NASA Technical Memorandum 85846

Simulation Applications at NASA Ames Research Center

Mamoru Inouye

March 1984

NASA
National Aeronautics and
Space Administration

NASA Technical Memorandum 85846

Simulation Applications at NASA Ames Research Center

Mamoru Inouye, Ames Research Center, Moffett Field, California

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

N84-19362#

SIMULATION APPLICATIONS AT NASA AMES RESEARCH CENTER

Mamoru Inouye*

1. Introduction

The National Aeronautics and Space Administration (NASA) is a civilian agency of the United States government responsible for planning, directing, and conducting aeronautical and space activities. Ames Research Center is one of ten major NASA field installations and was originally established in 1940 as a laboratory of the National Advisory Committee for Aeronautics (NACA). The Center is located 60-km southeast of San Francisco and occupies 168 hectares adjacent to San Francisco Bay as shown in Fig. 1. A staff of 1,800 Civil Service employees and over 1,000 contract workers and students are involved with research and technology programs in aeronautics, space science, life science, and spacecraft missions.

One of Ames Research Center's prime goals is to conduct aeronautical research that will be required to develop future aircraft and other flight vehicles. Specific research areas include aerodynamics, computational fluid dynamics, and flight simulation applicable to subsonic and transonic aircraft, including STOL and rotorcraft technology. The objective of this paper is to describe applications of simulation technology in aeronautics, including wind tunnels for investigating aerodynamics, computers for solving fluid dynamics problems, and flight simulators for studying aircraft handling qualities.

---

*Assistant Chief, Computational Fluid Dynamics Branch, NASA Ames Research Center, Moffett Field, California 94035, U.S.A.

2. Wind Tunnel Simulation

The testing of small-scale models in a wind tunnel dates back many years. In the early 1900's, the Wright brothers constructed a wind tunnel and tested various airfoil shapes before their first successful manned flight in 1903. Today, Ames Research Center has a collection of more than a dozen wind tunnels that operate at subsonic to hypersonic speeds. These facilities are used to determine the flow-field and aerodynamic characteristics for various aircraft, helicopter, and missile configurations.

The largest wind tunnel at Ames Research Center and in the world has two test sections, the older measuring 12- by 24-m and the newer, 24- by 36-m. The newer test section with its inlet may be seen attached to the original closed-circuit tunnel in the upper left-hand corner of Fig. 1. Both test sections are driven by the same set of six variable-pitch, 15-bladed fans, each powered by 18,000-hp electric motors. The maximum air speed is 555 km/hr in the 12- by 24-m test section and 185 km/hr in the 24- by 36-m test section. The large size permits testing of full size aircraft or very large models such as the one-third scale model of the Space Shuttle shown in the 12- by 24-m test section in Fig. 2. Although operable only at subsonic speeds, tests in this tunnel are required because supersonic aircraft must land and take off at subsonic speeds.

Wind-tunnel simulations have limitations because the model support and wind-tunnel walls interfere with the air flow around the model. In addition, the flight conditions may not be completely reproducible. Cost is also a factor. As aircraft become more complicated, the amount of wind-tunnel testing required has increased markedly as shown in Fig.3.

Combined with the rising cost of energy, labor, and materials, extensive wind-tunnel tests become prohibitive. As a result, computer simulations are becoming increasingly attractive for preliminary design studies with wind-tunnel tests providing final verification of a design prior to a flight prototype.

3. Computer Simulation of Aerodynamics

During the past decade, advances in computer technology have made it practical to solve problems that were previously considered impossible. As shown in Fig. 4, the cost of performing the same calculations has dropped markedly as each new generation of computers offers considerably more speed and capacity without a proportionate increase in cost. Concurrent advances in numerical methods have produced algorithms that make the most effective use of the available computer hardware. As a result, computational fluid dynamics has become an important tool in the study of basic fluid flow phenomena and the solution of practical aerodynamics problems. NASA's goal is to predict the flow field around aircraft configurations and to design airplanes better, faster, and at lower cost.

The steps in a computer simulation of a fluid dynamics problem are the following:

1) The governing partial differential equations are formulated. In the most general form, the unsteady Navier-Stokes equations are used. With turbulent flow, the turbulence terms are usually time-averaged to yield the Reynolds-averaged Navier-Stokes equations. When the viscous effects can be neglected or accounted for separately, the equations reduce to the Euler equations.

2) A grid including the body geometry and exterior boundary is created. The mesh must be orderly, and the points must be clustered where large gradients of flow properties occur.

3) A solution method, usually of finite-difference form, is selected. The method must be accurate, stable, and fast.

4) A computer program is written to obtain numerical solutions on a large computer. For large programs the specific computer architecture must be considered to take maximum advantage of the speed and storage capacity.

5) The results are processed to obtain graphical displays of the output for analysis. The accuracy of the computer simulations must be verified by comparisons with analytical solutions or experimental data.

Current computer simulation capabilities will be illustrated by the following three examples: large eddy simulation of turbulence, aileron buzz, and exhaust jet.

Turbulent flow is present in practically all aerodynamics problems. The calculation of the details of turbulence, however, is not possible even with the supercomputers expected to be available in the near future. Consequently, turbulence modeling based mainly on experimental data is required. For simple geometries, however, turbulence calculations are possible. Moin and Kim (Ref. 1) have performed computer simulations of incompressible turbulent flow between two parallel plates using the large eddy simulation method. In this approach, the subgrid scale eddies are modeled and the large eddies are solved using the unsteady, three-dimensional Navier-Stokes equations. Typical results are the contours of streamwise velocity fluctuations shown in Fig. 5 for a plane parallel

to the two walls. Over half a million grid points were used in these calculations, and the small rectangle in the lower right-hand corner depicts a grid cell. The computer simulations reveal minute details of the flow that are difficult or impossible to obtain experimentally and they provide information for turbulence models required in solving the Reynolds-averaged Navier-Stokes equations. The large eddy simulation method has also been used to show the effects of blowing or suction, compliant walls, and rotation on the viscous drag at the wall.

At transonic speeds the movement of the shock wave over a wing can cause an oscillation of the aileron known as aileron buzz. This phenomenon was observed initially toward the end of World War II, and experiments were conducted at that time in the Ames 16-foot Wind Tunnel as shown in Fig. 6. Steger and Bailey (Ref. 2) have recently reproduced this phenomenon by solving the unsteady, two-dimensional, Reynolds-averaged Navier-Stokes equations. The computer simulations show good agreement with the old experimental data for the buzz boundary as a function of the free-stream Mach number and angle of attack. A typical solution for aileron buzz is shown in Fig. 7, where the aileron deflection angle is plotted against time for a free-stream Mach number of 0.82. Both the amplitude and frequency of the oscillations agree with the experimental data.

Three-dimensional solutions of the Reynolds-averaged Navier-Stokes equations are possible using the supercomputers available today. Deiwert and Rothmund (Ref. 3) have used a CDC CYBER 205 computer to calculate the flow over a conical afterbody with an exhaust jet. The resolution of flow details near the surface and jet required over 200,000 grid points and a data base of 5-million words. The surface streamlines on the

afterbody are shown in Fig. 8, together with the density contours in the symmetry plane for a free-stream Mach number of 2, angle of attack of 6 degrees, and jet pressure three times the free-stream value. The expansion of the jet has caused the flow to separate on the conical afterbody.

To develop the capability to solve more complex problems, Ames Research Center is acquiring advanced computer hardware and supporting research on new numerical methods and computer codes. A long-range goal is to develop a Numerical Aerodynamic Simulator with the capability to calculate the viscous flow over an aircraft configuration in a matter of minutes. The computer requirements include a sustained processing speed of one-billion floating-point operations per second, main memory with 32-million words, and secondary memory with 256-million words. The planned operational date is 1987.

4. Flight Simulation

Aircraft operation involves the interaction between an airplane and its pilot. Flight simulators provide this experience without leaving the ground. Ames Research Center has more than eight flight simulators in operation, the largest of which are the Flight Simulator for Advanced Aircraft and the Vertical Motion Simulator. Different aircraft can be "flown" safely by a pilot in a flight simulator to assess their handling qualities, particularly during takeoff and landing. Existing designs can be tested to establish emergency procedures in the event of engine failure or other malfunction, and to recreate accidents and determine their cause. New designs can be studied to evaluate their performance and correct any deficiencies.

The most advanced flight simulators have six degrees of motion and they are provided with all the aircraft instruments and controls, the visual display of the terrain on the windshield, and the audio output of engine or the rotor blade noise and landing gear rumble. A high-speed digital computer provides the flight dynamics from aerodynamic data obtained from wind-tunnel tests or computer simulations and controls the various subsystems. The scene displayed on the windshield is produced by a closed-circuit television system with the camera mounted on a carriage that traverses a miniature landscape.

The Flight Simulator for Advanced Aircraft, shown in Fig. 9, was completed in 1969 for research on large aircraft. Lateral motion is emphasized — the cockpit can be moved 24-meters laterally and can be reconfigured for different aircraft with all the flight controls and instruments. The Concorde and the Space Shuttle operations have been some of the aircraft simulations which have been conducted.

The Vertical Motion Simulator was completed in 1979 for research on STOL and VTOL aircraft, including helicopters. The cockpit can be moved 18 meters vertically and 12 meters laterally. The large vertical motion permits simulation of maneuvers such as helicopter autorotations.

References

1) Moin, P., and Kim, J.: Numerical Investigation of Turbulent Channel Flow, Journal of Fluid Mechanics, Vol. 18, 1982, pp. 341-377.

20 Steger, J. L., and Bailey, H. E.: Calculation of Transonic Aileron Buzz, AIAA Journal, Vol. 18, No. 3, March 1980, pp. 249-255.

3) Deiwert, G. S. and Rothmund, H.: Three-Dimensional Flow over a Conical Afterbody Containing a Centered Propulsive Jet: A Numerical Simulation: AIAA 16th Fluid and Plasma Dynamics Conference Paper No. 83-1709, Danvers, Massachusetts, July 12-14, 1983.

FIGURE TITLES

Fig. 1 Aerial view of Ames Research Center.

Fig. 2 One-third scale model of Space Shuttle in 12- by 24-m wind tunnel.

Fig. 3 Wind-tunnel testing time for commercial aircraft.

Fig. 4 Computing costs for new computers relative to IBM 360/67.

Fig. 5 Contours of streamwise velocity fluctuations in plane parallel to walls for incompressible turbulent flow between two parallel plates.

Fig. 6 Aileron buzz test in old Ames 16-foot wind tunnel.

Fig. 7 Variation of aileron deflection angle with time for aileron buzz conditions.

Fig. 8 Surface streamlines on conical afterbody and density contours in symmetry plane of jet exhaust.

Fig. 9 Flight Simulator for Advanced Aircraft.

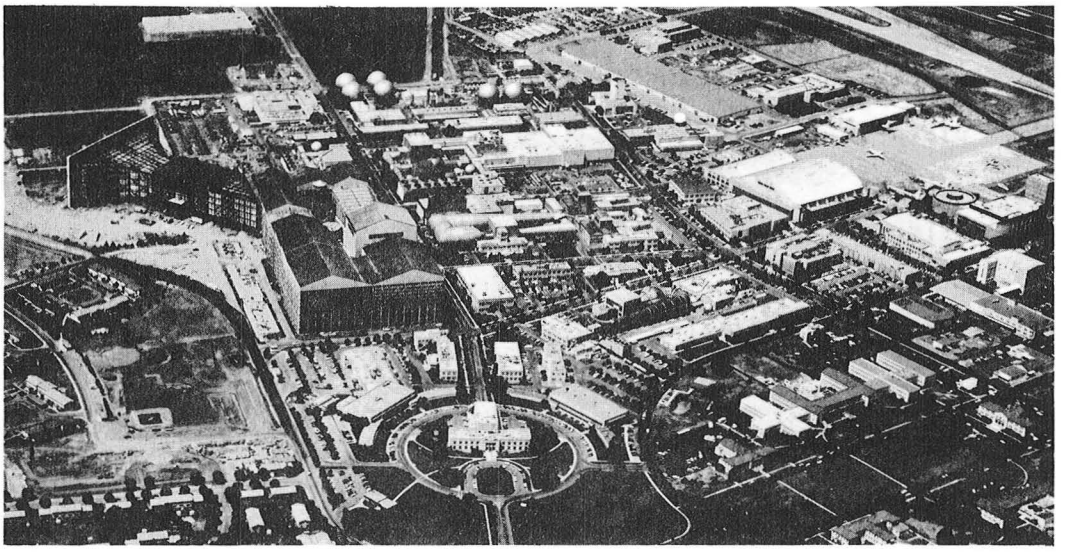

Fig. 1

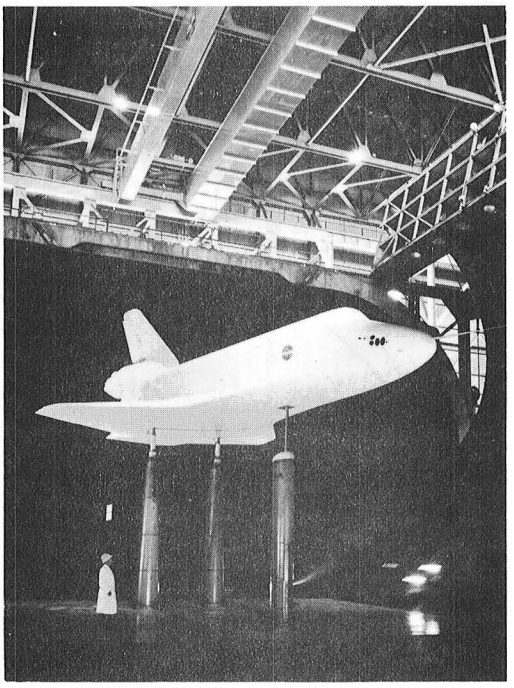

Fig. 2

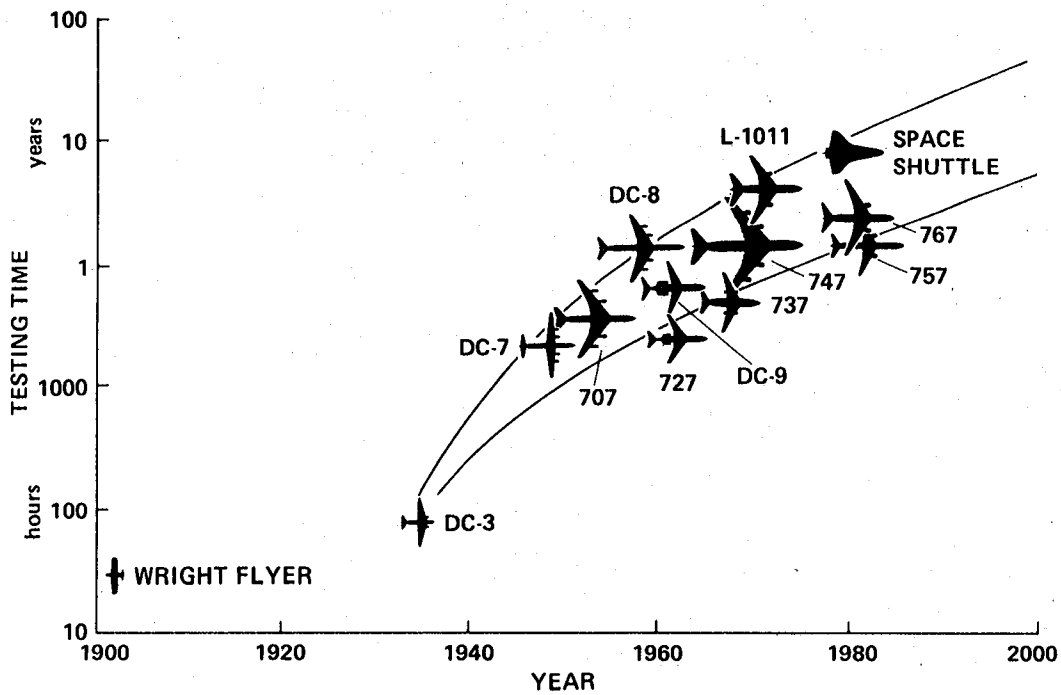
100
10
1
1000
100
10
years
hours
TESTING TIME
1900
1920
1940
1960
1980
2000
YEAR
WRIGHT FLYER
DC-3
DC-7
DC-8
L-1011
SPACE
SHUTTLE
707
727
DC-9
737
747
757
767

Fig. 3

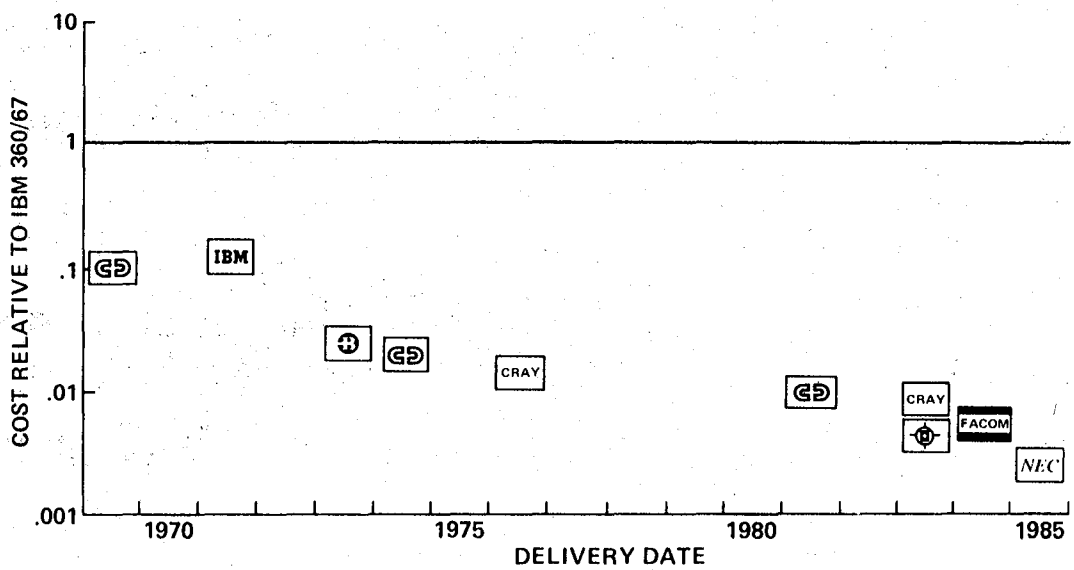
10
1
.1
.01
.001
COST RELATIVE TO IBM 360/67
IBM
CRAY
CRAY
FACOM
NEC
1970
1975
1980
1985
DELIVERY DATE

Fig. 4

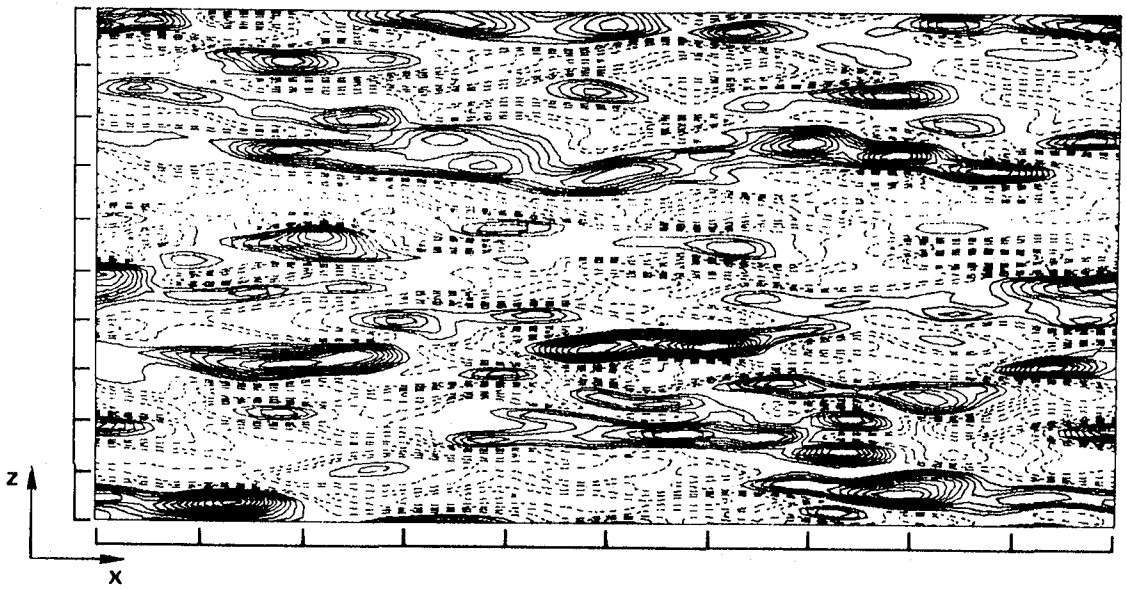

Fig. 5

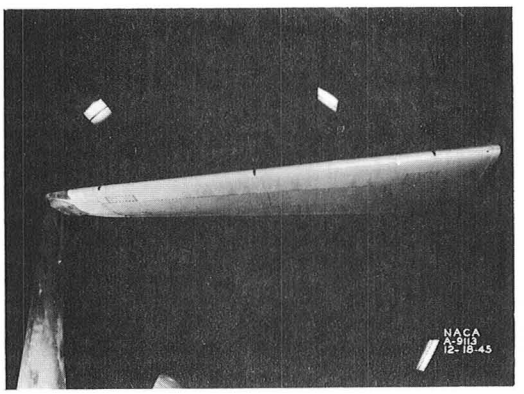

Fig. 6

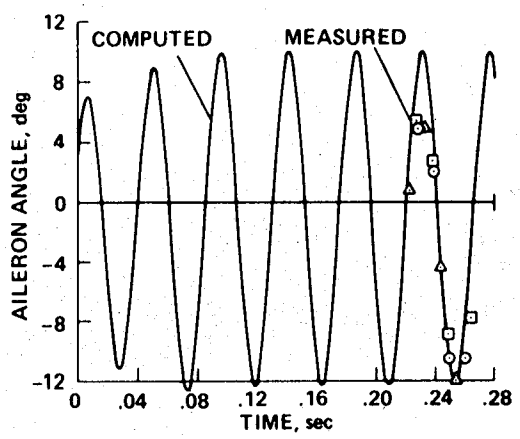
COMPUTED
MEASURED
AILERON ANGLE, deg
12
8
4
0
-4
-8
-12
0
.04
.08
.12
.16
.20
.24
.28
TIME, sec

Fig. 7

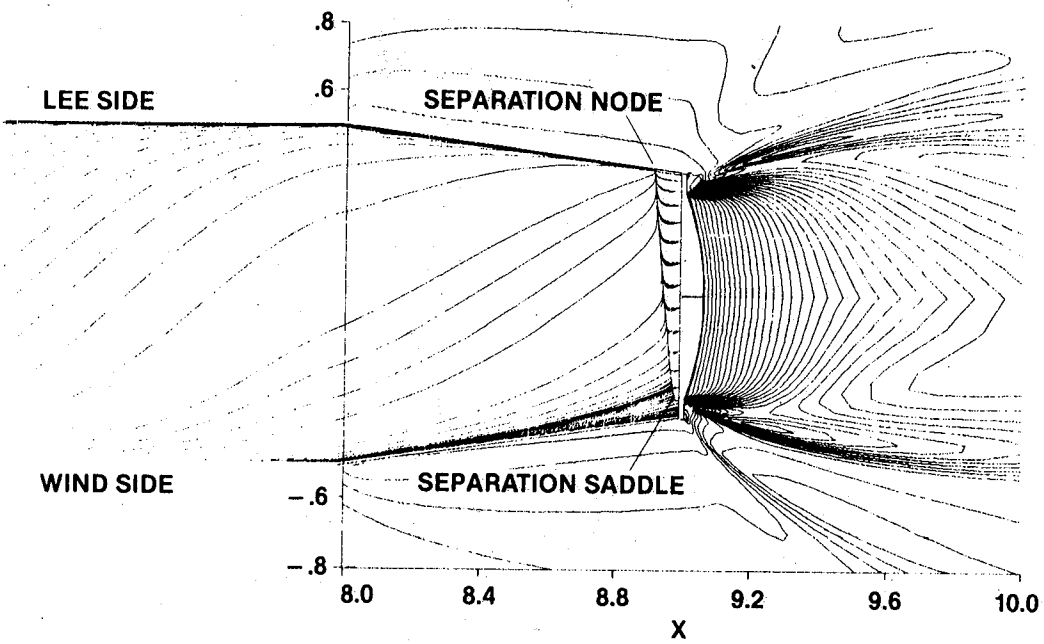
.8
.6
LEE SIDE
SEPARATION NODE
WIND SIDE
SEPARATION SADDLE
−.6
−.8
8.0
8.4
8.8
9.2
9.6
10.0
X

Fig. 8

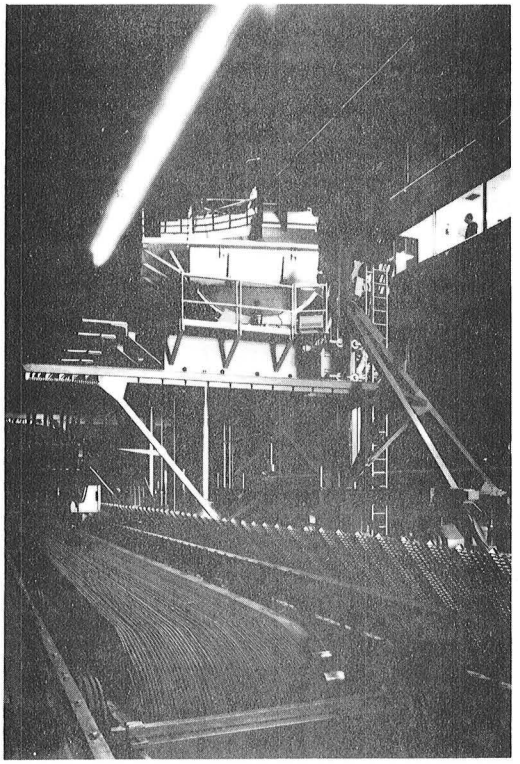

Fig. 9

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM 85846		

4. Title and Subtitle	5. Report Date
SIMULATION APPLICATIONS AT NASA AMES RESEARCH CENTER	March 1984
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
Mamoru Inouye	A-9503
	10. Work Unit No.
9. Performing Organization Name and Address	T-6465
Ames Research Center Moffett Field, CA 94035	11. Contract or Grant No.
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Technical Memorandum
National Aeronautics and Space Administration Washington, DC 20546	14. Sponsoring Agency Code 505-31-01

15. Supplementary Notes

Point of Contact: M. Inouye, MS 202A-1, Ames Research Center, Moffett Field, CA 94035, (415)965-5126 or FTS 448-5126

16. Abstract

The paper describes aeronautical applications of simulation technology at Ames Research Center, which is one of ten major field installations of the National Aeronautics and Space Administration, the civilian agency of the United States government responsible for planning, directing, and conducting aeronautical and space activities. The largest wind tunnel in the world is used to determine the flow field and aerodynamic characteristics of various aircraft, helicopter, and missile configurations. Large computers are used to obtain similar results through numerical solutions of the governing equations. Capabilities are illustrated by computer simulations of turbulence, aileron buzz, and an exhaust jet. Flight simulators are used to assess the handling qualities of advanced aircraft, particularly during takeoff and landing.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Wind tunnels Computational fluid dynamics Flight simulation	Unlimited Subject Category - 01

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	22	A02

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