NASA Technical Memorandum 102221

Computer Simulation of Aircraft Aerodynamics

Mamoru Inouye

October 1989

(NASA-TM-102221) COMPUTER SIMULATION OF AIRCRAFT AERODYNAMICS (NASA) ~~11 p~~ CSCL 01A
16 p
N90-11699
547197
Unclas
G3/02 0239287

NASA
National Aeronautics and
Space Administration

NASA Technical Memorandum 102221

Computer Simulation of Aircraft Aerodynamics

Mamoru Inouye, Ames Research Center, Moffett Field, California

October 1989

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

COMPUTER SIMULATION OF AIRCRAFT AERODYNAMICS

Mamoru Inouye

Ames Research Center

SUMMARY

The role of Ames Research Center in conducting basic aerodynamics research through computer simulations is described. The computer facilities, including supercomputers and peripheral equipment, represent the state of the art. The methodology of computational fluid dynamics is explained briefly. Fundamental studies of turbulence and transition are being pursued to understand these phenomena and to develop models that can be used in the solution of the Reynolds-averaged Navier-Stokes equations. Four applications of computer simulations for aerodynamics problems are described: subsonic flow around a fuselage at high angle of attack, subsonic flow through a turbine stator-rotor stage, transonic flow around a flexible swept wing, and transonic flow around a wing-body configuration that includes an inlet and a tail.

INTRODUCTION

Since the introduction of jet aircraft to regular commercial service 30 years ago, the world's fleet has grown to over 7,000 airplanes. Although the United States' market share has eroded over the years, most of these aircraft were manufactured in the United States. New aircraft are needed now to replace aging airframes, to comply with noise restrictions, and to improve operating efficiencies. Manufacturers plan to build over 8,000 new aircraft during the next 15 years. One of NASA's objectives is to provide the aircraft industry with the basic technology required to design new aircraft that will maintain the country's preeminence in the world marketplace.

COMPUTER FACILITIES

Ames Research Center has two computer facilities available for the simulation of aircraft aerodynamics. The Central Computing Facility serves the general research needs of the Center with a Cray Y-MP/832 supercomputer. The Numerical Aerodynamic Simulation (NAS) facility provides a computational capability to the whole country, including researchers in industry, universities and government agencies. The high-speed processors in the NAS facility are currently a Cray-2 and a Cray Y-MP/832. Thirty-five scientific workstations are distributed among users at the Center; other workstations are distributed around the country. The complete system includes mass storage, support processing, and long-haul communications subsystems connected by a high-speed data network. The long-term goal is to replace the high-speed processors periodically with the fastest available supercomputers.

COMPUTATIONAL FLUID DYNAMICS

The Navier-Stokes equations which describe the behavior of viscous flows have been known for over 150 years. Computational fluid dynamics is the relatively new field in which these partial differential equations are solved using high-speed computers. The flow field, including any solid bodies or boundaries, is encased within a grid of mesh points. Difference methods are then used to solve the flow properties for the specified boundary and initial conditions.

TURBULENCE

Nearly all practical aerodynamics problems involve turbulent flow near surfaces and in wake regions. The combination of the fine grid required to resolve the turbulence and the processing speed required to perform the calculations rules out exact solutions of the Navier-Stokes equations for complex configurations. In order to obtain solutions, the turbulence terms are time-averaged and approximated with empirical models.

The most commonly used turbulence model is the Baldwin-Lomax algebraic eddy viscosity model (Ref. 1), which works well for flows with little or no separation. Improved models are necessary for flows with large separated regions and steep pressure gradients. Computer simulation of turbulence is being used to study the detailed structure for the purpose of developing new turbulence models. Spalart (Ref. 2) recently performed direct simulations of a turbulent boundary layer on a flat plate for Reynolds numbers, Re_θ, based on momentum thickness, up to 1410. The detailed quality of the calculations may be observed in the vorticity contours shown in Fig. 1.

APPLICATIONS

Current capabilities for computer simulation of aerodynamics problems are demonstrated by the following applications made by members of the Ames Applied Computational Fluids Branch. The equations solved are usually the thin-layer, Reynolds-averaged Navier-Stokes equations, and the Baldwin-Lomax model is used for turbulent flow.

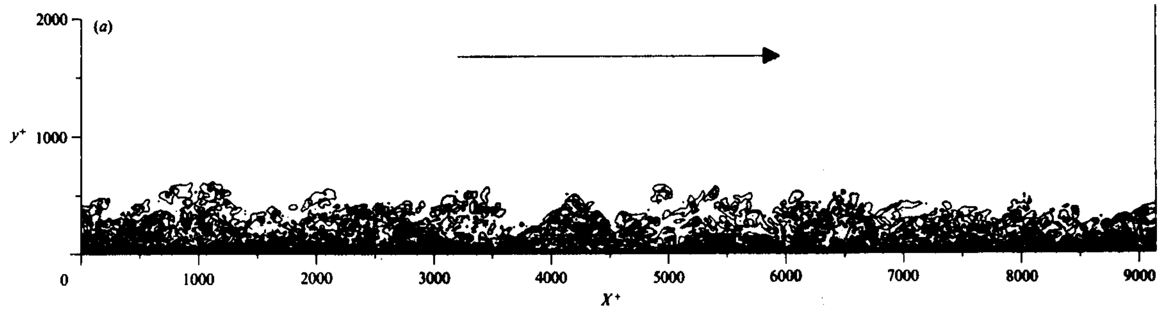

Fig. 1 Vorticity contours on flat plate, $Re_\theta = 1410$ (Ref. 2)

Aircraft forebody

Subsonic flow around a slender fuselage at a high angle of attack was calculated by Schiff, Cummings, Sorenson, and Rizk (Ref. 3) to study the vortical flow on the leeward side, which affects the maneuverability of the aircraft. A closeup of the two-block grid is shown in Fig. 2; the interface is located at the beginning of the wing leading-edge extension (LEX). The aft block has a larger number of circumferential points to define the LEX. Nearly a quarter of a million grid points were used.

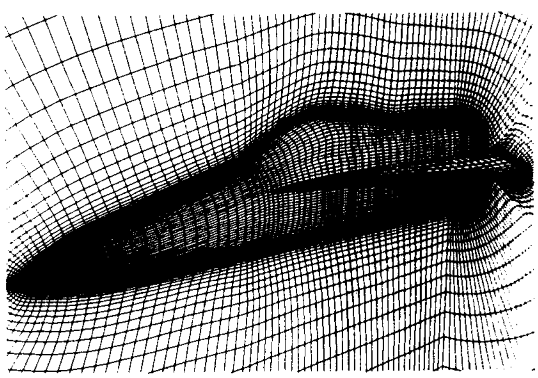

Fig. 2 Two-block grid around aircraft forebody (Ref. 3)

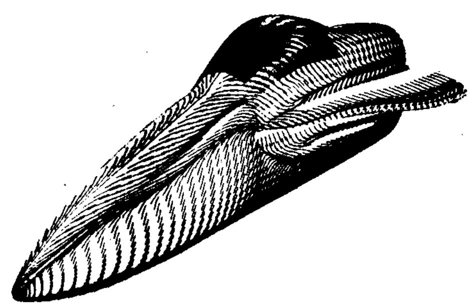

Fig. 3 Surface flow pattern on aircraft forebody (Ref. 3)

The surface flow pattern is shown in Fig. 3 for $M_\infty = 0.2$, $\alpha = 30^\circ$, and Reynolds number based on mean aerodynamic chord, $Re_c = 11.5 \times 10^6$. The primary and secondary separation lines on the forebody and the secondary crossflow separation line on the LEX are in good agreement with flight-test data.

Turbine stage

Subsonic flow through a turbine stator-rotor stage was calculated by Rai (Ref. 4) to understand the flow processes in order to design more efficient turbomachinery. A patched-grid system was used with the grid about the rotor moving relative to the grid about the stator. This is shown in Fig. 4 for a midspan location. The complete geometry included the hub and outer casing with some tip clearance. Equal number of rotor and stator blades were used to reduce the calculations to one rotor and one stator blade. The total number of grid points was 200,000.

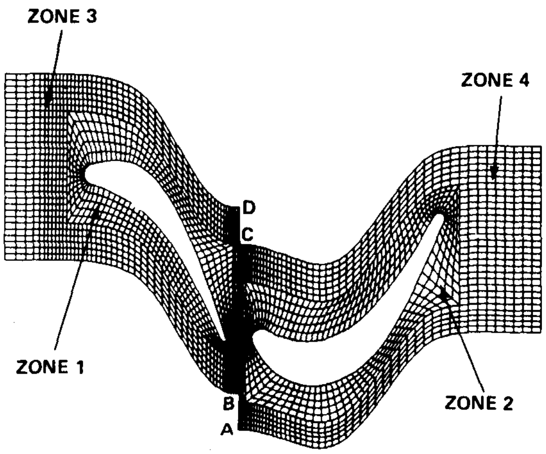

Fig. 4 Patched grid around stator and rotor at midspan (Ref. 4)

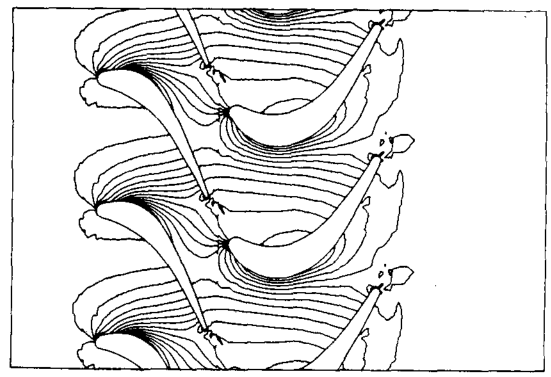

Fig. 5 Instantaneous pressure contours at midspan (Ref. 4)

Calculations were performed for an inlet Mach number, $M_i = 0.07$, and $Re = 10^5$ per inch. Instantaneous pressure contours at midspan are shown in Fig. 5.

Flexible wing

Transonic flow around a flexible swept wing was calculate by Guruswamy (Refs. 5,6) to demonstrate the coupling between the vortical flow and the wing deformation. The unsteady Navier-Stokes equations were solved simultaneously with the structural equations of motion. A new grid was generated at each time step to conform with the deformed wing. The initial and deformed grids at the 50% semispan airfoil section are shown in Fig. 6.

Calculations were performed for ramp motion of a rectangular wing, using a NACA 0015 airfoil section with an aspect ratio of 4 and a sweep angle of 30^o. The effects of flexibility on unsteady lift coefficients at four span locations are shown in Fig. 7 for $M_\infty = 0.5$, Reynolds number based on chord, $Re_c = 2 \times 10^6$, and pitching rate $A = 0.1$. The higher lift for the flexible wing is attributed to the increase in angle of attack caused by the flexibility.

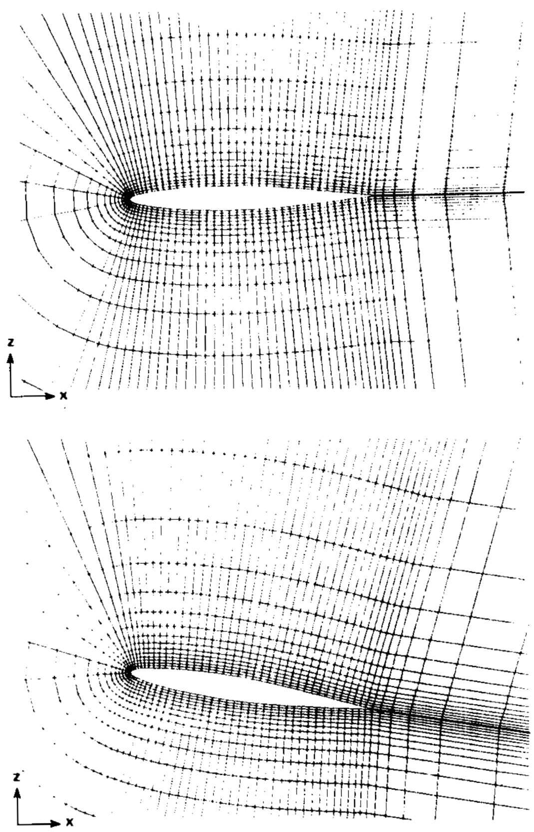

Fig. 6 Initial and deformed grid at 50% semispan of flexible wing (Ref. 5)

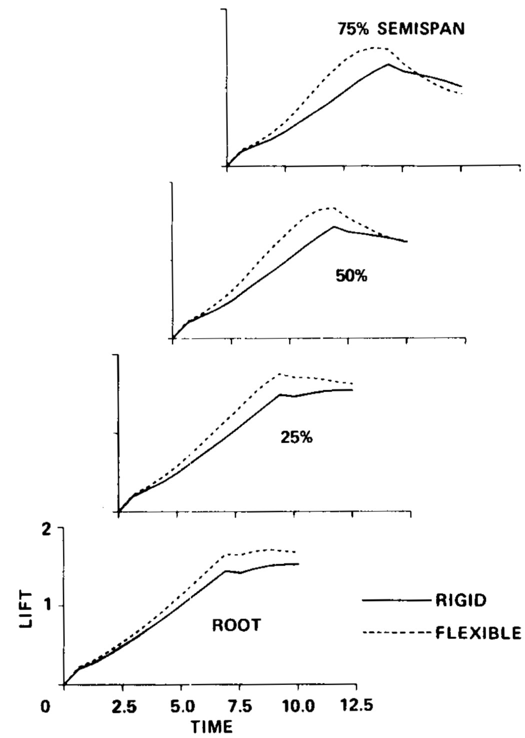

Fig. 7 Unsteady lift coefficient for rigid and flexible wing (Ref. 6)

Complete aircraft

Transonic flow around a wing-body configuration, including an inlet and a tail, was calculated by Flores and Chaderjian (Ref. 7). The purpose was to demonstrate the ability to calculate transonic viscous flow over a complete aircraft. The flow field was divided into 27 zones, determined by geometrical or flow conditions, for a total of half a million grid points. The surface grid covering the forward portion of the fuselage, including the inlet and wing, is shown in Fig. 8.

Computations for $M_\infty = 0.9$, $\alpha = 6^o$, and Reynolds number based on wing root chord, $Re_c = 4.5 \times 10^6$, required 25 hours for 5,000 iterations, using one processor of a Cray X-MP/48. Pressure coefficients on the vertical tail at three-quarters height are shown in Fig. 9. Comparison with experimental results (Ref. 8) shows good agreement. Calculations for the wing and fuselage without the tail surfaces (Ref. 9) also show good agreement (Fig. 10).

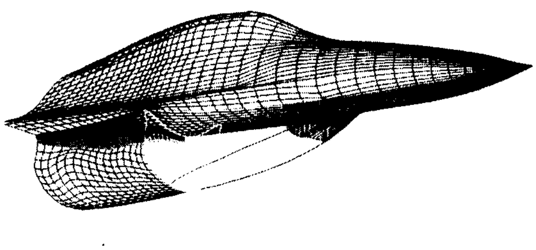

Fig. 8 Grid for fuselage forebody (Ref. 7)

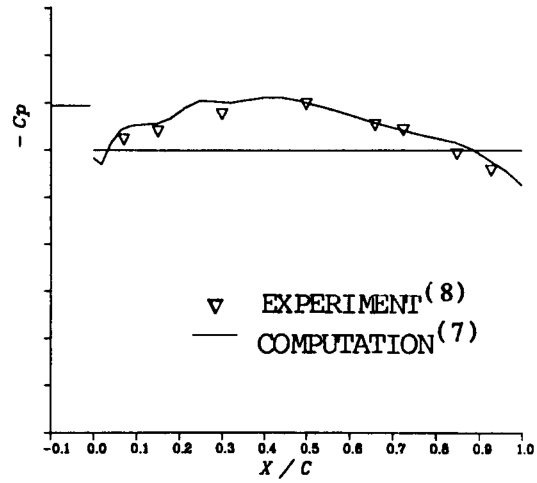

Fig. 9 Pressure coefficient on vertical tail (Ref. 7)

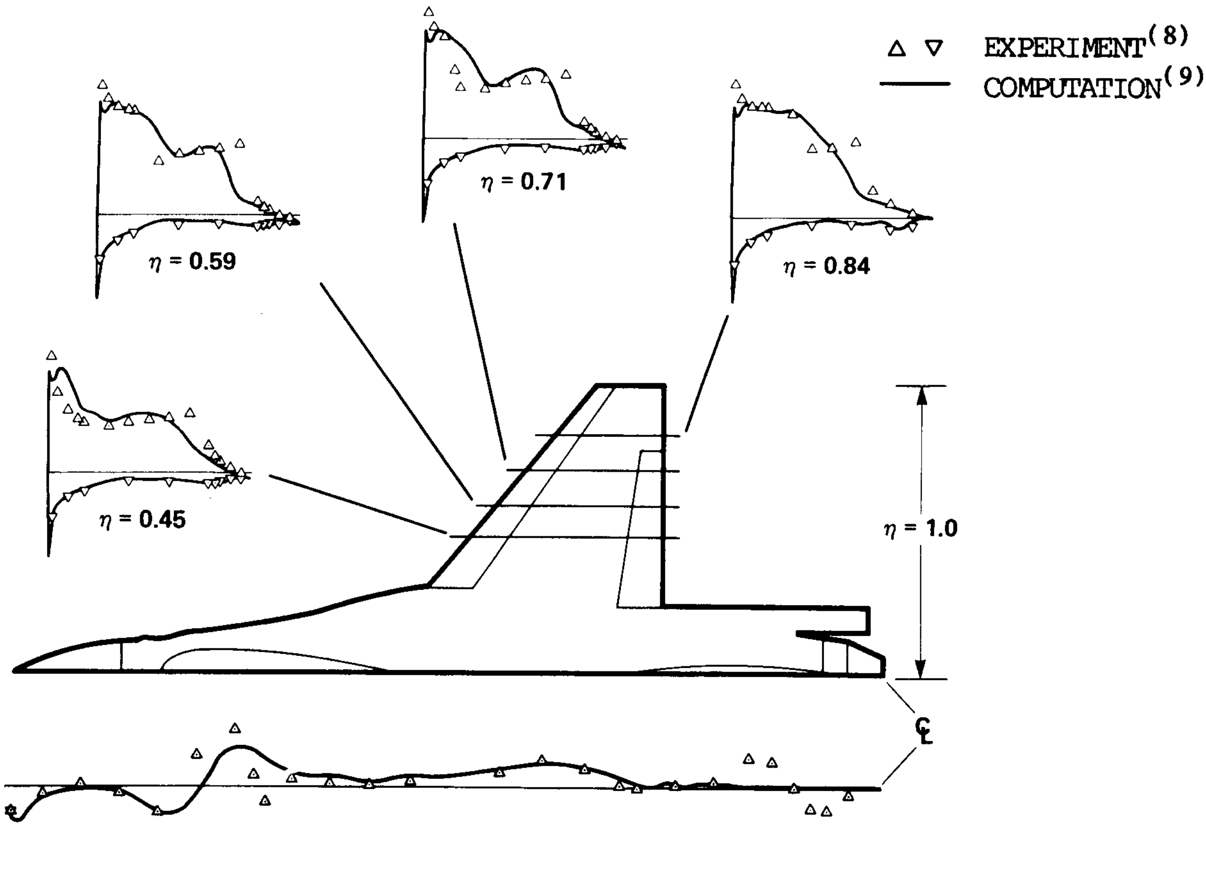

Fig. 10 Pressure coefficient on wing and upper fuselage center-line (Ref. 9)

CONCLUDING REMARKS

Many challenges still remain in computational fluid dynamics. Steady-state solutions are sought using time-dependent methods that require thousands of steps for convergence. The efficiency of numerical methods must be improved to make the best use of available computers. New turbulence models are needed; the most commonly used model--the Baldwin-Lomax model--is over ten years old and is inadequate for separated flows. Single-processor supercomputers are approaching their inherent performance limits, and multiprocessors offer a means for increasing performance. Algorithms and codes must be constructed to take advantage of the supercomputer architecture. Finally, aerodynamics must be combined with other aspects of aircraft design. Structure and materials, control and guidance, and propulsion systems are equally important, and coupling between disciplines will be required in future designs.

REFERENCES

1. Baldwin, B.S. and Lomax, H., Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper No.78-257, 1978.

2. Spalart, P.R., Direct Simulation of a Turbulent Boundary Layer up to R_θ = 1410, J. Fluid Mechanics, Vol.187, 1988, pp.61-98.

3. Schiff, L.B., Cummings, R.M., Sorenson, R.L., and Rizk, Y.M., Numerical Simulation of High-Incidence Flow over the F-18 Fuselage Forebody, AIAA Paper No.89-0339, 1989.

4. Rai, M.M., Unsteady Three-Dimensional Navier-Stokes Simulations of Turbine Rotor-Stator Interaction, AIAA Paper No.87-2058, 1987.

5. Guruswamy, G., Time-Accurate Unsteady Aerodynamic and Aeroelastic Calculations of Wings Using Euler Equations, AIAA Paper No. 88-2281, 1988.

6. Guruswamy, G., Vortical Flow Computations on Swept Flexible Wings Using Navier-Stokes Equations, AIAA Paper No.89-1183, 1989.

7. Flores, J., and Chaderjian, N.M. The Numerical Simulation of Transonic Separated Flow about the Complete F-16A, AIAA Paper No.88-2506, 1988.

8. Reue, G.L., Doberenz, M.E., and Wilkins, D.D., Component Aerodynamic Load from 1/9-Scale F-16A Loads Model, General Dynamics Rep. 16PR316, Fort Worth, TX, May 1976.

9. Reznick, S.G., and Flores, J., Strake-Generated Vortex Interactions for a Fighter-Like Configuration, J. Aircraft, Vol.26, No.4, Apr. 1989, pp.289-294.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-102221		

4. Title and Subtitle	5. Report Date
Computer Simulation of Aircraft Aerodynamics	October 1989
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
Mamoru Inouye	A-89227
	10. Work Unit No.
	505-60-01
9. Performing Organization Name and Address	11. Contract or Grant No.
Ames Research Center Moffett Field, CA 94035	
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Technical Memorandum
National Aeronautics and Space Administration Washington, DC 20546-0001	14. Sponsoring Agency Code

15. Supplementary Notes

Point of Contact: Mamoru Inouye, Ames Research Center, MS 202A-1, Moffett Field, CA 94035
(415) 694-5126 or FTS 464-5126

Presented at JSASS 27th Aircraft Symposium, Fukuoka, Japan, Oct. 18-20, 1989.

16. Abstract

The role of Ames Research Center in conducting basic aerodynamics research through computer simulations is described in this paper presented at the International Sessions of the 27th Aircraft Symposium, sponsored by the Japan Society for Aeronautical and Space Sciences and held at Fukuoka, Japan, during October 18-20, 1989. The computer facilities, including supercomputers and peripheral equipment that represent the state of the art, are described. The methodology of computational fluid dynamics is explained briefly. Fundamental studies of turbulence and transition are being pursued to understand these phenomena and to develop models that can be used in the solution of the Reynolds-averaged Navier-Stokes equations. Four applications of computer simulations for aerodynamics problems are described: subsonic flow around a fuselage at high angle of attack, subsonic flow through a turbine stator-rotor stage, transonic flow around a flexible swept wing, and transonic flow around a wing-body configuration that includes an inlet and a tail.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Aerodynamics Computational fluid dynamics	Unclassified-Unlimited Subject Category - 02

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	10	A02

NASA FORM 1626 OCT 86

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