Computational Challenge: Euler Solution for Ellipses

Thomas H. Pulliam*
NASA Ames Research Center, Moffett Field, California

Abstract

THIS paper presents a difficult flow problem for the numerical solution of the Euler equations: the inviscid flow past an ellipse. The basic result obtained here is a lifting solution for any combination of grid and/or angle of attack which is nonsymmetric. The purpose of this paper is to present this flow as a challenge to the computational fluid dynamics community, where the hope is that someone can explain this unusual behavior.

I. Introduction

The purpose of this paper is to introduce a flow problem for the Euler equations that should be relatively simple to solve numerically, but turns out to be very difficult. The problem is inviscid subcritical flow past an elliptical two-dimensional surface at angle of attack. In particular, a 6:1 ellipse at $M_\infty = 0.2$, $\alpha = 5$ deg is considered, where conventional finite-difference or finite-volume schemes are employed. Assuming initial and boundary conditions which are irrotational and contain no circulation, one would expect that the inviscid flow at any angle of attack would remain irrotational, and not generate any circulation. In practice, every numerical solution obtained to date by this author and many others has produced results with nonzero lift. The results obtained are very sensitive to algorithm parameters, mesh definition, and algorithm type. Even on the same mesh, various codes produce a wide scatter of lift values all of which are nonzero. It can be shown, however, that the solutions obtained are equivalent to potential flow past an ellipse with added circulation defined by the lifting numerical solution. This suggests that the numerical solutions obtained are reasonable solutions to the Euler equations with some mechanism which sets the lift. The mechanism responsible for the circulation generation is not understood at this time, and this paper stands as a challenge for other researchers to define the mechanism or to produce numerical results which are nonlifting for this class of problems.

II. Potential and Euler Solutions

Theoretically, the inviscid irrotational flow (i.e., potential flow) past an ellipse or cylinder is unique up to specification of the circulation or equivalently, the location of the stagnation points on the body. By setting the circulation or stagnation point locations, a one-parameter family of flowfield solutions can be obtained.

Two exact potential solutions are given below. In both cases, $M_\infty = 0.2$, $\alpha = 5$ deg and the solution is plotted on an "O" grid of 129×49 grid points. The ellipse has a major to minor axis ratio of 6:1. Figure 1 shows pressure contours and coefficient of pressure for a nonlifting case ($C_l = 0.0$) and for a lifting case ($C_l = 1.545$), which will be a useful choice in the later discussions. In the case of no lift, as one would expect, stagnation occurs at the front below the leading edge and at the rear antisymmetrically at the top. In the lifting case, stagnation occurs below the leading edge at the front and below the trailing edge at back; see Fig. 1.

This is all straightforward and can be found in any reference on potential flow, e.g., Ref. 1. The question that will be posed here is: What is the Euler solution to this problem? In particular, what is the numerical Euler solution, since a closed-form solution to the nonlinear equations is unknown?

Any procedure is affected by errors such as truncation, added terms (artificial dissipation), insufficient mesh resolution, and approximations in boundary conditions. As a computational experiment, the numerical solution on the preceding mesh was obtained using the Euler code, ARC2D.[2] Details of the various numerical experiments applied to this problem are presented in the original AIAA paper[3] and (due to space limitations) will not be repeated here.

The computations were performed at the preceding conditions. All residuals were driven to machine zero to assure stability of the converged solution. The basic numerical algorithm employs central differencing with added artificial dissipation, $\kappa^{(4)}$. The artificial dissipation coefficients were chosen at their nominal values for the results shown in Fig. 2. Pressure contours and coefficient of pressure are shown in Figs. 2a and 2b for this case. The contour levels for pressure were chosen to be the same for all plots presented here.

The converged value of lift coefficient for this case is $C_l = 1.545$, which corresponds to the lifting potential solution (Fig. 1). In fact, the two solutions are almost identical. The convergence history for the preceding case is shown in Figs. 2c and 2d. This is a very curious result; the initial solution has no circulation associated with it and the final solution, one would have assumed, should have zero lift. A nonlifting result can be obtained only if the oncoming flow is directly aligned with the grid line of upper and lower symmetry. In general, whenever the flow is not exactly aligned with a mesh line of symmetry, a lifting result is obtained.

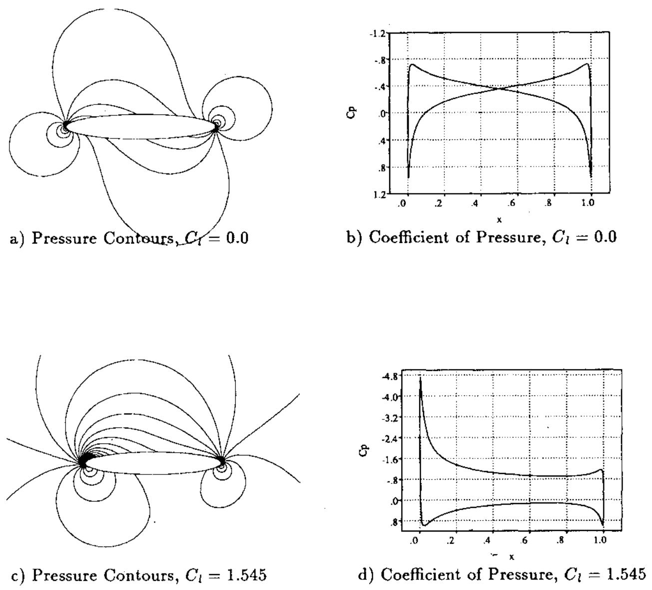
It is remarkable that the numerical solution and the potential solution at the same lift compare so well. This implies that the numerical solution is a solution to the Euler equations (since potential solutions also satisfy the Euler equations for cases where there are no shocks). The question now posed is: What causes the resulting circulation and what determines the final value of lift? Various numerical experiments show that many different final solutions can be obtained for the same physical problem, either by changing some of the algorithm parameters, the grid, or the differencing scheme. In all cases, some converged, very stable solution is obtained and there is no explanation of the mechanism which produces the final results.

Sensitivity to initial conditions can be one indication of nonuniqueness. For the preceding solution, various initial conditions were used ranging from an impulsive start from freestream flow to using the potential solution as initial conditions. The numerical time integration was also extensively modified, where time accurate and non-time accurate paths were taken to convergence. In all tests, for a given mesh, M_∞, α, and $\kappa^{(4)}$, the final steady-state result was the same as the preceding; i.e., $C_l = 1.545$.

Some of the numerical parameters which may influence the results include mesh definition, distribution, and refinement.

Received March 1, 1989; Synoptic received Oct. 10, 1989. Full paper available from National Technical Information Service, Springfield, VA 22151, at the standard price (available upon request).

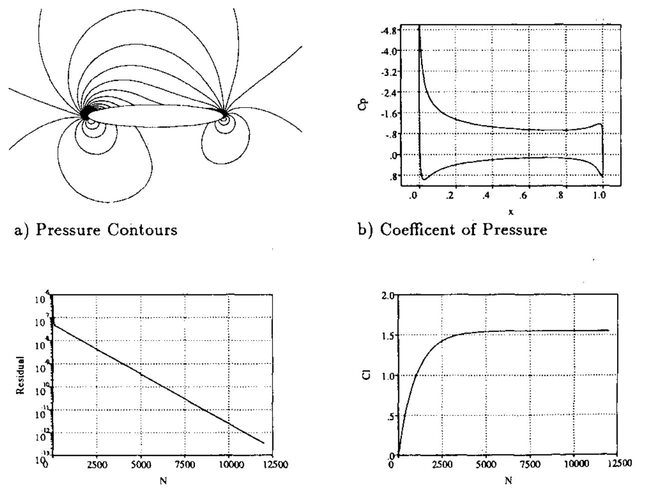
*Section Head, Computational Fluids Division, Computational Fluid Dynamics Branch. Member AIAA.



a) Pressure Contours, $C_l = 0.0$ b) Coefficient of Pressure, $C_l = 0.0$

c) Pressure Contours, $C_l = 1.545$ d) Coefficient of Pressure, $C_l = 1.545$

Fig. 1 Potential solution.



a) Pressure Contours b) Coefficent of Pressure

c) Convergence History for Residual d) Convergence History for C_l

Fig. 2 Euler solution, $\rho^{(4)} = 0.01$.

The fourth order artificial dissipation coefficient was varied by factors of 2 and ½ with some unexpected but interesting results. One possible explanation for the inability of the numerical code to produce the zero-lift case for nonzero α is the error associated with the artificial dissipation. Possible problems include false boundary layers, large total pressure losses, and base flow separation. In none of the calculations performed, even with some of the other codes mentioned below, have there ever been any false boundary layers, separation, or large total pressure losses observed in the results. A detailed study across various values of artificial dissipation coefficients shows a continuous trend from high lift at low values of $\kappa^{(4)}$ to lower lift at higher values of $\kappa^{(4)}$. This is exactly opposite to the trend one would have expected if the magnitude of artificial dissipation were directly related to the production of lift.

Another area of concern and possible error in the computations is the boundary conditions. In the preceding cases, the surface boundary conditions are tangency for the velocity, solution of a normal momentum equation for pressure, and constant freestream stagnation enthalpy for density. The accuracy of these conditions is controlled by the normal mesh spacing at the surface; studies where the mesh spacing was refined showed similar results to those just presented, although the absolute values of lift were different. The boundary conditions were also modified to zero normal gradients of enthalpy and entropy; this did not improve the results, i.e., no zero-lift solutions or even solutions close to zero lift were obtained.

The far-field boundary conditions used were local one-dimensional Riemann invariants (see Ref. 2), with or without the addition of a circulation correction based on the surface lift. The outer boundary location for the preceding cases was at 6 chords. Various cases were run where the outer boundary location was increased up to 100 chords producing results similar to those just given, i.e., again large values of lift. In general, there seems to be an insensitivity to boundary conditions in terms of obtaining a zero-lift solution, with no consistent trends in the final values of lift.

Mesh-refinement studies have also been carried out. These are made difficult by the slow convergence characteristic of this type of flow. The mesh used can be varied as to number and clustering of the nodes. The basic result is that the converged solutions vary inconsistently. As the mesh is refined, the lift may increase or decrease depending on the value of artificial dissipation coefficient, the type of spatial differencing (central or upwind), or the mesh-generation scheme. Similar results have been obtained by other researchers using anything from time accurate to steady-state codes such as multigrid. Others who have attempted this problem have initially reported low values of lift and what seemed to be good convergence. Further examination, however (where they continued their runs to machine zero), resulted in large lift solutions. Therefore, the author recommends that all future attempts at this problem be reported at machine zero.

Since artificial dissipation seems to have a large effect, computations were performed with a battery of[5,6] codes including upwind, total variation diminishing (TVD)-type algorithms. The basic result is a similar inconcsistency, with all codes producing very mesh- and parameter-dependent final solutions with differing values of lift.

These studies have been carried out for varying aspect ratios down to a circular cylinder. Even in the case of the cylinder, lifting results can be obtained if nonsymmetric conditions are employed; i.e., flow angularity misaligned with mesh symmetry. Nonlifting results can be obtained but only for sufficiently fine meshes in the normal direction and even then most codes have difficulty.

III. Summary

The results presented in this paper open a Pandora's box as to the general capability and validity of numerical Euler results, at least for conventional difference methods. The case of inviscid flow past simple bodies (ellipses) for simple conditions (subcritical flow at angle of attack) has produced some rather unusual and disturbing results. The mechanism which produces the lifting results is not understood and the resolution of this problem is unknown. The purpose of this paper is to define the flowfield problem and demonstrate the disturbing results. It is left as a challenge to the computational community to compute an accurate nonlifting result.

References

[1]Hughes, W. F., and Brighton, J. A., *Fluid Dynamics*, Schaum's Outline Series, McGraw-Hill, New York, 1967.

[2]Pulliam, T. H., *Efficient Solution Methods for the Navier-Stokes Equations*, Lecture Notes for the von Kármán Institute for Fluid Dynamics Lecture Series: Numerical Techniques for Viscous Flow Computation in Turbomachinery Bladings, von Kármán Institute, Rhode-St-Genese, Belgium, 1985.

[3]Pulliam, T. H., "A Computational Challenge: Euler Solution for Ellipses," AIAA Paper 89-0469, Jan. 1989.

[4]Pulliam, T. H., "Artificial Dissipation Models for the Euler Equations," *AIAA Journal*, Vol. 24, No. 12, 1986.

[5]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite-Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, 1981.

[6]Anderson, W. K., Thomas, J. L., and Van Leer, B. A., "A Comparison of Finite-Volume Flux Vector Splittings for the Euler Equations," AIAA Paper 85-0122, Jan. 1985.