

NASA Technical Memorandum 104466  
ASME-91-GT-217

# CFD Analysis of Jet Mixing in Low NO<sub>x</sub> Flametube Combustors

M.V. Talpallikar and C.E. Smith  
*CFD Research Corporation*  
*Huntsville, Alabama*

M.C. Lai  
*Wayne State University*  
*Detroit, Michigan*

and

J.D. Holdeman  
*Lewis Research Center*  
*Cleveland, Ohio*

Prepared for the  
36th International Gas Turbine and Aeroengine Congress and Exposition  
sponsored by the American Society of Mechanical Engineers  
Orlando, Florida, June 3-6, 1991

The NASA logo, consisting of the word "NASA" in a bold, sans-serif font with a stylized "A" that has a horizontal bar extending to the right.

CFD ANALYSIS OF JET MIXING IN LOW NO<sub>x</sub>  
FLAMETUBE COMBUSTORS

M. V. Tatpallikar and C. E. Smith  
CFD Research Corporation  
Huntsville, AL 35805

M. C. Lai  
Wayne State University  
Detroit, MI 48202

J. D. Holdeman  
NASA Lewis Research Center  
Cleveland, OH 44135

ABSTRACT

The Rich-burn/Quick-mix/Lean-burn (RQL) combustor has been identified as a potential gas turbine combustor concept to reduce NO<sub>x</sub> emissions in High Speed Civil Transport (HSCT) aircraft. To demonstrate reduced NO<sub>x</sub> levels, cylindrical flametube versions of RQL combustors are being tested at NASA Lewis Research Center. A critical technology needed for the RQL combustor is a method of quickly mixing by-pass combustion air with rich-burn gases.

In this study, jet mixing in a cylindrical quick-mix section was numerically analyzed. The quick-mix configuration was five inches in diameter and employed twelve radial-inflow slots. The numerical analyses were performed with an advanced, validated 3-D Computational Fluid Dynamics (CFD) code named REFLEQS. Parametric variation of jet-to-mainstream momentum flux ratio ( $J$ ) and slot aspect ratio was investigated. Both non-reacting and reacting analyses were performed.

Results showed mixing and NO<sub>x</sub> emissions to be highly sensitive to  $J$  and slot aspect ratio. Lowest NO<sub>x</sub> emissions occurred when the dilution jet penetrated to approximately mid-radius. The viability of using 3-D CFD analyses for optimizing jet mixing was demonstrated.

NOMENCLATURE

A	Pre-exponential Factor
D	Diameter of Quick-Mix Section
D <sub>h</sub>	Hydraulic Diameter
EI	Emission Index
E/R	Activation Energy/Gas Constant
J	Jet-to-Mainstream Momentum Flux Ratio
m <sub>i</sub>	Mass Flow in Each Cell i
m <sub>j</sub>	Jet Mass Flow
m <sub>∞</sub>	Mainstream Mass Flow
σ <sub>T</sub>	Mass Weighted Standard Deviation of Temperature
n	Optimum Number of Slots
NO <sub>x</sub>	Oxides of Nitrogen
T <sub>avg</sub>	Mass-Weighted Average Temperature
T <sub>i</sub>	Temperature in Each Cell i
u'	RMS of U Velocity
U	Averaged Axial Velocity
v'	RMS of V Velocity
V	Averaged Radial Velocity

INTRODUCTION

In order to meet the growing need for faster transportation, High-Speed Civil Transport (HSCT) aircraft and associated propulsion systems have been under study in recent years. One major concern that has surfaced concerning HSCT engines is their impact on deteriorating the earth's ozone layer. Using current technology, a fleet of HSCT aircraft would produce large amounts of oxides of nitrogen (NO<sub>x</sub>) while cruising in the stratosphere. Such high levels of NO<sub>x</sub>, through a series of well known reactions, would drastically reduce ozone levels. In order to reduce NO<sub>x</sub> emissions, technology must be developed to design advanced, low emission combustors.

One combustor concept that has been identified as a leading candidate to reduce NO<sub>x</sub> emissions is the Rich-burn/Quick-mix/Lean-burn (RQL) combustor. Originally conceived and developed for industrial combustors (Mosier and Pierce, 1980 and Pierce *et al.*, 1980), the RQL concept utilizes staged burning, as shown in Figure 1. Combustion is initiated in a fuel rich zone at equivalence ratios between 1.2 and 1.8, thereby reducing NO<sub>x</sub> formation by depleting the available oxygen. Bypass combustion air is introduced in a quick-mix section and lean combustion occurs downstream at an overall equivalence ratio between 0.5 and 0.7. The quick-mix section usually has a smaller geometric cross-section area than the rich burn zone in order to prevent backflow and enhance mixing.

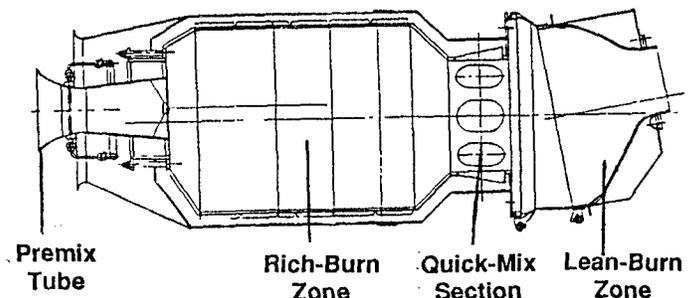


Fig. 1. Industrial Rich-burn/Quick-mix/  
Lean-burn (RQL) Combustor (Pierce *et al.*, 1980)

Perhaps the single most important issue of the RQL concept is the design of the quick-mix section. For previous laboratory combustors,

Tacina (1990) has shown RQL NO<sub>x</sub> levels to be higher than lean, premixed combustor NO<sub>x</sub> levels. The higher NO<sub>x</sub> emissions for RQL was attributed to stoichiometric burning in the quick-mix section, thus emphasizing the need for optimized rapid mix concepts. Indeed, Nguyen *et al.* (1989) have shown that if instantaneous mixing is assumed in the quick-mix section, low NO<sub>x</sub> emission index can be obtained at HSCT cruise flight conditions. Hence, one challenge of the RQL concept is to identify quick-mix sections with rapid mixing.

This study sought to investigate the influence of jet-to-mainstream momentum flux ratio (*J*) and slot aspect ratio (SAR) on mixing effectiveness in a RQL flametube combustor to be tested at NASA Lewis Research Center (LeRC). Conventionally, dilution air in can combustors has been introduced through radial inflow holes. According to Holdeman's correlation (Holdeman *et al.*, 1987), optimum mixing occurs when the following expression is satisfied:

$$n = \pi \frac{\sqrt{2J}}{C} \quad (1)$$

where

- n* = optimum number of holes
- C* = experimentally derived constant ~2.5
- J* = jet-to-mainstream momentum flux ratio ( $\rho_j V_j^2 / \rho_\infty V_\infty^2$ ).

Unfortunately, this correlation was developed for circular holed dilution jet mixing and jet mass flow-to-mainstream mass flow ratios (*m<sub>j</sub>/m<sub>∞</sub>*) of approximately 0.5. The RQL combustor requires *m<sub>j</sub>/m<sub>∞</sub>* of 2.0, thus necessitating slots instead of holes around the can's perimeter. The design of slots for optimum mixing needs further investigation, and was the focus of this study.

#### CFD CODE

The approach in this study was to perform 3-D numerical computations on a cylindrical quick-mix section. The goal of the study was to provide improved understanding of slot injection and mixing. An advanced CFD code, REFLEQS, was used to perform the computations. REFLEQS was developed by CFD Research Corporation (Przekwas *et al.*, 1990 and Smith *et al.*, 1988) to analyze turbulent, reacting flows. The basic capabilities/methodologies in REFLEQS include:

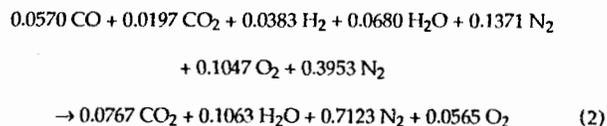
1. solution of two and three-dimensional Navier-Stokes equations for incompressible and compressible flows;
2. cartesian, polar, and non-orthogonal body-fitted coordinates;
3. porosity-resistivity technique for flows with internal blockages;
4. fully implicit and strongly conservative formulation;
5. three differencing schemes: upwind, hybrid, and central differencing with damping terms;
6. standard (Launder and Spalding, 1974) and extended (Chen and Kim, 1987) *k-ε* turbulence models, the two-scale turbulence model of Kim and Chen (1988), and the low-Reynolds number *k-ε* model of Chien (1985);
7. instantaneous, one-step, and two-step combustion models;
8. modified form of Stone's Strongly Implicit Solver; and
9. pressure-based solution algorithms including SIMPLE and a variant of SIMPLEX.

REFLEQS has undergone a considerable amount of systematic quantitative validation for both incompressible and compressible flows. Over 30 validation cases have been performed to date, and good-to-excellent agreement between benchmark data and predictions has been shown (Smith *et al.*, 1989; Ratcliff and Smith, 1989; and Avva *et al.*, 1990). The good agreement gives confidence in the numerical results of this study.

#### HEAT RELEASE MODEL

After reviewing the time scales for heat release at cruise-type conditions in RQL mixers, it was determined reaction rates were much faster than mixing rates. Hence, the combustion process was considered mixing controlled, and instantaneous reaction rates for heat release were assumed.

When rich burn gases (composed of equilibrium concentration of CO, CO<sub>2</sub>, H<sub>2</sub>, H<sub>2</sub>O and N<sub>2</sub>) were mixed with air, they were assumed to react according to the equation:



Accordingly, any CO concentration that remained in the mixer exit was the result of unmixedness.

#### NO<sub>x</sub> MODEL

It was assumed that the NO<sub>x</sub> reactions did not contribute to the overall heat release in the combustor, thus allowing the NO<sub>x</sub> reactions to be "decoupled" from the heat release reactions. NO<sub>x</sub> was calculated as a passive scalar after the computation of the reacting flowfield.

A simple Zeldovich reaction scheme was used to model the NO<sub>x</sub> formation. According to the mechanism, NO formation can be described by:



and



The first reaction is much slower than the second one and hence controls the rate of NO formation. If the concentration of NO is much smaller than the corresponding equilibrium value, the rate equation for NO can be written as:

$$\frac{d(\text{NO})}{dt} = K [\text{N}_2] [\text{O}] \quad (5)$$

Approximating the concentrations of N<sub>2</sub> and O by the local equilibrium values, the rate equation is given by

$$\frac{d(\text{NO})}{dt} = A e^{-\left(\frac{E}{RT}\right)} [\text{N}_2] [\text{O}_2]^{\frac{1}{2}} \quad (6)$$

From Quan *et al.*, (1972), the rate constants were determined to be:

$$A = 5.74 \times 10^4 \frac{1}{\text{sec}} \sqrt{\frac{\text{moles}}{\text{m}^3}} \quad (7)$$

$$\frac{E}{R} = 6.7 \times 10^{11} \text{ }^\circ\text{K} \quad (8)$$

The NO<sub>x</sub> model was calibrated against the experimental results of Anderson (1975), who used a premixed, prevaporized laboratory combustor. The REFLEQS test case consisted of premixed propane and air reacting in a straight channel and instantaneous heat release. The rate constants had to be modified to give good agreement with Anderson's data. The final constants used in this study were:

$$A = 3.3 \times 10^4 \frac{1}{\text{sec}} \sqrt{\frac{\text{moles}}{\text{m}^3}} \quad (9)$$

$$\frac{E}{R} = 1.03 \times 10^{14} \text{ }^\circ\text{K} \quad (10)$$

Figure 2 shows the computed results compared to Anderson's data of Emission Index (EI) as a function of adiabatic flame temperature.

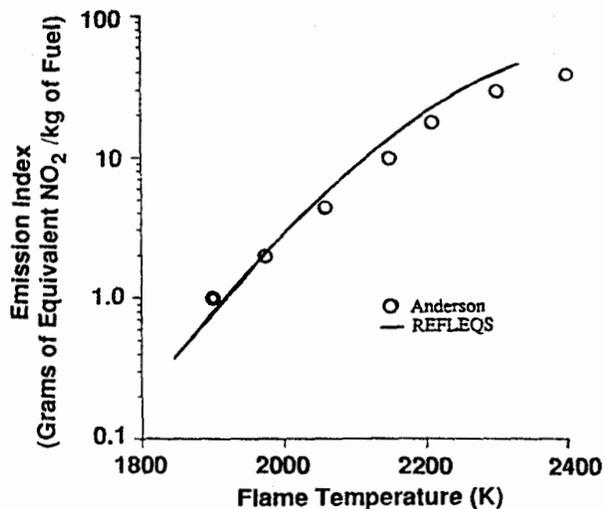


Fig. 2. Calibration of NO<sub>x</sub> Model in REFLEQS

#### DETAILS OF NUMERICAL MODEL

A geometry compatible with the NASA LeRC flametube combustor was selected for analysis. The geometry, numerical grid, numerical details, boundary conditions, grid independence, and convergence criteria are discussed below.

#### Geometry

The geometry of the numerical model consisted of three components: an inlet pipe, converging section and a quick-mix section (see Figure 3). The inlet pipe was 0.152 m (6.0 in.) in diameter and 0.076 m (3.0 in.) in length. The inlet pipe converged into the quick-mix section which was 0.127 m (5.0 in.) in diameter ( $D$ ). The length of the quick-mix section was 0.333 m (13.0 in.). In reality, the length of actual quick-mix section hardware is approximately 6 inches, but the computational domain is extended for better understanding of NO<sub>x</sub> formation and to eliminate flowfield contamination by exit boundary conditions.

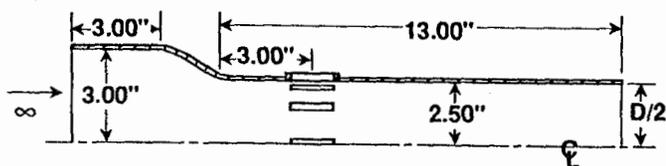


Fig. 3. Schematic of Quick-Mix Geometry

Twelve slots were located symmetrically around the perimeter of the quick-mix section. The axial location of the slot centerline was 0.076 m (3.0 in.) from the inlet of the quick-mix section. The baseline slots were rectangular in shape with an aspect ratio of four and aligned in the streamwise direction. Three variations in slot aspect ratio (SAR) were tested: 1, 4 and 16.

Due to geometric symmetry, only one slot was modeled with planes of symmetry set up halfway between adjacent slots. This allowed greater grid resolution and reduction of computer turnaround time. The  $r\theta$  domain was reduced to a pie section with a central angle of thirty degrees.

#### Grid

A baseline grid of 9,216 cells (32x16x18 in  $x, r, \theta$  directions) was selected and used for modeling the mixer. The grid is shown in Figure 4. Note that the origin of the coordinate system is located at the center of the slot. The axial grid spacing is dense near the slot, and gets coarser

upstream and downstream of the slot. The grid in the radial direction was non-uniform with greater density near the combustor wall (power expansion of 1.2). The grid in the transverse direction was uniform in the slot, and slowly expanding away from the slot. The baseline slot was represented by a 6x6 mesh. As will be discussed in grid independence studies, this rather coarse grid is not grid-independent, but it does capture all of the relevant flow features. For comparative studies, it was felt sufficient.

#### Numerical Details

The numerical details of the calculations included:

1. Whole field solution of u-momentum, v-momentum, w-momentum, pressure correction, turbulent kinetic energy  $k$ , dissipation rate  $\epsilon$ , total enthalpy, and mixture fraction;
2. Upwind Differencing for parametric studies of  $J$  and central differencing for parametric studies of SAR;
3. Variable Fluid Properties (i.e. temperature dependency of specific heat, laminar viscosity, etc.);
4. Adiabatic Walls;
5. Standard  $k-\epsilon$  Model with wall functions;
6. Turbulent Prandtl number of 0.9;
7. Instantaneous heat-release model; and
8. Six active chemical species.

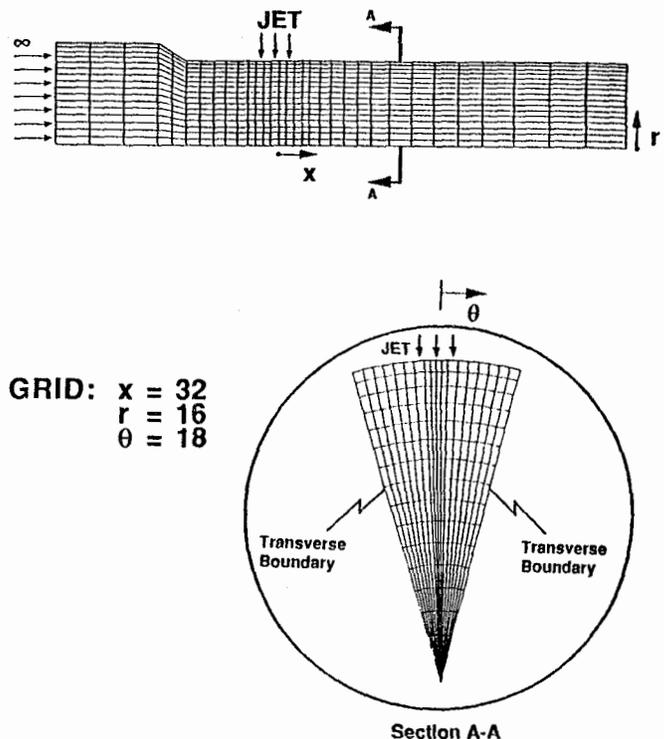


Fig. 4. Baseline Grid

#### Boundary Conditions

**Mainstream Boundary.** At the mainstream inlet boundary, propane and air are assumed to have completely reacted at an equivalence ratio ( $\phi$ ) of 1.6. The species and temperature of the reaction products were taken from the JANNAF-standard rocket code named One Dimensional Equilibrium (ODE) (Nickerson *et al.*, 1989).

Velocity and pressure were obtained from experimental test plans for the RQL flametube combustor. A uniform velocity profile was assumed with a turbulent intensity typical of primary zones in gas turbine combustors and a turbulent length scale corresponding to a turbulent viscosity 1000 times greater than laminar viscosity. The mainstream inlet conditions (at 6.0 in. diameter) were:

Axial velocity	=	35.5 m/s
Temperature	=	2221°K
Density	=	2.32 kg/m <sup>3</sup>
Composition (mass fraction)	=	0.134 CO, 0.068 CO <sub>2</sub> , 0.006 H <sub>2</sub> , 0.096 H <sub>2</sub> O, 0.696 N <sub>2</sub>
Turbulent intensity ( $u'/U$ )	=	50%
Turbulent length scale ( $l_t/D_\infty$ )	=	0.02

Since equilibrium NO<sub>x</sub> levels are very low for  $\phi$  of 1.6 ( $\approx$  4 ppm), no NO<sub>x</sub> was assumed in the mainstream inlet.

**Jet Inlet.** The composition at the dilution jet inlet was assumed to be air. A uniform velocity profile was assumed and turbulent properties were selected using the same logic as discussed for mainstream turbulence. The jet inlet flow conditions were:

Mass flux ratio ( $m_j/m_\infty$ )	=	1.94
Jet temperature	=	811°K
Density	=	6.35 kg/m <sup>3</sup>
Composition (mass fraction)	=	0.232 O <sub>2</sub> , 0.768 N <sub>2</sub>
Turbulent intensity ( $v'/V$ )	=	10%
Turbulent length scale ( $l_t/D_h$ )	=	0.13

The momentum flux ratio ( $J$ ) was varied parametrically from 16 to 64 by variation of jet velocity from 120 m/s to 240 m/s. The jet velocity variation corresponded to liner pressure drops ( $\Delta P/P$ ) of 3 to 12 percent. For each jet velocity, the slot flow area was modified to maintain constant jet flow.

**Exit Boundary.** The exit boundary condition was a zero gradient boundary condition.

**Transverse Boundaries.** The transverse boundaries were assumed to be symmetry planes. These boundaries were also tested for possible outflow by setting them to be periodic boundaries. No discernable difference was seen between cases with symmetric and periodic transverse boundaries.

**Combustor Wall.** The combustor wall was treated as a no-slip adiabatic wall (zero enthalpy gradient). Wall functions were used for the calculations of wall shear stress and near-wall turbulent quantities ( $k$  and  $\epsilon$ ).

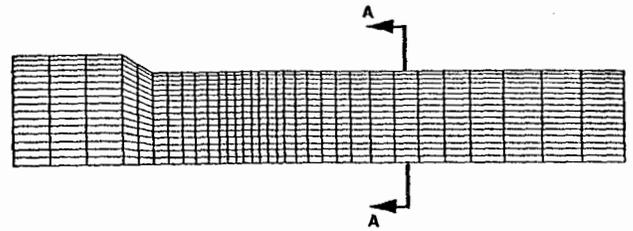
**Centerline.** The computational boundary at the centerline was assumed to be a symmetry plane.

**Grid Independence**

Two different sizes of grids were run to test grid independence: 9,216 and 52,650 cells. The finer grid was obtained by increasing the grid density by ~75% in all three directions. Comparison of the two grids is shown in Figure 5.

Computational results from the two grids are presented in Figure 6 for a momentum flux ratio ( $J$ ) of 32.0. The isotherms in an  $rx$  plane through the jet centerline are shown and compared. Qualitatively they exhibit similar features, although the jet penetrated a little further in the case of the fine grid. Isotherms are also shown for two axial planes:  $x/D = 0.0$  and 2.0. The isotherms at  $x/D = 2.0$  show slightly higher temperatures ( $\sim 22^\circ K$ ) for the fine grid. Also, the cold region in the fine grid solution is located closer to the centerline, indicating greater penetration. However, overall the coarse grid solution is very similar to the fine grid solution.

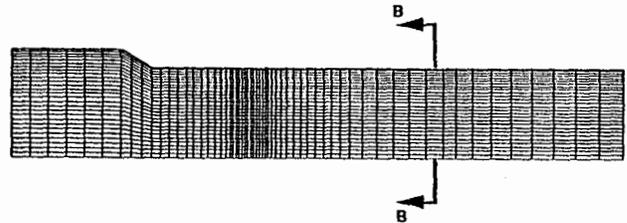
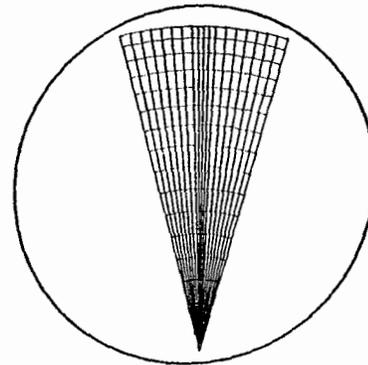
Based on this grid-independence study, it appears the coarse grid captures the overall physics of the problem, and can be used to qualitatively compare quick-mix designs.



GRID:  $x = 32$   
 $r = 16$   
 $\theta = 18$

9216 CELLS

Section A-A



GRID:  $x = 60$   
 $r = 30$   
 $\theta = 27$

52,650 CELLS

Section B-B

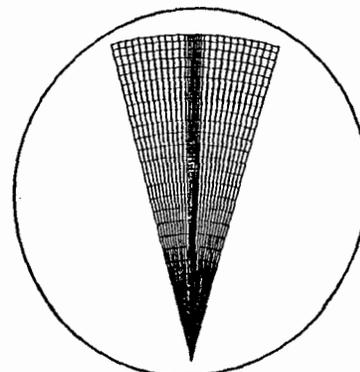


Fig. 5. Comparison of Coarse and Fine Grids

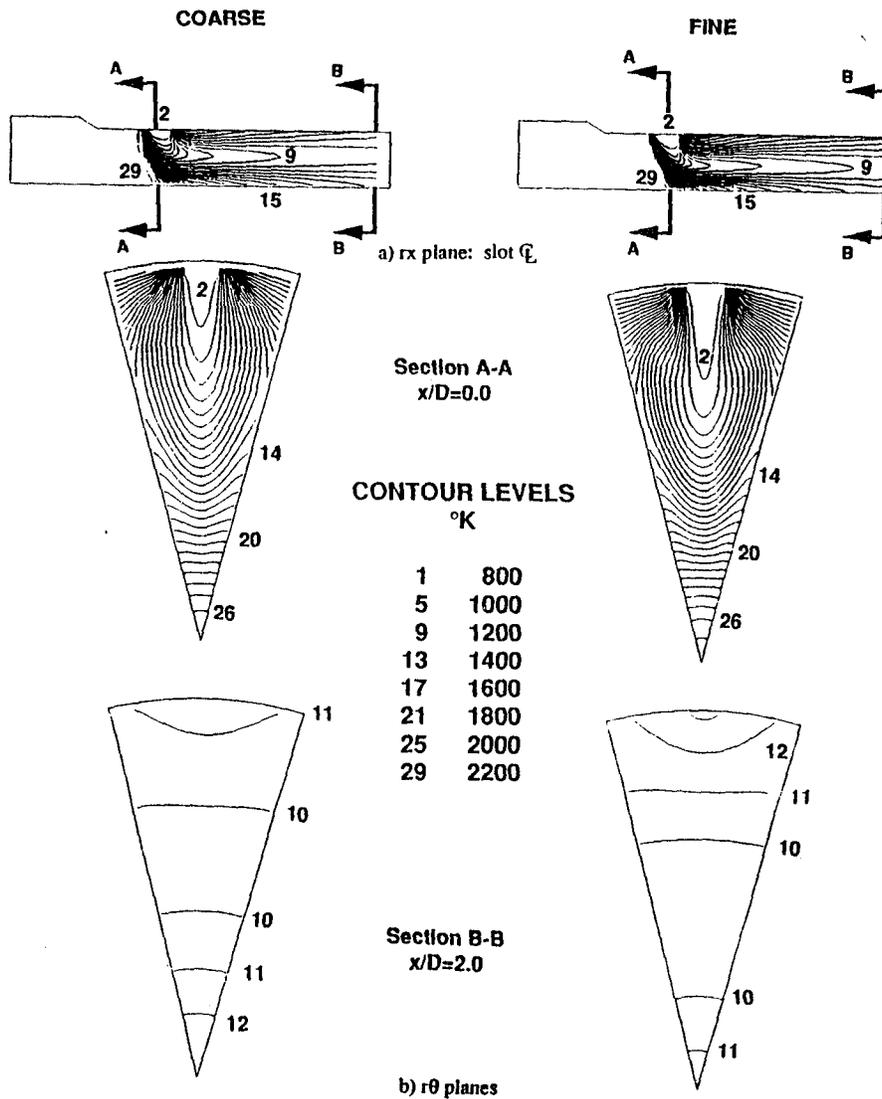


Fig. 6. Comparison of Isotherms for Coarse and Fine Grids:  $J=32$

### Convergence

The summations of all error residuals were reduced five orders of magnitude, and continuity was conserved in each axial plane. Typically, convergence required approximately 150 iterations as shown in the Figure 7. The relaxation on the velocity components ( $u$  and  $v$  only) was continuously varied during the run through a user specified input file. The repeated variation of relaxation allowed resolution of different scales of numerical error. This was found to speed up convergence by a factor of six compared to constant relaxation. Approximately 3 CPU hours were required on an Alliant FX/8 mini-supercomputer (operating on one computational element). Fine grid calculations took approximately 500 iterations and 40 CPU hours. For comparison, the ALLIANT computer speeds are  $\sim 20$  times slower than a CRAY X-MP.

### RESULTS

Parametric numerical tests were performed for jet-to-mainstream momentum flux ratios ( $J$ ), for both non-reacting and reacting gases. Parametric variation of slot aspect ratio (SAR) was also studied. Discussion of the findings are reported below.

### Variation of $J$ : Non-Reacting Flow

Five jet-to-mainstream momentum flux ratios were parametrically tested: 16, 32, 40, 48, and 64. All other flow conditions were held constant, including mass flow ratio (jet-to-mainstream) at 1.94. To maintain a constant mass flow, the slot size was changed for each  $J$ . The slot aspect ratio was held constant at four, and was always centered at the same location. The same number of grid cells were used in all cases. However, since the slot size was changing, the grid density had to be slightly altered for each case. This variation is thought to have a minimal effect on the results discussed below.

Computed temperature contour maps are presented at  $x/D$  of 1.0, as shown in Figure 8. The radial location of the lowest temperatures indicates the penetration location of the cold jet. As expected, increased jet penetration can be seen for larger values of  $J$ . Best mixing seems to occur when the jet penetrates to approximately mid-radius.  $J$  of 32 and  $J$  of 40 appear to be optimum mixers. For comparison, the optimum  $J$  is 45.6 using Eq. (1).

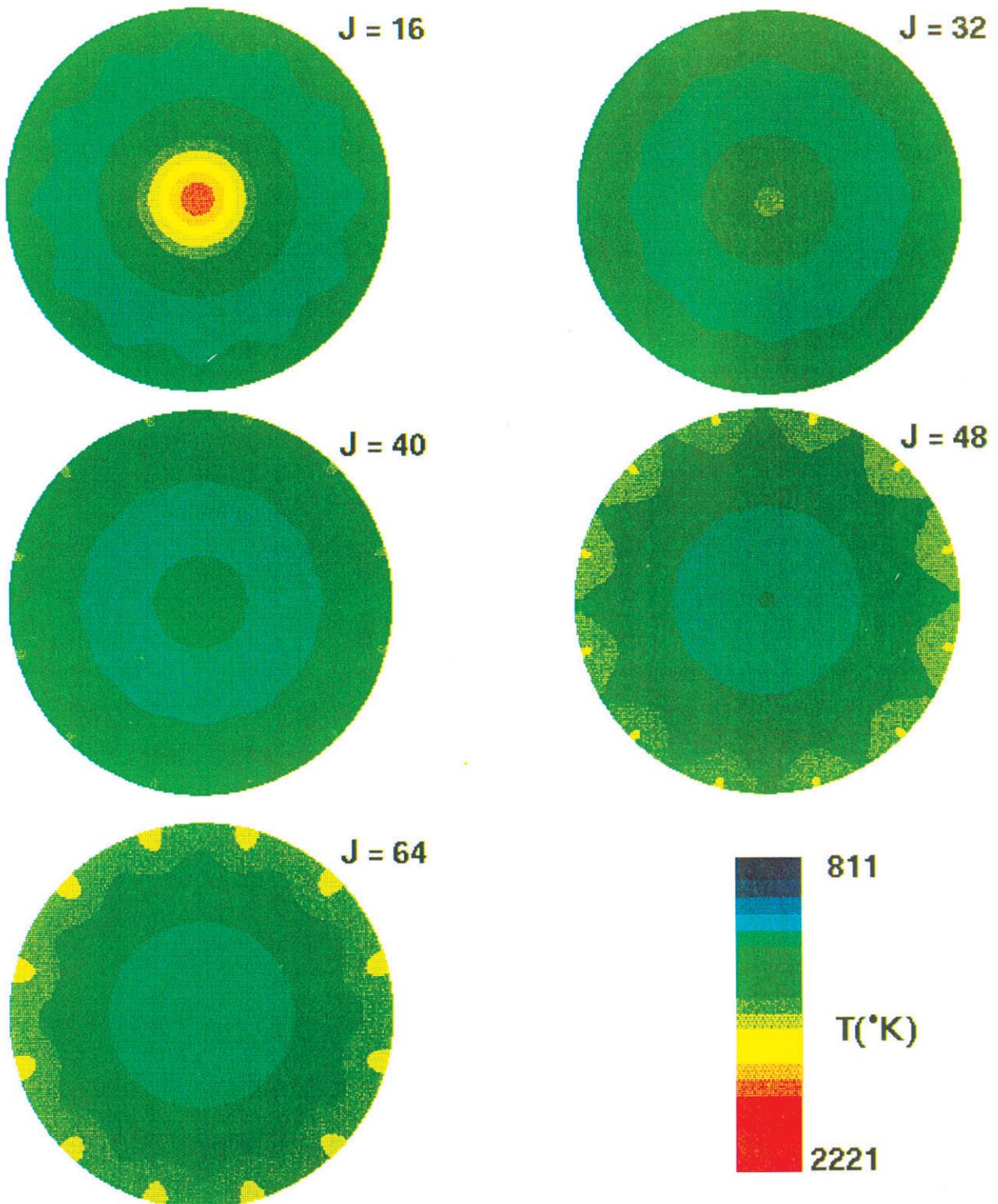


Fig 8. Temperature Contour Maps for Non-Reacting Conditions:  $x/D=1.0$

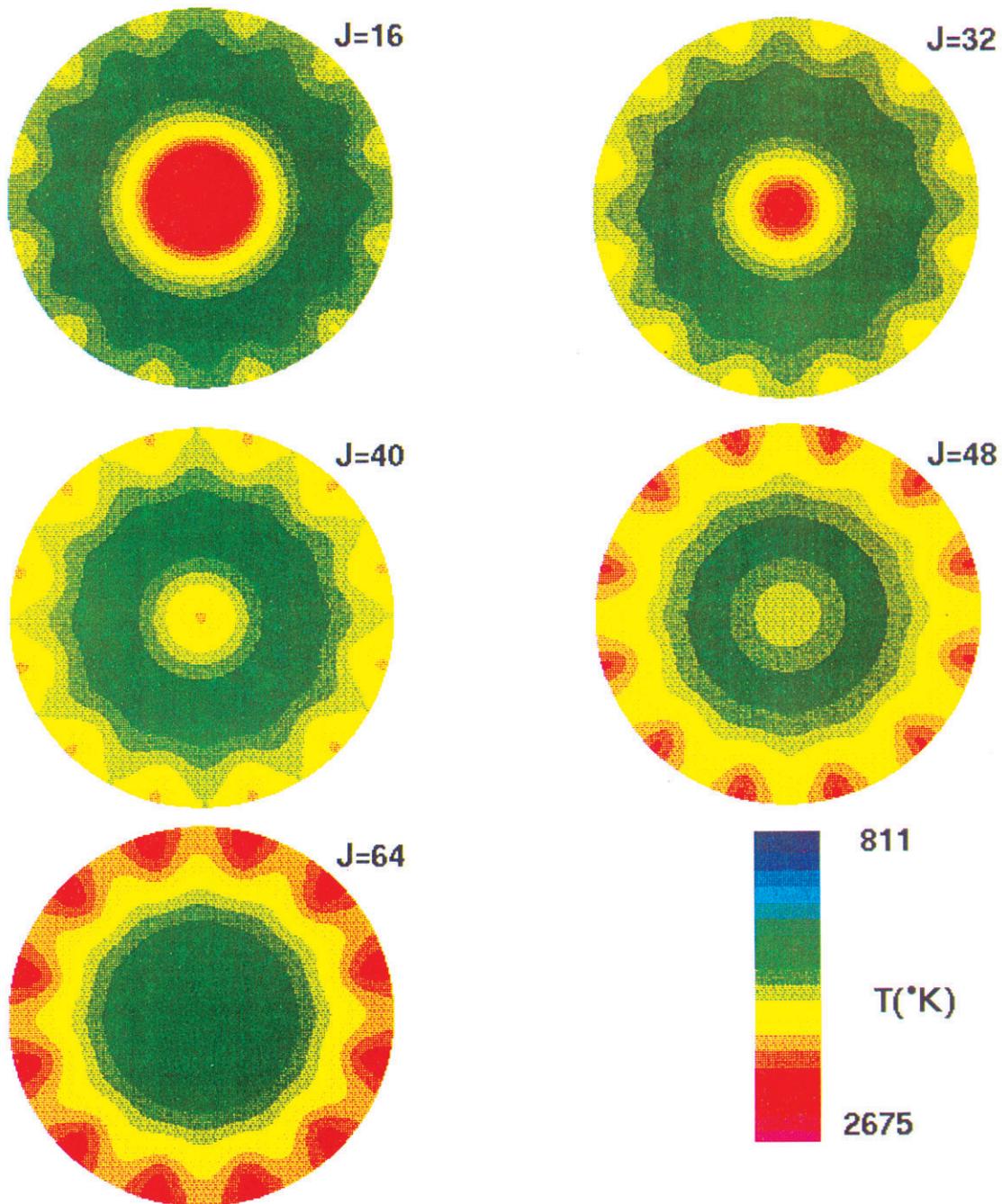


Fig 10. Temperature Contour Maps for Reacting Conditions:  $x/D=1.0$

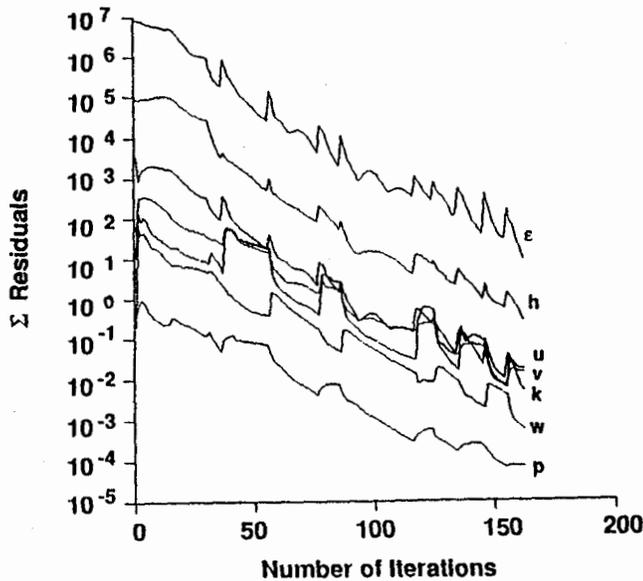


Fig. 7. Convergence History for a Typical Run

For a more quantitative comparison of mixing effectiveness, the mass-weighted standard deviation of temperature ( $\sigma_T$ ) was calculated for each case.  $\sigma_T$  was defined as:

$$\sigma_T = \frac{\sqrt{\frac{\sum_i m_i (T_i - T_{avg})^2}{\sum_i m_i}}}{T_{avg}} \quad (11)$$

In Figure 9,  $\sigma_T$  is presented versus  $J$ . It can be seen that  $J$  of 32 has the lowest  $\sigma_T$  at  $x/D = 2.0$ . Underpenetration is worse than overpenetration in terms of  $\sigma_T$ .

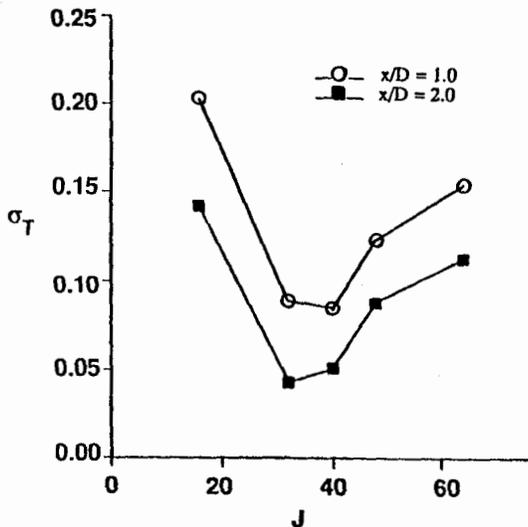


Fig. 9. Mixing Effectiveness: Non Reacting Flow

#### Variation of $J$ : Reacting Flow

The same cases were analyzed as discussed above, except chemical reaction was turned on. Due to reaction, the overall mass-averaged exit temperature increased from 1301°K for non-reacting flows to 1790°K for reacting flows. Figure 10 shows temperature contour maps for the reacting cases one diameter downstream of the jet center. From this figure, it appears that  $J = 40$  is the best mixer. This can be further elucidated by looking at the mixing effectiveness ( $\sigma_T$ ) shown in Figure 11. Figure 11 shows  $J = 40$  to be the best mixer at  $x/D = 1.0$  and 2.0.

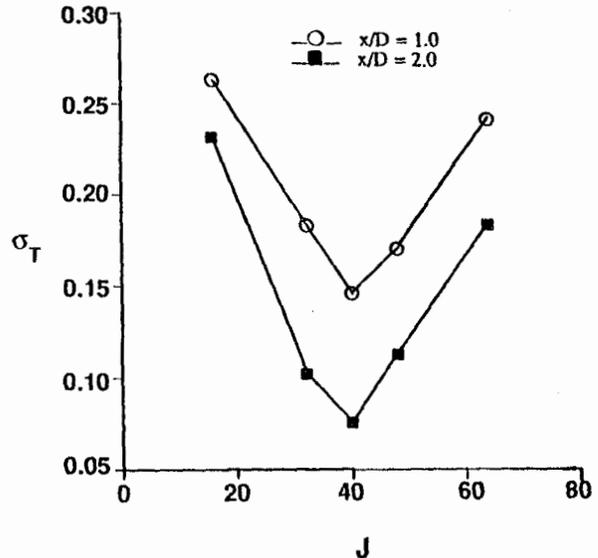


Fig. 11. Mixing Effectiveness: Reacting Flow

In addition to mixing effectiveness, another important criterion for evaluation of quick-mix sections is combustion efficiency. In particular, CO concentrations should be essentially eliminated from the combustor exit. The CO emission level in various axial planes downstream of the dilution jet is displayed in Figure 12. For all cases except  $J = 16$ , it can be seen that the CO species has been oxidized (to  $\text{CO}_2$ ) by  $x/D = 0.25$ . For  $J = 16$ , unreacted CO remains in the flowfield even at  $x/D = 2.0$ . This is due to jet underpenetration, thus allowing rich burn gases (containing CO) on the centerline to pass through the quick-mix section without contact with dilution air.

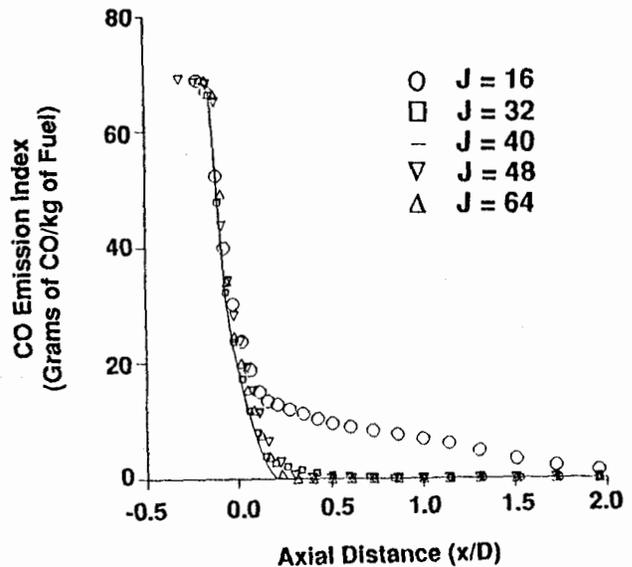


Fig. 12. Predicted CO Emissions

The  $\text{NO}_x$  results are presented in terms of Emission Index (EI) in Figure 13. For the optimum case ( $J_{opt}=40$ ), EI is 2.9 at  $x/D = 2.0$ . Significant increase in EI is predicted as  $J$  is increased or decreased from the optimum value. For  $J$  greater than  $J_{opt}$ , jet overpenetration causes jet backflow on the centerline, resulting in higher  $\text{NO}_x$  emissions. For  $J$  less than  $J_{opt}$ , underpenetration of the jet results in reaction (and high temperatures) on the combustor centerline.

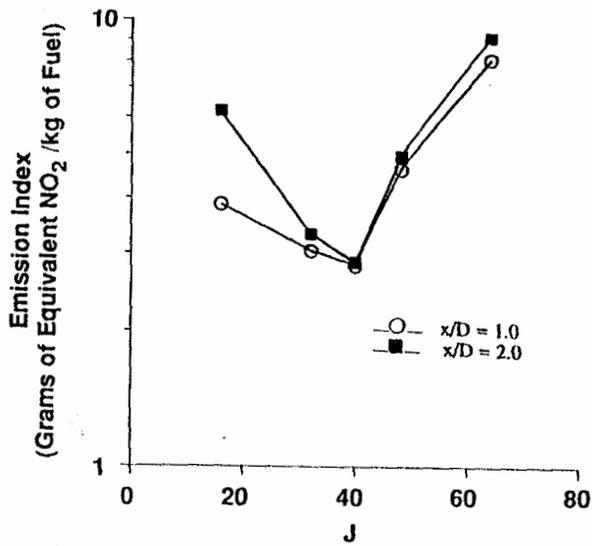


Fig. 13. Predicted NO<sub>x</sub> Emission Index

Figure 14 shows NO<sub>x</sub> concentrations convected out of each axial plane. Except for J = 16, all the cases show very little NO<sub>x</sub> formation downstream of x/D = 1.0. This indicates that high temperature zones are no longer existent. For the J = 16 case, NO<sub>x</sub> formation is increasing significantly downstream of x/D = 1.0, indicating high temperature and chemical reaction is still taking place.

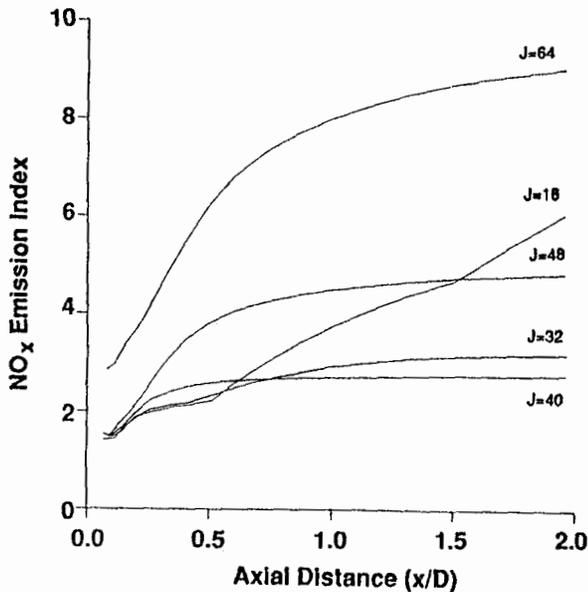


Fig. 14. History of NO<sub>x</sub> Formation in Mixer

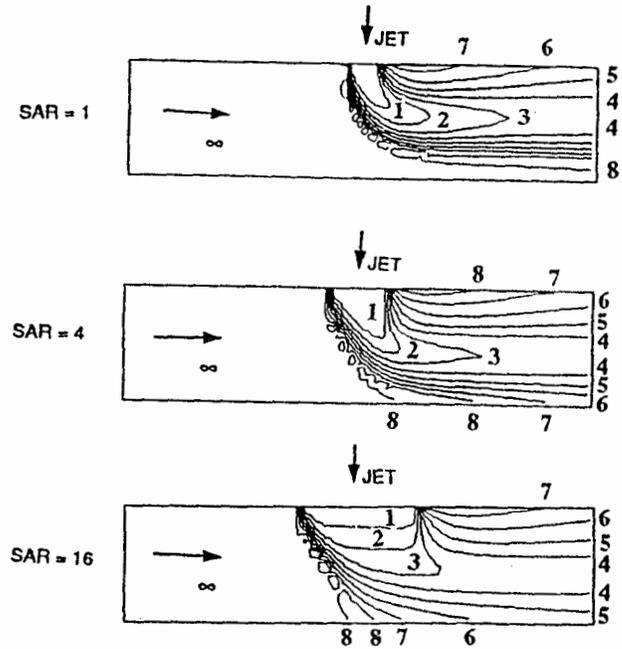
**Variation of Slot Aspect Ratio**

Three slot aspect ratios (SAR) were numerically analyzed: 1, 4 and 16. The long dimension was aligned in the mainstream flow direction. The numerical grid was slightly modified for each slot, and central differencing was employed for increased accuracy. The jet-to-mainstream momentum flux ratio and mass flow ratio was maintained constant at 32 and 1.94 respectively.

Isotherms in the rx plane through the jet centerline are shown in Figure 15. As SAR increased, jet penetration increased (as seen in Figure 15). This is due to reduced flow blockage as SAR is increased. Figure 16 shows the effect of SAR on NO<sub>x</sub> emissions. For SAR of 1, predicted NO<sub>x</sub>

levels are less than those for SAR of 4, but chemical reaction and NO<sub>x</sub> formation is still occurring at x/D of 1 due to jet underpenetration. This is evidenced by the steep slope of the NO<sub>x</sub> curve at x/D of 1. A similar effect of delayed NO<sub>x</sub> formation on the centerline caused by jet underpenetration was shown in Figure 14 for J of 16 and SAR of 4. Hence, the best SAR is 4, with jet overpenetration for SAR of 16 (and corresponding higher NO<sub>x</sub> levels).

These results suggest the importance of SAR on NO<sub>x</sub> emissions. As was shown earlier for J variation, the jet must penetrate to approximately mid-radius for optimum mixing, and hence lowest NO<sub>x</sub>.



**CONTOUR LEVELS**

Contour Level	Temperature (°K)
1	1000
2	1200
3	1400
4	1600
5	1800
6	2000
7	2200
8	2400

Fig. 15. Isotherms in rx Plane Through Jet Centerline: Effect of Slot Aspect Ratio (SAR)

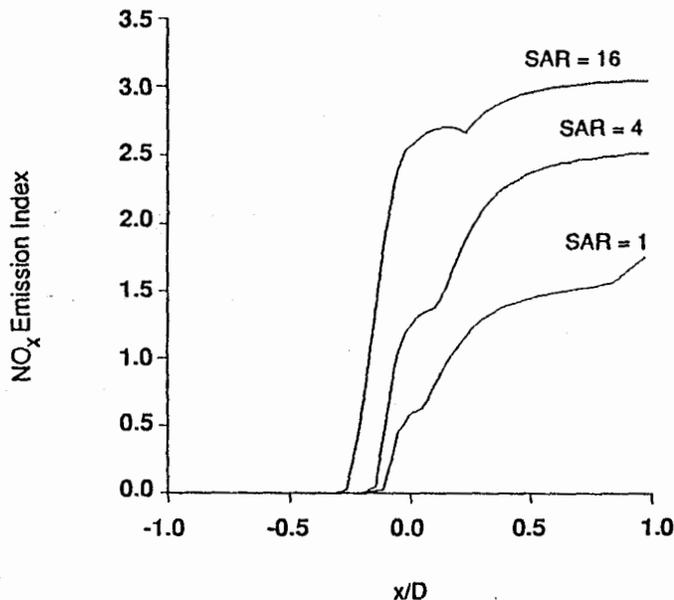


Fig. 16. Effect of Slot Aspect Ratio (SAR) on NO<sub>x</sub> Emissions

## CONCLUSIONS

The overall conclusions of this study were:

1. The viability of using 3-D CFD to model and screen quick-mix concepts of low emission combustors was successfully demonstrated.
2. A five-inch diameter quick-mix section compatible with the NASA LeRC Low Emission Combustor Program was numerically analyzed. The configuration consisted of twelve, radial-inflow slots uniformly distributed around the perimeter of the quick-mix section. Optimum mixing for non-reacting flow occurred for a jet-to-mainstream momentum flux ratio ( $J$ ) between 32 and 40. For reacting flow, the NO<sub>x</sub> emission index was shown to be highly sensitive to  $J$ , with the lowest value of 2.9 calculated for  $J$  of 40 (at  $x/D = 2.0$ ).
3. The numerical results suggest that slot aspect ratio has a pronounced effect on jet penetration and mixing effectiveness. Conventional correlations for optimum mixing effectiveness for holes may not be applicable for slots.

## ACKNOWLEDGEMENTS

The authors wish to thank NASA Lewis Research Center for funding this work under NASA Contract NAS3-25834. Our thanks also are extended to Drs. Laurence Keeton, H.Q. Yang, Andrzej Przekwas, Anantha Krishnan and Mr. Sami Habchi of CFD Research Corporation.

## REFERENCES

- Anderson, D., 1975, "Effect of Equivalence Ratios and Dwell Time on Exhaust Emissions from an Experimental Premixed Pre vaporizing Burner," NASA TMX-71592.
- Avva, R. K., Smith, C. E., and Singhal, A. K., 1990, "Comparative Study of High and Low Reynolds Number Versions of  $k$ -Models," AIAA-90-0246, 28th Aerospace Sciences Meeting.
- Chen, Y. S., and Kim, S. W., 1987, "Computation of Turbulent Flows Using an Extended  $k$ - $\epsilon$  Turbulence Closure Model," NASA CR-179204.

Chien, K. Y., 1985, "Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Turbulence Model," *AIAA Journal*, Vol. 23, No. 2.

Holdeman, J. D., Reynolds, R. and White, C., 1987, "A Numerical Study of the Effects of Curvature and Convergence on Dilution Jet Mixing," AIAA-87-1953, AIAA/SAE/ASME/ASEE 23rd Joint Propulsion Conference.

Kim, S. W., and Chen, C. P., 1988, "A Multiple Time Scale Turbulence Model Based on Variable Partitioning of Turbulent Kinetic Energy Spectrum," AIAA-88-1771.

Launder, B. E., and Spalding, D. B., 1974, "The Numerical Computation of Turbulent Flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, pp. 269-289.

Mosier, S. A., and Pierce, R. M., 1980, "Advanced Combustion Systems for Stationary Gas Turbine Engines," Vol. I, EPA Contract 68-02-2136.

Nguyen, H., Bittker, D., and Niedzwiecki, R., 1989, "Investigation of Low NO<sub>x</sub> Staged Combustor Concept in High Speed Civil Transport Engines," AIAA-89-2942, AIAA/ASME/SAE/ASEE 25th Joint Propulsion Conference.

Nickerson, G. R., Coats, D. E., Dang, A. L., Dunn, S. S., and Kehtarnavaz, H., 1989, "Two-Dimensional Kinetics TDK Nozzle Performance Computer Program; Vol II: Programming Manual," NAS8-36863.

Pierce, R. M., Smith, C. E., and Hinton, B. S., 1980, "Advanced Combustion Systems for Stationary Gas Turbine Engines," Vol. III, EPA Contract 68-02-2136.

Przekwas, A. J., Habchi, S. D., Yang, H. Q., Avva, R. K., Talpallikar, M. V., and Krishnan, A., 1990, "REFLEQS-3D: A Computer Program for Turbulent Flows With and Without Chemical Reaction, Volume 1: User's Manual," CFD Research Corp., Huntsville, AL, CFDRS Report GR-89-4.

Quan, V., Marble, F. E., and Kliegel, J. R., 1972, "Nitric Oxide Formation in Turbulent Diffusion Flames," 14th Symposium on Combustion, pp. 851-860.

Ratcliff, M. L., and Smith, C. E., 1989, "REFLEQS-2D: A Computer Program for Turbulent Flow with and without Chemical Reaction; Volume 2: Validation Manual," CFD Research Corp., Huntsville, AL, CFDRS Report GR-88-4.

Smith, C. E., Ratcliff, M. L., Przekwas, A. J., Habchi, S. D., and Singhal, A. K., 1988, "Modeling of Turbulent Combustion in Liquid Rocket Engine Components," NASA MSFC Contract NAS8-37619, SBIR Phase I Final Report, CFDRS Report 4045/1.

Smith, C. E., Ratcliff, M. L., Przekwas, A. J., and Habchi, S. D., 1989, "Validation of an Advanced Turbulent Combustion Code: REFLEQS," 7th SSME CFD Workshop, NASA MSFC.

Tacina, R. R., 1990, "Low NO<sub>x</sub> Potential Of Gas Turbine Engines," AIAA-90-0550, 28th Aerospace Sciences Meeting.

# REPORT DOCUMENTATION PAGE

*Form Approved*  
*OMB No. 0704-0188*

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

<b>1. AGENCY USE ONLY</b> ( <i>Leave blank</i> )	<b>2. REPORT DATE</b> June 1991	<b>3. REPORT TYPE AND DATES COVERED</b> Technical Memorandum	
<b>4. TITLE AND SUBTITLE</b>  CFD Analysis of Jet Mixing in Low NO <sub>x</sub> Flametube Combustors		<b>5. FUNDING NUMBERS</b>  WU-537-02-21-00	
<b>6. AUTHOR(S)</b>  M.V. Talpallikar, C.E. Smith, M.C. Lai, and J.D. Holdeman			
<b>7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)</b>  National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		<b>8. PERFORMING ORGANIZATION REPORT NUMBER</b>  E-6313	
<b>9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)</b>  National Aeronautics and Space Administration Washington, DC 20546-0001		<b>10. SPONSORING/MONITORING AGENCY REPORT NUMBER</b>  NASA TM-104466 ASME-91-GT-217	
<b>11. SUPPLEMENTARY NOTES</b>  Prepared for the 36th International Gas Turbine and Aeroengine Congress and Exposition sponsored by the American Society of Mechanical Engineers, Orlando, Florida, June 3-6, 1991. M.V. Talpallikar and C.E. Smith, CFD Research Corporation, Huntsville, Alabama 35805 (work funded by NASA Contract NAS3-25834). M.C. Lai, Wayne State University, Detroit, Michigan 48202. J.D. Holdeman, NASA Lewis Research Center, (216) 433-5846.			
<b>12a. DISTRIBUTION/AVAILABILITY STATEMENT</b>  Unclassified - Unlimited Subject Category: 07  Available electronically at <a href="http://gltrs.grc.nasa.gov/GLTRS">http://gltrs.grc.nasa.gov/GLTRS</a> This publication is available from the NASA Center for AeroSpace Information, (301) 621-0390.		<b>12b. DISTRIBUTION CODE</b>	
<b>13. ABSTRACT</b> ( <i>Maximum 200 words</i> )  The Rich-burn/Quick-mix/Lean-burn (RQL) combustor has been identified as a potential gas turbine combustor concept to reduce NO <sub>x</sub> emissions in High Speed Civil Transport (HSCT) aircraft. To demonstrate reduced NO <sub>x</sub> levels, cylindrical flametube versions of RQL combustors are being tested at NASA Lewis Research Center. A critical technology needed for the RQL combustor is a method of quickly mixing by-pass combustion air with rich-burn gases. In this study, jet mixing in a cylindrical quick-mix section was numerically analyzed. The quick-mix configuration was five inches in diameter and employed twelve radial-inflow slots. The numerical analyses were performed with an advanced, validated 3-D Computational Fluid Dynamics (CFD) code named REFLEQS. Parametric variation of jet-to-mainstream momentum flux ratio (J) and slot aspect ratio was investigated. Both non-reacting and reacting analyses were performed. Results showed mixing and NO <sub>x</sub> emissions to be highly sensitive to J and slot aspect ratio. Lowest NO <sub>x</sub> emissions occurred when the dilution jet penetrated to approximately mid-radius. The viability of using 3-D CFD analyses for optimizing jet mixing was demonstrated.			
<b>14. SUBJECT TERMS</b>  Dilution; Jet mixing flow; Gas turbines; Combustion chamber; Can; Emissions		<b>15. NUMBER OF PAGES</b> 14	
		<b>16. PRICE CODE</b> A03	
<b>17. SECURITY CLASSIFICATION OF REPORT</b> Unclassified	<b>18. SECURITY CLASSIFICATION OF THIS PAGE</b> Unclassified	<b>19. SECURITY CLASSIFICATION OF ABSTRACT</b> Unclassified	<b>20. LIMITATION OF ABSTRACT</b>