measured data trends for the transonic turbine are similar to those reported in the paper for the subsonic turbine. In closing, the authors wish to express their thanks to Mr. Lister for his learned critique.

Effect of Rotor Design Tip Speed on Aerodynamic Performance of a Model VTOL Lift Fan Under Static and Crossflow Conditions

U. W. Schaub.[2] The present paper goes a long way toward providing a better understanding of lifting fan aerodynamics and much needed, but expensive to obtain, test data. Hopefully, these experimental data, along with other published data, will lead to analytical studies that will ultimately culminate in design methods appropriate for lifting fans under crossflow conditions.

In discussing the crossflow effects, the authors fully recognize the importance of back-pressure distribution. It is relevant to observe that another important variable determining back pressure distribution, if the discharge is not fully axial, is efflux swirl. With exit louvers present, this would lead to a severe malincidence condition on the louvers, and significant losses due to misalignment and secondary flow effects. Without louvers, there would be a significant Magnus effect that would *(a)* change the circumferential distribution of static pressure, and *(b)* reduce the efflux wake in the crossflow. In this connection, it is important to realize that the back-pressure condition which open louvers provide is not the same as if there were no louvers at all. This may, in part, account for the demonstrated differences in the levels of back pressure for the three fan stage builds tested.

As the authors point out, the shown variation of exit total pressure with circumferential position (Fig. 11) cannot be explained in terms of back-pressure effects because the reported axial discharge and the exit louver arrangement used support a side to side flow symmetry. This total pressure distribution can, however, be directly associated with fan reaction to "potential inflow" distortion, and particularly to circumferential variations in rotor incidence angles.

The inflow distortion is, in fact, a potential flow distortion by virtue of the observation that the bulk of entering flow, no matter how badly distorted, is irrotational. However, the front-to-back variation in velocity the authors discuss, which is a flow bending phenomenon, is significantly adjusted by the advancing and retreating blades of the fan in reaction to *(a)* the distortion swirl (a crossflow residual), and *(b)* the back-pressure distribution. This results in a side-to-side variation of velocity with advancing blades seeing higher velocities than retreating blades, as shown in reference [3].

The attached figure shows some measured rotor blade mean radius incidence angles in a similar 12 in. lifting fan stage (tip speed and pressure ratio being 680 ft/sec and 1.16, respectively), also 0.75 in. upstream of the fan inlet face. Several observations can be made:

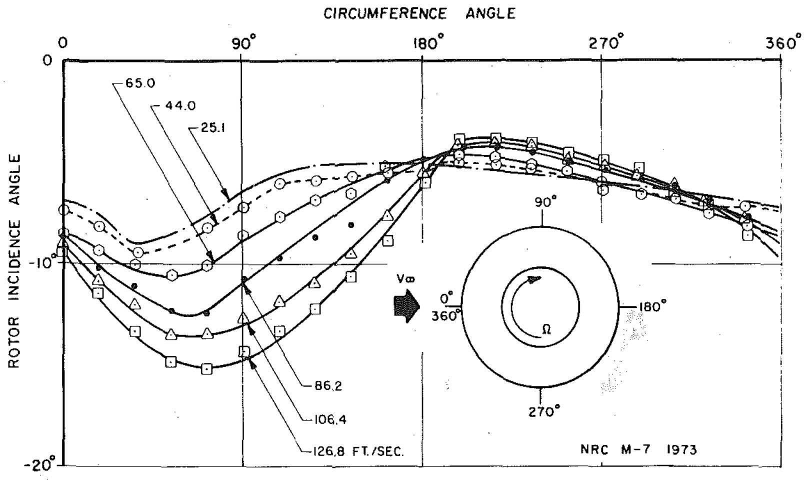
1 Crossflow has a profound influence on rotor blade incidence: the excursions in incidence angle increase rapidly with increasing crossflow condition at any given fan speed.

2 The actual circumferential distribution is not sinusoidal, as predicted from potential flow distortion theory [1].

3 The excursions towards the negative on the retreating blades far exceed those towards the positive on the advancing blades.

The significance of rotor blade incidence variation with crossflow condition and fan build tested suggests some difficulty in meaningfully explaining crossflow performance data in terms of static performance characteristics when blading incidence angles are at design (low loss) values.

Although the authors demonstrate that their low speed fan (750 ft/sec stage) featured the smallest surge margin of the three builds, it would seem that in the absence of exit louvers at least (but probably in general as well) this feature is not relevant to fan operating conditions normally expected because crossflow drives rotor incidence angles towards the negative, that is to say the fan tends away from surge.



CIRCUMFERENTIAL DISTRIBUTION OF ROTOR INCIDENCE AT MEAN RADIUS AND A FAN SPEED OF 10,000 R.P.M.

Fig. 1

---

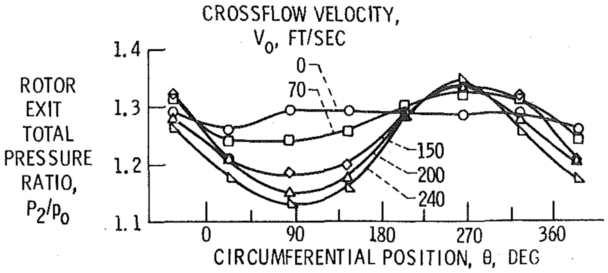
[1] By N. O. Stockman, L. S. Loeffler, and S. Liebein, published in the Oct. 1973 issue of JOURNAL OF ENGINEERING FOR POWER, TRANS. ASME, Series A, Vol. 95, No. 4, pp. 293-300.

[2] Research Officer, Engine Laboratory, National Research Council of Canada, Ottawa, Canada.

Authors' Closure

The experiments conducted by the authors at NASA and by the discusser and his colleagues at the National Research Council of Canada have amply demonstrated the complex nature of the flow through lift fans with two-dimensional exit louvers under forward velocity conditions. This is not surprising when it is considered that, within a relatively very short axial distance, the approaching free-stream flow is required to first turn 90 deg into the annular constraint of the inlet section, then be compressed by two rows of blading in the fan stage, and then immediately be coerced into a turned rectilinear flow by the exit louvers. Superimposed on this tortuous path are the interaction effects of the circulation developed about the aerodynamic body containing the fan. The lift fan in crossflow, therefore, embodies a complex closely coupled flow system with large-amplitude disturbances.

Internal interaction effects in the inlet are clearly demonstrated by the measured circumferential variation of rotor incidence angle supplied by Dr. Schaub for a fan in wing. The nonsinusoidal incidence variation in crossflow can well explain similar forms for the circumferential variation in rotor outlet total pressure observed in the NASA fan-in-wing tests as shown in Fig. 2 (taken from Fig. 10 of reference [3]). However, significantly less deviation from a sinusoidal variation was observed for the fan-in-pod tests shown in Fig. 11 of the authors' paper. Furthermore, the degree of deviation from sinusoidal variation for the fan-in-pod data appears to vary with design tip speed, with less deviation at the lower design tip speeds. The prediction of interaction or coupling effects in lift fan flow may therefore be difficult.



Fifteen-inch fan-in-wing rotor outlet total pressure in crossflow. Mid-passage radius, design tip speed (980 ft/sec).

Fig. 2

It is believed, however, that a static thrust performance map can still provide a very meaningful baseline for the qualitative trend prediction of the mean performance in crossflow, providing the effect of crossflow on back-pressure ratio is known or can be estimated. This can be illustrated by the application of Fig. 13 (back-pressure ratio) to the static map of Fig. 7 to provide a satisfactory rational synthesis for the measured crossflow trends of Fig. 9.

Although the crossflow effect does reduce the fan back pressure and tends to move the stage away from surge, a low-stall-margin fan can be potentially troublesome if louvers with large deflection angles are required (e.g., Fig. 9(*a*)).

A Wake and an Eddy in a Rotating Radial Flow Passage Part I: Experimental Observations; Part II: Flow Model[1]

W. Pfenninger.[2] Dr. Moore's experiments clearly show that the secondary flow on the top and bottom walls of his rotating diffusor are largely responsible for the low energy "wake" flow on the trailing surface of rotating diffusers. With the purpose in mind of raising the efficiency of radial flow compressors the question arises as to how to control and preferably eliminate this wake flow. This may be accomplished as follows:

In stationary diffusers of elliptic cross section, curved in the direction of the major axis of the ellipse, Sprenger has been able to suppress the accumulation of low energy air on the convex wall of the diffuser by introducing suitable artificial secondary flows in the diffuser by means of carefully placed small guide vanes (reference [1].[3] To minimize diffuser losses at lower axis ratios of the ellipse these guide vanes should preferably generate a counter-secondary flow, partially compensating the original secondary flow in the curved diffuser. To minimize diffuser losses at larger axis ratios, though, it usually was advantageous to generate a strong angular momentum in the diffuser flow passage by means of Sprenger's vanes. The efficiency of curved diffusers could thus be raised close to the values of straight diffusers of the same axis ratio (reference [1]). Since Coriolis forces in a rotating diffuser induce similar pressure gradients between its driving and trailing side, as the pressure gradients between the convex and concave surfaces in curved stationary diffusers, resulting from centrifugal forces, Sprenger's guide vanes, as well as, perhaps, suitable boundary layer stator vanes, might be equally effective in eliminating the accumulation of a low energy wake on the trailing side of rotating diffusers. Higher efficiencies should then be expected in rotating diffusers. At the same time, the velocity distribution at the exit of rotating diffusers would be more uniform to enable a correspondingly more efficient pressure recovery in the downstream stator diffusers of radial compressors.

Even though Sprenger's small auxiliary airfoils control secondary flow in curved stationary, and probably in rotating diffusers, boundary layer separation on the trailing side of rotating diffusers and the convex surface of stationary diffusers may still occur in strongly decelerated diffuser flows as a result of the reduced local turbulent boundary layer mixing under the action of Coriolis and centrifugal forces, respectively. Such separation might be avoided by enhancing the turbulent boundary layer mixing of these particularly critical diffuser surfaces. This might be accomplished by introducing longitudinal vorticity into the boundary layer, for example, by means of vortex generators or (at smaller losses), by redirecting part of the existing boundary layer vorticity into axial direction, using for example Kuethe's oblique wave vortex generators (reference [3]). Alternatively, longitudinal boundary layer vortices may be generated artificially in the inner layer of a turbulent boundary layer by means of suitably spaced shallow surface protuberances, with their chordwise and spanwise dimensions corresponding approximately to the chordwise and spanwise extent of Kline's longitudinal vortices (reference [4]) in the inner layer of a turbulent boundary layer. These artificially introduced longitudinal vortices may then intensify Kline's aforementioned longitudinal vortices to thin the laminar sublayer, raise the turbulent skin friction and increase the turbulent mixing within the boundary layer to raise accordingly the turbulent pressure rise without separation.

Secondary flow and pressure drag losses may be further reduced by means of boundary layer suction. Based on the experience with swept low drag suction wings (reference [5]), where spanwise pressure gradients induce a boundary layer crossflow in

---

[1] By J. Moore,[4] published in the July 1973 issue of JOURNAL OF ENGINEERING FOR POWER, TRANS. ASME, Series A, Vol. 95, No. 3, Part I pp. 205–212, Part II pp. 213–219.

[2] Boeing Commercial Airplane Co., Seattle, Wash.

[3] Numbers in brackets designate Additional References at end of Discussion.

[4] Present address: University Engineering Department, Cambridge, England.