A98-31596

ICAS-98-4,6,1

A METHOD FOR THE RAPID PREDICTION OF UNSTEADY LOADS OVER WINGS AT TRANSONIC SPEEDS

by

David Nixon

NWING Inc.

2425 Park Blvd.: Suite B102

Palo Alto: CA 94306

USA

Abstract

Aeroelastic effects, including flutter, at transonic speeds are often predicted using linear theory rather than a nonlinear cfd method. The reasons for this include lack of confidence in the results of nonlinear cfd methods, the excessive set up time, and a long computing time. The method discussed here attempts to address some of these issues by using a new variant of strip theory. Preliminary results are encouraging.

Introduction

Some of the most critical aerodynamic limitations to performance of a modern commercial aircraft are consequences of unsteady, rather than steady, flow. For example, the demand for flutter avoidance or active alleviation of flutter can determine the structural weight of the wing; flutter is a consequence of unsteady flow. Another demand is the avoidance of shock induced buffet, again a phenomenon associated with unsteady flow. The most critical speed regime for flutter is often the transonic regime and shock induced buffet also occurs at transonic speeds. Consequently the accurate prediction of unsteady transonic flows is essential if modern aircraft flying at transonic speeds are to be designed efficiently. Although research into unsteady transonic aerodynamics has been in progress for nearly 30 years, it is suprising that existing prediction methods ,such as those described in Ref.[1]&[2],are often not used in the aerospace industry. The linear doublet lattice theory(3) is frequently used for all speed regimes, including transonic.

The prediction methods that are available in the USA range from two dimensional small disturbance models, such as XTRAN2L(4), to three dimensional small disturbance models, such as CAPTSD(1), to methods, such as ENSAERO(2), that solve the Euler and Navier Stokes equations. Some of these, for example CAPTSD, have been in development for over a decade but still do not see wide use in industry. There are several rational reasons for this lack of use and some of these reasons are outlined below. Before proceeding in this direction, it is helpful to point out a few realities about aeroelastic calculations.

In any single design a considerable number of aeroelastic calculations must be done for flutter clearance. For a simple general aviation aircraft this number can be 10000+(5), while for military aircraft, with various combinations of store carriage, the number can be ten times this value. Hence any prediction method must be fast enough to be able to do this number of calculations in a reasonable time. A second point is that aeroelasticians are not CFD practitioners and should not have to understand the arcane rules of this art in order to do their job. A third point is that the aerodynamic loads should be in a form that allows the use of standard aeroelastic analyses to avoid the cost of unnecessary learning curves.

Practically all methods for predicting unsteady transonic aerodynamics are based on finite difference approaches and these have generic problems. One severe problem is that the set up time to generate the grid for a complex geometry can be several weeks(6). Another problem is that the accuracy can vary between the use of one computer code and another, even though they both solve the same set of equations. In fact the accuracy can depend on the skill of the user. It should be pointed out that transonic aerodynamics is very sensitive to errors in algorithms or formulation and this can cause errors in the solutions. An error specific to unsteady aerodynamics is that the propagation of waves from the body is difficult to model since there can be wave reflections off the grid cells, especially if they vary in size, thus contaminating the solution. Reflections of the waves off the outer computational boundary also contaminate the solution. There have been many attempts to formulate "nonreflecting" boundary conditions at the outer boundary but none of these seems to be adequate for a very wide range of test cases.

Unsteady aerodynamics generally consists of several different phenomena, such as the inviscid flow, shock waves and the viscous boundary layer, having an unsteady motion each with its own characteristic time scales. Each of these phenomena interact with the others. The boundary layer can interact with the shock wave causing quite complex time lags in the shock motion. The shock motion creates a pressure step that moves over the wing causing significant changes in the loads and thus the shock motion must be properly estimated. The physical model must (for attached flow) represent the inviscid wave propagation, the shock motion, and the unsteady boundary layer, including shock boundary layer interaction.

For a comprehensive CFD model a body fitted grid is generally used and this must be re-computed for each time step. This can use up valuable computer resources and one way to avoid this is to use a small disturbance model in which the boundary conditions are applied on a fixed boundary and do not require a moving grid. This can give rise to errors. It can be shown[7] that for an infinite swept wing the following relation must hold for potential flow.

$$v=\tan(\Lambda)(1-u) \qquad [1]$$

where Λ is the sweep angle, v is the velocity normal to the freestream direction in a spanwise sense and u is the velocity in the freestream direction. This formula can be used as a guide to see if a small disturbance model is accurate. In Fig.[1] the spanwise velocity, v, over a swept wing of high aspect ratio, for steady flow, computed exactly and by using the formula Eq.[1] is shown for CAPv12[1] (a specific version of the small disturbance code CAPTSD) and FLO22[8], a full potential code. It can be seen that the comparison between the calculated value and the formula is very good for FLO22 but not really that good for CAPv12, indicating that CAPv12 (or any conventional small disturbance method) cannot represent the flow over an infinite swept wing properly unless specific actions are taken. This is not a suprising result since a small disturbance method has the greatest error when U is close to zero (stagnation region) which is where the three dimensional effect, characterized by V, is largest.

Practically all numerical algorithms used to compute transonic flow have a shock capturing scheme. While the extremities of this region are physically realistic, the solution in the interior of the capture is non physical[9]. The extent and nature of this non physical region can change with time and hence a time marching solution will have a variable error in this region. This error can produce changes[10] in the flutter speed comparable to those generated by the effect of viscosity.

There are two approaches to incorporating unsteady transonic aerodynamics into an aeroelastic calculation, namely the time domain approach and the frequency domain approach. In the time domain approach the aerodynamic calculation is coupled with the structural model and the combined calculation, and, for a flutter calculation, the system is essentially "flown" until a flutter point is found. Because of the nonlinearity of at least one of the equation sets, the aerodynamic calculation lags the application of the structural calculation by a time step (or vice versa). The effect of this time lag is not necessarily trivial [11]. There is also the issue of actually finding the (singular) flutter point.

Frequency domain methods solve a linearized version of the nonlinear equations governing transonic flow; it is this linearization that allows the necessary harmonic decomposition to develop a frequency domain method. Frequency domain methods compute the aerodynamic loads in a form that is compatible with subsonic aeroelastic calculations but some of these methods do not represent the shock motion correctly in the linearization, if at all. It is noted earlier that it is essential to get the shock motion correctly.

For a time domain calculation, the computer time to obtain a single aerodynamic result can be excessive. In Ref.[12] some computational run times for ENSAERO[2] are given. For a relatively simple wing one calculation for one Mach number, one geometry change and one reduced frequency, takes over 1500 hours on a SGI workstation; about four runs are necessary to get the flutter point for one set of flow conditions and one frequency. This is unrealistic for routine flutter calculations.

One of the problems with time domain methods is the difficulty of getting aerodynamic influence coefficients (AIC), the aerodynamic load at all stations of the wing due to a change in geometry on a specified part of the wing. This can require a very considerable amount of computing power to construct the AIC if the number of geometry changes is large.

Viscous effects can be important in the prediction of unsteady transonic flow, in particular, the modeling of the shock-boundary layer interaction must be accurate. Unfortunately this is an area that is not completely understood. Part of the problem is the turbulence model used in the calculations; shock turbulence interaction is not well understood[13]. However, for flutter calculations all that is required is an accurate estimate of the effect of the boundary layer on the mean position of the shock and the effect of the unsteady boundary layer on the magnitude and time lag of the shock motion. Details such as prediction of the viscous drag are not usually required for aeroelastic purposes.

Shock induced buffet is still difficult to predict over modern supercritical wings. This is because the fundamental cause is not well understood. It is shown in Ref.[14] that a simple model of shock induced separation and reattachment, with phase lags, will set up a limit cycle akin to buffet. However, the most common approach in the USA is to use unsteady transonic flow prediction methods to try to map out a buffet boundary. This is an expensive undertaking and suffers from the inaccuracies in turbulence models noted above. It is possible that simpler methods can be developed if only the buffet *boundary*, rather than the finer details, are required.

In summary, in spite of the considerable effort in trying to develop prediction methods for unsteady transonic aerodynamics, it has still not reached the stage where such methods are used in aeroelastic calculations in a routine way. The reasons outlined above can be condensed into the following.

(a)The methods are too difficult to set up, especially for complex geometry's.

(b)The computing time is too long.

(c)The output is often not in the form required by traditional aeroelastic analyses.

(d)The accuracy of the calculation is often not good enough.

The approach described below attempts to address some, if not all, of these issues.

Approach

The approach is based on the premise that a modern CAD system can generate the surface paneling require for a panel method and that accurate panel methods are available. The operation from CAD to steady aerodynamic loads via the panel method can be made seamless; in other words, the user need not provide any input. Any change in geometry will lead to an estimation of the change in loads automatically. Unsteady aerodynamic loads can be computed using modifications of the panel method (for example that described in Ref.[15]) or doublet lattice[3] methods. The idea is to extend this capability to include steady and unsteady transonic flow. This method of approach addresses requirement (a) above. The computing time is reduced by using a variation of strip theory that is described briefly below. This addresses requirement (b).

The results given at the end of the paper for unsteady flow use a time domain method but current research is directed towards the development of a frequency domain method which should address requirements (c) and (d).

Strip Theory

Recent research[16] has indicated that for typical aircraft designs the three dimensionality of the airflow is adequately represented by linear theory, either incompressible theory with added compressibility corrections[17] or solutions of the Prandtl Glauert equation. Both of these equation sets can be solved using a panel method which is relatively easy to set up and run. The prediction method evaluated here makes use of this fact to develop a means of estimating the steady transonic flow around an aircraft using only two dimensional CFD methods and a panel method to represent the three dimensional effects. If a doublet lattice method is also used, then unsteady flows can be computed. Because of the "field nature" of transonic flow, data for the flow field is required, in addition to the surface data. This additional data is easily computed from a panel code. In some respects the present method is similar to a classic "strip theory" approach but it is based on an entirely different concept. The present method is also applicable to complex geometry because the panel code represents this accurately. If the input to the panel code is derived from a CAD program, then the geometry and the loads can then be generated directly from instructions to the CAD program.

In essence, the complex geometry is modeled by the panel method and the nonlinear effects, such as shock waves, are predicted by the CFD code. The CFD code can be a potential flow, Euler or (possibly) Navier Stokes solver. In the current work, a full potential equation solver is used. In principle, potential codes are adequate for most problems if "strong shock"[18] theories and boundary layer models are included. In practice "strong shock" theories can cause numerical problems and it is better to just use a boundary layer model. The boundary layer model weakens the shock sufficiently to avoid most of the problems associated with the excessively strong shocks associated with potential flow.

Strip theory approaches are not new and for many years the unsteady strip theory approach of Yates[19] was used to predict flutter. Recently Liu [20] described an alternative formulation. In spite of the effort in the development of strip theory the idea has not been entirely successful. The main reason for this is that the physics of three dimensional flow has not been fully understood. Classic strip theory is based on the idea that the flow effect due to the third dimension can be directly related to a change in geometry in the plane of the other two dimensions. This hypothesis has not been proven theoretically, although many results, some of them quite good, have been computed. In the present research a more recent understanding of the physics[16] is used to develop a prediction method that requires only a two dimensional CFD calculation. The approach is based on the finding that a sufficient degree of information about the three dimensionality of the flow is contained in the results from a panel method, either for incompressible flow or from a solution of the Prandtl Glauert equation. Because the method is based on strip theory, the transonic effect at any location can be predicted without predicting the flow at all stations on the vehicle as would be the case with conventional CFD. This allows easy diagnostics.

Computer Code(RAPTL)

As mentioned above, the computer code used in the research solves the full potential equation. For convenience, especially for unsteady flows, thin wing boundary conditions are used. Although it can be shown[7] that thin wing theory in its classic form may not adequately represent swept wing, the modified thin wing boundary conditions noted below do not have this disadvantage. To give a better representation of the flow, especially the shock location, a simple boundary layer model is incorporated.

Since one of the major goals of the present research is to produce a prediction method for unsteady flows, small disturbance boundary conditions are used. Small disturbance boundary conditions relate the actual boundary conditions on the body surface to equivalent boundary conditions on a mean surface. This removes the need for costly regridding during a calculation for unsteady flow. However, small disturbance boundary conditions in a finite difference algorithm are a source of uncertainty since the body or wing leading edge is usually placed between two grid points; introducing a fictitious leading edge radius. This problem can be alleviated by using an "analytic continuation" boundary condition[21] in which the conditions external to the wing or body are analytically continued inside the body to a mean surface to give a consistent small disturbance boundary condition. This is the approach used in the present research. Although small disturbance boundary conditions are used, a full potential formulation is used to solve the transonic problem. This does not necessarily mean that the disturbances from the freestream are small only that the direction of the dominant compressibility effects is not known in advance. The use of a full potential equation covers all possibilities.

The algorithm used is a variation of one reported by Shankar et al[22] for time accurate calculations. A simple Nash - McDonald (see Ref.[15] for details) boundary layer model is incorporated to represent viscous effects. This boundary layer model is accurate only for steady flow and is included mainly to locate the shock waves more accurately.

A grid of 80x61 is used. The grid points are clustered near the leading edge.

Compressibility Correction for Panel Code

The panel code used for most of the steady flow calculations is PMARC[15], which solves Laplaces equation. In the present method the solution of the Prandtl Glauert equation is required and this is obtained by using a post processor that converts an incompressible solution to a compressible solution without having to use the panel code again. The formulation is based on the infinite swept wing result reported by Kuchemann[23] with modifications[17] to account for finite wing effects.

Operation (Steady Flow)

The operation of the prediction method for steady flow is as follows.

(1)Use the panel code to compute the results for the flow over the configuration. Calculate the velocities in planes in the flow field as well as on the surfaces.

(2)Using the compressibility correction construct a "Prandtl-Glauert" solution.

(3)Using this Prandtl Glauert solution and RAPTL, compute the transonic flow at the desired stations. It is not necessary to solve for the entire flowfield if only one part of it is of interest.

Operation (Unsteady Flow)

The operation of the prediction method for unsteady flow is the same as for steady flow with the additional step of using the harmonic result from a solution of the unsteady Prandtl Glauert equation in step (3). At present this information is found from using CAPv12[1] in its "linear" form, but it is anticipated that a standard doublet lattice[3] code will be used in practice.

Validation Process

Because of the number of new features in the system a considerable amount of testing is required. These start with simple two dimensional cases to determine the accuracy of the boundary condition treatment and the suitability of the algorithm for unsteady flows and proceed through relatively simple swept wings with conventional sections to more realistic wings with supercritical sections. Simple wing body geometry's are also represented. Most of the three dimensional cases presented are for steady flow. In order to avoid the consistency problems with "tailoring" the code for different problems, the test cases were always run as a block with the same code and with no user initiated massaging. If one "bad" result occurred then any fix to the code must be capable of predicting accurately all of the other cases in the test matrix. This philosophy is applied also to the codes used for the validation; the "standard" grid is used for all cases.

Due to a lack of availability of certain computer codes results of the present method were compared to the results from the three dimensional code FLO22[8]. Although this uses a "nonconservative" algorithm in that it does not conserve mass across a shock, FLO22 is used widely in industry because of its speed and the fact that it generally gives good agreement with experimental data. However, it should be noted that it will give a weaker and further forward shock than a conservative algorithm. However, for a tapered wing the formal accuracy of this code is reduced at the wing tip. The other code used to generate results for comparison is CAPTSD[1] which uses a time accurate, conservative, algorithm that solves the transonic small disturbance equation. The code has the ability to represent approximately the effects of bodies. Its main usefulness in the present context is that it allows a self consistent solution of steady and unsteady transonic and linear subsonic governing equations. As pointed out earlier, small disturbance methods have accuracy problems for three dimensional cases. A point worth noting is that there can be a considerable difference in the results obtained from FLO22 and CAPTSD. CFD is not yet an exact art[6].

The present results are compared to results from both FLO22 and CAPTSD. If the present results are in general agreement then it is assumed that, *as a first step in validation*, the present results are accurate and that the basic concept is valid.

Test Cases

Two Dimensional Cases

The two dimensional cases are fairly conventional and are mostly NACA airfoil sections at a range of Mach numbers and angle of attack. A list of some of the cases is given in Table 1.

Wing Alone Cases

A number of wing alone cases were tested ranging from a 20 deg swept, untapered wing to wings with up to 37 deg. sweep and taper ratio of 0.33. The sections range from conventional sections to more modern supercritical sections. A list of the test cases is given in Table 2

Wing/Body Combinations

Two wing body combinations were tested, the RAE Wing/Body C[24] and the NACA Wing/Body RML51F07[25]. Both of these have bodies of revolution and conventional wing sections. The RAE Wing/Body has a cylindrical section at the wing junction. The NACA Wing/Body has a Sears-Haack body. Further details are given in Table 3. These cases are used only for calibration purposes as the 45 deg. swept wing is beyond a 37 deg. limit imposed on the theory by (temporary) computational constraints and the PMARC body model for the RAE Wing/Body case is not as accurate as it could be, and this makes the accuracy of the calculations close to the body difficult to determine..

Results

Two Dimensional Case

In Fig.[2] the pressure distribution around a RAE 2822 airfoil at a Mach number of 0.73 and angle of attack of 3.19 deg. is shown. The present results are for inviscid flow and much of the difference between the present results and the experimental data can be attributed to viscous effects. However, the "analytic continuation" boundary condition does lead to a lower level of suction on the lower surface, a trait that appears in all of the subsequent cases.

Higher Order Compressibility Correction

Since most of the results generated in the test matrix used PMARC[15] with a compressibility correction as a base, it is necessary to illustrate the accuracy of the method. In Fig.[3] the pressure distribution around the LANN wing at a Mach number of 0.655 and 0.62 deg. angle of attack is shown. These results were computed using a "higher order" correction[17] that can be applied as a post processor to the incompressible results generated by PMARC. The agreement between the present results and those generated from FLO22 is good.

Swept Wing

In Fig.[4] the pressure distribution around the ONERA M6 wing at M=0.84, and 3.06 deg. angle of attack is compared to results computed using CAPV12 and FLO22. It can be seen that there is a considerable variation in the results and that the present "strip theory" results are in the vicinity of the results from the more established methods. Similar results for some stations on the LANN wing are shown in Fig.[5]. In the ONERA M6 wing case the boundary layer model was used; in the case of the LANN wing the boundary layer model did not converge so the result presented is for inviscid flow.

A general trend in these results is that the suction on the lower surface is not as large in the present case as it should be. This is due to the error introduced by the boundary condition, similar to the two dimensional case, and the fact that the stagnation point in the panel code result is not at the same location as for the CFD code result. This can lead to considerable error and is the subject of ongoing investigation.

It should be noted that "better" results for each case can be obtained by "tuning" the prediction method but this is not a universal tuning and such results are not presented here.

Unsteady Cases

The unsteady cases used in this preliminary testing were the F5 wing[27] and the LANN wing[28].and the results are shown in Fig.[6] & [7]. For the F5 wing the results from CAPTSD[1], run in its linear mode, were used to generate the steady state result because PMARC would not converge for such a thin section. PMARC was used to generate the steady state results for the LANN wing. For both cases CAPTSD was used to generate the linear time dependent results. This was done purely for convenience; in the final operational method a doublet lattice code will be used. The conditions for the F5 wing are a Mach number of 0.9 and a reduced frequency of 0.275 based on root chord; the wing is pitching about 50% root chord. The result shown in Fig.[6] is at approximately 35% semi span. The conditions for the LANN wing are a Mach number of 0.82 and a reduced frequency of 0.205 based on root chord: the wing is pitching about 62% root chord. The result shown in Fig.[7] is at approximately 70% semi span. The results are calculated by running "RAPTL" in a time accurate mode and post processing the answers to give the real and imaginary parts of the pressures etc.. The results are shown in Fig.[6] and [7] using the conventional Fourier decomposition, as opposed to that described in Ref.[10]. This is to allow a more "conventional" comparison between results. It may be seen that the results are not too bad. The results are very dependent on changes to the algorithm and the boundary conditions. Because of this, attention is now directed at developing a frequency domain model.

The results shown in Fig.[5] & [6] do not indicate very good agreement and this is due to the sensitivity of the results, from both codes, to changes in grid density, boundary conditions and algorithms. The region in the vicinity of the shock wave dominates the flow and is mostly numerical error[10]. A better means of decomposing the results needs to be developed.

Concluding Remarks

A method of using strip theory to do steady and unsteady transonic flow calculations has been developed and the results contained here are the consequences of the preliminary testing of the method. It is clear that further research is required but it is felt that the basic concept has been proven. Because of the problems with time accurate calculations it is suggested that further research be concentrated on frequency domain versions of the method. The method is computationally very fast and is easy to use because of the strip theory concept. The method can be adapted easily to produce aerodynamic influence coefficients.

References:

1) Batina, J. "Efficient Algorithm for the Solution of the Unsteady Transonic Small Disturbance Equation" J Aircraft Vol. 25 No. 7 1988

2)Guruswamy, G.P. "User's Guide for ENSAERO - A Multidisciplinary Program for Fluid/ Structure Interaction Studies" NASA TM 108853 1994

3)Albano, E.D. & Rodden, W.P. "A Doublet Lattice Method for Calculating Lift Distribution on Oscillating Surfaces in Subsonic Flow" AIAA Journal Vol.7 1969

4)Whitlow,W. "XTRAN2L A Program for Solving the General Frequency Transonic Small Disturbance Equation" NASA TM 85723 1983

5)Samuel C McIntosh Private Communication 1995

6)Nixon,D. "Advanced Technology Assessment Report on Computational Fluid Dynamics" NWING Inc. Tech. Rpt. No. 3 1995

7)Nixon,D. "A Revised Theory for the Flow over Infinitely Swept Wings" NWING Inc. Note in preparation.

8)Jameson, A. & Caughey, D.A. COO-3077-140 New York University, June 1977

9)Nixon,D "Behavior of the Flow through A Numerically Captured Shock Wave" AIAA Journal Vol.22 No.1 1984
10)Nixon, D "Shock Capture Errors in Fourier Decomposition of Unsteady Transonic Flow Computations" NWING Inc. Report No.4 1997
11)Nixon,D "A Preliminary Study of the Errors in Aerodynamic Structural Coupling" NWING Inc. Report No.5 1998
12)Vermeersch, S.A. et al "Towards Cost-Effective Aeroelastic Analysis on Advanced Parallel Computing Systems" AIAA Paper 97-0646 1997
13)Nixon, D. et al "Turbulence Amplification Through a Shock Wave" AIAA Paper 92-0313 1992
14)Nixon, D & Reisenthel, P.H. "A Theoretical Study of the Causes and Control of Buffet" AIAA Paper 94-0312 1994
15)Ashby, D.L. et al "Potential Flow Theory and Operation Guide for the Panel Code PMARC_12 obtained from COSMIC 1992
16)"RAPTL Users Manual" in preparation NWING Inc.
17)Nixon,D "Higher Order Compressibility Effects" NWING Inc. Report No.6 1998
18) Klopfer, G.H. & Nixon, D. "Nonisentropic potential Formulation for Potential Flows" AIAA J. Vol.22 No.6 1984
19)Yates, E.C. "Modified Strip Theory Analysis Method for Predicting Wing Flutter at Subsonic to Hypersonic Speeds" J. Aircraft, Vol.3 No.1 1966
20)Lui, D.D. "Computational Transonic Equivalent Strip Method for Application to Unsteady Aerodynamics" AIAA Paper 83-0261 1983
21)Nixon, D "An Alternative Treatment of the Boundary Conditions for the Flow over Thick Wings" Aero. Quart. Vol.28 Part 2. 1977
22)Shankar, V. et al "A Fast Time Accurate, Unsteady Full Potential Scheme" AIAA J. Vol.25 No.2 1987
23)Kuchemann, D. "The Aerodynamic Design of Aircraft" Pergamon Press 1976
24)"Experimental Database for Computer Program Assessment" AGARD AR 138 1979
25)Loving,D.L. & Estabrooks, B.B. "Analysis of Pressure Distribution of Wing-Fuselage Configuration Having a Wing of 45deg Sweepback, Aspect Ratio 4, Taper Ratio 0.6 and NACA64a006 Airfoil Section: NACA RM L51f07, 1951
26)Bland, S.R. "AGARD Three Dimensional Aeroelastic Configurations" Private Communication
27)Borland, C.J. "XTRAN3S Transonic Steady and Unsteady Aerodynamics for Aeroelastic Applications" AFWAL TR 85 3124 Jan. 1986
28)Horsten, J.J. et al "Unsteady Transonic Pressure Measurements on a Semi-Span Wind Tunnel Model of a Transport Type Wing" AFWAL TR 83 3039 1983

Case	Section	M	α
1	NACA0012	0.72	0.0
2	NACA0012	0.63	2.0
3	NACA0012	0.75	2.0
4	NACA0010	0.85	1.0
5	RAE2822	0.73	3.19

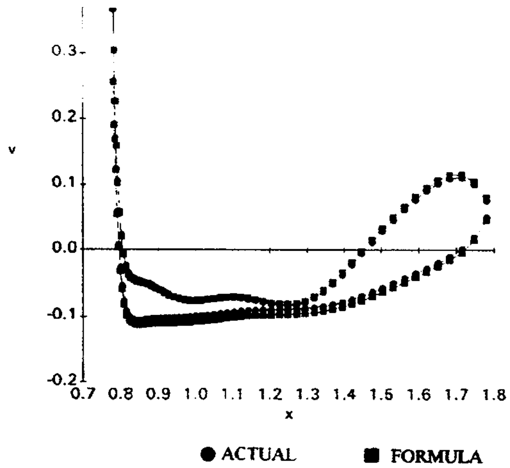
Table 1 Two Dimensional Test Cases

Case	Section	M	α	Taper Ratio	Twist	Sweep
1	NACA0012	0.8	0.0	1.0	0.0	20.0
2	NACA0012	0.826	2.0	1.0	0.0	20.0
3	NACA0010	0.85	1.0	0.33	0.0	30.0
4	LANN	0.82	2.62	0.45	-4.0	27.0
5	ONERA M6	0.84	3.06	0.45	0.0	29.0

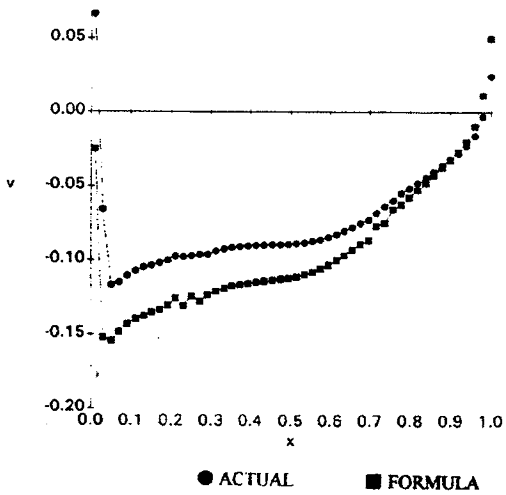
Table 2 Swept Wing Test Cases

Case	Section	Body Radius	Span	M	α	Taper Ratio	Twist	Sweep
1	RAE 101	0.33	1.82	0.9	1.0	0.4	0.0	37.0
2	NACA65A006	0.4	1.6	0.93	0.0	0.0	0.0	45.0

Table 3 Wing - Body Test Cases



(a) FLO22



(b) CAP v12

Fig.[1] Difference Between Numerical Results & Formula

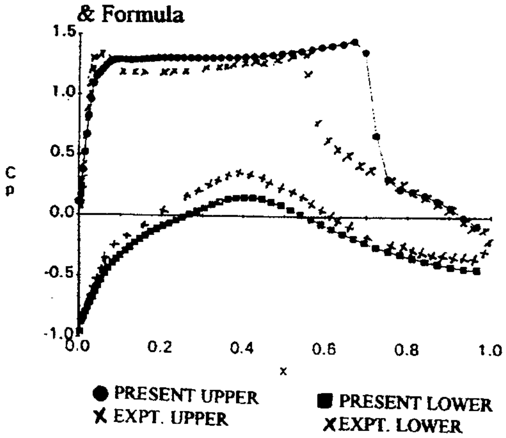
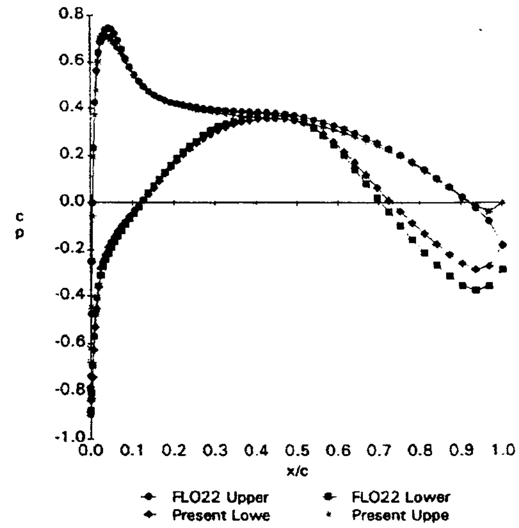
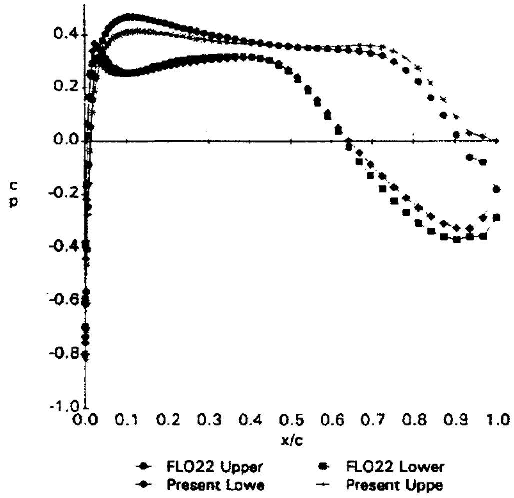


Fig.[2] Pressure Distribution around RAE 2822 Airfoil M=0.73; alpha=3.19 (Inviscid Calculation)

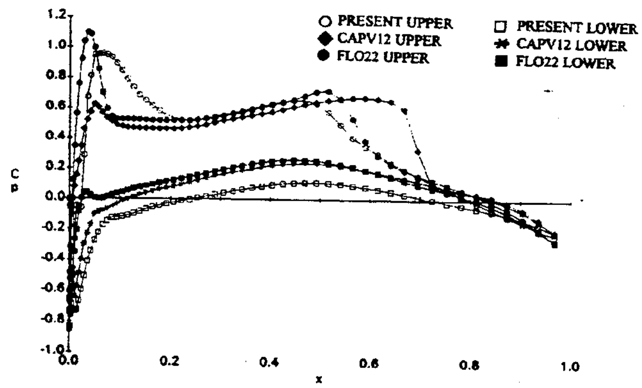


(a)Eta=0.07

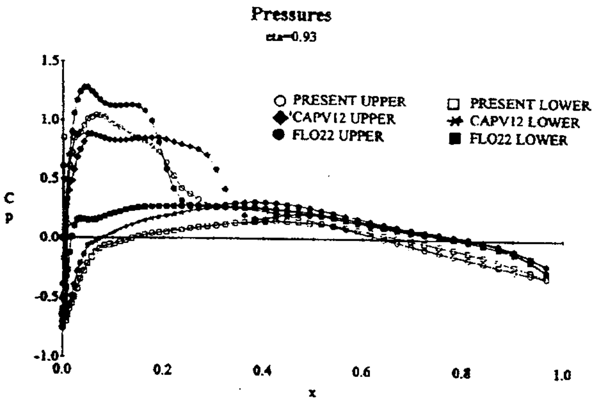


(b)Eta=0.736

Fig.[3] Higher Order Compressibility Correction LANN Wing. M=0.655; alpha=0.62

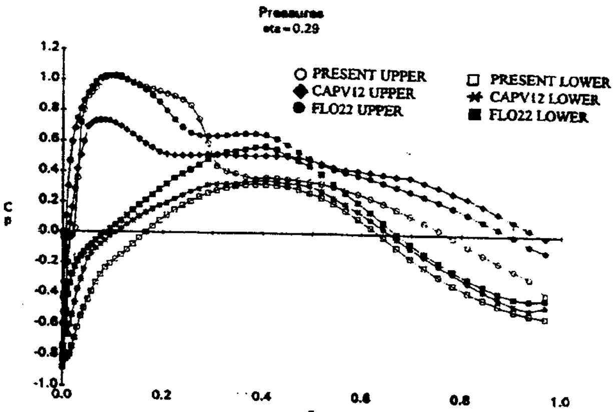


(a)Eta=0.16

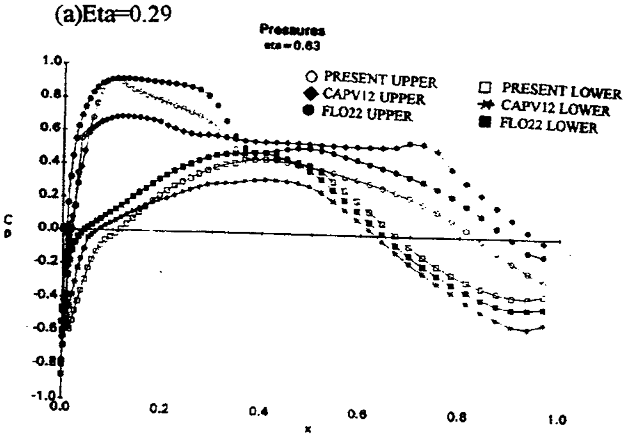


(b)Eta=0.93

Fig.[4] Pressure Distribution around ONERA M6 Wing M=0.84;alpha=3.06

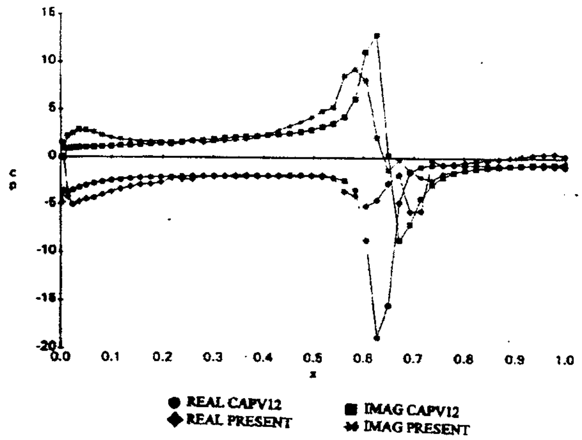


(a)Eta=0.29

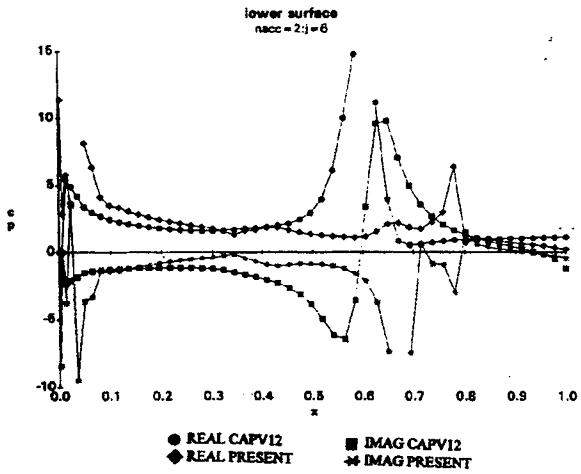


(b)Eta=0.63

Fig.[5] Pressure Distribution around LANN Wing; M=0.82; alpha=0.62

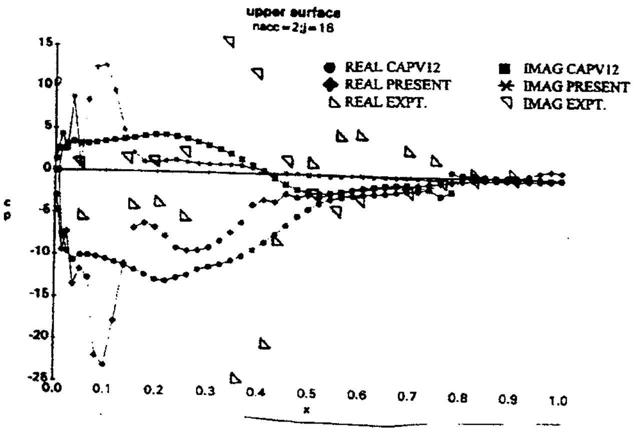


(a)Upper Surface

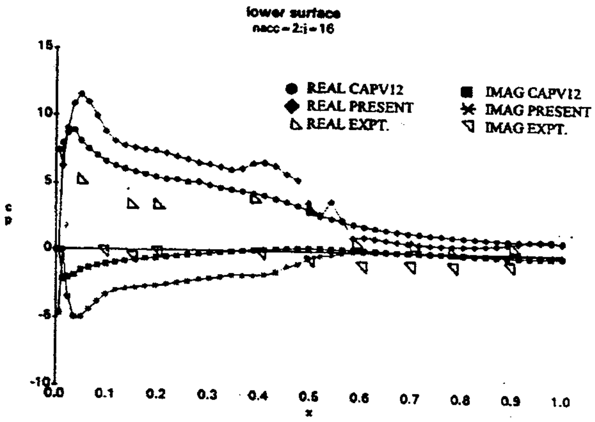


(b)Lower Surface

Fig.[6] Unsteady Pressures around F-5 Wing M=0.9; alpha=0.0; freq=0.275



(a) Upper Surface



(b)Lower Surface

Fig.[7] Unsteady Pressure Distribution for LANN Wing.
M=0.82; alpha=0.62; freq=0.205