N94- 33507

TRANSONIC AIRFRAME PROPULSION INTEGRATION

520-07
12050

Robert E. Coltrin & Bobby W. Sanders
NASA Lewis Research Center
Cleveland, Ohio

Daniel P. Bencze
NASA Ames Research Center
Moffett Field, California

First Annual High Speed Research Workshop
(Airframe/Propulsion Integration)
Williamsburg, Virginia
May 16, 1991

[redacted]

GOAL OF THIS PAPER

This paper will pose some issues related to transonic propulsion integration testing in HSR Phase II. It is intended to raise awareness and to generate discussion within the HSR propulsion/airframe community.

GOAL OF THIS PAPER

TO GENERATE AWARENESS IN THE HSR PROPULSION/AIRFRAME COMMUNITY OF THE ISSUES RELATING TO TRANSONIC PROPULSION/AIRFRAME INTEGRATION TESTING DURING HSR PHASE II

Figure 1

HSR PROPULSION/AIRFRAME INTEGRATION

This chart shows the time line for HSR propulsion/airframe integration program. HSR Phase I efforts are underway in both propulsion and aerodynamics. The propulsion efforts focus on cycles, inlets, combustors and nozzles that will be required to reduce nitrogen oxide (NOX) at cruise and noise at takeoff and landing to acceptable levels. The aerodynamic efforts concentrate on concepts that will reduce sonic booms and increase the lift/drag (L/D) ratio for the aircraft. The Phase II critical propulsion component technology program will focus on large scale demonstrators of the inlet, fan, combustor and nozzle. The hardware developed here will feed into the propulsion system program which will demonstrate overall system technology readiness, particularly in the takeoff and supersonic cruise speed ranges. The Phase II aerodynamic performance & vehicle integration program will provide a validated data base for advanced airframe/control/integration concepts over the full HSR speed range. The results of this program will also feed into the propulsion system demonstration program, particularly in the critical transonic arena.

HSR PROPULSION/AIRFRAME INTEGRATION

PHASE II

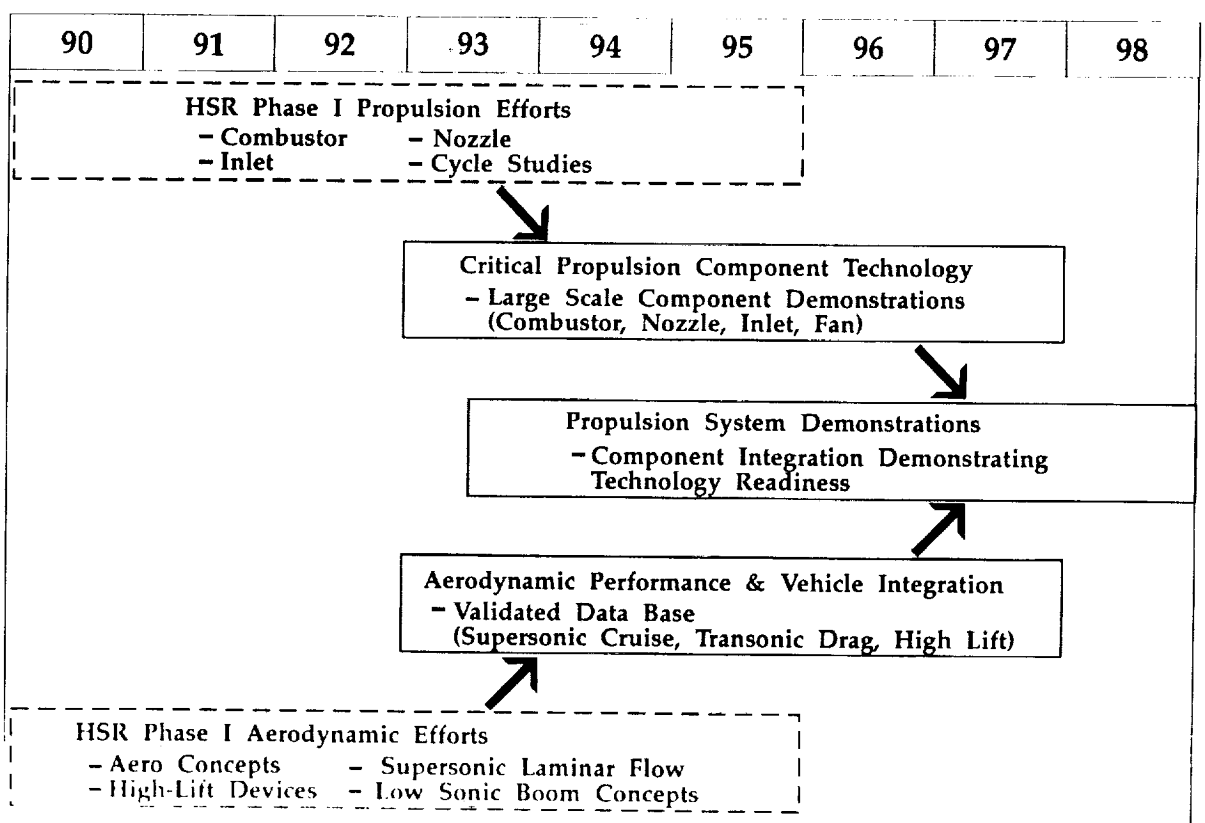

Figure 2

BACKGROUND

During the High Speed Research (HSR) Phase II planning exercise leading to the July 1990 nonadvocate review (NAR) process, the main thrust of the propulsion system effort was to ground test a full propulsion system over the entire speed range. The goal is to integrate the complex, highly coupled subsystems (inlet, nozzle, fan, engine core) into a testbed propulsion system to confirm overall system compatibility and operability and to acquire a knowledge base of subsystem interactions and system dynamics. The testbed engine would be based on an existing engine of the Advanced Technology Fighter (ATF) class. This system would be tested supersonically in the LeRC 10X10 foot SWT to obtain inlet and nozzle performance and to study inlet/engine stability and compatibility. Subsonic tests would be conducted in the Ames 40X80 foot WT with the engine pod installed with a wing simulator. The objectives will be to study inlet and nozzle performance and fan and nozzle acoustics at takeoff and approach conditions.

Transonicaly it was determined that the critical issues are more related to installed drag, than they are to internal inlet and nozzle performance. Testing for installed transonic drag requires a full configuration wing/body/nacelle model. There is no facility in the USA that is large enough to handle a full span or half span model sized for an ATF size engine and still be able to obtain data near Mach one. Therefore, the planned transonic testing will focus on a smaller scale wing/body/nacelle model in the Ames 11X11 foot TWT.

Background

HSR Non-Advocate Review (7/90) :
Experimental Validation of Propulsion System Performance Across the Mach Number Range

- **Supersonic → Large Scale Demonstration Engine Pod in Lewis 10-by 10-ft WT**
  - * **Internal Inlet & Nozzle Performance**
- **Subsonic (TO & L) → Large Scale Demonstration Engine Pod with Simulated Wing in Ames 40-by 80-ft WT**
  - * **Internal Inlet & Nozzle Performance**
  - * **Acoustics**
- **Transonic → Integrated Wing/Body/Nacelle Configuration in Ames 11-by 11-ft WT**
  - * **Transonic Drag**

Figure 3

TRANSONIC VALIDATION

This chart displays the goal and the strategy for the transonic validation part of the HSR Phase II Propulsion System Program. This strategy was developed during the NAR Phase II review that took place in July of 1990. Since no USA propulsion transonic wind tunnel is capable of testing a large scale wing/body/nacelle, a smaller scale model must be employed. The 11 foot transonic tunnel at Ames is most suitable for this type of testing. The proper test rigs and test techniques have been developed over years of testing in this facility. Therefore, the wing/body/nacelle models should be sized to be compatible with this facility. Two types of models were envisioned. A full span model with flow-through nacelles to establish the reference force and moment data and a semi span model with two propulsion simulators to obtain inlet/nozzle interactions with both flows established at the same time. Increments to the data with the full span model will be obtained with the powered semi span model. Therefore, models must be sized small enough to be compatible with the 11 ft. wind tunnel but large enough to employ propulsion simulators.

HSR PHASE II - PROPULSION SYSTEM
TRANSONIC VALIDATION

GOAL: TO DEMONSTRATE TECHNIQUES FOR PROPULSION-AIRFRAME INTEGRATION WHICH WILL MINIMIZE INSTALLED AIRPLANE DRAG

NAR STRATEGY

- NO CURRENT U.S. WIND TUNNEL CAN PROPERLY TEST A LARGE SCALE WING/BODY/ENGINE POD AT TRANSONIC SPEEDS
- THEREFORE, SMALLER SCALE WING/BODY/NACELLE MODELS MUST BE EMPLOYED
- SELECT SCALES THAT ARE COMPATABLE WITH AMES 11 FT. WIND TUNNEL
  - FULL SPAN FLOW THROUGH - REFERENCE
  - SEMI SPAN WITH TWO PROPULSION SIMULATORS - INCREMENT
- SEMISPAN SCALE MUST BE LARGE ENOUGH TO UTILIZE PROPULSION SIMULATORS
  - INLET/NOZZLE INTERACTIONS

Figure 4

SCHEDULE
TRANSONIC CRUISE

This chart shows the proposed schedule for the transonic cruise portion of the aerodynamic performance & vehicle integration HSR Phase II Program. This is shown to demonstrate that the airframe will be developed through a series of tests at LaRC and Ames leading up to the integrated configuration testing that is the subject of this presentation. At the same time, the inlet and nozzle will be developed through a series of tests at LeRC and LaRC. It is envisioned that three full span integrated models will be built and tested; a blown nacelle model for the LaRC 16 ft. TWT, a flow-through model for the Ames 11 ft. TWT (reference model for simulator model), and a high Reynolds number flow-through model for the LaRC NTF. The main subject of this paper is the integrated semi span simulator model for the 11 ft.

HSR PHASE II - AERODYNAMIC PERFORMANCE & VEHICLE INTEGRATION

Schedule

Transonic Cruise

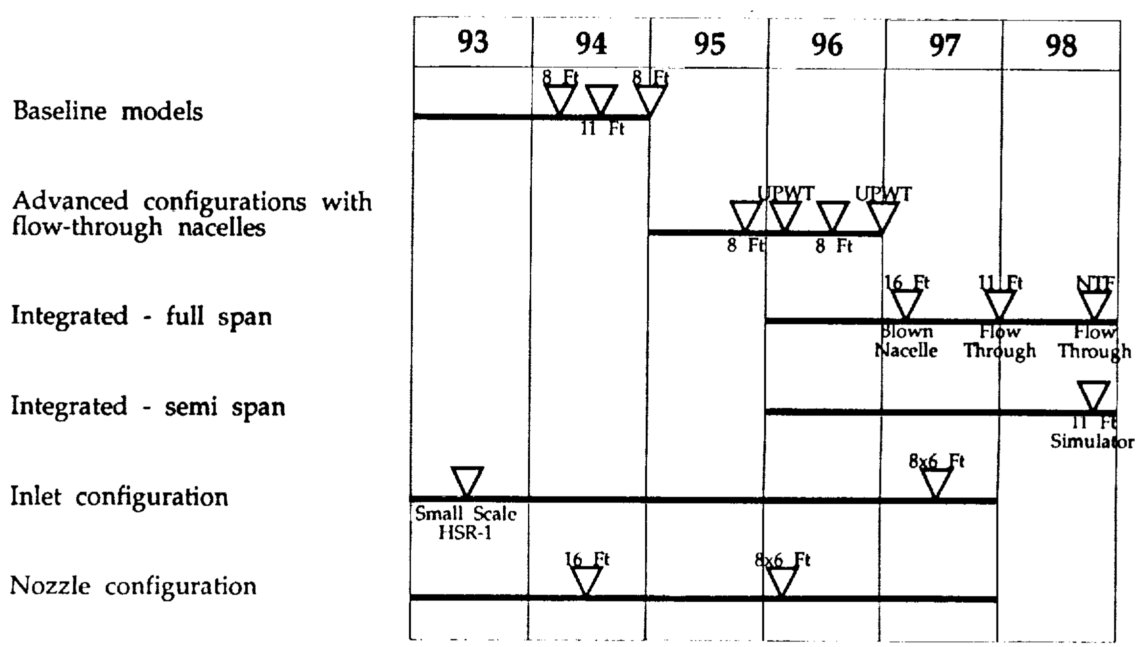

Figure 5

ISSUES

Several issues need to be resolved in planning for HSR Phase II wing/body/nacelle transonic tests. The test objective is defined to be the determination of installed drag rather than internal inlet and nozzle performance. However, the test technique to obtain this data is still open to discussion. Several questions need to be resolved:

1). Can conventional flow through inlet and blown-nozzle models be used or is a more sophisticated powered simulator model required?
2). How should the model be sized and should it be a full span model or a half span model?
3). What effect does Reynolds number have on the applicability of the proposed test results?
4). What practical issues such as data accuracy requirements and feasibility of plumbing installation need to be resolved?

ISSUES

- TEST TECHNIQUE
  - CONVENTIONAL VS POWERED SIMULATOR
  - FULL VS SEMI SPAN
- REYNOLDS NUMBER
- PRACTICAL ISSUES

Figure 6

ALTERNATIVE TEST TECHNIQUES

Generally there are two alternatives to measuring propulsion related increments to the aerodynamic characteristics of a vehicle. The first, termed conventional, uses individual inlet and nozzle models to obtain the increments associated with the inlet and nozzle streams respectively. The second approach attempts to model both the inlet and nozzle streams simultaneously, using some type of simulator device to pump the inlet and pressurize the nozzle. Both use a reference flow through aero model to obtain the basic aerodynamic characteristics of the vehicle. The conventional approach uses an inlet model with a fixed nozzle simulation to obtain the increments associated with variations in inlet mass-flow ratio (MFR) and a nozzle model with a faired over inlet to obtain the effects of nozzle pressure ratio (NPR). In the simulator approach both streams are modeled simultaneously and typically varied independently. The conventional approach is simpler but cannot resolve any mutual interactions between the inlet and nozzle flows and introduces extraneous effects with the faired inlet and fixed nozzle simulation. The simulator approach has the potential for capturing all the aerodynamic effects but is much more complicated and requires extensive flow calibrations that may compromise the ultimate data.

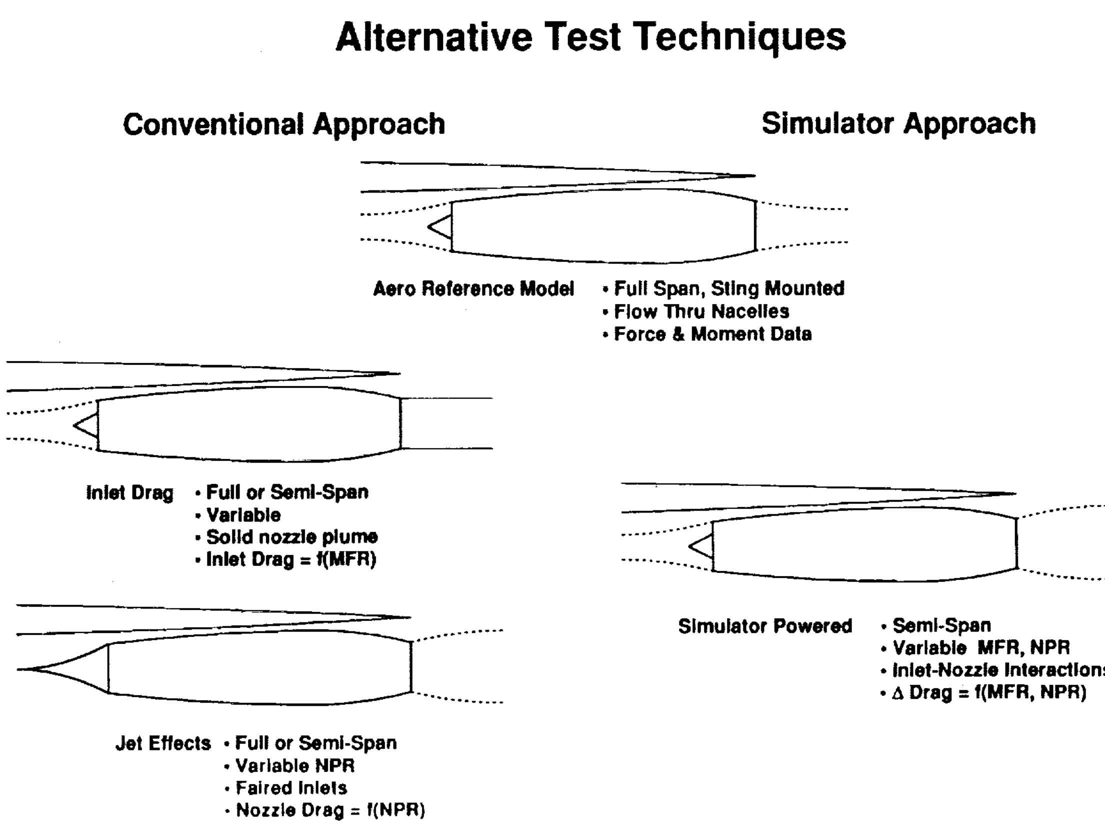

Figure 7

NOZZLE INTERACTIONS ON A SUPERSONIC STOVL CONFIGURATION

The results shown here compare similar data obtained using the conventional technique (reference aero model plus inlet and nozzle models) and a powered simulator approach (Ref. 1). Results are shown at Mach numbers of 0.9 and 1.4. The largest discrepancy between the two techniques occurred at M = 1.4 and corresponded to 20 drag counts or 4.5% of the drag of configuration. At this Mach number the trends with nozzle pressure ratio are similar, therefore the discrepancy appears to be associated with an interaction of the inlet and nozzle flow fields or possibly an effect associated with the inlet fairing.

Nozzle Interactions on a Supersonic STOVL Configuration

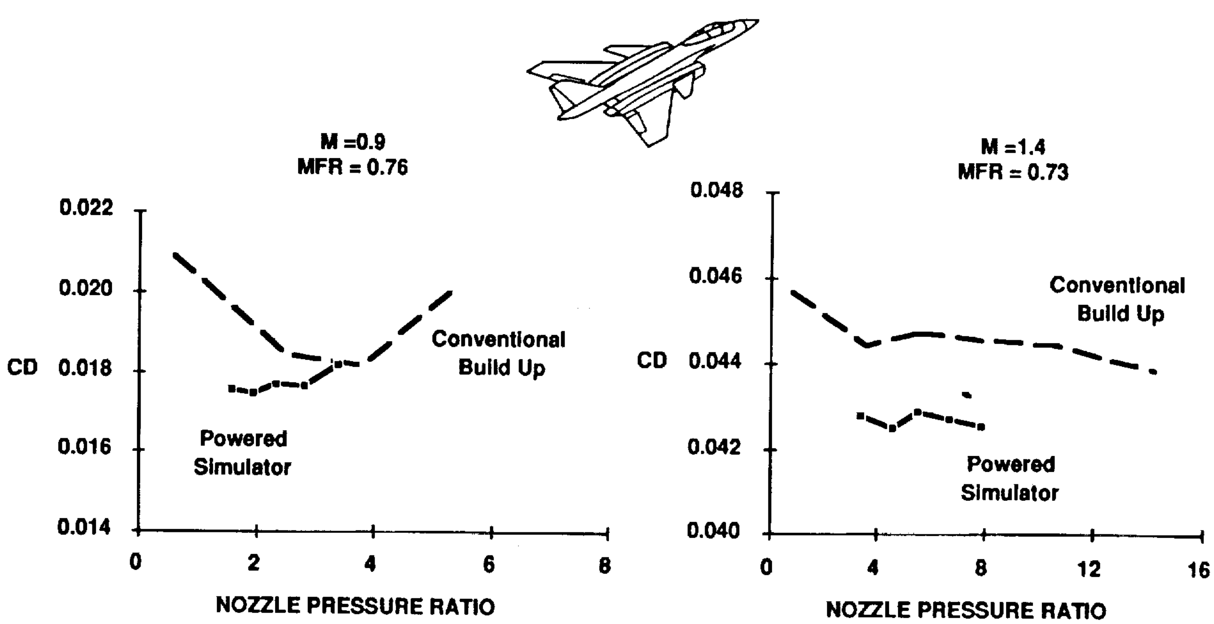

Figure 8

TRANSONIC MASS FLOW EFFECTS/BOEING SA 1150 MODEL

Shown here are the effects of inlet mass flow ratio on the overall wing-body-nacelle interference drag of the Boeing SA 1150 model with four axisymmetric nacelles located abreast at $X/C_{root} = 0.74$ (Ref. 2). The interference drag is defined as the total drag of the combination minus the isolated drag of the components at the corresponding mass-flow ratio. Since the nacelles were located relatively far aft on the wing, the overall interference effects are favorable. At Mach 1.15 reducing the inlet mass-flow ratio enhanced the favorable interference, while at Mach 0.9 and 1.4, reductions in mass-flow ratio decreased the favorable interference effects. The variations in drag over the mass flow ratios shown are 5 counts at M=1.4, 10 counts at M=1.15, and 2 counts at M=0.9. The changes in inlet mass flow represented in the figure provides a variation in system drag. If the inlet mass flow was reduced to zero as obtained by a faired inlet the effect could be expected to be rather large.

Transonic Mass Flow Effects
Boeing SA1150 Model

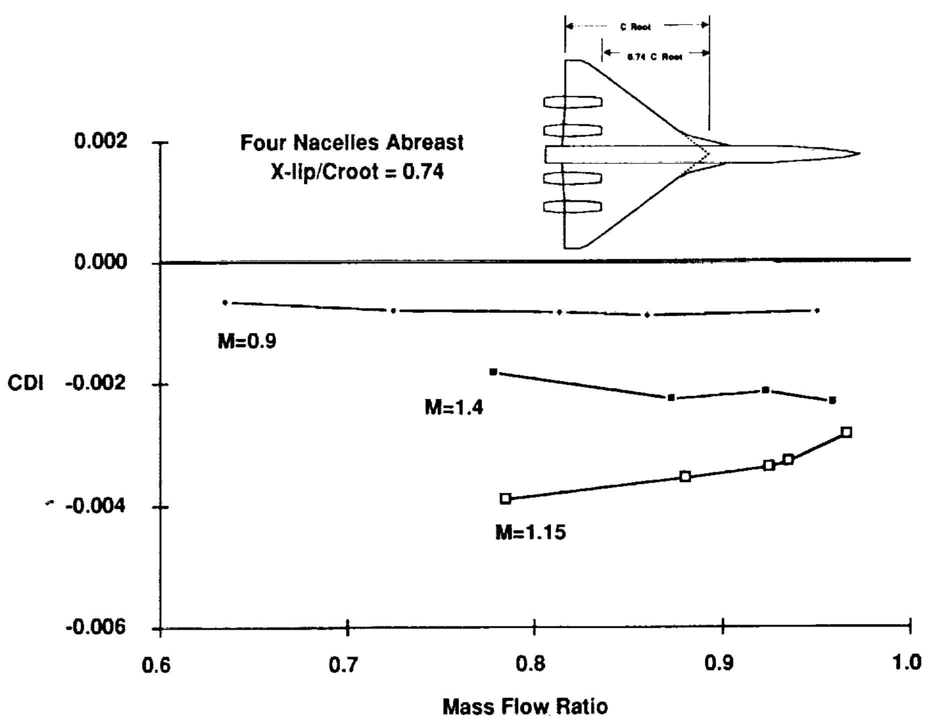

Figure 9

VARIETY OF TEST INSTALLATIONS

In the late 1960's and the early 1970's, Lewis conducted an extensive series of nozzle tests in both the wind tunnel and in flight. The F-106 aircraft was modified with two underslung J-85 engine pods, one under each wing. A wide variety of nozzle types were tested. Nozzles were first run isolated in the 8X6 Ft. SWT. Selected configurations were then tested with a 5% full span flow through model and a half-span model with a turbojet simulator in the 8X6 ft. SWT. Finally, flight tests were conducted with the F-106 aircraft.

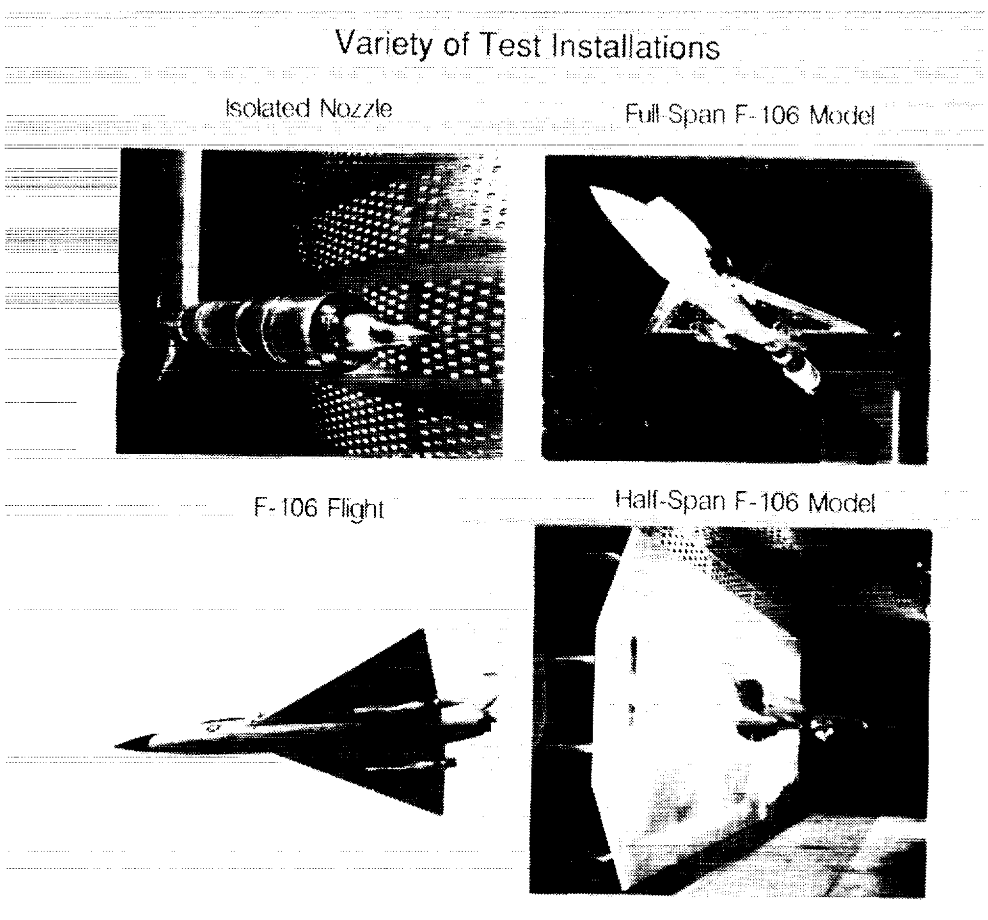

Figure 10

INSTALLED NOZZLE PERFORMANCE

This chart shows nozzle gross thrust coefficient data that was obtained from the NASA LeRC F-106 program in the late 60's and early 70's. The figure compares data obtained in flight to data obtained in the 8X6 SWT using a 22% scale semi-span model incorporating a turbojet simulator (Ref 3). The upper data was obtained for a variable flap ejector (VFE) nozzle and the lower data was obtained for an auxiliary inlet ejector (AIE) nozzle. The flight and 22 percent scale model data for the VFE nozzle agree very well from Mach 0.6 to 0.9 and agree fairly well from Mach 1.1 to 1.27. At Mach 0.95, the flight data rises above the model data and then falls below the model data at Mach 1.0. In this Mach range, a terminal shock moves off the rear of the nacelle, and the boattail flow becomes supersonic. Model blockage effects retard the passage of this shock system over the wind tunnel model with increasing Mach number, and the drag rise of the model is delayed until Mach 1.0 or higher.

The same sort of blockage effect is also present in the AIE nozzle data, but, in addition, the flight and model performance data for the AIE nozzle do not agree at Mach numbers below 0.9. Wind tunnel model data indicate that the flow through the auxiliary inlet doors of the nozzle is separated. Therefore, to be sure of the performance of nozzles which may have regions of separated flow, it may be necessary to test at the full-scale Reynolds number.

INSTALLED NOZZLE PERFORMANCE

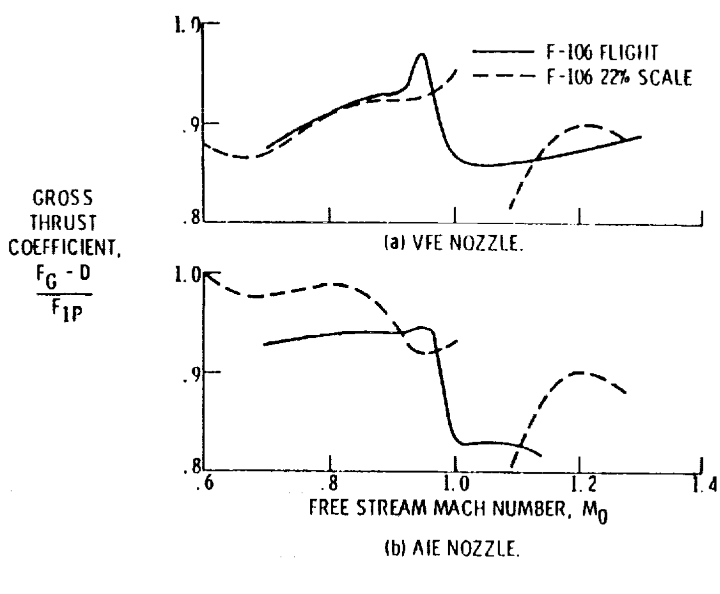

Figure 11

RELATIVE MERITS OF CONVENTIONAL VS. POWERED SIMULATOR MODELS

The decision to employ powered simulators to better model the propulsion streams is a complex one. On the surface the use of powered simulators appears to be an attractive approach, but there are many other factors to be considered. This chart outlines a number of Test Characteristics and compares the Relative Merits of the Conventional vs. the Powered Simulator approaches. Inherent in the chart is the assumption that the powered simulator model must be a semi-span model to be compatible with the existing simulator hardware. Both approaches would require very comprehensive test programs with extensive calibrations (balances, internal drag, nozzle thrust, simulator airflow and thrust) and elaborate bookkeeping schemes to achieve the required level of data quality. The simulator approach has the greatest potential of providing the best simulation, however the use of a semi-span model and attendant splitter plate in the tunnel can introduce tunnel effects that compromise the data and are very difficult to assess. On the other hand, the conventional approach must use faired inlets and reference nozzle configurations that may introduce extraneous effects that can not be sorted out. The conventional approach can use a full span model, while the powered simulator would be a semi-span model approximately twice the size of the full span model. The full span model could be tested at 2 atmospheres total pressure (Ames 11'X11' Tunnel) to achieve maximum Reynolds number. Although the powered simulator model would be approximately twice the size of the full span model, the simulators (CMAPS) themselves are limited to 1 atmosphere total pressure. Therefore, the maximum Reynolds number of the two approaches would be essentially the same. The appropriate choice is not obvious. Many factors have to be carefully considered in light of the overall test objectives.

Relative Merits of Conventional vs Powered Models

Conventional	Test Characteristic	Powered Simulator
ONE with Multiple Nacelles • Flow Thru w/ Variable MFR • Blown Nacelle w/ Faired Inlets	Number of Models	TWO: 1) Full Span Reference Aero 2) Powered Semi-Span
Complex Small Diameter Flow Thru 6-Component Balance	Balances	Conventional Airplane Balance + Simple 5 Component Floor Balance
• Internal Drag • Thrust of Blown Nacelle • Flow Thru Balance	Calibrations	Detailed Thrust and Mass Flow Calibrations of Simulators
Very Complex	Bookkeeping Scheme	Very Complex
Moderate, Potential Non-Linear Interactions of Nacelle Geometries	Degree of Simulation	High, Potential Adverse Splitter Plate and Boundary Layer Contamination
None	Internal Flow Measurements	Very Limited Inlet Data
Pt x L = 2 atm x L = 2 atm x L	Reynolds Number	Pt x L = 1 atm x 2L = 2 atm x L
Great	Level of Complexity	Very Great w/ Rotating Machinery

Figure 12

SIMULATOR/ENGINE MATCHING

Many organizations have utilized propulsion simulators during the past 25 years. At present there are two existing simulator designs within NASA that can be used to represent the engine for a system similar to the HSR. Ames has a 3 inch simulator design which has a design compressor corrected airflow of 1.65 lbm/sec. This design is referred to as CMAPS (compact multimission propulsion simulator). There are four of these simulators in existence. Lewis has a 4.3 inch simulator design which has a corrected design compressor corrected airflow of 2.85 lbm/sec. There is one of these simulators in existence. This chart shows how these two simulators would scale based on a full scale engine corrected air-flow of 550 lbm/sec. Since the prime scaling parameter would be based on corrected airflow, the CMAPS simulator would represent a 5.5% scale and the Lewis simulator a 7.2% scale.

SIMULATOR/ENGINE MATCHING

Simulator or engine	D2 in.	W #/sec	L in.	Scaling Based on: D2 %	W %	L %	Max EPR	P2 Max psia
Ames(CMAPS)	3.0	1.65	10.4	5.3	5.5	8.6	3.6	16.0
Lewis	4.3	2.85	17.7	7.5	7.2	14.6	2.8	17.0
HSR Engine	57.1	550	121	100	100	100	5.0	—

Figure 13

CMAPS AIRFLOW SCHEMATIC

The airflow through the Compact Multimission Aircraft Propulsion Simulator (CMAPS) is shown in the figure below (Ref. 1). The drive air powers the single stage turbine and drives the four stage compressor. The design compressor corrected air flow is 1.65 lbm/sec. The compressor airflow is a function of compressor RPM and be varied from approximately 1.0 lbm./sec to the design value. Compressor discharge air is mixed with the turbine drive air and exhausted either through the nozzle or bleed out of the simulator. This ability to remove air from the exhaust stream, allows the nozzle pressure ratio to be varied independent of the compressor air flow. At the design airflow the engine pressure ratio can be varied from approximately 1.6 to 3.6. The maximum physical rotor speed is 88,000 RPM.

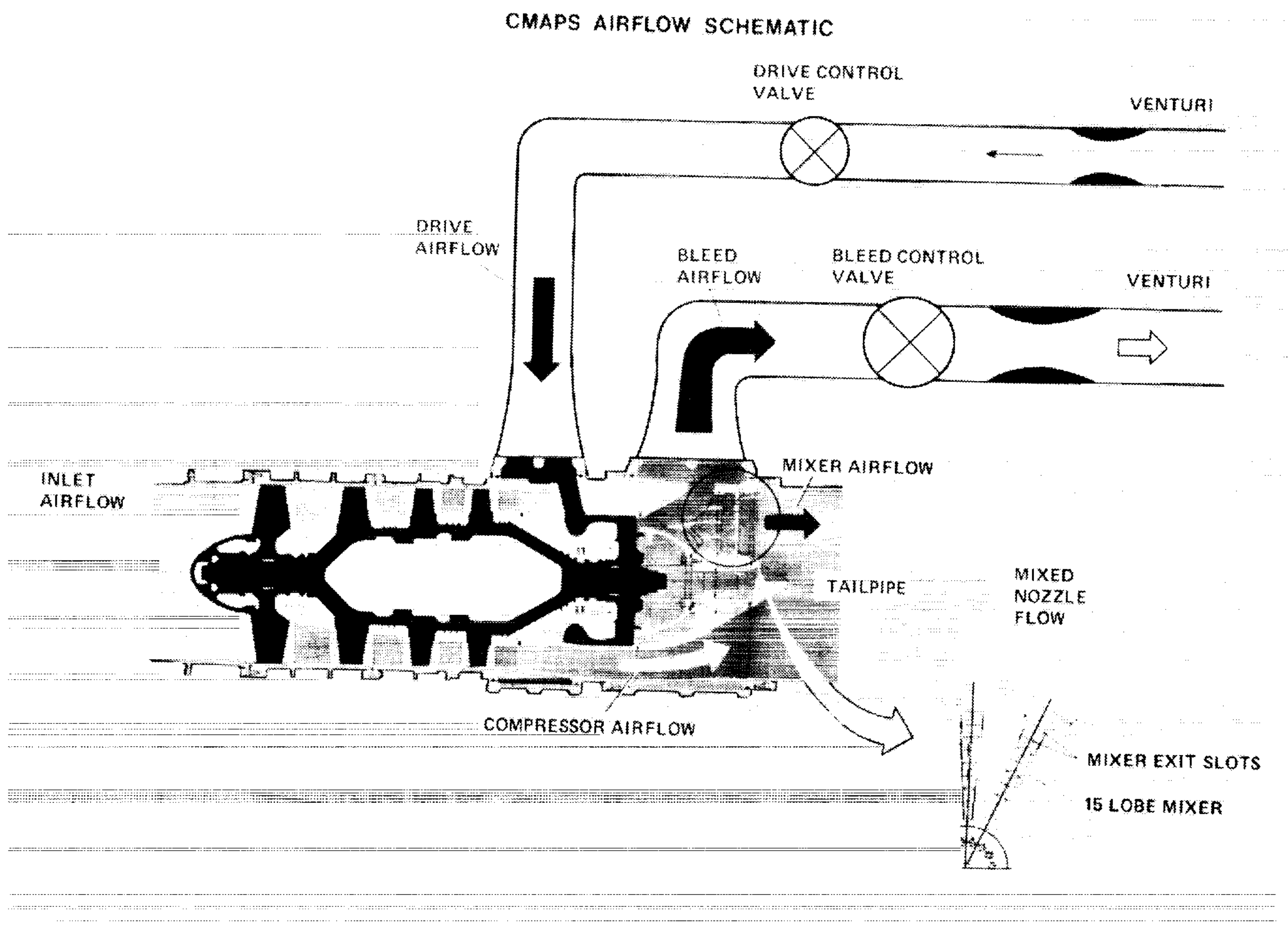

Figure 14

LEWIS PROPULSION SIMULATOR

The aerodynamic design of the Lewis turbojet simulator is based on the use of the six-stage axial compressor from the Allison T63 turboshaft engine. (Ref. 3). Its compact design and its relatively high mass flow and pressure ratio characteristics, plus the fact that it was a developed compressor in production, were the factors that lead to its selection as the critical component on which to base the simulator design. It's maximum corrected weight flow is 2.85 lbm/sec. The inlet air is compressed by the compressor and supplied to the nozzle through an annulus around the three-stage turbine. The turbine is powered by an external supply of 450-psia air that could be heated to 700 F. It's maximum physical rotor speed is 63,000 RPM. The drive air was supplied to an annular chamber around the engine and then through five of the six struts of the mid frame to an inner chamber feeding the turbine. (The top strut, which was aligned with the turbine air supply line, was blocked to obtain better distribution of the flow.) The air expands through the turbine and discharges into an annulus and then is mixed with the stream from the compressor. To obtain a desired ratio of nozzle throat area to engine inlet area and maintain proper nozzle pressure ratios, makeup air is supplied to fill the nozzle. The makeup air is supplied to an annular chamber from which it is fed to the nozzle through a 1/8-inch annulus concentric with the annulus from the compressor turbine. To improve uniformity of the flow, the three concentric streams are passed through a "daisy" mixer before entering the nozzle. The mixer was designed to rearrange the flow into eight radial lobes while maintaining a constant flow area in each of the three flow passages.

LEWIS PROPULSION SIMULATOR

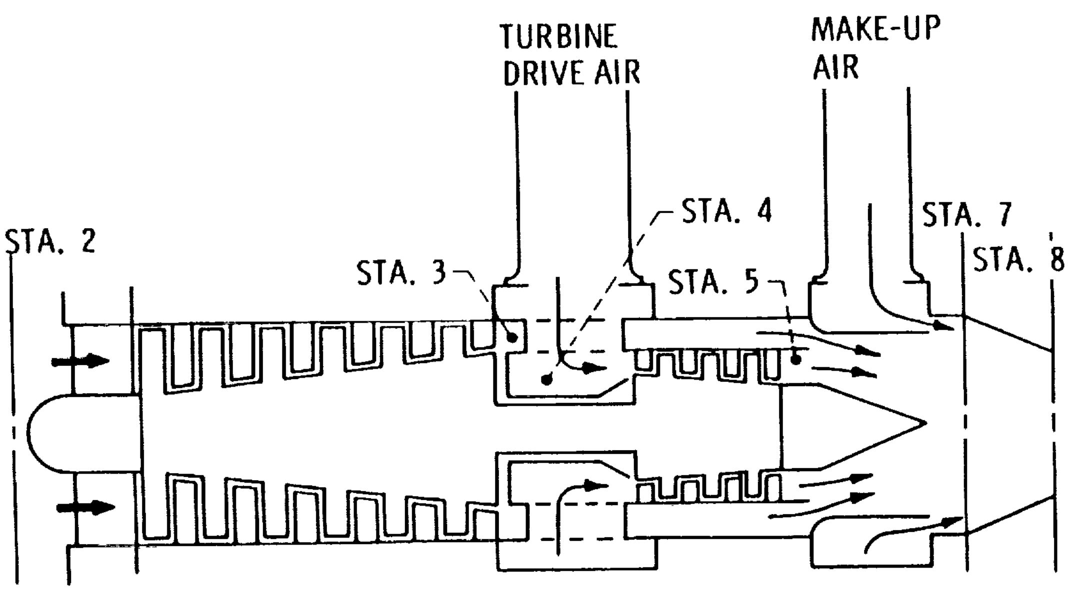

Figure 15

MODEL SCALING

The appropriate scale for various test models is a function of the type of test and the proposed test facility. This chart illustrates the resulting model characteristics as a function of various scaling parameters for the Ames 11 by 11 ft. wind tunnel and a full scale aircraft that is 300 feet long, has a wing span of 135 feet, a maximum cross sectional area of 225 square feet and an engine that has a maximum corrected air flow of 550 lbm. sec. The first three categories correspond to typical constraints in the Ames 11 ft. tunnel for full span models, namely, a blockage of 1/2%, a span of half of the tunnel width (5.5 ft.), and an overall model length of 6 ft. The only one of these categories that meet all three of the full-span criteria is the model scaled to the 6 ft. length which results in a very small 2% scale model. The blockage of this model would be .08% and the wind span would be 2.7 ft. The next category assumes a semi-span model scaled to a 16 ft. length which is a reasonable semi span length for the 11 ft. test section which is 22 ft. long. This model would be at 5.3% scale with a semi-span of 3.6 ft. and a blockage of .26%. As with the full span models, the length is the critical parameter in determining the maximum semi-span scale. The fourth category is a model sized to the 2.85 lbm/sec of the Lewis powered simulator. This results in a 7.2% scale model that is 21.6 ft. long with a wind semi-span of 4.9 ft. and a blockage of .49%. This model is too long for the 11 ft. tunnel. The last category is sized to the 1.65 lbm/sec of the Ames CMAPS simulator. This results in a 5.5% scale model that is 16.4 ft. long with a semi-span of 3.7 ft. and a blockage of .28%. When considering each of the resulting models from this scaling exercise, this semi-span model sized to match the CMAP airflow seems to be the best choice.

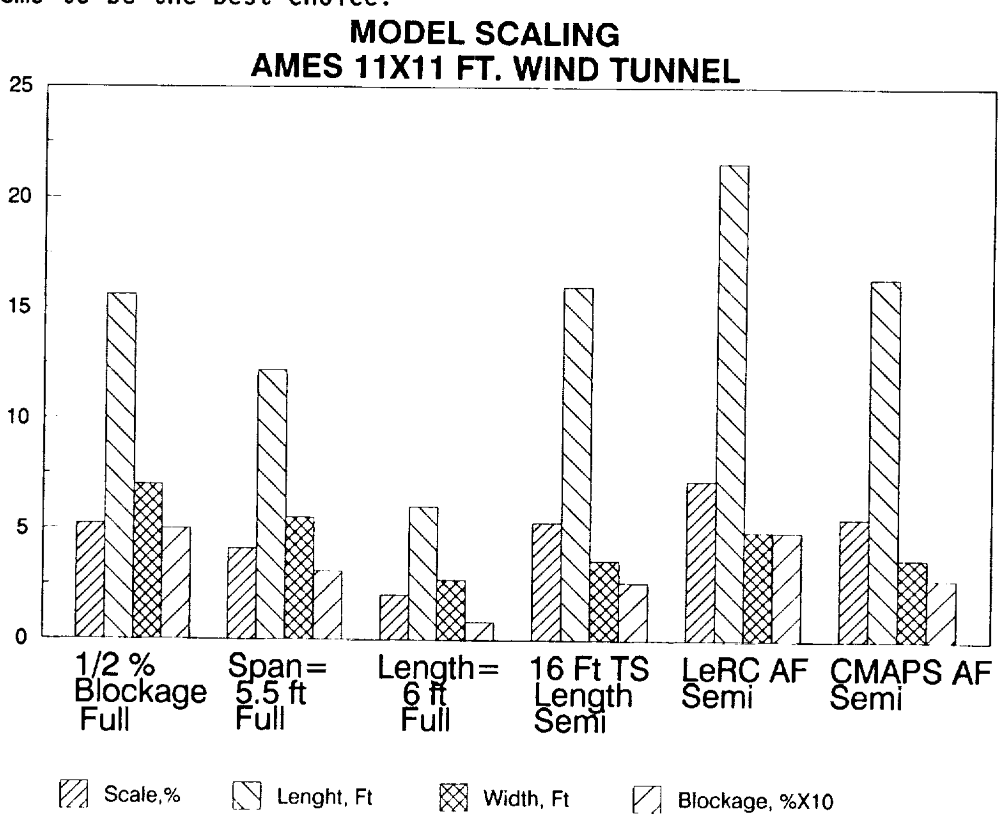

Figure 16

BOATTAIL DRAG

During the F-106 nozzle program, it was found that for a given configuration, boattail drag could be a strong function of Reynolds Number. This figure shows a generic curve of boattail drag vs. Reynolds Number that was generated from the F-106 Program for an arc-conic boattail at subsonic Mach numbers of 0.6 to 0.9 (Ref. 4). The observed drag variation with Reynolds number is the result of changes in the boundary layer thickness and separation on the aft part of the boattail. Pressure distributions on a typical nozzle boattail are shown schematically in this figure for three values of Reynolds number. The solid lines are typical of the observed pressure distributions. The dashed lines represent the pressure distribution for inviscid flow. Drag is low at the very high Reynolds numbers. Due to thin boundary layer, the flow remains attached over a major portion of the boattail. This results in a large expansion at the boattail shoulder but allows the flow to recompress to relatively high pressure on the aft boattail, which offset the low pressures at the shoulder. As the Reynolds number is decreased the boundary layer becomes thicker. With the thicker boundary layer the flow cannot traverse the adverse pressure gradient as far and will separate sooner. As the separation on the aft boattail increases, the recompression is lost and drag increases. As the Reynolds number is lowered still further the boundary layer becomes thicker causing separation to occur closer to the boattail shoulder which decreases the overexpansion. Eventually the beneficial effects of increasing pressure at the shoulder become large enough to offset the adverse effects of increased separation on the back of the boattail. Drag thus reaches a peak and then begins to decrease with further lowering of Reynolds number.

BOATTAIL DRAG

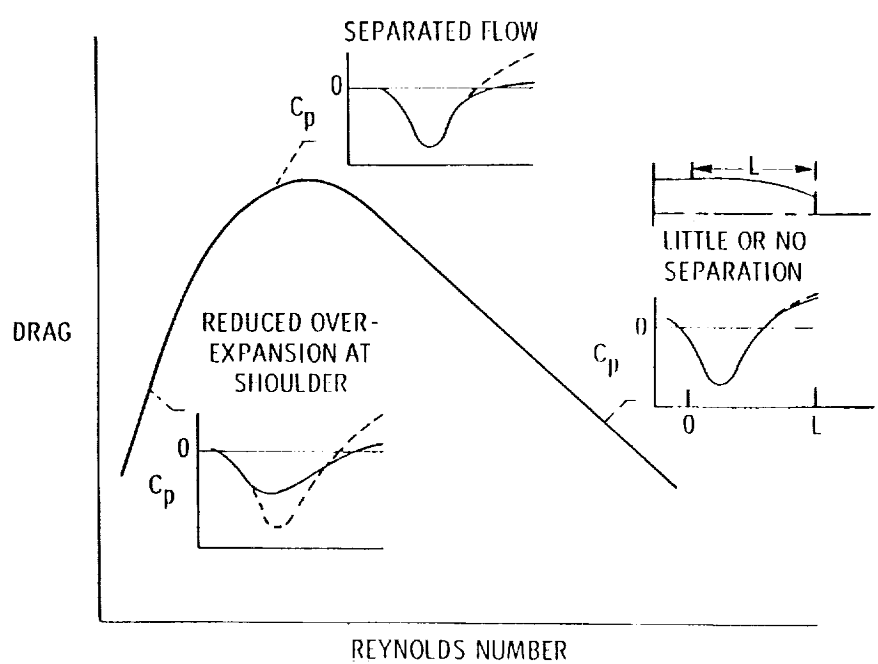

Figure 17

e-5

PRACTICAL ISSUES

There are several practical issues that must be addressed for either the conventional three model approach or the simulator approach. Of prime importance is the question of what data accuracy is required. First the mission sensitivity must be known so that the significance of a drag count can be determined. Knowing the mission sensitivity, the required model accuracy in drag counts can be determined. The type of model (full span or semi span, conventional or simulator model) will determine the number of models required, the balance configuration and the accounting system to be used. If a simulator approach is chosen, the issue of mounting the simulator and plumbing the required airflow lines through the wing without violating the mold lines of the configuration must be addressed. This will be more of a problem for an HSCT type of configuration than for past efforts with fighter configurations which had greater internal volume available for instrumentation and plumbing.

PRACTICAL ISSUES

- MOUNTING AND PLUMBING OF SIMULATOR(S) WITHOUT VIOLATING MOLD LINES OF VEHICLES
- ACCEPTABLE LEVEL OF ACCURACY
  - MISSION SENSITIVITY
  - $\pm$ X DRAG COUNTS
  - ABSOLUTE VS INCREMENTS

Figure 18

CONCLUSIONS/RECOMMENDATIONS

Reviewing past conventional models versus powered model data reveals that powered models appear to offer an accuracy advantage. Models sized for the ARC 11 ft. will be constrained by length but a semi-span model sized to the CMAPS airflow appears to be a reasonable size for this facility. Low Reynolds number compared to flight may be a problem for some propulsion system configurations and the CMAPS powered model does not offer any Reynolds number advantage. The information presented in this paper resulted from a very cursory look at the overall issue of transonic airframe propulsion integration testing for HSR. The purpose of this paper is to create an awareness of these transonic testing issues within the HSR propulsion/airframe community. The recommendation is that a much more detailed study of the practical issues is required either in HSR Phase I or early in HSR Phase II.

CONCLUSION/RECOMMENDATIONS

CONCLUSIONS:

- POWERED MODEL APPEARS TO OFFER ACCURACY ADVANTAGE
- MODELS WILL BE CONSTRAINED BY LENGTH
- CMAPS POWERED SEMI MODEL APPEARS REASONABLE FOR ARC 11-FT WIND TUNNEL
- REYNOLDS NUMBER MAY BE PROBLEM FOR SOME NOZZLE CONFIGURATIONS
  - NO ADVANTAGE FOR CMAPS POWERED SEMI SPAN MODEL

RECOMMENDATIONS:

- HSR PHASE I OR EARLY PHASE II STUDY TO INVESTIGATE PRACTICAL ASPECTS OF ALTERNATIVES

Figure 19

REFERENCES

1. Zilz, D.E.; Wallace, H.W.; Hiley, P.E.: Propulsion and Airframe Aerodynamic Interactions of Supersonic V/STOL Configurations Volume IV: Final Report - Summary. NASA CR-177343, 1985.

2. Bencze, Daniel P.: Experimental Evaluation of Nacelle-Airframe Interference Forces and Pressures at Mach Numbers of 0.9 to 1.4. NASA TM X-3321, 1977.

3. Steffen, F.W.; Satmary, E.A.; Vanco, M.R.; Nosek, S.M.: A Turbojet Simulator for Mach Numbers up to 2.0. NASA TM X-67973, 1972.

4. Wilcox, F.A.; Chamberlin, R.: Reynolds Number Effects on Boattail Drag of Exhaust Nozzles From Wind Tunnel and Flight Tests. NASA TM X-71548, 1974.

THIS PAGE INTENTIONALLY BLANK