Key Topics for High-Lift Research:
A Joint Wind Tunnel / Flight Test Approach

REPORT

for

NASA-Ames University Consortium
Joint Research Interchange

May 1, 1995 - September 30, 1996

COLLABORATORS

David Fisher
NASA Dryden
Flight Research Center

Flint O. Thomas
Associate Professor,
and
Robert C. Nelson
Professor,
Department of Aerospace and
Mechanical Engineering
University of Notre Dame

CONTENTS

Abstract

Future high-lift systems must achieve improved aerodynamic performance with simpler designs that involve fewer elements and reduced maintenance costs. To expeditiously achieve this, reliable CFD design tools are required. The development of useful CFD-based design tools for high lift systems requires increased attention to unresolved flow physics issues. The complex flow field over any multi-element airfoil may be broken down into certain generic component flows which are termed *high-lift building block flows* . In this report a broad spectrum of key flow field physics issues relevant to the design of improved high lift systems are considered. It is demonstrated that in-flight experiments utilizing the NASA Dryden Flight Test Fixture (which is essentially an instrumented ventral fin) carried on an F-15B support aircraft can provide a novel and cost effective method by which both Reynolds and Mach number effects associated with specific high lift building block flows can be investigated. These in-flight high lift building block flow experiments are most effective when performed in conjunction with coordinated ground based wind tunnel experiments in low speed facilities. For illustrative purposes three specific examples of in-flight high lift building block flow experiments capable of yielding a high payoff are described. The report concludes with a description of a joint wind tunnel / flight test approach to high lift aerodynamics research.

I. Background and Motivation

The low speed/high lift performance of commercial transport aircraft is a primary safety issue and also poses a major constraint regarding both sizing and overall configuration performance. In a recent study by the National Research Council entitled, "Aeronautical Technologies for the Twenty-First Century" (1992), the importance of low speed, high-lift technology to the future of the United States commercial transport aircraft industry was identified. The report concludes that for the United States aircraft industry to retain its leadership role, it must reduce the acquisition and maintenance costs of future designs.

a. High Lift System Design Issues for Transport Aircraft

High-lift systems are used in order to achieve the required take-off and landing performance for commercial jet transports. The high lift system must provide both high lift for landing and high L/D for take-off, climb out and noise reduction. The aerodynamic performance of the high lift system also plays an important role in the payload-range capability of an aircraft for a given field length.

The high lift system designer is faced with a demanding set of requirements and constraints. The high lift system can be thought of as a subsystem of the wing. Its sole purpose is to give the airplane the desired field performance. As in most system designs, constraints are generated for subsystems by other groups. While the high-lift system is essential for field performance of the airplane it does not set the geometric parameters of the wing. Rather, the wing geometric parameters are set by required cruise performance. The performance group determines the wing geometric characteristics to meet the payload, range and cruise performance specifications. They along with the structural design define most of the constraints on the system.

What are some of the constraints faced by the high lift system designer? As stated earlier, cruise performance establishes many of the high lift design parameters. For example, chord, thickness distribution, aspect ratio, etc. are determined by optimizing the wing for cruise performance. The challenge for the high lift designer is then to come up with an optimum design measured by merit functions based on weight and complexity to meet the field requirements, takeoff and landing performance. This must be accomplished with minimum effect on the cruise performance. In addition, the high lift system must be as light as possible and of minimum complexity.

The high lift designer must determine the type of high lift devices to use, their geometry, spanwise extent and the optimum flap and slat rigging (in terms of gap and overhang). Structural considerations typically place constraints on the chordwise extent

of the high lift system. Wing spar location (forward and aft) are set by wing stiffness and internal fuel volume requirements. This aspect is further complicated by the space required for internal storage of the main landing gear. The space within the wing required by the main landing gear has an impact on the size and location of the inboard flap. Another challenge for the designer is the support and deployment mechanisms required to move the slat and trailing flaps to their various positions during take-off and landing. The support and deployment system again must be light weight and when retracted have a minimum influence on the wing drag count. It should also be as simple as possible in order to minimize maintenance costs.

Given the constraints mentioned above how does the high lift system designer determine an optimum high lift system for a given wing? In the initial design phase the designer typically makes use of two-dimensional data bases and CFD codes. As we shall see, two dimensional CFD codes really have a quite limited predictive capability due to limitations in our understanding of the viscous flow physics encountered in high lift systems. The 2-D data bases available are obtained at Reynolds numbers well below flight (which for a commercial transport aircraft can extend to 50×10^6, based on mean aerodynamic chord). In addition, when a flap or slat is incorporated into a finite wing, the performance is degraded due to three-dimensional effects. Three-dimensional flow effects are associated with sweep, engine nacelle and pylon flow interactions and segmentation of the slats and trailing flaps due to spanwise limitations. The designer must account for these three dimensional effects in the design. At present 3-D design codes are neither easy to use nor have they been shown to produce truly reliable results.

Because of limitations in CFD and the restricted availability of high Reynolds number test facilities, the high lift designer is forced to rely on cut and try experiments (usually at low Reynolds numbers) and intuition about the flow physics associated with a finite wing high lift system. Due to its inherent complexity and the consequent emphasis on empiricism in its design, the high-lift system has invariably been a large lead time item in the development of new transport aircraft. For the current generation of commercial transport aircraft, the high-lift system is also a notoriously high maintenance cost item.

b. Flow Field Physics Issues for High Lift Systems

It is imperative that future high-lift systems achieve improved aerodynamic performance with simpler designs that involve fewer elements and reduced maintenance costs. To expeditiously achieve this goal, more reliable CFD design tools are required.

In the past, limitations in 2-D CFD capability for high lift systems stemmed both from the geometric complexity of multi-element airfoils as well as limitations inherent in

the flow physics modeling. *Great strides have been made in terms of grid generation methodologies and in algorithm development so that the issue of geometric complexity has been largely, if not completely, resolved. Unfortunately most of the flow field physics issues associated with high-lift systems remain unresolved.* This stems from the fact that the flow over a multi-element airfoil is quite complex and includes numerous viscous dominated effects for which our present understanding is extremely limited. For example, despite the desirability of doing so, one cannot simply "dial in" a new Reynolds number and repeat a high lift computation in order to study Reynolds number scaling issues. Changing the Reynolds number requires the computational fluid dynamicist to go back and examine the transition model or other physical assumptions built into the analysis. This is because both the transition mechanism and multiple viscous interactions can significantly change their character over the operating Reynolds number range of the high-lift system. Our understanding of these viscous flow phenomena in three dimensions is extremely limited.

Currently, CFD is capable of matching 2-D high-lift wind tunnel test results at fixed Reynolds number only after the appropriate model constants have been properly adjusted. Recourse to such "postdiction" is obviously unsatisfactory for a dependable high lift system design tool. In order for CFD to assume its desired role in the high-lift design process, the many unresolved flow physics issues that limit predictive capability must be addressed.

c. Introduction to the High-Lift Building Block Flow Concept

The flow field over any multi-element airfoil may be broken down into certain generic component flows which we will term *"high-lift building block flows"* (Thomas, 1995). These include: (1) laminar separation bubbles, (2) large-scale cove flow separation (3) boundary layer-wake interactions (i.e. the confluent boundary layer), (4) boundary layer development under influence of both arbitrary streamwise pressure gradient and surface curvature, (6) multiple wake interactions, (7) wake development in strong pressure gradients and with streamline curvature, (9) boundary layer transition, and (10) relaminarization of turbulent boundary layers.

The fluid dynamicist will recognize the above list to contain some of the most challenging unsolved problems of fluid mechanics. *It is important to recognize that in a multi-element airfoil flow field these building block flows are also strongly coupled.* Their interaction will determine the global structure of the high-lift flow which, in turn, will determine the high lift system performance. This can give rise to extremely complex, nonintuitive aerodynamic behavior. Indeed, this may well lie at the heart of so-called
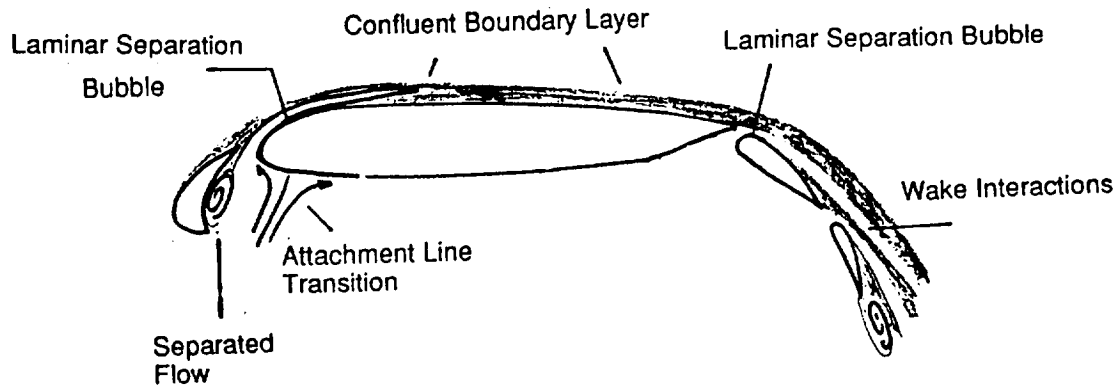
Figure 1. Viscous Interactions in a High Lift System
(from Mack & McMasters, 1992)

"inverse Reynolds number effects" which involve the unexpected deterioration of high-lift system performance with increased Reynolds number. The existence of inverse Reynolds number effects gives rise to fundamental questions about how one rationally extrapolates wind tunnel test results to flight Reynolds number.

d. Advancing State-of -the Art in High Lift Aerodynamics

The coupling of *high-lift building block flows* in high-lift systems poses a real impediment to gleaning useful flow field physics information from wind tunnel test data. In effect, the coupling can often obscure the "cause" and "effect" observations which are so essential in scientific experimentation. Consequently, it is the position of the authors that the most useful approach to advance state-of-the-art in high-lift aerodynamics is to perform benchmark experiments involving *individual* high-lift building block flows.

The objective of such research is clear: to develop an improved predictive capability for individual high-lift building block flows. It seems quite naive to expect to predict the flow over a multi-element airfoil if we cannot reliably predict the associated building block flows! *Understanding high-lift building block flows individually is a prerequisite to understanding their integrated behavior in a high-lift system.* As used in the previous sentence, the term "understanding" is taken as synonymous with the ability to perform predictions that are made without prior knowledge of the experimental results but which are in good agreement with experiment. We would argue that some of our major limitations in high-lift design methodologies and confusion regarding extrapolation of subscale wind tunnel test results to flight stem from a lack of fundamental research into these key building block flows.

8

II. A Joint Wind Tunnel / Flight Test Approach: A New Role for Flight Test in High Lift Research

The approach that needs to be taken to improve CFD capability in high-lift is to develop a series of joint wind tunnel / flight test experiments to address fundamental flow physics issues surrounding each of the building block flows cited above. Studying the individual building block flows through coordinated wind tunnel and flight experiments is probably the most expeditious and cost effective way of developing CFD capability for high-lift system design. To attempt to compute the complicated viscous flow interactions associated with lift production on three-dimensional multi-element wings without a better understanding of the flow physics will at best produce solutions of dubious quality and may, in fact, ultimately impede the acceptance of CFD tools by the design community. Although the characteristic high-lift system building blocks listed above are obviously interrelated, we must be able to first predict them independently before we can hope to integrate their effects into a high-lift system flow field calculation. The investigation of the high-lift building block flows should be performed in the spirit of providing relevant flow field physics in a form that is useful for inclusion into current CFD design tools.

Flight Reynolds numbers for commercial transports (based on mean aerodynamic chord) can extend to 50×10^6. As far as wind tunnel testing is concerned, achieving high Reynolds number at subsonic speeds requires either a very large wind tunnel (which poses problems in terms of fabrication cost and power consumption) or a more moderate size tunnel in which the working fluid is under pressure and/or cooled. The use of pressure and/or cryogenic tunnels has typically been reserved primarily for 2-D or semi span tests seeking to optimize specific configurations. Disadvantages of the use of such facilities for more fundamental work on building block flows includes high operational costs, difficulty in model access, limitations on viewing access to the model which tend to prevent the use of sophisticated laser based diagnostics like LDA or PIV. On the other hand, low speed wind tunnels are inexpensive and allow an extremely wide variety of sophisticated diagnostics to be implemented relatively easily. This provides the type of detailed flow field information required in order to address key flow physics issues. However, such facilities pose very real limitations in terms of Reynolds number capability that often lead one to question both the relevance and generality of the results obtained.

We feel that flight test could provide a key role in addressing Reynolds and Mach number effects on high lift building block flows, thereby complimenting ground based experiments. In the recent past, much of the high lift flight test research activity has

centered around the NASA Transport Systems Research Vehicle, a Boeing 737-100 (e.g. van Dam et al, 1993). Research of this type has provided important flight data concerning the performance of a particular high lift system. However, what we propose is an alternate approach in order to investigate high lift building block flows.

Of particular interest in the context of this report is the utilization of flight tests to be performed at NASA Dryden Flight Research Center in order to improve our understanding of the flow physics of high lift building block flows. The proposed flight tests would utilize the NASA Dryden Flight Test Fixture (FTF) carried on an F-15B support aircraft in order to perform fundamental high lift building block flow experiments which will compliment wind tunnel experiments at lower Reynolds number. The FTF is essentially an instrumented ventral fin that effectively provides a "flying wind tunnel" capability. The use of the FTF provides a novel and cost effective way to explore Reynolds and Mach number effects associated with key high lift building block flows. Advances in onboard instrumentation now allow relatively sophisticated flow field diagnostics to be utilized in flight, thereby allowing data acquisition comparable to that previously attainable only in low Reynolds number ground based facilities.

III. Sample High Lift Building Block Flow Experiments
Using the NASA Dryden Flight Test Fixture

In order to illustrate how the NASA Dryden Flight Test Fixture could be exploited in order to address flow physics issues relevant to high lift building block flows, this section outlines three potential experiments. Although meaningful experiments could be designed and conducted for each of the high lift building block flows listed in section Ic., we choose to illustrate the application of the NASA Dryden FTF by describing three experiments involving:

- Boundary Layer Relaminarization
- The Confluent Boundary Layer
- Wake Development in Pressure Gradient

It is useful for the reader to keep in mind that although the primary focus of the following sections will be on describing the flight test experiment itself, this should ideally be performed in conjunction with complimentary ground based experiments.

a. High Lift Building Block Flow 1: Boundary Layer Relaminarization
1. Concerning Boundary Layer Relaminarization in High Lift Systems

It is well known that if a turbulent boundary layer is exposed to a sufficiently large favorable pressure gradient the boundary layer can return to a laminar - like state and this

process has been termed "relaminarization" or "reverse transition". The relaminarization process is associated with a thinning of the boundary layer as illustrated schematically in Figure 2, a reduction in the bursting rate and the development of a new laminar sublayer near the wall. Laws of the wake and wall cease to be valid and laminar - like mean velocity profiles develop along with an associated reduction in skin friction coefficient. Far from the wall, the relative turbulent kinetic energy decays because of local flow acceleration but the absolute magnitude of the Reynolds stresses hardly change at all through the flow acceleration. Narasimha and Sreenivasan (1973) have shown that the relaminarized boundary layer can be described in terms of an outer stress free but rotational layer riding over an inner laminar sub-boundary layer. The thinning of the viscous layer is then a consequence of conservation of vorticity in the outer layer.

It has been speculated that relaminarization could play a significant role in the performance of commercial high lift systems. There can be little doubt regarding the important role played by leading edge flow physics in general. The question is whether relaminarization is a part of this process. The flow over a swept wing will experience a very strong favorable pressure gradient as it circumnavigates the leading edge from the attachment line. In addition, Garner et al. (1991) present evidence which suggests that
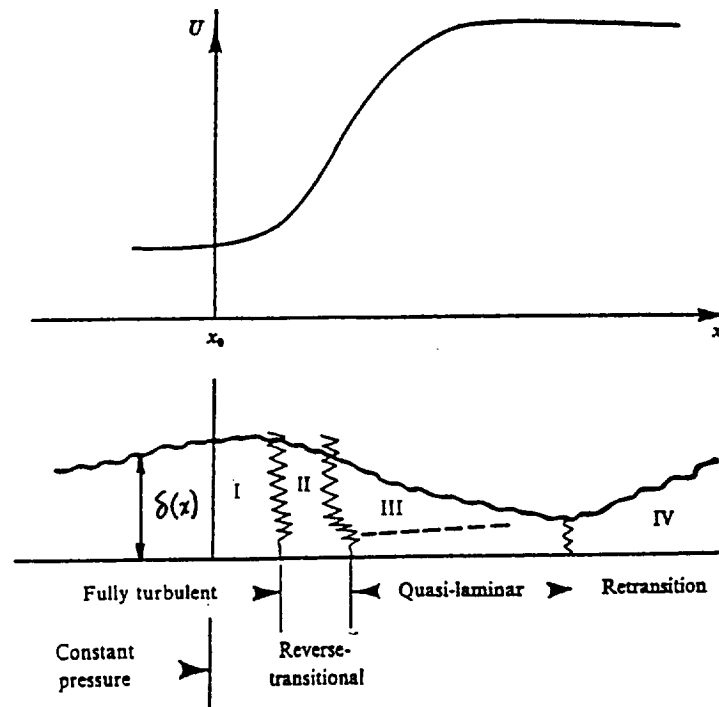
Schematic of Turbulent Boundary Layer Relaminarization
from Narasimha and Sreenivasan (1973)

failure of the primary wing leading edge flow to relaminarize at large Reynolds number may be responsible for adverse Reynolds number effects on C_{Lmax}. In-flight experiments on the NASA Transport Systems Research Vehicle (Boeing 737-100) by van Dam et al (1993) provide some experimental evidence based on Preston tube measurements which suggest the occurrence of boundary layer relaminarization on the upper slat surface of the Boeing 737.

The most widely used criterion for the occurrence of relaminarization is the parameter K (Launder and Jones, 1969) which is defined,

$$K = \frac{v}{U_e\, \xi} \qquad (1)$$

where U_e is the local inviscid velocity, v is the kinematic viscosity and ξ is a characteristic length scale associated with the flow acceleration which is defined as,

$$\xi = \frac{U_e}{dU_e\, /\, ds} \qquad (2)$$

where s is the spatial coordinate describing the local inviscid streamline. Combining eqn. 1 and eqn. 2 we have,

$$K = \frac{v}{U_e^2}\frac{dU_e}{ds}. \qquad (3)$$

Laboratory experiments have shown that relaminarization is likely when $K \geq 3\ X\ 10^{-6}$. Measurements by van Dam et. al. (1993) on the NASA Boeing 737-100 show peak K values on each of the five elements of the high lift system in excess of this value, further suggesting the possibility of relaminarization in high lift systems.

For flow over a 2-D body of chord length c where coordinate x denotes the distance over the surface of the body, it may be shown that the relaminarization parameter can be expressed as,

$$K = -\frac{1}{2}\left(\frac{1}{Re_c}\right)\frac{1}{\left(1 - C_p\right)^{3/2}}\frac{dC_p}{d(x\,/\,c)} \qquad (4)$$

where Re_c denotes the chord Reynolds number and C_p is the body surface pressure coefficient. It is apparent from (1) through (4) that K is determined completely from the inviscid external flow and can therefore be estimated from surface pressure measurements. Equation (4) also shows a tendency for the K value to be reduced with increased chord Reynolds number.

One troubling aspect of the relaminarization parameter is that it does not contain any information regarding the duration over which the flow is exposed to the favorable pressure gradient. Since there must be some finite time scale inherent to the relaminarization process, the K parameter alone seems an incomplete criterion. However,

an even more fundamental and troubling aspect is the recognition that the relaminarization criterion contains no boundary layer parameters at all! Indeed when written in the form of equation (4) it is very difficult to see why chord Reynolds number should be a relevant scaling parameter.

There have been some attempts to include boundary layer parameters in a criterion for relaminarization . For example, Patel and Head (1968) suggest the following parameter, Δ_p , defined as,

$$\Delta_p = -K\left(\frac{C_f}{2}\right)^{-3/2} \tag{5}$$

where C_f is the skin friction coefficient and it was noted that the log-law of the wall behavior disappears near $\Delta_p \approx -0.025$.

Laboratory experiments suggest that the boundary layer relaminarization process is not due to either dissipation or negative turbulence production. However, in all cases where relaminarization has been observed to occur, order of magnitude estimates show that the parameter,

$$\Lambda = -\frac{dP}{ds}\frac{\delta}{\tau_w}, \tag{6}$$

is very large. Here δ is the boundary layer thickness and τ_w is the wall shear stress. A large value of Λ implies that the pressure gradient is quite large in relation to the stress gradient in a relaminarizing flow. In other words, the Reynolds stress field does not change rapidly enough to respond to the external flow acceleration. This suggests the possibility of an alternate basis for a relaminarization criterion.

In order to illustrate how boundary layer relaminarization could play a role in inverse Reynolds number effects in high lift systems, consider the following speculative scenario: From the attachment line studies by Poll (1979) and the flight test results by van Dam et al (1993) it may be expected that for swept wing flows at all but the lowest Reynolds numbers the attachment line boundary layer is turbulent. As it circumnavigates the nose of the airfoil, the flow accelerates and consequently the boundary layer is initially exposed to a strong favorable pressure gradient. If this negative streamwise pressure gradient is strong enough as measured by the parameter K then relaminarization can occur. The effect of the relaminarization process will be to reduce the thickness of the boundary layer, and this is favorable for C_{Lmax}. Perhaps more importantly, it will also serve to move the location of onset of confluence with the slat wake downstream. The net effect will be to move the separation location on the primary airfoil aft. It has been shown (Garner et al, 1991) that as the Reynolds number increases, K is reduced to such a level that relaminarization may not take place. Thus, for sufficiently high Reynolds

13

numbers, relaminarization of the leading edge boundary layer may suddenly cease. As a result the leading edge boundary layer flow thickens, the location of confluence between the slat wake and turbulent boundary layer on the wing moves forward and gives rise to rapid mixing and the generation of a very thick viscous layer that will readily separate due to the adverse pressure gradient aft of the primary airfoil pressure peak. In this manner C_{Lmax} may be limited at high Reynolds numbers. It should be pointed out, however, that the scenario just described suggests that once relaminarization is lost C_{Lmax} would continue to drop with further increases in Reynolds number. In reality some experiments have actually shown a tendency for C_{Lmax} to recover and again increase at the highest Reynolds numbers. This suggests that the description of inverse Reynolds number effects described above may be incomplete and additional factors in the leading edge flow physics need to be accounted for.

It's important to point out that hard evidence for the occurrence of relaminarization in actual high lift flow fields is still lacking. Although the 737 flight test experiments of van Dam et al (1993) suggest the possibility of relaminarization, most work involving relaminarization has been done at low Reynolds numbers in academic wind tunnels. In the next section we describe basic boundary layer relaminarization experiments to be performed in flight using the FTF experimental platform. This will allow a fundamental investigation of relaminarization at Reynolds numbers and pressure gradients characteristic of those occurring in high lift systems.

2. In -Flight Relaminarization Experiments Using the FTF

The in-flight experiments will involve examination of the conditions giving rise to relaminarization of a turbulent boundary layer which develops on the FTF fixture. The favorable pressure gradient and associated local flow acceleration will be induced by an airfoil placed off-surface as shown schematically in Figure 3. In this figure the leading edge of the airfoil is located near the mid chord of the FTF platform so that flow acceleration commences near this location. The relaminarization process would take place on the FTF surface in the region of flow acceleration under the off surface airfoil. The boundary layer on the FTF would retransition at some point after the strong favorable pressure gradient is removed. Note, however, that any possible interaction between the airfoil wake and FTF boundary layer downstream of the of the airfoil is of no importance as far as this relaminarization study is concerned.

For the arrangement shown in Figure 3 lateral loads are small except perhaps for flight trajectories which involve large sideslip. At any rate, these effects could always be

Boundary Layer Trip

Turbulent Boundary Layer

Symmetric 2-D Airfoil

Airfoil Support and Boundary Layer Fence

FTF Platform (side view)

Leading Edge

Symmetric 2-D Airfoil

Instrumented FTF Surface: Pressure Taps and Flush Mounted Hot-Film Sensors

Airfoil Support and Boundary Layer Fence
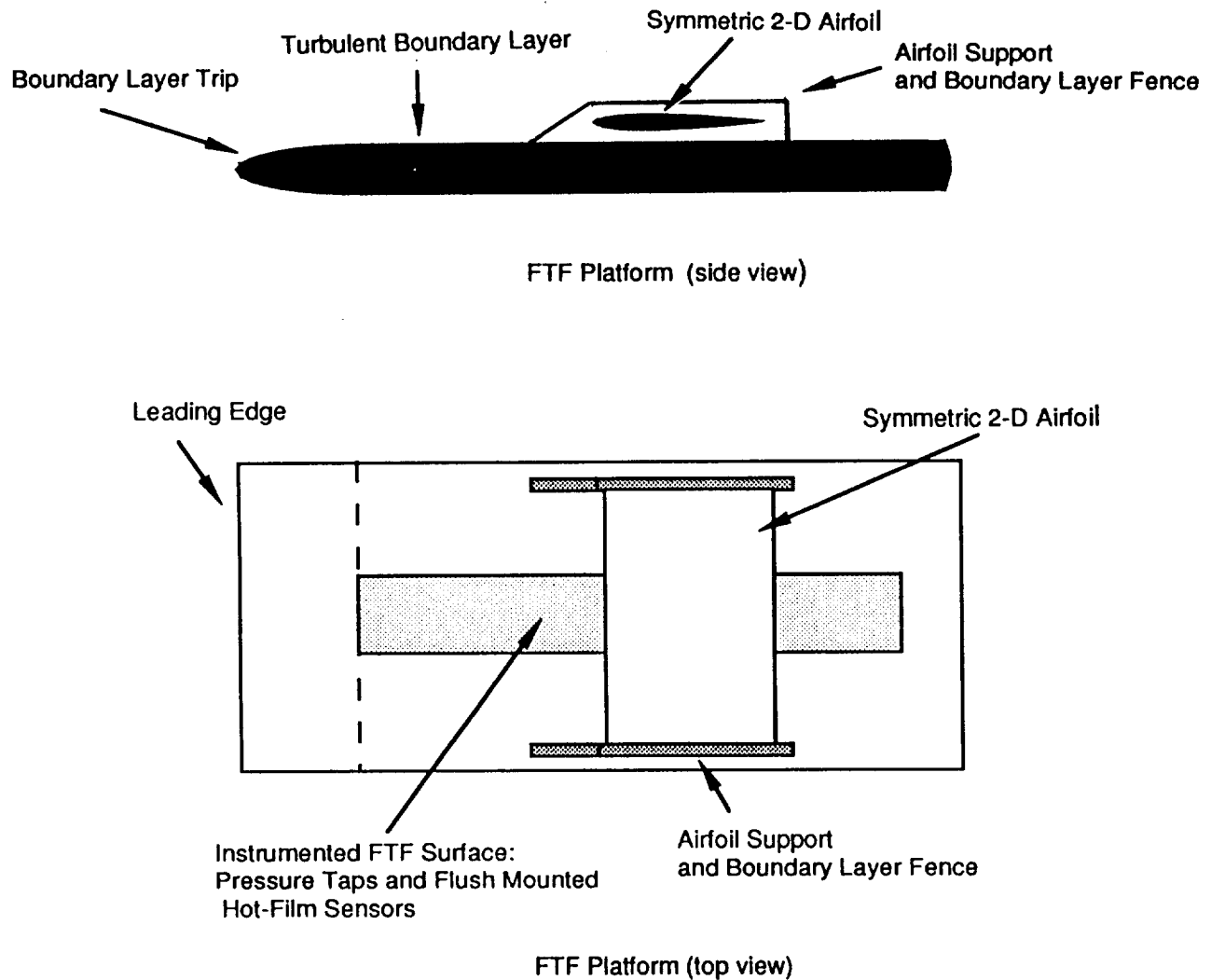
FTF Platform (top view)

Figure 3. Schematic of the FTF Relaminarization Experiment

minimized by mounting a dummy airfoil on the opposite side of the FTF in order to provide symmetry.

In order to perform the experiment it is necessary to develop a turbulent boundary layer on the FTF surface. The FTF will be equipped with an elliptic leading edge with distributed roughness in order to expedite boundary layer transition. Fences will be used to insure the development of a nominally two-dimensional turbulent boundary layer near the onset of favorable pressure gradient. A symmetric airfoil whose leading edge is located near the mid-chord of the FTF is positioned off-surface and this distance is fully

15

adjustable. By varying the airfoil - FTF surface gap width, the magnitude and duration of the favorable pressure gradient imposed on the boundary layer may be controlled. Streamwise and spanwise arrays of surface pressure taps on the FTF surface will allow documentation of the imposed pressure gradient dP/dx and evaluation of the associated K parameter. It is desirable to perform the experiments for K values both above and below the 3×10^{-6} threshold value. By selective positioning of the airfoil for a given flight speed it is possible to set appropriate K values.

An array of flush surface mounted hot - film probes on the FTF will be used to assess whether relaminarization has occurred. For cases where relaminarization is indicated, profiles obtained via off surface hot-film and/or pitot tube rakes will be used to assess the thickness and characteristic profile shape of the boundary layers.

Computations using panel methods indicate that the experimental arrangement shown in Figure 3 is capable of producing large suction pressures on the FTF surface, similar to those that characterize the leading edge of high lift systems. For a given flight Mach number, the limitation in local flow acceleration is ultimately set by the need to avoid reaching local sonic conditions. A computed surface pressure distribution is shown in Figure 4 for a NACA 0010 airfoil whose centerline is placed a distance $\Delta z / c = 0.036$ above the FTF surface (where c is the chord of the FTF platform). This pressure gradient will be imposed on a high Reynolds number turbulent boundary layer. For example, assuming a pressure altitude of 10, 000 ft., and flight Mach number of M = 0.25, the
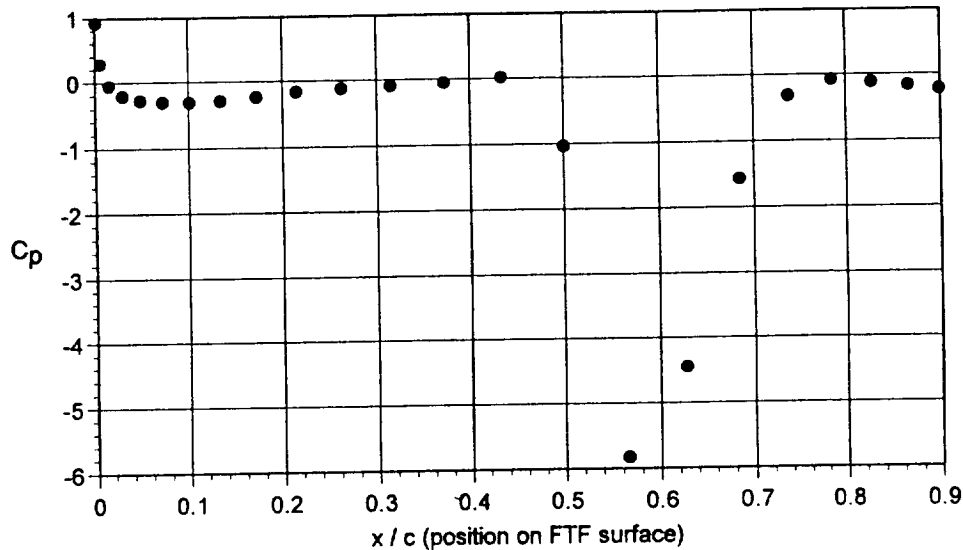

Figure 4. Example Surface Pressure Distribution on the FTF

Reynolds number based on the length of the FTF platform is over 13 million while the local Reynolds number, Re_x, at the onset of the flow acceleration shown in Figure 4 will be approximately 6.5×10^6. It is in this regard that the FTF possesses real advantages over ground based wind tunnel experiments.

For the geometry shown in Figure 3 equation (4) applies and one notes that for a given pressure distribution on the FTF surface a range of K values follow depending upon the chord Reynolds number and hence, the flight speed of the FTF platform. This seems questionable physically and again calls into question the relevance of K as an indicator of relaminarization for high lift flows. It will be of great interest to document whether K is indeed the proper correlating parameter for occurrence of relaminarization on the FTF platform.

It is important to point out that the flight envelope of the FTF platform allows relaminarization experiments to be performed over the range of Reynolds and Mach numbers relevant for high lift systems.

The objectives of the relaminarization experiments are to address the following questions:

• For the Reynolds numbers and external pressure gradients characteristic of high lift flows, is K a reliable criterion for occurrence of relaminarization? Can a more suitable scaling parameter involving boundary layer characteristics be developed from the experimental results?

• What is the effect of Reynolds and Mach number on relaminarization?

• What is the characteristic response time of the turbulent boundary layer relaminarization process ? How long must the flow be exposed to the imposed flow acceleration for relaminarization to occur?

• Is the nature of the relaminarization process observed in flight different from that observed in low speed academic wind tunnels? What is the character of the boundary layer during relaminarization?

• Based upon the FTF flight experiments, is relaminarization a likely player in leading edge flow physics for actual high - lift systems? If so, how can it be modeled?

b. High Lift System Building Block Flow 2: The Confluent Boundary Layer

1. Concerning Confluent Boundary Layers

Despite the important role played by confluent boundary layers in high lift system performance, our understanding of the physics associated with this flow is actually quite limited, especially at high Reynolds numbers.

Experimental work on confluent boundary layers generally falls into two broad categories. There have been studies on specific 2-D high lift configuration models in both low speed and pressure tunnels. These studies have typically been carried out as part of a high lift system configuration optimization and performance testing program. As such, the focus is often not on the confluent layer itself but rather the resulting integrated aerodynamic forces. Measurements of the confluent boundary layer typically involve total head tube surveys which provide mean velocity profiles. In addition, such studies include effects attributable not only to the confluence itself but also pressure gradient and streamwise curvature.

More fundamental studies of confluent boundary layers have been performed and reported in a series of papers by the group at Cambridge University (Zhou and Squire (1983, 1985), Agoropoulos and Squire (1988) and Moghadam and Squire(1989)). These experiments have primarily examined the interaction between a wake generated by either a flat plate or symmetric airfoil and the neighboring wind tunnel wall boundary layer. In each case both the airfoil and tunnel wall boundary layers were artificially tripped to produce turbulent flow. Such studies have shown that the level of turbulence in the wake has a very strong influence on the wake / boundary layer interaction. In cases where there is strong vortex shedding from the wake generating airfoil, the mixing in the interacting flow is found to be quite strong. The resulting confluent boundary layer is much thicker than the turbulent boundary layer would be in the absence of the upstream wake-generating body. Cases such as these also presented the greatest difficulties in computations since they involved counter-gradient momentum transport which violates standard eddy viscosity-based turbulence models. That is, in the initial region of the interaction, the shear stress and mean velocity gradient normal to the wall can have opposite sign which implies that the effective eddy viscosity is negative! Obviously such flows cannot be modeled with any type of standard eddy viscosity model. Further, the counter gradient transport tends to occur in the initial stages of the wake/boundary layer interaction which has the effect of "contaminating" numerical solutions obtained via streamwise marching methods. Even in regions without counter-gradient transport, the nature of the interaction is quite complex and the model constants are not known a priori.

18

Experiments have also shown that the effect of an adverse pressure gradient on the wake/boundary layer interaction is to accelerate the thickening of the confluent layer. This will obviously have important implications for high lift systems.

The confluent boundary layer "model flows" described above simulate many of the essential features of the leading edge confluent boundary layer characterizing commercial high lift systems. However, because the experiments described above were performed in low speed academic wind tunnels, the Reynolds number based on the chord of the wake generating body was quite low; typically in the range of 0.2 - 0.3 million. This may be compared, for example, to the Reynolds number based on slat chord for the Boeing 737-100 which is nominally in the range of 1.8 - 3.4 million with a corresponding Mach number range of 0.2 - 0.33. Flight test provides an ideal opportunity to address the effect of Reynolds number and Mach number on confluent boundary layer structure, thereby complimenting results from ground based experiments.

2. In -Flight High Reynolds Number Confluent Boundary Layer Experiments Using the FTF

The experiment using the FTF will provide measurements of the effect of Reynolds and Mach number on confluent boundary layer structure in the absence of surface curvature. The geometry is similar to that employed in the cited studies by the group at Cambridge. However, an important difference will be that the FTF experiments will be performed at significantly higher Reynolds and Mach numbers. Comparison of the resulting confluent boundary layer flight data with complimentary measurements from low speed wind tunnel facilities will allow an objective assessment of the effects of Reynolds number and Mach number on the nature of wake - boundary layer interactions. There certainly appears to be a general consensus that both Reynolds and Mach number effects are important. However, there seems to be very few specifics regarding in what ways the confluent boundary layer structure is likely to differ in high and low Reynolds number experiments. Comparison of low speed wind tunnel data for confluent boundary layers with FTF flight data will provide important information for modelers in assessing Reynolds and Mach number effects and incorporating them into new predictive models.

Figure 5 presents a schematic of the FTF confluent boundary layer experiment. The boundary layer on the FTF surface is artificially tripped by use of distributed roughness near the leading edge. As shown in the figure a wake generating airfoil is located off the FTF surface just downstream of the FTF leading edge. The airfoil wake will widen and eventually merge and interact with the turbulent boundary layer on the FTF surface. The

Airfoil Support
and Boundary Layer Fence

Leading Edge   FTF Platform (top view)

Instrumented FTF Surface

Wake Generating
2-D Body

Airfoil Support
and Boundary Layer Fence

Confluent Boundary Layer
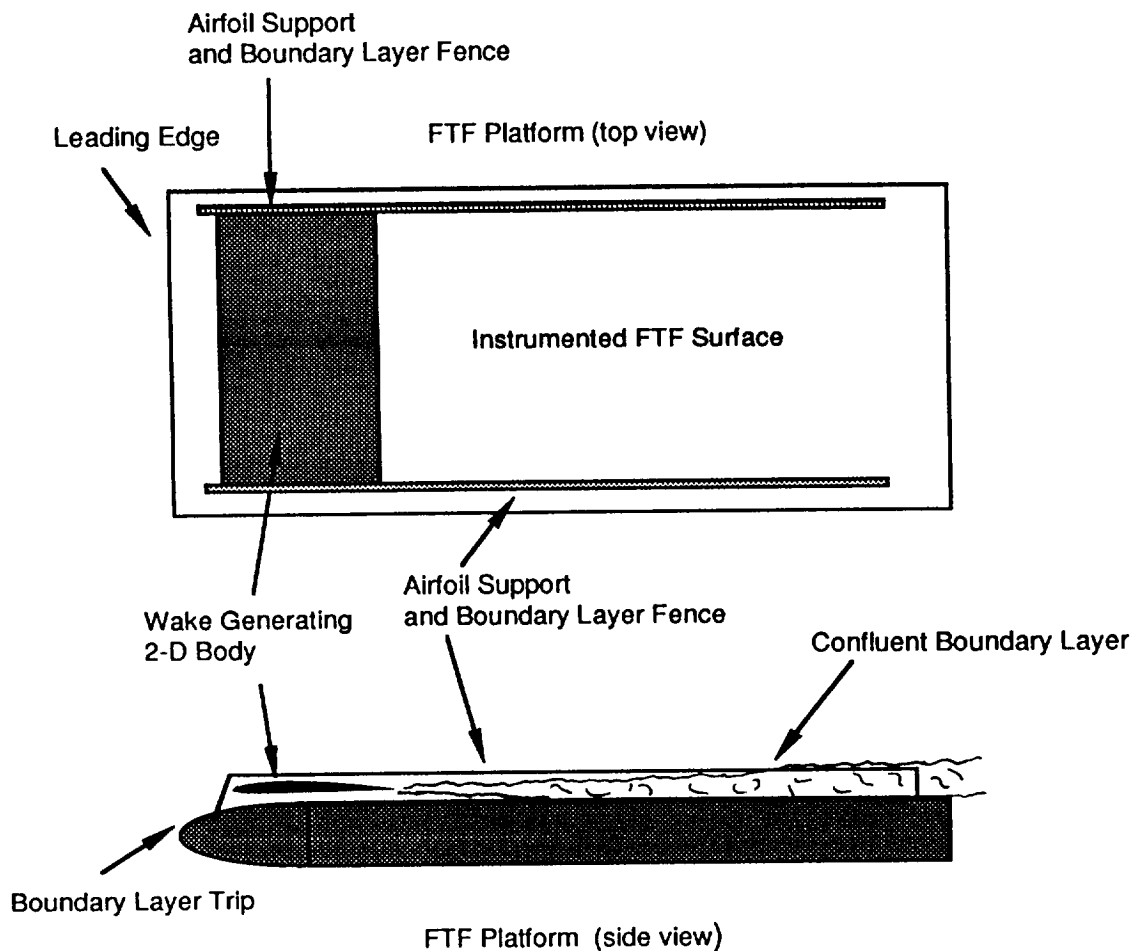
Boundary Layer Trip

FTF Platform (side view)

Figure 5. Schematic of FTF Confluent Boundary Layer Experiment

location of onset of this confluence will be determined by the wake widening and boundary layer growth. However, the vertical positioning of the wake generating body above the FTF surface provides a degree of user control over the location of confluence and insures a sufficiently long streamwise run of confluent layer for the experiment. Fences can also be used to provide a nominally 2-D confluent boundary layer interaction. Unlike the previously described in-flight experiment, the off surface airfoil in Figure 5 will not give rise to relaminarization. Computations using panel methods show that by using an airfoil of either sufficiently small thickness ratio or of the appropriate camber, the local flow acceleration on the FTF surface can be kept small.

20

Consider a wake generating body whose chord length is 20% that of the FTF platform. Assuming that it is flown at 10, 000 ft. pressure altitude at $M_\infty = 0.3$, the resulting Reynolds number based upon the chord of the wake generating body would be approximately 3×10^6. This Mach number - Reynolds number combination provides a very good match to actual high lift systems (like the NASA Boeing 737-100 Research Vehicle). An advantage of this approach over testing in an actual high lift system is the ability to decouple effects due to confluence alone from those due to streamwise curvature and pressure gradient.

By flying the appropriate altitude-flight speed combinations within the F-15B envelope, the Reynolds number and Mach number may each be varied over a range appropriate for high lift systems and the resulting effect on the confluent boundary layer structure assessed. Increased altitude for a fixed flight speed has the effect of increasing the Mach number and decreasing the Reynolds number. A fixed Reynolds number experiment in which Mach number is varied could be performed by appropriately increasing the flight speed for each selected altitude increase. Conversely, by decreasing flight speed by the appropriate amount for selected increases in cruise altitude, an experiment could be performed at a fixed Mach number and varying Reynolds number. The objective of such experiments would be to assess the structure of the confluent boundary layer as either Mach number or Reynolds number is varied over a range appropriate for high lift systems.

It is desirable to be able to investigate the interaction of both low and comparably high momentum deficit wakes with the turbulent boundary layer on the FTF surface. To this end, the geometry of the wake generating body can be varied in the experiments. Initial work, for example, might involve a low momentum deficit wake generated by a flat plate or symmetric airfoil with small thickness ratio. Higher momentum deficits would be investigated by using appropriately cambered airfoils of larger thickness ratio.

In recognition of the fact that the slat wake in a high lift system typically exhibits a high degree of cross-stream asymmetry, it is desirable to be able to control the degree of initial wake symmetry. In this manner the interaction of symmetric and asymmetric wakes with the FTF boundary layer could be investigated. This could be accomplished by using boundary layer control in the form of trips placed on one side of the wake generating body. Use of a trip on one side of the wake generating body would give rise to a disparity in top and bottom momentum thicknesses at the trailing edge, thereby leading to an initially asymmetric wake.

In order to characterize the confluent boundary layer for a particular Reynolds and Mach number combination, it is desirable to document the streamwise evolution of the

mean velocity profiles on the FTF surface. These mean profiles are essential for establishing both the location of onset of confluence as well as the streamwise growth of the confluent layer. The profiles would be used to obtain key integral parameters such as momentum thickness, $\theta(x)$, displacement thickness, $\delta^*(x)$ and shape factor H. Mean velocity profile measurements can be made off surface with hot-film probe or pitot tube rakes. The use of X-film probes would also allow both the normal and Reynolds stress variation through the confluent layer to be documented. These measurements are important for advancing turbulence models for confluent layers. For example, the local turbulence production term, $-\overline{uv}\,\partial\overline{U}/\partial y$, can be obtained from measurement of both Reynolds stress and mean velocity profiles. Another required quantity in terms of characterizing the confluent boundary layer structure is the local skin friction coefficient. This can be obtained by performing Preston tube measurements at selected streamwise locations.

c. High Lift Building Block Flow 3: Wake Widening and Structure in Arbitrary Pressure Gradient

1. Relevance to High Lift System Performance

A key parameter in the performance of a high-lift system involves the wake development from upstream elements. For example, the growth rate of the slat wake will determine, in part, the location of onset of confluence with the main element boundary layer. Even in cases where there is no strong confluence on the main element, the slat wake will have the effect of moderating the surface pressure peak on the trailing flap(s). Flap pressure peak moderation helps maintain flow attachment and improves C_{Lmax}. The degree of flap surface pressure peak moderation is related directly to the wake width. In general, the thicker the wake, the more the flap pressure peak is moderated. This is due to an associated streamline displacement effect. Consider, for example, Figure 6 which shows the effect of the main element wake widening rate on downstream flap element surface pressures. The wake widening was arbitrarily manipulated in the computation and serves to demonstrate the important effect wake growth rate can have on the aft element surface pressure distributions.

Another example of the profound influence of wake widening on high-lift performance is shown in Figure 7, which is taken from Lin, Robinson and McGhee (1992). This figure shows the lift coefficient variation with angle of attack for a two-dimensional single-flap, three-element, high-lift system at stowed chord Reynolds numbers of 5 X 10^6 and 9 X 10^6. A spanwise array of sub-boundary layer-scale surface vortex generators positioned near the quarter-chord position of the trailing flap was used to maintain attached flow over the flap. Figure 7 shows that the benefit of this is apparent only at moderate angles
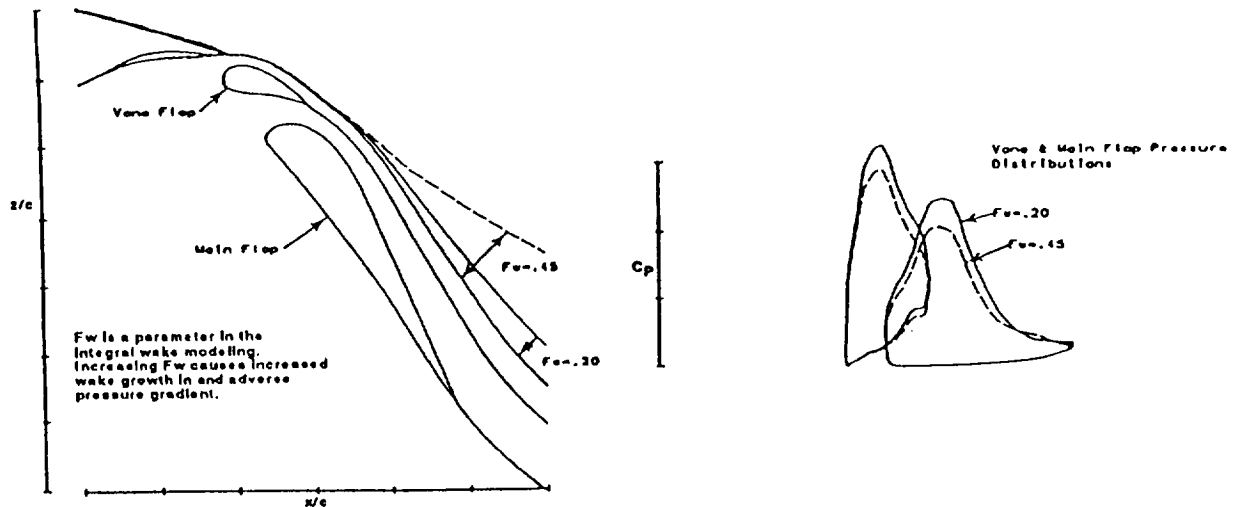
Figure 6. Effect of Wake Widening on Flap Surface Pressure Moderation
(from Garner, Meredith and Stoner, 1991)

of attack where lift is clearly augmented. Flow attachment is maintained at high angles of attack even without the use of vortex generators. This is due to the fact that at the highest angles of attack prior to stall, the main element wake thickens appreciably and has the effect of suppressing the trailing flap surface pressure peak. As a consequence the flap boundary layer flow remains attached. This serves as an example of the somewhat surprising and non-intuitive behavior encountered in high lift systems as a consequence of the viscous flows involved.
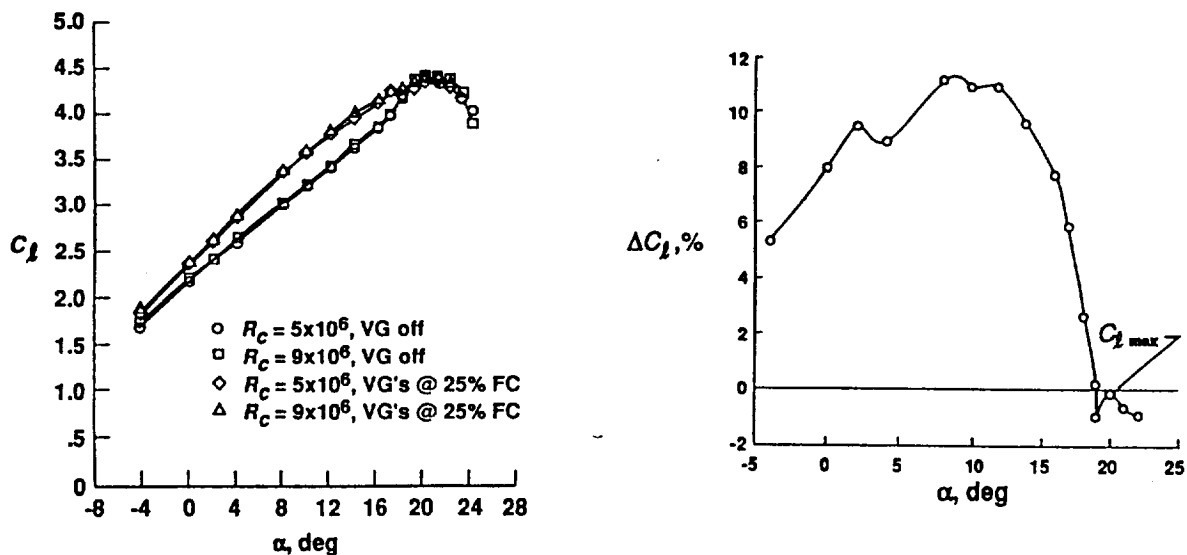

Figure 7. Maintenance of Flap Flow Attachment at High α Due to Wake Widening
(from Lin, Robinson and McGhee, 1992)

23

In addition to issues related directly to wake widening, Smith (1975) notes that off surface flow reversal can occur if the wake encounters a sufficiently strong adverse pressure gradient. This has been termed by some as "wake bursting". Indeed some measurements from the LTPT (e.g. Chin et al, 1993) show wake profiles over the trailing edge flap of a Douglas three element airfoil that appear very close to exhibiting off surface reversal.

2. In -Flight Experiments on Wake Widening in Pressure Gradient.

Unlike the often studied case of the symmetric wake in zero pressure gradient for which the underlying structure and model constants are well known, our understanding of the physics of wakes which develop in arbitrary pressure gradients is poor. Due to its importance in high-lift systems, we must be able to reliably compute wake growth under arbitrary pressure gradient conditions. Wake development in high-lift systems will likely depend upon the following factors (and possibly their interaction):(1) pressure gradient, (2) the degree of initial wake asymmetry, (3) streamline curvature and (4) unsteady effects.

It is desirable to form joint ground based and in-flight experiments whose goal it is to address the following questions:

• What is the structure and growth rate of wakes under arbitrary pressure gradient conditions? How is wake widening related to the imposed streamwise pressure gradient dP / dx?

• What role does initial wake asymmetry play in wake growth? Do symmetric and asymmetric wakes develop differently in a given adverse pressure gradient condition? If so, how and why?

• How do Reynolds number and Mach number effects affect wake growth and structure?

• Does off surface flow reversal play a significant roll in high-lift systems?

Figure 8 presents a schematic of one possible arrangement for wind tunnel experiments to investigate wake structure in pressure gradient as envisioned by the authors. The 2-D wake experiments would be performed in low speed wind tunnel facilities. By using the adjustable tunnel wall contour shown in Figure 8, the boundary layer development on side 1 of the wake generating plate can be controlled through the imposed pressure gradient. As shown in the figure, when the boundary layer momentum thicknesses on opposite sides of the plate at the trailing edge are disparate, $\theta_1 \neq \theta_2$, an initially asymmetric wake is generated which will subsequently interact with the adverse

pressure gradient in the diverging section. Conversely, by relaxing the side 1 wind tunnel wall, $\theta_1 = \theta_2$, and the wake is initially symmetric. In this manner the role of adverse pressure gradient on both symmetric and asymmetric wakes can be established. The degree of initial asymmetry would be selected to approximate that observed in slat wakes on actual high-lift systems. By adjustment of the diverging section sidewalls shown in Figure 8, the wake may be exposed to a variety of adverse pressure gradients of varying strengths. In this manner the response of both the wake's mean flow structure and turbulence quantities to the imposed pressure field will be determined. Flow diagnostics would involve detailed two component LDA traverses. The resulting LDA database would be particularly useful for Reynolds averaged Navier-Stokes modeling efforts since it provides not only benchmark data for model validation, but also more fundamental quantities like local turbulence production, mixing lengths, as well as terms in the equation for the wake energy budget. Similarly, by utilizing particle image velocimetry (PIV) techniques, experiments can be performed which will suggest how the essential aspects of wake unsteadiness can be incorporated into current models. For example, by application of the Proper Orthogonal Decomposition to the PIV results, one may find that the essential aspects of the unsteady character of the wake is captured by only a few low order modes. Such a result would have important implications for modeling.
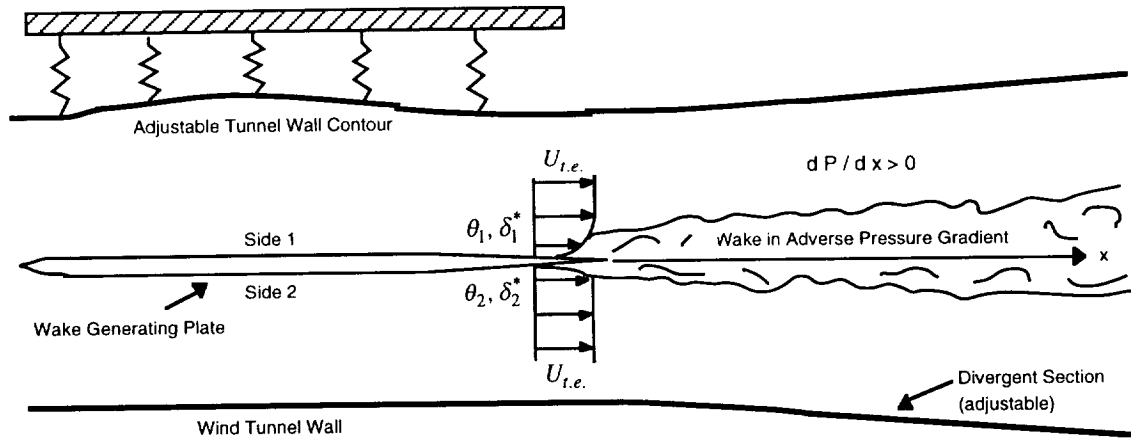

Figure 8. Wind Tunnel Experiment to Study Wake in Pressure Gradient

In order to address Reynolds and Mach number scaling issues related to wake development in pressure gradient, coordinated in-flight experiments using the NASA Dryden FTF would be used. A schematic of the flight test experiment is shown in Figure 9. In this arrangement, a 2-D wake generating body is positioned off the FTF surface as in

the confluent boundary layer experiment. One important difference is that in this experiment the distance between the wake generating body and the FTF surface is sufficiently large so that no confluence with the boundary layer on the FTF surface is allowed to take place. This may be facilitated by not tripping the FTF surface boundary layer as in the previous experiments and perhaps by use of passive laminar boundary layer control on the FTF surface. It is the focus of this experiment to examine the growth of the upstream airfoil wake in response to variations in pressure gradient. The pressure
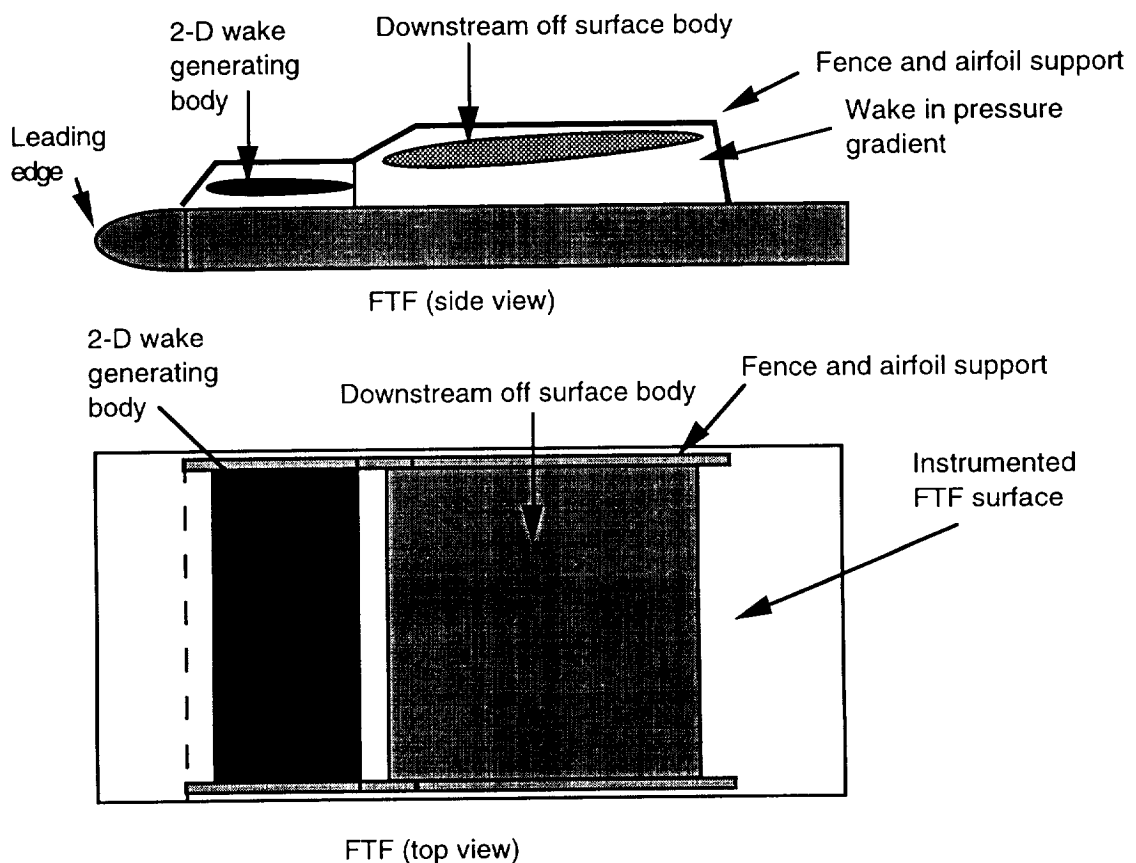
Figure 9. Schematic of In-Flight Wake in Pressure Gradient Experiment

gradient is imposed by a customized airfoil contour also mounted above the FTF surface but farther downstream. As the wake from the upstream airfoil flows between the FTF surface and the downstream airfoil contour it will first encounter a favorable and then adverse pressure gradient (as does the slat wake of a high lift system). For a given geometry of the downstream off surface body, the pressure gradient environment may be controlled by adjusting the orientation (i.e. angle of attack) of the downstream body. In order to minimize lateral loads on the FTF platform a dummy airfoil could be placed on the opposite side of the FTF in order to preserve symmetry.

Since the primary focus of the experiment is to document Reynolds and Mach number effects on wake widening in pressure gradient environments, the wake width will be monitored at preselected streamwise locations, particularly in the adverse pressure gradient region. It is possible to do this intrusively using, for example, pitot tube rakes or hot-film probe rakes. An alternate approach might be to use laser imaging similar to that employed in flight test experiments involving transverse gas jets by Karagozian et al. (1996). In this technique, nitrogen gas is seeded with iodine vapor for visualization purposes. The N_2 - I_2 mixture would be passed through a flow injection slit on the surface of the wake generating body. A sheet of laser light from a pulsed diode pumped Nd:YAG laser at 532.25 nm wavelength would then be used to illuminate the wake. Images of the wake may then be captured by a high speed gated video camera synchronized to the laser pulse rate. Provided that a convenient length scale is included in the video frames, the wake widening could then be obtained from video images.

IV. Summary and Recommendations

The development of a useful CFD-based design tool for high lift systems requires increased attention to unresolved flow physics issues. Only through an improved understanding of "high lift building block flows" which are an integral part of the commercial high lift system can a rational design strategy be developed. In this report we have considered a broad spectrum of key flow field physics issues relevant to the design of improved high lift systems. This report demonstrates how in-flight experiments utilizing the NASA Dryden FTF carried on an F-15B support aircraft can provide a novel method by which both Reynolds and Mach number effects associated with specific high lift building block flows can be investigated. In-flight experiments like the three examples described in this report provide a cost effective and, to this point, completely unexplored way of improving our understanding of high lift system flow physics issues at realistic Reynolds and Mach numbers. In many ways this "flying wind tunnel" approach appears more attractive than performing similar experiments in pressure and/or cryogenic

wind tunnels. These in-flight high lift building block flow experiments will be most effective when performed in conjunction with coordinated ground based wind tunnel experiments in low speed facilities. This unified approach seems to be an expeditious way to advance state-of-the-art predictive capability for high lift system design.

It is recommended that a joint wind tunnel / flight test partnership between NASA Dryden Flight Research Center and Hessert Center for Aerospace Research at Notre Dame be established in order to address unresolved issues pertaining to high-lift building block flows. Although any of the building block flows are amenable to this approach, we believe that each of the example experiments outlined in this report are particularly capable of yielding a high payoff from the proposed joint wind tunnel / flight test approach. One of these high lift building block flows should be selected for investigation based on joint discussions between NASA Dryden and Notre Dame personnel in a phase 2 effort. A primary objective of the phase 2 research effort would be to demonstrate the utility of the joint wind tunnel / flight test approach in advancing state-of-the-art high lift aerodynamics. For the particular high lift building block flow selected, this research would involve the detailed design and implementation of a joint wind tunnel and flight test program. The wind tunnel experiments would be carried out in facilities at Notre Dame and coordinated in-flight experiments using the FTF platform would be performed at NASA Dryden Flight Research Center. The flight test program would be used to address key Reynolds and Mach number scaling issues related to the selected building block flow. The wind tunnel experiments would be used to provide complimentary
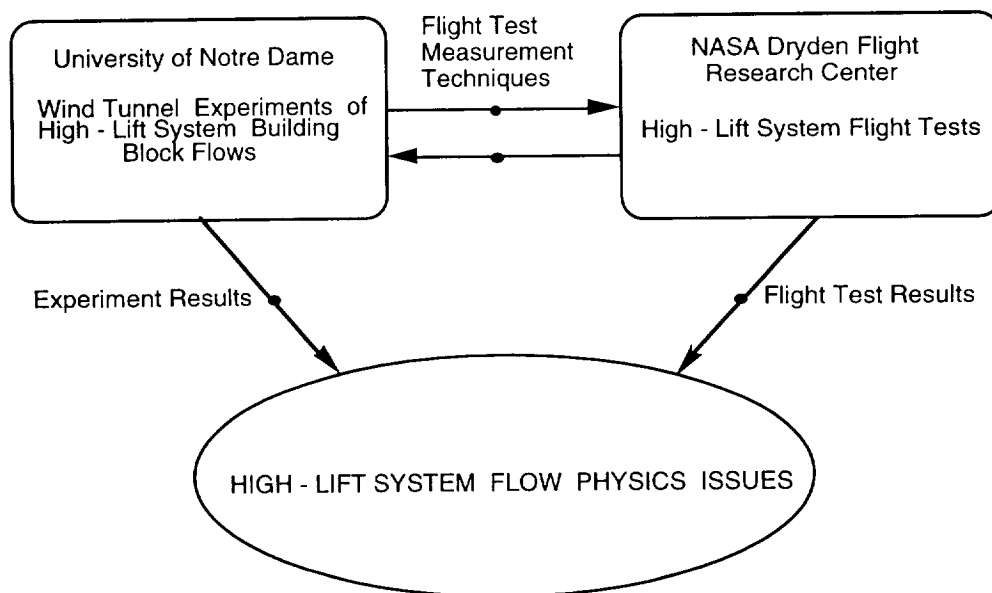

Figure 10. Schematic of Proposed Research Partnership

28

benchmark LDA and PIV data. The wind tunnel work would also serve to support NASA Dryden in the development of improved measurement techniques and diagnostics for the FTF experiments. An illustration of the envisioned NASA Dryden / Notre Dame high lift partnership is shown in Figure 10.

V. References

"Aeronautical Technologies for the Twenty-First Century", 1992, National Academy Press, Washington, D. C.

Agoropoulos, D. and Squire, L.C., 1988, "Interactions Between Turbulent Wakes and Boundary Layers, "AIAA Journal, 26,10, pp. 1194-1200.

Chin, V.D., Peters, D.W., Spaid, F.W. and McGhee, R.J., 1993, "Flowfield Measurements About a Multi-Element Airfoil and High Reynolds Numbers," AIAA paper 93-3137.

Chin, V.D., Peters, D.W., Spaid, F.W. and McGhee, R.J., 1993, "Flowfield Measurements About a Multi-Element Airfoil and High Reynolds Numbers," AIAA paper 93-3137.

Garner, P. L., Meredith, P. T., and Stoner, R. C., 1991, "Areas for CFD Development as Illustrated by Transport Aircraft Applications," AIAA paper 91-1527.

Garner, P. L., Meredith, P. T., and Stoner, R. C., 1991, "Areas for CFD Development as Illustrated by Transport Aircraft Applications," AIAA paper 91-1527.

Karagozian, A.R., Wang, K.C. , Le, A.T. , and Smith, O.I. , 1996, "Modeling and In-Flight Imaging of Transverse Gas Jets Injected Behind a Rearward-Facing Step, AIAA Paper 96-0914.

Launder, B. E. and Jones, W. P. (1969) " On the Prediction of Laminarisation," Great Britain Aeronautical Research Council, CP 1036.

Lin, J.C., Robinson, S.K., and McGhee, R.J., 1992, "Separation Control on High Reynolds Number Multi-Element Airfoils",AIAA paper 92-2636.

Mack, M.D., and McMasters, J. H., 1992, "High Reynolds Number Testing in Support of Transport Airplane Development," AIAA   paper 92-3982.

Moghadam, A., and Squire, L.C.,1989, "The Mixing of Three-Dimensional Turbulent Wakes and Boundary Layers," Aeronautical Journal, pp. 153-161.

Narasimha, R. and Sreenivasan, K. R., 1973, "Relaminarization in Highly Accelerated Turbulent Boundary Layers," J. Fluid Mech., 61, pp. 417-447.

Patel, V.C. and Head, M. R., 1968,"Reversion of Turbulent to Laminar Flow," J. Fluid Mech., 34,2, pp. 371-392.

Petrov, A. V., 1978, "Certain Types of Separated Flow Over Slotted Wings," Fluid Mechanics-Soviet Research, 7,5, pp. 80-89.

Poll, D. I. A., 1979, "Transition in the Infinite Swept Attachment Line Boundary Layer," Aeronautical Quarterly, 30, pp. 607-629

Smith, A.M.O., 1975, "High-Lift Aerodynamics," AIAA Journal, 12, 6, pp. 501-530.

Thomas, F. O., 1995, "Flow Physics of Subsonic High-Lift Systems," in notes from *An Overview of High Lift Aerodynamics*, AIAA Short Course , June 23-24, San Diego, CA.

van Dam, C.P., Vijgen, P.M.H.W., Yip, L.P., and Potter,R.C., 1993, "Leading Edge Transition and Relaminarization Phenomena on a Subsonic High-Lift System," AIAA paper 93-3140.

Zhou, M. D. and Squire, L. C., 1983, "The Interaction of a Wake with a Boundary Layer," in *Structure of Complex Turbulent Shear Flow*, R. Dumas and L. Fulachier, Ed., Springer-Verlag, New York.

Zhou, M. D. and Squire, L. C., 1985, "The Interaction of a Wake with a Turbulent Boundary Layer," Aeronautical Journal, pp. 72-81.