ISSN 0389-4010
UDC 533.6.04:
533.6.011.3

TECHNICAL REPORT OF NATIONAL AEROSPACE LABORATORY

TR-911T

Comparison of Transonic Airfoil Characteristics by Navier-Stokes Computation and by Wind Tunnel Test at High Reynolds Number

Naoki HIROSE, Nobuhiro KAWAI and Jun-ichi MIYAKAWA

September, 1986 → 1986.09

NATIONAL AEROSPACE LABORATORY

CHŌFU, TOKYO, JAPAN

Comparison of Transonic Airfoil Characteristics by Navier-Stokes Computation and by Wind Tunnel Test at High Reynolds Number*

By

Naoki HIROSE** Nobuhiro KAWAI** Jun-ichi MIYAKAWA***

ABSTRACT

The two-dimensional Navier-Stokes code, NSFOIL, is validated in terms of analysis of practical transonic advanced technology airfoil compared to data obtained from wind-tunnel tests. The comparison is carried out at a high Reynolds number of 23 million which corresponds to the acutal flight condition of a transonic transport aircraft. The numerical data show satisfactory agreement in both aerodynamic forces and pressure coefficients, so the code is concluded to be a very effective tool for predicting nonlinear characteristics of transonic airfoils, which has not been possible with any existing inviscid code.

In the course of careful comparison, some future improvements of the code are probed in order to obtain even better simulation with this promising computer code.

概　要

翼型に対する高レイノルズ数遷音速流の時間平均ナビエ・ストークス解析コードＮＳＦＯＩＬを用いて，実用目的の遷音速先端技術的翼型の空力特性を解析し，風洞試験の結果との比較により，コード検証を行なった。比較は遷音速輸送機の実機飛行条件に対応する高いレイノルズ数，23×10^6，で行ない，空力係数，圧力分布ともに良好な一致を得た。その結果，従来の非粘性コードでは不可能であった遷音速翼型の非線形特性の予測にきわめて有効な設計ツールであることが明らかにされた。さらに，本コードに対するいくつかの改良点について，比較解析の結果，明らかにされ，さらに良い予測法への手がかりが示された。

INTRODUCTION

Recent rapid advances in computational aerodynamics with growing computer hardware capability have made variety of numerical air flow analyses possible.[1] This progress has brought dramatic changes in aircraft industries. Numbers of transonic flow analysis codes have been carefully verified by comparing to wind tunnel experiments, and have matured to be practical tools for actual aircraft development. Aerodynamic engineers, who had to rely solely upon wind tunnel testings, have now obtained a powerful alternative which is expected to grow even more into the next century.

It is commonly agreed that CFD (Computa-

* Received 30, June 1986

** The Second Aerodynamics Division

*** Joint Collaborative Research Member (Mitsubishi Heavy Industries, Ltd.)

tional Fluid Dynamics) technology is superior to the conventional technique in many ways.[2] One is the cost effectiveness. Computer hardware has been improved dramatically for the last decade and computing cost keeps decreasing exponentially, whereas wind tunnel test models still need fairly large amount of man power work. Another is the free from interference effects. CFD analysis is by its nature free from any type of interference, although wind tunnel testings are always disturbed by wind tunnel wall and model support system interference or sometimes turbulence of the air flow.

In order to make full use of this new sophisticated design tool, an original aerodynamic CAD system has been established at the company to which the third author belongs by combining many of these computer codes into an efficient interactive design system. When the engineers design aerodynamic configuration, they only need to sit down in front of a graphic work station and communicate to the main frame computer where dozens of CFD codes and large memory of data base are ready for the appropriate call by the engineers. Owing to this CAD system, aerodynamic engineers are able to achieve their challenging task to design aircraft configurations in shorter period of time, as well as with more accuracy to meet the high performance requirements.

This current CFD technique applied to practical aircraft design, however, has been rather restricted to inviscid codes with some exceptions of boundary layer computation. This restriction does not become a major drawback as long as the computer code is used in the vicinity of the aircraft design point. Because the air flow near the design point would not contain a serious separation anyway, and under these circumstances boundary layer correction presents reasonable accuracy of analysis. However, as the aircraft design requirements become more and more demanding, aerodynamic designers are forced to challenge even more severe trade-offs between design point performances ank off-design region characteristics.

In order to evaluate the aerodynamic characteristics in off-design domains, wind tunnel testing is still the only one reliable design tool for the designers. Or in other words, wind tunnel tests are only required to evaluate the off-design performances of aircraft, since the current CFD technique takes good care of the design point performance. In fact, during the development of F-16 fighter aircraft, for instance, only 15 percent of the whole wind tunnel test hours are for attached flow analysis.[3] The more critical trade-offs apparently necessitate another breakthrough in CFD application of aircraft industries, namely more accurate computing capability to analyse the air flow at outer limits of flight envelope. It really should be the major revolution of CFD technique to replace the remaining 85 percent of the wind tunnel test hours conducted for vortex dominating and largely separated air flow analysis.

The never ending progress of CFD technology has again responded to this necessity by offering more sophisticated Navier-Stokes computation. The first breakthrough was made by Deiwert in 1975 for biconvex airfoil calculation using explicit finite difference scheme.[4] Steger then proposed more time-efficient implicit code for arbitrary shape airfoils using Implicit Approximate Factorization of Beam and Warming.[5,6] This numerical method has been applied to various types of air flow analyses ever since. It includes transonic aileron buzz by Steger,[7] transonic afterbody flow by Deiwert,[8] transonic thick airfoil by Levy[9], and unsteady transonic flow in diffusers by Liou and Coakley[10]. Other informative efforts are airfoil near subsonic maximum lift by Anderson, et al[11] and some exotic airfoils by Barth, et al[12]. These pioneer application efforts have shown the promising capability of the Navier-Stokes computation, which is more than attractive for the aerodynamic engineers of ailcraft industries who were thirsty for a technology revolution of viscous flow computation. Although they still need to undertake large amount of work of the validation of this relatively new computing technology.

Talking from the aircraft designer's point of view, the computer code validation work should be always related to performance prediction of actual aircraft. Therefore the computational results are hopefully to be compared to flight test data. Measuring detailed flow properties during flight tests, however, is not always an easy task. Engineers then sometimes have to settle with the comparison to wind tunnel test data. Even so, it is highly desirable that the comparison is to be carried out at actual flight Reynolds number.

National Aerospace Laboratory (NAL) and Mistubishi Heavy Industries, Ltd. (MHI) have jointly conducted this desirable validation work as a part of collaborative joint researches on transport aircraft aerodynamic design using computational aerodynamics. Both Navier-Stokes computer code and high Reynolds number wind tunnel facility are available at NAL for the detailed validation work. A transonic advanced technology airfoil designed at MHI was chosen for the comparative study, not only because it is relatively easy to test at the wind tunnel, but also because "the airfoil problem in essence is a microcosm of most the difficulties in numerical simulation one would encounter for the full scale aircraft investigation."[13]

NSFOIL is a Navier-Stokes code developed at NAL for two-dimensional transonic viscous flow analysis by two of the co-authors of this paper.[14] It employs Implicit Approximate Factorization scheme and the turbulence model of Baldwin and Lomax with options of full or thin layer Navier-Stokes computation. Computational grid was generated by AFMESH, a code for generating C-grid by elliptic type partial differential equation method originated by Thompson,[15] and resulted mesh was partially enhanced by the geometrical method.

NAL High Reynolds Number Two-Dimensional Transonic Wind Tunnel (HR2DTWT) has the capability in offering airfoil test data at actual flight Reynolds number.[16] It was constructed in 1978, and since then it has been contributed for both fundamental research work and aircraft development projects.

The present collaborative work has two purposes; 1) to assess the validation of the Navier-Stokes code as a viscous flow solver of transonic airfoil analysis, and 2) to probe the future improvements of the code to obtain better numerical simulation. This paper is essentially the summary of the collaborative work. A brief description of the Navier-Stokes computation is first presented, then the outline of the wind tunnel tests is addressed. After comparing the two simulations in detail, the validation and the future improvements of the code are finally discussed.

NAVIER-STOKES COMPUTATION

The governing equations of NSFOIL are the two-dimensional Navier-Stokes equations, which are written as the strong conservation form in Cartesian coordinates. NSFOIL is based on the Implicit Approximate Factorization scheme for Navier-Stokes equations originally developed by Beam, Warming and Steger. Numerical stability is controlled by the fourth order explicit dissipation terms and the second order implicit dissipation terms. The final equations to solve numerically are

$$
\begin{aligned}
&[\, I + h\partial_\xi \hat{A}^n - J^{-1}\alpha h \nabla\xi\Delta\xi J \\
&\qquad - Re^{-1} h\partial_\xi J^{-1}\hat{N}^n \,] \\
&[\, I + h\partial_\eta \hat{B}^n - J^{-1}\alpha h \nabla\eta\Delta\eta J \\
&\qquad - Re^{-1} h\partial_\eta J^{-1}\hat{M}^n \,]\, \Delta\hat{Q}^n \\
&= -h\,[\, \partial_\xi \hat{E}^n + \partial_\eta \hat{F}^n - Re^{-1}\,(\partial_\xi \hat{R}^n + \partial_\eta \hat{S}^n)] \\
&\qquad - \alpha' h J^{-1}\,[(\nabla\xi\Delta\xi)^2 + (\nabla\eta\Delta\eta)^2]\, J\hat{Q}^n
\end{aligned}
$$

In order to improve the airfoil and wake boundary conditions, they are treated implicitly in the formulation of block-tridiagonal system. The turbulence modeling is also applied as the algebraic model of Baldwin-Lomax type to simulate turbulence.[17]

The computational grid, on the other hand, is constructed using AFMESH, a computer code for two-dimensional numerical mesh developed also at NAL.[18] AFMESH basically makes use of three mesh generating techniques, i.e. 1) elliptic type PDE method originated by Thompson, 2) geometric method using polynomial functions, and 3) exponential expansion to have the boundaries clustered. The resulted grid is again enhanced near the airfoil surface and the trailing edge by improving the distortions generated by the above mentioned grid.

Fig. 1 shows the generated grid for an advanced technology airfoil, which is designed at Mach number 0.77 and lift coefficient 0.65. The computational mesh of 125 x 51 points gives fine resolution near the body boundary and also reasonable space of 10 chord lengths for upstream and downstream directions. The minimum spacing of the grid adjacent to the surface is 1.0×10^{-5} of the chord length.

The transonic characteristics of the airfoil is analysed at Reynolds number of 7 million and 22 million. The higher number is corresponding

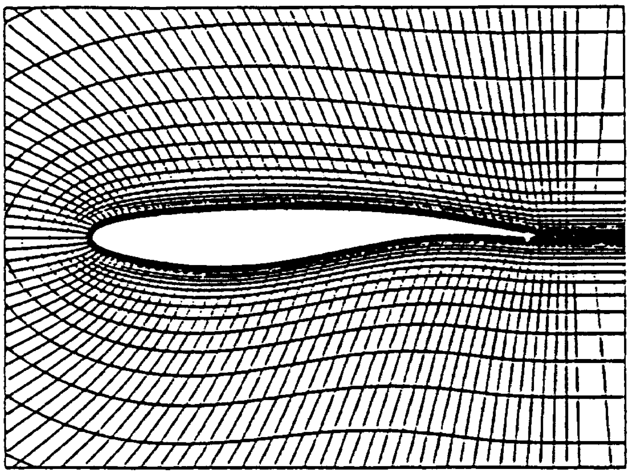
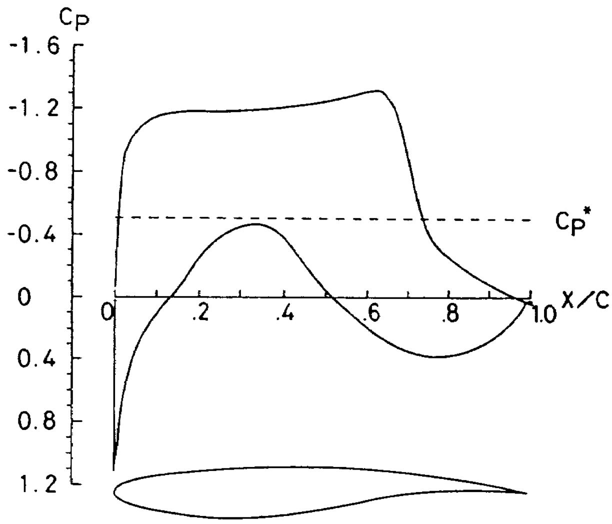
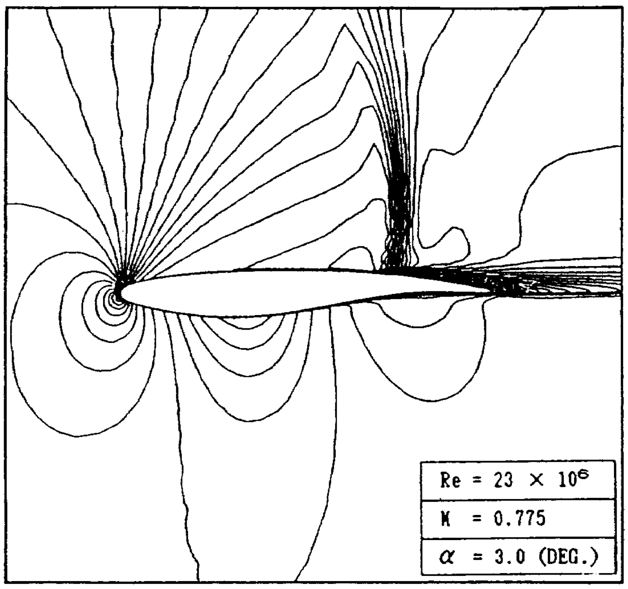


Fig. 1. Computational grid generated about transonic airfoil.



(a) Pressure distribution



(b) Mach number contour

Fig. 2. Typical outputs of numerical simulation (Re = 22×10^6, M = 0.775, alpha = 3.0°).

to that of actual cruising condition of a transonic transport aircraft, and the other is to that of the lowest number the wind tunnel is capable for. The numerical experiment is conducted in two modes, the angle of attack sweep mode at constant Mach numbers (M = 0.75, 0.775) and the Mach number sweep mode at constant angle of attack (alpha = 1.0 deg.). Fig. 2 shows a set of typical output formats of the numerical simulation of NSFOIL, including the pressure distribution and Mach number contours.

WIND TUNNEL TESTING

The transonic advanced technology airfoil was also analysed by physical simulation at NAL High Reynolds Number Two-Dimensional Transonic Wind Tunnel. The facility was constructed in 1978 as one of the several wind tunnels in the world which are capable of offering aerodynamic test data at the actual flight Reynolds number. It has been extremely active in contributing to the basic research works and the aircraft development projects.

The wind tunnel test setups is illustrated in Fig. 3. This tunnel is a blow-down type with solid walls on both sides and variable slotted walls for the upper and lower sides. The slot ratio is 3 percent for this specific wind tunnel test. The test section is 1.0 m high, 0.3 m wide and 1.5 m long. The airfoil model of 0.25 m in chord length is installed between the two Schlieren windows and supported by the rigid steel blocks outside the flow passage. The carefull sealings are provided between the model and the walls. The facility is capable of Reynolds number up to 40 million with more than 20 seconds of measuring time. The current test was carried out at Reynolds number of 23 million based on the chord length. The wind tunnel covers Mach number ranging from 0.2 to 2.0, whereas this particular test was conducted at transonic region from 0.6 to 0.85. The high pressure air, reaching up to 118×10^4 Pa (12 kg/cm^2) in total pressure at the test section, is supplied by three sphere tanks, which provide the test interval of 30 minutes typically.

As illustrated in Fig. 3, the test data are obtained in terms of pressure measurement by four sets of scanivalves. The measured data are 1) pressure distribution on the airfoil at 58 pressure taps lined up along the center line of the model, 2) wake pressure distribution by the wake traverse system located also at the center line and 1.5 chord length downstream from the

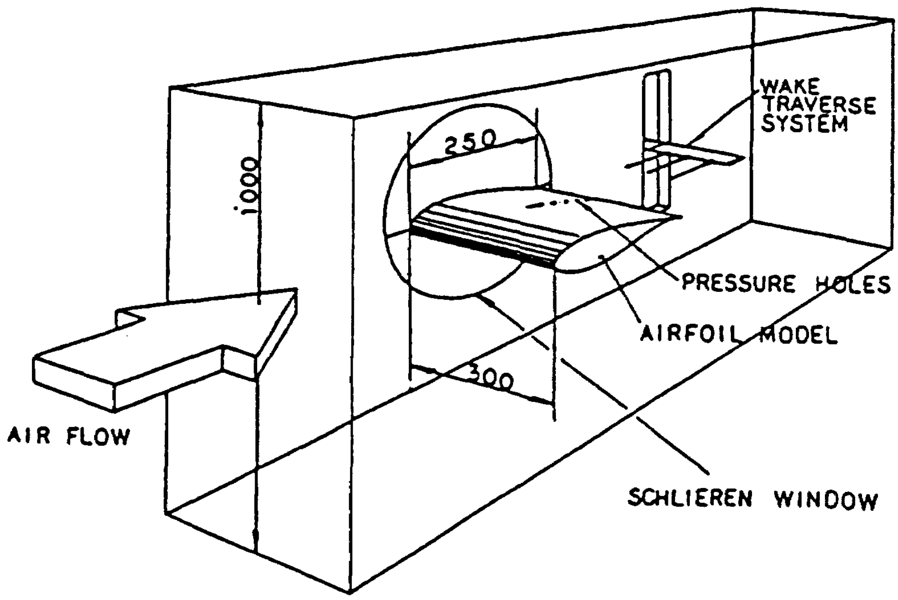
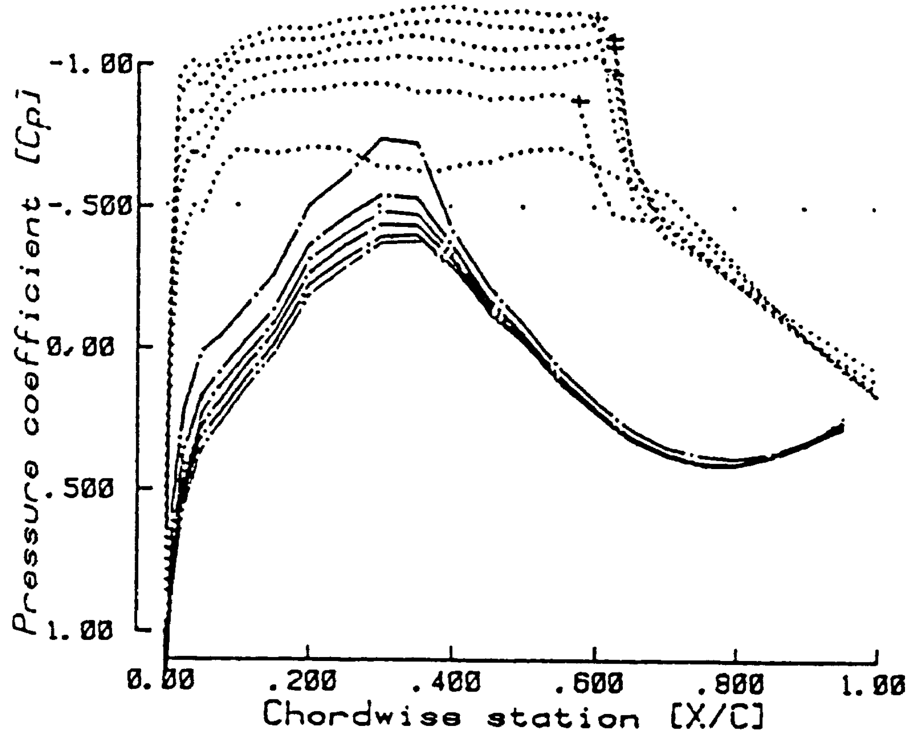
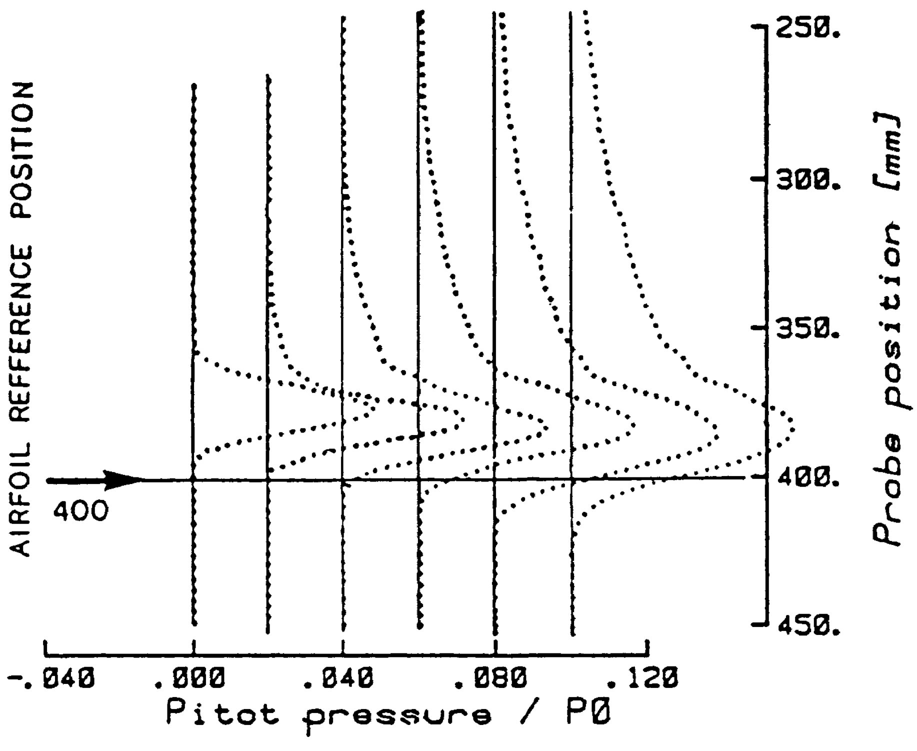


Fig. 3. Schematics of wind tunnel test.



(a) Pressure distribution. Dotted line for upper surface and solid line for lower surface.



(b) Wake profile. Dotted line for wake profile and solid line for reference.

Fig. 4. Typical outputs of wind tunnel test

(Re = 23 x 10^6, M = 0.775, alpha = 0.57 - 3.19°).

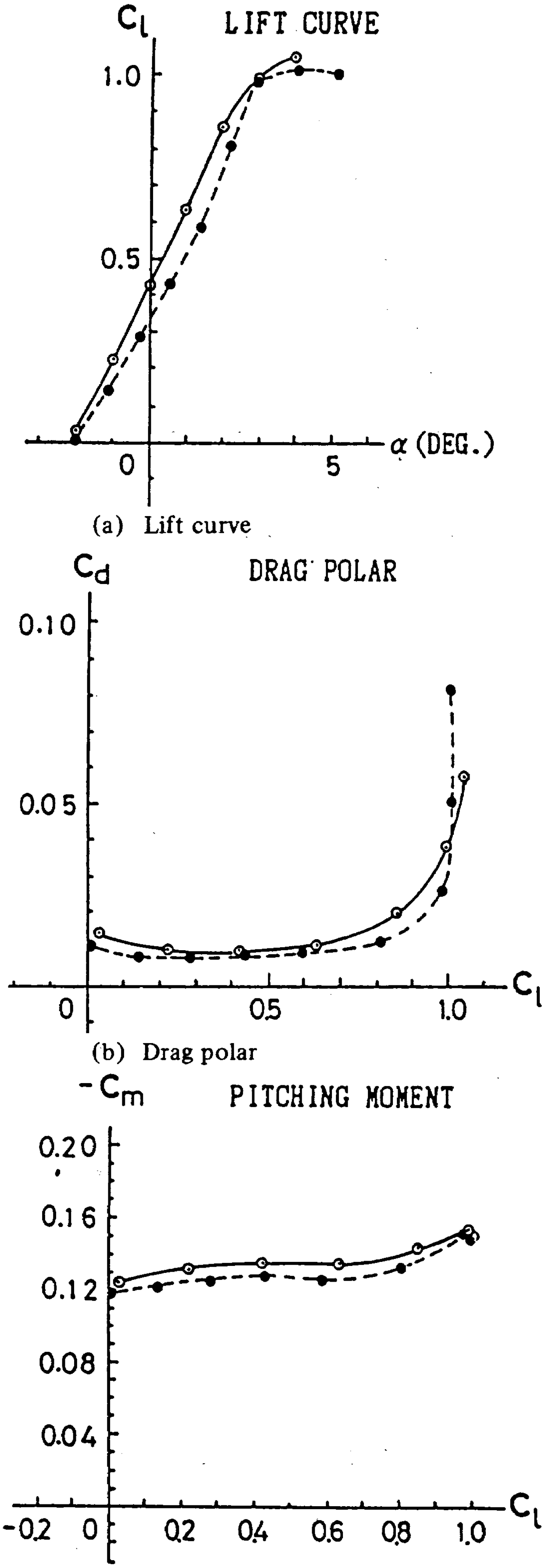
trailing edge. Also color Schlieren photographs are taken through the glass windows of both side walls. Lift, drag and pitching moment, are then obtained by integrating the corresponding pressure distribution. These data are acquired and processed by the dedicated on-line computer system, and the final results are prepared in the form of printed and plotted data soon after a blow is finished. Fig. 4 shows a set of typical outputs of the wind tunnel test, which include pressure distribution and wake profile.

Since the height of the tunnel and the model chord length are the ratio of 4 to 1 which presents some wall interference effects, a series of careful study for the interference are conducted by Sawada, et al.[19] Their research work has established a reliable wall correction formula, and it has been applied to all of the test data obtained in this wind tunnel.

COMPARATIVE STUDY

FORCE DATA

The comparison of lift curve is shown in Fig. 5(a) and Fig. 6(a) for flow Mach number 0.75 and 0.775 and the Reynolds number 23×10^6 respectively. The nonlinear characteristics of the transonic airfoil is well predicted by the computation. The value of $C_{\ell\,max}$ is almost identical between the two simulations. This agreement would be more than enough for the practical application to airfoil design, and it certainly is the property which the conventional inviscid codes have never predicted before. Our experiences show that as long as it is applied to the linear region, a transonic full potential code with boundary layer correction gives fairly good agreement to both Navier-Stokes computation and the experiments, although it never predicts the $C_{\ell\,max}$ point and simply extrapolates the linear prediction into large angle of attack regions. From the practical application point of view, it does not make any sense to use the Navier-Stokes computation to analyse these linear characteristics which the less expensive inviscid codes are proven to be accurate enough. The nonlinear region is the place where the

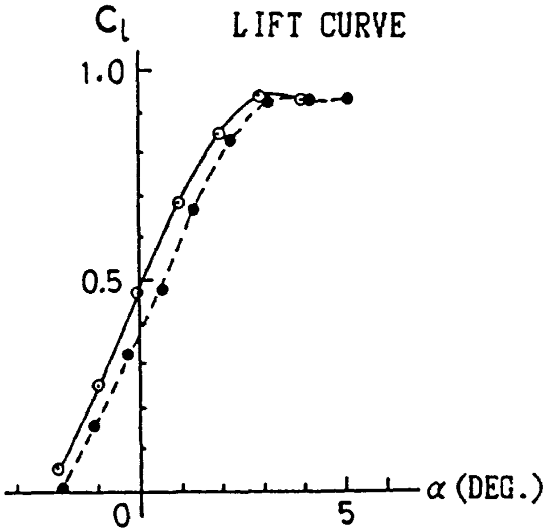


(a) Lift curve

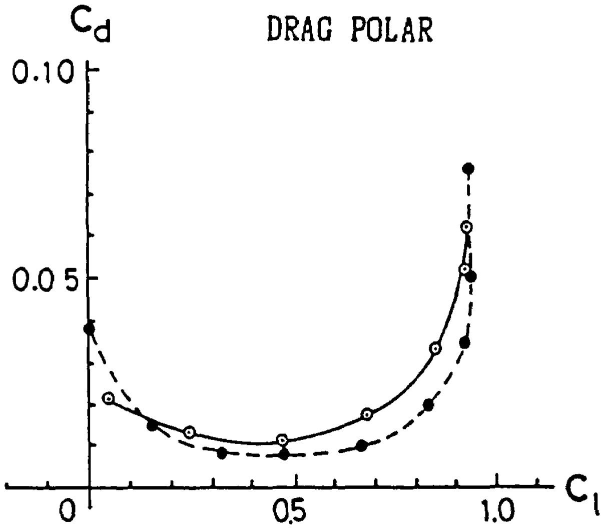
(b) Drag polar

(c) Pitching moment

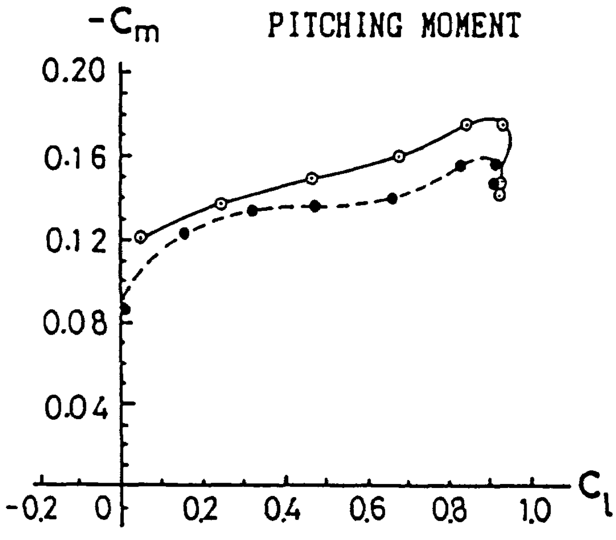
Fig. 5. Aerodynamic force comparison ($Re = 23 \times 10^6$, $M = 0.75$). Open circles for NSFOIL computation and filled circles for experiments.



(a) Lift curve



(b) Drag polar



(c) Pitching mement

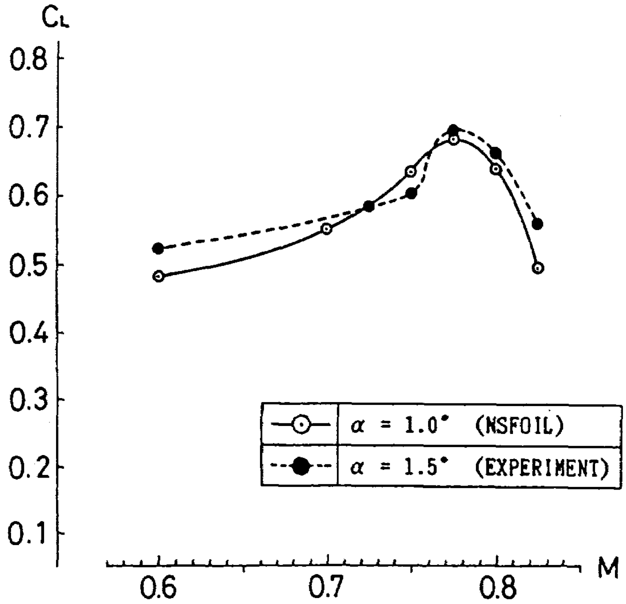
Fig. 6. Aerodynamic force comparison (Re = 23×10^6, M = 0.775).
Open circles for NSFOIL computation and filled circles for experiments.

Navier-Stokes code displays its powerful capability. In the range of our investigation, this ability does not seem to be affected either by Mach number change or by Reynolds number change.

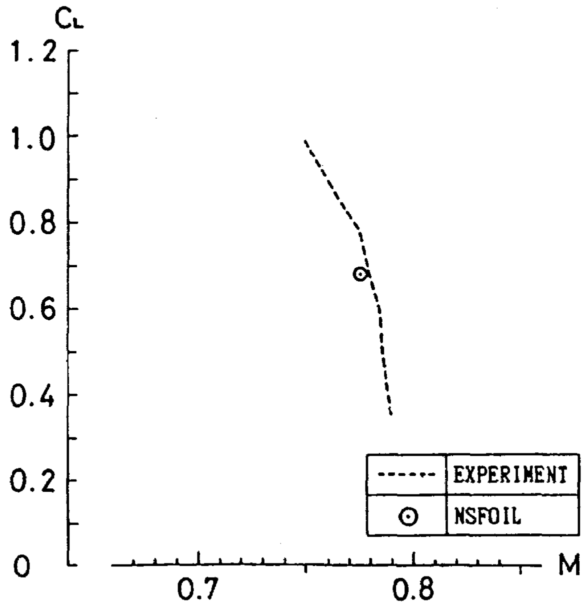
The drag polar curves are compared in Fig. 5(b) and Fig. 6(b). The nonlinear phenonenon is again well predicted at the higher lift region, where the polar curve demonstrates so called "polar break" and deviates from the theoretical drag curve for attached flow. The Navier-Stokes computation shows reasonable accuracy in predicting the drag coefficient of the airfoil with large separation, whereas the conventional inviscid codes fails to predict the polar break and simply overestimate the drag characteristics. It apparently is extremely useful in evaluating drag characteristics of the airfoil at higher lift coefficient. Unfortunately, it can be also pointed out that the computed minimum drag coefficient does not really agree with the wind tunnel test data. Since the skin friction dominates the drag at this regin, the accuracy of viscous simulation is responsible for this discrepancy. The detailed discussion is presented later at the section of pressure data comparison.

Fig. 5(c) and Fig. 6(c) show the pitching moment comparison. The Navier-Stokes computation predicts the pitch-up movement at the higher lift region, where the shock induced separation gives some lift losses in the rear loading of this advanced technology airfoil and results the pitch-up tendency. The conventional inviscid codes could never predict this transonic nonlinear characteristics as they simply keep obtaining strong rear shocks which present the increase of negative pitching moment. The level of the pitching moment coefficient is also reasonably well predicted and the computation would be very useful in evaluating pitching moment characteristics of airfoil section.

The lift divergence characteristics is illustrated in Fig. 7(a) and Fig. 7(b) for comparison. The NSFOIL computation predicts the lift divergence Mach number with fairly good accuracy. It is well known that the popular inviscid codes are able to calculate accurate drag divergence Mach number and have contributed quite large to airfoil design and analysis, although they are not capable of computing lift divergence which is basically shock induced separation and stall.



(a) Lift divergence curve.
Solid line for NSFOIL computation. (alpha = 1.0 for computation and alpha = 1.5 for experiment)



(b) Lift divergence boundary.
Open circle for NSFOIL computation

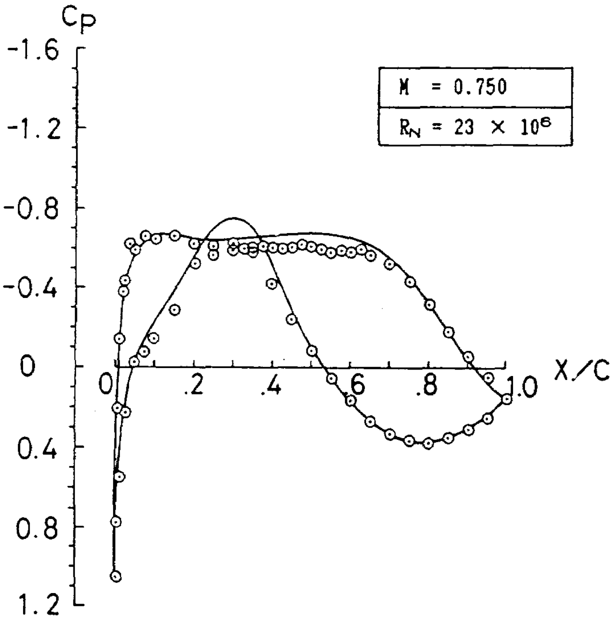
Fig. 7. Aerodynamic force comparison (Re = 23×10^6)
Dashed line for experiments.

PRESSURE DATA

Pressure distribution on the airfoil is compared in Fig. 8, where cases are chosen such that the pressure distribution of the same condition would be compared. Fairly good agreement is observed in Fig. 8(a) for a subcritical condition. Fig. 8(b) shows reasonable agreement of the two simulations at the drag divergence point. The Navier-Stokes computation gives satisfactory agreement in every aspect of upper and lower surface pressure level, shock location and trailing edge pressure.

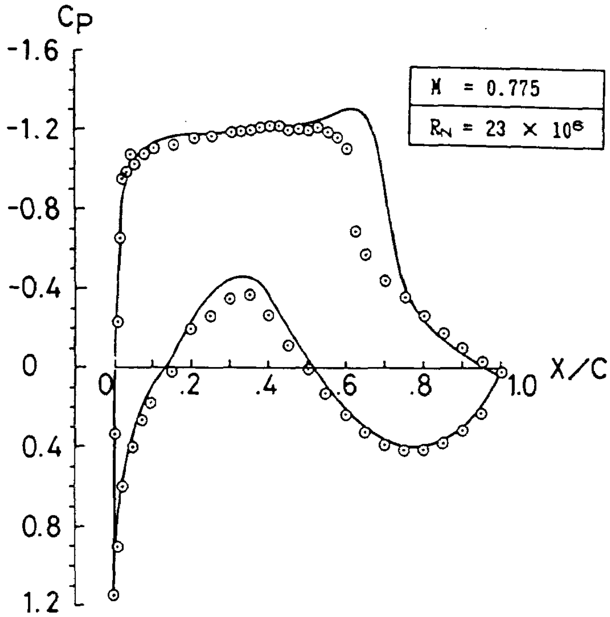
Fig. 8(c) illustrates the comparison just above the lift divergence point and at the buffet onset point. The agreement is still quite good especially at the trailing edge, and so is the pressure level of upper and lower surface. NSFOIL computation predicts shock location closer to the trailing edge compared to the physical simulation, but the difference in lift coefficient should account some part of the discrepancy. That is, if the lift coefficient of the wind tunnel test data increases by 0.02, then the shock location should come closer to achieve better agreement. The rather coarse mesh near the shock location, of course, has to be improved for the even better simulation. It is under progress to have the grid clustered at the shock position, possibly using the solution adaptive technique introduced by Deiwert and Nakahashi.[20]

The pressure comparison at further above the buffet onset point is presented in Fig. 8(d). The Navier-Stokes computation seems to fail in simulating the physical flow field at this point, and this looks like the limit of the application



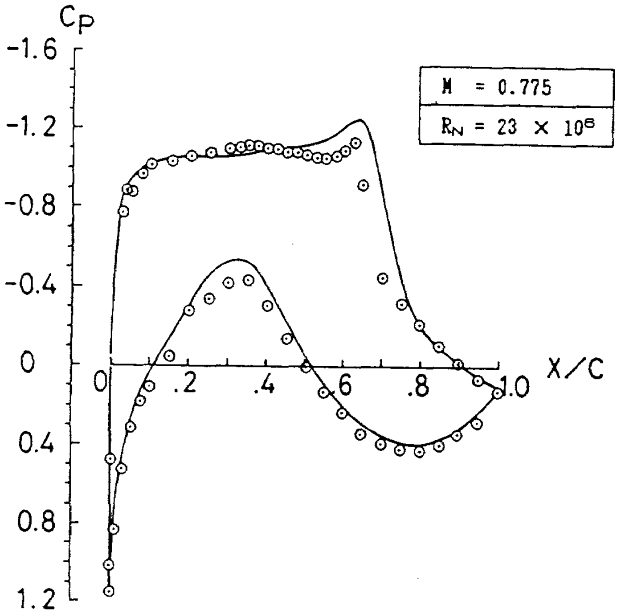
SYM.	DATA	α°	C_L	T_N
——	NSFOIL	0.0	0.423	M7500
⊙	EXP.(NAL 2D)	0.56	0.435	2365/2

(a) M = 0.75, alpha = 0.0° (comp.), 0.56° (exp.)



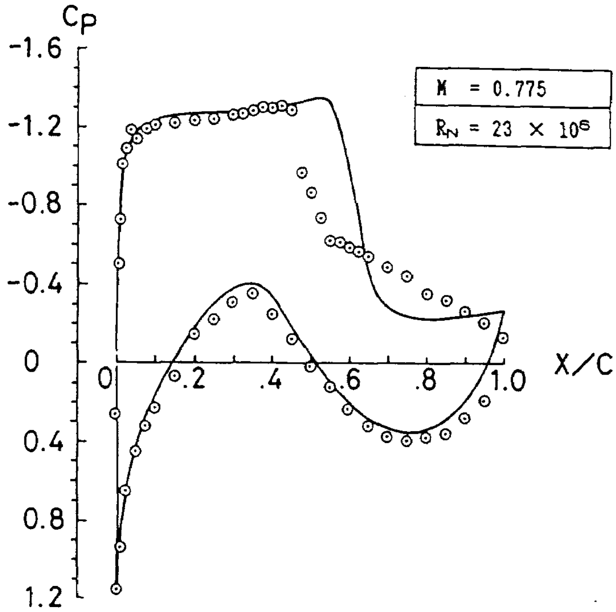
SYM.	DATA	α°	C_l	T_N
——	NSFOIL	3.0	0.939	M77531
⊙	EXP.(NAL 2D)	3.21	0.921	2370/1

(c) M = 0.775, alpha = 3.0° (comp.), 3.21° (exp.)



SYM.	DATA	α°	C_l	T_N
——	NSFOIL	2.0	0.850	M77521
⊙	EXP.(NAL 2D)	2.28	0.831	2369/3

(b) M = 0.775, alpha = 2.0° (comp.), 2.28° (exp.)



SYM.	DATA	α°	C_l	T_N
——	NSFOIL	4.0	0.929	M77541
⊙	EXP.(NAL 2D)	4.19	0.926	2370/2

(d) M = 0.775, alpha = 4.0° (comp.), 4.19° (exp.)

Fig. 8. Pressure distribution on airfoil (Re = 23 × 10^6).

Solid lines for NSFOIL computation and circles for experiments.

of this computing method. The grid clustering would be still worth trying for this large separation flow field. It turned out to be, however, that this is not the limit of the NSFOIL simulation, after a series of additional wind tunnel tests in which unsteady pressure distribution was measured on the airfoil model. This will be further detailed in the following section of unsteady data comparison.

Another pressure comparison is the one for wake profile in Fig. 9. As described in the section of wind tunnel testing, the wake traverse system is located about 1.5 chord length downstream, where the corresponding numerical result is compared in total pressure coefficient. The global scale of the wake is well predicted by the computation, especially the lower side of the wake. The difference at the center of the wake is supposedly due to the turbulence model switching problem. The model is switched from the wall type to the wake type immediately after the trailing edge, though it takes several grid points by relaxation process for numerical stability. There is a good change that this switching causes the overprediction of skin friction effect at the wake core. This leads to another possible future enhancement of the NSFOIL code.

The last pressure comparison is presented in Fig. 10, where the trailing edge pressure is plotted with respect to lift coefficient. Although there is slight discrepancy at the pressure level in the lower lift range, the pressure divergence point is predicted almost exactly by the numerical simulation. This is nothing but the result of the accurate computational simulation of shock induced separation flow. This trailing edge pressure divergence is known to be corresponding to the buffet onset point of an airfoil, which is found in Fig. 11. One of the most important off-design characteristics is now to be predicted by the computational method, which has never been possible by any inviscid code. This should enable aerodynamic engineers to evaluate buffet onset characteristics accurately enough without expending any wind tunnel test.

UNSTEADY DATA

The numerical result of the last figure in the pressure comparisons (Fig. 8(d)) was not in

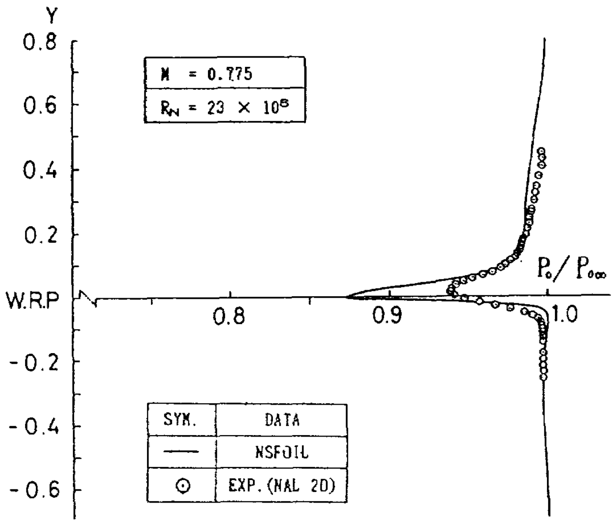


Fig. 9. Wake profile comparison (Re = 23 × 10^6, M = 0.775, alpha = 3.0°). Solid line for NSFOIL computation and circles for experiments.

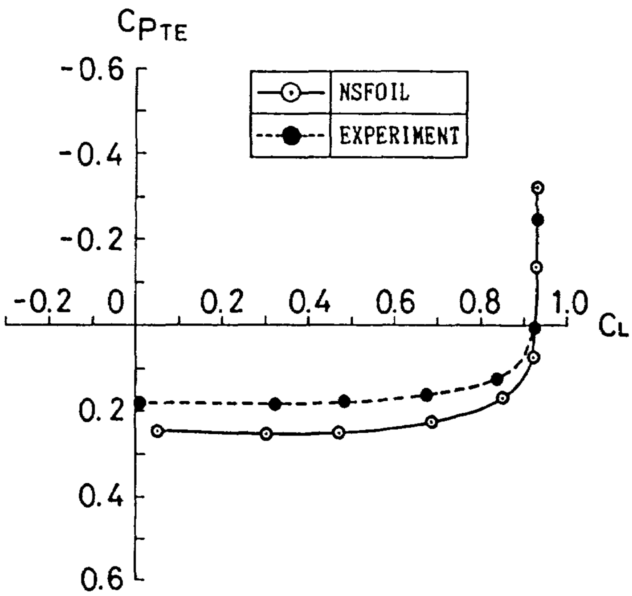


Fig. 10. Trailing edge pressure comparison (Re = 23 × 10^6, M = 0.775). Open circles for NSFOIL computation and filled circles for experiments.

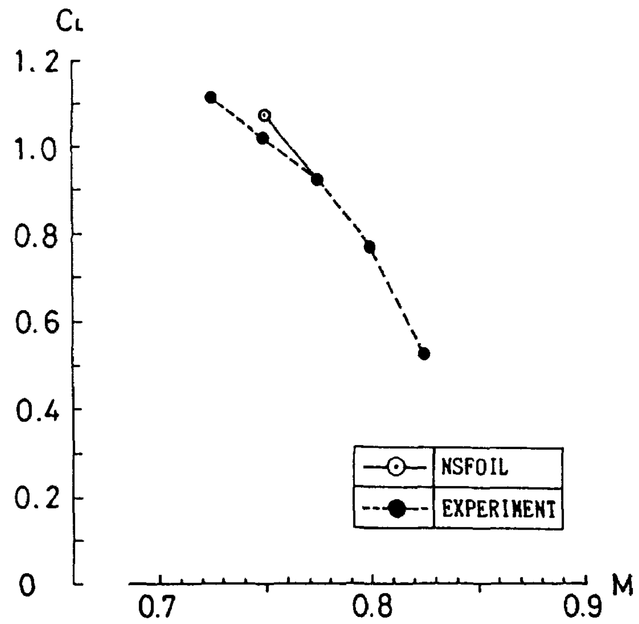


Fig. 11. Buffet boundary comparison (Re = 23×10^6).
Open circles for NSFOIL computation and filled circles for experiments.

fact a steady state solution. After more than 6,000 iterations, the numerical simulation has reached an oscillatory result, without going into either divergence or convergence. Fig. 12 shows the whole cycle of the oscillatory motion in terms of pressure distribution, where the upper surface shock is moving back and forth interacting with the separation thereafter. Here the question arises if this time dependent solution is actually the simulation of physical flow field or just a numerical oscillation, considering the fact that this difference formulation is first order accurate in time for the governing equation.

A series of additional wind tunnel tests are then conducted in order to find out the answer to this question. The instrumentation of test is exactly the same as described before, except twelve of unsteady pressure sensors are embedded near the center line of the airfoil model for the purpose of acquiring time dependent pressure data on the upper surface. The experimental results are still under careful reduction and analysis, although a typical result is presented in Fig. 13, where the test condition is identical to the case of numerical simulation. The energy power spectrum measured at 45 per cent chord station on the upper surface where is upsteam of the oscillating shockwave shows a distinct peak at about 60 Hz in frequency, which is not far from the frequency of 45 Hz NSFOIL has predicted.

At this stage, it is still too early to say that NSFOIL is capable of simulating even transonic unsteady flow field of strong shock and boundary layer interaction, even though the frequency is predicted. But it is also too hasty to call the heavy buffet of Fig. 8(d) is beyond the validity of NSFOIL computation.

CONCLUSION

NSFOIL, a Navier-Stokes computer code for two-dimensional transonic viscous flow, was validated by the detailed comparisons with the experimental data of NAL High Reynolds Number Two-Dimensional Transonic Wind Tunnel.

The numerical simulation of the flow around a practical transonic airfoil reasonably agrees to the physical simulation, and the validation work shows the code is a very effective tool for viscous flow analysis of advanced technology airfoils. It is capable of predicting nonlinear characteristics accurately, such as buffet boundary and lift divergence boundary. It has never been possible by any inviscid computer code.

Some future improvements of the code, however, are still required in order to get even better numerical simulation. The first one would be the grid clustering or possibly the grid adaptation for the better resolution of the shock wave and the flow separation. Another improvement would be the turbulence model switching

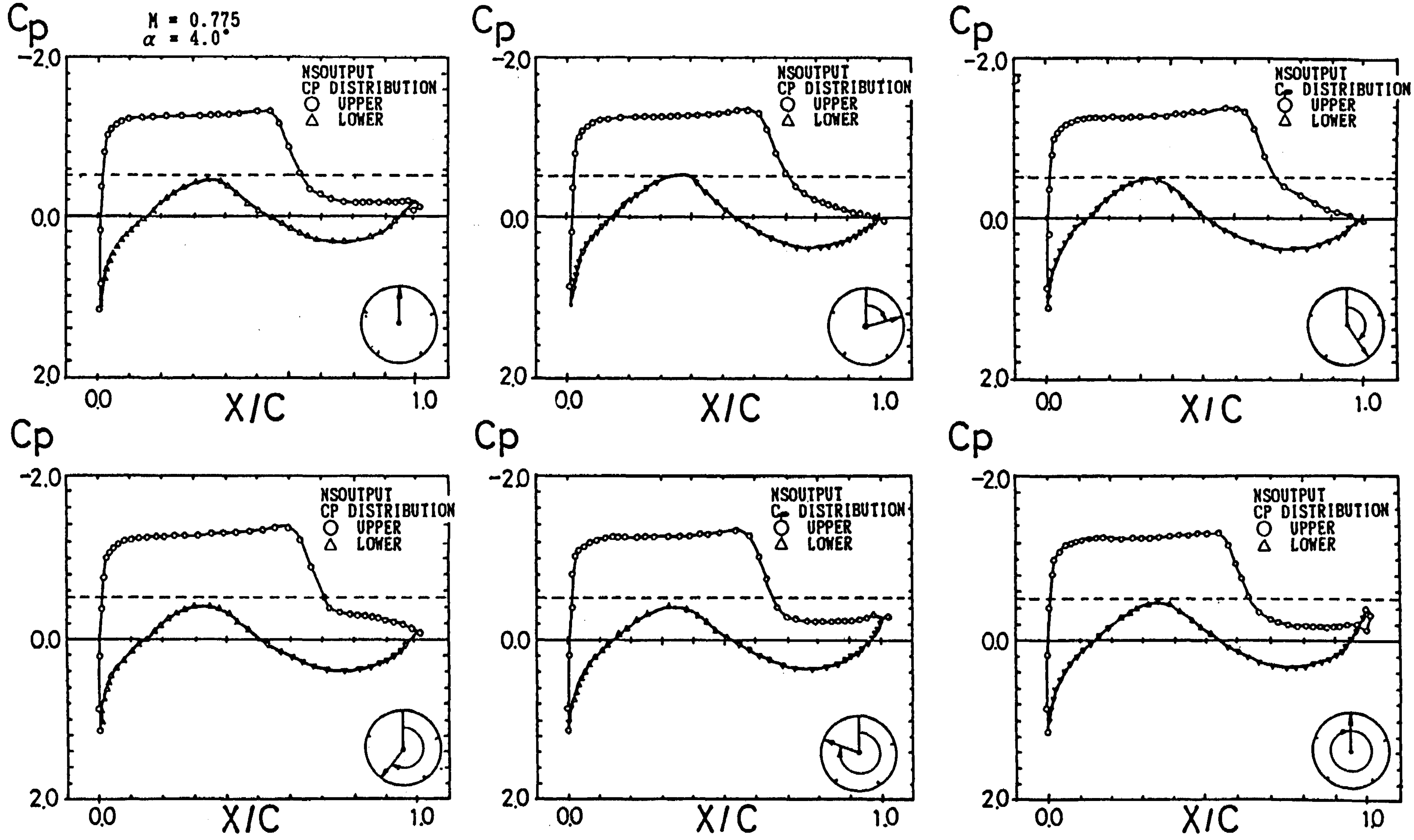


Fig. 12. Numerical results of oscillatory motion – a cycle in pressure distribution (Re = 23 × 10⁶, M = 0.775, alpha = 4.0°).

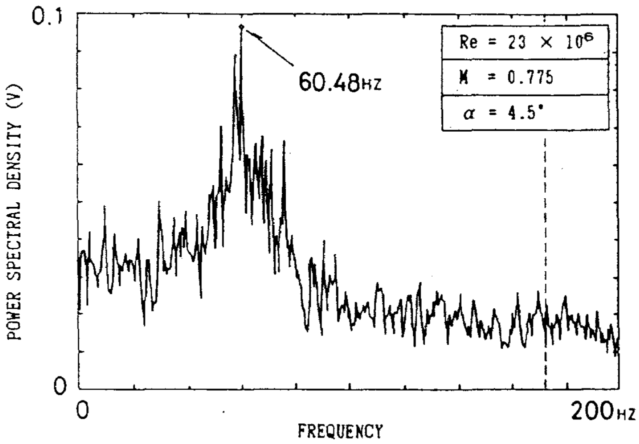


Fig. 13. Experimental result of oscillatory motion – power spectrum density (Re = 23×10^6, M = 0.775, alpha = 4.5°).

right after the trailing edge in order to obtain more accuracy in drag estimation.

The series of Navier-Stokes code validation is going to continue as the collaborative research work of NAL and MHI. The next subject will be the detailed unsteady data comparison.

The present work is a part of the collaborative joint researches on computational aerodynamics between National Aerospace Laboratory and Mitsubishi Heavy Industries, Ltd. starting from 1983. The author express their thanks to Mr. Kazuaki Takashima and the members of Two-Dimensional Transonic Wind Tunnel Laboratory for providing wind tunnel testings and advising on the evaluation of test data. The authors are also thankful to Mr. Hajime Miyoshi and the members of the Computer Center for providing special computing conveniences, and to Dr. Nobuhiko Kamiya for coordinating this joint researches. A part of present work was presented on AIAA 3rd Applied Aerodynamics Conference, Colorado Springs, Colorado, U.S.A., October, 1985.

REFERENCES

1) Chapman, D.R., "Computational Aerodynamics Development and Outlook," AIAA Paper 79-0129, Jan., 1979.

2) Kutler, P., "A Perspective of Theoretical and Applied Computational Fluid Dynamics," AIAA Paper 83-0037, Jan., 1983.

3) Korkegi, R.H., "The Impact of CFD on Development Test Facilities – National Research Council Projection," AIAA Paper 83-1764, July, 1983.

4) Deiwert, G.S., "Numerical Simulation of High Reynolds Number Transonic Flows," AIAA Jour., Vol.13, No.10, Oct., 1975, pp.1354–1359.

5) Steger, J.L., "Implicit Finite Difference Simulation of Flows About Arbitrary Geometries with Application to Airfoils," AIAA Paper 77-665, June, 1977.

6) Beam, R.M. and Warming, R.F., "An Implicit Finite Difference Algorithm for Hyperbolic Systems in Conservative Law Form," J. of Computational Physics, Vol.22, 1976, pp.87–110.

7) Steger, J.L. and Bailey, H.E., "Calculation of Transonic Aileron Buzz," AIAA Jour., Vol.18, No.3, Mar., 1980, pp.1409–1410.

8) Deiwert, G.S., "Numerical Simulation of Three-Dimensional Boattail Afterbody Flowfields," AIAA Jour., Vol.19, No.5, May, 1981, pp.582–588.

9) Levy, L.L. Jr., "Experimental and Computational Steady and Unsteady Transonic Flows About a Thick Airfoil," AIAA Jour., Vol.16, No.6, June 1978, pp.564–572.

10) Liou, M. and Coakley, T.J., "Numerical Simulation of Unsteady Transonic Flow in Diffusers," AIAA Jour., Vol.20, No.8, Aug., 1984, pp.1139–1145.

11) Anderson, W.K., Thomas, J.L. and Rumsey, C.L., "Application of Thin-Layer Navier-Stokes Equation Near Maximum Lift," AIAA Paper 84-0049, Jan., 1984.

12) Barth, T.J., Pulliam, T.H. and Buning, P.G., "Navier-Stokes Computation for Ex-

otic Airfoils," AIAA Paper 85-0109, Jan., 1985.

13) Shang, J.S., "An Assessment of Numerical Solution of the Compressible Navier-Stokes Equations," AIAA Paper 84-1549, June 1984.

14) Kawai, N. and Hirose, N., "Development of the code NSFOIL for analysing High Reynolds Number Transonic Flow Around an Airfoil," NAL TR-816, 1984.

15) Thompson, J.F., "Automatic Numerical Generation of Body-Fitted Curvilinear Coordinate System for Field Containing Any Number of Arbitrary Two-Dimensional Bodies," Jour. of Computational Physics, Vol.15, No.3, 1974, pp.229–319.

16) Takashima, K., "Experimental Works in the NAL High Reynolds Number Two-Dimensional Wind Tunnel on Advanced Technology and NACA Airfoil," ICAS Paper 82-5.4.4., 1982.

17) Baldwin, B.S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan., 1978.

18) Hirose, N., Kawai, N., Isawa, T. and Kikuchi, M., "Development of Grid Generator Code AFMESH for Transonic Airfoil Analysis Codes," Proc. 13th Ann. Meeting, Jap. Soc. Aero. & Space Sci., 1982, pp.158–161.

19) Sawada, H., Sakakibara, S., Sato, S. and Kanda, H., "Wall Interference Estimation of the NAL's Two-Dimensional Wind Tunnel," NAL TR-829, Aug., 1984.

20) Deiwert, G.S., Andrews, A.E. and Nakahashi, K., "Theoretical Analysis of Aircraft Afterbody Flow," AIAA Paper 84-1524, June 1984.

TECHNICAL REPORT OF NATIONAL AEROSPACE LABORATORY TR-911T

航空宇宙技術研究所報告911T号(欧文)

昭和61年9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 武蔵野三鷹(0422)47-5911(大代表)

印刷所　株式会社 実業公報社
東京都千代田区九段南4-2-12

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

Printed in Japan