ICAS Paper No. 70-47

PROBLEMS ON INLETS AND NOZZLES

by
D. Zonars, Chief Scientist
Air Force Flight Dynamics Laboratory
Air Force Systems Command
Wright-Patterson AFB, Ohio, USA

The Seventh Congress of the International Council of the Aeronautical Sciences

CONSIGLIO NAZIONALE DELLE RICERCHE, ROMA, ITALY / SEPTEMBER 14-18, 1970

Price: 400 Lire

PROBLEMS ON INLETS AND NOZZLES

D. Zonars

Air Force Flight Dynamics Laboratory
U.S. Air Force Systems Command

Abstract

The wide range of Mach number and altitude requirements placed on high performance aircraft has resulted in a number of inlet and nozzle problems reflecting adversely upon flight performance and maneuverability. These circumstances have emphasized the need for greater understanding of the airframe induced flow fields and how these fields interact with inlet and nozzle systems. Inlet-airframe interactions involving local flow angularities and Mach number effects measured about a wing-body wind tunnel model at transonic and supersonic speeds are presented. Also, the influence of internal inlet flow turbulence is assessed since this phenomenon can contribute significantly to engine instability during supersonic flight. The nozzle portion of the paper examines the drag losses associated mainly with twin-jet configurations. Attendant variations in lateral nozzle spacing, aft-end fineness ratio and nozzle pressure ratio are evaluated as a function of transonic Mach numbers.

I. Introduction

Historically, the engine inlet and nozzle have played a secondary role in the design and development of aircraft. However, recent flight vehicle operational experience has shown the need for proper integration of the airframe and propulsion systems to achieve trouble free and effective flight performance. Specifically, engine compressor stalls have been associated with complex, distorted inlet flow fields, while less than desired aircraft range characteristics have resulted from excessive aft-end drag. Development emphasis of the inlet and nozzle systems cannot be overlooked since these components are of primary importance in the thrust producing mechanism for transonic and supersonic flight. The details of inlet and nozzle systems involve major geometrical variations which must function efficiently in a complex, changing flow environment dependent upon Mach number and aircraft orientation. These circumstances have emphasized the need for greater understanding of the airframe induced flow fields and how these fields interact with inlet and nozzle systems.

II. Inlet Systems

In the past the inlet and the engine have been developed on a component basis. Emphasis was placed on the inlet system to generate the proper pressure recovery with an acceptable steady state distortion. The experience factor of current day aircraft clearly indicates that flight vehicle performance including stability and control must be treated on an integrated basis with due consideration for the large variations of inlet airflows. Experience has also taught us that a substantial similarity exists between the characteristics of the captured flow and the resultant flow to the compressor face. Since the inlet operates in an external flow environment which is strongly dependent upon the shape of the airframe, it behooves engineers to examine such influences and sensitivities of inlets to local flow angularities and nonuniformities of the oncoming flow.

In recent years a considerable amount of wind tunnel work has been published concerning inlet systems.[1-9] More recently, the Air Force Flight Dynamics Laboratory has undertaken a number of programs to investigate flows about fuselage and fuselage-wing configurations throughout the subsonic, transonic and supersonic speed regimes. The objectives of these programs are to develop a clearer understanding of inlet-airframe interactions. A typical effort is reported herein with an ultimate goal of developing an experimental data bank and a corresponding analytical approach for assessing the flight performance characteristics of present day and future flight vehicles.

The basic wind tunnel model system employed in one of these studies is shown in Figure 1. Testing was accomplished on a family of 7 fuselage configurations including a wing stub which was varied in 2 sweep positions. Representative inlets were also tested ahead of and under the wing. The approximate one-twelfth scale model system was tested at Mach numbers of 0.8, 1.35, 1.8, 2.2 and 2.5 throughout an angle of attack range varying from -3 to 24 degrees. In addition sideslip angles were varied from -4 to +4 degrees.

The flow field survey system utilized in the wind tunnel tests, also shown in Figure 1, consisted of 3 conical pitot-static pressure probes mounted on a remotely actuated drive system. Each probe was 0.125 inches in diameter with a 40 degree included cone angle. The probes consisted of a pitot orifice in the nose along with 4 pressure taps placed equidistantly around the cone surface for determining local flow angularities.

FIGURE 1 WIND TUNNEL MODEL AND FLOW FIELD SURVEY SYSTEM.

The model and survey systems described above were utilized to develop the external flow field in terms of local values of flow direction and Mach number. Data presented herein treats the case of inlet location under the simultaneous influence of the fuselage and wing. Although this representative data is for one specific configuration, a number of variations in nose droop, canopy, and fuselage cross-section were examined with and without wing glove influences. Figure 2 shows such characteristics for a free stream Mach number of 0.8 and body angles of attack of 10, 15, and 19 degrees. The vectors show the local flow directions in the cross-plane with the length of vector denoting the magnitude of the angle relative to the free-stream. A reference length equivalent of 20 degrees local flow magnitude is shown inside the representative body-wing shape for comparative analysis. For the most part Figure 2 shows the cross-flow pattern one would expect with increasing angle of attack including the obvious influence of the wing section. The flow pattern shown for 19 degrees angle of attack also suggests the development of some secondary flow at the body-wing intersection. Constant Mach number profiles are also presented in the same figure. For all three (3) angles of attack, the local Mach number profiles show a goodly amount of similarity. There is very little cross-flow acceleration which is interpreted as the result of the decelerative influence caused by the wing. As might be expected, the wing section serves to turn and align the oncoming flow to the inlet and thus aid in the compression assignment required of the inlet.

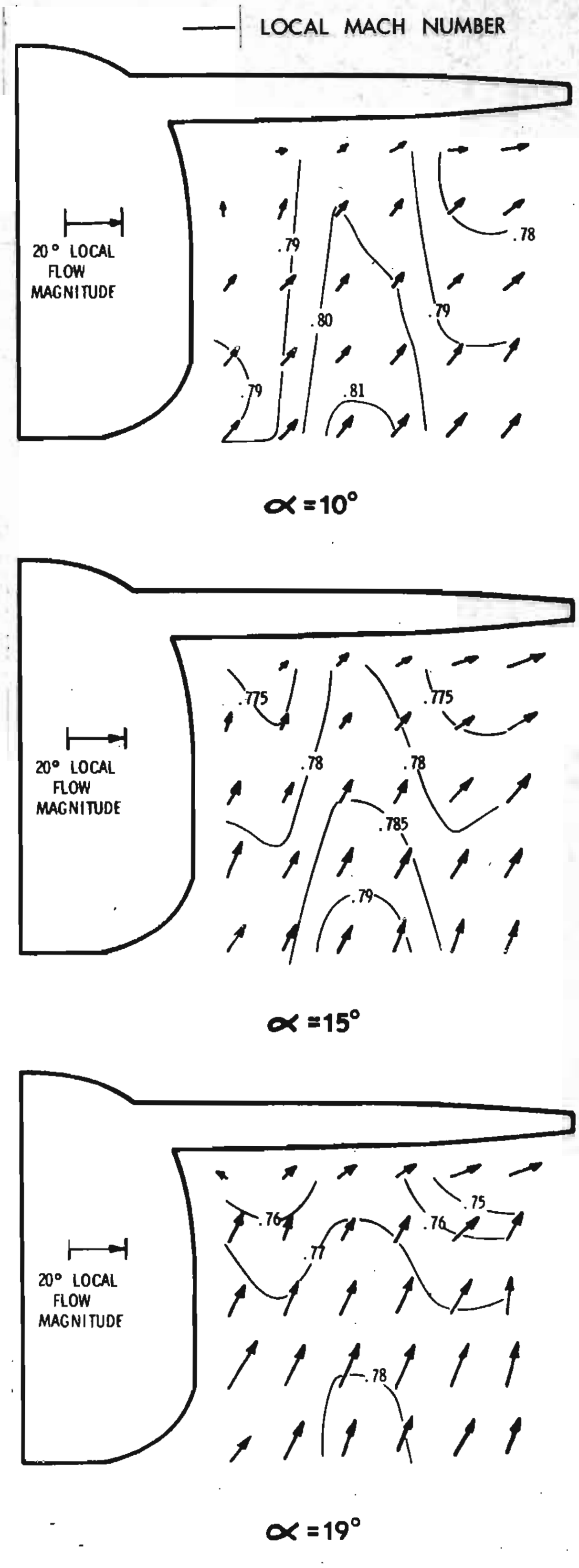

FIGURE 2 LOCAL FLOW CHARACTERISTICS: MACH NUMBER = 0.8, ANGLE OF ATTACK = 10°, 15° AND 19°.

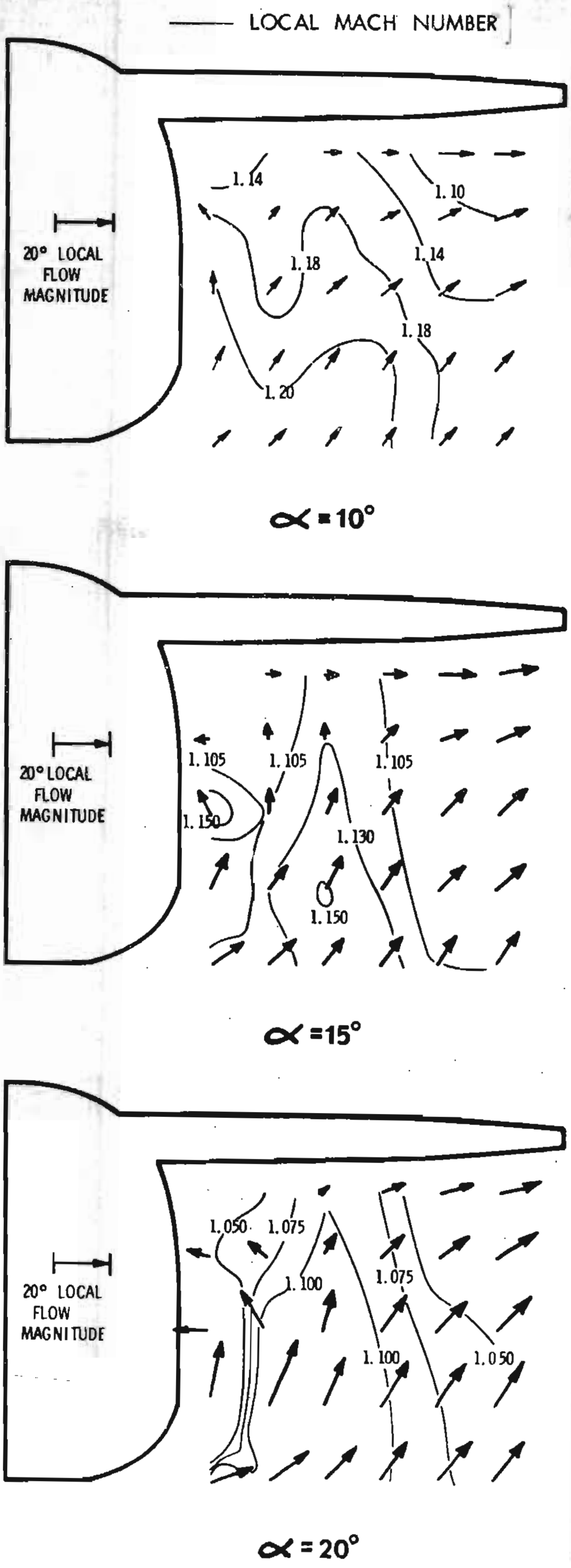

FIGURE 3 LOCAL FLOW CHARACTERISTICS: MACH NUMBER = 1.35, ANGLE OF ATTACK = 10°, 15° AND 20°.

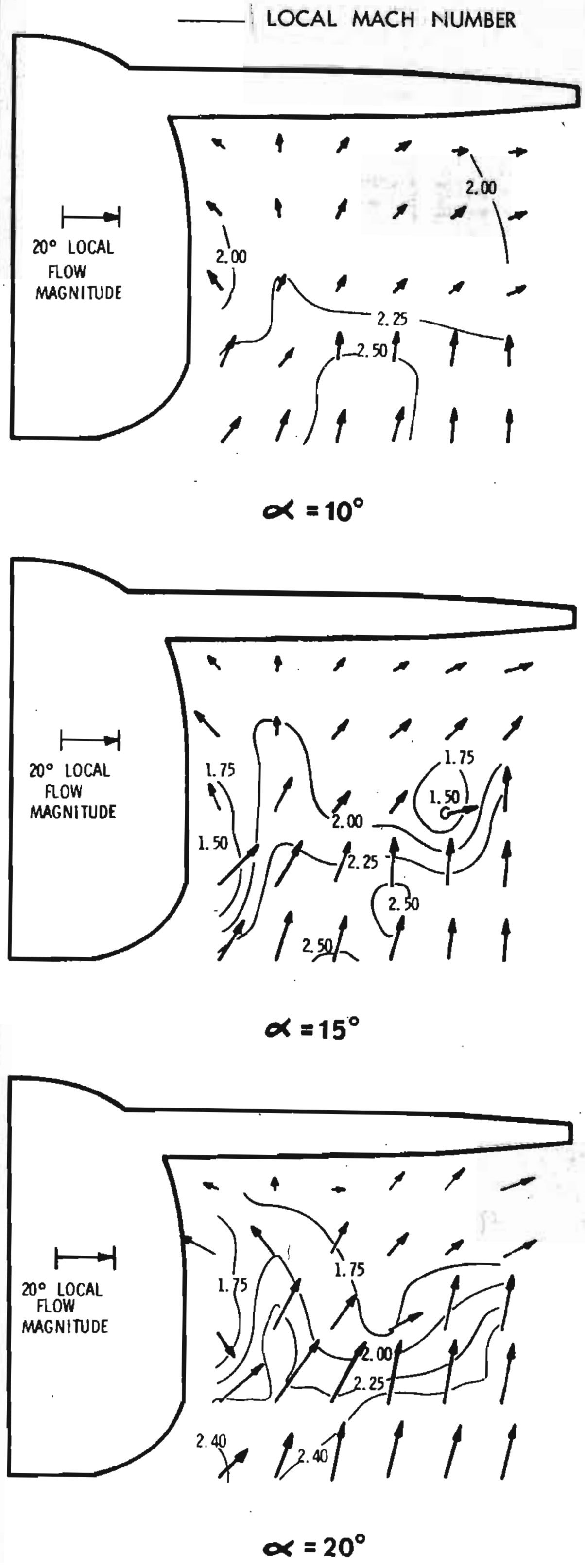

FIGURE 4 LOCAL FLOW CHARACTERISTICS: MACH NUMBER = 2.50, ANGLE OF ATTACK = 10°, 15° AND 20°.

Figure 3 shows the local flow characteristics about the same body-wing combination for Mach number 1.35. The flow field near the clipped portion of the wing tip displayed the same characteristics as shown for Mach number 0.8. However, the flow field adjacent to the body is considerably different in that there is a strong tendency for the flow to split, depending on angle of attack, and to develop into a vortex pattern. Here again, the Mach number is reduced by approximately 0.3 as the flow nears the underside of the wing.

The effects of increasing Mach number to 2.5 is shown in Figure 4. There is a strong tendency for the flow to develop in a very similar fashion as was experienced at Mach number 1.35. Flow splitting effects appear to prevail at a lower angle of attack with the development of vortical flow next to the fuselage. In addition, another flow disturbance is identified in the outer portion of the flow field. This is first seen at an angle of attack of 15 degrees. Attendant with this condition is a substantial reduction in Mach number with an outward flow tendency. This disturbance tends to shift slightly in-board as the angle of attack is raised to 20 degrees.

The wing-body combination reported above was considered one of the more desirable configurations for inlet location since the inlet entrance was shielded by the lower wing surface. In addition, the wing provided a compression of the on-coming supersonic flow and consequently reduced the capture area required of the inlet. Subsonic flow characteristics were as might be predicted, with good wing shielding characteristics. However, the beneficial effects expected supersonically were somewhat degraded due to the development of the vortex adjacent to the body. This vortex development was due to the shockwave-boundary layer interaction brought on by the conical shockwave system emanating from the wing leading edge intersection with the body. The magnitude of this vortex and the degree to which it extended into the inviscid inlet flow field was substantially greater than the boundary layer thickness. This phenomenon and the associated problem of properly positioning inlets relative to the airframe must be given serious consideration by future aircraft designers.

III. Inlet Pressure Fluctuations - Engine Instability

Inlet steady-state distortion resulting from both external and internal flow conditions has been one of the primary causes of propulsion instabilities. The solution to this problem has been achieved through ground test techniques in which screens, flow deflectors, etc. have been used to simulate loss of total pressure head to the engine. Many combinations of circumferential and radial patterns have been investigated to determine the effects of these flow irregularities on engine stability and performance. However, with the advent of supersonic flight, the unsteady nature of inlet flows has had a profound effect in reducing the engine operational stability margin and consequently causing compressor stalls. The primary source of these inlet flow pulsations or fluctuations has been identified as shock-boundary layer interactions and flow separations. These fluctuations are generally random in both time and space and are often referred to as "turbulence".

Early experimental investigations on inlet "turbulence" were performed at the NASA Lewis Research Center[10] and the Air Force Arnold Engineering Center[11]. In these two studies, turbojet engines operating at static conditions were subjected to fluctuating inlet flows. Tests showed a reduction in compressor surge margin due to the "turbulence" encountered. Subsequent to these two basic studies, a limited number of investigations[12-16] have been carried out in order to shed light on this problem area. A typical example of the more recent programs was reported by Plourde and Brimelow[17]. In this effort a fan and low pressure compressor of the Pratt and Whitney Aircraft TF30 turbofan engine was selected as the test article to study the effects of "turbulence" on engine stall margin. Figure 5 shows a schematic of the TF30 3-stage fan and 6-stage low pressure compressor system. The forward section of the compressor was connected to a "turbulence" generator duct similar to the convergent-divergent device developed by Kimsey[18].

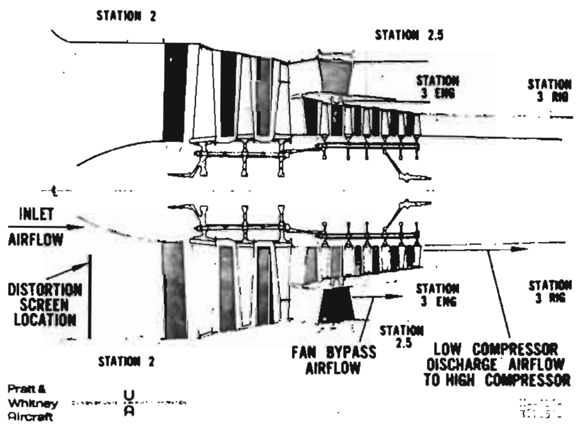

FIGURE 5 TF30 FAN AND LOW PRESSURE COMPRESSOR SYSTEM.

Figure 6 shows a cutaway of this "turbulence" generator which included a movable plug centerbody followed by a constant area duct. The purpose of this plug was to develop a sonic throat followed by supersonic flow and a normal shock system. The interaction of the shockwave with the duct boundary layer generated the fluctuating or "turbulent flow" conditions. The "turbulence" generator included a section just ahead of the compressor wherein a variety of stream obstructions such as 1/2-inch rods or 3-inch pipes could be placed in front of the compressor face to further increase or change the "turbulence" spectrum. Figure 7 shows the 3-inch rod system.

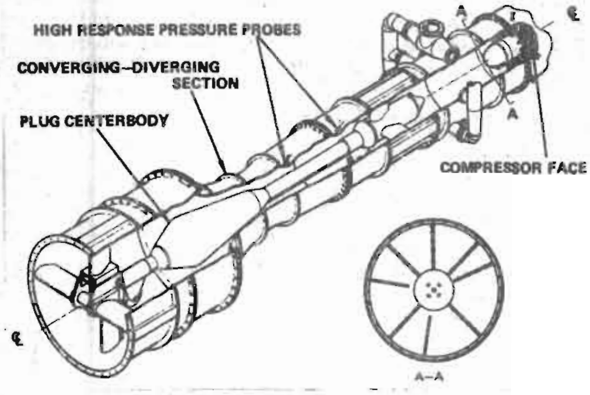

FIGURE 6 "TURBULENCE" GENERATOR.

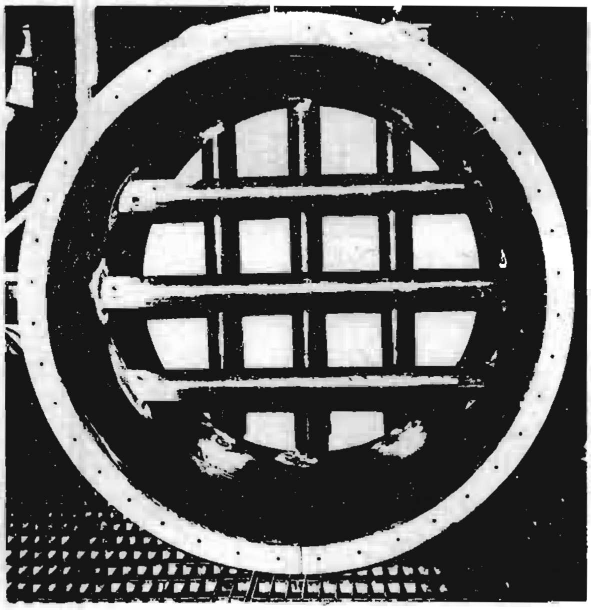

FIGURE 7 TWO ROW GRILL OF THREE-INCH DIAMETER PIPE.

High frequency response pressure transducers were used to measure both static and total pressures at the compressor face. A typical inlet rake and total pressure probe utilizing Kistler transducers and low frequency response sensing tubes is shown in Figure 8. These total pressure rakes were positioned around the compressor inlet at 0°, 45°, 135°, 225°, 292.5°, and 315° when facing upstream.

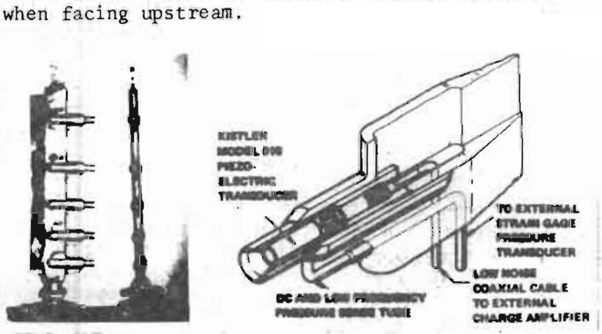

FIGURE 8 HIGH RESPONSE INLET RAKE AND TYPICAL TOTAL PRESSURE PROBE.

The power spectral densities resulting from the turbulence generators along with the spectra produced by the 1/2-inch and 3-inch rods is shown in Figure 9. The installation of the 1/2-inch grill system generated a fairly flat spectrum over the entire compressor face. The spectrum established from the 3-inch rods contained discrete frequencies as a result of shed vortices which were not yet dissipated to small scale "turbulence".

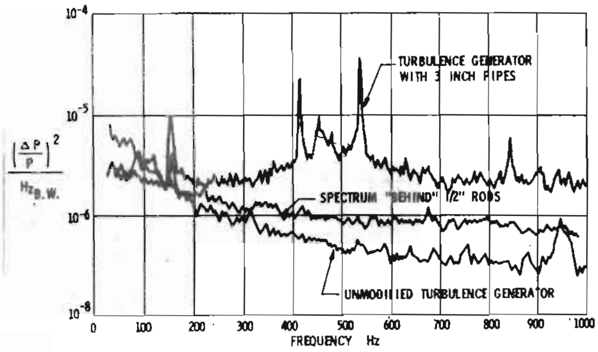

FIGURE 9 POWER SPECTRAL DENSITY COMPARISON.

The effects of unsteady flow on compressor performance is now assessed. Gabriel, Wallner, and Lubick[10] first showed that a sinusoidal varying plane flow displayed detrimental effects on the compressor stall characteristics. In addition, their analog simulation of a turbo-jet axial flow compressor utilizing volumetric dynamics and steady-state total pressure air flow relationships was sufficient to establish the unsteady flow characteristics through an engine. A comparison of the analytical procedure with experiment showed excellent agreement. Now, the effects of "turbulence" can be described by an instantaneous spatial pressure distortion which is a function of pressure variation in amplitude and geometric location of the peak to peak pressure regions over the compressor face. The effects of this "turbulence" on the compressor performance, is shown in Figure 10. Base line characteristics for the compressor performance were determined from bellmouth tests and are so indicated. The unmodified "turbulence" generator characteristics (in other words, without the 1/2" and 3" pipe or other grill installations) shows a reduction in primary airflow, and more importantly a reduction in the stall line. Figure 10 also shows the effects of 1/4-inch and 3-inch pipe installations. As expected, further decreases in primary flow were experienced along with some reduction in the operating stall line. Figure 11 shows the loss of compressor surge line as a function of "turbulence" level intensity. Here it is clearly shown that compressor stall is related to instantaneous spatial distortion.

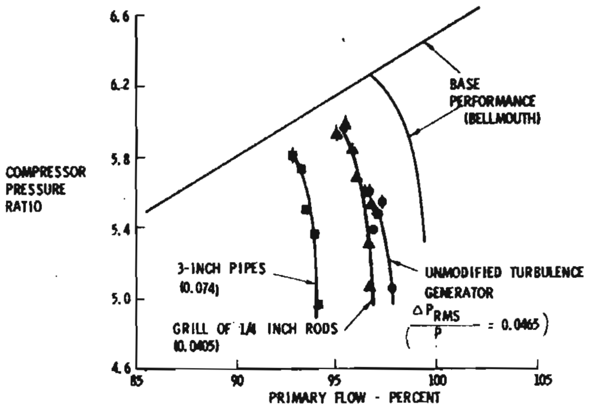

FIGURE 10 FAN LOW PRESSURE COMPRESSOR PERFORMANCE.

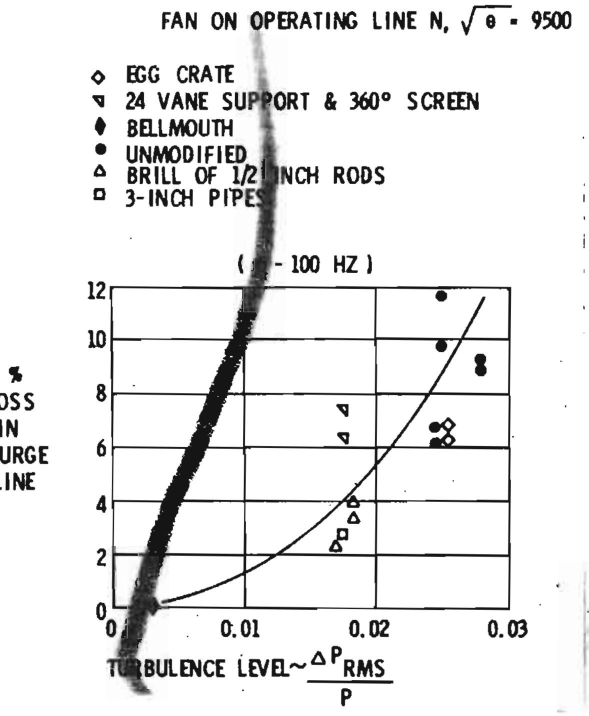

FIGURE 11 SURGE LINE REDUCTION VS "TURBULENCE" LEVEL.

Today, there are a number of different theories advanced by numerous investigators to predict compressor stall. One effective method of describing the phenomenon is an instantaneous spatial distortion pattern. That is, although the time dependence of the fluctuations at a point is important, this effect may be approximated by the instantaneous spatial variation. With this assumption, the description of "turbulence" reduces to a weighted spatial integration producing an instantaneous distortion parameter which can be used to correlate the effects of "turbulence". The instantaneous distortion parameter K_θ[17] is expressed as follows:

$$K_\theta = \frac{\sum_{i=1}^{i} \left[\left(\frac{A_n}{n^2} \right)_{max} \right]_i \frac{P_{t_{av}}}{Q_{av}} \frac{1}{D_i}}{\sum_{i=1}^{i} \frac{1}{D_i}}$$

where

i = number of pressure instrumented ring

D = diameter of the pressure instrumented ring

Q_{av} = average inlet velocity head at compressor face

$$A_n = \sqrt{a_n^2 + b_n^2}$$

θ = circumferential angle

$$a_n = \frac{1}{\pi} \int_{-\pi}^{\pi} \frac{P_t}{P_{t_{av}}} (\theta) \cos n\theta \, d\theta$$

$$b_n = \frac{1}{\pi} \int_{-\pi}^{\pi} \frac{P_t}{P_{t_{av}}} (\theta) \sin n\theta \, d\theta$$

with

$$\frac{P_t}{P_{t_{av}}} = 1 + \sum_{n=1}^{\infty} a_1 \cos \theta + a_2 \cos 2\theta + \dots a_n \cos n\theta + b_1 \sin \theta + b_2 \sin 2\theta + \dots b_n \sin n\theta$$

and

P_t = impact pressure

$P_{t_{av}}$ = average impact pressure

Correlation of the computed instantaneous circumferential distortion parameter with experiment is shown in Figure 12. This figure shows reasonably good agreement between computation and the various configurations employed to develop "turbulence".

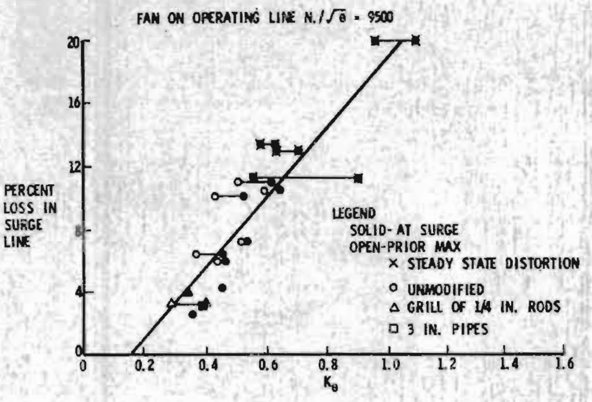

FIGURE 12 SURGE LINE LOSS VERSUS INSTANTANEOUS SPATIAL DISTORTION.

Recently, Burcham and Hughes[19] have modified and utilized the Pratt and Whitney K_{DA} distortion factor for predicting surge. The engine compressor face was sub-divided into 5 equal-areas through concentric circles or rings. Probes were placed on rings which were maintained at a constant radii from the compressor centerline. The modified K_{DA} distortion parameter was defined as follows:

$$K_{DM} = \frac{\frac{1}{2}\sum_{i=1}^{5}\left[\frac{P_{t_{max}} - P_{t_{min}}}{P_{t_{av}}}\right]_i \Theta_i^- \; C_i}{\sum_{i=1}^{5} C_i} \times 100$$

where

C = ratio of compressor inlet radius to ring radius

i = number of ring

Θ^- = largest continuous arc of the ring over which the total pressure is below the ring average pressure

$P_{t_{max}}$ = ring maximum total pressure

$P_{t_{av}}$ = ring average pressure

$P_{t_{min}}$ = ring minimum total pressure

This modified distortion parameter was found to be 80% effective in identifying surge when engine stall occurred within approximately 90% of the maximum steady state distortion value. Needless to say, more exacting methods must be developed to predict engine instability due to dynamic inlet conditions. For the time being, the loss of compressor stability margin can be attributed to random inlet pressure fluctuations which vary in amplitude and location and which are within the frequency sensitivity range of a compressor.

IV. Nozzles

The high performance characteristics of modern day aircraft require substantial propulsion jet area variations through the transonic and supersonic flight regimes. A minimum jet area is required for subsonic speeds; however, boat-tail effects become an important consideration since there is a large amount of aft-facing aircraft area involved. The greatest aggravation in this respect is flow separation wherein the aircraft aft section is engulfed in unsteady and low surface pressures associated with the flow separation point. Increasing the flight speed to transonic values requires higher nozzle exhaust pressure ratios. This results in under-expanded nozzle flow plumes which can be beneficial in the development of a recompression flow field acting on the aft-facing projected area, or, the recompression can serve to cause flow separation on the same surface and thereby induce significant losses due to drag. Further speed increases to supersonic flight velocities tend to improve matters slightly providing there is no after-body flow separation. Figure 13 shows the range of convergent-divergent nozzle performance as a function of Mach number. The transonic portion of the flight regime should be noted with interest since the nozzle characteristics are least efficient near Mach number unity. This situation, in conjunction with aircraft drag rise, becomes a serious problem area for vehicles designed to fly in this speed regime.

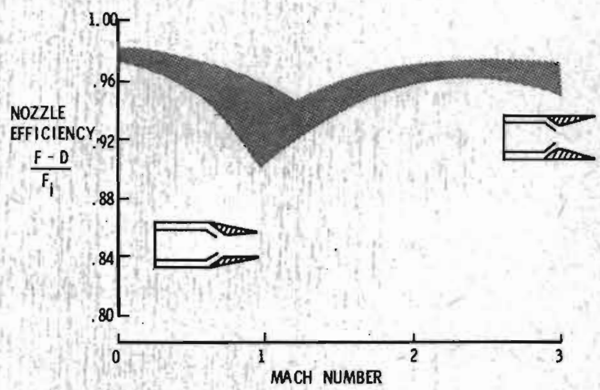

FIGURE 13 CONVERGENT-DIVERGENT NOZZLE PERFORMANCE CHARACTERISTICS.

The transonic flight regime has always been one of great analytical complexity. Even the simplest of practical flight configurations has not been amenable to any reasonable analytical exactness. Hence, one must turn to ground and flight test means of the past[20-23] to investigate all flight configurations prior to development. Although this has been the only practical recourse in the past, one must constantly be aware of the fact that such configurations have suffered some degree of data inexactness due to low Reynolds number properties of present day wind tunnels along with blockage effects when testing very near the speed of sound.

Many experimental nozzle programs have been performed to assess the aircraft performance sensitivities resulting from the integration of the airframe and nozzle. This situation is a compromise between nonuniform external flow generated by body and tail and the variable geometry requirements set up by nozzle area ratios. Isolated, internal nozzle performance characteristics are fairly well understood today. However, the influence of an irregular external flow shrouding the aft aircraft section has been very difficult to determine analytically[24]. To a greater extent, the characteristics of a twin-jet or multiple exhaust nozzle system is virtually impossible to correctly assess. A number of comprehensive boattail studies on multi-exhaust nozzle systems have been conducted by Runckel[25] in order to assess the interference effects between nozzles. Figure 14 shows the results of such interference effects. It is observed that the employment of a closely spaced multi-exhaust nozzle system further reduces the flight performance efficiency.

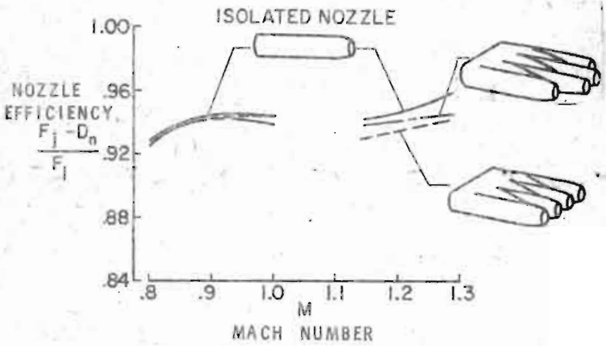

FIGURE 14 PERFORMANCE CHARACTERISTICS OF MULTIPLE NOZZLE SYSTEM COMPARED TO ISOLATED NOZZLE.

Wilcox, Samanich and Blaha[26] have recently reported on nozzle installation effects of an F-106B aircraft which was modified to test two underwing nacelles containing J85-GE-13 afterburning turbojet engines. One of the objectives of this program was to compare isolated, cold flow nozzle data from wind tunnel tests with flight characteristics on various nozzle configurations. Figure 15 shows the cold flow isolated nozzle installed in the NASA Lewis Research Center 8 by 6-Foot Supersonic Wind Tunnel. The flight data as compared to the isolated cold flow wind tunnel data for a nozzle system with and without rounding at the boattail juncture is shown in Figure 16. Classical transonic drag rise effects are shown here with lesser boattail pressure drag observed on the flight article than measured on the isolated nozzle system. In addition, a substantial subsonic boattail drag reduction was achieved by merely rounding the juncture point. This data shows conclusively that one can not rely upon isolated nozzle flow characteristics to quantitatively predict flight nozzle performance. A strong dependence exists on nozzle configuration in conjunction with airframe interference effects.

FIGURE 15 COLD-FLOW ISOLATED NOZZLE IN NASA 8 X 6 FOOT SUPERSONIC WIND TUNNEL.

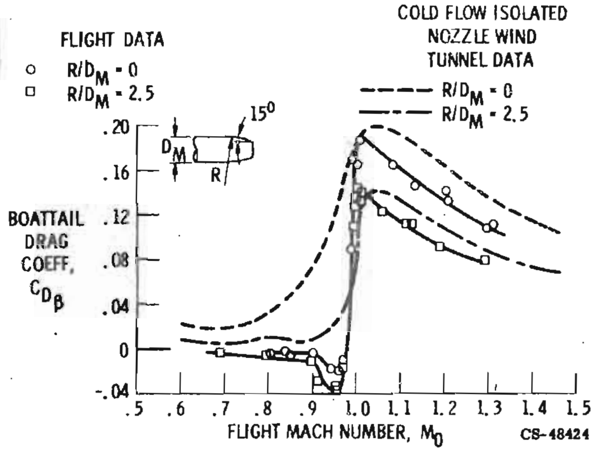

FIGURE 16 COMPARISON BETWEEN FLIGHT AND ISOLATED WIND TUNNEL NOZZLE DATA INCLUDING EFFECT OF AFTERBODY SHAPE.

Development of exhaust nozzle technology for application to various emerging advanced aircraft designs is lagging behind induction system studies. Comprehensive studies are needed to fill technological gaps involving viscous and divergence losses, effects of nozzle operation on vehicle stability and control, and many other parametric variations for optimizing aircraft performance. Swavely[27] recently undertook a study to systematically determine the aft-end performance characteristics of a twin-jet configuration. Figure 17 shows the specific configuration examined for variations in Mach number and nozzle pressure ratio. Tests of this model in the United Aircraft Research Laboratories 8-Foot Wind Tunnel involved afterbody and nozzle thrust minus drag measurements, boundary layer characteristics at the metric plane and static pressure distributions for equivalent bodies of revolution.

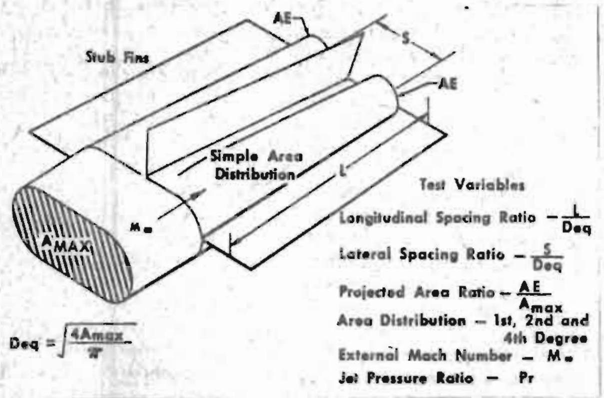

FIGURE 17 TWIN-JET CONFIGURATION AND TEST VARIABLES.

Figure 18 shows the aft-end drag characteristics at Mach number 0.7, a nozzle pressure ratio of 2.5, lateral spacing ratio of 0.61 and 1.10 and for indicated ranges of AE/A_{max} and L/D_{eq}. The carpet plot shows that increasing the lateral spacing ratio decreases the aft end drag while decreasing the longitudinal spacing ratio also decreases the drag. The need for a shortened length of longitudinal run clearly indicates the effect of reduced wetted area and hence a lower skin friction drag contribution.

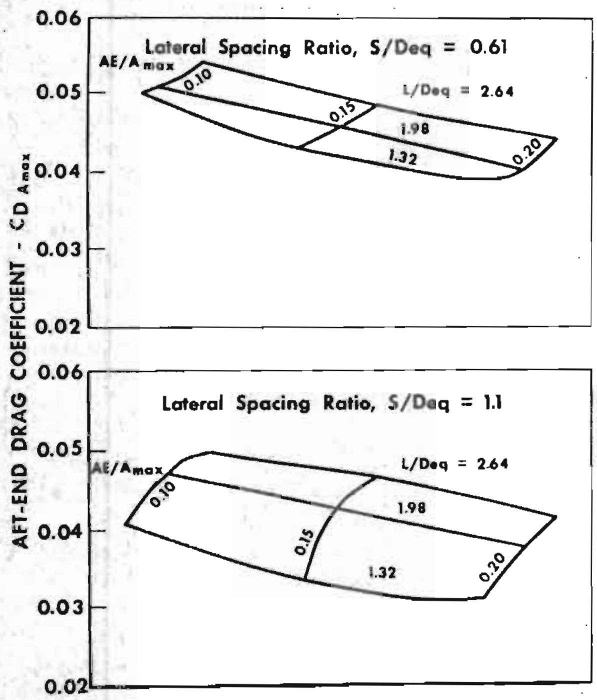

FIGURE 18 AFT-END DRAG CHARACTERISTICS: MACH NUMBER = 0.7, EXHAUST NOZZLE PRESSURE RATIO = 2.5.

The same type of plot is shown in Figure 19 for Mach number 0.9. The beneficial effects of increased lateral spacing are similar as experienced at Mach number 0.7. However, a dilemma arises in that inverse longitudinal spacing characteristics are experienced. This fact is undoubtedly associated with the rate at which transonic flows can be recompressed as a function of effective slenderness ratio. Also, an increase in skin friction at Mach number 0.9 contributes to higher drag for the increased length.

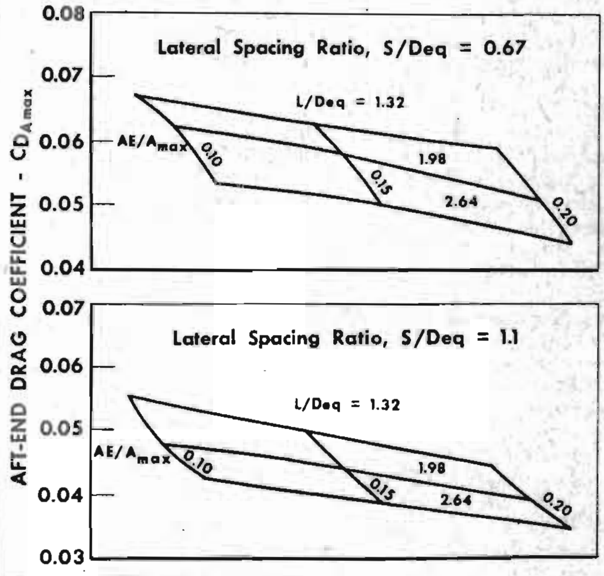

FIGURE 19 AFT-END DRAG CHARACTERISTICS: MACH NUMBER = 0.9, EXHAUST NOZZLE PRESSURE RATIO = 2.5.

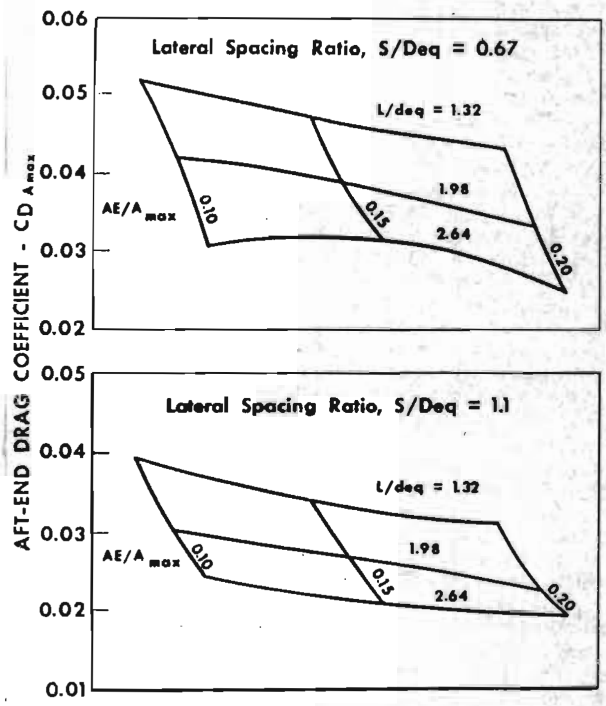

FIGURE 20 AFT-END DRAG CHARACTERISTICS: MACH NUMBER = 0.9, EXHAUST NOZZLE PRESSURE RATIO = 6.0.

Increasing the nozzle jet pressure ratio from 2.5 to 6.0 at Mach number 0.9 serves to pressurize the aft section of the configuration as shown in Figure 20. This would be as expected since the pluming effects of the jet would tend to establish a decelerating flow field which acts on the aft-facing body surfaces and thus provides a reduced drag condition.

V. Discussions and Conclusions

Airframe-propulsion compatibility has become a critical problem area for both commercial and military high performance aircraft. Classically the solution to overcoming the problem of compressor stall has been through reduction of the pressure distortion generated by the inlet and increased distortion tolerance of the engine. Intensive efforts are presently underway in ground and flight test facilities to understand the effects of coupled steady-state and dynamic inlet distortion. Also, considerable research is being directed toward the cause and effect relationship of non-uniform flow fields entering the inlet system of turbo jet engines. Many of these flow field examinations show local angles of attack and yaw which far exceed aircraft attitude values. Inlet designers are presently faced with a very difficult task to match inlet geometry with the large variations in flow conditions developed about many reasonable airframe geometries. Flow field studies will continue on many airframe configurations to determine optimum inlet positioning.

The effects of "turbulence", or specifically, the fluctuating nature of the measured total pressures at the compressor face have been found to have a strong influence on the stall margin of most engines. This phenomenon normally commences at low supersonic speeds with increasing disturbance intensity as a function of increasing Mach number. This "turbulence" exhibits a wide range of amplitude-frequency content. For low frequency disturbances, engine performance is basically similar to steady-state operation since the engine outlet pressures will follow the inlet flow variations in magnitude and phase, such that overall compressor pressure ratio will remain the same. However, the majority of time dependent total pressure fluctuations are found to be significantly faster than the aforementioned flow properties. Under these circumstances, outlet pressures lag the inlet pressure variations in both amplitude and phase. Consequently, the pressure ratio across the compressor can differ considerably from a steady-state value, and conditions can develop wherein compressor stall margin is completely negated. For years the use of a frequently referred to "turbulence factor", $(\Delta P_t)_{rms}/Pt_{ave}$, averaged over the compressor face has raised many doubts concerning its usefulness. The results of the study presented in this paper clearly indicates that instantaneous spatial distortion calculations are necessary to judge the performance characteristics of combined inlet and engine. Future couplings between inlets and engines must account for the dynamic or time dependent characteristics of the ducted flow as caused by a number of physical phenomenon such as shock wave-boundary layer interaction and flow separation. The development of small-scale powered simulators for wind tunnel use can be very beneficial in establishing inlet-engine compatibility.

The exhaust nozzle of military and commercial airplanes alike is a critical airframe-propulsive component utilized in attaining the desired aircraft performance. Many nozzle systems have been studied to insure compatibility with advanced turbo-jet and turbo-fan engines; however, the influence of external air flow upon the performance characteristics of these nozzles has received little attention. In this paper an effort was made to show the primary problem in installation effects and exhaust nozzle integration with a twin-jet aircraft. It was found that closely spaced nozzles interfere with each other, and at times, with surrounding tail surfaces. Also, it was observed that extreme care must be taken in integrating the nozzle system with the aircraft during wind tunnel tests in order to minimize installation effects. New test techniques utilizing small powered engines with combustion and correct exhaust gas temperatures appear necessary in order to evaluate nozzle and engine installation effects. Although generalized techniques for predicting the aft-end performance characteristics have been developed for the correlation of experimental twin-jet data, there are still some inherent limitations in the test methods utilized. More extensive model and flight test data are needed to improve our experimental techniques and to establish appropriate analytical models. A concentrated effort involving Reynolds number effects on interactions between the airframe and the nozzle system must be pursued. Model data can certainly be used to check analytical procedures; however, in-flight tests are necessary to establish the sensitivities of boundary layer development, shock-boundary layer interactions, divergence losses and external flow non-uniformities.

References

1. Sorenson, N.E., Cubbison, R.W., and Smeltzer, D.B., "Study of a Family of Supersonic Inlet Systems," AIAA Preprint 68-580 (1968).

2. Chun, K.S., "A Control System Concept and Substantiation Test for an Axisymmetric Mixed Compression Supersonic Inlet," AIAA Preprint 68-581 (1968).

3. Richey, G.K., Stava, D.J., Surber, L.E., and Zonars, D., "F-111A Inlet-Engine Compatibility Investigation," AIAA 4th Propusion Joint Specialist Conference, (June 1968).

4. Anderson, W.E. and Wong, N.D., "A Two-Dimensional, Mixed Compression Inlet System Designed to Self-Restart at a Mach Number of 3.5," AIAA Preprint 69-447 (1969).

5. Hancock, J.P. and Hinson, B.L., "Inlet Development for the Lockheed 500," AIAA Preprint 69-448 (1969).

6. MacMiller, C.J., "Investigation of Subsonic Duct Distortion," AIAA Preprint 69-449 (1969).

7. Callahan, G.M. and Stenning, A.H., "Attenuation of the Inlet Flow Distortion Upstream of Axial Flow Compressors," AIAA Preprint 69-485 (1969).

8. Rall, F.T., "Aircraft and Propulsion Operational Considerations Related to Inlet Design," AGARD Presentation to the Flight Mechanics Panel, Goettingen, Germany, (September 1967).

9. Daniel, B.R., "Wind Tunnel Investigations of the F-111A and FB-111 Inlet at Mach Numbers from 0.75 to 1.20," AEDC TR 66-118, (July 1966).

10. Gabriel, D.S., Wallner, L.E., and Lubrick, R.J., "Some Effects of Transients in Inlet Pressures and Temperatures on a Turbojet Engine," Institute of the Aeronautical Sciences Twenty-Fifth Annual Meeting, (January 1957).

11. Kimzey, W.F. and Lewis, R.J., "An Experimental Investigation of the Effects of Shock-Induced Turbulent In-Flow on a Turbojet Engine," AIAA Second Propulsion Joint Specialists' Conference, (June 1966).

12. Martin, A.W. and Kostin, L.C., "Propulsion System Dynamic Test Results," North American Aviation, Inc., Report No. NA-67-386, (April 1967).

13. Winslow, L.J., et. al., "Inlet Distortion Investigation, Upstream Engine Influence and Screen Simulation," The Boeing Company, Technical Report AFAPL-TR-68-140, (January 1969).

14. Wasserbauer, J.F. and Willoh, R.G., "Experimental and Analytical Investigation of the Dynamic Response of a Supersonic Mixed-Compression Inlet," AIAA Preprint 68-651 (1968).

15. Martin, A. and Kostin, L., "Dynamic Distortion at the Exit of a Subsonic Diffuser of a Mixed Compression Inlet," North American Rockwell, Report TFD-69-588 (1969).

16. Oates, G.C., Sherman, D.A., Motycka, D.L., "Experimental Study of Inlet-Generated Pressure Fluctuations," PWA Report 3682 (1969).

17. Plourde, G.A. and Brimelow, B., "Pressure Fluctuations Cause Compressor Instability," Air Force Airframe/Propulsion Compatibility Symposium, (June 1969).

18. Kimzey, W.F., "An Investigation and Calibration of a Device for the Generation of Turbulent Flow at the Inlet of a Turbojet Engine," AEDC TR 65-195, (October 1965).

19. Burcham, F.W., Jr. and Hughes, D.L., "Analysis of In-Flight Pressure Fluctuations Leading to Engine Compressor Surge in an F-111A Airplane for Mach Number to 2.17," AIAA Preprint 70-624 (1970).

20. Mikkelson, D.C. and Blaha, B.J., "Flight Investigation of Installation Effects on Underwing Supersonic Cruise Exhaust Nozzles at Transonic Speeds," AGARD Aerodynamic Interference Specialists' Meeting, Washington, D.C., (September 1970)

21. Byrd, K.F., "Experimental Investigation of Twin Jet Aircraft/Exhaust Nozzle Installation Effects," AIAA Preprint 69-429 (1969).

22. Migdal, D., Miller, E.H., and Schnell, W.C., "An Experimental Evaluation of Exhaust Nozzle/Airframe Interference," AIAA Preprint 69-430 (1969).

23. Samanich, N.E. and Burley, R.R., "Flight Performance of Auxiliary Inlet Ejector and Plug Nozzle at Transonic Speeds," AIAA Preprint 70-701 (1970).

24. Laughrey, J.A., "Calculation of the Performance of Installed Exhaust Nozzles on Supersonic Aircraft," AIAA Preprint 69-428 (1969).

25. Runckel, J., "Aerodynamic Interference between Exhaust System and Airframe," AGARD Aerodynamic Interference Specialists' Meeting, Washington, D.C., (September 1970).

26. Wilcox, F.A., Samanich, N.E., and Blaha, B.J., "Flight and Wind-Tunnel Investigation of Installation Effects on Supersonic Cruise Exhaust Nozzles at Transonic Speeds," AIAA Preprint 69-427 (1969).

27. Swavely, C.E., "Twin-Jet Aircraft Aft-Fuselage Performance Prediction," Air Force Airframe/Propulsion Compatibility Symposium, (June 1969).