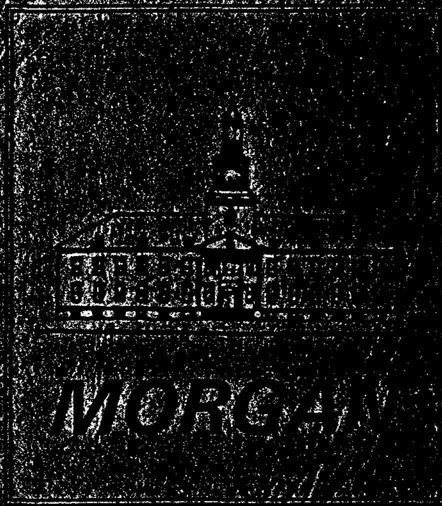


MORGAN STATE UNIVERSITY
School of Engineering
Cold Spring Lane
Baltimore, MD

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DOE/MT/93006--T4

Technical Progress Report No.9

Investigation of Heat Transfer and Combustion in the Advanced Fluidized Bed Combustor (FBC)

to

U.S. Department of Energy
Pittsburgh Energy Technology Center
P.O. Box 10940, MS 921-118
Pittsburgh, PA 15236-0940

RECEIVED
USDOE/PETC
96 JAN 31 AM 10:16
ACQUISITION & ASSISTANCE DIV.

for

Project No: DE-FG22-93MT93006

by

Dr. Seong W. Lee, Principal Investigator

Morgan State University
School of Engineering
Baltimore, MD 21239
(phone) 410-319-3137

January 1996

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

SUMMARY

This technical report summarizes the research performed and progress achieved during the period of October 1, 1995 to December 31, 1995. The measurements of gas flow in the advanced FBC test chamber (10" I.D.) was continued to better understand and utilize the fluid dynamics of gas and particle flows in the advanced FBC.

Measurements showed that the gas flow field in the test chamber is characterized by strongly swirling flow in tangential direction and developing flow in axial and radial directions. In addition, multiple secondary air injection caused significant effects on gas flow in the freeboard of the test chamber.

Numerical simulation of typical gas flow patterns in the freeboard was conducted using a computational fluid dynamics (CFD) code, FLUENT. The axial velocities resulting from theoretical prediction were smaller than the tested results. However, the predicted radial velocities at the exit zone of the test chamber were greater than that of the tested results. The calculated results showed the non-isotropic structure with vigorous fluctuating in axial and radial directions. Generally speaking, the predictions of the theoretical calculation agreed with the experimental results.

The measurements of gas and particle flows will be continued under different test conditions. In addition, the numerical simulation on gas and particle flows will be continued, which will be compared with the experimental results.

TABLE OF CONTENTS

PAGE

SUMMARY...ii

SECTION

1. Measurements of Gas Flow Field..............................1
   1.1 Test Conditions...1
   1.2 Test Results and Discussions............................1
2. Numerical Simulations and Discussions.......................7
3. Research Continuation......................................15

References..16

SECTION 1

Measurements of Gas Flow Field

Measurements of gas flow were continued in the bench-scale advanced FBC test chamber (10" I.D.) [1] to better understand how gas recirculating flow is affected by swirling flow in the freeboard of the test chamber.

1.1 Test Conditions

The tests were carried out under a constant primary gas velocity of 2.55 ft/s. All of eight secondary air injecting nozzles were fully opened while the tests conducted.

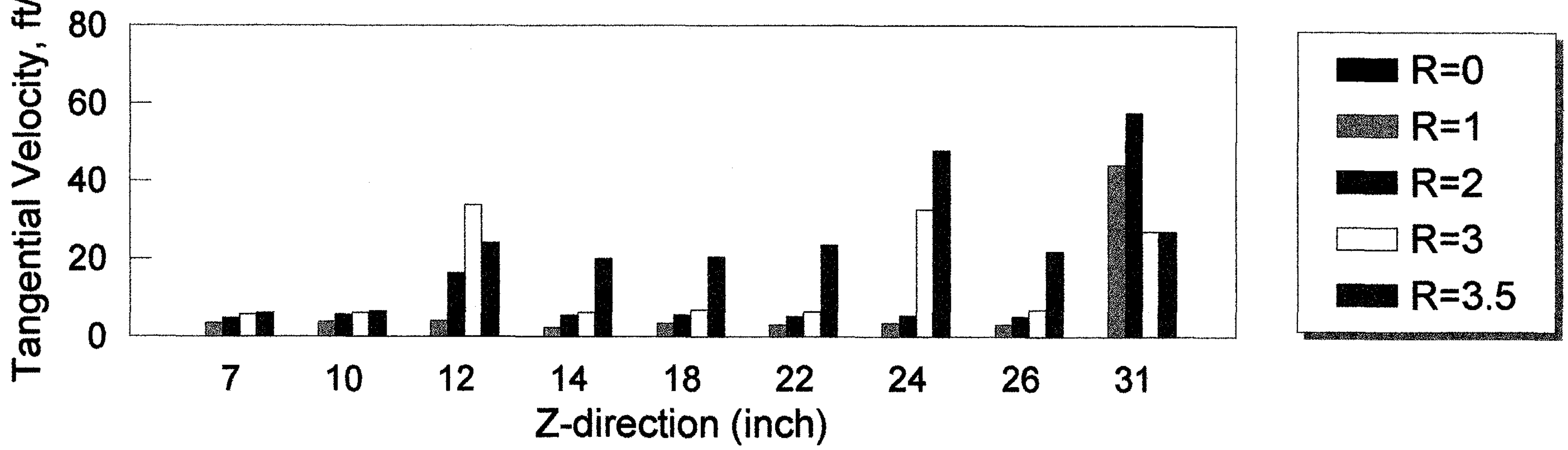
1.2 Test Results and Discussions

The flow characteristics profiles with three direction components, vertical velocity (Z-direction), radial velocity (R-direction), and tangential velocity (Q-direction) were measured and discussed.

Figures 1.1 through 1.4 show the effects of the secondary air flow rate at both the upper and lower sections on the gas flow pattern in the test chamber. In this case, with all secondary air nozzles open, the flow pattern was very stable. Tangential velocities could be predicted as the function of both R-direction and Z-direcion. In the Z-direction, larger tangential velocities were found around the nozzles (12-inch, 24-inch) and exit region of top (31-inch) as shown in Figure 1.1. In the R-direction, the larger tangential veocities occurred at the near the wall region. The tangential velocity is almost one order of magnitude larger

Figure 1.1 Tangential Velocity vs. Z-location

Test case No.3: Upper-Nozzle open, Lower-Nozzle open



than the vertical/radial velocities. These results indicated that the strong swirling feature of gas flow is expected in the test chamber.

Figure 1.2 shows the changes of radial velocities versus Z-location. The changes of the radial velocities are relatively stable, since the stream of gas flows into the center of the chamber from the wall region within the lower Z-direction (less than 14-inch).

Due to multi-stage injection of secondary air, the vertical and radial velocities changed rapidly and frequently as shown in Figures 1.2 and 1.3. These results are desirable for gas-gas mixing and gas-particle mixing in the combuster [2]. A few small eddies formed in the radial direction. As shown in Figure 1.3, larger vertical velocities were obtained at 31-inch of Z-direction, which is the effect of mass balance [3] at the top exit of test chamber. Measurements showed that the vertical velocity gradually decreased from the center of chamber to the wall region.

The gas recirculating flow existed below the lower level secondary air injection nozzles. This result is expected to contribute the solid particle recirculating in the freeboard and to enhance gas-particle mixing and particle resident time [4]. A relative lower static pressure formed at the top exit region, and the pressure fluctuation was small in the lower section as shown in Figure 1.4. The test conditions for this case have been taken as the input data for the numerical simulation.

Figure 1.2 Radial Velocity vs. Z-location

Test case No.3: Upper-Nozzle open, Lower-Nozzle open

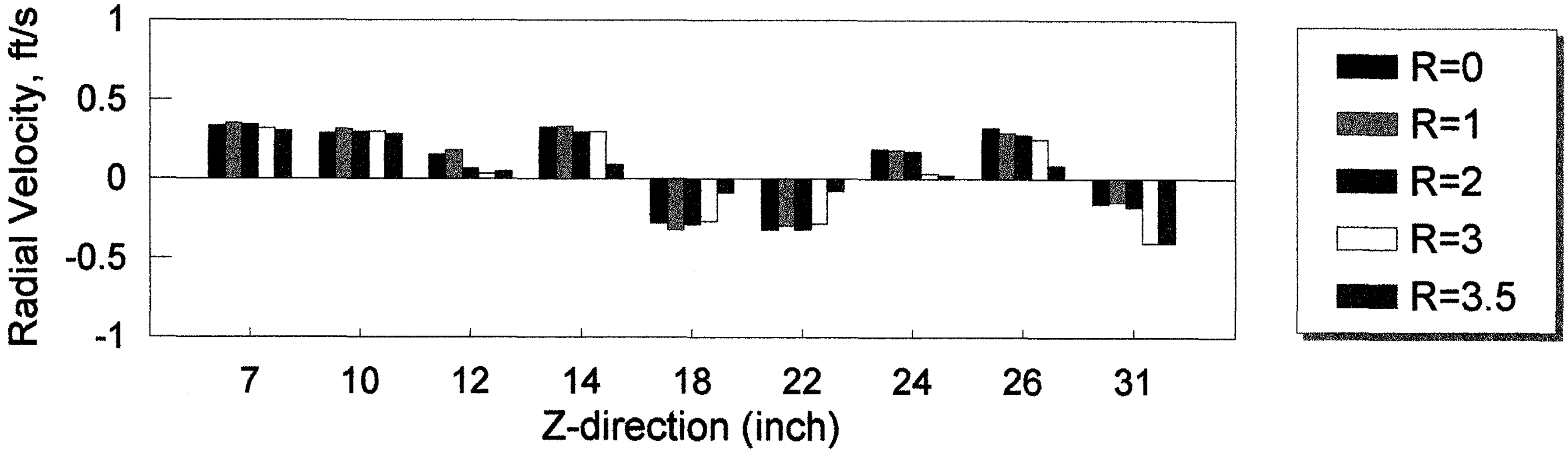
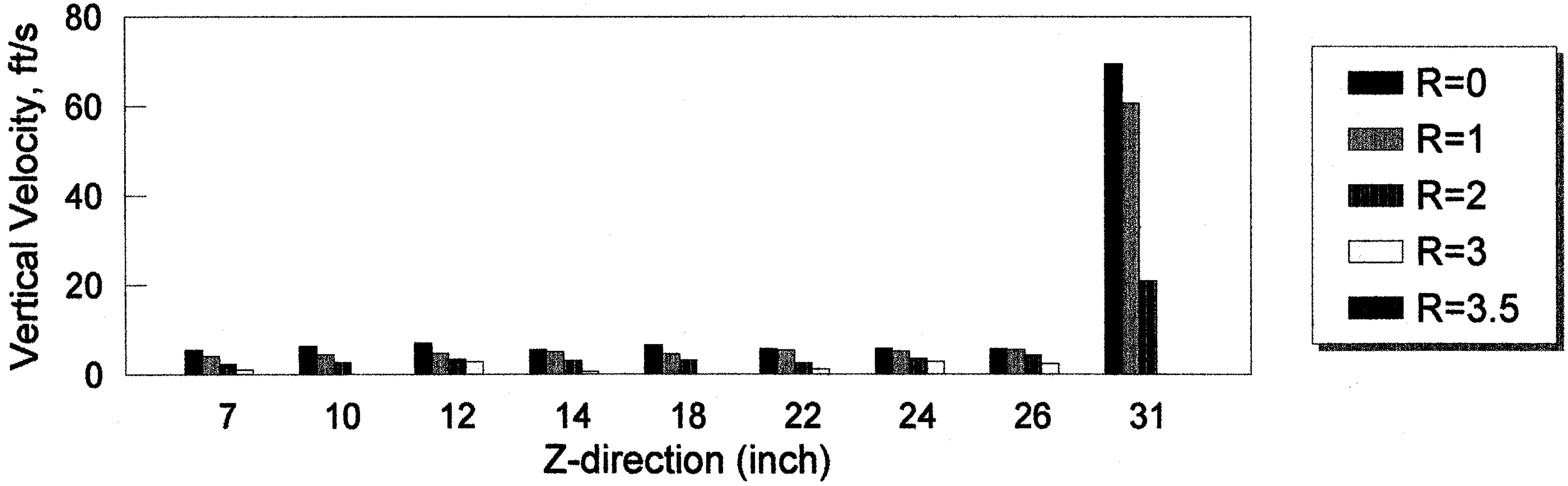


Figure 1.3 Vertical Velocity vs. Z-location

Test case No.3: Upper-Nozzle open, Lower-Nozzle open



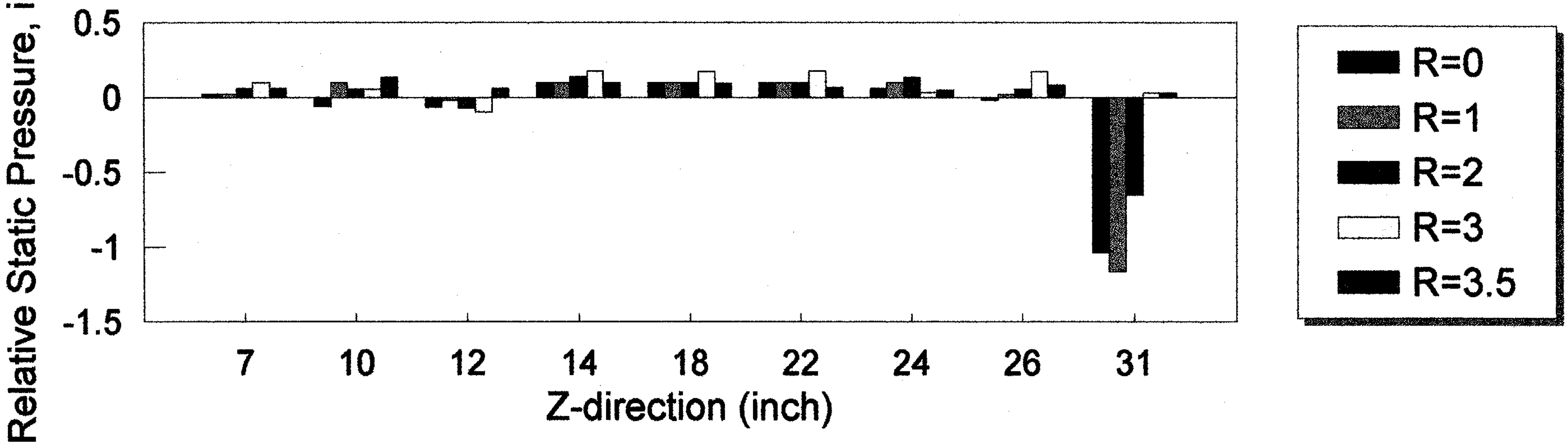


Figure 1.4 Static Pressure Change in Z-direction

Test case No.3: Upper-Nozzle open, Lower-Nozzle open

SECTION 2

Numerical Simulations and Discussions

The purpose of this simulation is to simulate the profiles of velocity with 3 direction components and relative static pressure in the combustion chamber under swirling flow condition. This was axi-symmetric, 3-dimensional turbulent flow problem involving a low speed primary air input through the gas distributor plate at the combustor bottom and a high speed secondary air input from eight secondary air nozzles. The secondary air nozzles were arranged in two axial-symmetrical levels: 12 inches and 24 inches high. In order to reduce the computation times, the cyclic method [5] was used to consider a quarter of the combustor cylinder as shown in Figure 2.

The calculation grid was a set of orthogonal lines arranged in the cylindrical coordinates, $\Theta(I)$, $R(J)$, $Z(k)$, directions superimposed on the solution domain. The lines in the three directions were uniformly spaced. The boundaries of the domain lay midway between grid lines. The computational cells and boundaries for the combustor calculation domain are also shown in Figure 2. The boundary and initial conditions for the computation were set up as follows: the walls were plates 4-3-5-6 and 7-8-4-6, the symmetric line was line 1-2, the cyclic plates were 1-2-3-4 and 1-2-5-6. The No.1 inlet for the primary air input was plate 2-3-5, the No.2 inlet for lower level secondary air input was point 9, and the No.3 inlet for upper level secondary air input was 10. The outlet plate was 1-8-7.

The input condition for inlet No.1 was vertical velocity is

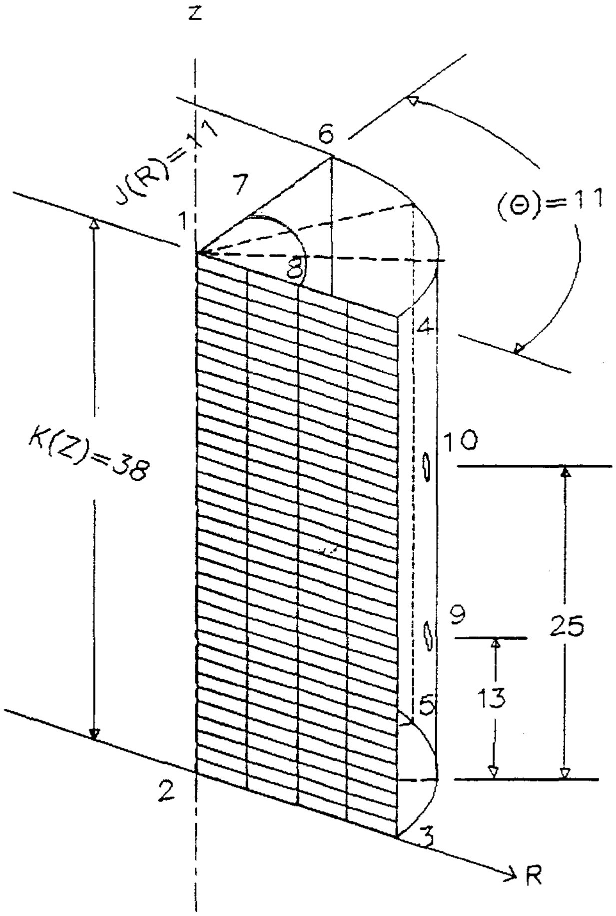


Figure 2 Flow System and Velocity Component in the Freeboard

2.5 ft/s, and the input conditions for inlets No.2 and No.3 were a radial velocity of 65.47 ft/s, and a tangential velocity of 37.8 ft/s. The yaw angle of the nozzle was 30 degrees. The average primary gas flow velocity, 2.55 ft/s, was given in the initial conditions for the computation process.

Since the swirling flow is a complex flow, the turbulent viscosity is strongly directional. The isotropic $k-\epsilon$ model is not adequate for such flow [6]. The Algebraic Stress Model (ASM) was selected to solve algebraic approximations of the differential transport equations for the Reynolds stresses in the polar coordinates. The ASM provided a non-isotropic descriptions of turbulence that is of particular importance in flow dominated by strong swirl [7].

The gas density and viscosity are computed based on the ideal gas flow and in pure-component flows derived from the kinetic theory [3].

After 4890 iterations by CFD code of Fluent 4.26, a good convergence was indicated, and the final results were obtained. Those profiles included the three components of velocity (vertical, radial, tangential components) and relative static pressure.

The vertical velocity as a function of Z-location is shown in Figure 2.1. A down-flow was shown in the lower section (7 and 10 inches) near the wall region (R=4.5 inches). The center region exhibited high up-flow. At the top center, vertical velocity increased to about 80 ft/s since the flow-cross area was reduced to the top outlet. This is also shown in Figure 2.2. A high negative radial velocity at the top section indicated the gas flow from the wall region to the center region. Also,

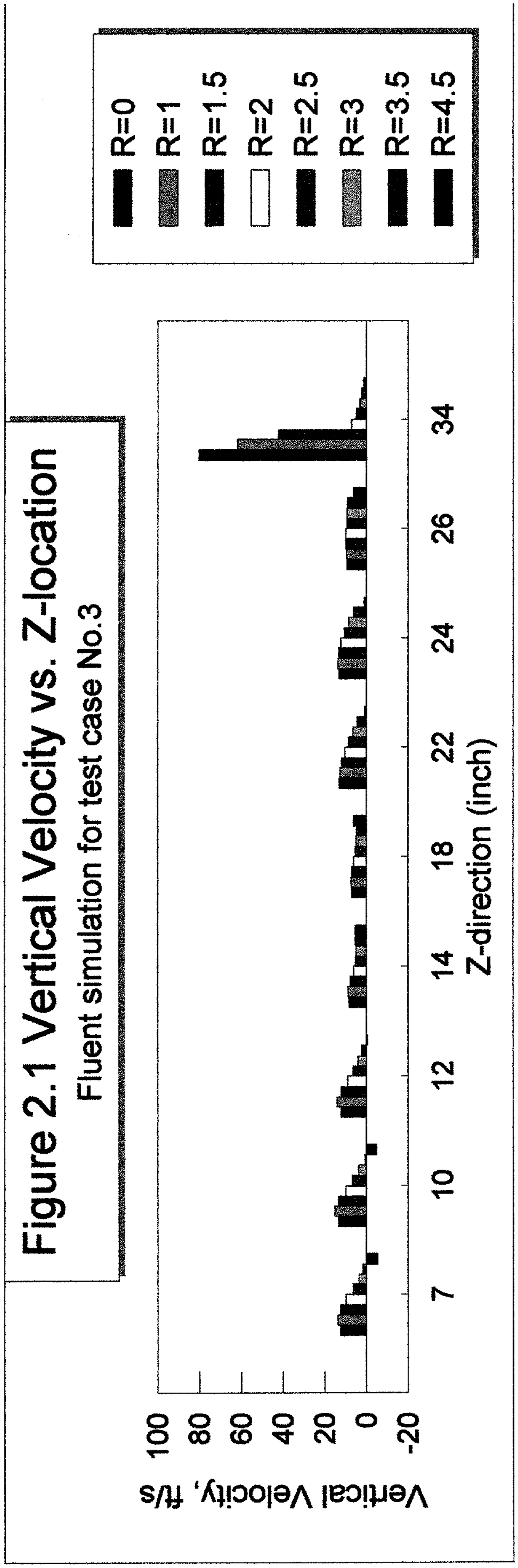


Figure 2.1 Vertical Velocity vs. Z-location

Fluent simulation for test case No.3

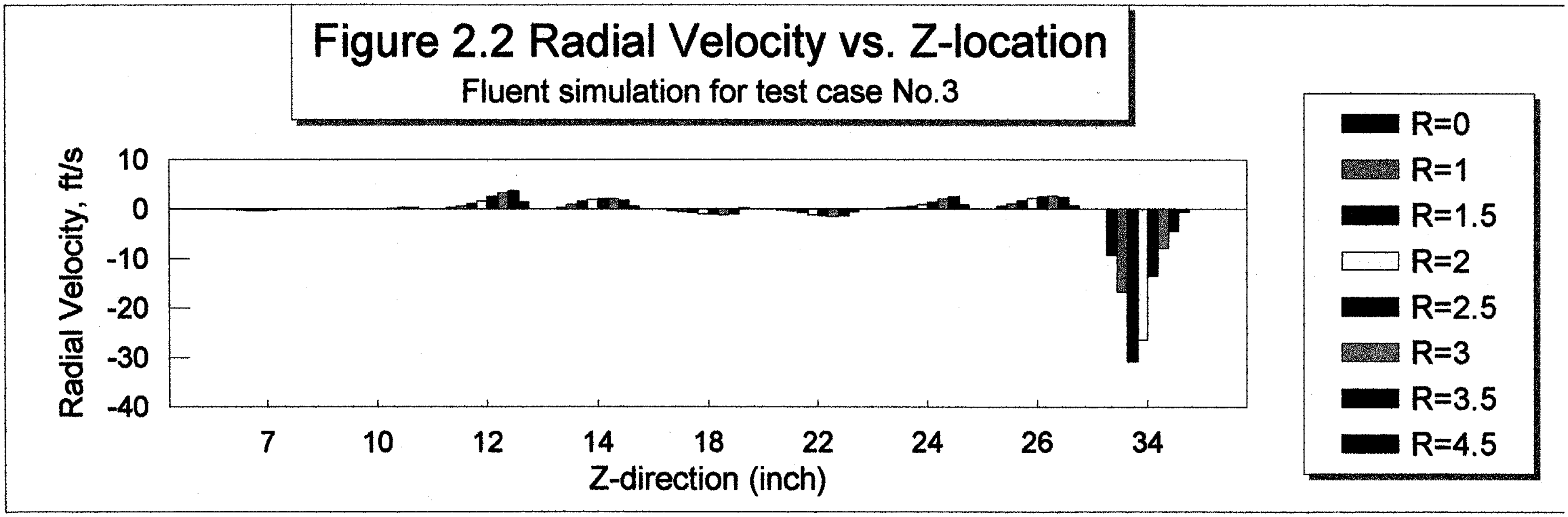


Figure 2.2 Radial Velocity vs. Z-location

Fluent simulation for test case No.3

there was a negative relative static pressure at the top center region. The tangential velocity as a function of Z-location and R-location is shown Figure 2.3. The highest tangential velocity, about 30 ft/s, was found around the secondary air injection nozzle region at Z-location 12 and 24 inches and R=3.5 inches.

The tangential velocity reduced as the radial distance from the center line decreased. At the center line, it should have been close to zero. However, Figure 2.2 does not show the zero value because the calculation value is based on the center point of each cell, therefore the radial distance shifts about 0.25 inches. Higher pressure was found near the wall region and lower pressure was found near the combustion chamber center as shown in Figure 2.4.

The axial velocities resulting from theoretical prediction were smaller than the tested results. However, the predicted radial velocities at the exit zone of the test chamber were a little greater than that of the tested results.

Figure 2.3 Tangential Velocity vs. Z-location

Fluent simulation for test case No.3

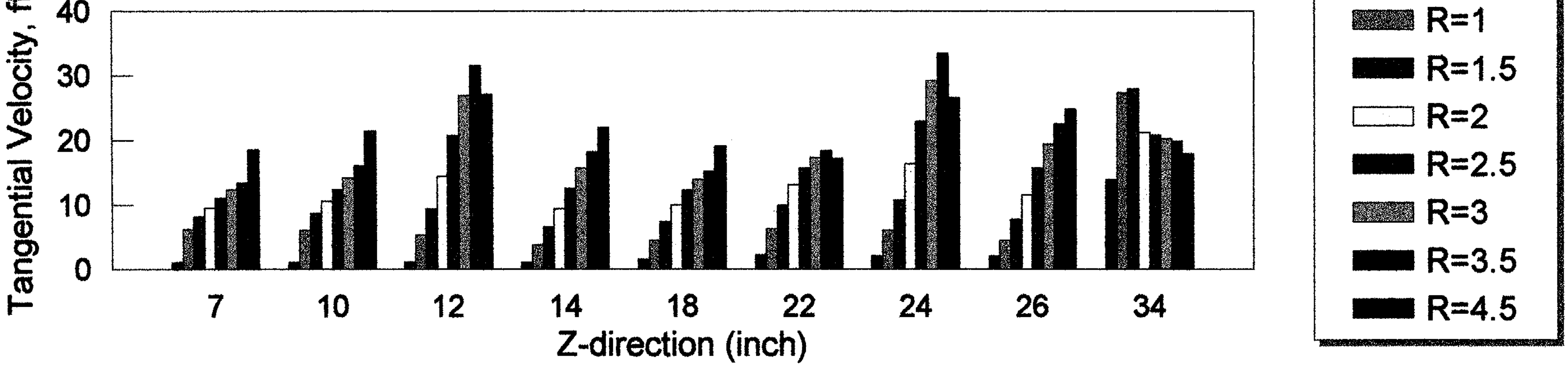
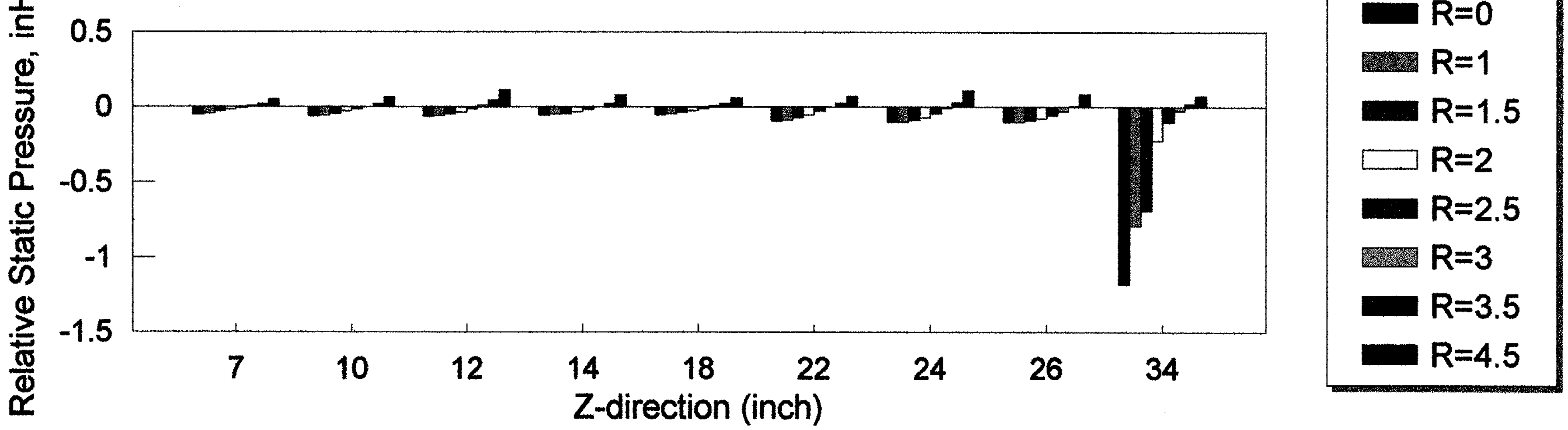


Figure 2.4 Static Pressure Change in Z-direction

Fluent simulation for test case No. 3



SECTION 3

Research Continuation

The progress of this project has been on schedule. The particle flow measurements will be continued under different test conditions. In addition, the numerical simulation on gas and particle flow will be continued, which will be compared with the experimental results.

REFERENCES

[1] Lee, S.W., Technical Progress Report, No.8 to U.S. DOE, Pittsburgh Energy Technology Center, Cctober 1955.

[2] Syred, N. and J.M. Beer, Combustion in Swirling Flows: a Review, Combustion and Flames, V.23, pp.143-201, 1974.

[3] Hetsroni Gad, Handbook of Multiphase Systems, McGraw-Hill book, Co, Chapters 1,8, 1982.

[4] Arena U. et al., Hydrodynamics of A Circulating Fluidized Bed with Secondary Air Injection, Pro. CFB Technology IV, pp. 899-905, 1993.

[5] Sloan, D.G., P. J. Smith, and L.D. Smoot, Modeling of Swirl in Turbulent Flow System, Progress in Energy and Combustion Science, Vol.12, pp. 163-250, 1986.

[6] Khalil, E.E., Numerical Computations of Turbulent Flow Structure in Cyclone Chamber, Joint ASME/AIChE 18th National Heat transfer Conf., ASME paper No. 79-HT-31, 1979.

[7] Boysan, F., J. Swithenbank, and C.J. Lawn, Modeling Coal-Fired Cyclone Combustor, Combustion and Flame, Vol.63, pp.73-85, 1986.