AIAA JOURNAL
Vol. 31, No. 10, October 1993

Zonal-Local Solution Method for the Turbulent Navier-Stokes Equations

D. Drikakis* and S. Tsangaris†
National Technical University of Athens, Athens, Greece

Abstract

A ZONAL-LOCAL solution method is developed for the solution of the compressible Navier-Stokes equations. The main feature of the method is coupling of the Navier-Stokes equations with the Euler equations and the local mesh solution procedure. Mesh sequencing (multilevel) technique is also used for the improvement of the efficiency of the algorithm. The methodology is applied to turbulent flowfields past an airfoil. A flux vector splitting method with an upwind scheme up to fourth-order accuracy is used for the discretization of the inviscid fluxes. The system of the equations is solved by an unfactored implicit method using Gauss-Seidel relaxation.

Contents

Techological improvements in supercomputer speed and memory size provided the means to solve the full compressible Navier-Stokes equations for turbulent flowfields and complex geometries. However, large amounts of computer time are required for the solution of the equations especially for problems in design practice. To reduce the computational time, a zonal-local solution method is presented for the solution of the two-dimensional Navier-Stokes equations in high Reynolds number flows. The equations are solved in time-dependent form and for a curvilinear body-fitted coordinate system

$$JU_t + (E_{\text{inv}})_\xi + (G_{\text{inv}})_\zeta = (1/Re)[(E_{\text{vis}})_\xi + (G_{\text{vis}})_\zeta] \tag{1}$$

where J is the Jacobian of the transformation of the Cartesian coordinates (x, z) to curvilinear coordinates (ξ, ζ), the subscripts inv, vis indicate the inviscid and viscous fluxes, respectively, and U is the conservative variables matrix. The equations are solved by a finite volume Navier-Stokes code[1] which uses a modified[1] Steger-Warming flux vector splitting (FVS) method for the discretization of the inviscid fluxes. The FVS method split the fluxes into two parts, positive and negative, in accordance with the sign of the eigenvalues

$$(E_{\text{inv}})_{i+\frac{1}{2}} = E^{+}_{i+\frac{1}{2}}(U^{-}_{i+\frac{1}{2}}) + E^{-}_{i+\frac{1}{2}}(U^{+}_{i+\frac{1}{2}}) \tag{2}$$

Similarly, the fluxes are discretized on the other cell faces. The definition of the fluxes on the cell faces improves the results in the boundary layers, in contrast with the definition of the fluxes on the center of the volume. The term of the energy equation of the inviscid fluxes is split in accordance with the total enthalpy. An upwind interpolation scheme up to fourth-order accuracy is used in combination with the FVS method for the calculation of the conservative variables on the cell faces. The hybrid scheme[2] is constructed by the superposition of the first-, second-, third-, and fourth-order extrapolation schemes

$$U^{\pm}_{i+\frac{1}{2}} = AU^{1,\pm} + (1-A)\{BU^{2,\pm} + (1-B)[CU^{3,\pm} + (1-C)]U^{4,\pm}\} \tag{3}$$

The superscripts 1–4 denote the varying order of the extrapolation; the terms A, B are limiter functions defined by the squares of the second-order pressure derivatives[2]; and the constant C is 2.25. Chakravarthy's scheme[3] is used for the discretization of the viscous fluxes. The system of equations is solved by an unfactored implicit method[4] using Gauss-Seidel relaxation sweeps. Implicit treatment of the boundary conditions is obtained by Newton subiterations.

The concept of zonal modeling results from the fact that, in high Reynolds number attached flows, diffusion effects are important near the body surface and in bounded shear layers in the wake of the flowfield. Therefore, in regions distant from those regions the flow can be considered as inviscid. In regions where the viscous effects are considered important, Navier-Stokes equations are solved, whereas Euler equations are solved elsewhere. The variation of the streamwise velocity is used as criterion for the treatment between the viscous and inviscid boundary. In this way, the inviscid zone is defined at the position where the streamwise velocity differs less than 10% from the freestream velocity. In the past, other authors have developed zonal methods coupling different kinds of equations in discrete regions of the flowfield.[5,6] The contribution of the present work is the combination of the zonal modeling with the mesh sequencing technique and the local solution of the equations.

In the mesh sequencing technique an initial guess on the fine mesh is obtained by first iterating the solution of the equations on a sequence of coarser grids and then interpolating the solution to the next finer grid. The coarse mesh is constructed by eliminating every second line of the fine mesh on each direction. Because in the mesh sequencing procedure the center of the volumes of the fine mesh is not a subset of the volumes of the coarse mesh, bilinear interpolation is used for the calculation of the conservative variables on the fine mesh using the corresponding variables on the coarse mesh.

The local solution method[7] originates from the nonuniformities of the flow variation toward a steady or unsteady solution. Thus the local solution of the equations can be applied only in regions where the disturbances are large, because in the remaining field the solution has been achieved. The local solution of the equations is obtained in subregions (partial meshes) of the fine mesh characterized by large values of the numerical disturbances. In the case of high Reynolds number flows large numerical disturbances are generated in the turbulent boundary layers. These disturbances are eliminated at the same rate around every solid boundary. Therefore, all of the solid boundaries must be contained in the partial mesh during the local solution of the equations.

Received April 27, 1992; synoptic received Nov. 5, 1992; accepted for publication March 10, 1993. Full paper available from AIAA, 555 W. 57th St., New York, NY 10019.

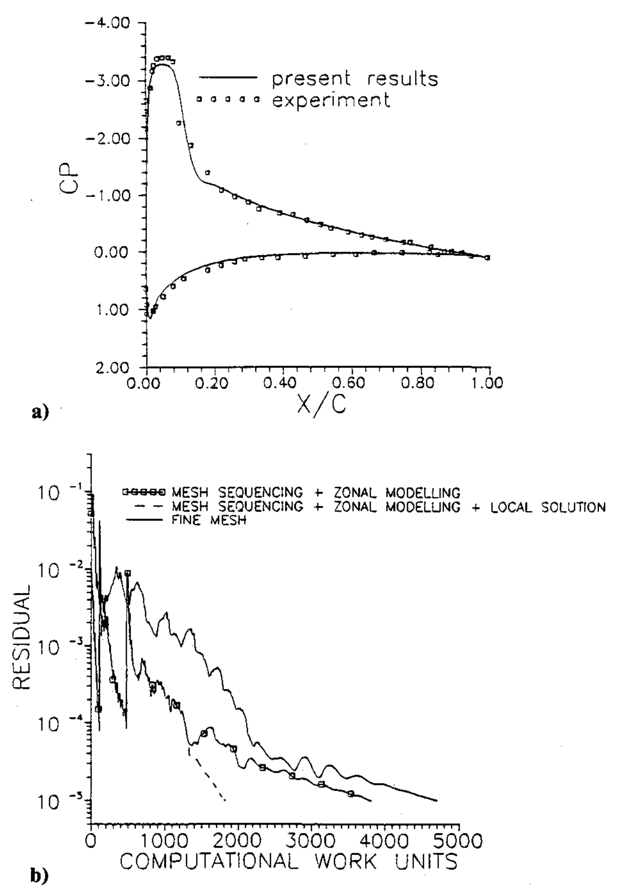
*Research Engineer, Laboratory of Aerodynamics; present address: Lehrstuhl für Strömungsmechanik, Universität Erlangen-Nürnberg, Cauerstrasse 4, D-91058 Erlangen, Germany. Member AIAA.

†Assistant Professor, Laboratory of Aerodynamics, P.O. Box 64070, 157 10 Zografou. Member AIAA.

The local solution technique can be applied in combination with the zonal method for the solution of turbulent flowfields. Thus there are two basic regions where the Euler and Navier-Stokes equations are solved, respectively, although the local solution is applied in both these regions. The local solution begins after a number of iterations on the fine mesh. The criterion for the beginning of the local solution procedure is an input value for the variation of the solution. This input value may be the maximum variation of the conservative variables on the whole flowfield. The partial mesh contains all of the numerical disturbances from the inviscid and viscous regions, which are larger than the convergence criterion. The partial mesh of the local solution may overlap a part of the inviscid region and the whole viscous region, or only the viscous region. Partial meshes may be reconstructed during the numerical solution. Each reconstruction of a partial mesh defines a local solution.

The general solution procedure involves the mesh sequencing technique, the zonal modeling, and the local solution of the equations. The zonal modeling and the local solution of the equations are applied only on the finest mesh of the solution. The steps of the solution strategy are the following: 1) initiation of the numerical solution on the coarsest mesh, 2) interpolation of the solution to the finer mesh and repetition of the interpolations and the numerical solution up to the finest mesh, 3) initiation of the numerical solution on the finest mesh using the zonal modeling, 4) continuation of the iterations on the fine mesh up to where the local solution criterion is satisfied, and 5) initiation of the local solution and reconstruction, if it is necessary, of the partial meshes up to the steady-state solution. The Baldwin-Lomax[8] algebraic turbulence model has been used for the present turbulent compressible flows. As mentioned earlier, the partial mesh of the local solution may contain the viscous region, or the partial mesh may be a subset of the viscous region. In the first case eddy viscosity is calculated in the whole profile using the standard Baldwin-Lomax procedure. In the second case the functions of the turbulence model may be calculated either in the whole viscous region or in the partial mesh, which is a subset of this region. If the calculation of the model is obtained in the partial mesh, the function F_{max} of the model[8] is defined, comparing it with its previous maximum value in a profile, including the whole viscous region. This value has been stored before the construction of a partial mesh into the viscous zone.

As a flow case example, the transonic turbulent flow around an NACA 0012 airfoil, with $M_\infty = 0.55$, $a = 8.34$ deg, and $Re = 9 \times 10^6$, has been studied. The computational mesh contains 240×60 grid points. In this flow case a shock wave is formed in the chordwise location $x/c = 0.1$. Comparison between numerical and experimental results[9] for the pressure coefficient distribution is shown in Fig. 1a. The convergence behavior has been tested for three different cases: 1) fine mesh solution, 2) mesh sequencing procedure and zonal modeling, and 3) zonal-local solution (mesh sequencing, zonal modeling, and local solution). The convergence histories (Fig. 1b) justify the efficiency of the zonal-local solution methodology for the present flow case where the shock wave and the separation region are formed. The computational cost is 4700 computational work units (CWU) using only the fine mesh. A work unit is defined as the computational time on the finest mesh without local solution or zonal modeling. Comparison between cases 1, 2, and 3 shows that the local solution essentially improves the convergence. The steady-state solution is succeeded after 3850 CWU using the case 2 and after 1800 CWU using the zonal-local solution.



a) b)

Fig. 1 Results for the compressible flow $M_\infty = 0.55$, $a = 8.34$ deg, and $Re = 9 \times 10^6$ over a NACA 0012 airfoil: a) pressure coefficient distribution and b) convergence histories.

References

[1]Drikakis, D., "Development of Upwind Computational Methods for High Speed Aerodynamics," Ph.D. Dissertation, National Technical Univ. of Athens, Greece, Nov. 1991.

[2]Eberle, A., "Characteristic Flux Averaging Approach to the Solution of Euler's Equations," Von Kármán Institute Lecture Series, Vol. 1987-04, March 1987.

[3]Chakravarthy, S. R., "High Resolution Upwind Formulations for the Navier-Stokes Equations," *Computational Fluid Dynamics,* Von Kármán Institute Lecture Series, Vol. 1988-05, 1988.

[4]Schmatz, M. A., Brenneis, A., and Eberle, A., "Verification of an Implicit Relaxation Method for Steady and Unsteady Viscous and Inviscid Flow Problems," AGARD CP-437, 1988, pp. 15-1–15-33.

[5]Flares, J., Holst, T. L., Kaynak, U., Gundy, K. L., and Thomas, S. D., "Transonic Navier-Stokes Wing Solution Using Zonal Approach: Part 1. Solution Methodology and Code Validation," *Application of Compuational Fluid Dynamics in Aeronautics,* AGARD CP-412, 1986.

[6]Schmatz, M. A., Mannoyer, F., and Wanie, K. M., "Numerical Simulation of Transonic Wing Flows Using Zonal Euler, Boundary Layer, Navier-Stokes Approach," *Zeitschrift für Flugwissenschaften und Weltraumforschung,* Weltraumforsch, Vol. 13, 1989, pp. 377-384.

[7]Drikakis, D., and Tsangaris, S., "Local Solution Acceleration Method for the Solution of the Euler and Navier-Stokes Equations," *AIAA Journal,* Vol. 30, No. 2, 1992, pp. 340-348.

[8]Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA 16th Aerospace Sciences Meeting, AIAA Paper 78-527, Huntsville, AL, Jan. 1978.

[9]Harris, C. D., "Two Dimensional Aerodynamics Characteristics of the NACA 0012 Airfoil in the Langley 8-Foot Transonic Pressure Tunnel," NASA TM 81927, 1981.