Cascade Flow Calculations Using the k–ω Turbulence Model with Explicit–Implicit Solver

Chang Ho Choi* and Jung Yul Yoo†
Seoul National University,
Seoul 151-742, Republic of Korea

Introduction

THE solution of the unsteady Navier–Stokes equations incorporates implicit or explicit time-marching techniques. Implicit methods enable larger time-step and better convergence characteristics than explicit methods. However, explicit methods combined with convergence acceleration techniques have recently proven to be very effective for solving steady flow problems.[1] Therefore, in the present study, a four-stage Runge–Kutta scheme, which is a typical explicit method, is adopted to solve the compressible Navier–Stokes equations.

Traditionally, many researchers have used algebraic stress models[1] in the cascade flow calculations that do not involve any source terms. But these models have difficulties in predicting flows associated with large adverse pressure gradient or separation region.[2] Among two-equation eddy-viscosity models, the k–ε model has been most widely used. However, a major difficulty with the k–ε model is related to its application to near-wall turbulent flows where inaccuracy and numerical stiffness may arise.[3] One of the most notable alternatives is the k–ω model proposed by Wilcox.[4] Very encouraging results for separated flows have been reported by Menter.[5] Because it does not require any damping functions in the viscous sublayer, it is mathematically simpler than the other two-equation models in the near-wall region. However, the k–ω model involves large source terms, which may affect the stability of the explicit time-stepping scheme. To alleviate this stability problem, an implicit approximate factorization scheme, which can be solved by a tridiagonal matrix (TDMA) solver, is developed in the present study. In fact, Turner and Jennions[6] also attemped this mixed time-stepping scheme using the k–ε model based on implicit staggered grid method as described by Patankar,[7] which is widely used in the calculation of incompressible flows.

In brief, a numerical method is suggested that solves the compressible Navier–Stokes equations and the k–ω turbulence model equations with a mixed explicit–implicit time stepping scheme.

Numerical Method

An explicit four-stage Runge–Kutta time-stepping scheme is adopted for the Navier–Stokes equations. Second-order central differences are used for the spatial discretization, and artificial dissipation terms[1] with proper scaling are included to stabilize the numerical solutions. Local time-stepping and residual smoothing methods are employed to accelerate convergence of the time-marching schemes, as in Ref. 1. The k–ω equations are numerically decoupled from the Navier–Stokes equations through the use of an approximate factorization scheme as follows:

$$\left[I + \Delta t\left\{\frac{\partial}{\partial \xi}\left(\frac{\partial \boldsymbol{E}_c}{\partial \boldsymbol{Q}} - \frac{\partial \boldsymbol{E}_v}{\partial \boldsymbol{Q}}\right)\right\}\right] \times \left[I + \Delta t\left\{\frac{\partial}{\partial \eta}\left(\frac{\partial \boldsymbol{F}_c}{\partial \boldsymbol{Q}} - \frac{\partial \boldsymbol{F}_v}{\partial \boldsymbol{Q}}\right) - \frac{\partial \boldsymbol{S}}{\partial \boldsymbol{Q}}\right\}\right] \cdot \Delta \boldsymbol{Q} = R^{*n} \tag{1}$$

$$R^{*n} = \Delta t\left[\frac{\partial}{\partial \xi}\left(\boldsymbol{E}_v^n - \boldsymbol{E}_c^n\right) + \frac{\partial}{\partial \eta}\left(\boldsymbol{F}_v^n - \boldsymbol{F}_c^n\right) + \boldsymbol{S}^n\right]$$

where the vectors $\boldsymbol{E}_c$ and $\boldsymbol{F}_c$ account for the convective terms, $\boldsymbol{E}_v$ and $\boldsymbol{F}_v$ are the viscous terms, and $\boldsymbol{S}$ is the source term. A first-order upwind scheme is used for the convective terms, with central differences used for the diffusive terms. Proper evaluation of source terms is crucial to the stability of the computation. In the present study, the source terms are linearized in time with the same approximation method as Wilcox.[8] Therefore, a TDMA or cyclic TDMA (CTDMA) solver can be used, depending on the boundary conditions, resulting in less computation time than other implicit methods. Local time stepping is also employed for the k–ω equations.

Results and Discussion

To validate the present numerical method, computations have been performed to simulate the flows through the von Kármán Institute (VKI) gas turbine cascade.[9] Because of the blunt trailing edge and large turning angle of the VKI turbine cascade, it is difficult to generate a grid that is smooth near the trailing edge by using an H-type or a C-type grid, particularly when a low-Reynolds-number-type turbulence model, which needs extremely stretched grid near the wall, is used. In the present work, a 200 × 56 O-type grid is used (as shown in Fig. 1) because it enables smooth grid distribution near the trailing edge and, thus, requires less grid points. An initial grid is generated using an elliptic grid generator, and then an algebraic grid generator is put in action to cluster grid lines effectively near the wall. Convergence histories for the transonic flow case are given in Fig. 2. For this case it took about 1850 time steps and 3196 s on the HP735 workstation for a residual drop of 10^{-5}, after which there were only negligible changes in the solutions. In Fig. 3, the computed and experimental isentropic surface Mach number distributions for subsonic and transonic flow conditions are shown to be in good agreement, particularly near the trailing edge of the blade, which are due to smooth distribution of an O-type grid around the trailing edge.

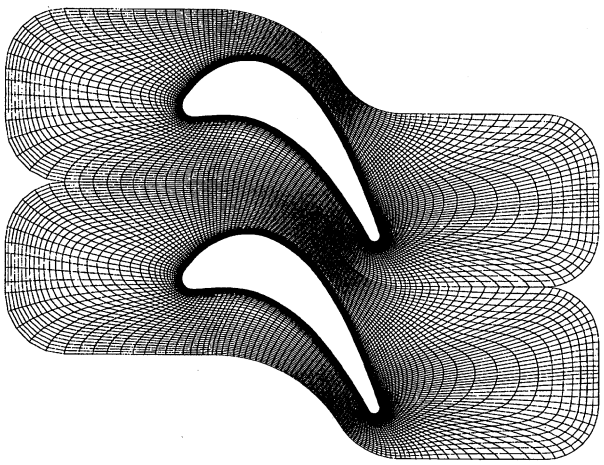

Fig. 1 VKI turbine cascade flow calculation, 200 × 56 O-type grid.

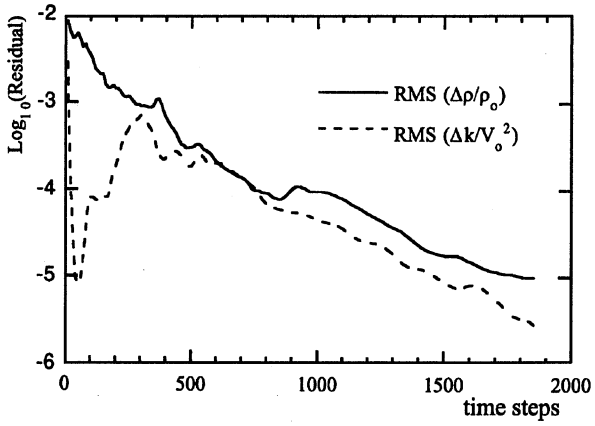

Fig. 2 Convergence histories for the VKI turbine cascade flow calculation.

Received Dec. 28, 1995; revision received May 10, 1997; accepted for publication May 27, 1997. Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Ph.D. Student, Department of Mechanical Engineering.
†Professor, Department of Mechanical Engineering.

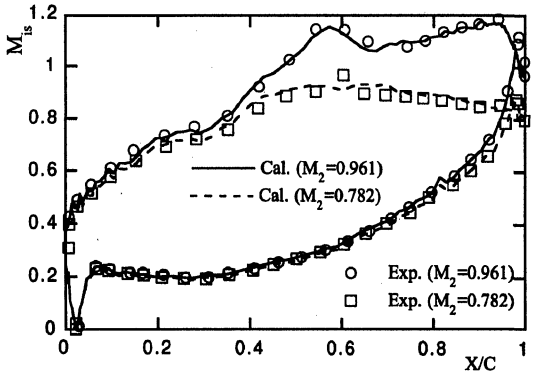

Fig. 3 Isentropic surface Mach number distributions for different values of M_2.

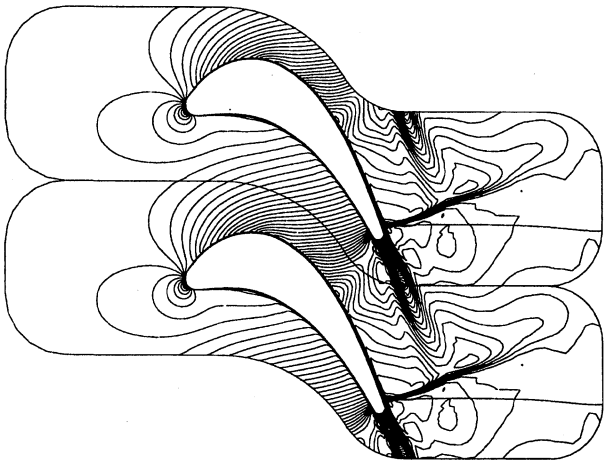

Fig. 4 Mach number contours for the VKI turbine cascade flow, M_2 = 0.961.

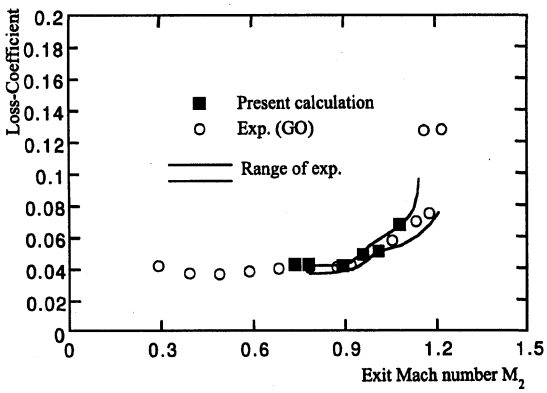

Fig. 5 Variation of loss coefficient with exit Mach number.

Figure 4 shows a clean capturing of the shock for transonic flow calculation. Figure 5 presents a comparison between experimental and computed loss coefficients. Calculated results agree favorably with the experimental data in both behavior and absolute values.

Conclusion

Subsonic and transonic flows through the VKI turbine cascade are calculated adopting the k–ω turbulence model. An explicit fourth-order Runge–Kutta solver for the Navier–Stokes equations and an implicit approximate factorization scheme for the k–ω equations are proposed. This mixed explicit–implicit time-marching scheme has proven to be fast, stable, and accurate, requiring less computer capacity than conventional fully implicit time-stepping schemes.

Acknowledgments

The authors would like to express their gratitude to the Turbo and Power Machinery Research Center of Seoul National University and Samsung Aerospace Industries, Ltd., for their support extended through the course of the present study.

References

[1]Arnone, A., and Swanson, R. C., "A Navier–Stokes Solver for Turbomachinery Applications," *Journal of Turbomachinery*, Vol. 115, No. 2, 1993, pp. 305–313.

[2]Lakshminarayana, B., "An Assessment of Computational Fluid Dynamic Techniques in the Analysis and Design of Turbomachinery—The 1990 Freeman Scholar Lecture," *Journal of Fluids Engineering*, Vol. 113, No. 3, 1991, pp. 315–352.

[3]Patel, V. C., Rodi, W., and Scheuerer, G., "Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review," *AIAA Journal*, Vol. 23, No. 9, 1985, pp. 1308–1319.

[4]Wilcox, D. C., "Reassessment of the Scale-Determining Equation for Advanced Turbulence Models," *AIAA Journal*, Vol. 26, No. 11, 1988, pp. 1299–1310.

[5]Menter, F. R., "Performance of Popular Turbulence Models for Attached and Separated Adverse Pressure Gradient Flows," AIAA Paper 91-1784, June 1991.

[6]Turner, M. G., and Jennions, I. K., "An Investigation of Turbulence Modelling in Transonic Fans Including a Novel Implementation of an Implicit k–ε Turbulence Model," *Journal of Turbomachinery*, Vol. 115, No. 2, 1993, pp. 249–260.

[7]Patankar, S. V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, New York, 1980.

[8]Wilcox, D. C., "Progress in Hypersonic Turbulence Modelling," AIAA Paper 91-1785, 1991.

[9]Kiock, R., Lehthaus, F., Baines, N. C., and Sieverding, C. H., "The Transonic Flow Through a Plane Turbine Cascade as Measured in Four European Wind Tunnels," *Journal of Engineering for Gas Turbines and Power*, Vol. 108, No. 2, 1986, pp. 277–284.

S. Fleeter
Associate Editor

Numerical Simulations of Three-Dimensional Trailing Vortex Evolution

Robert E. Robins* and Donald P. Delisi†
NorthWest Research Associates, Inc., Bellevue, Washington 98009-3027

I. Introduction

IT is well known that trailing vortices in unstratified, unsheared fluid, out of ground effect, will undergo an instability that results in the periodic linking of the initially parallel vortex tubes and the formation of vortex rings. Evidence of this instability was first noted by Scorer.[1]

Crow[2] first explained this phenomenon (hence, it is now called Crow instability) by performing a perturbation analysis of the kinematic relation between velocity and vorticity and found that 1) the wavelength of maximum perturbation growth (which we denote as λ^*) is $8.6b_0$, where b_0 is the initial distance between the parallel vortices; 2) the perturbation amplitude grows by a factor of e ($\cong$ 2.72) in a time equivalent to $1.21T_0$, where T_0 is the time required for the vortices to descend a distance equal to b_0, i.e., $T_0 = b_0/V_0$, where V_0 is the initial descent speed, given by $\Gamma_0/2\pi b_0$, Γ_0 being the initial magnitude of the circulation about each of the vortices; 3) the angle to the horizontal of the plane in which the initial maximum growth occurs is 48 deg; and 4) λ^* increases as the size of the vortex cores decreases. Result 2 is characterized by

Received Nov. 7, 1995; revision received March 3, 1997; accepted for publication May 27, 1997. Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Research Scientist, P.O. Box 3027. Member AIAA.
†Senior Research Scientist, P.O. Box 3027. Member AIAA.