Third International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2003

NUMERICAL ANALYSIS OF FLOW VELOCITY DISTRIBUTION IN HIGH TEMPERATURE AIR COMBUSTION FURNACE

Li-Jun WANG[1, 2], Zong-Shu ZOU[2], Jiu-ju CAI[2] and Yu-Ling HAO[3]

[1]Shenyang Architectural and Civil Engineering University, Shenyang 110015, China
[2]School of Materials and Metallurgy, Northeastern University, Shenyang 110004, China
[3]School of software, Northeastern University, Shenyang 110004, China

ABSTRACT

The three-dimensional velocity distribution in a small scale high temperature air combustion furnace was numerically simulated with a developed CFD program. Three case for an isothermal flow configuration were predicted under the conditions of three different velocity ratios of fuel to air injection. The flow fields of iso-thermal calculations are compare to those of a combusting flow. It is shown that under the certain geometry parameters, the lower oxygen concentration in the re-circulating flow and a lower uniform temperature level can be obtained in high temperature air combustion furnace by selecting the proper velocity ratio of the fuel to air jet. The predicted results are in agreement with the results of experimentation.

NOMENCLATURE

u10	air inlet velocity
u20	fuel gas inlet velocity
k	kinetic energy of turbulence
u, v, w	velocity component
m_{fu}	fuel mass fraction
f	mixture fraction
g	turbulent pulse of *f*
h	total enthalpy
S_ϕ	source term
CEBU	EBU model coefficient
Γ_ϕ	effective mass diffusion coefficient
ρ	fluid density
ε	dissipation rate of turbulence kinetic energy
μ_e	effective viscosity

INTRODUCTION

A new combustion technology in which gas fuel is combusted by high temperature air (800-1000℃) with a low oxygen concentration (<15%) was developed (Morita, 1996, Yoshiawa, 2000, Gupta, 1999). This high temperature air combustion (HiTAC) technology has the following advantages.

(1) A uniform combustion heat release and uniform heat transfer is realized.

(2) Low NO_X emission is achieved by in-furnace exhaust gas recirculation.

3) Exhaust heat recovery is possible by regenerative burners (Newby, 1987).

In this study, a 3d-simulation for the model furnace was performed using a developed computational fluid dynamics calculation program. The flow fields of isothermal and combustion occurring in the model furnace were calculated by simultaneously solving equations for continuity, momentum, mixture mass fraction and it's turbulent pulse transport, fuel mass fraction and energy with a radiation transport equation (RTE) as a source term. The simulated velocity profile properties were analyzed and found to agree with the previous experimental velocity profile structure (Huang et al., 1997).

MODEL DESCRIPTION

Physical Model

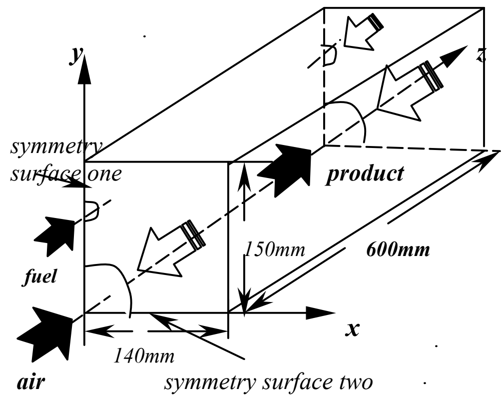

Figure.1 1/4 symmetry chart of furnace model

Figure 1 is the schematic diagram of the HiTAC experimental furnace whose dimensions are *600mm×280mm×300mm*. The black and white arrows of each side represent that combustion takes place alternatively. Switching period is about 30 seconds. Natural gas was used as fuel. The calculating domain is a 1/4 symmetry body for the geometry and the fuel inlet velocity is split using half symmetry.

The main jet flow velocity of the high temperature air is *u10* and the nozzle diameter is 32mm. The fuel jet flow velocity is *u20* and the nozzle diameter is 11mm. The distance between the air nozzle and fuel nozzle centers is 112mm. An uneven grid distribution is used here with a total number of cells of *44×18×17* in the z, y, x directions respectively.

Mathematical Model

The governing equations to be solved in the calculation domain are three-dimensional steady-state equations. In this study, the turbulent flow field, the temperature field, and the concentration of the species of the reactants and products have been simulated. The turbulence model used was the standard $k-\varepsilon$ model and a probability density distribution (PDF) is assumed for the mixture fraction and it's turbulent pulse distribution (Spalding, 1979). The combustion is modeled using the SCRS assumption (Spalding, 1979) and the Eddy Break-up (EBU) model is adopted. The discrete ordinate model (DOM) is adopted for radiative heat transfer in absorbing-emitting gray medium using a S4 approximation. (Fiveland, 1984, Truelove, 1988).

Governing equations can be expressed by a generalized time-averaged transport equation:

$$\frac{\partial}{\partial x_j}\left(\rho u_j \phi\right)=\frac{\partial}{\partial x_j}\left(\Gamma_\phi \frac{\partial \phi}{\partial x_j}\right)+S_\phi \qquad (1)$$

where, ϕ is the general flow variable such as velocity components, temperature, mixture fraction and its turbulent pulse, etc. The effect of turbulence in the fluid is accounted for in the transport coefficient Γ_ϕ by a turbulence model, and S_ϕ is a general source term which to be given in table 1.

Boundary Conditions

Dirichlet conditions are imposed at the inlet boundary, the Numann conditions were applied at symmetry boundary conditions, no-slip wall boundary conditions with a wall function approach were used at the wall and k, ε are handled by the wall function. The mass conservation equation was imposed at the outlet plane to obtain the outlet velocity. The intensity boundary condition for the radiative transfer equation (RTE) for the symmetry condition is a reflecting boundary, and a diffusely emitting-reflecting boundary for gray surface.

Numerical Approach

The model equations are solved by a finite volume method employing the SIMPLE algorithm (Patankar, 1980). The couple solving method is used in the whole numerical simulation. Using this approach the governing equation of continuity, momentum, mass fraction and it's pulsing combined PDF, energy, RTE heat transfer, and gas physics property such as density, thermal specific capacity, and absorption coefficient are coupled together and solved simultaneously. A FORTRAN program was developed by the authors to obtain the solution of coupled equations.

RESULTS

In order to investigate the change in flow patterns change due to the different velocity ratios of fuel to air jet, the isothermal flow field was simulated and analyzed under

Table 1: The expressions of Γ_ϕ , S_ϕ

ϕ	Γ_ϕ	S_ϕ
u	μ_e	$\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial x}\right)-\frac{\partial p}{\partial x}$
v	μ_e	$\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial y}\right)-\frac{\partial p}{\partial y}$
w	μ_e	$\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial z}\right)-\frac{\partial p}{\partial z}$
k	μ_e/σ_k	$\mu_e\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\rho\varepsilon$
ε	μ_e/σ_ε	$\left(C_1\mu_e\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-C_2\rho\varepsilon\right)\varepsilon/k$
f	μ_e/σ_f	0
g	μ_e/σ_g	$C_{g1}\mu_e\left[\left(\frac{\partial f}{\partial x}\right)^2+\left(\frac{\partial f}{\partial y}\right)^2+\left(\frac{\partial f}{\partial z}\right)^2\right]-C_{g2}\rho g\varepsilon/k$
m_{fu}	μ_e/σ_{fu}	$-C_{EBU}\overline{m}_{fu}\rho\varepsilon/k$
h	μ_e/σ_h	$k_a\sum_{i=1}^{N}w_iI_i-4k_a\sigma T^4$

different fuel and air jet conditions. Based on this, the combustion flow pattern was simulated and analyzed. For isothermal flow, three ratios of fuel to air jet conditions were modeled and are given in table 2.

Table 2: The operation parameters

Parameters	Condition1	Condition2	Condition3
Air jet velocity u_{10} (m/s)	20.00	20.00	20.00
Fuel jet velocity u_{20} (m/s)	0.000	20.00	8.30
Jet ratio U_{20}/u_{10}	0.00	1.00	0.42

Figure 2 shows the three flow patterns on the symmetry surface one for each of the given three operating conditions. The circulation loops were observed as figure 2(a) when no fuel gas jet into the furnace. Figure 2(b) for a jet ratio u20/u10≤0.42, i.e. u20 magnitude is less then u10, the fountain flow pattern is formed. In figure 2(c), for a jet ratio u20/u10=1, the penetration flow pattern is shown, in this case the momentum of the fuel jet is high enough to overcome the momentum of the reverse flow of the recirculating fluid so that the penetration flow pattern is formed. After penetration, the size of the recirculation shrinks with the increase in the fuel jet velocity. High temperature air combustion (HiTAC) may occur in this zone because of the recirculating combustion products mix with air and generate the lower oxygen concentration.

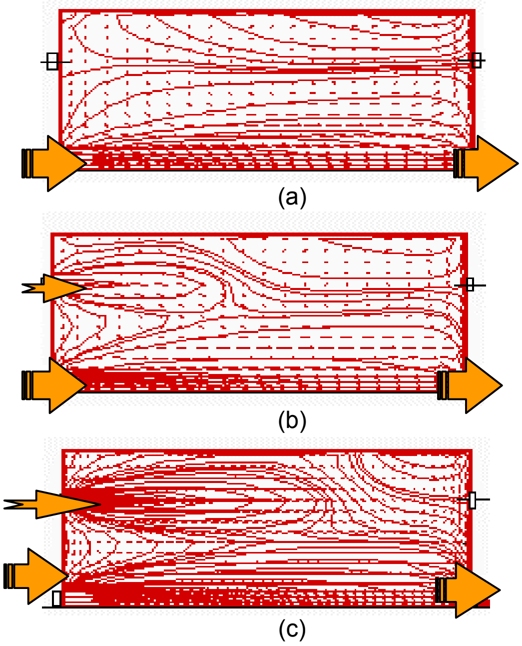

Figure 2: Effect of different jet ratios on flow field (a) u_{20}/u_{10}=0.00, (b) u_{20}/u_{10}=0.42, (c) u_{20}/u_{10}=1.00

The influence of combustion on the flow pattern on symmetry surface one is shown in figure 3. Air inlet temperature is 1000℃ with u20/u10=1 the same as the cold flow situation. The penetration distance of the fuel jet is hence shorter in burning than for the cold flow simulation.

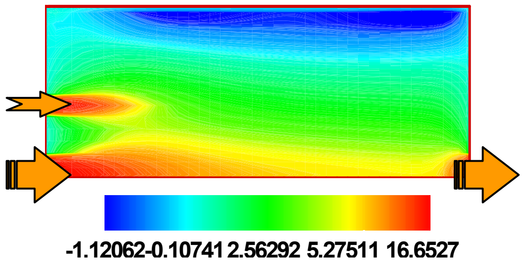

Figure 3: Influence of thermal drag on the flow

In the same case, the penetration depth is strongly influenced by combustion because of the thermal drag effects on the fluid flow (Guo Z. Y.,1991). For this case, fuel preheated to a high temperature may penetrate the thermal drag of recirculation zone. Where it can react with the lower oxygen concentration to generate a high temperature combustion (HiTAC) region.

For the final case the fuel inlet temperature is set to 1000℃, the same as the air inlet, and the velocity ratio is u20/u10=16/20. For this case the fuel jet may penetrate the recirculating zone as shown in figure 4 on symmetry surface one, and HiTAC may occur under this condition.

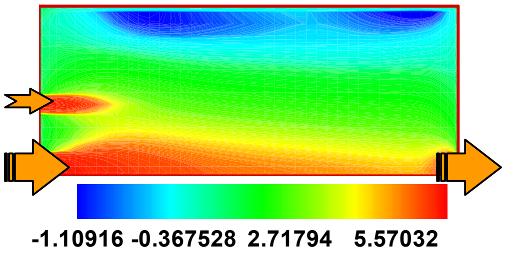

Figure 4: Preheated fuel alleviates thermal drag

Compared with the experiment results (Huang, 1997) under the condition of double concentric jets but with a similar profile, the predicted flow patterns agree with the experiment flow patterns.

CONCLUSION

Three-dimensional simulation for a model HiTAC small scale furnace has been undertaken to determine the flow pattern of isothermal flow and combusting flow. The simulated results show that the jet velocity ratios influence the velocity profile. The fountain flow and penetrate flow pattern are predicted at the ratio of u20/u10≤0.42 and u20/u10=1 respectively. The analysis shows that when the air jet velocity magnitude is equal to that of fuel jet velocity, the fuel jet may penetrate combustion product zone and HiTAC may occur under this circumstance. For combusting flow, the thermal drag influence the velocity profile, the fuel jet penetration length is shorter in than in cold flow case. The predictive flow patterns are agreement with the experimental results under similar conditions.

REFERENCES

MORITA M., (1996). " Present status of high temperature air combustion and regenerative combustion technology ", *The Energy Conservation*, 48, 21-28.

KOBAYASHI H., YOSHIKAWA K., (2000), " Numerical simulation for thermal performance of high temperature air combustion boiler ". *Proc. Int. Cong. on ACFD,* Beijing, China, October 714-719.

GUPTA A. K., BOLZ S., (1999). " Effect of air preheat temperature and oxygen concentration on flame structure and emission ". *Journal of Energy Resources Technology*, 121, 209-216.

NEWBY J.N., (1987), " High-performance heat recovery with regenerative burners ". *Iron and Steel Engineer*, 1987, 64, 20-24.

HUANG R. F., YANG J. T., LEE P. C., (1997), " Flame and flow characteristics of double concentric jets ", *Combustion and Flame*, 108, 9-23.

ELDING M.A.S., SPALDING D.B., (1979), " Computations of three-dimensional gas-turbulent combustion chamber flows", *Journal of Engineering for Power*, 101, 326-336.

FIVELAND W.A., (1984), " Discrete-ordinates solutions of the radiative transport equation for rectangular enclosures ". *J. of Heat Transfer*, 106, 699-706.

TRUELOVE J.S., (1988), " Three-dimensional radiation in absorbing-emitting-scattering media using the discrete-ordinates approximation ". *J. Quant. Spectros. Rad. Transfer,* 39, 27-31.

PATANKAR S V., (1980), " Numerical Heat Transfer and Fluid Flow ", New Yoyk, Hemisphere Pub. Corp.

GUO Z.Y., (1991), "Thermal drag in forced duct flow ", *J. Heat and mass Transfer, 34, 229-236.*