Final Technical Report

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

To:

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

For

Project No: DE-FG22-93MT93006

By

Dr. Seong W. Lee, Principal Investigator

Morgan State University
School of Engineering
Baltimore, MD 21239
(phone) 443 885 3106

ACQUISITION & ASSISTANCE
2000 MAR -1 A 9:44
USDOE-FETC

October, 1998

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, make any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.

TABLE OF CONTENTS

LIST OF TABLES.....	Page iv
LIST OF FIGURES.....	v
<u>CHAPTER</u>	
1. INTRODUCTION.....	1
2. DESIGN AND FABRICATION OF THE EXPLORATORY COLD MODEL (6" ID).....	2
(a) Design and Fabrication of the Test Chamber/Nozzles.....	2
(b) Design Considerations.....	4
(c) Auxiliary System for the 6" ID Cold Model.....	6
(d) Design and Calibration of the Electrostatic Impact Probe.....	7
3. MEASUREMENT OF GAS FLOW FIELD IN COLD MODEL (6" ID).....	9
(a) Experimental Apparatus and Instrumentation.....	9
(b) Gas Recirculation Flow in the Freeboard.....	16
(c) Axial Velocity Distribution.....	19
(d) General Gas Flow Pattern in the Freeboard.....	21
4. MEASUREMENT OF PARTICLE FLOW FIELD IN COLD MODEL (6" ID).....	22
(a) Experimental Considerations/Test Conditions.....	22
(b) Distribution of Particle Mass Flux in Axial Direction.....	23
(c) Distribution of Particle Mass Flux in Radial Direction.....	25
(d) Particle Suspension Layer in the Freeboard.....	25
(e) Particle Residence Time	27
(f) Particle Velocity Measurement by Modified Dual Static Pressure Probe (MDSPP).....	28
5. DESIGN AND FABRICATION OF THE BENCH-SCALE COLD MODEL (10" ID)	41
6. MEASUREMENT OF GAS FLOW FIELD IN THE BENCH-SCALE COLD MODEL (10" ID)	44
(a) Experimental Apparatus and Instrumentation	44
(b) Test Conditions	45
(c) Test Result and Discussion	45
(d) Numerical Modeling and Simulation for Bench-Scale Cold Model	56
7. DESIGN AND FABRICATION OF THE EXPLORATORY HOT MODEL	84

8. AUXILIARY SUBSYSTEMS OF EXPLORATORY HOT MODEL	86
(a) Air Supply Subsystem	86
(b) Water Supply Subsystem	86
(c) Ignition Subsystem	88
9. SFBC HOT MODEL TESTING	88
(a) Preliminary Test Results and Discussions	88
(b) Proof-of-Concept Test	97
(1) Instrumentation for Flue Gas Composition Measurement	97
(2) Procedures of Flue Gas Composition Measurement	102
(3) Effect of Cooling Water	102
(4) Combustion Procedures and Pollution Performance	107
(5) Modification of the Exploratory Hot Model	107
(c) Results and Discussion	115
(1) Heat Balance Calculation Results	118
(2) Thermal Analysis and Heat Transfer Effect	124
(3) Effect of Secondary Air Flow Ratio and Heat Transfer Coefficient	127
10. NUMERICAL MODELING AND SIMULATION FOR THE EXPLORATORY HOT MODEL	132
(a) Overall Description of Numerical Modeling and Simulation	132
(b) The Flow Patterns in the Hot Model	138
(c) The Velocity Profiles in the Hot Model	140
(d) The Pressure Profiles in the Hot Model	148
(e) The Temperature Profile in the Hot Model	150
(f) The Gas Concentration Distribution and Characteristics	157
(g) Heat Transfer Characteristics	159
11. CONCLUSIONS	166
References	168

LIST OF TABLES

	<u>Page</u>
1. Test Conditions for FLUENT Simulation.....	58
2. Curvature-Related Source Terms in the RSM.....	61
3. Test Conditions of Single Particle Injection	78
4. Test Conditions	91
5. Test Conditions	92
6. Test Conditions	93
7. Summary of Test Conditions (with Cooling Water) (Test A)	105
8. Summary of Test Conditions (without Cooling Water) (Test B)	106
9. Summary of Hot Model Test (Case A)	116
10. Summary of Hot Model Test (Case B)	117
11. Summary of Hot Model Test Result (1)	128
12. Summary of Hot Model Test Result (2)	129
13. Test Conditions for Simulations	138

LIST OF FIGURES

	<u>Page</u>
1. Schematic Diagram of Test Chamber (6"ID) and Nozzles of the Exploratory Cold Model	3
2. Schematic Diagram of Parameters of Swirl Number in Exploratory Model...	5
3. Schematic Diagram of the Electrostatic Impact Probe/Associated Signal Processing Unit	8
4. Schematic Diagram of 3-Dimensional Probe System.....	10
5. Calibration Curve of the Pitch Angle Coefficient.....	12
6. Calibration Curve of Immersion Effect of 3-Dimensional Probe.....	13
7. Calibration Curve of Immersion Effect with Different Pitch Angles.....	14
8. Calibration Curve of Immersion Effect with Different Angles.....	15
9. Characteristics Curve of Blower for Fluidized Bed System (Single Blower Arrangement).....	17
10. Measured Axial and Tangential Air Velocity Distributions in the Exploratory FBC Model.....	18
11. Gas Axial Velocity Distribution affected by Swirl Number.....	20
12. The Axial Distribution of Normalized Particle Mass Flux in the Exploratory Model.....	24
13. The Radial Distribution of Normalized Particle Mass Flux in the Exploratory Model.....	26
14. Schematic Diagram of Modified Dual Static Pressure Probe.....	30
15. Background Signal from Channel #0 (0.08 psi)	33
16. Background Signal from Channel #1 (0.2 psi).....	34
17. Signal Shift Time in 6" Unit.....	35
18. Background Signal from Channel #0 (0.08 psi).....	36
19. Background Signal from Channel #1 (0.2 psi).....	37

20. Signal Shift Time in 6" Unit.....	38
21. Background Signal from Channel #0 (0.08 psi).....	39
22. Background Signal from Channel #1 (0.2 psi).....	40
23. Signal Shift Time in 6" Unit.....	42
24. Schematic Diagram of the Bench-Scale Advanced FBC Cold Model System.....	43
25. Tangential Velocity vs. Z-location. (Test case No. 1).....	46
26. Radial Velocity vs. Z-location. (Test case No. 1).....	46
27. Vertical Velocity vs. Z-location. (Test case No. 1).....	46
28. Static Pressure Change in Z-direction. (Test case No. 1).....	47
29. Tangential Velocity vs. Z-location. (Test case No. 2).....	49
30. Radial Velocity vs. Z-location. (Test case No. 2).....	49
31. Vertical Velocity vs. Z-location. (Test case No. 2).....	50
32. Static Pressure Change in Z-direction. (Test case No. 2).....	50
33. Tangential Velocity vs. Z-location. (Test case No. 3).....	51
34. Radial Velocity vs. Z-location. (Test case No. 3).....	52
35. Vertical Velocity vs. Z-location (Test case No. 3).....	53
36. Static Pressure Change in Z-direction. (Test case No. 3).....	54
37. Flow System and Velocity Component in the Freeboard (Slices:K=1, J=8, K=37).....	59
38. Gas Velocity Vectors(feet/s).(K=1, 6, 12, 24, 18, 30, 33).....	62
39. Gas Pressure Profile, Static Pressure (psi). (K=1, 6, 12, 18, 24, 30, 33).....	63
40. Gas Velocity Vectors(feet/s) with secondary Air Injection.....	64
41. Gas Pressure Profiles(psi).....	65

42. Gas Velocity Vectors(feet/s).....	66
43. Gas Velocity Vectors (feet/s). (K=6).....	67
44. Gas Velocity Vectors (feet/s). (K=12).....	68
45. Gas Velocity Vectors (feet/s).(K=18).....	69
46. Gas Velocity Vectors (feet/s). (K=24).....	70
47. Gas Velocity Vectors (feet/s). (K=30).....	71
48. Gas Pressure Profile (psi). (K=6).....	72
49. Gas Pressure Profile (psi). (K=12).....	73
50. Gas Pressure Profile (psi). (K=18).....	74
51. Gas Pressure Profile (psi). (K=24).....	75
52. Top View of Particle Trajectory in the Combustor.....	80
53. Side View of Particle Trajectory in the Combustor.....	81
54. Three-Dimensional Plot of Particle Trajectory in the Combustor.....	82
55. Schematic Diagram of the Exploratory Hot Model.....	85
56. Schematic Diagram of the Gas Distributor.....	87
57. Schematic Diagram of Fuel Nozzle with a Cone.....	89
58. Temperature Profile in the Combustion Chamber.....	94
59. Temperature Profile in the Combustion Chamber.....	95
60. Temperature Profile in the Combustion Chamber.....	96
61. Test Conditions and Temperature Profile in Combustion Chamber (Test A)...	98
62. Test Conditions and Temperature Profile in Combustion Chamber (Test B)...	99
63. Test Conditions and Temperature Profile in Combustion Chamber (Test C)...	100
64. Test Conditions and Temperature Profile in Combustion Chamber (Test D)...	101

65. Test Conditions and Temperature Profile in Combustion Chamber.....	103
66. Stack Temperatures vs Combustion Gas Analyzer.....	108
67. Stack Temperatures vs Combustion Gas Analyzer.....	109
68. Schematic Diagram of Air Injection Nozzles (Type B).....	111
69. Schematic Diagram of Nozzle and Sub-Assembly System.....	112
70. Schematic Diagram of Fuel Injection Nozzle.....	113
71. Schematic Diagram of Igniter System.....	114
72. The Changes of the Flame Enthalpy/Heat Loss along the Combustor Height...	123
73. The Changes of the Flame Enthalpy/Heat Loss along the Combustor Height...	123
74. The Changes of the Flame Enthalpy/Heat Loss along the Combustor Height...	126
75. Local Heat Transfer Coefficients for the Three Sections in the Combustion Chamber.....	126
76. Local Heat Transfer Coefficients for the Three Sections in the Combustion Chamber.....	131
77. Flow System and Velocity Component in Combustor Chamber.....	135
78. Top View of the Computational Domain in Combustor Chamber.....	136
79. Computational Domain and Different Grid Spacings in Combustor Chamber..	137
80. Flow Patterns of the Side View in the Combustor.....	141
81. Flow Patterns of the Top View in the Combustor.....	142
82. Velocity Profiles of the Vertical Direction in the Combustor.....	143
83. Velocity Profiles of the Vertical Direction at Different Levels (K = 6, 10, 20, 30, 40, 50, 60, and 70).....	144
84. Velocity Profiles at Level K = 10 in the Combustor.....	145
85. Velocity Profiles at Level K = 30 in the Combustor.....	146
86. Velocity Profiles at Level K = 50 in the Combustor.....	147

87. Side View of Static Pressure Profiles.....	149
88. Top View of Pressure Profiles at Different Vertical Levels; K = 6, 10, 20, 30, 40, 50, 60, 70.....	151
89. Side View of Temperature Profiles.....	152
90. Top View of Temperature Profiles at Different Vertical Levels; K = 6, 10, 20, 30, 40, 50, 60, 70.....	153
91. Top View of Temperature Profiles at Level, K = 6.....	154
92. Top View of Temperature Profiles at Level, K = 30.....	155
93. Top View of Temperature Profiles at Level, K = 50.....	156
94. The Concentration Profiles of the Methane.....	158
95. The Concentration Profiles of the Carbon Dioxide.....	160
96. The Concentration Profiles of the Oxygen.....	161
97. The Side-view of the Grid Profiles for Heat Flux.....	162
98. The Heat Flux Profiles near the Combustor Wall.....	163
99. The Heat Transfer Coefficients along the Combustor Wall Height.....	165

1. INTRODUCTION

The existing coal combustion and gasification technologies, namely stoker-fired combustion, pulverized coal-fired combustion, and fluidized bed combustion must be reevaluated and modified to meet the ever-stringent environmental and economic needs.

Fluidized bed combustion (FBC) is of continuously increasing interest as an attractive alternative to conventional stoker-type boilers because their attractive operation efficiencies and reliable method for direct combustion of high-sulfur, low grade coals in an economically and environmentally acceptable manner [1].

Over 20 years of the development, several undesirable features were found to be inherent with this first generation FBC boiler system; problems with solid feeding, relatively low combustion efficiency due to unburned coal particles existing in the freeboard, low calcium utilization of sorbents for sulfur retention that affects the emission of pollutants and elutriations of fines, erosion of in-bed tubes, and poor loading capability [2]. High fly ash recycle design may improve the combustion efficiency and sulfur capture. However, it may add complications to the feed system. The bed slumping technique may give better turndown operation. Because of that, a large margin of fan power may be required, and the slumped bed surface may overheat and result in clinker formation. Load turndown is the ability to vary the firing rate of a combustor to match the system of energy demands. Maximum load turndown ratio is defined as the ratio of maximum to minimum fuel firing rates.

Inherent to conventional FBC design is an inability to produce large variations in heat transfer rate that are modest and are accompanied by degradation in combustion. Thus, innovative concepts in bed design are highly desired to control heat transfer

independent of combustion. This capability is especially important for small-scale boilers and furnaces.

The objective of this research is to predict the heat transfer and combustion performance in a newly designed fluidized bed combustor and to provide design guidelines and innovative concepts for advanced small-scale boilers and furnaces. The research consists of five major phases of work:

- (1) design/fabrication of the exploratory/bench-scale test facility (cold model)
- (2) Prediction of the gas recirculating flow/particle suspension flow in the freeboard of the cold model
- (3) Design, fabrication, and combustion test of the combustor (hot model)
- (4) Prediction of the heat transfer characteristics in the combustor
- (5) Numerical modeling and computer simulation for the cold model tests and the hot model tests.

2. DESIGN AND FABRICATION OF THE EXPLORATORY COLD MODEL (6" ID)

(a) Design of the Test Chamber/Nozzles

This test chamber was made of transparent acrylic tube (Plexiglas) with 6" I.D. to facilitate visual observation. The test chamber is physically divided into several parts by a Plexiglas perforated distributor plate, the wind box below the distributor, the exhaust pipe, and the freeboard above the distributor. Eight (8) nozzles were mounted at the freeboard in different levels to provide secondary air, as shown in Figure 1.

Figure 1 Schematic Diagram of Test Chamber (6" ID) and Nozzles of the Exploratory Cold Model

(b) Design Considerations

According to the commonly adopted definition [3], the dimensionless swirl number for the test chamber can be expressed as follows;

$$\text{Swirl Number} = (\text{Inlet Angular Momentum}) / [(\text{Outlet Axial Momentum}) * D_e / 2]$$

$$\text{swirlNumber} = \frac{\pi \tau^2 (D_i D_e)}{4 A_t}$$

where D_e = Diameter of exit tube,

D_i = Diameter of imaginary circle of secondary air injection,

A_t = Total outlet area of secondary air nozzle, and

τ = Volumetric flow rate ratio between secondary air flow and total air flow

Figure 2 shows the schematic diagram for parameters of the swirl number. The swirl number can be changed by adjusting the volumetric flow ratio and secondary air injection angle. In order to maintain a stable combustion process and good fluidization, the volumetric flow ratio is required to vary within the limited range. Therefore, the nozzle along with an adjustable injection angle was considered and designed for the appropriate airflow.

The nozzle is well sealed into the Plexiglas box, which is glued with a test chamber. Secondary air is introduced from the bottom by a 2" ID tube and horizontally injected into the test chamber via a 1" ID tube manipulated by an angle deflector. The injection angle of each nozzle can be independently varied and easily read from an angle indicator on the nozzle.

The swirl number and the secondary air injection angles are considered as two major parameters affecting the gas recirculating flow.

Figure 2 Schematic Diagram of Parameters of Swirl Number in Exploratory Model

(c) Auxiliary System for the 6" ID Exploratory Cold Model

The exploratory of the 6" ID model has an auxiliary system, which includes the air supply system, test chamber, and instrumentation for the particle measuring system.

Two 1-hp, two-speed centrifugal blowers were used to supply the primary air and secondary air separately during the test. Primary air was sent into the wind box of the test chamber from the bottom, and passed through a distributor to fluidize the bed material. Secondary air was injected into the freeboard of the test chamber tangentially through a number of injection nozzles to generate a strong swirling flow field. The volumetric flow rate is controlled by adjusting the blower's voltage through the AC variac transformer. A particle feeder loads the test particles into the primary air pipelines and feeds the particles into the lower chamber. A dust collector with a paper bag filter was used for particle collection and removal for the cold flow tests.

The particle sampler was made of copper tubing, which was mounted at the freeboard with connection of a vacuum pump. This system collected test particles for microscopic analysis. Sample particles at various locations in the test chamber were classified and connected with the aid of a mono-specular microscope to determine the dynamic size distributions in the system.

An electrostatic impact probe and the associated signal processing units measured the particle mass flux, which was designed and fabricated in our laboratory. In order to measure the gas velocity, a three (3) dimensional probe was used along with manometers. Measurement of the residence time is generally difficult for mixed size particles in a complex flow field. A simplified gross measurement of particle residence time, namely distribution function method was used in this study.

(d) Design and Calibration of Electrostatic Impact Probe

The electrostatic probe was designed and fabricated, which could serve as the input transducer to generate impact charge signals. The series of converting/amplifying network circuits was built to count the number of charge signals. The charge signals are first converted into voltage signals by means of a charge-to-voltage (Q-to-V) converter, which is combined by an operational amplifier (TL084 CN), two resistors, and a feedback capacitor (100 PF), as shown in Figure 3. A simple amplifier circuit [4] was used to apply the converted signals forward to the pulse sampler, which corresponds to the particle-sensor collision. The digital signals are recorded and displayed by a digital frequency counter. A copper wire with a 1-mm diameter was used along with a probe sensor. As shown in Figure 3, a 1mm diameter metal ball was used as the impact probe sensor, which is electrically isolated from the probe stem. Under this condition, the probability of having two or more particles striking on the sensor at the same time will be very rare.

To verify the working principle of the electrostatic impact probe and to establish the reliability of the probe readings, several tests were performed. When the compressed air jet was suddenly directed to the probe sensor tip, no electric charge signals were generated or detected by the impulse force. This phenomenon indicates that variation of the probe tip caused by the impact of moving particles will not affect the probe readings. When the fluidizing air increased, the readings of the impact probes increased with the increasing air velocity. The impact signals can not be generated without the particle surface contact. The number of impact particles can be conveniently read from the

Figure 3 Schematic Diagram of the Electrostatic Impact Probe/Associated Signal Processing Unit

frequency counter by the impact probe. By multiplying the total number of impacts per unit time with the local averaged particle mass and dividing by the projected area of the metal sensor, the local particle mass flux can be determined [5].

3. MEASUREMENT OF GAS FLOW FIELD IN COLD MODEL (6" ID)

(a) Experimental Instrumentation and Apparatus

1.1 Air Flow Measuring System

The flow measuring system was carefully arranged with the auxiliary system. A pitot tube and an inclined manometer were used to measure the primary air velocity and thus flow rate. A DA & DAT 3-dimensional directional probe with four manometers was arranged with the bench-scale cold test model to measure the local air velocities.

1.2 Working Principle and Calibration of Static Pressure Probe

Figure 4 shows the schematic diagram of the 3-dimensional velocity probe system. The flow pitch and yaw angles can be found based upon the four readings of four U-type manometers [6]. Results of the pitch angle of the gas flow, stagnation and static pressures, and axial, radial, and tangential components of the gas velocity are derived from the readings of the pressure differentials, $P_4 - P_5$, P_1 , $P_1 - P_2$, and $P_2 - P_3$ [7]. The probes can transverse in the test chamber along three major axes to give an adequate spatial resolution of measured quantities.

Indicated static pressure (P_2 or P_3) is more subject to errors in measurement than indicated total pressure (P_1). Introducing a probe into a small passage immediately changes the static pressure of the passage itself by reducing the cross-flow. The drop in

(a) DA&DAT Probe System

(b) Configuration of DA&DAT Probe

Figure 4 Schematic Diagram of 3-Dimensional Probe System

static pressure at the probe cross-section and downstream from it has been experimentally found to be approximately,

$$\frac{P_t - P_{s1}}{P_t - P_{s2}} = 1.2 \frac{a}{A}$$

Where P_{s1} and P_{s2} are the original and modified static pressures, P_t is total pressure and a/A is the fraction of passage area (A) blocked by the projected probe area (a). Figure 5 shows the calibration curve of the pitch angle coefficient under ambient conditions (temperature: 84 °F; barometric pressure: 1004 mbar; relative humidity: 31.2%).

In addition various errors are related to the Mach number, pitch angle, yaw angle, and immersion depth. Yaw angle errors are zero when the probe is rotated ($P_2 = P_3$). Mach number errors are shown in the individual calibration charts, as shown in Figures 6 to 8. Generally the indicated static pressure (P_2 or P_3) increases with the Mach number, $(P_2 - P_s) / (P_t - P_s)$ varying approximately linearly with the quantity $(P_t P_s)/P_t$, abs. Immersion errors are due to the effects of boundaries of the passage and secondary flow along the upstream edge of the probe that in turn is influenced by the total pressure gradient in the passage, as shown in Figures 6 to 8.

The accompanying curves show the magnitude of errors when the probes are placed near the boundaries. For comparison, a curve for a standard pitot-static tube is inclined. This indicates that even this type can read as much as 4% low near a boundary.

1.3 Instrumentation of the Primary Air Flow

Pitot tube and an inclined manometer were used to measure the primary air velocity and flow rate. Two 1-hp, two-speed centrifugal blowers were connected to supply the primary air and secondary air separately to the bench-scale cold test model.

Figure 5 Calibration Curve of the Pitch Angle Coefficient

Figure 6 Calibration Curve of Immersion Effect of 3-Dimensional Probe

Figure 7 calibration Curve of Immersion Effect with Different Pitch Angles.

Figure 8 Calibration Curve of Immersion Effect with Different Angles

These blowers can be arranged in series, in parallel, or in a combination of series-parallel to meet the needs of desired airflow rate and pressure. The variac transformer controls each blower, which is used to vary the input voltage (0 to 125 volt) and thus the rotating speed (0 to 1750 rpm) of the driving motors. Two valves and 2-inch PVC tubes are arranged to conveniently adjust the testing flow conditions. The measured blower characteristics in terms of the gas flow rate and static pressure head for the single blower could reach 110 cm-of-water. The maximum air velocity was 35 m/s under an input voltage of 125 volt, as shown in Figure 9.

(b) Gas Recirculation Flow in the Freeboard

A DA & DAT 3-dimensional directional probe with four (4) manometers was used for the gas velocity measurements. This probe was able to transverse in the test chamber along three major axes (axial, radial, and tangential) to provide an adequate spatial resolution of the measured quantities. The pitot tube and the inclined manometers were also used to measure the primary air velocity and flow rate.

Measurements were conducted based upon the assumption that the flow field was axi-symmetric. The following parameters were tested and observed during the experimental work: (a) mass flow ratio of primary and secondary air, (b) angles of secondary air jets, and (c) the number of secondary air nozzles.

Figure 10 shows the typical test results of tangential and axial velocity distributions in the freeboard of the bench-scale FBC. The tangential velocity profiles show rapid decay from the wall toward the center in the freeboard region. There was no

Figure 9 Characteristics Curve of Blower for Fluidized Bed System (Single Blower Arrangement)

Figure 10 Measured Axial and Tangential Air Velocity Distributions in the Exploratory FBC Model

significant tangential velocity change in the axial direction. However, the tangential velocity was detected in the cross-section near the exit opening of the exhaust pipe.

A recirculating flow in the axial direction exists in the area below the lower nozzles of the bottom two cross-sections. This gas internal circulation will enhance the solid circulation, which is desirable for coal particle combustion by increasing coal particle residence time and solid/gas mixing. Basically, the gas recirculating flow results in the sudden expansion of the chamber. In the mean time, the swirling flow may significantly affect the region of recirculating flow. The swirl number and the secondary air injection angles are considered as two major parameters that affect the gas recirculation flow.

(c) Axial Velocity Distribution

The measurement of gas flow continued with the assumption of axi-symmetric flow in the laboratory-scale FBC. The gas velocity and pressure measuring system consisted of the DA&DAT 3-dimensional (5-hole directional) probe, a universal traverse probe holder, four manometers, and one pressure differential gage.

The gas flow field in the test chamber of the FBC was assumed to be axi-symmetric. Measurements were made primarily in a vertical plane passing through the chamber axis. Figure 11 shows the gas axial velocity distributions at two cross-sections. Three sets of data from the upper part of the FBC were obtained at a cross-section 3" below the air inlet of the lower level (designated as section 1), and the other three sets were measured at a cross-section 3" below the location of section 2. The axial locations of the two sections are indicated in the bottom area of Figure 11.

Figure 11 Gas Axial Velocity Distribution affected by Swirl Number

As shown in Figure 11, the circulating flow is relatively strong (swirl number of 3.0) near the freeboard wall for the two cross-sections because the flow has high reversal gas velocity. The recirculation flow had a weak air velocity when the swirl number was 0.60. For the non-swirling flow, the axial velocity was low near the freeboard wall. However, no circulating flow was observed at the two cross-sections.

(d) General Gas Flow Pattern in the Freeboard

The measurements of the gas flow field in the test chamber were continued to understand the flow pattern in the FBC freeboard. The existing axial velocity distributions showed that the peak of axial velocity detected in the vicinity of the wall was above the nozzles level. It was found that the gas recirculating flow existed below the lower level secondary air injection nozzles. This recirculating flow is suspected of contributing to the solid particle recirculation in the freeboard and enhances the particle/gas mixing, and solid particle resident time[8, 9]. A relative uniform velocity distribution was exhibited in the freeboard region.

For the ideal tangential gas distribution, there was a higher tangential velocity in the core region, which is able to allow solid particles to move to the outward of the freeboard. The measured gas tangential velocity in the exploratory model is nearly zero in the core region (close region from the centerline to $1/4 R$). This zero tangential velocity region is referred to as dead zone since the particles leave this zone.

Although the tangential momentum energy is supplied from secondary air injection, the tangential gas velocity in the two cross-sections below the lower level nozzles was not significantly smaller than that in the cross-sections above the nozzles,

which is caused by a recirculating flow. This tangential gas velocity is believed to increase the combustion performance due to the early solid particle separation, which can require a lower freeboard height.

4. MEASUREMENT OF PARTICLE FLOW FIELD IN COLD MODEL (6"ID)

(a) Experimental Considerations and Test Conditions

A series of visual observations and measurements on particle flows were designed in the exploratory and bench-scale models. These experimental considerations and test conditions are summarized here:

- Systematic measurements of particle mass flux and density distributions at different measuring sections in the exploratory and bench-scale model were conducted by the improved probe system;
- Observations and measurements of particle suspension regions in the freeboard were made;
- Visual observations and video taping of particle flow patterns in the exploratory and bench-scale models were conducted by VCR/Camera/TV System;
- Measurement of particle entrainment in the exploratory model by sampling / sieving method;
- Measurement of particle residence time in the bench-scale model by a sampling/sieving method,
- Systematic measurements of particle size distributions remained in and exited from the test chamber were conducted using the bench-scale model.

The following assumptions and arrangements are made in the particle flow measurement:

- The particle flow field in the test models was assumed to be axi-symmetric, the same as the gas flow field. Measurements of particle mass flux and density will be made primarily in the vertical plane passing through the chamber.
- The depletion effect of the particle diameter and the material density during the combustion were not considered in particle flow tests.

(b) Distribution of Particle Mass Flux in Axial Direction

The normalized particle axial mass flux measured 1cm from the wall with the electrostatic impact probe system traversing along the freeboard height in the 6" ID test chamber. Two dilute regions with small mass fluxes near the injection nozzles and two dense regions of the peaks in the suspension regions were shown. It is worth noting that the time-averaged mass flux in the lower dilute region is about four (4) times that of the upper dilute region, as shown in Figure 12; although little difference between the regions can be seen through visual observation. The particle mass flux decreases as the height increases and forms a valley near the upper nozzles. The mass flux at the upper nozzle is about an order of magnitude less than that of the lower suspension region. Above the upper dilute region, the mass flux increases again in the upper suspension region, as shown in Figure 12. The axial distribution of the mass flux exhibits a peak, which indicates the plane of higher density of particles. This distribution often shows maximum value near the chamber wall. This is desirable because more oxygen would be needed at

Figure 12 The Axial Distribution of Normalized Particle Mass Flux in the Exploratory Model

the dense particle region near the wall, hence improving gas (oxygen) particle (char) mixing and combustion [9].

(c) Distribution of Particle Mass Flux in Radial Direction

Measurements of particle mass flux were conducted in the 6 inch-diameter test chamber of the laboratory-scale cold model. Figure 13 shows the particle mass flux distribution in the radial direction. Curve A is measured at $H^*=6$ and curve B at $H^*=5$. The particle dispersion is dense near the wall and relatively dilute in the core region. It was found that the particle mass flux generally increases along the radial direction adjacent to the wall of the test chamber.

It is noticed that the mass flux difference in curve B amounts to more than one order of magnitude. As shown in Figure 13, a high mass flux was detected near the wall and core region. Under the interaction of strong centrifugal force, particles in the chamber undergo a spiral trajectory and are thrown towards the chamber wall by the swirling gas flow. On the other hand, when the particles bounce from the chamber wall, turbulent diffusion and inward radial fluid drag tend to push the particles back toward the chamber center.

An interface at a certain radius adjacent to the chamber wall is expected where interacting forces are dynamically balanced. At this balancing interface, the particle mass flux exhibits a peak, which indicates the plane of higher particle density.

(d) Particle Suspension Layer in the Freeboard

When primary and secondary air velocities are maintained in certain specific

Figure 13 The Radial Distribution of Normalized Particle Mass Flux in the Exploratory Model

ranges, dense suspension layers and dilute suspension regions were observed in the freeboard of the exploratory cold model (6" I.D.).

The characteristics of the suspension layers are radial particle distribution, layer thickness, and location, which can be controlled by changing the primary and secondary air velocities. Based upon observations and measurements, the general behavior of suspension layers can be summarized as follows:

(1) Tangential air injection is necessary to generate the suspension layer. This layer is formed immediately above the plane of the injected air. The suspended particles are relatively dense near the freeboard wall and dilute in the core region. Radial air injection does not directly affect the formation of the suspension layers. Instead, it has a detrimental effect to the existing suspension layers.

(2) Once the suspension layers are established, their location and thickness are insensitive to small changes in the secondary air velocity. Excessively high secondary velocities may cause flow instability and fluctuation of the suspension layer.

(3) Particles are fluidized and entrained into the freeboard by the primary air. Primary air velocity affects the total number of particles and hence the particle density in the suspension layers. High primary air is not desirable to the formation of suspension layers because of serious particle elutriation.

(e) Particle Residence Time

Measurement of particle residence time is generally difficult for mixed-size particles in a swirling flow field. A simplified gross measurement of particle residence time, namely the distribution function method, was used in this study.

According to stochastic process theory, fuel particles passing through a test chamber follow a random process, but possess a deterministic residence time distribution. This distribution function $F(t)$ is defined as the accumulated number fraction of particles whose residences are less than t . The distribution density function $f(t)$ is defined as the number percentage, dN , of particles whose residence times lie between t and $t + dt$. when N is the total number of particles. Therefore, dN/N equals to $f(t)dt$. With this notation, normalization condition can be shown as follows:

$$\int_0^{\infty} f(t)dt = \int_0^{\infty} dN / N = 1$$

and the particle residence time distribution function $F(t)$ is

$$F(t) = \int_0^t f(t)dt$$

Thus, the averaged particle residence is then given by

$$\bar{t} = \int_0^{\infty} tf(t)dt$$

$F(t)$, $f(t)$ are generally discrete when measured experimentally.

Thus, the above equation becomes

$$\bar{t} = \frac{\sum t_i f(t_i)}{\sum f(t_i)}$$

(f) Particle Velocity Measurements by Modified Dual Static Pressure Probe (MDSPP)

In the advanced fluidized combustor under swirling flow condition, the combustion of coal-based fuel particles takes place in a highly swirling, developing, and turbulent flow. In order to better understand and control particle behavior, and further improve the combustion performance, it is necessary to investigate the particle dynamic characteristics in the combustor chamber.

1.1 Probe Development and Working Principle

The probe used for the particle velocity measurement is a modified dual static pressure probe (MDSPP). The MDSPP included four copper tubes with 0.19" of diameter and 30" of length, as shown in Figure 14. This probe measures two differential pressures, ΔP_1 and ΔP_2 , at the same time. The distance between the pair of tubes for P_1 and P_2 is 1/2 inches; the distance between the centers of the pair of tubes for P_1 and P_2 is 2 inches. The signals by differential pressure were transmitted and measured by two differential pressure transducers, Validyne P3ODIN2OS4 and ODIN22S4. These pressure transducers produce an analog voltage output with a range of ± 5 volts. The voltage signal is converted to a digital signal through the analog-to-digital (A/D) board and recorded in a computer file. Then the signals are analyzed by the cross-correlation function, which is a program 'xcorr', in MATLAB language. The cross-correlation function is used to find the time delay (or shift time) between the delta P signals. The shift time can be considered as the time for particles moving from the sensor point of ΔP_1 to the sensor point of ΔP_2 . The particle velocity can be calculated by the distance between the two ΔP and the signal shift time, tsf. If we assume that the first signal delta P is $R_x(t)$ at time t , and the secondary signal delta P is $R_y(t + tsf)$, the results can be presented as follows,

$$R_{xy} = \frac{1}{t_1} \int_0^{t_1} R_x(t) R_y(t + t_{sf}) dt$$

The time-tag, tsf, is such that the maximum value of the cross-correlation function is an

Figure 14 Schematic Diagram of Modified Dual Static Pressure Probe

indication of the dominant peak in the plot. The particle moving velocity can be calculated by using the following function:

$$V_p = \frac{L}{t_d}$$

1.2 Experimental Setup and Instrumentation

Based on signal transport technology [10], an experimental method was developed to measure the particle tangential velocity in the 6" ID cold flow model. The MDSPP probe is inserted into the chamber at 1.5" from the gas distribution plate and set at three radial locations at 1", 2", and 2.25" from the axis line of the combustion chamber. The particles used in this test are tubular macaroni with an average size of 5 mm, and particle density of about 1.8 g/cc without holes and 0.9 g/cc with the inside hole. The particles terminal velocity is 12.21 m/s, and the minimum fluidization velocity is 1.15 m/s. In the test, about 300g of the particles are put in the chamber. The primary airflow rate into the wind box is 0.0343 m³/s. The primary air pressure is about 1 psi. The total secondary airflow rate into the chamber is 0.0239 m³/s, and the secondary air pressure is close to the atmospheric pressure. The primary superficial velocity in the bottom of the test chamber is 1.88 m/s, and the velocity at the secondary air nozzle outlet is 5.24 m/s. Under these test conditions, a good solid tangential movement was observed. Then particle tangential velocities at the three locations (Z = 1.5", R = 1", 2", and 2.25", Z and R are the vertical distance from the distributor plate and radial distance from the test chamber, respectively) were measured by the MDSPP probe.

1.3 Test Results and Discussions

Three test cases, with different probe locations, have been conducted on the 6" ID cold flow model.

For the first test case, the probe was set at a location of $Z=1.5''$ and $R=1''$. Figure 15 shows the pressure fluctuation signal from channel #0 along with a 0.08-psi pressure transducer, the average pressure drop is about 19.5 mmH₂O with an average pressure fluctuation of 3 mmH₂O. Figure 16 shows the pressure fluctuation signal from channel #1 along with a 0.2-psi pressure transducer. The average pressure drop is about 1 mmH₂O with an average pressure fluctuation of 1 mmH₂O. By comparing Figure 15 and Figure 16, the two figures have similar regions which indicate the same particle cloud. This particle cloud passed the two delta P sensor points with a shift time. The shift time of 0.33 second was determined by cross-correlation function and shown in Figure 1.7. The statistical velocity of particle or particle cloud is 15.3 cm/s.

For the second test case, the probe was set at the location of $Z=1.5''$ and $R=2''$. Figure 18 shows the pressure fluctuation signal from channel #0 along with a 0.08-psi pressure transducer. The average pressure drop is about 17 mmH₂O with an average pressure fluctuation of 3 mmH₂O. Figure 19 shows the pressure fluctuation signal from channel #1 along with a 0.2 psi pressure transducer. The average pressure drop is about -1 mmH₂O with an average pressure fluctuation of 0.5 mmH₂O. By comparing Figures 18 and 19, they have similar regions. The shift time of 0.77 second was determined and shown in Figure 20. The statistical velocity of particle or particle clouds is 6.5 cm/s.

For the third test case, the probe was set at a location of $Z=1.5''$ and $R=2.25''$. Figure 21 shows the pressure fluctuation signal from channel #0 along with a 0.08-psi pressure transducer. The average pressure drop is about 3 mmH₂O with an average pressure fluctuation of 1 mmH₂O. Figure 22 shows the pressure fluctuation signal from channel #1 along with a 0.2-psi pressure transducer. The average pressure drop is 4

Figure 15 Background Signal from Channel #0 (0.08 psi)

Figure 16 Background Signal from Channel #1 (0.2 psi)

Figure 17 Signal Shift Time in 6" unit

Figure 18 Background Signal from Channel #0 (0.08 psi)

Figure 19 Background Signal from Channel #1 (0.2 psi)

Figure 20 Signal Shift Time in 6" unit

Figure 21 Background Signal from Channel #0 (0.08 psi)

Figure 22 Background Signal from Channel #1 (0.2 psi)

mmH₂O with an average pressure fluctuation of 2 mmH₂O. The shift time of 0.33 second is determined and shown in Figure 23. The statistical velocity of particle or particle cloud is 15.3 cm/s.

Case 3 and case 1 indicate that the moving velocity of the particle near the wall region is faster than that near the axis (or inner region). It is believed that the gas velocity near the wall region is faster than that of the center region.

5. DESIGN AND FABRICATION OF BENCH-SCALE COLD MODEL (10" ID)

As shown in Figure 24, the bench-scale advanced FBC test chamber was designed and fabricated to better understand how the gas recirculating flow, particle suspension flow, and the particle elutriation rate are affected by swirling flow in the freeboard of the test chamber.

This test chamber was made of a transparent acrylic tube (Plexiglas) with 10" I.D. to facilitate visual observation. The test chamber was physically divided into several parts by a Plexiglas perforated distributor plate, the wind box below the distributor and the exhaust pipe, and freeboard above the distributor. The key components of the cold models, such as the secondary air nozzles were designed to be adjustable in the tests. Eight nozzles were mounted in the freeboard at different levels to provide secondary air. Each level of the secondary air injection consists of four-equally-spaced nozzles along the circumferential wall of the test chamber. Two high-pressure blowers were used to provide primary air and secondary air to the test chamber.

Figure 23 Signal Shift Time in 6" unit

Figure 24 Schematic Diagram of the Bench-Scale Advanced FBC Cold Model System

6. MEASUREMENT OF GAS FLOW FIELD IN THE BENCH-SCALE COLD MODEL (10" ID)

(a) Experimental Apparatus and Instrumentation

The bench-scale cold model of the 10" I.D. is shown in Figure 24. The primary air supply was connected to the bottom of the chamber. There are two sets of secondary air injectors with two blowers. Each has four nozzles equally spaced on the circumferential wall. An exhaust draft fan with dust collector was installed and connected to the top of the combination chamber. With the exhaust draft fan and a bypass-controlling valve, the chamber pressure was controlled to keep a constant pressure level.

Three pressure taps were installed on the system. The first pressure tap was inserted at the bottom wind chamber to measure the total pressure of primary air. The second tap was inserted at the main frame pipe of the nozzle injectors for measuring the total pressure of the secondary air. The third one was put on the top the of combination chamber to measure the static pressure in the chamber. A pitot static probe was installed on the exhaust pipe to measure the total air flow rate through the system. A DA&DAT 3-dimensional directional probe was used for the measurements of gas velocity and pressure.

A differential pressure transducer and the computer-assisted data acquisition system [10] were employed to measure pressure fluctuations in the freeboard of the test chamber. The differential pressure gage, P305D of Validyne Engineering Corp. was connected to the excitation power supply. The output is 5 VDC for full scale of 0.08 psi and 0.2 psi. The analog signal is sent to the Analog Digital Converter (ADC) board, RTI800 of the Analog Devices, Inc. This ADC board converts the analog signal into a digital signal to be recognized and analyzed by a PC computer. The fundamental

frequency of the pressure fluctuation, the time average size, motion speed of gas bubble and solid cluster can be determined.

(b) Test Conditions

The tests were carried out under three different operating conditions by changing the secondary airflow rate, but with a constant primary flow rate of 2.55 ft/s. In the first case, the upper four nozzles were closed and the lower four nozzles were fully opened. In the second case, the upper nozzles were fully opened and the lower four nozzles were closed. The gas velocity at the nozzle outlet for both cases was 76.5 ft/s. The third case was carried out under a constant primary gas velocity of 2.55 ft/s. All of the eight secondary air injecting nozzles were fully opened while the tests were conducted.

(c) Test Results and Discussion

The flow characteristic profiles with three direction components, vertical velocity (Z-direction), radial velocity (R-direction), and tangential velocity (Q-direction), were measured in the first two cases. The test results for the first case are shown in Figures 25 to 28. These results showed that the secondary air flow rate at the lower section has effects on the gas flow pattern in the test chamber by closing the upper four air nozzles and fully opening the four lower secondary nozzles. In this case, a large tangential velocity, about 55 ft/s, and a large vertical velocity, 22 ft/s were found out near the lower nozzles. Also a stronger vortex flow circulation was formed between the center and near the wall region in the vertical direction. A large tangential velocity occurred at the top center since the top cover forced the air flow into the center exit duct, which had a similar

Figure 25 Tangential Velocity vs. Z-location

Test case No.1: Blower 2# off, 3# low; Valves Upper closed, lower open

Figure 26 Radial Velocity vs. Z-location

Test case No.1: Blower 2# off, 3# low; Valves Upper closed, lower open

Figure 27 Vertical Velocity vs. Z-location

Test case No.1: Blower 2# off, 3# low; Valves Upper closed, lower open

Figure 28 Static Pressure Change in Z-direction

Test case No.1: Blower 2# off, 3# low; Valves Upper closed, lower open

diameter to that of the combustor. The static pressure changed in the Z-direction with about a 0.2 inch water fluctuation.

Figures 29 to 32 show the effect of the secondary air flow rate at the upper section on the gas pattern in the freeboard of test chamber. Thus, conduction was made possible by closing the lower four secondary air nozzles and fully opening four upper secondary nozzles. In this case, a large tangential velocity, about 75 ft/s, was found around the upper nozzles. A large size of swirl pool above the nozzles was formed between the center and near the wall region in the vertical direction. The circulating flow was rather stable with a near zero radial velocity and small relative pressure fluctuations.

At the lower section of the combustion chamber, there was another large size vortex flow down the near the wall, then into the center at the bottom, and then up at the center region. The relative static pressure fluctuation was small in the middle section of the chamber, increasing to both ends of the top and bottom. These results indicate that the vigorous turbulence, such as swirling, recirculating, and developing gas-particle flows with intensified mixing and slip motion can be contributed to intensify the heat/mass transfer and large firing intensity, and high combustion efficiency.

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 in the third case. Figures 33 through 36 show the effects of the secondary airflow 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 a function of both R-direction and Z-direction. In the Z-direction, larger tangential velocities were found around the nozzles

Figure 29 Tangential Velocity vs. Z-location

Test case No.2: Blower 2# off, 3# low; Valves Upper open, lower closed

Figure 30 Radial Velocity vs. Z-location

Test case No.2: Blower 2# off, 3# low; Valves Upper open, lower closed

Figure 31 Vertical Velocity vs. Z-location

Test case No.2: Blower 2# off, 3# low; Valves Upper open, lower closed

Figure 32 Static Pressure Change in Z-direction

Test case No.2: Blower 2# off, 3# low; Valves Upper open, lower closed

Figure 33 Tangential Velocity vs. Z-location

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

Figure 34 Radial Velocity vs. Z-location
 Test case No.3: Upper-Nozzle open, Lower-Nozzle open

Figure 35 Vertical Velocity vs. Z-location

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

Figure 36 Static Pressure Change in Z-direction

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

(12-inch, 24-inch) and exit region of the top (31-inch), as shown in Figure 33. In the R-direction, the larger tangential velocities occurred at the near the wall region. The tangential velocity is almost one order of magnitude larger than the vertical/radial velocities. These results indicated that the strong swirling feature of gas flow is expected in the test chamber.

Figure 34 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 34 and 35. These results are desirable for gas-gas mixing and gas-particle mixing in the combustor. A few small eddies formed in the radial direction. As shown in Figure 35, larger vertical velocities were obtained at 31-inch of Z-direction, which is the effect of mass balance at the top exit of the 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 to the solid particle recirculating in the freeboard and to enhance gas-particle mixing and particle resident time [11]. 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 36. The test conditions for this case have been taken as the input data for the numerical simulation.

(d) Numerical Modeling and Simulation for Bench Scale Cold Model

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 conditions. This was an axi-symmetric, 3-dimensional turbulent flow problem involving low speed primary air input through the gas distributor plate at the combustor bottom and high speed secondary air input from eight secondary air nozzles. The secondary air nozzles were arranged in two axi-symmetrical levels: 12 inches and 24 inches high. In order to reduce the computation times, the cyclic method [12] was used to consider a quarter of the combustor cylinder.

The cold flow pattern in the swirling combustor chamber was simulated by using the Computational Fluid Dynamics code, Fluent [15], to determine the three velocity component profiles in three different directions: vertical direction (K), radial direction (J), tangential direction (I); and the static pressure profile in the chamber. In order to simulate the problem completely, the following basic procedures were carefully considered.

Using a numerical calculation procedure, the governing fluid flow equations, continuity and momentum equations in cylindrical coordinates are employed to determine the velocity and pressure in the combustion chamber. The continuity and three direction momentum equations are the following equations [15,16]: equation (1) through equation (10)

Continuity:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{1}{r} \frac{\partial(r \rho v)}{\partial r} + \frac{1}{r} \frac{\partial(\rho w)}{\partial \theta} = RSM \quad (1)$$

Axial Direction Momentum:

$$\begin{aligned}
& \frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u u)}{\partial x} + \frac{1}{r} \frac{\partial(r \rho v u)}{\partial r} + \frac{1}{r} \frac{\partial(\rho w u)}{\partial \theta} \\
& = -\frac{\partial P}{\partial x} + \frac{\partial(\tau_{xx})}{\partial x} + \frac{1}{r} \frac{\partial(r \rho v)}{\partial r} + \frac{1}{r} \frac{\partial(r \rho v)}{\partial r} + \rho g_x + F_x
\end{aligned} \tag{2}$$

Radial Direction Momentum:

$$\begin{aligned}
& \frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho u v)}{\partial x} + \frac{1}{r} \frac{\partial(r \rho v v)}{\partial r} + \frac{1}{r} \frac{\partial(\rho w v)}{\partial \theta} \\
& = -\frac{\partial p}{\partial r} + \frac{\rho w^2}{r} + \frac{\partial(\tau_{xr})}{\partial x} + \frac{1}{r} \frac{\partial(r \tau_{rr})}{\partial r} + \frac{1}{r} \frac{\partial(\tau_{r\theta})}{\partial \theta} - \frac{\tau_{\theta\theta}}{r} + \rho g_r + F_r
\end{aligned} \tag{3}$$

Circumferential Direction momentum

$$\begin{aligned}
& \frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho u w)}{\partial x} + \frac{1}{r} \frac{\partial(r \rho v w)}{\partial r} + \frac{1}{r} \frac{\partial(\rho w w)}{\partial \theta} \\
& = -\frac{1}{r} - \frac{\rho v w}{r} + \frac{\partial(\tau_{x\theta})}{\partial x} + \frac{1}{r^2} \frac{\partial(r^2 \tau_{r\theta})}{\partial r} + \frac{1}{r} \frac{\partial(\tau_{\theta\theta})}{\partial \theta} + \rho g_\theta + F_\theta
\end{aligned} \tag{4}$$

where u is the axial velocity component, v is the radial velocity component, and w is the circumferential velocity component. The stress tensor in cylindrical coordinates is:

$$\tau_{xx} = 2\mu \frac{\partial u}{\partial x} - \frac{2}{3} \mu \left(\frac{\partial u}{\partial x} + \frac{1}{r} \frac{\partial(rv)}{\partial r} \right) + \frac{1}{r} \frac{\partial w}{\partial \theta} \tag{5}$$

$$\tau_{rr} = 2\mu \frac{\partial v}{\partial r} - \frac{2}{3} \mu \left(\frac{\partial u}{\partial x} + \frac{1}{r} \frac{\partial(rv)}{\partial r} \right) + \frac{1}{r} \frac{\partial w}{\partial \theta} \tag{6}$$

$$\tau_{\theta\theta} = 2\mu \frac{1}{r} \frac{\partial w}{\partial \theta} + \frac{v}{r} - \frac{2}{3} \mu \left(\frac{\partial u}{\partial x} + \frac{1}{r} \frac{\partial(rv)}{\partial r} \right) + \frac{1}{r} \frac{\partial w}{\partial \theta} \tag{7}$$

$$\tau_{xr} = \mu \frac{\partial u}{\partial r} + \frac{\partial v}{\partial x} \tag{8}$$

$$\tau_{x\theta} = \mu \frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{\partial w}{\partial x} \tag{9}$$

$$\tau_{x\theta} = \mu \frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{\partial w}{\partial x} \quad (10)$$

1.1 System Configuration

The solution process required a geometry modeler, grid generator, and system configuration. The system was configured in 3-D cylindrical coordinates with uniform mesh grids. The combustor chamber was 36 inches in height and 10 inches in diameter. The computation grid configuration is shown in Figure 37. There are a total of 12,136 grids in the system configuration, 41 pieces are in the tangential direction, I, 8 pieces are in the radial direction, J, and 37 pieces are in the vertical direction, K. Only three slices for the surfaces of the cylinder were shown in the Figure. They are the bottom surface (K=1), top surface (K=37), and side wall surface (J=8).

1.2 Test Conditions

The test conditions and input boundary conditions are summarized in Table 1.

Table 1. Test Conditions for FLUENT Simulation

Reactor Diameter	Inch	10
Reactor Height	Inch	36
Top Secondary Nozzle	Inch	24
Bottom Secondary Nozzle	Inch	12
Primary Air Flow Rate	ft ³ /s	1.39
Gas Velocity at Bottom	ft/s	2.55
Secondary Air Flow Rate	ft ³ /s	0.2323
Gas Velocity at Nozzle	ft/s	75.6
Nozzle Yam Angle	Degree	30
Nozzle Pitch Angle	Degree	0
Temperature	°C	22
Test Pressure	Atm	1
Gas Density	Lb/ft ³	0.08

Figure 37 Flow System and Velocity Component in the Freeboard
(Slices: $K=1$, $J=8$, $K=37$)

Since the swirling flow is a strong turbulence flow with anisotropic behaviors, the k-ε turbulence model is not suitable for this case. The simplified Algebraic Stress Model (ASM) was selected and tested for the swirling turbulence flow simulation. The gas density was simplified to take the constant value of 0.08 lb/ft³. After 6340 iteration calculations, a good convergence was indicated, for the gas flow profiles. The Reynolds stress models are presented in equations (11), (12), (13), and Table 2.

In the Reynolds averaging of the momentum equations, the velocity at a point is considered as a sum of the mean (time averaged) and fluctuating components:

$$u_i = \overline{u_i} + u'_i \quad (11)$$

Substituting expressions of this form into the basic momentum balance (and dropping the overbar on the mean velocity, u) yields the ensemble-averaged momentum equations applied by FLUENT for predicting turbulent flows:

$$\begin{aligned} \frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \right. \\ \left. \left(\frac{2}{3} \mu \frac{\partial u_i}{\partial x_i} \right) \right) - \frac{\partial p}{\partial x_i} + \rho g_i + F_i + \frac{\partial(\rho u_i u_j)}{\partial x_j} \end{aligned} \quad (12)$$

Equation (12) has the same form as the fundamental momentum balance with velocities now representing time-averaged (or mean-flow) values and the effect of turbulence incorporated through the "Reynolds stresses", $\rho u'_i u'_j$. Note that $u'_i u'_j$ is a symmetric second order tensor since:

$$\overline{u_i u_j} = \overline{u_j u_i} \quad (13)$$

and hence has six unique terms. The main task of turbulence models is to provide expressions or closure models that allow the evaluation of these correlations in terms of mean flow quantities. The turbulence closure models used in FLUENT are:

$$B = \left(\frac{1}{r^2} \partial_{\theta} \mu_t \right) + \frac{2\mu_t}{r^2} \partial_{\theta}$$

Table2. Curvature-Related Source Terms in the RSM

ij	S _{ij}	D _{ij} Algebraic	D _{ij} Differential
xx	_____	_____	_____
Xr	$\rho \frac{v\theta}{r} v_x'v_{\theta}'$	$-v_x'v_r' \frac{\mu_t}{r^2}$	$-B v_x'v_{\theta}'$
xθ	$-\rho \frac{v\theta}{r} v_x'v_r'$	$-v_x'v_{\theta}' \frac{\mu_t}{r^2}$	$B v_x'v_r'$
rr	$2\rho \frac{v\theta}{r} v_r'v_{\theta}'$	$-2(v_r'^2 - v_{\theta}'^2) \frac{\mu_t}{r^2}$	$-2B v_r'v_{\theta}'$
rθ	$\rho \frac{v\theta}{r} (v_{\theta}'^2 - v_r'^2)$	$-4v_r'v_{\theta}' \frac{\mu_t}{r^2}$	$B (v_r'^2 - v_{\theta}'^2)$
θθ	$-2\rho \frac{v\theta}{r} v_r'v_{\theta}'$	$2(v_r'^2 - v_{\theta}'^2) \frac{\mu_t}{r^2}$	$2B v_r'v_{\theta}'$

1.3 Simulation Results for Gas Flow Pattern in the Combustor Chamber

The simulation results are shown in Figures 38 through 51. Figure 38 shows the gas velocity-vectors at five levels. K=6, 12, 18, 24, and 30, and the scaling factor to be equal to 1(dimensionless). The details of the velocity-vectors for the seven levels are shown in Figures 43 through 47 (Figure 43 for level 6, Figure 44 for level 12, Figure 45 for level 18, Figure 46 for level 24, and Figure 47 for level 30). Figure 39 shows the pressure profiles at six levels; K=6, 12, 18, 24, 30, and 33, and the scaling factor to be equal to 1 (dimensionless). The details of the pressure profiles for the six levels are shown in Figures 48 through 51 (Figure 48 for level 6, Figure 49 for level 12, Figure 50

Figure 38

Gas Velocity Vectors (feet/sec)
(K = 1, 6, 12, 24, 18, 24, 30, 33)

Figure 39 Gas Pressure Profile, Static Pressure (psi)
(K = 1, 6, 12, 18, 24, 30, 33)

Figure 40 Gas Velocity Vectors(feet/s) with Secondary Air Injection

Figure 41 Gas Pressure Profiles (psi)

Figure 42 Gas Velocity Vectors (feet/s)

Figure 43 Gas Velocity Vectors (feet/sec)
(K = 6)

Figure 44 Gas Velocity Vectors (feet/sec)
 (K = 12)

Figure 45 Gas Velocity Vectors (feet/sec)
(K = 18)

Figure 46 Gas Velocity Vectors (feet/sec)
 (K = 24)

Figure 47 Gas Velocity Vectors (feet/sec)
(K = 30)

Figure 48 Gas Pressure Profile (psi)
(K = 6)

Figure 49 Gas Pressure Profile (psi)
 (K = 12)

Figure 50 Gas Pressure Profile (psi)
(K = 18)

Figure 51 Gas Pressure Profile (psi)
(K = 24)

for level 18, Figure 51 for level 24). Figure 40 and Figure 42 show a 2-D velocity-vector in slide plates I=2 and I=22. The pressure profiles in the same 2-D slides are shown in Figure 41.

1.3.1 *The Flow Pattern*

At the bottom of the chamber, the velocity at the center is greater than it is nearer the wall region. At the secondary air injection levels, K=12 and K=24 (see Figure 49 and Figure 51), the outside velocity is greater than at the center. In the level between the two secondary air injectors, K=18 (see Figure 50), the outside velocity is reduced to less than the center velocity. At the top region (above the top secondary air injectors), the velocity increases from the wall to the center region.

At level K = 6 (see Figure 43), the velocity at the near wall is about 32 ft/s. By closing the center, it increases to 37 ft/s. However, in the center, it is reduced to about 20 ft/s.

At level K=12 (see Figure 49), the velocity of the secondary air nozzle outlet is 60 ft/s. When the air is injected into the chamber, it decreases rapidly but the whole chamber achieves a swirling flow. The swirling velocity is about 4 ft/s at J=6, radial position, and are reduced in both directions, to the wall and to the center of the chamber. In the center, however, and behind the nozzle, there are some lesser velocities (about 20 ft/s).

It is more interesting to note Figure 45 that shows the details of velocity-vector at the 18-inch level. At the level place, the highest velocity is about 49 ft/s at the center region and is reduced to about 34 ft/s at the near wall region (see the color scales).

However, the tangential velocity component is increased from the inner region to the outer region (see the arrow scales).

It is noted that velocity-vectors in Figure 40 and Figure 42 indicate the velocity profile in a vertical plate ($I=2$ and 22). Figure 40 is a front view of the profile, and Figure 42 is the side view of the profile. The profiles clearly show that the gas at the near wall region flows down to the bottom and flows up at the center region, and the velocity at the center region increases up along the axis of the combustor (see the arrow scales in Figure 40). Two larger tangential velocities are found at the secondary air injection levels (see arrow scales in Figure 42). The overall velocity in the chamber changes from 15.8 ft/s to 93.8 ft/s (see the color scales in the two Figures).

1.3.2 *The Pressure Profiles*

The overall static pressure profiles are shown in Figure 39 and Figure 41. The details for pressure profiles at each level are shown in Figures 48 through 51. In general, the pressure at the outer region near the wall is greater than that at the inner region near the axis; and the pressure at the bottom is greater than that of the top region. The pressure profiles for the vertical plate ($I=2$ and 22) are clearly shown in Figure 41. A negative pressure is formed at the top center region. The pressure is about -0.092 psi . We recognize that a lower pressure is always located at the chamber's center region and in the upper one. Since a higher velocity is always found at the center region, according to the Bernoulli equation, the increasing velocity is coming from pressure potential energy and is transferred into kinetic energy. It was noted that a higher-pressure zone is formed surrounding the secondary airflow into the center region, then a dead zone or a local

swirling flow was formed near the wall region, as shown in Figures 37 and 51.

1.4 *Numerical Simulations for Particle Trajectories*

1.4.1 *Numerical Simulations for Single Particle Injection*

An understanding of particle flow characteristics in the strongly swirling turbulent flow field is very important to control the particulate emissions and fuel burnout in the swirling fluidized bed combustor.

The single particle injection into the combustor chamber was simulated by the CFD code, FLUENT. The test conditions for the single particle injection are summarized in Table 1. Test conditions for the gas phase flow are the same as those shown in Table 3.

Table 3. Test Conditions of Single Particle Injection

Particle type	Glass beads
Size (mm)	0.04
Density (lb/ft ³)	156.05
Particle injection locations And initial velocity: (degree)	45
I (inch)	1
J (inch)	3
K at the fluid bed surface Particle injected velocity in K-direction (ft/s)	2.55

Since the swirling flow is a strong turbulent flow with anisotropic behaviors, the k- ϵ turbulence model is not suitable for this case. The Reynolds Stress Model (RSM), with a general algebraic expression, was selected and tested for the swirling turbulence flow simulation [17]. The gas density was determined by the universal gas law, which

takes the gas density as a function of pressure and temperature. The single particle injection simulation was conducted in the whole reactor chamber in cylinder coordinates. The simulation results for the single particle-moving trajectory, in the combustion chamber, are shown in Figures 52, 53, and 54. Figure 52 is the top view, Figure 53 is the side view, and Figure 54 is the isolated 3-dimension view. The particle trajectory showed that when the particle was injected from the surface of the fluidized bed, it swirled up. The swirling diameter increased as it rose.

Below the lower secondary air injection nozzle level, at about 8 inch levels, the particle moved toward the wall, bounced against the wall several times, then fell into the dense phase fluidized bed and finally escaped from the reaction region (see Figure 53 and Figure 54). Particles moved up spirally, but stayed closer to the wall due to a stronger centrifugal interaction. After they reached a certain height in the combustion chamber, they circulated around the wall, as shown in Figure 53. For a given flow condition, particles of certain diameters will be confined at an equilibrium height under the balance of gravity and drag force of up-flowing gas.

The particle moved in two stages: stage I from point A to point B1 is the ascending stage, stage II from point B1 to B17 is the colliding/bouncing stage, as shown in Figure 52. In the bouncing stage, the particle bounced on the wall seventeen times before it reached the bottom of the reactor chamber.

In summary, the basic flow pattern of the particles in the combustion chamber includes; (i) uprising spiral flow following the gas, (ii) horizontal circulation around the combustor wall, and (iii) slowly sliding flow at the bottom.

Figure 52 Top View of Particle Trajectory in the Combustor

Figure 53 Side View of Particle Trajectory in the Combustor

Figure 54 Three-Dimensional Plot of Particle Trajectory in the Combustor

1.4.2 Numerical Calculation and Basic Governing Equations

Equations (14) through (17) were used to calculate the particle trajectory. The numerical calculation predicts the trajectory of a dispersed phase particle (or droplet or bubble) by integrating the force balance on the particle, which is written in a Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the particle, and can be written (for the x-direction in Cartesian coordinates) as:

$$\frac{du_p}{dt} = F_D(u - u_p) + g_x(\rho_p - \rho / \rho_p) + F_x \quad (14)$$

where $F_D(u - u_p)$ is the drag force per unit particle mass and:

$$F_d = \frac{18\mu}{\rho_p D_p^2} \frac{CD Re}{24} \quad (15)$$

Here, u is the fluid phase velocity, u_p is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_p is the density of the particle, and D_p is the particle diameter.

Re stands for the relative Reynolds number, which is defined as:

$$Re = \frac{\rho D_p |u_p - u|}{\mu} \quad (16)$$

The drag coefficient, C_D , is a function of the relative Reynolds number of the following general form:

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (17)$$

where the a 's are constants that apply over several ranges of Re .

7. DESIGN/FABRICATION OF THE EXPLORATORY HOT MODEL

The exploratory hot model was designed under the assumptions that the understanding of swirling-flow combustion processes was purely based upon theoretical considerations and no relevant technical information except for gas-particle flow characteristics was existent.

Figure 55 shows the schematic diagram of the combustion chamber and secondary air nozzles. The combustion chamber is made of stainless steel cylinder of 20" height and 6" diameter, which includes 1" thickness of the refractory line of the inside chamber. Heat transfer surfaces, such as water cooling tube, are provided to remove the excess heat and control the combustor temperature to maintain the stable ignition and good burnout of fuels. In addition, heat transfer surfaces assist the turndown operation and turndown ratio.

For the exploratory hot model, the copper tube of 0.51, diameter was covered to the outside wall of the combustion chamber as the water-cooling tube. Three sections of independently controlled water-cooling tubes were arranged to identify the local heat transfer coefficient along the flow directions of the combustion chamber.

Eight secondary air nozzles are arranged with two different levels 6" and 12", respectively, from the bottom of the chamber as shown in Figure 55. Each level has four nozzles. All of the nozzles have 30-degree yaw angle to produce the tangential velocity. The combustion air will be tangentially injected to the chamber to form a strong swirling, recirculating, turbulent flow field. One of the major design features of the test chamber is strongly swirling flow, which is characterized by the swirl number. The calculated swirl number is 12 for this design.

Figure 55 Schematic Diagram of the Exploratory Hot Model

As shown in Figure 56, the gas distributor is designed by the flat perforated stainless steel plate. The hole size of the distributor is 0.45" with 152 holes. The gas distributor was fixed to the bottom flange with 3/8" bolts.

8. AUXILIARY SUBSYSTEMS OF THE EXPLORATORY HOT MODEL

(a) Air Supply Subsystem

The primary air for this hot model was supplied by an air blower. The blower's output is varied by changing the input voltage via a variable AC transformer. The primary air supply enters at the bottom of the combustion chamber through a wind box. The wind box is a small chamber with a perforated plate. Its purpose is to diffuse the air supply across the entire bottom of the combustion chamber.

The secondary air supply consists of two sets of four nozzles. The nozzles are entering the combustion chamber at 90 degrees. Each nozzle has a 45-degree angle cut on it. The air is supplied via the building at a pressure of 90 psi. Control valves can vary the air output. Two flow meters monitor the airflow in each of the two sets of nozzles.

(b) Water Supply Subsystem

The water circulation subsystem for removing excess heat from combustion consists of a series of three different copper coils, water connecting pipeline, control valve, and flow meter. Each of three coils has a control valve. These coils wrap around the outside of the chamber and provide surface cooling. The wastewater is deposited into a drain. A main control valve varies the water input, while each coil's valve controls the coil flow. A flow meter monitors the total water flow.

Figure 56 Schematic Diagram of the Gas Distributor

(c) Ignition Subsystem

Natural gas was used as the preliminary testing fuel. The ignition subsystem consists of regulators, flow meter, propane burner, fuel nozzle, and connecting tubing. The natural gas is supplied via the building at a pressure of 90 psi. The fuel nozzle was specially designed with varying configurations (i.e., one hole, four holes) to supply/burn the natural gas effectively in the chamber. A cone was added to the top of the fuel nozzle, which can protect the flame. The cone is constructed of a steel alloy approximately .015" thick and has a top diameter of 1.5" and a height of 1.5", as shown in Figure 57. It is attached to the fuel supply tubing with a small hose clamp.

9. SFBC HOT MODEL TESTING

(a) Preliminary Test Results and Discussions

According to the established safety and health guideline, the auxiliary subsystems were inspected carefully. All instruments are checked and calibrated for the preliminary test of exploratory hot model.

- Verify that all main control valves are closed. The water, secondary air, and gas valves are closed. Start the blower to supply primary air of the desired amount and mix with fuel (natural gas).
- Light the propane torch through a hole in the combustion chamber. Observe the flame in an angled mirror. Adjust control valves to the desired settings while constantly observing the combustor flame.

Figure 57 Schematic Diagram of Fuel Nozzle with a Cone

- According to the test plan, adjust the test parameters, such as primary/secondary airflow rate, and total excess air to the desired values.
- Allow the system to stabilize for a minimum of 10 minutes prior to recording any temperatures.
- After moving any thermocouple probe to measure temperature, let the temperature stabilize for a minimum of 5 minutes before recording data.

Temperature was measured via K-type thermocouples connected to a rotary switch and displayed on a digital display. Thermocouples were located in the holes in the chamber at distances of 8", 16", and 24.5" from the bottom of the combustor. The thermocouples' positions were varied radially within the chamber. Their radial positions used were the center, 1" from the center, 2" from the center, and 2.5" from the center. Thermocouples were also located at the top of the chamber to measure flue gas temperature.

Figure 58 shows a typical distribution of combustion temperatures at the different location of the combustor. The fuel nozzle is located at 8" from the bottom of chamber. Secondary flow rates for top and bottom regions are 4 CFM and 8.5 CFM, respectively. Table 4 shows detailed test conditions for this test. The averaged vertical temperature gradient along the longitudinal axis of the combustion chamber decreased.

Figures 59 and 60 show test results with the primary air on and secondary air nozzles at 90 degrees. Tables 5 and 6 show detailed test condition for each test. Results showed as before that the highest temperatures were obtained in the radial center of the chamber and at the 8" vertical location. Having the primary air at 100% and approximately 55%, respectively, lowered the radial center temperature by approximately

Table 4 Test Conditions

Hot Model Preliminary Test Data Conducted 6/14/96

Straight 4 hole Fuel Nozzle height at 8" from bottom

NO Primary air flow

FUEL: Natural Gas

Water flow rate 1.5GPM / all valves open

Secondary air top 4 cfm

Secondary air top 8.5 cfm

Cooling Water in 70.9 F

Cooling Water out 70.5 F

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	Temp (F) Surface	Avg temp
K1	8	630	1131	1144	444	360	837.25
K2	16	464	606	583	281	254	483.5
K3	24.5	523	530	500	195.8	205	437.2
	avg 3 therm.cpls	539.00	755.67	742.33	306.93		585.98

Table 5 Test Conditions

Hot Model Preliminary Test Data Conducted 6/28/96 B

With Primary Air 1.7 in H₂O

Straight 5 hole Fuel Nozzle at 5"

FUEL: Natural Gas Fully open

Water flow rate All coils @ 100%

secondary nozzle angle 90

Secondary air top 11 cfm

Secondary air top 11 cfm

Cooling Water in 71.2

Cooling Water out 71.2

Cooling water flow 1.5 gpm

Flue Temp 216

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	Temp (F) Surface	Avg temp
K1	8	680	320	109	102	87.5	302.75
K2	16	270	135	126	105	87.5	159
K3	24.5	240	191	141	127.7	110	174.925
	avg 3 therm.cpls	396.67	215.33	125.33	111.57		212.23

Table 6 Test Conditions

Hot Model Preliminary Test Data Conducted 6/28/96 C

With Primary Air .68 to .85 in H2O

Straight 5 hole Fuel Nozzle at 5"

FUEL: Natural Gas Fully open

Water flow rate 1.5 gpm All coils @ 100%

secondary nozzle angle 90

Secondary air top 11 cfm

Secondary air top 11 cfm

Cooling Water in 71.2

Cooling Water out 71.2

Cooling water flow 1.5 gpm

Flue Temp 216

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	Temp (F) Surface	Avg temp
K1	8	660	440	143	108.5	89.6	337.875
K2	16	263	207	165.1	129	92.6	191.025
K3	24.5	284	226.3	176.9	166	112.6	213.3
	avg 3 therm.cpls	402.33	291.10	161.67	134.50		247.40

Figure 58 Test 6/14/96

Test 6/28/96 B

Figure 59

Test 6/28/96 C

Figure 60

900 °F. This could be a result of too high primary air velocity with fully opened exit of the combustor.

Another three tests were conducted at a primary flow of approximately 45% (0.6 in. H₂O). The ratio of the secondary air was the varying factor. The range of flow rate at both the top and the bottom was 5 CFM to 10 CFM. Test A of Figure 61 was exactly opposite to Test B of Figure 62. The detailed test conditions were shown in Figures 61 to 63. Varying the secondary airflow rate as in Tests B and C had no appreciable effect in either the combustor temperature or inflame stability or color. The significant effect seen when the three tests are compared is the increase in combustor temperature due to the decreased airflow. Tests B and C with a secondary air flow of 15 CFM averaged 271.95°F and 275.8°F, respectively, while, test 1 with a 20 CFM secondary airflow has an average temperature of 254.8°F, as shown in Figures 61 to 63. The least airflow yields the highest combustor temperatures.

Test D of Figure 64 was conducted to determine the conditions at the lowest combined primary and secondary airflow. The average temperature was 530° F. Primary air versus secondary air shows the relationship between the amount of primary and secondary air versus average combustor temperature. The more primary air injected into the system lowered the combustor temperatures. The amount of primary air had a greater influence than did the secondary air.

(b) Proof-of-concept Test

(1) *Instrumentation for Flue Gas Composition Measurements*

Figure 61 Test Conditions and Temperature Profile in Combustion Chamber

Hot Model Preliminary Test Data

With Primary Air 0.6 in H2O

Straight 5 hole Fuel Nozzle at 5"

FUEL: Natural Gas Fully open

Water flow rate 1.5 GPM All coils @ 100%

secondary nozzle angle 90

Horizontal angle 45

Secondary air top (CFM) 10.5

Secondary air top (CFM) 10.5

Cooling Water in 71

Cooling Water out 71.2

Cooling water flow (GPM) 1.5

Flue Temp 249.6

Dist:(in) from Bottom Center 1" from center 2" from center 2.5" from center

8 550(F) 515 121.6 102.7

16 331.7 257 160.9 129

24.5 277.6 245 199.5 167.7

avg 3 therm.cpls 386.433 339.000 160.667 133.133

Surface Avg temp

94.2 322.33

98 219.65

114.6 222.45

Figure 62 Test Conditions and Temperature Profile in Combustion Chamber

Figure 63 Test Conditions and Temperature Profile in Combustion Chamber

Figure 64 Test Conditions and Temperature Profile in Combustion Chamber

A computer-assisted data acquisition system for the flue gas compositions / stack temperature measurements was employed to accelerate the data taking process and to eliminate human errors.

The ENERAC 2000 gas analyzer was used to measure the composition of flue gases. This analyzer is designed to have an automatic self-calibrating system that can sample, condition, and measure oxides of nitrogen, carbon monoxide, sulfur dioxide, and oxygen on a continuous basis [18].

The ENERAC 2000 gas analyzer interfaced with an on-site personal computer with software, ENERCOMP. This software includes a program that allows us to translate stored stack data into LOTUS program format for further manipulation. To take the average values, set the Log-In period of ENERCOMP program to the shorter average period.

(2) Procedures of Flue Gas Composition Measurements

Measurements of combustion products by the gas analyzer start at the sampling probe that is inserted into the outlet of the exhaust tube of the exploratory hot model. The sampled flue gases are transported to the monitor console of the gas analyzer.

(3) Effect of Cooling Water

Figure 65 shows one of the system test results to analyze the thermal performance on the exploratory hot model. This test was conducted with the top combustor plate sealed with a ¼" thickness of the steel plate.

The data shows an average temperature of 657 °F and a close to homogeneous heat distribution. This average temperature is higher than that of the test at the lowest combined primary and secondary airflow. This result indicated that the exhaust location

Figure 65 Test Conditions and Temperature Profile in Combustion Chamber

Hot Model Preliminary Test Data
 With Primary Air (in of H₂O) 0.05 (20 cfm)
 Straight 5 hole Fuel Nozzle at 5"
 FUEL: Natural Gas Fully open
 Water flow rate 1.5 GPM All coils @ 100%
 secondary nozzle angle 90
 Horizontal angle 45
 Secondary air top (CFM) 0
 Secondary air top (CFM) 5
 Cooling Water in 70.8
 Cooling Water out 75.7
 Cooling water flow (GPM) 1.5
 Flue Temp 900?

Dist. (in) from Bottom	Center	1" from center	2" from center	2.5" from center
8	1380 (F)	1051	720	539
16	574	583	564	599
24.5	420.9	415	416	617
avg 3 therm. cpls	791.633	683.000	566.667	585.000

Sealed combustor top
 Exhaust size 5/8 "
 Exhaust Location 5.5" from bottom

Surface Avg temp	186	922.5
	185	580
	184	467.23
656.575 Total Average		

influenced the mixing of the chamber gas and heat similarly. The vertical location of the exhaust was very close to the fuel nozzle, which also affected the increasing of the combustion temperature.

Two tests were conducted to examine the effect of cooling water using natural gas. The gas analyzer, ENERAC 2000, was used to measure the composition of flue gases. Test A was run on the exploratory hot model with the cooling water. The cooling water flow rate was 1.5 gallon/min. For the three levels of the cooling water tubes. These cooling tubes consisted of three runs of independently controlled water cooling tubes. The secondary airflow rates for the upper and lower nozzles are 3 CFM and 3.5 CFM, respectively. The average combustion temperature was 326.8 °F. The detailed test conditions and results were shown in Table 7.

Table 8 summarized the test results without the cooling water. The secondary airflow rates were same as the test with the cooling water. The average combustion temperature was 301.4 °F. The detailed test conditions and results were shown in Table 8. Without cooling water, overall combustion temperatures were increased. Especially, the average temperature at the center of the combustor was increased by a range of 27%. When the cooling water was provided, the combustion temperatures decreased. In addition, the top portion of the combustor absorbed the least amount of heat from the hot combustion gases.

It is believed that the heat removal rate by water at each section of heat transfer surface affects the local heat transfer rate in the combustion chamber [19].

Table 7 Summary of Test Conditions (With Cooling Water) (Test A)

Hot Model Preliminary Test Data
 With Primary Air 0.40 in (55 cfm)
 Straight 5 hole Fuel Nozzle at 5" Fully open
 FUEL: Natural Gas
 Water flow rate 1.5 gpm All coils @ 100%
 secondary nozzle angle 90
 Horizontal angle 45
 Secondary air top 3 cfm
 Secondary air bottom 3.5 cfm
 Cooling Water in 46.5
 Cooling Water out 47.7
 Cooling water flow 1.5 GPM
 Flue Temp 235.7 238.1 233.4 233.6 235.2 AVG

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	Temp (F) Surface	Avg temp
K1	8	255.1	250.2	228.9	209.8	98.9	236.0
K2	16	297.8	278.5	230.2	196.2	104.1	250.7
K3	24.5	721	601.8	196	151.6	117.8	417.6
°	avg 3 therm.cpls	424.63	376.83	218.37	185.87	106.93	301.4

Test performed with dome on top & fixed analyzer position

Table 8 Summary of Test Conditions (Without Cooling Water) (Test B)

Hot Model Preliminary Test Data
 With Primary Air 0.40 in (55 cfm)
 Straight 5 hole Fuel Nozzle at 5"
 FUEL: Natural Gas Fully open
 Water flow rate none All coils @ 0%
 secondary nozzle angle 90
 Horizontal angle 45
 Secondary air top 3 cfm
 Secondary air bottom 3.5 cfm
 Cooling Water in n/a
 Cooling Water out n/a
 Cooling water flow 0
 Flue Temp 235.4 234.6 233.6 233.9 234.375 AVG

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	Temp (F) Surface	Avg temp
K1	8	255.8	244.4	234	205.4	103	234.9
K2	16	301	279.8	219.7	191.3	107.3	248.0
K3	24.5	1065	587	171.6	166.7	117	497.6
	avg 3 therm.cpls	540.60	370.40	208.43	187.80	109.10	326.8

Test performed with dome on top & fixed analyzer position

(4) Combustion Products and Pollution Performance

Figures 66 and 67 show the concentration of oxygen, carbon dioxide, carbon monoxide, oxides of nitrogen, and sulfur dioxide. The sampling was collected from the exhaust at the top portion of the combustor. The carbon monoxide was found to be about 14% to 20%. As expected, the CO and NO_x levels were very low (0 ppm-0.3 ppm) because the fuel was gas, which is a clean fuel. Natural gas is practically free from noncombustible gas or solid residue.

Methane (CH₄) and ethane (C₂H₆) are the principal combustible components of natural gas. The existing SO_x level was found to be relatively low (13 ppm to 16 ppm). It may be considered that the sampling gas included some saturated moisture. Usually, natural gas that is delivered from a pipeline has often been "rehydrated," that is saturated with water vapor by means of steam jet.

(5) Modification of the Exploratory Hot Model

Based upon the data and operating experience obtained from exploratory hot model tests, the exploratory combustor model was modified to explore the operational limits, fuel flexibility, and the role of heat transfer in combustion control.

- **Design of Air Injection Nozzles**

Air injection nozzles were originally designed with a 90 degree input angle and yaw, which could be varied. However, varying roll was difficult. One would have to place their arm inside the chamber and manipulate the roll by hand.

Eight air injection nozzles were incorporated into the new design. Four nozzles were set at a 30-degree angle and the remaining four at a 60-degree angle. These nozzles

Figure 66 Stack Temperatures vs Combustion Gas Analyzer

Figure 67 Stack Temperatures vs Combustion Gas Analyzer

can be rotated to a 360-degree direction. The nozzle is approximately 1" in length with a tapered outlet with a 0.125" diameter, as shown in Figure 68. Round bar stocks with a 0.125" diameter and a measured length of 1/16" were incorporated into the nozzle placement. These stocks were attached and held in place with coins acting as a sliding lock. This coin was drilled and tapped into place, as shown in Figure 69. In addition, an eccentric locking disc positioned the nozzles at the correct depth in the chamber wall. High temperature tubing attached the nozzles to the air supply.

- Design of Water Cooling Tube

Heat transfer surfaces, such as water jacket or water cooling tube, are provided to remove the excess heat and control for stable ignition and good burnout in the combustor. For our hot model, three runs of independently controllable water-cooling tubes with 0.25" diameter copper tube wound on the outer wall for studying the local heat transfer characteristics along the height of the combustor.

- Design of Fuel Injection Nozzle

The atomization quality and condition of a fuel nozzle significantly affect the ignitability, flame stability, and combustion efficiency of the fuel. Type B is carefully designed to atomize fuel effectively at a 45-degree angle from the vertical direction, as shown in Figure 70. Eight holes were drilled to keep a stable flame.

- Design of Igniter System

The igniter system was designed with a safe and dependable ignition of natural gas for the combustion test. Electrodes were placed over the fuel nozzle path. The igniter push button was attached to the ground screw, as shown in Figure 71. Whenever the igniter button was pushed, the spark appeared in the collector box. This igniter system

Air Injection Nozzles

Figure 68 Schematic Diagram of Air Injection Nozzles (Type B)

NOZZLE AND STOPPER SUB-ASSEMBLY

Figure 69 Schematic Diagram of Nozzle and Sub-Assembly System

Figure 70 Schematic Diagram of Fuel Injection Nozzle

Figure 71 Schematic Diagram of Igniter System

could ignite within 3 seconds.

- **Design of Combustion Chamber**

The combustion chamber is made from the carbon steel cylinder of 30" height, 6.625" outer diameter, and 6" inner diameter. A 0.5"(thickness) refractory coating was installed inside the combustion chamber. Eight holes of secondary air input are arranged with two different levels 11" and 19", respectively, from the bottom of the combustion chamber. In addition, three holes were cut for the temperature measurements at different levels 11", 19", and 27", respectively, from the bottom of the combustion chamber.

(c) Results and Discussions

Two different tests were analyzed to understand the thermal performance on exploratory hot model under the exact same condition with the exception of the amount of fuel.

For Test A, fuel (natural gas) flow rate was 19.5 cubic feet per hour (CFH), which is almost two times higher than that of Test B. The detailed test conditions are shown in Table 9. The average cooling water flow rate was 1.6 gallon per minute (GPM). Temperature was measured at different locations of the combustor chamber. As shown in Table 9, the change of temperature was decreased along the increase of the distance from the bottom of the combustor chamber.

For Test B, fuel flow rate was reduced to 10 CFH, which was almost half of Test A. The average combustion gas temperature decreased from 1394 °F to 1015 °F while the fuel flow rate decreased, as shown in Tables 9 and 10. When examining the data there is only 6% difference of temperature at the 8" thermocouple location from the bottom of

Table 9 Summary of Hot Model Test (Case A)

Primary Air Pressure (inH2O)	0			
Primary Air Flow (CFM)	0.00			
FUEL: Natural Gas flow (CFH)	19.5		Maximum	
Fuel Nozzle (description)	5 hole nozzle 5" from chamber floor			
Sec. Nozzle Yaw Angle	TOP	90	BOTTOM	90
Sec. Nozzle Roll Angle	TOP	45	BOTTOM	0
Secondary Air Flow (CFM)	TOP	2	BOTTOM	2
Total Air Flow (%)	TOP	50	BOTTOM	50
			PRIMARY	0
Cooling Water Temp (F)	IN	52.7	OUT	66.9
Cooling water flow(GPM)	1.6			
Water valve opening (%)	TOP	10	MIDDLE	10
			BOTTOM	10
LENGTH OF BURN-IN	50 min			

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	AVG TEMP (F)	Temp (F) Surface	FLUE TEMP
K1	8	1560	1857	1777	1712	1726.50	442	1012.0
K2	16	1475	1374	1551	1179	1394.75	437	1001.0
K3	24.5	1200	1196	1124	723	1060.75	440	1008.0
AVG	TEMP (F)	1411.67	1475.67	1484.00	1204.67	1394.00	439.67	1007

Enerac 2000 Gas Analysis

Position	Stk temp (F)	Amb temp (F)	Efficiency (%)	O2(%)	CO2 (%)	CO(ppm)
Flue gas analysis	723	79	48.8	15.2	3.3	33

Table 10 Summary of Hot Model Test (Case B)

Primary Air Pressure (inH2O)	0	
Primary Air Flow (CFM)	0.00	
FUEL: Natural Gas flow (CFH)	10	Maximum
Fuel Nozzle (description)	5 hole nozzle 5" from chamber floor	
Sec. Nozzle Yaw Angle	TOP 90	BOTTOM 90
Sec. Nozzle Roll Angle	TOP 45	BOTTOM 0
Secondary Air Flow (CFM)	TOP 2	BOTTOM 2
Total Air Flow (%)	TOP 50	BOTTOM 50 PRIMARY 0
Cooling Water Temp (F)	IN 73.3	OUT 75.5
Cooling water flow(GPM)	1.6	
Water valve opening (%)	TOP 100	MIDDLE 100 BOTTOM 100
LENGTH OF BURN-IN	55 min	

Thermocouple No.	Distance (in) From Bottom	Temp (F) Center	Temp (F) 1" from center	Temp (F) 2" from center	Temp (F) 2.5" from center	AVG TEMP (F)	Temp (F) Surface	FLUE TEMP
K1	8	1475	1123	1721	1406	1431.26	273.7	724.8
K2	16	928	905	880	774	871.75	264.3	733.5
K3	24.5	806.5	848	762	557	743.38	259	732.4
AVG TEMP (F)		1069.83	958.67	1121.00	912.33	1015.46	265.67	730.23

Enerac 2000 Gas Analysis

Position	Stk temp (F)	Amb temp (F)	Efficiency (%)	O2(%)	CO2 (%)	CO(ppm)
Flue gas analysis	723	79	48.8	15.2	3.3	33

combustor chamber. However, the 16" thermocouple location exhibits a temperature on the magnitude of 62.48%. The 24" thermocouple location exhibits a temperature difference on the magnitude of 45.76%. The flue gas temperature has a 65.5% decrease, as shown in Test A vs. Test B. When decreasing the fuel flow rate, the overall temperature decreases.

(1) Heat Balance Calculation Results

The heat balance calculations are summarized as follows;

1) The first law of thermodynamics (Energy Balance) .

The flow enthalpy increasing is equal to the sum of total heat exchange and mechanical works which done by the system.

$$H_i + Q_i - (H_o + Q_o) = W \quad (1)$$

$$H_o - H_i = Q_r - Q_w - Q_l + W \quad (2)$$

2) For the combustion system the mechanical work is zero

$$W = 0 \quad (3)$$

3) The flow enthalpy is defined as:

For single flow component

$$H_j = Cp_j * \rho_j * q_j * \Delta T_j \quad (4)$$

For mixture of flow components

$$H = \sum_{j=1}^k m_j H_j \quad (5)$$

4) Flow density, ρ

For water:

$$\rho_w = 1000 \text{ kg} / \text{m}^3 \quad (6)$$

For gases:

The gas density is a function of gas temperature and pressure

$$\rho_{gas} = \frac{M_{gas}}{22.4} \left(\frac{T^o}{T_{gas}} \right) \left(\frac{P_{gas}}{P^o} \right) \quad (7)$$

5) Heat capacity, Cp

For Water:

$$Cp_w = 4.18 kJ / kg.^{\circ}C \quad (8)$$

For Gases:

The gas heat capacity is a function of gas temperature.

$$Cp / R = a + bT + cT^2 + dT^3 + eT^4 \quad (9)$$

There a, b, c, d, e is constant values for each gas components and shown in the following table.

	a	b	c	d	e
CO ₂	2.401	8.735e-3	-6.607e-6	2.002e-9	0
H ₂ O	4.07	-1.108e-3	4.152e-6	-2.964E-9	8.07E-13
N ₂	3.675	-1.208e-3	2.324e-6	-6.32e-10	-2.26e-13
CH ₄	3.826	-3.979e-3	2.456e-5	-2.273e-8	6.963e-12
air	3.653	-1.337e-3	3.294e-6	-1.913e-9	2.76e-13

6) The average heat transfer coefficient from hot gas to the cooling water can be estimated by using the following equation:

$$h_{g-w} = \frac{Q_w}{A(T_g - (\frac{\Delta T_w}{2}))} \quad (10A)$$

7) The average heat transfer coefficient from hot gas to the environment at room temperature can be estimated by using the flowing equation [20]:

$$h_{g-L} = \frac{Q_L}{A(T_g - (\frac{\Delta T_w}{2}))} \quad (10B)$$

(A) Heat Balance Calculation Results for Case A:

Based on 1 minute of time period. The fuel is natural gas (95% of CH₄)

Fuel combustion heat, Q _r :	468 kJ
Fuel input enthalpy, H _f :	0.355 kJ
Air input enthalpy, H _a :	3.573 kJ
The input flow enthalpy, H _f :	3.928 kJ
Flue gas enthalpy, H _o :	92.71 kJ
Heat loss from cooling water, Q _w :	201.63 kJ

Heat loss from the reactor wall, Q_L can be calculated using the equation (2):

$$Q_L = Q_r - Q_w - (H_o - H_i) = 177.58(KJ) \quad (11)$$

The overall average heat transfer coefficient from hot gas to the cooling water,

$$h_{g-w} = 7.28 \text{ w/m}^2\text{C}$$

The overall average heat transfer coefficient from hot gas to the cooling water,

$$h_{g-l} = 6.497 \text{ w/m}^2\text{C}$$

The flame enthalpy and flame heat loss changing along the reactor height are shown in the following table.

Distance from the bottom (inch)	Flame enthalpy (kJ)	Flame heat loss (kJ)
8	173.34	37.51
16	135.83	37.3
24	98.53	5.82

The dimensionless height based on the reactor height, H , for the three distance from the bottom are 0.28, 0.55, and 0.83. The Flame enthalpy and flame heat loss as a function of the dimensionless height are shown in Figure 72 for case A.

(B) Heat Balance Calculation Results for Case B:

Based on 1 minute of time period. The fuel is natural gas (95% of CH_4)

Fuel combustion heat, Q_r :	240.08 kJ
Fuel input enthalpy, H_f :	0.1823 kJ
Air input enthalpy, H_a :	3.573 kJ
The input flow enthalpy, H_i :	3.755 kJ
Flue gas enthalpy, H_o :	59.55 kJ
Heat loss from cooling water, Q_w :	31.24 kJ

Heat loss from the reactor wall, Q_L can be calculated using the equation(2):

$$Q_L = Q_r - Q_w - (H_o - H_i) = 153.02(\text{kJ})$$

The overall average heat transfer coefficient from hot gas to the cooling water,

$$h_{g-w} = 0.634 \text{ w/m}^2\text{C}$$

The overall average heat transfer coefficient from hot gas to the cooling water,

$$h_{g-L} = 7.854 \text{ w/m}^2\text{C}$$

The flame enthalpy and flame heat loss changing along the reactor height are shown in the following table:

Distance from the bottom (inch)	Flame enthalpy (kJ)	Flame heat loss (kJ)
8	129.82	65.8
16	73.03	13.48
24	60.8	1.25

The dimensionless height based on the reactor height, H , for the three distance from the bottom are 0.28, 0.55, and 0.83. The Flame enthalpy and flame heat loss as a function of the dimensionless height are shown in Figure 73 for case B.

Based on the heat balance calculation results for both case A and case B, the heat loss from the reactor wall to the environment is a great portion of the total heat transfer. For the case A, it is about 47 percent of the total heat loss; for the case B, it is about 83 percent of the total heat loss. In order to reduce the heat loss from reactor wall to the environment, it is necessary to increase the water cooling coil surface area to cover more of the reactor wall. The heat loss into the cooling water for case A is about two times larger than that of case B, since the fuel input for case B is about half of the fuel injected for case A. The enthalpy of the flame at the reactor center is changing along the reactor height that was measured at the three height levels from the reactor bottom, 8 inch, 16 inch, and 24 inch. The flame enthalpy can be used to estimate the flame heat losses in combustion chamber that may caused by the gas mixing process. They are 37.5 kJ, 37.3 kJ, and 5.82 kJ for case A; and 65.8 kJ, 13.48 kJ, and 1.25 kJ for case B, as shown in Figures 72 and 73. It is believed that the better gas mixture was achieved for case A.

For case A, the overall average heat transfer coefficients are $7.28 \text{ w/m}^2\text{C}$ from hot gas to the cooling water, and $6.497 \text{ w/m}^2\text{C}$ from hot gas to the environment. For case B, the overall average heat transfer coefficients are $0.634 \text{ w/m}^2\text{C}$ from hot gas to the cooling water, and $7.854 \text{ w/m}^2\text{C}$ from hot gas to the environment. Comparing case A and case B, the overall heat transfer coefficient from hot gas to the cooling water decreased. The heat transfer coefficient from hot gas to the environment increased as the flame average temperature increased.

(2) Thermal Analysis and Heat Transfer Effect

The combustion test result was analyzed to understand the thermal performance and heat transfer characteristics on the modified exploratory hot model. The fuel flow rate was 21 cubic per hour (cfh). The secondary air was provided evenly, 2 cubic feet per minute (cfm) for upper and lower levels. The cooling water flow rates for the three different sections of the heat transfer surface were 0.81, 0.73, and 0.2 gallon per minute from the bottom to the top surface of the combustor chamber. The cooling water inlet temperature was 71°F . The average gas combustion gas temperature was 1209°F . The flue gas temperature was 843°F .

The results of heat balance calculation are summarized as follows:

Based on 1 minute of time period. The fuel is natural gas (95% of CH_4)

Fuel combustion heat, Q_r :	328.07 kJ
Fuel input enthalpy, H_f :	0.3828 kJ
Air input enthalpy, H_a :	3.573 kJ
The input flow enthalpy, H_i :	3.955 kJ
Flue gas enthalpy, H_o :	76.17 kJ

Heat loss from cooling water, Q_w :

Bottom Section	44.57 kJ
Middle Section	70.61 kJ
Top Section	14.2 kJ

Heat loss from the reactor wall, Q_L , can be calculated using the equation (2):

$$Q_L = Q_r - Q_w - (H_o - H_i) = 127.7 \text{ (kJ)}$$

The local average heat transfer coefficient from hot gas to the cooling water, h_{g-w} :

Bottom Section	10.15 w/m ² C
Middle Section	16.11 w/m ² C
Top Section	3.23 w/m ² C

The overall average heat transfer coefficient from hot gas to the environment,

$$H_{g-L}: \quad 6.81 \text{ w/m}^2\text{C}$$

The flame enthalpy and flame heat loss changing along the reactor height.

Distance from the bottom (inch)	Flame enthalpy (kJ)	Flame heat loss (kJ)
8	176.14	21.67
16	154.47	35.72
24	118.75	42.58

The dimensionless height based upon the combustor height, H are 0.28, 0.55, and 0.83. The flame heat loss increased along the combustor height. However, the flame enthalpy decreased, as shown Figure 74. It is believed that the changes of heat loss/flame enthalpy depended on the combustion temperature and location.

The axial variation of heat transfer coefficient along the combustor chamber is shown in Figure 75. The heat is removed by the cooling water at different zones during the combustion test. The top portion of combustor absorbed less heat from hot combustion gases. The heat transfer coefficient is generally lower in the top area than in the bottom of the combustor, as shown in Figure 75.

(3) Effect of the Secondary Air Flow Rate and Heat Transfer Coefficient

Two different tests were analyzed to understand the thermal performance of the exploratory hot model. For Test 1, fuel (natural gas) flow rate was 25 cubic feet per hour (cfh). The primary air flow rate was 2.9 cubic feet minute (cfm). The secondary air flow rates for the top and the bottom levels were both 1 cfm. The cooling water flow rate was 0.5 gallon per minute (gpm). The detailed test conditions are shown in Table 11. For Test 2, the test conditions were the same as Test 1 except the secondary air flow rates, as shown in Table 12. The secondary air flow rates for the top and the bottom levels were 1.65 cfm and 1.1 cfm, respectively.

For Test 1, the ratio of the top to the bottom secondary air flow rate was chosen to be 1. The average combustion temperatures for the upper part, middle part, and bottom part were 1347.5 °F, 1150 °F, and 1419 °F.

For Test 2, the ratio of the top to the bottom secondary air flow rate was increased by 1 to 2. The average combustion temperatures for the upper part, middle part, and bottom part were 1414 °F, 1347 °F, 1372 °F. These results indicate that the average combustion temperature of Test 2 is higher than that of Test 1. The secondary air flow rate may be a very sensitive factor to the combustion temperature profile.

Table 11 Summary of Hot Model Test Result (1)

Hot Model Preliminary Test Data

Primary Air Flow (CFM)	2.9	Room Temp. (F)	74
		Air Temp. (F)	74
		Fuel Temp. (F)	74
Fuel: Natural Gas Flow (CFH)	25	Primary (bottom)	16
Fuel Nozzle (Description)	5 holes nozzle 5" from chamber floor	Secondary (top)	9
Sec.Nozzle Yaw Angle	Top	Middle in	30°
Sec.Nozzle Roll Angle	Top	Bottom	30°
Secondary Air Flow (CFM)	Top	Bottom	1
Total Air Flow (%)	Top (Sec.) 20.408	Bottom (Sec.) 20.408	Primary 59.184
Cooling Water Temp. (F)	Top in 68	Out	Cooling Water Flow Rate (GPM) 0.5
	Middle in 68	Out	Cooling Water Flow Rate (GPM) 0.5
	Bottom in 68	Out	Cooling Water Flow Rate (GPM) 0.5
Length of Burn-in	30 min		

Thermocouple No.	Distance (in)	Temp (F)	Temp (F)	Temp (F)	Temp (F)	Temp (F)	Temp (F)	Flue Temp
K1	From Bottom	Center	1" from center	2" from center	2.5 from center	Average	Surface	
K2	8"	1550	1400	1480	1320	1437.5	180	903
K3	12"	1570	1470	1310	1250	1400	182	940
K4	16"	1490	1250	1025	920	1171.25	174	950
K5	20"	1320	1200	1080	940	1135	215	890
K6	24"	1560	1490	1320	1200	1392.5	236	905
	28"	1410	1390	1250	1160	1302.5	300	934
AVG	Temp (F)	1483.3	1366.7	1244.2	1131.7	1306.5	214.5	920.3

Enerac 2000 Gas Analysis

Position	Stk Temp (F)	Amb Temp (F)	Efficiency (%)	O2 (%)	CO2 (%)	CO (ppm)
Flue Gas Analysis	919	79	90.2	1.9	10.8	0

Table 12 Summary of Hot Model Test Result (2)

Hot Model Preliminary Test Data

Primary Air Flow (CFM)	3	Room Temp. (F)	74
		Air Temp. (F)	74
		Fuel Temp. (F)	74
Fuel: Natural Gas Flow (CFH)			
Fuel Nozzle (Description)	25	Primary (bottom)	16
	5 holes nozzle 5" from chamber floor		Secondary (top) 9
Sec.Nozzle Yaw Angle	30°	Middle in	30°
Sec.Nozzle Roll Angle	30°	Bottom	30°
Secondary Air Flow (CFM)	1.65	Bottom	1.1
Total Air Flow (%)	28.696	Bottom (Sec.)	19.13
		Primary	52.174
Cooling Water Temp. (F)	68	Out	92
	68	Out	88
	68	Out	85
Cooling Water Flow Rate (GPM)			0.5
			0.5
			0.5
Length of Burn-in	30 min		

Thermocouple No.	Distance (in)	Temp (F)	Temp (F)	Temp (F)	Temp (F)	Temp (F)	Flue Temp
K1	8"	Center	1" from center	2" from center	2.5 from center	Average	Surface
K2	12"	1550	1450	1250	1030	1320	166
K3	16"	1670	1570	1320	1130	1422.5	178
K4	20"	1530	1430	1200	1040	1300	176
K5	24"	1570	1450	1300	1250	1392.5	270
K6	28"	1620	1540	1370	1280	1452.5	270
AVG	Temp (F)	1540	1440	1320	1200	1375	290
		1580	1480	1293.3	1155	1377.1	225
							959.8

Enerac 2000 Gas Analysis

Position	Slk Temp (F)	Amb Temp (F)	Efficiency (%)	O2 (%)	CO2 (%)	CO (ppm)
Flue Gas Analysis	924	79	91.5	2.3	10.5	0

Once the secondary air flow rate is fixed to be 1 cfm (Test 1), the swirling flame in the upper/middle part of the chamber was observed as the same shape. The flame at the bottom part of the combustor was smaller than that of the upper or the middle part of the combustor.

When the secondary air flow rate was increased by 1 cfm to 2 cfm (Test 2), the swirling flame in the upper part was much stronger than that of the middle or the bottom part of the combustor. This flame was dominant in the combustor chamber which affected the combustion temperature profile. So, the large ratio of the secondary air flow rate causes the increase in the combustion temperatures.

The heat transfer coefficients in the combustion chamber were computed by our developed C program. This computer program was based upon the energy/heat balance equations. Figure 76 shows the heat transfer coefficient changes along the combustor height for Tests 1 and 2. For the fixed ratio of the secondary air flow rate (Test 1), the heat transfer coefficients at the top and bottom parts of the combustor are $23.7 \text{ w/m}^2\text{C}$ and $21.3 \text{ w/m}^2\text{C}$, respectively. The swirling/vortex flame in the middle section of the combustion chamber was produced by the top/bottom second swirling flow. So, it causes the highest heat transfer coefficient in the middle section of the combustion chamber, as shown in Figure 76.

For the increased ratio of the secondary air flow rate (Test 2), the heat transfer coefficients at the top and bottoms are $25.7 \text{ w/m}^2\text{C}$ and $18.8 \text{ w/m}^2\text{C}$. The swirling/vortex flame in the top section was dominant in the combustion chamber. So, it affects the higher heat transfer in the top section of the combustion chamber.

Figure 76 Local Heat Transfer Coefficients for the Three Sections in the Combustion Chamber

10. NUMERICAL MODELING AND SIMULATION FOR THE EXPLORATORY HOT MODEL

(a) Overall Description of Numerical Modeling and Simulation

Numerical modeling/simulation of gas-particle flows, heat transfer, and combustion process in the fluidized bed combustor has gradually increased with the development of modern computers. Numerical simulation has also been recognized as a powerful tool for design verification and operational guidance for the fluidized bed combustors. The successful simulation work may significantly reduce the efforts in the experimental study.

The purpose of the numerical modeling/simulation on the advanced swirling fluidized bed combustor (hot model) is to determine the hot flow patterns, velocity profiles, static pressure profiles, species concentration profiles, and temperature profiles in the combustor chamber.

The basic governing equations for swirling, turbulent gas-particle flows and combustion in the swirling fluidized bed combustor can be formulated based upon the continuity and momentum equations, and energy equation in the cylindrical coordinates. The continuity and three direction momentum equations were introduced in the cold flow modeling/simulation.

Energy conservation equation is as follows:

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho u_i h)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(k \frac{\partial T}{\partial x_i} \right) - \frac{\partial}{\partial x_i} \sum h_i J_i + \frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} + \tau_{ij} \frac{\partial u_i}{\partial x_j} + S_h$$

Where T is the temperature, l is the heat flux of species, and k is the mixture thermal conductivity. S_h is a source term that includes sources of enthalpy due to a

chemical reaction (combustion reaction) and radiation heat exchange between the gas and the wall. The h , is the static enthalpy which is defined as:

$$h = \sum m_i h_i$$

The conservation of species 1 is determined by:

$$\frac{\partial}{\partial t}(\rho m_i) + \frac{\partial}{\partial x_i}(\rho u_i m_i) = \frac{\partial}{\partial x_i}(J_{i,i}) + S_i$$

Where m_i is the mass fraction of species 1, $J_{i,i}$ is the diffusive mass flux of species 1 in the i th direction and S_i is the net rate of production of species 1 per unit volume due to the chemical reaction.

In general, the diffusive mass flux, $J_{i,i}$ is composed of diffusion due to thermal effects and diffusion due to species concentration and pressure gradients.

The chemical reaction and the radiation heat transfer were also considered in this modeling/simulation. The chemical reaction and the diffusion due to concentration gradients and thermal effects are included. Two radiation models are available in CFD code, Fluent, including the Discrete Transfer Radiation Model (DTRM) and the P-1 Radiation Model. The DTRM for prediction of surface-to-surface radiation heat transfer with or without a participating medium were employed in our modeling/simulation.

In this model, the simplest case of a constant absorption coefficient is determined by the local concentrations of CO_2 and H_2O species in the gas phase. The change in radiation intensity, DI along with a path ds is defined by:

$$\frac{dI}{ds} = -aI + \frac{a\sigma T^4}{\pi}$$

Where a = absorption coefficient (1/m), σ = Stefan-Boltzmann constant ($\text{W/m}^2\text{K}^4$),

T = gas temperature (K)

The radiation intensity approaching a point on a combustor wall surface is integrated to yield the incident radiation heat flux q_{rad} as:

$$q_{rad} = (1 - \epsilon_w) \int_{\Omega} I d\Omega + \epsilon_w \sigma T_w^4$$

Where T_w is the surface temperature at a point P on the surface and ϵ is the emissivity of the combustor wall.

The swirling hot flow in the combustion chamber is an axi-symmetric 3-D turbulent flow problem involving chemical (combustion) reaction and radiation heat transfer. The system was configured in 3-D cylindrical coordinates with the uniform mesh grids. The computational cells and boundaries for the calculation domain are shown in Figure 77.

There are a total of 24,192 grids in the system configuration including 14 slices of the tangential direction (I), 24 slices of the radial direction (J), and 72 slices of the vertical direction. Figure 78 shows the top view of computational domain in the combustor.

A typical slices in radial direction, $I = 1$, indicates the increased grid sizes in the radial direction, as shown in Figure 79. Figure 79 shows a variable grid system with 1.5 increments of non-uniform spacings in the tangential direction and 2 increments of non-uniform spacing in the radial direction. This arrangement was to improve the overall accuracy of computation while still keeping the total number of grids, and the computer time and storage at reasonably low levels.

A pressure gradient was expected to show in the across the cyclic planes at $I=0$ degree and $I=90$ degrees under the swirling flow in Figure 79. An initial mass flow rate across the boundaries was provided and identified in the simulation process. The center

Figure 77 Flow System and Velocity Component in Combustor Chamber

Figure 78 Top View of the Computational Domain in Combustor Chamber

Figure 79 Computational Domain and Different Grid Spacings in Combustor Chamber

line (or axis) of the combustor chamber is an axi-symmetry line. Based upon the symmetry and cyclic boundaries, the simulation procedure was simplified to save a lot of computer time.

(b) The Flow Patterns in the Hot Model

The combustor chamber has a height of 73 cm, an inner diameter of 14 cm, and an outer diameter of 17.6 cm. Two sets of secondary air nozzles are installed on the combustor wall at different levels of the height; 22.2 cm height for bottom nozzles and 41.6 cm height for top nozzles, respectively. Each nozzle is separated at 90 degrees around combustor wall. The secondary air is injected into the combustor chamber from the nozzles with a 45 degree yaw angle and zero degree pitch angle. The fuel nozzle is installed at the bottom of the combustor. The primary air is provided to the bottom part of the combustor along with the fuel supply pipe.

The test conditions and input boundary conditions are summarized in Table 13.

Table 13. Test Conditions for Simulations

Combustor Inner Diameter (ID)	cm	14.0
Combustor Outer Diameter (OD)	cm	17.6
Combustor Height	cm	73.0
Top Secondary Nozzle	cm	41.6
Top Nozzle No.		4
Bottom Secondary Nozzle	cm	22.2
Bottom Nozzle No.		4
Nozzle Size (OD/ID)	mm	10/8.5

Nozzle Yam Angle	degree	45
Nozzle Pitch Angle	degree	0
Secondary Air Flow Rate	m ³ /min (total)	0.11
Primary Air Flow Rate	m ³ /min	0.0036
Fuel Supply Pipe (OD/ID)	mm	9.52/7.62
Height of Fuel Nozzle	cm	7.63
No. of Fuel Nozzle Holes		5
Hole Size of Fuel Nozzle	mm	1.8
Fuel Flow Rate	m ³ /min	0.0092
Type of Fuel	Natural Gas	CH ₄
For Input Air/Fuel;		
Temperature	C	22
Pressure	atm	1

The flow patterns of the side view/top view on the advanced swirling fluidized bed combustor are shown along with stream lines in Figures 80 and 81, respectively. The flow starts from the fuel nozzle and swirling flow along the combustor height. The diameter of the swirling flow increased as the gas flow moved upward along the combustor chamber axis.

When the flow reached the secondary air input of the lower air injection nozzles, the flow pattern was changed from a laminar flow pattern to a turbulent flow pattern. When the flow reached the secondary air input of the upper air injection nozzles, a horizontal flow circle was formed along the nozzles. These results showed the pressure balance in K direction where the swirling flow could reach the combustor wall.

The stream lines at the bottom section of Figure 80 are the same stream lines as the center zone of Figure 81. Similarly, the stream lines at the upper zone in Figure 80 are the same stream lines as near the combustor wall in Figure 81. Four stream lines at the bottom/center zone sharply stretched out on the region where four nozzles are located along with a fuel injector. The results indicated that the flow patterns of the radial velocity components at the fuel injection could be simulated by our test conditions.

The swirling flow patterns in Figures 80 and 81 are very similar to the observed flow patterns in the combustion test. During the experimental test, the fire flame patterns and conditions are observed and recorded by VCR.

(c) The Velocity Profiles in the Hot Model

Figure 82 shows the side view of the velocity profile in the vertical direction. The velocity at the center is greater than that of the wall region. The velocity of the top section is also greater than that of the lower section.

Figure 83 shows velocity profiles at eight levels; $K=6, 10, 20, 30, 40, 50, 60$ and 70 . Figure 84 shows the velocity profiles for the fuel nozzle injector at the level of $K=10$. Figure 85 shows velocity profiles of the lower secondary air nozzle at the level of $K=30$. Figure 86 shows velocity profiles of the upper secondary air nozzle at level $K=50$.

At level $K=10$, the velocity near the wall is about 0.32 m/s. The velocity is reduced to about 3.77 m/s near the center, as shown in Figure 84. It is worth noting that the swirling velocity is reduced in both directions of the wall and the center of the combustor chamber. The highest velocity is about 4 m/s near the wall.

Figure 80 Flow Patterns of the Side View in the Combustor

Figure 81 Flow Patterns of the Top View in the Combustor

Figure 82 Velocity Profiles of the Vertical Direction in the Combustor

Figure 83 Velocity Profiles of the Vertical Direction at Different Levels (K=6, 10, 20, 30, 40, 50, 60, and 70)

Figure 84 Velocity Profiles at Level K=10 in the Combustor

Figure 85 Velocity Profiles at Level K=30 in the Combustor

Figure 86 Velocity Profiles at Level K=50 in the Combustor

At the secondary air injection levels, $K=30$ and $K=50$, the outside velocity is greater than that of the center as shown in Figures 85 and 86. The outside velocity is reduced to less than the center velocity in the level between the two secondary air injectors at $K=40$, as shown in Figure 83. In addition, the velocity increased from the wall region to the center region at the top section of the secondary air injectors.

At level $K=30$, the velocity of the secondary air nozzle outlet is about 3.4 m/s. When the air injected into the combustor chamber, the velocity decreased rapidly. However, the whole chamber achieved the swirling flow with a core diameter, as shown in Figure 85.

At level $K=50$, the velocity of the secondary air nozzle outlet is about 3.44 m/s as shown in Figure 86. The velocity formed the swirling flow with a core diameter, which is about half size at $K=30$.

(d) The Pressure Profiles in the Hot Model

A side-view of static pressure profiles is shown in Figure 87. The ranges of pressure changes are 1.692 pascals to -3.525 pascals. Some top view of pressure profiles at eight different vertical levels; $K=6, 10, 20, 30, 40, 50, 60, 70$ are shown in Figure 88.

The pressure values in the Figures are relative values based upon the reference point at the primary air inlet. The pressure change at the outer region of the wall is greater than that of the inner region of the axis. The pressure change at the bottom is greater than that of the top region because of a large amount of primary input, as shown in Figure 87. The higher velocity was found at the center region. It is believed that the secondary air injection affected the pressure profile of the center region. The increasing velocity from

Figure 87 Side View of Static Pressure Profiles

the pressure potential energy is transferred into kinetic energy, which caused the higher velocity at the center region.

The higher pressure zone is formed where the secondary air flows into the center region of the combustion chamber as shown in Figure 88. A dead zone or a local swirling flow was formed near the wall region, as shown in Figure 87.

(e) The Temperature Profiles in the Hot Model

Figure 89 shows the side view of the temperature profiles in the vertical direction. Figure 90 shows the top view of temperature profiles at eight different vertical levels; $K=6, 10, 20, 30, 40, 50, 60, 70$.

The high temperature zone (1400 °F) of the bottom section is located at the fuel injector nozzle outlet, which caused the ignition of the primary air and fuel mixing as shown in Figure 89. Figure 91 shows the temperature profiles of the top view near the fuel injector nozzle outlet ($K=6$). The low temperature profiles were near the combustor wall because of the cooling effect of the heat exchanger as shown in Figure 91.

At the vertical level, $K=30$, the temperature was relatively high (1300 F) because of the secondary air input at the lower level nozzles as shown in Figure 90. Figure 92 shows more detailed temperature profiles of the top view at level, $K=30$. The temperature profiles are changed by the secondary air input as shown in Figure 92. Figure 93 shows the temperature profiles of the top view at the upper secondary air nozzles.

The center temperature of the combustor at the upper secondary air nozzles is higher than that at the lower secondary air nozzles as shown in Figures 92 and 93. The

Figure 88 Top View of Pressure Profiles at Different Vertical Levels; K=6, 10, 30, 40, 50, 60, 70, 20

Figure 89 Side View of Temperature Profiles

Figure 90 Top View of Temperature Profiles at Different Vertical Levels; K=6, 10, 20, 30, 40, 50, 60, 70

Figure 91 Top View of Temperature Profiles at Level, K=6

Figure 92 Top View of Temperature Profiles at Level, K=30

Figure 93 Top View of Temperature Profiles at Level, K=50

upper secondary air was used effectively to burn the rest of the fuel in top combustor chamber. These results show the relatively large temperature gradient along the combustion chamber height and the radial direction. The combustion temperature was affected by the cooling water of the heat exchanger.

(f) The Gas Concentration Distribution and Characteristics

The basic covering equations for swirling, turbulent gas-particle flows and combustion in the swirling fluidized bed combustor were formulated and described in the former chapters. The species conservation equations included the mass fraction of the species, diffusive mass flux of species, and the net rate of production of the species due to the chemical reaction. Mass concentration is the mass of species per unit volume of the solution. The various chemical species in a diffusing mixture move at different velocities.

Figure 94 shows the methane (CH_4) concentration profiles. The center zone of the combustor has relatively high methane mass concentration. It is seen that for the present case the combustion of methane mainly takes place in the center zone of the combustor. The final burnout of methane further extends into the upper part of the center tube. It is believed that the inactive zone at the bottom is primarily caused by an insufficient oxygen supply. The injection of secondary air squeezes the uprising flue gas and methane that causes a fuel-lean zone (inactive reaction) near the secondary air nozzles. It is of interest to note that the active reaction zone generally coincides with the high gas temperature zone.

Figure 94 The Concentration Profiles of the Methane

Figure 95 shows the carbon dioxide (CO_2) concentration profiles. The carbon dioxide concentration increased with the increasing of the combustor height. The carbon dioxide concentration near the combustor wall is lower than that of the combustor center zone. As shown in Figure 95, the lower part of the combustor has lower carbon dioxide concentration.

Figure 96 shows the oxygen (O_2) concentration profiles. It can be seen that the oxygen in the primary air was completely consumed, which implies an efficient combustion of fuel during the early stage in the combustor. The addition of secondary air supplies the needed oxygen for the continued combustion of fuel. The peak concentration of oxygen remains at the combustor wall because of the secondary air injection. The oxygen concentration increased with the increasing of the combustor height. However, the oxygen concentration of the combustor center zone decreased.

(g) Heat Transfer Characteristics

Heat transfer data between hot flue gases and combustor walls are important factors for the design and operation of combustors.

The simulation results of the heat flux and heat transfer coefficients in the combustor chamber are shown in Figures 97 to 98. Figure 97 shows a side-view of the grid profiles. Figure 98 shows the profiles of the heat flux near the combustor wall zone. A large amount of heat is generated at the bottom and wall zone of the combustor. The heat flux on the wall of the combustor chamber is relatively higher than that of the center region of the combustor.

Figure 95 The Concentration Profiles of the Carbon Dioxide

Figure 96 The Concentration Profiles of the Oxygen

Figure 97 The Side-view of the Grid Profiles for Heat Flux

Figure 98 The Heat Flux Profiles near the Combustor Wall

The heat flux on the wall of the upper chamber is much higher than that of the lower chamber. It is believed that the designed strong swirl and secondary air injection affected this special characteristic of heat flux. An averaged wall heat flux amounted to 550 W/m^2 in the upper chamber of the combustor.

The gaseous fuel in the lower part of the combustor is largely depleted. The heat removal is reduced to much lower extent of 240 W/m^2 . For a given fuel, a proper design of heat transfer surfaces in the fluidized bed combustor can match with the combustion process of the fuel.

Figure 99 shows the heat transfer coefficient changes along the combustor height. It is a similar trend that the heat transfer coefficient changes based on the combustion test results described before. The heat transfer coefficients at $0.4 (X/H)$ and $0.75 (X/H)$ are $1.2 \text{ W/m}^2\text{C}$ and $2.2 \text{ W/m}^2\text{C}$. When the secondary air was provided effectively, the swirling flame in the upper chamber of the combustor was much stronger than that of the middle or the bottom part of the combustor. These swirling and vortex flame were dominant in the upper chamber of the combustor, which affected higher heat transfer in the upper chamber. Multiple secondary air injection can cause significant effects on gas-particle flow in the combustor. This air injection was found to be the best arrangement to strengthen the swirling flow, increase mass fluxes, and retain more particles in the combustor chamber.

Figure 99 The Heat Transfer Coefficients along the Combustor Wall Height

11. CONCLUSIONS

The objective of this project is to predict the heat transfer and combustion performance in newly-designed fluidized bed combustor (FBC) and to provide the design guide lines and innovative concept for small-scale boiler and furnace. The major accomplishments are summarized below:

- o Establish the test facilities of the advanced FBC cold models and gas-particle measuring systems;
- o Conduct systematic measurements of gas flow field in the cold models;
- o Conduct systematic measurements of particle flow field in the cold models.
- o Cold test models, namely exploratory model, bench-scale model, and associated auxiliary subsystems for air supply, particle collection have been successfully developed and tested.
- o Measurements showed that the gas flow field in the advanced FBC is characterized by strongly swirling/turbulent flow in tangential direction and developing flow in axial and radial directions. Axial and radial recirculation zones were found in the secondary air injecting nozzles.
- o Particle flow field is characterized by circulating dense layers and dilution zones, which appear alternatively along the test chamber height.
- o Secondary air injections could cause significant effects on gas-particle flow in the FBC. Measurements showed that an increase in air flow rate and injection angle can accelerate the gas tangential velocity.
- o Numerical simulation of swirling, turbulent gas-particle flows has been successfully performed for predicting gas-particle flow in the advanced FBC.
- o The exploratory hot model has been designed to better understand the combustion processes and the local heat transfer phenomena in the combustor chamber. The auxiliary subsystems and measuring devices for this test model has been successfully developed.
- o The systematic combustion tests have been conducted on the heat transfer characteristics with regard to temperature variation, heat removal rate, and air distribution.
- o The combustion temperature is controllable by the heat transfer surface consisting of the water-cooling tube. So the heat removal rate can be controlled by the cooling water, which can be served to find the desirable local heat transfer coefficients along the axial combustor.

- o The primary air fraction and secondary air fraction could be arranged appropriately to provide good parameters including ignition, combustion, burnout of fuel, and timing of oxygen supply.
- o Numerical simulation has explored the gas concentration distribution, combustion temperature profiles, heat flux, and heat transfer coefficient in the combustor.
- o Many inherent advantages and unique features of the advanced FBC technology in gas flow, particle flow, and combustion have been confirmed by our continued efforts in cold flow measurements and hot flow measurements, numerical simulation, which shows a promising potential for future use.

References

- [1] Singer, J, " Combustion; Fossil Power System", Combustion Engineering, Inc., pp. 24/19-28, 1981.
- [2] Stringer J. and W. Stallings, "Wastage in Bubbling Fluidized Bed Combustors", Proceedings of 10th International FBC Conference, PP. 857-858, 1989.
- [3] Syred, N. and J.M. Beer, "Combustion in Swirling Flows: a Review of Combustion and Flame", Vol. 23, pp. 143-201, 1974.
- [4] Fink, G.D., Thermal Physics, 2nd Edition, pp. 194-196, Chapman & Hall, 1993
- [5] Cheng, L., S.L. Soo, and S.K. Tung, "Electrical Measurement of Flow rate of Pulverized Coal Suspensions", J. Eng. Power, Vol. 92, pp. 135 –143.
- [6] Manual of Probe Catalog, United Electric Controls Company, Watertown, MA.
- [7] Hestroni, G., Handbook of Multi-Phase Systems, McGraw-Hill Book Co., New York, Chapter 3.
- [8] Boysan, F. J. et al, Modeling Coal-Fired Cyclone Combustor, Combustion and Flame, Vol. 63, pp.73-85, 1986.
- [9] Smooth, L.D. and P.J. Smith, Coal Combustion and Gasfication, Plenum Press, New York, NY, 1985
- [10] User's Manual of RTI-800/815, Analog Devices, Inc.
- [11] Arena U. et al., Hydrodynamics of a Circulating Fluidized Bed with Secondary Air Injection, Pro. CFB Technology IV, pp. 899-905, 1993
- [12] 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.
- [13] Khalil, E.E., Numerical Computations of Turbulent Flow Structure in Cyclone Chamber, Joint ASME/AIChE 18th National Heat Transfer Conference, ASME Paper No. 79-HT-31, 1979.
- [14] Boysan, F., J. Swithenbank, and C.J. lawn, Modeling Coal-Fired Cyclone Combustor, Combustion and Flame, Vol.63, pp. 73-85, 1986.
- [15] Fluent User's Guide, Vol. 4, Chapter 19, pp.19/7-19/10, 1995.

- [16] Hoffmann, K.A. and S.T. Chiang, Computational Fluid Dynamics, 3rd Edition, Vol.1; Chapter 9, Vol.2; Chapters 11 & 13, Engineering Education System, KS, 1995.
- [17] Launder, B.E. and D.B. Spalding, Mathematical Models of Turbulence, Academic Press, London, 1972.
- [18] Instruction Manual for Enercomp M2000/M2000E, Energy Efficiency Systems, Inc. 1992.
- [19] Nag, P.K. and B.V. Reddy, Some Investigations on Heat Transfer in a Hot CFB, Proceedings of the 13th International Conference on FBC, Vol.2, pp. 1327-1336, 1995.
- [20] Hewitt, G.F. et al, Process Heat Transfer, CRC Press, 1994.