References

[1]Farassat, F., and Succi, G. P., "The Prediction of Helicopter Rotor Discrete Frequency Noise," *Vertica*, Vol. 7, No. 4, 1983, pp. 309–320.

[2]Katz, J., and Plotkin, A., *Low-Speed Aerodynamics*, McGraw–Hill, New York, 1991, pp. 237–263.

[3]Lee, J., "A Potential Based Panel Method for the Analysis of Marine Propellers in Steady Flow," Ph.D. Dissertation, Dept. of Ocean Engineering, Massachusetts Inst. of Technology, Cambridge, MA, July 1987.

P. J. Morris
Associate Editor

Navier–Stokes Prediction of Internal Flows with a Three-Equation Turbulence Model

S. Duranti* and F. Pittaluga†
University of Genoa, 16145 Genoa, Italy

Nomenclature

G = compressible shear-layer growth rate g normalized by its incompressible counterpart g_i ($G = g/g_i$; $g = d\delta_\omega/dx$)
K = turbulent kinetic energy
K_ρ = density variance ($K_\rho = \langle\rho'^2\rangle$)
M_C = convective Mach number in a two-stream mixing layer ($M_C = |U_1 - U_2|/(a_1 + a_2)$, a being the sound speed)
U^* = normalized X-component mean velocity [$U^* = (U - U_2)/(U_1 - U_2)$]; U_1 and U_2 are U in either stream of a two-stream mixing layer
$-\langle u'v'\rangle$ = Reynolds stress (ensemble mean)
V = Y-component mean velocity
Y_2^* = nondimensional y coordinate $\{Y_2^* = [(y - y_0)/\delta_\omega]\}$
δ_ω = vorticity thickness $\{\delta_\omega = (U_1 - U_2)/[\partial U/\partial y]_{MAX}\}$
ε = dissipation rate
ρ' = density fluctuation

Introduction

THE strategy most usually adopted for improving the numerical prediction of turbulent flows within the frame of classical K–ε, or similar, models is that of better calibrating these latter, or simply their coefficients, by pursuing interpretation of, or seeking accordance to, flow features such as velocity profiles, turbulence levels, dissipation rates, etc., with reference to specific flow situations. At difference with the preceding, the modelization effort here pursued numerically has been that of reinterpreting the classical two-equation turbulence model with the aim of capturing, at a fundamental level, the effects of the turbulence scales interaction: basically, this implies finding a proper analytical means suitable to represent the most relevant statistical quantities of the fluctuating field. In incompressible turbulence this strategy has already allowed both to improve the eddy viscosity formulation by taking care of the anisotropy of the turbulence field and to capture the higher-order effects in the shear-stress behavior.[1–3] In more general terms, it is here believed that the implications of the preceding perspective are very much promising and, at the same time, usually overlooked by the turbulence modelers, inducing the result that otherwise sophisticated schemes, such as the full Reynolds-stress models, turn out intrinsically weakened if this kind of statistical information is lacking. Such a point appears even more important when the turbulent flowfield is compressible or subject to thermally induced density variations.[4,5]

Received 22 June 1999; presented as Paper 99-3372 at the AIAA 14th Computational Fluid Dynamic Conference, Norfolk, VA, 28 June–1 July 1999; revision received 8 November 1999; accepted for publication 19 November 1999.

Copyright © 2000 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Ph.D. Student, Department of Thermal Machines, Energy Systems, and Transportation.

†Professor, Department of Thermal Machines, Energy Systems, and Transportation. Associate Fellow AIAA.

Navier–Stokes Solver with the Three-Equation Turbulence Model

To the said end, the approach adopted is the so-called TSDIA[1,4] method (two-scale direct-interaction approximation), extended with the latest compressible-turbulence developments [Markovianized two-scale method (MTS[5])]. Scale-expansion equations are written and solved directly in terms of fluctuations, and all of the involved correlations appear ready to be evaluated. Several correlations show up and require modelization, both in the governing Navier–Stokes equations and in the compressible-turbulence model: this latter turns out made up by a transport equation for each one of the three main turbulence quantities K, ε, K_ρ, namely the turbulent kinetic energy, the dissipation rate, and the density variance.[4–6]

From a numerical point of view, the preceding three-equation turbulence model has been implemented into a new version of NAST,[6,7] a three-dimensional time-dependent Navier–Stokes structured solver belonging, at large, to the Arbitrary Lagrangian–Eulerian codes' family, of which the series of Kiva[8] codes are the main representatives. To implement correctly the turbulence model has required conversion of the Favre averages previously adopted in NAST into ensemble-mean expressions: it is exactly this procedure that allows us to explicitly arrive at the fluctuating-field correlations to be modeled via the TSDIA/MTS methodologies, yielding a second-order representation of the Reynolds stresses.[4–6] In the last version of NAST, the governing equations are discretized explicitly for the diffusive and the convective terms, while an implicit Poisson-like pressure equation is obtained and solved via multigrid technique.[6]

Results and Discussion

A few comparisons are now presented between the (ensemble-mean) predictions of the NAST code and two turbulent test cases, one (numerical, direct numerical simulation) referred to as an isothermal, low-Mach-number, backward-step flow situation[8] and the other (experimental) to a compressible variable-property free-shear flow regime.[9] The complete results have recently been presented in all details.[6]

With reference to the first test case, the Reynolds number based on freestream velocity and height of the step is 5.1×10^3, whereas the Reynolds number based on momentum thickness is 6.67×10^2. The calculations were performed on a 161×71 rectangular mesh, with some grid coarsening for Y/H higher than 3.

Figures 1 and 2 show, respectively, the horizontal velocity component and the Reynolds shear stress. The picture turns out everywhere good and in many instances excellent. The normal Reynolds-stress

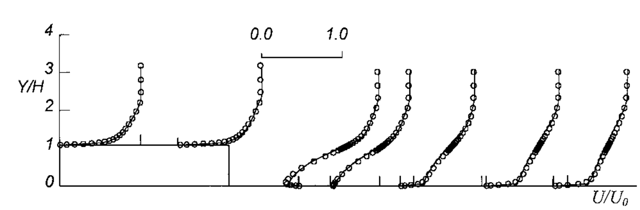

Fig. 1 Horizontal velocity (——, NAST; ⊘, Le and Moin[9]).

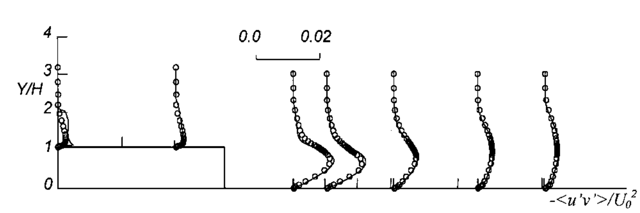

Fig. 2 Reynolds shear stress (——, NAST; ⊘, Le and Moin[9]).

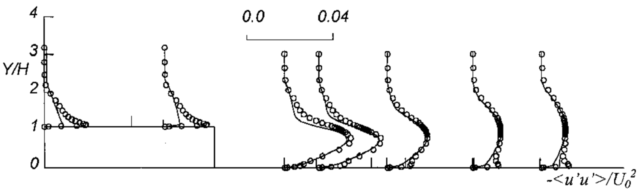

Fig. 3 Horizontal Reynolds stress (——, NAST; ⊘, Le and Moin[9]).

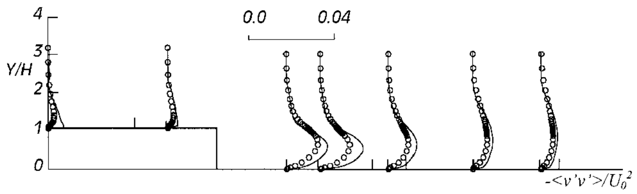

Fig. 4 Vertical Reynolds stress (——, NAST; ⊘, Le and Moin[9]).

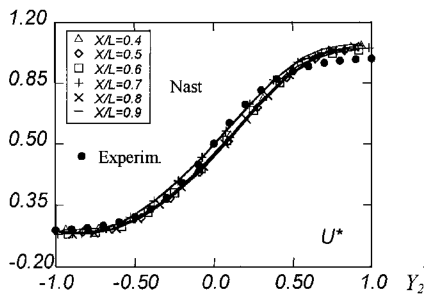

Fig. 5 Normalized velocity for $M_C = 0.51$.

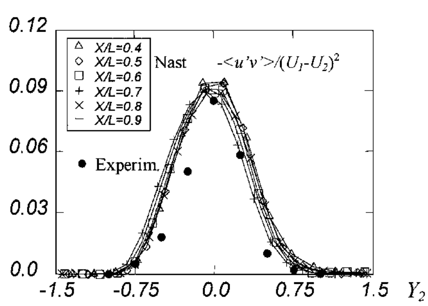

Fig. 6 Normalized Reynolds stress for $M_C = 0.51$.

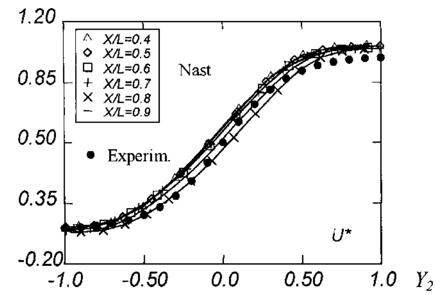

Fig. 7 Normalized velocity for $M_C = 0.64$.

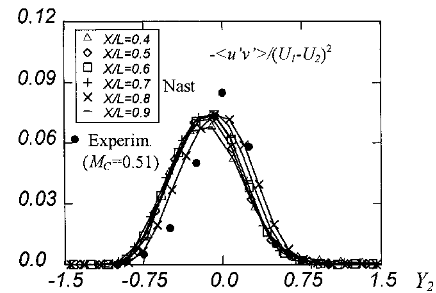

Fig. 8 Normalized Reynolds stress for $M_C = 0.64$.

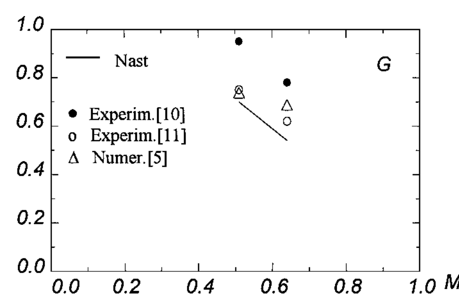

Fig. 9 Normalized growth rate.

behavior (very seldom presented and discussed) is shown in Figs. 3 and 4, respectively, for the horizontal and vertical components. Some underpredictions are visible upstream of the step, but better predictions are attained downstream, where the turbulence level is higher. Though the vertical velocity fluctuation correlations are generally overpredicted, observe how the second-order extension to Reynolds-stress representation allows us to describe quite well the anisotropy in the turbulence field.

For compressible turbulence and variable-density flows the dilatation effects and the gas property fluctuations bring in some entirely new turbulence characteristics, making the picture much more complicated. Moreover, the lack of suitable boundary conditions for the new correlations to be modeled hardly help investigating flow features with the same accuracy as in the incompressible field. In this context the choice of a suitable test case has fallen upon a wall-free compressible and variable-density mixing layer,[10] owing also to its relatively simple overall flowfield geometry.

Two series of calculations were performed: one with a convective Mach number M_C equal to 0.51 and the other with M_C equal to 0.64. A uniform, 81×51 cells, rectangular grid was adopted. The inlet boundary conditions could be set directly with the two freestream velocities without any boundary layer. The inlet K was set to 7% of mean velocity squared (improved inlet conditions are presently under consideration). Results for $M_C = 0.51$ show a quite satisfactory prediction of velocity profiles (Fig. 5), although they do not collapse exactly onto the experimental results, probably because of the rather poor resolution of the mesh or to still inadequate settings for the nondimensional constants of turbulence model. The shear-stress prediction is rather good with the peak value nicely predicted, although the exact shape of the bell is not caught (Fig. 6). Results for $M_C = 0.64$ are shown in Figs. 7 and 8. The normalized growth rate is slightly lower than the experimental (Fig. 9), but hopes are that things will improve in a very near future, as the research here discussed is still under way. Additional reference data, both experimental[11] and theoretical,[5] are given, for further comparison, in the same Fig. 9. The trend of G as a function of M_C is correctly captured.

Conclusions

The three-equation turbulence model presented here, belonging at large to the eddy-viscosity models family, appears as featuring substantial variances from it. These can be traced both in the possibility of interpreting the anisotropic effects as induced by the largest eddies, and in the treatment, in terms of turbulence modelization performed at a fundamental level, of density-fluctuation-related correlations. This means that such a model is suitable for general classes of flows, having no a priori physical limitations but the ones imposed by the adoption of the energy spectrum dependence for compressible (and incompressible) turbulence. The test-case validations here reported attest the high predictive capacity of the solver, which appears excellent in the first case and good in the second one. Even in this case the most relevant flow features and tendencies are correctly captured.

References

[1]Yoshizawa, A., "Statistical Analysis of the Deviation of the Reynolds Stress from Its Eddy-Viscosity Representation," *Physics of Fluids*, Vol. 27, No. 6, 1984, pp. 1377–1387.

[2]Duranti, S., and Pittaluga, F., "Improvement of Navier–Stokes ALE Predictions by Adoption of a TSDIA Turbulence Model," *Proceedings of COST Action F1 Workshop on 3D Navier–Stokes Codes: Numerics and Modeling*, Paper 7/2, Courchevel, France, Jan. 1997.

[3]Duranti, S., Pezzuto, G., and Pittaluga, F., "Numerical Prediction of Neutral Atmospheric Boundary Layers," Symposium EACWE-2 on Wind Engineering," *Proceedings of the 2nd EACWE (European and African Conference on Wind Engineering)*, S. G. E., Padua, Italy, pp. 341–348; also *International Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 74–76, Elsevier, Amsterdam, 1998, pp. 263–273.

[4]Yoshizawa, A., "Simplified Statistical Approach to Complex Turbulent Flows and Ensemble-Mean Compressible Turbulence Modeling," *Physics of Fluids*, Vol. 7, No. 12, 1995, pp. 3105–3177.

[5]Yoshizawa, A., Liou, W. W., Yokoi, N., and Shih, T. H., "Modeling of Compressible Effects on the Reynolds Stress Using a Markovianized Two-Scale Method," *Physics of Fluids*, Vol. 9, No. 10, 1997, pp. 3024–3036.

[6]Duranti, S., and Pittaluga, F., "Prediction of Complex Internal Flows by Means of a Three-Equation Turbulence Model," AIAA Paper 99-3372, July 1999.

[7]Pittaluga, F., "NAST: An Advanced 3D Compressible Navier–Stokes Solver," *Proceedings of COST Action F1 Workshop on Complex Viscous Flows*, Paper 13/2, Ecole Polytechnique Federale de Lausanne (EPFL), Lausanne, Switzerland, Oct. 1993.

[8]Amsden, A. A., Butler, T. D., and O'Rourke, P. J., "KIVA-II: A Computer Program for Chemically Reactive Flows with Sprays," Los Alamos National Lab., Rept. LA-11560-MS, Los Alamos, NM, May 1989.

[9]Le, H., and Moin, P., "Direct Numerical Simulation of Turbulent Flow over a Backward Facing Step," Center for Turbulence Research, Annual Research Briefs, Stanford Univ., Stanford, CA, 1992, pp. 161–173.

[10]Samimy, M., and Elliott, G. S., "Effects of Compressibility on the Characteristics of Free Shear Layers," *AIAA Journal*, Vol. 28, No. 3, 1990, pp. 439–445.

[11]Goebel, S. G., and Dutton, J. C., "Experimental Study of Compressible Turbulent Mixing Layers," *AIAA Journal*, Vol. 29, No. 4, 1991, pp. 538–546.

R. M. C. So
Associate Editor

Vortex Method Simulation of the Flow Around a Circular Cylinder

Angelo A. Mustto* and Gustavo C. R. Bodstein†
Federal University of Rio de Janeiro,
21945-970 Rio de Janeiro, Brazil
and
Miguel H. Hirata‡
Federal Engineering School of Itajuba,
37500-000 Itajuba, Minas Gerais, Brazil

Nomenclature

C_D, C_L	= drag and lift coefficients, respectively
k	= numerical parameter
N	= number of vortices created every time step
N_T	= total number of vortices present in the flow
P, Q	= random numbers between 0 and 1, drawn from a uniform density distribution
Re, Sr	= Reynolds and Strouhal numbers, respectively, based on the cylinder diameter
r	= radial distance from a vortex
$t, \Delta t$	= time and time step, respectively
u, v	= x and y components of velocity field
x, y	= Cartesian coordinates
$z, \bar{z}$	= complex variable, $x + iy$, and complex conjugate
Γ	= vortex strength (positive counterclockwise)
$\Delta r, \Delta\theta$	= radial and polar angle random increments, respectively
ε	= distance between generated vortices and cylinder surface
σ	= vortex core radius
ω	= vorticity

Subscripts

c	= convection
d	= diffusion
im	= inverse image of a vortex
θ	= circumferential direction

Introduction

THE flow around a circular cylinder is characterized by several different regimes, depending on the value of the Reynolds number. These regimes range from steady Stokes-type flows to strongly unsteady flows, where a von-Kármán-type periodic wake is formed. Among the methods reported in the literature that have been used to simulate this flow, the (Lagrangian) discrete vortex method[1–3] has been playing an important role. In this work, we propose a novel algorithm of the vortex method that is able to produce effective calculations of global quantities for flows around bluff bodies. Our objectives are 1) to develop an alternative algorithm within the Lagrangian discrete vortex method framework, where a new vorticity generation scheme is implemented; 2) to identify the influence of the numerical parameters; and 3) to calculate mean quantities of the flow and compare the results to others available in the literature.

Mathematical Background

We consider the two-dimensional, incompressible, unsteady and high-Reynolds-number flow around a circular cylinder, immersed in an unbounded region with a uniform flow upstream. This flow, started impulsively from rest, is governed by the continuity and the Navier–Stokes equations, subject to the impermeability and no-slip conditions on the cylinder, and uniform flow at infinity. Alternatively, we can use the nondimensional vorticity equation in the form

$$\frac{\partial\omega}{\partial t} + \boldsymbol{u}\cdot\nabla\omega = \frac{2}{Re}\nabla^2\omega \tag{1}$$

All of the equations are made nondimensional by the freestream speed and the cylinder radius. Because the vorticity is modeled as a cloud of discrete vortices, we can use the circle theorem[4] to construct the velocity field such that it satisfies the continuity equation, the condition at infinity, and the impermeability condition. Thus, a general expression for the complex velocity is

$$u - iv = \left(1 - \frac{1}{z^2}\right) - \frac{i}{2\pi}\sum_{n=1}^{N_T}\frac{\Gamma_n}{z - z_n(t)} + \frac{i}{2\pi}\sum_{n=1}^{N_T}\frac{\Gamma_n}{z - z_{\mathrm{im},n}(t)} \tag{2}$$

where $z_{\mathrm{im},n}(t) = 1/\bar{z}_n(t)$. From the unsteady Blasius formula,[4] the aerodynamic forces can be written as

$$C_D + iC_L = -i\sum_{n=1}^{N_T}\Gamma_n[(u_n + iv_n) - (u_{\mathrm{im},n} + iv_{\mathrm{im},n})] \tag{3}$$

Numerical Method and Implementation

The numerical method is implemented in five steps: 1) vorticity generation, 2) calculation of the forces on the body, 3) convection of the vortices, 4) diffusion of the vortices, 5) elimination of some vortices, and 6) stepping in time. The vorticity generation is carried out as follows. At each time step, N new vortices are created a small distance ε off the body surface, at a radial distance $1 + \varepsilon$, with a uniform angular distribution. Their strengths are determined by imposing the no-slip condition at $N - 1$ points on the cylinder

Presented as Paper 98-2409 at the AIAA 16th Applied Aerodynamics Conference, Albuquerque, NM, 15–18 June 1998; received 5 February 1999; revision received 3 November 1999; accepted for publication 16 December 1999. Copyright © 2000 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Graduate Student, Department of Mechanical Engineering, EE/COPPE, C.P. 68503.

†Associate Professor, Department of Mechanical Engineering, EE/COPPE, C.P. 68503. Senior Member AIAA.

‡Visiting Professor, Department of Mechanics, IEM, C.P. 50.