Aerodynamic Anomalies: Can CFD Prevent or Correct Them?

Richard S. Shevell*
Stanford University, Stanford, California

Computational fluid dynamics (CFD) has become a standard design tool for modern sophisticated aircraft. This paper reviews design methods that were used for some existing jet transport aircraft at a time when theoretical tools had very limited capability. Early designers combined available theory and empirical data with sufficient skill to develop highly successful aircraft. They also overcome serious problems that sometimes arose only after first flight, problems due to aerodynamic phenomena that were not predicted. Some of these problems are discussed in detail in this paper. The questions of whether modern CFD would have warned of these problems and provided necessary solutions is addressed. It is concluded that, in many cases, the answer is negative. Comparisons between two-dimensional CFD airfoil compressibility drag rise, converted to finite wing drag rise using simple sweep theory, and flight test results show good agreement. A rapid method of compressibility drag rise prediction based on two-dimensional CFD results is suggested. The method will define the drag that can be expected if wing designers do a proper job, thereby allowing early performance prediction. Actual wing design is a task for three-dimensional wing and body CFD code including viscous and shock wave effects.

Introduction

THE ability to analyze theoretically the flow over an airplane and its components and to calculate the resultant forces and moments has been the goal of aerodynamicists since the beginning of manned flight. Solutions to the general problems of determining lift and induced drag were successfully found in the first decades of this century, but the solutions were limited to incompressible, inviscid fluids. The problem of determining turbulent skin friction drag was solved in form by theory, but the numerical values could be found only by combining empirical results with the theory. Modest empirical modifications to the elegant theories for lift and drag due to lift were also required. Nevertheless, analytical methods plus wind tunnel studies allowed many airplanes to be developed and to meet predicted performance with acceptable accuracy.

Since the 1950s it has been correct to say that the experienced aerodynamicist could predict the drag and lift of a high-subsonic-speed transport airplane with analytical tools over almost all of the possible speed and angle-of-attack conditions. Only at the highest angles of attack near stall and at the highest speeds would the predictions be of doubtful value and wind tunnel tests urgently required. Unfortunately, for most aircraft designs, these limited regions were the most important. Takeoff and landing field lengths depended on stall speeds and cruising speed and efficiency depended on drag in the compressible flow speed regime after shocks began to appear. When flow separation was involved (as at the stall) or shock waves were present on the wing surface, the theories broke down.

The development of computational fluid dynamics (CFD) and the high-speed, large-capacity computers it requires changed that. Theoretical calculations of good accuracy can now be made including the effects of boundary layers and shock waves.[1-3] CFD produces more detailed aerodynamic information faster than previously possible, particularly on simple configurations such as an airfoil section, a fuselage, or even a complete wing. However, modeling complex complete configurations such as a swept wing/nacelle/pylon combination is much more difficult and, when vortex interactions and spanwise boundary flows are involved, a practical and economic CFD solution is not yet available. Many of the performance and flying quality problems that plagued aerodynamicists in the pre-CFD period are of this complex character.

To gain perspective on this issue, it is useful to study some of the aerodynamic difficulties that have afflicted high-subsonic-speed transport aircraft—difficulties that analysis did not and could not predict. The physical nature of the problems and their resolution is discussed. The question of whether these problems would have been avoided or their solution readily found if current CFD capability had been available during the last 30 years is explored.

It should be noted that "CFD" as used in this paper means the latest computational methodology available for practical design application today and in the near future.

The DC-8 Experience

A Bit of History

Development efforts toward high-speed jet transports began in the late 1940s. Basic aerodynamic lift, drag, and pitching moment data for the initial studies were obtained primarily from NACA tests of swept-wing models. Two-dimensional sweep theory was studied and generally formed the analytical framework upon which the test results were draped to arrive at useful analytical prediction methods. Although some engineers liked to stay close to a configuration for which specific test data existed, others tried to develop general methods that fit the range of test points, e.g., different wing sweepback angles, airfoil thickness ratios, and Mach numbers.

Early studies showed that an economical jet transport of adequate range was not possible with engines available in the late 1940s.[4] Engine fuel efficiency and thrust-to-weight ratios just did not permit ranges greater than New York to Chicago and the operating costs were well above those of the reciprocating engine powered aircraft of that time. Nevertheless, these studies pointed out many fundamentals of jet airplane performance that are still basic to modern operations.

Presented as Paper 85-4067 at the AIAA Applied Aerodynamics Conference, Colorado Springs, CO, Oct. 14-16, 1985; received Oct. 26, 1985; revision received March 6, 1986.

*Professor of Aeronautics. Fellow AIAA.

The development of the Pratt & Whitney JT3 (military designation, J-57) twin-spool turbojet engine was the needed breakthrough in efficiency to permit the development of economical jet transports. For the first time, the specific fuel consumption was reduced below 1.0. Boeing plunged into a prototype development, while Douglas activities were limited to analysis and wind tunnel testing.

Airfoil Studies

At Douglas we started an empirical study of the drag divergence Mach numbers, M_{div}, and the section maximum lift coefficients of every airfoil family for which we could find data. M_{div} is the Mach number at which the parasite drag starts to rise abruptly. It also has an analytical definition such as the Mach number at which $dC_D/dM=0.035$, 0.05, 0.1, or some other value selected by the researcher. Of course, a consistent definition must be used for all comparisons. The data sources were primarily NACA wind tunnel tests.

The study showed that NACA modified four-digit airfoils, with designations such as 0010-1.10-40/1.051, had a significantly higher M_{div} than the generally accepted NACA six-series airfoils. These data were found in an obscure NACA report primarily featuring hinge moment information. Drag data just happpened to be included. We visited the NACA Ames Research Center, where the tests had been done, and showed these results to some skeptical engineers. NACA was sufficiently interested to run some direct comparisons between six-series and modified four-digit airfoils. The results of these tests are summarized in Fig. 1 and verified the superiority of the modified four-digit series.[5]

An excellent aerodynamicist, the late Harold Kleckner, was assigned to determine what the pressure distribution characteristics of these airfoils were and why a higher M_{div} resulted. Kleckner first established that these airfoils had "peaky" pressure distributions and carried a lot of lift on the forward portion. For a given total airfoil lift, this permitted the local velocity at the "crest" of the airfoil to be lower. The crest of an airfoil is the point on the upper surface to which the freestream is tangent; see Fig. 2. It is, therefore, the point that separates the forward-facing airfoil surface from the aft-facing portion. It had been shown earlier[6] that M_{div} occurs at a Mach number a few percent above the crest critical Mach number, i.e., the Mach number at which the speed of sound first occurs, perpendicular to the isobars, at or behind the crest. This gave a method for predicting M_{div} and Kleckner found that supersonic flow ahead of the crest caused very little drag provided the flow, as calculated with incompressible methods corrected for compressibility by the Karman-Tsien approximation, decelerated to a subsonic speed prior to reaching the crest. He then sought further improvements in the modified four-digit airfoils to maximize the gain due to peaky airfoils. Kleckner used an empirically established maximum peak negative pressure coefficient that could be realized in practice, probably from Ref. 7. The limit was based on obtaining a pressure coefficient corresponding to attaining 70% of a vacuum, i.e., $C_{p\max}=-1/M^2$ (see also Ref. 8). Kleckner then designed airfoils to achieve this maximum usable peakiness and to maximize the maximum lift coefficient, under airfoil labels such as DSMA 87 (Fig. 3). The name stood for Douglas Santa Monica airfoil.

In the early 1950s there were no methods for calculating three-dimensional compressible pressure distributions for swept wings. Kleckner developed empirical ratios relating the pressure coefficient for a three-dimensional swept-back wing at a Mach number, near the drag divergence Mach number $(C_P)_{M,3}$, to the calculated pressure coefficient for the corresponding two-dimensional airfoil in incompressible flow, $(C_p)_{M=0,2}$. NACA tunnel tests of swept wings were the data sources. The pressure coefficient ratios were plotted vs the chordwise position for various spanwise stations and were determined for several available wing sweep angles. This procedure accounted for effects of Mach number, root and tip interference, and sweep.

There were three 3.5% scale complete wind tunnel models, as opposed to the half-span, built before the DC-8 design was frozen. The first two had NACA modified four-digit airfoils and the third and final model had specially designed DSMA airfoils.

High-Speed Results

Original wind tunnel tests used for performance predictions were run at a Reynolds number of about 1.8×10^6 with a 3.5% scale model in the Caltech Cooperative Wind Tunnel. The results showed an excellent drag rise curve with the high anticipated M_{div} and a very gradual drag rise prior to M_{div}; see Fig. 4. A 7% scale semi-span model with a Reynolds number of 6.5×10^6 was run long after the design was frozen. This test showed the same M_{div}, but also a serious excessive drag creep prior to the steep rise at M_{div}. We spent the ensuing months before the first DC-8 flight writing each other memos on the unreliability of half-span tests. Unfortunately, this high-technology self-delusion did not work and the flight tests confirmed the excessive drag

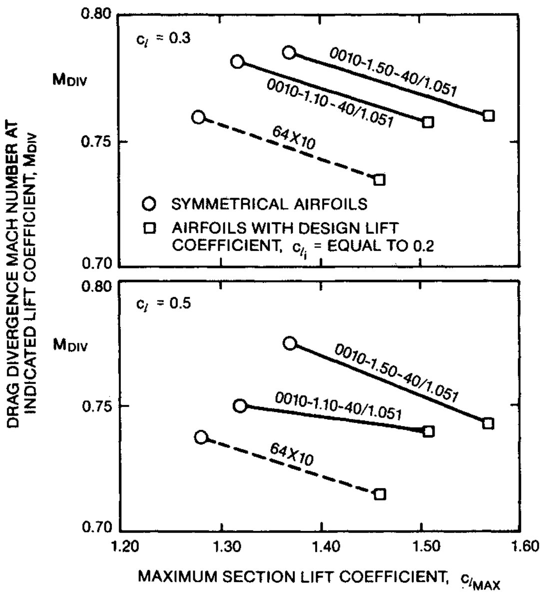

Fig. 1 Drag divergence Mach number and maximum lift comparison.

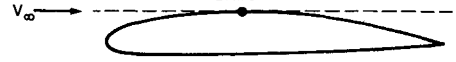

Fig. 2 Definition of the airfoil crest.

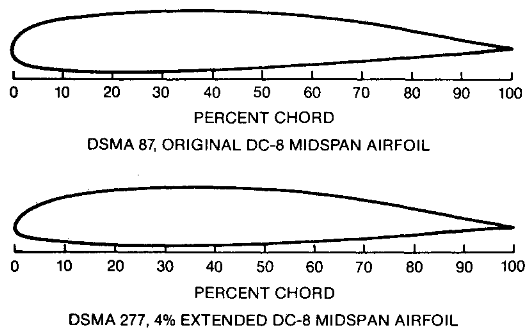

Fig. 3 Contours of original and 4% extended DC-8 airfoil.

rise of the half-span model. Later, full-span model tests with a 3.4% model in the Ames Research Center 11 ft Pressurized Transonic Tunnel, at a Reynolds number of over 6×10^6, also showed the excessive drag rise.

We concluded that for some airfoils, and certainly with our peaky ones, the Reynolds number was important: exceed 6×10^6 Reynolds number or run a risk of getting the wrong drag rise answer. The problem was also attributed to an excessive nose peak. Therefore, a modified airfoil was designed to lower the nose peak pressure in the cruise condition. The modification was limited to the leading edge ahead of the front spar and increased the chord by 4%; see Fig. 3. This change eliminated about 70% of the problem, although a modest jump in the drag still occurred well below M_{div}; see Fig. 4. Production was changed to the 4% extended leading edge and most of the early DC-8 aircraft were retrofitted with it.

During this period of agonizing reappraisal, we became aware of the theoretical airfoil work of Pearcey[9] in England. From Pearcey's studies we decided that our real problem was that the incompressible airfoil pressure distribution did not have a sufficiently sharp nose peak and that the slope of the pressure coefficient curve with distance along the airfoil had to become quite flat forward of the crest. This, in our opinion, was necessary to avoid excessive drag creep prior to M_{div}. Our ability to attain this on the modified DC-8 airfoil was limited by the existing structure, which could be changed only forward of the front spar, and the need to maintain the same stall speeds as originally achieved. Later, when the airfoil was further modified for the DC-9-30 series and subsequent versions of the DC-9 and fitted with a leading-edge extension of about 6% equipped with a leading-edge slat to handle the more stringent $C_{L\max}$ requirements, a sharper peak was achieved. We shall see, however, that the modified DC-8 airfoil was really quite good in itself. Much of the residual drag creep problem later turned out to be due to pylon interference.

Could CFD techniques currently available have warned of the excessive drag creep on the original DC-8 configuration? The answer is "probably not." Figure 5 compares recently computed two-dimensional CFD compressibility drag rise characteristics of the defining "midspan" airfoils for the original DC-8, the 4% extended DC-8, the DC-9-30, and the DC-10 airfoils from Ref. 10. The method used is the Bauer-Garabedian-Korn code (program H). Data are compared at a section lift coefficient, c_l, of 0.55 that corresponds, for airplanes with 25-35 deg of sweepback, approximately to an airplane lift coefficient, C_L, of 0.4, representative of the cruise condition. Simple sweep theory states that the section c_l, perpendicular to the quarter chord sweep line, equals $C_L/\cos^2$. The Mach number shown in Fig. 5 is the component of Mach number in the direction perpendicular to the quarter-chord sweep line and, therefore, will be equal to the freestream Mach number reduced by the cosine of the sweep angle. The curves do show improvement in drag rise with each step in airfoil development.

In examining Fig. 5, note that the DC-8 (4% extended) and the DC-10-10 airfoils have essentially the same thickness ratios. The DC-8 (original) has a thickness ratio about 0.005 greater, while the DC-9-30 is about 0.010 greater. To compare the *quality* of the airfoils at approximately equal thickness ratios, the DC-8 (original) curve may be shifted to a higher Mach number by $\Delta M = 0.006$ while the DC-9-30 curve is corrected by $\Delta M = +0.011$.

Figure 6 shows that the original DC-8 airfoil drag rise was much worse in flight tests than predicted by CFD. Drag coefficients are shown for an airplane lift coefficient of 0.4. The Mach number for the flight test data has been reduced by cos Λ to give an equivalent unswept drag rise curve. In Fig. 6, the drag coefficients shown as predicted by two-dimensional CFD are the section drag rise values converted to equivalent airplane values by multiplying them by $\cos^3 \Lambda$

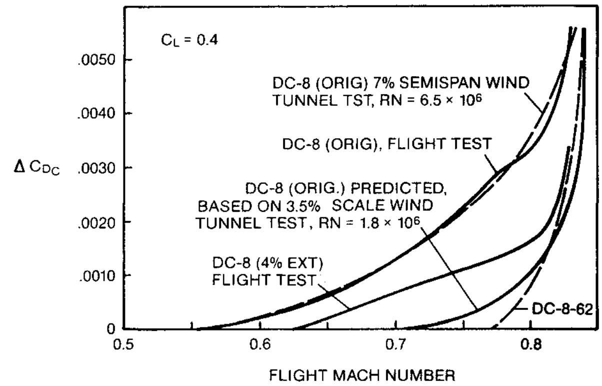

Fig. 4 DC-8 comparison of compressibility drag rise wind tunnel tests at low and high Reynolds numbers and flight test.

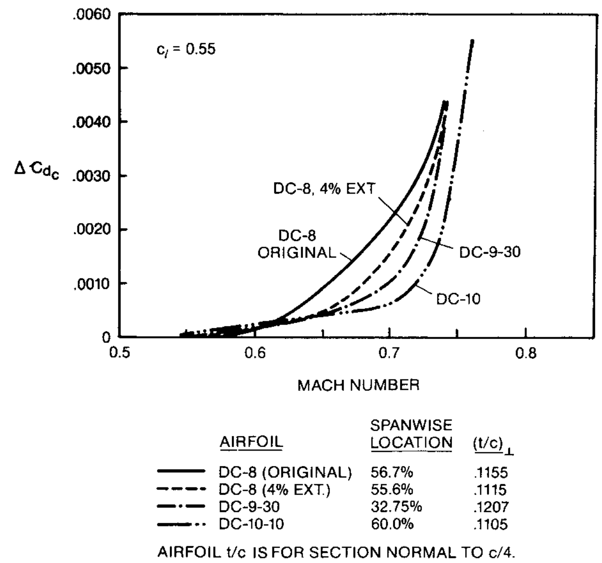

Fig. 5 1985 CFD two-dimensional airfoil drag rise for defining midspan airfoils.

in accordance with simple sweep theory. The Mach numbers have been reduced by 0.013 to correct from the defining airfoil thickness to the larger mean thicknesses of the wing, 11.2%, freestream, for the original DC-8 and 10.8% for the DC-8 (4% extended). A reduction in Mach number of 0.005 was used to account for the fuselage flow field. As a further refinement, the section lift coefficients were slightly adjusted to represent exactly the airplane C_L of 0.4 using the quarter-chord sweep angle of each configuration. With these corrections, the CFD predicted differences between the original DC-8 airfoil and the 4% extended section are only about three drag counts, less than 1.5% of total airplane drag. It is quite probable that the higher $C_{L\max}$ of the original airfoil would have made it the airfoil of choice anyway. The lower wing area required for takeoff and landing field length requirements would have saved more in weight and drag than the drag rise penalty would have cost. In any case, the DC-8 would not have had an airfoil change if the two-dimensional CFD prediction corrected by simple sweep theory, shown in Fig. 6, had prevailed.

We have been using two-dimensional CFD results for the defining midspan airfoils, applying simple sweep theory, and treating them as if they applied to a real three-dimensional wing. This is an approximation in many ways. First, second-order terms cause errors in simple sweep theory after the occurence of shock waves. It would be desirable to have three-dimensional CFD results for the actual wing with the

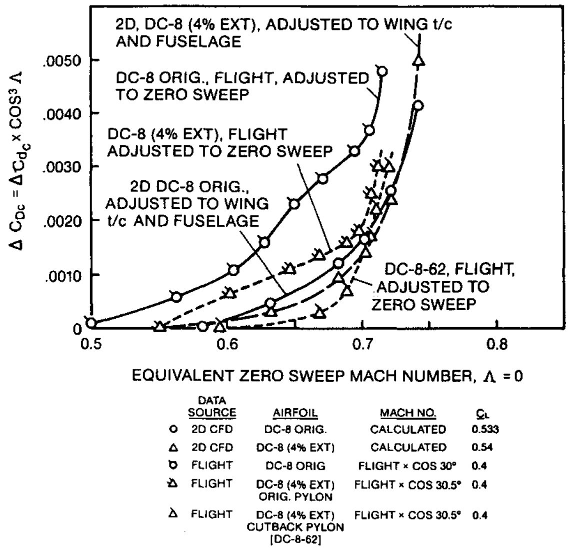

Fig. 6 Comparison of DC-8 flight test compressibility drag with drag rise predicted from two-dimensional CFD.

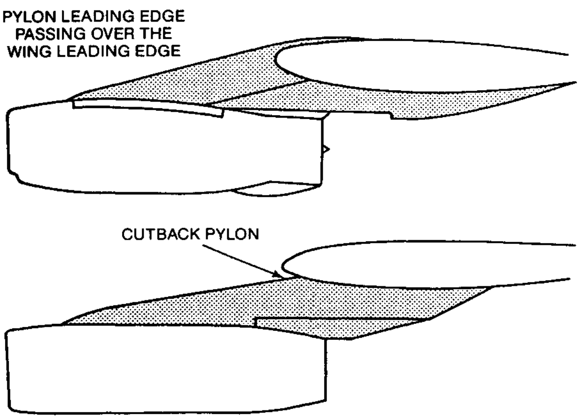

Fig. 7 Over-the-wing and cutback pylons for wing mounted jet or turbofan engines.

thickness ratios and airfoils varying along the span. Three-dimensional CFD methods can obtain moderately good results for wing configurations. How well a configuration with pylon leading edges going over the top of the wing leading edge can be modeled is not so clear. In any case, the three-dimensional results are not available. If wings of these aspect ratios were properly designed, however, the results would be close to two-dimensional airfoil results modified for sweep. An objective of wing design is to compensate for root and tip effects by varying the airfoils to obtain isobars that, for cruise conditions, follow constant-percent chord lines. This simulates the two-dimensional airfoil and simple theory. The good agreement between flight tests and the two-dimensional CFD plus simple sweep prediction for the DC-8-62, which uses the DC-8 (4% extended) airfoil but has a cutback pylon leading edge, suggests that this approach is valid; see Fig. 6. Of course, this also assumes that the wing is the critical element in the compressibility drag rise. It is interesting to note that the drag rise achieved with the DC-8-62 is as good or better than the original prediction; see Fig. 4.

The Cutback Pylon

In 1963, development of the DC-8-62 and -63 series was initiated. Preceded by much wind tunnel testing, the design featured cutback pylons whose leading edge intersected the wing lower surface at about 5% chord, as opposed to going over the top of the leading edge (Fig. 7), long fan ducts on the nacelles that eliminated the external scrubbing by the high-speed fan exhaust air, and a 6 ft span extension. The airfoil sections were not changed. The result was a surprisingly good drag rise characteristic, with a minimum of drag rise prior to M_{div}, see Fig. 6

It is interesting to note that, around 1954 during early development of the DC-8, serious pylon interference drag was found in one wind tunnel test. A cambered pylon that seemed to eliminate the problem was designed. However, later tests showed no interference drag from a symmetrical pylon and the cambered configuration was dropped. History shrouds the details of this, but it may have been the wind tunnel Reynolds number that confused the issue. Certainly, shaping the pylon to conform to the local wing flow is a good way to reduce or eliminate interference. The problem is that the "local flow" varies with flight condition.

We must really conclude that the remaining excessive drag rise of the DC-8 with the 4% extended leading edge and the original pylons was due to pylon interference, not to the airfoil at all. Was the serious compressibility drag creep of the original airfoil also due to the pylon? Obviously, it was more sensitive to pylon interference, but whether it would have been very successful if the airplane had been built with cutback pylons will never be known. However, a hint can be found in the DC-9 experience.

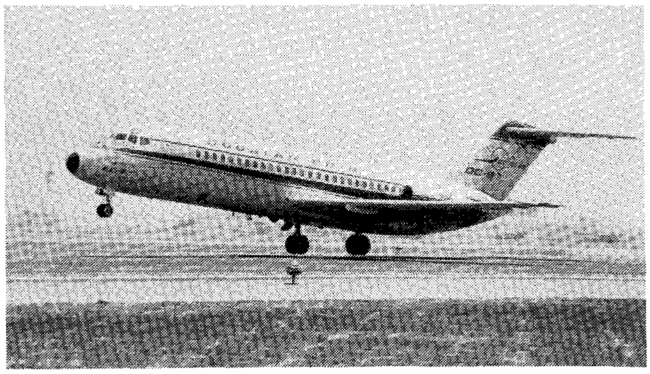

Fig. 8 The DC-9-10.

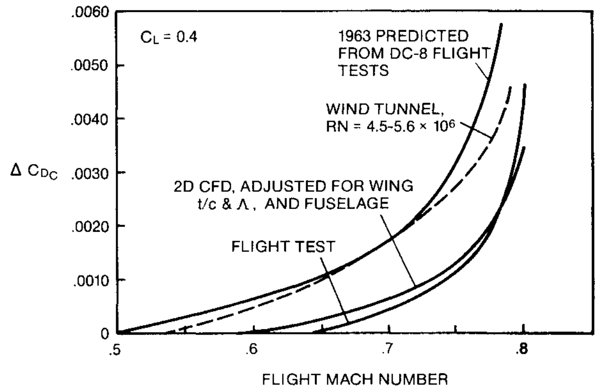

Fig. 9 DC-9-10 comparison of predicted, wing tunnel test, and flight test.

The DC-9 Airfoil Selection

When the DC-9 configuration studies began in about 1962 (Fig. 8), we who had just weathered the miserable airfoil experience with the DC-8 were sure of at least one thing—we would not use the DSMA 87 airfoil that had caused all the drag problems on the DC-8. The original DC-9 concepts emphasized simplicity and short-field length capability; no leading-edge devices were to be considered. Nevertheless, the performance analyses showed that the high maximum lift coefficient of the DSMA 87 airfoil saved so much drag and weight, by permitting a smaller wing area, that it more than paid for the drag penalty due to the high-drag creep of the airfoil as measured in the original DC-8 flight tests. After checking and rechecking these results, and with some uneasiness, we went along with the analysis and used the same basic airfoil for the original DC-9, the series 10. Moderately high Reynolds number (4.5–5.6×10^6) wind tunnel tests showed more or less drag rise than expected depending upon the lift coefficient, but were modestly better at the long range flight conditions. To our great delight, however, when the airplane flew, the drag creep was much better than the predicted or the wind tunnel values; see Fig. 9. This windfall was attributed to the lower wing sweepback angle, 24 deg rather than 30 deg on the DC-8. Only later did it seem reasonable to conclude that the lack of wing pylons was the cause of our success. Why the wind tunnel drag rise was so high remains a mystery.

Pylon Cruise Effects

A cruise drag problem arises from the pylon or fence blocking leading-edge cross flow at the cruise angle of attack. This leads to an increase in local velocity and Mach number inboard of the pylon or fence. A distortion of isobars, which should follow contours at constant percentages of the wing chord, early shocks, and premature compressibility drag creep result. A cutback leading edge on the pylon eliminates this problem, as illustrated in Fig. 10. Alternate approaches to the problem involve drooping and applying a fillet to the leading edge inboard of the pylon. Assuring that such a fillet will handle a wide range of flight conditions such as high Mach number cruise and stall is difficult.

Could modern CFD warn us of the pylon interference compressibility drag problem? I think the answer is a conditional "yes" in theory and "no" in practice. If the wing/pylon geometry were properly modeled, CFD could theoretically generate the correct solution. However, the modeling is difficult and time consuming. When I asked this question of leading applied CFD experts, I was told, "Yes, one could do it, but it is probably not practical." I conclude that the answer, from an applied engineering standpoint, is "no."

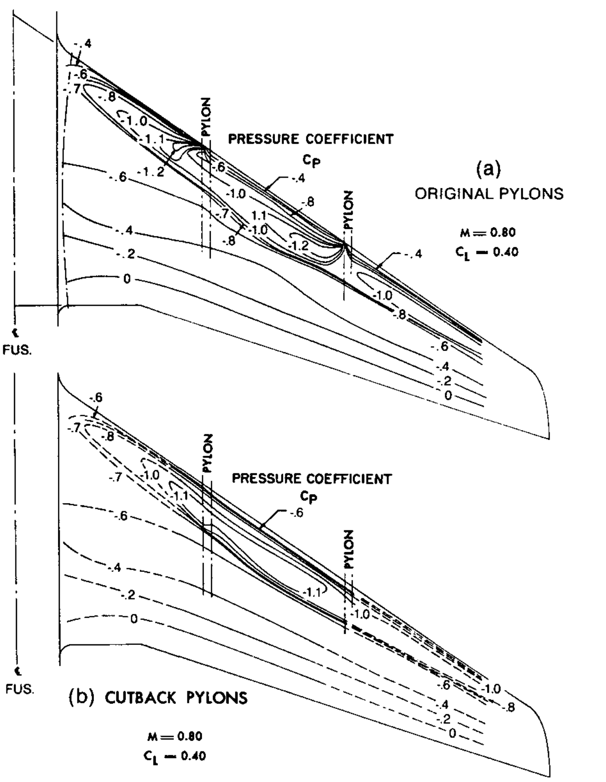

Fig. 10 DC-8 wing pressure distributions, 4% leading-edge extension/7% semispan model.

From Two Dimensions to Three Dimensions: Additional Cases

In Fig. 6, the drag rise curves for two DC-8 airfoils calculated by two-dimensional CFD are compared with flight test results converted to an equivalent zero sweep Mach number using simple sweep theory. The DC-8-62 case, free of pylon interference, showed good agreement. In Fig. 9 we compared the 1963 predicted, wind tunnel, and flight drag rise characteristics for the DC-9-10. Also shown is the two-dimensional CFD airfoil calculation adjusted to the wing average thickness and sweep. In addition, a fuselage correction of—0.015 is applied to the Mach number.[3] Here, the airfoil data Mach numbers are increased to the equivalent airplane Mach numbers by the factor, $1/\cos \Lambda$. The agreement between the CFD airfoil calculation adjusted by simple sweep theory and flight test is excellent.

Figures 11 and 12 compare three-dimensional drag rise predicted from CFD two-dimensional drag rise calculations with flight tests for the DC-9-30 and the DC-10-10 airplanes. The shape of the curves are generally in agreement, but flight tests for these airplanes show more drag rise than the predictions. The DC-9-30 is higher by about 5 drag counts, a relatively small error, while the DC-10-10 shows lower Mach numbers for any specified drag rise level by as much as 0.03. At Mach 0.83, however, the DC-10-10 increased drag is only 5 drag counts. The method of converting airfoil section drag rise includes corrections for thickness and sweep, as described above, and a fuselage Mach number correction of -0.011 for the DC-9-30 and -0.013 for the DC-10-10. The fuselage corrections are functions of the fuselage fineness ratio, decreasing with increased fineness ratio.[3]

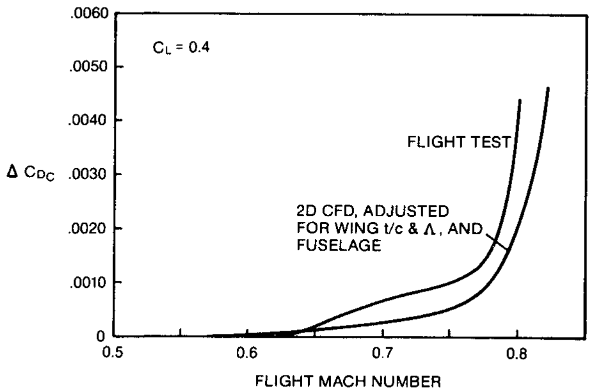

Fig. 11 DC-9-30 comparison of flight test and prediction from two-dimensional CFD.

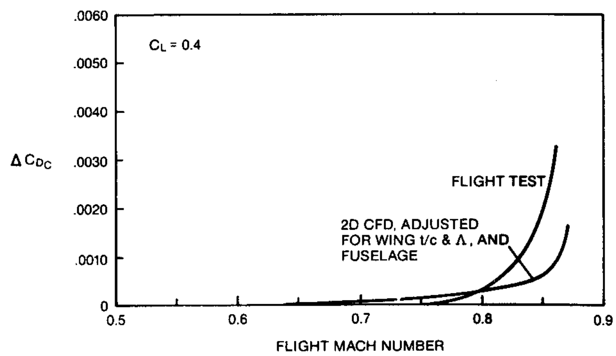

Fig. 12 DC-10-10 comparison of flight test and prediction from two-dimensional CFD.

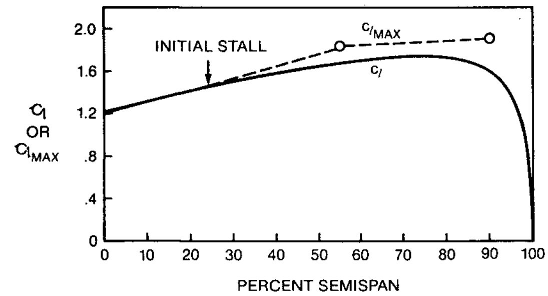

Fig. 13 Spanwise variation of c_l and $c_{l_{\max}}$.

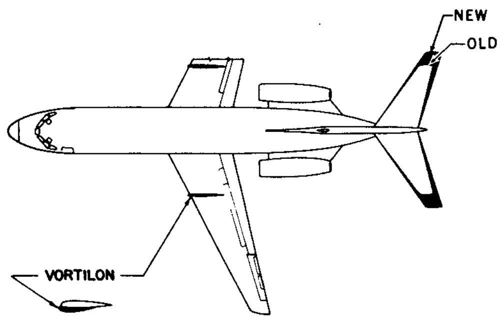

Fig. 14 DC-9 modified horizontal tail and vortilon.

Considering the four cases studied, it appears that the approximate simple sweep theory works surprisingly well, giving excellent results for two cases, a good result for the DC-9-30, and a larger error for the DC-10-10. The discrepancy may be due to the wing, nacelles, tail, or a combination of them. Available drag rise flight data on the Lockheed 1011, which has essentially the same sweep and thickness ratio as the DC-10 and a similar airfoil, shows good agreement with the DC-10-10 CFD prediction made here. Thus, this method can provide rapid compressibility drag rise predictions for early performance studies. It would be difficult for the final design to be better, but relatively easy to be worse.

Designing for Good Stall Characteristics

The DC-8 was designed for good stall characteristics using sound flying quality principles and proper aerodynamic fundamentals. A properly stalling airplane will stall first at the root. This reduces the rolling moment produced by asymmetric stalling, since the stalled area has a smaller moment arm and maintains good flow over the ailerons on the outer wing, thereby providing good lateral control to minimize any rolling motion. Furthermore, with the loss of lift occurring first inboard, the downwash over the horizontal tail is reduced, resulting in a reduced tail download and an airplane nose-down pitching moment that tends to push the airplane out of the high-angle-of-attack stalled condition. On a swept-back wing, the loss of lift inboard at the stall is forward, causing an additional favorable pitch down moment on the wing itself.

To comply with these design requirements, the DC-8 was designed with about 4.5 deg of aerodynamic washout to counteract the tendency of the highly tapered swept-back wing to produce higher section lift coefficient, outboard, and with the airfoils varied from the root to the tip to lower the maximum lift coefficient inboard. Figure 13 shows the general concept aimed at having the wing stall inboard first. When the stall first occurs inboard, a margin of 0.2–0.3 between the actual lift coefficient and the maximum section lift coefficient is maintained over the outer panel. When the airplane flew, excellent pitch down and roll control were demonstrated at the stall, leading the aerodynamicists to

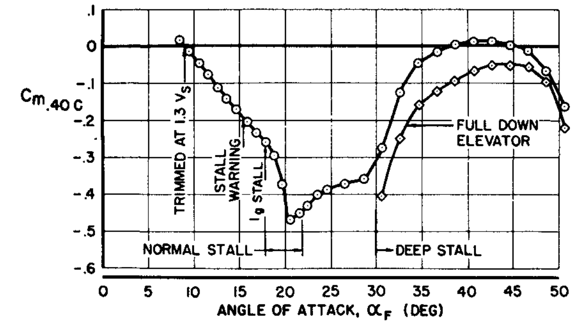

Fig. 15 Pitching moment coefficients for DC-9-10 configuration, flaps up (from Ref. 11).

ponder their brilliance. It was later found that the good stall characteristics were the result of a happy accident.

The maximum lift coefficients used in the DC-8 wing design were two-dimensional airfoil section values. We did not understand at the time the magnitude of the effect of the spanwise flow in the boundary layer. This spanwise flow transports the boundary layer away from the root, raising the maximum lift coefficients inboard. The outboard flow also thickens the boundary layer over the outboard panel and lowers the maximum lift coefficients in that portion of the wing. The result would be a nasty tip stall, no matter how many pretty pictures we drew, such as Fig. 13.

So why did the wing stall properly? The answer lay in the vortices created at the pylon/wing intersection. The spanwise flow along the leading edge of a swept wing produced a side force on the pylon. The pylon is really a highly swept lifting surface, with the lift sideways. A vortex results from this side force. The vortex is carried around the leading edge and over the top of the wing. The vortex mixes the boundary layer with freestream air and effectly interrupts the outboard flow of the boundary layer. Thus, the degradation of the outer panel $c_{l_{\max}}$ is greatly alleviated. In addition, the direction of the pylon trailing vortices, which become stronger as the angle of attack is increased, acts to increase the angle of attack at the tail which adds to the pitch-down moment on the airplane.

There is a further effect of the pylons. The blocking of the wing spanwise flow at the leading edge by the wing/pylon intersection also serves to produce a substantial increase in induced velocity and negative pressure coefficient at the inboard side of that intersection (Fig. 10). The result is that the stall first occurs at the pylon locations and the outer panel stalls later.

We had learned previously in the wind tunnel that the pylons reduced the wing maximum lift coefficient substantially without realizing the important favorable contribution of pylons to proper spanwise progression of the stall. When the reduced $C_{L_{\max}}$ caused by the pylons showed up in flight, was designed and tested wing leading-edge slots, not slats, and placed them just inboard of the pylons. The slots were 80 in. in span and eliminated the early stall at the pylon/wing juncture. The result was a violent tip stall and a very unhappy pilot. The inboard slot was reduced in spanwise length to 32 in., a configuration which eliminated much of the pylon $C_{L_{\max}}$ loss but still assured that the stall would start inboard of the inboard pylon. The airplane pitched down at the stall and had excellent lateral control. The slots, open only when the flaps are deflected, became part of the production configuration.

When the aft-engined DC-9 was developed about 6 years later, we found that the logic so successfully used for the DC-8 stall characteristics did not seem to work at all.[11] In early wind tunnel tests, the airplane had a strong pitch up at the stall. The DC-9-10, the first of the series, had 24 deg of sweep as opposed to 30 deg for the DC-8. This should have

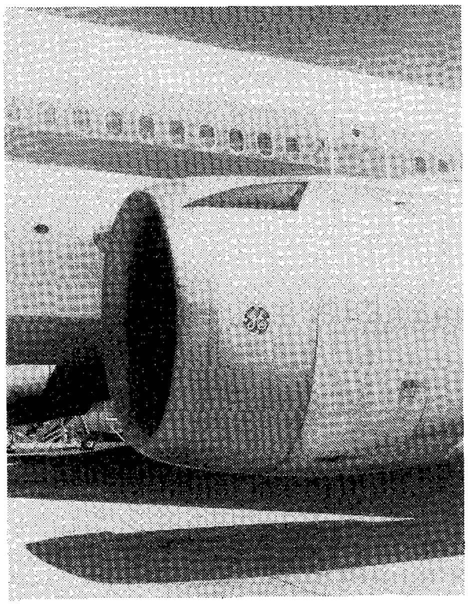

Fig. 16 DC-10 nacelle strakes.

resulted in better pitch down for the DC-9. The only other major difference in the wings of the two airplanes was the lack of pylons and nacelles on the DC-9 wing. Two pylons with nacelles were borrowed from the DC-8 model and rigged to the DC-9 model. The result was a nice pitch down at the stall. The nacelles were removed so that only the pylons remained. The pitch down also remained. We then gradually cut away the pylon and found only a small part of it was required to obtain the desired pitch-down characteristic. These pylons had leading edges that went over the top of the wing and produced the usual large loss in maximum lift. The leading edge was then cut back so that the stagnation point on the leading edge of the wing (the point of greatest spanwise flow on the swept wing) was ahead of the leading edge of the pylon almost until the stall angle was reached. In this way, the cause of the early stall, i.e., the peak suction created on the inboard side of the pylon, is delayed almost until the normal airfoil $c_{l_{max}}$ at that spanwise station occurs. Therefore, loss in wing maximum lift due to the pylons is almost eliminated, but when the wing stall first occurs at the truncated pylon, the vortex that reduces spanwise flow of the boundary layer is produced and tip stall does not occur. The truncated pylon was streamlined for drag and aesthetic reasons and appears on every DC-9 (Fig. 14) to assure initial pitch down at the stall. The device was patented and called a vortilon (vortex generating pylon) which properly describes its function. Proper spanwise placement of vortilons assures that upwash from the vortilons' trailing vortices is felt by the tail and assists the pitch down.

All swept wing airplanes have either wing-mounted pylons or some kind of wing fence, usually a leading-edge fence, to create the vortex. Use of the cutback leading edge, whether on the pylon or the fence, minimizes the adverse effect on $C_{L_{max}}$ and cruise drag.

Can CFD give adequate solutions at the stall when boundary-layer cross-flow effects on local $c_{l_{max}}$ are dominant? Can CFD describe and quantify the effects of a pylon or vortilon vortex on flying qualities and $C_{L_{max}}$? I believe that the answer is "no" at this time and for some time in the future.

The Deep Stall

Another aerodynamic difficulty has been the "deep stall," a problem generally associated with T-tail aircraft. To have a deep stall potential, an airplane will either be unstable at the initial stall angle of attack of about 16–21 deg or will become unstable at an angle well above the initial stall. If the unstable airplane pitches up to an angle above the normal stall angle, it may then become stable and trimmed in a fully stalled condition at 35-45 deg angle of attack; see Fig. 15. At such high angles, aerodynamic control surfaces such as elevators and ailerons are sufficiently stalled to have severely reduced effectiveness and a highly swept rudder may have no effect at all because its effective sweep angle is approaching 90 deg. The controls may then lack the capability to pitch the airplane nose down to escape from this stable, very high drag, high sink rate condition.

This "locked-in" deep stall is very dangerous. The instability is caused by a combination of fuselage vortices creating an enormous downwash over the inner part of the horizontal T-tail surface, where the local velocity is close to the freestream value, and a helpful upwash over the outer part of the tail, where the local velocity is greatly reduced by the wing wake and, if the engines are aft mounted, by the nacelle wakes at higher angles of attack. The stability that reappears at very high angles of attack results from the tail surface passing through the wakes and emerging into the freestream below the wakes.

The deep stall problem was tragically demonstrated by the BAC-111 crash during a flight test in 1965. Details of the causes of this accident were disseminated and led to a new series of wind tunnel tests for the DC-9, which was then in an advanced stage of development. These tests showed a previously unknown deep stall threat. Extensive testing showed the solution to be a horizontal tail span extension beyond the nacelle wakes in order to maintain sufficient tail effectiveness to assure recovery in the event of a deep stall. Fortunately, the DC-9 had not yet flown, but several sets of tail surfaces were already complete. These tails were scrapped and larger span tails hastily built (Fig. 14). A standby power system was developed to provide full nose-down elevator capability for deep stall recovery, because the normal aerodynamic tab system would not be capable of providing sufficient elevator angle at these abnormal angles of attack.

The DC-9 has a strong stable pitch down, compared to the moment at the initial stall, for about 7–10 deg beyond the stall, but then becomes unstable. The airplane is protected against the stall by a stick shaker, a visual and aural stall warning, a strong aerodynamic initial pitch down, and the standby elevator power system in the event of a deep stall. As far as I know, the deep stall has never been entered.

The physics of this problem is a mixture of fuselage boundary layer and fuselage vortex interaction, fuselage vortex flow at the tail, wakes from wing and nacelles, and the effectiveness of stalled or partially stalled tail surfaces. Could CFD do much with this? I doubt it.

Controlling Vortices with Strakes

The DC-10 Nacelle/Stalling Speed Problem

DC-10 wind tunnel tests showed significant loss in maximum lift coefficient in the flap deflected configurations, with landing slat extended, compared to predictions. This resulted in a stall speed increase of about 5 knots in the approach configuration.[12-14] Initial wing stall occurred behind the nacelles and forward of the inboard ailerons. The problem was traced by flow visualization techniques to the effects of the nacelle wake at high angles of attack and the absence of the slat in the vicinity of the nacelle pylons. The solution was developed in the NASA Ames Research Center 12 ft pressurized tunnel and turned out to be a pair of strakes mounted forward on each side of the nacelles in planes about 45 deg above the horizontal. The final strake shape was optimized in flight tests, as shown in Fig. 16. The strakes are simply large vortex generators. The vortices mix nacelle boundary-layer air with the freestream and disperse the momentum loss in the wake. The vortices then pass just

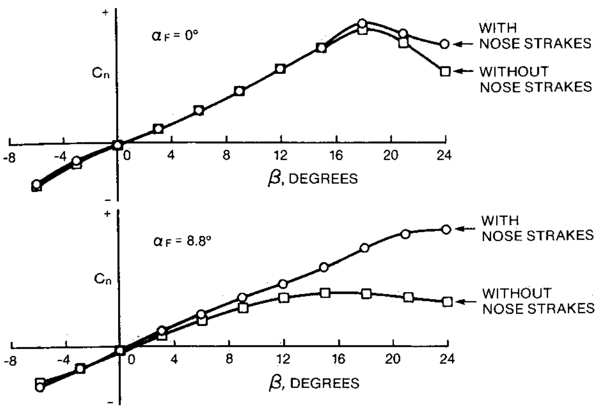

Fig. 17 Effect of nose strakes on DC-9-50 directional stability.

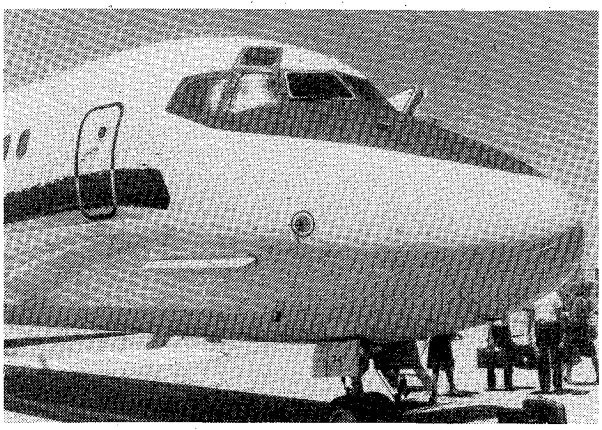

Fig. 18 MD-80 fuselage nose strake.

over the upper surface of the wing, continuing this mixing process. The counterrotating vortices also create downwash over the wing region unprotected by the slat, further reducing premature stall. The effect of the strakes is to reduce required takeoff and landing field lengths by about 6%, a very large effect. Here again, we have a problem involving wakes, separation, and vortices acting on a configuration involving nacelle, pylon, wing, and slats. Contemporary applied CFD cannot adequately cope with this complex aerodynamic difficulty.

DC-9-50 and MD-80 Fuselage Strakes

Another example of complex aerodynamic interactions was found in the DC-9-50 and MD-80 directional stability at high angles of attack.[15] Vortices shed from the nose of the extended fuselages of these airplanes in sideslips at high angles of attack degraded directional stability; see Fig. 17. The problem was caused by interaction of these vortices and the vertical tail. The solution was found to be fuselage strakes (Fig. 18), developed and optimized in wind tunnel testing. The strakes, when placed in a low forward position on the fuselage, generate vortices that alter the flow past the vertical tail so that good directional stability is maintained over the required ranges of angles of attack, sideslip, and flap deflection. The original problem involved interaction between the fuselage boundary layer and the high-angle-of-attack flow. These phenomena plus the interaction with the vertical tail could not be handled analytically. The wind tunnel remains essential for such complex flow interactions.

A Note on Wind Tunnels

Experience has shown that high Reynolds number wind tunnels, with Reynolds numbers of about 6×10^6 or greater, are generally required to assure a high likelihood of agreement with flight tests. The flight test Reynolds numbers for large aircraft are 3-10 times as large. The high Reynolds numbers are required for conditions where the effect of the boundary layer is important. This includes angles of attack near stall and compressibility drag rise. Having made this sweeping statement, it is sad to have to emphasize that even Reynolds numbers of close to 6×10^6 do not guarantee a correct wind tunnel result, particularly where compressibility drag rise is involved. For example, the DC-9-10 and the DC-8-62 showed poor agreement between wind tunnel and flight compressibility drag rise, while the DC-8 (original) and the DC-9-30 showed good agreement. As a result of this experience, the author developed a basic wind tunnel hypothesis which states: wind tunnels are correct half the time so that the only chore remaining for the aerodynamicist is to determine which half to believe.

In spite of these problems, the wind tunnel will almost always be successful in indicating the better of two configurations, although the magnitude of the improvement may remain in doubt. For problems that modern CFD can handle efficiently, the calculated result may be more reliable than the wind tunnel. But when the solution can not be found analytically for the reasons discussed above, the wind tunnel remains indispensable.

Conclusions

Early swept-wing transport designers made remarkably successful use of inadequate theory and empirical data. This required a sound understanding of the physics of the flow, a talent that will always be important. Computational fluid dynamics, by itself, may tend to discourage this, since the computational art is largely devising acceptable grids and mathematical procedures. Grasping the nature of a problem flow, and being able to judge what kind of changes in the configuration will improve the flow will remain an important part of the applied aerodynamicist's skills. The design change can then be evaluated relatively quickly with CFD procedures in many cases, although a wind tunnel may be necessary in others.

The wind tunnel will remain an essential, although imperfect, tool for the foreseeable future. The remarkable progress in computational capability over the past decade, extrapolated into the future, gives hope that this situation may change. It has been said that verification of CFD results in the next 15 years will depend largely on comparisons with wind tunnel data.[16] The desired result is really to match flight test results. It might be more convincing to compare CFD codes to existing aircraft for which well-established flight data exist than to compare only with wind tunnel results.

References

[1]Miranda, L. R., "Application of Computational Aerodynamics to Airplane Design," *Journal of Aircraft,* Vol. 21, June 1984, pp. 355-370.

[2]Ruppert, P. E., and Tinoco, E. N., "Impact of Computational Methods on Aircraft Design," AIAA Paper 83-2060, Aug. 1983.

[3]Henne, P. A., Dahlin, J. A., and Peavey, C. C., "Applied Computational Transonics-Capabilities and Limitations," *AIAA Progress in Astronautics and Aeronautics: Transonic Aerodynamics,* Vol 81, edited by D. Nixon, AIAA, New York, 1982.

[4]Shevell, R. S., "Operational Aerodynamics of High Speed Transport Aircraft," *Institute of the Aeronautical Sciences Journal* Vol. 15, No. 3, 1948, pp. 133-143.

[5]Thomas, G. B., "Maximum Lift and High Mach Number Drag Characteristics of NACA Modified Four-Digit Airfoil Sections," Douglas Aircraft Co. Rept. SM-14585, Dec. 5, 1952

[6]Edwards, G. and Boetz, F. W., "An Analysis of the Forces and Pressure Distributions on a Wing with the Leading Edge Swept Back 37.25 deg.," NACA RM A9K01, 1950.

[7]Mayer, J. P., "A Limit Pressure Coefficient and an Estimation of Limit Forces on Airfoils at Supersonic Speeds," NACA RM. L8F23, 1948.

[8]Smith, A. M. O., "High Lift Aerodynamics," *Journal of Aircraft,* Vol. 12, June 1975, pp. 501-530.

[9]Pearcey, H. H., "The Aerodynamic Design of Section Shapes for Swept Wings," *Advances in the Aeronautical Sciences,* Vol. 3, Pergamon Press, New York, 1961, p. 277.

[10]Henne, P., Douglas Aircraft Co., private communication, June 1985.

[11]Shevell, R. S. and Schaufele, R. D., "Aerodynamic Design Features of the DC-9," *Journal of Aircraft,* Vol. 3, Nov-Dec., 1966, pp. 515-523.

[12]Well, O. D., Douglas Aircraft Co., private communication, June 1985.

[13]Callaghan, J. G., "Aerodynamic Prediction Methods for Aircraft at Low Speeds with Mechanical High Lift Devices," AGARD Lecture Series No. 67, Von Kármán Institute, Brussels, May 1974.

[14]Bates, R. E., "DC-10 Design Development," Douglas Aircraft Co. Paper 5996, presented to ATA Engineering Maintenance Conference, Dec. 1971.

[15]Brislawn, M., Douglas Aircraft Co., private communication, July 1985.

[16]Korkegi, R. H., "Impact of Computational Fluid Dynamics on Development Test Facilities," *Journal of Aircraft,* Vol. 22, March 1985, pp. 182-187.