Propulsion System Installation Design for High-Speed Prop-Fans

B.H. Little Jr.*
Lockheed-Georgia Company, Marietta, Georgia

The prop-fan propulsion system has great potential for fuel savings in high-subsonic-speed cruise. Major problems barring the acceptance of this concept—demonstration of structural integrity and noise reduction—are being attacked along several lines, and attention is now swinging to design of the propulsion installation that will permit the prop-fan to realize its potential. This paper presents a survey of the problems in prop-fan installation design, discusses the tools available to deal with these problems, and points to areas where additional work is needed. The work is based on design studies for a prop-fan aircraft.

Introduction

IT has been almost 30 years since a new commercial transport was designed to fly with a turboprop propulsion system. Two things have recently happened, however, to renew interest in the turboprop: fuel costs have skyrocketed, and the prop-fan has been demonstrated in wind tunnel tests to operate at cruise speeds competitive with turbofan-powered aircraft and efficiencies almost 20% higher.

Programs are currently underway to clear the major obstacles barring acceptance of the prop-fan as a viable propulsion system. NASA's proposed testbed program[1] will demonstrate structural integrity of large-scale prop-fan hardware and measure prop-fan noise in a realistic flight environment. The testbed program will not, however, provide a good vehicle for optimizing installed propulsive efficiency, since it must start with an "off-the-shelf" engine, gearbox, and airframe and put them together in the best way possible. Other programs are needed to address specifically the problems of propulsion installation and provide the technology base for system optimization.

In the years since the last turboprop propulsion system was designed, advances have been made in all related technologies. It is appropriate therefore that prop-fan installation design be approached without preconceived ideas about what constitutes the best configuration. Furthermore, the characteristics of the prop-fan system are such that an optimized design will only be reached by close collaboration between engine and airframe manufacturers.

The need for collaboration between engine and airframe designers stems from the existence of the speed reduction gearbox as a necessary part of the system. The gearbox may have a frontal area as great as, or greater than, that of the gas generator and may weigh as much as the gas generator. The gearbox is pivotal; it plays a part in determining the mechanical efficiency of the system, and it affects the shape of the air inlet and diffuser, which in turn affect the shape of the nacelle.

Reference 2 presents prop-fan propulsion system design from the standpoint of an engine manufacturer, while this paper addresses the same general subject from the standpoint of an airframe manufacturer. The authors have coöperated in an attempt to make the two papers complementary.

To discuss prop-fan installation design in a completely general sense, allowing for both tractor and pusher arrangements and for the several places on the aircraft where the propulsion system might be mounted, is too extensive a task to be addressed here. Attention is focused, therefore, on tractor propellers and wing-mounted nacelles.

The focus of attention is also confined primarily to those areas of installation design most impacted by the interaction between engine and airframe manufacturer: design of the forebody and design of the inlet and diffuser duct. This constraint leaves some important areas of installation design, such as nacelle/wing interaction, for discussion in future papers.

Engine/Gearbox Arrangement

Two general arrangements are possible for the engine/gearbox assembly: in-line and offset. As the names imply, and as illustrated in Fig. 1, the in-line arrangement is characterized by the engine, gearbox, and propeller lying on a common centerline; in the offset arrangement, the prop-fan may be higher or lower than the engine centerline. These options afford a degree of flexibility in positioning the bulk of the nacelle relative to the wing reference plane and have a strong impact on installation design.

There are two turboprop systems in production in the United States: the Detroit Diesel Allison DDA T 56 and the General Electric GE T 64. Both use the offset engine/gearbox arrangement, and each has been used in both pinion-high and pinion-low configurations. Figure 2 shows the DDA T 56 mounted pinion-low in the Lockheed C-130, and mounted pinion-high in the Lockheed P-3. The only in-line engine/gearbox arrangement currently in production is the Rolls-Royce Tyne.

In selecting the engine/gearbox arrangement for a new prop-fan design, trade studies must consider both a mechanical optimization and an installation optimization. The mechanical optimization must consider factors such as reliability, maintainability, gearbox efficiency, and the control system, as well as the conventional cost and weight factors. For installation optimization, trade studies must consider such factors as the need for nacelle symmetry, inlet-diffuser efficiency, and nacelle/wing interference.

For cruise at Mach 0.8, it has also been found[3] that the prop-fan performance is sensitive to the shape of the spinner and nacelle forebody. Unless this region is carefully contoured, supersonic flow can form in the passages between the blades at the spinner surface. This is obviously undesirable because of the wave drag and boundary-layer separation that might result. Guidelines for avoiding supercritical flow in this region have been developed empirically, but all of the experimental work on which these guidelines are based has used body-of-revolution nacelles. It is not known how far the

Presented as Paper 81-1649 at the AIAA Aircraft Systems and Technology Conference, Dayton, Ohio, Aug. 11-13, 1981; submitted Sept. 18, 1981; revision received July 26, 1982.

*Staff Specialist, Propulsion and Acoustics Department.

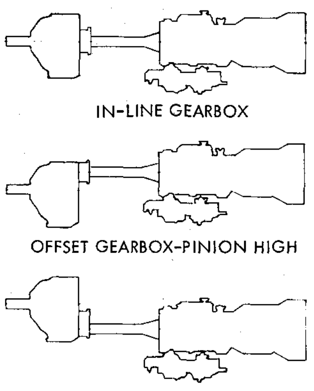

Fig. 1 In-line and offset engine/gearbox arrangements.

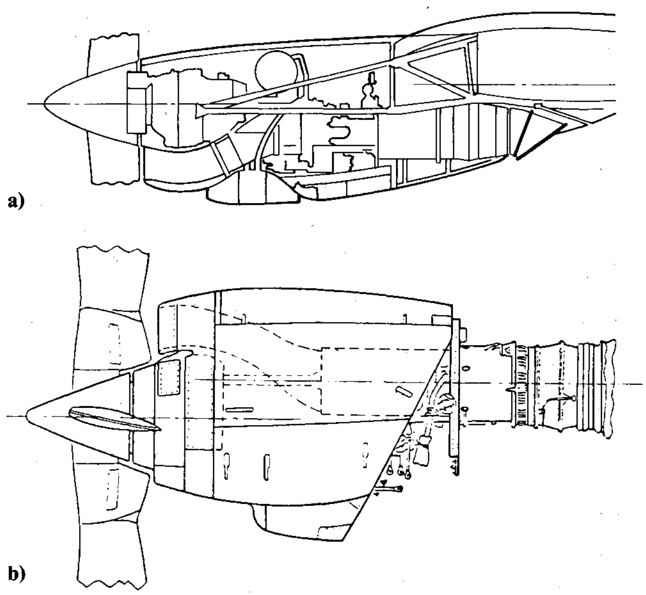

Fig. 2 DDA T 56 engine in a) Lockheed C-130 and b) P-3 nacelles.

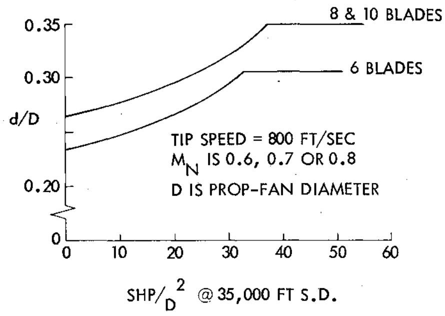

Fig. 3 Minimum nacelle diameter to avoid blade root choking.

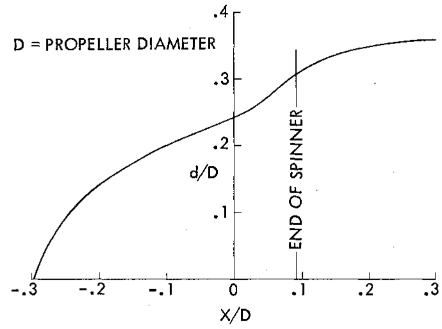

Fig. 4 Forebody shape to avoid blade root choking.

nacelle may depart from axisymmetry and still obtain the desired flow patterns. If it is found that departures from axisymmetry lead to trouble in the blade root region, an in-line engine/gearbox arrangement might be preferred.

There is a second concern with asymmetric nacelles: that the nacelle may induce an effective inflow angle to the prop-fan or that "bumps" on the nacelle may introduce a one-per-revolution (1-P) or higher-order disturbance to the prop-fan blades. The prop-fan blades are expected to be sensitive to 1-P dynamic loads because they are thin and relatively flexible. A need for nacelle symmetry might then be established by the dynamic effects with the result that the in-line engine/gearbox is the preferred arrangement.

The inlet-diffuser duct configuration selected will depend strongly on the engine/gearbox arrangement. An annular inlet is appropriate to an in-line arrangement, but would be difficult to design for an offset arrangement. A scoop-type inlet, on the other hand, appears best suited for the offset engine/gearbox. The complex nature of the design trade studies that must be made in this area will be dealt with in some detail later.

The third major area of nacelle installation design impacted by the choice of engine/gearbox arrangement is nacelle/wing interference. Wings designed for cruise at Mach 0.8 are generally sensitive to deviations from the design environment. The swirling flow from the prop-fan (swirl angles up to about 7 deg) will by itself introduce a strong positive incidence angle on one side of the nacelle and a strong negative incidence on the other—an effect for which compensation must be made. In-line engine/gearbox arrangements appear best suited for nacelles centered in the wing reference plane, while offset engine/gearbox arrangements appear best suited for nacelles mounted over or under the wing.

It is believed that the examples cited above are sufficient to make the point that the airframe designer and the engine manufacturer must work together in designing an optimum propulsion system for the prop-fan aircraft, and to show that there is not, at this point, an obvious best choice. There are some interesting technical questions to be answered in a number of areas before that choice can be made. Those questions that relate to nacelle forebody, inlet, and diffuser design are addressed in the remainder of this paper.

Nacelle/Forebody Design

For an ogival-shaped body, flow near the surface will stagnate at the nose and reaccelerate to freestream velocities as local streamlines turn back parallel to the freestream direction. The general principle followed in design of prop-fan forebodies is to delay reacceleration enough that local Mach numbers are low in the vicinity of the prop-fan blade root region. This is done by careful attention to forebody shape. If this is not accomplished, the contraction of streamlines in the blade root region will cause local acceleration to sonic velocity, which in turn can result in supersonic flow departing the blade root region. This is highly undesirable, since it can result in a rather messy flow on the nacelle forebody in the area where the engine air inlet will most likely be located.

In the development of the prop-fan, some guidelines have been derived to generate acceptable forebody flow patterns. These guidelines are summarized in Figs. 3 and 4. In Fig. 3 (from Ref. 4), the minimum nacelle diameter to prevent blade root choking is shown as a function of prop-fan disk loading SHP/D^2. For eight- and ten-bladed prop-fans in the disk loading range of from 35 to 40, it is shown that the nacelle diameter should be 0.35 times the prop-fan diameter. The data in Fig. 3, however, provide only part of the information needed for forebody design; the shape of the forebody is prescribed by the curve of Fig. 4. This shape was derived from wind tunnel tests of a body-of-revolution nacelle, and was found to give an acceptable surface flow distribution, not necessarily an optimized flow distribution.

Little or nothing is available to provide better design tools than those represented by the curves of Figs. 3 and 4. One of the obvious shortcomings of the available information is that it was derived from body-of-revolution tests and analyses. One might expect that, as long as a realistic nacelle does not depart too greatly from axisymmetry, transonic area ruling would permit the development of an acceptable nacelle forebody shape by following an area distribution equivalent to that represented by the curve of Fig. 4. Even if this is true, however, it does not guarantee that a given design will be free from local flow problems that might occur in critical regions such as the environs of the engine air inlet.

NASA-Lewis has recently initiated work to explore the effect of nacelle asymmetry with a solid model of a prop-fan nacelle on the propeller test rig. Tests with real inlet simulation must await special equipment such as the core inlet test rig currently being designed also by NASA-Lewis. Data from this test rig will probably not be available before late 1983, at the earliest.

It is imperative, therefore, that the development of analytical tools to aid prop-fan nacelle design be continued and accelerated. To be completely adequate these tools must 1) predict realistic prop-fan slipstream flowfields, 2) predict pressure distributions on complex body shapes immersed in the slipstream flow, 3) predict the inlet and internal duct flow, and 4) predict the total effect of these strongly interacting flows.

Any one of the subroutines mentioned above contains formidable obstacles to completion, so the desired objective of a single program to predict the total flowfield seems somewhat remote at this time. Some of the obstacles are discussed in the following paragraphs.

Slipstream analysis in compressible flow is currently possible via several methods. Aljabri of Lockheed[5] has developed a code, somewhat typical of those generally available, that is based on classical vortex theory modified to account for departure from the optimum Betz loading and to include corrections for finite blade width, finite blade thicknesses, and compressibility effects. This and the other similar techniques, however, are not capable of dealing with supersonic tip speeds and do not allow the slipstream flow to be modified as it interacts with the bodies on which it impinges. Thus these methods provide only a first approximation to flowfield definition.

For prediction of pressures and flows over the nacelle and wing, several three-dimensional, transonic, complex-geometry potential flow codes are available.[6-8] These are currently limited, however, by inability to deal with viscous flows and with the complex interactions that occur in transonic viscous flows.

The inlet and internal duct flow is another difficult problem area. If an annular inlet were selected, it presumably could be treated two-dimensionally with a reasonably good approximation. A scoop inlet, however, with a resulting diffuser duct that changes cross-sectional shape, bends, contains internal obstructions, and is strongly sensitive to viscous effects, appears even to an optimist as a computational challenge of first magnitude. Studies dealing with duct flow analysis are reported in Refs. 9-11.

The first step in developing a complete analytical prop-fan nacelle methodology will be the consolidation of the above component design codes to treat the entire problem with interaction of the individual parts so that interactions to an optimized solution can be performed. At this point, that methodology appears to be at least 4 to 5 years in the future.

Inlet/Diffuser Design

Inlet Types

Generally, there are only three basic types of inlets for prop-fan drive systems: annular, branched (bifurcated, trifurcated, etc.), and scoop inlets. Examples of these three types are shown in Fig. 5. All possible inlet configurations may be classified as variations of these three basic types.

The annular inlet adapts best to the in-line engine/gearbox arrangement and has several advantages from the standpoint of internal flow losses and flow distortion. The duct from inlet to compressor face can have favorable turn-radius-to-duct-diameter ratios and should produce little circumferential flow distortion at the compressor face at least in the cruise condition. This inlet, however, will be more sensitive to boundary-layer and blade root flow distortion; and the larger wetted area of the duct will produce higher friction losses than scoop or bifurcated inlets. Figure 6 shows annular inlets on a version of the Lockheed Constellation that was modified for turboprop drive.

There is one peculiar geometric characteristic of the prop-fan, relative to earlier turboprops, that has a large influence on inlet type. The eight- and ten-bladed prop-fans have somewhat larger hubs (to contain the blade-angle-changing mechanism) than the hubs for four-bladed propellers. With

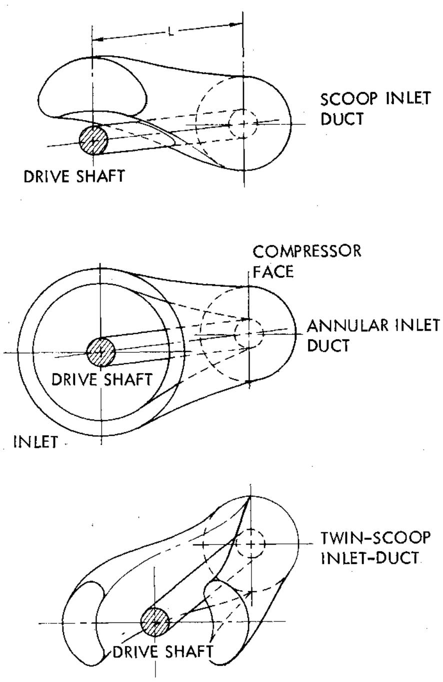

Fig. 5 Prop-fan inlet duct configurations.

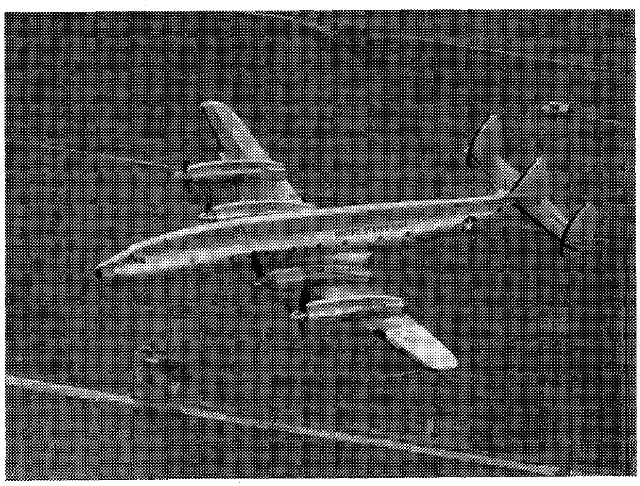

Fig. 6 Lockheed Constellation with annular inlets on turboprop engine installation.

equal throat areas, therefore, an annular prop-fan inlet will have a smaller throat height than for a turboprop. In fact, based on the boundary-layer data of Ref. 3, it appears that throat height for an annular prop-fan inlet will typically be somewhat less than the boundary-layer thickness. This penalizes the internal flow performance for an annular inlet.

The scoop-type inlet is most compatible with the offset gearbox arrangement and also offers a simple method for diverting spinner boundary-layer air. Figure 2 shows the scoop inlets used by Lockheed on the P-3 aircraft, where boundary-layer diversion was accomplished by elevating the scoop to a position off the surface of the nacelle. This also moves the scoop into a region of higher energy air, and thus further improves pressure recovery.

Scoop inlets can be used for in-line gearbox arrangements but, in that application, may require a larger nacelle frontal area than an annular inlet. Use of a scoop also results in a departure from body-of-revolution shape and may be undesirable from that standpoint. A major disadvantage to the scoop inlet is that the engine drive shaft passes through the diffuser duct, thereby producing additional pressure losses and flow distortion.

Branched inlets offer a compromise between annular and scoop inlets and consequently possess some of the advantages and disadvantages of both. Figure 7 shows sketches of two of many possible branched inlet duct designs. As shown, this inlet type can be applied to either in-line or offset engine/gearbox arrangement, and it offers several potential advantages: 1) as in the case of the annular inlet, flow is not required to traverse the engine drive shaft; 2) the inlets can be elevated off the surface for boundary-layer control; and 3) inlets can be placed peripherally at positions where crosswind and other adverse effects are minimized. An obvious disadvantage of branched inlet ducts is that surface area per unit of flow area is greater than for a single scoop, so that friction losses will generally be greater.

Inlet Design

Inlet size and location must be selected to produce minimum nacelle drag, minimum dynamic loading on the prop-fan blades, and the best possible flow to the engine compressor face. Inlet location must also recognize the need for an unobstructed airflow in static operation and in the reversed-thrust mode. Inlet size, or cross-sectional area, also plays a large role in determining both nacelle drag and the quality of flow delivered to the engine.

Flowfield Definition

The flowfield in which prop-fan engine inlets will likely be located has been the subject of both analytic and experimental studies. In studies to design a new prop-fan inlet test rig,[12] NASA-Lewis has defined the flowfield, which will be described with the help of Figs. 8-10.

Figure 8 shows local Mach numbers behind an SR 3 prop-fan plotted against a radial position function defined by the percentage of total slipstream flow intercepted. This is a convenient vertical scale, since an inlet can also be defined in terms of the percentage of slipstream flow that it captures in a local region. An annular inlet, for instance, for an engine scaled to the following conditions: a Mach number of 0.8, an altitude of 35,000 ft, and $SHP/D^2 = 37.5$ will capture about 1.3% of the total prop-fan slipstream. This is indicated in Fig. 8. A scoop inlet contained within an azimuthal angle θ would capture the same percentage of total slipstream flow, but a

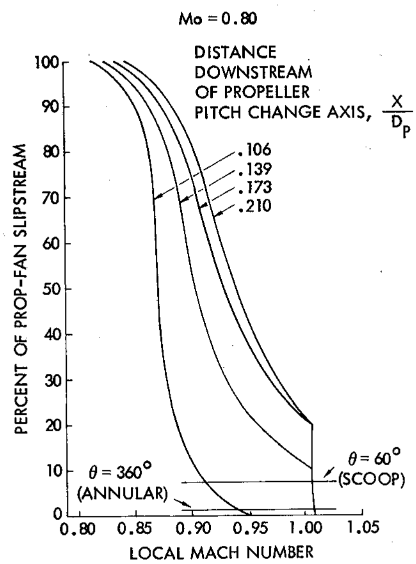

Fig. 8 Predicted Mach numbers behind SR 3 prop-fan.[10]

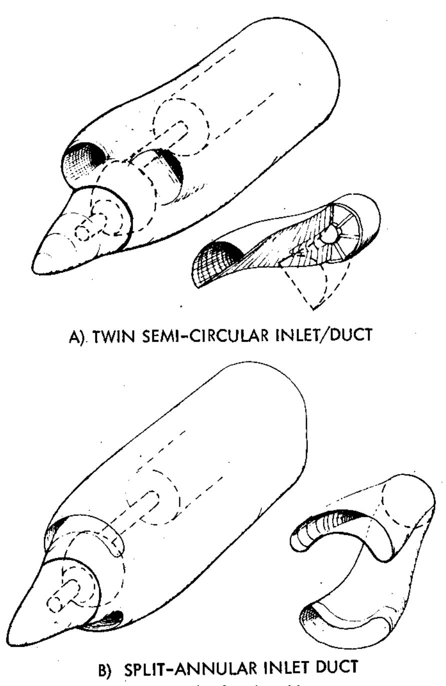

Fig. 7 Branched inlets.

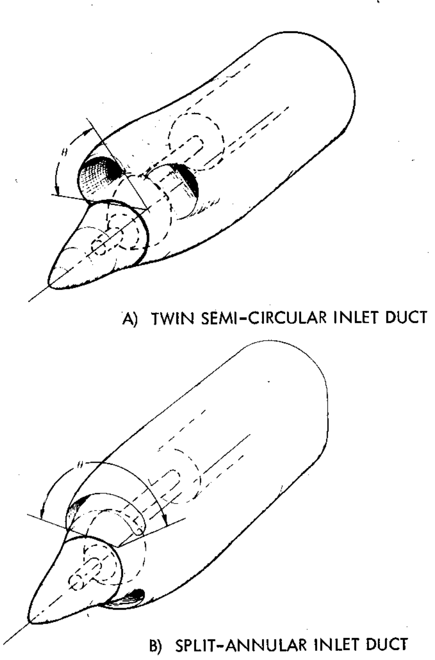

Fig. 9 Branched inlet duct with definition of inlet parameter.

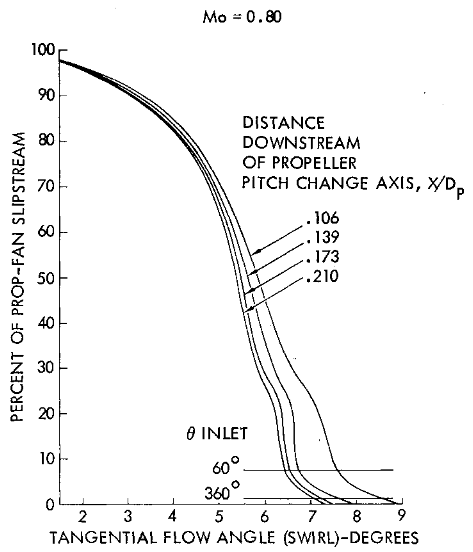

Fig. 10 Predicted swirl behind SR 3 prop-fan.[10]

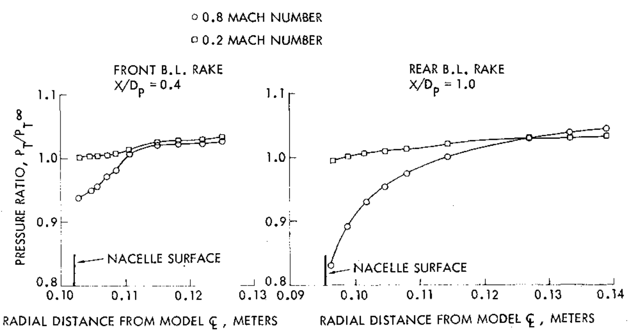

Fig. 11 Measured boundary layers behind SR 1 prop-fan.[11]

larger percentage of the local slipstream flow by the ratio $360/\theta$. This concept is depicted by the sketch of Fig. 9.

Referring back to Fig. 8, it is evident that in the region behind the prop-fan, local Mach numbers approach and slightly exceed unity—even at freestream Mach numbers of 0.8. These data were calculated, however, for clean axisymmetric nacelles without inlets. The presence of an inlet will probably reduce these local Mach numbers.

The flow behind the prop-fan will also be characterized by a significant amount of swirl. This is shown in Fig. 10 using the same calculated data as were used for local Mach numbers. It can be seen that the inlet must be located in a region where swirl angles of from 6 to 9 deg are predicted.

The calculations upon which the flowfield data of Figs. 8 and 10 are based did not include viscous effects. When the boundary layer is thick relative to inlet height, its effect will be to reduce the predicted local Mach numbers. At the same time, however, the boundary layer will reduce the mean total pressure at the inlet face. Boundary-layer profiles were measured at two locations behind a prop-fan in the wind tunnel tests of Ref. 3. Results for a freestream Mach number of 0.8 are shown in Fig. 11. The boundary-layer thickness in the forward survey position is greater than the annular inlet height depicted in Figs. 8 and 10.

Figure 12 shows one other aspect of the prop-fan flowfield that must be considered: the supercharging effect of the propfan. It can be seen that total pressure is greater than freestream total pressure by significant amounts in the outer radial regions behind the blade. This effect gives advantage to inlets located at greater radial distances from the nacelle surface.

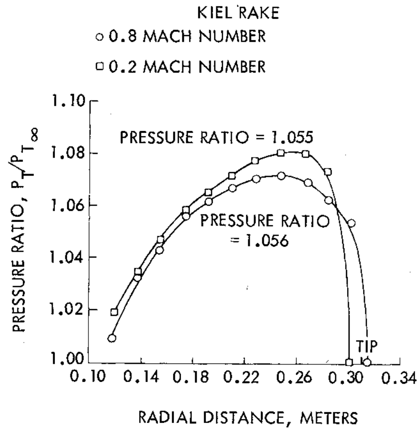

Fig. 12 Supercharging effect of prop-fan in slipstream.[11]

Inlet Area Selection

Contemporary and advanced-design engines are operating with compressor face Mach numbers in the general vicinity of 0.5. The prop-fan inlet, as was shown in Fig. 8, will be located in a region where local flow Mach numbers are at least 0.8. Obviously then, somewhere in the air induction system, the ingested air must be slowed from Mach 0.8 to 0.5 before it is delivered to the engine. The designer faces the choice of taking this diffusion internally or externally, and he exercises this choice in the selection of inlet area. If he opts for a smaller inlet area and internal diffusion, that diffusion must take place in a duct that makes an S-bend around the gearbox and changes cross-sectional shape. With a larger inlet area, and resulting external diffusion and spillage, nacelle drag penalties may be excessive. These penalties could be quite high if the spillage results in shock waves in the transonic flowfield.

The problem of selecting the best inlet area remains basically the same regardless of whether the inlet is annular, branched, or scoop. However, the internal flow problems may be greatest for a scoop-type inlet, since the scoop duct must carry flow around the engine drive shaft and the S-bend offset will be more severe.

Earlier turboprop installation designs have not had to deal with this inlet problem to any significant degree because they have been designed for cruise at Mach numbers of 0.6 or less. Generally, the engine air ducts for these installations have been designed with a contraction from the inlet to the compressor face. The necessary diffusion was taken external to the inlet, but at Mach number 0.6 or less, this can easily be accomplished without significant spillage drag.

Boundary-Layer Effects

The inlet type must be selected with another factor in mind: the effect of boundary layer on inlet performance. The boundary-layer thickness shown in Fig. 11 is of the same order of magnitude as the height of an annular inlet for an engine scaled to this model. As has already been pointed out, this means that the pressure recovery of an annular inlet would be significantly penalized by the presence of the boundary layer. For these annular inlets there appears to be no easy way of removing boundary-layer air whose quantity is as great or greater than the engine airflow.

If we examine the extreme opposite of an annular inlet—a scoop with circular cross section, the boundary-layer thickness shown in Fig. 11 would not greatly impact total pressure recovery unless the boundary layer caused separation inside the inlet duct. Even this could be avoided by raising the lower lip of the scoop off the nacelle surface by a slight amount. This type of inlet could avoid boundary-layer problems completely, but perhaps at the expense of unacceptable external drag levels.

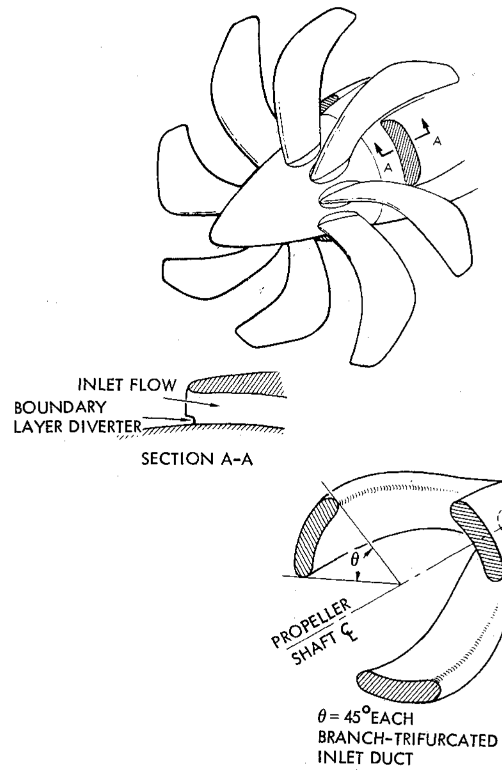

Fig. 13 Trifurcated inlet duct for in-line engine/gearbox. The advantages are 1) branched ducts are quasi-two-dimensional; 2) good bend radii; 3) moderate transition from inlet to compressor face; 4) boundary layer diversion under scoop.

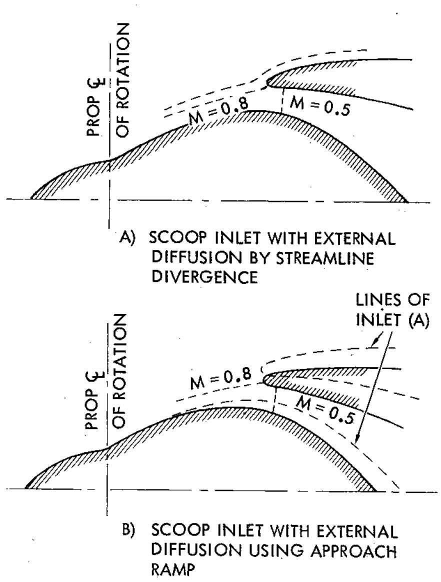

Fig. 14 External diffusion concepts.

For an in-line engine/gearbox arrangement, branched inlet ducts (bifurcated, trifurcated, etc.) may provide the best compromise. One such inlet duct is depicted in Fig. 13—a trifurcated or three-branch inlet duct. With this arrangement the internal duct bends and contortions are not at all bad, and the inlets can be kept large relative to the approaching boundary layer.

Inlet Spillage Drag Consideration

As stated earlier, the nacelle designer has a choice in inlet area selection concerning where and how to diffuse from about Mach 0.8 to the compressor face Mach number of about 0.5. The reasons for not diffusing in complex S-ducts

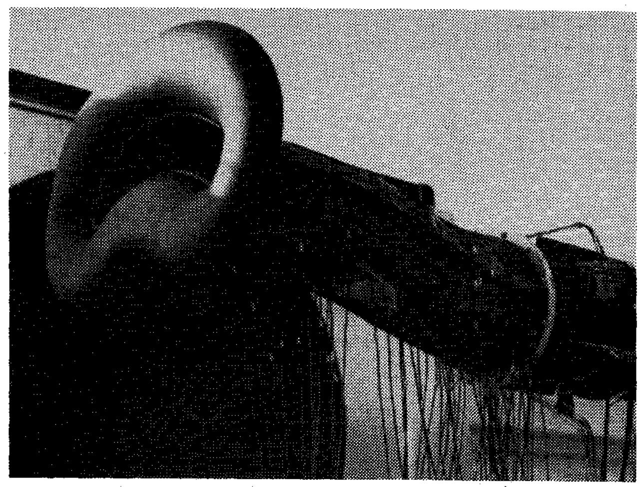

Fig. 15 Inlet duct diffuser model—scoop inlet.

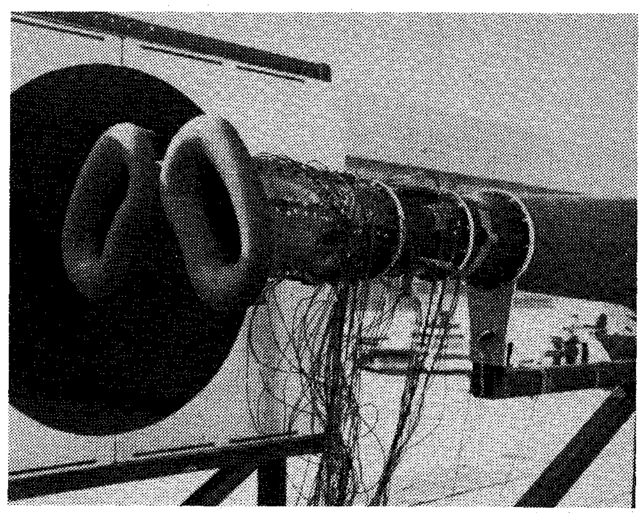

Fig. 16 Inlet duct diffuser model—bifurcated inlet.

have already been mentioned. The alternative, however, of diffusing in the flow ahead of and external to the inlet must be carefully done if spillage drag is to be avoided. From previous work, we know that spillage drag will be zero if the inlet lips are shaped so that suction on the lips balances drag forces in the externally diffused slipstream. The larger the spillage, however, the fatter the lip must be and the greater the danger of transonic flow (and drag-rise effects) on the inlet cowl.

An alternate inlet concept has arisen from consideration of submerged inlets and their possible application to prop-fans. This concept would encourage external (to inlet) flow diffusion by pulling the nacelle surface inward and away from the slipstream. The concept is illustrated in Fig. 14, where it may also be compared with the more conventional external diffusion geometry. The projected advantages of the semisubmerged inlet concept would be a lower nacelle frontal area and a higher drag-rise Mach number.

The concept will be limited, however, by the ability of the approaching boundary layer to follow the ramp. This inlet would, as do most submerged inlets, probably have a lower pressure recovery than the more conventional scoop configuration.

Diffuser Duct Design

Subsonic diffuser design has received considerable attention for a good many years, and for many installations reasonably accurate predictions of total pressure loss and flow distortion can be made. For axisymmetric and other simple shapes without strong secondary flows, there are now some computer programs to predict viscous flow development.

If an annular inlet duct were selected for a prop-fan installation, diffuser performance could probably be predicted by a combination of empirical and analytical methods, assuming a quasi-two-dimensional flow. For some branched inlet ducts, the same might be true. For scoop inlets like those in Fig. 2, with distorting S-ducts penetrated by an engine drive shaft, there are neither empirical nor analytical methods sufficient to predict performance accurately.

It has already been pointed out that the turboprop aircraft currently flying have been able to use engine inlet ducts with contraction from the inlet to the compressor face without paying an excessive external drag penalty. There has, therefore, been little pressure to generate technology for the duct systems needed today. Methods are badly needed now to predict internal flow performance for complex S-ducts. These methods must be able to account for inlet boundary-layer effects and for the effects of swirl in the flow approaching the inlet.

The Lockheed-Georgia Company has recently begun experimental studies of complex S-duct diffusers appropriate to prop-fan installation. Test apparatus for two of the configurations are shown in Figs. 15 and 16. In the first phase of this program, bell-mouth inlets were used, producing an ideal inlet condition with no boundary-layer or inlet swirl effects. In later test phases, an attempt will be made to simulate both effects. Some of this work is reported in Ref. 13. Additional research in this area is needed.

For prediction and analysis of duct flows, the program developed by Anderson[9] for axisymmetric viscous flows is currently the best available tool. It has been used for approximate analysis of an S-shaped subsonic diffuser by transforming the three-dimensional geometry into a section of an equivalent annular duct. This effort, part of the HiMat program, gave good results for conditions where the flow was not separated. It provided indication of regions where flow separation might be expected, and these predictions correlated well with experimental data. However, the geometry analyzed did not contain a penetrating drive shaft.

Concluding Remarks

This survey has presented some of the challenging design problems associated with prop-fan propulsion installation. The survey has been confined to nacelle forebody and air induction system design, but with full awareness that integration of the nacelle into the wing and aircraft environment is yet another major problem area.

In designing a propulsion system for advanced prop-fan aircraft, the airframe and engine manufacturer must work together to decide whether the engine and gearbox should be in-line or offset, because this choice will have a large bearing on the design of the external nacelle and the air induction system.

The Mach 0.8 cruise speed, as an objective in prop-fan aircraft design, adds new dimensions to the problems of designing turboprop nacelles. The complex air duct required to take air around the gearbox may not be able to afford the luxury of contraction afforded at lower cruise Mach numbers. Even a constant-area duct will require the inlet to operate with spillage in the high-speed cruise condition. Finally, the prop-fan blades themselves may be quite sensitive to both static and dynamic loads associated with asymmetric forebodies. All of these problems interact, adding to the overall complexity of the design.

There are several vital gaps in the experimental data base for optimized design, and existing computational codes are of limited value. There is a strong need for advances in both areas.

References

[1]Bradley, E.S. and Little, B.H. Jr., "The Role of Flight Research Vehicles in Prop-Fan Technology Development," ASME Preprint 81-GT-216 presented at the ASME Gas Turbine Conference and Products Show, Houston, Texas., March 1981.

[2]Banach, H.J. and Reynolds, C.N., "Prop-Fan Engine Propulsion for the 1990's," AIAA Paper 81-1648, Cleveland, Ohio, June 1982.

[3]Black, D.M., Menthe, R.W., and Wainauski, H.S., "Aerodynamic Design and Performance Testing of an Advanced 30° Swept, Eight Bladed Propeller at Mach Numbers from 0.2 to 0.85," NASA CR 3047, Sept. 1978.

[4]"Prop-Fan Data Support Study," NASA CR 15241, prepared by Hamilton-Standard, Feb. 28, 1978.

[5]Aljabri, A.S., "Prediction of Propeller Slipstream Characteristics," Lockheed-Georgia Rept. No. LG79ER0120, Oct. 1979.

[6]Caughey, D.A. and Jamison, A., "Accelerated Calculation of Transonic Nacelle Flow Field," *AIAA Journal,* Vol. 15, Oct. 1977, pp. 1474-1480.

[7]Hess, J.L., Mack, D., and Stockman, N.O., "An Efficient User-Oriented Method for Calculating Flow In and About Three-Dimensional Inlets," NASA CR 159578, 1979.

[8]Rizk, M.H., "Propeller Slipstream/Wing Interaction in the Transonic Regime," Flow Research Rept. No. 132, June 1979.

[9]Anderson, O.L., "Finite-Difference Solution for Turbulent Swirling Compressible Flow on Axisymmetric Ducts with Studies," NASA CR 2365, 1974.

[10]Eiseman, P.R., Levy, R., McDonald, H., and Briley, W.R., "Development of a Three-Dimensional Turbulent Duct Flow Analysis," NASA CR-3029, 1978.

[11]Neumann, H.E., Povinelli, L.A., and Coltrin, R.E., "An Analytical and Experimental Study of a Short S-Shaped Subsonic Diffuser of a Supersonic Inlet," AIAA Paper 80-0386, Pasadena, Calif., Jan. 1980.

[12]"Aerodynamic and Mechanical Design of Advanced Turboprop Core Inlets for the NASA-Lewis Research Center Propeller Test Rig," NASA-Lewis RFP 3-153420, July 1980.

[13]Little, B.H. Jr. and Trimboli, W.S., "An Experimental Investigation of S-Duct Diffusers for High-Speed Prop-Fans," AIAA Paper 82-1123, Cleveland, Ohio, June 1982.