A Comprehensive Comparison of Turbulence Models in the Far Wake

by

Professor John M. Cimbala
Mechanical Engineering Department
The Pennsylvania State University
University Park, PA 16802

Abstract

In the present study, the far wake was examined numerically using an implicit, upwind, finite-volume, compressible Navier-Stokes code. The numerical grid started at 500 equivalent circular cylinder diameters in the wake, and extended to 4000 equivalent diameters. By concentrating only on the far wake, the numerical difficulties and fine mesh requirements near the wake-generating body were eliminated. At the time of this writing, results for the K-ϵ and K-ω turbulence models at low Mach number have been completed, and show excellent agreement with previous incompressible results and far-wake similarity solutions. The code is presently being used to compare the performance of various other turbulence models, including Reynolds stress models and the new anisotropic two-equation turbulence models being developed at NASA Langley. By increasing our physical understanding of the deficiencies and limits of these models, it is hoped that improvements to the universality of the models can be made. Future plans include examination of two-dimensional *momentumless* wakes as well.

Introduction

Purpose The author is an experimentalist by background, so the primary purpose of this ASEE Summer Faculty Fellowship was for him to obtain some experience with computational fluid dynamics (CFD). Towards this end, the two-dimensional far wake was chosen as a good test flow on which to practice computations and learn more about turbulence modeling. The author will remain here at NASA Langley during the academic year 1993-94 (on sabbatical leave) to continue these studies and to perform numerical calculations on other flows as well.

Flow Field The turbulent far wake of a two-dimensional body was chosen since there has been extensive experimental study of this flow, and since there is a known analytical solution very far downstream[1]. If turbulence in the wake could be properly modeled, a CFD code would predict both spreading rate and velocity profiles which would match those of experiment. Unfortunately, even for such a simple flow, none of the popular turbulence models in use today yield CFD predictions which match experimental results. For example, the standard two-equation turbulence models (K-ϵ and K-ω) perform reasonably well at predicting the mean velocity and turbulent kinetic energy profile shapes in the two-dimensional far wake, but they do a poor job at predicting the *spreading rate* of the far wake[2].

In the present study, the wake-generating body was not included in the computational space; rather, the inlet was chosen at a location far enough downstream where the wake is already fully developed. This choice enabled comparison of the performance of turbulence models, without the requirement of a huge number of grid points. The inlet conditions were taken from the experimental data of Browne, Antonia, and Shah[3,4], who conducted extensive measurements in the far wake of a circular cylinder.

Grid The numerical grid began at approximately 500 equivalent circular cylinder diameters downstream of the wake-generating body, where the wake is already fully developed. 101 grid points were sufficient in the streamwise (x) direction, and 51 in the normal (y) direction. Parametric studies of grid resolution showed that neither a 201x51 grid (twice the number of grid points in the x direction) nor a 101x101 grid (twice the number of grid points in the y direction) had any significant effect on the results. The 101x51 grid corresponded to an executable program which was small enough (< 10 Mwords) to run on NASA Langley's Cray Y-MP ("sabre") interactively. In the code, both x and y were normalized by b_0, the wake half width at the inlet. b is defined as the y location where the mean streamwise velocity is half-way between its minimum at the centerline and its maximum in the freestream. The grid was stretched geometrically in the y direction from $y/b_0 = 0$ (centerline) to $y/b_0 \approx 10$ times the *local* wake half width, b, which was assumed to grow as $\sqrt{x}$, as is known from both experiment and similarity solutions of the far wake. The grid was also stretched in the x direction as $\sqrt{x}$. For the case studied here, the inlet corresponded to a distance of 100 inlet wake half widths from the wake-generating body. The grid extended therefore from $x/b_0 = 100$ to $x/b_0 = 800$, which is equivalent to 500 to 4000 circular cylinder diameters. A plot of the grid is shown in Figure 1.

CFD Code The code used in the present study was written by Joe Morrison, and is an implicit, upwind, finite-volume, compressible Navier-Stokes code. Details about the numerical scheme and the performance of the code are described in Morrison[5] and Morrison and Gatski[6]. Although the code was written for compressible flows, the Mach number was set very low (typically 0.2 or lower) so that the effects of compressibility were negligible. (This was verified by repeating the calculations at several Mach numbers.) All calculations to date have been steady-state, and have been restricted to two dimensions, although the code is capable of solving three-dimensional, time-dependent flows.

Results

To date, the code has been run with both the K-ϵ and K-ω turbulence models. Self-similar wake profiles were obtained by the following scheme:

1. The best available data[2,3] were supplied at the inlet of the computational domain (i.e. at $x/b_0 = 100$).
2. The code was run until convergence. However, the downstream profiles (U, V, K, and either ϵ or ω, depending on which turbulence model was used) did not attain self-similarity with the profiles supplied as inlet conditions. Note that this was not entirely unexpected, since some of the input variables (particularly dissipation ϵ and pressure P) are extremely difficult to measure, and were not correct self-similar profiles at the start.
3. The profiles at a downstream station of $x/b_0 = 400$ were re-normalized and fed back into the code as modified inlet conditions. The code was then run again, using these modified inlet conditions, and using the output from the previous run as the initial guess for the new run (to speed up convergence).
4. Step 3 was repeated several times (typically six or seven runs were required) until the entire solution converged. In other words, when the correct inlet profiles for a self-similar far wake were specified as inlet conditions, the wake developed further downstream in an exactly self-similar manner, consistent with the specified inlet conditions.

Upon convergence of the solution, plots of the normalized profiles of U, K, ϵ, and ω at every x location collapsed onto the same curves, indicating complete self-similarity. Examination of the spreading rate of the wake could then be performed. Results for the K-ϵ turbulence model are shown in Figures 2 and 3. Figure 2 shows how the centerline velocity defect decays as $\sqrt{x}$, while the wake half width grows as $\sqrt{x}$. Figure 3 shows the curve fit used to determine the nondimensional spreading rate for the wake. The spreading rate was found to be 0.257. For comparison, Wilcox[2] found a spreading rate of 0.256 using the same turbulence model, but assuming a self-similar solution from the start. This agreement is encouraging since no such assumption was necessary in the present calculations; i.e. the self-similar state was predicted by the full Navier-Stokes code, provided that the inlet conditions were correct.

Conclusions and Plans for Future Work

At the time of this writing, it has been verified that the CFD code being used can accurately predict the growth of a simple shear flow, such as a turbulent far wake. The solutions, though consistent with previous calculations of others, do not match experimentally observed growth rates. The reason for this discrepancy is not due to the code itself, but rather to non-universality of the turbulence models. In other words, the turbulence models do not contain enough information to adequately model the physics of the flow, and thus the numerical predictions are only as good as the turbulence model itself. This is an ideal situation for comparison of various turbulence models.

In the next few months, several other turbulence models will be tested on this same flow field. These will include the more sophisticated Reynolds stress models and the new anisotropic two-equation turbulence models being developed at NASA Langley[7]. Only by increasing our physical understanding of the deficiencies and limits of these models can improvements to the universality of the models be made. Future plans include examination of the two-dimensional *momentumless* wake as well. This case is even more difficult to predict numerically, since the mean shear decays extremely rapidly downstream.

References

1. Schlichting, H. *Boundary Layer Theory* (1962) 6th ed., McGraw-Hill, New York.
2. Wilcox, D. C. *Turbulence Modeling for CFD* (1993) DCW Industries, Inc., La Canada, CA.
3. Browne, L. W. B., Antonia, R. A. Anisotropy of the Temperature Dissipation in a Turbulent Wake, *J. Fluid Mech.* (1986) **163**, pp. 393-403.
4. Antonia, R. A. and Browne, L. W. B., Shah, D. A. Turbulent Energy Dissipation in a Wake, *J. Fluid Mech.* (1987) **179**, pp. 307-326.
5. Morrison, J. H. (1992) A Compressible Navier-Stokes Solver with Two-Equation and Reynolds Stress Turbulence Closure Models, NASA Cont. Rep. 4440.
6. Morrison, J. H. and Gatski, T. B. (1993) *Bull. Am. Phys. Soc.*, **38**.
7. Gatski, T. B. (1993) Private Communication, NASA Langley Research Center.

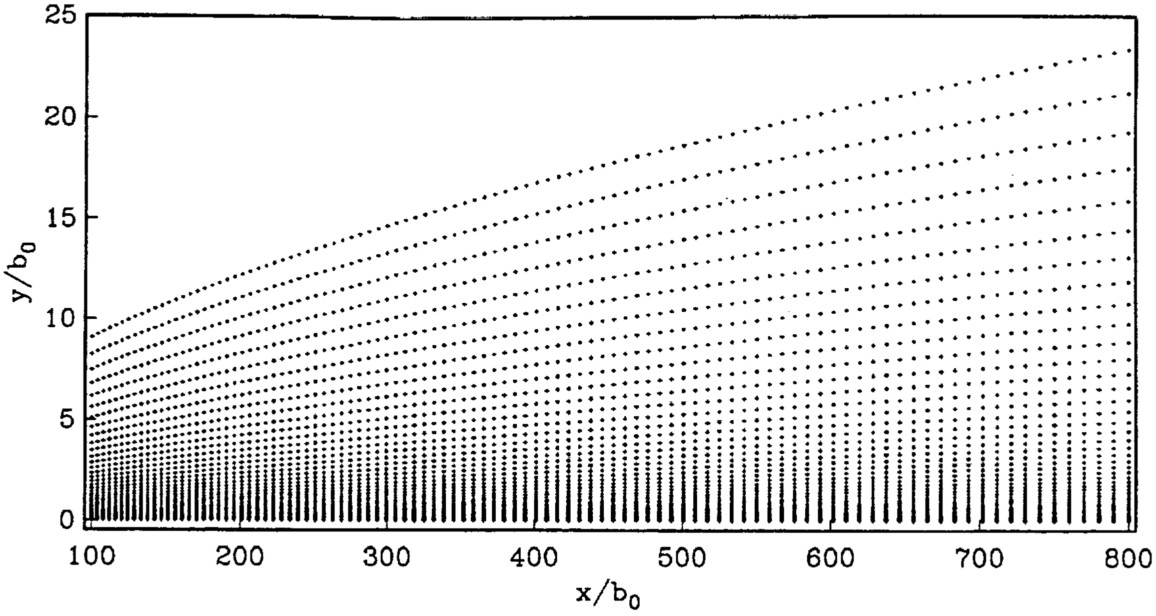

Figure 1. Grid generated for turbulence model study; 2–D wake.

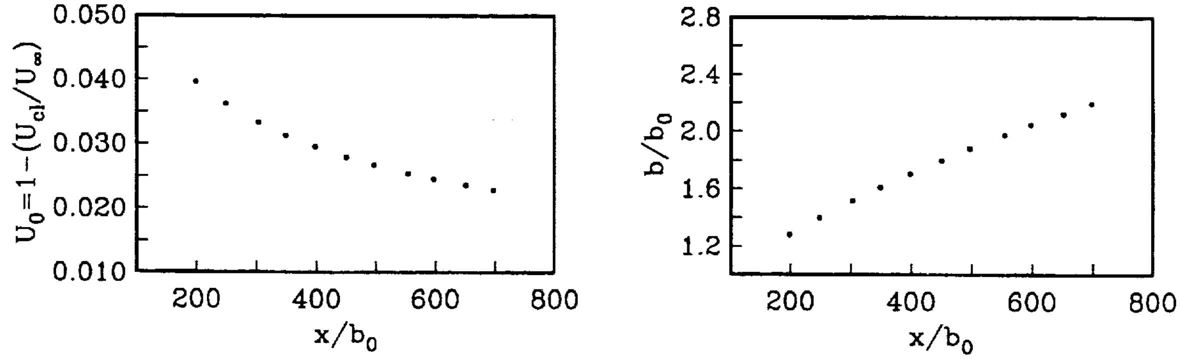

Figure 2. Streamwise development of velocity defect and wake halfwidth.

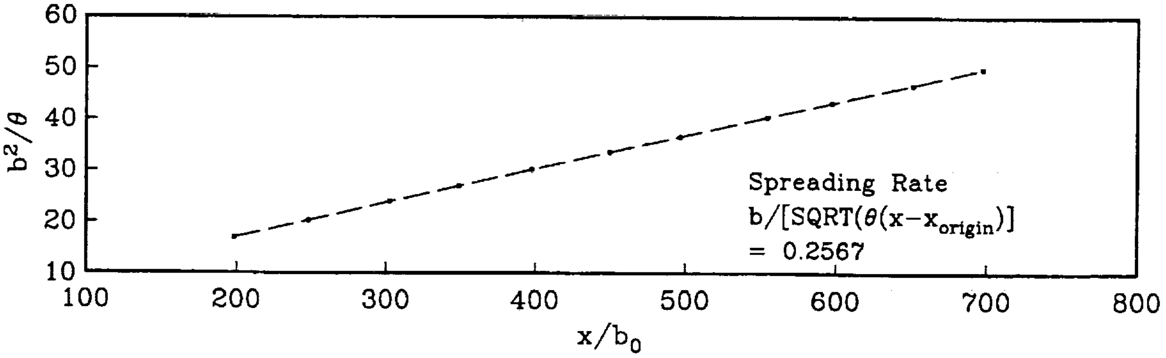

Figure 3. Curve fit for calculation of wake spreading rate.

C-2