DOE/MT/93006--T7

Technical Progress Report No.12

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

RECEIVED
USDOE/PETC
96 DEC 16 AM 10:41
ACQUISITION & ASSISTANCE DIV.

to

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

for

Project No: DE-FG22-93MT93006

MASTER

by

Dr. Seong W. Lee, Principal Investigator

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

November 1996

DISCLAIMER

SUMMARY

This technical report summarizes the research conducted and progress achieved during the period from July 1, 1996 to September 30, 1996.

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

Numerical simulation was acquired from the particle trajectories by means of the Reynolds Stress Model (REM) with general algebraic expressions. The typical particle trajectories for single particle injection were predicted by the top view, the side view, and the isolated 3-dimensional view. The simulation of particle trajectories showed three different stages: ascending, colliding/bouncing, and slipping stages.

Numerical simulation for the bunch particle injection will be continued to understand the particle characteristics in the combustion chamber.

The preliminary system test was continued on the hot model. Thermal performance of the preliminary test results was analyzed and predicted. Based upon the preliminary test results, the auxiliary subsystems were modified and improved for the systematic test. The development of the computer-assisted data acquisition system will be continued for the instrumentation of the temperature measurement, the flow measurement, and the emissions measurement.

TABLE OF CONTENTS

PAGE

SECTION

SECTION 1

Numerical Simulations for Particle Trajectories

1.1 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 is summarized in Table 1. Test conditions for the gas phase flow are the same as those shown in Table 1 in the previous report [1]. Experimental method and results for the particle velocity measurements were stated in the previous report [2]

Table 1. Test Conditions of Single Particle Injection

Particle type		Glass Beads
Size	mm	0.04
Density	lb/ft^3	156.05
Particle Injection Location and Initial Velocity:		
I	degree	45
J	inch	1
K at the Fluid Bed Surface	inch	3
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 [3]. 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 1, 2, and 3. Figure 1 is the top view, Figure 2 is the side view, and Figure 3 is the isolated 3-Dimensions 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 2 and Figure 3). 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 2. 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 upflowing gas.

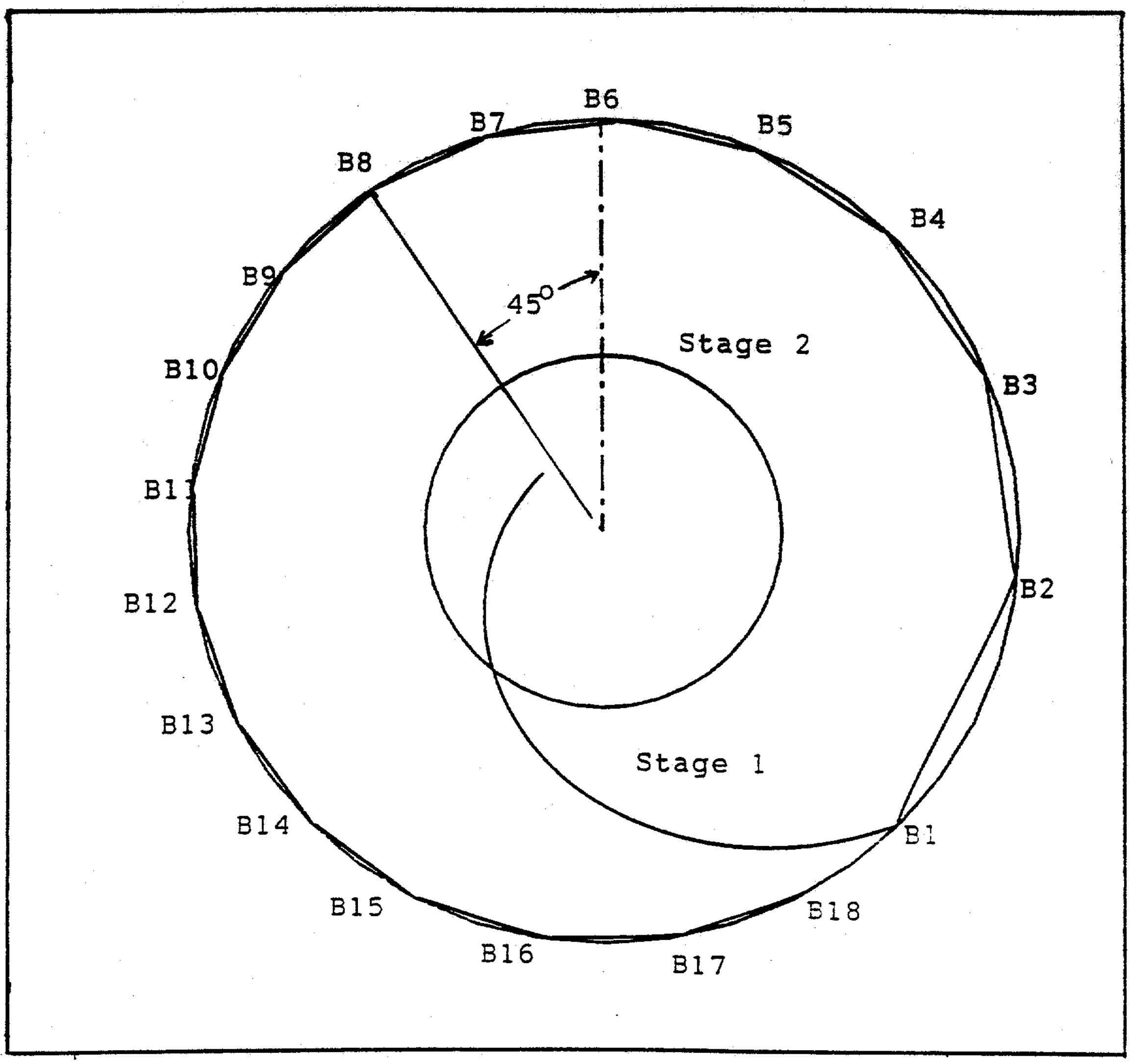


Fig. 1 Top View of Particle Trajectory in the Combustor

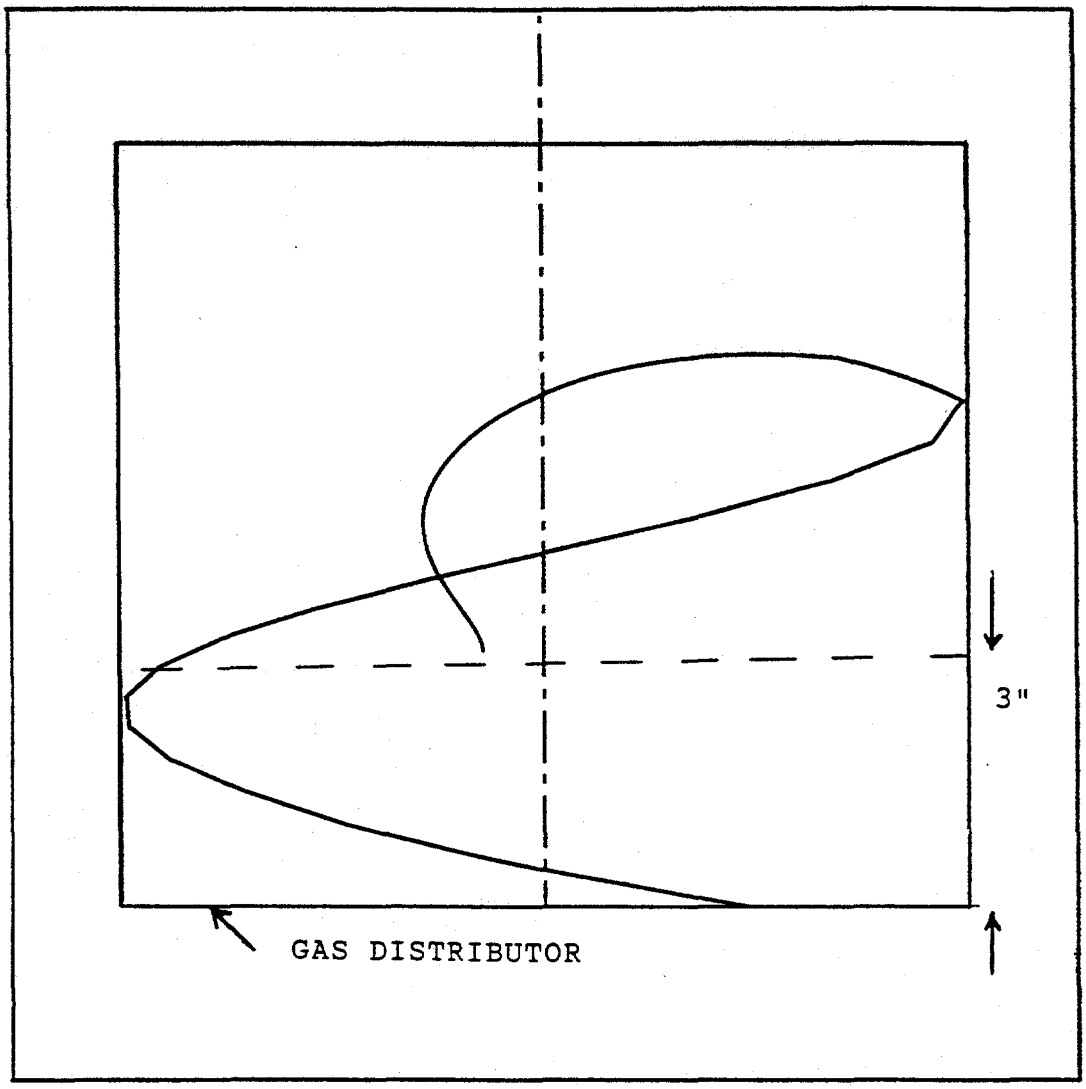


Fig. 2 Side View of Particle Trajectory in the Combustor

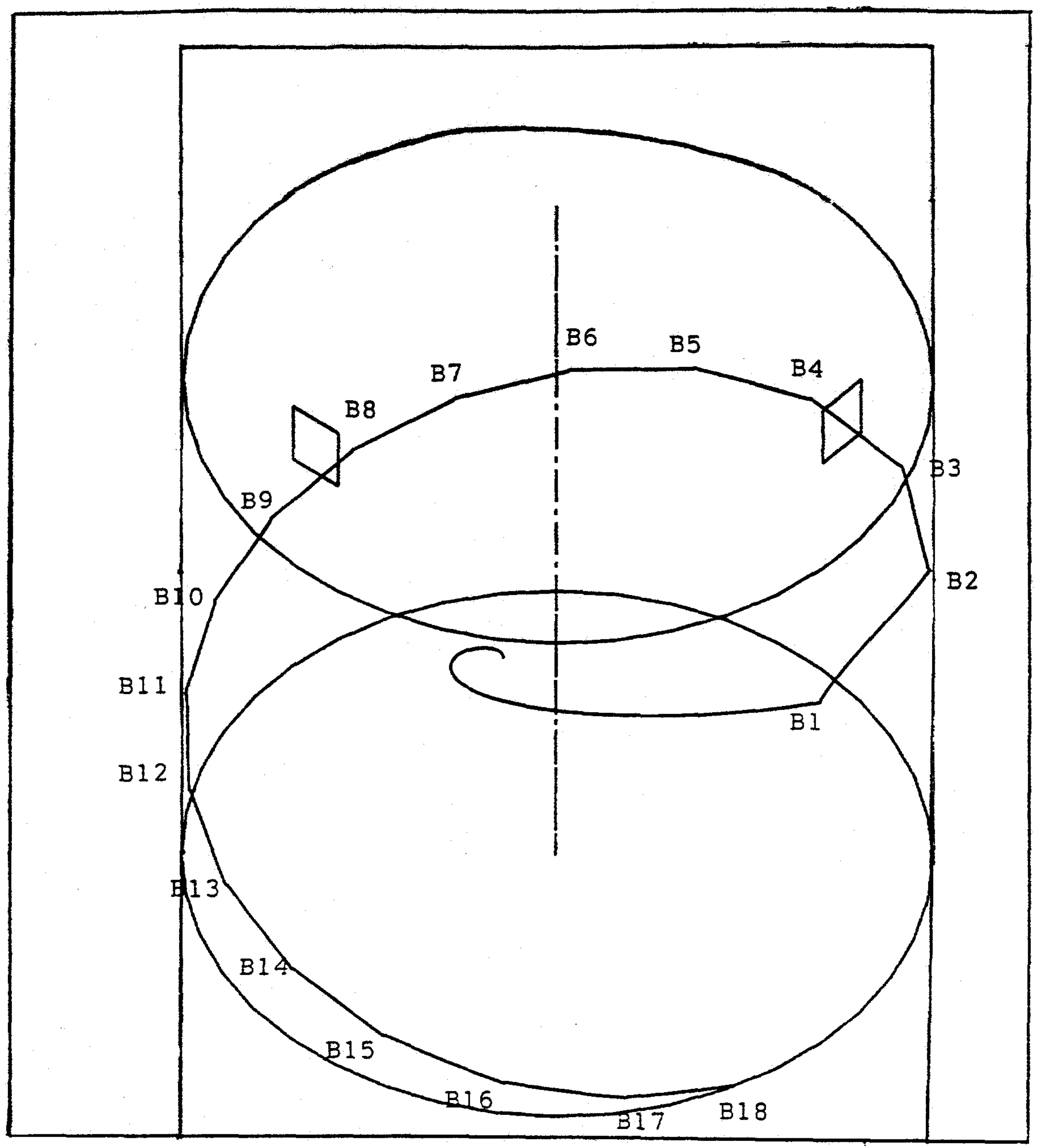


Fig. 3 **Three-Dimensional Plot of Particle Trajectory in the Combustor**

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

1.2 Numerical Calculation and Basic Governing Equations

Equations (1) through (4) were used to calculate the particle trajectory [4,5]. 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:

$$\frac{du_p}{dt}=F_D(u-u_p)+g_x(\rho_p-\rho/\rho_p)+F_x \qquad (1)$$

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

$$F_d=\frac{18\mu}{\rho_p D^2_p}\frac{CDRe}{24} \qquad (2)$$

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:

$$Re=\frac{\rho D_p|u_p-u|}{\mu} \qquad (3)$$

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

$$C_D=a_1+\frac{a_2}{Re}+\frac{a_3}{Re^2} \qquad (4)$$

where the a's are constants that apply over several ranges of Re [6].

SECTION 2

The Preliminary System Test of Hot Model

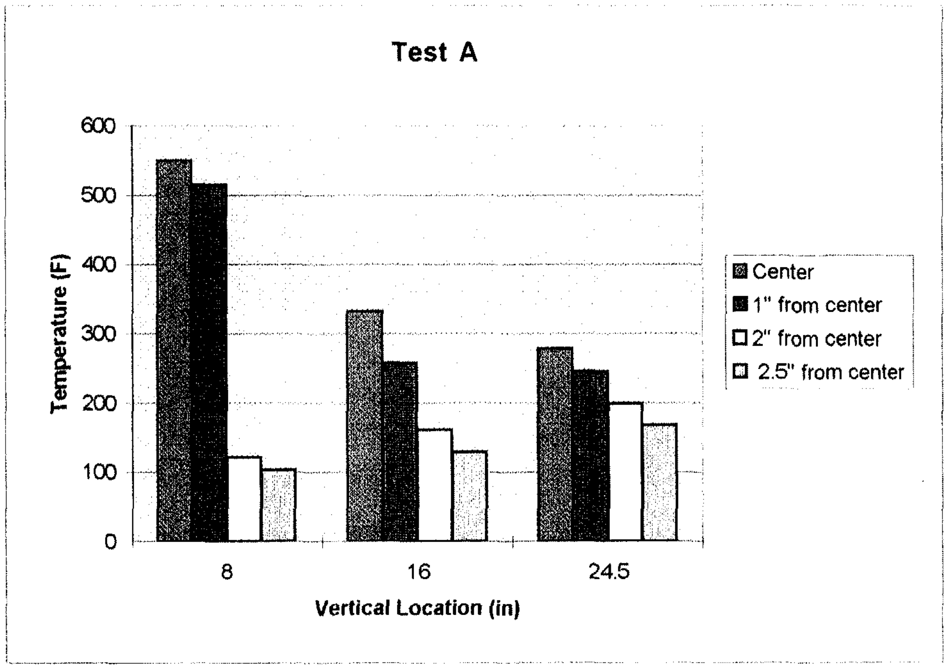
The preliminary system test was continued to analyze the thermal performance on the exploratory hot model. The auxiliary subsystems including air supply, water supply, and fuel supply were carefully inspected by the safety and health guideline [2]. All instruments are checked and calibrated for the tests.

Three tests were conducted at a primary flow of approximately 45% (0.6 in. H2O) as shown in test conditions of Tables 4 to 6. 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 4 was exactly opposite to Test B of Figure 5. The detailed test conditions were shown in Figures 4 to 6.

Varying the secondary air flow 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 air flow. 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 air flow has an average temperature of 254.8 F as shown in Figures 4 to 6. The least air flow yields the highest combustor temperatures.

Test D of Figure 7 was conducted to determine the conditions at the lowest combined primary and secondary air flow. 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.

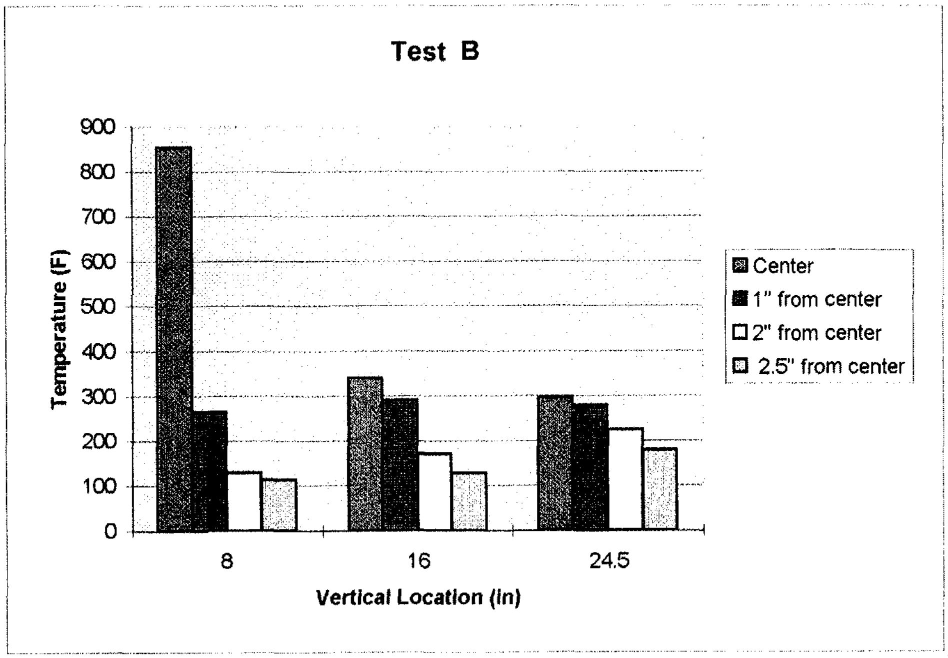
Fig. 4 Test Conditions and Temperature Profile in Combustion Chamber



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 cei	2" from center	2.5" from cent	Surface	Avg temp
8	550(F)	515	121.6	102.7	94.2	322.33
16	331.7	257	160.9	129	98	219.65
24.5	277.6	245	199.5	167.7	114.6	222.45
avg 3 therm.cpls	386.433	339.000	160.667	133.133		

Fig. 5 **Test Conditions and Temperature Profile in Combustion Chamber**

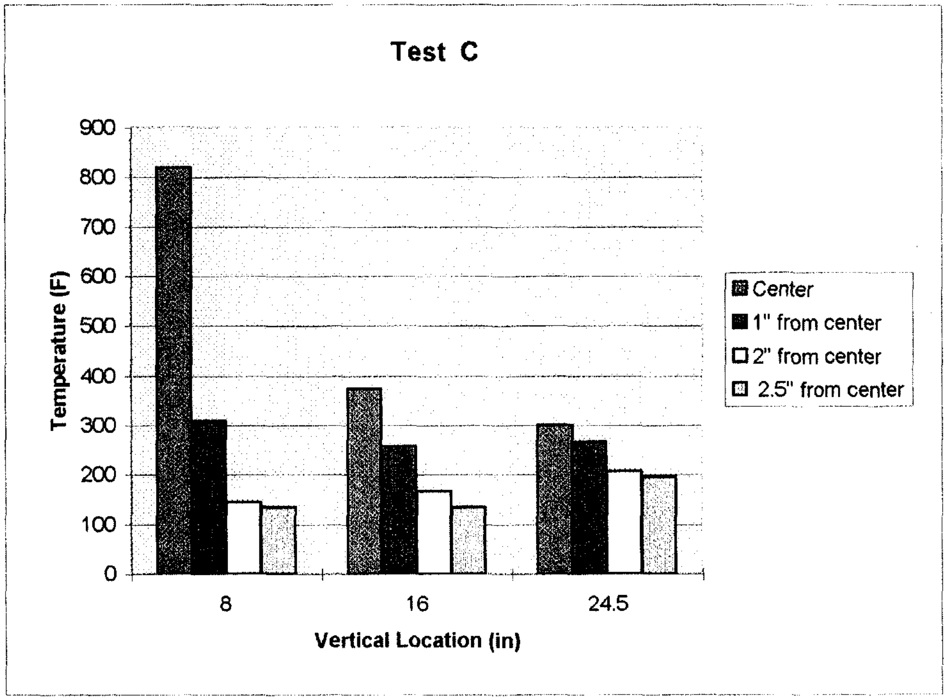


Hot Model Preliminary Test Data

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)	5	
Secondary air top (CFM)	10	
Cooling Water in	71	
Cooling Water out	71.2	
Cooling water flow (GPM)	1.5	
Flue Temp	229.2	

Dist.(in) from Bottom	Center	1" from cei	2" from center	2.5" from cent	Surface	Avg temp
8	855 (F)	265	128	113.2	97.5	340.3
16	340	290	170	126.5	102.2	231.63
24.5	295	278	225	178	123.7	244
avg 3 therm.cpls	496.667	277.667	174.333	139.233		

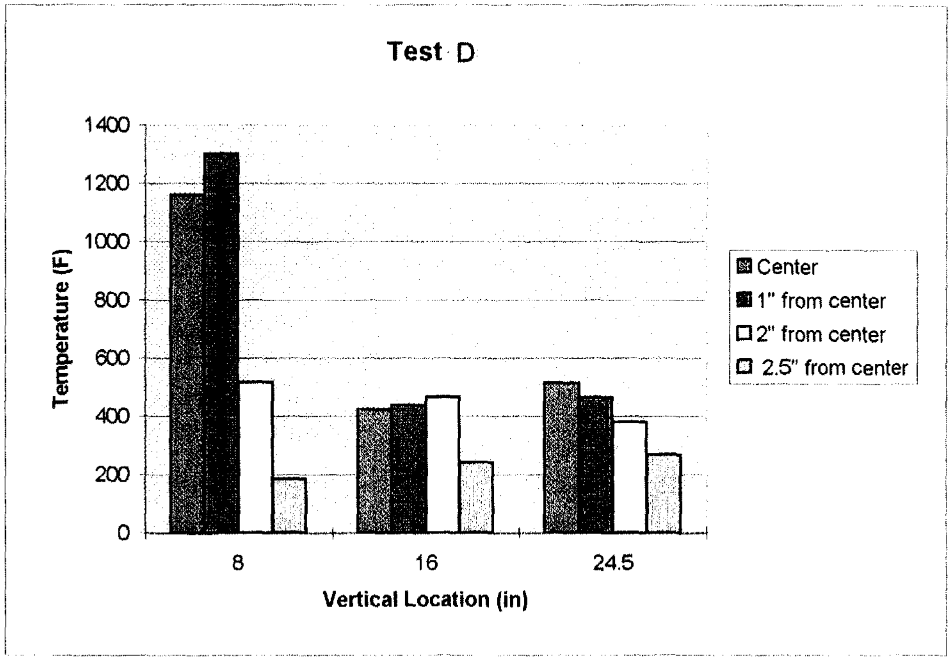
Fig. 6 Test Conditions and Temperature Profile in Combustion Chamber



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	
Secondary air top (CFM)	5	
Cooling Water in	71	
Cooling Water out	71.2	
Cooling water flow (GPM)	1.5	
Flue Temp	255.9	

Dist. (in) from Bottom	Center	1" from cei	2" from center	2.5" from cent	Surface	Avg temp
8	820(F)	309	145	134.5	103.5	352.13
16	373.8	257	166.3	134.4	102.5	232.88
24.5	300.2	266.8	207	195.5	121.4	242.38
avg 3 therm.cpls	498.000	277.600	172.767	154.800		

Fig. 7 **Test Conditions and Temperature Profile in Combustion Chamber**



Hot Model Preliminary Test Data		
With Primary Air (in of H2O)	0.1	
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	72.3	
Cooling water flow (GPM)	1.5	
Flue Temp	487	

Dist. (in) from Bottom	Center	1" from cei	2" from center	2.5" from cent	Surface	Avg temp
8	1160 (F)	1300	518	184.7	137	790.68
16	423	438	466	242	145	392.25
24.5	515	465	380	268	164.7	407
avg 3 therm.cpls	699.333	734.333	454.667	231.567		

REFERENCES

[1] Lee, S.W., Technical Progress Report, No. 10 to U.S. DOE PETC, April 1996.

[2] Lee, S.W., Technical Progress Report, No. 11 to U.S. DOE PETC, July 1996.

[3] Launder, B.E. and D.B. Spalding, Mathematical Models of Turbulence, Academic Press, London, 1972.

[4] Fluent User's Guide, Vol.4, Chapter 19, 19/7-19/10, 1995.

[5] Hoffman, K.A. and S.T.Chiang, Computational Fluid Dynamics, 3rd Ed. Vol. 1; Chapt.9, Vol.2; Chapters 11 & 13, Engineering Systems, KS, 1995.

[6] Hetsroni gad, handbook of Multiphase Systems, McGraw-Hill book, Co., Chapter 3, 9, 1982.