TECHNICAL PROGRESS REPORT NO.7

DEVELOPMENT OF THE INSTRUMENTATION
AND MODELING FOR HEAT TRANSFER
CHARACTERISTICS IN CFBC

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-FG22-95MT950

BY

DR. SEONG W. LEE, PRINCIPAL INVESTIGATOR

MORGAN STATE UNIVERSITY
SCHOOL OF ENGINEERING
BALTIMORE, MD 21239
(phone) 443-885-3106

April 1999

ABSTRACT

This technical report summarizes the research conducted and progress achieved during the period from October 1, 1998 to March 31, 1999.

Numerical simulation was conducted to predict the flow pattern, velocity and pressure, temperature, and heat transfer characteristics in the CFB system.

The 2-D air velocity profiles showed the axial and tangential velocity profiles in the CFB riser. The small flow boundary layers were found near the CFB riser. The tangential velocity profile is characterized by injection of aeration air. The highest air pressure at the bottom of the heat transfer probe caused a strong gas mixing process in the CFB riser.

The heat absorbing water-cooled heat transfer probe enclosing the CFB riser of the cold model was assumed. The gas temperature decreased along the flow direction of the heat transfer probe.

The heat transfer characteristics was described by the heat flux changes in the CFB chamber. The higher heat flux was found at the bottom of the heat transfer probe. A large amount of heat is generated and removed via the neighboring the heat transfer probe.

Numerical simulation will be continued to predict the flow patterns, velocity, pressure, temperature, and heat transfer characteristics in the CFB system.

TABLE OF CONTENT

ABSTRACT.....	ii
---------------	----

SECTION

1. NUMERICAL SIMULATION FOR THE CFB MODEL WITH HEAT TRANSFER.....	1
1.1 Numerical Simulation Procedure.....	1
1.2 Calculation Conditions for Isothermal Gas Flow.....	2
2. RESULTS AND DISCUSSION.....	3
2.1 The Effect of Air Velocity.....	3
2.2 The Effect of Air Pressure.....	4
2.3 The Effect of Temperature.....	4
2.4 The Heat Transfer Characteristics.....	5
3. CONCLUSIONS.....	6
REFERENCES.....	12

SECTION 1

NUMERICAL SIMULATION FOR THE CFB MODEL WITH HEAT TRANSFER

A series of numerical simulation of the gas flow in the CFB in the absence of particles and combustion were conducted.

The basic physical modeling for flow/heat transfer was introduced to determine velocity, pressure, temperature, and heat flux in the CFB chamber [1]. The system configuration with 3-D cylindrical coordinates was introduced for the solution process.

1.1 Numerical Simulation Procedure

Due to the distinct nature of gas-phase and particle-phase governing equations, different numerical simulation techniques were developed and applied for the CFB process. In the present study, the gas flow in the CFB riser is a turbulent, swirling, developing, reacting flow in a system of complex geometry with a secondary (aeration) air inlet. The standard k - ϵ turbulence was not appropriate for the strong turbulent flow with unisotropic flow behaviors. Thus the Renormalization Group (*RNG*) k - ϵ model was employed to this study [2].

The flow system and the solution domain including the coordinates axes and velocity components were introduced [1]. The system was configured in 3-D cylindrical coordinates with uniform mesh grids. There are a total of 30422 grids including 41 grids of the tangential direction (I), 14 grids of the radial direction (J), and 53 grids of the vertical direction (K). The wall functions were used to deal with the boundary layer and turbulence interactions at the mesh grids along the walls.

1.2 Calculation Conditions for Isothermal Gas Flow

The detailed calculation conditions of the CFB configuration air injection, and input boundary conditions for numerical simulation are summarized in Table 1. The CFB chosen in this study is the bench-scale cold CFB model presented in the previous report [3].

Table 1 Calculation Conditions for Numerical Simulation

Combustor Diameter	cm	11.25
Reactor Height	cm	135.0
Aeration Nozzle	cm	15.0
Heat Transfer Probe Diameter	cm	1.50
Heat Transfer Probe Height	cm	35.0
Primary Air Sources;		
Flow Rate	m ³ /s	0.1059
Gas Velocity	m/s	10.65
Turbulence, K-ε	m ² /s ²	0.3
Turbulence Dissipation	m ² /s ³	0.3
Aeration Air Sources;		
Flow Rate	m ³ /s	0.00202
Gas Velocity	m/s	1.03
Turbulence, K-ε	m ² /s ²	0.5
Turbulence Dissipation	m ² /s ³	0.5
Heat Transfer Probe;		
Water Flow Rate	g/min	4.0
Inlet Water Temp.	F	180
Outlet Water Temp.	F	170

SECTION 2

RESULTS AND DISCUSSION

2.1 The Effect of Air Velocity

Figures 1 and 2 show the 2-D velocity profiles. In the vertical direction, the lower gas velocity was found at the end of the heat transfer probe, which is ranged from 7.4 m/sec to 8.5 m/s. The higher gas velocity, near the probe, is ranged from 9.6 m/sec to 12.1 m/sec as shown in Figure 1.

Figure 2 shows that the velocity profiles below the probe were relatively uniform. When the air flow reached the probe, the uniform flow pattern changed. The air velocity at the center of the probe decreased from 12.1 m/s to 5.4 m/s.

The small flow boundary layers were found near the CFB riser wall and the probe wall as shown in Figure 2 (see green color region). Since the tangential velocity exhibits a uniform profile near the CFB riser and the probe wall. The tangential velocity profile is characterized by injection of aeration air.

The axial and radial velocities change frequently, which indicates the gas flows in the CFB riser is in the "developing" mode with repeated accelerations and decelerations. It is believed that these velocity profiles are advantageous for an enhanced gas-particle slip motion and heat/mass transfer in the CFBC [4]. The same results were also found in our cold experimental measurements in the previous reports [2]. The higher gas velocity between the heat transfer probe and the CFB riser ranges from 10.4 m/s to 12.1 m/s.

It is worth noting that the turbulent kinetic energy, k and the absolute values of Reynolds stresses increase along the flow direction, which is certainly desirable for gas mixing and combustion. From a turbulent mixing point of view, the region near the probe, having the most vigorous turbulence, provides an ideal place for the burnout of fuel particles [5].

2.2 The Effect of Air Pressure

Figure 3 shows the static pressure profiles in the CFB riser. The positive numbers represent when the tube pressure is larger than the system pressure. Similarly, the negative numbers indicate that the tube pressure is smaller than that of the system pressure. The air pressure was affected by the aeration air flow. The air pressure was reduced with lower aeration air flow rate as shown in Figure 3.

On the other hand, the highest air pressure at the bottom of the probe caused a strong gas mixing process in the CFB riser. It is believed that the turbulent flow characteristics produced some eddies and vortex flow at the bottom region of the probe. This should be considered as one of the unique features of the circulating fluidized bed combustor gas flow, which is desirable for gas-gas mixing and gas-particle mixing. It is found that the calculated air pressure field resumes good dynamic similarities with the experimental measurements [6].

2.3 The Effect of Temperature

Figure 4 shows the temperature changes along the vertical direction in the CFB chamber. Heat absorbing water-cooled heat transfer probe enclosing the CFB riser of the cold model was assumed.

It is found that the gas temperature decreased along the flow direction of the heat transfer probe.

In Figure 4, temperature is changed from 353 K to 340 K along the heat transfer probe. The temperature of the probe bottom region is relatively higher than that of the probe top region.

The lower temperatures are found near the CFB riser wall which ranges from 315 K to 330 K. It is believed that the riser wall is cooled down by the ambient temperature.

2.4 Heat Transfer Characteristics

Heat transfer data between hot flue gas and combustor water wall tubes are important for the design and operation of combustors [7].

Figure 5 shows the heat flux changes along the vertical direction in the CFB chamber. In this calculation, it was assumed that the heat transfer tube in contact with gas flow was water cooled. As shown in Figure 5, a large fraction of heat is removed in the lower part of the heat transfer probe, where rapid combustion of gaseous fuel may occur. A large amount of heat is generated and removed via the neighboring the probe. The gas flow in the upper part of CFB riser is largely depleted. The continued combustion gives little heat to this upper part. The heat removal is reduced to a much lower extent of 3.9 W/m². The higher heat flux was found at the bottom of the heat transfer probe, which is about 50.3 W/m². The heat flux was changed from 50.3 W/m² to 16.1 W/m² along the heat transfer probe.

SECTION 3

CONCLUSIONS

The numerical simulation was conducted to predict the flow pattern, velocity and pressure, temperature, and heat transfer characteristics in the CFB system.

The axial velocity radial velocities change frequently, which indicates the gas flows in the CFB riser is in the "developing" mode with repeated accelerations and decelerations. It is believed that these velocity profiles are advantageous for an enhanced gas-particle slip motion and heat/mass transfer. The air pressure was affected by the aeration air flow. The highest air pressure at the bottom of the heat transfer probe caused a strong gas mixing process in the CFB riser.

Heat absorbing water-cooled heat transfer probe enclosing the CFB riser of the cold model was assumed. It is found that the gas temperature decreased along the flow direction of the heat transfer probe.

A large fraction of heat is removed in the lower part of the heat transfer probe, where rapid combustion of gaseous fuel may occur. The higher heat flux was found at the bottom of the heat transfer probe, which is about 50.3 W/m^2 . The heat flux was changed from 50.3 W/m^2 to 16.1 W/m^2 along the heat transfer probe.

Numerical simulation will be continued to predict the flow patterns, velocity, pressure, temperature, and heat transfer characteristics in the CFB system.

Figure 1 Velocity Profiles along the CFB Riser

Figure 2 Velocity Profiles in the CFB Riser with a Heat Transfer Probe

Figure 3 Static Pressure Profiles along the CFB Riser

Figure 4 Temperature Changes along the CFB Riser

Figure 5 Heat Flux Profiles along the CFB Riser

REFERENCES

- [1] Lee, S.W., Technical Progress Report, No.6, U.S. DOE, Federal Energy Technology Center (FETC), October 1998.
- [2] Lee, S.W., Technical Progress Reports, Nos.4/5, U.S. DOE, Federal Energy Technology Center (FETC), Oct. 97/May 98.
- [3] Lee, S.W., Technical Progress Report, No.1, U.S. DOE, Federal Energy Technology Center (FETC), April 1996.
- [4] Haler, P.K. et al, Axial and Radial Variation of Bed to Wall Heat Transfer in a CFB, Proceedings of 4th Intl' Conference on CFBS, PP. 319-323, August 1993.
- [5] Dasgupta, S. et al, Turbulent Gas-Particle Flow in CFB Riser, Proceedings of 4th Intl' Conference on CFBS, PP. 367-372, August 1993.
- [6] Hetsroni, G., Ed. in Chief, Handbook of Multiphase Systems, McGraw-Hill Book Co., New York, NY, 1982.
- [7] Anderson, B. et al, Local Lateral Distribution of Heat Transfer on Tube Surface Membrane Walls in CFB Boilers, Proceedings of 4th Intl' Conference on CFBS, PP.331-318, August 1993.