NASA Technical Memorandum 102811

Progress and Challenges in Modeling Turbulent Aerodynamic Flows

Joseph G. Marvin

July 1990

(NASA-TM-102811) PROGRESS AND CHALLENGES IN MODELING TURBULENT AERODYNAMIC FLOWS (NASA) 13 p CSCL 20D

N90-27981

Unclas
H1/34 0302484

National Aeronautics and
Space Administration

NASA Technical Memorandum 102811

Progress and Challenges in Modeling Turbulent Aerodynamic Flows

Joseph G. Marvin, Ames Research Center, Moffett Field, California

July 1990

NASA
National Aeronautics and Space Administration

Ames Research Center
Moffett Field, California 94035-1000

PROGRESS AND CHALLENGES IN MODELING TURBULENT AERODYNAMIC FLOWS

JOSEPH G. MARVIN
NASA Ames Research Center
Moffett Field, CA 94035

ABSTRACT

Progress in modeling external aerodynamic flows achieved by using computations and experiments designed to guide turbulence modeling is presented. The computational procedures emphasize utilization of the Reynolds-averaged Navier-Stokes equations and various statistical modeling approaches. Developments for including the influence of compressibility are provided; they point up some of the complexities involved in modeling high-speed flows. Examples of complementary studies that provide the status, limitations, and future challenges of modeling for transonic, supersonic, and hypersonic flows are given.

INTRODUCTION

Computational fluid dynamics (CFD) has matured over the past decade and its potential for solving a wide variety of fluid dynamics problems is well recognized. It is rapidly becoming an important tool in the design of aerospace systems [1]. However, turbulent flows still present a formidable challenge, and turbulence modeling still paces CFD's development. Turbulence must be modeled in numerical simulations, because limits imposed by computer speed and memory make it impractical to compute all scales of turbulent motion, except for low-Reynolds-number flows. The governing equations are the Reynolds-averaged Navier-Stokes equations, and computations of realistic aerodynamic flows using them fall within the capabilities of today's supercomputers [2].

Experiment plays an important role in developing CFD and turbulence modeling. Code development progresses from research codes through pilot codes and ultimately to production codes, and each progressive stage is linked to a type(s) of experiment [3]. Research codes require data from building-block experiments undertaken to understand flow physics, to guide flow modeling processes, and to validate their incorporation into the codes. An additional new development (briefly discussed later) is the use of full- and large-eddy simulations to provide understanding of the physics and guide modeling [4]. Pilot codes require data from benchmark experiments designed to provide the parametric data needed to calibrate and validate a code's range of applicability and accuracy, including the turbulence model. Subsequently, the code would be advanced to a production code, in which form it could be used in aerodynamic design applications.

In this paper, some highlights of the progress of turbulence modeling toward the development of Reynolds-averaged Navier-Stokes codes are discussed. The statistical averaging process is briefly described as it relates to the compressible flow problems that are common in aerodynamic applications, and examples of modeling complex flows are described with emphasis on separation caused by shock wave interactions. Comparisons with experiment provide an evaluation of the modeling progress. The paper concludes by presenting some of the remaining challenges of turbulence modeling.

RESULTS AND DISCUSSION

Governing Equations and Models

Solutions to the Reynolds-averaged form of the Navier-Stokes equations written in mass-averaged (or Favre-averaged) variables are presented in the examples to follow. The goal of turbulence modeling is to relate the turbulent stresses and heat-fluxes to known mean-flow quantities such as velocity and temperature [5]. Equations can be derived to describe each of the Reynolds stresses [6]. If the Reynolds stresses and heat fluxes are related algebraically to the mean-flow variables, the corresponding models are called algebraic stress models (ASMs). Eddy-viscosity models are an important subclass of these models; they relate the Reynolds stresses to the strain rates in a manner identical with molecular stresses. This approach is generally referred to as the Boussinesq approximation. The models may be expressed in terms of an eddy-viscosity function and a turbulent Prandtl number. Compressibility is included by introducing mean density when mass-averaging is used. Aside from the use of mass-averaging instead of time-averaging, there is very little difference in form from the incompressible models. Eddy-viscosity models have been of primary interest for developing the Reynolds-averaged Navier-Stokes codes to be discussed. Such interest results from considerations of computational efficiency and the limited improvement afforded by applying Reynolds stress models to complex flows.

Compressibility and Averaging

Many external aerodynamic flow problems involve speeds such that compressibility must be taken into account. Investigations by the author [7] and his colleagues, restricted to attached transonic and moderately supersonic flows, showed that compressibility effects could be accounted for by including the mean density in the mass-weighted model equations developed for incompressible flows [8]. This is illustrated in Fig. 1 where results for skin friction predicted by various models are shown for a flat surface with and without pressure gradient. The predicted skin-friction variations without pressure gradient are compared with the Van Driest prediction, which represents all available data to within 10%. Aside from showing that all models predict the trends of compressibility and pressure gradient with reasonable accuracy, it is concluded that the choice of mass- or time-averaging has no significant influence. The latter finding led to the decision to use mass-weighted variables in compressible Reynolds-averaged Navier-Stokes codes, since algorithms developed for laminar flows could be used and since modeling a large number of higher-order correlations could be avoided.

Transonic Flow Modeling

Transonic flow computations are sensitive to the choice of a turbulence model. The high-speed flow leads to large displacement thicknesses, and shock waves can form that lead to separation. Each or both of these effects can influence the development of the entire flow field because of its elliptic character. Initial attempts to solve airfoil problems with Reynolds-averaged Navier-Stokes codes were disappointing [9]. In retrospect, the poor predictions resulted because the grids used were too coarse and because the turbulence models were inadequate. The situation has been ameliorated in both aspects.

Fig. 2 shows predicted results using four eddy-viscosity models compared with the Bachalo-Johnson experimental data (case 8611[9]). These recent computations were made with the same computer code using grids sufficiently fine to minimize numerical errors [10]. The Johnson-King [11] and Viegas-Rubesin [12] eddy-viscosity models do an excellent job of predicting this flow, which separates due to a strong shock wave.

The deficiency of the Jones-Launder model is a result of the low-Reynolds-number modeling terms and their integration near the surface. Viegas and Rubesin eliminated this deficiency by developing a wall-function model that includes the pressure-gradient effect and allows for reversed profiles in the separated region [12]. In addition to the model improvement, the use of wall functions reduced the required computer time by an order of magnitude by eliminating grid points and the associated restrictive time-step required to integrate the low-Reynolds-number model terms.

The deficiency of the Cebeci-Smith model is a result of its inability to model turbulence history effects through shock waves. The Johnson-King model modifies the inner layer description of the Cebeci-Smith model by adding an ordinary differential equation that describes the growth of the maximum shear stress. It accounts for the lag in turbulence adjustment to the mean flow in regions of strong interaction. Compared with the Cebeci-Smith model, it contains two additional modeling constants, one that scales the rate of development of the maximum shear stress and another that scales the effect of turbulence diffusion.

The Johnson-King model is easily put into Reynolds-averaged Navier-Stokes codes already using the Cebeci-Smith model (or variations of it) and for that reason it is being used extensively in transonic airfoil and wing codes.

Fig. 3 compares predictions using the model with transonic airfoil data [13]. For the attached-flow case, the model and that of Cebeci-Smith agree with the data. For the separated case, only the improved model agrees with the data. In the application shown, some modifications to the original Johnson-King [11] model have been made so that it behaves more closely to the law-of-the-wall in pressure gradient regions [14]. This model has also been extended successfully to wing computations [15]. Fig. 4 compares predictions for the ONERA M-6 wing flow at an angle of attack of 6° where strong shock waves form at outboard span locations on the upper surface. Predictions of the new model agree with the data, whereas those using the 0-Eq. model of Baldwin-Lomax (a form of the Cebeci-Smith model) do not.

Supersonic Flow Modeling

Attached supersonic turbulent flows can be predicted with engineering accuracy using either 0-Eq. or 2-Eq. turbulence models [5]. Flows involving strong shock waves are still a matter of concern, but progress has been made. An example is shown in Fig. 5. In the flow, which is over a compression surface, a strong shock wave forms owing to flow turning and separation occurs in the corner region. Results of computations using the Jones-Launder model are compared with data (case 8631[9]). Solutions with integration of the low-Reynolds-number terms near the wall fail to provide adequate predictions, yielding little upstream influence and only a small zone of separation. Modifications using the wall-function model developed by Viegas et al. provide improved predictions [12]. This model has been applied to three-dimensional swept shock-wave flows with similar success.

Hypersonic Flow Modeling

New aerospace vehicles, such as the U.S. National Aero-Space Plane, have renewed interest in hypersonic flows. Turbulence modeling is an important issue for the Space Plane, which will use an air-breathing engine.

Attached, simple hypersonic flows can be predicted satisfactorily with 0-Eq. models [5]. Fig. 6 shows comparisons of predictions with skin-friction and heat-transfer data. Computations are given for two 0-Eq. model formulations of the turbulent shear stress, the usual equation and another that includes the higher-order correlations involving vertical velocity fluctuations. Differences between the model predictions are not large, and agreement with experiment is good.

Hypersonic flows involving shock interaction are difficult to predict. Concern for compressibility is a major issue, and some advances are being made toward understanding its influence; an example is shown in Fig. 7. Predictions of pressure and heat transfer are compared with data [16] taken on a compression ramp deflected 15° into the flow. Predictions from two 0-Eq. models (Cebeci-Smith; Baldwin-Lomax) and one 2-Eq. model are presented.

Predicted results from the 0-Eq. models are not adequate, whereas the 2-Eq. results provide an improvement. In this model advance [5], the constant used to multiply the dilation term in the dissipation rate equation was modified to insure that the product of density and turbulent length scale remained constant during compression. Also, an upper limit was placed on the length scale in the eddy viscosity such that it never exceeded the Prandtl length scale in the near-wall region. This limit was necessary to control the maximum heating level at and near reattachment.

Other studies of the influence of compressibility on models have concentrated on the problem of shear-layer development at supersonic speeds where significant compressibility effects are known to be present. Vandromme [17] used Rubesin's compressibility modification and modified the Jones-Launder model equations and in Fig. 8 his computations are compared with shear-layer data (case 8500[9]). The spreading-rate from an unmodified Jones-Launder model does not show the decrease observed experimentally. However, adding more compressibility terms to the model equations did give predictions that agreed with the data. More recently, Zeman has modified the dissipation rate equation to account for compressibility effects arising from shocklets that form in supersonic shear layers, and he also predicts the proper spreading-rate [18]. The applicability of both of these model changes to the shock interaction problem discussed previously is being evaluated.

CONCLUDING REMARKS

Emphasis on model improvement for compressible aerodynamic flows using Reynolds-averaged Navier-Stokes codes has been directed toward eddy-viscosity formulations. As mentioned earlier, this stems from the requirement for computational efficiency and the perception that Reynolds stress models have not yet provided more accurate predictions. Computational efficiency will remain a requirement and, therefore, further improvement in eddy-viscosity modeling is needed. Some flows cannot be modeled with this approach, however; as a result, more accurate and computationally efficient Reynolds stress models are also needed. The challenge for the future is in the means used to achieve these needs. As the following examples illustrate, I suggest that further work on the full- and large-eddy numerical simulations of turbulence will be the most fruitful.

Most 2-Eq. model formulations are unable to model the near-wall behavior of turbulence, as some of our previous examples showed. Fig. 9 shows computations using various model formulations in the near-wall region of a turbulent channel flow compared with results from a full numerical simulation. Shih [19] (present model) was able to provide better formulations, using the simulation as guidance. Overall, his model predicts the proper behavior of the turbulent kinetic energy, of other turbulence quantities, and of the eddy viscosity. This formulation has not been tried on any aerodynamic flow problems, however.

Many aerodynamic flow computations will require Reynolds stress modeling to provide more accurate predictions. For three-dimensional flows, the cross-flow turbulent stress lags in response to changes in the cross-flow velocity gradient, and eddy-viscosity formulations may fail, depending on the severity and extent of the three dimensionality. Fig. 10 shows data [20] from a three-dimensional boundary layer experiment that illustrates the limitations of eddy-viscosity modeling. The cross flow, generated with a forward-spinning section of a cylinder aligned with the free stream, relaxes toward two-dimensional flow over the stationary portion of the cylinder. The angles of the flow direction (0° coincides with the free stream), strain rate, and turbulent stress are shown for a downstream location on the stationary surface. Significant differences in angle between the stress and strain rate occur. The Rotta (1979) T-model (Fig. 10) predicts this difference between the stress and strain-rate directions, provided a judicious choice of reference frames is used (e.g., the spinning reference frame). However, such a choice in this case depended on knowing the desired result and was not predictive. The Rodi (1976) algebraic stress model fails to predict the stress direction any better than an isotropic eddy-viscosity model, as indicated by the stress direction being nearly identical to the strain-rate direction. The Launder-Reece-Rodi (1975) model produces a better prediction, but is still not adequate. Numerical simulations for a flow analogous to this experiment have been made [21]. The simulation data are now being used to help guide statistical model improvements that will be compared with these data.

The future challenge in numerical simulation is to solve more complex problems, for example, three-dimensional flows with severe pressure gradients, with separation, and with compressibility. And the challenge remains to incorporate any statistical model improvements into codes used to predict aerodynamic flows in order to assess their behavior and success.

REFERENCES

1. Bradley, R. G., Bhateley, I. C., and Howell, G. A., Computational Fluid Dynamics—Transition to Design Applications, Supercomputing in Aerospace, NASA CP-2454, pp. 69-76 (1987).
2. Rubesin, M. W., Turbulence Modeling for Aerodynamic Flows, AIAA Paper 89-0606 (1989).
3. Marvin, J. G., Accuracy Requirements and Benchmark Experiments for CFD Validation, AGARD-CP-437, Vol 1 (1988).
4. Moin, P. and Kim, J., The Structure of the Vorticity Field in Turbulent Channel Flow. Part I. Analysis of Instantaneous Fields and Statistical Correlations, JFM, *155*, p. 441 (1985).
5. Marvin, J. G. and Coakley, T. J., Turbulence Modeling for Hypersonic Flows, NASA TM-101079 (1989).
6. Rubesin, M. W. and Rose, W. C., The Turbulent Mean Flow, Reynolds-Stress, and Heat-Flux Equations in Mass-Averaged Dependent Variables, NASA TM X-62248 (1973).
7. Marvin, J. G., Turbulence Modeling for Computational Aerodynamics, AIAA J., *21*, No. 7 (Jul. 1983).
8. Morkovin, M. V., "Effects of Compressibility on Turbulent Flows," *The Mechanics of Turbulence*, Fordon and Breach, NY, p. 367 (1961).
9. Kline, S. J., Cantwell, B. J., and Lilley, G. M., eds, The 1980-81 AFOSOR-HTTM-Stanford Conference on Complex Turbulent Flows; Comparison of Computation and Experiment, Thermophysics Div., Mech. Eng. Dept., Stanford Univ., Stanford, Calif. (1982).
10. Rubesin, M. W. and Viegas, J. R., Turbulence and Modeling in Transonic Flow, NASA CP-3020, Vol. I, Pt. 2, p. 581 (1988).
11. Johnson, D. A. and King, L. S., A Mathematically Simple Turbulence Closure Model for Attached and Separated Turbulent Boundary Layers, AIAA J., *23*, pp. 1684-1692 (1985).
12. Viegas, J. R., Rubesin, M. W., and Horstman, C. C., On the Use of Wall Functions as Boundary Conditions for Two-Dimensional Separated Compressible Flows, AIAA Paper 85-0180 (1985).
13. Cook, P. H., McDonald, M. A., and Firmin, M. C. P., Aerofoil RAE 2822—Pressure Distributions, and Boundary Layer and Wake Measurements, AGARD AR 138 (1979).
14. Johnson, D. A. and Coakley, T. J., Improvements to a Nonequilibrium Algebraic Turbulence Model, To appear in AIAA J., Oct. 1990.
15. Abid, R. and Johnson, D. A., Effects of Turbulence Models on the Prediction of Transonic Wing Flows, SAE Technical Paper Series 892224, Anaheim, Calif., Sept. (1989).
16. Coleman, G. T., A Study of Hypersonic Boundary Layers over a Family of Axisymmetric Bodies at Zero Incidence: Preliminary Report and Data Tabulation. I.C. Aero Report 73-06, Imperial College of Sci. and Tech., London, Sept. 1973.
17. Vandromme, D., Contribution to the Modeling and Prediction of Variable Density Flows, Ph.D. thesis presented at the U. of Science and Technology, Lille, France (1983).
18. Zeman, O., Dilatation Dissipation: The Concept and Application in Modeling Compressible Mixing Layers, Phys. Fluids A 2(2), pp. 178-188 (Feb. 1990).

19. Shih, T. -H., Turbulence Modeling: Near-Wall Turbulence and Effects of Rotation on Turbulence, CTR Annual Research Briefs, Center for Turbulence Research, NASA Ames Research Center and Stanford University, pp. 12-26 (1989).
20. Driver, D. M., Three-Dimensional Boundary Layer Flow with Streamwise Adverse Pressure Gradient, Tenth Australasian Fluid Mech. Conf., Dec. 1989.
21. Moin, P. Shih, T. -H, Driver, D. and Mansour, N. N., Numerical Simulation of Three-Dimensional Turbulent Boundary Layer, AIAA Paper 89-0373 (1989).

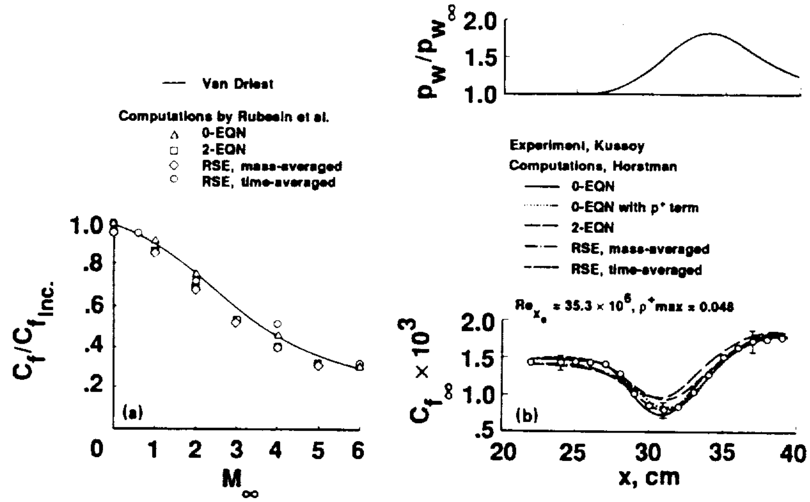

FIG. 1 Turbulent skin friction (from Ref. 7). (a) No pressure gradient, $R_L = 10^7$; (b) pressure gradient, $M_\infty = 2.3$.

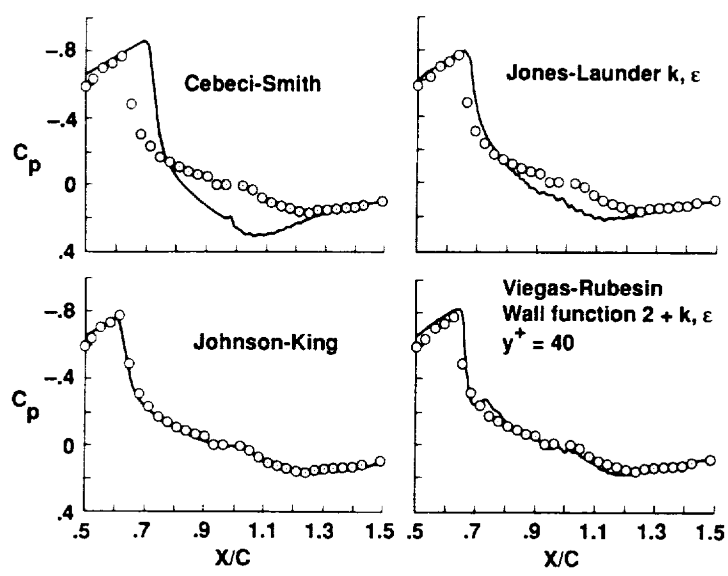

FIG. 2 Predicted pressure distributions over an axisymmetric circular arc bump compared with data.

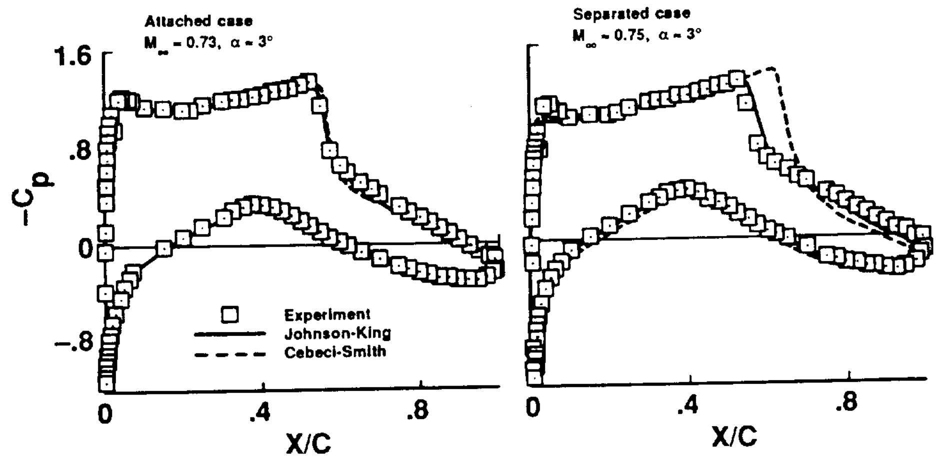

FIG. 3 Pressure distributions on the RAE 2822 airfoil.

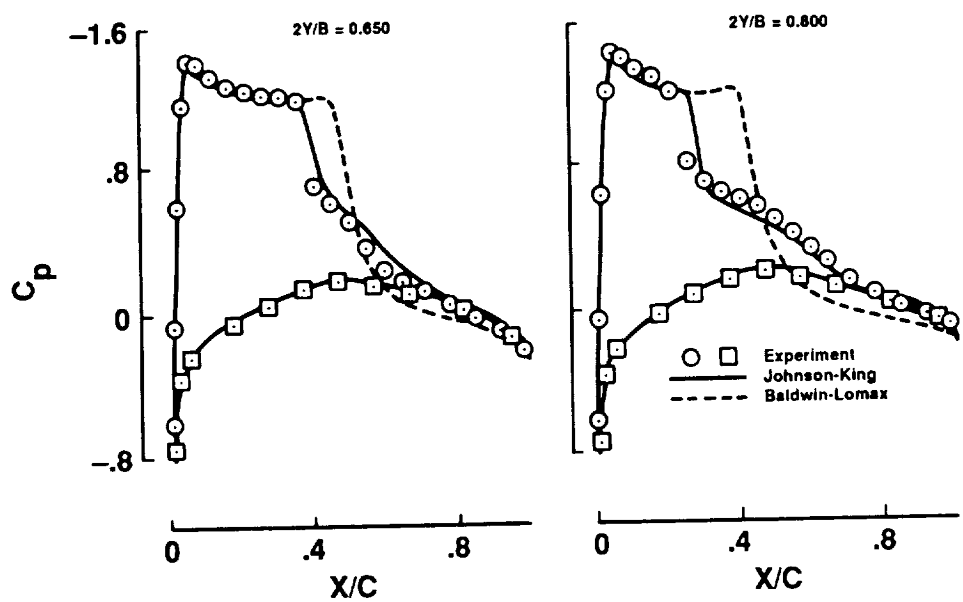

FIG. 4 Pressure distribution on the ONERA M-6 Wing at $M_\infty = 0.84$ and $\alpha = 6°$.

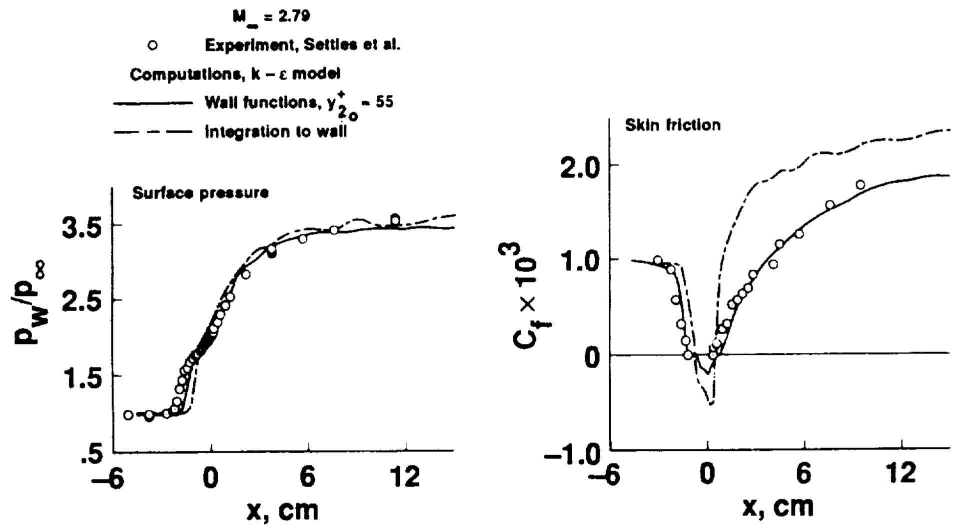

FIG. 5 Pressure and skin friction distributions on a two-dimensional compression corner deflected 20° (from Ref. 5).

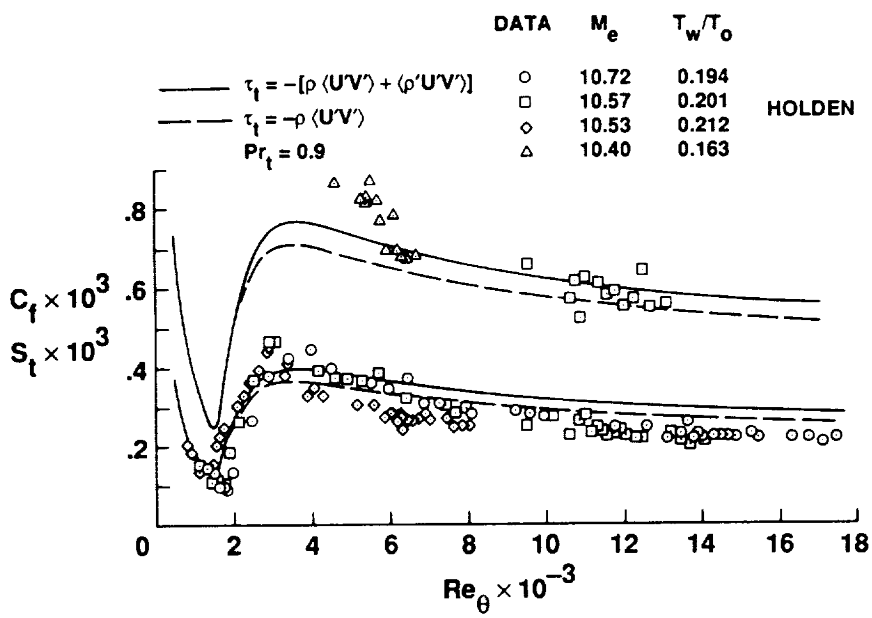

FIG. 6 Turbulent skin friction and heat-transfer distribution on a flat plate (from Ref. 5).

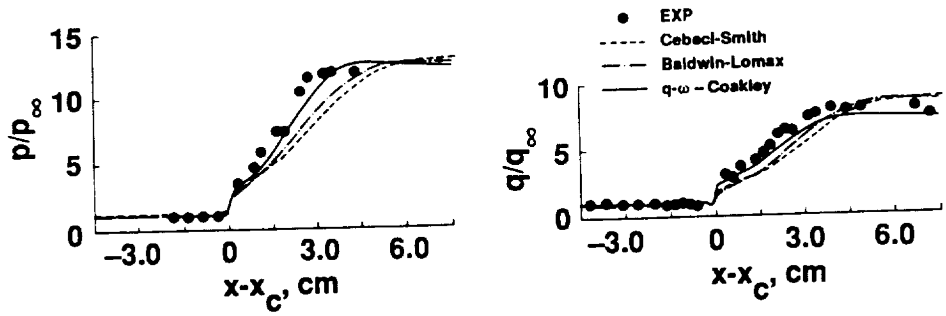

FIG. 7 Pressure and heat-transfer distribution on a two-dimensional compression corner deflected 15° at $M_\infty = 9.2$ (from Ref. 5).

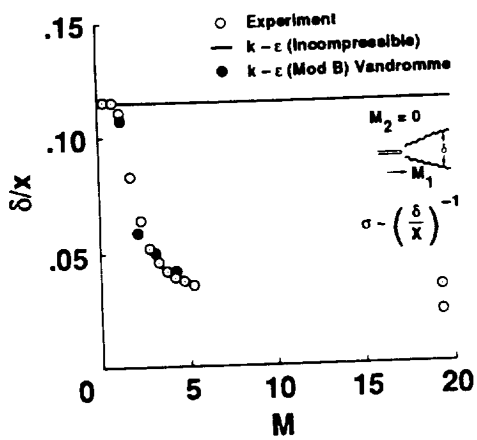

FIG. 8 Spreading-rate versus Mach number (from Ref. 17).

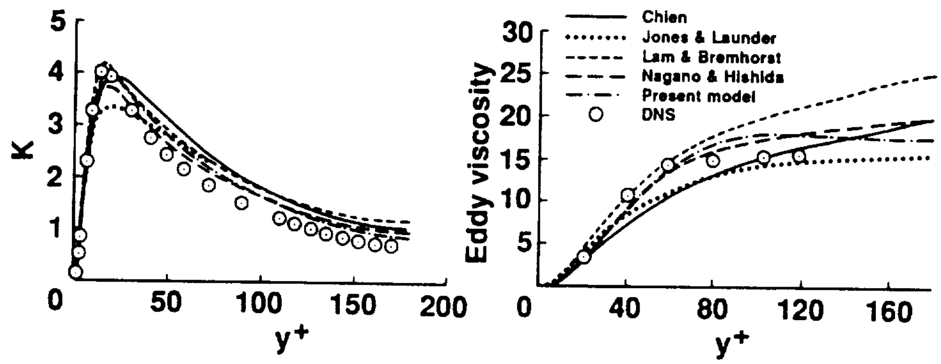

FIG. 9 Near-wall turbulent kinetic energy and eddy-viscosity formulations for a channel compared with direct Navier-Stokes simulations (DNS) (from Ref. 19).

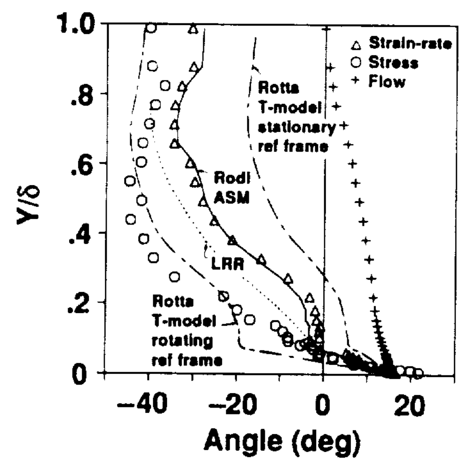

FIG. 10 Principal stress, strain rate, and flow direction in a three-dimensional boundary layer (from Ref. 20).

NASA

National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM-102811	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Progress and Challenges in Modeling Turbulent Aerodynamic Flows		5. Report Date July 1990
		6. Performing Organization Code
7. Author(s) Joseph G. Marvin		8. Performing Organization Report No. A-90133
		10. Work Unit No. 505-60-11
9. Performing Organization Name and Address Ames Research Center Moffett Field, CA 94035-1000		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001		14. Sponsoring Agency Code

15. Supplementary Notes

Point of Contact: Joseph G. Marvin, Ames Research Center, MS 229-1, Moffett Field, CA 94035-1000
(415) 604-5390 or FTS 464-5390

To be presented at the Symposium on Engineering Turbulence Modeling and Measurements, Dubrovnik, Yugoslavia, September 24-28, 1990.

16. Abstract

Progress in modeling external aerodynamic flows achieved by using computations and experiments designed to guide turbulence modeling is presented. The computational procedures emphasize utilization of the Reynolds-averaged Navier-Stokes equations and various statistical modeling approaches. Developments for including the influence of compressibility are provided; they point up some of the complexities involved in modeling high-speed flows. Examples of complementary studies that provide the status, limitations, and future challenges of modeling for transonic, supersonic, and hypersonic flows are given.

17. Key Words (Suggested by Author(s)) Turbulence modeling, Computational fluid dynamics, Experimental fluid dynamics, Compressible flow		18. Distribution Statement Unclassified-Unlimited Subject Category - 34	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 14	22. Price A02

NASA FORM 1626 OCT 86

For sale by the National Technical Information Service, Springfield, Virginia 22161