ISSN 0389-4010
UDC 533. 6. 04
533. 6. 011. 3

TECHNICAL REPORT OF NATIONAL AEROSPACE LABORATORY

TR-1018T

Some Topics in Computational Transonic Aerodynamics A Revised Paper Based on a Presentation given at the IUTAM Symposium TRANSSONICUM III

Naoki HIROSE and Susumu TAKANASHI

April 1989

NATIONAL AEROSPACE LABORATORY

CHŌFU, TOKYO, JAPAN

Some Topics in Computational Transonic Aerodynamics*

—A Revised Paper Based on a Presentation given at the IUTAM Symposium Transsocicum III—

Naoki HIROSE** and Susumu TAKANASHI**

ABSTRACT

Transonic aerodynamics research has been extensively conducted at NAL, JAPAN. Progress in CFD application is remarkable. N-S codes for airfoil, wing and full-configuration have been developed. The codes revealed various viscous effects which otherwise were not predicted by past inviscid codes and were validated with wind tunnel test results. These N-S codes proved to be excellent as practical design tools for designing such aircraft as the Boeing-Japan 7J7. A versatile wing design code was combined with various flow solvers and used for practical purpose. The paper covers some topics of recent computational aerodynamics in transonic research.

概　　要

国際理論応用力学連合 (IUTAM) 主催『シンポジウム・トランソニカムIII』（第３回遷音速空気力学シンポジウム）が1988年５月24日～27日，西ドイツのゲッチンゲンにあるDFVLR-AVA（西独航空宇宙研究機構航空力学研究所）において開催された。本報告は，その時の発表に基づき，内容に必要な見直し，修正を施した物である。我国における遷音速空気力学研究の歴史的発展と前回シンポジウムまでの経過を示し，ついで，今回までの13年間の研究進展，そして最近の著者達の研究を含む計算空気力学的方法を使った遷音速空気力学研究の幾つかについて，遷音速航空機に対する応用空気力学を中心として，概要をとりまとめた。

1. INTRODUCTION

Japan has a long tradition of aerodynamics and fluid dynamics researches. In the area of theoretical transonic aerodynamics, classical works of TOMOCHIKA-TAMADA$^{1)}$, and I. IMAI$^{2)}$ are well known. When computer came in, computer simulation of fluid flow was tried by several people. At its early days, computer power was so limited and the number of researchers accessible to large computers was also limited. NATIONAL AEROSPACE LABORATORY (NAL) is one of such institutions besides few number of national universities.

NAL has been engaged in developing computational transonic aerodynamics for the design of aircraft during the past 20 years. The computer codes developed are now being applied to the aerodynamic design of Japanese civil transport YXX (Boeing-Japan 7J7), etc. As the advanced flow simulation codes became available, the code validation evaluations with the existing codes and the wind tunnel experiments are extensively made. The computational aerodynamics is now an established new discipline besides the existing theoretical and experimental aerodynamics for the research of transonic flow. In the present paper, only the practical transonic CFD develop-

---

* Received 21, July 1988

** The Aircraft Aerodynamics Division

ment for aircraft pursued around the authors is briefly described although various academic research works on transonic aerodynamics and CFD fields other than transonic flow problems have been made both at NAL and other various academic institutions because the purpose of the Symposium limits the range of coverage.

2. HISTORICAL ASPECTS

2.1 1960's

When the experimental works of PEARCEY³⁾ and WHITCOMB⁴⁾ revealed the practical possibility of transonic airfoil design in '60s, both of the experimental and theoretical researches for transonic airfoil and wing were on the way at NAL. The computer was used to numerically solve the transonic flow equation by hodograph method by the senior author (TAKANASHI⁵⁾). The details of those early day works can be found in the ICAS-74 paper by SHIGEMI⁶⁾. Computational fluid dynamics (CFD) using finite difference methods was investigated mainly for the supersonic blunt body and jet flow problems in '60s and early '70s at NAL⁷⁾⁸⁾. In those days, transonic flow application of CFD was not practical although we noticed MAGNUS-YOSHIHARA's early pioneering work⁹⁾ as the very important accomplishment.

2.2 1970's

MURMAN's success¹⁰⁾ of transonic small disturbance equation (TSD) type-dependent relaxation method and practical application of the method to C-141 wing analysis by BALLHAUS et al.¹¹⁾ brought us practical interest to develop transonic aerodynamic analysis and design tools utilizing CFD in '70s. Transonic airfoil and wing analysis codes based on TSD and full potential equations similar to GARABEDIAN-KORN's¹²⁾ and BAILEY-BALLHAUS's¹³⁾ were developed by KAMIYA, et al.¹⁴⁾ The latter required 10 hours of CPU time on then-available FACOM-230-75 (nearly equivalent to CDC-6600) to obtain one data-point of NAL-designed transonic swept wing with favorable agreement with wind tunnel experiment. Euler analysis code for airfoil was also developed and applied to both steady and unsteady flow by ISHIGURO¹⁵⁾. ISOGAI¹⁶⁾ developed transonic flutter analysis code based on TSD equation. TAKANASHI¹⁷⁾ developed an efficient wing analysis method using 3-D transonic integral equation. An efficient and effective airfoil design method was developed by ISHIGURO¹⁸⁾. When the surface pressure distribution was specified, the code iteratively solves airfoil geometry including viscous effects utilizing inverse and analysis steps for full potential equation. KAMIYA¹⁹⁾ developed his transonic airfoil design methodology and utilized the fore-mentioned CFD tools to design supercritical airfoils and clarified the applicability of the tools. In '79. High-Reynolds Number 2-D Transonic Wind Tunnel was introduced to the transonic aerodynamics research. The wind tunnel provides high-Reynolds number flow up to 4×10^7 and presents high-quality experimental data of airfoils to researchers and designers at the industries. The details are described in ICAS-80 paper by KAMIYA & HIROSE²⁰⁾ and ICAS-82 paper by TAKASHIMA.²¹⁾

2.3 1980's

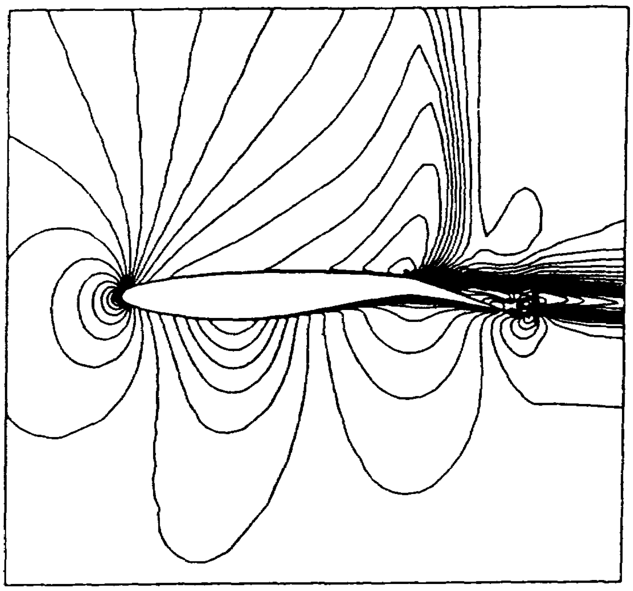
7th ICNMFD, at Stanford/NASA Ames²²⁾ revealed that CFD outgrew the engineering status just like FEM used to do. The potential codes are used easily and revolutionary advancement will not be made even in Navier-Stokes (N-S) solvers. Only high-speed computers and massive complicated grid-generation were needed although turbulence modeling remains unsolved. At NAL, the trends of research followed similar to the one at other countries. N-S codes for airfoil, wing, full-configuration were developed and applied to the practical computations. Improvements to the Navier-Stokes and Euler schemes were made. New design method were developed and applied to various flow regimes. To enhance the code development and applications, supercomputers were installed at NAL. The new computer facility is named 'Numerical Simulator (NS)' and it consists of FACOM VP-400, VP-200 and front-end processor M-780 with various graphic terminals. Some of the early achievements are described in KAMIYA's paper at ISCFD, Tokyo$^{23)}$.

3. NAVIER-STOKES ANALYSIS OF AIRFOIL, WING, AND COMPLETE AIRCRAFT

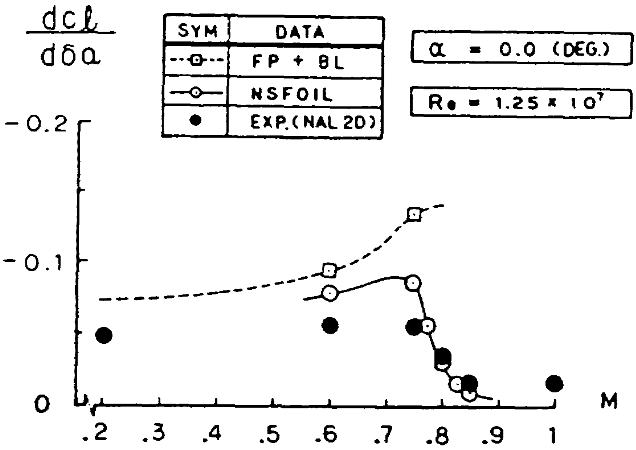
3.1 Airfoil Analysis

Time-averaged Navier-Stokes analysis for high Reynolds number transonic flow has been one of the major research topics. Navier-Stokes code was rather pursued than Euler code because Euler code is not worth-while for transport type aircraft aerodynamics in which shock wave is not so strong compared with the military aircraft. Also viscous interaction will be inevitable when shock wave becomes strong. As the selection of scheme type is different for Euler and viscous flow, N-S approach was selected.

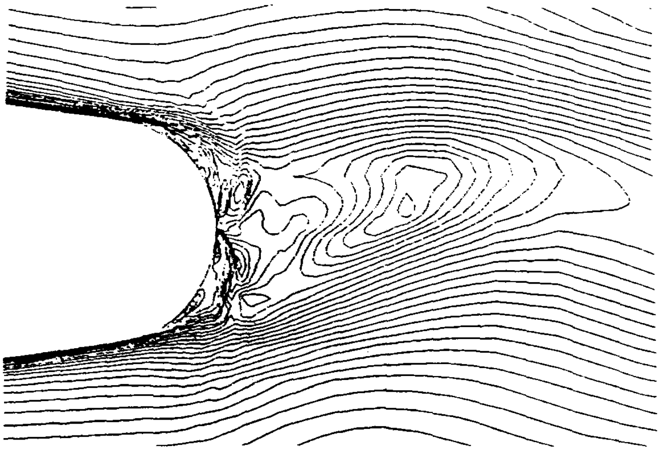
Two-dimensional airfoil analysis code NSFOIL based on Beam-Warming-Steger Implicit Approximate Factorization (IAF) algorithm$^{24)}$ developed by KAWAI and HIROSE$^{25)}$ has been applied to various supercritical airfoil analyses for practical purposes. High-Reynolds 2-D Transonic Wind Tunnel data was used for the code validation. One data point computation requires less than 8 min. and the code now is served as preliminary analysis of the transport wing designs before wind tunnel testing by the industries. Figure 1 shows some result of such application. Flow field for airfoil with aileron deflection is shown in (a), aileron effect dC$_1$/d α vs. Mach number was compared with the experiment in (b)$^{26)}$. Blunt trailing edge flow of supercritical airfoils was analyzed using fine mesh distribution in this particular region and Karman-vortex shedding was discovered in (c)$^{27)}$. Recent 2-D analysis covers Laminar Flow Control (LFC) and Natural Laminar Flow (NLF) applications$^{28)}$, and the



(a) Flowfield for aileron deflection.



(b) Aileron effect $dC_1/d\,\delta\,\alpha$ vs. M_∞



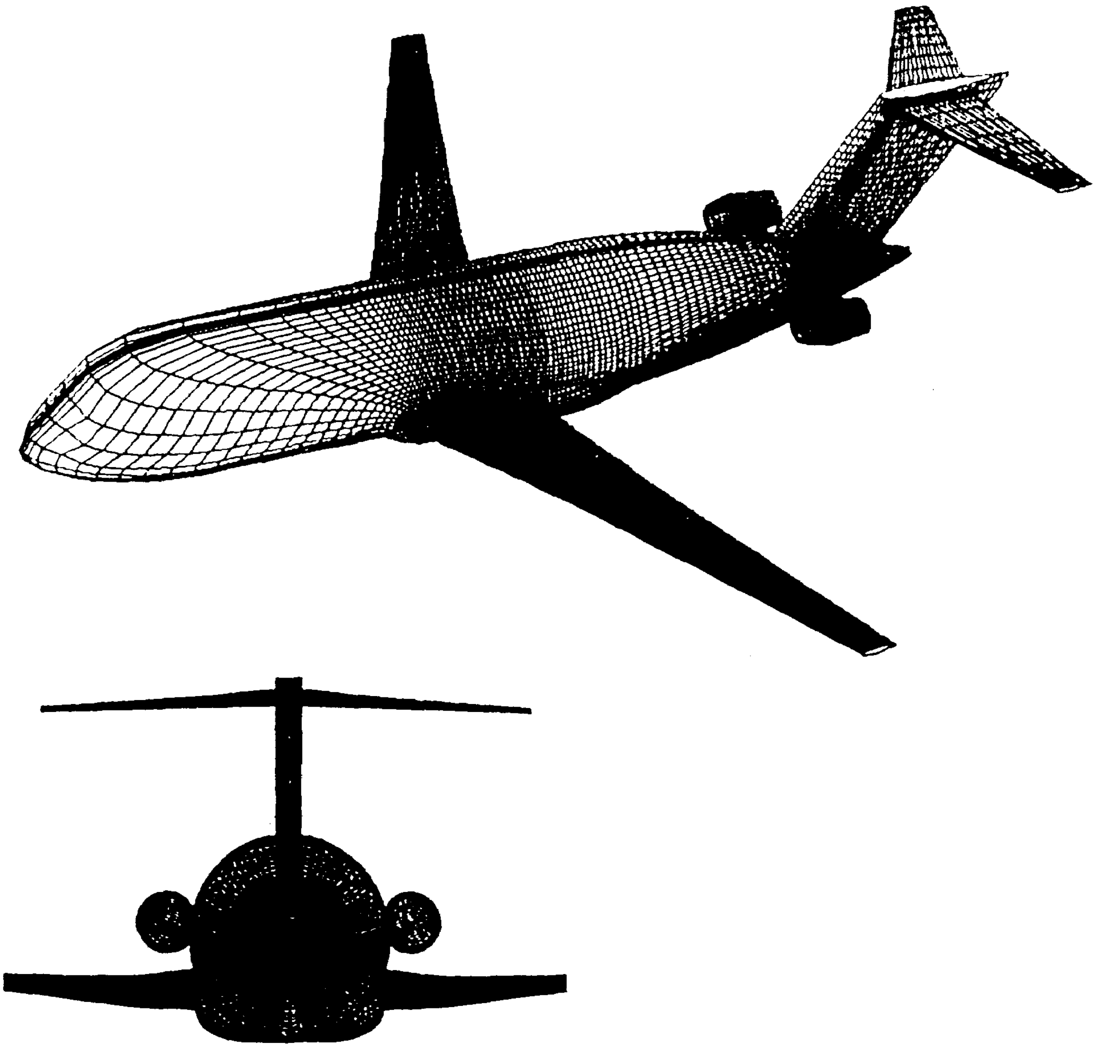
(c) Detailed blunt trailing edge flow of NACA0012.

Fig. 1 N-S analysis of airfoils (ref. 26, 27).

evaluation of the more accurate turbulence modeling and transition is the pacing item.

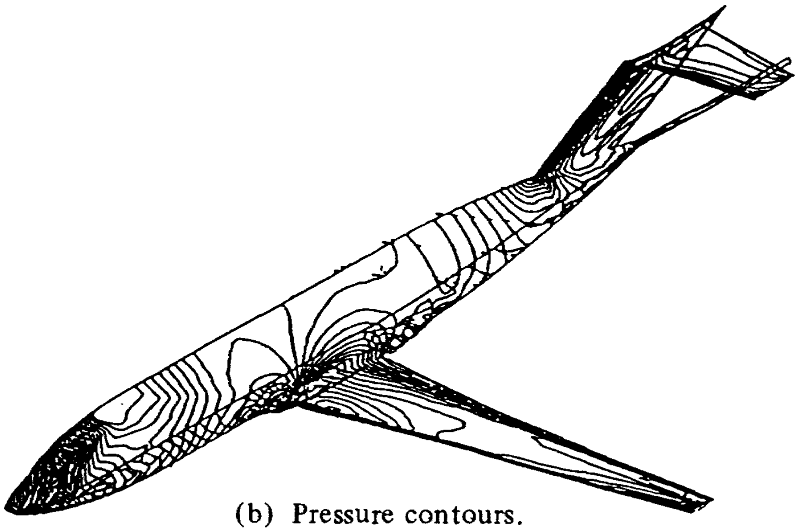
3.2 Wing and Complete Aircraft Analysis

OBAYASHI[29)] et al. proposed an LU-ADI factorization scheme for Navier-Stokes equations. This gives an efficient improvement to the original IAF scheme, since the algorithm is reduced to the scalar bidiagonal inversions suitable to supercomputers. FUJII & OBAYASHI[30)] developed the 3-D version code. New ideas of the scheme improvements are incorporated everyday and the latest version includes a TVD upwind scheme using Roe's averaging. A MUSCL interpolation is applied to obtain the smooth shock wave and high-order accurate resolution. Mean-while, TAKANASHI[31)] developed an efficient and versatile grid-generation method for a realistic 3-D configuration. Analytic functions to map the physical space covering a basic cylindrical body and unswept wing into the cubic computational space are used. To accommodate to the realistic aircraft with swept wing, horizontal and vertical tails and canards, grid adjustment and clustering are made using algebraic functions. The grid generation only requires 20 sec. Some of the latest works by TAKANASHI et al.[32)] are shown in Figure 2. (a) is the surface grid for T-tailed transport configuration with rear-prop-fan. To accurately evaluate viscous effect, fine mesh clustering is made over the every body surface although not shown in this figure. Preliminary

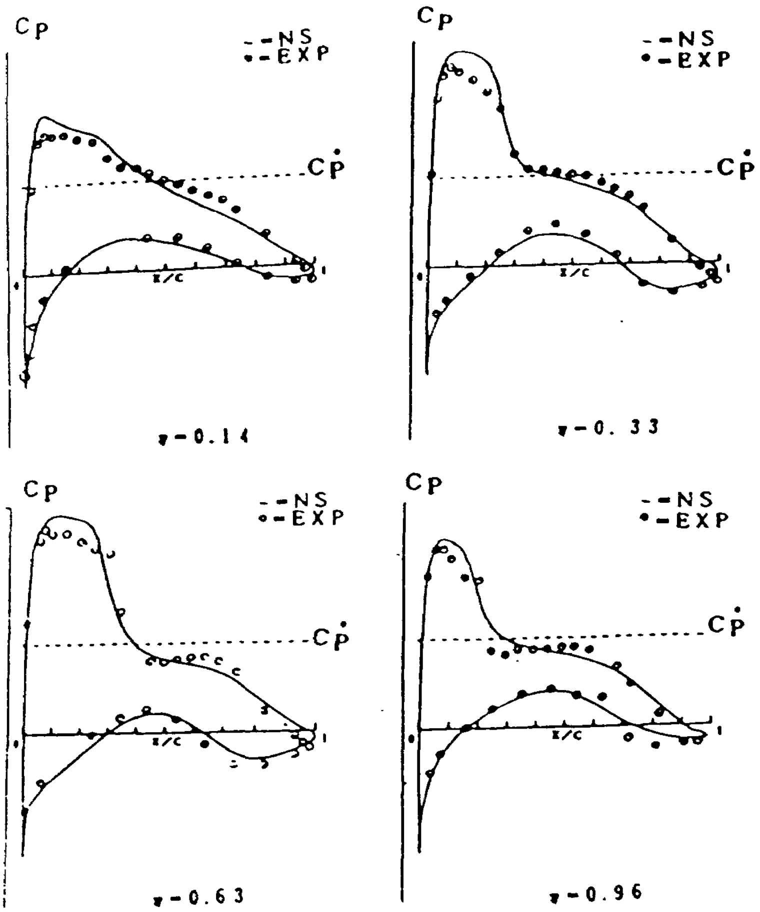


(a) Surface grid.

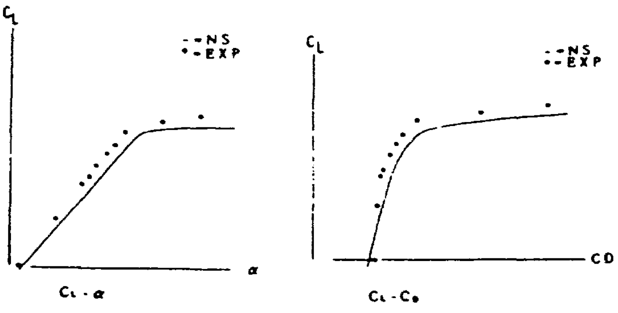
Fig. 2 N-S anaysis of complete aircraft (ref. 32).



(b) Pressure contours.



(c) Cp distributions on wing.



(d) CL-α, CL-CD curves.

Fig. 2 (Continued)

Navier-Stokes analysis for wing-body-T-tail is shown in (b)-(d), at several flow Mach numbers and angles of attack. Wing surface pressure distributions, CL vs. α and CL-CD curves show excellent agreement with the wind tunnel result. CPU Time for 221 x 74 x 62 grid computation requires 6 to 20 hours on VP400 depending on flow conditions. Further code validation study is now on-going.

Another application of present code is the flow analysis of SPACE-PLANE. The authors do not go into the details since SPACE-PLANE is not an appropriate topic to concern at the present SYMPOSIUM TRANSSONICUM, unless the detailed analysis of the transonic flow characteristics is of main concern of the work. Here brief presentation of the figure, however, is made only to show the versatility of the code. The Navier-Stokes code covers low-speed to hypersonic flow regimes and the grid generation is easy to handle for this configuration. The leading edge vortex at large angle of attack will be the only one item to care. The Figure 3 shows an example of supersonic flow computation. A parametric flow analyses at the various Mach number are on the way.

3.3 Unsteady Wing Analysis

Another 3-D Navier-Stokes research is NAKAMICHI's[33] unsteady flow analysis past a swept wing in pitching motion for an aeroelastic problem. 3-point implicit time-accurate version of Beam-Warming diagonalized scheme for a time-dependent moving 3-D grid was applied to NORA swept wing with a low aspect ratio about which unsteady aerodynamics were measured in various European wind tunnels in 1978 by LAMBOURNE[34]. Computations were made for Mach numbers: 0.9~1.1, Reynolds number: 6×10^6, angle of attack: 4°, amplitude of pitching motion: 1°, and reduced frequencies: 0.238~0.428, A 161 x 29 x 35 grid point computation required

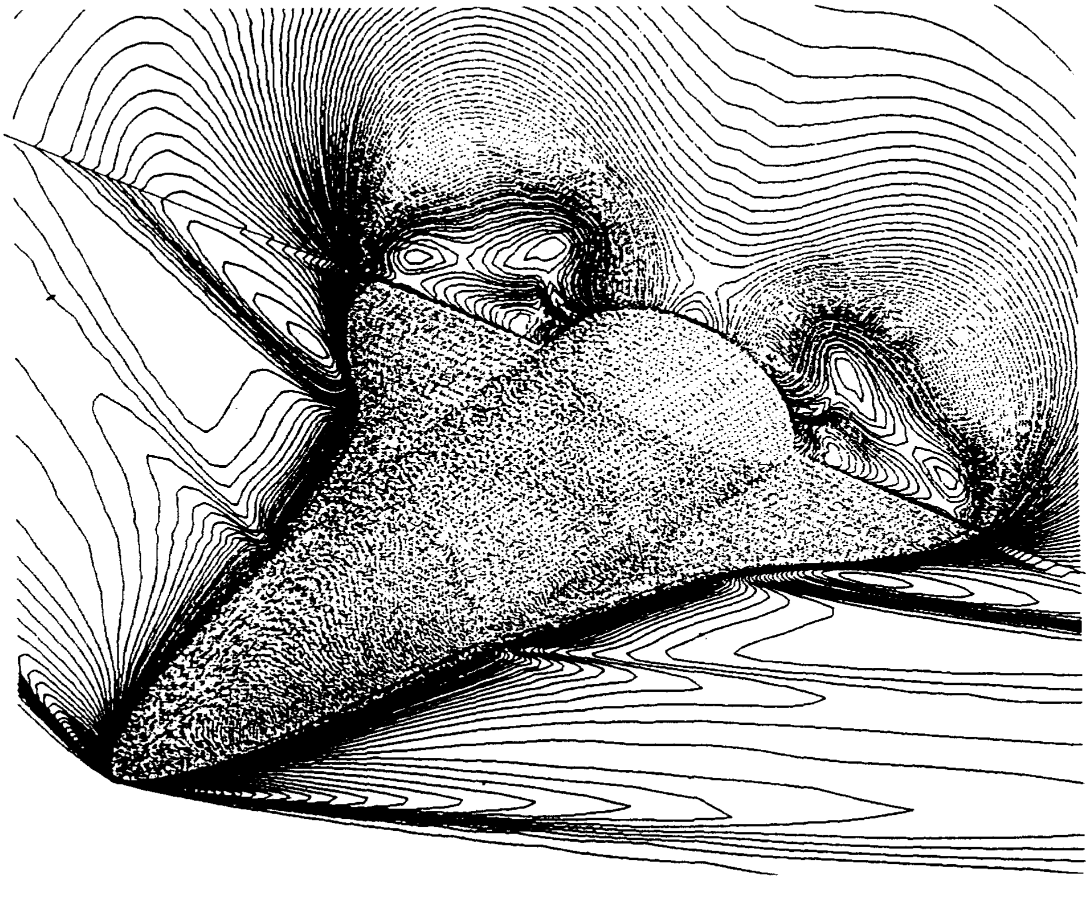
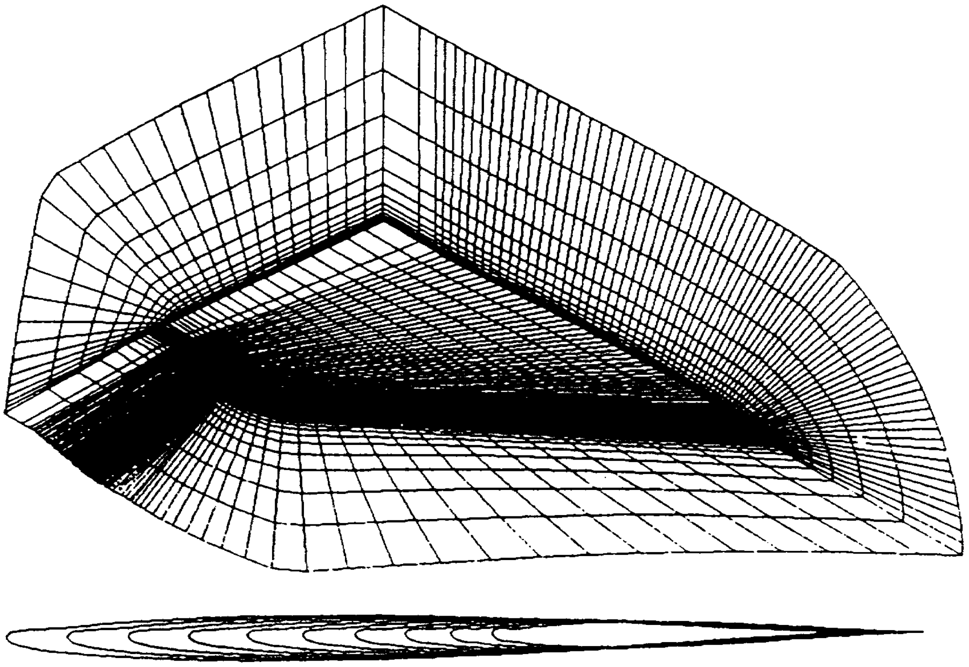


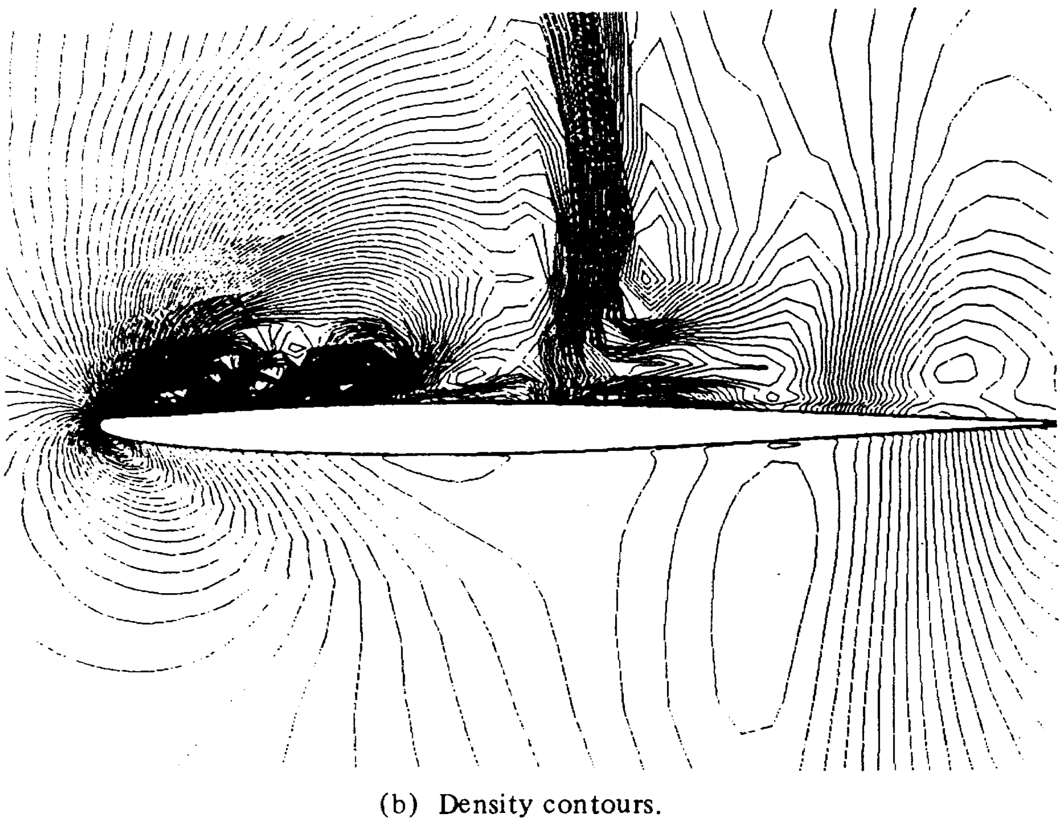
Fig. 3 Flow past SPACEPLANE (ref. 32).

about 7-10 hours on FUJITSU VP400 per one case. Figure 4 shows the mesh used and density contours at one span station. Vortex shedding from up-drooped leading edge interacts with the shock wave. Good agreement with experiment was obtained when compared with ISOGAI's full potential code USTF3[35], although more detailed work remains for the quantitative agreement. Here ISOGAI's USTF3, itself, is an excellent work to be mentioned.

The other unsteady Navier-Stokes analysis researches concerning on the dynamic stall phe-



(a) Grid for analysis.



(b) Density contours.

Fig. 4 Unsteady analysis of NORA wing (ref. 33).

nomena have been made by several groups outside of NAL. One of such works is by SHIDA & KUWAHARA36). They computed a transonic dynamic stall flow past NACA0012 airfoil at the oscilating large angles of attack. Using the assumption of direct simulation without turbulence model the vortex shedding from the airfoil and the interaction with the shock wave are clearly captured. The details are left in the reference.

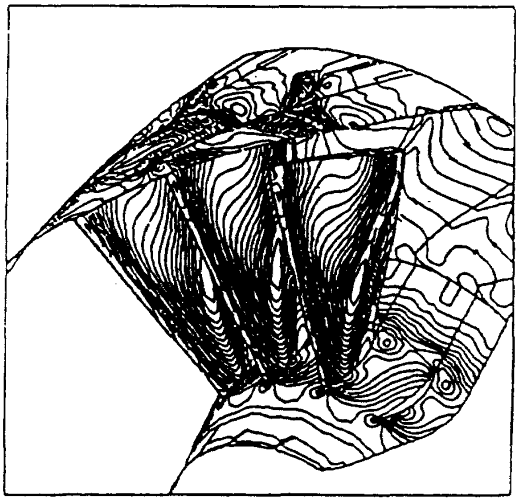
3.4 3-D Fan Analysis

3-D Navier-Stokes analysis application to the aero-engine problems was made by NOZAKI & NAKAHASHI et al.37). 3-D flow field past the rotating turbine and fan cascades were computed applying similar scheme as the previous ones. Figure 5 shows a 3-D view of surface density contours around strongly twisted fan blade. The hub wall rotates with blades, while the outer casing remains fixed. The inlet Mach number is (0.61 at hub, 0.62 at tip) and the outlet Mach number is (0.72, 0.57). Reynolds number is 5×10^5. Figure (b) shows the density contour at the tip section. A strong trailing shock wave from the suction surface reaches to the midchord of the pressure surface of the neighbouring blade and produces shock-induced separation. On the suction surface, the shock wave location advances to the midchord at the mid-span station and the flow is separated. At the hub region, 3-D flow separation is observed. Although not shown here, a good agreement with the experiment were obtained in the cases of FJR-710 R&D fan engine turbine cascade flows. In all of the Navier-Stokes analyses, Baldwin-Lomax turbulence model is used.

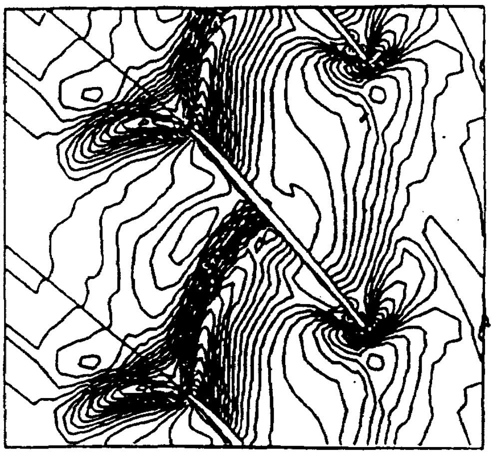
4. EULER ANALYSIS OF AIRCRAFT AND COMPONENT

4.1 Fan-Jet Engine Simulation

HIROSE, ASAI & IKAWA38) analyzed the transonic flow field around a Fan-Jet Engine at an angle of attack. Turbine Powered Simulator wind tunnel test were also made for the same geometry. Figure 6 shows the grid and pressure contours for a flow at Mach number: 0.9, angle of attack: 6°, mass flow ratio: 0.8. Shock wave appears on the leeside of the intake nacelle. And the fan-jet and core-jet interaction is visible. At some distance from nozzle exit, a pair of horseshoe like vortices appear due to the crossflow. The details are not clear because of the crude grid spacing. Surface pressure distribution agreed with experiment. It was confirmed that the temperature effect of the core-jet to the outside flow is negligibly small as far as TPS testing is concerned.

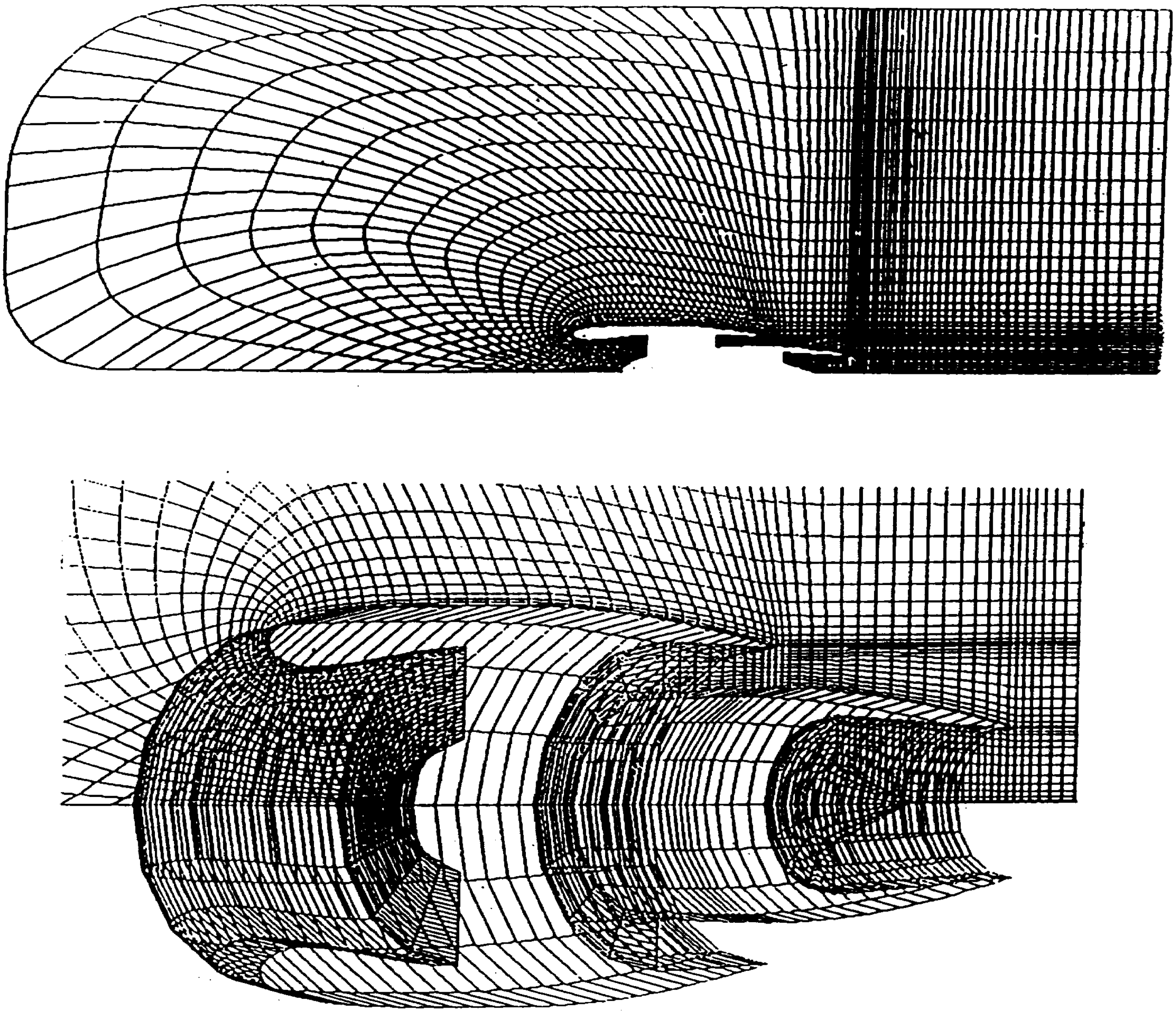


(a) 3-D view of density contours.

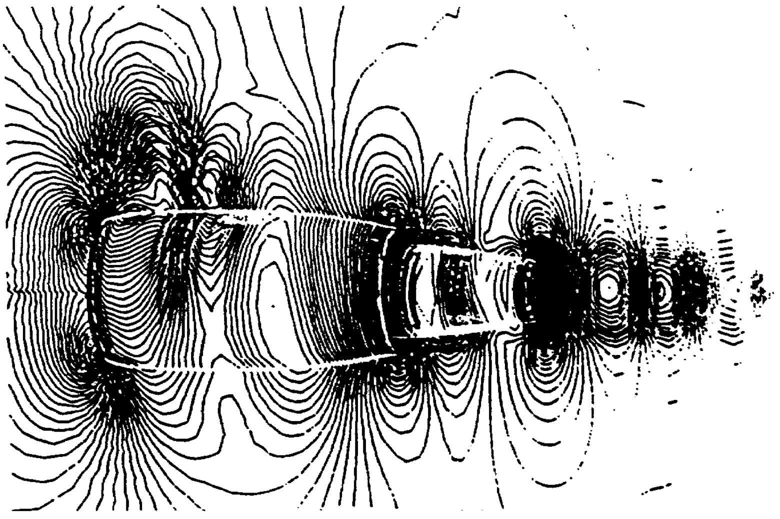


(b) Tip section.

Fig. 5 3-D fan-blade analysis (ref. 37).

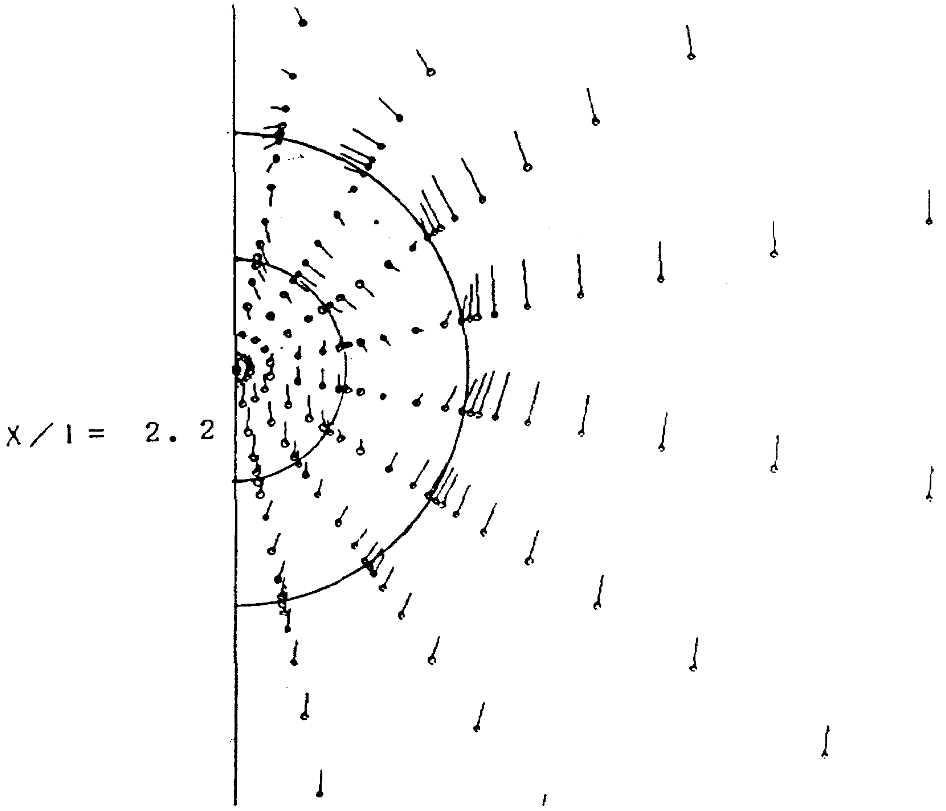


(a) Grid.



(b) 3-D pressure contours.

Fig. 6 Euler analysis of fan-jet engine (ref. 38).



(c) Velocity vectors of exhaust jet.

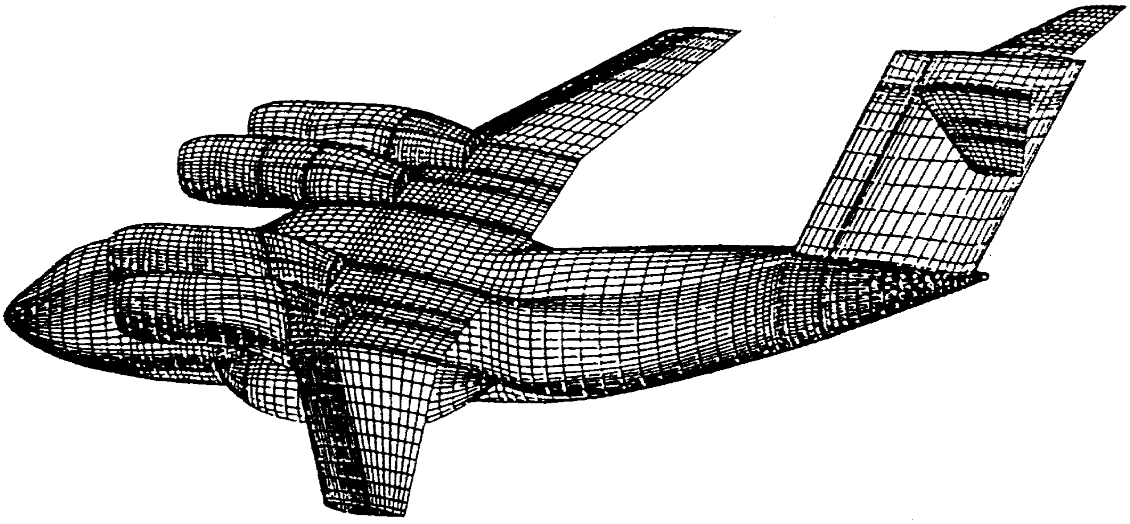
Fig. 6 (Continued)

4.2 Complete Quiet-STOL Research Aircraft ASKA Configuration

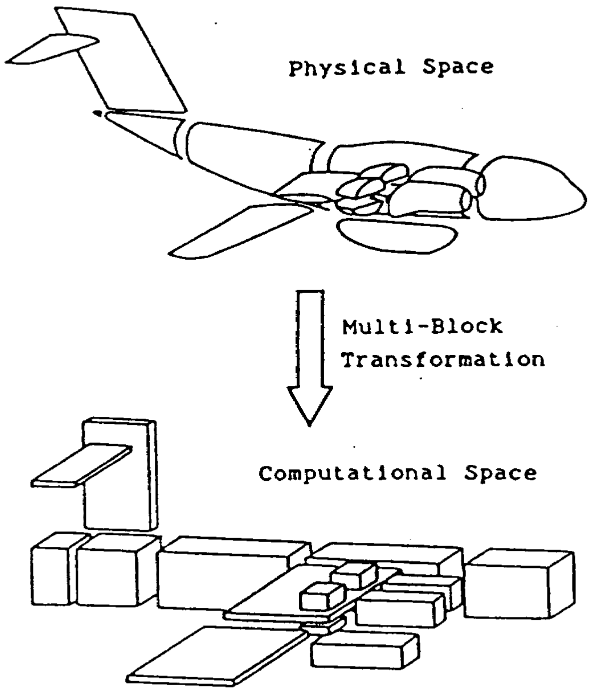
Complete configuration analysis of NAL Quiet-STOL "ASKA (飛鳥)" was made by TAKANASHI & SAWADA39) applying the second-order Roe's Riemann solver. A multi-block transformation technique was used to transform this complicated geometry to a single structured computational space. To construct a good mesh distribution, an interactive procedure was deviced rather than automatic grid generation. Figure 7(a) shows the surface of the aircraft. The effect of four-jet, twin-jet and clear wing geometries were analyzed. 162 x 103 x 76 grid points were used. Figure (b) shows isobar pattern on surfaces and (c) shows shock wave patterns on the upper surface of the wing at Mach number 0.8. It is clear that flow between the nacelles chokes and complex shock waves are formed.

4.3 Other Works

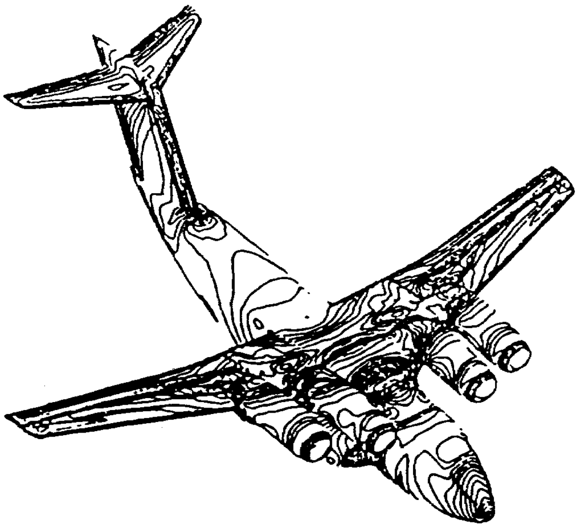
Many works on TVD-related scheme development and applications of both Euler and Navier-Stokes codes to SPACE-PLANE and America's SPACE SHUTTLE-like vehicle have been made at the various research groups in NAL. Works by OGAWA, et al.40), ISHIGURO, et al.41) of the Computational Sciences Division and their coworker, TAKAKURA, et al.42), and YAMAMOTO43) of the Aerodynamics Division are among such excellent examples. Since the complete coverage of CFD is not the object of the present paper and the transonic aerodynamics is concerned at the SYMPOSIUM, the details are not shown here and the interested readers are reffered to the respective NAL reports.



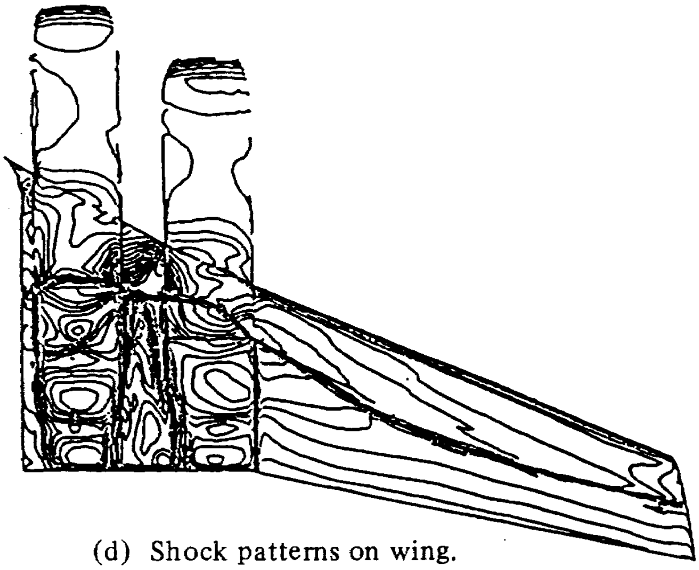
(a) Surface grid.



(b) Multi-block transformation.



(c) 3-D pressure contours.



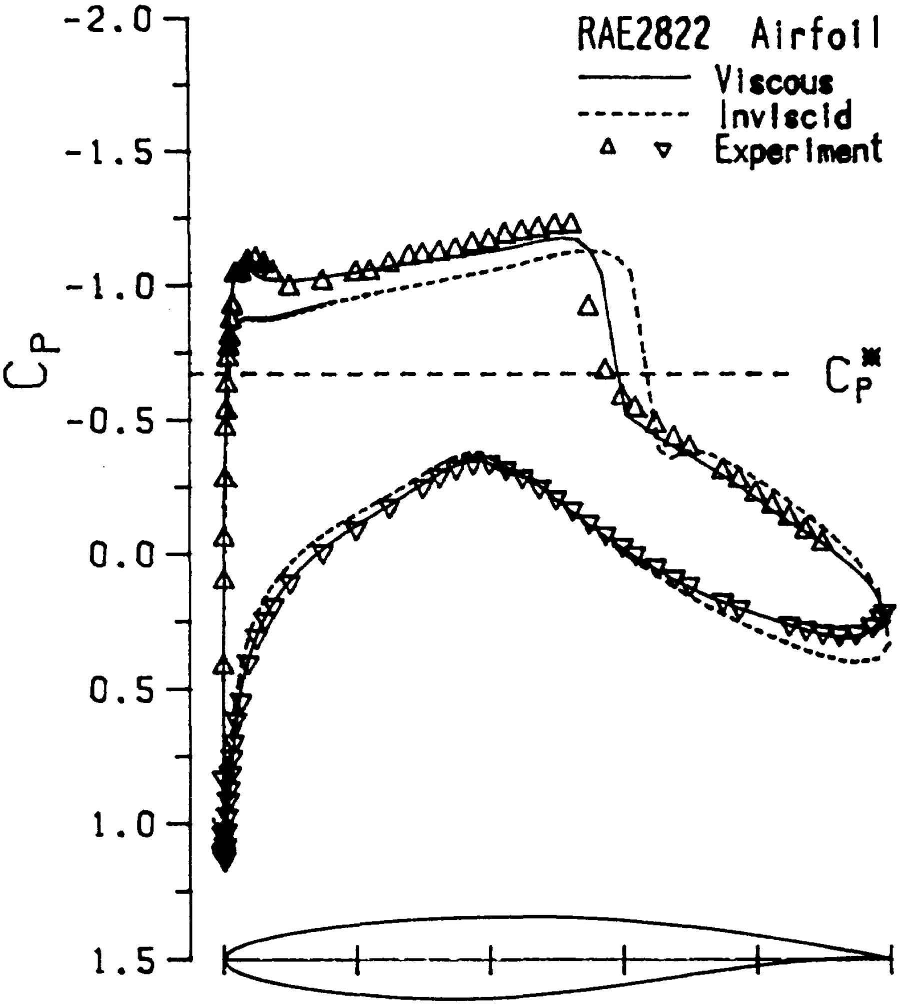
(d) Shock patterns on wing.

Fig. 7 Euler analysis of NAL Quiet-STOL "ASKA (飛鳥)" (ref. 39).

5. VISCOUS-INVISCID INTERACTION

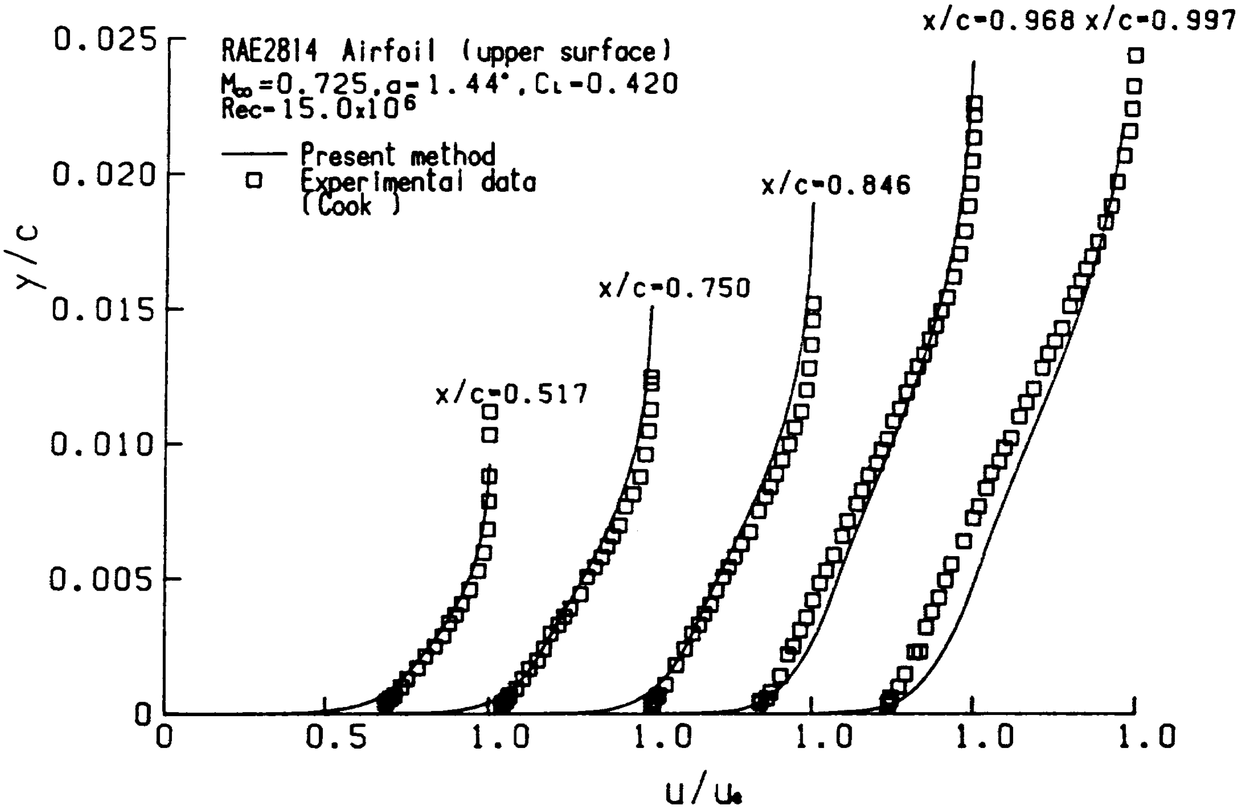
Viscous-inviscid interaction method is a fast and easy analysis tool to predict the aerodynamic performances of the airfoil and wing in engineering point of view although the N-S analysis is exploited recently among researchers. MATSUNO[44)] developed a predictor-corrector Crank-Nicolson scheme to solve 3-D compressible turbulent boundary layer equations over a sweptwing called code 'BLAY'. The scheme is vector-oriented. The code was incorporated with the various inviscid wing analysis codes including JAMESON's[45)] FLO27, FLO30 and similar homemade potential codes. The iterative interaction effects of transonic wings including forward-swept wing for a transport has been extensively evaluated at NAL and the industries[46)]. Also the code served for swept wing design which will be described in the next paragraph.

To advance more refined study, MATSUNO[47)] developed an advanced interaction method for airfoil which incorporates turbulent wake thickness and curvature effects as well as the improved algebraic turbulence model for lifting wake region. The result for RAE2822 is shown in Figure 8. The velocity profiles on the airfoil and wake show excellent agreement with COOK's experiment[48)]. The drag-polar curve comparison also shows good agreement.



(a) Result for RAE2822.

Fig. 8 Advanced interaction method (ref. 47).



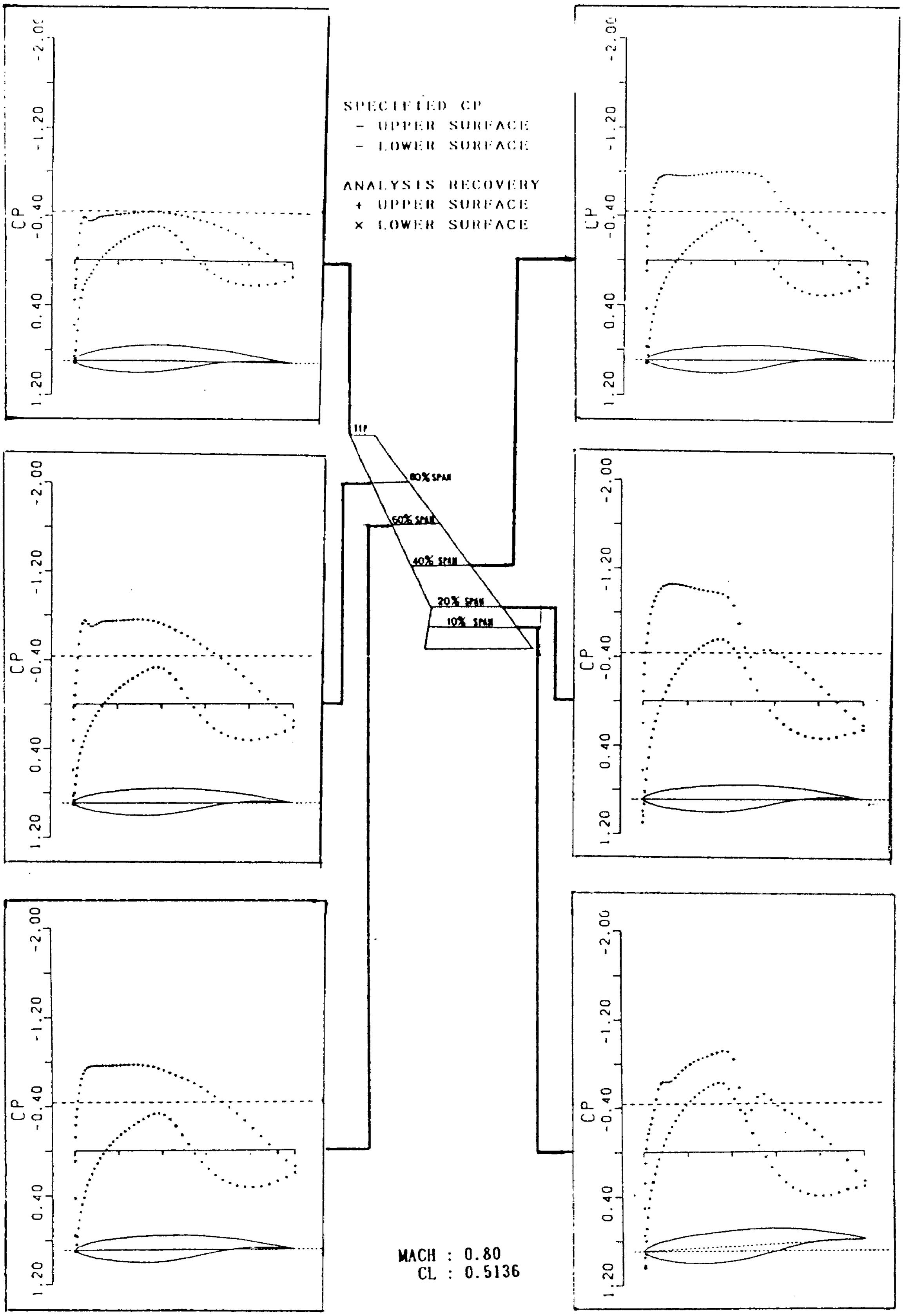
(b) Velocity profiles.

Fig. 8 (Continued)

6. DESIGN METHOD AND ITS APPLICATIONS

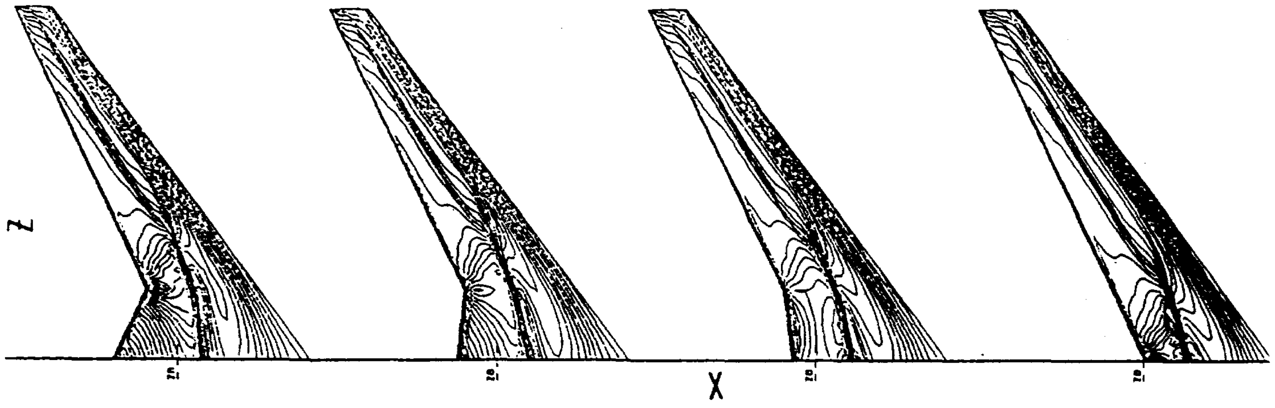
One of the most practical and versatile design methods for transonic swept wing was developed by TAKANASHI[49]. The method is a residual-correction type procedure and is based on the solution of the transonic integral equation of a potential perturbed around a known solution of the flow past a wing. When the pressure distribution on wing with fixed planform is specified as the target, arbitrary flow solver used gives initial pressure distribution for initially guessed geometry. The difference between the target and the initial distributions is related with the geometry correction through the integral equation. Therefore the geometry correction is easily solved and the corrected geometry is used for the flow solver to obtain new pressure distribution. The procedure continues iteratively until the target pressure distribution is realized. The significant feature of the method is that: (1) the geometry correction step is completely independent from the flow solver in the analysis mode, i.e., any flow solver can serve in the analysis mode. (2) a mixed inverse/direct calculation is possible, i.e., only arbitrarily sepcified part of span station can be designed. (3) the design covers the entire wing section from the leading edge to the trailing edge. (4) it is robust even a strong shock wave is specified. (5) computing time of design mode is negligibly short.

The present code, 'WINDES' was linked with various flow solvers such as Jameson's FLO22, FLO27 (with BLAY), FLO30, Boppe's WIBCO[50] and used as a research tool for designing practical wings by NAL and industries. Figure 9 (a) shows an example of a forward-swept wing design. For high lift design at high Mach number, strong shock wave is formed at the wing root and the geometry to alleviate shock strength becomes unrealistic shape. The present authors are investigating how to cure the problem. The leading edge extension is one of device for such treatment. Figure 9 (b) shows the effect of L.E. extension

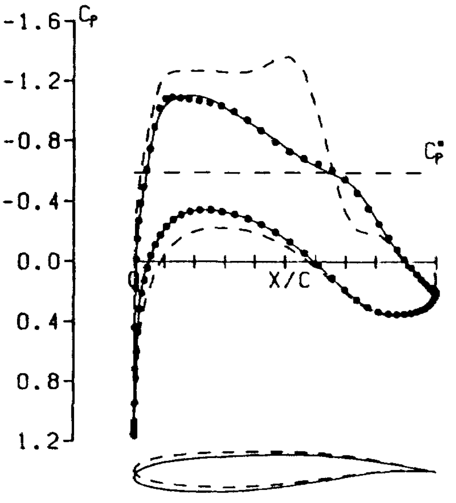


(a) Forward-swept wing design.

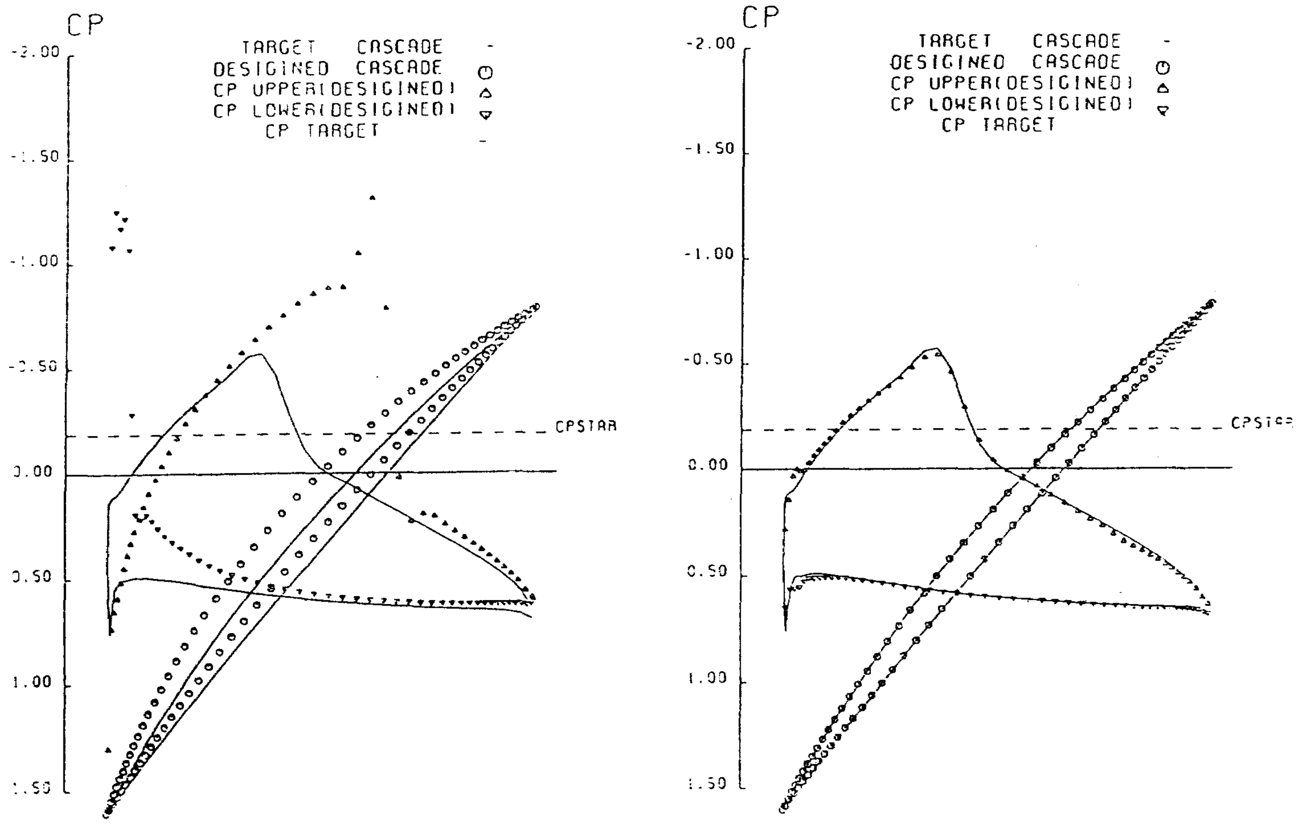
Fig. 9 Examples of new design method (Ref. 49, 51).



(b) Effect of L.E. extension.



(c) N-S airfoil design.



(d) Compressor blade design.

Fig. 9 (Continued)

to a forward-swept wing. By moderate L.E. extension, shock wave can be diffused spanwise and its strength can be weakened. BLAY analysis revealed favorable viscous surface streamlines flowing inward.

The applicability of the design method to other than potential codes was demonstrated by HIROSE, TAKANASHI & KAWAI(51). Navier-Stokes analysis code for airfoil was linked with WINDES. Supercritical airfoils with strong shock wave and weakly-separated boundary layer were modified utilizing the present method. Figure 9 (c) shows an example. Low-speed thick airfoil for LFC was also designed easily. When the work was done, it required more than 10 hours of CPU Time on FUJITSU M-380. But now it can be done less than 20 minutes on a supercomputer. Another application is the cascade design version. Figure 9(d) shows a thin compressor blade design at high Mach number 0.9. Thick blade design which is more difficult to apply was also succesfully tried.

7. SYMPOSIUMS AND CFD WORKSHOP

There are no specialist meetings for the transonic reasearches because researchers devoted purely in this field are quite few even at NAL. Various fluid dynamics meetings, however, are held quite often in Japan. The first CFD Workshop sponsored by JSME was held in 1987. Among the several basic flow problems, transonic airfoil analysis was the theme organized by HIROSE. Applicants from the industries and universities presented their results for NACA0012, RAE2822 and GK-75-06-12 obtained by their N-S and Euler codes. Comparison in detail clarified various aspects of present status of CFD application.(52)

NAL hosts Symposia on Aircraft Computational Aerodynamics (SACAD) annually since 1983. And their proceedings are published(53), NAL is promoting cooperative researches on CFD and transonic aerodynamics with industries and universities. Some of the works in the present paper were the products of these cooperations. The relation with universities were not tight unlike the one in U.S.A. or Europe. Same was the one with industries except wind tunnel testing. One major reason of this comes from the small size of Japanese aerospace industry. But recent progress in this area is stimulating more close relations between NAL, industries and universities. The authors hope more cooperation expected in near future.

8. CONCLUDING REMARKS

Transonic aerodynamics researches are reviewed in a view of CFD. Historical background and recent activities mainly in the field of practical applications are presented. No extensive and comprehensive review of the entire CFD was intensionally avoided because such is not the tradition and the purposes of the authentic SYMPOSIA TRANSSONICUM held every other 13 years. The authors, however, are afraid that not all of the excellent works on transonic aerodynamics research were reffered in the present paper due to the time and space limitation. The authors should be responsible for such is the case.

ACKNOWLEDGEMENTS

The present paper is based on a presentation given at the IUTAM SYMPOSIUM TRANSONICUM III, Goettingen, FRG, May 24-27, 1988. The paper was revised extensively then. The paper contains references of some transonic CFD research works done by the authors' colleague research scientists. Without them, the purpose of the present paper to review transonic CFD researches would not be possible. The authors express their sincere thanks to Drs. KEN-ICHI MATSUNO, JIRO NAKAMICHI and OSAMU NOZAKI for providing their materials.

The authors also express their sincere gratitudes to Prof. ISAO IMAI, the Member of IUTAM International Committee. Their gratitudes also go to the NAL Director, Mr. HIDEO NAGASU and

Division Chief, Mr. KAZUAKI TAKASHIMA for their strong support of the presentation at the Symposium. Finally the authors express their sincere thanks to the unnamed reviewers for their valuable comments and suggestions.

REFERENCES

1) TOMOTIKA, S. & TAMADA, K., "Studies on Two-Dimensional Transonic Flows of Comparessible Fluids. Part III", Quarterly Appl. Math., 9, 129-147, (1951).

2) IMAI, I., "Transonic Flow Research in Japan", SYMPOSIUM TRANSSONICUM AACHEN, 3-7. SEPT. 1962, Springer Ver., pp. 370-393, (1964)/also see "Approximation Methods in Compressible Fluid Dynamics", Inst. Fluid Dyn. & Appl. Math., Univ. Maryland, Tech. Note BN-95, (1957).

3) PEARCEY, H.H., "The Aerodynamic Design of Section Shapes for Swept Wings", ADVANCES IN AERONAUTICAL SCIENCES, Vol. 3, Pergamon Press, N.Y., pp. 277-322, (1962).

4) WHITCOMB, R.T.,"Review of NASA Supercritical Airfoils", ICAS-Paper #74-10, (1974).

5) TAKANASHI, S., "A Method of Obtaining Transonic Shock-Free Flow Around Lifting Aerofoils," Trans. Jap. Soc. Aero. Space Sci., **16**, pp. 246-263, (1973).

6) SHIGEMI, T., "Research on the Transonic Aerofoil Sections at the National Aerospace Laboratory, Japan", ICAS-Paper #74-12, (1974).

7) HIROSE, N., "Computer Experiment of Supersonic Opposing Jet by a Time-Dependent Finite Difference Method," Proc. 10th Int. Symp. on Space Tech. & Sci., TOKYO, 1973, pp. 497-504, (1973).

8) ISHIGURO, T., "Finite-Difference Calculations for Two-Dimensional Unsteady Expanding Flows," AIAA J., **10**, pp. 217-219, (1972).

9) MAGNUS, R., GALLAHER, W. & YOSHIHARA, H., "Inviscid Supercritical Airfoil Theory," AGARD-CP-35 'SPECIALIST MEETING ON TRANSONIC AERODYNAMICS', Paper No. 3, (1968).

10) MURMAN, E.W. & COLE, J.D., "Calculation of Plane Steady Transonic Flows," AIAA J., **9**, pp. 114-121, (1971).

11) BALLHAUS, W.F. & BAILEY, F.R., "Numerical Calculation of Transonic Flow About Swept Wings," AIAA Paper #72-677, (1972).

12) GARABEDIAN, P. & KORN, D., "Analysis of Transonic Airfoils, "Comm. P. & Appl. Math., **24**, (1971).

13) BAILEY, F.R. & BALLHAUS, W.F., "Relaxation Methods for Transonic Flow About Wing-Cylinder Combinations and Lifting Swept Wings," Proc. 3rd Int. Conf. on Num. Met. in Fluid Dyn. Paris, July 3-7, 1972, Springer Ver., LECTURE NOTES IN PHYSICS **19**, pp. 2-9, (1972).

14) KAMIYA, N., et al., "Numerical Computation of Transonic Flow past Swept Wing," Proc. 9th JASS Fluid Dynamics Conf. (in Japanese), pp. 90-93, (1977).

15) ISHIGURO, T., "Finite Difference Calculations of Inviscid Transonic Flows over Oscillating Airfoils," NAL-TR-632 (in Japanese), (1980).

16) ISOGAI, K., "Numerical Study of Transonic Flutter of a Two-Dimensional Airfoil," NAL-TR-617T, (1980).

17) TAKANASHI, S., "Numerical Method for 3-D Transonic Integral Equation," Proc. 11th JASS Fluid Dynamics Conf. (in Japanese), pp. 86-89, (1979).

18) ISHIGURO, T., "Transonic Airfoil Design of Full Potential Flow, Part I & II," NAL-TR-672/673 (in Japanese), (1981).

19) KAMIYA, N., "On the Drag Divergence of Two-Dimensional Airfoils at Transonic Speeds," NAL-TR-299 (in Japanese), (1973).

20) KAMIYA, N. & HIROSE, N., "Research on Transonic Wings at the National Aerospace Laboratory, Japan", ICAS-Paper #80-11-1, (1980).

21) TAKASHIMA, K., "Experimental Works in the NAL High Reynolds Number Two-Dimensional Transonic Wind Tunnel on Advanced Technology and NACA Airfoils", ICAS-Paper #82-5-4-4, (1982).

22) REYNOLDS, W.C. & MACCORMACK, R.W. (Editors), PROC. 7th INT. CONF. ON NUM. Met. IN FLUID DYNAMICS, STANFORD, June 23-27, 1980, Springer Ver., LECTURE NOTES IN PHYSICS 141, (1981).

23) KAMIYA, N., "Several Topics of Computational Fluid Dynamics Research at the National Aerospace Laboratory", Proc. Int. Symp. on CFD Tokyo (ISCFD Tokyo), (1985).

24) STEGER, J.L., "Implicit Finite Difference Simulation of Flow about Arbitrary Two-Dimensional Geometries," AIAA J., **16**, pp. 679-686, (1978).

25) KAWAI, N. & HIROSE, N., "Development of the Code NSFOIL for Analyzing High Reynolds Number Transonic Flow Around An Airfoil", NAL-TR-816, (in Japanese), (1984).

26) MIYAKAWA, J., HIROSE, N. & KAWAI, N., "Transonic Aileron Computation in Comparison on Wind Tunnel Test", Proc. 3rd NAL Symposium on Aircraft Computational Aerodynamics, June 13-14, 1985, NAL-SP-5, (in Japanese), pp. 235-242, (1985).

27) HIROSE, N., "Navier-Stokes Analysis of Blunt Trailing Edge Flow of Supercritical Airfoils, First Report: Preliminary Results," Proc. 20th JASS Fluid Dynamics Conf. (in Japanese), pp. 198-201, (1988).

28) KAWAI, N., HIROSE, N. & ISHIDA, Y., "Navier-Stokes Analysis of Laminar Flow Control of a Two-Dimensional Airfoil," Proc. 4th NAL Symposium on Aircraft Computational Aerodynamics, June 19-20, 1986, NAL-SP-7, (in Japanese), pp. 277-280, (1986).

29) OBAYASHI, S. & KUWAHARA, K., "An Approximate LU Factorization Method for the Compressible Navier-Stokes Equations," J. Comp. Phys., **63**, No. 1, (1986).

30) FUJII, K. & OBAYASHI, S., "Navier-Stokes Simulation of Transonic Flow Over Wing-Fuselage Combinations", AIAA P. 86-1831, (1986).

31) TAKANASHI, S. & ISAWA, T., "Analytical Method for Generating Computational Grids about Complete Aircraft Configurations", Proc. 4th NAL Symposium on Aircraft Computational Aerodynamics, June 19-20, 1986, NAL-SP-7, (in Japanese), pp. 281-288, (1986).

32) TAKANASHI, S., OBAYASHI, S., MATSUSHIMA, K. & FUJII, K., "Numerical Simulation of Compressible Viscous Flows around Practical Aircraft Configurations", AIAA P. 87-2410, (1987).

33) NAKAMICHI, J., "Calculations of Unsteady Navier-Stokes Equations around an Oscillating 3-D Wing Using Moving Grid System", AIAA P. 87-1158, (1987).

34) LAMBOURNE, J., "Compendium of Unsteady Aerodynamic Measurements," AGARD Report No. 702, (1982).

35) ISOGAI, K., "Calculation of Unsteady Transonic Potential Flow Over Oscillating Three-Dimensional Wings", NAL-TR-706T, (1982).

36) SHIDA, Y. & KUWAHARA, K., "Computation of Flow around NACA0012 Airfoil at High Angle of Attack," Proc. 5th NAL Symposium on Aircraft Computational Aerodynamics, June 25-26, 1987, NAL-SP-8, (in Japanese), pp. 73-80, (1987).

37) NOZAKI, O., NAKAHASHI, K. & TAMURA, A., "Numerical Analysis of Three-Dimensional Cascade Flow by Solving Navier-

Stokes Equations," Proc. 1987 Int. Gas Turbine Conf., 87-TOKYO-IGTC-43, (1987).

38) HIROSE, N., ASAI, K. & IKAWA, K., "Euler Analysis of 3-D Fan-Jet Engine Flow Field", NAL TR- in print, (1988).

39) TAKANASHI, S. & SAWADA, K., "Numerical Simulation of Compressible Flow Field about Complete ASKA Aircraft Configuration", Proc. Int. Powered Lift Conf., Santa Clara, (1987).

40) OGAWA, S., ISHIGURO, T. & TAKAKURA, Y., "Numerical Simulations of Flow around Shuttle Orbiter," Proc. 15th NAL Symposium on Aircraft Computational Aerodynamics, June 25-26, 1987, NAL-SP-8, (in Japanese), pp. 19-28, (1987).

41) ISHIGURO, T., OGAWA, S. & TAKAKURA, Y., "Numerical Calculation of Three-Dimensional Flow by TVD Scheme," Proc. 5th NAL Symposium on Aircraft Computational Aerodynamics, June 25-26, 1987, NAL-SP-8, (in Japanese), pp. 137-146, (1987).

42) TAKAKURA, Y., ISHIGURO, T. & OGAWA, S., "On the Stationary Solutions of Three-Dimensional Euler Equations by the TVD Difference Schemes," Proc. 5th NAL Symposium on Aircraft Computational Aerodynamics, June 25-26, 1987, NAL-SP-8, (in Japanese), pp. 211-216, (1987).

43) YAMAMOTO, Y. & KUBO, S., "Numerical Simulation of Hypersonic Flow around a Space Plane," NAL-TR-976T, (1988).

44) MATSUNO, K., "A Vector-Oriented Finite Difference Scheme for Calculating Three-Dimensional Compressible Laminar and Turbulent Boundary Layers on Practical Wing Configurations", AIAA P. 81-1020-CP, (1981).

45) JAMESON, A. & CAUGHEY, D.A., "A Finite Volume Method of Transonic Potential Flow Calculations," AIAA P. 77-635, (1977).

46) HIROSE, N., MATSUNO, K., KOMATSU, K. & KAWAMURA, R., "Boundary Layer-Weak-Interaction Analysis Method for Transonic Wing", Proc. 16th JSASS Annual Meeting, (in Japanese), (1985).

47) MATSUNO, K., "A Viscous-Inviscid Interaction Method to Predict the Aerodynamic Performance of Airfoils at Transonic Speed", NAL-TR-873, (1985).

48) COOK, P.H., MCDONALD, M.A. & FIRMIN, M.C.P., "Aerofoil RAE2822-Pressure Distributions, and Boundary Layer and Wake Measurements," AGARD-AR-138, Paper No. A6, (1979).

49) TAKANASHI, S., "Iterative Three-Dimensional Transonic Wing Design Using Integral Equations," J. Aircraft, 22, pp. 655-660, (1985).

50) BOPPE, C.W., "Transonic Flow Field Analysis for Wing-Fuselage Configurations," NASA CR-3243, (1980).

51) HIROSE, N., TAKANASHI, S. & KAWAI, N., "Transonic Airfoil Design Utilizing Navier-Stokes Analysis Code," AIAA J., 25, pp. 353-359, (1987).

52) HIROSE, N., "Organizer's Compendium on workshop of 2-D Airfoil Flow analyses," in PROC. FIRST CFD WORKSHOP, JSME (Inoue, M., ed.), (1988).

53) Proceedings of NAL Symposia on Aircraft Computational Aerodynamics, held annually and published on NAL-SP Reports, (in Japanese).

**TECHNICAL REPORT OF NATIONAL
AEROSPACE LABORATORY
TR-1018T**

航空宇宙技術研究所報告1018T号（欧文）

平成元年4月発行

発行所　航空宇宙技術研究所
東京都調布市深大寺東町7丁目44番地1
電話三鷹(0422) 47-5911（大代表）〒182
印刷所　株式会社　東京プレス
東京都板橋区桜川2－27－12

Published by
NATIONAL AEROSPACE LABORATORY
7-44-1 Jindaijihigashi-Machi Chōfu, Tokyo
JAPAN

Printed in Japan