References

[1]Dolling, D. S., and Grossman, B. (eds.), "CFD in Design," AIAA 12th Computational Fluid Dynamics Conf., San Diego, CA, June 1995.

[2]Vanderplaats, G. N., *Numerical Optimization Techniques for Engineering Design: With Applications*, McGraw–Hill, New York, 1984, pp. 153–226.

[3]Press, W. H., Teukolsky, S. A., Vetterling, W. T., and Flannery, B. P., *Numerical Recipes in FORTRAN: The Art of Scientific Computing*, 2nd ed., Cambridge Univ. Press, Cambridge, England, UK, 1992, pp. 436–448.

[4]Goldberg, D. E., *Genetic Algorithms in Search, Optimization and Machine Learning*, Addison–Wesley, Reading, MA, 1989, pp. 59–88.

[5]Quagliarella, D., and Cioppa, A. D., "Genetic Algorithms Applied to the Aerodynamic Design of Transonic Airfoils," *Journal of Aircraft*, Vol. 32, No. 4, 1995, pp. 889–891.

[6]Doorly, D., "Parallel Genetic Algorithms for Optimization in CFD," *Genetic Algorithms in Engineering and Computer Science*, edited by G. Winter, J. Periaux, M. Galán, and P. Cuesta, Wiley, Chichester, England, UK, 1995, pp. 251–270.

[7]Katz, J., and Plotkin, A., *Low-Speed Aerodynamics: From Wing Theory to Panel Methods*, International ed., McGraw–Hill, New York, 1991, pp. 301–377.

[8]Vanderplaats, G. N., "ADS—A FORTRAN Program for Automated Design Synthesis, Version 3.00," Engineering Design Optimization, Inc., Santa Barbara, CA, March 1988.

A. Plotkin
Associate Editor

Suitability of Upwind-Biased Finite Difference Schemes for Large-Eddy Simulation of Turbulent Flows

Rajat Mittal* and Parviz Moin†
Stanford University, Stanford, California 94305

Introduction

LARGE-EDDY simulation (LES) with the dynamic model[1,2] produces good results when used in conjunction with spectral-method-based solvers.[3] The dynamic modeling procedure utilizes information from the small scales of the flowfield, which are typically not corrupted by numerical errors in spectral simulations. However, spectral methods are usually limited to simple geometries, and for complex configurations, conventional finite difference methods are used. A fifth-order one-point upwind-biased scheme has been successfully used in well-resolved direct numerical simulations of turbulent flows,[4] and it was thought that the high resolving power and relatively low numerical dissipation of such a scheme would make them useful in LES of flows in complex geometries. To study the utility of these schemes for LES, Beaudan and Moin[5] employed them in a series of simulations of flow past a circular cylinder at $Re = 3.9 \times 10^3$, where the Reynolds number is based on the diameter D and freestream velocity U_∞. This is a challenging flow for the LES methodology because it contains features such as thin laminar boundary layers, unsteady separation, and transitional shear layers. The particular Reynolds number was chosen owing to the availability of two experimental data sets. Lourenco and Shih used a particle image velocimetry technique to measure turbulence statistics in the near-wake region including the recirculation zone. Ong and Wallace[6] used a hot-wire probe for measuring mean velocity and stress profiles in the downstream wake region from $x/D = 3$ to 10.

Beaudan and Moin[5] carried out simulations with no subgrid-scale (SGS) model, with a fixed coefficient Smagorinsky model, and with the spanwise averaged version of the dynamic model[1,2] and observed that mean wall statistics such as drag, wall pressure coefficient, wall shear stress, and separation angles were not significantly different in the three simulations and all showed reasonable agreement with experimental data. The most significant finding of these simulations came from the comparison of the computed solution with the experiments in the region downstream of the vortex formation region ($5.0 < x/D < 10.0$) where the mesh was relatively coarse. It was found that, in this region, numerical dissipation overwhelmed the contribution of the SGS model, and the three computed solutions were virtually indistinguishable beyond $x/D > 7.0$. The simulation with a seven-order scheme also showed that energy in a substantial portion of the resolvable wave number range was damped due to numerical dissipation, and it was concluded that these high-order upwind-biased schemes were unsuitable for use in LES.

Analysis of the truncation error[5] indicates that higher-order upwind-biased schemes provide good resolution in about two-thirds of the wave number range, and the upper-half of the wave number range is affected by numerical dissipation. In contrast to upwind-biased schemes, which control aliasing errors through numerical dissipation, in central schemes aliasing must be controlled by enforcing kinetic energy conservation. Such schemes do not exhibit numerical dissipation and, therefore, there is no spurious damping of the smaller scales. This feature makes the schemes attractive for use in LES, and a number of complex flows have been successfully simulated using a second-order central difference scheme on a staggered mesh.[7,8] One disadvantage of using these schemes is the dominance of dispersive error, which makes them extremely sensitive to aspects such as the grid stretching factors and outflow boundary conditions. These issues apparently do not present difficulties in simulations of relatively simple flows such as channel flow or flat plate boundary layers but are critical when simulating flows in complex geometries. Thus, even though the central schemes seem more attractive for LES, the advantage of these schemes over the higher-order upwind schemes for LES in complex flows needs to be established, and this is the main motivation for the current study.

We have simulated flow past a circular cylinder at a Reynolds number of 3.9×10^3 using a solver that employs an energy-conservative second-order central difference scheme for spatial discretization. Detailed comparisons of turbulence statistics and energy spectra in the downstream wake region ($7.0 < x/D < 10.0$) have been made with the results of Beaudan and Moin[5] and with experiments[4] to assess the impact of numerical diffusion on the flowfield. Based on these comparisons, conclusions are drawn on the suitability of higher-order upwind schemes for LES in complex geometries.

Simulation Methodology

The solver used in the current work is based on the numerical scheme developed by Choi et al.,[9] which solves the incompressible Navier–Stokes equations in generalized coordinates on a spanwise periodic domain. The governing equations are written in terms of the volume fluxes, and the in-plane (x–y plane) volume fluxes and pressure are discretized on a fully staggered grid using a second-order central difference scheme. The spanwise volume flux is collocated at the pressure node, and a Fourier spectral collocation method is used in the spanwise direction. Dealiasing is performed in the spanwise direction using a two-thirds truncation rule to make the numerical scheme energy conservative.

A C-mesh is used for the present simulation. The branch cut of the mesh is located along the wake centerline and the inflow and outflow boundaries are located at $19D$ and $17D$, respectively. Furthermore, the vertical extent of the outflow boundary is about $25D$ from the wake centerline. Uniform freestream velocity is prescribed at the inflow and far-field boundaries, and a convective boundary condition is employed at the outflow boundary to smoothly convect the disturbances out of the computational domain. The spanwise domain size L_z is chosen equal to πD, which is the same as that used by Beaudan and Moin.[5] It has been found that in LES, where the resolution is at best marginal, central schemes can tolerate only a small streamwise stretching factor (< 3%); higher stretching factors can lead to the amplification of grid-to-grid oscillations (2-Δ waves). If an O-type

Received Dec. 9, 1996; revision received April 16, 1997; accepted for publication April 30, 1997.

*Research Fellow, Center for Turbulence Research; currently Assistant Professor, Department of Mechanical Engineering, University of Florida, Gainesville, FL 32611-6300.

†Director, Center for Turbulence Research.

mesh similar to the one used by Beaudan and Moin[5] were to be used, the flow in the region of the separated shear layer would experience large streamwise grid stretching as it would go from being aligned with one family of grid lines to the other, and the flowfield would be contaminated with 2-Δ waves. This problem does not arise when a C-mesh is used. In the downstream region of the wake where detailed comparisons between the simulations will be made, both grids are nearly Cartesian and, therefore, the grid topology is not a factor.

The solution is advanced in time using a second-order accurate, semi-implicit fractional step scheme where the convection-diffusion terms are advanced followed by the pressure correction step.[9] A version of the dynamic model suitable for application in generalized coordinates has been used. Details of the dynamic SGS modeling procedure can be found in Refs. 1 and 2. Test filtering is performed in the streamwise and spanwise directions, and the least-squares minimization approach[10] is used for obtaining the model coefficient. Furthermore, the model coefficient is obtained as a spanwise averaged quantity, and the total viscosity is constrained to be positive through clipping of large negative eddy viscosity values. The current simulation has been carried out on a 401 × 120 × 48 (streamwise × wall normal × spanwise) mesh with a time-step size ($U_\infty \Delta t/D$) of about 0.007. All statistics for the current simulation have been averaged over about 12 shedding cycles. The data of Beaudan and Moin[5] used here are from their simulation that employed a fifth-order upwind-biased scheme.

Description of Results

It is found that key wall statistics (mean base pressure coefficient, mean drag coefficient, Strouhal number, and mean separation angle) obtained from the current central difference-based simulation are in good agreement with experiments and show less than 2% deviation from the results of the upwind-biased simulations.[11] Furthermore, mean velocity profiles in the near wake ($x/D < 3.0$) also compare reasonably well with the profiles obtained from the upwind-biased simulations.[11] Inasmuch as the drag and base pressure coefficient depend strongly on accurate prediction of near-wake features such as vortex rollup and formation of streamwise vortical structures, good prediction of these quantities suggests that the development and evolution of the vortical structures in the near wake is being simulated reasonably accurately.

Because the flowfields in the near wake obtained from the two simulations are in reasonable agreement with each other, we expect that difference in the downstream portion of the flow ($5.0 < x/D < 10.0$) will be primarily due to differences in the resolution. Thus, comparison of the statistics in the downstream portion of the wake should allow us to compare the performance of the different schemes. Also, based on the estimate of the Kolmogorov length scale at these downstream locations provided by Ong and Wallace,[6] Beaudan and Moin[5] determined that the flow was about three to four times better resolved in the vertical and spanwise directions as compared to the streamwise direction in their simulations. In the current simulation, too, the flow is better resolved in these directions, thus allowing us to base the comparison between the two simulations solely on the streamwise resolution. The streamwise grid spacing in the current simulation is 20–30% smaller than Beaudan and Moin's between $x/D = 5$ and 7. However, at $x/D = 10.0$ both simulations have roughly the same streamwise grid spacing. This difference cannot be avoided because, as mentioned earlier, the grid cannot be stretched in the streamwise direction as fast in the central difference simulation as was done in the upwind-biased simulations.

In Fig. 1 we have plotted the one-dimensional frequency spectra of the streamwise velocity at two locations in the downstream region of the wake. The current solver utilizes a variable time-step size and, therefore, the time series obtained is not evenly sampled. To obtain the spectra from these unevenly sampled data, we have used the Lomb periodogram with an oversampling factor of four.[12] Spectra from both simulations and experiment[6] are plotted together for comparison. The streamwise grid spacing limits the highest frequency that can be locally resolved in the simulation, and this represents the implicit filter that is imposed by the grid on the flowfield. The vertical lines in the plots indicate the grid cutoffs for the two simulations. The experimental spectra shows about half a decade of inertial range

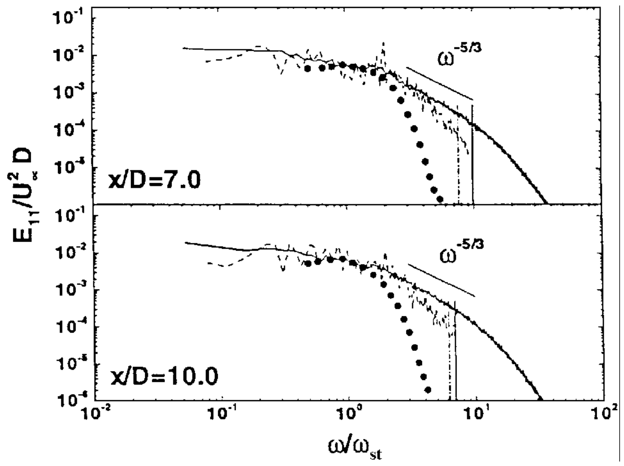

Fig. 1 One-dimensional streamwise velocity spectra E_{11} along the wake centerline: ——, Ong and Wallace[6]; – – –, central difference; and ●, upwind biased.[5] Vertical lines indicate the grid cutoff: ——, central difference and — · —, upwind biased.[5]

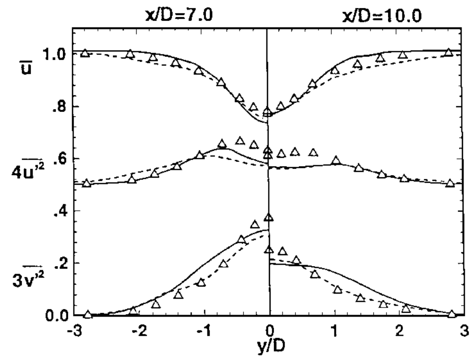

Fig. 2 Mean velocity and Reynolds stress profiles at two downstream locations: △, Ong and Wallace[6]; ——, central difference; and – – –, upwind biased.[5]

extending from about $\omega/\omega_{st} = 2$–7. Figure 1 clearly shows that the spectra from the current simulation match the experimental spectra much better than the simulation of Beaudan and Moin.[5] A closer look at the spectra at the three locations obtained from the upwind-biased simulation[5] shows that only the energy in the lower 20–25% of the resolved wave numbers matches with the experiment. On the other hand, in the current simulation the damping at the higher wave numbers is not as severe and spectra in the lower 40–50% of the resolved wave number range match well with the experiment. It might be more appropriate to compare the spectra obtained from LES with a suitably filtered experimental spectra. However, there is no straightforward way of accurately determining the grid filter function, and we have chosen instead to just indicate the grid cutoff on the spectra plot. Beaudan and Moin[5] attributed the marginal performance of the upwind-biased schemes in the downstream wake region to the dominance of numerical dissipation. Thus, given the fact that the spectra for the current simulation show better agreement with the experiment than Beaudan and Moin's,[5] it is reasonable to expect that the turbulence statistics obtained from current simulation will also be better predicted.

In Fig. 2 the mean streamwise velocity and normal stress profiles at these two locations are shown. We observe that the streamwise velocity profiles obtained from both simulations agree reasonably well with the experiment. Furthermore, at $x/D = 7.0$, the simulation of Beaudan and Moin[5] underpredicts the peak streamwise normal stress, whereas the current simulation shows better agreement in both the magnitude of the peak stress and shape of the stress profile. However, at $x/D = 10.0$, streamwise stress profiles from both simulations match quite well, and both underpredict the experimental stress level. Because streamwise Reynolds stress at the wake centerline is directly related to the area underneath the spectra shown in Fig. 1, it is somewhat surprising that the current simulation does not predict a streamwise stress level that is significantly higher than the simulation of Beaudan and Moin.[5] However, this can be explained by noting that, for this flow, most of the contribution to the Reynolds stress comes from fluctuations in a narrow frequency band extending from about $0.5\omega_{st}$ to $3.0\omega_{st}$, and in this frequency band, the energy in both simulations is comparable. Thus, even though the simulations of Beaudan and Moin[5] exhibit significant damping of the higher frequencies, this does not have a significant impact on the low-order turbulence statistics.

By comparing the vertical stress profiles at these locations, we observe that prediction from the two simulations at $x/D = 7.0$ is quite similar. At $x/D = 10.0$, the two simulations predict roughly the same peak stress level; however, the shape of the experimental profile matches the profile of Beaudan and Moin[5] better than it does for the current simulation. Furthermore, we have found that vertical velocity and shear stress profiles (not shown here) from both the simulations are also in reasonable agreement with experiments.[11]

Conclusions

It is found that in the downstream portion of the wake, where the grid is relatively coarse, the numerical dissipation inherent in the higher-order upwind-biased schemes removes substantial energy from roughly three-quarters of the resolved wave number range. In the central difference simulation, because there is no numerical dissipation, the smaller scales are more energetic. Because of this reduction in the damping of smaller scales, we find that the computed power spectra agree well with the experiment up to about half of the resolved wave number range. However, the enhanced energy in the small scales has no significant effect on the low-order statistics, and the mean velocity and Reynolds stress profiles in this region obtained from the two simulations are comparable. This is because most of the contribution to the normal stress comes from fluctuations whose frequency is centered in a narrow band around the shedding frequency and change in the energy of the small scales does not have a significant effect on the magnitudes of the Reynolds stresses. However, in applications such as flow generated noise and reactive flows, small-scale fluctuations play a crucial role, and it is, therefore, critical to retain the energy in the small scales. In such applications, energy conservative schemes would be preferable over upwind schemes. We also find that with about 20–30% smaller grid spacing, the second-order central difference scheme gives results that are comparable to those obtained by the high-order upwind biased schemes. The higher-order upwind based solver is more expensive on a per-point basis than the second-order central difference solver, and this partially offsets the additional cost of the increased resolution required by the second-order method. A drawback of the second-order central scheme is that the simulations are sensitive to numerical factors such as grid discontinuities and outflow boundary conditions and, thus, grids and boundary conditions have to be designed with extreme care.

Acknowledgments

This work was supported by the Office of Naval Research under Grant N00014-95-1-0221. We thank L. M. Lourenco and C. Shih for their private communication, "Characteristics of the Plane Turbulent Near Wake of a Circular Cylinder. A Particle Image Velocimetry Study." We would also like to thank Thomas Lund for his careful reading of the manuscript and helpful suggestions.

References

[1]Germano, M., Piomelli, U., Moin, P., and Cabot, W. A., "Dynamic Subgrid-Scale Eddy Viscosity Model," *Physics of Fluids A*, Vol. 3, No. 7, 1991, pp. 1760–1765.

[2]Ghosal, S., Lund, T. S., Moin, P., and Akselvoll, K., "A Dynamic Localization Model for Large-Eddy Simulation of Turbulent Flows," *Journal of Fluid Mechanics*, Vol. 286, March 1995, pp. 229–255.

[3]Piomelli, U., "High Reynolds Number Calculations Using the Dynamic Subgrid-Scale Stress Model," *Physics of Fluids A*, Vol. 5, No. 6, 1993, pp. 1484–1490.

[4]Rai, M. M., and Moin, P., "Direct Simulation of Turbulent Flows Using Finite-Difference Schemes," *Journal of Computational Physics*, Vol. 109, No. 2, 1991, pp. 169–192.

[5]Beaudan, P., and Moin, P., "Numerical Experiments on the Flow Past a Circular Cylinders at Sub-Critical Reynolds Numbers," Dept. of Mechanical Engineering, Rept. TF-62, Stanford Univ., Stanford, CA, Dec. 1994.

[6]Ong, L., and Wallace, J., "The Velocity Field of the Turbulent Very Near Wake of a Circular Cylinder," *Experiments in Fluids*, Vol. 20, No. 6, 1996, pp. 441–453.

[7]Akselvoll, K., and Moin, P., "Large-Eddy Simulation of Turbulent Confined Coannular Jets," *Journal of Fluid Mechanics*, Vol. 315, May 1994, pp. 387–411.

[8]Lund, T. S., and Moin, P., "Large-Eddy Simulation of a Concave Wall Boundary Layer," *International Journal of Heat and Fluid Flow*, Vol. 17, No. 3, 1996, pp. 290–295.

[9]Choi, H., Moin, P., and Kim, J., "Direct Numerical Simulation of Flow Over Riblets," *Journal of Fluid Mechanics*, Vol. 255, Oct. 1994, pp. 503–539.

[10]Lilly, D. K., "A Proposed Modification of the Germano Subgrid-Scale Closure Method," *Physics of Fluids A*, Vol. 4, No. 3, 1992, pp. 633–635.

[11]Mittal, R., "Progress in LES of Flow Past a Circular Cylinder," *CTR Annual Research Briefs*, Center for Turbulence Research, Stanford Univ., Stanford, CA, 1996, pp. 233–242.

[12]Press, W. H., Teukolsky, S. A., Vetterling, W. T., and Flannery, B. P., *Numerical Recipes in Fortran*, Cambridge Univ. Press, New York, 1992.

C. G. Speziale
Associate Editor

Three-Dimensional Finite Difference Method for Rotordynamic Fluid Forces on Seals

Morgan Williams,* Wei Chen,† Laura Brozowski,‡ and Anthony Eastland§
The Boeing Company, Canoga Park, California 91309-7922

Nomenclature

d	= amplitude of shaft whirl motion eccentricity
$\boldsymbol{E}, \boldsymbol{F}, \boldsymbol{G}$	= flux vectors in x, y, and z coordinate directions, respectively
F_n	= shaft normal force, $F_y y(t) + F_z z(t)$
F_t	= shaft tangential force, $F_y z(t) - F_z y(t)$
F_y, F_z	= shaft fluid pressure forces in y and z directions, respectively
k	= turbulence kinetic energy
p	= static pressure
$\boldsymbol{Q}$	= vector of flow variables
$\boldsymbol{S}$	= vector of momentum equation source terms
t	= time
$\boldsymbol{U}$	= vector of velocity variables
u, v, w	= Cartesian velocity components in the x, y, and z coordinate directions, respectively
x, y, z	= Cartesian coordinates
x_t, y_t, z_t	= mesh speed (partial differentiation of position with respect to time)

Presented as Paper 96-2738 at the AIAA/ASME/SAE/ASEE 32nd Joint Propulsion Conference, Lake Buena Vista, FL, July 1–3, 1996; received Dec. 16, 1996; revision received March 24, 1997; accepted for publication April 7, 1997.

*Engineering Specialist, Computational Fluid Dynamics, Rocketdyne Division, Boeing North American, Mail Stop IB39, 6633 Canoga Avenue, P.O. Box 7922. E-mail: morgan.williams@boeing.com.

†Senior Engineering Specialist, Rotating Machinery Analysis, Rocketdyne Division, Boeing North American, Mail Stop IB49, 6633 Canoga Avenue, P.O. Box 7922. E-mail: wei-chung.chen@boeing.com.

‡Project Engineer, Rotating Machinery and Valves Development, Rocketdyne Division, Boeing North American, Mail Stop IA34, 6633 Canoga Avenue, P.O. Box 7922.

§Project Manager, RAPID, Rocketdyne Division, Boeing North American, Mail Stop IA34, 6633 Canoga Avenue, P.O. Box 7922.