single blade passage or imposes periodic boundary conditions, so axisymmetry is automatically imposed in the specification of the problem. Although the predicted peak efficiency occurs where the speedline turns and the shock is about to detach from the leading edge of the blades, surprisingly the flow does not seem to operate in this way.

Wilson et al. [13] reported CFD carefully studied at a full set of rotor blades, using a well-established program called AU3D (a cell-vertex, finite-volume unsteady Navier-Stokes method). Because the program is aimed at aeroelastic behavior, the CFD also predicted the force on each blade in the row, the steady aerodynamic deflection of the blades, and the flutter (vibration) stability conditions. The blade geometry was therefore specified after allowing their deformation by centrifugal loads but the additional and consequent aerodynamic deflection was calculated by the program. The variation in aerodynamic deflection gives a measure of the variation in aerodynamic performance in terms of force on each blade. Calculations were carried out first with identical blades, when the predicted flow was axisymmetric, but then calculations were performed including small imposed variations in tip stagger, typically in a range of up to 0.2 deg; the stagger variation is random but constant for all cases considered.

Figure 4 shows the blade stagger variation for two cases. The first, shown in red, is when the only deformation of the blades is from subjected is the centrifugal loads (the centrifugal load being taken as identical for each blade the variation in stagger is that imposed). The second case, shown in black, is when the blades are

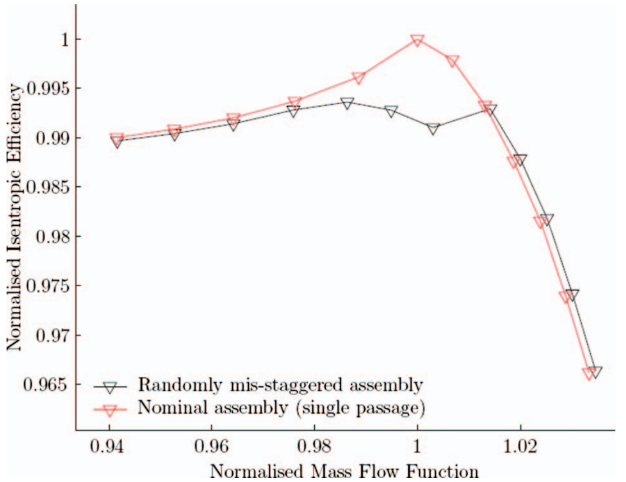

Fig. 5 Efficiency versus normalized flow rate for a supersonic rotor having small levels of misstagger [13]

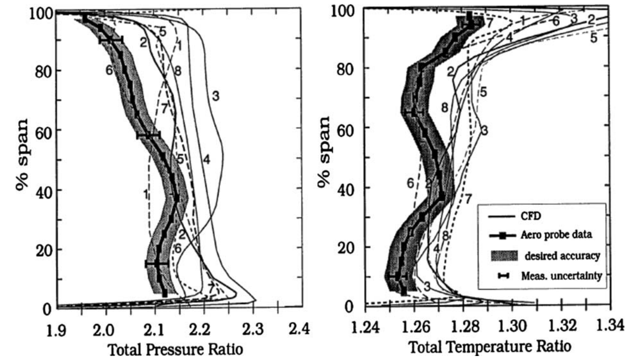

Fig. 6 CFD and measurement of pressure ratio for NASA Rotor 37. The numbers provide a key to the authors of the different codes used [15]

also subjected to the aerodynamic loading and the stagger of each blade is altered by differences in blade pressure distribution. When the blade row is "unstarted" (case a), which, it will be recalled, is at a corrected mass below design (peak efficiency), the deflection of each blade corresponds closely with the imposed centrifugal-only variation. Likewise, when the blade row is "started" (case c), which is for a mass flow higher than the peak efficiency, the deflection again closely follows the imposed variation in blade stagger. However, for the intermediate case, corresponding to the "corner," the variation in blade deflections is substantially greater (note larger scale in Fig. 4(*c*)), and the variation no longer follows the pattern that was imposed. This intermediate case, when the shocks just move ahead of the rotor leading edge, is where the efficiency is predicted to be highest.

The inference from this is that the flow does not actually settle at the intermediate condition where the shocks in all the passages are about to be expelled, which is the intermediate condition which gives the highest efficiency. Instead, it settles to a pattern that it finds more stable, which is nonaxisymmetric with some blades started and others unstarted.

Wilson et al. [13] predicted the efficiencies in two different ways. When the blades are identical at the nominal stagger, the efficiency can be found from a single-passage calculations, Fig. 5. This figure also shows results when the full assembly is calculated with the random variation in stagger no bigger than 0.2 deg. As Fig. 5 shows, the two calculations for efficiency agree well when the row is either fully started or fully unstarted. For the intermediate case, however, the non-axi-symmetric rotor has a lower efficiency because the flow with blades of variable stagger is able to split between some being started and some being unstarted. (It should be noted that in Fig. 5 the scale of the abscissa is very large, so the variation in mass flow is quite small).

Rotating stall and supersonic blades are just two examples where nonaxisymmetric behavior is the norm for compression systems. It is probable that many annular diffusers also operate this way, but begs the question, "Do turbines always operate axisymmetric?" Perhaps turbine blades which require very special circumstances, for example, precise deceleration on the uncovered rear portion of the blades, may find it more stable to adopt a nonuniform pattern.

4 Gaps and Leakage Paths

One obviously cannot get rid of all the gaps in the annulus walls of a real engine. There have to be axial gaps between the rotors and stators, and some of these gaps will vary as the temperature and loads change. Normally the gaps will have been made as small as possible and any net flow into the well under the gap is reduced to a minimum—at this point about all that can be done is that the gap can be moved axially to where it does least harm.

In 1990s, at the instigation of Denton [14], the ASME/IGTI organized a blind CFD test case of NASA Rotor 37. The monumental task this entailed was handled by Strazisar and others in NASA Glenn Research Center. A comparison summary of the measurement and calculation is shown in Fig. 6. Because this was a well tested NASA rotor there was considerable confidence in the measurements and in the geometry—things like rotor untwist were well understood. The remarkable thing about the comparison in Fig. 6 is the similarity between many CFD results, with pressure ratio rising toward the hub, and considerable difference between almost all the CFD and the measurements, with the latter having a dip toward the hub.

Because most CFD pointed to the same trend, the team at NASA Glenn explored effects of turbulence modeling in the CFD, since this is a recognized weakness, but were unable to reproduce differences of the size shown with plausible turbulence models. As a result they went back to the rig and concluded that they should check the effect of the clearance ahead of the rotor, which was 0.75 mm or 1.8% of the rotor root axial chord. This investigation [16] was both experimental and computational. The experiments varied the amount of net leakage through the gap and also the effect of a gap over an enclosed volume with no net leakage. The experimental results with different levels of clearance and net leakage are shown in Fig. 7. The CFD was able to vary the input in a somewhat more controlled way and was able to explore the impact on calculation of leakage.

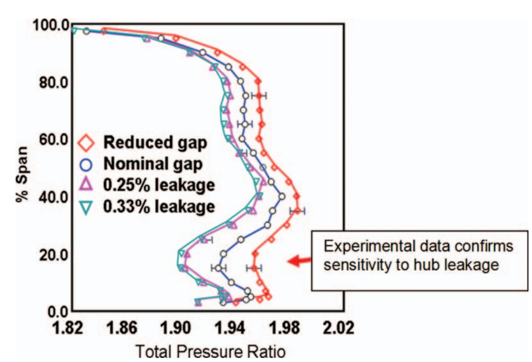

Fig. 7 Variation in rotor hub leakage and its effect on stagnation pressure ratio across the rotor [16]. Reduced gap and nominal gap refer to zero net flow from the gap; other cases are for net outflow.

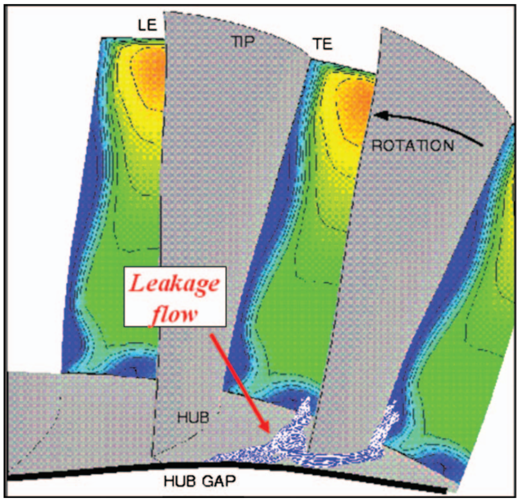

Fig. 8 Computed flow in NASA Rotor 35 showing leakage flow entering hub separation region [16]

The effect of a gap with no net leakage is evidence that flow goes into the slot where the outside static pressure is high, and emerges where the outside static pressure is low. This emerging flow became involved with the complex three-dimensional flow separation near the suction surface of the blade close to the hub wall, as shown in Fig. 8. This, of course, is a wholly three-dimensional effect, which would not be anticipated by any two-dimensional consideration—indeed the thinking about the leakage and three-dimensional behavior was altered by this experience. The sensitivity of the 3D separated region to the ingress of flow from the leakage far exceeded what most people had imagined.

This combined CFD exercise and experimental study together form one of the most important lessons in turbomachinery. It was one of the first examples where the experiments were found to be in error[2] because of the predictions of CFD, but it also showed incontrovertible evidence of the deleterious nature of gaps in the endwalls. Since the time of this work computers have become bigger and cheaper and CFD has become better as well, so calculating the flow to include the primary flow and the out-of-flow-path regions (i.e., the flow in gaps and slots) together is possible. Probably the primary limitation now on predicting the effect of gaps and seal leakage flow is knowing the details of geometry in hot running engines.

5 CFD and the Limitation of Perfection

This is written by someone who is an appreciator of CFD rather than an expert and who, during his career, has seen it progress from two-dimensional inviscid calculations around blades through Euler calculations in 3D to unsteady Navier–Stokes solutions in three-dimensions for multistage machines. At each stage it was a wonder and one strove to make the most of the results. Quite often, if there was an experiment for comparison, the agreement would not be very good and the CFD was then always suspected. As discussed in Sec. 4, this faith in the experiment but suspicion of CFD may sometimes have been misplaced. Particularly, if several different CFD codes predict trouble it is probable that something is not right.

In the limit one can carry out a thought experiment in which perfect CFD is employed. The grid is refined to the point where no alteration takes place with further refinement. The turbulence is modeled in a manner that fully accounts for all effects, so errors due to this are negligible. The switch from the stationary frame of reference for stators to the moving frame of reference for rotors is carried out accurately and without the erroneous introduction of mass, enthalpy, or entropy. The upstream and downstream boundary conditions perfectly match the experimental setup. The running geometry is known and correctly input. What one has is a perfect analog for the experimental setup. This may not be altogether a good thing.

In the case of a newly designed multistage compressor there is a considerable risk that some aspect will not be right—very likely this is the matching between some of the stages. As already noted, getting the matching right (the desired nondimensional mass flow rate) at each stage is difficult and is often at first in error. How will the perfect CFD respond to this? Probably by failing to converge or else converging to some different flow pattern from the one intended. If there were serious mismatching, one or more stages might be in the rotating stall and the perfect CFD would predict this. Unfortunately, knowing correctly from high-resolution CFD that the compressor as designed would be stalled may give very little help in sorting out the underlying problem.

If 3D CFD is to be used in the early stages of design, then CFD of quite low resolution and fidelity is appropriate; what is needed is a converged solution of sufficient accuracy to show where the problem lies. In the case of the compressor design, which high-fidelity CFD predicts to be stalled, the designer wants to know what blade rows or stages would be in trouble if the compressor were somehow able to operate unstalled. In the hands of an expert user the robust but less accurate calculations can lead to corrections in geometry and then rapidly to refined and precise calculations. (It is also true that some highly effective design systems still rely on a good two-dimensional throughflow method, incorporating a body of empirical knowledge). Such a 3D CFD design method would have a relatively coarse mesh and would be quick; its robustness would come from using a high level of turbulent stress. Only when the robust CFD has allowed the machine to be matched adequately can the more refined CFD be used to estimate efficiency and to carry out more subtle optimization of the geometry. It appears to be a general rule, borne out by 3D CFD, that one needs not one tool but several, each having different strengths.

6 Conclusions

1. CFD has transformed the way that turbomachinery aerodynamic design, development, and analysis are carried out. It has even led to an alteration in the way one thinks about the aerodynamics because one can now look at three-dimensional effects or unsteady behavior in a way that would have been impossible only a few years ago. What is most beneficial is that there is no longer a need to be trapped by two-dimensional steady thinking about flows that are inherently three-dimensional and unsteady.
2. Flow in 3D is very different from 2D. We have almost no understanding of the 3D flow, and intuition gives little guidance. In 3D regions of flow separation are normal and necessary, but not necessarily very damaging in terms of loss. CFD appears to be remarkably good at predicting the complicated 3D flow patterns.
3. Having drawn attention to the possibilities for three-dimensional and unsteady analysis, it is fair to say that the most serious deficits in performance in high-speed machines probably have a one-dimensional character to them. If efficiency is well down on expectation, it is likely that somehow the matching between stages, or even between blade rows, has gone wrong.
4. Stage matching is a one-dimensional idea based on corrected mass flow. Since corrected mass flow depends on pressure rise and on blockage, the corrected mass flow is itself dependent on the three-dimensional details. Raising the flow

[2]Describing the experiments as in error is deliberately provocative. However, the interpretation of the experimental results and the assessment of what geometric features of the rig were important were certainly in error. Arguably so too was the CFD modeling, since no one bothered, or was able, to include the clearance gap in their calculations.

Mach number stage pressure ratio, and overall compressor pressure ratio inevitably increases the difficulty of matching compressor stages and maintaining the match under various operating conditions.

5. It is possible to focus too much on the shape of the annulus and the blades, and overlook gaps. The flow from a gap usually occurs where the static pressure is low, and this flow can add to the complex three-dimensional separation in the endwall corner. The effect can be out of all proportion to the open area of the gap, as was demonstrated so effectively by the NASA Rotor 37 test. Serious damage can occur to flow performance even when there is no *net* flow out of the gap, but flow enters in regions of higher pressure and leaves in regions of lower pressure.
6. Robust CFD, which can be used to calculate the flow in machines at an early stage of design or development, is necessary. This does not need to be high fidelity, nor should the assessment of usefulness be wholly based on the accuracy of prediction—the key purpose of the robust CFD is to identify where there are defects so that the machine can be adequately matched. When the obvious defects have been corrected there is an opportunity for more refined CFD, which would probably have failed, or predicted the stalled flow, if used on the uncorrected early designs.
7. For a company or organization CFD is a tool and how they verify it is only an issue for them. When results are published based on CFD some way of assessing or calibrating its accuracy is necessary. What is required depends on the use made of the computations in the publication, but some code of practice needs to evolve rather urgently.
8. Experiments are definitely still needed, but the objectives have shifted somewhat over the years. Collecting loss, turning, and pressure rise for two-dimensional cascades, to take an extreme example, is not often needed now, but even for these there can be some surprises around such aspects as transition. Since the modeling of turbulence and transition is still weak in two-dimensional flows, and for complex three-dimensional flows virtually nonexistent, experiments still have a place. Because the computed results are now so much more reliable than they used to be, the experiments will also be required to be more precise and more closely targeted to the specific questions to be answered.
9. With few exceptions it is no longer interesting to present a measurement for a compressor or turbine without very clear information on how the machine was designed and without sufficient geometric definition that the measurements can be used to test CFD. Nor is comparison of CFD with data necessarily interesting either—what is required is CFD and experiment targeted to understand some specific issue, as exemplified in the untangling of the effect of leakage by Shabbir et al. [16].

Acknowledgment

My gratitude to Dr. Leroy Smith was the motivation underlying this paper but I am also grateful to many other people for establishing the knowledge on which this paper is based and direct assistance in the preparation of the paper. Of great importance are colleagues and former students, particularly in the Whittle Laboratory, in the MIT Gas Turbine Lab, and in Imperial College. I would like to single out a small number who have particularly helped me. Chris Freeman, who until he retired worked for Rolls-Royce, has taught me a great deal over many years and was an important contributor to the success of the Whittle Laboratory. Dave Wisler, who until he retired worked for General Electric, has helped me on many occasions, and in many capacities. My debt to Ed Greitzer, often expressed, is familiar to many. I have gained much from a number of people from NASA Glenn, but in the context of the present paper special mention is due to John Adamczyk and Tony Strazisar.

References

[1] Wu, C. H. A., 1952, "General Theory of Three-Dimensional Flow in Subsonic or Supersonic Turbomachines of Axial-, Radial- and Mixed-Flow Type," Paper No. NACA TN 2601.
[2] Adkins, G., and Smith, L. H., 1982, "Spanwise Mixing in Axial Flow Turbomachines," ASME J. Eng. Power, **104**, pp. 97–111.
[3] Gallimore, S. J., and Cumpsty, N. A., 1986, "Spanwise Mixing in Multistage Axial Flow Compressors Part 1: Experimental Investigations," J. Turbmach., **108**, pp. 2–9.
[4] Wisler, D. C., Bauer, R. C., and Okiishi, T. H., 1987, "Secondary Flow, Turbulent Diffusion and Mixing in Axial-Flow Compressors," J. Turbomach., **109**, pp. 455–482.
[5] Wisler, D. C., 1988, Advanced Compressor and Fan Systems, GE Aircraft Engines, Copyright.
[6] Wadia, A. R., and Beacher, B. F., 1990, "Three-Dimensional Relief in Turbomachinery Blading," J. Turbomach., **112**, pp. 587–598.
[7] Gbadebo, S., Cumpsty, N. A., and Hynes, T. P., 2005, "Three-Dimensional Separations in Axial Compressors," ASME J. Turbomach., **127**(2), pp. 331–339.
[8] Dong, Y., Gallimore, S., and Hodson, H. P., 1987, "Three-Dimensional Flow and Loss Reduction in Axial Compressors," ASME J. Turbomach., **109**, pp. 354–361.
[9] Gbadebo, S., Hynes, T. P., and Cumpsty, N. A., 2004, "Influence of Surface Roughness on Three-Dimensional Separations in Axial Compressors," ASME J. Turbomach., **126**, pp. 455–463.
[10] Lei, V.-M., Spakovszky, Z. S., and Greitzer, E. M., 2008, "A Criterion for Axial Compressor Hub-Corner Stall," ASME J. Turbomach., **130**, p. 031006.
[11] Adamczyk, J. J., Hansen, J. L., and Prahst, P. S., 2007, "A Post Test Analysis of a High-Speed Two-Stage Axial Flow Compressor," ASME Paper No. GT2007-28057.
[12] Freeman, C., and Cumpsty, N. A., 1992, "Method for the Prediction of Supersonic Blade Performance," J. Propul. Power, **8**(1), pp. 199–208.
[13] Wilson, M. J., Imregun, M., and Sayma, A. I., 2007, "The Effects of Stagger Variability in Gas Turbine Fan Assemblies," ASME J. Turbomach., **129**, pp. 404–411.
[14] Denton, J. D., 1996, "Lessons Learned From Rotor 37," *Proceedings of the Third International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows (ISAIF)*, Beijing.
[15] Suder, K. L., 1996, "Experimental Investigation of Flow Field in a Transonic Axial-flow Compressor With Respect to Development of Blockage and Loss," Ph.D. thesis, Case Western Reserve University, OH.
[16] Shabbir, A., Adamczyk, J. J., Strazisar, A. J., and Celestina, M. L., 2007, "The Effect of Hub Leakage on Two High Speed Axial Flow Compressor Rotors," ASME Paper No. 97-GT-346.