8.4 Propulsion Airframe Integration

D. Mikkelson
NASA Lewis Research Center

Figure 1. 20" Turbofan Simulator. Proper simulation of the propulsion system can be quite important for both isolated propulsion system work and propulsion/airframe integration work. We found this out in an isolated simulator test using a 20" powered nacelle (shown in this figure). This simulator had a fan pressure ratio of 1.15 and was tested in the Lewis 8-by-6-foot Supersonic Wind Tunnel. (Reference: NASA TMX-3064.)

20" FPR 1.15 NACELLE

$D_m/D_f = 1.08$ $\quad$ $X_i/D_m = .175$

$L_c/D_m = 1.51$ $\quad$ $D_n/D_m = .884$

$D_{h\ell}/D_m = .935$

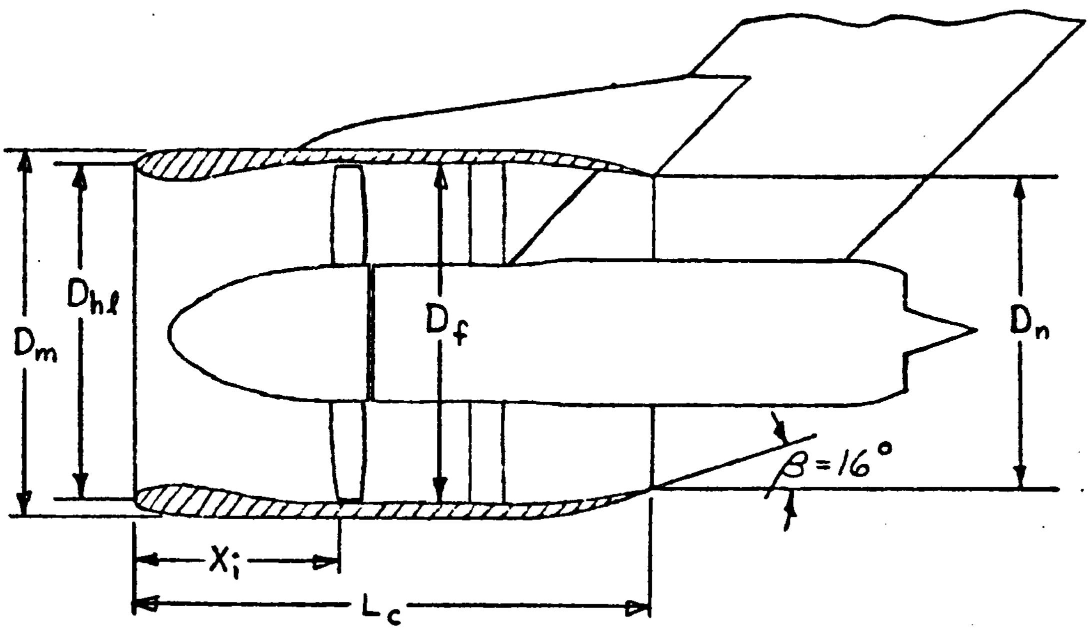

Figure 2. 20" Fan Pressure Ratio 1.15 Nacelle .
Design drag divergence Mach number: 0.8
Tightly cowled: $D_m/D_f = 1.08$
Inlet: $D_{h\ell}/D_m = .935$
Inlet capture mass flow ratio: $A_o/A_{max.})_{des., M=.75} = 0.66$
Inlet cowl length: $X_i/D_m = .175$
Fan Boattail angle: 16°

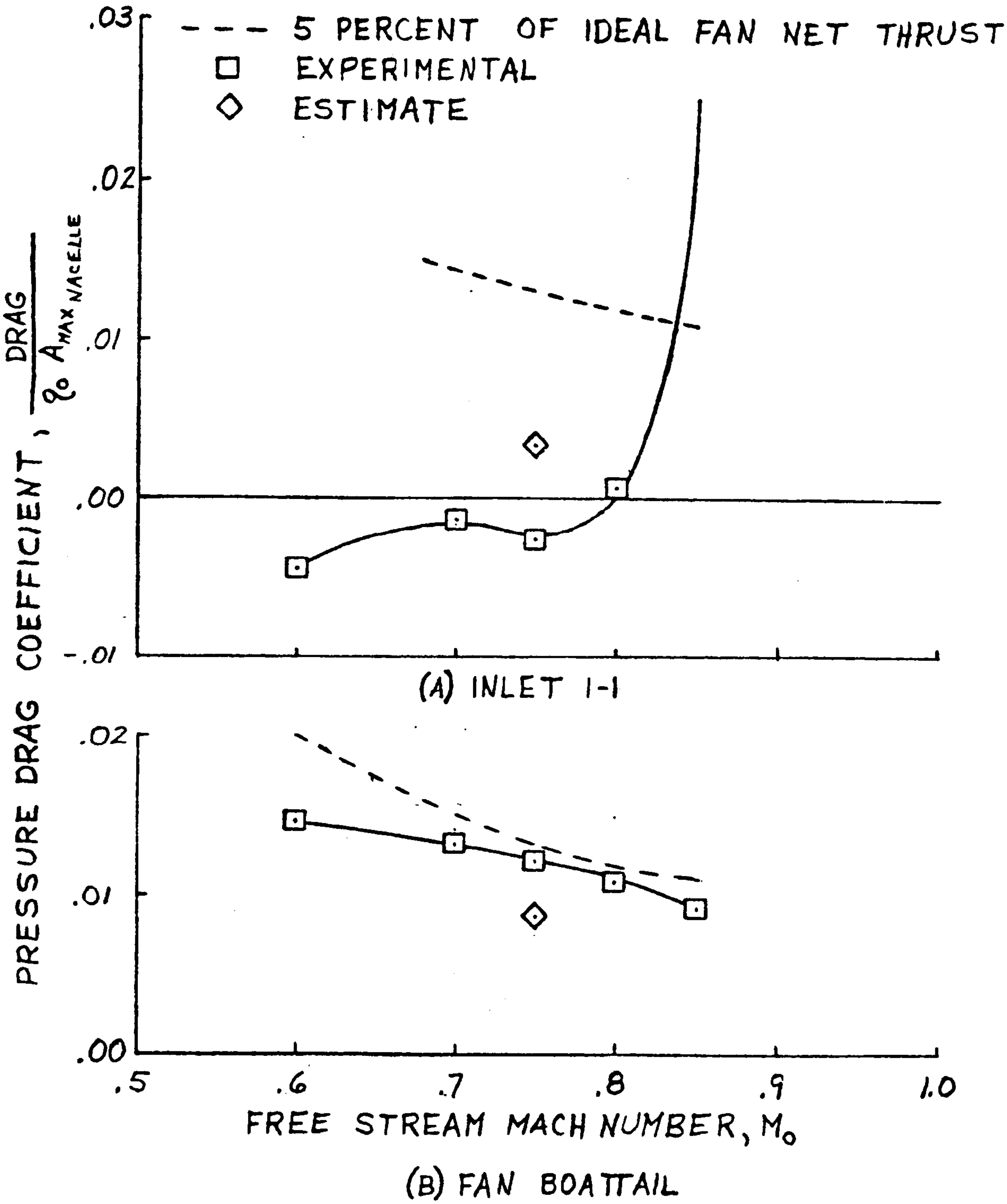

Figure 3. Variation of Inlet and Fan Boattail Pressure Drag with Mach Number

1. Below drag rise the inlet pressure drag was less than the estimated value. However, in the same speed range boattail drag was higher than the value estimated from model boattail tests.
2. In addition, as the inlet went into drag rise (above M=0.8) the boattail drag decreased somewhat.
3. Both of the above trends indicate the possibility of an interaction between the inlet and aft end flow fields for close coupled propulsion systems like this one.

EFFECT OF BOATTAIL PROXIMITY ON INLET PRESSURE DRAG

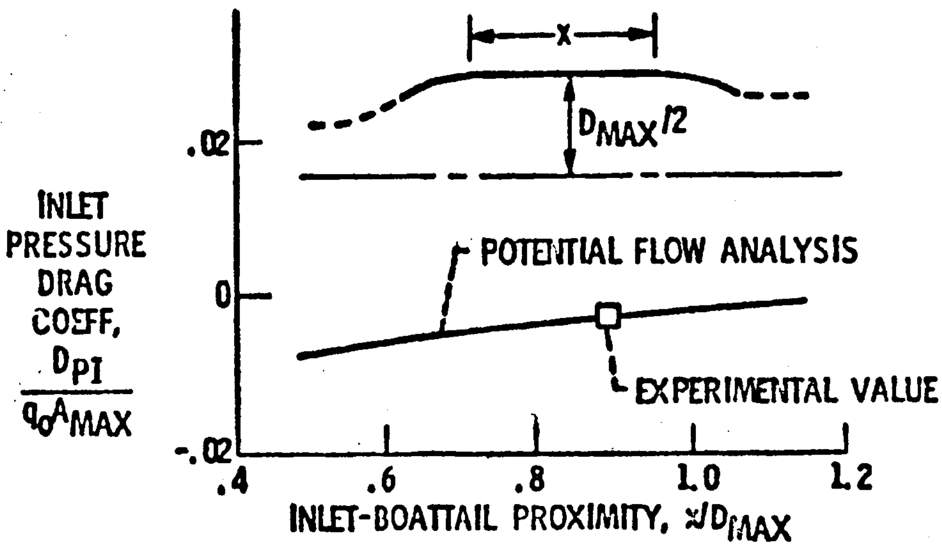

Figure 4. Effect of Boattail Proximity on Inlet Pressure Drag.

To verify the interaction between inlet and exit flow fields, we analyzed the effect of the proximity between the inlet and nozzle on the inlet pressure drag using a 2D potential flow program. This was done by varying the distance between the inlet and boattail. The calculated pressure force was adjusted to pass through the experimental value shown at $X/D_{max} = .9$. As the boattail was moved closer to the inlet (decreasing X/D_{max}), it resulted in a reduction of the inlet drag; thus indicating that there is an interaction between these two flow fields.

In light of this interaction between the inlet and aft end flow fields, it may be quite important to simulate the proper flow fields of both components simultaneously when doing isolated propulsion system and propulsion system/integration work. Three propulsion simulation techniques that are commonly used are:

1. Flow thru nacelle--which is normally used to properly model the inlet flow field.
2. The blown nacelle --for proper simulation of nozzle flow only (correct NPR).
3. Powered turbofan simulator--close simulation of both inlet and nozzle flow fields.

At NASA Lewis we have a program underway to evaluate and compare these three simulation techniques (both isolated and installed with the airframe) in terms of their degree of simulation and their relative accuracy. This program will be conducted for both conventional and unconventional types of airframe installation.

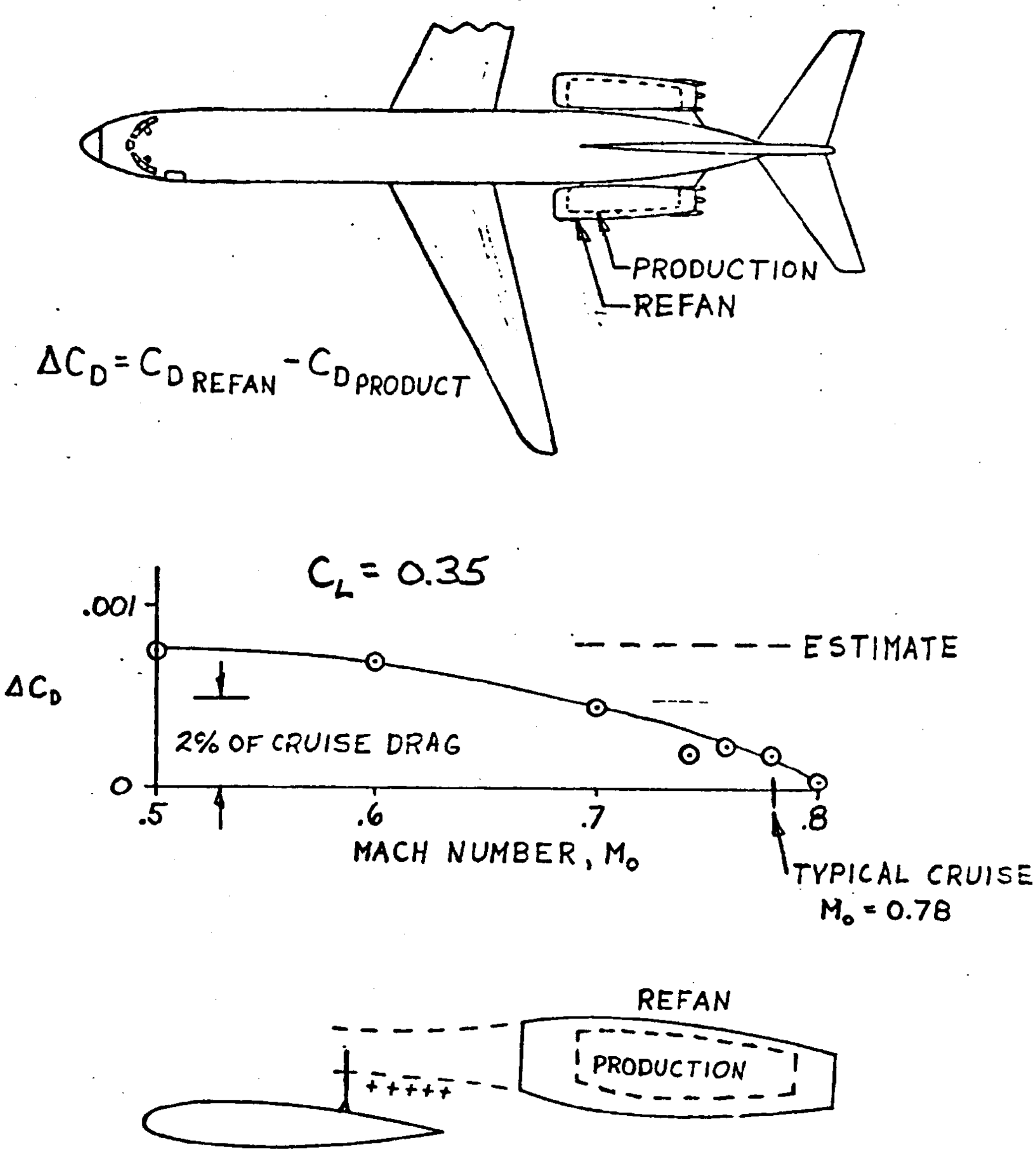

Figure 5. Effect of Nacelle Size on DC-9-30 Cruise Drag.

Flow-thru nacelles are normally adequate when concerned about the interaction between the inlet and wing flow fields like aft fuselage installations. Some of the results from the DC-9 refan program are shown above. Shown is the drag penalty associated with the larger refan nacelle poltted as a function of free-stream Mach number, M_0(at $C_L = 0.35$). An estimate was made of the drag penalty and is shown as a dashed line. Based on these wind tunnel results, the drag increment decreased as M_0 was increased.

At the cruise Mach number of 0.78, most of the estimated drag increment was cancelled out due to a favorable interference effect. This favorable effect was associated with the larger refan inlet and its closer proximity to the wing. This effect most likely occurs because the positive pressures on the stream tube suppress the wing upper surface velocities, thereby moving the wing shock forward and reducing the Mach number at the shock with subsequent reduction in wing compressibility drag. This trend was observed from wing pressure data. (Reference: NASA CR-121219.)

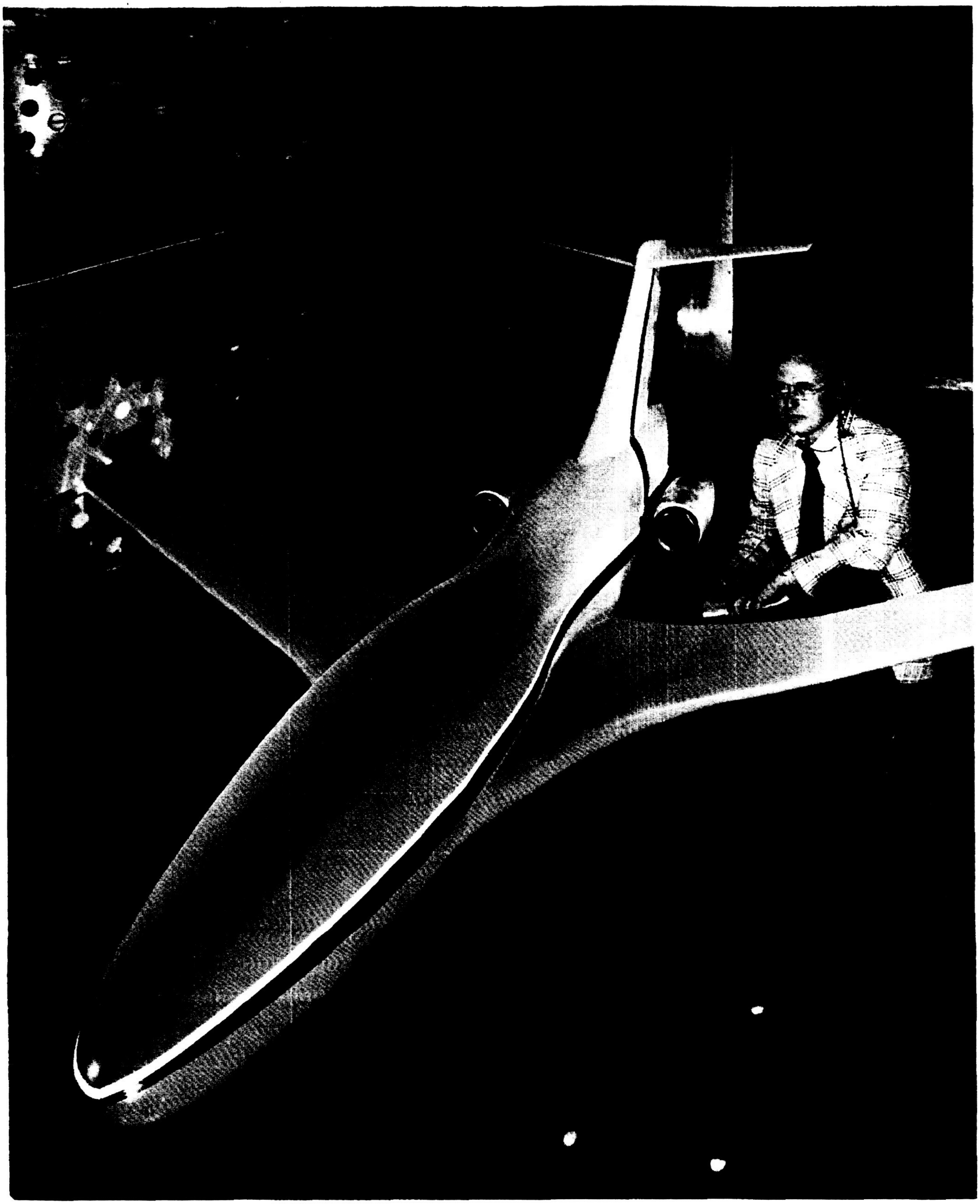

Figure 6. CTOL Full Span Model. At higher Mach numbers it may also be possible to achieve a favorable interference effect. Shown here is a full span CTOL model which was tested in the Lewis 8 x 6 SWT to investigate propulsion system/airframe integration at cruise speeds above M = 0.9 for the same type of aft fuselage nacelle installation. This model was also tested with flow thru nacelles and incorporated local area ruling in the nacelle vicinity. The part of the model aft of the wing trailing edge was on a balance. (Reference: NASA TMX-3178)

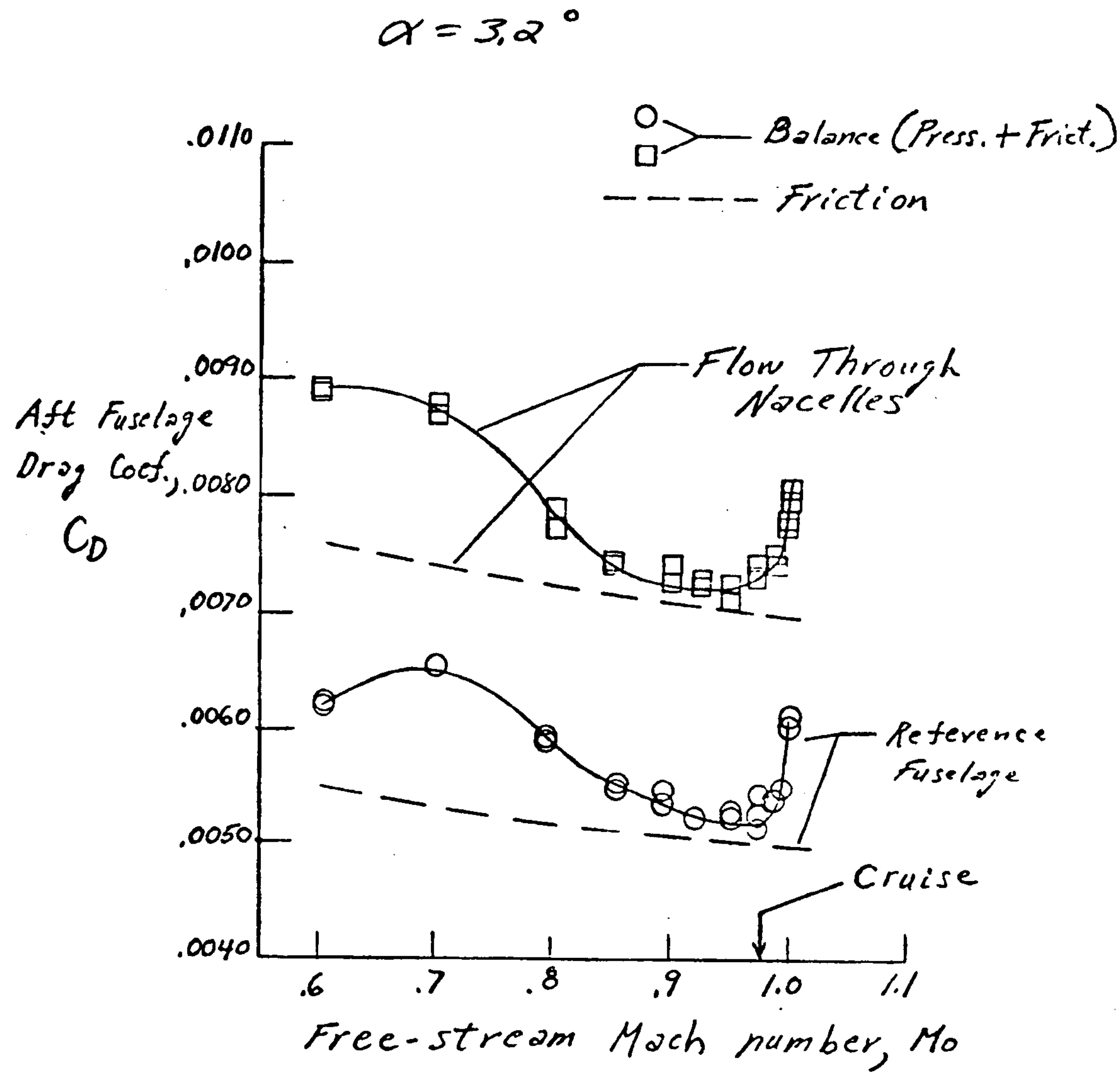

Figure 7. Full Span CTOL Aft Fuselage Drag

Shown above is the total aft fuselage drag with and without nacelles installed. The nacelles had NACA-1 inlet cowl contours and relatively low boattail angles (8 to 10°). The estimated friction drag (flat plate type calculation) is shown as a dashed line. The reference fuselage did not have nacelles; however, it had the same total area distribution as the fuselage with nacelles installed.

For the reference fuselage, the measured drag was quite close to the calculated skin friction level. With the nacelles installed, the incremental increase in the drag at Mach numbers from 0.7 to 0.97 was approximately equal to the increase in skin friction drag associated with the larger wetted area nacelles installed case. This comparison indicates that the pressure drag of the isolated nacelles was essentially cancelled out when the nacelles were installed with the airframe. We found that this favorable effect was quite sensitive to inlet cowl geometry. When a cowl with a more blunt contour than the NACA-1 was tested, a relatively large adverse effect occurred at these speeds.

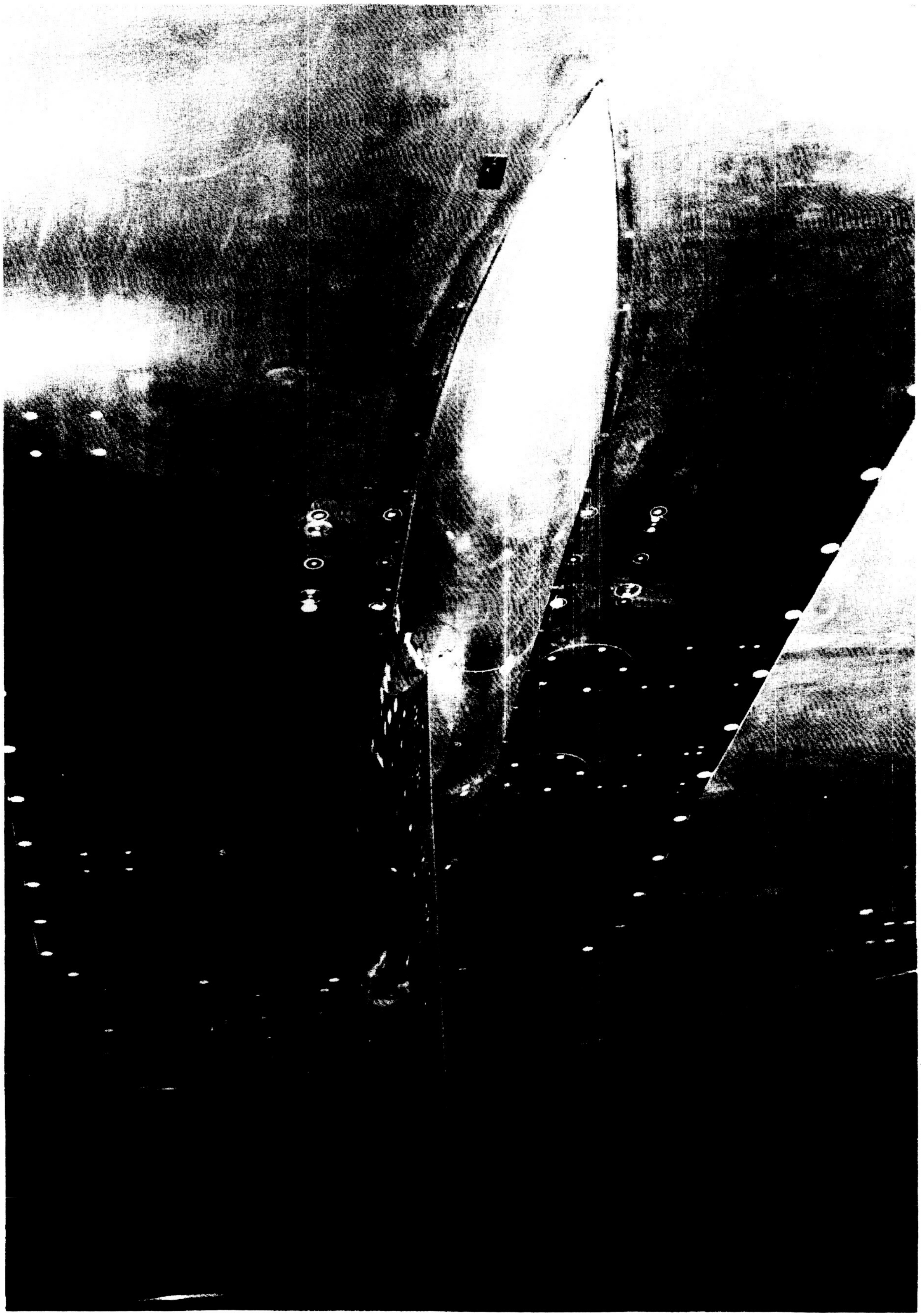

Figure 8. Over-the-Wing Half Span Model (see next page for explanation)

Figure 8. Over-the-Wing Half Span Model

An example of an unconventional propulsion system installation is shown in Figure 8. This model is presently being tested in the Lewis 8 x 6 SWT to investigate integration of over-the-wing (OTW) type of nacelle installation. This test is a joint NASA-Douglas Aircraft test to support the QCSEE engine program. The cruise Mach number for this particular installation is 0.72. Both flow thru and powered simulator nacelles will be tested. Some of the important features of this model are:

1. One and two nacelles with variation in the inboard nacelle spanwise position.
2. Four different nozzle designs.
3. Variations in local wing geometry in the region where the exhaust flow passes over the wing.
4. Supercritical wing.

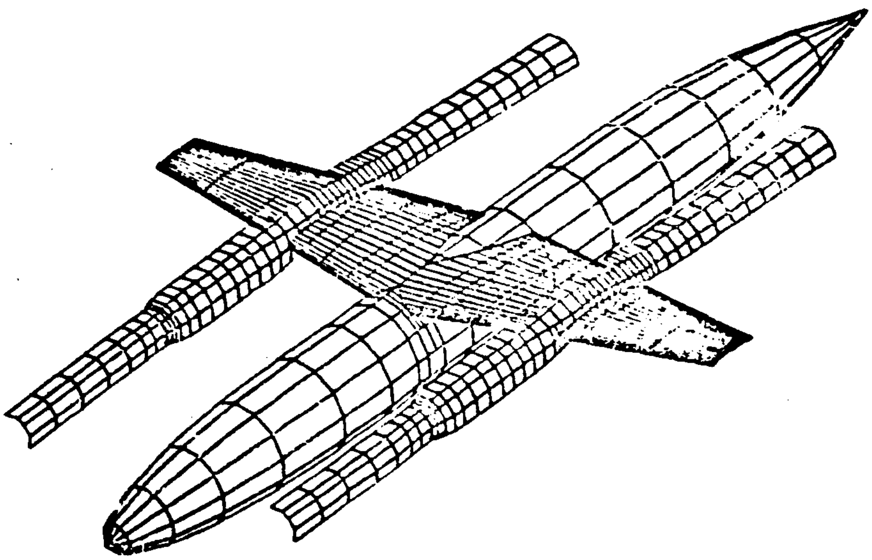

Figure 9. 3-D Newmann Representation of OTW Model

An extensive aerodynamic design effort was done on this model. The main analytical tool used in this design was the 3-D Neumann Lifting Potential flow program. Shown here is a graphical representation of how the model was paneled up for this program.

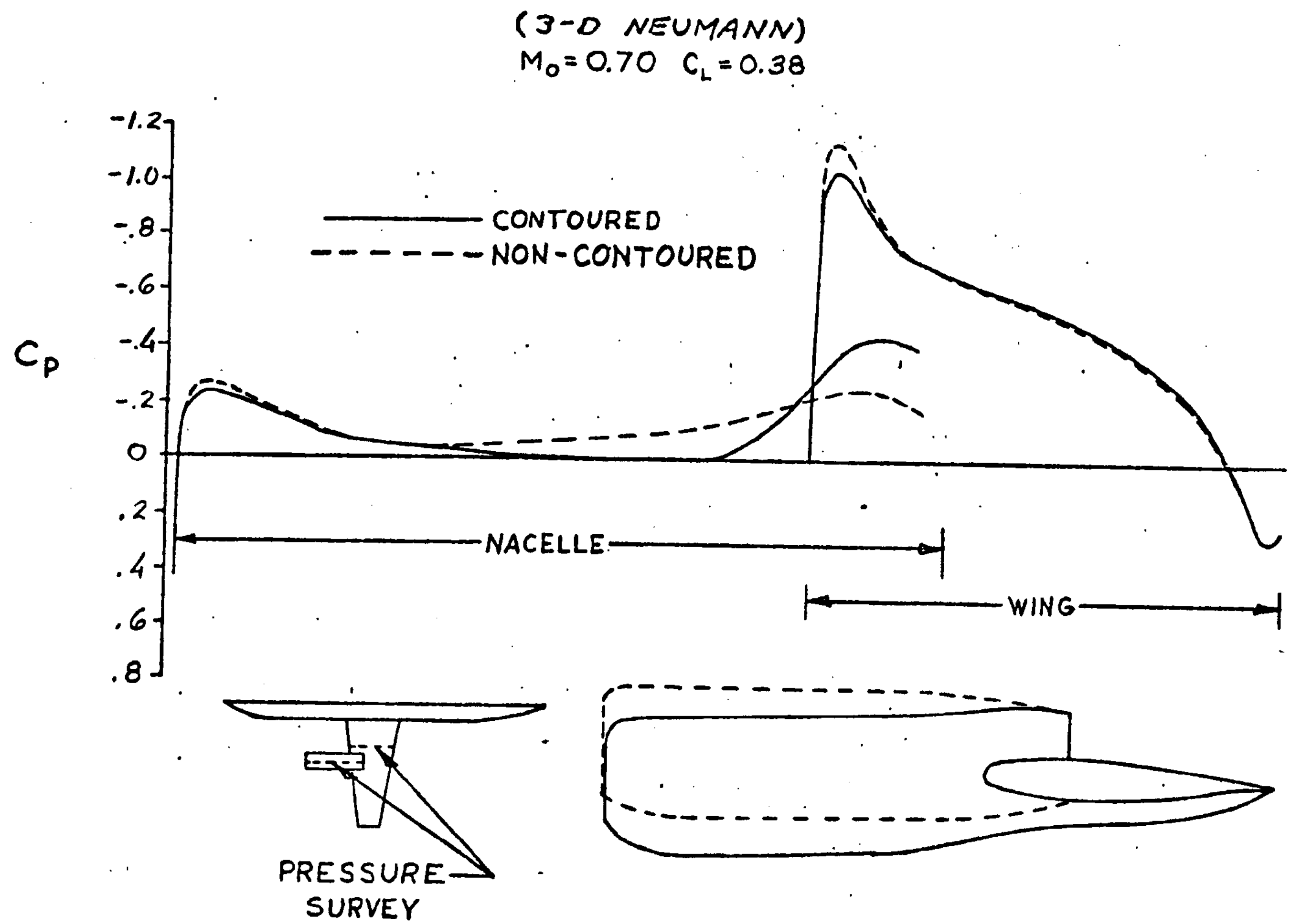

Figure 10. Effect of Nacelle Streamline Contouring on Nacelle and Wing Pressure Distributions

Some of the results from the 3-D Neumann are shown above. Computed pressure distributions on a straight nacelle are compared with those for a nacelle that was contoured to follow the stream sheets that pass over and under the wing at the cruise condition. The streamline contouring resulted in an improvement in the flow quality in the vicinity of the nozzle and wing region. The minimum pressure on the wing was increased (reduced Mach number). The pressures on the nozzle were more closely matched to wing surface pressures.

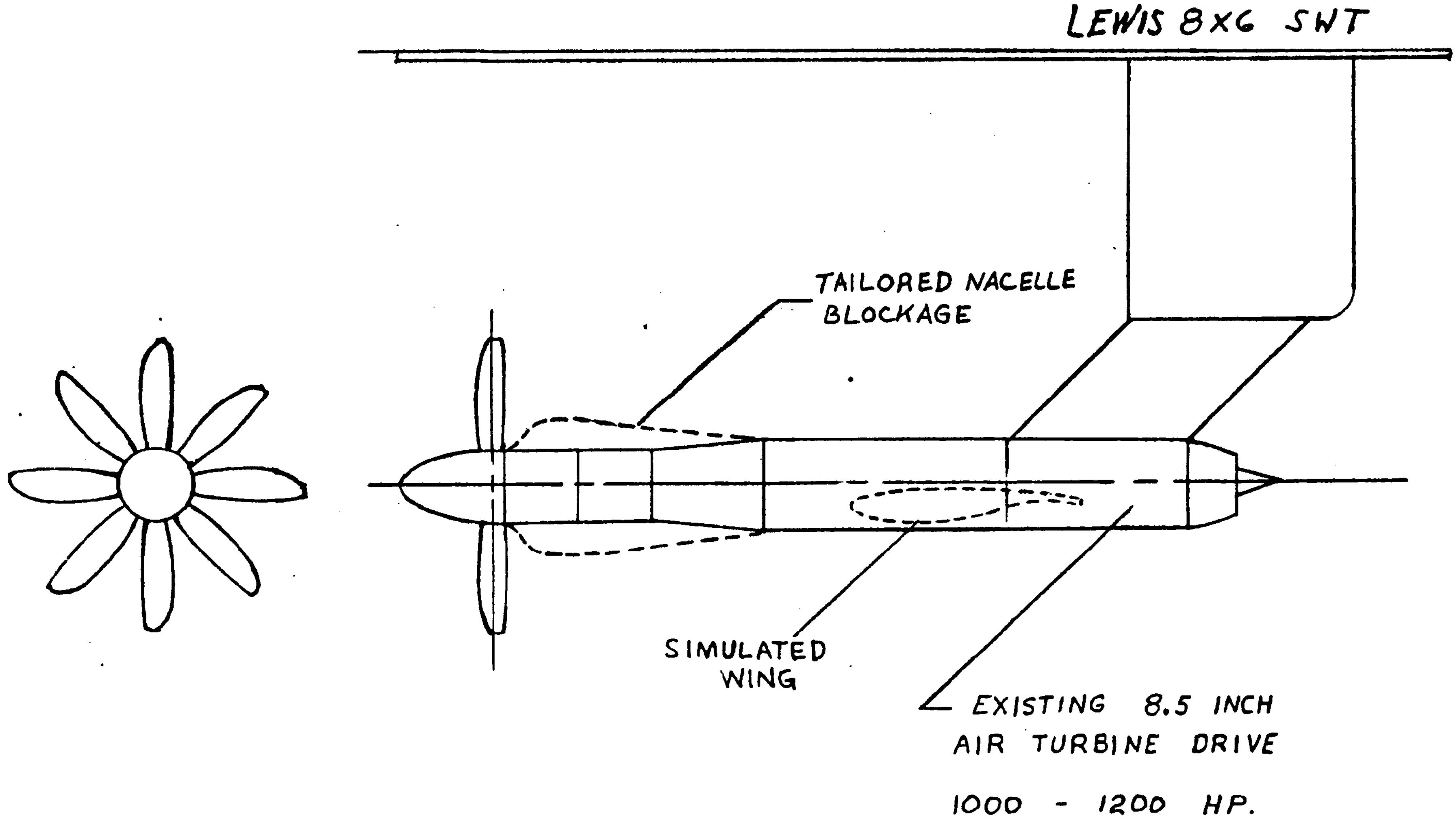

Figure 11. Isolated Turboprop Tests - M_{cr} = 0.75 - 0.9 (see explanation next page)

Figure 11. Isolated Turboprop Tests - M_{cr} = 0.75 - 0.9

Because of the energy shortage and high fuel prices it is highly desirable to reduce aircraft drag and improve propulsion system efficiency. Two new propulsion system concepts we have come up with at the LeRC are the high speed turboprop and the ducted fan. These concepts show a large potential for reducing energy consumption (10 to 25%) compared to the conventional high BPR turbofan. These potential improvements would be obtained through increased propulsive efficiency.

NASA Lewis has recently initiated a high speed (M_{cr} = 0.8) turboprop aerodynamic technology program. The model shown in the figure will be used to do part of this work. An 8½" air drive turbine will be used to drive 30" diameter highly loaded propellers. This turbine is capable of producing over 1000 hp. The propellers will have eight blades and be designed for M = 0.8 cruise at 35,000 feet. They will be tested in the Lewis 8 x 6 SWT. Increased propeller efficiencies may be achieved if tailored nacelle blockage shapes behind the propellers can be designed to suppress the Mach number in the propeller plane (and reduce propeller compressibility losses) without incurring high drag themselves. These blockage shapes will be investigated on this model.

These very high power loading propellers (SHP/D^2_{prop}) = 70 at take off) have significant swirl thrust losses (6 to 8%) at cruise. It may be possible to recover part of these losses using a second counter rotating propeller, fixed stators, or even the wing. These areas will also be investigated in this program. A simulated wing is shown mounted on the model. The integration of the high speed turboprop with the airframe will be further investigated on a small half-span aircraft model.

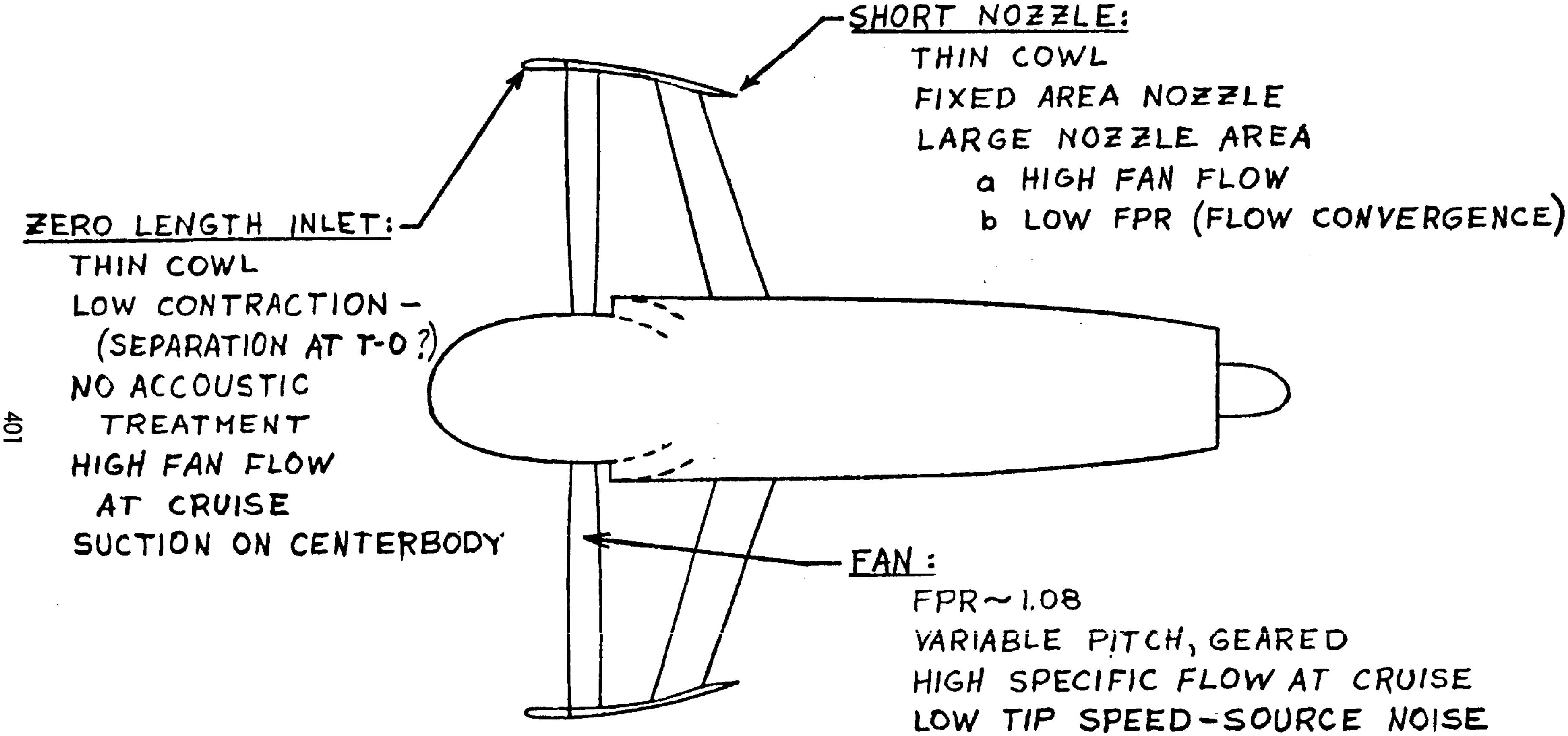

Figure 12. Ducted Fan (see explanation next page)

Figure 12. Ducted Fan

The unconventional ducted fan concept shown in Figure 12 may offer some advantages compared to the high speed turboprop. Some of these advantages are: smaller fan diameter, and reduction in swirl and tip losses. In order to make this concept viable, it must have a minimum size fan cowling as shown. Conventional size cowlings (relative to fan diameter) that are utilized with existing high BPR engines would have very high cruise drags. This drag would cancel any thrust improvement obtained by going to this very high bypass ratio concept. A short cowl requires a fixed area nozzle and a thin inlet. The fixed nozzle with a large exit area results in a high fan flow at cruise where ram effects increase the nozzle pressure ratio, and low fan flow at takeoff where there is little ram recovery. This wide weight flow range requires a variable pitch fan. Also, the thin cowl requires small flow spillage and therefore high fan flow to avoid drag problems at cruise. In addition to the high cruise flow, the low fan pressure ratio (FPR ~ 1.08) would minimize the amount of internal flow convergence at the nozzle exit and would result in a large exit stream tube size and short boattail length. At takeoff, the low fan flow dictated by the fixed nozzle minimize the sharp lip inlet losses, but some separated flow would occur during static operation.

Another concern that would have to be evaluated is the aeroelastic stability of the fan blades and cowling during operation with separated flow. This separated region would diminish as forward speed increased. No appreciable amount of acoustic treatment can be utilized with this fan cowling due to its small size. Therefore, a relatively low tip speed fan with low inherent noise would have to be incorporated. This concept is currently being analyzed for NASA by Pratt and Whitney and General Electric under two study contracts (NAS3-19121 and NAS3-19201).