COMPUTATIONAL TECHNIQUES FOR UNSTEADY AERODYNAMICS

J. Szmelter and A. Pagano

British Aerospace Airbus Limited - PO Box 77, Bristol BS99 7AR, UK

Abstract

The ability to model time dependent flows over complex geometries is becoming increasingly important to the civil aircraft design process, with the prediction of flutter as a prime application, particularly when combined with structural codes.

In this paper only two dimensional codes are discussed, as a necessary initial step towards three dimensional generalisations. Solution techniques for two dimensional time dependent, viscous and inviscid flows around complex geometries with moving bodies are presented. A pilot version of a viscous-inviscid coupled method has been developed. It uses a combination of time dependent Euler solvers coupled to an integral boundary layer code.

An inviscid solution is obtained on inviscid triangular meshes which are ideally suited to avoid problems of complex geometry modelling. The unstructured mesh generator and Euler solver have moving meshes and local refinement capabilities, thus allowing for general body movement and for adaptive capturing of flow features such as shocks. Finite element discretisation in space and Runge-Kutta time stepping are used. Numerical examples and computational results are provided.

1. Introduction

In recent years there has been increased interest in the development of aeroelastic and flutter analysis methods involving Computational Fluid Dynamics. This research was mainly concerned with the development of fast response methods such as Vortex Lattice and Unsteady Transonic Small Perturbation methods assuming a very simplified description of flows together with empirical corrections. These methods are well established within the aerospace industry and are sufficient for many types of problems when the requirements imposed on the accuracy of aerodynamic predictions are moderate.

Increasing sophistication in aircraft design requires further CFD developments allowing for better flow modelling, together with the potential to simultaneously integrate both aerodynamics and structural equations of motion in the same time domain and with the revision of the adequacy of the traditional method of flutter analysis, where the aeroelastics matrix equation is solved. Rapid progress in mesh generation and in flow solvers have resulted in the first successful unsteady solutions for three dimensional Euler with moving meshes and for two dimensional Navier-Stokes methods [1-3]. This progress is particularly limited by very high CPU and memory requirements for which the introduction of parallel computing provides new opportunities.

The ability to model time dependent flows over complex geometries with moving bodies is becoming increasingly important to the civil aircraft design process, particularly when combined with structural codes, with the prediction of flutter seen as a prime application.

The study of complex configurations and complex flows, also for steady state problems, is receiving increasing attention by the aerospace industry. The flexibility and generality offered by unstructured meshes (tetrahedral or triangular) to model geometrically very complex configurations have been well recognised, particularly as the method offers relatively short lead times to generate new meshes. Examples of three dimensional applications of tetrahedral unstructured meshes to typical Airbus configurations are presented in fig. 1. In addition many forms of adaptivity such as mesh enrichment/derefinement, mesh regeneration and mesh movement are easily introduced within unstructured meshes methods and they maintain the uniform data structures within the flow solver, thus having good potential for porting codes into massively parallel computers.

However, the successful application of unstructured meshes techniques within the aircraft design process is still limited to the solution of the Euler equations since the generation of highly stretched unstructured meshes suitable for the solution of the Navier-Stokes equations over three dimensional complex geometries is not readily available, although initial achievements for both three dimensional and two dimensional cases have been reported [4-6].

Alternatively viscous-inviscid interaction techniques can be used in which the viscous layer (boundary

Copyright © 1994 by ICAS and AIAA. All rights reserved.

layer and wake) and the external inviscid flow are calculated separately and then coupled together through an iterative, interactive process. Although they are based on the simplified descriptions of the flow, viscous coupled techniques can be used for many aeronautical applications. These techniques have been proven to be very effective in predicting the high speed performance of civil transport aircraft and they are one of the mainstays of the CFD prediction/design environment at British Aerospace.

The viscous coupling of the unsteady unstructured meshes based Euler solvers to boundary layer solvers is proposed in this paper.

2. Numerical Procedure

The novel application of viscous coupling to unstructured meshes allows for viscous modelling and avoids the difficulty of generating the highly stretched tetrahedral or triangular elements required for Navier-Stokes solvers. In this paper two dimensional flows are discussed, as a necessary initial study leading towards three dimensional generalisations. A suite of codes has been developed, which consists of two steady state unstructured meshes Euler solvers with the option of coupling to an integral boundary layer code. Work to use a finite difference boundary layer code is also under way. The semi-inverse approach [7] is used. At the present initial stage of the work the coupling technique and boundary layer codes are steady state.

Both the inviscid solvers and the unstructured mesh generator originated at Swansea University. The mesh generator used during the unsteady calculations is based on the advancing front technique [8,9]. This technique requires the user to specify the so called background mesh which is responsible for the density of point distribution in the generated computational mesh. To improve user friendliness and to ensure good quality of the initial mesh used at the beginning of the calculation in the steady state mode, the first background meshes used in this work have been generated using a different technique based on the Delaunay Triangulation [9]. Such approach allows to model easily important two dimensional complexities such as multiple aerofoils for high lift configurations and blunt trailing edges as illustrated on a typical Airbus geometry in fig.2.

The unsteady calculation is performed using the advancing front technique, since the method, with its local remeshing capability [10], allows for arbitrary movement of bodies and for mesh refinement in the vicinity of flow features, such as moving shocks, characterised by high gradients. Alternative treatment of moving meshes by means of the 'spring analogy' [1,10] technique is also available within the suite, although this method imposes limitations on the amplitude of the movement.

Two versions of the flow solver based on the Runge -Kutta time stepping and on Taylor-Galerkin discretisation in time [10] are used. Their performance has been compared for a series of meshes of different density using an Airbus type aerofoil section for the inviscid steady state calculation. The representative results are given in Table 1.

The integral laminar/turbulent boundary layer code uses a prescribed pressure distribution (direct mode) or the predicted growth in displacement thickness (inverse mode) to drive the turbulent boundary layer equations. The method is based on the 'lag-entrainment' approach [11,12], pioneered at DRA, which has been further developed to incorporate surface curvature [13], compressibility effects [14] as well as wake calculations [15] and other enhancements.

For applications to realisitc civil aircraft configurations it is important that the boundary layer methods used are able not only to calculate well behaved attached laminar and turbulent flows, but also to handle 'off design' behaviour such as for instance: shock induced boundary layer interaction and separation, trailing edge separation and interaction between shock induced and trailing edge separation. The techniques advocated in this paper are able to handle limited regions of separation [15,16]. Also Le Balleur has extended the semi-inverse coupling method to treat massively separated flows [17].

Although the semi-inverse viscous-inviscid coupling technique has been used, other forms of coupling as well as time dependent boundary layer methods are under investigation. For the limited range of frequencies of oscillating aerofoils the present coupling to the steady boundary layer is valid, however the confidence of better identification of the appropriateness of this assumption requires further evaluation of the presented capabilities. Work is also well advanced to introduce viscous coupling to the three dimensional unstructured meshes based Euler solvers, initially in the steady-state mode.

3. Numerical Results

Three unsteady inviscid runs using the adaptive Runge-Kutta flow code are presented for the

NACA0012 Aerofoil. Incidence becomes a function of time:

$$alpha = a0 + a1 \sin(kt)$$

Testcase	Mach N.	a0	a1	k
1	0.600	2.89	2.41	0.0808
2	0.796	0.00	1.01	0.2020
3	0.599	4.86	2.44	0.0814

The results in the form of C_p plots and C_L history varying with incidence are given in figs.3-5.

The results of a steady state viscous run using the Runge-Kutta flow code with two levels of adaptivity employed to capture the shock is presented for the RAE 2822 aerofoil in fig.6. The adapted mesh, and correspoding Mach number contours and C_p plot are shown.

Finally results of the viscous quasi-steady run for NACA0012 aerofoil are provided in fig.7, in the form of curves of C_L and C_M versus incidence.

4. Conclusions and Future Prospects

The novel approach described in this paper has demonstrated the feasibility of combining unsteady unstructured meshes Euler solvers with boundary layer methods to develop a technique capable of predicting viscous unsteady non-linear flows associated with flutter and aeroelastic phenomena.

Work is well advanced at BAe Airbus towards the creation of a new generation of highly effective, accurate unsteady aerodynamic prediction tools for application to the design and development of civil transport aircraft. The principal elements of this work are :

- Further evaluation and development of the present 2D unsteady viscous coupled suite
- Development of a 3D unsteady unstructured meshes Euler solver with moving bodies
- Development of 2D and 3D unsteady boundary layer codes and of viscous coupling techniques
- Coupling of the new unsteady aerodynamic prediction methods to structural analysis codes.

This work is part of the BAe Airbus Ltd long term Technology Development Strategy to fulfil the requirements of its Product Plan. This strategy involves collaboration with external parties at national and international level.

5. References

1. J.T. Batina, D.A. Seidel, S.R. Bland, R.M. Bennett, 'Unsteady Transonic Flow Calculations for Realistic Aircraft Configurations.' Journal of Aircraft, Vol.26, No.1, pp-21-28, 1989.

2. B.A. Robinson, J.T. Batina, H.T.Y. Yang, 'Aeroelastic Analysis of Wings Using the Euler Equations with a Deforming Mesh.' Journal of Aircraft, Vol.28, No.11, 1991.

3. R. Lohner, 'Three-Dimensional Fluid-Structure Interaction using a Finite Element Solver and Adaptive Remeshing'. Computing Systems in Engineering, Vol.1. Nos.2-4, pp-257-272, 1990.

4. D.J. Mavriplis, 'Turbulent Flow Calculations Using Unstructured and Adaptive Meshes'. ICASE Report No.90-61, 1990.

5. T.J. Baker, 'Point Placement and Control of Quality for Inviscid and Viscous Mesh Generation'. Proceedings of the 4th International Conference on Numerical Grid Generation in CFD and Related Fields, The Cromwell Press Ltd., pp-137-150, 1994.

6. O. Hassan, E.J. Probert, K. Morgan, J. Peraire, 'Line Relaxation Methods for the Solution of 2D and 3D Compressible Viscous Flows Using Unstructured Meshes'. Proceedings of 1993 European Forum on Recent Developments and Applications in Aeronautical CFD, Published by Royal Aeronautical Society - London, pp-7.1-7.11, 1993.

7. J.C. Le Balleur, 'Strong Matching Method for Computing Transonic Viscous Flows Including Wakes and Separation'. Rech. Aerosp, 1981, (3), 21-45.

8. J. Peraire, J. Peiro, L. Formaggia, K. Morgan, O.C. Zienkiewicz, 'Finite Element Euler Computatons in Three Dimensions'. Int. J. Num. Meth. Engng, Vol.26, pp-2125-2159, 1988.

9. N.P. Weatherill, 'Delaunay Triangulation in Computational Fluid Dynamics'. Computer Meth. Applic., Vol.24, No.516, pp-129-150, 1992.

10. O. Hassan, E.J. Probert, K. Morgan, J. Peraire, 'Domain Decomposition Combined with Adaptive Remeshing for Problems of Transient Compressible Flow'. Proceedings 13th International Conference on Numerical Methods in Fluid Dynamics, Rome 1992.

11. J.E. Green, D.J. Weeks, J.W.F. Brooman, 'Prediction of Turbulent Boundary Layers and Wakes in Compressible Flow'. RAE Technical Report TR72231, 1973.

12. J.E. Green, 'Applications of Head's Entrainment Method to the Prediction of Turbulent Boundary Layers and Wakes in Compressible Flow'. RAE Technical Report TR72079, 1972.

13. P. Bradshaw, 'The Analogy Between Streamline Curvature and Buoyancy in Turbulent Shear Flow'. J. Fluid Mech., Vol.36, pp-177-191, 1969.

14. J.E. Green, 'The Prediction of Turbulent Boundary Layer Development in Compressible Flow'. J. Fluid Mech., Vol.31, part 4, pp-753-778, 1968.

15. B.R. Williams, 'The Prediction of Separated Flows Using a Viscous-Inviscid Interaction Method'. RAE Tech Memo Aero 2010, 1984.

16. A. Pagano, 'A Viscous Coupled Euler Method'. Presented at the AGCFM Workshop on Separated Flows, RAE Farnborough, 1985.

17. J.C. Le Balleur, 'Viscous-Inviscid Calculation on High-Lift Separated Compressible on High-Lift Separated Compressible Flows over Aerofoils and Wings'. Proceedings AGARDCP-415, 1992.

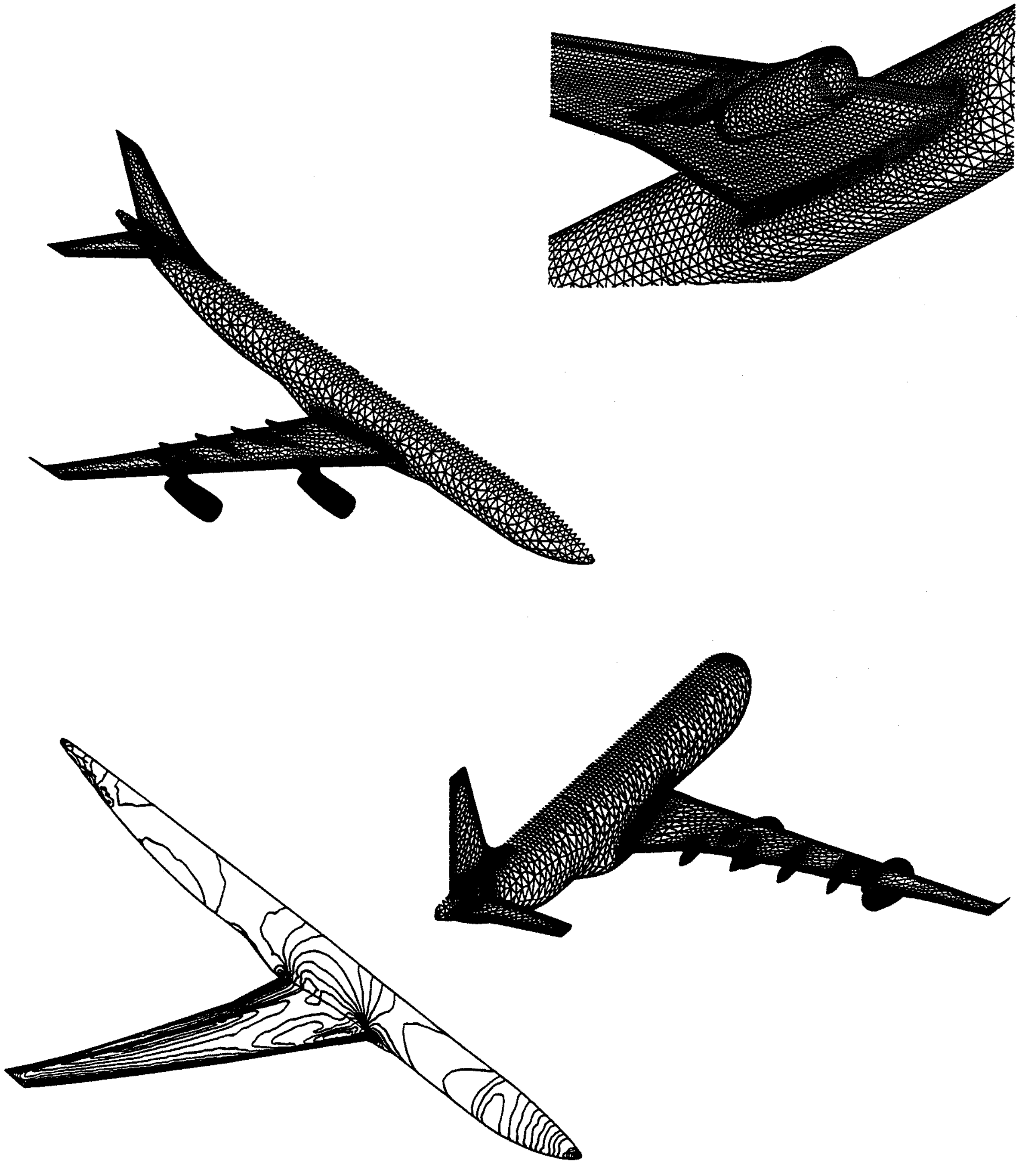


Figure 1 Surface meshes and Cp contours on typical Airbus configurations.

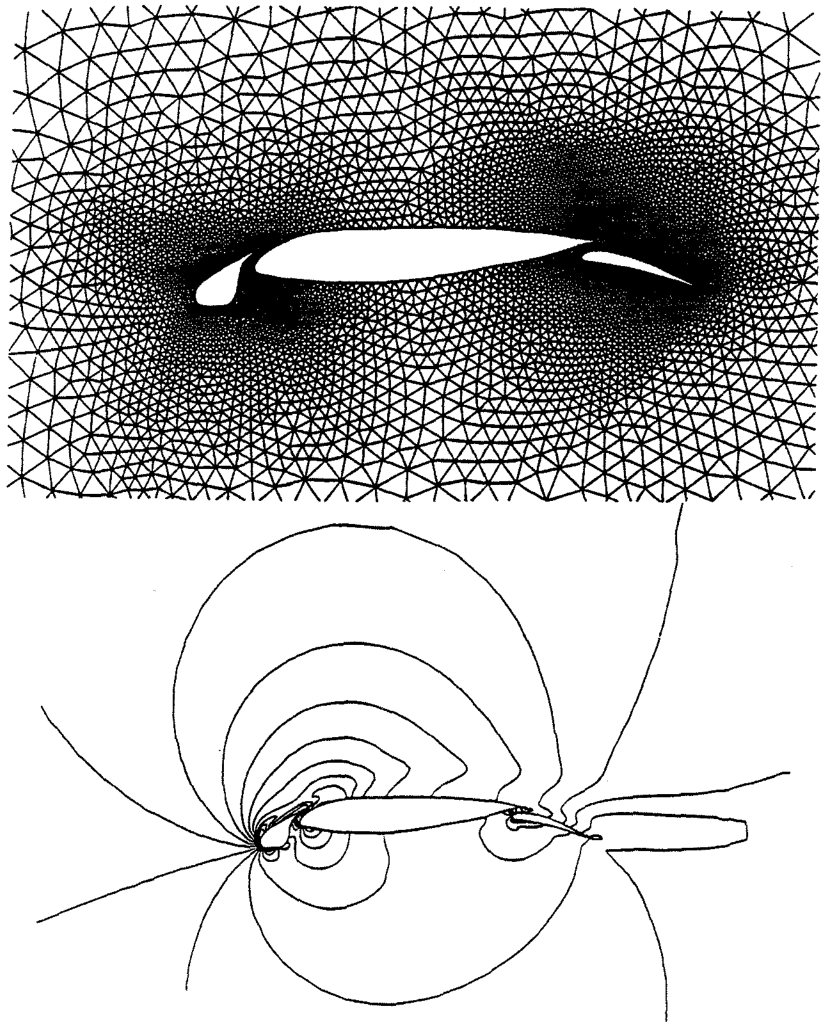
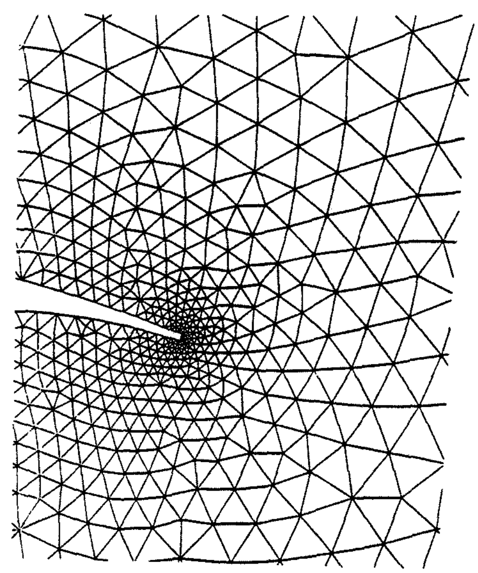


Figure 2

Multicomponent Configuration and detail of aerofoil with blunt trailing edge.



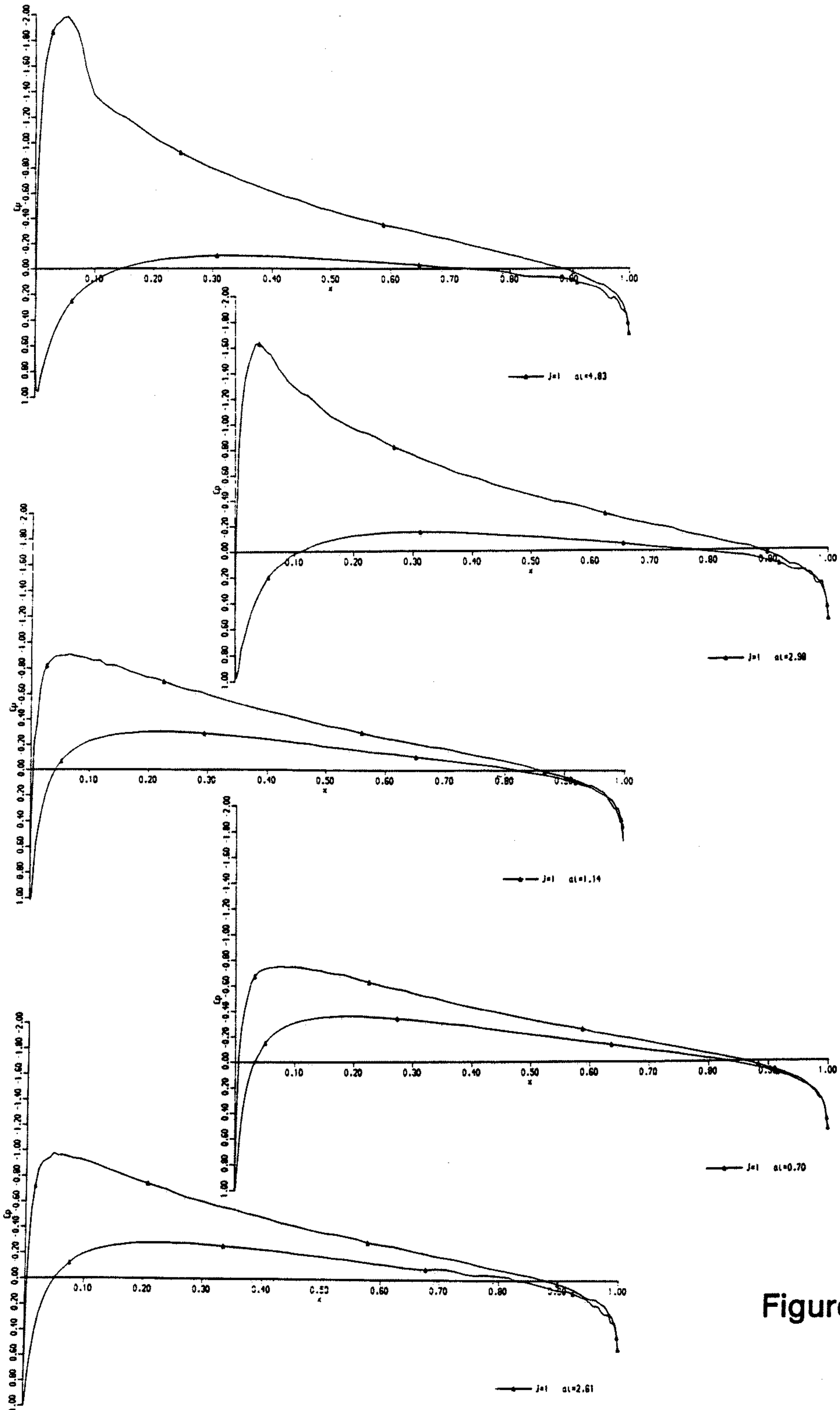
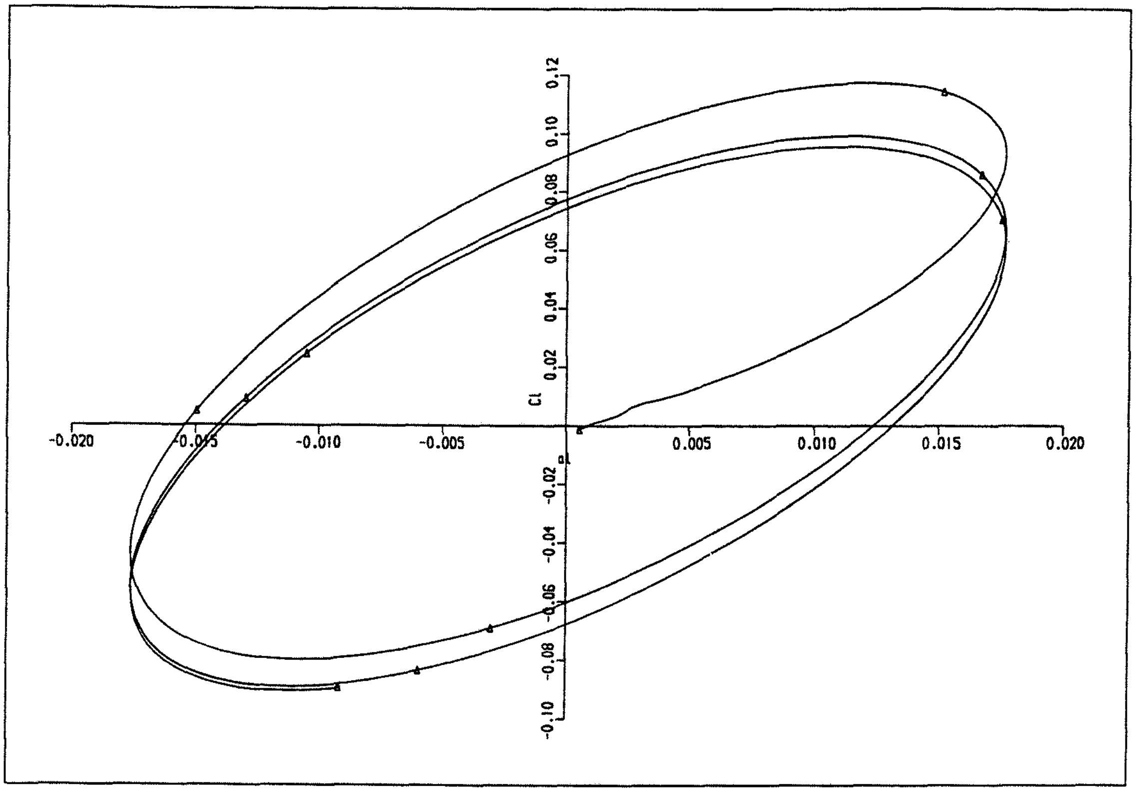
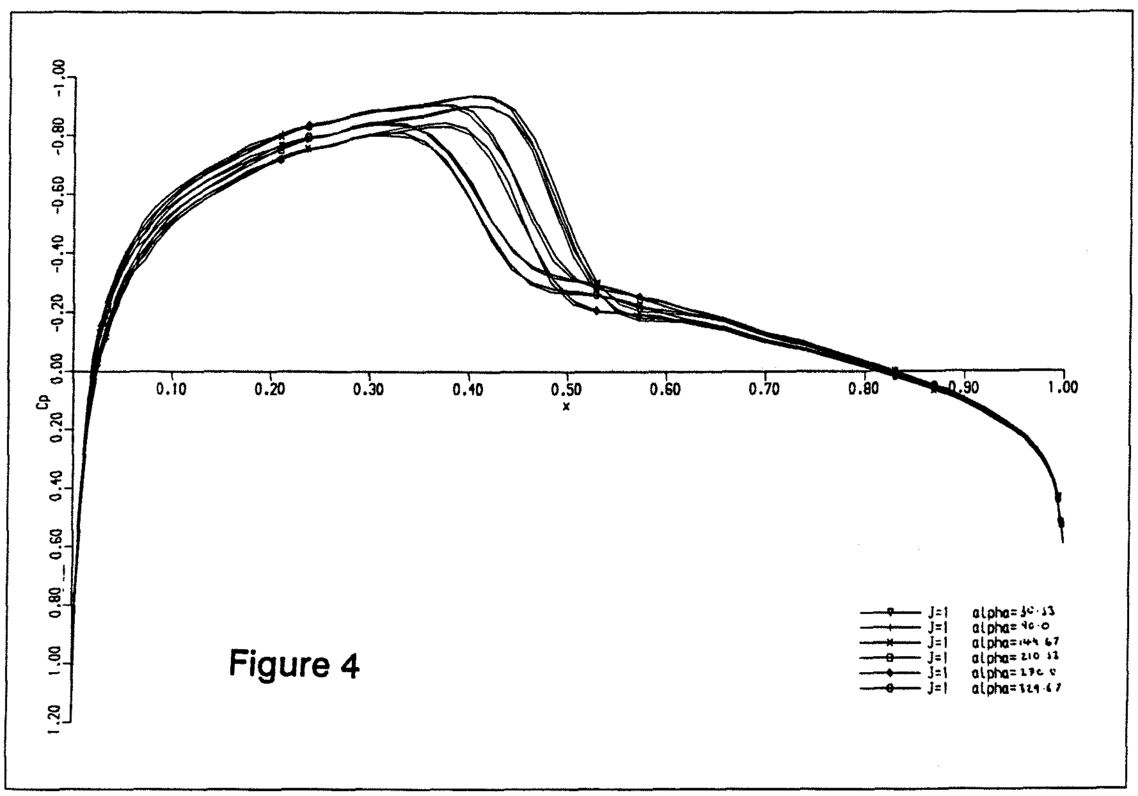


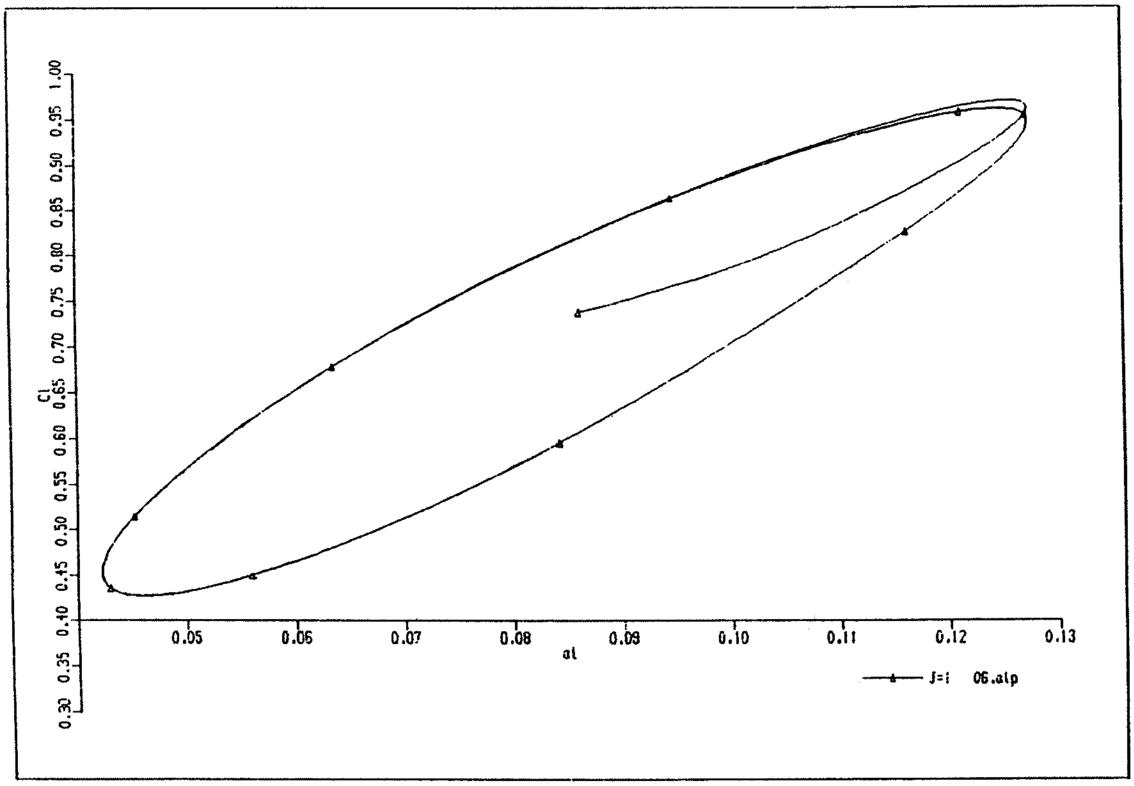
Figure 3

Unsteady Inviscid Calculation NACA0012 Aerofoil, TEST CASE 1.





Unsteady Inviscid Calculation NACA0012 Aerofoil, TEST CASE 2.



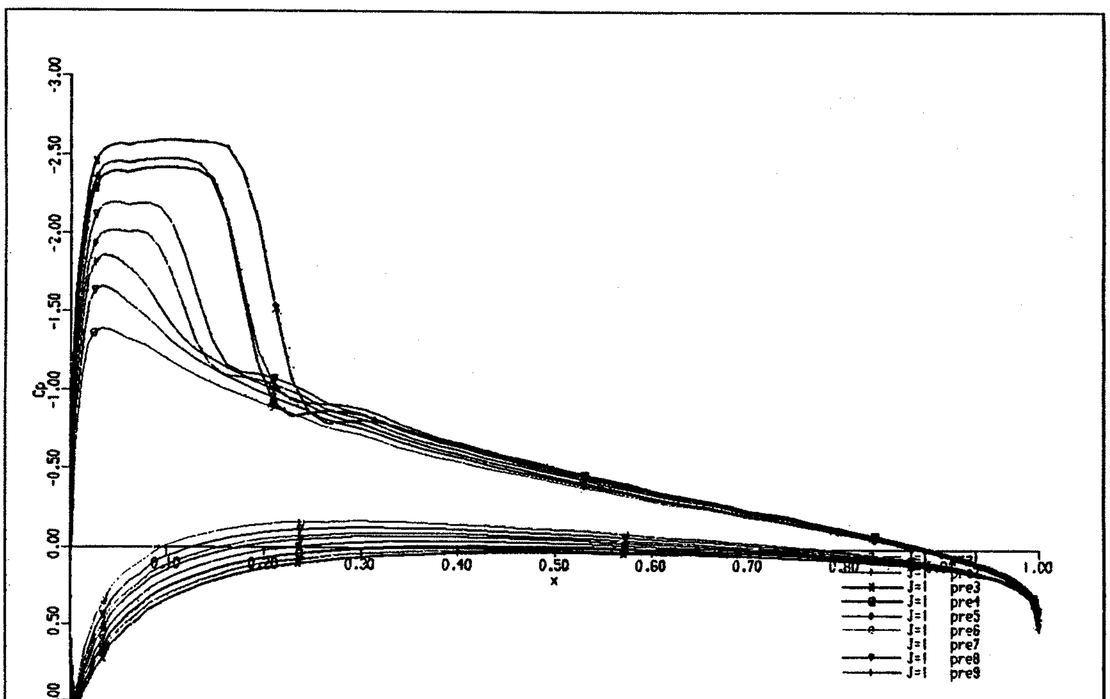
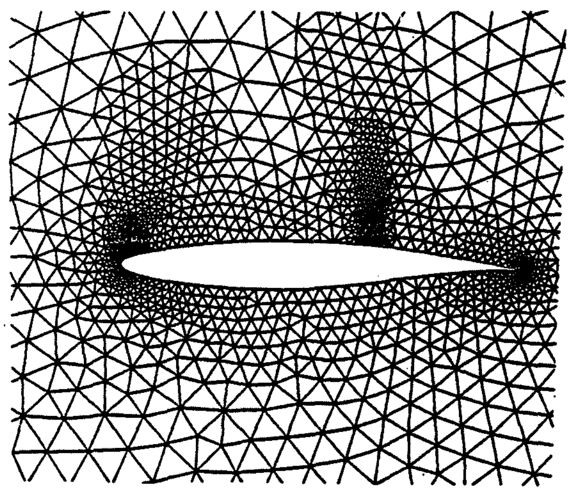
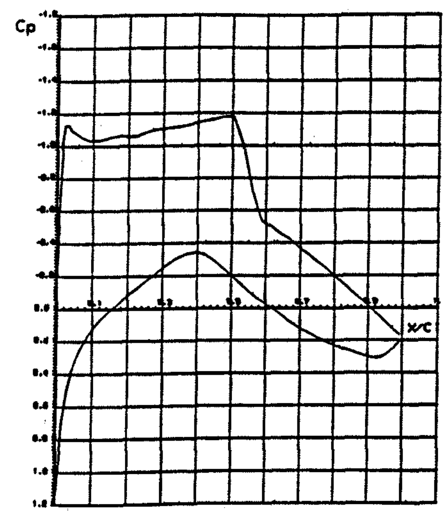


Figure 5

Unsteady Inviscid Calculation NACA0012 Aerofoil, TEST CASE 3.





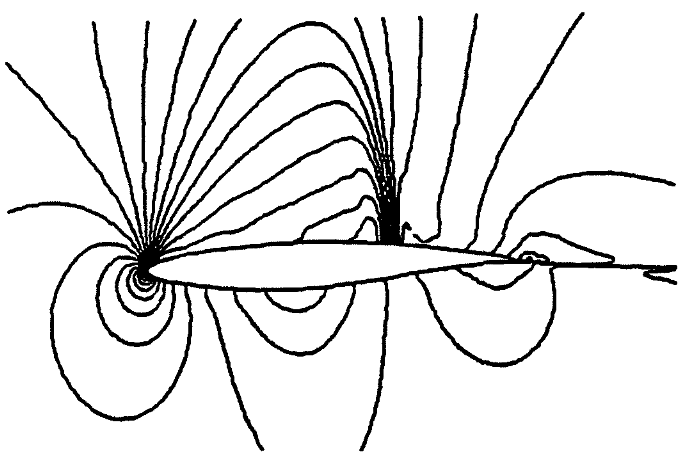


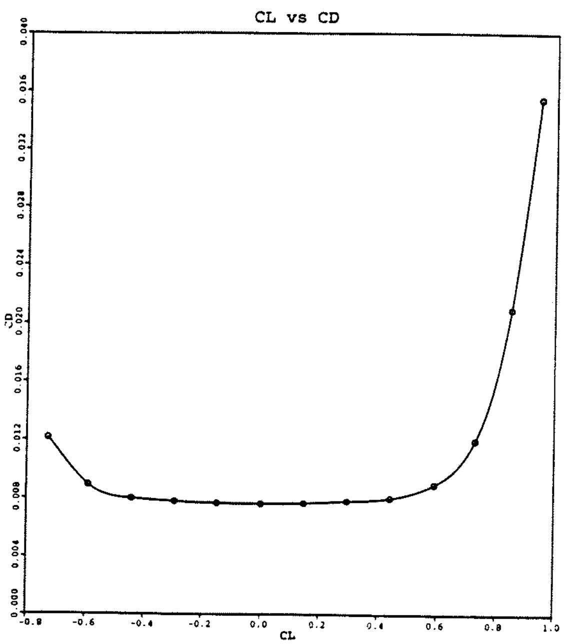
Figure 6

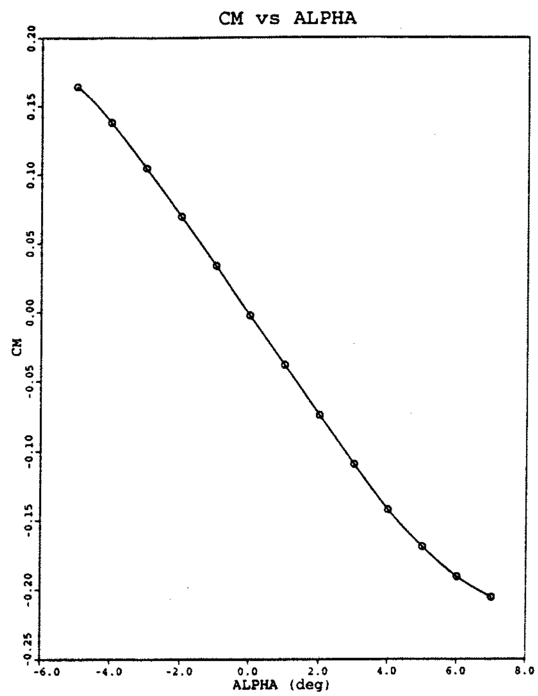
RAE2822 Aerofoil - Adapted Unstructured Triangular Mesh, Mach contour plot ans Cp plot computed at M=0.729, alpha=2.46 and Re=6500000.

	-RUNGE-KUTTA SCHEME			TAYLOR-GALERKIN SCHEME		
	M2S COARSE MESH	M1S STANDARD MESH	M3S FINE MESH	M2S COARSE MESH	M1S STANDARD MESH	M3S FINE MESH
No BOUNDARY POINTS	127	161	196	127	161	196
No POINTS	2337	3916	5927	2337	3916	5927
No ELEMENTS	4547	7671	11658	4547	7671	11658
Cl −6	−0.37640	−0.38270	−0.37890	−0.35634	−0.36616	−0.36959
Cl −4	0.01350	0.01920	0.02634	0.01559	0.01848	0.01614
Cl −2	0.45580	0.46340	0.46645	0.41766	0.42326	0.43177
Cl 0	0.89430	0.89300	0.89210	0.82379	0.84769	0.85022
Cl 2	1.28750	1.28510	1.28880	1.18120	1.20770	1.20550
Cl 4	1.54840	1.55240	1.55310	1.44520	1.47800	1.47710
Cd −6	0.05376	0.05558	0.05426	0.05572	0.05402	0.05338
Cd −4	0.00953	0.00494	0.00470	0.01707	0.01398	0.01252
Cd −2	0.00340	0.00265	0.00140	0.01106	0.00908	0.00642
Cd 0	0.01289	0.01353	0.01432	0.02261	0.02010	0.02040
Cd 2	0.06535	0.06330	0.06402	0.05979	0.05808	0.05600
Cd 4	0.13432	0.13082	0.13343	0.12125	0.12242	0.12275
Cm −6	−0.06691	−0.06788	−0.07281	−0.06524	−0.06636	−0.06553
Cm −4	−0.20055	−0.20286	−0.20839	−0.19230	−0.19931	−0.19982
Cm −2	−0.31779	−0.31814	−0.32186	−0.30354	−0.30877	−0.31361
Cm 0	−0.44866	−0.44833	−0.44796	−0.42991	−0.44011	−0.43769
Cm 2	−0.60969	−0.60421	−0.60959	−0.55123	−0.56386	−0.56260
Cm 4	−0.72474	−0.72097	−0.72513	−0.66012	−0.67527	−0.67373

TABLE 1

Viscous Testcase
NACA0012 Aerofoil
M = 0.599, Re = 8.5e6, Trans = 5%





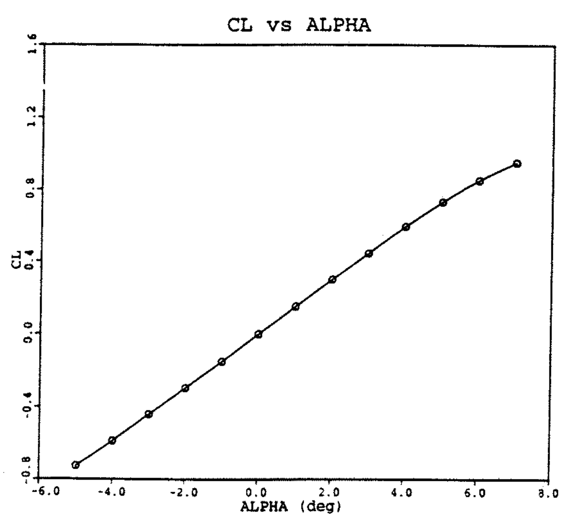


Figure 7