AERODYNAMICS OF AIRCRAFT ENGINES – STRIDE AND STUMBLES

by

Nicholas A. Cumpsty

GAS TURBINE LABORATORY
MASSACHUSETTS INSTITUTE OF TECHNOLOGY
CAMBRIDGE, MASSACHUSETTS

AERODYNAMICS OF AIRCRAFT ENGINES – STRIDE AND STUMBLES

by

Nicholas A. Cumpsty

GTL Report #213   September 1992

AERODYNAMICS OF AIRCRAFT ENGINES – STRIDE AND STUMBLES*

N.A. CUMPSTY[†]
Jerome C. Hunsaker Visiting Professor
Department of Aeronautics and Astronautics
Massachusetts Institute of Technology
Cambridge, MA 02139

SUMMARY

Attempts to understand and predict the aerodynamic behaviour of compressors and turbines in aircraft gas turbines have been encouraged by the intense competitive pressure which exists. Many of the apparently most difficult problems have been overcome using suitable numerical analysis, for example the calculation of three-dimensional transonic flows has been particularly successful. It is seemingly paradoxical that the numerical methods are relatively very good for flows which are traditionally regarded as difficult, but do less well at predicting efficiency when the flow is well behaved in the conventional aerodynamical sense, such as fully attached flows. The numerical methods do not necessarily give insight into the flow that is seen as most helpful to the designer and it can be useful to complement them with simpler approaches to the problem which seek to capture the essential features of the flow. The aerodynamics of aircraft engine fans are used to illustrate these points.

Although numerical methods have been very successful with aeroengine fan blading they have been less successful with the multistage compressor. The reasons for this, primarily the difficulty of prescribing the boundary conditions, are discussed in this paper. The analysis of flow in multistage compressors still stumbles along with empirical methods, much of it based on data published over twenty years ago. A second area where stumbling has occurred is the prediction of flutter of blading, particularly fan blading; as recently as 1990 there were major in-flight failures due to flutter of a fan in civil airline service. To this day there is no reliable method of predicting the operating boundaries of flutter, and testing the engine over the entire operating range of altitude and speed is the only reliable method of ensuring safe operation.

1.0 INTRODUCTION

On March 21, 1956, W.R. Hawthorne presented the first Minta Martin Lecture which was subsequently published in revised form, Hawthorne (1957). A principal feature of the lecture was to draw attention to the particular aerodynamic problems associated with the flows confined to ducts or passages inside engines; this interest will be followed here with the perspective of 1991. Another message to be drawn from Hawthorne's lecture, which was given before the real dawn of civil jet transport, was the importance of propulsion to the development of aeronautics. Since then there has been an astonishing increase in civil air travel. Much of this increase has been made possible by the availability of jet engines, and some comment on this seems fitting as the opening theme of this paper.

To get some measure of the extent of travel, it is useful to look back to one of the great human migrations, the immigration of Europeans into the United States. According to the US Bureau of Census, see Lerner (1975), the peak year for immigration during the 19th century was 1882, when there were 648,148 European immigrants. At that time most would have begun their ocean crossing from Liverpool, even if their original country was in continental Europe, and arrived in New York after a voyage taking typically three weeks. The normal immigrant fare was around $300 in 1991 dollars, but then as now, fares could be much lower at times when the traffic was below the carrying capacity.

The total of 648,148 translates into an average of 1775 people carried per day; to relate this to modern capacity, one Boeing 747 carries about 450 people in comparative comfort, so four such planes could have carried the entire traffic. In fact the immigrants travelled in very little comfort. Something nearer to the conditions under which immigrants travelled probably occurred in 1991 when El Al evacuated Ethiopian Jews; they were able to convey 1200 people in one 747, and on this basis two such planes would have been able to carry the total immigrant traffic with some reserve capacity. The movement of immigrants was referred to at the time by terms like "flood" or "tide". Having in mind the hundreds of Boeing 747s, and hundreds of other wide body jet aircraft, in the air at any time of the day or night, one can obtain some impression of the extraordinary increase in travel that has taken place in the last century[#].

[#] Between 1900 and 1914 there were four years in which more than one million Europeans immigrated into the United States. (Europeans overwhelmingly dominated immigration in that period.) Immigration into the USA reached an all-time peak in 1907 when there were 1,199,566 European immigrants and the total was only slightly less in 1914. During the first World War immigration virtually stopped, and after the war a number of laws were passed which restricted immigration, culminating in the 1927 amendment of the National Origins Act, which set an upper limit of 150,000 immigrants per year. Shipping companies had invested in ships to carry the previous large immigration traffic. To make use of their expensive assets, they therefore encouraged a new type of passenger, the tourist, who travelled for interest or to visit relations but was not an immigrant, a business traveller or one of the relatively rich who had previously travelled across the ocean. For this they modified their ships to accommodate large numbers of passengers in reasonable comfort but at relatively low cost. In "inventing" the tourist the shipping companies were creating the trade which in a later era was to fill many aircraft seats, particularly on international flights.

Travel across the north Atlantic ocean for many years set the pace for air travel, just as it had for travel by ship, and London to New York was one of the most important routes. When conditions returned to normal after World War II, the scheduled journey time from London to New York in a piston-engine Constellation was 22 hours and the cost, in 1991 dollars, of a return trip was about $5000. Two refueling stops were necessary. The first big reduction in travel time was around 1956 with the gas turbine powered turbo-prop aircraft. Because of the much greater power available, the turbo-prop travelled faster and eliminated the need to refuel for the London to New York journey, reducing the time to about 11 hours. By this time the cost was down to around $3500.

The introduction of the Boeing 707 reduced the east-west flight time to about 7 hours. (Later subsonic aircraft have actually increased the journey time relative to the 707: the 747 by about 15 minutes, the 767 by about 45 minutes.) The cost has continued to fall, so that by 1990 a full (no-restrictions) economy return fare was typically about $1000 and, by accepting some restrictions, one could travel for much less. The improvement in speed and cost were a result of the increases in power and efficiency which the jet engine produced, particularly with the introduction of the high bypass engines from about 1970. There were also other important changes, the most significant being the enormous increase in reliability, which not only reduced the delays and costs associated with failure, but more recently has allowed the use of twin engine aircraft on long over-water flights. (A summary of the history of jet propulsion is given in the appendix.)

The performance of jet engines used in subsonic civil transport aircraft engines has improved substantially since the introduction into airline service of the high-bypass engines towards the end of 1969. For the first six or so years there was little improvement in efficiency, measured in distance for which a Boeing 747 would be propelled per unit measure of fuel. About 1975, however, two alternative engines became available for this aircraft and the scene was set for intense competition. Since then, fuel utilisation for the Boeing 747 has improved by up to 24%, as shown in Fig. 1; of this improvement only about 3% arises from improvements in the airframe aerodynamics. Within quite a narrow range, all three competing engines offer similar performance. Competition between the engine manufacturers remains fierce, and there is every reason to expect performance to continue to improve, though perhaps at a slower rate than in the past seventeen years, since it may be assumed that the easier gains have already been achieved and that it will become harder to find ways to improve efficiency in the future. By way of contrast, the lift-drag ratio for the Boeing 747-400 has been determined from measurements at cruise to be about 17, whereas estimates for the Airbus 330 and 340 are between 20 and 21, demonstrating that a further 20% or so fuel saving is possible with existing airframe technology.

Much of the improvement in the engines has been in the form of better aerodynamic

performance, in particular, higher efficiency from the compression system (fan and compressor) and the turbine. These strides are extremely impressive, as is some of the underlying technology, notably in the application of computational fluid dynamics (CFD) to these components. In some situations, there has been less impact from the advances in computational methods, notably in multistage compressors, where the underlying technology is not enormously greater than it was twenty years ago.

The embracing of CFD has not necessarily led to a greater understanding of the flow, or appreciation of its constraints, and to allow progress to continue in a rational manner it is important that a solidly based understanding does exist. To obtain such understanding, it is normally necessary to simplify the flow to a model in which the relationship between the major parameters can be expressed in a simple, though approximate, form. This paper will look at some achievements that have been made in the last few years, beginning with the introduction of CFD and moving to a variety of simpler models. They will be followed by two examples of where the progress has been less like a stride than a stumble: modelling and prediction of fan flutter and the prediction of multistage compressor performance.

2.0 THE AIRCRAFT ENGINE FAN

The fan at the front of a high bypass ratio fan is one of the most conspicuous components of the civil aircraft engine and one of the most influential for overall performance. All of the air flow passes through the fan, whereas only about one sixth of the total passes through the core of the machine. Figure 2 shows photographs of two different fan blades. The one on the left is typical of 1960's design, having part-span shrouds to give it the necessary stiffness to resist vibration and flutter. The one on the right is a design of the late 1980's with lower aspect ratio, that is a much wider chord in relation to its span, and no part-span shrouds. (What cannot be seen for the more modern design is that the blade is actually hollow.) Normally these blades would be part of a rotor assembly, having a total of between 20 or 40 blades. The flow about such a blade provides the aerodynamicist with almost all imaginable difficulties. The blades can be seen to be highly three-dimensional, and until fairly recently, when numerical methods began to be able to treat three-dimensional flows, the calculation of even incompressible potential flow could not be carried out for the correct geometry. The Mach number of the entry flow relative to the blades varies from about 1.5 near the tip to slightly less than unity near the hub. Calculation of such transonic flows was also not possible until suitable numerical methods were available. The flow is further complicated by the presence of strong non-planar shock waves; the shock wave extends across the passage and intersects the suction surface (the convex surface) of the blade, where a strong shock-boundary layer interaction occurs, with local separation. Finally, there are boundary layer effects near the hub and near the casing, with those near the casing being complicated by the presence of the tip clearance and the flow through this gap.

2.1 Three-Dimensional Numerical Solutions

For all of the flow features listed above, the traditional analytical approach for calculating the flow would have led to enormous difficulties and probably little useful result. Over the last few years, numerical procedures have become generally available for computing the behaviour of blade rows using Reynolds-averaged versions of the Navier-Stokes equations. In the Reynolds-averaged versions, the turbulence effects are represented by equivalent time-averaged Reynolds shear stresses. Some empiricism is required to estimate the level of these stresses, and this has been an area in which a great deal of work has been done, but it turns out that knowing the turbulent stresses is not a very crucial issue in most turbomachines; the flow is primarily a balance between pressure and inertia. Viscous effects, including turbulent stresses, contribute little directly to this force, though the blockage produced by the boundary layers (i.e. the reduction in the effective flow cross-sectional area) can play a crucial part, especially in transonic flows which are extremely sensitive to flow area. Successful methods for predicting the steady flow in

turbomachinery components using the Navier-Stokes equations have been developed by several people, but of particular note here are those by Dawes (1987) and Adamczyk et al. (1990).

As an illustration of the capability, Fig. 3 shows the predicted and measured pressure ratio for a NASA high-speed rotor. The stagnation pressure ratio is the ordinate and the corrected mass flow the abscissa. The calculations were performed with the Adamczyk et al. (1990) code using a fairly coarse mesh of 41 points along the chord, 31 across the pitch and 31 across the span. Figure 4 shows the corresponding points for efficiency. The agreement between the measured and calculated values is generally good, particularly when it is noted that there is considerable scatter in the measurements. There is also some uncertainty about the actual blade geometry when the machine is operating, because the blade untwists under the centrifugal acceleration, and this may well explain the small discrepancy in the mass flow at the right of the figure, essentially the choking condition.

The results of three-dimensional calculations may also be presented as three-dimensional views, and two examples are shown in Figs. 5 and 6. In Fig. 5 the static pressure distribution is shown on the suction surface at a condition near the design point. The rapid change in flow variables across the shock wave intersecting the surface is apparent. Plots of this type can be used to diagnose problems such as separation near the tip or choking near the hub. Other views may also be selected, such as that in Fig. 6 which shows an axial plane about 40% of the chord back from the leading edge. In Fig. 6 the quantity shown is the Mach number of the flow relative to the blades. The increments in Fig. 6 have been chosen to emphasise the rapid change across the shock; towards the hub the blue corresponds to a Mach number of 0.95 and towards the casing the red to a Mach number of 1.05. A jagged pattern in the shock structure, partly associated with the mesh spacing, is evident.[#]

Although it is fair to describe the agreement in Figs. 5 and 6 as good, accuracy of prediction is not good enough to provide an effective alternative screening technique for designs of fan to replace the need for building and testing. This is because the benefit to overall engine performance from small improvements in component efficiency (less than one percent, for example) is so large. The precision required to discriminate between the efficiency of designs is therefore greater than that achievable by current calculation methods. Two reasons may be advanced for the current limitation on accuracy of prediction. The most obvious is that the computational grids are too coarse to give accurate numerical treatment; as will be explained later, this does not appear to be a very serious cause of inaccuracy for well-constructed methods using variables in conserved form. The other reason is that the transition from laminar to turbulent flow

---

[#] Actually the irregularity is a combined effect of the discretisation and the subroutine used to determine the colours; the jagged shape was not visible when contour lines were plotted.

cannot be accurately predicted at present, and very crude assumptions for its occurrence are used in all calculation methods. When loss mechanisms such as shock loss or loss associated with the tip clearance are the dominant cause of loss in efficiency, the significance of the prediction of laminar-turbulent transition may be negligible. For machines in which there are only small contributions to loss from shocks or tip clearance flow, however, a large part of the loss in efficiency will be caused by the attached boundary layers, and in these machines the location of transition is important in fixing the level of efficiency. The significance of transition will be discussed further below.

Figure 7 shows results of three different three-dimensional calculations and one set of measurements for the hub region of another rotor, also tested by NASA. The calculations (which were performed by Professor J. D. Denton) are, from left to right and in increasing order of sophistication, an inviscid Euler calculation, a Euler calculation in which the blockage effect of the boundary layer is allowed for and, finally, a full viscous calculation. What is noticeable and important is the similarity of the viscous calculation and the inviscid calculation including blockage, not only with one another but also with the measurements. The purely inviscid calculation, however, predicts an entirely different flow, in which a second strong shock occurs near the trailing edge. The crucial point is that the inclusion of a realistic estimate of blockage in a transonic calculation such as this is absolutely essential; this is not surprising in view of the well-known importance of area changes for flow near Mach one.

Lord Kelvin is supposed to have stated that one only understands a phenomenon if one can calculate it; such an idea would be widely accepted now. The apparent corollary, that if one can calculate a situation one understands it, has also sometimes come to be accepted. That this is false has been amply demonstrated by three-dimensional calculations of turbomachinery flows. The crucial point about the circumstances to which Kelvin was referring was that the physical situation had been simplified to the point at which the calculations could be carried out by hand or mathematical analysis could be performed. It was the act of simplifying in order to allow the calculations to be carried out which required that the essence of the problem had been extracted. Simple analyses still have a place and two examples of simple approaches to aspects of fan and compressor performance, which give insight into the underlying problem, are discussed in the next section.

2.2 Simple Analysis of Features of the Fan Flow

2.2.1 Shock loss and pressure rise in the inlet region

The shock wave produced by fan or compressor blades has been a source of difficulty in predicting fan or compressor blade performance for as long as supersonic relative inlet flow has

been used. The problem is compounded by the shock-boundary layer interaction on the suction surface, which very often involves a region of separated flow. It also is common that not one single shock is involved but a complicated system of shocks, separations and expansions. At first sight, the numerical methods should not be expected to cope with this because of the fairly coarse grid that it is necessary to use. As noted above, the numerical methods based on the Navier-Stokes equations do predict the shock-related loss quite well and it is interesting to consider why. It is also desirable to have a method which looks only at the shock structure to understand better what controls the loss and pressure rise.

A simple method was produced by Freeman and Cumpsty (1992) which took advantage of the special geometry and conditions around the leading edge of typical high-speed blades. Such blades are thin and have little camber in the forward region. This makes it possible to approximate the momentum equation parallel to the blade suction surface simply and accurately. The control volume used is shown in Fig. 8; flow enters the inlet region with Mach number and flow direction (measured from axial) of M_1 and β_1 respectively. There is at least one strong shock in the inlet region, and the flow leaves the inlet region subsonically with Mach number M_2 and flow direction equal to the blade direction in the inlet, χ_1. The incidence is therefore $\beta_1-\chi_1$. It is assumed that the flow is uniform at outlet from the control volume. Camber and loss downstream of the inlet region can be included, but will not be here.

The analysis consists of applying conservation of mass flow, momentum flux and stagnation enthalpy between inlet and outlet of the inlet control volume. The three algebraic equations (not differential equations) can be solved by hand. If the inlet relative Mach number is given, and if the flow is unchoked, the static pressure at station 2 affects the inlet flow direction. If, on the other hand, inlet relative Mach number and direction are prescribed, which corresponds to a fixed Mass flow and rotational speed, the downstream static pressure, the loss, and the Mach number leaving the inlet region M_2 are prescribed. For a particular Mach number and blade geometry (taking for simplicity the blades to have no camber or thickness), the loss coefficient is plotted against incidence in Fig. 9.

Also shown in Fig. 9 is the loss given by the method still commonly used in design (Miller et al., 1961). In this method a normal shock wave is assumed at the average Mach number across the entry to the blade passage; because the shock is taken to be normal to the flow, this approach is usually and erroneously assumed to be an *upper* bound on the loss. The two methods agree at zero incidence, but the conservation law analysis applied to the simple control volume shows a much stronger dependence on incidence. Figure 10 compares the prediction by the Freeman and Cumpsty method with measured losses. There is reasonably good agreement for the absolute level of loss and, perhaps more importantly, for the trends with both incidence and with

rotational speed.

The mechanism of loss in the inlet region is highly complicated. To obtain losses much higher than that of a normal shock at the inlet Mach number requires a system of shock boundary layer interactions, separations and reattachments. The agreement between the measured losses with those predicted by the simple conservation law method, using such a small amount of input information about the geometry of the blade, is remarkable. It confirms that the flow is determined by overall conservation constraints on mass flow, momentum flux, and stagnation enthalpy. These are the same conservation variables which are used in the numerical solution of the Navier-Stokes equations; the good agreement provided by finite difference methods for which the mesh is not fine enough to resolve boundary layers and shocks is associated with the use of a formulation in terms of these variables.

2.2.2 Tip clearance flow and loss

A significant fraction of the loss in compressors is associated with the tip clearance flow, and most of the uncertainty in the prediction of the performance is associated with this, as discussed further below. Here, two different approaches towards understanding the flow in basic terms will be described: the first, by Chen et al. (1991), allows the position of the clearance flow vortex to be predicted by simple inviscid theory; while the second, by Storer (1991), allows the loss to be predicted. Both these approaches are founded on the work done at Caltech, reported by Rains (1954).

The analysis by Chen et al. is based on the recognition that, for thin blades, the *steady* three-dimensional flow about a tip can be approximated by an *unsteady* inviscid flow in *two*-dimensions: this correspondence is illustrated in Fig. 11. The development of the equivalent unsteady flow is tracked in time by considering discrete vortices shed from the tip. Figure 12 shows the geometry of the tip and Fig. 13 the comparison of the predicted and measured positions of the vortex core for a large number of measurements. The agreement is generally very satisfactory, confirming that the view of the flow implied by the method is reasonable, in particular that viscosity plays only a very small part in the development of the flow. As further confirmation of the approach, Fig. 14a shows a Navier-Stokes prediction for the streamlines over a rotor tip (performed by J.A. Storer using the Dawes (1987) code) while Fig. 14b shows a comparison of the envelope of the predicted streamlines with the prediction of Chen et al.

The second tip clearance approach began with a Ph.D project begun by Storer in about 1987 which was to see how Navier-Stokes calculation methods could be modified to allow satisfactory predictions to be made of the tip clearance flows. It was accepted at the time that the grid would inevitably be too coarse to model the clearance flow processes at all well. As a first

step, measurements of the clearance flow associated with a cascade of compressor blades were made and compared with calculations using the Dawes (1987) code. It turned out that good predictions of many features of the flow and the losses associated with the clearance could be made. This is illustrated in Fig. 15, in which Storer's (1991) measurements and computations for stagnation pressure loss for a cascade with zero clearance and with clearance equal to 2% of chord are shown. For this case, there were only five grid points across the clearance height. The corner separation observed with no clearance is predicted, as is the absence of the separation and the presence of a vortex when there is a clearance. If a method with only about five grid points across the clearance gap can predict the flow reasonably well, it indicates that the global issues determine it and that details of the local flow structure are largely irrelevant[#] .

The flow normal to the blade through the clearance gap is similar to the potential flow shown in Fig. 16. The flow experiences negligible loss while it passes through the clearance, so the *stagnation* pressure of most of the clearance flow is equal to that of the freestream. On the downstream side of the clearance gap, along the suction surface of the blade, the curvature of the streamlines in the cross-section of Fig. 16 is negligible, so the *static* pressure is uniform across the gap and equal to that on the suction surface at this position along the blade chord. As a result, the *speed* of the flow in most of the clearance flow is equal to that of the freestream on the suction surface: the direction, however, is not the same and a shear layer or vortex sheet exists between the flow emerging from the clearance gap and the freestream flow which has passed near the suction surface. The situation is illustrated in Fig. 17, where a shear layer is shown with flows of equal speeds but with a difference in direction equal to ζ. The direction difference depends on the local pressure difference across the blades and varies somewhat along the blade chord. An average value of ζ can be specified, and this depends almost entirely on the overall turning of the flow outside the endwall region and on the blade row solidity.

If there is no significant loss experienced by the flow in passing through the clearance, the loss must be produced downstream by a mixing process. Most of it actually occurs in the shear layer very close to the edge of the blade, but it is not necessary to examine any of the details of the flow to estimate how large this loss is. Figure 18 shows a generic mixing problem, where the difference between the two streams is only in direction, and using this one can estimate the loss produced by the tip clearance flow. A comparison of the measurements and estimates for the cascade tested by Storer are shown in Fig. 19 for a range of tip clearance magnitudes. Results are

---

[#] Quite separate from the issue of predicting the tip clearance flow, the comparison between measurement and calculation with clearance in Fig. 15 shows the weakness of the calculation method in predicting the fully attached flow. Well away from the endwall the flow is more or less two-dimensional and a narrow wake is measured. The calculated wake, however, is substantially thicker and this discrepancy is primarily because the transition is not properly modelled.

shown for two different thicknesses of boundary layer on the endwall, but this introduces only an amount added on and does not affect the estimates of the simple model. The model accurately predicts the trend of loss with clearance everywhere except for the thin endwall boundary layer with small clearances. In this regime a corner separation, which is an entirely different flow phenomenon, dominates. The validity of the simple theory helps to explain why the Navier-Stokes scheme with only five grid points across the clearance can produce good predictions of flow structure and loss - the overall pattern is determined by global conservation and is insensitive to the details of the flow. Indeed, the situation may be stated more precisely by saying that the details must adapt to be compatible with the overall constraints on mass, momentum and energy.

2.2.3 *Condensation in long engine intakes*

Another problem showing the utility of a simple model that gives a clear physical understanding arose during ground testing of a high bypass jet engine for which the thrust was increased without increasing the diameter of the inlet. The phenomenon here involves a rather different type of simplified modelling to those addressed above. The engine had a long, straight intake, extending approximately 8 m from the front of the intake to the engine face. The increased thrust required an increase in the mass flow rate, and as a result the Mach number of the axial flow along the duct was close to 0.8 over most of the length. Figure 20 shows the inlet, the inlet control volume and the Mach number contours calculated for dry air with the engine operating at zero forward speed. When the engine was tested under sea level static conditions, it was found that the fan overspeeded, there was a decrease in stagnation pressure along the inlet (well away from the walls where boundary layer effects were expected), and water was found in the pitot tubes used ahead of the fan. Condensation was indicated by the presence of water in the tubes and Rolls Royce approached Dr. J.B. Young, an expert in the field of steam condensation, to examine the processes and phenomena involved.

The mass fraction of water vapour in the air is proportional to the relative humidity and a strong function of temperature: for a relative humidity of 70%, the mass fraction of water vapour is about 0.5% at 10°C, rising to about 1.9% at 30°C. When the air is accelerated into the inlet, the static temperature falls and, for most conditions met in sea level test conditions, drops below the saturation temperature. Given a long enough time, with an adequate supply of nucleation sites, the water vapour would condense, although this does not necessarily occur in an intake. For an under-wing intake on a large engine, the time the flow is in the intake before reaching the engine inlet face may be less than 0.01 seconds. The supersaturated air can therefore remain in a metastable condition (out of equilibrium) until reaching the engine fan. A long intake makes condensation more probable, however, since it provides more time for the condensation process.

The significance of condensation is the latent heat which is released by the water droplets and transferred to the gas stream. Heat flow to a subsonic gas stream leads to an acceleration of the flow and sufficient heat transfer will raise the Mach number to unity, at which condition the flow is choked. The conditions at which choking due to condensation occurs are shown in Fig. 21 as a function of inlet flow Mach number, calculated by treating the air as a dry gas. The results are for various values of inlet stagnation temperature and relative humidity of the inlet gas at stagnation conditions. Because the dry-air Mach number was about 0.8 for much of the duct in this case (see Fig. 20) a broken line is drawn on Fig. 21 at this level. For choking to occur because of condensation to the equilibrium condition with an incoming dry-air Mach number of 0.8, an incoming relative humidity of 40% would be needed at 30°C, rising to 75% at 10°C. Table 1 shows parameters at three test conditions for which choking occurred; what is of particular note is the large increase in the axial velocity after the condensation, denoted by V_2, compared to the value ahead of it, denoted by V_1. It is this increase in axial velocity which led to the overspeed of the fan, while the heat addition and the irreversibility of the condensation process gave the decrease in stagnation pressure.

TABLE 1
LIMITING FLOW CONDITIONS FOR THERMAL CHOKING

	Inlet Stagnation		Ahead of Condensation		After Condensation $M_2 = 1$	
Test	T_0 (°C)	Rel. Humidity	M_1	V_1 (m/s)	V_2 (m/s)	% Condensed
R4	20.0	48	0.80	260	302	66
R1	25.3	52	0.78	257	306	58
R3	18.8	72	0.77	251	301	67
			Out of Equilibrium		In Equilibrium	

Whether condensation occurs depends on the degree of supercooling (i.e. the difference between the temperature of the supersaturated gas and the saturation temperature), the time available for condensation (which is greater in a long duct), and whether there are suitable dust particles to act as nucleation sites. Around the lips of the intake, the Mach numbers get as high as 1.2, and at this condition the supercooling is large enough for condensation to occur even without dust (homogeneous condensation). For a Mach number of 0.8, it is essential that dust of the correct size is present in the air, but most engine test facilities are in industrial areas where dust is plentiful and measurements confirmed that, for the test described here, the atmosphere was suitably seeded with small dust particles.

The calculations performed by Young were one-dimensional, and therefore highly simplified, but they clearly pointed out the mechanism for the observed behaviour. They demonstrated the importance of water vapour in tests and drew attention to the features which are most significant in defining the magnitude of the observed effects. This example serves as a reminder of the importance of one-dimensional gas dynamics, especially that with heat addition; it also points to the likelihood that there are technologically interesting two- and three-dimensional flows of moist air in which condensation can occur.

2.3 An Unresolved Problem for Fans – Flutter

Up to this point in the paper, the emphasis has been on successes, either from the numerical solution of the three-dimensional Navier-Stokes equations or from simple analyses which can predict important features of the flow and provide insights into the controlling mechanisms and parameter trends. This section addresses flutter, a self-excited aero-elastic vibration of the blades which remains a phenomenon where the mechanism is still neither adequately understood nor predictable. Being self-excited it can occur with no outside disturbance to drive it; the aero-elastic aspect is that the deformation of the blades is an important part of the process which leads to the fluctuating aerodynamic pressures and forces on the blades.

Many fans still have part-span shrouds, such as those shown in Fig. 2a, to stiffen the blades. These shrouds lock tightly with the shrouds of adjacent blades so that if flutter occurs the entire fan assembly of some thirty blades vibrates together. A holographic interferogram of such a vibration is shown in Fig. 22; in this particular mode there are three diametral nodes. There are some aspects of the solid mechanics of flutter which are still incompletely understood, notably the damping, but it is with the aerodynamics of flutter where the inadequacies are most obvious.

Figure 23 shows a sketch of pressure ratio versus mass flow for a fan on which Mikolajczak et al. (1975) shaded regions of possible flutter. The solid lines denote operating lines of constant corrected rotational speed and a possible operating line. The regions indicated as flutter are where it had been found in past experience. Names are given to some of the regions, for example stall flutter and choke flutter, but these are not necessarily descriptive of the actual processes involved and may indeed be misleading. Of particular note for the operation of fans is the narrowing of the operating range at high speed between regions of flutter, so at the 100% design speed there is only a narrow range between so-called choke flutter and surge flutter.

In January 1989, a Boeing 737-400 crashed after only a few weeks of airline service and the primary cause of the accident was determined to be fan flutter (Trimble, 1990). The

circumstances clearly demonstrate that flutter is not understood. The engine thrust for the 737-400 was approximately 10% higher than for the 737-300 and to obtain this from engines of the same type it was necessary to increase the rotational speed of the fan by between 2 and 3%. Because this was seen as a minor increase in speed, it was decided by the manufacturer that flight testing was unnecessary. This decision was approved by the Federal Aviation Authority (the body which approves procedures for testing of aircraft and engines). The engine was therefore tested at the increased fan corrected speed required for the extra thrust, but on the ground. To reach the increased fan speed at sea level without overheating the turbine, it was necessary to increase the bypass nozzle area and the effect of this is illustrated in Fig. 24, a plot# of fan pressure ratio versus mass flow rate. The solid curves with circular symbols are measured lines of constant rotational speed, while a hand-drawn line indicates what the 103% line might be like (speed sufficient for the extra thrust). The two lines stretching diagonally across the diagram are the working lines as designed and with the increased nozzle area. Increasing the bypass nozzle at a given rotational speed reduces the pressure ratio and therefore also the power needed to drive the fan. The shaded regions are guesses for where the flutter boundaries might occur, based on the ideas of Fig. 23. With the increased nozzle, the working point at 103% speed is just between flutter regions, whereas with the correct nozzle the flutter boundary is just crossed. Consequently, when maximum corrected speed was reached, which would be during climb when fan speed was high and the altitude was sufficient for the temperature to be substantially below that at sea level, flutter could and did occur.

After the fatal crash, the damage to the engine was too great to definitively determine the cause of initial failure. Just six months later, however, failures occurred on identical engines on two separate flights. Because the pilots took the correct action to deal with the situation, no crash occurred. As a result, high-cycle fatigue failure of the outer part of a blade was unequivocally determined, with flutter as the probable cause. Subsequent flight testing of the engine demonstrated that flutter indeed occurred at maximum climb thrust near 20,000 ft.

At the present time, all the aerodynamic mechanisms actually involved in flutter at the transonic conditions are not known. Various possible mechanisms have been postulated, and some have been demonstrated with two-dimensional models in wind tunnels, but quite how the mechanical movement of the blades locks into the aerodynamic field is not understood. Flutter is a possibly fatal flaw in an engine, yet at present one can only ensure that it does not occur through a costly and time-consuming flight test programme. Flutter cannot be designed out aerodynamically

---

Although Fig. 24 is for a real fan of similar duty, it is not one of the type for which the failure occurred.

by any rational procedure, since it is not known what the mechanism is, and the "fixes" are at present restricted to mechanical changes to alter the frequencies or to increase the damping. The proper prediction of flutter probably requires an unsteady, three-dimensional, transonic, viscous calculation and this needs to be performed for every likely mechanical mode of the system. Such methods are nearly ready, though it is not known how the coupling of the mechanical dynamics and fluid dynamics can be most effectively carried out.

3.0 THE MULTISTAGE COMPRESSOR

Most of the material described above for fans has been in the nature of success, with aerodynamic problems overcome despite the apparent intractability of the highly three-dimensional transonic flow which characterises these devices. The treatment of multistage compressors would, at first sight, appear to be more straightforward than fans: the flow is often fully subsonic and the blades are much less twisted, so the flow may be expected to be more nearly two-dimensional. The reality is that many aspects of the flow in multistage compressors are not understood, and the prediction of the flow using computational fluid dynamics (CFD) has been less helpful.

There are two principal reasons why prediction using CFD has been less successful in multistage compressors, both of which are associated with the difficulty of imposing the inlet and outlet boundary conditions into each blade row. The first difficulty is outlined diagrammatically in Fig. 25. If the flow about the middle blade row is to be computed, it is necessary to take into account the influence of the adjacent blade rows. The rows nearest to the one under consideration exert the largest influence, the ones further removed exert a weaker influence; likewise the flows about the adjacent blades are affected by all other rows. In some respects, numerical methods which rely on iterative solutions are well suited to solving this problem. Recently, methods have become available which take into account the most important part of the interaction between the rows, which is the radial (i.e. spanwise) variation in velocity, stagnation pressure, and stagnation enthalpy (Dawes, 1992; Denton, 1992). The variation in the circumferential direction is averaged out upstream and downstream of each blade row in order to solve for a steady flow in that row. This has the advantage of maintaining the convenience of a steady solution, and it is *probable* that this approximation introduces smaller errors than those arising from other sources, as discussed below. This procedure is not, of course, valid if the unsteady properties of the flow, such as blade vibration, are needed. When it is necessary to look at the flow with the circumferential variations in the inlet and outlet boundary conditions retained, the problem is very much more complicated, and is going to take very much longer to calculate; steps have been taken in formulating this problem (Adamczyk, 1985; Giles, 1992), but there are still no firm estimates of how important the various terms are.

The second difficulty is shown schematically in Fig. 26 and is associated with the boundary layers which form on the hub and casing walls. The term boundary layer is conventionally used for these endwall regions because the velocity, stagnation pressure, and stagnation temperature vary rapidly in the vicinity of the walls, but in reality the regions near hub and casing are very different from the normal boundary layers growing on, for example, wings. The flow is highly unsteady and three-dimensional, with regions of three-dimensional separation, usually corner separations, in many of the blade rows. The tip clearance flow, discussed above in connection with the fan, exer-

cises a large (and sometimes a dominant) influence on the behaviour of the endwall region. In addition, the flow in the endwall region is affected by the inlet conditions. Taken together, these effects produce a flow of great complexity, and the difficulty is exacerbated in the multistage compressor where the endwall flow leaving one blade row becomes inlet flow to the next. In this environment, the variations, uncertainties and errors in prediction can increase cumulatively. A modern core compressor might have a pressure ratio of twenty-to-one, with twenty or more rows of blading. Any calculation method introduces some inaccuracy or error, particularly in the endwall region, and the cumulative uncertainty from errors in calculating the flow in one blade row being passed to the next row is so great that the prediction of the flow is reliable only for the first few rows.

Some understanding of the endwall flow can be obtained by considering only the axial component of the momentum, drawing a control volume upstream and downstream of a single blade row, as is shown in Cumpsty (1989). The pressure gradient in the axial direction is high, when non-dimensionalised in terms of the parameters of the endwall boundary layer. This gradient would certainly separate the endwall flow were it not for the presence of the blades; it is only because most of the pressure gradient is balanced by the axial force of the blade that flow reversal does not take place. Outside the endwall region, near midspan, the pressure gradient and the blade force balance exactly, but near the endwalls there is no reason for this to be the case. If for any reason the blade force in the endwall region drops below that needed to balance the pressure gradient, the endwall boundary layer grows rapidly across the blade. As mentioned above, small local changes in the aerodynamic behaviour of the blade can thus have a very large effect on the endwall boundary layer. At the present time, the factors which control the blade force near the endwall are not anywhere near being understood; though the three-dimensional Navier-Stokes methods can predict much of the endwall behaviour, their predictions cannot be exact and the cumulative effect of errors renders the estimates after the first few rows useless.

The endwall boundary layer quantity which has most effect is the blockage, equivalent to the loss in flow area caused by the reduced velocity. The best estimates for this are still obtained from correlations produced by Smith (1969) in terms of tip clearance, blade row staggered gap, and the ratio of pressure rise to the pressure rise at stall. The intractability of the endwall flow is demonstrated by the very small progress in the last 20 years for an issue widely recognised as the biggest single limitation on the accurate design of multistage compressors. To allow high pressure ratio designs to succeed, it is common to include many rows of variable and adjustable stator rows. Variable rows can be altered according to an experimentally determined schedule as the speed of the machine changes, while the adjustable rows can be fixed at suitable settings determined by experiment during development. It is in this way that industry has learned to live with its inadequate predictive ability for the endwall regions of multistage compressors.

4.0 CONCLUSIONS

Competitive pressure has led to improvements in fuel efficiency of about 20% for high bypass engines since their introduction into airline service. At the same time, the thrust of the engines has been greatly increased. Much of the increase in performance has been achieved by improvements in the compression system efficiency. The enormous strides made in the last twenty or so years in the aerodynamic performance of compression systems are most pronounced in the case of the fan, where numerical methods have made possible prediction of many aspects of the steady flow and given the designer the ability to deal with truly three-dimensional geometry. However, even the most advanced solvers of the three-dimensional Navier-Stokes equations are unable to obtain adequately precise predictions of the efficiency. This is principally because these methods are not very good for predicting the viscous losses of fully attached flows, which predominate in good fan designs. The same methods are surprisingly good at predicting behaviour, including loss, associated with strong shocks, regions of separation and tip clearance flows, even though the grids used are too coarse to resolve details of the flow at all accurately. The three-dimensional methods do not always provide a convenient and clear insight into the controlling mechanisms and constraints on the flow, and for this there is still scope for simple methods which do provide insight into the principal processes and mechanisms.

As examples of simple methods it is reasonable to quote the following:

i) A method is able to predict many aspects of the flow caused by the shock structure around the leading edge of fans, including the loss. The predicted trends of loss with speed and incidence, as well as estimates for the absolute level of loss, agree reasonably with measurement. The loss is always higher than that of a normal shock and therefore also higher than the method normally used in design.

ii) An inviscid method is able to predict the position and strength of the tip clearance vortex.

iii) A model treating the tip clearance flow as a case of simple mixing is able to predict the level of tip clearance loss, to demonstrate its dependence on the magnitude of the tip clearance area in relation to blade passage area and on the overall loading of the blades. The same method shows the insensitivity of clearance loss to the details of the main flow about the tips.

Simple methods such as these not only demonstrate the controlling features of the flow, but point to the underlying reason why some features of the flow (e.g. shock loss, clearance loss) are well predicted by three-dimensional numerical methods, even when the grid is too coarse to resolve details: the formulation of the numerical methods in terms of conservation variables reflects the underlying constraint on the flow, conservation of mass, momentum and energy.

If water vapour condenses in the inlet during static operation, there are surprisingly large effects on performance. A simple one-dimensional analysis has shown how the transfer of the latent heat of the condensing vapour can accelerate and even choke the inlet flow.

Flutter of fans is one of the areas where there has been relatively little progress so far in understanding the aerodynamics of the flow. It seems very likely that, within the next few years, three-dimensional computations will be developed to predict some aspects of the flow, including the involvement of the boundary layer. At present, there is no alternative but to test a fan over its whole operating range to demonstrate absence of flutter; even then, the variability in mechanical damping with operating altitude and with time of service gives grounds for concern.

Another area in which there has been disappointing progress is the aerodynamics of multistage compressors. The predictive capability is much less well-developed than for the fan. The interaction between blade rows, at least in the time-mean, circumferentially-averaged sense, seems to be accomplished. The biggest single stumbling block is the behaviour of the boundary layers on the hub and casing, since these exert a very large effect on the performance of the multistage compressor. There has been no significant increase in predictive ability for the endwall boundary layers since about 1969, notwithstanding the widespread recognition of their importance.

5.0 <u>ACKNOWLEDGEMENTS</u>

6.0 <u>REFERENCES</u>

Adamczyk, J.J., 1985, "Model Equations for Simulating Flows in Multistage Turbomachinery", Paper 85-GT-226 presented at the 30th ASME Gas Turbine Conference and Exposition, Houston.

Adamczyk, J.J., Celestina, M.L., Beach, T.A., Barnett, M., 1990, "The Simulation of Three-Dimensional Viscous Flow Within a Multistage Turbine," *Trans ASME Journal of Turbomachinery,* Vol. 112, pp. 370-376.

Chen, G.T., Greitzer, E.M., Tan, C.S., Marble, F.E., 1991, "Similarity Analysis of Compressor Tip Clearance Flow Structure," *Trans ASME Journal of Turbomachinery,* Vol. 113, pp. 260-271.

Dawes, W.N., 1987, "A Numerical Analysis of Three-Dimensional Viscous Flow in a Transonic Compressor Rotor and Comparison With Experiment," *Trans ASME Journal of Turbomachinery,* Vol. 109, pp. 83-90.

Cumpsty, N.A., 1989, *Compressor Aerodynamics,* Longman Scientific and Technical Publishers, Essex, England.

Dawes, W.N., 1992, "Toward Improved Throughflow Capability: The use of Three-Dimensional Viscous Solvers in a Multistage Environment," *Trans ASME Journal of Turbomachinery,* Vol. 114, pp. 8-17.

Denton, J.D., 1986, "The Use of a Distributed Body Force to Simulate Viscous Effects in 3D Flow Calculations," Paper 86-GT-144, presented at the ASME International Gas Turbine Conference and Exhibit, Düsseldorf.

Denton, J.D., 1992, "The Calculation of Three-Dimensional Viscous Flow Through Multistage Turbomachines," *Trans ASME Journal of Turbomachinery,* Vol. 114, pp. 18-26.

Freeman, C., Cumpsty, N.A., 1992, "Method for the Prediction of Supersonic Compressor Blade Performance," AIAA *Journal of Propulsion,* Vol. 8, pp. 199-208.

Giles, M., 1992, "An Approach for Multi-Stage Calculations Incorporating Unsteadiness," Paper 92-GT-282, presented at the 37th ASME International Gas Turbine and Aeroengine Congress and Exposition, Cologne.

Hawthorne, W.R., 1957, "Some Aerodynamic Problems of Aircraft Engines," *Journal of the Aeronautical Sciences,* Vol. 24, pp. 713-30.

Lerner, W., 1975, *Historical Statistics of the US. Colonial times to 1970, Part 1,* US Department of Commerce, Bureau of the Census.

Mikolajczak, A.A., Arnoldi, R.A., Snyder, L.E., Stargardter, H., 1975, "Advances in Fan and Compressor Blade Flutter Analysis and Prediction," AIAA *Journal of Aircraft,* Vol. 12, pp. 325-32.

Miller, G.R., Lewis, G.W., Hartmann, M.J., 1961, "Shock Losses in Transonic Rotor Rows," *Trans ASME Journal of Engineering for Power,* Vol. 83, pp. 235-42.

Rains, D.A., 1954, "Tip Clearance flow in Axial Flow Compressors and Pumps," California Institute of Technology, Hydrodynamics and Mechanical Engineering Laboratories, Report No. 5.

Smith, L.H., 1969, "Casing Boundary Layers in Multistage Compressors," *Proceedings of the Symposium on Flow Research on Blading,* Brown Boveri & Co Ltd, Baden Switzerland. (Also in Dzung, L.S. (ed), *Flow Research on Blading,* Elsevier Publishing Company, 1970.)

Storer, J.A., 1991, "Tip Clearance Flow in Axial Compressors," PhD Dissertation, University of Cambridge.

Trimble, E.J., 1990, *Report on the Accident to Boeing 737-400 G-OBME near Kegworth, Leicestershire on 8 January 1989.* Aircraft Accident Report 4/90, Air Accidents Investigation Branch, Department of Transport, HMSO London (ISBN 0 11 550986 0).
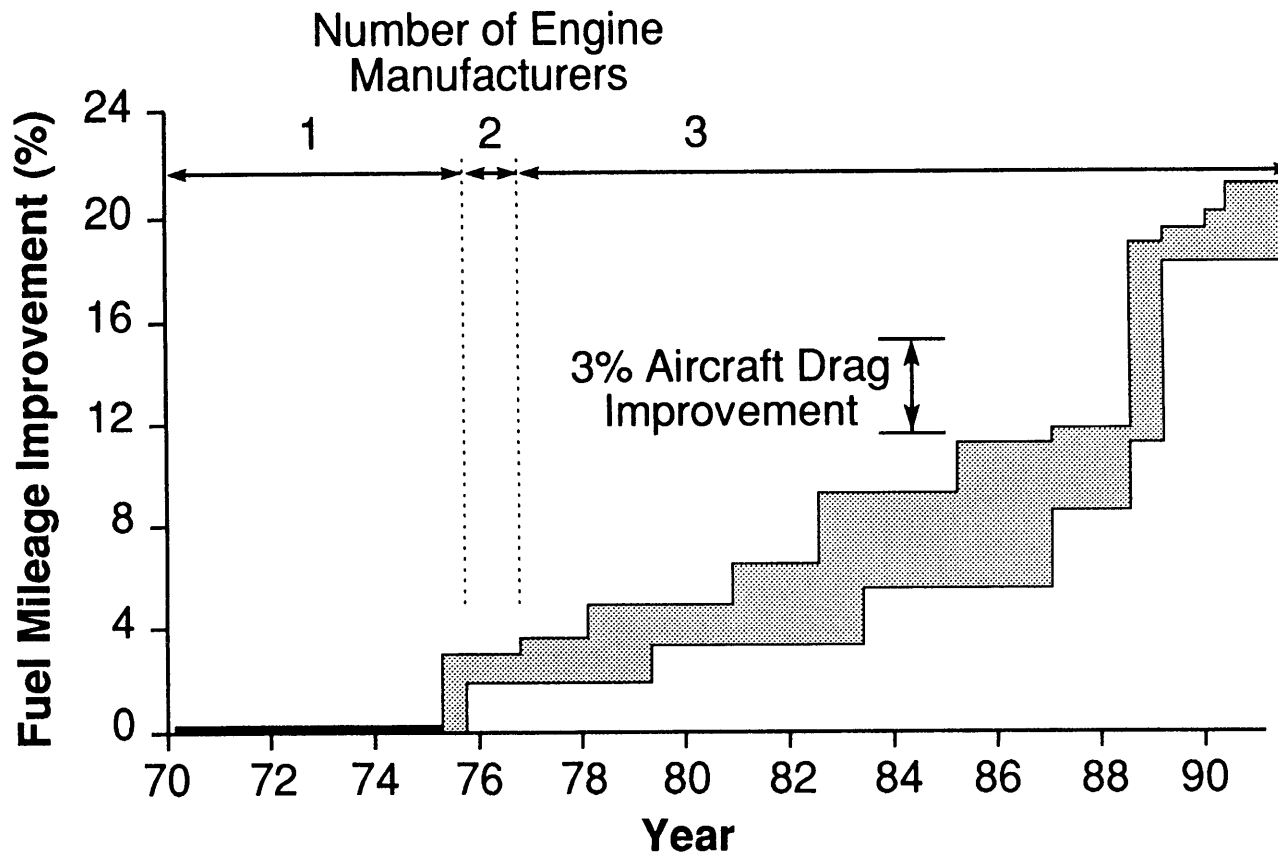
Fig. 1: Improvements in fuel consumption for Boeing 747. Shaded region shows range of performance for different engine manufacturers. Discontinuities in border correspond to new engine type introduction.
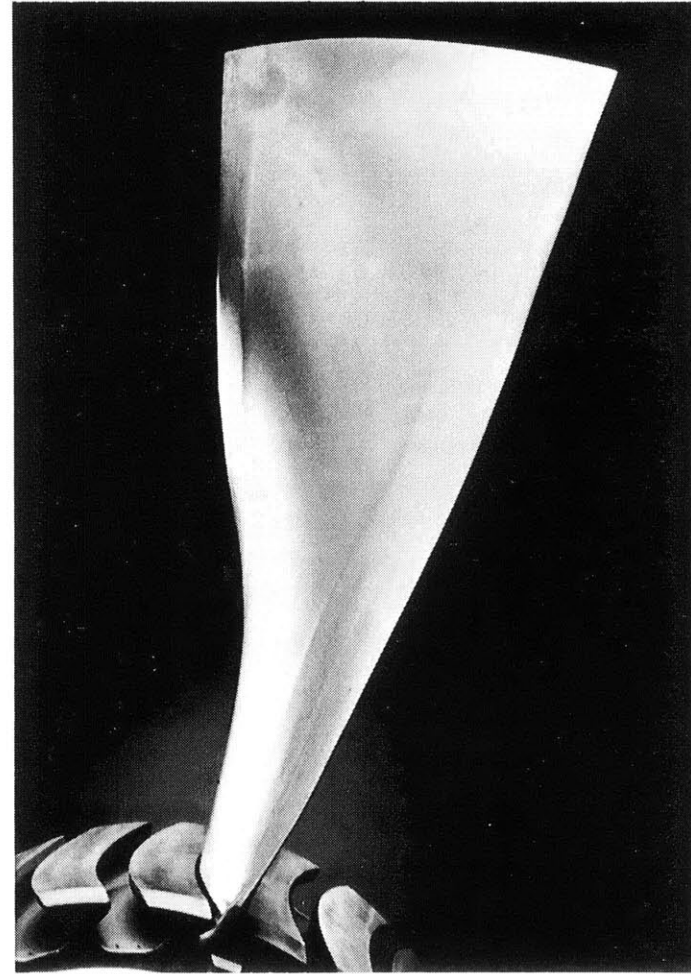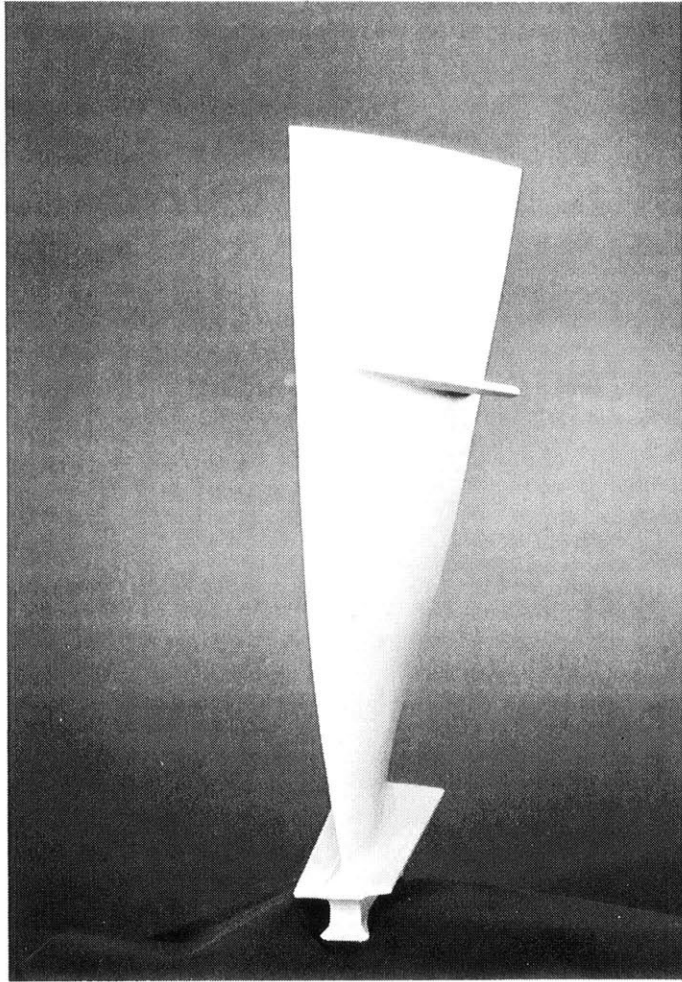
Fig. 2: Two aircraft engine fan blades: on the left, a blade from the 1960's; on the right, a modern hollow blade.
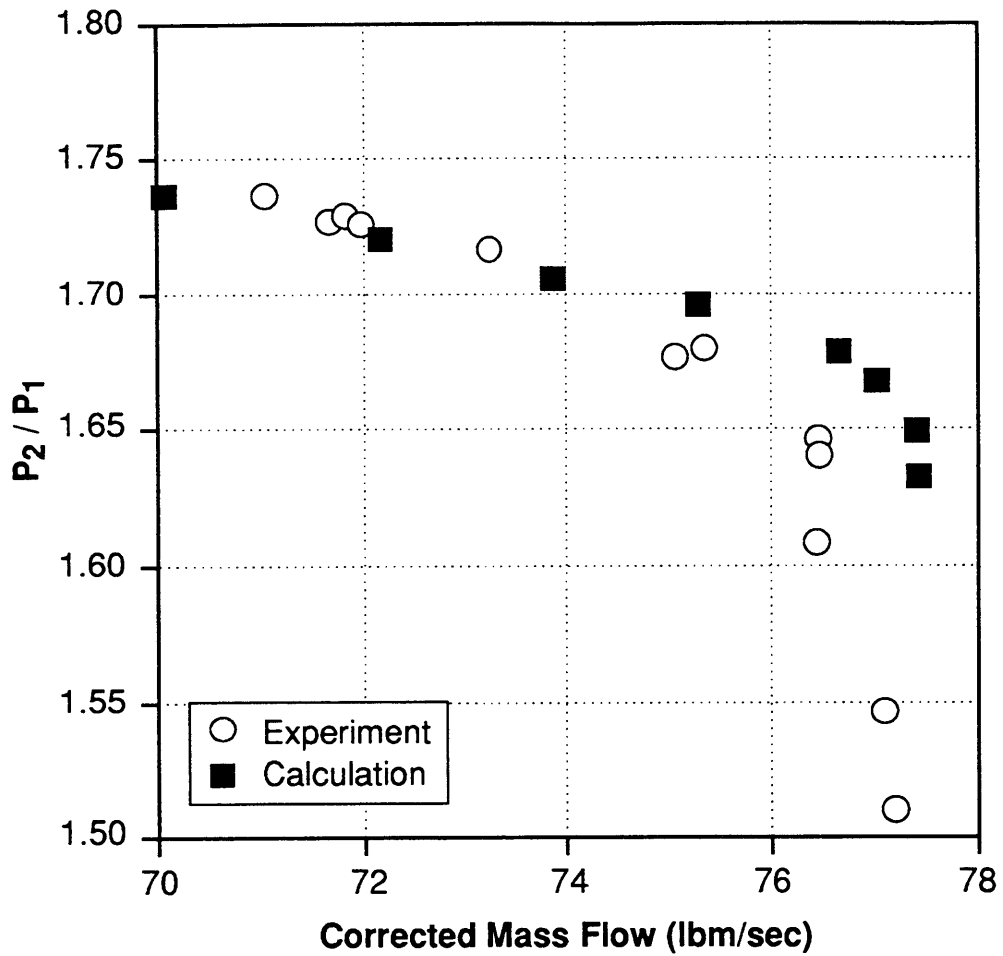
Fig. 3: Stagnation pressure ratio for NASA Rotor 67. Calculation by Adamczyk using Navier-Stokes code.
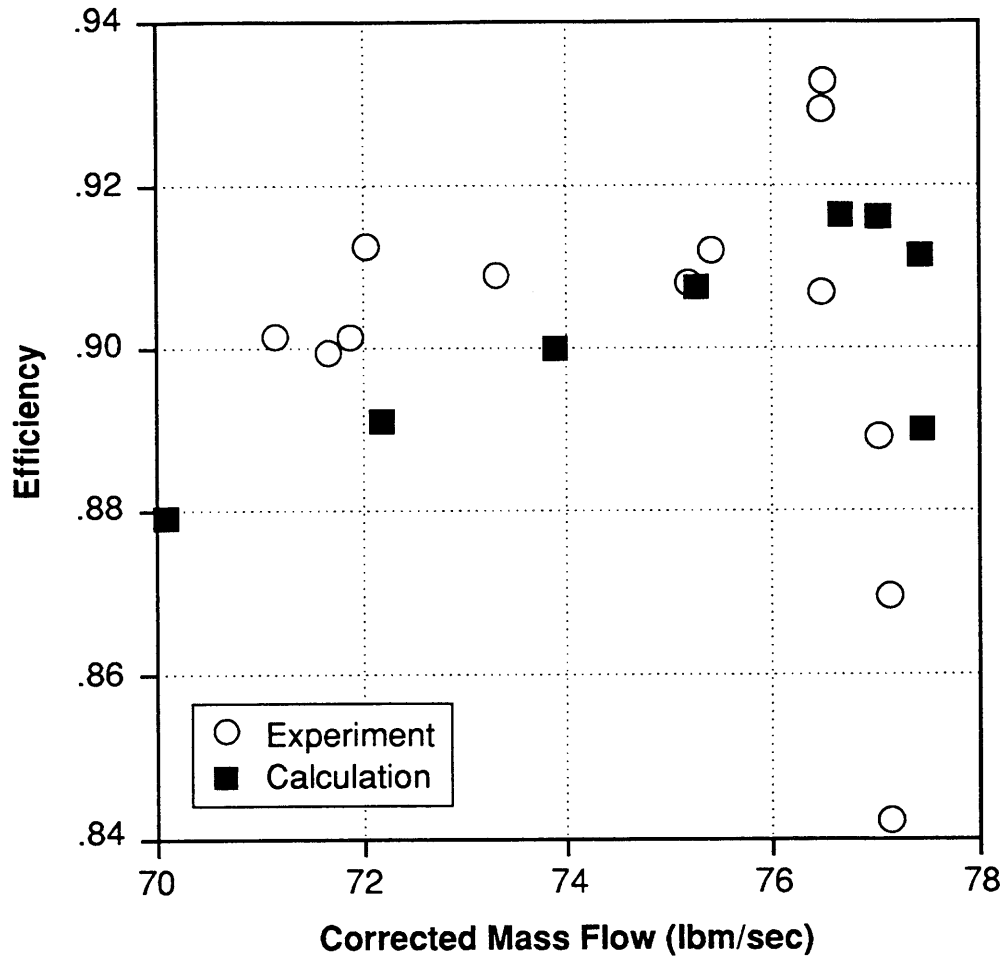
Fig. 4: Efficiency for NASA Rotor 67. Calculation by Adamczyk using Navier-Stokes code.
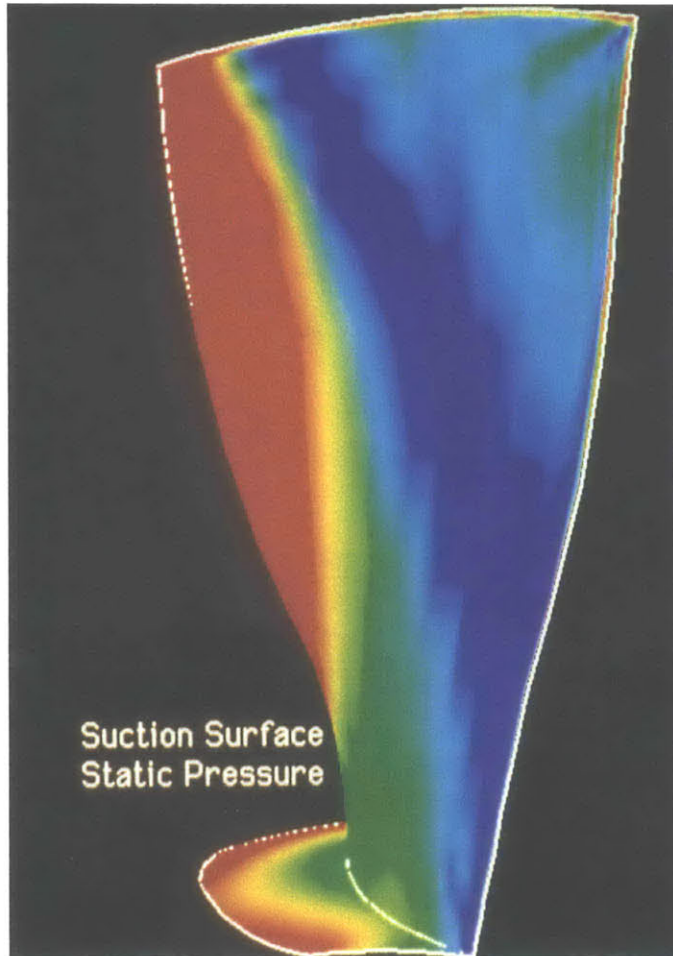
Fig. 5: A view of the suction surface of NASA Rotor 67 showing static pressure on the surface. Blue and red indicate low and high pressure respectively.
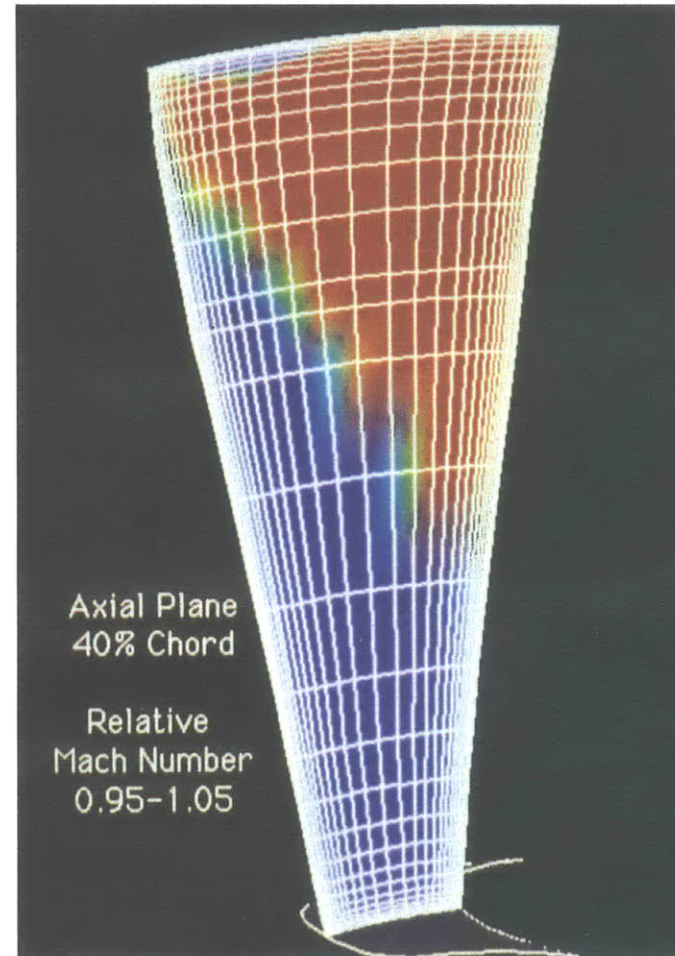
Fig. 6: An axial section through the passage of NASA Rotor 67 showing the variations in relative Mach number and superimposed computation mesh. Red: $M_{rel} \geq 1.05$; blue: $M_{rel} \leq 0.95$.
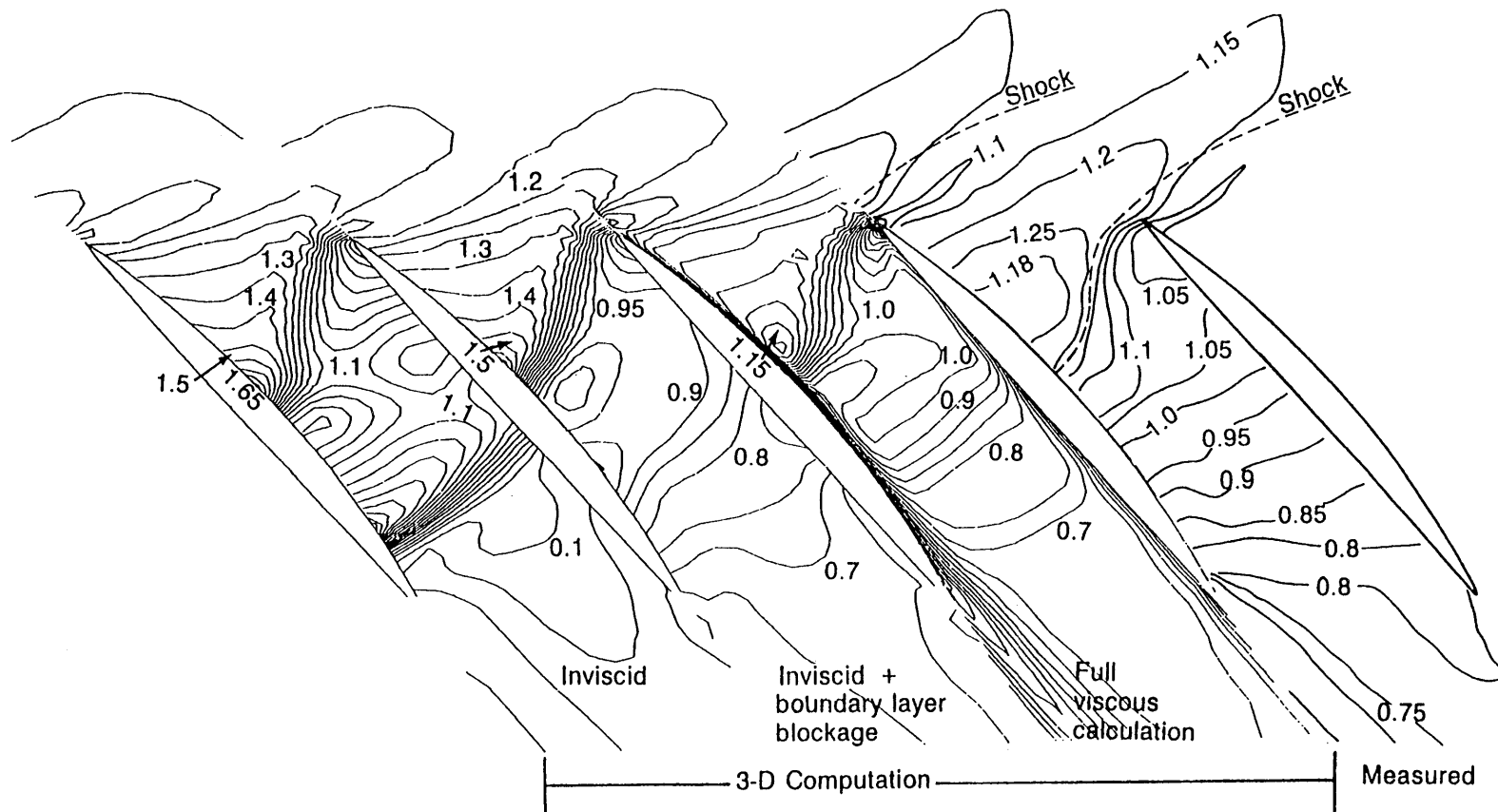
Fig. 7: Mach number contours near hub of NASA Rotor 33. Calculations by Denton, measurements from Chima and Strazisar (1983).
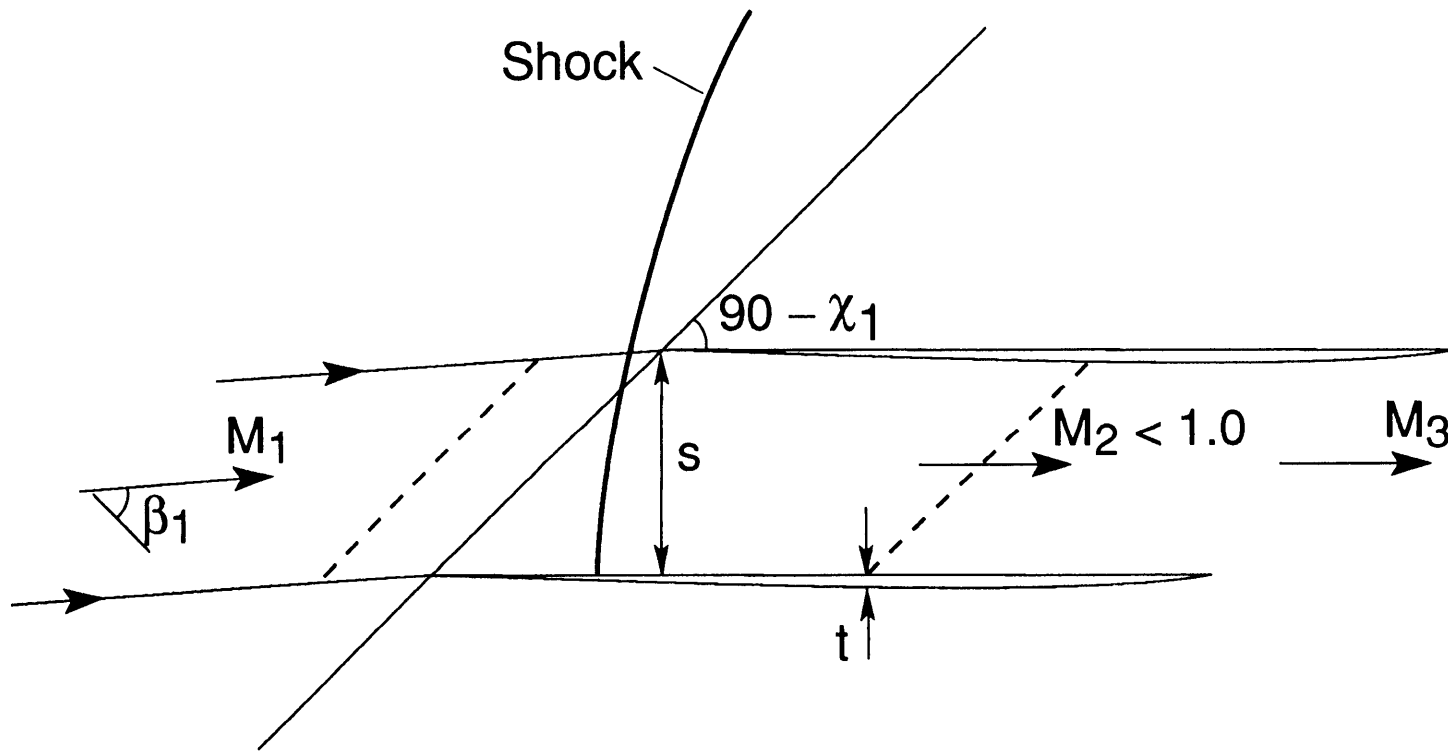
Fig. 8: Unchoked blades with control surface around leading edge and inlet region.
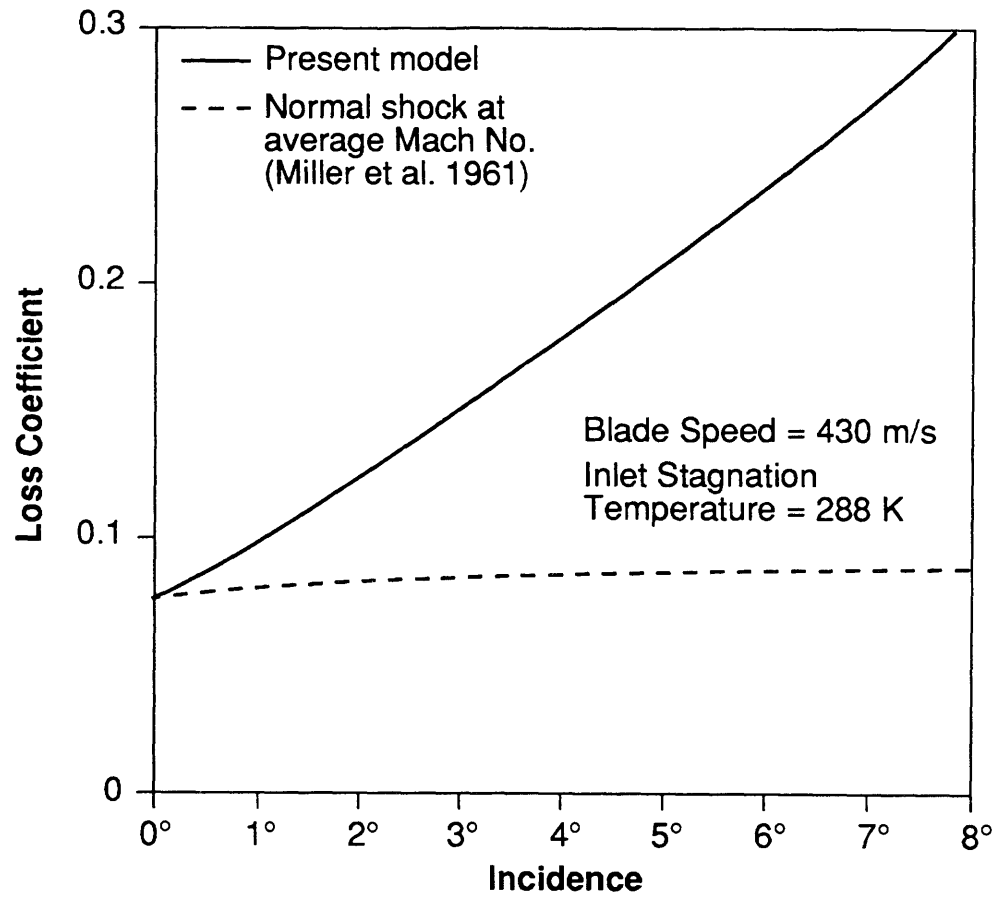
Fig. 9: Loss coefficient vs. incidences for uncambered blades of zero thickness.
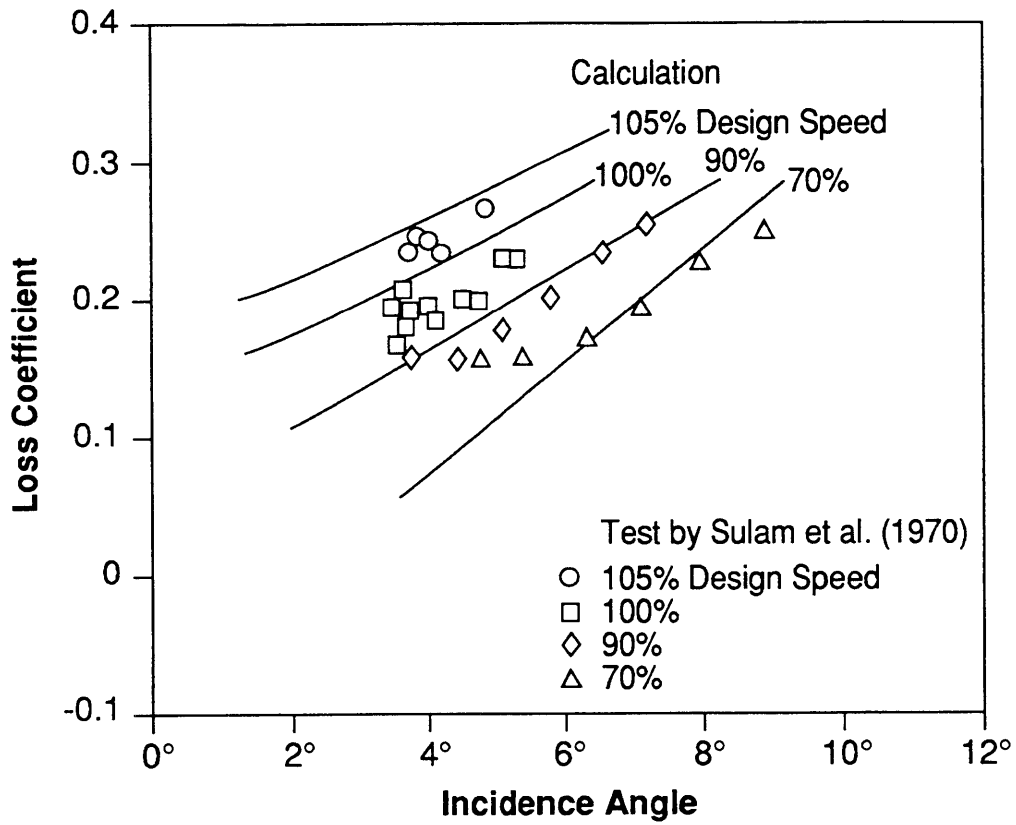
Fig. 10: Loss coefficient vs. incidences; comparison of calculated and measured values.
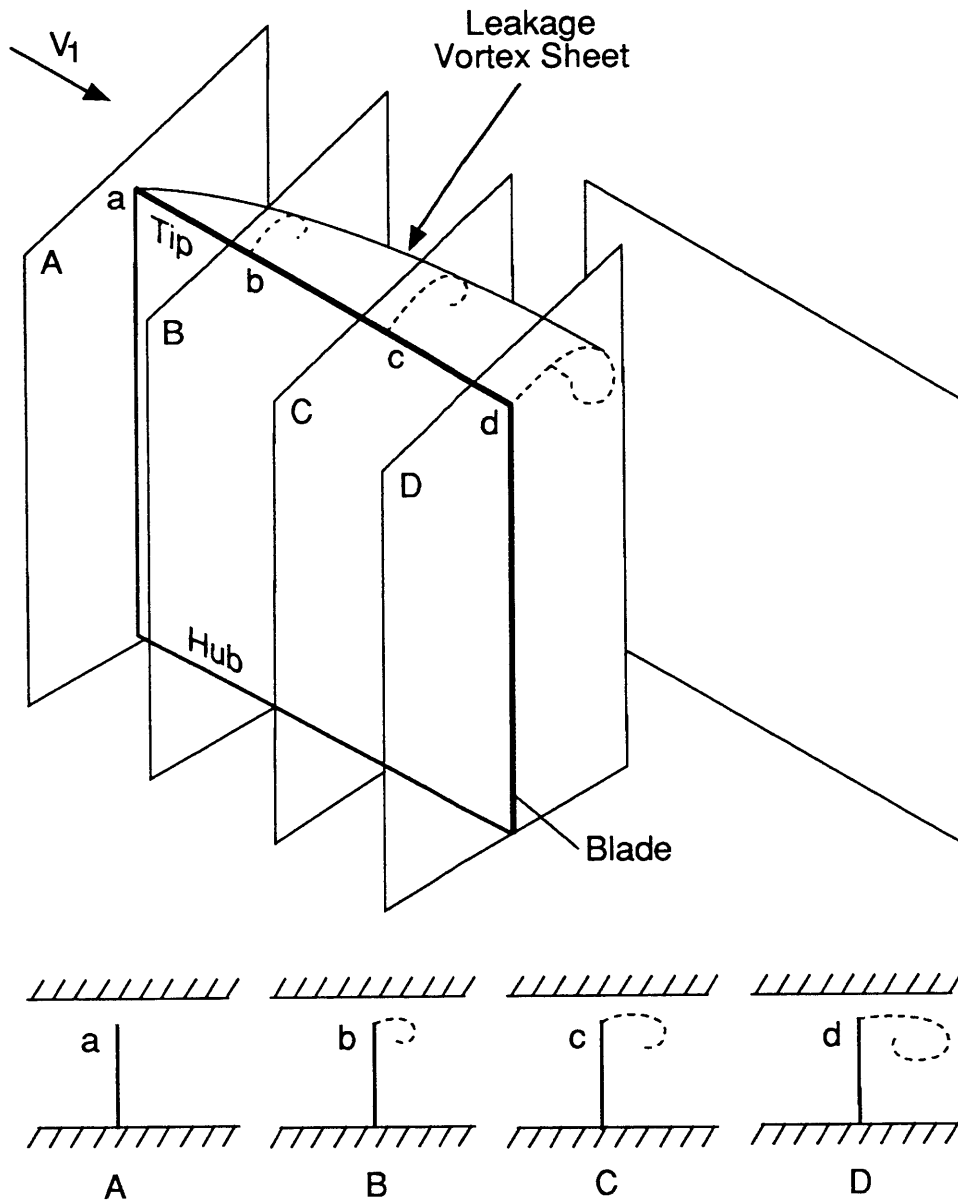
Fig. 11: Correspondence between 2-D steady tip clearance flow
and unsteady 2-D model flow.
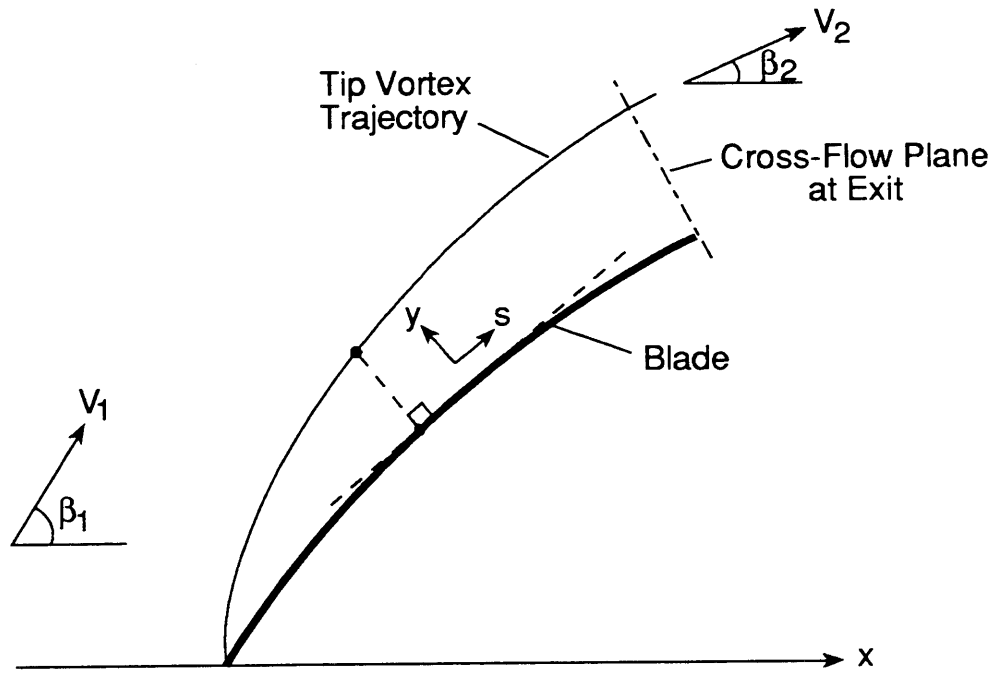
Fig. 12: Schematic of projection of vortex trajectory on constant radius surface.
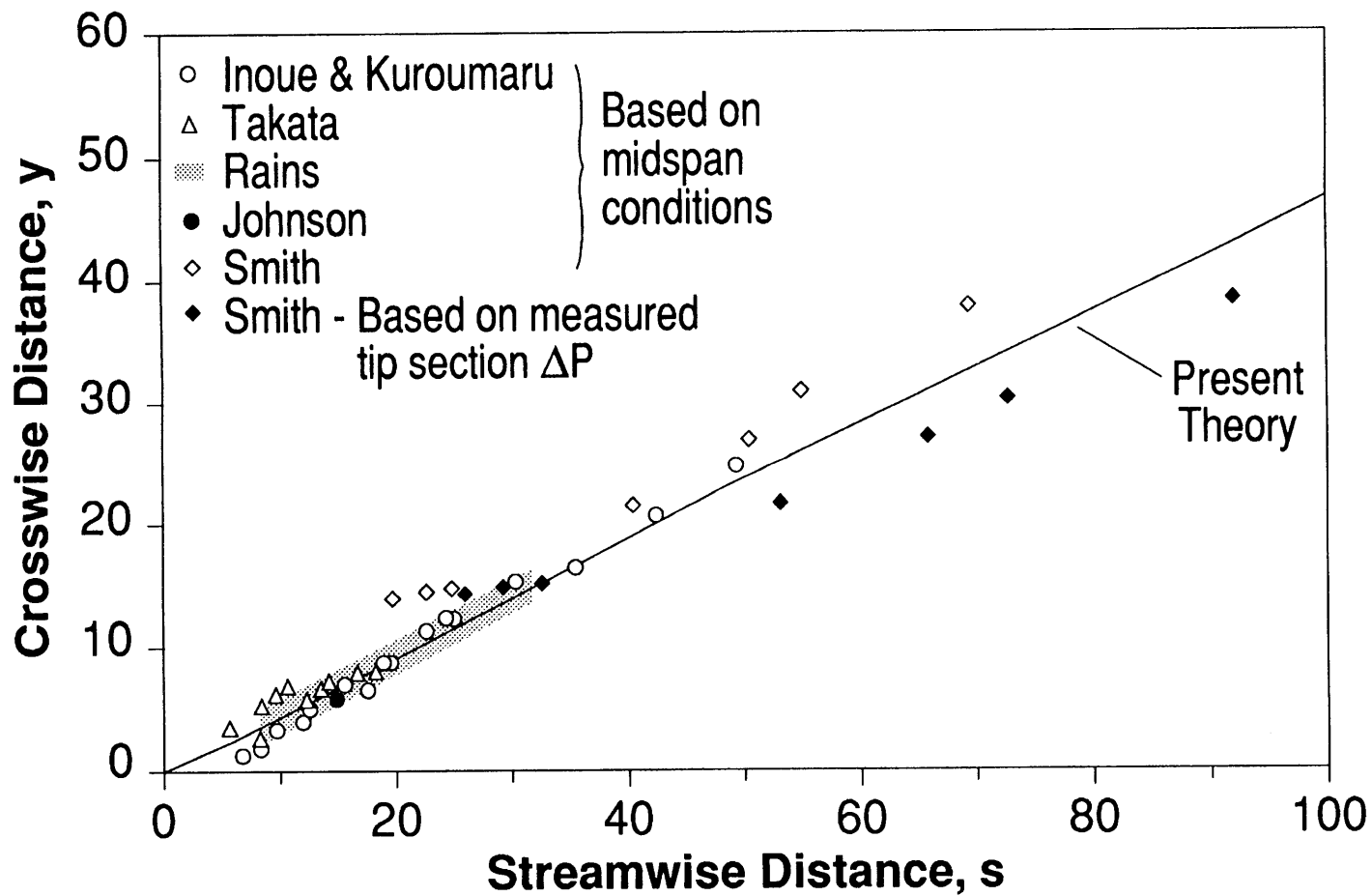
Fig. 13: Generalized tip clearance vortex core trajectory. Non-dimensional distance projected on constant radius surface.

(a)

(b)

Outline of vortex
predicted by
N-S calculation

Vortex predicted
by simple model
of Chen et al. (1991)

y_c

Camberline of blade
profile at hub

x
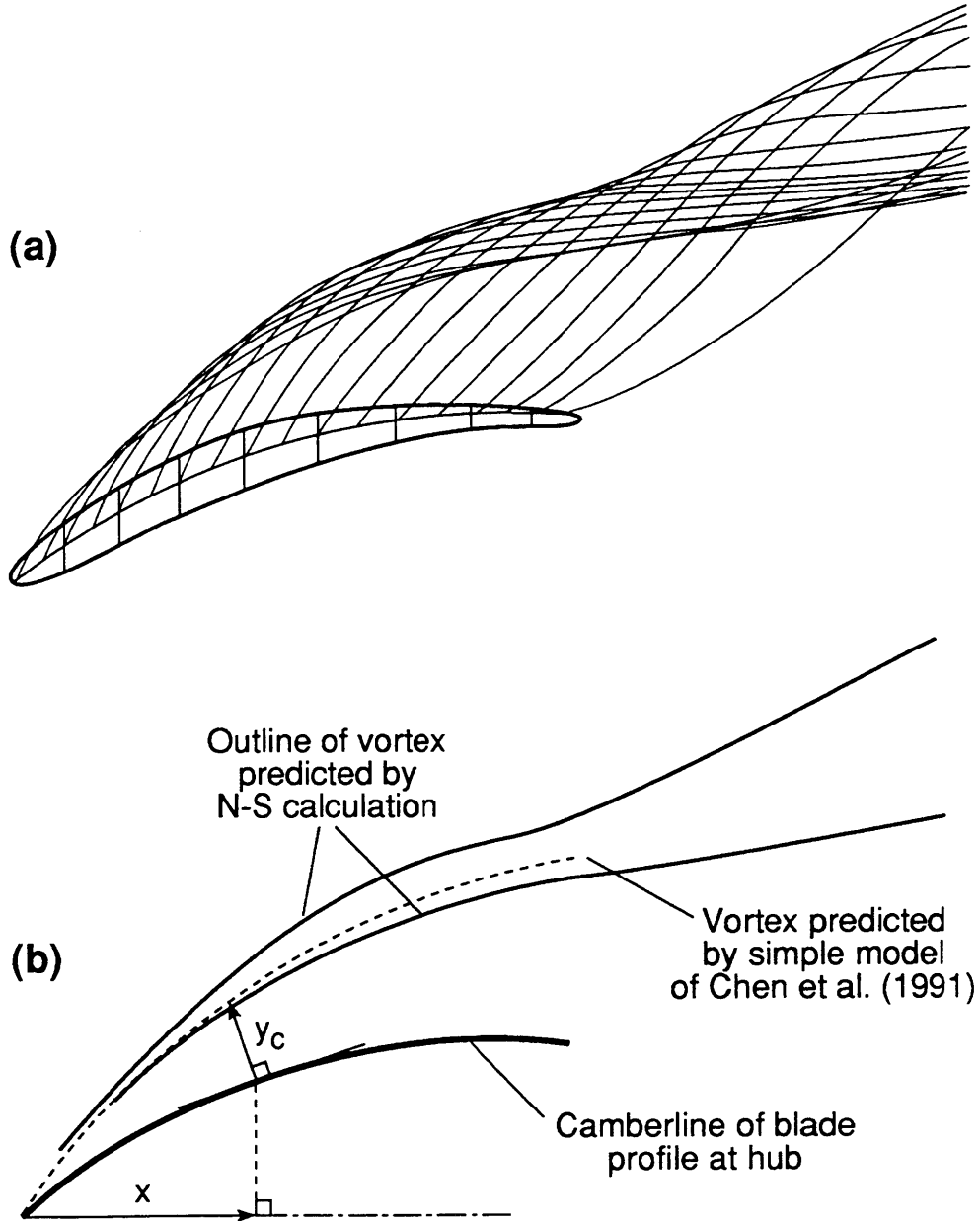
Fig. 14: Tip clearance flow: comparison of Navier-Stokes calculation
with simple model of Chen et al. (1991).

0.5
0.4
z/c 0.3
0.2
0.1
0.0

Measured

Zero Clearance

Clearance 2% of Chord

0.5
0.4
z/c 0.3
0.2
0.1
0.0

N-S
Predicted
(Dawes code)

(Contour intervals 5% inlet dynamic pressure)
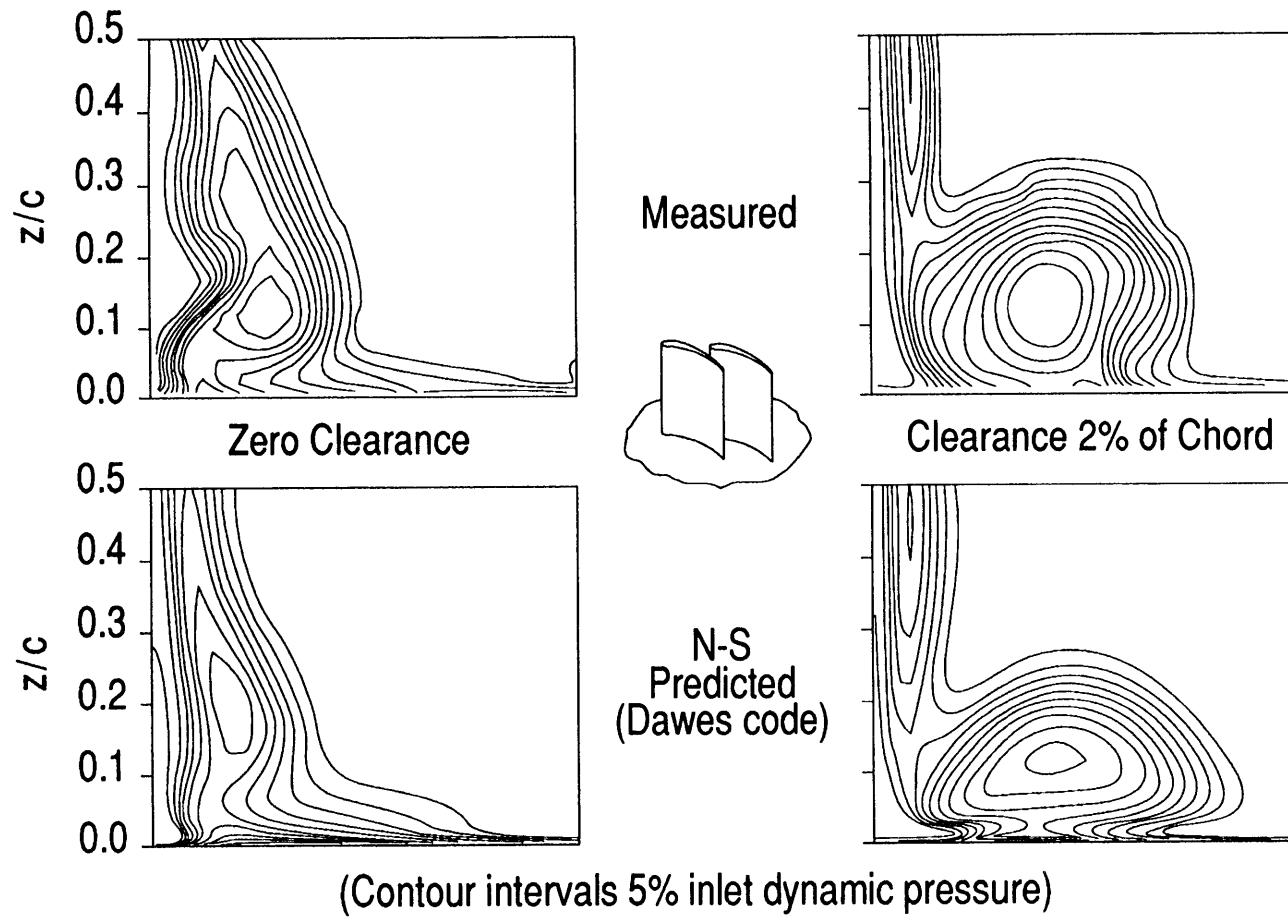
Fig. 15: Contours of loss 50% chord downstream of cascade (Storer, 1991).
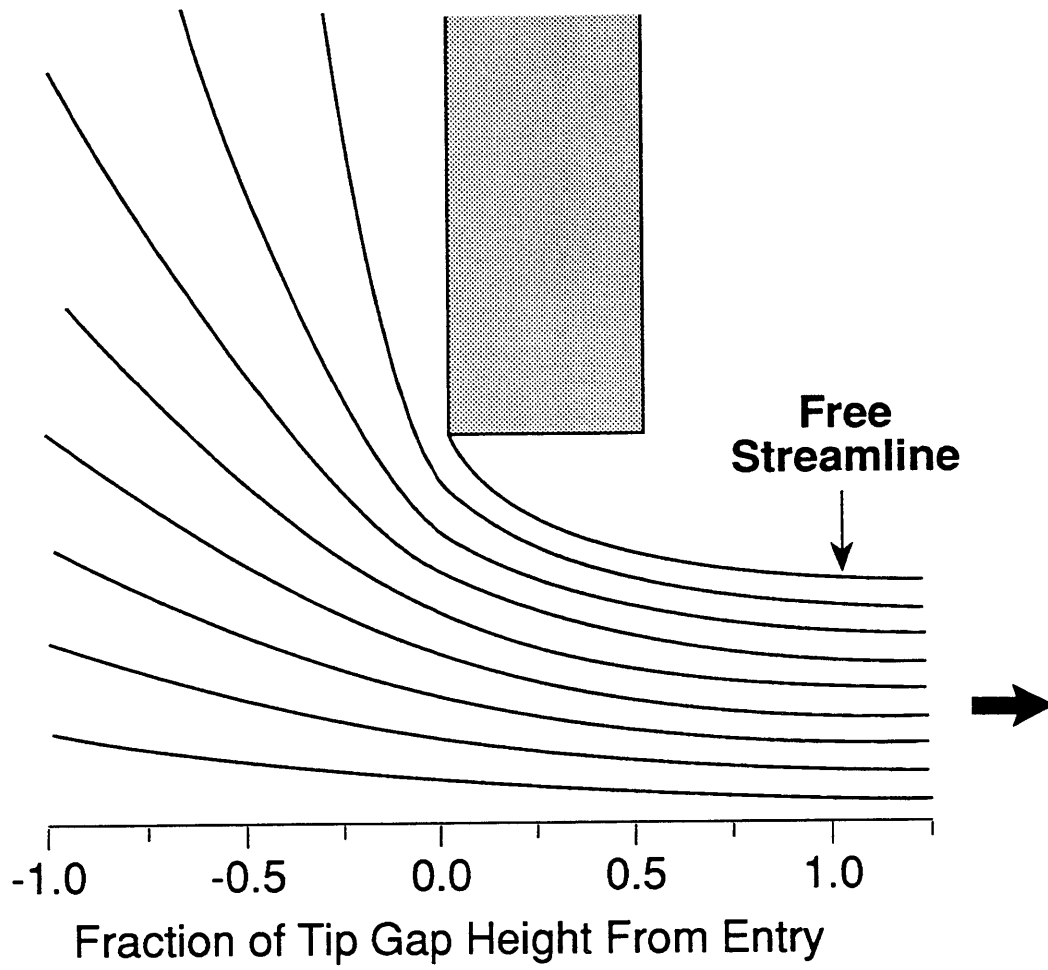
Fig. 16: Streamline pattern for 2-D inviscid flow through a square-edged orifice.
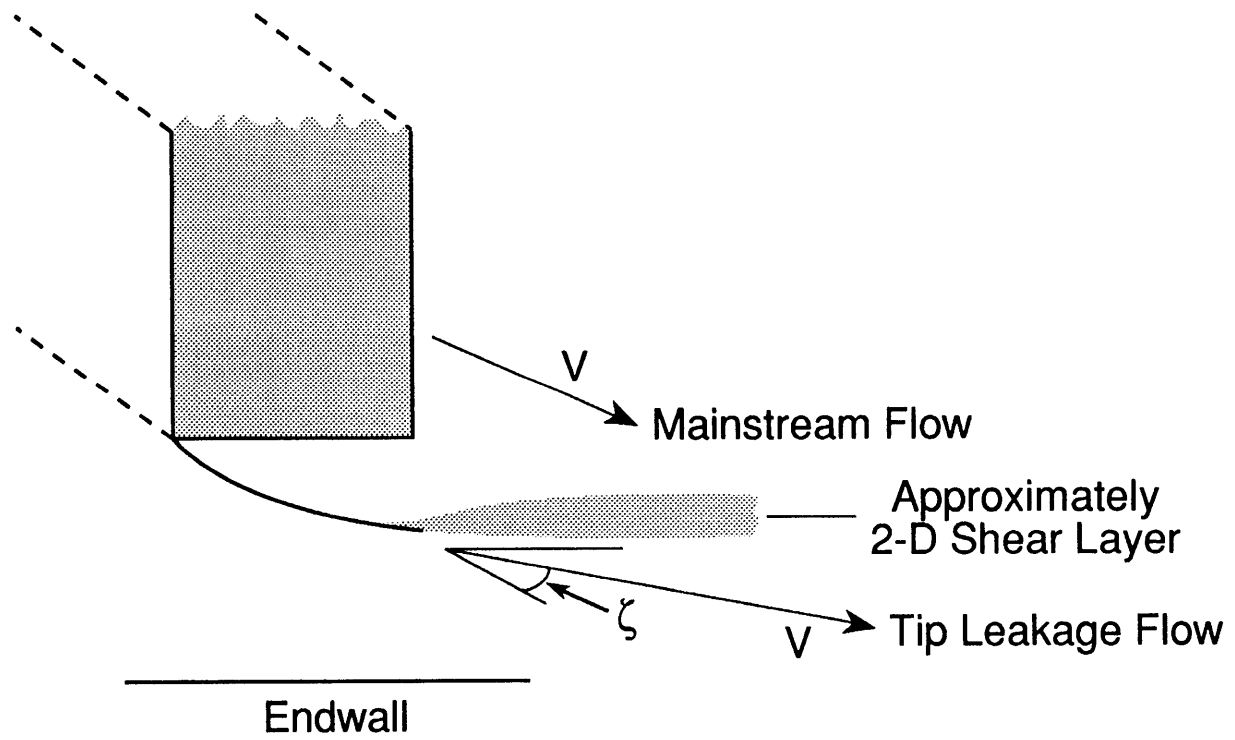
Fig. 17: Shear layer forming outside tip gap for simple model.
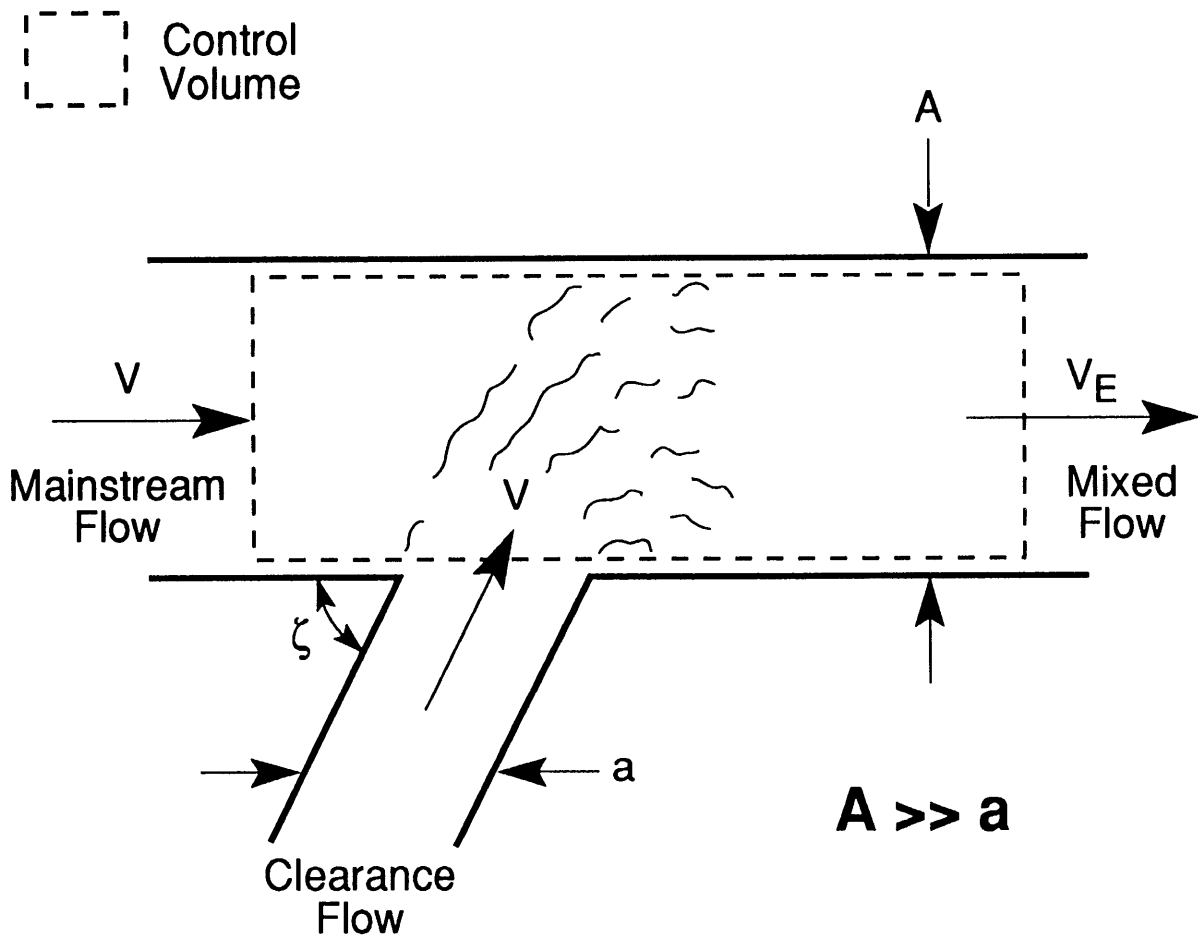
Fig. 18: Simple model for mixing of leakage jet with mainstream flow.
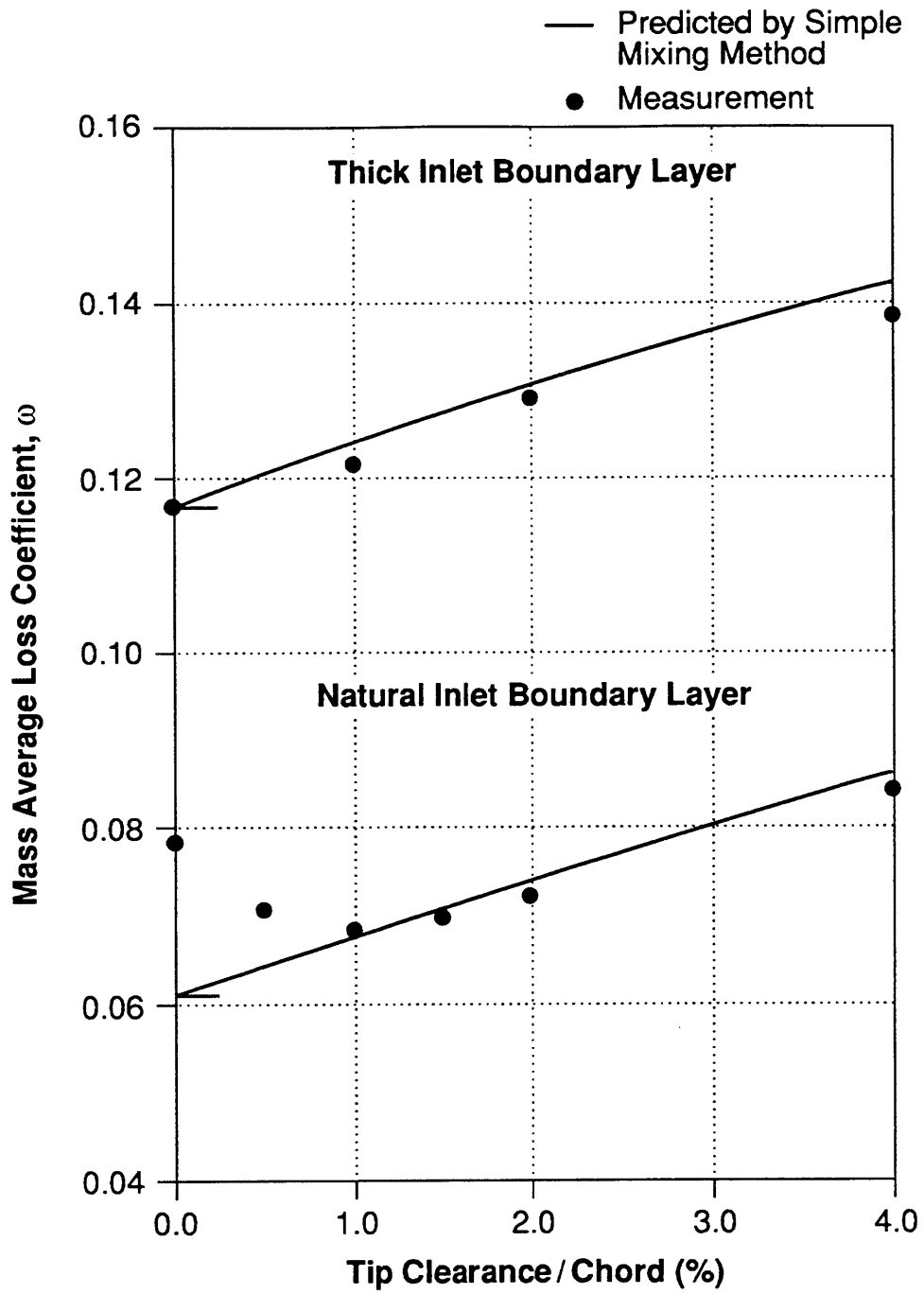
Fig. 19: Tip clearance loss 50% chord downstream of cascade.
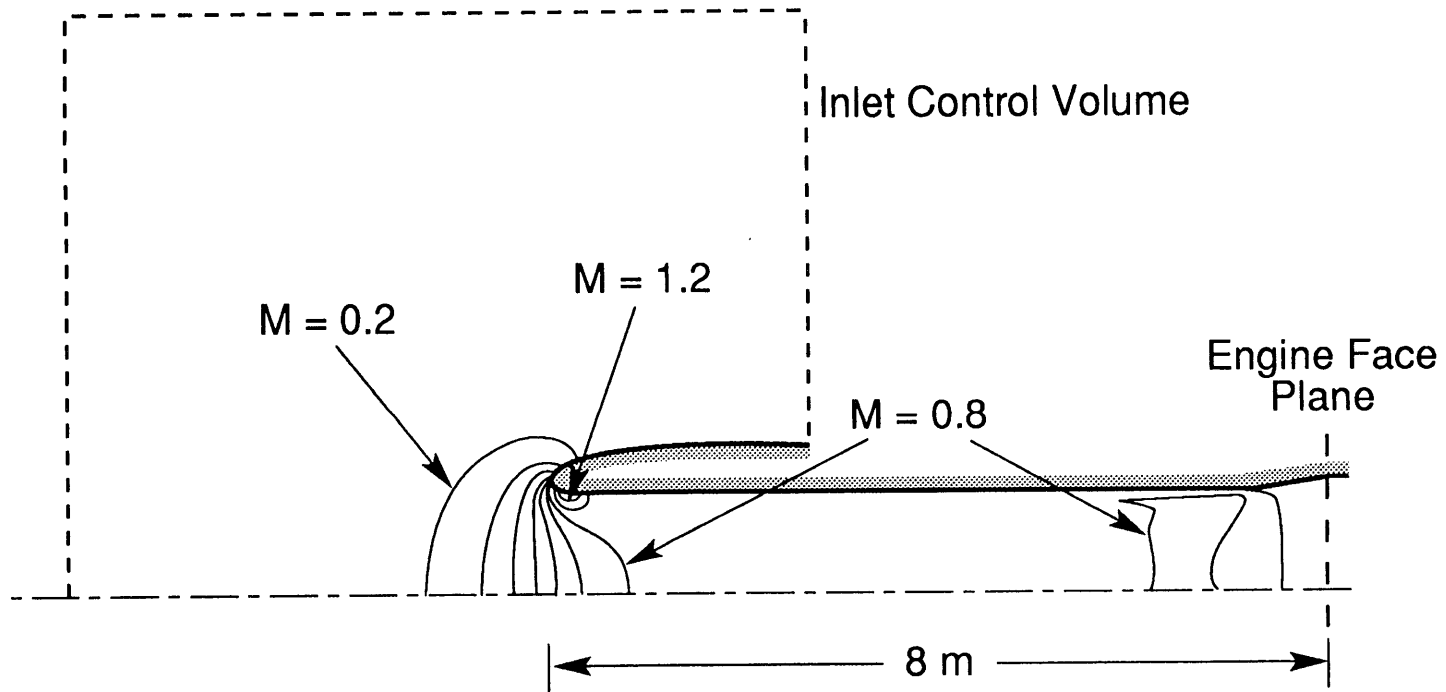
Fig. 20: Contours of Mach number in a long intake during sea level static tests, calculated for dry air.
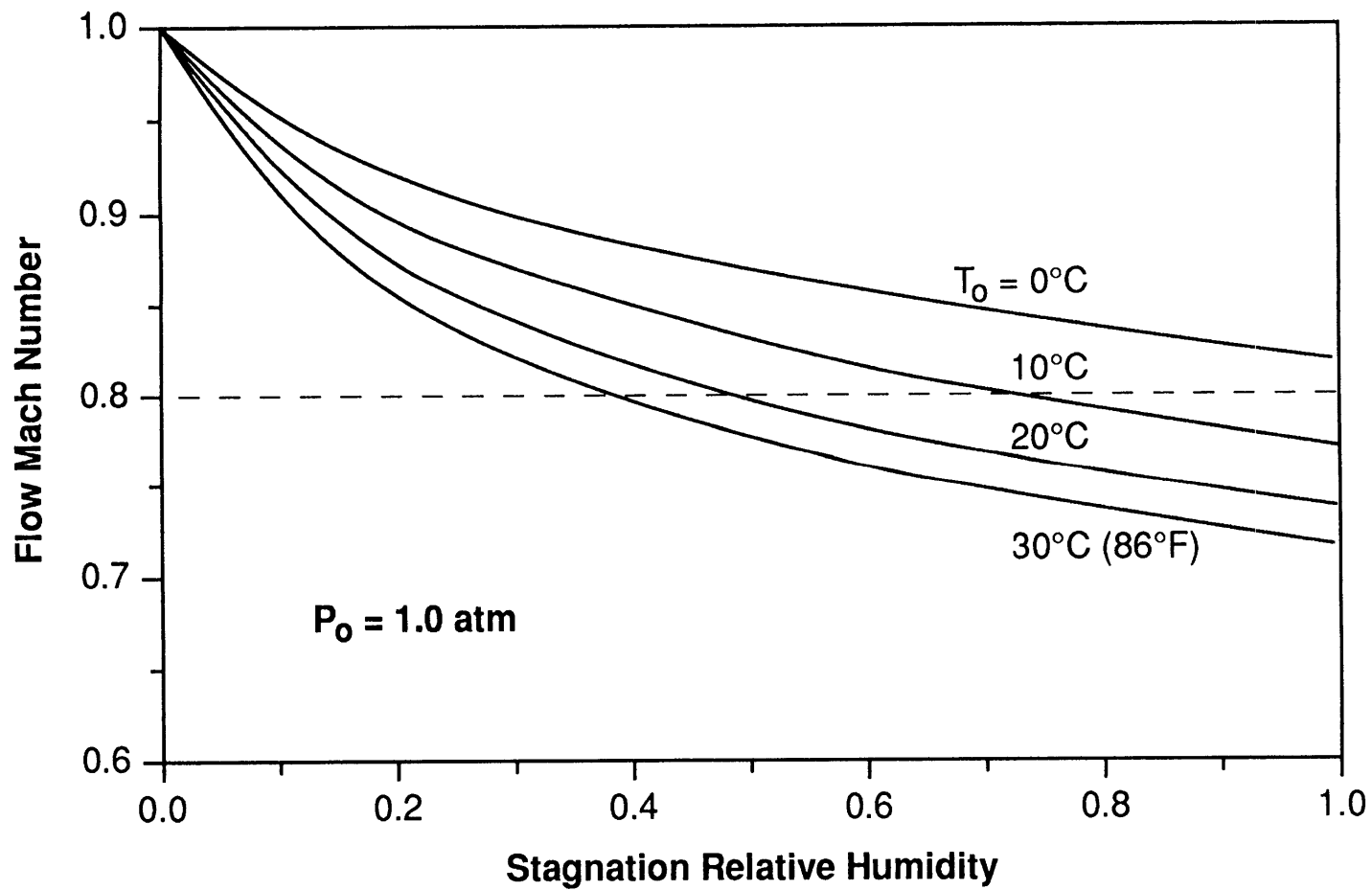
Fig. 21: Upstream Mach numbers required to produce thermal choking if water vapour condensation takes place to a new equilibrium condition.
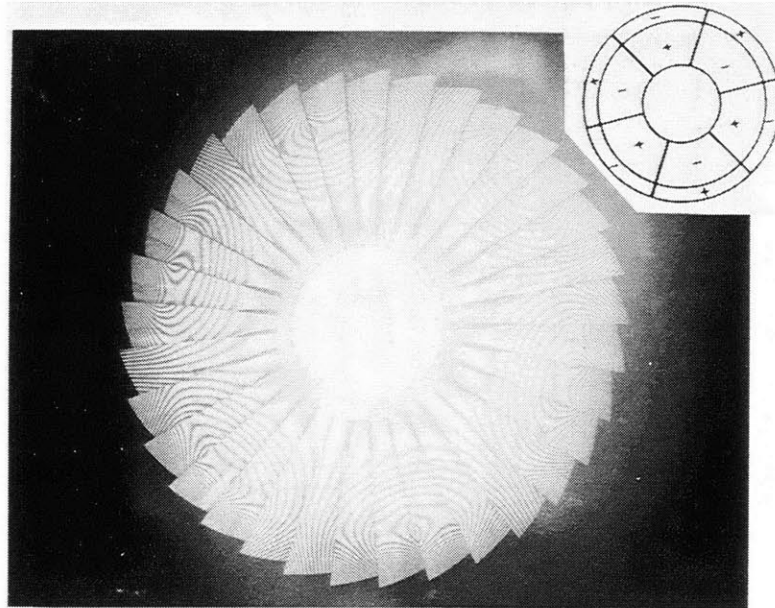
Fig. 22: A stationary fan rotor with part-span shrouds excited with three diametral modes. Visualisation obtained by holographic interferometry. (Cartoon included for clarification of relative phasing.)
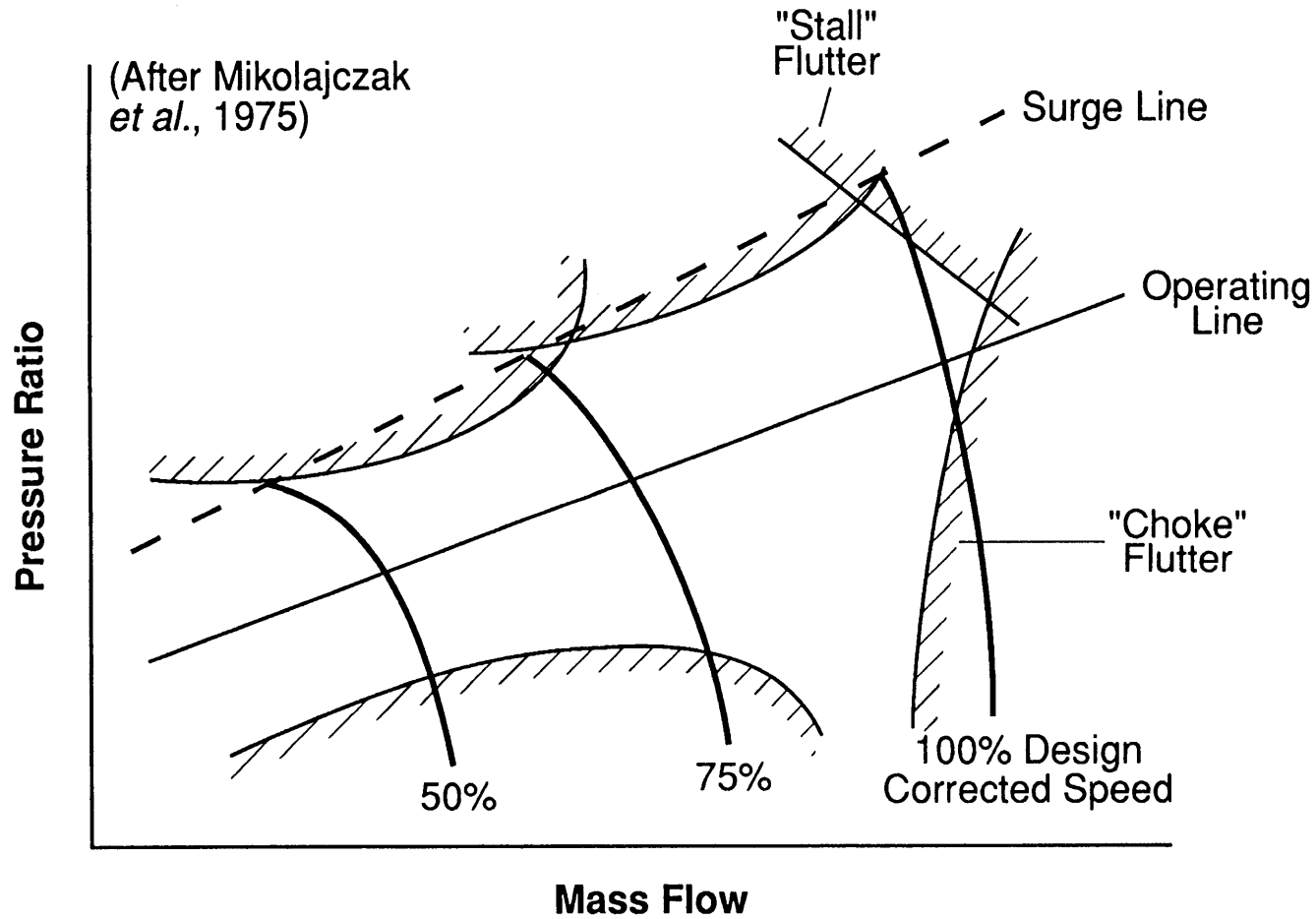
Fig. 23: Schematic operating map of transonic compressor showing various regions where flutter has been experienced or reported.
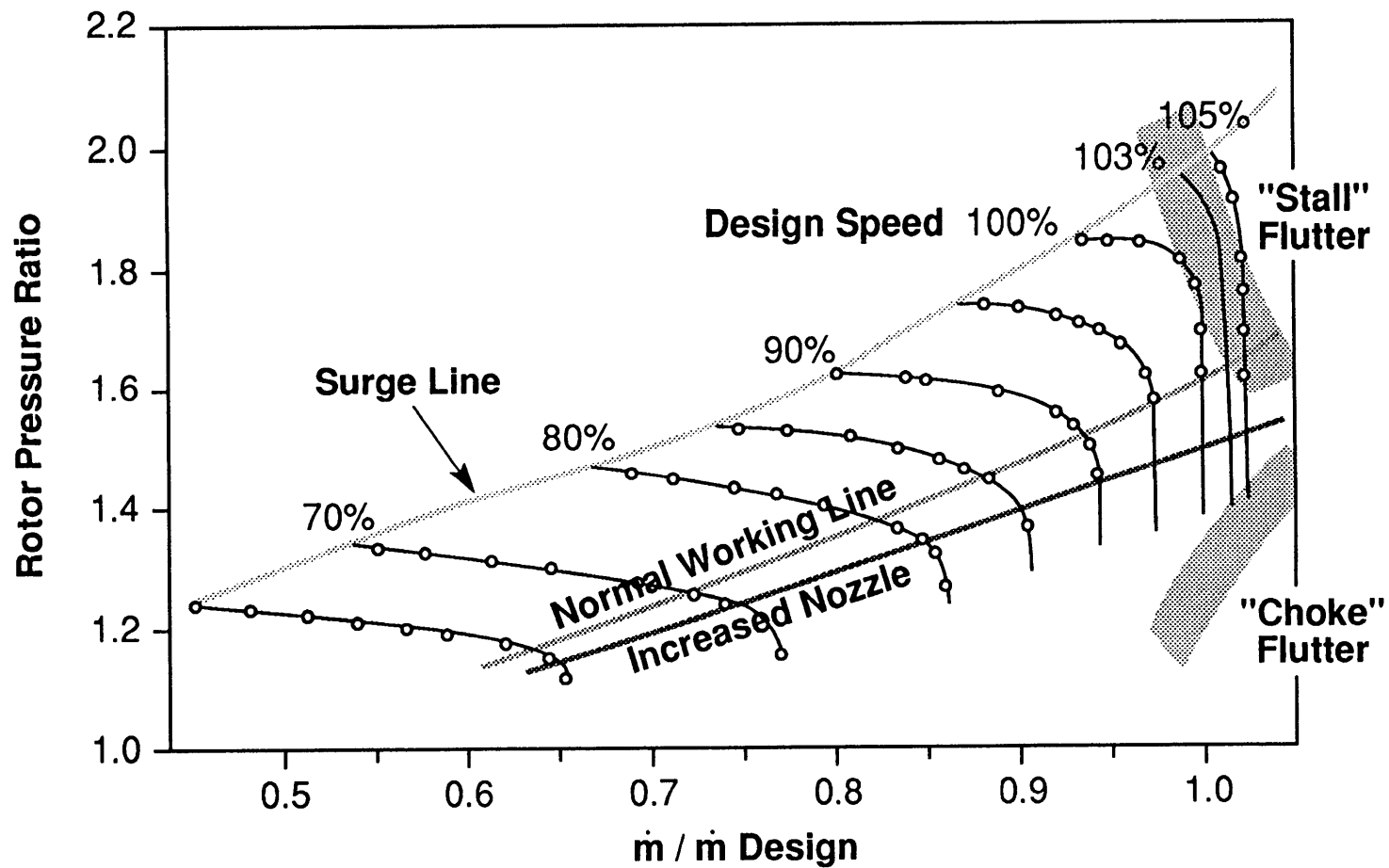
Fig. 24: Operating map for a modern civil aircraft fan. Working lines and flutter regions drawn in by hand for illustrative purposes.

Blade To Be Studied

These Blades Exert
a Strong Influence
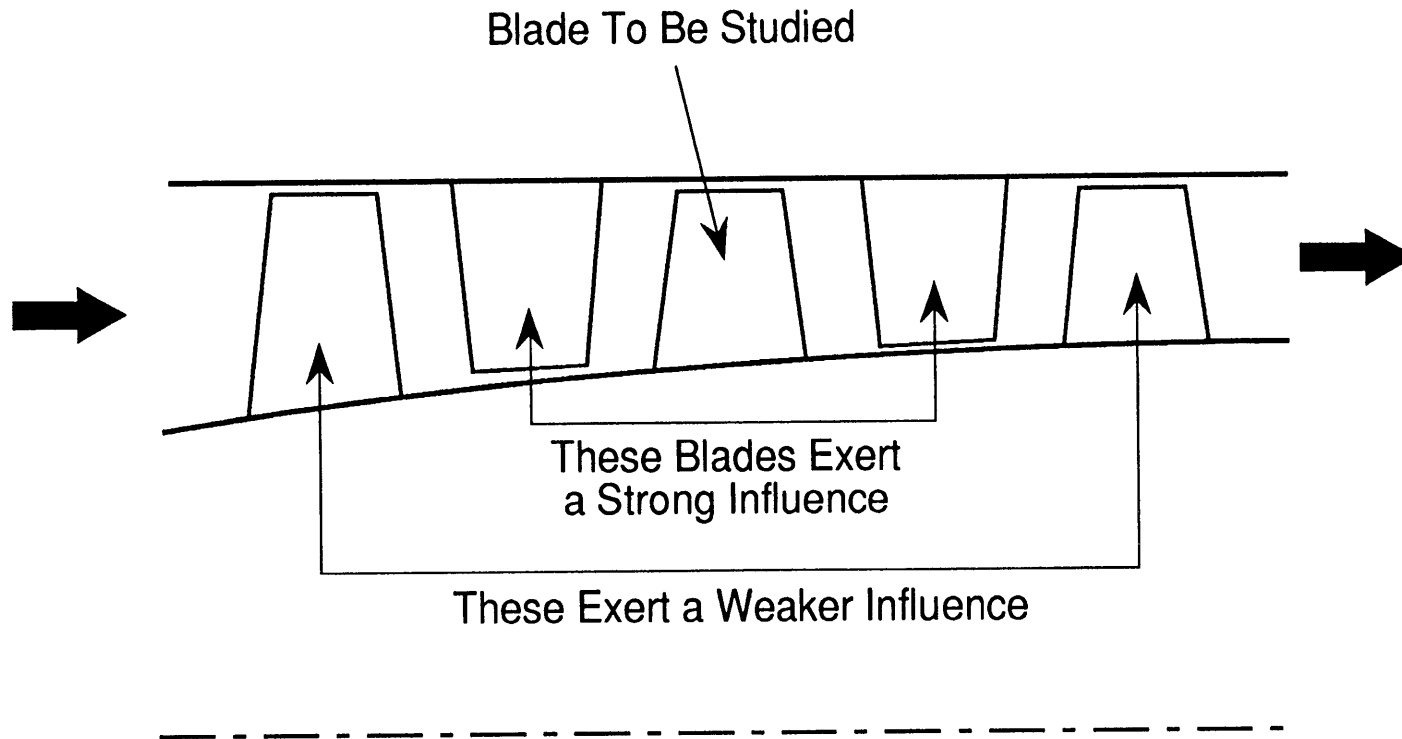
These Exert a Weaker Influence

Fig. 25: Schematic cross-section through a multistage compressor indicating the interaction between the rows.
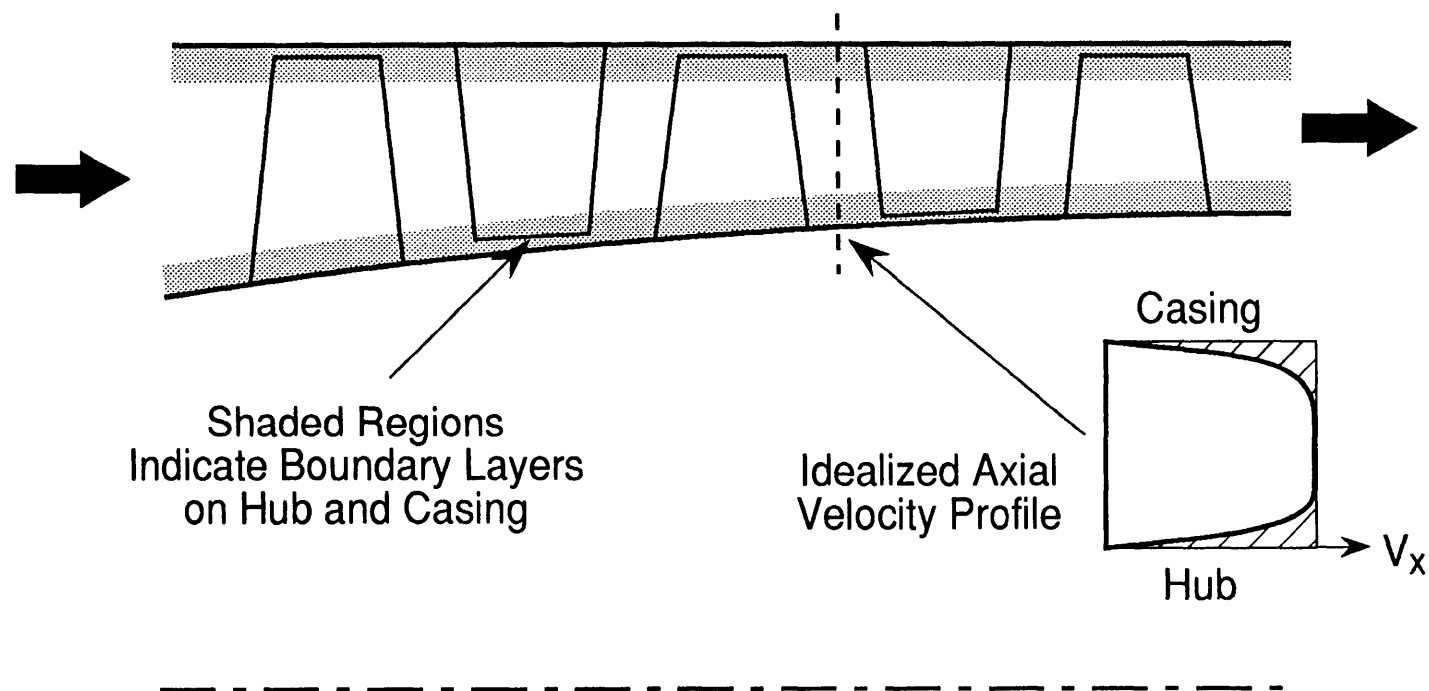
Fig. 26: Schematic cross-section through a multistage compressor indicating endwall regions, so-called endwall boundary layers.

APPENDIX

Figure A1 shows some highlights of the development of jet aviation during the 20th century. The very rapid development in the period between 1930 and 1950, when work was active in both Britain and Germany, is to be contrasted with the slow evolution between 1970 and 1990. The German work came to an abrupt stop with the end of the war, whereas the British work led directly to the ill-fated Comet airliner. The "jet age" really began with the introduction of the Boeing 707, when passenger-carrying planes could first cross the Atlantic non-stop. The first Minta Martin lecture, given by Professor Hawthorne, was before this.

It is notable in Fig. A1 that the United States played no part in the early development of the jet engine, in contrast to the crucial part it played in almost every other aspect of air travel. One reason for this may be found in a report from a committee of the National Academy of Science to the Secretary of the Navy, for which an extract from the conclusions is given below.

"In its present state, and even considering the improvements possible.... the gas turbine could hardly be considered a feasible application to airplanes..."

Max Mason (Chairman)	*A.G. Christie*	
Theodore von Karman	*C.F. Kettering*	
L.S. Marks	*R.A. Millikan*	10th June 1940

Notwithstanding the conclusions of this committee of most distinguished men, the first flight of the von Ohain plane had already taken place by then and the Whittle engine intended for flight was under construction.
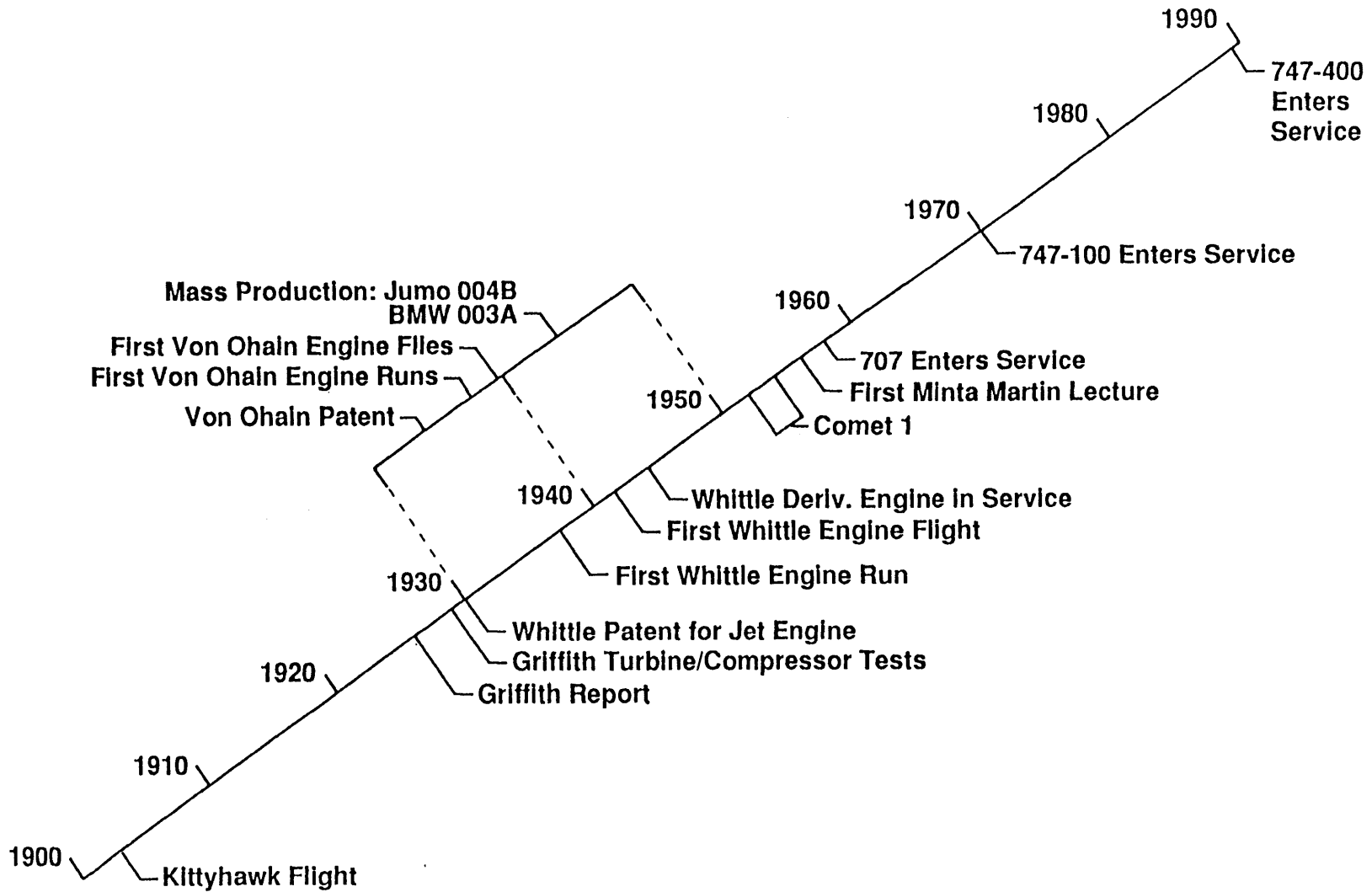
Fig. A1: A brief history of the 20th century.