IN-09
26557
P-39

NASA Technical Memorandum 102698

Overview of the Applied Aerodynamics Division

JUNE 1991

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665-5225

FOREWORD

Effective February 12, 1989, a reorganization of the NASA Langley Research Center Aeronautics Directorate took place. One of the results of this action was the formation of the Applied Aerodynamics Division (AAD). This division is composed of elements of the former Low-Speed, Transonic, and High-Speed Aerodynamics Divisions. One of the goals of this reorganization is to place most of the major aerodynamic test facilities in the same organization.

The purpose of this publication is to present a brief overview of this new division and to describe the missions, facilities, programs, and recent research highlights of each of its branches. It is anticipated that this document will be of use to those interested in testing in the facilities as well as those only interested in learning more about the existing capabilities.

Your comments and suggestions for improvement or additions to this document are welcome. When major changes in the division occur, this document will be revised. At that time, any suggested improvements can be incorporated. For additional information, contact the AAD at Mail Stop 285, NASA Langley Research Center, Hampton, VA, 23665-5225, (804) 864-3520 or (804) 864-5023 (facsimile).

William P. Henderson

Langley Research Center

Organization Chart - May 1991

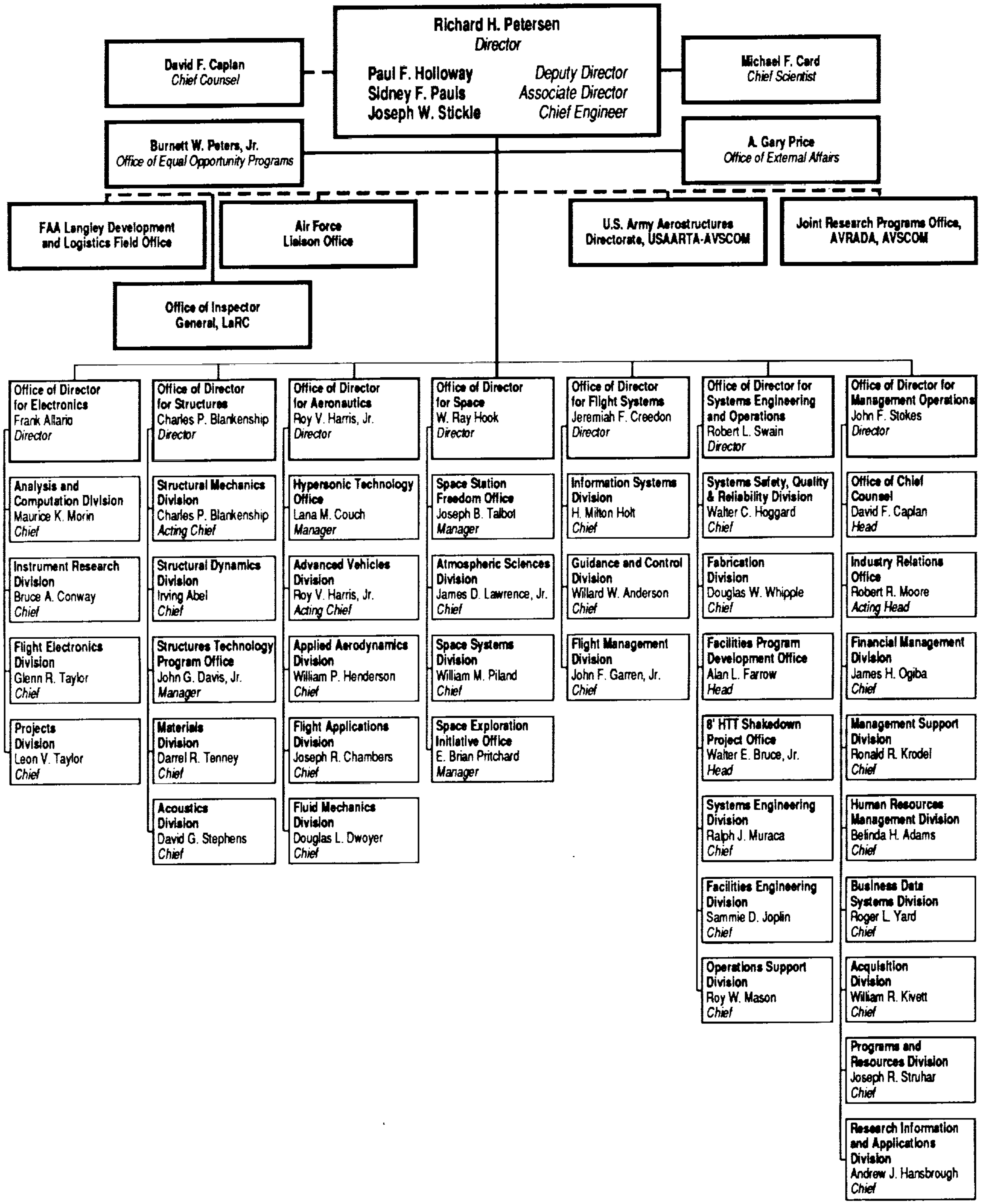


Figure 1

APPLIED AERODYNAMICS DIVISION

The Applied Aerodynamics Division (AAD) is one of four divisions under the Office of Director for Aeronautics at the NASA Langley Research Center (LaRC). This is illustrated in the LaRC organization chart presented in figure 1. As a result of the 1989 reorganization, most of the major subsonic, transonic, and supersonic aerodynamic test facilities, as well as some of LaRC's hypersonic facilities, are now the responsibility of this division.

The AAD consists of about 115 full-time civil service employees distributed in five branches, one office, and the division office. This organization is illustrated in the division organization chart presented as figure 2. This chart gives the names of the heads of the organizations as well as a list of the facilities assigned to each branch or office. The distribution of the personnel and skill mix among the offices and branches is illustrated in a bar graph. (See fig. 3.) The Subsonic Aerodynamics Branch (SAB) is supplemented by eight U.S. Army employees under a formal agreement with the Aviation Systems Command. These personnel are responsible for basic rotorcraft aerodynamic research in the SAB facilities.

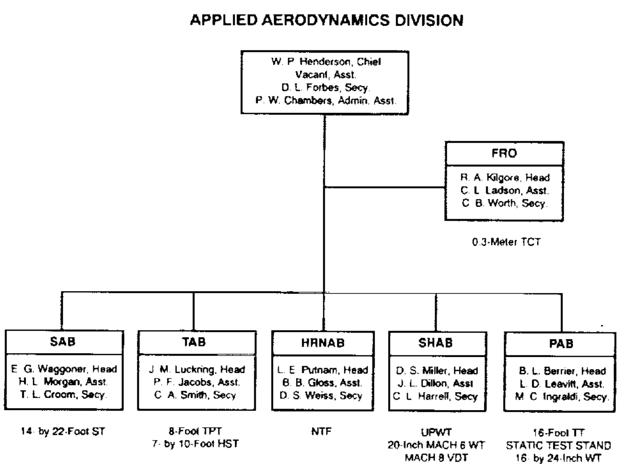


Figure 2

It is also important to note in figure 3 that the majority of the division personnel are research engineers. Most support services, as well as additional research effort, are provided to the division by personnel of other LaRC divisions, support-service contractors, and universities.

The wind-tunnel facilities assigned to the AAD are situated in seven locations in the West Area and one location in the East Area of the LaRC. The aerial photographs in figures 4 and 5 indicate the locations of the facilities. In most cases, the research personnel associated with the facilities are located in office space adjacent to the facility.

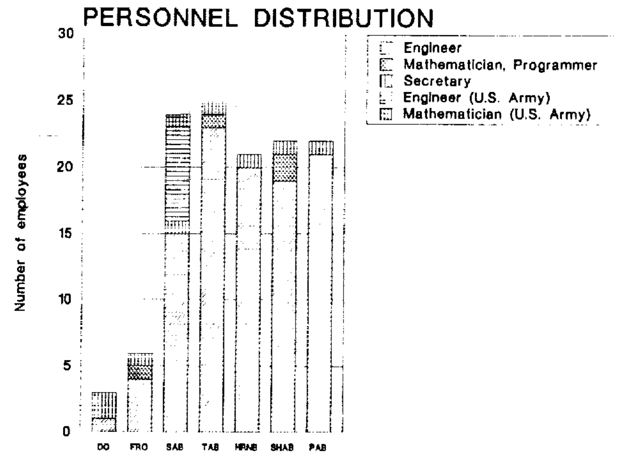
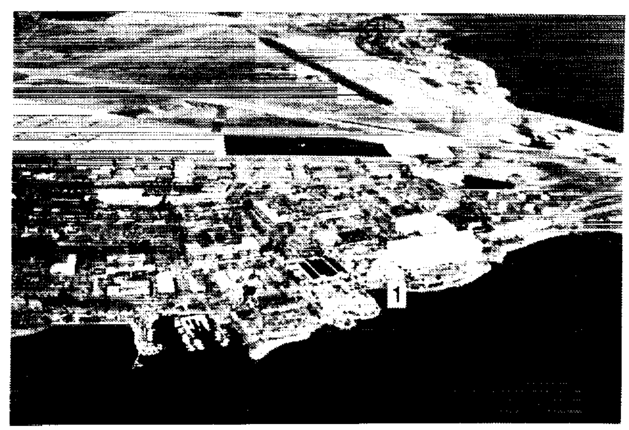


Figure 3

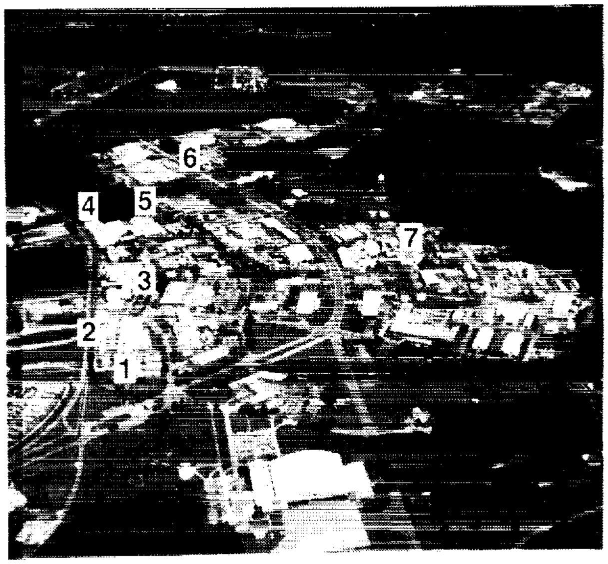


1. 8-Foot Transonic Pressure Tunnel

Figure 4

THE MISSION

The mission of the AAD is to advance the state-of-the-art of aircraft and missile technology from subsonic to hypersonic speeds by developing and applying new innovative aerodynamic technologies. Particular emphasis is placed on Reynolds-number effects, high angle-of-attack flows, stability and control, performance, innovative configurational concepts, new multifunctioning nozzles, advanced engine installation concepts, and the interaction of these technologies with technologies from other disciplines in a multidisciplinary manner.



1. National Transonic Facility
2. 0.3-Meter Transonic Cryogenic Tunnel
3. 16-Foot Transonic Tunnel
4. 14- By 22-Foot Subsonic Tunnel
5. 7- By 10-Foot High-Speed Tunnel
6. Unitary Plan Wind Tunnel
7. 20-Inch Mach 6 Wind Tunnel
   Mach 8 Variable-Density Tunnel

Figure 5

The research emphasis in the AAD is on the development of advanced aerodynamic technology which will play a key role in future aerospace vehicles, incorporating such advanced concepts as active flight controls, thrust vectoring for control, laminar flow and turbulent drag-reduction concepts, closely-coupled propulsion/airframe integration, and flexible composite structures. Particular areas of emphasis included improving the aerodynamic efficiency and reducing the fuel consumption of conventional jet and turboprop transports, developing the technology for advanced military combat aircraft and missile concepts, developing theoretical and analytical methods for predicting aerodynamic characteristics for aircraft with both separated and/or attached flows, for nozzle/afterbodies, pylon/nacelle and inlet flows, jet/ejector ground interactions, and a data base for theory validation. In addition, the AAD is responsible for developing advanced experimental techniques including advanced-wall concepts for transonic wind tunnels, cryogenic wind-tunnel technology, advanced instrumentation, and magnetic-suspension systems.

Research information is generated through advanced analytical techniques making use of LaRC's large digital computer complex and through the use of unique experimental facilities, such as the 16-Foot Transonic Tunnel, the 8-Foot Transonic Pressure Tunnel, the 7- by 10-Foot High-Speed Tunnel, the 0.3-Meter Transonic Cryogenic Tunnel, the 14- by 22-Foot Subsonic Tunnel, the 20-Inch Mach 6 Tunnel, and the Mach 8 Variable Density Tunnel.

Also, the AAD is responsible for the operation of the cryogenic National Transonic Facility and the supersonic Unitary Plan Wind Tunnel.

The development of new test methods and data-reduction techniques for these facilities and the coordination of in-house research programs and cooperative studies in support of industry, the Department of Defense (DOD), and other Government agencies are areas of continuous activities.

FACILITY RESEARCH OFFICE

THE MISSION

The Facility Research Office (FRO) was created early in 1991 in response to the recognition of a need to focus and coordinate several existing research programs related to improving the design and use of wind tunnels. The primary purpose of the FRO is to advance the state-of-the-art of wind-tunnel testing with particular emphasis on cryogenic-nitrogen, liquid-helium, and heavy-gas tunnels. Areas of emphasis directly related to wind-tunnel testing include data accuracy and tunnel productivity. Areas of emphasis directly related to test techniques include elimination or reduction in tunnel-wall and model-support interference and developing improved flow diagnostic techniques.

The FRO will coordinate within AAD all in-house research and development related to facilities, test techniques, and instrumentation. Through the Small Business Innovation Research (SBIR) Program, the FRO involves industry researchers in selected areas related to improving the quality and quantity of data from wind tunnels. As required, the FRO will develop additional programs to involve industry, university, and government researchers in selected research projects.

The FRO is responsible for establishing and maintaining technical contact with researchers around the world working in the areas of tunnel development and testing techniques. As a part of this effort, the FRO oversees the production and distribution of various bibliographies and newsletters related to cryogenic wind tunnels, adaptive-wall test sections, and magnetic suspension and balance systems. The FRO is responsible for identifying areas of foreign technology related to tunnel development and testing techniques. Once identified, in concert with NASA Headquarters, the FRO will seek to implement cooperative programs related to areas of mutual interest.

The FRO is responsible for the management of the 0.3-Meter Transonic Cryogenic Tunnel (0.3-Meter TCT).

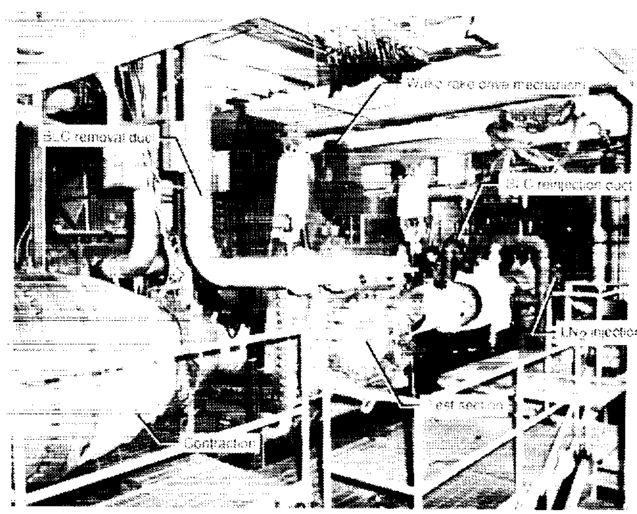
THE PERSONNEL

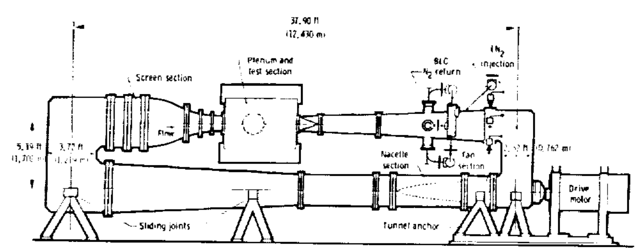
FLECHNER, STUART G.
HILL, ACQUILLA S.
KILGORE, ROBERT A. (Office Head)
LADSON, CHARLES L. (Assistant Office Head)
LEE, EDWIN E., JR.
WORTH, CATHERINE B.

THE FACILITY

0.3-Meter Transonic Cryogenic Tunnel

The 0.3-Meter Transonic Cryogenic Tunnel (0.3-Meter TCT) is a closed-circuit, fan-driven cryogenic pressure tunnel. The 0.3-Meter TCT operates over the Mach number range of 0.20 to 0.95 at stagnation temperatures and pressures from 150° F to approximately −300° F and from 1 to 6 atm., respectively. The wide ranges of pressure and temperature allow the study of Reynolds-number effects on flow phenomena up to Reynolds numbers of 100 million/foot.





The tunnel was placed in operation in 1973 as a three-dimensional pilot tunnel to demonstrate the cryogenic wind-tunnel concept at transonic speeds. During more than 15 years of operation, the 0.3-Meter TCT has run with three different test sections. Currently, the facility is operating with an adaptive-wall test section which is nominally 13 inches square and has an effective length of 55.8 inches. This test section has four solid walls with the horizontal walls (floor and ceiling) being flexible. A system of 21 computer-controlled jacks supports each of the flexible walls. The facility has motorized model-support turntables and a traversing wake-survey probe, both of which are computer controlled. For two-dimensional testing, the 0.3-Meter TCT has provisions for both active- and passive-sidewall boundary-layer control. Porous plates can be fitted into the rigid sidewalls just upstream of the model location.

CURRENT PROGRAMS

A major component of the current program of the FRO is to bring the 0.3-Meter TCT back on line following modification of the liquid nitrogen supply system. Once the tunnel is operating, we will use it to support several research and development projects. These include tests related to flow visualization and transition detection in cryogenic tunnels. We will make additional mechanical modifications to the 0.3-Meter TCT to improve the flow quality and add a 3-dimensional model support system. We will continue a program to improve both software and hardware related to the adaptive-wall test section to improve data accuracy and increase productivity.

We will evaluate the use of heavy gases for high-lift testing, possibly using the 0.3-Meter TCT for experimental verification of theoretical studies related to heavy-gas tunnels. As a part of this effort, we will publish a bibliography on heavy-gas wind tunnels. Finally, we will compare and contrast the relative merits of heavy-gas and cryogenic tunnels for high-lift testing.

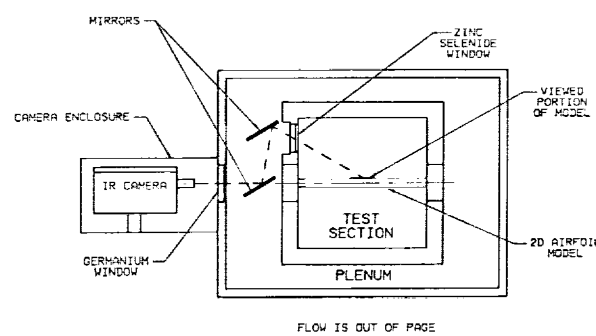
An ongoing effort of the FRO is to provide administrative and technical support to both national and international meetings related to tunnel and test-technique development. Another ongoing effort is the publication of bibliographies and newsletters related to cryogenic tunnels, adaptive-wall test sections, and magnetic suspension and balance systems. Currently in preparation are bibliographies on wall interference, support interference, magnetic suspension and balance systems, and heavy-gas wind tunnels.

HIGHLIGHTS OF RECENT RESEARCH

Transition Detection by IR Imagery

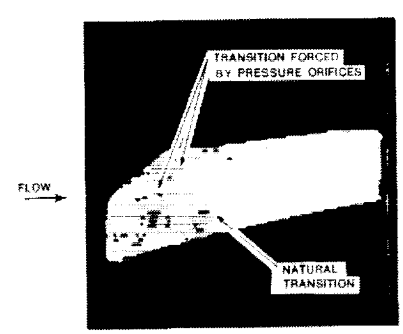
According to the Reynolds analogy, the heat-transfer coefficient and the skin-friction coefficient behave similarly under fully-developed boundary-layer flows. Therefore, the boundary-layer characteristics can be inferred from the thermal signature on the airfoil surface. To enhance the thermal signature of the transition, a positive temperature perturbation (smaller than 1 percent of the free-stream temperature) was imposed on the flow. The subsequent temperature response of the surface showed the effects of the difference between the heat-transfer coefficients under the laminar and turbulent boundary-layer regime; that part of the airfoil under the turbulent boundary layer heated faster than the part under the laminar regime. Further visual enhancement was achieved by compressing the range of temperature differences associated with the laminar-flow regime into one color, in this case blue, and those temperature differences associated with the turbulent-flow regime into white. The schematic of the test setup in the 0.3-Meter TCT is shown in the first figure below, and the second figure below shows the result of a pixel-by-pixel subtraction of two thermograms taken after and before the temperature perturbation. This approach focuses on the temperature difference between the laminar and turbulent areas of the airfoil rather than on the absolute temperature of its surface. It also filters out apparent temperature differences on the airfoil surface to local variations in the emittance. Using this method, natural and forced transition were detected at free-stream and surface temperatures down to $-154°$ F.

SCHEMATIC OF TEST SETUP
IN 0.3-METER TCT



TRANSITION DETECTION BY IR IMAGERY
IN 0.3-METER TCT

$T_\infty = -63°F \quad M_\infty = 0.56 \quad Re_c = 4.56 \times 10^6$

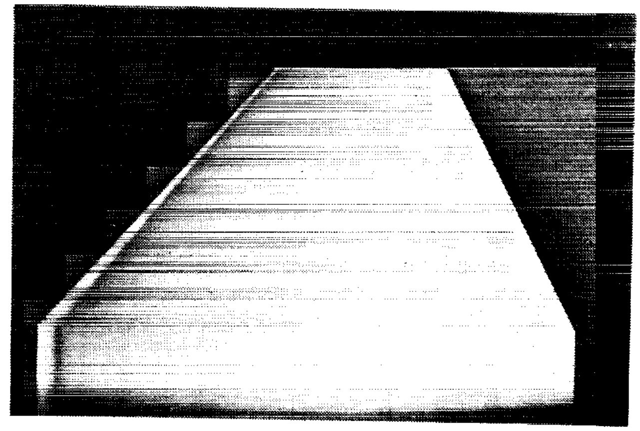


ORIGINAL PAGE IS
OF POOR QUALITY

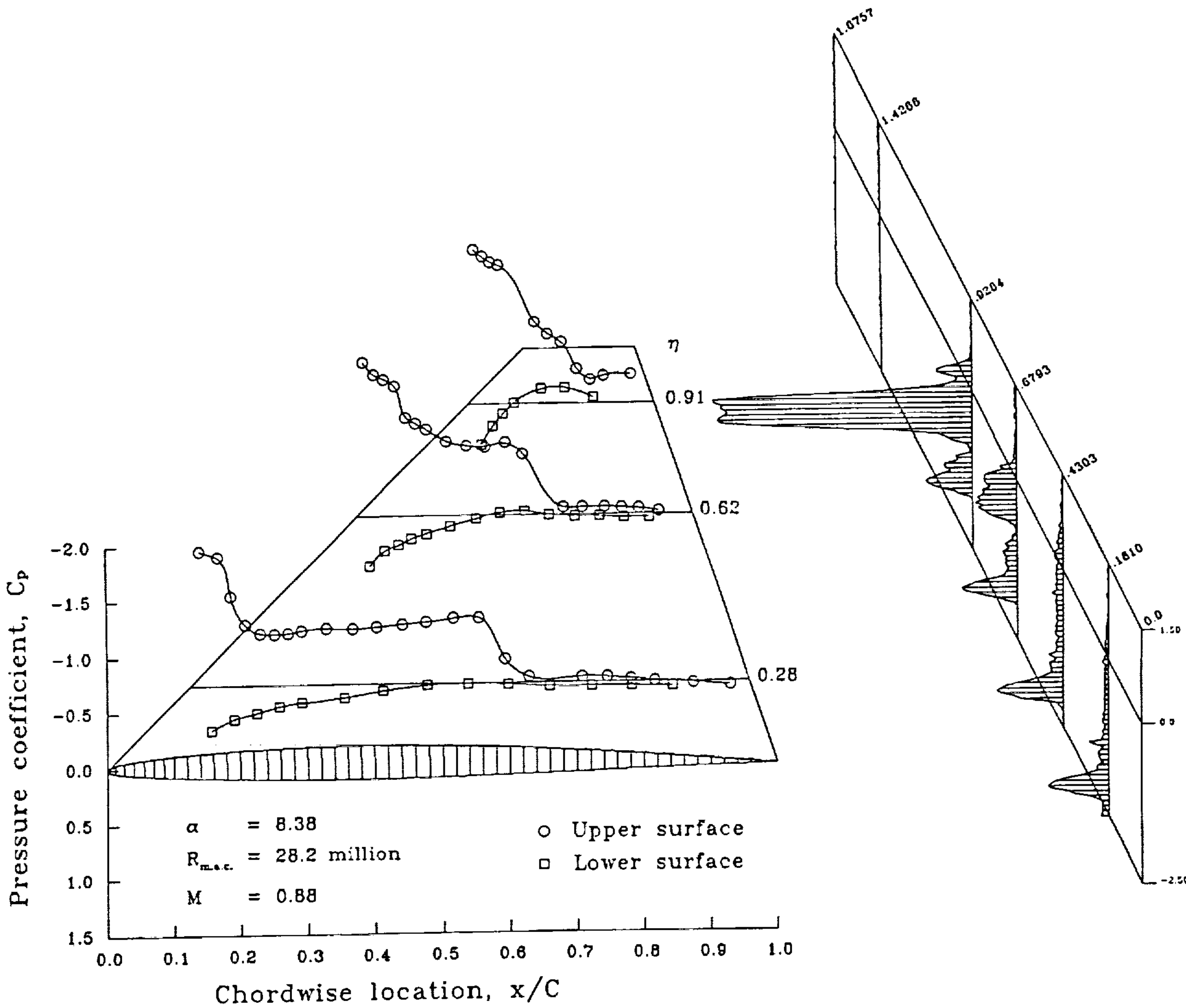
Construction and Transonic Testing of a Thin Wing at High Reynolds Numbers

Airfoils used in supersonic fighter aircraft are typically very thin. Building thin airfoil models with pressure instrumentation for cryogenic testing is unusually difficult. However, a technique which uses chemically-etched pressure channels in the bond planes between multiple sheets of metal has been developed. The canard of the X-29 fighter aircraft was chosen as a proof-of-concept model to build. This configuration has a maximum thickness of only 5 percent of chord; and it is highly tapered from root to tip, making pressure instrumentation very challenging.

The next figure is a photograph of the X-29 canard upper surface. Three rows of orifices with a total of 56 orifices on the upper surface exist and six thin plates form the upper surface of the airfoil. The outcrop of the five bond planes is clear in the figure. There are 37 additional orifices on the bottom surface. The choice of model size to test resulted in a root chord of 5.71 inches and a corresponding maximum thickness of 0.285 inch.



The model was mounted on the sidewall turntable in the 0.3-Meter TCT, and a special computational method was used to adapt the flexible test-section floor and ceiling for minimum interference. The aerodynamic data (surface pressures and wake survey) shown in the figure on the following page were taken for the wing at Mach numbers from 0.30 to 1.07. The data in the figure are for an angle of attack of 8.38°, Mach number of 0.88, and Reynolds number (based on mean aerodynamic chord) of 28.2 million.



Pressure coefficient, C_p
-2.0
-1.5
-1.0
-0.5
0.0
0.5
1.0
1.5
0.0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1.0
Chordwise location, x/C
η
0.91
0.62
0.28
α = 8.38
$R_{m.a.c.}$ = 28.2 million
M = 0.88
○ Upper surface
□ Lower surface

SUBSONIC AERODYNAMICS BRANCH

THE MISSION

The Subsonic Aerodynamics Branch (SAB) conducts research and provides technical expertise in subsonic aerodynamics, including fundamental fluid mechanics, performance and efficiency, stability, control, and aerodynamic loads for general aviation, rotorcraft, transport, and military high-performance aircraft. The research emphasis is on the development of advanced aerodynamic technology which will play a key role in future aerospace vehicles incorporating such concepts for efficient high lift and maneuverability, thrust vectoring for control, drag reduction, closely-coupled propulsion/airframe integration, and minimization of ground effects. The research is conducted using the latest state-of-the-art theoretical methods for prediction and analysis, experiments in the 14- by 22-Foot Subsonic Tunnel, and selected flight experiments. Research results include advanced aerodynamic concepts, new insight into aerodynamic phenomena important to advanced vehicles, and prediction of advanced-vehicle aerodynamic characteristics including propulsion-induced effects and ground effects. The SAB also provides technical and facility assistance to the DOD and other civil organizations (as appropriate) and provides facility support for Army rotorcraft research. Other NASA organizations work closely with the SAB in conducting research in areas of common interest.

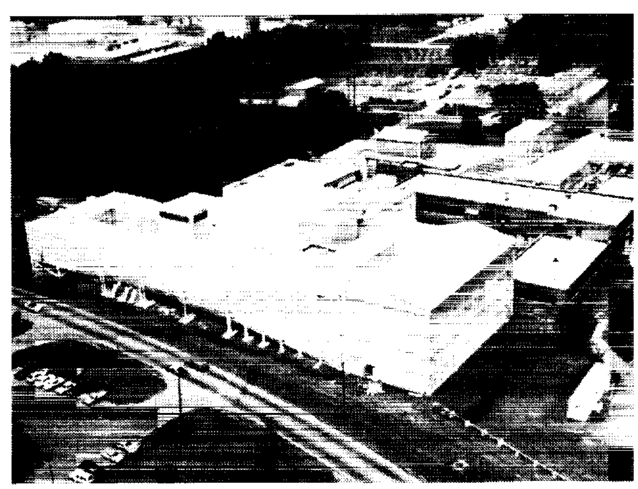
THE PERSONNEL

ALTHOFF, SUSAN L.
APPLIN, ZACHARY T.
BANKS, DANIEL W.
BERRY, JOHN D.
BEZOS, GAUDY M.
CAMPBELL, BRYAN A.
CROOM, TAMARA L.
ELLIOTT, JOE W.
GATLIN, GREGORY M.
GENTRY, GARL L., JR.
HODGES, WILLIAM T.
JONES, KENNETH M.
KELLEY, HENRY L.
KEMMERLY, GUY T.
KJERSTAD, KEVIN J.
MORGAN, HARRY L., JR. (Assistant Branch Head)
ORIE, NETTIE M.
PAULSON, JOHN W., JR.
PHELPS, A. E., III
QUINTO, P. FRANK
WALKER, GREGORY W.
WAGGONER, EDGAR G. (Branch Head)
WILSON, JOHN C.
WOOD, RICHARD M.
YAROS, STEVEN F.

THE FACILITY

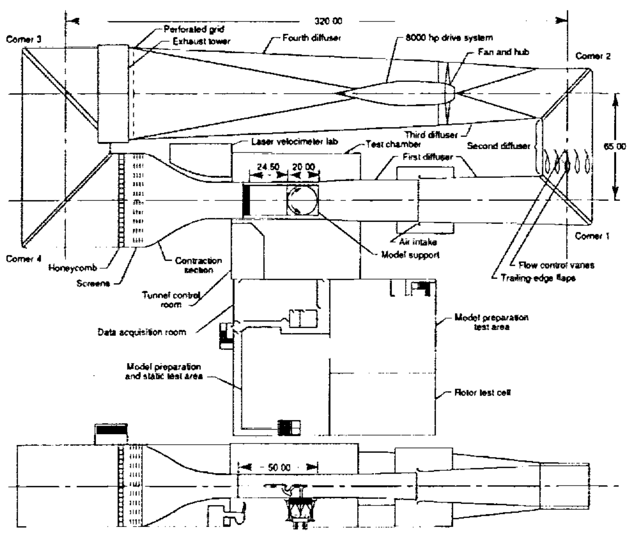
14- by 22-Foot Subsonic Tunnel

The LaRC 14- by 22-Foot Subsonic Tunnel (14- by 22-Foot ST) (formerly the 4- by 7-Meter Tunnel) is used for low-speed testing of powered and unpowered models of various fixed- and rotary-wing civil and military aircraft. The tunnel is powered by an 8000-hp electrical drive system which can provide precise tunnel speed control from 0 ft/s to 318 ft/s with the Reynolds number per foot ranging from 0 to 2.1×10^6. The test section is 14.5 feet high, 21.8 feet wide, and approximately 50 feet long. The tunnel can be operated as a closed tunnel with slotted walls or as one or more open configurations when the side walls and ceiling are removed to allow extra testing capability, such as flow-visualization and acoustic tests. The tunnel is equipped with a two-component laser velocimeter system. Furthermore, boundary-layer suction on the floor at the entrance to the test section and a moving-belt ground board for operation at test-section-flow velocities to 111 ft/s can be installed for ground-effect tests.



LaRC has completed significant modifications to the 14- by 22-Foot ST to improve and expand its aerodynamic and acoustic test capability. One of the more significant aerodynamic improvements was achieved through the use of flow deflectors installed downstream of the first corner of the tunnel circuit to improve the performance of the tunnel fan. The deflectors resulted in a more uniform velocity distribution into the tunnel drive system and eliminated regions of large-scale flow separation in the return leg of the tunnel circuit.

Langley 14-by 22-Foot Subsonic Tunnel



A turbulence-reduction system consisting of a grid, a honeycomb, and four fine-mesh screens dramatically reduced the level of longitudinal turbulence intensity in the tunnel test section. This system provided a reduction in turbulence of 50 percent or more for the closed test-section configuration. Periodic flow pulsations which occurred at several speeds in the unmodified configuration of the open test section were eliminated by installation of a new flow collector.

Acoustic reverberations in the open test section were reduced through the use of sound-absorbing panels on the test-chamber walls. A major operational improvement was achieved through the construction of a specially-designed laser velocimeter laboratory for setup and maintenance of the two-component laser velocimetry system. Finally, an addition to the model preparation area, which includes a support system and rotor test cell, provides the capability to assemble and test rotor models in hovering conditions prior to actual entry into the tunnel.

THE CURRENT PROGRAM

The SAB is currently involved in both dynamic- and static-testing activities related to improving the low-speed aerodynamic performance of military fighters, commercial transports, and rotorcraft aircraft. The dynamic testing activities include fundamental research to investigate the effects of rate of descent and ground proximity on the aerodynamic performance of high-speed fighter and supersonic commercial aircraft. As a result of this research, a unique model-support system is being fabricated which will properly simulate descent dynamics in the wind tunnel. Considerable dynamic testing is also being conducted to study the effects of various rotor-blade tip and fuselage modifications on the aerodynamic performance of a typical rotorcraft. Detailed flow-field measurements of the rotor inflow and of the interaction of the rotor wake on the fuselage have been obtained using a two-component LV system. These measurements are being used to validate several computer codes which predict rotor performance. Counter-rotating- and tilt-rotor configurations are also being tested to provide baseline data for future rotorcraft development efforts.

The static testing activities are mainly involved with studies related to commercial transport and military aircraft. The subsonic-transport activities consist primarily of experimental investigations of the effects of turboprop wakes on wing spanload distributions, turboprop engine placement on low-speed performance, and of a heavy-rain environment on high-lift performance. It is planned that this research be expanded to include the integration of advanced very high bypass ratio turbofan engines into a next generation subsonic transport. The supersonic and hypersonic transport activities consist of investigations to establish baseline low-speed powered and unpowered take-off and landing performance characteristics of advanced High-Speed Civil Transport (HSCT) and National Aero-Space Plane (NASP) configurations. The military fighter activities consist primarily of investigations of the effects of vectored thrust on low-speed take-off and landing and on high-angle-of-attack maneuvering. Additional high-angle-of-attack research is being conducted to establish the effects of scaling on forebody flow fields. Several researchers within the SAB are also actively involved with the application of existing computational methods for the low-speed aerodynamic analysis of the various configurations being tested.

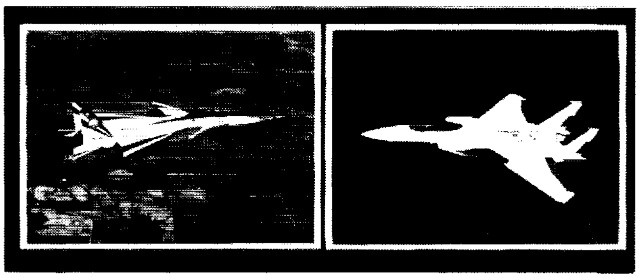
HIGHLIGHTS OF RECENT RESEARCH

Static and Dynamic Ground Effects

Historically, there have been discrepancies between the ground effects predicted from the wind-tunnel tests of many aircraft configurations and those actually experienced in flight. These differences have been greatest on vectored-thrust configurations. A wind-tunnel research program has been conducted to determine whether the absence of rate-of-descent simulation in conventional ground-effects testing could be the source of this discrepancy. Early tests, using a model of the F-15 S/MTD, as shown in the following figure, confirm the existence of the influence of rate of descent on a thrust-vectoring configuration of current interest. In general, the data indicate moderate effects on the lift coefficient due to rate of descent at low-thrust-vector angles; however, they indicate much larger differences between the results from the two test techniques at higher vector angles. Similar trends were seen in pitching moment.

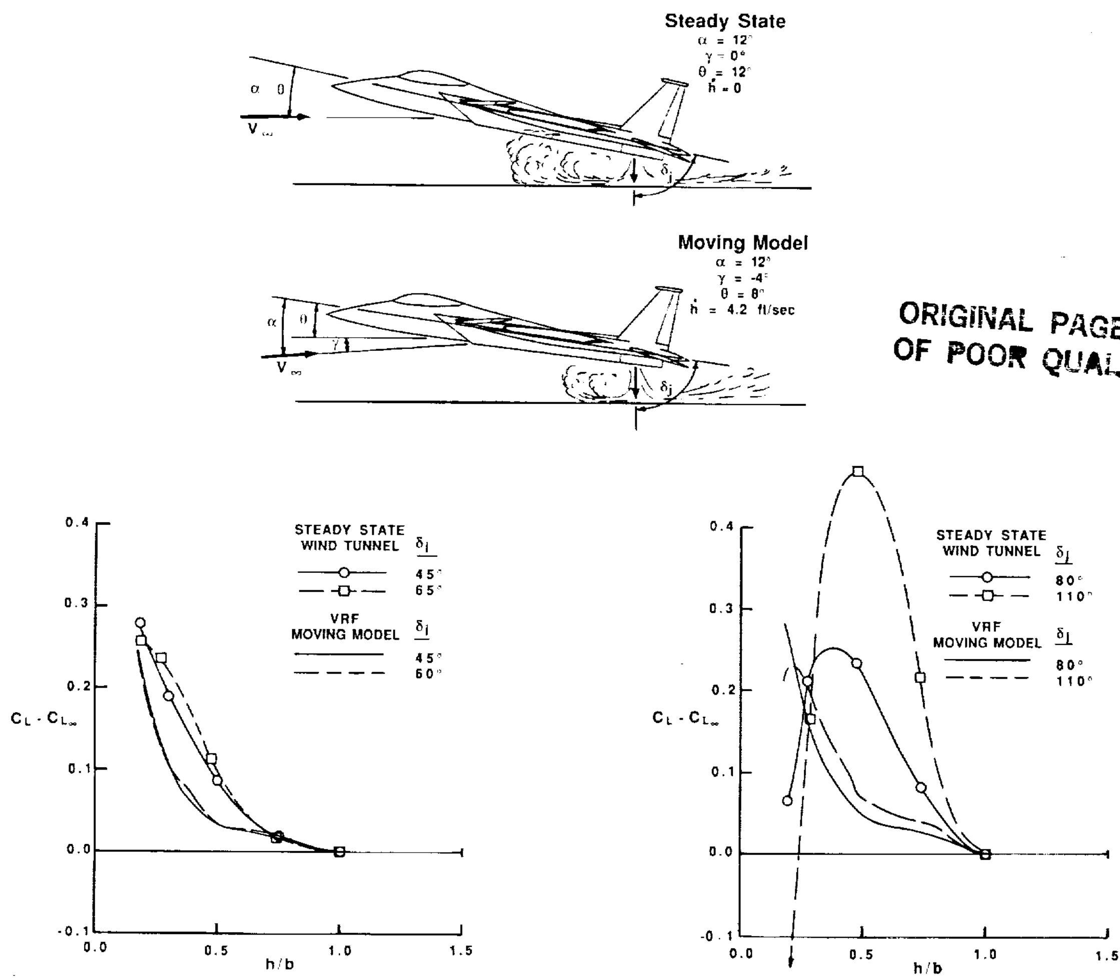
STOL AND MANEUVER TECHNOLOGY DEMONSTRATOR

- S/MTD probably sensitive to rate of descent
- Ground effects in simulation data base are based on steady-state wind-tunnel results
- LaRC measured ground effects dynamically



To achieve a simulated rate of descent, an inclined ground board was constructed within the test section of the LaRC Vortex Research Facility. As the model, traveling horizontally through the facility, passed over the inclined ground board, the height of the model above the ground reduced at a constant rate–the simulated rate of descent. A comparison of these results with those obtained using conventional steady-state ground-effects testing techniques is presented in the figure below. Similar comparisons have been made for a variety of other configurations and show, conclusively, that the effect of sink rate should

DYNAMIC GROUND EFFECTS on F-15 S/MTD



be represented in predicting ground effects for advanced vehicle designs. Based on the results to date, a special dynamic-model-support system is being developed for the 14- by 22-Foot ST to study dynamic ground effects.

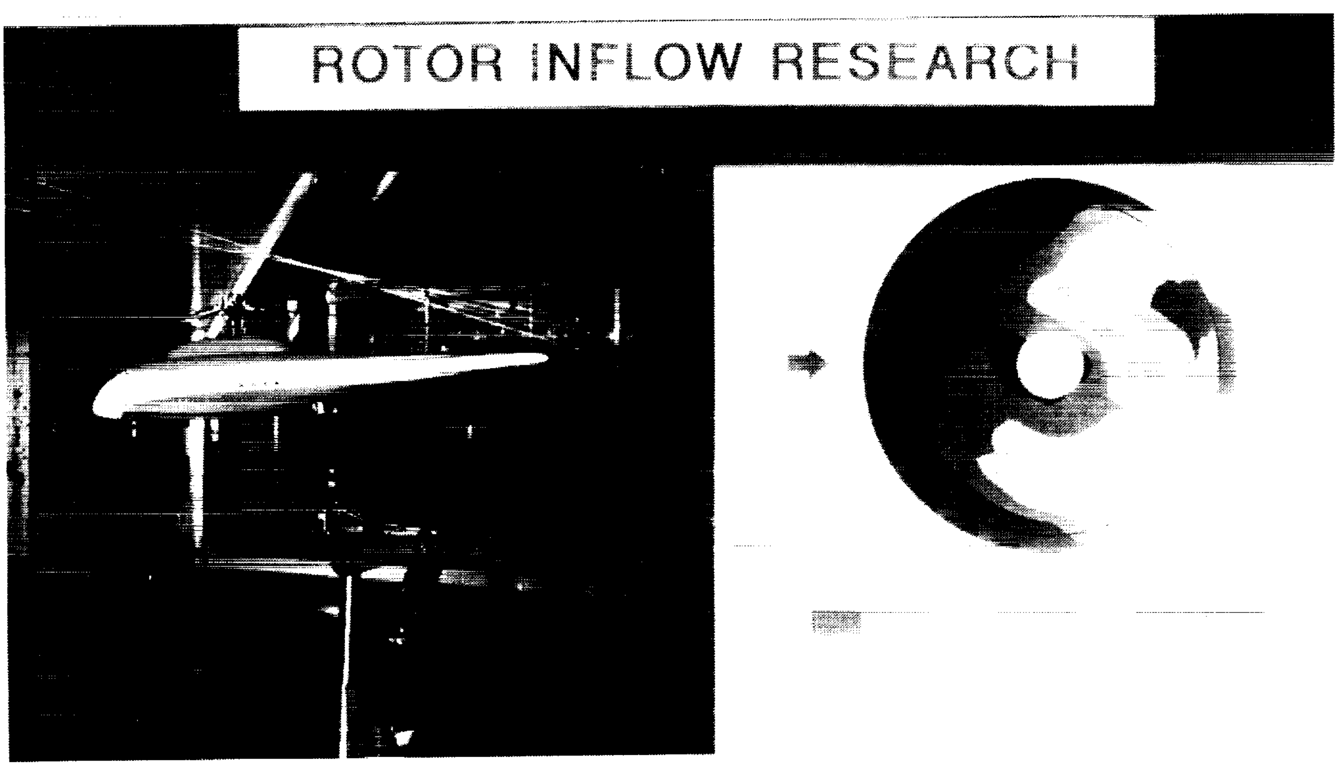
Rotor Inflow Research

The fifth in a series of rotor inflow measurements tests was conducted in the Langley 14- by 22-Foot ST using a 2-Meter Rotor Test System and a laser velocimeter for measuring flow velocities. The purpose of this U.S. Army/NASA program is to establish an experimental data base of rotor inflow and wake velocities which can be used for the validation of computational methods that predict flow velocities near a rotor. In the most recent test, rotor inflow data were acquired for a four-bladed rotor with a generic research fuselage. The data were measured at 180 locations in a plane approximately 3 inches above the plane formed by the rotating blade tips for advance ratios (ratio of flight speed to rotor-tip speed) of 0.23 and 0.30. Both average and time-dependent data were acquired for each measurement location. The predictions of various computational analyses of rotor inflow were compared to the experimental data. A photograph of the test model and a contour plot of the mean-induced-inflow ratio are presented in the figure below.

The experimental data show that as wind speed is increased, the area of upflow induced by the rotor moves progressively from the far-forward region of the rotor disk to cover the complete forward half of the disk. The induced-inflow characteristics at all wind speeds are asymmetric about the longitudinal axes of the rotor with the maximum downwash concentrated in the aft portion of the rotor disk, skewed to the advancing blade side. The computational methods show significant differences from the experimental data, indicating that improvements in the methods are necessary for the proper calculation of the flow conditions affecting rotor performance.

Advanced Turboprop Integration Research

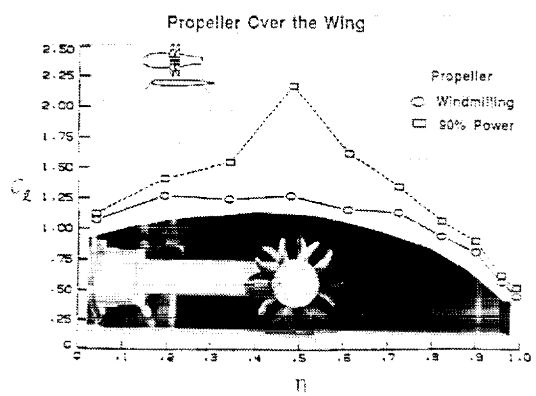
An exploratory research program has been conducted in the 14- by 22-Foot ST to investigate propeller installation effects on the low-speed aerodynamic characteristics of an advanced counter-rotation turboprop configuration mounted in several different chordwise locations above and behind a wing. A 2-foot-diameter model of the General Electric Unducted Fan (UDF) was used in combination with a 1- by 3-meter semispan wing as shown in the figure; and tests were conducted for simulated takeoff, cruise, and landing configurations. A total of seven different propeller/wing positions were tested for angles of attack ranging from −4° to 20°.



Aerodynamic loads were measured on the wing with a six-component balance, and oscillatory blade stresses were measured on each row of the UDF simulator by means of blade-mounted strain gages. Several hundred pressure measurements were made on the surfaces of both the wing and nacelle to aid in identifying the sources of the aerodynamic interference effects. A sample plot showing the effect of propeller slipstream on the spanwise distribution at lift on the wing is presented in the following figure.

ATP INSTALLATION AERODYNAMICS AT LANGLEY

UDF Semi-Span Wing Experiment



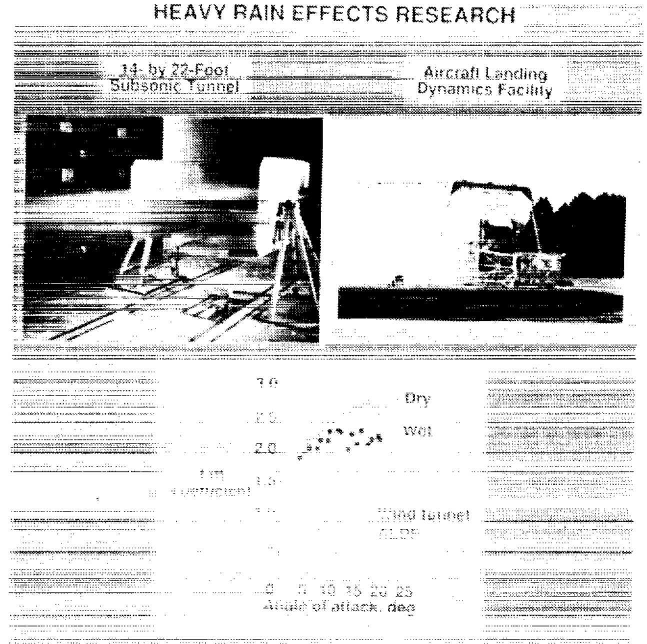
Early analysis of the results of the experiments indicated that blade stresses were very high for those configurations where the wing wake passed through a significant portion of the propeller disk, but the stresses were reduced as the wake was moved toward the outer edges of the disk. When the propeller was positioned over the wing forward of the wing trailing edge, increases in power caused increases in the basic wing lift which increased the overall lift for the wing-propeller combination. An analysis of the pressure data indicated a favorable effect of the propeller on the wing upper-surface pressure distribution for those cases, but the measured propeller performance was essentially unaffected by the wing pressures.

Data from this investigation are being incorporated into a data base which is being used to identify and develop needed improvements in computational fluid dynamics (CFD) capabilities for design and analysis.

Heavy-Rain Research

Since 1982, NASA has been studying the influence of heavy rain on airfoil aerodynamic performance. Small-scale airfoil tests in the 14- by 22-Foot ST and large-scale airfoil tests at the Aircraft Landing Dynamics Facility (ALDF) showed that a high-intensity rainfall adversely affects airfoil performance. Photographs of an NACA 64-210 wing section mounted in the tunnel and on the ALDF test carriage are presented in the figure below. Aerodynamic lift data were obtained with and without the rain simulation system turned on for an angle-of-attack range of 7.5° to 19.5° and for rainfall conditions of 9 inches/hour and 40 inches/hour. The results obtained at the 9-inch/hour rainfall condition indicate a small reduction in maximum lift and only a slight influence in the stall angle of attack. The test results shown in the figure were obtained at the 40-inch/hour rainfall condition and show a 15-20 percent reduction in observed maximum lift and a reduction from a dry-air stall angle of approximately 6°. These results compare well with the previous small-scale wind-tunnel results for the same airfoil section. It appears that to first order, scale effects are not large and the wind-tunnel research technique can be used to predict rain effects on airplane performance.

HEAVY RAIN EFFECTS RESEARCH



TRANSONIC AERODYNAMICS BRANCH

THE MISSION

Develop and conduct research programs which will improve understanding of transonic flows over airplane and missile configurations. Conduct basic and applied research using experimental and theoretical methods to develop and improve the performance, load characteristics, and stability and control of all types of aircraft, missiles, and spacecraft in both cruise and maneuvering flight at subsonic and transonic speeds. Develop verified capability for computer-aided design of aircraft at transonic cruise and maneuver conditions. Methods must be capable of designing complex configurations with flexible wings, fuselages, stores, canards and/or horizontal tails, and single or twin vertical tails. Develop verified prediction capability and understanding of transonic attached and vortical flows over wings, bodies, complete aircraft, and missiles. Conduct fundamental flow-modeling experiments to obtain data required to verify advanced transonic CFD methods. Exploit laminar-flow technology and vortical flow to enhance aircraft performance. Develop improved understanding of and methods for reducing induced drag.

Responsible for the administration, operation, and enhancement of the 8-Foot Transonic Pressure Tunnel (8-Foot TPT) and the 7- by 10-Foot High-Speed Tunnel (7- by 10-Foot HST). Support research and development requests for experimental investigations originated by other NASA organizations and government agencies.

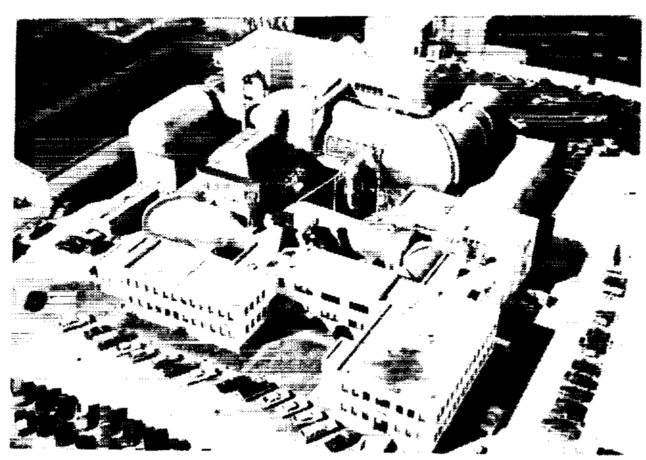
THE PERSONNEL

ALLISON, DENNIS O.
BROOKS, CUYLER W., JR.
CAMPBELL, JAMES F.
CAMPBELL, RICHARD L.
CLUKEY, PATRICIA G.
ERICKSON, GARY E.
FERRIS, JAMES C.
FOX, CHARLES H., JR.
FRINK, NEAL T.
GHAFFARI, FARHAD
HALL, ROBERT M.
HALLISSY, JAMES B.
HARRIS, CHARLES D.
HUFFMAN, JARRETT K.
JACOBS, PETER F. (Asst. Branch Head)
LAMAR, JOHN E.
LUCKRING, JAMES M. (Branch Head)
MINECK, RAYMOND E.
PHILLIPS, PAMELA S.
PLENTOVICH, ELIZABETH B.
SCHOONOVER, WARD E., JR.
SEWALL, WILLIAM G.
SMITH, CONNIE A.
SMITH, LEIGH A.
TRACY, MAUREEN B.

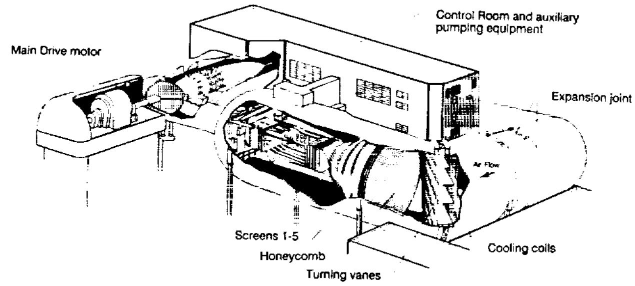
THE FACILITIES

8-Foot Transonic Pressure Tunnel

The Langley 8-Foot Transonic Pressure Tunnel (8-Foot TPT) is a variable-pressure, slotted-throat wind tunnel with controls that permit independent variations of Mach number, stagnation pressure and temperature, and dew point. See figures below. Air is circulated through the circuit by an axial compressor located downstream of the test-section diffuser and driven by an electrical drive system. The test section is square with filleted corners and a cross-sectional area approximately equivalent to an 8-foot-diameter circle. The floor and ceiling of the test section are axially slotted (approximately 6.9-percent porosity in the calibrated test region) to permit continuous operation through the transonic speed range. The side walls are solid and fitted with windows for schlieren flow visualization. The contraction ratio of the test section is 20:1.



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



Tunnel stagnation pressure can be varied from a minimum of about 0.25 atm. at all test Mach numbers to about 1.0 atm. at a Mach number of 1.2, about 1.5 atm. at high subsonic Mach numbers, and about 2.0 atm. at Mach numbers of 0.4 or less. Temperature is controlled by water from an outside cooling tower circulating through cooling coils across the corner of the tunnel circuit upstream of the settling chamber. The tunnel air is dried until the dew point temperature is reduced enough to prevent condensation in the flow by use of dryers using silica gel desiccant.

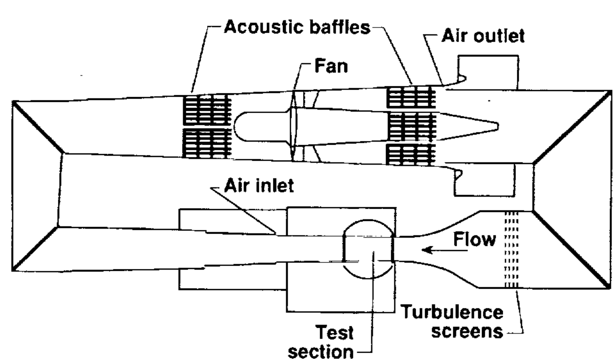
Based upon both centerline probe and wall pressure measurements, generally uniform flow is achieved over a test-section length of at least 50 inches at Mach numbers 0.20 to 1.20. The higher the Mach number, the shorter the region of uniform flow becomes. The tunnel is capable of achieving Mach numbers to about 1.3, but most testing is limited to a maximum Mach number of 1.2 since the calibrated region of the test section for $M = 1.3$ is further downstream than for lower Mach numbers and requires that a model be located further aft in the test section.

The 8-Foot TPT is a very versatile wind tunnel capable of supporting basic fluid dynamics research as well as a wide range of applied aerodynamic research. With the installation of screens and honeycomb in conjunction with the recently completed Laminar Flow Control Experiment, the quality of the flow in the test section is suitable for performing reliable code-validation experiments. The test section is already instrumented with many ceiling, floor, and side-wall pressure orifices and more could be easily added if desired. In addition, fixed chokes and test-section slot covers are currently being designed which would permit data to be obtained on both open- and closed-tunnel configurations as well as improve the flow quality in the test section by blocking upstream propagation of diffuser noise.

7- by 10-Foot High-Speed Tunnel

The figures below show the LaRC 7- by 10-Foot HST. This tunnel is a closed-circuit, single-return, continuous-flow atmospheric tunnel with a solid-wall test section 6.6 feet high, 9.6 feet wide, and 10 feet long. The tunnel is fan driven and is powered by a 14,000-hp electric motor. It operates over a Mach number range from 0.0 to 0.9 to produce a maximum Reynolds number of 4×10^6/foot. In addition to static testing of models to high angles of attack and large sideslip angles, the facility is equipped for both steady-state roll and oscillatory stability testing.

7- BY 10-FOOT HIGH-SPEED TUNNEL



The facility has an important role in a wide range of basic and applied aerodynamic research, including advanced vortex-lift concepts, drag-reduction technology, highly-maneuverable aircraft concepts, and the development of improved aerodynamic theories, such as the difficult separated-flow and jet-interaction effects needed for computer-aided design and analysis. The facility's flow-visualization capability has been upgraded through the installation of a permanent laser-vapor-screen system.

THE CURRENT PROGRAM

Currently, the Transonic Aerodynamics Branch (TAB) is conducting experimental, computational, and applied aerodynamic research to develop improved understanding of transonic attached and vortical flows over bodies, wings, complete aircraft, and missiles. Research is also underway to develop methods for controlling the vortical flows to improve high angle maneuvering performance of fighter aircraft. As part of this effort, extensive studies of the ability of advanced CFD methods for predicting subsonic/transonic vortical flows are underway.

Research studies are underway to develop methods and procedures for design of advanced airplane concepts. The current focus of this work is to develop the capability for computer-aided design of advanced subsonic/transonic transport configurations with emphasis on cruise performance. Methods to design at multiple design conditions are also being developed. An important element of this research is the development of fast, accurate flow solvers for predicting transonic flow over complex configuration geometries. Research is underway toward developing and assessing the capabilities of unstructured Euler solvers for accomplishing these objectives. Research is also underway to develop understanding and procedures for applying the aerodynamic-design methodology in a multidisciplinary environment.

Experimental studies are being conducted to develop increased understanding of the causes of induced drag. Once understanding is obtained, concepts for reducing drag due to lift will be developed. Preliminary research for induced-drag reduction is underway with studies of winglets on advanced high-aspect-ratio transport wings and the use of wing-tip blowing as a means of increasing the "effective" aspect ratio of a wing.

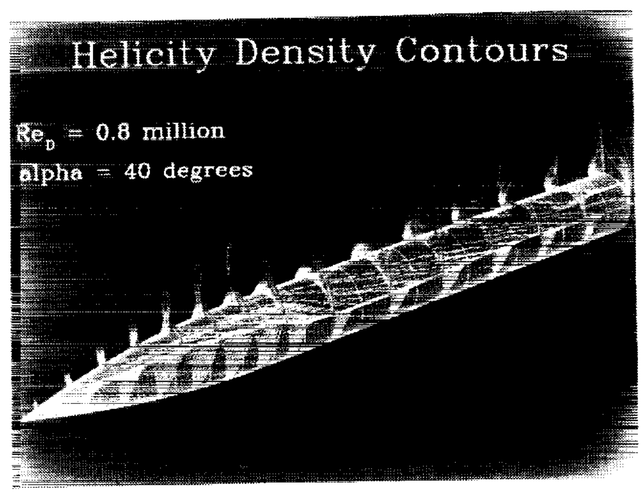
Experimental studies in support of the HSCT Initiative and the National Aero-Space Plane (NASP) Program are also underway.

HIGHLIGHTS OF RECENT RESEARCH

Navier-Stokes Solutions For Asymmetric Vortical Forebody Flows

Asymmetric vortex shedding arises when slender bodies, typical for missiles and fighter aircraft noses, are set at large angles of attack (typically greater than 30°). Under these conditions, side forces occur which become so large that they dominate the lateral stability of these vehicles. Experimental surface-pressure data suggested that small geometric imperfections, such as an out-of-round tip of a sharp-nosed wind-tunnel model control, caused such vortex asymmetries along the entire body. This is confirmed in the present computational study, where the surface geometry of a 3.5-caliber tangent-ogive cylinder is perturbed into slightly elliptic cross sections just at the nose tip.

Steady-state solutions for vortical flows with 0.2 million $\leq Re_D \leq$ 3.0 million (D: maximum diameter) and $20° \leq$ alpha $\leq 40°$ have been obtained using FMC1, a time-implicit upwind method for the three-dimensional, incompressible Navier-Stokes equations. This solver comprises flux-difference splitting, a TVD-like discretization of the inviscid fluxes, and an extension to the algebraic turbulence model by Baldwin and Lomax which allows, for the first time, computational modeling of transitional cross-flow separation (i.e., flows with three-dimensional, laminar, equatorial separation bubbles and subsequent transition in the separating shear layers which roll up into two primary vortices).

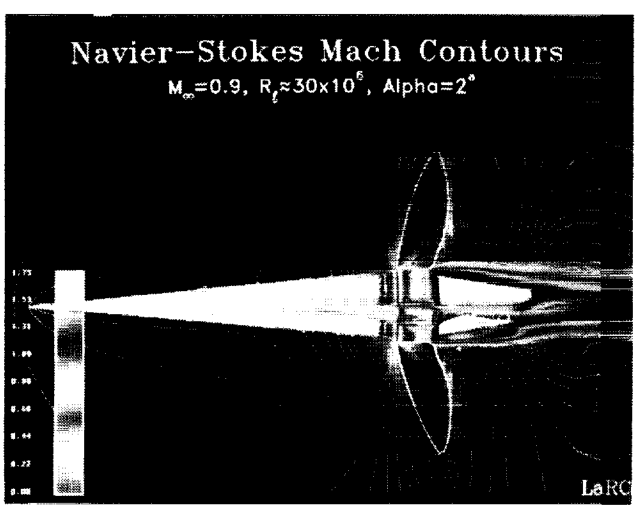


By rotating the perturbed nose tip, the computed surface pressures indicate an almost sinusoidal variation of side force with roll angle while the vortex shedding frequency remains close to a fixed value. These two integral properties and the computed surface-pressure distributions themselves agree well with experimental data. A typical result is shown for alpha = 40° and Re_D = 0.8 million. The computed flowfield is visualized by means of helicity density contours (helicity density is defined as the scalar product of the local velocity and vorticity components) which are superimposed on a surface-flow pattern. The figure above shows four vortex shedding events and their spatial frequency. Further analysis, using computational flow-visualization techniques, showed that maximum local side force is associated with one dominant primary vortex in the wake and an asymmetric arrangement of the lateral primary separations. Minimum local side force is distinguished by primary vortices of about equal strength and an almost symmetric separation pattern.

Transonic Navier-Stokes Solution About A High-Speed Accelerator Configuration

The design of the proposed NASP will inevitably rely heavily on CFD to complement and extend information obtained in current ground test facilities. The present investigation is directed toward applying an advanced CFD code to a generic NASP-like configuration at transonic-flow conditions which may be conducive to flow separation.

An accelerator configuration recently tested in the LaRC 16-Foot Transonic Tunnel (16-Foot TT) was selected for the study. This model was comprised of a cone-cylinder-frustum body, a wrap-around engine nacelle, forebody and aftbody engine fillets, and a 70° delta wing at incidence. The configuration surface was represented analytically; a blocked flow-field domain of approximately 373,000 points was then constructed with hybrid topologies using established transfinite interpolation methodology. Steady-state solutions to the compressible thin-layer Navier-Stokes equations were obtained with an implicit finite-volume algorithm (CFL3D) developed at LaRC.



These solutions were achieved using Van Leer's upwind-biased, flux-vector-splitting technique and an extended version of the Baldwin and Lomax algebraic turbulence model.

Turbulent results have been obtained at an angle of attack of 2°, a Reynolds number of approximately 30×10^6 (based upon the total body length), and a Mach number of 0.9. Mach contours on the surface and in the plane of symmetry demonstrate a smooth solution in the figure above for the blocked representation of this configuration. After a subsonic and mainly attached forebody flow, the flow accelerates supersonically at the cowl-lip of the faired-over engine inlet and subsequently shocks down at the exhaust cowl-lip. (The sonic line is represented with a white contour line in the plane of symmetry to highlight the supersonic flow region.) The shock produces an adverse pressure gradient which causes the flow to separate massively and envelop the boattail region. Predicted forebody pressures agree with experimental data reasonably well; a qualitative prediction of the separated boattail flow is also achieved.

Navier-Stokes Solutions For The F-18 Forebody-LEX

Advances in numerical solution methodology along with increased computer speed and capacity have made it feasible to seek numerical solutions to the three-dimensional Navier-Stokes equations at flight Reynolds numbers for relatively complicated aircraft geometries. As a step toward modeling a complete aircraft, turbulent Navier-Stokes solutions have been achieved for the forebody leading-edge extension (LEX) portion of the F-18 High Alpha Research Vehicle (HARV).

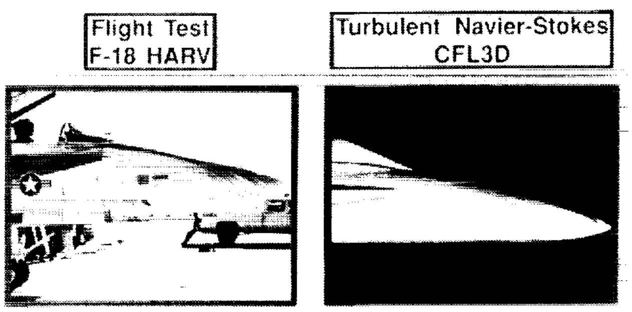
The surface definition of the configuration was obtained from a detailed CAD/CAM description of a 6-percent F-18 wind-tunnel model. A longitudinally-blocked grid of approximately 185,000 points was generated with an H-O topology using established transfinite interpolation methodology. A solution for the flow was then obtained from a version of CFL3D recently extended for longitudinally-blocked grids. CFL3D models the compressible full Navier-Stokes equations by a finite-volume technique which incorporates an upwind-biased, flux-difference-splitting approach. Turbulence effects were represented by an extended version of the Baldwin and Lomax algebraic turbulent model.

A representative turbulent solution is shown in the following figure for a Mach number of 0.34, a Reynolds number of 13.5 million (based on the wing mean-aerodynamic chord), and an angle of attack of 19°. These conditions correspond to recent flight tests of the NASA F-18 HARV at the Dryden Flight Research Facility. The flight tests were focused on documenting the forebody surface-flow pattern; this result qualitatively compares well with the computed turbulent flow pattern as shown in the figure.

The flow pattern on the aircraft was generated by emitting a fluid mixture of propylene glycol monomethyl ether (pgme) and dye from surface orifices. After flowing down the forebody, the pgme evaporates leaving the dye pattern shown. Numerical convergence corresponded to a two to three order-of-magnitude reduction of the residuals. This required approximately 2400 cycles corresponding to approximately two hours of CRAY-2 time. Computations for other flight conditions, as well as for a more complete configuration representation, are underway.

NAVIER-STOKES PREDICTION OF FLIGHT TEST

$M_\infty = 0.34$, $R_c = 13.5\times10^6$, $\alpha = 19°$

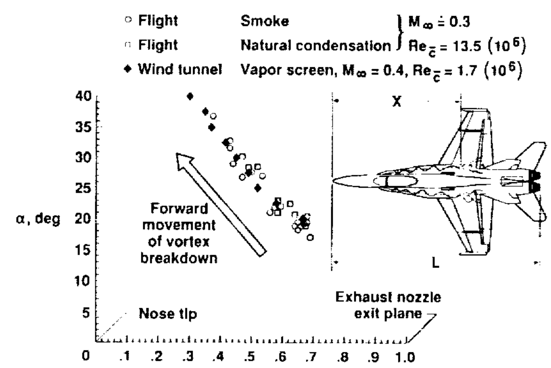


Wind Tunnel-To-Flight Correlations

Correlations have been made of the vortex flows visualized in wind-tunnel and flight experiments of the Navy/McDonnell Douglas F/A-18. This work was conducted to establish the degree to which the vortical flows in wind-tunnel testing of subscale fighter models represent the flow-field behavior in flight at full scale. The wing LEX vortices about a 6-percent-scale model of the F/A-18 were visualized in the David Taylor Research Center (DTRC) 7- by 10-Foot Transonic Wind Tunnel using a laser-vapor-screen technique. The vortex structure, trajectories, and breakdown characteristics observed at subsonic speeds ($M = 0.30$ and 0.40) in the wind tunnel were compared to the off-body flow visualizations obtained on the full-scale F-18 HARV at the NASA Ames/Dryden Flight Research Facility. (See first figure below.) The Reynolds number based on the wing mean aerodynamic chord was approximately 1.7×10^6 in the wind tunnel and 13.5×10^6 in flight. The experiments were conducted as part of the NASA F-18 High-Alpha Technology Program.

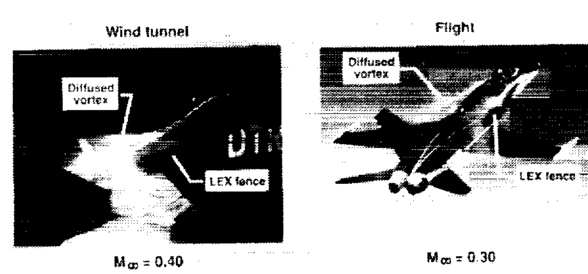
The present study showed a high degree of correlation of the vortex flows about the small-scale wind-tunnel model and the F/A-18 aircraft. The structure and location of the LEX vortical flows were similar. The onset of vortex breakdown, its forward progression with increasing angle of attack, and the interaction of the burst vortex with the twin vertical stabilizers compared well in the wind-tunnel and flight experiments. The physical mechanism for the effectiveness of the LEX upper-surface fences in reducing the vertical tail buffet was first identified in the wind-tunnel testing and, subsequently, confirmed in the flight experiments. The off-body flow-field observations on the 6-percent-scale model and the F-18 HARV showed an upward displacement and restructuring of the LEX vortices with the fences on. (See second figure below.) The test results also demonstrated the usefulness of applying different flow-visualization techniques to improve the understanding of complex vortical flows. For example, the diffuse nature of the LEX vortex downstream of the fence resembled the "classical" vortex breakdown phenomenon due to the rapid increase in the size of the vortical region. However, illumination of the vortex cross flow with the intense sheet of laser light in the wind tunnel revealed a weakened system of co-rotating vortices that had not burst.

COMPARISON OF F-18 LEX VORTEX BREAKDOWN CHARACTERISTICS – FLIGHT AND WIND TUNNEL RESULTS



F-18 VORTEX FLOW FIELD WITH LEX FENCES
WIND TUNNEL-TO-FLIGHT CORRELATION

$\alpha = 20°$

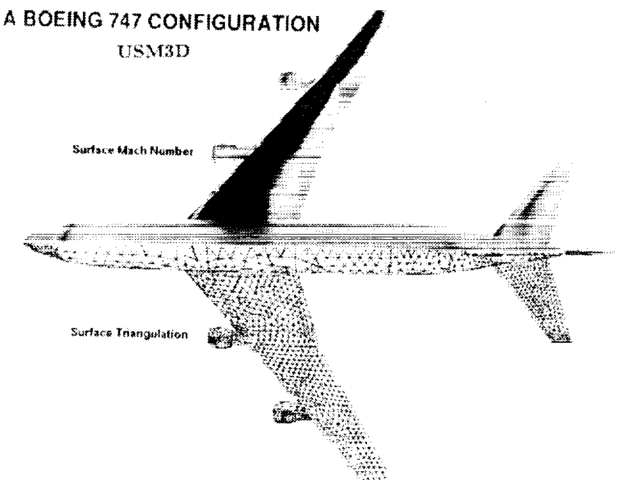


A Fast Upwind Solver For The Euler Equations On Three-Dimensional Unstructured Grids

One of the major difficulties in computational fluid dynamics is the accurate and efficient representation of flow fields about complex configurations. A computer code, USM3D, has been developed to solve the steady-state Euler equations about arbitrary configurations in three dimensions on unstructured tetrahedral grids using a finite-volume technique that incorporates an upwind-biased, flux-difference-splitting approach. The code uses a new algorithm which achieves computational efficiencies comparable to those of structured codes with nominally 50 percent more memory than typical structured codes.

A BOEING 747 CONFIGURATION

USM3D



A sample result for a Boeing 747-200 with flow-through nacelles is portrayed in the figure above which shows both the surface grid as well as shaded Mach contours for $M = 0.84$ and alpha = 2.73°. The grid consists of 105,372 cells, 19,698 nodes, 8402 boundary faces, and 4195 boundary nodes. The original figure shows red contours on the wing which indicate a region of supersonic flow which is terminated by a shock wave where the contours transition to yellow. This solution was obtained with a CFL number of 3 in 1600 cycles for a decrease in the L2-norm of 3.7 orders of magnitude. The solution required 1 hour and 40 minutes of Voyager CRAY-2S time and used less than 8 megawords of memory.

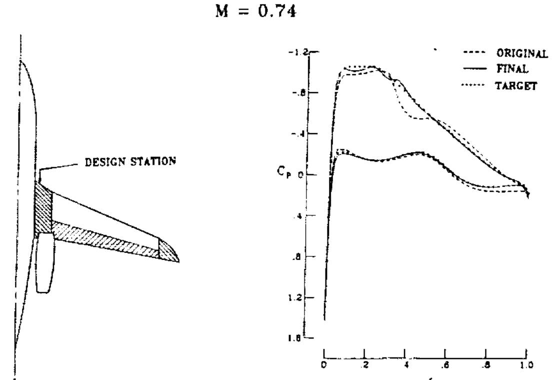
Application Of A Transonic Design Method To Complex Geometries

During the past 5 years, NASA LaRC has been involved in several research programs which had the requirement of designing or modifying a wing to achieve a given pressure distribution at transonic speeds. In order to meet this requirement, a design method was developed which modifies the surface curvatures and slopes of an initial airfoil geometry so that a target pressure distribution is matched. The Direct Iterative Surface Curvature (DISC) Method was then extended to wings and found to be robust and efficient. The method has been useful in designing wings in the presence of other aircraft components such as winglets, fuselages, and nacelles. It can be used to achieve a given pressure distribution and thus reduce or eliminate undesirable flow characteristics or adverse interference effects between different components.

The surface curvature method has been used in a number of design exercises involving complex geometries. Results for an executive transport with fuselage-mounted nacelles are shown in the figure below. The sketch on the left shows a wing root plug (cross-hatched area ahead of the nacelle) to be added to the existing wing. The flow in this region is strongly influenced by both the fuselage and nacelle. The original wing plug has a fairly strong shock near 40 percent chord (right half of figure). The design method was used successfully to modify the wing to eliminate this shock and to give a more uniform isobar pattern in this region.

WING ROOT DESIGN TO ELIMINATE SHOCK

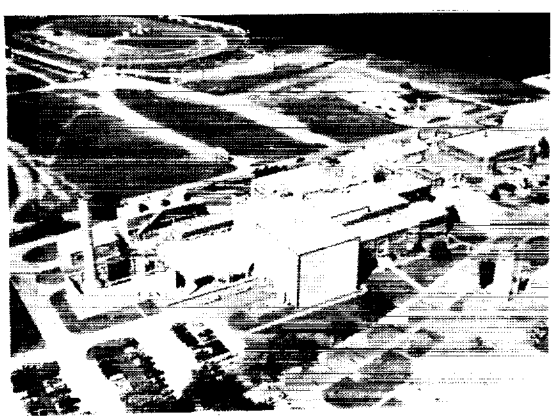
M = 0.74



HIGH-REYNOLDS-NUMBER AERODYNAMICS BRANCH

THE MISSION

Conducts theoretical and experimental research to develop Reynolds-number-scaling techniques for configurational aerodynamics as well as basic fluid mechanics phenomena at subsonic and transonic speeds. This research includes investigations of Reynolds-number scaling for: shock/boundary-layer interaction, vortical flows, juncture flows, attached flows (skin friction), control effectiveness, high-lift systems, stores integration, and propulsion integration. The research is conducted utilizing the National Transonic Facility (NTF) (see figure below), the latest computational techniques, and flight experiments. Because of the cryogenic and high-pressure environment of the facility, supporting research includes the development of experimental techniques and instrumentation which are compatible with the harsh environment of this facility; development of new instrumentation includes flow-visualization systems and a boundary-layer transition detection system. In an effort to continually improve the quality of the experimental data obtained from the NTF, studies are carried out to develop better wall and support interference-correction procedures.



The HRNAB is responsible for the administration, operation, and enhancement of the NTF to accomplish the national high-Reynolds-number test requirements; and, as such, support research and development requests for experimental investigations originated by other NASA organizations and government agencies.

THE PERSONNEL

ADCOCK, JERRY B.
AL-SAADI, JASSIM A.
BOYLES, GEORGE B., JR.
CARTER, ANGELA M.
CHU, JULIO
FOSTER, JEAN M.
FULLER, DENNIS E.
GLOSS, BLAIR B. (Assistant Branch Head)
HANNON, JUDITH A.
HILL, JEFFREY S.
IGOE, WILLIAM B.
JOHNSON, WILLIAM G., JR.
LAWING, PIERCE L.
OWENS, LEWIS R., JR.,
POPERNACK, THOMAS G., JR.
PUTNAM, LAWRENCE E. (Branch Head)
SNOW, DANIEL B.
THIBODEAUX, JERRY J.
TOMEK, WILLIAM G.
WEISS, DINA SLACK
WILLIAMS, M. SUSAN

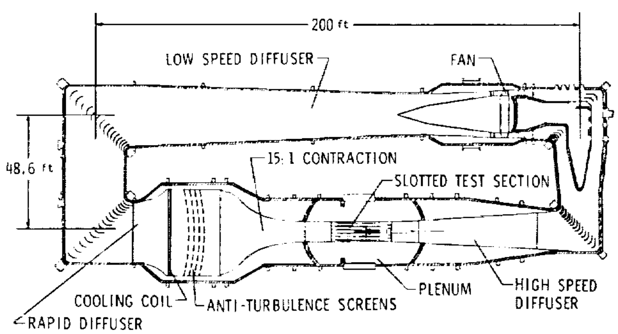
THE FACILITY

National Transonic Facility

The National Transonic Facility (NTF) is a transonic wind tunnel primarily intended to provide a high-Reynolds-number capability for aerodynamic research and the developmental testing of commercial and military aircraft configurations. The NTF is a closed-circuit, single-return, fan-driven wind tunnel capable of continuous operation and capable of achieving a chord Reynolds number of 145 million/ft at a Mach number of 1.0 by operating at elevated pressures and cryogenic temperatures. The NTF can operate at Mach numbers from 0.2 to 1.2, stagnation pressures from 1 to 9 atm., and stagnation temperatures from 140° F to approximately −260° F. In the cryogenic mode of operation, nitrogen is used as the test gas, with cooling accomplished by the injection of liquid nitrogen directly into the tunnel circuit. At ambient temperatures, air is used as the test gas with cooling accomplished by a conventional water-cooled heat exchanger inside the tunnel circuit. See figure below for plan of tunnel circuit.

The test section is square, 8.202 feet wide, with small flat fillets at 45° angles in the corners resulting in a test-section cross-sectional area of 66.77 ft^2. There are six longitudinal slots in the test section top and bottom walls. The length of the slotted region is approximately three test-section widths. The vertical walls are parallel and fixed during testing; the top and bottom walls have flexures at the upstream end which permit variation in wall angle from about 0.5° converged to 1.0° diverged. Generally, test models will be sting supported from a circular arc strut permitting a pitch range of 30°, nominally −11° to 19°; but this can be varied using offset sting supports. The model roll-angle range is ±180°; sideslip angles are obtained from combinations of the pitch and roll. Plans are currently underway to provide for wall-mounted half-span models in the test section for tests where large model sizes are required. The maximum allowable model loads for a sting-mounted model are: normal - 19,500 lb; axial - 9,356 lb; and side - 10,000 lb.

NATIONAL TRANSONIC FACILITY
PLAN OF TUNNEL CIRCUIT



Construction of the NTF was completed in September 1982; it was declared operational in August 1984.

CURRENT PROGRAMS

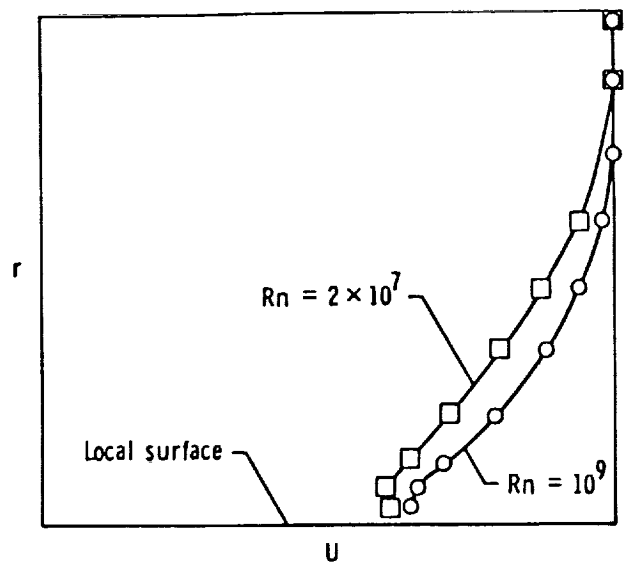
The basic challenge for the HRNAB is the development of Reynolds-number-scaling techniques for subsonic (incompressible and compressible) and transonic flow fields. This area of research is very broad and presented below is a brief discussion of some of the specific areas of study. One area of research is validating and developing scaling procedures for: (1) drag associated with transport configurations at near-design conditions and (2) supercritical pressure distributions obtained from a high-wing-transport test in the NTF. In order to obtain the best possible data from the cryogenic high-Reynolds-number facility, there is a continuing effort to develop better instrumentation and test techniques; in this regard, there is an effort to develop an infrared transition detection system capable of operation at cryogenic temperatures. To better understand Reynolds-number effects on control effectiveness, there is a study underway to develop a numerical procedure to accurately characterize the flow fields around subsonic and transonic expansion corners at high-Reynolds numbers; this study has direct application to aileron flow fields. Utilizing a high-speed civil transport configuration and a delta-wing model, an investigation is underway to develop vortical scaling techniques for leading-edge vortices. The investigation of juncture flows from a configurational, as well as a basic fluid mechanics perspective, has been initiated with experimental studies. To assist in not only aircraft design but also in improved wind-tunnel-testing procedures, an investigation is underway to develop a technique to predict buffet onset for subsonic transport wings across the Reynolds number range of the NTF.

HIGHLIGHTS OF RECENT RESEARCH

High-Reynolds-Number Wake Measurement on Submarine Model

A research program sponsored by NASA and the Navy was conducted in the NTF to investigate Reynolds number (Rn) effects up to 50 million/foot on the complicated flow field around the stern of a 20 foot-long submarine model as shown in the figure below. Near the stern of the model, five-hole pitot-pressure probes and triaxial hot-film probes were located at 10 different radii from the hull centerline to the maximum hull radius. These probes were rotated 360°, in steps of 1.8°, to map the wake flow. Additionally, surface and boundary-layer measurements on the hull were obtained.

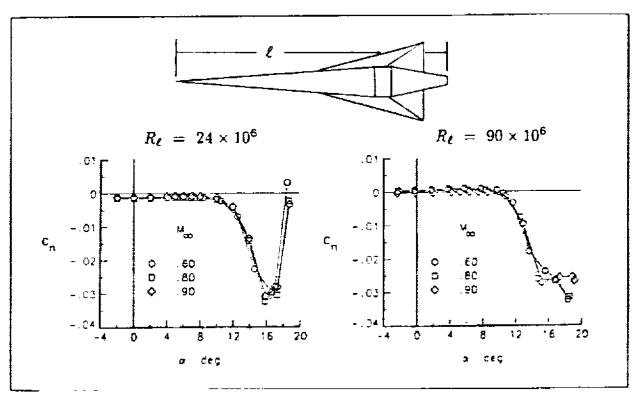
The data presented are the circumferential mean axial velocity, U, at each probe distance from the surface, r. These data indicate that the mean velocity flows are significantly affected by Reynolds number. (See figure below.)



Reynolds number effects on mean axial velocity in wake.

Transonic Reynolds Number Effects for a Slender Wing/Body Configuration

A wind-tunnel investigation was conducted in the NTF to determine the Reynolds-number effects on the transonic aerodynamics of a slender wing/body configuration. The model was comprised of a cone-cylinder-frustum body along with a unit aspect-ratio delta wing and was representative of a class of vehicles capable of very high-speed flight. Tests were conducted at Mach numbers ranging from 0.3 to 1.15 and Reynolds numbers ranging from 18×10^6 to 180×10^6 (based on body length). Both longitudinal and lateral-directional force and moment data were obtained. At 0° sideslip and high angles of attack, asymmetric flow separation occurred causing significant lateral-directional forces and moments. A sample result is presented for the yawing moment at 0° sideslip. The data demonstrate significant Reynolds number effects above an angle of attack of approximately 14°. The low Reynolds number data show a nonlinear reversal in the yawing moment trend with angle of attack above 14°, whereas the high-Reynolds-number data do not evidence this effect in the angle-of-attack range investigated. The high-Reynolds-number data also show evidence of increased compressibility effects at the high angles of attack. See figures below.



Reynolds number effect on yawing moment.

SUPERSONIC/HYPERSONIC AERODYNAMICS BRANCH

THE MISSION

The Supersonic/Hypersonic Aerodynamics Branch (SHAB) conducts theoretical and experimental research to improve the high-speed aerodynamic performance for civil transports, high-performance military aircraft, and missiles. The research efforts are focused both on developing the fundamental technology base and on specific technology applications. This research includes investigation of basic flow phenomena as well as the development and assessment of new concepts related to aerodynamic loads, performance, heating, stability, and control for these vehicles. Supporting research includes the development of new experimental techniques and computational methods for design and analysis.

Performs research directed toward the development of analytical methods for the aerodynamic design and analysis of advanced missiles and aircraft. The work is accomplished by adaptation of established procedures, independent in-house development, and contract and grant activities. When possible, analytical work is supplemented by related experimental studies. Emphasis is placed on computer-implemented methods capable of a reasonable degree of accuracy compatible with requirements of user convenience, solution speed, and breadth of applicability. The work encompasses studies of wave drag, skin friction, drag due to lift, static and dynamic stability and control, aerodynamic interference, and flow-field properties.

Originates and develops new and innovative aerodynamic-missile concepts and provides the technology base required for these concepts to be incorporated into future system designs. A companion activity evaluates both experimentally and analytically the performance of existing missile systems and executes further studies which will lead to the development of technology to improve these systems.

Responsible for the operation of the Unitary Plan Wind Tunnel (UPWT), the 20-Inch Mach 6 Wind Tunnel (20-Inch Mach 6 WT), and the Mach 8 Variable Density Tunnel (Mach 8 VDT). Develops new test methods and data-reduction techniques for these facilities and coordinates in-house research programs, cooperative studies, and studies in support of industry, DOD, and other government agencies.

THE PERSONNEL

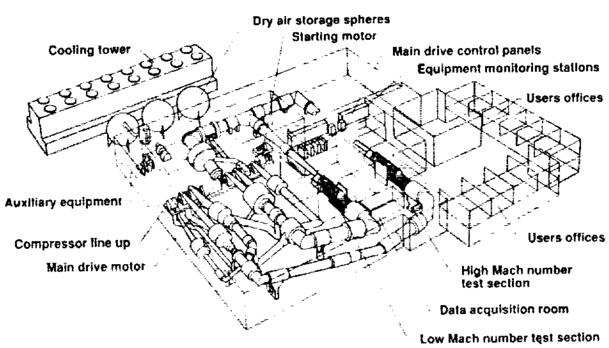
ALLEN, JERRY M.
BAUER, STEVEN X. S.
BLAIR, ADOLPHUS B., JR.
BOYDEN, RICHMOND P.
COCKRELL, CHARLES E., JR.
COLLINS, IDA K.
CORLETT, WILLIAM A.
COVELL, PETER F.
DILLON, JAMES L. (Assistant Branch Head)
DRESS, DAVID A.
FLAMM, JEFFREY D.
FORREST, DANA K.
HARRELL, CHERYL L.
HERNANDEZ, GLORIA
HOWELL, DOROTHY T.
HUEBNER, LAWRENCE D.
MCMILLIN, SUSAN NAOMI
MILLER, DAVID S. (Branch Head)
MONTA, WILLIAM J.
SHAW, DAVID S.
WATSON, CAROLYN B.
WILCOX, FLOYD J., JR.,
WITTE, DAVID W.

THE FACILITIES

Unitary Plan Wind Tunnel

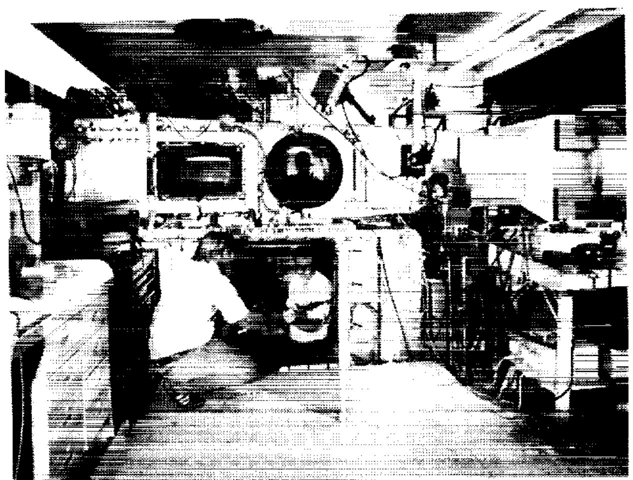
Immediately following World War II, the need for wind-tunnel equipment to develop advanced airplanes and missiles was recognized. The military and the National Advisory Committee for Aeronautics (NACA) developed a plan for a series of facilities which was approved by the United States Congress in the Unitary Wind Tunnel Plan Act of 1949. This plan included five wind-tunnel facilities, three at NACA laboratories and two at the Arnold Engineering Development Center. The LaRC UPWT was among the three built by NACA. (See figures below.) The UPWT is a closed-circuit, continuous-flow, variable-density tunnel with two 4- by 4- by 7-foot test sections. The low-range test section has a design Mach number range of 1.5 to 2.9, and the high-range section Mach number varies from 2.3 to 4.6. The tunnel has sliding-block-type nozzles which allow continuous variation in Mach number while on-line. The maximum Reynolds number per foot varies from 6×10^6 to 12×10^6 depending on Mach number. The tunnel is used for force and moment, pressure distribution, jet effects, dynamic stability, and heat-transfer studies. Flow-visualization data, which are available in both test sections, include schlieren, oil flow, and vapor screen. Since this facility came on-line in 1955, it has averaged over 1000 hours of operation per year. A major portion of the investigations performed in this facility deals with determining stability, control, and performance characteristics of supersonic aircraft.

LANGLEY UNITARY PLAN WIND TUNNEL

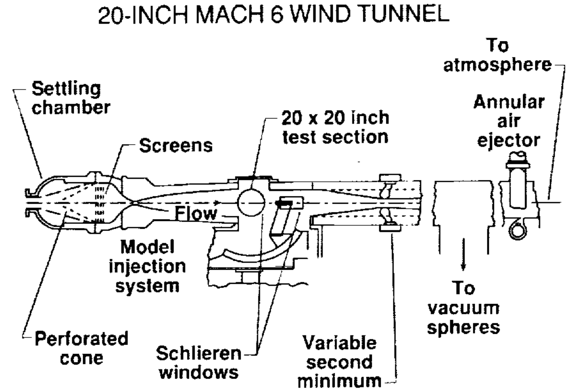


20-Inch Mach 6 Wind Tunnel

The 20-Inch Mach 6 WT is located in the Gas Dynamics Laboratory. This facility operates in the blow-down mode and has a fixed two-dimensional nozzle with parallel sidewalls which lead to the 2- by 25-inch test section. Pressure can be varied from approximately 30 to 470 psia and total temperature up to 500° F. The supply air is heated with an electric resistance heater. The tunnel discharges to either vacuum spheres or to the atmosphere with the aid of an annular ejector. The model support system has remote controls for angles of attack and yaw and can be injected/retracted. A remotely controlled three-degree-of-freedom flow-field survey mechanism is available. Typical tests include force and moment, pressure, and heat transfer. Flow visualization includes schlieren, oil flow, and vapor screens. Optical access includes two 16-inch-diameter windows on each side of the test section near the model support system center of rotation, two 9- by 17-inch rectangular windows aft of the round ones, one 11.5- by 17.5-inch rectangular window on top of the test section, and one 12-inch-diameter window slightly aft of the rectangular one. This facility was constructed in 1958. A major upgrade is currently underway which includes on-site data acquisition/reduction, a closed-loop model-positioning system, and an improved flow-field survey mechanism. See figures below.



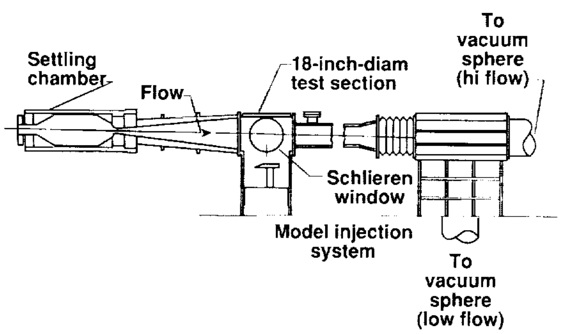
20-INCH MACH 6 WIND TUNNEL



Mach 8 Variable Density Tunnel

The Mach 8 VDT is located in the Gas Dynamics Laboratory (see figures below). This facility has an axisymmetric nozzle which leads an 18-inch-diameter test section and exhausts to a vacuum sphere. Total pressure can be varied from 15 to 3000 psia and total temperature up to 1050° F. Reynolds number per foot varies from 0.1×10^6 to 12×10^6. The supply air is heated with an electric resistance heater. The model support system is remotely controlled for angle of attack and is manually adjustable for angle of yaw. Optical access includes two 12-inch-diameter windows on each side of the test section and two 5- by 14-inch rectangular windows in the test-section ceiling. Typical tests include force and moment, pressure, and heat transfer. Flow visualization includes schlieren and oil flow.

M 8 VDT



This facility was constructed in 1952. It has been on standby status since the mid-1970's. A major rehabilitation is currently underway which includes supply air filters, a new nozzle, and a new test section. It is anticipated that this facility will return to an operational status in mid-1991.

THE CURRENT PROGRAM

The current supersonic program is focused on HSCT and high-performance military aircraft research. In support of HSCT, NASA LaRC has initiated a High-Speed Airframe Integration Research (HiSAIR) activity to combine the technology developments from aerodynamics, structures, electronics, flight controls, etc. The SHAB is presently providing the supersonic aerodynamic information for this activity by testing an HSCT wind-tunnel model and using computational methods to scale the data from wind-tunnel model to aircraft-flight conditions. In support of both HSCT and high-performance military aircraft research, the SHAB has ongoing research to develop performance-improvement technologies such as friction-drag reduction using natural laminar flow, leading-edge vortex flow management techniques, conical-flow wing designs, and component arrangements for positive interference. Also in support of high-performance military aircraft, experimental and computational studies to determine store-carriage drag and separation characteristics are producing new and innovative performance improvement concepts; and, in cooperation with several DOD agencies, studies of foreign missiles are being performed to assess the present threat. Studies are also underway to upgrade existing U.S. missiles.

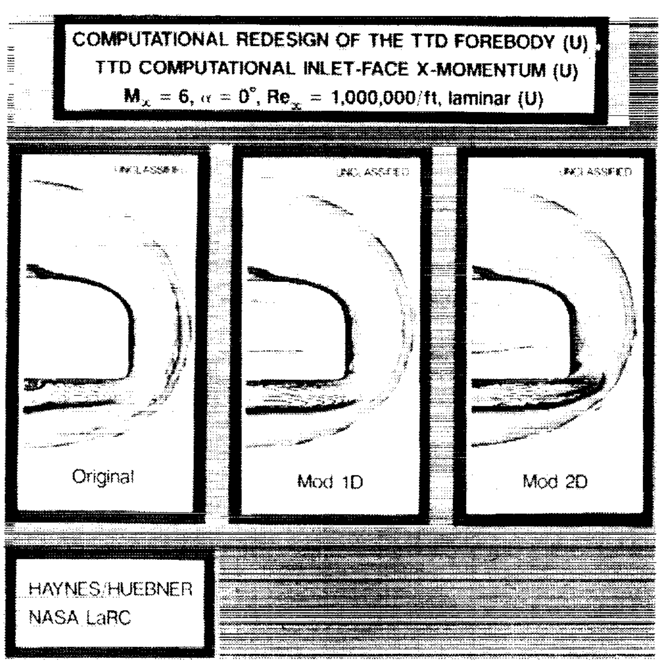
For the past several years, the hypersonic program's primary focus has been in developing technologies directly in support of the NASP Program; however, there has also been a small but very successful effort in developing a new class of waverider aerodynamic shapes which are optimized including viscous effects. The NASP activity is addressing the very difficult problem of studying the airframe-propulsion integration issues by wind-tunnel testing of powered models in small hypersonic facilities and applying appropriate CFD codes. To facilitate this activity without compromising the NASP contractor's proprietary information, NASA LaRC created the Test Techniques Demonstrator (TTD) which is a generic NASP-like concept. The SHAB is presently testing and analyzing the following three TTD models: (1) an unpowered model of the complete TTD to determine static and dynamic stability and unpowered performance, (2) an unpowered forebody-inlet model to study forebody-inlet interactions and determine forebody forces, and (3) a powered model of the complete TTD with metric afterbody to determine external nozzle characteristics. The waverider research is proceeding with the experimental and computational evaluation of both a Mach 4 and Mach 6 design; if the evaluation produces promising results, airframe-propulsion integration studies will be initiated.

HIGHLIGHTS OF RECENT RESEARCH

Test Technique Demonstrator Forebody Redesign

The design of scramjet-powered hypersonic vehicles is mainly driven by the necessity to obtain good propulsion/airframe integration. The airframe must provide efficient compression and expansion surfaces for the scram-jet propulsion system. In support of the NASP Technology Maturation Plan

(NASP TMP), the LaRC's TTD was designed to address these issues. Unfortunately, studies of the original TTD geometry revealed undesirable forebody-flow characteristics. Therefore, an effort was undertaken to computationally redesign the TTD forebody to eliminate or minimize the undesirable characteristics. The flow-characterization study of the original TTD geometry identified three undesirable characteristics: (1) unexpected forebody shocks, (2) significant lower-surface inflow, and (3) substantial centerline boundary-layer accumulation. To address these problems, geometry modifications to the TTD were parametrically evaluated using the CFL-3DE Parabolized Navier-Stokes (PNS) computational method. As shown in the figure below, the geometry modifications which were incorporated influenced the x-momentum and resulted in a more uniform boundary layer at the inlet face. Furthermore, examination of particle trace, surface pressure, mass flow, and drag data indicates that the Mod 2D geometry modifications eliminate the undesirable forebody shocks as well as significantly increasing inlet-face mass flow and reducing forebody drag. Wind-tunnel models of the modified TTD were fabricated and a validation test was conducted in the 20-Inch Mach 6 facility. Oil-flow, schlieren, and inlet-face pitot-pressure data were obtained and compared with similar data for the original TTD. Comparison of these data validated the modified TTD redesign which greatly enhances the TTD inlet-face flow quality, making it more representative of NASP TMP goals and more suitable for conducting scramjet propulsion/airframe integration studies.

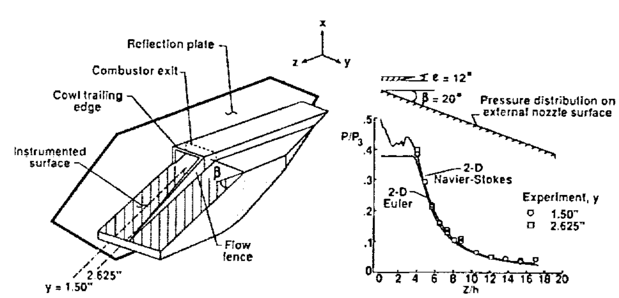


Scramjet Exhaust Simulation Studies

Hypersonic, airbreathing vehicles require careful integration of the scramjet propulsion system and the airframe since the airframe afterbody also serves as an external nozzle surface for the scramjet. The exhaust flow pressure field yields a large thrust and pitching moment that affect the total vehicle performance. It is impractical to perform combustion in a small-scale wind-tunnel model; and, therefore, other means of simulating the scramjet-exhaust pressure field must be used.

A study was performed at Grumman Aerospace Corporation under NASA contract to identify cold gases which correctly simulate the pressure field of a hot scramjet exhaust. This study showed that a mixture of Argon and Freon correctly matches the inviscid exhaust simulation parameters of combustor exit Mach number, static-pressure ratio, and the ratio of specific heats. These findings were experimentally validated against hydrogen-air combustion products in a shock tunnel at Mach 6 and 8. Subsequently, the Argon/Freon scramjet-exhaust-simulation technique was adopted for use in long-duration conventional hypersonic tunnels. The following figure shows a generic hypersonic vehicle afterbody model which uses substitute gases. The cross-hatched area on the external nozzle surface indicates the area instrumented with pressure ports. Experimental surface-pressure data are shown compared with computed results from a two-dimensional Euler code, SEAGULL, and a two-dimensional full Navier-Stokes code developed by Dr. Oktay Baysal of Old Dominion University. At the free-stream condition of Mach 6, the surface pressures inside the flow fence are well predicted by both computational methods.

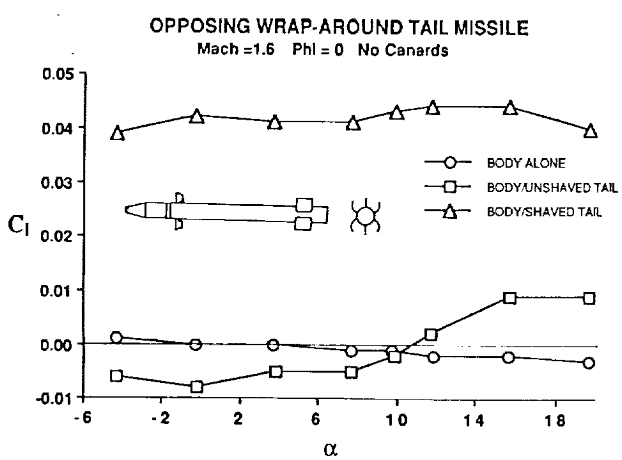
2-D CFD PRESSURE PREDICTION
ON EXTERNAL NOZZLE
Argon/Freon Exhaust Simulation
$M_\infty = 6$ NPR = 120



Aerodynamics Of A Tactical Missile With Opposing Wrap-Around Tail Fins

Tube-launched tactical missiles often employ wrap-around fins which can be folded against the missile body for compact storage within the launch tube. These fins are normally curved in the same direction to match the body surface, with each fin covering about one-fourth of the body circumference when folded. When these curved fins are used as tail surfaces, mechanical deployment can be a problem due to the limited space available in the aft region of a missile. Another method of deploying this type of fin is to use the energy of the rocket exhaust to blast the fins into their deployed position. This method would require four side-mounted exhaust ports to deploy the four conventional wrap-around fins.

A study was conducted on an alternate design whereby the fins are mounted on the body in opposing pairs, with one folded beneath the other prior to deployment. In this manner, only two nozzles would be required to deploy all four fins. This configuration has vertical symmetry; thus, it will not possess the inherent rolling motion of the conventional wrap-around design. To permit this configuration to be controlled with simple planar canards, a rolling moment must be produced on the configuration. One unconventional method of achieving this moment is to shave very shallow bevels on each tail fin.



Tail fins with and without beveling were tested at low supersonic Mach numbers, over a range of angles of attack, roll angles, and canard-deflection angles. Typical rolling-moment characteristics for this configuration are shown in the figure above. The unbeveled tails and the body alone produce very little rolling moment. The beveled tails, however, introduce a substantial rolling moment to the configuration. Moreover, this moment is virtually constant with angle of attack. Thus, shallow beveling of the opposing wrap-around tail fins appears to be an effective method of producing the desired rolling motion for this configuration.

Multigrid Solutions Of Navier-Stokes Equations On Overlapped Grids

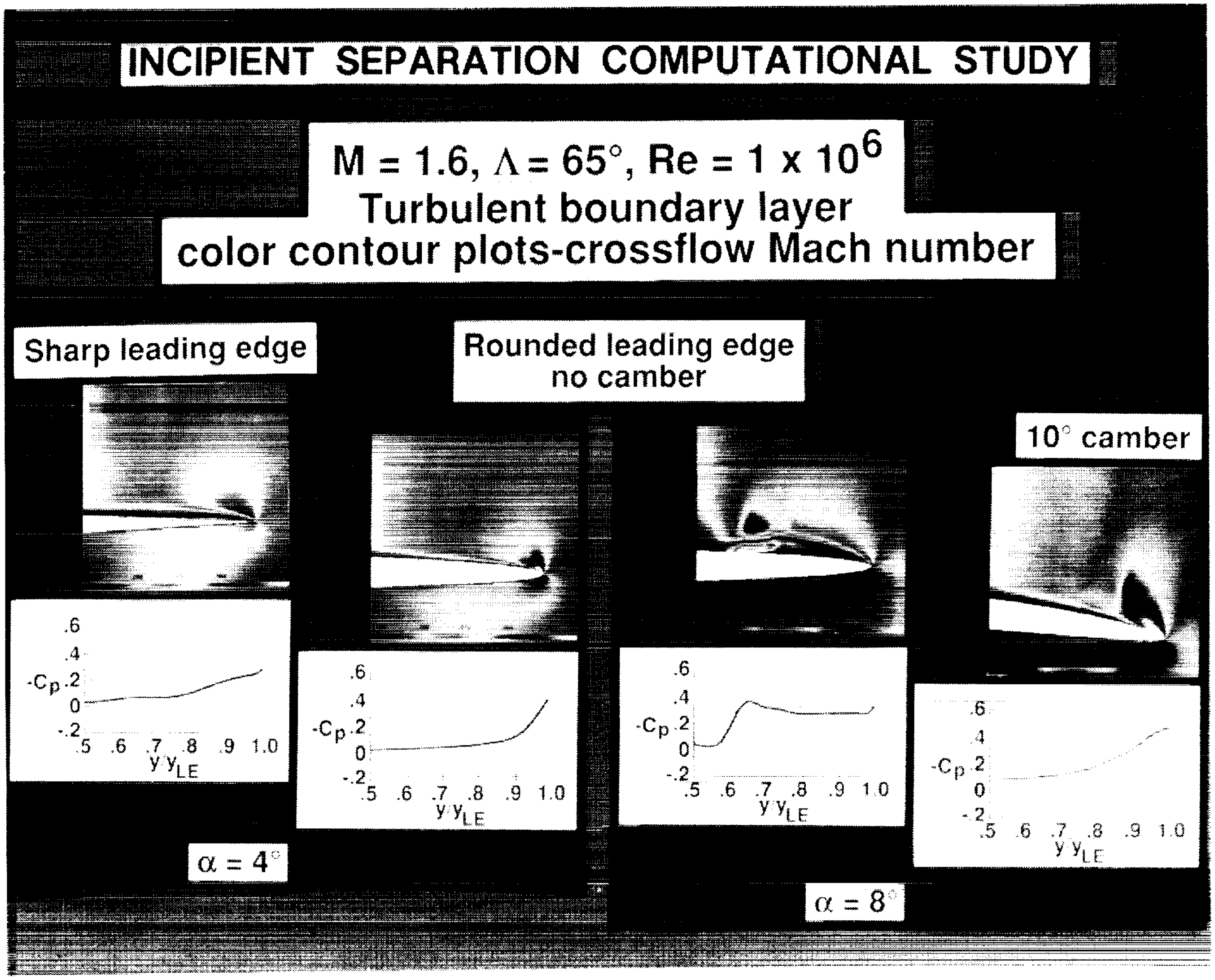
One of the limitations in solving the partial-differential equations of the fluid flow is the adequate discretization of the physical domain about complex configurations. One method of avoiding this limitation is to decompose the global-flow domain into overlapped subdomains, which can accept easily generated, fairly smooth, and curvilinear component grids with no singularities. The Multi-Geometry-Grid-Embedder (MaGGiE) computer code is developed, based on the Chimera code of NASA Ames Research Center and Calspan AEDC. It generates overlapped composite grids at sequentially coarser levels for a multigrid and finite-volume solution scheme. Regions of a component grid common to the others are removed; thus creating holes, to avoid excessive interpolation.

The CFL3D computer code is modified to solve the complete Navier-Stokes equations and perform multigrid convergence acceleration despite the existence of holes and overlapped regions. Hence, this CFL3D-based computer code (VUMXZ3) solves the viscous-flow equations using an upwind, multigrid scheme on Chimera-type overlapped grids and zonal grids. It combines the advantages of an efficient, geometrically conservative, and minimally dissipative solution algorithm with the flexibility of the domain decomposition using overlapped or nonoverlapped grids.

The supersonic flow past an ogive-nose cylinder in the proximity of a flat plate has been simulated to test the applicability, accuracy, and convergence attributes of MaGGiE and VUMXZ3 codes. (See figure below.) Currently, the flows past an ogive-nose cylinder with an L-shaped sting in and near a rectangular cavity are being simulated. Wind-tunnel tests have been conducted for this configuration for the validation of these computer codes.

Incipient Leading-Edge Separation

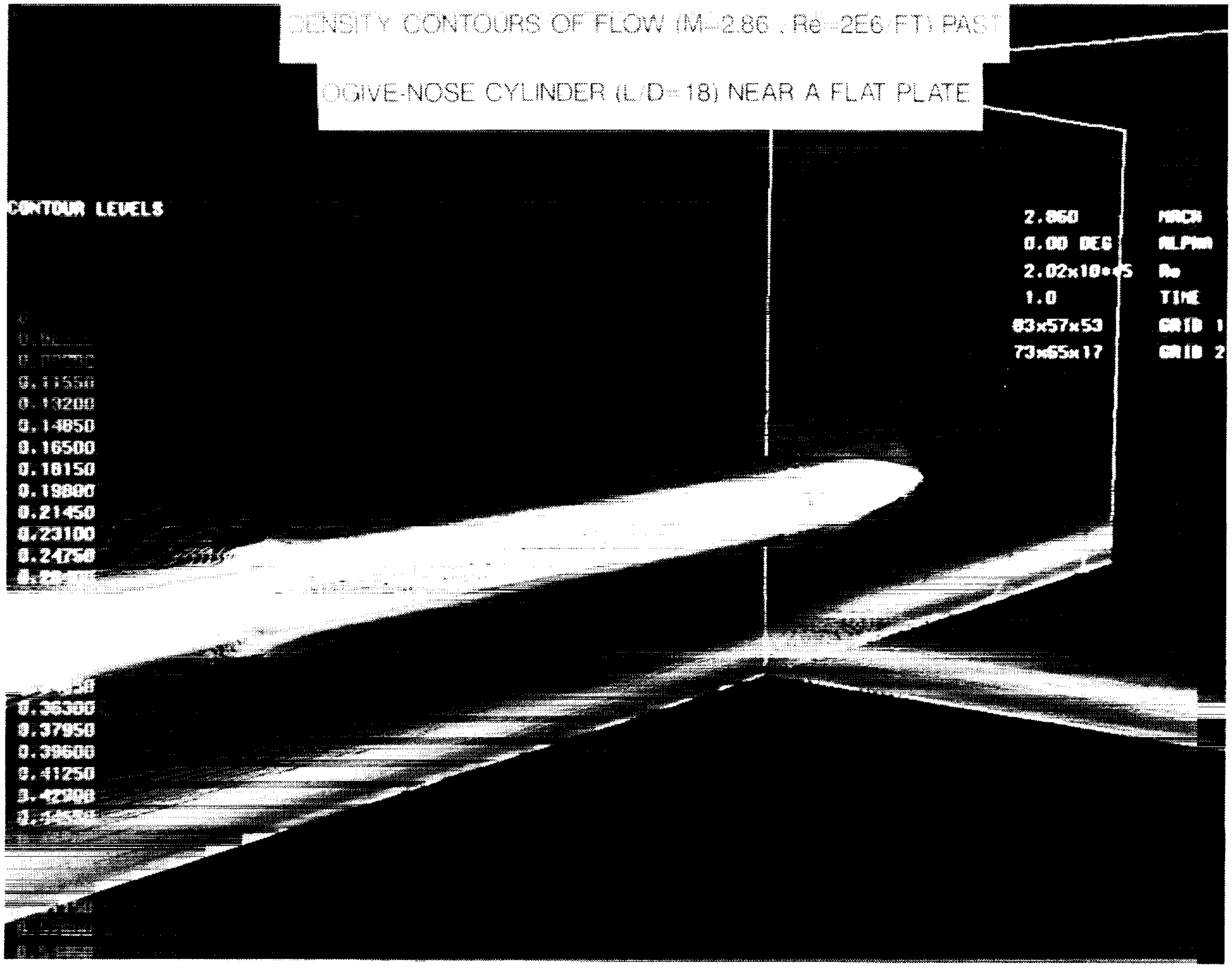
In supersonic wing design, the aerodynamicist would like to have control over the formation of leading-edge separation in order to make optimum use of both separated and attached-flow conditions at the leading edge. Thus, it becomes important to be able to predict and understand the effects of certain geometric parameters on the initial formation of leading-edge separation. A computational study was conducted to determine the effects of leading-edge radius and camber on the initial formation of leading-edge separation on the leeside of a 65° conical delta wing at Mach 1.6. Conical Navier-Stokes solutions were obtained on geometries which varied in leading-edge radius and/or spanwise camber.



A summary of the results is shown in the figure below. The test conditions were Mach 1.6, a local Reynolds number of 1×10^6, and a turbulent boundary-layer condition. The computational data are presented in the form of cross-flow Mach number contours and surface-pressure distributions. The data show that at an angle of attack of 4°, the leading-edge separation observed on the sharp leading-edge geometry is prevented through the use of leading-edge radius. At the higher angle of attack of 8°, the rounded leading-edge geometry has a leading-edge separated-flow pattern which can be prevented through the use of spanwise camber. Based on this study, three wind-tunnel models are being designed to verify these results.

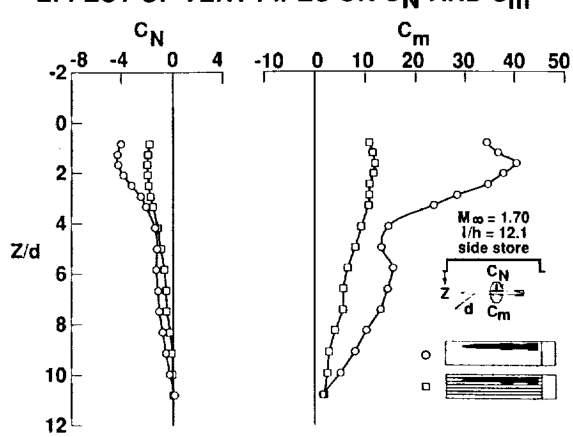
A Passive Venting Technique To Facilitate Store Separation From Shallow Cavities

A study was recently completed in the LaRC UPWT to evaluate the effectiveness of a passive venting technique for improving the aerodynamic characteristics of stores separating from shallow cavities. The large pressure gradients and flow-turning angles that normally occur for shallow cavity flow fields can create large pitching moments on a store during separation and make it pitch up; and, in some cases, fly back into the cavity. A characteristic pressure distribution for shallow cavities consists of low pressures occurring over the forward part due to the flow expanding into the cavity and large pressures occurring over the rear part due to the flow compressing as it leaves the cavity. The existence of this pressure difference led to the passive-venting concept investigated in this study. This concept consists of small pipes installed on the cavity floor which permit high pressures at the rear to vent to the low-pressure region at the front of the cavity. The resulting increase in pressure at the front will then reduce the extent of flow expanding into the cavity and decrease the local flow-turning angles, which should reduce the store pitching moments.



Typical data results from the present study for a store separating from a shallow cavity are presented in the figure and show that the store pitching moments at a free-stream Mach number (M) of 1.70 were significantly reduced with the vent pipes installed as indicated by the square symbols. These results are for the case of the store separating from a side carriage-position in the cavity. Similar results were also obtained at this Mach number with the store separating from the center carriage position.

EFFECT OF VENT PIPES ON C_N AND C_m



PROPULSION AERODYNAMICS BRANCH

THE MISSION

The Propulsion Aerodynamics Branch (PAB) conducts theoretical and experimental research on aerodynamic phenomena at subsonic, transonic, and supersonic speeds, with special emphasis directed toward studies of engine-airframe integration problems. Research is aimed at understanding and accounting for the mutual interference effects which exist between components of the propulsion system and the airframe and installation concepts which will result in a significant increase in aircraft cruise and maneuvering performance.

Perform research directed toward the development of analytical methods for predicting the propulsion/airframe integration characteristics of advanced aircraft concepts. While the goal of this research is to develop methods to address the complete aircraft configuration with propulsion effects, the current status is aimed at developing methods to predict the characteristics of the integrated nozzle/afterbodies, inlet forebodies, and turboprop/turbofan-wing integration. The approaches under consideration include the utilization of both the approximate patched methods and the more complex Euler and Navier-Stokes equations. Where necessary, the analytical work is supplemented by related experimental studies which will be used to verify and determine the utility of test analytical methods.

Develop new and innovative propulsion integration schemes which will result in significant increases in aircraft performance. In this research, the emphasis will be on integrating advanced axisymmetric and nonaxisymmetric nozzles into fighter aircraft configurations and turbofan and turboprop nacelles into transport aircraft. Studies will be conducted at subsonic, transonic, and supersonic speeds.

Responsible for the operation of the 16-Foot Transonic Tunnel complex, which includes the 16-Foot Transonic Tunnel (16-Foot TT), the Static Test Facility, and the 16- by 24-Inch Water Tunnel (16- by 24-Inch WT). In this capacity, the program objective is to coordinate the requirement for research time to conduct in-house research programs, cooperative studies, and support of industry and other government agencies.

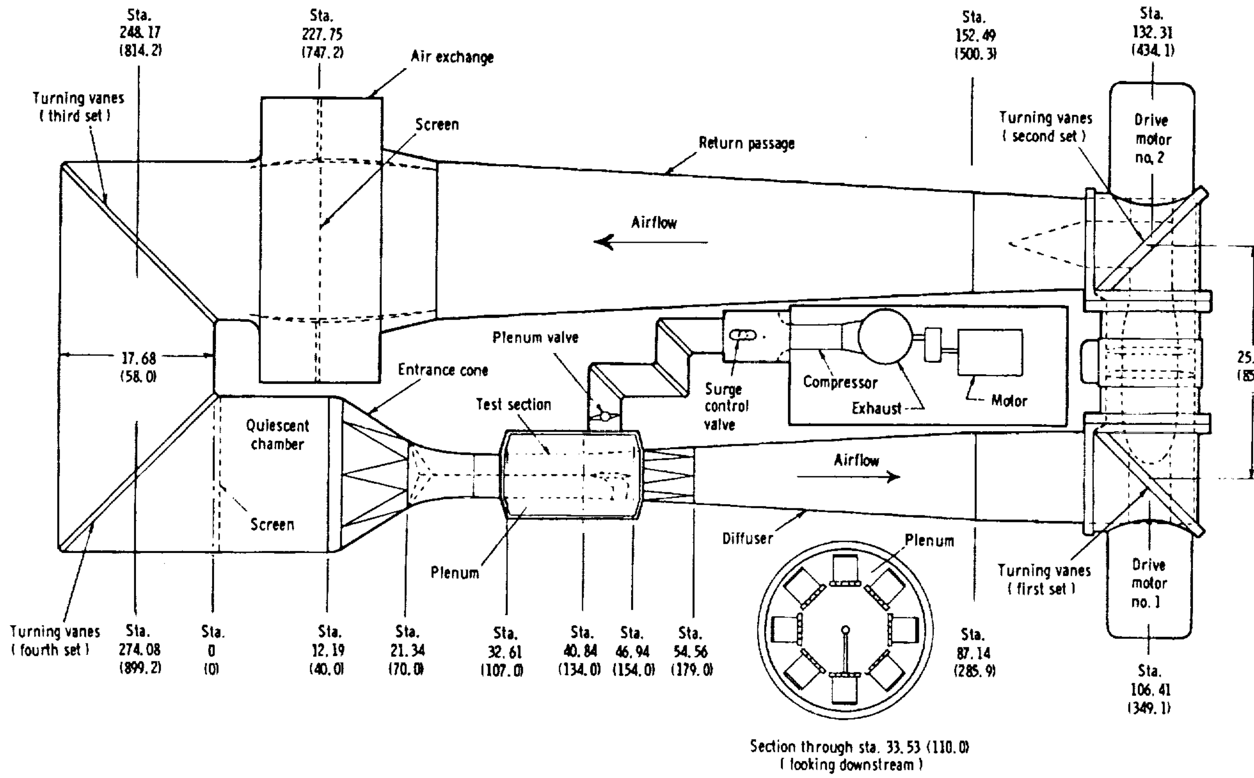
THE PERSONNEL

ABEYOUNIS, WILLIAM K.
ASBURY, SCOTT C.
BANGERT, LINDA S.,
BARE, E. ANN
BERRIER, BOBBY L. (Branch Head)
CAPONE, FRANCIS J.
CARLSON, JOHN R.
CARSON, GEORGE T., JR.
CLER, DANIEL L.
COMPTON, WILLIAM B., III
INGRALDI, ANTHONY M.
INGRALDI, MARSHA C.
KRIST, STEVEN E.
LAMB, MILTON
LEAVITT, LAURENCE D. (Assistant Branch Head)
MASON, MARY L.
MERCER, CHARLES E.
PAO, S. PAUL
PENDERGRAFT, ODIS C., JR.
RE, RICHARD J.
WING, DAVID J.
YETTER, JEFFERY A.

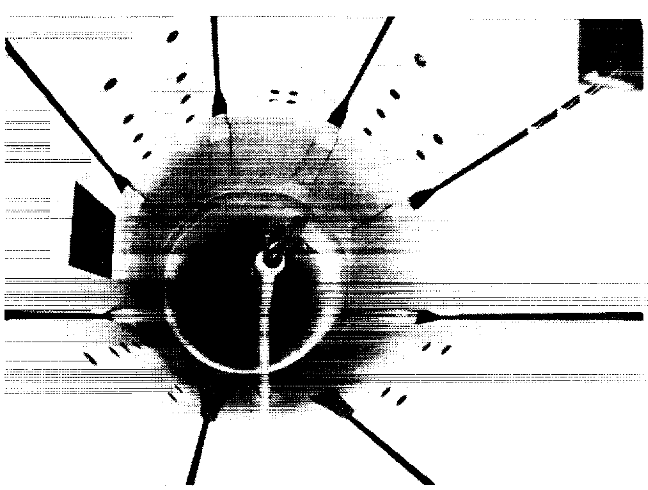
THE FACILITIES

16-Foot Transonic Tunnel

The LaRC 16-Foot TT shown in the figure below, which was originally placed into operation in 1941, is a closed-circuit, single-return, continuous-flow atmospheric tunnel. Speeds up to Mach 1.05 are obtained with the tunnel main-drive fans, and speeds from Mach 1.05 up to Mach 1.30 are obtained with a combination of main-drive and test-section plenum suction. The slotted octagonal test section shown in the figures on the next page, measures 15.5 feet across the flats. The tunnel is equipped with an air exchanger with adjustable intake and exit vanes to provide some temperature control. This facility has a main-drive power system consisting of two 30 000-hp motors driving counter-rotating fans. A 36 000-hp compressor provides test-section plenum suction.



The tunnel is used for force, moment, pressure, flow-visualization, and propulsion-airframe integration studies. Model mounting consists of sting, sting-strut, and semispan support arrangements; propulsion simulation studies are made with dry, cold, high-pressure air or with air-driven engine simulators.



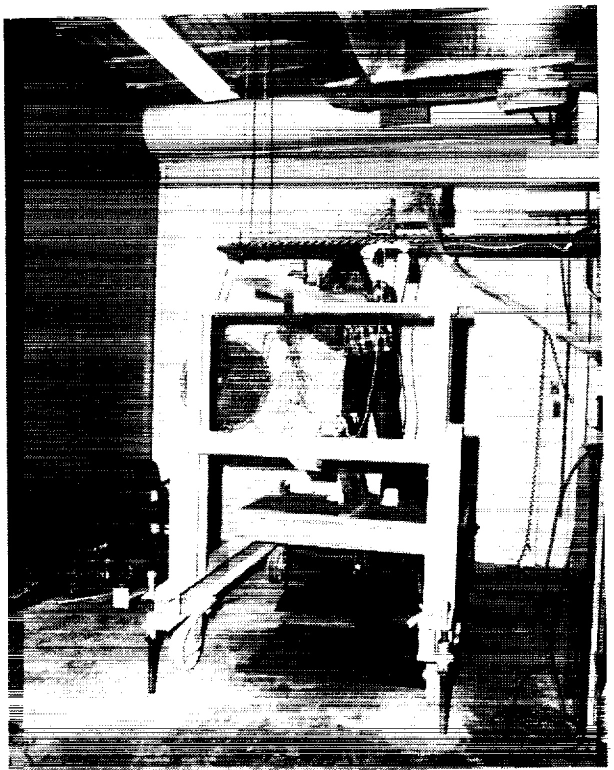
The 16-Foot Transonic Tunnel has recently undergone several major modifications. These include a floor-mount system to facilitate semispan model testing, a model-preparation area for model build-up and calibration, and a new model support strut with an angle-of-attack range from -10° to 25°, remote roll capability, and dual high-pressure air systems.

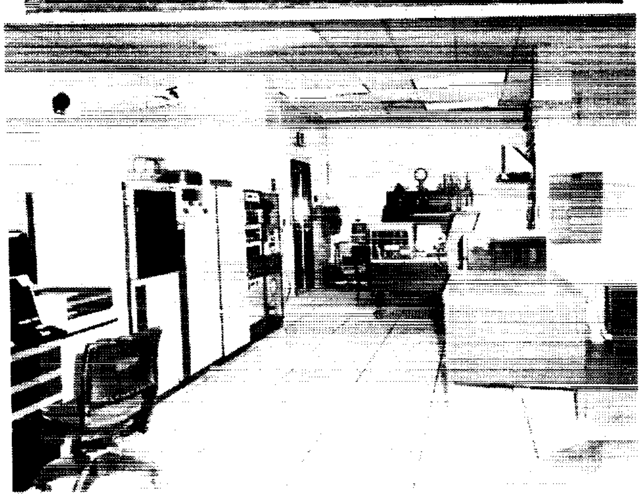
Static Test Facility

The Static Test Facility (STF) of the Langley 16-Foot TT complex shown in the first figure on the next page, has been used for nozzle internal-performance testing since the middle 1950's. Early testing was conducted using hydrogen peroxide to simulate jet exhaust; however, since 1976, high-pressure air has been used for exhaust simulation. Subscale nozzle performance tests are conducted in a high-ceiling room with the jet exhausting to the atmosphere. The control room shown in the second figure on the next page, is remotely located from the test area and a closed-circuit television camera is used to observe the model. This facility uses a high-pressure air system (similar to the one used in the 16-Foot TT) to simulate jet exhaust.

The facility is typically used for force, moment, pressure, and flow-visualization studies on multi-function nozzles. The impact of pitch and yaw thrust vectoring, thrust reversing, and novel exhaust nozzle concepts on internal performance is generally the objective of research conducted in the STF. Special features of this facility include an acoustically-triggered schlieren flow-visualization system for obtaining flow details resulting from particular acoustic structures.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



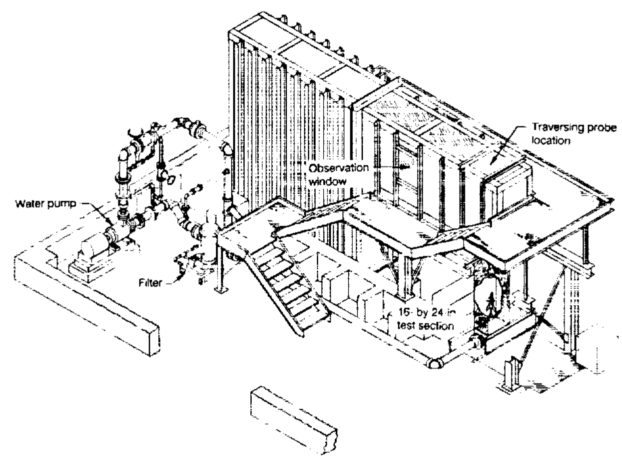


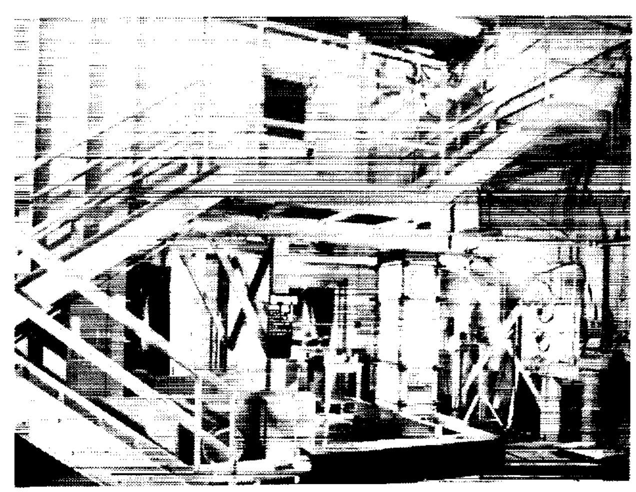
The facility has undergone recent upgrades, including the addition of a stand-alone data acquisition system to allow operation independent of the 16-Foot TT data-acquisition system. The test-bay-area walls have been acoustically treated to reduce noise in the test bay and control room. A dual-flow thrust stand is expected to be in use by the end of FY 1991, allowing performance testing of multistream-flow exhaust-nozzle concepts. An upgrade planned for FY 1992 will increase the mass-flow capability of the facility from 15 to 45 lb/s.

16- by 24-Inch Water Tunnel

The Langley 16- by 24-Inch WT is used for (see two figures below) flow-visualization studies at low Reynolds numbers. The tunnel has a vertical test section with an effective working length of approximately 4.5 feet. The test section is 16 inches high by 24 inches wide. All four sidewalls are Plexiglas to provide optical access. A pump transfers the water from the test-section exit to the reservoir upstream of the test section. The test section velocity can be varied from 0 ft/s to 0.75 ft/s. The unit Reynolds-number range for water at 78° F for this velocity range is 0 to 7.7×10^4/ft. The normal test velocity that produces smooth flow is 0.25 ft/s.

SKETCH OF THE LANGLEY 16- BY 24-INCH WATER TUNNEL





A sting-type model-support system positions the model. The model attitude can be varied in two planes over angle ranges of ~ 33° and ~ 15°. Operator-controlled electric motors are mounted outside of the test section to control the model position. The model position is read by the operator on a protractor mounted to the model support. Semispan models are mounted on a splitter plate supported by a sting with a lateral offset.

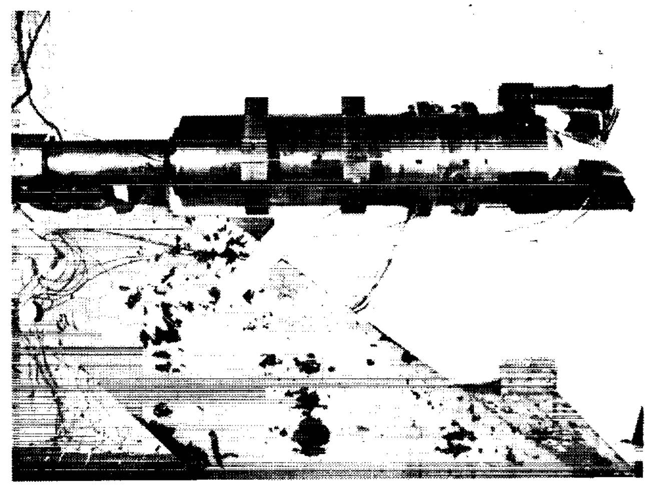
Ordinary food coloring is used as a dye to visualize the flow. The dye is supplied by three reservoirs under pressure so that up to three dye colors may be used. Dye may be ejected from small orifices on the model surface or injected upstream of the test section. The water tunnel was placed in operation in 1987 and has primarily been used to study vortical flows associated with forebodies, nose strakes, wings and wing-leading-edge extensions, and flaps. In some cases, these flows have been studied in the presence of flowing nozzles and inlets. By the end of FY 1990, a laser-light-sheet flow-visualization technique using fluorescent dye and a laser fluorescence anemometer will be available to obtain more quantitative flow information.

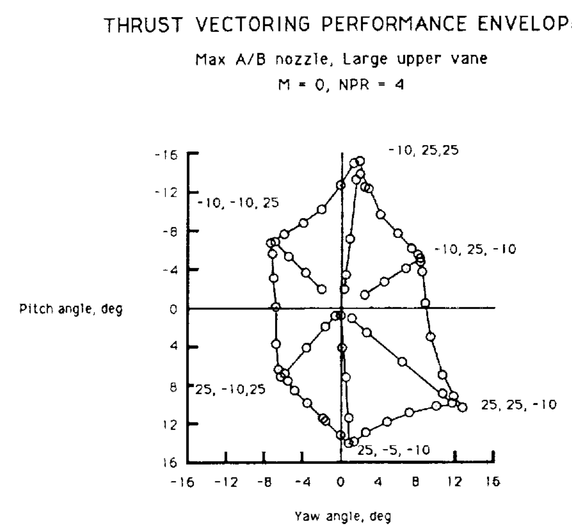
CURRENT PROGRAMS

The PAB maintains a balanced propulsion airframe integration research program in both high-performance aircraft and civil transports. The current emphasis in the high-performance-aircraft arena is centered around improving internal and installed performance of multifunction nozzles (nozzles capable of simultaneous pitch and yaw vectoring and/or thrust reversing), and the integration of these nozzles for propulsive control of highly-maneuverable aircraft. This work involves both experimental and computational efforts at subsonic, transonic, and supersonic speeds.

The civil transport research conducted within the PAB is also a multifaceted effort. Computational techniques are extensively used to address both turboprop and turbofan propulsion system installation issues. Early program emphasis was placed on the development of Euler analysis codes to allow researchers to address turboprop integration problems. These codes are being used by PAB researchers to verify the CFD methods and to assess performance of designs capable of reducing and possibly eliminating unfavorable turboprop installation effects. More recently, these same Euler codes are being used to address the complex integration issues associated with incorporating ultra-high-bypass ratio turbofans and ducted propellers in future transonic transport designs.

The PAB is also involved in experimental and computational research in support of the National Aerospace Plane (NASP). The current research effort addresses the off-design performance of the nozzle/afterbody and forebody/inlet. In particular, transonic nozzle drag reduction and transonic/supersonic forebody flowfields at the inlet face are being worked to provide as efficient a propulsion airframe integration as possible. Both experimental research models and three-dimensional Navier-Stokes computational techniques are being used.





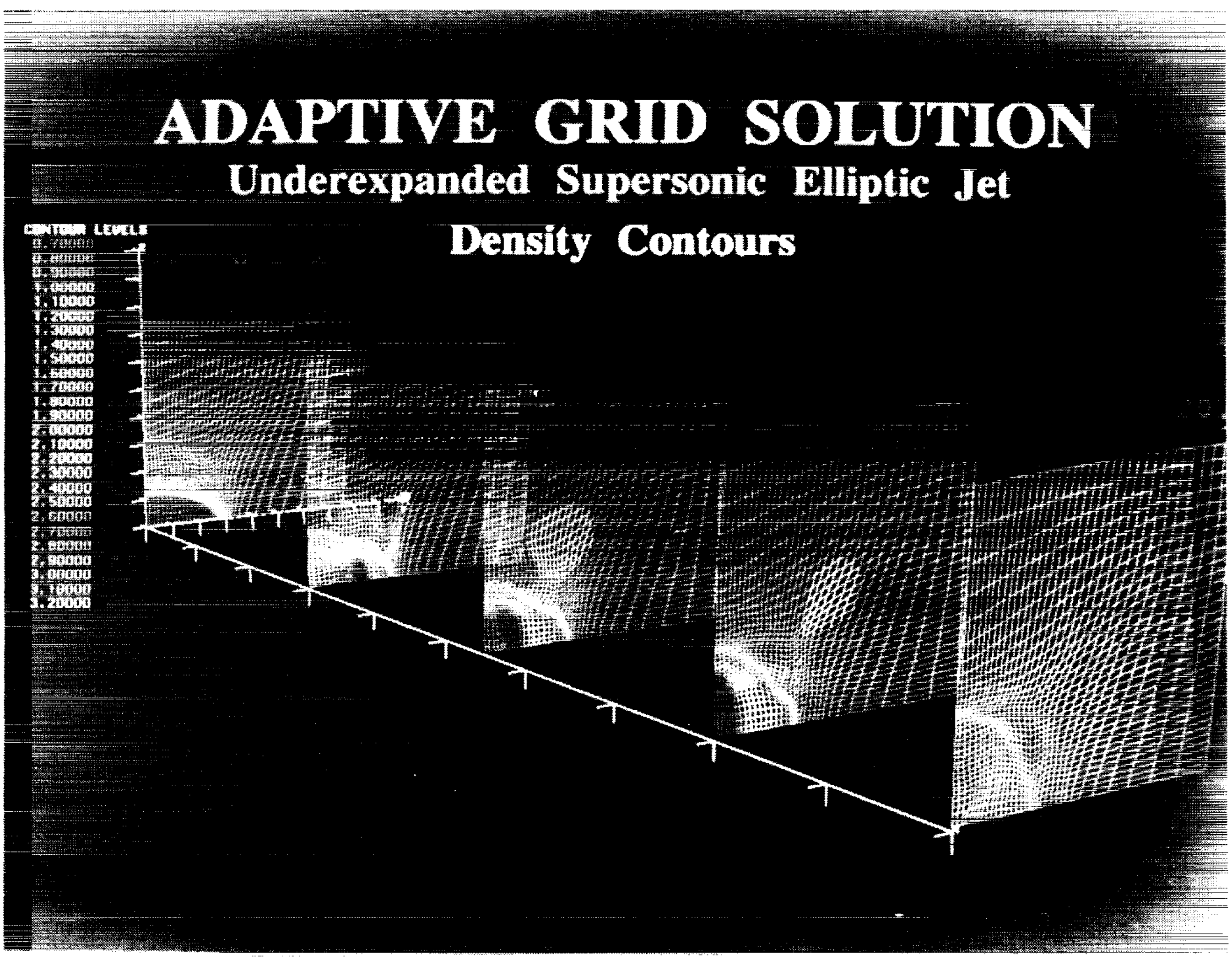
HIGHLIGHTS OF RECENT RESEARCH

Static Thrust Vector Envelope for F-18 HARV Multiaxis Thrust Vectoring System

An investigation has been conducted to evaluate the effectiveness of the multiaxis thrust-vectoring system for the F-18 HARV. This thrust vectoring system consists of three externally mounted vanes on each exhaust nozzle of the F-18 aircraft. The investigation was conducted in the STF of the NASA LaRC 16-Foot TT. Four hundred configurations consisting of various combinations of vane deflection at two nozzle power settings were tested at nozzle pressure ratios up to 6.0. The model used for this investigation is shown in the first figure above. The measured thrust-vector envelope from this investigation is shown in the plot of pitch angle versus yaw angle (second figure above). These results indicated that the thrust-vectoring system will provide the required pitch-thrust-vectoring capability. However, only about 85 percent of the required yaw-thrust vectoring will be supplied by this system. Of course, external flow effects on this type of multiaxis control system could be substantial. A future test utilizing an F-18 propulsion model will be used to investigate these external flow effects on the thrust-vectoring system.

Navier-Stokes Code (PAB3D) for Jet Plume Aerodynamics

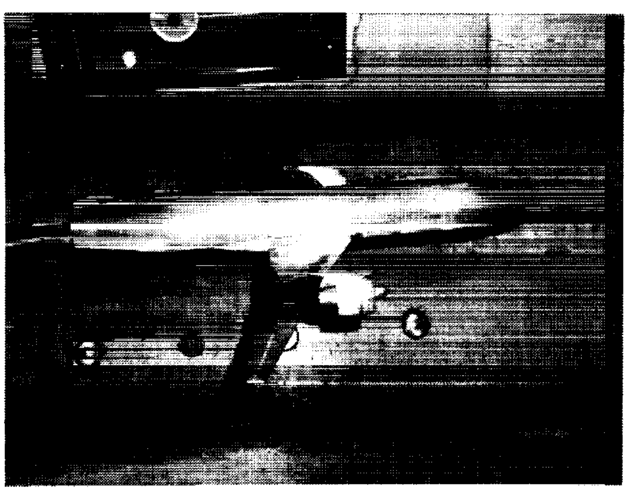
The Navier-Stokes code (PAB3D) for jet-plume aerodynamics is based on the 3-D time-dependent Reynolds-averaged Navier-Stokes equations written in strong conservation form. Finite-volume, upwind-biased flux-splitting schemes are used in the implementation. Solver options include time-dependent, PNS, and space-marching modes. The multiblock, multizone interface algorithm built into this code allows general application to complex configurations and propulsion integration. Options for several turbulence models (mixing length, Baldwin and Lomax, Johnson and King, and Goldberg) and an adaptive-grid option for jet-plume calculations have recently been added to the code. The figure below shows a successful application for calculations of an elliptic supersonic jet. Unique features such as the nonsymmetrical growth of the shear layer leading to axis-switching and the reduced shock-cell volume have been captured in this calculation. This code has been validated for circular-jet aerodynamic analysis, and data comparisons for square and elliptic jets are in progress. The PAB3D code has been recently used for jet/afterbody flow analysis to evaluate the accuracy of several turbulence models: Baldwin and Lomax, Johnson and King, and Goldberg. Surface-pressure distributions obtained with the Johnson and King model compared favorably with experimental data.



Effects of Advanced Superfan Nacelles on Low-Wing-Transport Model

An experimental study was conducted on a twin-engine, low-wing-transport model (see figure below) in the 16-Foot TT to determine wing/nacelle interference effects of two different designs of flow-through nacelles simulating very high bypass ratio advanced technology turbofans (BPR 18). For comparison, current technology turbofan nacelles (BPR 6) were also tested. Mach number (0.5 to 0.8) and angle of attack (-4° to 6°) were tested over ranges appropriate for a subsonic transport designed to cruise at $M = 0.77$ and $CL = 0.55$. Measurements included 6-component forces and moments and extensive external-surface static-pressure data on the wing, pylon, and nacelles. Minimum drag with each set of nacelles installed was determined by varying nacelle incidence angle from -3° (nose down) to 4° and nacelle tow-in angle from 0° to 2°, in 1° increments to get the optimum combination. Current technology nacelles were tested at 0.34 and 0.40 of the 1/2 span, while the two superfan nacelle configurations were only tested at the 0.40 position.

Results from this investigation indicate that although the drag of the large superfan nacelles was higher than drag for current technology nacelles, adverse interference effects from the superfan nacelles on the wing/body/pylon were negligible. Improved SFC performance of the superfan engines is expected to more than offset the increased absolute drag values.



Development of Euler Code for Turboprop Integration of Full Airplane Configurations

A transonic, three-dimensional, inviscid (Euler) CFD code was developed for the study of turboprop engine/airframe integration. This code may be used for either aft-mounted or wing-mounted full-airplane configurations. The propeller power effects are simulated by an actuator disk, where either components of force and work distributions or total pressure, total temperature, and swirl distributions are prescribed along the disk as boundary conditions for the flow solver.

Computational grids are generated, either algebraically or by solving elliptic partial-differential equations, for a given surface-geometry specification. The flow solver has multiblock capability and employs a multigrid scheme with successive mesh refinement to accelerate convergence. A separate embedded flow solver provides detailed flow characteristics in the vicinity of the propulsive unit. This embedded solver inherits its starting values from the global solution.

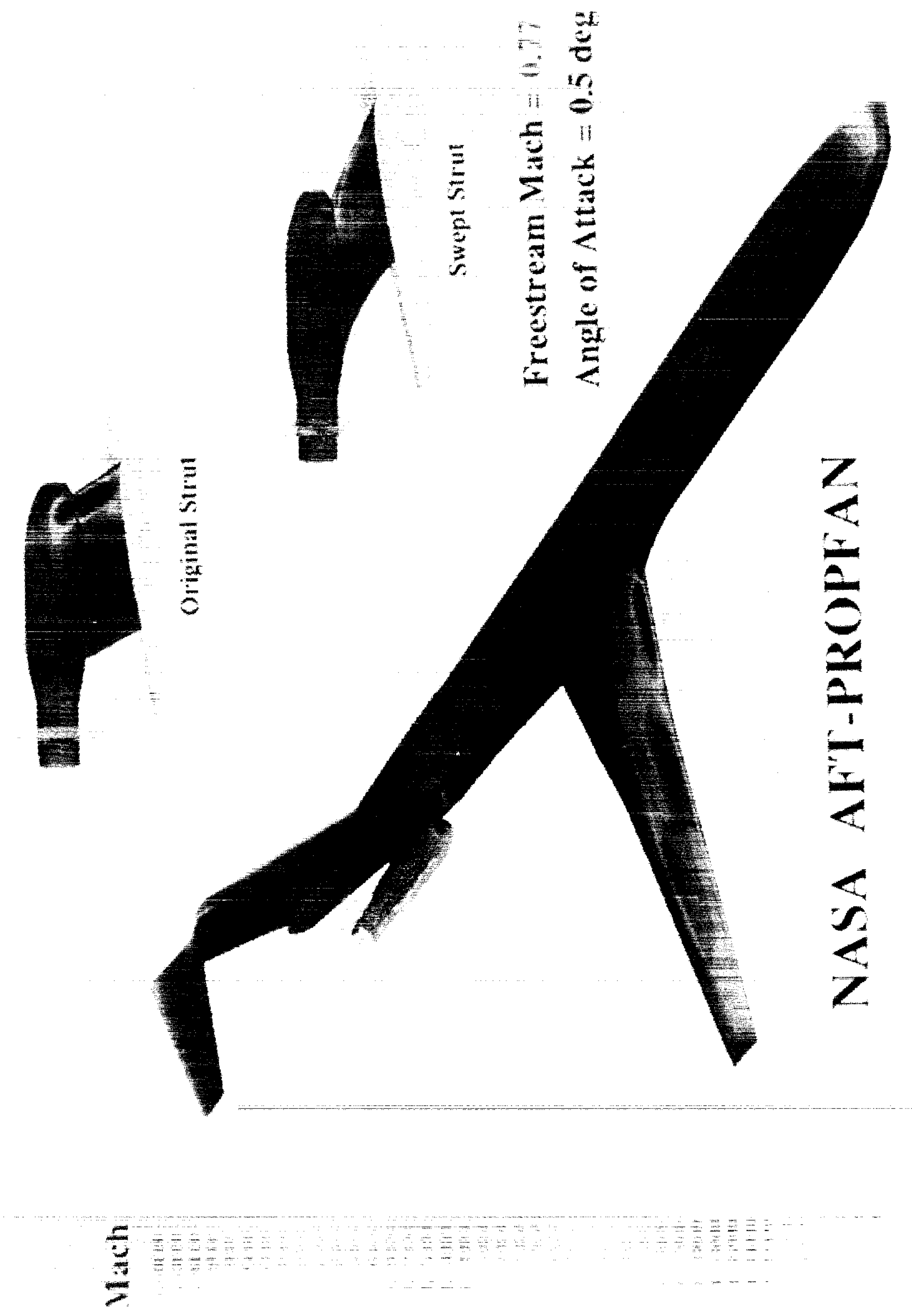
The code has been successfully applied to the NASA aft-turboprop model shown in the figure on the following page. Qualitative analysis of the CFD solutions has aided in the selection and modification of configurations for quantitative experimental analysis. For the example, shown in the figure, increasing the sweep of the strut is found to significantly reduce the adverse installation effects of propulsion integration.

A typical analysis takes about 2 hours on a CRAY-2, uses 295,000 grid points, and requires 12 megawords of memory in the single-block mode or 3 megawords of memory in the multiblock mode. Embedded mesh solutions consume less than 10 minutes of CPU on the CRAY-2. This code is currently being applied to the development of a CFD code for turbofan/superfan installations. Additionally, an inverse (target-pressure distribution) design code is being developed for transport applications.

Concepts for Alleviation of Adverse Inlet Spillage Interactions on External Stores

The spillage flow around inlets on fighter-type aircraft at reduced engine throttle settings has potential for adversely interacting with external stores. The resultant effect on aerodynamic surfaces due to the interaction with the vortical spillage flows can be structural damage due to severe buffeting and release of the stores into an unsteady flow environment. As a first step in addressing this problem, the personnel at the 16- by 24-Inch WT were asked to study the flow using a 1/48-scale twin-engine fighter model.

A test was conducted in the 16- by 24-Inch WT at a Reynolds number based on mean geometric chord of 8400. The model had flowing inlets which were connected to a series of valves and flowmeters so that the inlet flow rates could be varied. Flow visualization using colored dye indicated the path of the vortical spillage flows.



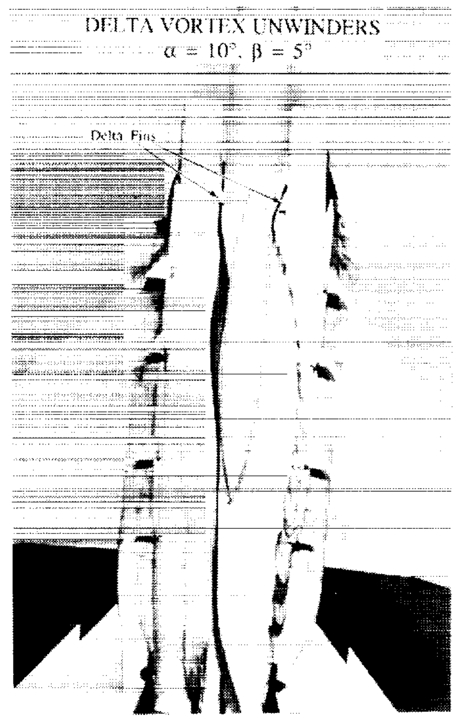
ORIGINAL PAGE IS
OF POOR QUALITY

A series of flow-control devices was studied for manipulating the spillage flows. Two approaches were taken to a solution of the problem. First, devices for deflecting the spillage flows were tested. These devices were vertical fins attached to the bottom of the fuselage and placed to deflect the spillage flow away from the external stores. The second approach was based on creation of auxiliary vortex flows to interact with the vortical spillage flows and alleviate their adverse interaction with the external stores. The devices which created these auxiliary vortex flows were also fins placed at various locations on the fuselage. The tip vortices from these devices were used to neutralize ("unwind") the spillage vortices, induce the spillage vortices away from the stores, or deflect and deform the spillage vortices.

The results of the study showed that the devices which created auxiliary vortices worked more effectively than the flow-deflecting fins. In particular, the fins that neutralized the spillage vortex and those that deflected and deformed it appeared most successful. The delta fins for neutralizing the spillage flow are shown in the figure below. They are located near the source of the vortical spillage flow and immediately neutralize the flow and slightly overcompensate it by reversing the direction of rotation. This flow-rotation reversal is not observable in the still photograph, but was verified during testing and recorded on videotape.

ORIGINAL PAGE IS
OF POOR QUALITY

The potential for effective utilization of devices for alleviation of adverse interactions of inlet spillage flows with external stores has been shown by the results of this test. Further studies, including detailed design, should be carried out in facilities where higher Reynolds numbers can be simulated and compressibility effects can be addressed.



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

National Aeronautics and Space Administration

Report Documentation Page

1. Report No.
NASA TM-102698

2. Government Accession No.

3. Recipient's Catalog No.

4. Title and Subtitle
Overview of the Applied Aerodynamics Division

5. Report Date
June 1991

6. Performing Organization Code

7. Author(s)

8. Performing Organization Report No.

9. Performing Organization Name and Address
NASA Langley Research Center
Hampton, VA 23665-5225

10. Work Unit No.
505-61-01-01

11. Contract or Grant No.

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, DC 20546-0001

13. Type of Report and Period Covered
Technical Memorandum

14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract
A major reorganization of the Aeronautics Directorate of the Langley Research Center occurred in early 1989. As a result of this reorganization, the scope of research in the Applied Aeronautics Division is now quite different than in the past. This report contains an overview of the current organization, mission, and facilities of this division. Also included are a summary of current research programs and sample highlights of recent research. This report is not intended to provide in-depth technical results but, rather, to provide a general view of the scope and capabilities of the division.

17. Key Words (Suggested by Author(s))
Subsonic wind tunnels
Transonic wind tunnels
Supersonic wind tunnels
Hypersonic wind tunnels

18. Distribution Statement
Unclassified—Unlimited

Subject Category 09

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	38	A03

NASA FORM 1626 OCT 86

NASA-Langley, 1991

For sale by the National Technical Information Service, Springfield, Virginia 22161-2171