N91-13330

The Semidiscrete Galerkin Finite Element Modeling of Compressible Viscous Flow Past an Airfoil

Andrew J. Meade, Ph.D

Assistant Professor
Department of Mechanical Engineering and Materials Science
Rice University
Houston, TX 77251-1892

Abstract

During the ASEE summer tenure at NASA Langley I participated in two projects that were of mutual interest to myself and the staff of the Transonic Aerodynamics Branch. The primary project was the numerical simulation, by a finite element/finite difference method, of the viscous flow about an airfoil. This project was a natural outgrowth of my Ph.D dissertation[1]. The secondary project involved the numerical simulation of the three-dimensional separated and vortex-dominated flow about a hemispherically capped cylinder in the transonic regime. This was a logical continuation of the experimental high-alpha study of the hemisphere – cylinder[2] at NASA Ames. Dr. Veer Vasta, of the Computational Aerodynamics Branch, and I have started preliminary calculations[3] for the hemisphere-cylinder at 0° and 5° angle of attack. It was thought that in view of the limited time given, a majority of my efforts would be best applied to the primary project. It was agreed that the secondary project should be handled as a long term task.

An appreciable amount of work done by the Transonic Aerodynamics Branch's (TAB) Applied Aerodynamics Group, is in the design of airfoils and wings in the transonic regime. The solution of the flowfield about these bodies is required to determine the important parameters of lift, moment, and drag. Viscous effects must be accounted for if the drag is to be accurately calculated. At present there are basically two approaches to the numerical simulation of the flowfield, the use of fully viscous models and the inviscid/viscous models.

The fully viscous models require the solution of an approximation of the Navier-Stokes equations and therefore should simulate most of the physical mechanisms. However, in all cases the fully viscous models require relatively large amounts of computer time and storage. Because of their relative speed and simplicity the inviscid/viscous models are still in wide use in design and theoretical investigations. The inviscid/viscous models assume that most of the flowfield can be considered inviscid and that the viscous effects about a body can be approximated by experiments or theoretical means. The most familiar theoretical method of accounting for the viscous effects, is the solution of Prandtl's boundary layer approximations.

A fast, accurate, and computationally efficient inviscid flow solver has recently been developed by Hartwich[4] of the TAB. It is thought that Hartwich's program coupled to a fast, accurate, and computationally efficient boundary layer code, will make an excellent tool for airfoil design. The purpose of the primary project was to develop a compressible boundary layer code using the semidiscrete Galerkin finite element method.

The numerical scheme employed used the combination of a Dorodnitsyn formulation of the boundary layer equations[5], with a finite difference/finite element procedure (semidiscrete Galerkin method[6,7]), in the solution of the compressible two-dimensional boundary layer equations.

Linear elements were chosen for computational efficiency while also providing adequate accuracy. The finite element discretization yielded a system of first order ordinary differential equations in the streamwise direction. The streamwise derivatives were solved by an implicit and noniterative finite difference marching scheme.

A laminar compressible boundary layer code has been developed and has been tested for a NACA 0012 airfoil at a Mach number of 0.5, a Reynolds number of 5000, and zero angle of attack (Fig. 1). At present the boundary layer program solves up to, but not beyond, separation. Also it does not, at present, interact with the inviscid flow solver. However, the code gives good results on the NACA 0012 airfoil, even for the very crudest grid of three nodes. Figure 1b, using seventeen nodes, gives the proper peak coefficient of friction, but separates too early[8]. The NACA 0012 test case is considered a preliminary and encouraging result. It it is believed that the early separation problem is a result of the lack of interaction between the inviscid and boundary layer codes.

Work will continue on the boundary layer program to impliment a turbulence model, extend calculation beyond flow separation, and to incorporate an inviscid code/boundary layer code interation model.

References

1. Meade, A.J., "Semidiscrete Galerkin Modelling of Compressible Viscous Flow Past a Circular Cone at Incidence",U.C. Berkeley Ph.D thesis, Dept. of Mech. Eng., Nov. 1989.
2. Meade, A.J. and Schiff, L.B., "Experimental Study of Three-Dimensional Separated Flow Surrounding a Hemisphere-Cylinder at Incidence", AIAA Paper 87-2492, 1987.
3. Vatsa, V.N. and Wedan, B.W., "Development of an Efficient Multigrid Code for 3-D Navier-Stokes Equations", AIAA Paper 89-1791, 1989.
4. Hartwich, P., "Split Coefficient Matrix (SCM) Method With Floating Shock Fitting For Transonic Airfoils", 12th Int. Conf. on Num. Meth. in Fluid Dynam., Oxford, UK, 1990.
5. Dorodnitsyn, A.A., "General Method of Integral Relations and Its Application to Boundary Layer Theory", Adv. Aero. Sci., 3, 207-219, 1962.
6. Popinski, Z. and Baker, A.J., "An Implicit Finite Element Algorithm For Laminar Boundary Layer Flow", Comp. and Fluids, 9, 43-62, 1981.
7. Fletcher, C.A.J., and Fleet, R.W., "A Dorodnitsyn Finite Element Formulation For Laminar Boundary Layer Flow", Int. J. Num. Meth. Fluids, 4, 399-419, 1984.
8. Radespiel, R., "A Cell-Vertex Multigrid Method For The Navier-Stokes Equations",NASA Technical Memorandum 101557, 1989.

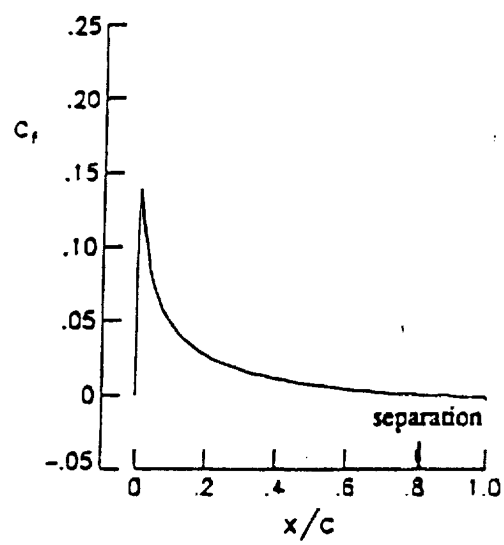

Fig. 1 a. R. Radespiel NASA TM 101557, 1989.

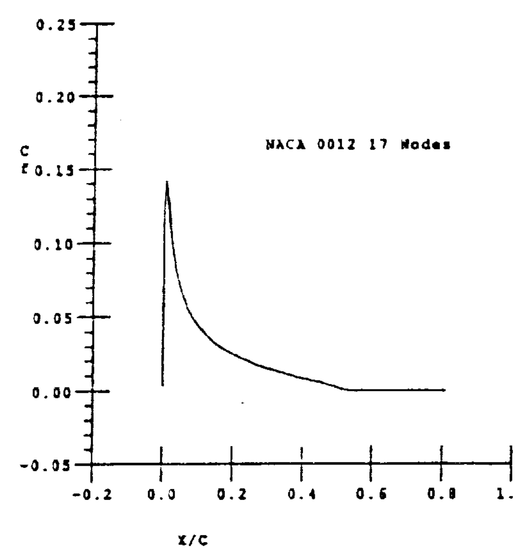

Fig. 1 b. S.D.G. Method