Numerical Investigation of the Entire Boiler System with SCR De-NO_x Reactor

CHENG Xingxing[1], JIN Baosheng[2]
(1 Southeast University, 210096, PR China. E-mail: xphcxx@yahoo.com.cn 2 Southeast University, 210096, PR China)

Abstracts: The numerical investigation of a 300 MW boiler with SCR de-NOX equipment is carried out with advanced methods. The simulation includes furnace, economizer, super-heater, re-heater, air pre-heater, SCR De-NO_x equipment, ESP, induced draft fan. A reasonable agreement has been attained when compared with the designed statistics, which proves the reliability of this simulation and implicates its utility value. As a result, an analysis tool is available to study the temperature and pressure characters of the whole boiler system, in a feasible and economic manner.

Keywords: Boiler system; SCR De-NO_x reactor; numerical simulation

1 INTRODUCTION

De-NO_x facilities have been used on many coal power plants, with much more concerns relating to environmental problems and more stringent legislations to control emission of NO_x. Selective catalytic reduction(SCR), with its high NO_x removal efficiency, has been used most widely among all the post-combustion gas clean-up processes. However, SCR reactor could change the pressure character of the flue gas pass, which would further impact the operation and safety of the whole boiler system.

Current simulations of boilers [1,3,6,7] are merely conducted on the furnace [1,3,6,7], and all of them are the systems without SCR reactor. Numerical simulation of boilers with SCR reactors has not been reported yet. There has been certain development on the simulation of flue gas system already, in which much simplification are used, such as assuming heat exchanging tubes to be porous material, and simplifying EPS to be one layer of meshes[8].

Advanced methods are used in this paper, and all the main parts related to flue gas, such as furnace, super-heater, re-heater, economizer, air pre-heater, SCR reactor, ESP, and induced draft fan , are included in the numerical model, which could be a helpful tool to analyze the temperature and pressure character of the whole boiler system. In the end of this paper, results gained from the simulation are compared to the actual values to validate the model, which show that errors are small and the methods used in this paper are feasible.

2 MATHEMATICAL MODELING OF THE BOILER SYSTEM

2.1 Case-study Boiler

Numerical simulation have been performed for a tangentially- fired coal boiler of 300 MW gross nominal load. Fig.1 displays a schematic arrangement of burners and heat exchanger sections. SCR reactor is located before air pre-heater and after economizer.

Air and coal enters the furnace from the four corners. The distribution of the nozzles is shown in Fig. 1, with 4 sections of primary air, 3 sections of secondary air and 3 sections of fuel-oil used during start-up. Fig. 2 displays furnace cross-sectional size and different orientations imposed for the entering streams.

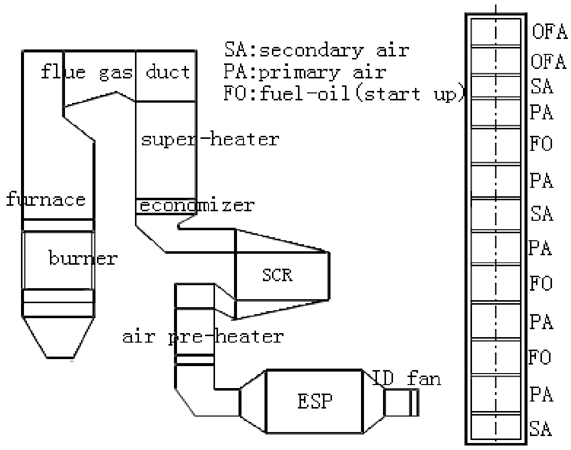

Fig. 1 Schematic arrangement of boiler system and burners

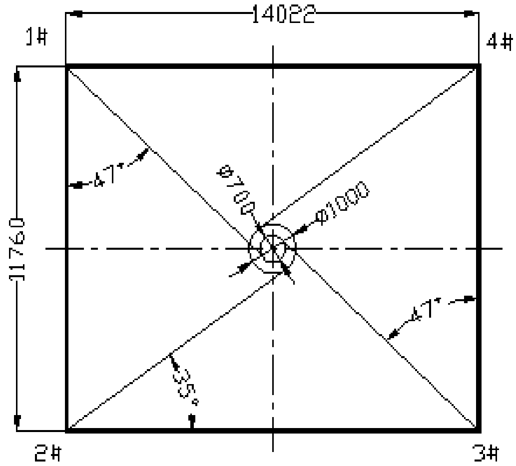

Fig. 2 Orientations of inlet streams

The furnace is designed to fire Shenhua coal, with the detailed parameters presented in Table 1.

Table 1 The furnace is designed to fire Shenhua coal, with the detailed parameters presented

V_{daf}	M_t	A_{ar}
23%	7.8%	28.95%
C_{ar}	N_{ar}	H_{ar}
54.246%	0.98%	3.060%
O_{ar}	S_{ar}	$Q_{net.ar}$
4.36%	0.644%	20916 kJ/kg

Parameters of primary and secondary air are as Table 2.

Table 2 Parameters of primary and secondary air

Velocity of primary air	25.4 m/s
Temperature of primary air	191 ℃
Total Mass flow rate of coal	47 kg/s
Velocity of secondary air	55.2 m/s
Temperature of secondary air	329 ℃

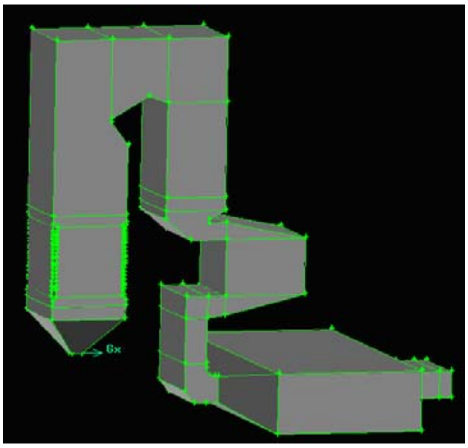

Fig. 3 Model of boiler system

2.2 Model Construction and Meshing

Three-dimension numerical model, as Fig. 3 illustrates, is constructed according to the real system with the ratio 1:1.

The tubes in flue gas ducts, which are numerous and only about 60 mm in diameter, are the most complex structure in the whole boiler. The effect of the tubes is too fine a geometric detail for any allowable discretization. Moreover, catalyst in SCR reactor is shaped as honeycomb with myriads thin walls and holes. Cases are similar in air pre-heater, which has a lot of heat exchanging panels in it, with the distances only about 3mm. If all theses sections are constructed exactly according to their real shapes, the number of the meshes will be too large to carry out the simulation. Therefore, in this paper, heat-exchanging and SCR regions are simplified. The meshing distribution is presented in Fig. 4.

The meshes are created separately in the sections of burner, upper part of furnace, upper flue pass, super-heater, economizer, SCR reactor and air pre-heater. By doing so, sizes and shapes of meshes in different sections could be determined to different geometry and calculation requirement. The meshing in burner and in the upper region of the furnace is displayed in Fig.5 and Fig.6, respectively.

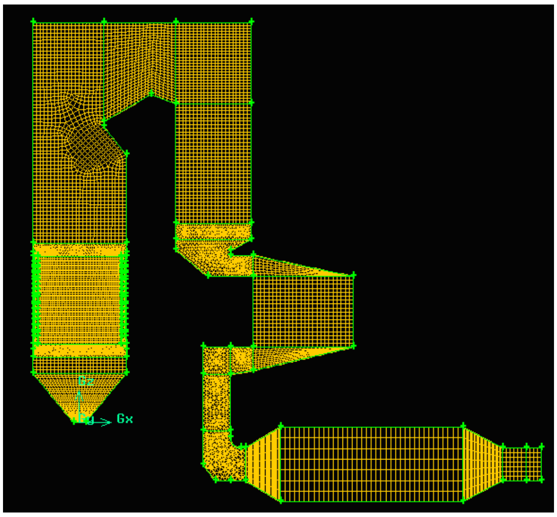

Fig. 4 Meshing of the whole system

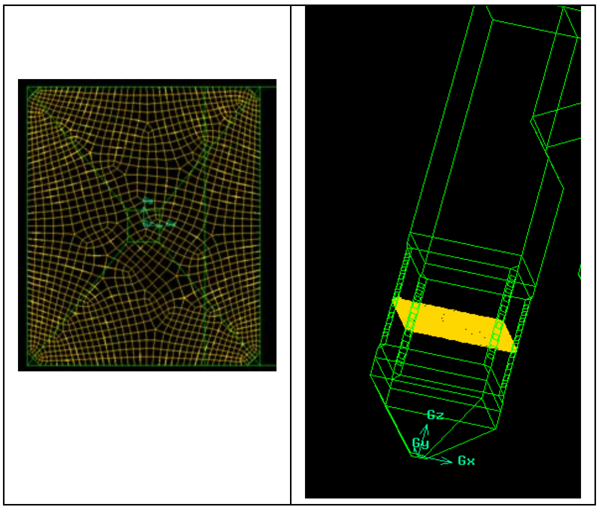

Fig. 5 Meshing scheme in burner region

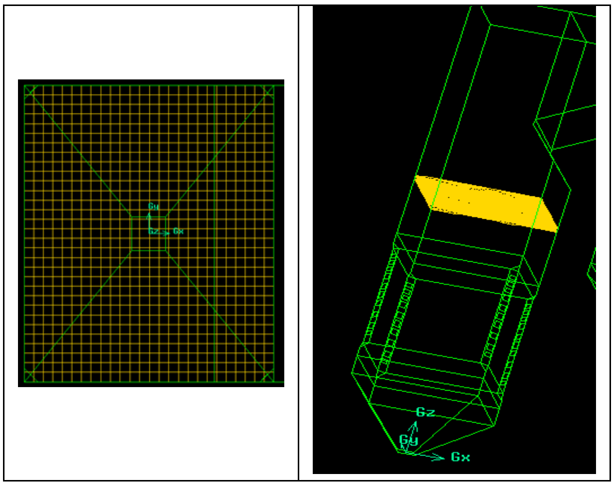

Fig. 6 Meshing scheme above burner region

3 MATHEMATICAL MODELING

A commercial CFD code has been used to simulate combustion, fluid and particle flow, heat and mass transfer in the system.

To calculate a numerical approximation to the fields of velocity, temperature and concentration of species, the discretized conservation equations for mass, momentum and energy are solved. Only time-averaged values are sought, assumed stationary, so that the Reynolds-averaged versions are adopted. Turbulence closure is effected by means of the standard k–e model. As for boundary conditions, axial velocities and temperatures are fixed on the inlet sections and standard wall functions are used for solid contours. Representative, constant values of temperature and emissivity are also imposed on the walls of the enclosure: 550℃ and 0.85, respectively.

3.1 Heat Exchanger Model

Simplified methods for the tubes in the gas pass were always used in the existing numerical simulations. Zhao[8] used a porous region to simulate the tubes, assuming both laminar (Poiseiuille) and inertial components of porous resistance. As for the heat exchanging, energy source was imposed to each mesh.

More accurate method is used in this paper as explained below. The fluid zone representing the heat exchanger core is subdivided into macroscopic cells or macros along the auxiliary fluid path, as in Fig. 7.

coolant passage

macro0	macro 1	macro 2	macro 21	macro 22	macro 23
macro 3	macro 4	macro 5	macro 18	macro 19	macro 20
macro 6	macro 7	macro 8	macro 15	macro 16	macro 17
macro 9	macro10	macro 11	macro 12	macro 13	macro 14

Fig. 7 Core discretized into macros

The streamwise pressure drop can be expressed as:

$$\Delta p = \frac{1}{2} f \rho_m U_{A\min}^2 \tag{1}$$

where,

Δp = streamwise pressure drop;

f = streamwise pressure loss coefficient;

ρ_m = mean gas density;

$U_{A\min}$ = gas velocity at the minimum flow area.

The NTU model calculates the effectiveness from the ratio of heat capacity and the number of transfer units using the relation.

$$\varepsilon = 1 - \exp\left[-\frac{1}{C_r} N_{tu}^{0.22} \left(1 - e^{-C_r N_{tu}^{0.78}} \right) \right] \tag{2}$$

This N_{tu} for the heat exchanger is scaled for each macro in the ratio of their areas.

The heat transfer for a macro is calculated from

$$q_{\text{macro}} = \varepsilon C_{\min} \left(T_{\text{in,auxiliaryfluid}} - T_{\text{in,gas}} \right) \tag{3}$$

where, εis macro effectiveness; $T_{\text{in,auxiliaryfluid}}$ is macro auxiliary fluid inlet temperature; $T_{\text{in,gas}}$ is macro gas inlet temperature.

The total heat rejection from the heat exchanger core is computed as the sum of the heat rejection from all the macros

$$q_{\text{total}} = \sum\nolimits_{\text{allmacro}} q_{\text{macro}} \tag{4}$$

The auxiliary fluid inlet temperature to each macro is computed based on the energy balance of the auxiliary fluid flow. For a given macro

$$q_{\text{macro}} = (\dot{m})_{\text{auxiliaryfluid}} (h_{\text{out}} - h_{\text{in}}) \tag{5}$$

where, h_{in} and h_{out} are the inlet and outlet enthalpies of the auxiliary fluid in the macro.

3.2 Fan Model

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon some larger flow field while structures in the fan is not considered. Empirical fan curve which governs the relationship between head (pressure rise) and flow rate (velocity) across a fan element is directly put into the model.

A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. the relationship is mostly of the form $P = \sum_{i=0}^{n} a_i v^i$, Where a_i is a coefficient, v is the flow velocity in the fan.

When fan speed is not changed, moving of fan operating point is achieved in a extremely short time. In this case, static fan equation could be used in dynamic model.

$$p = p_0 \left(\frac{n}{n_s} \right)^2 \left(\frac{\rho}{\rho_s} \right) - \frac{k^2 v^2 \rho}{\mu^2} \tag{6}$$

where p_0 is the static pressure when flow rate is 0, Pa; n_s is relative speed; v is gas flow velocity, m/s; andρis gas density, kg/m^3. Therefore, a_i in the fan model could be defined by equation (6).

4 RESULTS: VALIDATION AND DISCUSSION

The results of the simulation of the combustion of 300 MW boiler system at full load are shown as below.

Fig. 8 displays gas flow velocity vectors, indication combustion at different furnace regions. In these figures arrows represent gas velocity directions and colors represent gas temperatures.

Gas temperatures are relatively low at the bottom of burner region. Primary air and coal enters furnace from four corners and forms a circle. Coal starts to burn at the bottom of furnace, where temperature begins to rise and small region of high temperature is formed at the center. When comes to the upper part of burner region, gas entangled in the cycle is increased and the cycle's radium is also increasing. Gas temperature and radiation rise dramatically and region with high temperature is larger, which is presented red and yellow in the figures. In the burner, when the region is higher, the average temperature is higher. In top region of the burner, OFA air enters the furnace, forming a cycle which is in the reverse direction of the previous one. This cycle could diminish the rotation of the combusting gas and make the distribution of furnace temperature more uniform. Fig. 8 shows that in the region above OFA inlets, rotation of gas disappears and the temperature is homogeneous. All these results of velocity and temperature distribution exhibit quite realistic trends.

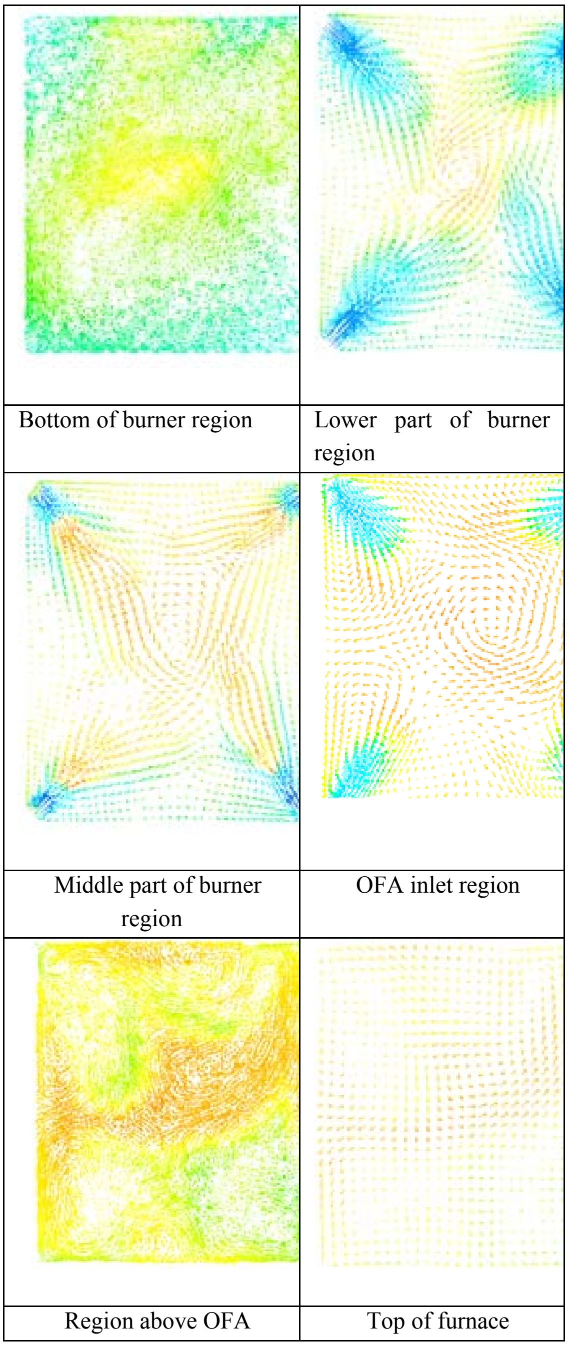

Fig. 8 Velocity vector in the furnace

Fig. 9 displays temperature throughout the entire computational domain. Temperature is quite high in furnace, where combustion occurs. Hot gas gives out heat and the temperature comes down through super-heater, economizer and air pre-heater. Temperature at the duct exist is about 120 □. Fig. 10 shows the comparison between plant values and simulated results of temperature at different sections of the boiler, which are A to G, as are represented in Fig. 9. The agreement is quite good, with all the errors less than 5%.

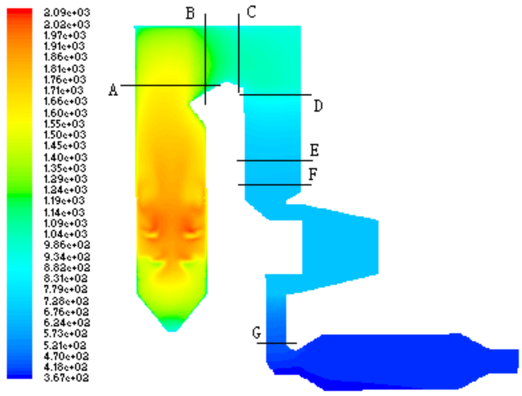

Fig. 9 Temperature distribution throughout the computational domain

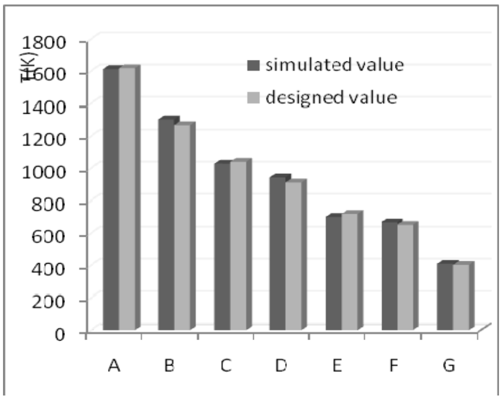

Fig. 10 Comparison between plant values and simulated results

Fig. 11 displays pressure throughout the entire computational domain. Average furnace pressure is -50 Pa, which could well agree with the actual values. Pressure decreases through super-heater, economizer, SCR reactor and air pre-heater. In all these sections, SCR reactor and air pre-heater have the biggest resistance and cause the largest pressure drop, about 1000 Pa. At the exit of the boiler, ID fan raises the gas pressure to about 0 Pa. Impact of stack is neglected here. Fig. 12 shows the comparison between plant values and simulated results of temperature at different sections of the boiler, which are A to H, as are represented in Fig. 11.

Most agreement is quite good, with errors less than 10%. The errors at C and D are both about 20%. But power plant data are possibly subjected to large uncertainties, that are difficult to estimate due to the physical effects involved (flow stratification, directed thermal radiation and backflow in heat balances, lack of calibration of large flow meters, etc.) The scarce literature dealing with this issue [2,4] indicates that errors in a commonly accepted value can surpass 10% or even reach 20%. So the errors at C and D are acceptable.

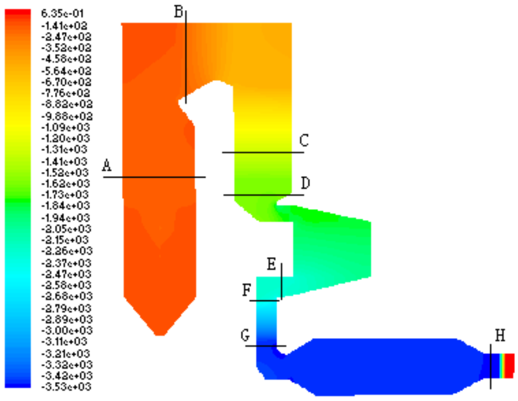

Fig. 11 Pressure distribution throughout the computational domain

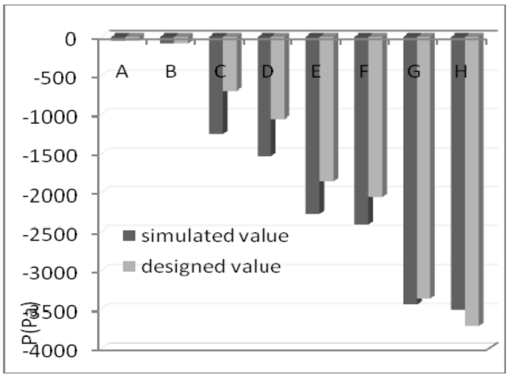

Fig. 12 Comparison between plant values and simulated results

5 CONCLUSIONS

A 3D numerical simulation for the thermal performance of a whole system of pulverized-coal, tangentially-fired utility boiler has been completed, specifically aiming at the prediction of temperature and pressure distribution. Both SCR reactor and ESP are taken into the simulation, which were never done before. Once compared with actual values in the designed system, validation of numerical results exhibits quite realistic trends. As a final result, a reliable computational tool is available to assist in the analysis for the flow field in the whole system of full-scale utility boilers.

REFERENCES

1. Abbas A S, Lockwood FC. Prediction of Power Station Combustors. 21st Symp. (Int.) on Combustion, the Combustion Institute, Pittsburgh, 1986: 285-292.
2. Chojnowski B, Soloukhin RI, Afgan NH. Research and operational measurements on modern power station boilers: a selective survey. In: 15th ICHMT. 1985. 465–489.
3. Coimbra C F M, Azevedo R A, Carvalho M G. 3-D numerical model for prediciting NOx emissions from an industrial pulerized coal combustor. Fuel, 1994, 7(3).
4. Dı′ez L, Corte′s C, Arauzo I, Valero A. Combustion and heat transfer monitoring in large utility boilers. Int J Therm Sci 2001; 40(5): 489-496.
5. Dı′ez L, Corte′s C, Campo A. Modelling of pulverized coal boilers: review and validation of on-line simulation techniques. Appl Therm Eng 2005, 25: 1516-1533.
6. Fiveland WA, Wessel RA. Numerical Method for Predicting Performance of Three Dimensional Pulverized-Fuel Fired Furnace. Journal of Engineering for Gas Turbines and Power, 1988, 110: 117-126.
7. Gorner K, WZinser. Prediction of Three-Dimensional Flows in Utility Boiler Furnaces and ComParison with Experiment . Comb. Sci. and Tech., 1988, 58: 43-47.
8. Zhao Zhen-zhou, Numerical and experimental simulation research on the influence of flue gas desulphurization system to boiler implosion. sep.2007.