Technical Notes

TECHNICAL NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Investigation of the Influence of Blowing and Combustion on Turbulent Wall Boundary Layers

Clemet Yam* and Harry Dwyer†
University of California, Davis, Davis, California

Introduction

THE main goal of this investigation is to study some of the properties of turbulent boundary layers with the influences of wall blowing and combustion. The principal tool utilized is "numerical experiments" with the use of the k-ϵ turbulence model.[1] These numerical experiments have been closely coupled to experimental resutls, but they were extended into regions where it is difficult to perform experiments. It is shown that the influence of blowing and combustion is dramatic on the boundary-layer flow, but that existing turbulence models can be used in a semiempirical way to model many features of such complex flows.

The basic flow geometry is the turbulent flow over a plate with a mixture of hydrogen and nitrogen injected at the wall. The conditions were the same as in two sets of experiments,[2-5] which contained detailed measurements of mean and turbulence quantities in an H_2/airflow. For both of the experiments it was reported that there was no mean pressure gradient, but it is quite plausible that an induced pressure gradient existed in the results of Ueda et al.,[2-4] as will be seen from the results in this paper. The k-ϵ turbulence model was supplemented with the diffusion flame approximation along with the assumption of chemical equilibrium.[6] In this model the conservation of injected atoms from the wall, the level of the concentration fluctuations, and a probability density function (PDF) (a beta function PDF has been used in the present study) for injected atom concentration determine in a unique way the mean and fluctuating temperatures in the boundary layer.

Results

The present paper will only present the results for combustion in the boundary layer, although a series of other calculations[7] were carried out for the isothermal injection experiments of Senda et al.[5] Initially,the calculations were carried out with the k-ϵ turbulence model[1] and low Reynolds number corrections were applied to resolve the near-wall region of the flow. The low Reynolds number damping terms are

$$\mu_t = C\mu k^2/\epsilon$$

where

$$C\mu = 0.09 \exp[-2.5/(1+R_T/50)]$$

and

$$C\epsilon_2 = 2.0[1.0 - 0.3\exp(-R_T{}^2)], \qquad R_T = k^2/(\nu\epsilon)$$

All attempts to carry out calculations at high-blowing rates were unsuccessful, and it was determined that the low Reynolds number terms had to be modified. In order to obtain some insight, a study of the corrections made in the mixing length model of Cebeci and Smith[8] was carried out. In this model the size of the damping region is decreased for blowing and increased for suction. The wall damping factor A^+ is modified as an exponential function of the dimensionless blowing velocity, and is given as

$$A^+{}_B = 26\exp(-c'v_w{}^+)$$

where $v_w^+ = (v_w/v^*)$, $c' = 5.9$, and v^* is the wall friction velocity.

The physical interpretation is that the turbulence length scale is damped less by the wall with increasing blowing. This result is consistent with the boundary-layer velocity profile being retarded and made more unstable with wall blowing. Therefore, for large amounts of wall injection the mixing length model is applied completely to the wall without damping due to viscous influence. The model can then be said to be a high Reynolds number one without a low Reynolds number correction.

The correction applied to the k-ϵ model with wall blowing followed almost directly the concepts developed for the mixing length model. All of the low Reynolds number terms were damped directly by the factor

$$D^* = A^+{}_B/26$$

and the high Reynolds number form of the model was obtained for large blowing rates. The boundary conditions for the k-ϵ model were not changed as a function of blowing, and it was possible to utilize the low Reynolds number boundary conditions for large wall blowing.

The initial attempts to model the experimental combustion conditions were begun with an effort to calculate the cold isothermal injection results of air into a 10 m/s air boundary layer at a large wall blowing velocity ratio of $F = v_w/u_e = 0.01$. For this condition the calculated flow tended to develop negative shear stress, and a large displacement thickness was calculated. The experimental results of Refs. 2–4 did not report a large displacement thickness, and also did not report an induced pressure gradient caused by displacement-thickness blockage. It appears from the present investigation that the induced pressure gradient caused by the displacement-thickness influences of blowing and combustion was not measured or reported in the experiments performed in Refs. 2–4.

In order to account for a displacement-thickness interaction with the main channel flow, an axial pressure gradient based on the boundary-layer displacement thickness was introduced into the simulation. The magnitude of the axial pressure gradient was determined from a calculated boundary-layer dis-

---

Presented as Paper 87-0226 at the AIAA 25th Aerospace Sciences Meeting, Reno, NV, Jan 12–15, 1987; received July 23, 1987; revision received July 27, 1988.

*Research Assistant, Department of Mechanical Engineering.
†Professor, Department of Mechanical Engineering. Member AIAA.

placement thickness, and the resulting freestream velocity increase was approximately 10 m/s/m. With the induced pressure gradient included in the calculation both the mixing length and the wall blowing modified k-ϵ model gave similar results for the mean velocity profiles.

The velocity profile for a typical space location in the hydrogen combustion experiment is shown in Fig.1. It is seen that the agreement with experimental measurements is good. The overshoot in the velocity profile is very unusual for boundary-layer flows and it is predicted quite well by the modified model. However, this overshoot in velocity has very little to do with the turbulence model and is primarily caused

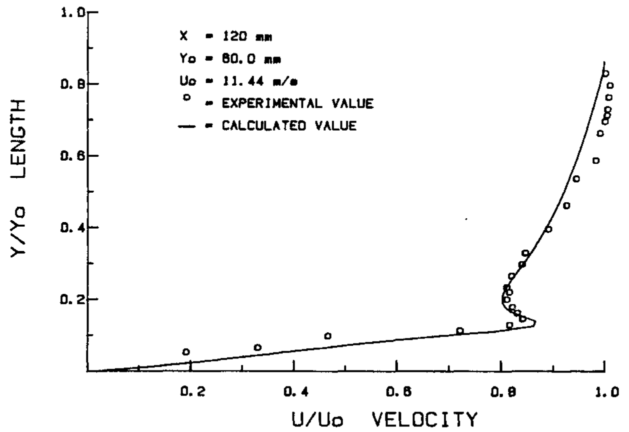

Fig. 1 Mean velocity profile with combustion and pressure gradient.

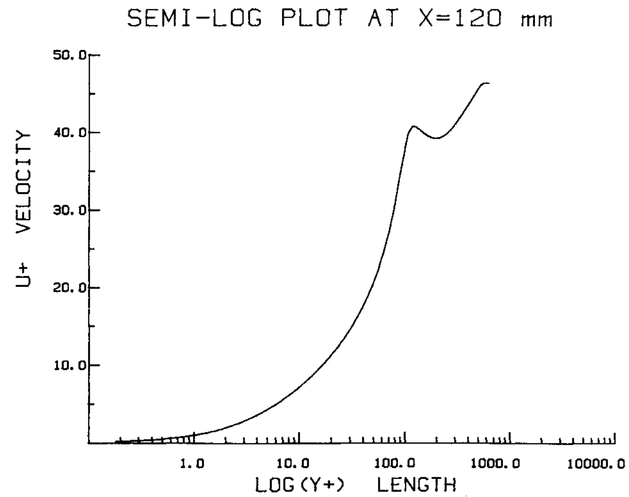

Fig. 2 Semilog mean velocity plot, combustion, and pressure gradient.

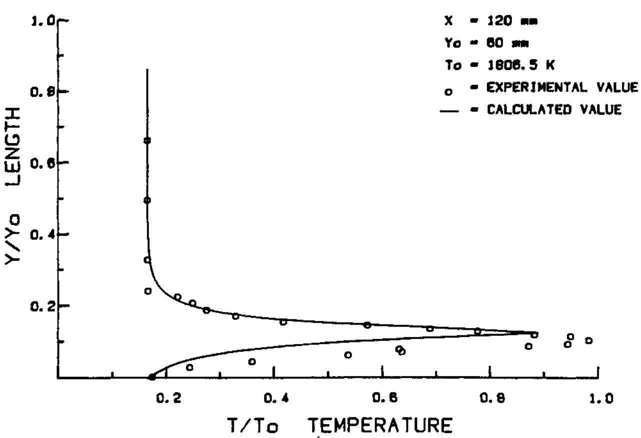

Fig. 3 Mean temperature profile with pressure gradient.

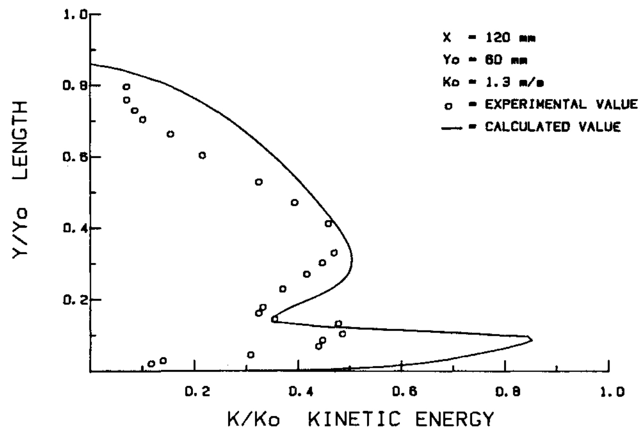

Fig. 4 Kinetic energy distribution with combustion.

by the pressure gradient acting on the flame. The low-density fluid in the flame is given an additional acceleration by the freestream pressure gradient, and the net result is a velocity overshoot. The fact that a pressure gradient was not reported in the experiments does raise some questions about these results (all attempts to calculate overshooting velocity profiles have failed without a favorable pressure gradient); however, related results of Wooldridge and Muzzy[9] do indicate that the velocity overshoot and the pressure gradient are closely connected.

The *extraordinary* nature of the combustion boundary-layer flow is easily illustrated with the use of a standard semilog plot (Fig. 2). The large values of u^+, $u^+ = u/v^*$, the lack of a law of the wall region, and the overshoot in the velocity profile give this plot a truly *extraordinary* appearance. The retardation of the boundary layer, due to expansion of the gas near the flame and wall blowing, causes the maximum values of y^+ to be very low. The temperature profile and the distribution of kinetic energy are presented in Figs. 3 and 4, and it is seen that the agreement with experiment is good. The predictions for the kinetic energy distribution are somewhat high with respect to the experimental data, but the double maximum in the profile is predicted by the computations.

Conclusions

The major conclusions of the investigation are:

1) The use of the low Reynolds number form of the k-ϵ model in flows with large wall blowing leads to an underprediction of the near-wall turbulent kinetic energy, and incorrect shapes for the mean velocity profiles.

2) The extension of the concepts of mixing length studies to the low Reynolds terms in the k-ϵ model improves the predictive capability for flows with wall blowing.

3) The model has been successfully extended to a boundary layer with hydrogen injection and combustion. The numerical calculations indicate that an induced pressure gradient existed in the experimental results of Ueda et al., and this induced pressure gradient is responsible for the velocity overshoots that were observed.

Acknowledgment

This research was supported by the U.S. Department of Energy, the Division of Chemical Sciences.

References

[1]Patel, V. C., Rodi, W., and Scheuerer, G., "Turbulent Models for Near-Wall and Low Reynolds Number Flows: A Review," *AIAA Journal*, Vol. 23, Sept. 1985, pp. 1308–1319.

[2]Ueda, T., Mizomoto, M., Kobayashi, T., and Ikai, S., "Velocity and Temperature Fluctuations in a Flat Plate Boundary Layer Diffusion Flame," *Combustion Science and Technology*, Vol. 27, 1982, pp. 133–142.

[3]Ueda, T., Mizomoto, M., Matsubatashi, Y., and Ikai, S., "Turbulent Properties of a Flat Plate Boundary Layer Diffusion Flame," AIAA Paper, Jan. 1983.

[4]Ueda, T., Mizomoto, M., and Ikai, S., "Thermal Structure of a Flat Plate Turbulent Boundary Layer Diffusion Flame," *Bulletin of the JSME*, Vol. 26, March 1983, pp. 399-405.

[5]Senda, M., Suzuki, K., and Sato, T., "Turbulent Structure Related to the Heat Transfer in a Turbulent Boundary Layer with Injection," Second Symposium on Turbulent Shear Flows, Imperial College, London, England, July 1979, pp. 9.17-9.22

[6]Kent, J. H., "Turbulent Jet Diffusion Flame," Ph.D. Thesis, Univ. of Sydney, Sydney, Australia, 1972.

[7]Yam, C., "A Study of a Turbulent Boundary Layer Diffusion Flame," M.S. Thesis, Univ. of California, Davis, Davis, CA, 1986.

[8]Cebeci, T. and Smith, A. M. O., *Analysis of Turbulent Boundary Layers*, Academic, New York, 1974.

[9]Wooldridge, C. E and Muzzy, R. J., "Boundary-Layer Turbulence Measurements with Mass Addition and Combustion," *AIAA Journal*, Vol. 4, Nov. 1966, pp. 1009-2016.

Boundary-Layer Predictions for Small Low-Speed Contractions

James H. Bell* and Rabindra D. Mehta†
Stanford University, Stanford, California

Introduction

CONTRACTION sections form an integral part of all wind tunnels, whether designed for basic fluid flow research or model testing. The main effects of a contraction are to reduce both mean and fluctuating velocity variations to a smaller fraction of the average velocity and to increase the flow mean velocity.[1] The most important single parameter in determining these effects is the contraction ratio *c*. Contraction ratios of between 6 and 10 are found to be adequate for most small, low-speed wind tunnels—defined here as tunnels with a test section cross-sectional area of less than about 0.5 m^2 and freestream velocities of less than about 40 m/s.

The wall shape design of a contraction of given area ratio and cross section centers on the production of a uniform and steady stream at its outlet. These conditions generally can be met by making the contraction section sufficiently long. On the other hand, another desirable flow quality, namely a minimum boundary-layer thickness (in a laminar state) at the contraction exit, suggests that the contraction length should be minimized. However, the risk of boundary-layer separation near the two ends of the contraction increases as the length is reduced. In general, the boundary layer is less liable to separate at the contraction exit, due to its reduced thickness caused by passage through the strong favorable pressure gradient. Also, the concave curvature at the contraction inlet has a destabilizing effect on the boundary layer, in contrast to the convex curvature near the exit that has a stabilizing effect.[2] In addition to unnecessary thickening of the boundary layer, separation also generally leads to flow unsteadiness, which cannot be easily eliminated from the test section flow. A design satisfying all criteria will be such that separation is just avoided (implying a minimum acceptable length), and the exit nonuniformity is equal to the maximum tolerable level for a given application (typically less than 1% variation in mean streamwise velocity outside the boundary layers).

Several papers have been published on the design or choice of contraction wall shapes using a variety of analytical and numerical techniques (see Ref. 3 for a review). Most recent studies have involved the calculation of the wall pressure distributions, using some potential flow numerical scheme, and then the application of a boundary-layer separation criterion based on a critical value of the pressure coefficient. The most popular separation criterion used is that due to Stratford[4] for turbulent boundary-layer separation. We propose in this Note that a boundary layer in a small, low-speed contraction is more likely to start in a laminar state and remain so, for the most part, in passage through it. The normally applied Stratford's criterion for turbulent boundary-layer separation therefore may be too liberal for these designs.

Computational Approach

A three-dimensional potential flow code (VSAERO) was used to compute the velocity distributions along the contraction walls.[3] VSAERO uses a singularity panel method employing sources and doublets to solve the Laplace equation.

It was hypothesized in this study that for small, low-speed wind tunnels the boundary layers enter the contraction in a laminar state. In most small wind tunnels, the flow entering the contraction comes through a honeycomb and a series of screens (usually at least three). The effect of a screen on a turbulent boundary layer is to significantly reduce its thickness and turbulence stress levels and scales, as shown by Mehta.[5] The results from that investigation showed that a turbulent boundary layer at moderate Reynolds numbers ($Re_\theta \sim 1600$) was effectively relaminarized immediately downstream of the screen. Note that the typical Re_θ encountered in small, low-speed settling chambers is likely to be lower by at least an order of magnitude. However, "forced" transition may still occur through either the effects of the Taylor-Gortler instabilities in the regions of concave curvature or a separation bubble. In either case, the strong favorable pressure gradient, encountered in contractions with reasonable area ratios ($c \sim 6$–10), would invariably relaminarize the boundary layer

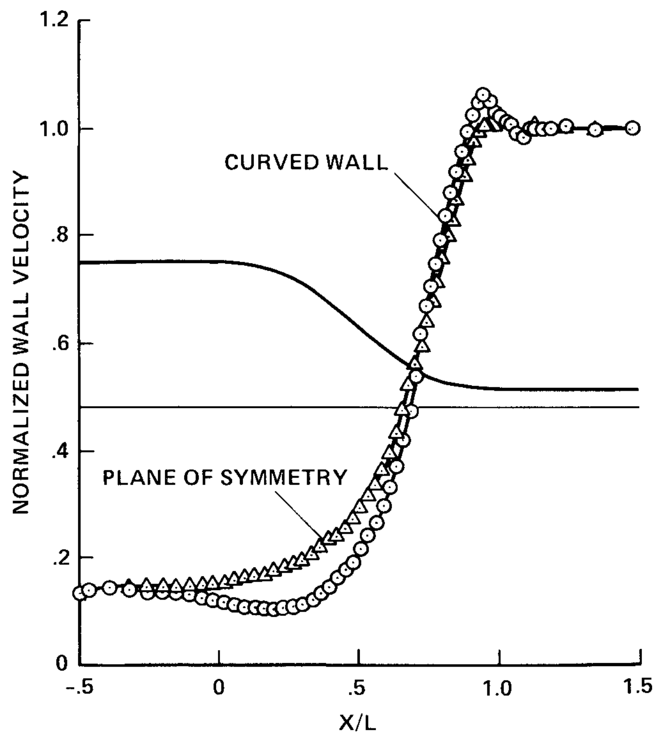

Fig. 1 Typical calculated wall velocities.

Received March 21, 1988; revision received June 2, 1988.

*Graduate Student, Department of Aeronautics and Astronautics.
†Research Associate, Department of Aeronautics and Astronautics. Member AIAA.