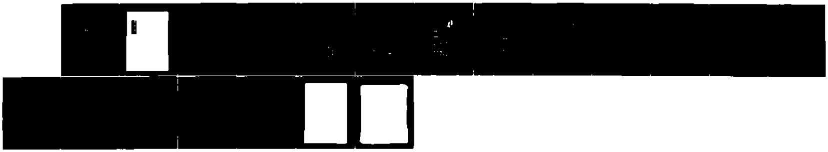
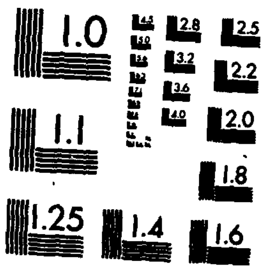
AD-A181 034 THE ROLE OF FLOW FIELD COMPUTATION IN IMPROVING TURBOMACHINERY(U) ROYAL AIRCRAFT ESTABLISHMENT FARNBOROUGH (ENGLAND) J DUNHAM JUN 86 RAE-TM-P-1086 1/1

UNCLASSIFIED DRIC-BR-99862 F/G 13/7 NL





MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

AD-A181 034

ROYAL AIRCRAFT ESTABLISHMENT

Technical Memorandum P 1086

Received for printing 9 June 1986

THE ROLE OF FLOW FIELD COMPUTATION IN IMPROVING TURBOMACHINERY*

by

J. Dunham

SUMMARY

A historical review is presented of the influence of computational fluid dynamics on the development of UK compressors and turbines. The ability to predict pressure distributions has led to increasingly successful attempts to tailor aerofoil shapes in such a way as to optimise performance. Once proven, new computer programs have rapidly been put to use by Rolls-Royce and by some non-aeronautical firms. Examples are given of improvements achieved by their application.

The present state of the art is assessed, and the prospects for future computational developments are discussed. The need for detailed experimental test cases is emphasized. The economical representation of viscous effects remains a key difficulty, especially when heat transfer predictions are needed. (Great Britain)

* *ICAS Paper No. ICAS-86-3.7.3 for presentation at the 15th Congress of the International Council of the Aeronautical Sciences, London, 10 September 1986.*

LIST OF CONTENTS

1 Introduction

One of the expanding areas of scientific research in the 1980's has been computational fluid dynamics. CFD presents a most stimulating intellectual challenge to the mathematically-gifted, and offers a way for the fluid dynamicist to develop his art despite the increasing cost and timescale of high speed aerodynamic experiments. But does CFD really help the designer produce better engineering products, or reduce development time and cost?

The purpose of this paper is to give a clear affirmative reply to that question, based upon reviewing the history of aeroengine turbomachinery research in the UK over the past 25 years. It is true that other fields can be seen in which CFD has not yet demonstrably helped. A study of the particular circumstances of this history leads to some general conclusions about conditions needed for the successful development and exploitation of CFD.

The paper is written from the point of view of a "user" of turbomachinery CFD rather than a developer of it; no attempt is made to review the analytical and computational schemes used, or to make judgements about their relative scientific merits.

It is important to note that in citing this UK case history, it is not suggested that the UK has uniquely achieved CFD success. Similar papers might well be written about corresponding experience in other countries. Graham Adamczyk, and Rohlik(1) have already done so for the USA.

The paper begins by outlining the broad pattern of evolution of aeroengine turbomachinery. A historical review of the development of turbomachinery CFD methods follows. The role these methods have played in improving aeroengines is then described, with examples; further examples are taken from turbomachinery in applications other than aero engines. After comments on this history, the future needs and possibilities of turbomachinery CFD are considered.

2 The Pattern of Aeroengine Evolution

It is first necessary to observe the nature of aero engine development, which is significantly different from that of aircraft or of power generation plant. An engine is first designed using the best technology available at the time, and some prototypes are made and tested. Despite the best efforts of the designers, these prototypes exhibit defects in the form of premature failures or inadequate performance. For several years, intensive development then follows, correcting defects until a formal type approval test is passed and engines are put into service. The development process does not end there. Progressive improvements in both durability and performance continue to be tried out on development engines, and, if appropriate, incorporated into production. Usually, the weight of an aircraft increases in successive versions as more passengers are provided for or more weapons carried; this requires more power from the engines. The economics of both civil and military aircraft operations justify far more changes in engine blading than could ever be justified in, say, the design of an aircraft wing. In the power generation industry, the quantities of steam turbines or gas turbines sold are not sufficient to justify a development programme of the aero-engine type. It is clear, therefore, that aero-engines have much more frequent opportunities to introduce new technology than many other fluid dynamic devices.

Looking back at the history of aeroengine turbomachinery in the UK, it is possible to identify a small number of major advances in compressor technology and turbine technology which, in combination with new materials and manufacturing processes, have enabled engine standards to move ahead. In between these steps, the progressive refinement of individual designs has improved them to the limit of the current technology. Development is the process of climbing up the asymptotes; research is the process of moving the asymptotes themselves.

The broad pattern of evolution of axial compressor technology in the UK is illustrated in Fig 1. Both the research stream (on the left) and the development stream (on the right) started from the early work of Griffith[2], Constant[3], and Howell[3] in the 1930's and 1940's. The major achievement of the 1950's was the improved understanding of stage matching, stall, rotating stall, and blade vibration on which multi-spool engines were based. The next major achievement was the evolution in the late 1960's of the streamline curvature method for analysing meridional flow (of which more later) which allowed designs to be prepared along streamlines, and so enabled transonic fans to be designed and developed to far higher standards of efficiency than previously achieved. No major new aerodynamic advances came after then until the 1980's when the ideas of "end bends" (local profile modifications near end walls) and supercritical profiles were first applied and when it became possible to include viscous effects in blade-to-blade calculations.

The origin of UK centrifugal compressors (Fig 2) lies in the superchargers of aircraft piston engines between the wars. These were developed by Whittle's team to a state of reliability and, for the time, high efficiency[3]. A major step forward was made possible in the 1960's by the evolution of methods for the analysis of inviscid, meridional flow and these formed the basis of considerable advances, especially in the USA and Canada. Swept-back blades were introduced.

Turbines for early jet engines (Fig 3) were designed using steam turbine practice, but adapted for vortex flow by Whittle and others. The first major research advance arose from Ainley and Mathieson's performance prediction method[4], because it enabled optimum velocity triangles to be selected for the specified duty. The next significant advance came when methods of calculating the velocity distribution around the blade surfaces were employed. Then later came the streamline curvature through-flow method, particularly helpful for turbines with highly flared annulus lines. New advances are now emerging in the 1980's, in the form of end wall shaping or blade stacking changes with the object of reducing secondary losses.

Looking back, many of the major advances in turbomachinery technology over the last thirty years have occurred when designers were able to estimate pressure distributions on wetted surfaces, and so express quantitatively the intuitive understanding of turbomachinery flows which research workers developed first. The methods for doing so will now be reviewed.

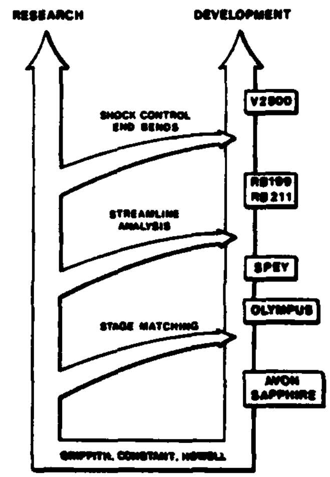


FIGURE 1. Axial Compressor History

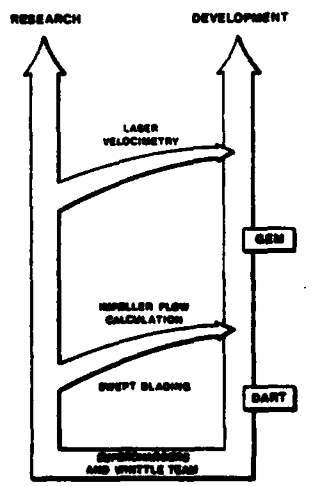


FIGURE 2. Centrifugal Compressor History

TM P 1086

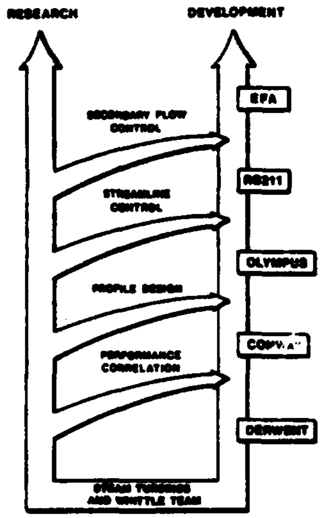


FIGURE 3. Axial Turbine History

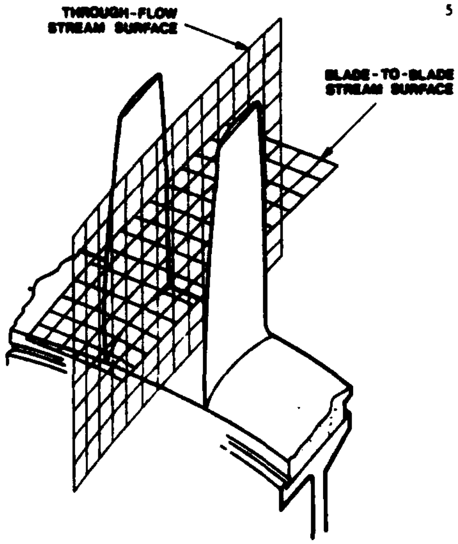


FIGURE 4. The Quasi-Three-Dimensional Approach

3 Computational Fluid Dynamics of Turbomachinery

Turbomachinery has always been designed on a "quasi-three-dimensional steady flow" basis (Fig 4). The design process starts by choosing a "through-flow" pattern on a meridional plane - a cross-section of the engine including the axis of rotation. The through-flow is conceived as a circumferential average of the real flow, or as a typical stream surface passing between the blades. Secondly, the design of the blade profiles (cross-sections) is considered in a "blade-to-blade" stream surface, assuming the flow to be steady in time. This implies that the unsteadiness of the flow leaving previous rows is suddenly time-averaged. This general approach was first formally justified by Wu[5] in 1951.

The through-flow calculation first generates the axisymmetric stream surface for the blade-to-blade program. That in turn generates the total pressure loss and flow direction required for the through-flow program. Finally, after iteration if needed, the through-flow program calculates the overall characteristics of the turbomachine: flow, pressure ratio, and efficiency. The role of the blade-to-blade program is to supply local flow angle and pressure loss, and to predict blade surface pressure distribution; some programs also predict boundary layers. The role of the through-flow program is to predict radial flow distribution and overall performance.

Until relatively recently, turbomachinery analysis could only be done on this quasi-three-dimensional steady flow basis, but now many fully three-dimensional and some unsteady flow computations are possible. It is therefore convenient to review the history of turbomachinery CFD under the sub-headings of blade-to-blade methods; through-flow methods; quasi-3D methods; and fully-3D methods. For the purpose of this paper, a CFD method is defined as a solution of the inviscid or viscous equations of fluid motion in two or three dimensions (as distinct from one dimension). Integral boundary layer methods and heat transfer methods are excluded to keep the length of the discussion within bounds.

The process so far described is the calculation of the flow-field within a turbomachine of specified geometry. It is also desirable to have CFD design programs, which will compute the shape of blading required to achieve a specified performance in an optimum way. Design programs are considered under a final sub-heading.

There has been a steady succession of books[6-9], papers[10-19], and Conferences and Lecture Series[20-22] devoted entirely to turbomachinery CFD. The reader is referred to them for detailed expositions of the equations used, the solution methods adopted, and the justification of the results by comparison with specific experiments. The purpose of the present paper is to stand back from the details and review the historical use and effectiveness of the methods.

Blade-to-blade methods

The earliest calculations of flow through a cascade of cambered blades of finite thickness were by Howell[23], and Merchant and Collar[24], both in 1941. They solved the inviscid incompressible steady plane flow equations (which reduce to Laplace's equation) by conformal transformation. Many others followed (see p268 of ref 7) including the extension to linearised compressible flow. The alternative classical method of solving Laplace's equation is the singularity method, and many such methods were evolved because they could analyse arbitrary profiles. The first method adopted in the UK for practical use was that of Martensen[25], adapted for compressible flow by Price[26].

In the 1960's, more versatile ways of solving compressible flow equations were developed, as the growing power of computers made such methods practicable. They were generally elaborated to solve the subsonic compressible flow around cascade blade profiles on a stream surface of arbitrarily-varying thickness and radius, as required for the quasi-3D approach. Four types of method evolved: finite difference, finite difference using matrix inversion, streamline curvature, and finite element.

By subsequently employing any convenient boundary layer method, predictions of total pressure loss could be attempted. The varying streamtube thickness and radius could easily be catered for in an integral boundary layer method[27]. Outlet angle prediction proved troublesome, because the trailing edge of a practical turbomachine blade is not sharp enough to apply the Kutta condition simply. Instead, a convergence of surface pressures towards a common trailing edge pressure was adopted.

There was much discussion at the time of the relative computational and practical merits of the four types of method listed. The amount of detail needed at the leading and trailing edges, especially at off-design incidence, was highly relevant. In the UK, the streamline curvature method was the only one to emerge at this time as a regular design tool. It is probably true that this was due not so much to the relative scientific merits of the four methods as to the fact that Rolls-Royce had invested a large amount of effort within the Company into developing streamline curvature programs into practical tools capable of being applied by specialists other than the program originator.

The singularity and matrix blade-to-blade methods have also been applied to radial turbomachines, by Railly[28] and Goulas[29] respectively.

Although the methods so far described can generate solutions with supersonic patches, modern turbomachines contain shock waves and supersonic regions. It was therefore a great step forward when Denton developed practical time-marching methods for turbomachines[30], following MacCormack's approach, which were capable of computing flow fields with embedded shock waves. He used a finite volume formulation, which he has since substantially improved[31]. His methods are now used by many firms throughout the world.

Still, the flow through a compressor cascade with supersonic inflow could not be realistically modelled. The leading edge shock impinges on the suction surface and either separates the boundary layer or brings it close to separation. The natural way of modelling the boundary layer is to compute the boundary layer displacement thickness, add it to the blade metal thickness, recalculate the velocity distribution, and iterate to convergence. Unfortunately any attempt to do so is found to be unstable with a separating or nearly separating boundary layer. This difficulty was at last overcome by Calvert[32] in 1982, by inverting the order of iteration along the suction surface after the shock wave. He obtained converged solutions to cases with separated or attached boundary layers which agreed well with experiments[33].

None of the methods so far described are aimed at computing unsteady flow. Whitehead[34] has developed a finite element method for this purpose, which has been used as a design and analysis method for steady flow, as well as forming the basis for a flutter prediction system.

These methods of Denton, Calvert, and Whitehead have been adopted by Rolls-Royce, extended where necessary, and applied regularly to analyse and design compressors.

Attention has also turned in the 1980's to solving the viscous equations, now that supercomputers are more widely available. The problem is how to model the turbulence economically and yet sufficiently realistically for the turbomachinery application. Dawes[35] adopted a simple mixing length model with some success.

Through-flow methods

The earliest through-flow assumption was that the flow was uniform from hub to tip. This was succeeded by the assumption of "radial equilibrium" which provided analytical values of radial variations. The first CFD method evolved was the actuator disc approach, in which the axisymmetric flow field equations in the annulus were solved, while the blade rows were represented by actuator discs generally located at the trailing edge of the actual blading. These methods, first suggested in 1944[36], were developed mainly by Hawthorne, Horlock, Ringrose, Lewis, and Railly in the UK[37]. Although some trial calculations on a real engine were done within Rolls-Royce, the initial restriction to incompressible flow in a parallel annulus, and the sheer labour involved in completing the calculations on the electro-mechanical desk-top calculators of the day, precluded the adoption of actuator disc methods for design use. By the time these drawbacks had been overcome more versatile methods had evolved.

The new methods of the 1960's fall into the same categories as the blade-to-blade methods: finite difference, finite difference using matrix inversion, streamline curvature, and finite element. Only matrix and streamline curvature methods have been developed in the UK and of these only the streamline curvature method has been widely used for axial turbomachines, probably for the reason given in the previous section.

The streamline curvature idea came as early as 1942[38], but practical methods were first developed nearly simultaneously by Smith[39] and Novak[40] in the USA and by Hetherington[41] and Ringrose in the UK. In its original form, calculation planes were located at every trailing edge plane, so that local outlet angles and total pressure losses could be inserted from cascade correlations. The Hetherington method was adopted by Rolls-Royce and has been widely used ever since for the design and analysis of both compressors and turbines.

The addition of calculating planes within rows caused much discussion in the early 1970's, when it became clear that the radial components of blade force (not accounted for when only trailing edge planes are used) could influence the flow considerably. Smith[42], using a matrix method with a grid within the blade rows, demonstrated a much better prediction of the radial variation of static pressure in a turbine (which would affect cooling airflow predictions). More recently, Ginder[43] has shown the importance of the extra planes in a streamline curvature program applied to a transonic fan.

Through-flow methods have also been developed for radial turbomachines. In 1967, Wood and Marlow[44] applied the streamline curvature method to a pump impeller. At the National Gas Turbine Establishment, an "Impeller Computer Design Package" was evolved[45] which combined a matrix through-flow calculation (with an approximate blade-to-blade assumption) with stressing and numerically-controlling machining elements. At the National Engineering Laboratory an incompressible flow pump design package was produced[46], also using a matrix method, and making use of Railly's singularity method for blade section design. Goulas[47] has also developed the matrix method for centrifugal compressors.

One of the features of multistage compressor performance which inviscid throughflow methods cannot predict, and which throughflow methods with annulus wall boundary layer allowances also fail to predict, is the "repeating stage" phenomenon identified by L H Smith[48]. The radial distribution of axial velocity settles down after a few stages to a fairly constant pattern. In 1981 Adkins and Smith[49] proposed an explanation based essentially on secondary flow phenomena, and introduced semi-empirical terms into their streamline curvature program to calculate the secondary flow effects. Their results were consistent with experimental observations. More recently, Gallimore[50] has proposed an alternative explanation based on mixing theory. This provides an even more convincing explanation in the particular cases tested, though as in all mixing theories the prediction of effective turbulent eddy viscosity is problematical. The full implications of mixing on turbomachinery design have yet to become clear.

<u>Quasi-three-dimensional methods</u>

Until larger and faster computers were developed, it was essential to avoid having to solve fully three-dimensional equations with the proper boundary conditions. One approach was fundamentally an actuator disc approach, in that fully three-dimensional solutions were obtained only in unbladed ducts (Hawthorne[51], Dunham[52], Lewis[53]). This approach has not led to a practical working method.

The universally adopted method was first suggested in 1951 by Wu[5], who derived the basic equations. It involves iterative solution of blade-to-blade flow and through-flow, as described earlier. This process converges in only a few iterations. In principle, it does not allow for the effects of streamwise vorticity, which distorts the assumed blade-to-blade stream surfaces. Goulas[54] has proposed extra terms for streamwise vorticity effects in centrifugal compressors. However, the basic method is used for design purposes, and has been limited primarily by the quality of the blade-to-blade and through-flow methods incorporated into it.

<u>Fully three-dimensional methods</u>

Since Stuart and Hetherington[55] and Oliver and Sparis[56] first proposed fully three-dimensional methods in 1970/1, at least sixty papers have been published, most of them based on the MacCormack time-marching approach. The main contributors of inviscid schemes have been Denton[57], Thompkins[58], Bosman[59] (who all first published in 1976), and Hirsch[60] (1980). The feature of all these methods is that the convergence in time is slow - needing of the order of 1000 steps - and this has to be controlled by selecting the right grid shape, and the best time step, and the best relaxation factors. The Denton finite volume formulation has been improved and extended[61] over the years since 1976. This numerical scheme conserves mass flow exactly but does not automatically conserve stagnation pressure. Thompkins uses a finite difference scheme and Hirsch a finite element scheme.

One of the first attempts at a fully three-dimensional viscous calculation appears to be that of Carrick (1975)[62] who added simple viscous terms to Stuart and Hetherington's method[55]. This was not pursued, however, and the major contributions have been made by Walitt (from 1975)[63], Dodge (from 1977)[64] and the Moores (from 1979)[65].

The Walitt and Dodge methods are essentially successive approximations. The equations are written with inviscid terms on the left hand side and viscous terms on the right hand side. Starting from an inviscid solution, the viscous terms can then be calculated, and further solutions obtained successively. Both axial and centrifugal flow fields have been calculated by these methods. The Moores use a finite difference scheme, which is confined to subsonic flow in principle. An improved version has recently been devised[66].

Recently, Denton[61] and Dawes[35] have proposed separate schemes for extending the time-marching approach to viscous flows. Dawes uses a mixing length model as in his two-dimensional code, and Denton has tried avoiding modelling eddy viscosity by using empirical blade force terms. The essential difficulty with all the viscous methods is, of course, accommodating a sufficiently fine grid. To resolve the flow near walls it is essential to have sufficient grid lines actually within the boundary layer; this requires a huge computer store.

<u>Design as a direct process</u>

No formal solution has been found for the design problem when it is expressed in the most general terms: compute the optimum turbomachine for a specified duty. The number of independent variables is effectively infinite, and the number of design objectives is considerable and their relative importance unquantified (in general). These design objectives can be stated as follows:

1. mechanical integrity

   This sets limits to leading and trailing edge thickness, and thickness in general (especially for cooled blading). It also requires freedom from significant vibration of any kind over the running range.

2. efficiency over the running range.

3. surge margin (for a compressor) and low susceptibility to inlet flow distortion, over the running range.

4. low size, weight and cost.

5. minimum design and development cost.

When undertaking a design, proposals must be assessed against each of these objectives so as to reach a final compromise; this is still a human judgement.

It is necessary to start by making an arbitrary trial choice of most of the independent variables. In practice this is done by selecting a through-flow pattern conforming to optimum values of stage loading parameters known from previous experience; that is, the "optimum" velocity triangles are selected. Any computer-based optimisation is undertaken in the blade-to-blade calculation, where the designer attempts to find the optimum blade profile for the required inlet and outlet flow angles and velocities. Even this is not easy.

Most approaches involve prescribing a "good" velocity distribution (PVD) and computing the blade profile required to generate it. Lighthill[67] proposed a PVD method in 1945, using a conformal transformation. In 1952, Stanitz[68] introduced a linearised compressible PVD method, which was developed by Payne[69] and applied within Rolls-Royce to design turbine blade shapes. Murugesan and Railly[70] wrote a design version of Martensen's methods[25], and Lewis[71] also wrote a design method using distributed singularities.

The fundamental difficulty in using the PVD approach is that the selection of the pressure distribution requires experience (and trial-and-error) if an unacceptably thin aerofoil (or even one of "negative thickness", since the Stanitz method actually designs a passage) is to be avoided. Wilkinson[72] adopted an interesting scheme to avoid this difficulty; his method designed the suction surface (with the more critical velocity distribution) to satisfy a prescribed velocity distribution but then prescribed the aerofoil thickness and calculated the resulting pressure surface velocity distribution. A streamline curvature method on these lines has been developed within Rolls-Royce, and used to design turbines.

The inversion of a singularity method into PVD form involves solving the same equations with different variables unknown. The inversion of more modern analysis methods (using grids) is more complicated and involves iteration of the analysis program. Paige[73] has written a PVD finite volume method using a hill-climbing scheme. Cedar and Stow[74] have written a PVD finite element method based on Ref (34), using a local transpiration model to avoid changing the grid.

Recognising that trial-and-error is necessary in applying even a PVD scheme, a workable alternative is to provide a flexible shape description in the form of one or more algebraic expressions involving arbitrary parameters. Then the choice of shape is controlled by the choice of a fixed number of parameters, say 8. Starting from guidelines established by experience, the parameters are varied until a satisfactory pressure distribution and mechanical shape are obtained. A method of this type for turbine design was proposed by Dunham[75] and this approach is also used by RAE for transonic fans.

A more radical approach was proposed by le Foll[76] who took the further step of prescribing the desired boundary layer development and hence working back to the profile shape. This method would presumably encounter the same difficulties as PVD in arriving at a mechanically acceptable shape; it has apparently not been adopted by any manufacturer.

4 Applications of CFD to Improving Turbomachinery

In this section, the historical role of CFD in improving turbomachinery is examined, and illustrated by some specific examples taken from aeronautical and industrial applications.

Axial Compressors

Early axial compressors were designed using "standard" profile shapes evolved from systematic cascade testing, and assuming simple radial equilibrium conditions. Although incompressible blade-to-blade design and analysis methods were available in the 1940's, and later compressible ones, they were never used to design engine blades, because of their inability to predict the key features of outlet flow angle and total pressure loss. The compressors used in engines designed in the 1940's, 1950's, and early 1960's were developed to acceptable performance and reliability by means of long expensive test programmes. Many cascade tests on standard section shapes were undertaken, in which the variations of outlet flow angle and total pressure loss were measured over a range of incidence. Tests of this kind remained the basis for blade profile selection right up to the 1980's.

The advent of high bypass ratio engines necessitated the development of the single stage transonic fan, which was very difficult both because of the supersonic relative inflow on the outer radii and because of the steeply curved flow path. The assumption of simple radial equilibrium and the use of traditional subsonic profiles were entirely inadequate. Coplin[77] presented the history of transonic fan evolution in the form of Fig 5. Before 1970, thin aerofoils with relatively sharp leading edges were introduced, but again only of arbitrary (double and later multiple circular arc) shape. A major advance was made when the streamline curvature through-flow method was introduced. This enabled blade profiles to be defined along stream surfaces. As can be seen in Fig 5 this improved the performance of RB211-type fans by some 3% and enabled the RB211-22B to be introduced at a satisfactorily competitive performance level.

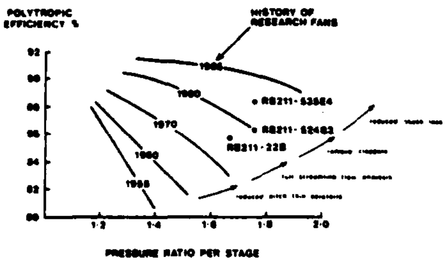


FIGURE 5. Transonic Single Stage Fans

It was not until the last few years that the next step improvement became possible, described in Fig 5 as "reduced shock loss", as a result of the possibility of designing high speed blade profiles by computational methods (rather than using circular arcs). The principle of the "supercritical" aerofoil for a high subsonic inflow, with a shape tailored to provide a shock-free slightly supersonic surface velocity distribution and a delayed diffusion, had been evolved for wings. Bauer, Garabedian, and Korn[78] published a computer program providing the corresponding solution for a cascade. Coplin reported the result of applying their method to an RB211 fan outlet guide vane. The number of vanes was reduced - saving weight and cost - and the efficiency improved by ½ to 1% in that case.

For a rotor blade section with supersonic relative inflow, no purely inviscid method proved effective, as explained earlier. The application of Calvert's method[32], which allowed for separating and nearly separating boundary layers as well as the shock pattern, enabled blading with lower shock and separation losses to be designed. Two quite different fan rotor blades (one military and one civil) have recently been designed using Calvert's method and tested by Rolls-Royce. Both displayed efficiency improvements at design speed over designs undertaken prior to the availability of the new computational tool. The gains ranged from 2 to 4% and were, in the multistage case, partly associated with more accurate matching. Indeed, this is a good example of how a well proven computational method can reduce trial-and-error; only one build was needed. This first time success was also due to another computational improvement, not this time in the fluid dynamics area: the application of a new Rolls-Royce method of calculating the deformation of a fan blade under centrifugal and aerodynamic loading.

The successful civil rotor design was undertaken by a proper quasi-three-dimensional procedure. The early 1980's designs had been done by first calculating the through-flow using calculating planes between blade rows (not within them) and then blading along the resulting stream surfaces. This new design was reached by iterating between the blade-to-blade calculation (on stream surfaces) and through-flow calculations with planes within the blade source, implementing the Wu approach in full. Ginder[43] had shown that simply interpolating conditions within the row (knowing those between the rows) could lead to significant errors, as the radial components of blade force are then unaccounted for.

CFD has played a less significant role, so far, in core compressor improvements. The through-flow is first calculated by a streamline curvature method, using planes between the blade rows only. With much straighter annulus wall lines than in a transonic fan, it seems less necessary to consider planes within the rows; and of course the computational grid for a multistage compressor might become too large with the extra planes.

The selection of blade profiles, for many years taken from "standard" shapes, has recently followed the "supercritical" route previously described for fan outlet guide vanes, with checks using other inviscid methods. Aerofoils of this type have consistently shown efficiency improvements around 1%, (equivalent to reducing the actual loss by the order of 10%) and reductions in the number of blades by more than 10%. This reduction in blade numbers accounts for the efficiency gain and economises directly in cost and weight. The new shapes could not have been generated by any simpler method. They are checked by finite difference or finite element methods.

Centrifugal compressors

The early UK centrifugal compressors were designed by essentially one-dimensional methods. In the late 1960's and early 1970's, finite difference, matrix, and streamline curvature methods became available to compute impeller vane surface velocity distributions on the assumption of unseparated flow. NGTE[45] and NEL[46] made their impeller design schemes available to UK industry in the late 1970's. The NEL methods were applied there to design various pumps and fans for commercial customers. For example, a quiet cooling fan was designed in 1973 which had a much better performance than its competitors. The NGTE package was used by Comp Air to design a most satisfactory fan[79], and was adopted and improved by Noel Penny Turbines for various designs[80].

The application of scientific design principles to centrifugal pumps can have a startling effect on performance. Fisher[81] quotes the case of an automotive water pump at least three times as efficient as its predecessor.

In the aeronautical field, the first application of a modern through-flow calculation was worth some 4% in efficiency. A more notable example occurred when the opportunity came to Rolls-Royce to redesign the Dart impeller, which had originally been developed from pre-war superchargers long before CFD had been introduced. The fuel consumption of the engine was reduced by a remarkable 8½%.

It is interesting to note that all the methods actually used, so far, assumed unseparated flow, whereas later measurements have confirmed that most impeller flows appeared to have local separations most of the time. The fully three-dimensional viscous methods now becoming available should cater for separations. Will it become possible then to design even better impellers and overcome the relatively low efficiency of most high pressure ratio units?

Turbines

Early turbines were designed assuming uniform flow from hub to tip, or later some form of radial equilibrium. The outlet gas angle from a turbine blade row is fairly well approximated by $\cos^{-1}$ (throat/pitch), so the blade profiles were designed on a drawing board by marking out the throat circle (to suit the required outlet angle) then fitting several circular arcs to blend into a smooth streamlined shape. Continuous contraction of the passage width up to the throat was ensured.

As in the case of compressors, the early conformal mapping methods were not used to design engine blades, though some exact solutions generated by these methods were used as test cases for validating approximate numerical methods.

CFD methods were first applied in the 1960's, to design better aerofoil shapes. Initially, the incompressible Martensen method[25] was used at NGTE. One lesson that the theory illuminated was that surface curvature (not slope) is the geometrical property appearing directly in the equations determining surface velocity. So to get a smooth surface velocity the curvature must be continuous. This conflicts with the instinctive feeling that only slope needs to be continuous. So profiles designed using contiguous circular arcs did not show favourable surface velocity distributions. The effect of using the Martensen approach was to improve efficiency. In one case, blading designed by NGTE for a small industrial turbine manufacturer showed a 7% improvement over a traditional design.

Rolls-Royce adopted the Stanitz PVD approach[68] and it was widely used to design turbine profiles. The method designs a passage, using a linearised compressible flow calculation; leading and trailing edges are added afterwards. It was first applied to the Olympus turbines, and direct comparison between old and new designs (conducted on various configurations) showed immediate gains of up to 6% in efficiency[82].

In the 1970's, Rolls-Royce adopted streamline curvature methods for both through-flow and blade-to-blade design and analysis. The principle of design was to start with a favourable pressure distribution and iterate manually until a coolable shape (that is, an aerofoil with sufficient thickness, leading and trailing edge radii and trailing edge wedge angle) was reached.

In the 1980's, both NGTE and Rolls-Royce turned to the Denton two-dimensional program[31], which enabled locally supersonic flows (as normally encountered in the trailing edge region) to be calculated. Unlike a compressor passage, a turbine passage is not greatly affected by the relatively thin boundary layers on the blade surfaces (except near the annulus walls). So an inviscid method provides a good basis for design. There is, unfortunately, a caveat to this. In 1983 Paige[73] designed a nozzle guide vane profile which was "better" than an existing profile in that the pressure distribution looked better and calculation of the surface boundary layers led to a lower loss prediction. When tested in cascade, the design velocity distribution was achieved but the overall loss was nevertheless higher than the old design because the base pressure was lower. The base pressure cannot be predicted by an inviscid method and the change was not in accordance with base pressure correlations. This experience sounded a note of caution.

In the last few years, the fully three-dimensional Denton method[61] has been employed by RAE and by Rolls-Royce to help design new turbines. The proximity of an annulus wall can significantly change the aerofoil surface flow. The availability of a three-dimensional method allows the designer to explore changes in blade stacking (the way aerofoils at each radius are relatively positioned) and changes in end wall shape. Morgan[83] has described the effect of changing the stacking of the RB211 hp nozzle guide vanes, guided by CFD methods, which increased efficiency by around 1%.

Another method used in recent years by Rolls-Royce is the Moore three-dimensional viscous program[84]. This is formally restricted to subsonic compressible flow, but it can be used to assess possible secondary flows. A particular example in which the inviscid and viscous three-dimensional methods were used to guide a design change was the RB211 IP nozzle guide vane. These vanes are mounted in a duct of rapidly increasing radius, which cannot adequately be catered for in a quasi-3D method. The ngv used in earlier engines was redesigned to remove a local three-dimensional flow separation, and the engine specific fuel consumption improved by some 1½%.

Turbines are of course widely used outside the aircraft industry, and the same CFD methods are available, for example, to steam turbine designers. It has recently been reported[85] that the LP turbine in one of the three Parsons 500 MW sets at Didcot Power Station was modified by Parsons for the Central Electricity Generating Board at a cost of £3M to incorporate a new last stage rotor blade designed with the help of a Denton 3D program, and new diaphragms. The unit efficiency rose some 3%, and the resulting saving in the coal bill is £1.7M per annum on that one set alone.

At the other end of the size range, Connor and Payne[86] have described how the application of PVD methods has increased the efficiency of a turbocharger axial turbine by over 3% at design point, increasing to over 10% at off-design.

5 Comments on CFD History

Looking back at what has happened, a number of general comments can be made:

(1) The invention of a fundamentally new method is rare and is done not by a team but by a gifted individual. It cannot be "scheduled" by a Research Manager; all he can do is to create the conditions under which a suitable research worker is attracted to the problem and equipped with the time and facilities to tackle it.

(2) The implementation of a new method is a long hard job which can be greatly helped by less gifted workers than the originator. When it comes to converting it into a "user friendly" program capable of being used by designers who do not understand the mathematics, there is a great deal of work to be done by a team, properly planned and professionally managed.

(3) The validation of a new program against experimental results is vital and requires a complicated (and probably expensive) experiment planned and executed with the help of the CFD analyst. Especially for high speed turbomachinery, the necessary experimental facilities can only be found at national Research Establishments or in industry. It must be the function of a Research Manager to plan such work.

(4) The decision to commit a new design evolved by CFD methods, first to an experimental demonstration and later to a production engine, is a Chief Engineer's decision, and a proposal has to be justified to him by unequivocal validation achievements, carefully planned.

(5) Many more methods have been evolved than have ever been put to good use. It is probably not true that "only the fittest survive". Much seems to depend on the accidents of history: which computer was used, how eloquent the originator was, which organisation he happended to work in. The most marked progress occurred when two or more methods were actually competing scientifically on the same problem. Research Managers should encourage such competition in developing new methods.

(6) The visual presentation of three-dimensional flow patterns, to enable the research worker himself to grasp them, and later to enable him to explain his results to others, presents some difficulty. Considerable effort has needed to be devoted to computer graphics.

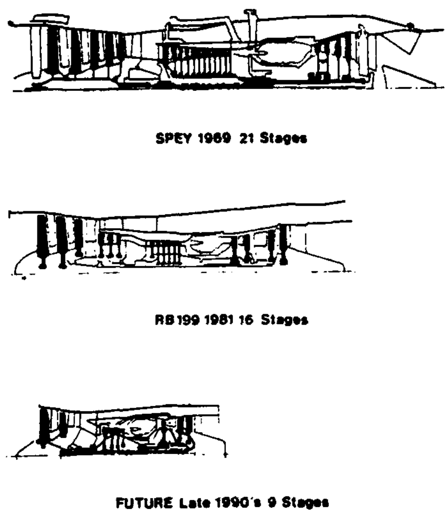


FIGURE 6. Combat Engines - Scaled to the Same Thrust

6 Future Needs and Prospects

Fig 6, showing three generations of combat aircraft engine scaled to the same thrust, illustrates the overall achievements over the last twenty years in engine design and the target for the immediate future.

Axial compressors

Despite the much improved ability to predict the design point flow field and performance of a transonic compressor stage, described earlier, the reliable prediction of off-design performance - and especially surge prediction - remains elusive. The first missing element is the accurate prediction of the total pressure loss of a sharp-leading-edge transonic profile at off-design conditions, and this seems achievable by the improvement of existing methods. A much more difficult problem - requiring a fully three-dimensional viscous method - is the prediction of flow separation (possibly leading to rotating stall or surge) in the end wall regions, including tip clearance effects. Another reason for needing a 3D viscous method is to predict the radial migration of aerofoil boundary layer fluid which effectively "transfers" loss from one radius to another, and may on occasions accumulate low energy fluid around part-span shrouds.

The trend of compressor design has been not so much towards increasing efficiency as maintaining efficiency at progressively increasing pressure ratio per stage. Fig 6 shows how this trend has reduced the size and weight of military engines and is continuing to do so; the three engine drawings illustrated have been scaled to the same thrust. The principal element in increasing stage pressure ratio is of course increasing blade speed as improved materials are developed; but the inevitable consequence is more and more supersonic flows. If a military engine is ever to have a single stage fan to achieve its pressure ratio of 3 or 4, it would be a fully supersonic compressor. A great deal of effort was devoted to abortive attempts to produce efficient supersonic compressors in the 1950's and 1960's; could the CFD tools now being developed enable much more successful attempts to be made in the 1990's?

Turbomachine flows are assumed to be steady in most analyses and designs. Unsteady calculations have been concerned primarily with flutter. It is not yet clear whether the development of unsteady flow computations (as computers become fast enough) will reveal a need to alter design philosophies to improve performance. Reliable CFD predictions of flutter and noise appear very difficult and distant targets still.

Turning to multistage core compressors, there is again a difference between the priority targets for civil engines (efficiency) and military engines (compactness and low weight), but the technology required to achieve them is common. The flow in the end wall regions, including the tip clearance flow, has a dominant influence on the annulus wall boundary layer blockage and hence stage matching. It also plays a large part in generating total pressure losses, and perhaps in surge initiation. Three-dimensional viscous programs are needed to give the designer a better insight into the phenomena, and especially to guide the evolution of "end bends".

The importance of mixing within a multistage compressor, by both secondary flow and turbulent eddies, has been explained, and further research in this area should contribute substantially to design techniques.

In addition to improved performance in the final version, the application of CFD will significantly reduce the number of trial builds required in development, so saving time and money.

Finally, the response of compressors to a distorted inlet flow (typically due to combat manoeuvres) has been extensively studied experimentally but to date only two-dimensional theories[87] (allowing circumferential and axial flow variation but no radial variation) have been employed successfully for analysis. A three-dimensional theory (allowing radial variation) is clearly essential for a low hub/tip ratio transonic fan, and possibly also for a core compressor. Attempts to date[32,88] have produced methods too restricted in scope or too difficult to apply. A solution of this problem appears possible using modern numerical methods and should be attempted.

Centrifugal compressors

In the 1960's, extensive research was devoted, especially in Canada, to high pressure ratio units, which were successfully developed only at the expense of lower efficiency. The losses appeared inevitable because of the high supersonic inflow to the narrow diffuser ring. Most small aeronautical gas turbines have therefore chosen to use several axial stages followed by a lower pressure ratio, more efficient, centrifugal stage. The other problem of high pressure ratio centrifugal stages was of course stressing, but improvements in materials will presumably continue. So there appear to be two lines of advance for centrifugal compressors, both heavily dependent on CFD improvements.

The first is the low pressure ratio unit - around 3 - when the flow reaches the diffuser subsonically. As already mentioned, the impeller flow is probably separated, and it seems a reasonable target to evolve unseparated designs and hence increase overall efficiency to axial compressor levels. A centrifugal stage could then become attractive even for a large civil engine. The diffuser introduces considerable loss because the wetted surface area is large, and there appears to be scope for the application of 3D viscous codes to improve diffusers.

The second line of advance could be a return to higher pressure ratio units (10?) to reduce engine weight and cost, but reducing the efficiency loss by tailoring the shock patterns with the help of 3D codes.

Axial turbines

Although the equations are the same, there are significant differences between compressors and turbines for the CFD analyst. The first is that the turbine blade surface boundary layers are small fractions of the passage width, so that inviscid methods give a good guide to optimum shape and predict the flow well, until near the trailing edge. The second difference is that the blade camber and hence the secondary flows are very much greater than in compressors, and are accentuated by the tip clearance rather than reduced. A third difference is that heat transfer is a key element in turbine design.

Most aeronautical turbine nozzle guide vane rows and many rotor rows have sonic or supersonic relative outflow, with a shock structure impinging on the boundary layer just ahead of the trailing edge. The resulting lambda-shock controls the base pressure. Currently, the base pressure is predicted empirically and it seems possible that, on a two-dimensional basis, a viscous-inviscid interactive method or a fully viscous method might be able to predict it theoretically. Because turbulence modelling of separated regions is still difficult, the interactive method seems more likely to succeed in the short term. However, there are two serious complications; one is the need to model some form of radial equilibrium in the nearly stagnant base region, and the other is the effect of cooling air discharge into the base region. Both phenomena appear to offer scope for improved overall efficiency if they could be well enough understood.

Considerable quantities of experimental data have been amassed on end wall and secondary flow. The secondary flow tends to strip the incoming wall boundary layer fluid off the wall and discharge it into the mainstream via the blade suction surface trailing edge. A new wall boundary layer starts. The ability to predict this type of flow depends upon the development of a 3D viscous code able to cater for the corners between wall and blade, and in due course also for tip clearance. The zero clearance case seems likely to be solved quite soon. It is generally accepted that end wall effects of this kind account for around half the total pressure loss in a turbine; surely that loss can be reduced by scientifically-chosen end wall profiling or "end bending" of the blades, when those CFD tools become available. For a multistage turbine, the type of mixing analysis described for axial compressors is surely also needed.

The prediction of external heat transfer is largely a question of boundary layer prediction, which becomes particularly difficult in the presence of film cooling or Görtler vortices. However, it has been demonstrated that the passage of upstream wakes (or even of downstream blades) has a major effect on the boundary layers[89,90]. Robson[91] showed the boundary layer switching from laminar to turbulent and back again as wakes passed. It is clear that an analysis of this situation requires an unsteady CFD model. At least on a two-dimensional viscous basis, this should be already possible.

As for compressors, CFD will not only improve turbine performance but also reduce development time and cost.

Radial turbines

Radial inflow turbines have not been used in UK aero engines, not on account of inefficiency but because of the mechanical design problem of making a cooled high speed rotor of adequate strength and life, and not too high a rotational inertia. As a result, they have not received the attention of many CFD specialists in the UK. There must be scope for applying the methods developed for centrifugal compressors, without the worries about large local flow separations. Some work of this kind has been done for application to large turbochargers(86).

The possibility of advanced ceramic materials, not needing cooling, could promote the radial turbine as an option for small aeronautical applications.

7 Conclusions

Computational fluid dynamics methods specific to turbomachinery have been developed since 1940 within the UK. They have usually employed the same mathematical methods used in other branches of CFD, as and when computers became large and fast enough. Initially, the models used were too simplified (incompressible inviscid planar or cylindrical flow) to be realistic and too laborious to apply and were not used in practice. The practicability of achieving realistic design and analysis results using the quasi-three-dimensional (Wu) approach meant that CFD could however be usefully employed as early as the 1960's although computers then could only calculate two-dimensional flow fields. By the 1980's, three-dimensional inviscid flow fields could be computed and the present decade is seeing great advances including viscous calculations.

The UK aeroengine firm (Rolls-Royce) has been quick to adopt CFD methods and to employ them to design and develop better compressors and turbines, just as other aeroengine manufacturers have done. The first key requirement for this to happen is that the firm should have a "core CFD team" large enough not only to develop its own new methods sometimes, but essentially to take a chosen method from any source and develop it into a proven working tool capable of use by a designer who does not understand the mathematical or programming details. The second requirement is that research managers in the firm or associated Research Establishments must organise a systematic methods validation programme, capable of convincing the most hard-headed Chief Engineer to commit a CFD design to his engine.

Examination of the history of UK turbomachinery technology since 1960 shows that some of the major advances in product quality were made as a direct result of the application of CFD, which arguably could not have been achieved without it. Typically, 1960's methods improved on 1950's methods by some 5 to 10% in efficiency, that is, reducing losses by 30 to 50%. The subsequent improvements have increased the stage loading levels at which high efficiency can be maintained. Three-dimensional flow field tailoring - only imperfectly understood and not yet predictable in CFD terms - has proved generally worth another 1 to 2%.

The future trend in CFD is inevitably towards 3D viscous flow, and unsteady effects. As computers improve, these new methods will make available to the turbomachinery designer on a more rational basis a wide range of options primarily in the end wall regions: wall profiling, end bends, varying stacks, winglets, tip treatment, casing treatment. There is obvious scope for reducing end wall region losses which amount to around half the total losses. The possibility of significant savings in the number of blades and stages - and hence in weight and cost - will arise from a better understanding of how to control supersonic flows. Finally, the escalating cost of engine development may be reduced considerably by getting the aerodynamics "right first time"; visible progress has already been made in this direction.

Examples of successful application of CFD methods to non-aeronautical turbomachines have also been given. There are two industries - aerospace and power generation - with a major economic justification for performance improvement through undertaking CFD research. The methods they generate will continue to be adopted by other turbomachinery industries.

It is concluded that the further advance of computational fluid dynamics for turbomachinery should be vigorously encouraged because it has a major role to play in advancing technological standards and reducing the time and cost of development.

Acknowledgements

The author acknowledges gratefully examples of CFD applications provided by Mr P M Came, Dr J D Denton, Dr E H Fisher, Dr D E Hobson, Prof R I Lewis, Dr A N Neal, and Rolls-Royce Ltd., and helpful discussions with colleagues.

Any views expressed are those of the author and do not necessarily represent those of the Ministry of Defence.

References

1. Graham, R. W., Adamczyk, J. J. and Rohlik, H. E.: Computational thermo-fluid dynamics contributions to advanced gas turbine engine design, NASA TM 86865, 1985.

2. Howell, A. R.: Griffith's early ideas on turbomachinery aerodynamics, Aero Jnl, Vol. 80, p. 521, 1976.

3. Constant, H., et al: The development of the internal combustion turbine, Proc I Mech E, Vol. 153, p. 409, 1945.

4. Ainley, D. G. and Mathieson, G. C. R.: A method of performance estimation for axial-flow turbines, ARC R&M 2974, 1951.

5. Wu, C. H.: General theory of fluid flow with subsonic or supersonic velocity in turbomachines of arbitrary hub and casing shapes, NACA TN 2302, 1951.

6. Horlock, J. H.: Axial flow compressors, Butterworths, London, 1958.

7. Scholz, N.: Aerodynamik der Schaufelgitter, Braun, 1965, (Aerodynamics of cascades, AGARDograph 220, translated and revised by A. Klein, 1977).

8. Horlock, J. H.: Axial flow turbines, Butterworths, London, 1966.

9. Woods, L. C.: The theory of subsonic plane flow, Camb Univ Press, 1961.

10. Gregory-Smith, D. G.: Methods of flow calculation for axial flow compressors, AGARD CP 34 Paper 6, 1968.

11. Gostelow, J. P.: Review of compressible flow theories for airfoil cascades, Trans ASME A, Vol. 195, Pt.4, p. 281, 1973.

12. Perkins, H. J. and Horlock, J. H.: Computation of flows in turbomachines. Finite elements in fluids, Vol. II, Wiley, London, p. 141, 1975.

13. Marsh, H.: Through flow calculations in axial turbomachinery: a technical point of view, AGARD CP 195, 1976.

14. Japikse, D.: Review-progress in numerical turbomachinery analysis, Trans ASME Jnl Fluids Engg, p. 592, 1976.

15. McNally, W. D. and Sockol, P. M.: Computational methods for internal flows with emphasis on turbomachinery, NASA TM 82764, 1981.

16. Kerrebrock, J. L.: Flow in axial compressors, AIAA J, Vol. 19, no. 1, p. 4, 1981.

17. Hirsch, C. and Denton, J. D. (eds.): Through flow calculations in axial turbomachines, AGARD AR-175, 1981.

18. Horlock, J. H. and Marsh, H.: Fluid mechanics of turbomachines: a review Int J Heat and Fluid Flow, Vol. 3, no. 1, 1982.

19. Freeman, C. and Stow, P.: The application of computational fluid mechanics to aero gas turbine compressor design and development, I Mech E CP 1984-3, p. 1, 1984.

20. I Mech E.: Computational methods in turbomachinery, I Mech E CP 1984-3, 1984.

21. Meauzé, G. (ed.): 3-D computational techniques applied to internal flows in propulsion systems, AGARD LS 140, 1985.

22. von Karman Institute: Seminar 1965, Lecture Series 6 (1968), 30 (1971), 59 (1973), 84 (1976), 1979-7, 1982-5, 1986-2.

23. Howell, A. R.: Note on the theory of arbitrary aerofoils in cascades, RAE Report No E3859, 1941, (Phil Mag, Vol. 39, p. 913, 1948).

24. Merchant, W. and Collar, A. R.: Flow of an ideal fluid past a cascade of blades, Parts I and II, ARC R&M 1890, 1940 and 1893, 1941.

25. Martensen, E.: Berechnung der Druckverteilung an Gitterprofilen in ebener Potentialstromung mit liner Fredholmschen Integralgleichungen, Arch Rat Mech An, Vol. 3, no. 3, 1959.

26. Price, D.: Two-dimensional, compressible, potential flow around profiles in cascade, Proc VKI Conf, Pt. I, Paper 6, 1965.

27. Dunham, J.: The effect of stream surface convergence on turbomachine blade boundary layers, Aero Jnl, Vol. 78, p. 90, 1974.

28. Railly, J. W.: Design of mixed flow cascades by Ackeret's method, The Engineer, Vol. 224, p. 405, 1967.

29. Goulas, A.: A blade-to-blade solution of the flow in a centrifugal impeller with splitters, ASME 79-GT/ISR-17, 1979.

30. Denton, J. D.: A time-marching method for 2- and 3-dimensional blade-to-blade flows, ARC R&M 3775, 1974.

31. Denton, J. D.: An improved time marching method for turbomachinery flow calculation, ASME 82-GT-239, 1982.

32. Calvert, W. J.: An inviscid-viscous interaction treatment to predict the blade-to-blade performance of axial compressors with leading edge normal shock waves, ASME 82-GT-135, 1982.

33. Calvert, W. J.: Application of an inviscid-viscous interaction method to transonic compressor cascades, AGARD CP 351, 1983.

34. Whitehead, D. S. and Newton, S. G.: A finite element method for the solution of two-dimensional transonic flows in cascades, Int J Numerical Methods in Fluids, Vol. 5, p. 115, 1985.

35. Dawes, W. N.: Application of full Navier-Stokes solvers to turbomachinery flow problems, VKI LS 2, 1986.

36. Ruden, P.: Investigation of single stage fans (translation), NACA TM 1062, 1944.

37. Horlock, J. H.: Actuator disc theory, McGraw Hill, London, 1978.

38. Traupel, W.: Neue allgemeine Theorie der mehrstufigen axialen Turbomaschinen, Leeman, Zurich, 1942.

39. Smith, L. H.: The radial equilibrium equation of turbomachinery, Trans ASME ser A, Vol. 88, p. 1, 1966.

40. Novak, R. A.: Streamline curvature computing procedures for fluid flow problems, ASME J Engg for Power, Vol. 89, no. 4, p. 478, 1967.

41. Silvester, M. E. and Hetherington, R.: Three-dimensional compressible flow through axial flow turbomachines, Numerical Analysis-an Introduction Ch. II, Pt. III, p. 182, Academic Press, New York, 1966.

42. Smith, D. J. L. and Barnes, J. F.: Calculation of fluid motion in axial flow turbomachines, ASME 68-GT-12, 1968.

43. Ginder, R. B.: A streamline curvature through-flow calculation for transonic axial-flow compressors including planes within the blade rows, I Mech E CP, 1984-3, 1984.

44. Wood, M. D. and Marlow, A. V.: The use of numerical methods for the investigation of flow in water pump impellers, Proc I Mech E, Vol. 181, p. 725, 1967.

45. Came, P. M.: The development, application and experimental evaluation of a design procedure for centrifugal compressors, Proc I Mech E, Vol. 192, no. 5, p. 49, 1978.

46. Neal, A. N.: Through-flow analysis of pumps and fans, NEL Rep 669, 1980.

47. Goulas, A. and Baker, R. C.: Through-flow analysis of centrifugal compressors, ASME 78-GT-110, 1978.

48. Smith, L. H.: Casing boundary layers in multistage axial-flow compressors, in: Flow research on blading, p. 275, Elsevier, Amsterdam, 1970.

49. Adkins, G. G. and Smith, L. H.: Spanwise mixing in axial-flow turbomachines, ASME 81-GT-57, 1981.

50. Gallimore, S. J.: Spanwise mixing in multistage axial compressors, Ph D Thesis, Cambridge Univ, 1985.

51. Hawthorne, W. R.: Rotational flow through cascades, Pt. I, The components of vorticity, Q J App Maths, Vol. 8, p. 266, 1955.

52. Dunham, J.: Non-axisymmetric flows in axial compressors, MES Monograph no. 3, I Mech E, London, 1965.

53. Lewis, R. I. and Horlock, J. H.: Flow disturbances due to blade thickness in turbomachines, JMES, Vol. 2, no. 1, p. 88 & Ht & Fld Flow Vol. 1, no. 1, p. 1, 1971.

54. Resnick, A. and Goulas, A.: The prediction of the flow field inside axial pump impellers, Isr J Tech, Vol. 19, no. 3, p. 98, 1981.

55. Stuart, A. R. and Hetherington, R.: The solution of the three variable duct flow equations, Int Symp, Penn State Univ, 1970.

56. Oliver, D. A. and Sparis, P.: A computational study of three-dimensional transonic shear flow in turbomachine cascades, AIAA 71-83, 1971.

57. Denton, J. D.: Extension of the finite area time-marching method to three dimensions, VKI LS 84, 1976.

58. Thompkins, W. T. and Oliver, D. A.: Three-dimensional flow calculation for a transonic compressor rotor, AGARD CP 195 Paper 6, 1976.

59. Bosman, C and Highton, J.: A calculation procedure for three dimensional, time dependent, viscous, compressible flow through turbomachines of any geometry, J Mech Eng Sci, Vol. 21, no. 1, p. 39, 1979.

60. Hirsch, C., Lacor, C. and Warzee, G.: Three-dimensional inviscid calculations in centrifugal compressors, AGARD CP 282 Paper 7, 1980.

61. Denton, J. D.: The calculation of fully-three-dimensional flow through any type of turbomachine blade row, AGARD LS 140, 1985.

62. Carrick, H. B.: Secondary flow and losses in turbine cascades with inlet skew, AGARD CP 214 Paper 9, 1977.

63. Walitt, L., Harp, J. L. and Liu, C. Y.: Numerical evaluation of the internal flow field in a centrifugal compressor impeller, NASA CR 134984, 1975.

64. Dodge, P. R.: Numerical method for 2-D and 3-D viscous flows, AIAA J, Vol. 15, no. 7, p. 961, 1977.

65. Moore, J., Moore, J. G. and Johnson, M. W.: On three-dimensional flow in centrifugal impellers, ARC CP 1385, 1977.

66. Moore, J. G.: Calculation of three dimensional flow without numerical mixing, AGARD - LS 140 Paper 6, 1985.

67. Lighthill, M. J.: A mathematical method of cascade design, ARC R&M 2104, 1945.

68. Stanitz, J. D.: Design of two-dimensional channels with prescribed velocity distributions along the channel walls, Pt. I - relaxation solutions, Pt. II - Green's function solutions, NACA TN 2593 and 2595, 1952.

69. Payne, D.: Axial turbine blading, Symposium on gas turbine blading design, Borough Poly, London, 1970.

70. Murugesan, K. and Railly, J. W.: Pure design method for aerofoils in cascade, J Mech Engg Sci, Vol. II, no. 5, p. 454, 1969.

71. Lewis, R. I.: A method for inverse aerofoil and cascade design by surface vorticity, ASME 82-GT-154, 1982.

72. Wilkinson, D. H.: A numerical solution of the analysis and design problems for flow past one or more aerofoils or cascades, Pt I - analysis problem, Pt. II - design problem, ARC R&M 3545, 1967.

73. Paige, R. W.: A computational method for the aerodynamic design of transonic turbine blades, Ph D Thesis, Cambridge Univ, 1983.

74. Cedar, R. D. and Stow, P.: A compatible mixed design and analysis finite element method for the design of turbomachinery blades, Int J Num Methods in Fluids, Vol. 5, p. 331, 1985.

75. Dunham, J.: A parametric method of turbine blade profile design, ASME 74-GT-119, 1974.

76. le Foll, J.: A theory of the representation of the properties of boundary layers on a plane, Proc VKI Seminar, Pt. I, Paper 9, 1965.

77. Coplin, J. F.: The accelerating pace of advancing aero engine technology, Aerospace, Vol. 11, no. 2, p. 17, 1984.

78. Bauer, F., Garabedian, P. and Korn, D.: Supercritical wing sections - Part III, Lecture notes in economics and mathematical system, Vol. 150, Springer-Verlag, New York, 1975.

79. Starmer, C.: Product development - CompAir's philosophy, Ch Mech Engr, p. 65, Apr, 1983.

80. Chevis, R. W.: Design and test of two 8:1 pressure ratio single stage centrifugal compressors of different flow size, Can J Aero & Space Inst 29th AGM, 1982.

81. Fisher, E. H.: Private communication, 1986.

82. Carter, A. F.: unpublished Rolls-Royce report, 1962.

83. Morgan S.: Low aspect ratio turbine design at Rolls-Royce, VKI LS 1984-05, 1984.

84. Moore, J. G.: An elliptic calculation procedure for 3D viscous flow, AGARD LS 140 Paper 4, 1985.

85. Turbine mods hot up efficiency race, Electrical Review, Vol. 218, no. 1, 1986.

86. Connor, W. A. and Payne, D.: Aerodynamic design of turbines for large turbochargers, I Mech E Paper C 325/31, 1986.

87. Mokelke, H.: Prediction techniques, AGARD LS 72 Paper 5, 1974.

88. McCune, J. E. and Hawthorne, W. R.: The effects of trailing vorticity on the flow through highly loaded cascades, J Fluid Mechs, Vol. 74, Pt. 4, 1976.

89. Doorly, D. J., Oldfield, M. L. G. and Scrivener, C. T. J.: Wake-passing in a turbine rotor cascade, AGARD CP 390 Paper 7, 1985.

90. Dring, R. P., et al.: Turbine rotor-stator interaction, ASME 82-GT-3, 1982.

91. Hodson, H. P.: Measurements of wake-generated unsteadiness in the rotor passages of axial flow turbines, ASME 84-GT-189, 1984.

END

DATE

FILMED

7-8