Conf-9103133--3

COMPUTER SIMULATION OF JET PENETRATION AND FLUID MIXING IN A CHANNEL FLOW WITH CROSS-STREAM JETS

ANL/CP--72679

DE91 010821

DISCLAIMER

S.A. Lottes and S.L. Chang

ARGONNE NATIONAL LABORATORY
Energy Systems Division
9700 South Cass Avenue
Argonne, Illinois 60439

Manuscript submitted to
The 7th International Conference on Advanced Science and Technology
March 30, 1991
Argonne National Laboratory, Argonne, Illinois

COMPUTER SIMULATION OF JET PENETRATION AND FLUID MIXING IN A CHANNEL FLOW WITH CROSS-STREAM JETS

S.A. Lottes and S.L. Chang
Argonne National Laboratory
9700 South Cass Avenue
Argonne, Illinois 60439

ABSTRACT

Multi-dimensional hydrodynamics computer codes are used to investigate jet penetration and fluid mixing patterns of main and jet flows in an MHD second stage combustor. The computer simulation is intended to enhance the understanding of flow and mixing patterns in the combustor, which in turn may improve downstream MHD channel performance. A two-dimensional code is used to study the effects of jet angle, jet velocity, and turbulence parameters on the fluid mixing and a three-dimensional code is used to examine the effects of the jet port configuration and inlet swirl on the fluid mixing. Both codes solve the conservation equations of mass, momentum, and energy, in conjunction with a turbulence model. The mixedness (degree of mixing) of jet and main fluids depends greatly on jet angle and jet velocity. Counter-flow injection of jet fluid and high jet velocity are required to have a satisfactory fluid mixing for MHD combustor applications.

Introduction

Much effort has been expended in the MHD community to explain and, thus, ultimately overcome the shortfall in MHD generator output observed in testing at the Component Development and Integration Facility [1-3]. As part of this effort, Argonne National Laboratory (ANL) has been using computer simulation to investigate combustion inefficiencies resulting from inadequate mixing in the second stage of the TRW combustor [4-6]. The computer simulation is intended to enhance understanding of the complex hydrodynamic and combustion phenomena in the combustor, to evaluate the effects of combustor operating parameters on fluid mixing and combustion performance, and to identify potential improvements in the design of the combustor. The work is being performed in two phases. The first phase focuses on the hydrodynamics and the second stage focuses on chemical reactions.

Two computer codes are used in the hydrodynamics modeling effort. A two-dimensional computer code (GEMCHIP) is used to study the effects of jet angle, jet velocity, and turbulence parameters on fluid mixing patterns. A three-dimensional hydrodynamics code (COMMIX) is used to assess the three-dimensional effects of fluid mixing, in particular, the jet port arrangement, and the inlet swirl. The hydrodynamics calculations performed by ANL are both qualitatively and quantitatively compared with TRW's cold flow experimental data [7-8].

Three-Dimensional Hydrodynamics Simulation

The COMMIX code solves the conservation equations of mass, momentum, energy, and a transport equation of turbulent kinetic energy to determine the velocities, pressure, density, and temperature of a single-phase flow in a staggered grid system [9]. An idealized combustor (length:height:width = 3.84:1:1) for three-dimensional simulation is shown in Figure 1a. The twelve solid dots indicate the jet port locations (at $x/D = 0.66$, jet angle = 90 degrees) and the four circles near the inlet represent openings for generating inlet swirl. The strength of the inlet swirl is represented by a swirl ratio defined as the ratio between swirl jet momentum and inlet axial momentum. In Figure 1b, a three-dimensional grid consisting of 41 by 21 by 13 nodes and its coordinate system are shown. The geometry of the combustor is summarized in Table I. Mixing patterns computed using the COMMIX code are compared for various jet port configurations, jet velocities, inlet swirls, and asymmetric inlet flows. Jet port configurations include a 12 vertical jets (12v) configuration, three configurations of 8 vertical jets (center-, side-, and mixed-8v), and three configurations of 8 vertical plus 4 horizontal jets (center-, side-, and mixed-8v4h). An illustrative set of these jet port configurations is shown in Figures 1a and 2.

Jet penetration, which depends on jet entry velocity, strongly affects fluid mixedness in the combustor. When the jet penetrates deeper, main and jet fluids mix better because of more effective convective mixing. For computed cases with flow symmetry and no inlet swirl, one vortex is found in each quadrant of the combustor for all 8v-jet and 8v4h-jet configurations except the center-8v4h-jet configuration. In most cases these vortices do not give these alternative configurations a mixing advantage over the 12v-jet configuration because the vortices primarily rotate the jet and main fluid layers around without stretching and folding jet fluid against main fluid to any significant degree. Except for the center-8v4h-jet case, the 12v-jet configuration appears to have better or equal fluid mixedness than the 8v-jet cases, as shown in Figure 3. Figure 3 compares the axial development of jet mixedness (σ_c) for various jet port configurations. The jet mixedness is defined as the standard deviation divided by the mean of the jet concentration in a cross-section. Thus, this measure of mixedness decreases and approaches zero as the two fluids become uniformly mixed. Asymmetric inlet flow does not change mixing patterns significantly but a strong inlet swirl can.

To validate the computer code, calculations are made to compare with TRW's experimental results. TRW's experimental data, taken at three axial locations, x/D = 1.8, 2.8, 3.8, is compared against results of a simulated COMMIX calculation in Figure 4. The contour numbers in Figure 4 represent the normalized jet concentration (jet concentration divided by the cross-sectional mean). Due to the physical equipment configuration, TRW was not able to measure velocity profiles at the inlet plane, and therefore, only mean values were know for the inlet conditions. Assuming flat profiles and selecting an inlet swirl ratio of 0.34, the computed fluid mixing patterns show reasonably good agreement with TRW's measurements. Asymmetries in the TRW measured concentration field due to asymmetries in the experimental inlet profiles are clearly visible in Figure 4. Because the inlet profiles of the experiment were not available, these asymmetries could not be readily modeled in the COMMIX calculations. As the flow moves downstream from X/D=1.8 to X/D=3.8, fluid structures in both the TRW experimental measurements and the COMMIX calculations are observed to rotate in the clockwise direction at approximately the same rate. In addition to the fluid mixing patterns, the computed and measured fluid mixedness also shows reasonably good agreement in Figure 5. The predicted fluid mixedness measure (σ_c) is generally higher than the measured value, and therefore the fluids in the experiment are more completely mixed than the computation indicates.

Two-Dimensional Hydrodynamics Simulation

The GEMCHIP code solves the conservation equations of mass, momentum, energy, and chemical species, as well as transport equations of turbulent kinetic energy and turbulent dissipation rate [5]. A two-dimensional grid system having 53 by 31 nodes in an idealized rectangular combustor is shown in Figure 6. Opposing jet flow openings on top and bottom walls are two-dimensional slots. The GEMCHIP code is used to predict mixing patterns for various jet angles, jet velocities, asymmetric inlet flows, and turbulence parameters. Simulated flow conditions for the parametric analysis are summarized in Table II.

The effect of jet angle, jet velocity, and turbulence model parameters on mixing patterns has been evaluated. The fluid mixedness depends on the jet angle because flow patterns, especially the size of the jet induced vortices, change according to the jet angle. In figure 7a, the best main/jet fluid mixing appears to occur at a jet angle in the interval 125 to 140 degrees. The fluid mixing also greatly depends on the jet velocity as shown in Figure 7b. Clearly, higher jet velocity, which causes deeper jet penetration, enhances fluid mixing.

The two-dimensional idealization which makes a slot out of the round jets of the three-dimensional configuration creates a significant jet Reynolds number miss match between the two and three-dimensional configurations when velocity and mass flow rate inlet conditions are matched. When inlet conditions are matched, distributing the round jets in a slot makes the slot very narrow in comparison to the round jet diameter. This difference makes the round jet Reynolds number a factor of five greater than the Reynolds number of the slot jet. One consequence of this Reynolds number miss match is that cross-stream momentum from the main flow penetrates the simulated jet slot much too quickly producing much shallower jet penetration and less mixing than in a three-dimensional configuration of the same jet velocity and mass flow rate. An additional problem with the slot idealization is that the slot jet creates a screen blocking the main flow and forcing it to accelerate through the chamber center as it turns the jets in the downstream direction. In the three-dimensional configuration a portion of the main flow flows around the sides of the jets reducing the screening effect of the jets and the size of vorticies which form behind the jets. Increasing the slot jet Reynolds number to achieve jet Reynolds number similarity produces chamber exit mixedness that is very close to experimental measurements, however the flow configuration in the chamber interior becomes markedly different due to the large screening effect of high momentum slot jets.

An alternative to adjusting jet Reynolds number in an effort to achieve a better overall similarity between three-dimensional configuration and two-dimensional idealization is to adjust turbulence parameters. In part to investigate this alternative and in part to investigate the computational sensitivity of the various empirical parameters a study of turbulence parameters was undertaken.

The GEMCHIP code employs a k–ε turbulence model to solve the transport equations of turbulent kinetic energy, k, and turbulent dissipation rate, ε, in conjunction with the continuity, momentum, species, and energy equations. The k–ε model has several empirical constants selected by matching computational results with experimental data. A sensitivity study to investigate the effects of turbulent empirical constants on the main/jet flow mixedness and flow configurations in the modeled MHD second stage combustor was performed and the following conclusions may be made: (1) the flow mixedness is sensitive to the change of the turbulent constants, (2) an increase of the viscosity constant increases the turbulent diffusivity and enhances the mixedness, (3) a decrease of the turbulent dissipation source constant increases the eddy size and enhances the mixedness, and (4) an increase of the turbulent dissipation sink constant increases the eddy size and enhances the mixedness.

The effect of the turbulent viscosity constant, C_μ, a scaling factor for determining turbulent viscosity, is illustrated in the jet mass concentration contours of Figure 8. Increasing the viscosity constant by an order of magnitude above the value suggested by Launder and Spalding (0.09) yields a very large increase in the mixing as shown in the compact concentration contours of Figure 8a. Reducing the constant to 0.05 significantly reduces the mixing as shown in Figure 8c.

Selection of an optimum combination of jet Reynolds number and turbulence parameter values requires more experimental data than is currently available. Results of sensitivity studies indicate that a reasonably good adjustment could be made if sufficient data were available, and comparisons with the available experimental data confirm the computed trends in the dependency of mixedness on jet injection angle and jet velocity.

Summary

Multi-dimensional hydrodynamics computer codes are used to investigate jet penetration and fluid mixing patterns of main and jet flows in an MHD second stage combustor. The two-dimensional GEMCHIP code is used to study the effects of jet angle, jet velocity, and turbulence parameters on the fluid mixing and the three-dimensional COMMIX code is used to examine the effects of the jet port configuration and inlet swirl on the fluid mixing. The computer codes are validated by comparing against experimental data. Simulation results show a number of mixing patterns for various operating conditions. Three-dimensional simulations show that 12 vertical jet and 8 centered vertical with 4 horizontal jet configurations produce the best mixing of the configurations tested. The two-dimensional simulations

show that fluid mixedness depends greatly on jet angle and jet velocity. Counter-flow injection of jet fluid and high jet velocity are required to have a satisfactory fluid mixing for MHD combustor applications.

ACKNOWLEDGEMENTS

This work is supported by U.S. Department of Energy, Assistant Secretary for Fossil Energy, under Contract W-31-109-ENG-38. The assistance by William Rimkus and Paul Lissac of ANL and valuable comments by Andrew Grove of TRW Inc. are greatly appreciated.

REFERENCES

[1] Listvinsky, G., J. Alpay, L. Hil, K. King, D. Paul, D. Vanevenhoven, K. Natesan and D.Y. Wang "Development of a Prototypical MHD Coal Combustor," Proc. of 24th Intersociety Energy Conversion Engineering Conference, Washington, D.C., 2:965-970 (August 6-11, 1989).

[2] McClaine, A., J. Pinsley, and B. Pote, "Experimental Investigation on the Effects of the TRW Two-Stage Coal Combustor on the Performance of the AVCO Mk VI MHD Generator," Proc. of 24th Intersociety Energy Conversion Engineering Conference, Washington, D.C., 2:971-977 (August 6-11, 1989).

[3] Burkhart, T., G. Funk, R. Glovan, A. Hart, A. Herbst, J. Joyce, Y.M. Lee, S. Lundberg, and I. Stepan, "Coal-Fired MHD Topping Cycle Hardware and Test Progress at the Component Development and Integration Facility," Proc. of 23rd Intersociety Energy Conversion Engineering Conference, Denver, CO, 4:445-453, (July 31-August 5, 1988).

[4] Lottes, S.A., S.L. Chang, and G.F. Berry, "Effects of Jet Ports Arrangement on Three-Dimensional Non-Reacting Jet-Gas Mixing in an MHD Second Stage Combustor," Proceedings of 25th Intersociety Energy Conversion Engineering Conference, Reno, Nevada, 2:468-473 (August 12-18, 1990).

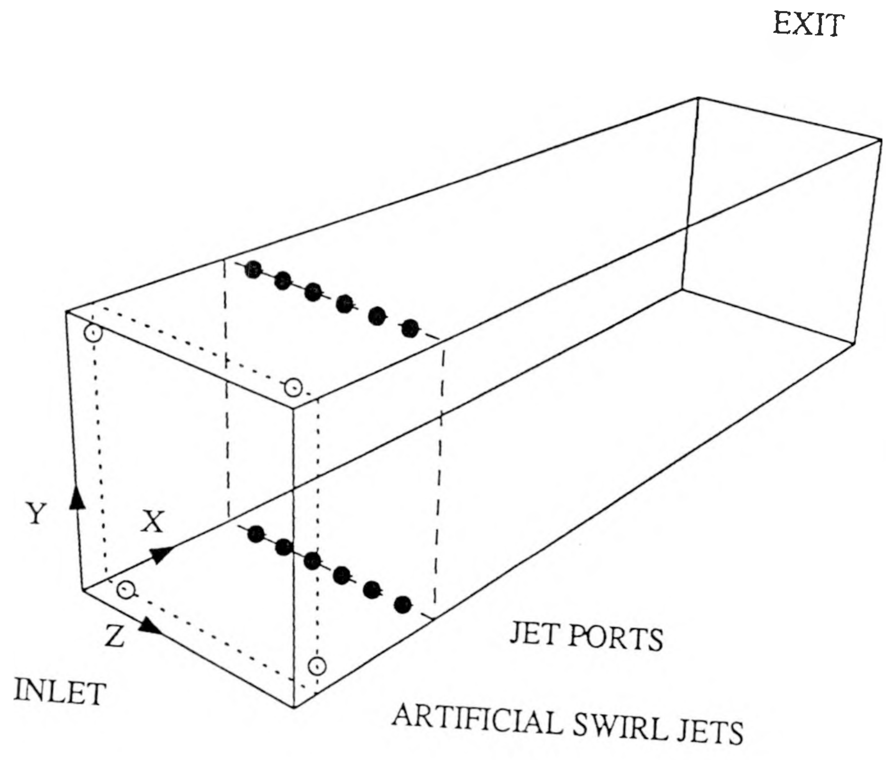
[5] Chang, S.L., S.A. Lottes, and G.F. Berry, "Two-Dimensional Non-Reacting Jet-Gas Mixing in an MHD Second Stage Combustor," Proceedings for 28th Symposium of Engineering Aspects of Magnetohydrodynamics, Chicago, IL, pp. V.3.1-V.3.10 (June 26-28, 1990).

[6] Chang, S.L., S.A. Lottes, and G.F. Berry, "Three-Dimensional Computer Simulation of Non-reacting Jet Gas Flow Mixing in an MHD Second Stage Combustor," Proceedings for 28th Symposium of Engineering Aspects of Magnetohydrodynamics, Chicago, IL, pp. IV.24.1-IV.24.9 (June 26-28, 1990).

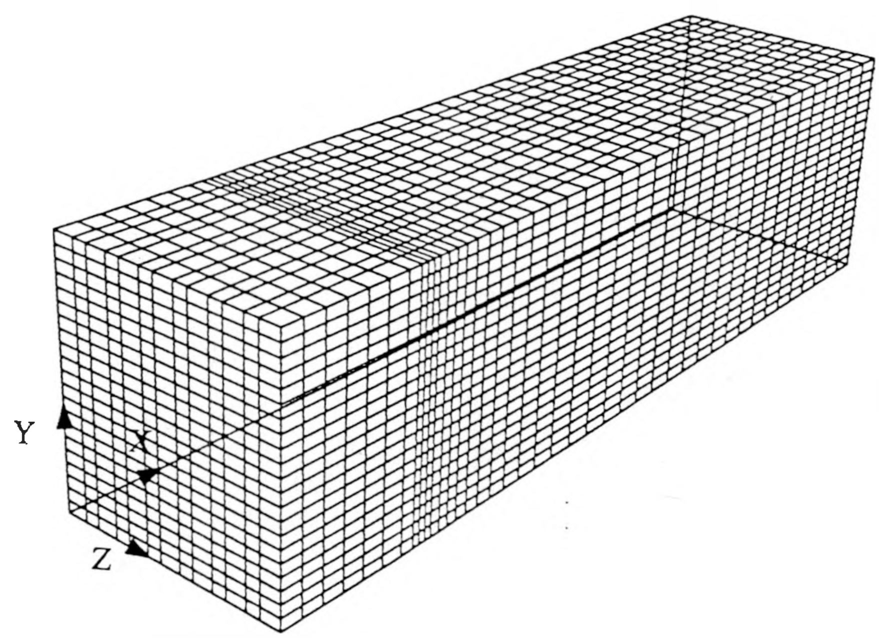
[7] Grove, A., "Cold Flow Mixing Study of 50 MWt MHD Coal-Fired Combustor Second Stage," 27th Symposium on Engineering Aspects of Magnetohydrodynamics, Reno, NV (1989).

[8] Wilson, W.W., D.V. Srikantaiah, and A. Grove, "Cold Flow Mixing Study of an MHD Combustor Model Using Laser Velocimeter and Concentration Measurements," 28th Symposium on Engineering Aspects of Magnetohydrodynamics, Chicago, IL (1990).

[9] Domanus, H.M., R.C. Schmitt, W.T. Sha, and V.L. Shah, "COMMIX-1A: A Three-Dimensional Transient Single-Phase Computer Program for Thermal Hydraulic Analysis of Single and Multicomponents Systems," NUREG/CR-2896, ANL-82-25 (1983).



(a) Geometry and coordinates



(b) Three-dimensional grid (41 x 21 x 13 nodes)

Figure 1 Combustor geometry and grid system

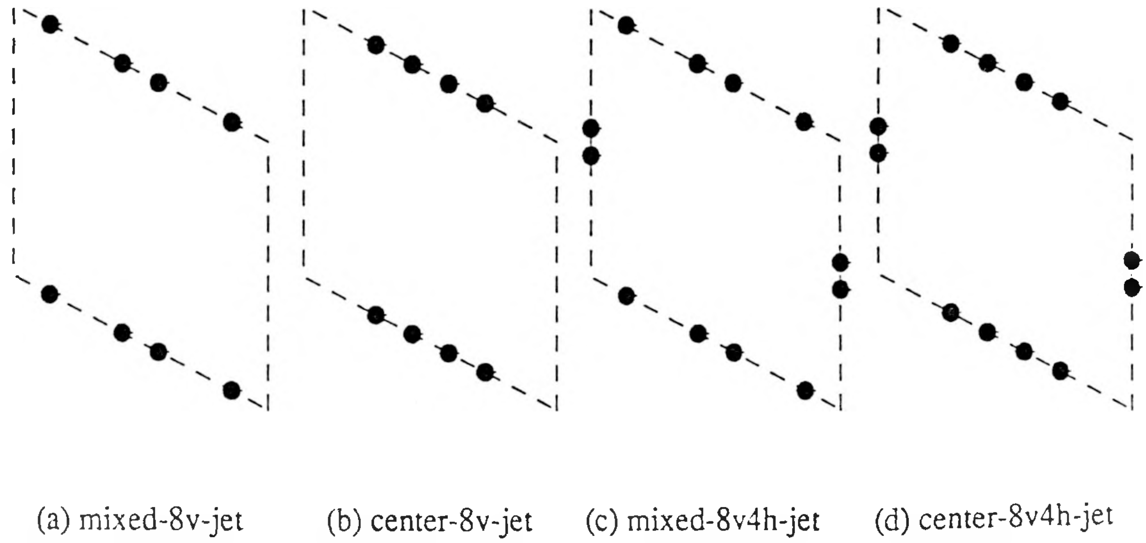


Figure 2 Various jet port configurations

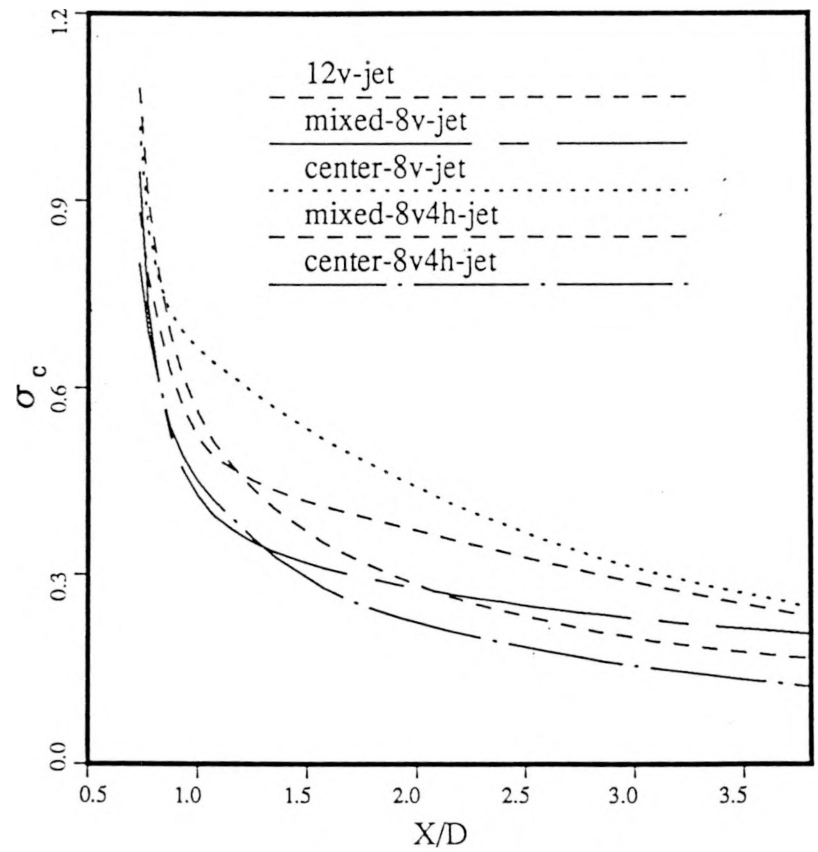


Figure 3 Comparison of fluid mixedness for various jet port configurations

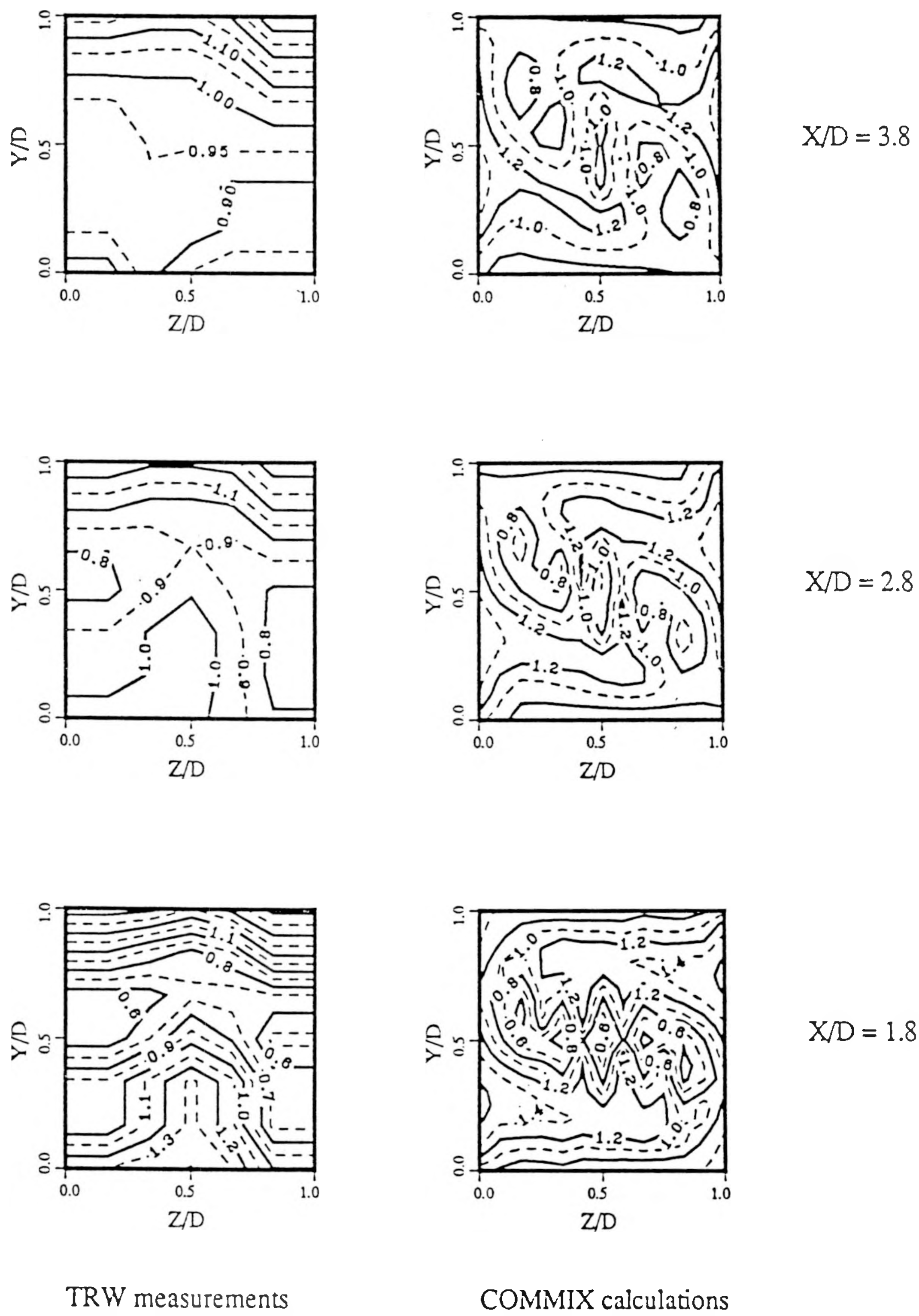


Figure 4 Comparison of fluid mixing patterns

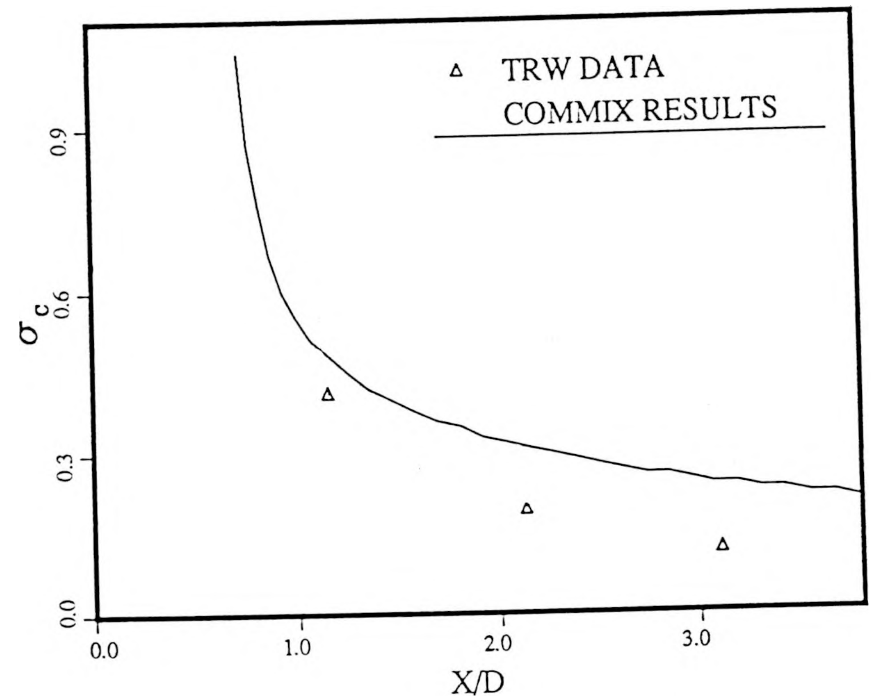


Figure 5 Comparison of fluid mixedness between 3-D results and TRW data

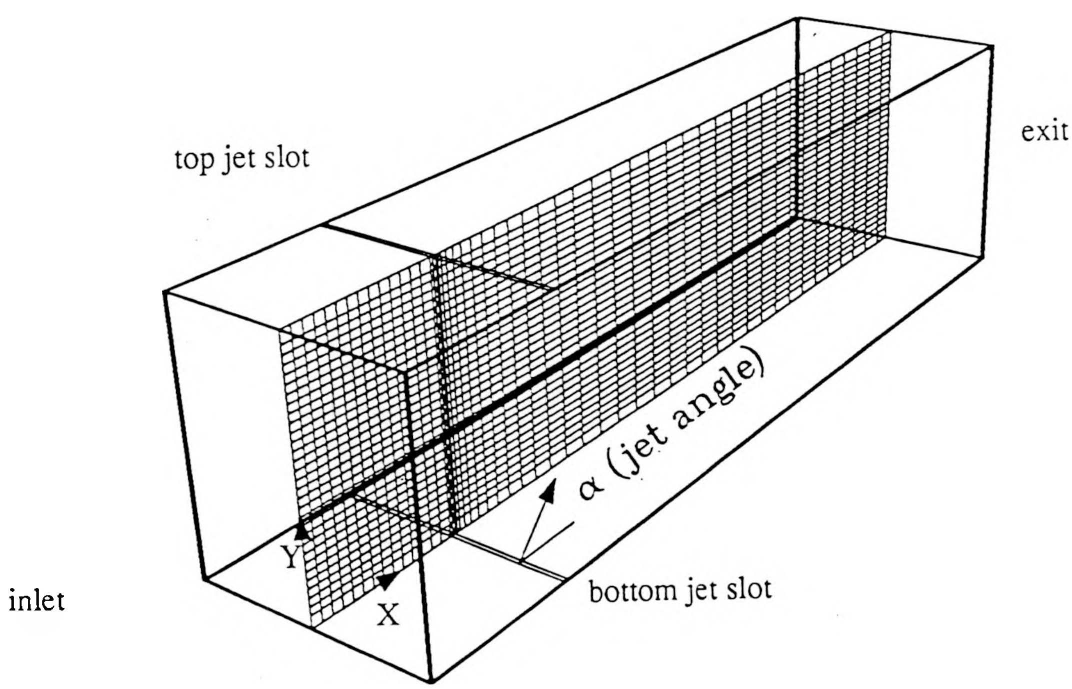
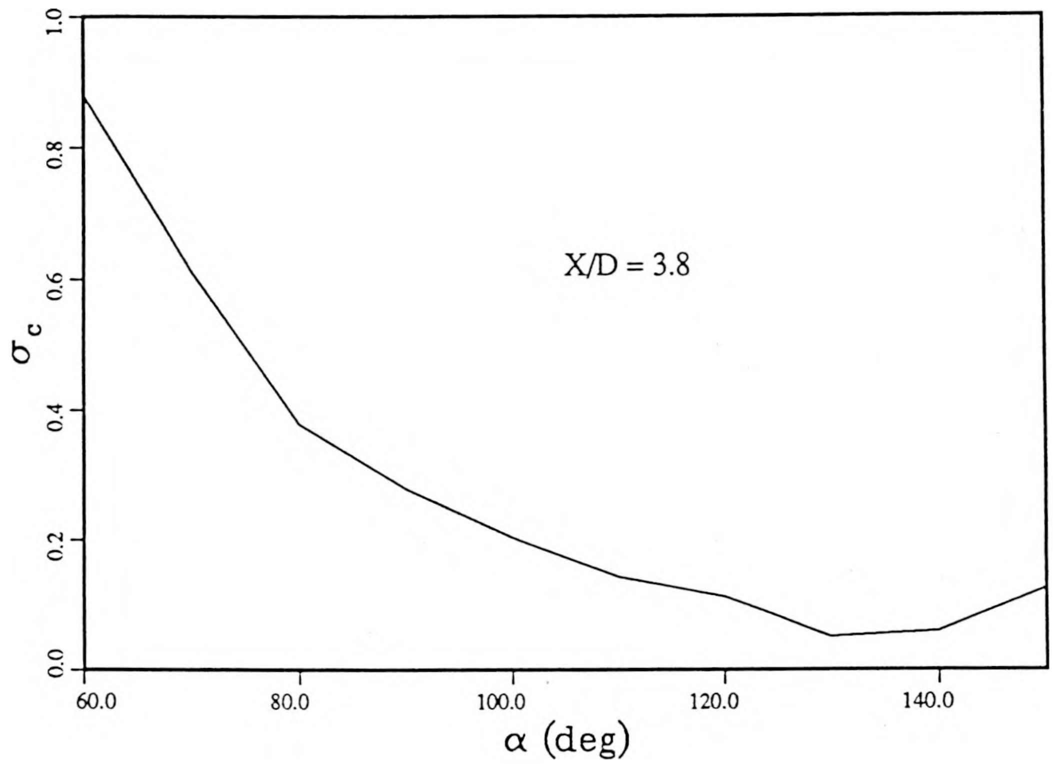
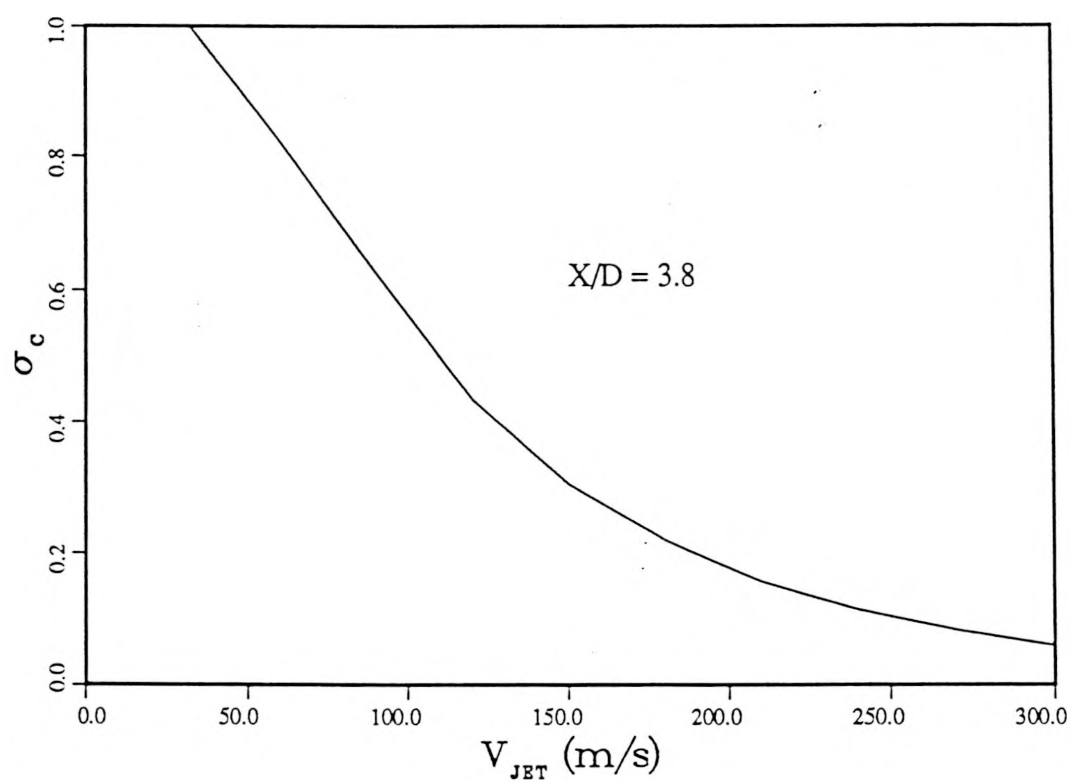


Figure 6 Two-dimensional computational grid

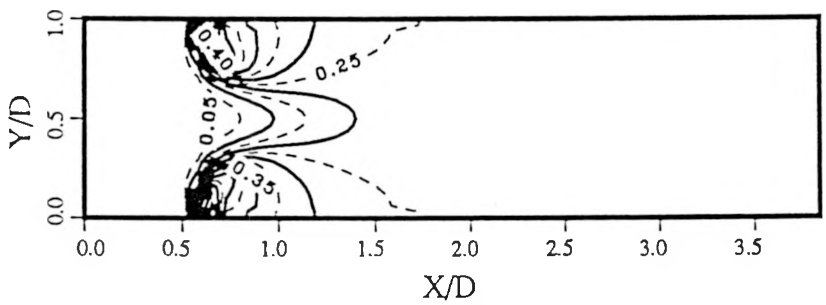


(a)

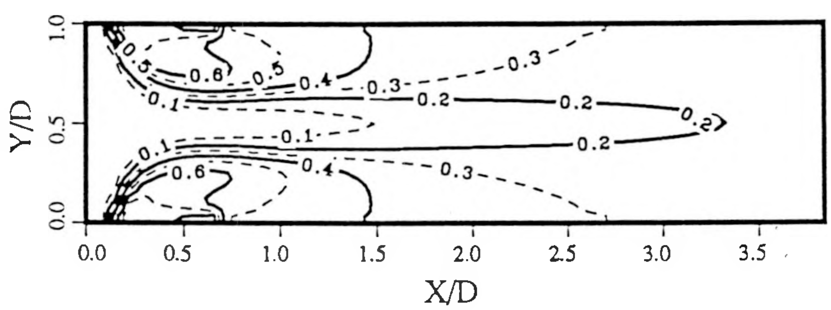


(b)

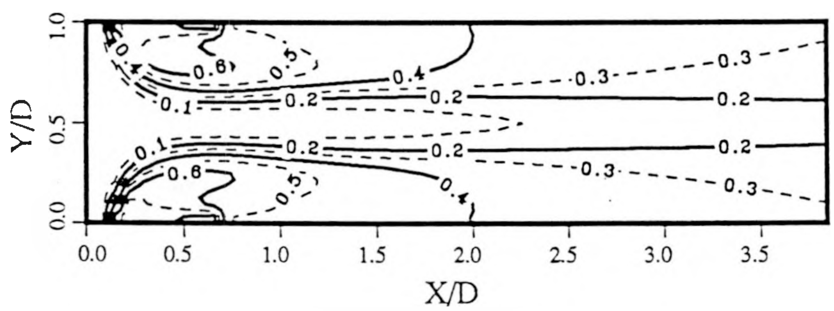
Figure 7 Dependence of exit jet mixedness on (a) jet angle and (b) velocity



(a) $C_{\mu} = 0.9$



(b) $C_{\mu} = 0.09$



(c) $C_{\mu} = 0.05$

Figure 8 Effect of turbulent diffusivity on fluid mixing