Prediction of an Axisymmetric Combusting Flow

S. M. Correa*
General Electric Company, Schenectady, New York

A numerical model for turbulent, recirculating combusting flow is described and applied to a research combustor. The model is based on the density-weighted averaged Navier-Stokes equations with an eddy-viscosity turbulence closure. The eddy viscosity is obtained from the two-equation (k-ϵ) turbulence model. Compositional fluctuations at each point are described probabilistically in terms of the mixture fraction. The probability density function is obtained from transport equations for its first two moments together with an assumption regarding its shape. A standard finite difference procedure and line relaxation algorithm are used to solve the resulting equations. Predictions of the model are compared with data from an axisymmetric, bluff-body-stabilized research combustor. The major discrepancies are similar to those found in isothermal flow comparisons. In particular, the k-ϵ turbulence model appears to be inadequate. Peculiar features of this flow which contribute to the errors are identified. The theory-data agreement deteriorates as the central jet velocity is increased, suggesting an enhanced role for the unsteady effects which are experimentally observed.

Introduction

COMPUTATIONAL fluid mechanics has been applied to a wide variety of isothermal, recirculating, turbulent flows. While major issues still remain, the overall performance of the models has encouraged extension to reacting flows. The purpose of this paper is to examine current modeling capability by comparison with data from a carefully designed combustion experiment.

There are several new physical problems in combusting flowfields. Large temperature and density gradients can make isothermal turbulence models inappropriate. Both the chemistry of combustion and its interaction with turbulence are very complex. Liquid fuel sprays may be present and undergo atomization, transport, vaporization, and other phenomena. Furthermore, combustion equipment often features strong swirl, radial jets, film cooling, etc. Because of the lack of detailed data on such flows, as well as the ambiguities caused by trying to model several new complex phenomena all at once, simpler experiments have been designed. One such experiment is the axisymmetric, propane-fueled, bluff-body-stabilized combustor[1] at the Air Force Aeronautical Propulsion Laboratory (AFAPL).

The AFAPL experiment is shown schematically in Fig. 1. It consists of a cylindrical flameholder in a tunnel. The flameholder has a narrow fuel jet at its center; propane at several different flow rates is used as the fuel. The annulus air inlet velocity may also be varied. Depending on the relative velocities of the jet and annular flows, very different overall flow patterns are observed.

For low fuel flow rates the central jet exhibits an on-axis stagnation point (S1). The reverse flow region along the centerline indicates the dominance of the annular flow; there is a second stagnation point further downstream (S2). Conversely, when the fuel jet is dominant there are no on-axis stagnation points. As the fuel flow rate is increased for a constant annular velocity, the stagnation points move downstream until they disappear at some critical jet velocity. Laser Doppler velocimetry (LDV) was used to support these qualitative observations.

Presented as Paper 83-1264 at the AIAA/SAE/ASME 19th Joint Propulsion Conference, Seattle, Wash., June 27-29, 1983; received July 12, 1983; revision received Dec. 21, 1983.

*Aerospace Engineer, Corporate Research and Development. Member AIAA.

This experiment provides a data base for assessing future models for gas turbine combustors. In simplifying the flow, however, some new sensitivities are introduced. The narrow fuel jet necessitates a highly nonuniform grid with attendant computational difficulties; furthermore, the spreading rate of a jet even in a coflowing stream is a difficult problem in its own right. Another important aspect of this flow is the stabilization of the flame by the recirculation in the wake of the bluff body. Although geometrically simpler than the combination of swirl and primary zone jets used in most combustors, the annular shear layer so formed plays a very important role in this flowfield. In gas turbine combustors, for example, there is no similar feature. Thus the dynamics of the shear layer may not be very relevant. In later experiments on the AFAPL combustor,[2,3] it became clear that the flame had a significant dynamic character. For this reason, all of the comparisons reported herein are for the maximum annulus (air) flow rate, which resulted in the most stable behavior when the fuel flow rate was a minimum.[3] Comparisons for the higher fuel flow rates are also presented.

Description of Model

The model is based on the density-weighted averaged form of the equations for the conservation of momentum, enthalpy, and species. Use of the density-weighted statistics ("Favre averaging") is more convenient in variable-density flows since correlations involving density do not appear. The equations apply to high Reynolds number flow only, so that molecular transport phenomena, such as preferential diffusion of hydrogen, are not allowed for. Closure of the mean flow equations is accomplished by an eddy-viscosity model in which the eddy viscosity

$$\mu_t = C_\mu \bar{\rho} k^2 / \epsilon \tag{1}$$

is obtained from the local density $\bar{\rho}$, turbulence kinetic energy k, and dissipation rate ϵ; C_μ is a constant in the model. The Favre-averaged equations for these two quantities are used. The resulting set of transport equations has been used in many previous studies of isothermal and reacting flows[4] and will not be reproduced here. It should be noted that the Favre-averaged equations resemble the unweighted averaged equations for uniform density flows and are closed with the same constants.

The use of density-weighted statistics rather than explicitly recognizing density correlations, the use of the k-ϵ model, and the use of an isotropic eddy viscosity all may be criticized. The present calculation uses these methods in the interest of practicality. Weaknesses will be seen from disagreement with the data.

A model for the turbulence-chemistry interactions must also be introduced. In turbulent nonpremixed hydrocarbon flames, the major species may be considered to be in equilibrium as far as density and temperature fields are concerned. Nonequilibrium radical concentrations and slow pollutant kinetics do not alter the density or temperature significantly. However, turbulent fluctuations in the local concentration lead to the phenomenon of "unmixedness."[5] Models must account for these random turbulent fluctuations. The average temperature (or density) is not simply the equilibrium value corresponding to the average local concentration, but rather the average of the local fluctuating temperature

$$\bar{T}\neq f^{eq}(\bar{c}) \qquad \bar{T}=\overline{f^{eq}(c)}$$

where $(\bar{c})$ denotes an average, c is the concentration, and $f^{eq}(c)$ is the instantaneous equilibrium temperature corresponding to c. The reason for this distinction is the strong nonlinearity of f^{eq}. Because the mean density "drives" the hydrodynamics, an accurate calculation of the average quantities is very important. The approach used herein has found wide application in practical calculation methods.[4]

The problem is considered in terms of probability. A probability density function (pdf) is obtained (by some particular method as discussed next) at each point in the flowfield. The pdf $P(c,x)$ is such that $P(c,x)\mathrm{d}c$ gives the probability of the instantaneous concentration of the fluid at x being in the range c to $c+\mathrm{d}c$. The normalization follows:

$$\int_0^\infty \cdots \int_0^\infty P(c,x)\mathrm{d}c=1 \tag{2}$$

There are several approaches of varying complexity to generating the probability $P(c,x)$. In each, any mean quantity at a point x is given by

$$\bar{Q}(x)=\int_0^\infty \cdots \int_0^\infty Q^{eq}(c)P(c,x)\mathrm{d}c \tag{3}$$

where $Q^{eq}(c)$ is the instantaneous equilibrium value of the quantity Q corresponding to the instantaneous concentration c. Favre averages result if the Favre pdf is used; the density $\rho^{eq}(c)$ then allows calculation of either average. The local pdf $P(c,x)$ thus may be considered as a weighting function encapsulating local turbulence/chemistry interactions. The model used herein consists of solving transport equations for the first two moments and using them to generate the local pdf. This involves assuming a shape for the pdf with unknown mean and standard deviation at each point.

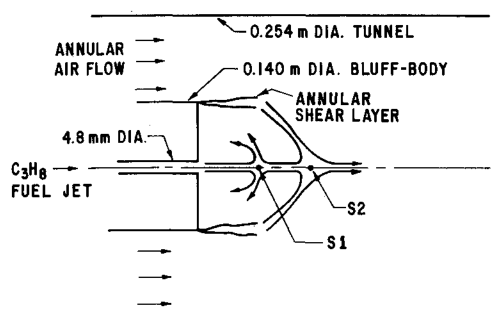

Fig. 1 Schematic of AFAPL combustor showing centerline stagnation points S1 and S2.

Rather than obtaining the pdf from a finite number of its moments and an assumption regarding its shape, pdf evolution equations may be derived from the Navier-Stokes and species conservation equations.[6] A joint pdf for the velocity components and scalars (concentrations, enthalpy) results. Turbulent transport and chemical kinetic source terms appear in closed form. Due to the large number of independent variables, a finite difference solution is not practical and a Monte Carlo scheme is required. Although promising, this model is not ready for application to recirculating flows. If only the scalar(s) pdf evolution is sought, a simpler (though still Monte Carlo) model results; however, turbulence assumptions are then required. The assumed-shape pdf model used here is felt to be the most appropriate for present use in combustor modeling, and similar approaches have been tested in simpler flows, for example, by Lockwood and Naguib.[7]

Since chemical equilibrium will be assumed, elemental rather than molecular species concentrations are considered. Elements and total (chemical plus sensible) enthalpy are neither created nor destroyed in reactions. Thus their conservation equations are homogeneous. If all the diffusion coefficients are equal, the "mixture fraction" ξ is usefully defined as

$$0\leq\xi\equiv\frac{Z-Z^a}{Z^f-Z^a}\leq 1 \tag{4}$$

where a and f refer to air and fuel streams and Z is any elemental mass fraction or total enthalpy. A value of the conserved scalar ξ corresponds to a known instantaneous concentration of the elements and total enthalpy. The Gibbs free energy for the system is a minimum in chemical equilibrium which, therefore, may be determined for a given ξ. The thermochemical package CREK[8] was built into the model to find the equilibrium density, temperature, and species concentration for each mixture fraction. These equilibrium quantities were then stored for use as required during solution of the conservation equations. This is more accurate than assuming a single-step irreversible reaction because large concentrations of CO exist for rich mixtures and make a large difference in the temperature.[4]

The definition of ξ implies that the elemental and total enthalpy transport equations and boundary conditions map into that for ξ. The transport equation for ξ is Favre averaged (density-weighted average) to obtain the equations for the mean $\tilde{\xi}$ and the variance $\widetilde{\xi''^2}$.[9]

Experiments in recirculating flows show that the fluctuations in ξ may be represented by the β probability density function (β pdf)[10,11]:

$$\tilde{P}(\xi)=\xi^{a-1}(1-\xi)^{b-1}\Big/\int_0^1 \xi^{a-1}(1-\xi)^{b-1}\mathrm{d}\xi \tag{5}$$

where the parameters at each point x are given by

$$a=\tilde{\xi}\left[\frac{(1-\tilde{\xi})\tilde{\xi}}{\widetilde{\xi''^2}}-1\right] \qquad b=\frac{1-\tilde{\xi}}{\tilde{\xi}}a \tag{6}$$

These relations have been obtained by solving for the mean $\tilde{\xi}$ and variance $\widetilde{\xi''^2}$ of the pdf given by Eq. (5) in terms of a and b and inverting. Realizability constraints on the fluctuations

$$0\leq\widetilde{\xi''^2}\leq\tilde{\xi}(1-\tilde{\xi}) \tag{7}$$

i.e., the variance cannot be greater than that due to alternating pure air and fuel eddies, lead to

$$a\geq 0, \qquad b\geq 0 \tag{8}$$

However, a or $b<1$ cause singularities $\xi=0$ or 1 which must be analytically removed before convoluting any quantity with the pdf. The β pdf is more rigorous than a clipped Gaussian which does not necessarily go to zero at $\xi=0, 1$.

The transport equations for the first two moments of the pdf are solved together with the equations for conservation of momentum, turbulence kinetic energy k, and dissipation rate ϵ. The continuity equation is not of the same convection-diffusion-source form and requires special treatment. Here the SIMPLE[12] (Semi-Implicit Method for Pressure Linked Equations) algorithm is used: the momentum and continuity equations are manipulated to obtain a Poisson equation for the pressure field. This equation is linearized and solved with the same line-by-line relaxation method used for the other equations. The velocity field (obtained from solution of the momentum equations with the last known pressure field) is then updated to satisfy continuity. While faster algorithms can be devised, they will not affect the issue of accuracy. The equations are discretized in the staggered grid system in which velocity and scalar nodes are offset by half a grid spacing.

Discretization of the convection term is another important issue. Here the hybrid differencing scheme is used; efforts to ensure second-order accuracy are described below. Finally, convergence of this iterative scheme was decided on the basis of the maximum residual—the sum of the absolute out-of-balance terms for each control volume due to the finite difference equations not being satisfied—in the system; iteration continued until this global measure was satisfied to within prescribed tolerances. The density at each node was also monitored and converged to within a specified tolerance; density corrections during iteration were underrelaxed.

Results

Isothermal flow modeling in the AFAPL combustor has been reported by other authors.[13] Those results were broadly confirmed by the present study and will not be discussed in detail here. For the combusting flow, the annulus-dominated case was modeled first. A baseline simulation was made after which the boundary conditions were varied as described below. The higher fuel flow rates were then predicted.

Measured boundary conditions were used where possible. Radial profiles of axial velocity (mean and rms) were obtained in the annulus from which the turbulence kinetic energy (tke) may be calculated. Dissipation rate boundary conditions were obtained by taking the length scale of turbulence ($\propto k^{3/2}/\epsilon$) to be 3% of the annulus height or the jet diameter as appropriate. In the jet a turbulent pipe flow was assumed with the same ratio of tke to mean velocity squared as at the first centerline measurement location.

The length of the computational domain (approximately six bluff-body diameters) was adjusted until the solutions were unaffected. The exit condition was that the axial gradients of the dependent variables should vanish. A Couette flow analysis was used at solid boundaries and symmetry was used at the centerline to complete the set of boundary conditions. The grid consisted of 51×36 nodes clustered in the jet and the annular shear layer. Numerical truncation error due to the use of hybrid differencing on this grid is analyzed below.

Baseline Simulation

Dividing streamlines for the baseline case (4 kg/h propane) show the overall size and shape of the recirculation zone (Fig. 2). The stagnation points are located further away from the bluff body than in the isothermal case with the same jet velocity. This is predicted by the model; however, as seen from the axial velocity profile along the centerline (Fig. 3), the first stagnation point (S1) is too near the bluff body, while the second (S2) is too far. The average annulus air velocity $\bar{u}_a$ and the bluff-body diameter D are used to scale the results. It should be noted that the prediction is for the Favre-averaged velocity.

The overprediction of the size of the centerline reverse flow region is seen in the isothermal flow also. The jet spreads too rapidly; numerical diffusion caused by the second-order truncation error incurred in upwind differencing may be responsible as discussed below. Another serious difficulty is that the standard k-ϵ model does in fact overestimate the spread rate of round jets. Pope[14] has suggested that "hoop stretching" in axisymmetric flows leads to a spectral energy transfer which enters the k-ϵ model as an additional source term in the dissipation rate equation. The modified model predicted the spread-rate difference between round and plane jets successfully. However, experience in recirculating flows[15] shows that in regions of negative radial velocity the realizability condition on ϵ, i.e., $D\epsilon/Dt\to0$ as $\epsilon\to0$, is violated and stable solutions cannot be obtained. Thus the jet spread can represent a significant source of error which affects the overall prediction.

Since the model gives the density-weighted turbulence kinetic energy, direct quantitative comparisons with LDV data cannot be made. This is in contrast to the situation for quantities related to the mixture fraction; the calculated pdf for the latter can be used to generate either the density-weighted or the conventional mean of any quantity. With this caveat, the centerline development of tke is compared with the measurements in Fig. 4. Combustion causes the levels of tke to increase and this was in fact predicted but the peak levels are too low. This discrepancy is found in the isothermal case also. Several reasons for the discrepancy have been identified in previous studies; for example, the k-ϵ model does not represent normal stresses correctly and fails to account for the transport of Reynolds stresses.

The assumption of isotropic eddy viscosity is another potential source of error in the model. Streamline curvature is of the stabilizing type in the annular shear layer and leads to a reduction in the radial transport of axial momentum. Algebraic modifications of the two-equation turbulence model have been developed for various flows[16] but the more fundamental approach—solution of the full set of Reynolds stress transport equations—is currently impractical. However, because the recirculation zone is larger for the combusting flow in the AFAPL experiment, the role of streamline curvature is expected to be less than in the isothermal case. Another weakness is seen in the region past

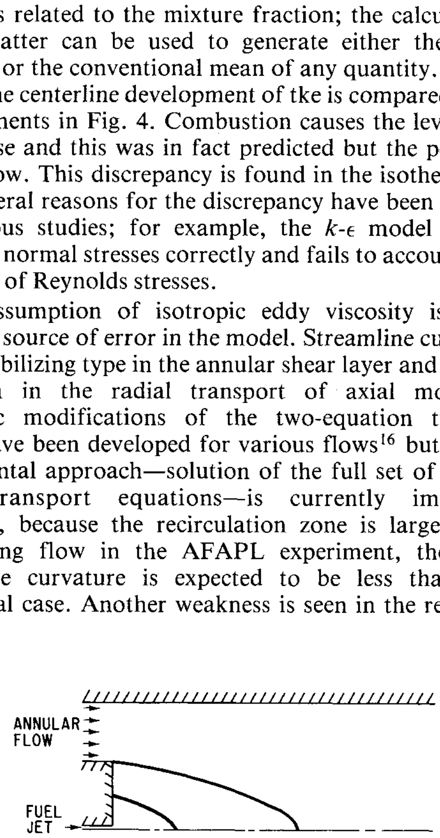

Fig. 2 Dividing streamlines from baseline simulation.

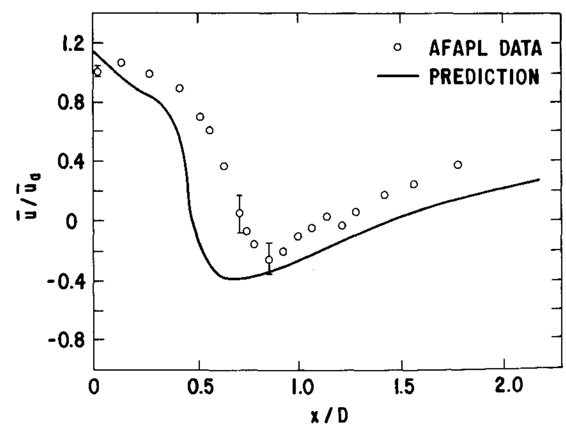

Fig. 3 Centerline axial velocity.

the recirculation zone, where the flow acquires the "weak-shear" character in which dissipation of tke exceeds production. This is also seen in simulations of other flows.[15] Under these conditions, the constant C_μ in the k-ϵ model becomes a function of the ratio of production to dissipation.[16] Failure to account for this in the standard k-ϵ model leads to the slow recovery of the predicted flow. Another source of error is the implicit assumption that density weighting accounts for the effects of heat release.

Contours of tke (Fig. 5) show that the peak in the shear layer occurs well downstream of the edge of the bluff body; the peak contours along the centerline are not shown because they are too closely spaced. Contour data were not available for the AFAPL experiment, but a downstream shift between isothermal and combusting flow was observed by Taylor[15] in a series of disk- and cone-stabilized premixed confined flows.

In isothermal confined flows the failure to properly predict the tke levels leads to incorrect turbulent transport terms which are often dominated by the mean pressure gradient.[17] In a combusting flow, however, the time scale for the dissipation of scalar fluctuations is obtained from the k-ϵ model, and, therefore, errors in the tke affect the solution for the variance $\widetilde{\xi''^2}$. Furthermore, $\widetilde{\xi''^2}$ is used together with the mean $\tilde{\xi}$ to generate the mean density field which occurs in all of the equations. The turbulence model is thus more consequential in a combustion flow.

Contours of the mean mixture fraction $\tilde{\xi}$ and the variance $\widetilde{\xi''^2}$ (Figs. 6 and 7) show that the flame forms a closed ellipsoidal shape. This is in accordance with the observations. A detailed assessment of the results requires instantaneous species concentration measurements which are not available at present. Such data would indicate the errors caused by the gradient-diffusion closure assumption, the influence of pressure-gradient correlations, and the assumption of a β pdf.

A radial density-weighted temperature profile in the recirculation zone shows general agreement with thermocouple data (Fig. 8). The temperature is a derived quantity which involves assuming the shape of the pdf taken here as a β function. Intermittency is known to cause "spikes" in the pdf; the spikes are located at $\xi=0$ near the air stream and at $\xi=1$ near the fuel stream. In the annular shear layer this could explain why the predicted temperature gradient is steeper than measured. Intermittency has been recognized in free combusting jets[18,19] by assuming that the total pdf is composed of a delta function plus a continuous part which represents the turbulent flow. The relative contributions are empirically based on free jet data and, therefore, could not be used here. Near the jet intermittency is less significant; near-stoichiometric values of ξ ($\simeq 0.06$) are not expected to occur, therefore, the temperature fluctuations will not be as large. The overprediction of centerline temperature may be due to assuming instantaneous chemical equilibrium. The fuel-rich and, therefore, relatively cool jet is probably kinetically limited.

The calculation of a recirculating flow must include an estimate of the numerical error incurred in approximating spatial derivatives with finite differences. In this study, the diffusion terms are approximated with second-order accurate central differences, while the well-known "hybrid" scheme is used for the convection terms. In regions where the grid is fine enough for the absolute cell Peclet (Reynolds) number to be less than two, the hybrid scheme uses a central difference representation of the convection term. This scheme is second-order accurate but leads to a loss of diagonal dominance and, therefore, instability for coarser grids. To obtain solutions in the latter case upwind differencing (first-order accurate) is used. The principal consequence is that numerical diffusion, caused by the second-order truncation error in the discretization, must be suspected wherever the absolute cell Peclet number exceeds two. Such regions do occur along the jet, in a lobe along the inner dividing streamline, and again outside the recirculation zone as seen from east-west (E-W), and north-south (N-S) Peclet number maps (Figs. 9 and 10). The jet, however, is convection-dominated and so upwinding in the E-W direction is acceptable; in the radial direction, central differencing was used except in the lobe. The region outside the recirculation zone is not expected to be significant.

The truncation error may be analyzed in several ways. A local measure of the error, which accounts for streamline-to-grid skewness as well as the cell size, was derived by deVahl Davis and Mallinson.[20] Another, more elaborate, technique entails plotting the various terms, e.g., net convection, pressure gradient, and so on, in the momentum (or any other) equation and comparing them with the local second-order truncation error. These analyses indicated that the major errors occurred in the lobe region. The grid was refined on this basis. This approach will not be practical in three-dimensional flows.

To reduce numerical diffusion errors for coarser grids the QUICK scheme[21] was also tried. The scheme uses quadratic

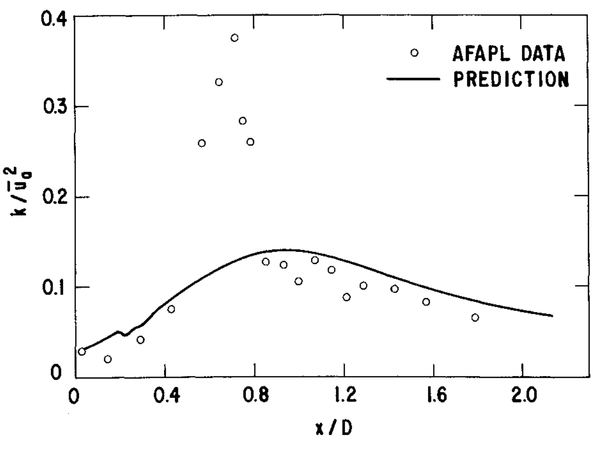

Fig. 4 Centerline turbulence kinetic energy.

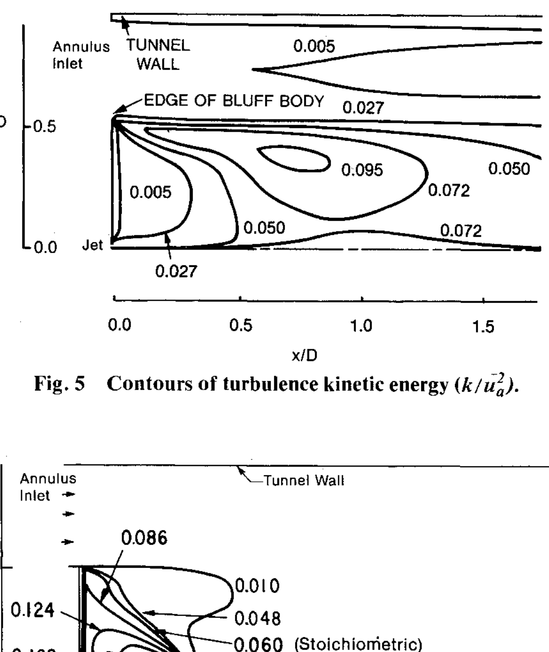

Fig. 5 Contours of turbulence kinetic energy ($k/\bar{u}_a^2$).

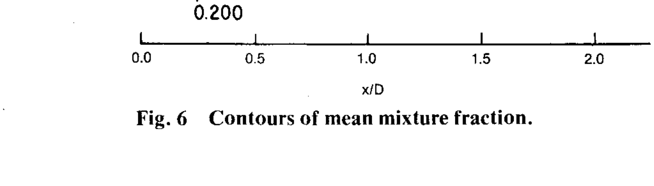

Fig. 6 Contours of mean mixture fraction.

interpolation over upstream nodes to generate control-volume face values. Most reported uses of this scheme have been in the momentum equations; however, in regions where the production and dissipation terms are not dominant, McGuirk et al.[22] have shown that the k equation also can be affected by numerical error. In the combusting case, the scheme is needed in the mean mixture-fraction equation, which has no source terms, and in the variance equation away from the regions of steep gradients. Attempts to implement the scheme in these other equations, however, generally lead to boundedness difficulties in which a variable takes on nonphysical values such as the variance becoming negative. Thus a reliable and accurate alternative to upwind differencing is required.

Variation of the Boundary Conditions

Annulus and jet exit velocity, turbulence kinetic energy, and dissipation rate are required inputs for the model. Although measurements or reasonable guesses are available for most of these, it is worthwhile to study their influence on the predictions.

In the baseline simulation there was no radial component to the annulus inlet flow. When this condition is modified by taking the radial velocity to be

$$\bar{v} = -5\% \bar{u}$$

the second stagnation point moves about 6% closer to the bluff body while the first stagnation point is unaffected. This causes better agreement with the data but cannot be used to replace the baseline simulation unless radial components are in fact measured. The levels of tke are unaffected except on the bluff-body face so that the extra strain appears insignificant.

When the inlet turbulence length scale in the annulus is halved, both of the stagnation points move away from the bluff body. Because the inlet dissipation rate is effectively doubled the upstream levels of tke are reduced, leading to less radial transport across the annular shear layer. By about 1.5 bluff-body diameters downstream the radial tke profiles are about the same as in the baseline case. This sensitivity is unfortunate because length-scale boundary conditions will entail two-point or autocorrelative measurements in the annulus.

Varying the inlet turbulence level in the jet does not alter the predicted stagnation points. The shear-generated turbulence at the edge of the jet essentially dominates the initial condition.

Higher Fuel Flow Rates

On increasing the jet exit velocity the stagnation points S1 and S2 are observed to move away from the bluff body until at a flow rate of about 8 kg/h the jet penetrates along the centerline.[1] The model was used to study these higher fuel flow rates. The stagnation points do move outward although the underprediction of S1 remains (Fig. 11). By monitoring the minimum centerline axial velocity as a function of the jet exit velocity (Fig. 12), the breakthrough point can be estimated. The predicted breakthrough flow rate of about 12 kg/h is much higher than the observed 7-8 kg/h; again, the prediction is for the density-weighted mean velocity.

The agreement between theory and data deteriorates as the jet velocity increases. In the "transitional" regime where neither the jet nor the annulus flow dominates, unsteady effects due to acoustic resonance, inflow perturbations, or heat-release feedback phenomena may make the predictions very inaccurate. Later experiments have revealed distinct axial slugs of flame, named "turbules,"[2] which convect down-

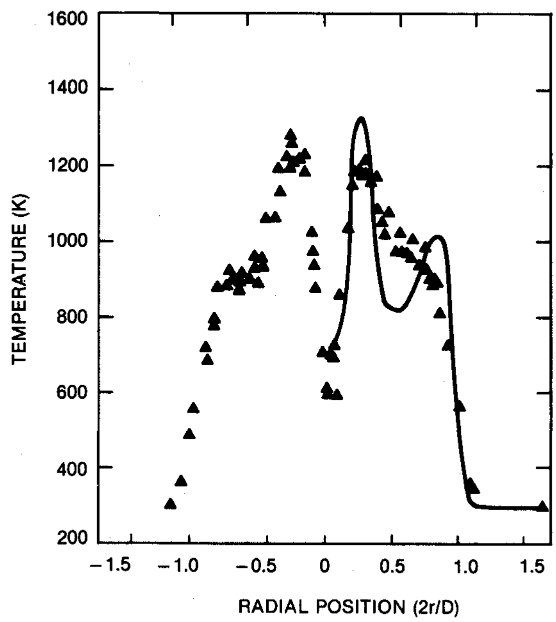

Fig. 8 Radial temperature profile at $x/D = 0.43$.

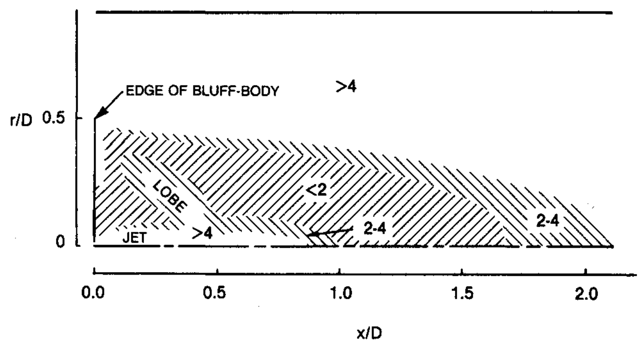

Fig. 9 Absolute east-west cell Peclet numbers.

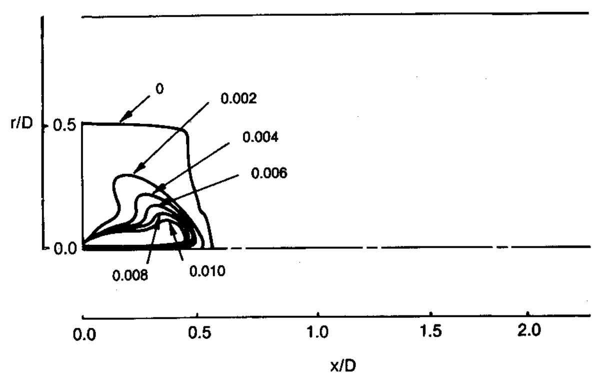

Fig. 7 Contours of variance in mixture fraction.

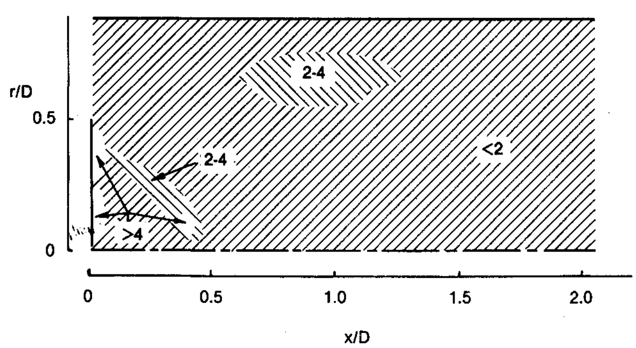

Fig. 10 Absolute north-south cell Peclet numbers.

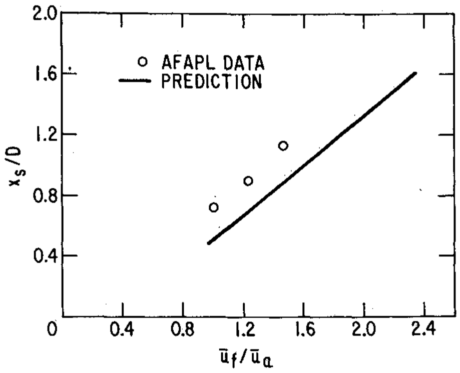

Fig. 11 Location of first stagnation point for higher jet velocities.

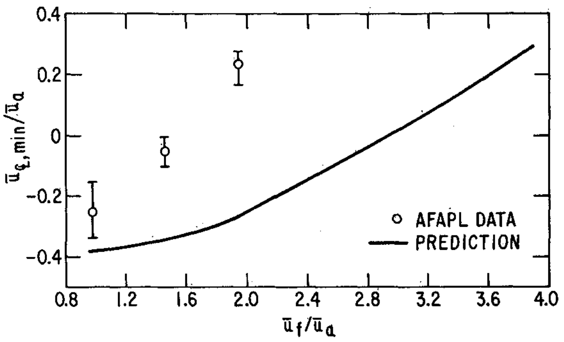

Fig. 12 Minimum centerline axial velocity for higher jet velocities.

stream. The axial velocity in the turbules, which were present about 74% of the time, is much greater than in the intervening nonluminous regions.[3] The flow is apparently unsteady with the possibility of intermittent jet penetration. This condition is most pronounced for the higher fuel flow rates.[3]

While the current model accounts for random fluctuations in velocity and concentrations, periodicity is not addressed. Any "coherent" structures in the flow thus affect the accuracy of the predictions. In a similar experiment[23] such structures were not observed unless the annular shear-layer was acoustically excited. The flow was then significantly affected. Whether these structures are significant in practical combustors and must therefore be included in numerical models is not entirely clear at present.

Conclusions

An axisymmetric bluff-body-stabilized research combustor has been modeled numerically. While general agreement with the measurements is obtained, there are some significant discrepancies. These may be divided into two categories: some are particular to this flowfield, while the others are of general significance.

Flame stability in the AFAPL experiment is obtained by the recirculation of hot combustion products in the wake of the axisymmetric bluff body. Simplification of the geometry, however, makes the entire flow very sensitive to the development of the annular shear layer. Periodic behavior and intermittency in the shear layer will affect the local temperatures and densities and the overall recirculation zone length. Another atypical aspect of the combustor is the sensitivity to the fuel jet. Proper prediction of the spread rate of round free jets is known to require a modification in the k-ϵ model. In the AFAPL combustor, an error in the jet spread affects the stagnation points and so the entire flowfield.

On the other hand, some general conclusions may be drawn:

1) The k-ϵ model performs about as well as in the isothermal flow. The observed peak in turbulence kinetic energy at the first stagnation point is not predicted. These errors can be attributed to many theoretical reasons; overall, the k-ϵ model seems to be inadequate in recirculating flows.

2) Contours of the mean and variance of mixture fraction for the lower fuel flow rates show that the flame has a closed, ellipsoidal shape as observed. Data for these quantities are needed before a meaningful assessment of the predictions can be made.

3) Radial temperature profiles show the effects of assuming equilibrium in rich regions and of neglecting intermittency in the annular shear layer.

4) Numerical diffusion errors have been minimized by selective refinement of the grid in important regions which permits central differencing of the convection term. In a three-dimensional flow this would require much storage and does not currently appear practical. Thus a reliable, i.e. "wiggle"-free, second-order accurate numerical scheme is required to replace upwind differencing on coarse grids.

5) Theory-data discrepancies are larger for the higher jet exit velocities. The unsteadiness observed in this "transitional" flow regime becomes increasingly significant. Whether this behavior is intrinsic to all such experiments and significant for practical combustors is not clear at present.

Acknowledgments

Part of this work was supported by NASA Contract NAS3-23525. Many helpful discussions with Dr. W. M. Roquemore of the AFAPL are also gratefully acknowledged.

References

[1]Lightman, A. J. et al., "Velocity Measurements in a Bluff-Body Diffusion Flame," AIAA Paper 80-1544, July 1980.

[2]Roquemore, W. M., Britton, R. L., and Sandhu, S. S., "Investigation of the Dynamic Behavior of a Bluff Body Diffusion Flame Using Flame Emission," AIAA Paper 82-0178, Jan. 1982.

[3]Magill, P. D., Lightman, A. J., Orr, C. E., Bradley, R. P., and Roquemore, W. M., "Flowfield and Emission Studies in a Bluff Body Combustor," AIAA Paper 82-0883, June 1982.

[4]Jones, W. P. and Whitelaw, J. H., "Calculation Methods for Reacting Turbulent Flows: A Review," *Combustion and Flame*, Vol. 48, 1982, p. 1.

[5]Hawthorne, W. R., Weddell, D. S., and Hottel, H. C., "Mixing and Combustion in Turbulent Gas Jets," Third Symposium on Combustion and Flame and Explosion Phenomena, The Williams & Wilkins Company, Baltimore, Md., 1949, p. 266.

[6]Pope, S. B., "Transport Equation for the Joint Probability Density Function of Velocity and Scalars in Turbulent Flow," *Physics of Fluids*, Vol. 24, 1981, p. 588.

[7]Lockwood, F. C. and Naguib, A. S., "The Prediction of the Fluctuations in the Properties of Free, Round-Jet, Turbulent, Diffusion Flames," *Combustion and Flame*, Vol. 24, 1975, p. 109.

[8]Pratt, D. T. and Wormeck, J. J. "CREK: A Computer Program for Calculation of Combustion Reaction Equilibrium and Kinetics in Laminar or Turbulent Flow," Washington State University, Seattle, Wash., 1976, WSU-ME-TEL-76-1.

[9]Spalding, D. B. "Concentration Fluctuations in a Round Turbulent Free Jet," *Chemical Engineering Science*, Vol. 26, 1971, p. 95.

[10]Richardson, J. M., Howard, H. C., and Smith, R. W., "The Relation Between Sampling-Tube Measurements and Concentration Fluctuations in a Turbulent Gas Jet," *Fourth Symposium (International) on Combustion*, The Williams & Wilkins Company, Baltimore, Md., 1953, p. 814.

[11]Rhodes, R. P., Harsha, P. T., and Peters, C. E., "Turbulent Kinetic Energy Analyses of Hydrogen-Air Diffusion Flames," *Acta Astronautica*, Vol. 1, 1974, p. 443.

[12]Patankar, S. V. and Spalding, D. B., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," *International Journal of Heat and Mass Transfer*, Vol. 15, 1972, p. 1787.

[13]Sturgess, G. J. and Syed, S. A., "Widely-Spaced Co-Axial Jet, Diffusion-Flame Combustor: Isothermal Flow Calculations Using the Two-Equation Turbulence Model," AIAA Paper 82-0113, Jan 1982.

[14]Pope, S. B., "An Explanation of the Turbulent Round-Jet/Plane-Jet Anomaly," *AIAA Journal*, Vol. 16, 1978, p. 279.

[15]Taylor, A. M. K. P., "Confined, Isothermal and Combusting Flows Behind Axisymmetric Baffles," Ph.D. Thesis, University of London, England, 1981.

[16]Rodi, W., *Turbulence Models and Their Application in Hydraulics*, International Association for Hydraulic Research, Delft, the Netherlands, 1980.

[17]Green, A. and Whitelaw, J. H., "Measurements and Calculations of the Isothermal Flow in Axisymmetric Models of Combustor Geometries," *Journal of Mechanical Engineering Science*, Vol. 22, 1980, p. 119.

[18]Kent, J. H. and Bilger, R. W., "The Prediction of Turbulent Diffusion Flame-fields and Nitric Oxide Formation," *Sixteenth Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, Pa., 1977, p. 1643.

[19]Drake, M. C., Bilger, R. W. and Starner, S. H., "Raman Measurements and Conserved Scalar Modelling in Turbulent Diffusion Flames," *Nineteenth Symposium (International) on Combustion*, The Combustion Institute, Pittsburgh, Pa., 1982, p. 459.

[20]deVahl Davis, G. and Mallinson, G. D., "An Evaluation of Upwind and Central Difference Approximations by a Study of Recirculating Flow," *Computers and Fluids*, Vol. 4, 1976, p. 29.

[21]Leonard, B. P., "A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation," *Computer Methods in Applied Mechanics and Engineering*, Vol. 19, 1979, p. 59.

[22]McGuirk, J. J., Taylor, A.M.K.P., and Whitelaw, J. H., "The Assessment of Numerical Diffusion in Upwind-Difference Calculations of Turbulent Recirculating Flows," *3rd Symposium on Turbulent Shear Flows*, Springer-Verlag, 1981, p. 206.

[23]Correa, S. M. and Pitz, R. W., "The Effect of Large-Scale Structures on Eulerian Models of Turbulent Combusting Flows," Paper 2, Eastern States Section of the Combustion Institute, Fall Technical Meeting, Atlantic City, N.J., 1982.