Performance Analysis on Combustion and Heat Transfer of the Ship Power Plant

Lili Liu 1, Liyang Zhang 2

1 Intelligent Manufacturing Institute, Qingdao Huanghai University, Qingdao 266427, China
2 Datang Huangdao Power Generation Co., Ltd, Qingdao 266427, China
E-mail: S311037008LLL@163.com

Abstract. Study results on the supercharged boiler numerical simulation are on the boiler, convection tube bundle or the super-heater. These study subjects in a single model, and simplified the boundary condition to a certain extent. This paper takes the boiler as a whole research object. On the furnace chamber for combustion simulation, using the profile boundary information file boundary condition setting, faithfully transmitting boundary parameter spatial distribution. Followed by the simulation of heat transfer between the convection tube bundle and super-heater modules, we can get a real fluid flow and heat transfer condition, thus reacting to the complex heat transfer and flow inside of boiler, knowing the distribution of temperature, speed, pressure and other parameters. We can also see the change rule of the combustion and heat transfer processes in that supercharged boiler. Then we discuss the impact of the not uniform heat load on the convection heating surfaces, and preliminary analyzes the mechanism of combustion and heat transfer in supercharged boiler.

1. Introduction
The most obvious difference between supercharged boiler and atmospheric pressure boiler is that the air, supporting combustion have a certain pressure and temperature. In order to meet the requirements, people must use the turbine compressor instead of turbine blower to conveying of high temperature and high pressure combustion air. As a result of supercharged boiler using pressurized combustion system, providing high pressure, high density, high temperature combustion air, the combustion pressure can reach 0.2~0.6MPa, strengthening the combustion and heat transfer significantly [1].

The supercharged boiler numerical simulation studies regard a module (such as furnace, super-heater) as the research object, simulate separately. Such as in the super-heater, set the flue gas inlet boundary condition, often assuming a uniform distribution or according to the experience, using of user defined function to define non homogeneous boundary conditions for simplification [2]. It will enlarge the simulation results and the actual deviation. Combustion is a complex process with chemical reaction, boiler furnace combustion flow change with different working conditions, simulation of convection tube bundle or super-heater flow and heat transfer must from the combustion results. If taking the burning conditions into account, the results will be more authentic and accurate. Based on it, this paper studies the whole supercharged boiler combustion and heat transfer, and it will be helpful to supercharged boiler design and operation.
2. Research methods

Using the profile method of document need to the putout module one simulation results as a profile boundary information file, then import it to module two, as its simulated initial boundary conditions. On the numerical simulation of furnace, put the exit flue gas’s temperature, velocity and some other main parameter as the boundary information file profile. Again, in the simulation of convection tube bundle, using the profile as the gas inlet boundary conditions. Realizing the furnace exit gas information is passed through the convection tube bundle. Similarly, using this method pass the outlet flue gas distribution information of convection tube to the super heater’s entrance successfully. Numerical simulation on the overall 3D super heater steam flow field, we can get the steam mass flow rate distribution in parallel tubes directly.

Use PRO/E to establish the simulation object--overall supercharged boiler, including hearth, convection tubes, as well as three processes super heater. The whole model structure was shown in Figure1. But limited by the memory and performance of computer, we have to divide it into three parts.

![Figure 1: Numeration model](image1)

We complete the grid by dividing the geometric model into three parts, then meshing them one by one. Using the hex grid or hexahedral mesh/wedge in the simple structure as far as possible[3]. Applying tetrahedron grid in the key areas where the speed or the temperature change much, such as flame holder, tubes wall, the inside structure of top cases. Next, we need to refine the grid. Figure 2 shows the grid of each model, the total number of the grid for hearth is 1.2 million, the grid number of flue gas area embodying the convection tubes is 830 thousand and the total grid number of superheater tubes and the flue gas area is 2.48 million.

![Figure 2: Grids of each model](image2)

(a) burner and end surface of the hearth
During the simulation, using a discrete phase model to simulate fuel injection; selecting the rapid reaction of the simplified PDF model simulate furnace combustion process with cyclone; using modified realizable turbulence model simulate the aerodynamic field in furnace; considering the heat radiation on the gas temperature and wall temperature influence, using DO radiation model for calculating radiated heat transfer[4]. Discrete the control equation with two order upwind. Use SIMPLE algorithm to discrete equation. On both sides of super heater tube wall, the different fluid coupling calculation[5].

3. Numerical simulation on furnace burning and heat exchange

3.1 Set of boundary conditions
The imported of fuel entrance and air entrance use mass flow type, respectively. Give the quality of the fuel flow and air quality flow, and give injection temperature, injection pressure, half atomization angle and excess air coefficient, also some necessary parameters; gas exit is pressure export type, export absolute pressure is 0.285MPa; Water wall is the isl-thermal wall, internal emission rate is 0.8. Other walls are heat insulation.

3.2 Simulation result analysis

3.2.1 Air dynamic field and the recirculation zone

As the streamline chart shown in Figure3, hedge arranged swirl burner for rotary flow jet cause six central re-circulation zone at the outlet of the negative pressure, the reason is the air distribution device mounted the tangential vanes. The furnace chamber combustion air along the tangential vanes produce rotational flow air, the result that the swirling flow of the radial and axial produced pressure gradient. When the jet is more intense, due to the axial pressure gradient increases, the fluid will occur on the back in the axial direction, and in the vicinity of the nozzle appears circumferential. The largest two re-circulation zone is located on both sides of the burner nozzle near a rotating jet diffusion angle, the left boundary length is about 1.0m, the right boundary length is about 0.9m. This is because the
two jets in the center of the flow turbulence enhancement collided, enhanced the late perturbation and mixed.

3.2.2 Exports of flue gas velocity, temperature distribution

Figure 4. Flue gas temperature distribution of furnace exit / K
Figure 5. Flue gas velocity distribution furnace exit m/s

Because of the effect of hedge arranged burners and re-circulation zone, the central section combusted most intense\cite{6}. Most of flue gas outflow from middle area. This part of the flue gas has high flow rate and mass flow rate. The temperature level is the highest. It also can be seen from Figure 4 and 5, there is a re-circulation zone near the burner. Thus the gas velocity of this section is low, forming the velocity distribution that middle is higher than both sides. While the flue gas temperature in bottom of furnace is lower due to cyclone burner drives the air more first passed through the top of furnace, flue gas did not filled with the bottom of furnace enough, so the temperature is relative lower.

3.2.3 Dealing with the furnace exit plane

Export parameters and composition of the flue gas from furnace exit surface, then form the boundary information file: profile1. The parameters of the flue gas will serve as the flue gas inlet boundary condition on numerical simulation of the behind convective evaporation tube bundle heat transfer; the mass fraction of each component as the working medium flue gas which flows through the convection tubes.

4. Numerical simulation of convection heat transfer on convection heating surface

4.1 Numerical simulation of convection tubes

In determining whether the success of passing a boundary condition by Profile1 file, we measured the mass of connection surface existing in different modules and connecting surface temperature.

|                  | Exit report of furnace | Entrance report of convection tube | Error % |
|------------------|------------------------|-----------------------------------|--------|
| Pressure /Pa     | 184989.8               | 185062.8                          | 0.04   |
| Temperature /K    | 1934.43                | 1930.92                           | 0.18   |
| Mass flow /kg/s   | 13.88                  | 13.883                            | 0.02   |

Table 1. Connecting surface error

Table 1 indicate that surface error of mass flow, temperature and pressure are very small. Therefore, conditions can be successfully delivered to the convection tubes from the furnace exit.

4.2 Simulation results
Figure 6. Flue gas temperature distribution of convection tubes outlet /K

Through the comparison of Figures 4 and 6, when the flue gas flow through convection tubes, as the effect of tubes, flue gas temperature difference between original outlet bottom of furnace and the central become smaller. But the temperature distribution form that intermediate temperature higher than temperatures on both sides has not changed. Moreover, large temperature gradients exist in central.

5. Numerical simulation results of super heater

It can be seen from Figure 7, the average velocity of the third process away from one end of main steam valve is a little smaller than the one closer to main steam valve side. Due to the former steam take a slender connecting pipe bending to reach the steam outlet, in this process the flow resistance increases, velocity dropped[7].

Figure 7. Steam outlet velocity distribution

Because the heat transfer coefficient in steam tube side bigger than flue gas side, therefore it plays a decisive role on heat transfer coefficient of the outer surface of the tubes, including convective heat transfer coefficient and radiate heat transfer coefficient[8][9].

Figure 8. Outside surface heat transfer coefficient of tubes

From the Figure 8, we can see the heat transfer coefficient outside of the tubes near the furnace wall has increased. This is because the certain distance between the furnace wall and the lateral tubes wall, it is known as smoke corridor. It formed in bundles, with a larger flow of flue gas section between the individual snake pipes. Then the flow resistance becoming smaller and flue gas velocity accelerated. So the convective heat transfer coefficient of the tubes wall close to the outside corresponding larger,
therefore overall surface heat transfer coefficients corresponding bigger.

The Figure 9 shows that the tubes wall temperature of the first two rows is minimum, the second process wall temperature is first-class ride high, the third process tubes wall temperature is highest. And the temperature of two rows tubes wall in first process are similar. This is because the wall temperature limited by steam temperature level and flue gas temperature level two aspects. The first-process steam has uniform intake condition. Although the first process two rows tubes contact flue gas in turn, steam temperature level is very low and flue gas velocity is fast, so wall temperatures do not separate layers. First process tubes wall temperatures are not consistent. This is because the flue gas temperature in middle is higher than both sides. The corresponding tubes wall temperatures are also higher than both sides. The steam continue to be heated through the three processes in turn, so the third process wall temperature is higher than the second process average level, the second process wall temperature average level is higher than the first process. The highest wall temperature is 880 K, located in the third row away from steam export side of the third process.

From the Figure 10, it is shown the flow deviation coefficient of the third process is bigger. The largest flow deviation coefficient located in third row tube which is near the steam export side. The branch tubes are II connection type, so the position of minimum flow tube is relative away from the exit of the steam, the maximum flow tube located in the end near the steam exit. And because the entrances of the third process gates box are oblique incision, the steam speed near the incision is higher, the mass flow is largest, so flow deviation is the largest.

Contrasting Figure 10 and 11, obviously, the flow deviation and the heat deviation distribution of third process have contrary trends. This is because absorb the same quantity heat, tubes with small steam flow, temperature elevated larger, enthalpy increases. It is shown from Figure 10, the flow deviation of first and second processes are small, so the thermal deviation is mainly affected by the influence of the not uniformity hot flue gas. The third process flow deviation is the largest, on the common influence of the uneven flow and heat, the thermal deviation is larger.

Combined with figure 11 and figure 9, it can be found that the maximum thermal deviation tube and the highest temperature tube is the same one, located in the third flow central, away from the steam export. It has small steam flow, high heat load and steam temperature, big thermal deviation and high wall temperature, so it becomes the most dangerous tube in super heater.

6. Conclusions
(1) The effect of furnace combustion can be taken into account through the successful delivered of the profile. Realize the pressurization boiler combustion and overall numerical simulation of heat.
(2) This paper reflects the fact that the heat uniformity of export decreased after the convection tubes, but its influence on thermal deviation of super heater is still big.
(3) Preliminary make sure the position of danger tube in super heater. This study will provide a reference to the design, operation and maintenance of super heater.
References
[1] Wang J.Z.(2006) Marine Pressurization Boiler Heat Balance Calculation. Thermal energy and power engineering,05:470-472+544.
[2] Purbolaksono J., Khinani A., Rashid A.Z., Ali A.A.Ahmad J., Nordin N.F.(2009) A New Method for Estimating Heat Flux in Superheater and Reheater Tubes. Nuclear Engineering and Design, 10:1879-1884.
[3] Wang F.J. (2004) Analysis of Computational Fluid Dynamics. Tsinghua University Press. Beijing.
[4] Li Z.G., Jing R.Q., Li Y.J., Sun B.Z. (2006) Marine Boosting the Boiler Overheating in the Flow Field Numerical Simulation and the Flow Rate Distribution Regularity Study. China Shipbuilding, 6:34-43.
[5] Li Y.J., Zhang G.L., Cao M.X. (2008) Numerical Modeling and Analysis of Thermal Deflection for the Overheater of Supercharged Boiler. Journal of engineering thermophysics. 12:2115-2118.
[6] Hwajin Kim, Jin Young Kim, J.S. Kim, Hyoun Cher Jin. (2015) Physicochemical and Optical Properties of Combustion-generated Particles from a Coal-fired Power Plant, automobiles, ship engines, and Charcoal kilns. Fuel, 161:120-128.
[7] Wu D., Li Q., Ding X., Sun J.F., Li D. Fu H.B. (2018) Primary Particulate Matter Emitted from Heavy Fuel and Diesel Oil Combustion in a Typical Container Ship: Characteristics and Toxicity. Environmental science & technology, 52:12943-12951.
[8] Byung Chul Choi. (2016) Thermodynamic Analysis of a Transcritical CO2 Heat Recovery System with 2-stage Reheat Applied to Cooling Water of Internal Combustion Engine for Propulsion of the 6800 TEU Container Ship. Energy, 107:532-541.
[9] Yu J., Kong L., Zhu Q.S., Zhu H.J., Wang H.Q., Guan J.L., Yan Q. (2020) Thermal Stress Analysis of a Segmented Thermoelectric Generator under a Pulsed Heat Source. Journal of Electronic Materials, 7:4392-4402.