An implicit finite volume nodal point scheme for the solution of two-dimensional compressible Navier-Stokes equations

Vimala Dutta

Computational and Theoretical Fluid Dynamics Division
National Aerospace Laboratories, Bangalore 560 017, INDIA

Abstract: An implicit finite volume nodal point scheme has been developed for solving the two-dimensional compressible Navier-Stokes equations. The numerical scheme is evolved by efficiently combining the basic ideas of the implicit finite-difference scheme of Beam and Warming (1978) with those of nodal point schemes due to Hall (1985) and Ni (1982). The 2-D Navier-Stokes solver is implemented for steady, laminar/turbulent flows past airfoils by using C-type grids. Turbulence closure is achieved by employing the algebraic eddy-viscosity model of Baldwin and Lomax (1978). Results are presented for the NACA-0012 and RAE-2822 airfoil sections. Comparison of the aerodynamic coefficients with experimental results for the different test cases presented here establishes the validity and efficiency of the method.

1. Introduction

Due to rapid advances achieved in numerical methods as well as computer technology, there has been considerable progress in the development of Navier-Stokes solvers in the past decade. Several finite-difference and finite-volume time stepping schemes have been sucessfully applied to a variety of two- and three-dimensional fluid flow problems.

The main advantage of the finite-volume method is the capability of handling arbitrary geometries through direct discretisation in the physical space. This approach has the flexibility of modifying the arrangement of control volumes and the evaluation of fluxes through the control surfaces. In the cell-centred finite-volume scheme, the dependent variables are associated with the centre of the cell in the computational mesh. In cell-vertex or nodal point schemes, the flow quantities are specified at cell vertices rather than at cell centres or as cell averages.

With a view to implement the cell-vertex finite-volume approach to implicit schemes, a new implicit finite-volume nodal point scheme for the solution of 2-D Navier-Stokes equations is presented here. The numerical scheme has been developed by combining the basic ideas of nodal point schemes due to Hall (1985) and Ni (1982) with those of implicit finite-difference scheme of Beam and Warming (1978). Since implicit schemes have significantly larger stability bounds than explicit methods, fewer time steps are needed to compute unsteady flow over a given interval of time or to converge to a steady state,

2. Finite volume formulation

In order to facilitate a finite-volume formulation, the two-dimensional Navier-Stokes equations for unsteady compressible flow are written in integral form after applying Euler's implicit time-differencing formula. The computational domain is partitioned into a finite

The integral conservative equations are applied to each control volume formed by joining the centres of the neighbouring cells of a nodal point.

The line integrals occurring in the resulting equations are evaluated by summing up the contributions due to the flux terms over the four edges of the computational cell. The terms containing inviscid flux vectors are calculated by using the flow variables at the four neighbouring points. The derivatives in viscous flux terms are evaluated by using Taylor's series expansion around the location at which the gradients are needed.

Second- and fourth-order dissipation terms are added to ensure convergence and to suppress oscillations near shock waves. The turbulence model of Baldwin and Lomax (1978) is employed for the computation of eddy viscosity in turbulent flows.

3, Compressible viscous flow around airfoils

The implicit finite-volume nodal point scheme thus developed is applied here for steady compressible viscous flow around airfoils. The computational domain considered for the flow past an airfoil is bounded by the airfoil surface, an outer far field boundary and a wake cut extending from the trailing edge of the airfoil. C-type grids are generated using an algebraic grid generation technique (Jain 1983). At the body surface, no-slip condition for velocity and adiabatic wall condition for temperature are imposed. Along the branch cut in the wake the conservative variables are obtained by averaging their values on either aide of the cut. At the outer boundary, viscous effects are neglected and non-reflecting farfield boundary conditions are constructed using the theory of characteristics for locally one-dimensional flow normal to the boundary.

4. Results

Computations have been carried out for steady laminar/turbulent flows past the NACA-0012 and RAE-2822 airfoil sections.

Figure 1 shows the C_p distribution, convergence history, streamlines, pressure and Mach contours for laminar subsonic flow past NACA 0012 airfoil at $M_\infty = 0.8$, $a = 10°$, $Re = 500$ and CFL=50. The lift, drag and moment coefficients compare well with those obtained by other methods (Müller 1986, Müller and Rizzi 1986, Chakrabartty 1987). Since it is an implicit method it has been possible to achieve fast convergence. The streamlines show a separation region with separation and reattachment points at $x/c = 0.39$, $x/c = 0.98$ respectively, which match well with those ($x/c = 0.37$, $x/c = 0.97$) predicted by Miiller (1986). The pressure and Mach contours are also very similar to those obtained by Müller (1986). Figure 2 presents the surface pressure distributions for high Reynolds number flows over the same airfoil. Good agreement with the experimental results (Thibert *et al.* 1979) is observed for the subsonic Mach number case with $M_\infty = 0.5$, $a = 1.77°$, $Re_\infty = 2.91 \times 10^6$. In the transonic flow case with $M_\infty = 0.799$, $a = 2.26°$, $Re_\infty = 9 \times 10^6$, the shock location is predicted farther downstream compared to experiment (Harris 1981).

Results for transonic turbulent flow around RAE-2822 airfoil section are presented in Figs. 3 and 4 Pressure distribution, shock location, upper surface skin friction distribution and the aerodynamic coefficients are well predicted in the first test case, *viz.*, $M_\infty = 0.73$, $\alpha = 2.79°$, $Re_\infty = 6.5 \times 10^6$. Even though no separation is reported in the experimental results (Cook *et al.* 1979), the skin friction distribution shows a small separation near the trailing edge. In the second test case, *i.e.*, $M_\infty = 0.75$, $\alpha = 2.81°$, $Re = 6.2 \times 10^6$, the position of the shock and the flow downstream of the shock wave are slightly different from the experimental data (Cook *et al.* 1979). The upper skin friction distribution, however, indicates separation at the foot of the shock wave from $x/c = 0.64$ to $x/c = 0.67$ in agreement with experimental results. Even th not remain separated upto the trailing edge, there is another separated flow region close to the trailing edge.

5. Conclusions

An implicit 2-D Navier Stokes solver has been developed for steady laminar/turbulent compressible flow past airfoils by employing a cell-vertex finite-volume approach. The 2-D Navier-Stokes equations governing the flow are solved by an implicit, finite-volume nodal point scheme. The numerical algorithm is derived by combining the basic ideas of the implicit finite difference technique of Beam and Warming (1978) with those of nodal point schemes due to Hall (1985) and Ni (1982). Results presented for NACA-0012 and RAE-2822 airfoil sections show good agreement with experimental results. The implicit nature of the numerical scheme permits the use of larger time steps and hence faster rate of convergence to steady state solution is possible.

Acknowledgement

The author is thankful to Prof. R. Narasimha for his keen interest and encouragement in this indigenous effort. She thanks Dr. S.S. Desai, Dr. M.D. Deshpande and Dr.P. Ramamoorthy for their support and encouragement at different stages of the present work. She is grateful to Dr.S.K. Chakrabartty for providing the grid generation program and turbulence model routine and also for allowing her to continue this work. Her heartfelt thanks are also due to her dear friend Mrs. K. Dhanalakshmi for the graphics routines in particular, and computational assistance in general.

References

Baldwin, B.S. and Lomax, H. (1978), "Thin layer approximation and algebraic model for separated turbulent flows," AIAA Paper No. 78-257.

Beam, R.M. and Warming, R.F. (1978), "An implicit factored scheme for the compressible Navier-Stokes equations," *AIAA J.*, 16(4), pp.393–402.

Chakrabartty, S.K. (1987), "Numerical solution of two-dimensional Navier-Stokes equations by finite volume method," DFVLR-IB-129-87/23.

Cook, P.H., McDonald, M.A., and Firmin, M.C.P. (1979) "Airfoil R.AE 2822 - pressure distributions and boundary layer and wake measurements," AGARD AR-138.

Hall, M. G. (1985), "Cell vertex multigrid scheme for solution of Euler equations," RAE-TM-Aero-2029, Proc. Conf. on Numerical Methods for Fluid Dynamics, UK, pp.303–345.

Harris, C.D. (1981), "Two dimensional aerodynamic characteristics of the NACA 0012 airfoil in the Langley 8-foot transonic pressure tunnel," NASA TM-81927.

Jain, R.K. (1983), "Grid generation about an airfoil by an algebraic equation method," DFVLR-IB-129-83/40.

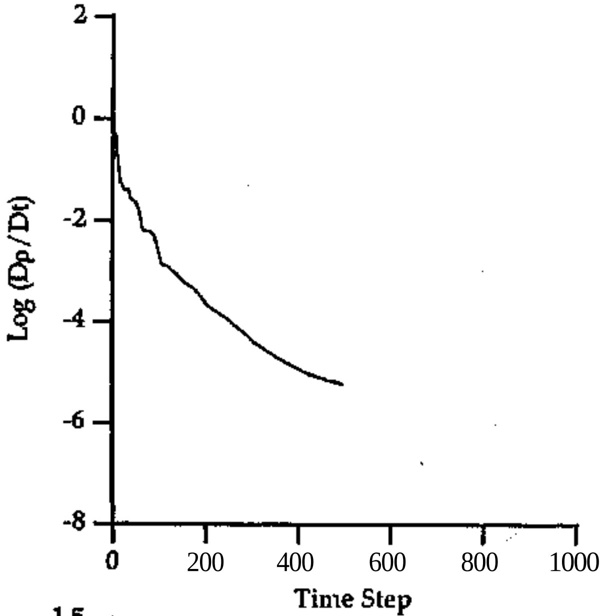
Müller, B. (1986), "Navier-Stokes solution of laminar transonic flow over a NACA 0012 airfoil," FFA Report-140.

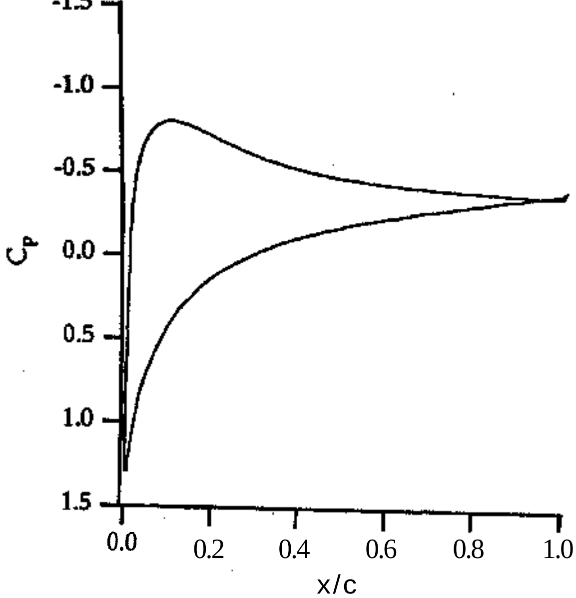
Müller, B., and Rizzi, A. (1986), "Runge-Kutta finite volume simulation of laminar transonic flow over a NACA 0012 airfoil using the Navier-Stokes equations," FFA TN-1986-60.

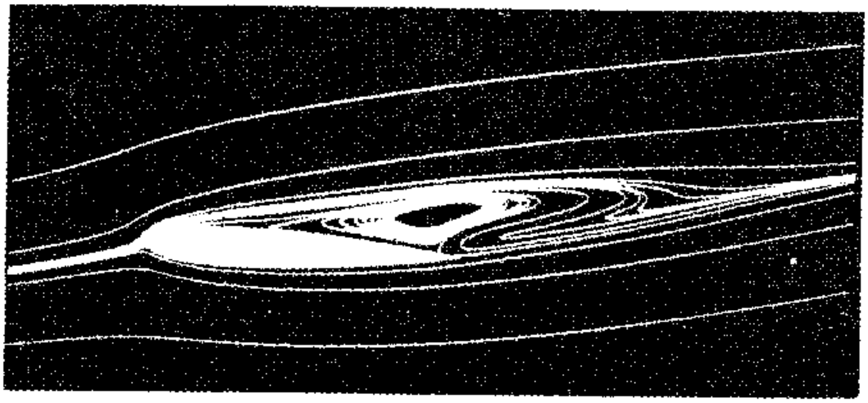
Ni, R.H. (1982), "A multiple grid scheme for solving the Euler equations," *AIAA J.*, **20**, 1565-1571.

Thibert, J.J., Grandjacques, M., and Ohman, L.H. (1979), "NACA 0012 airfoil," AGARD AR-138.

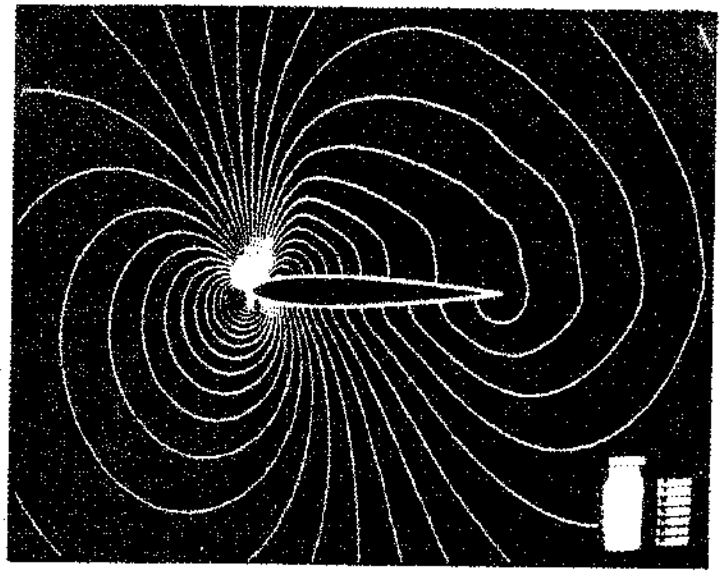
	Present	Müller	Müller & Rizzi	Chakrabartty
CL •	0.4119	0.4199	0.4327	0.4711
CDp	0.1412	0.1411	0.1455	0.1365
CDf	0.1138	0.1221	0.1239	0.1440
CD	0.2550	0.2632	0.2694	0.2805
CM (le)	-0.1111	0.1145	0.1199	-0.1140



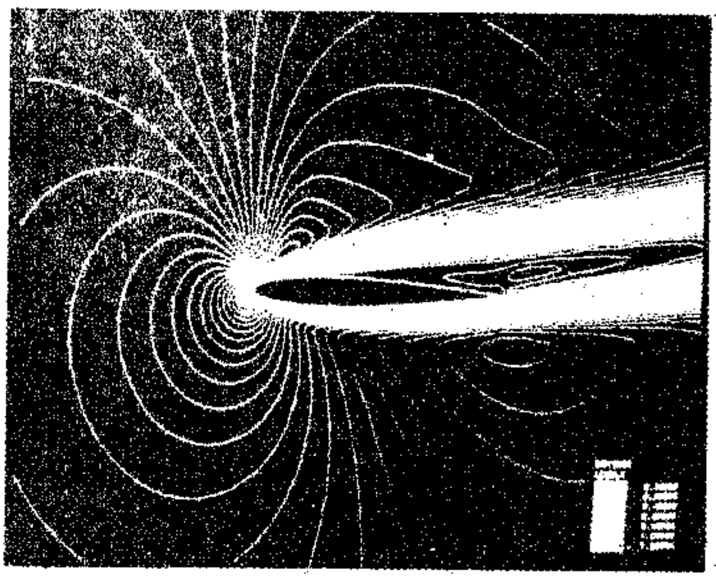


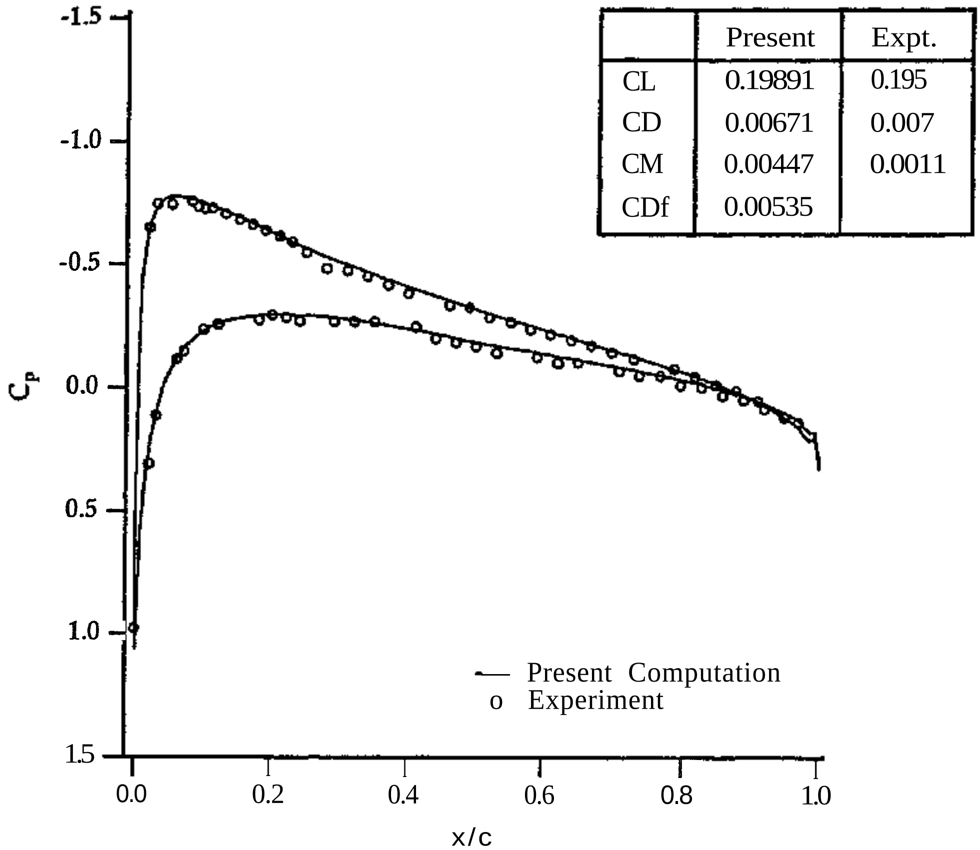


(i) Streamlines



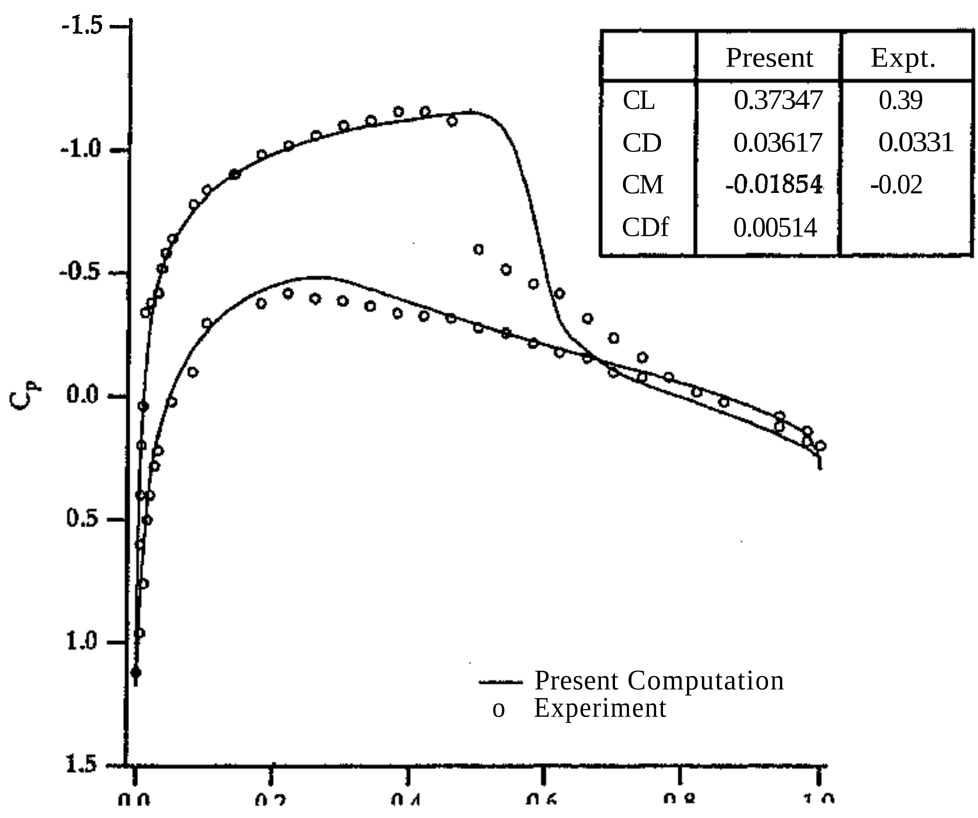
(ii) Pressure contours



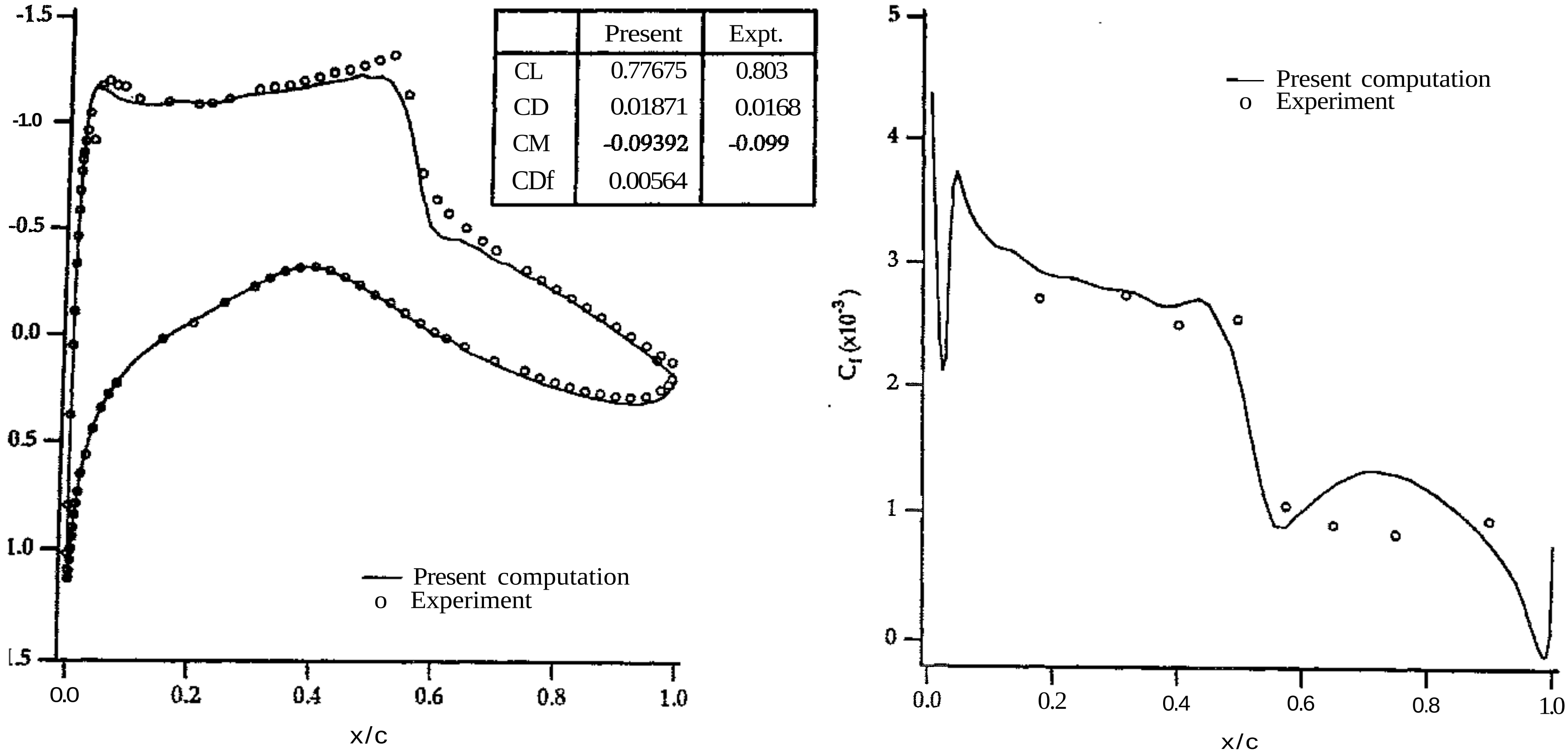


	Present	Expt.
CL	0.19891	0.195
CD	0.00671	0.007
CM	0.00447	0.0011
CDf	0.00535	

Figure 2a. Surface pressure distribution for a NACA-0012 airfoil.
(M_∞ = 0.5, a = 1.77°, Re = 2.91 x 10^6, Grid = 165 x 61)

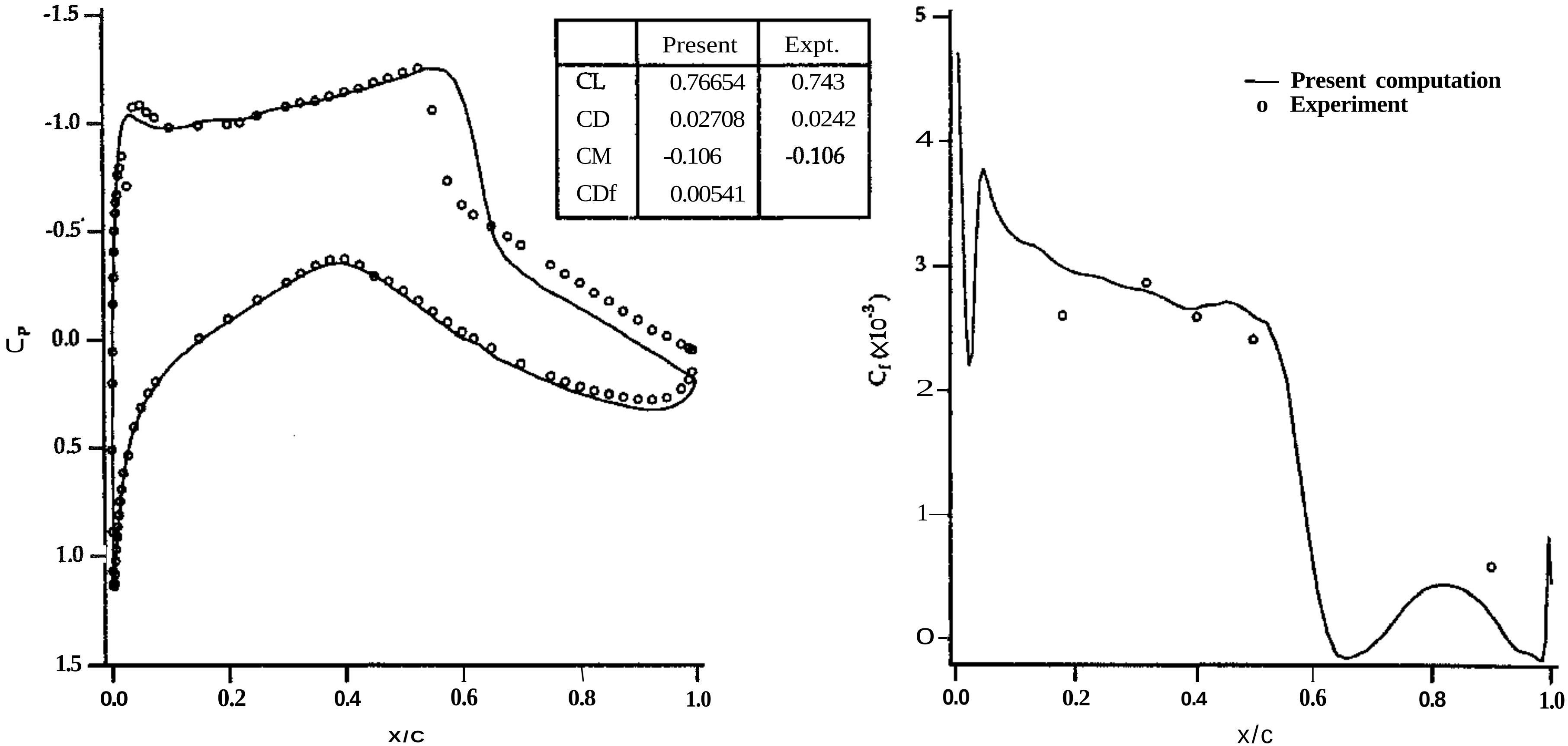


	Present	Expt.
CL	0.37347	0.39
CD	0.03617	0.0331
CM	-0.01854	-0.02
CDf	0.00514	



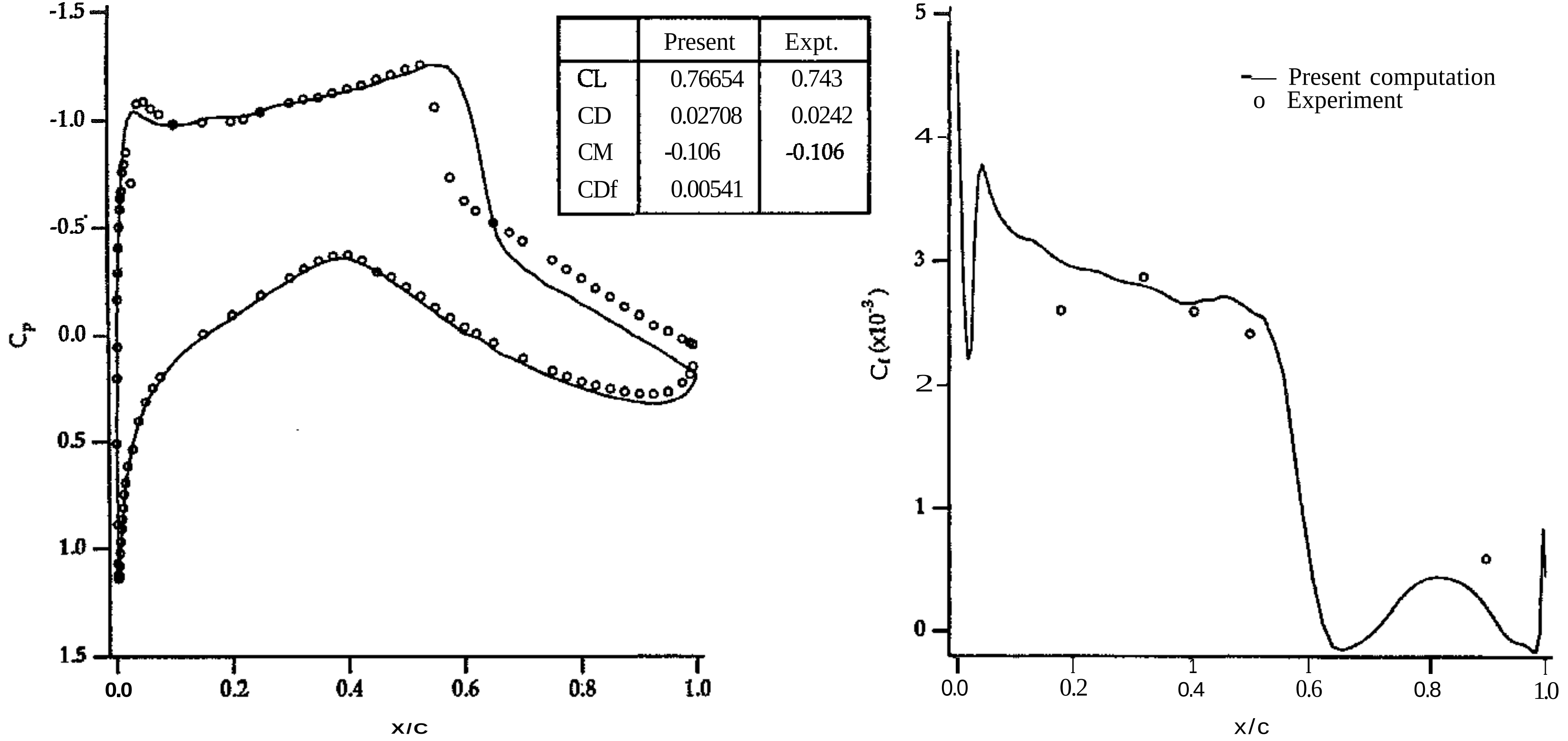
	Present	Expt.
CL	0.77675	0.803
CD	0.01871	0.0168
CM	-0.09392	-0.099
CDf	0.00564	

Figure 3. Surface pressure and skin-friction distributions for an RAE-2822 airfoil. (M_∞ = 0.73, a = 2.79°, Re = 6.5 x 10^6, Grid = 247 x 65)



	Present	Expt.
CL	0.76654	0.743
CD	0.02708	0.0242
CM	-0.106	-0.106
CDf	0.00541	

Figure 4. Surface pressure and skin-friction distributions for an RAE-2822 airfoil, (M_∞ = 0.75, α = 2.81°, Re = 6.2 x 10^6, Grid = 247 x 65)



	Present	Expt.
CL	0.76654	0.743
CD	0.02708	0.0242
CM	-0.106	-0.106
CDf	0.00541	

Figure 4. Surface pressure and skin-friction distributions for an RAE-2822 airfoil.
(M_∞ = 0.75, a = 2.81°, Re = 6.2 x 10^6, Grid = 247 x 65)