Simulation study on combustion of biomass

M L Zhao[1], X Liu[1], J W Cheng[2,3], Y Liu[2] and Y A Jin[2]

[1]Jilin Province Electric Research Institute Co. LTD, Changchun, 130021, China
[2]College of Automotive Engineering, Jilin University, Changchun, 130025, China

Email: 1055236456@qq.com

Abstract. Biomass combustion is the most common energy conversion technology, offering the advantages of low cost, low risk and high efficiency. In this paper, the transformation and transfer of biomass in the process of combustion are discussed in detail. The process of furnace combustion and gas phase formation was analyzed by numerical simulation. The experimental results not only help to optimize boiler operation and realize the efficient combustion of biomass, but also provide theoretical basis for the improvement of burner technology.

Keywords: biomass combustion, numerical simulation analysis, boiler.

1. Introduction

The burning phenomenon is the interaction of complex physical and chemical processes. Combustion is a violent exothermic oxidation reaction, but the related physical processes, specifically the exchange of energy, mass and momentum, also play an important role in the combustion system. The combustion model of the actual processes is determined almost entirely by turbulence. There is a strong interaction between turbulence and the gas phase. Heat released by the reaction can affect turbulence [1]. The turbulence is also influenced by the mixing of the reactant and the product. The k−ε equation model is most often used to solve the flow problems associated with turbulence [2-3].

2. Simulation calculation conditions

In this paper, the combustion process of a small biomass boiler is simulated by CFD software. In the calculation process, the mass fraction of O_2 is set to 23.3%, the mass fraction of N_2 is 76.7%, the initial temperature of each region is 283.15 K, and the pressure relative to the air pressure is 0 Pa. The boundary conditions are set as follows.

2.1. Boundary condition

Solid wall, velocity slip and no-penetration conditions are assumed. Therefore, relative to the air, solid wall tangential velocity and normal velocity components are zero. The boundary condition for the k−ε equation is $\frac{\partial k}{\partial y} = 0$.

2.2. Entrance boundary conditions
Inlet boundary conditions are the combustion chamber and the chamber of a wind and the two winds at the entrance. The turbulence parameters at the entrance are $I = 10\%$, $l = 0.01m$. I is the turbulent intensity; l is the turbulent mixing length.

The turbulent kinetic energy κ and the dissipation rate of ε are given by the following formula:

$$\kappa = 1.5\times(U\times I)^2 \varepsilon = C_u^{0.75}\times\frac{\kappa^{1.5}}{l} C_u = 0.09$$

2.3. Exit boundary conditions
The exit boundary is taken as the cross section of the furnace outlet. In the field of export, the distribution of fluid parameters is not generally known in advance. In many cases, the mixture parameter distribution in the outlet section is needed to figure out the exact content [4].

3. Physical model and grid division
Figure 1 is an engineering drawing of the boiler. Figure 2 shows the three-dimensional geometry of the boiler. Figure 3 is a simplified diagram of the combustion chamber network and flue gas flow channel. The construction of these models has greatly improved the accuracy of the experimental results.

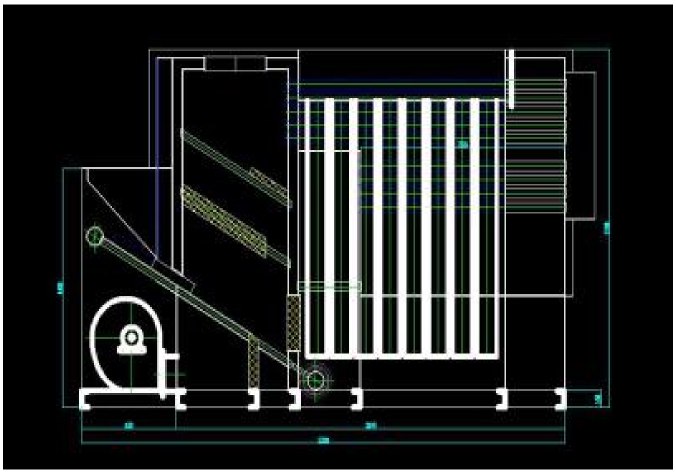

Figure 1. Engineering drawing of the boiler.

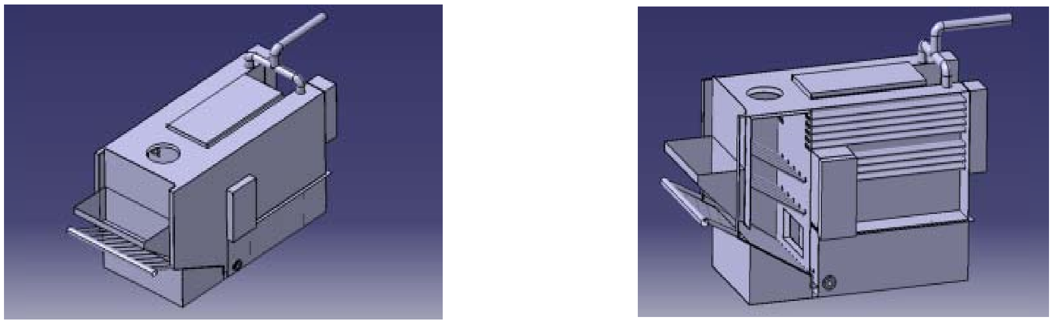

Figure 2. The three-dimensional geometry of the boiler.

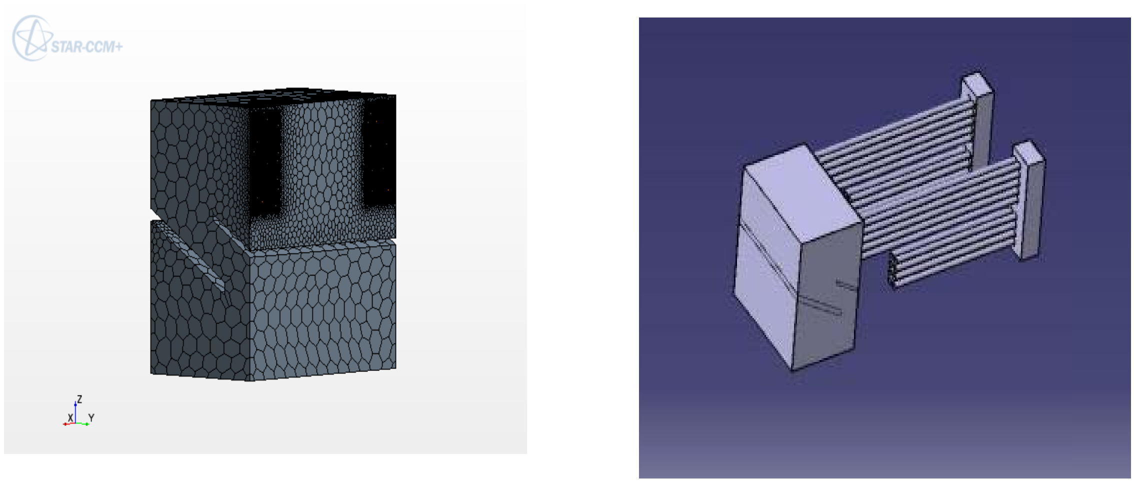

Figure 3. Simplified diagram of combustion chamber network and flue gas flow channel.

3.1. The establishment of the model and the analysis of the operation process

The model shown above was built by three-dimensional CATIA software, based on the design specifications shown in the drawing of the boiler. The operation process is simulated by setting specific inlet and outlet boundary conditions. For the control equation, the simulation equation is K. Due to its demonstrated effectiveness, the two equation model for solving flow problems was used to simulate the actual process.

4. Simulation results and analysis

4.1. Flow characteristics analysis

The flow line trajectory of the flue gas is S-shaped, as shown in Figure 4.

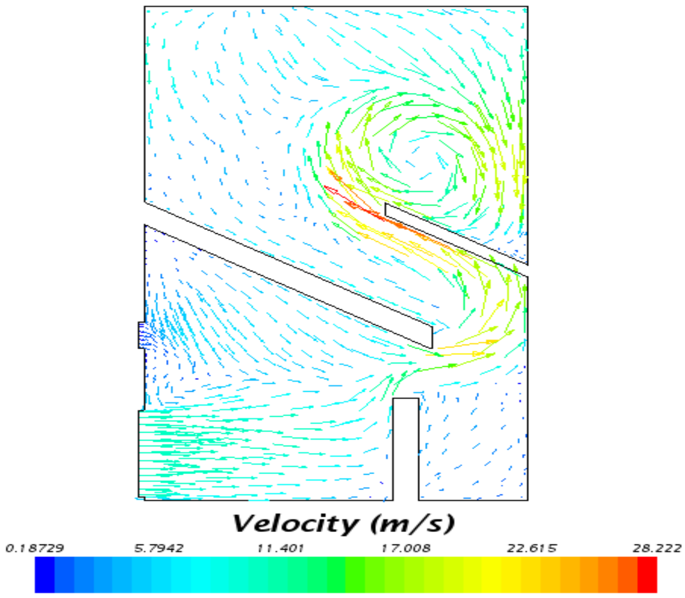

Figure 4. Velocity distribution in the combustion chamber.

It shows that the installation of fire-retardant material on the inclined water discharge in the combustion chamber has played a role in prolonging the distance of smoke. This can increase the heat transfer time and improve the efficiency of heat transfer. Therefore, more of the heat of the flue gas can be transferred to the water in the pipe wall.

4.2. Product distribution analysis of combustion

The product components distribution is shown in Figure 5 and Figure 6.

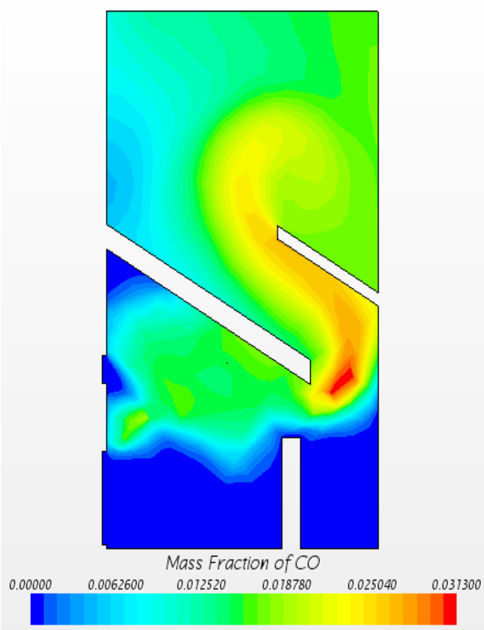

Figure 5. Distribution of CO in combustion chamber.

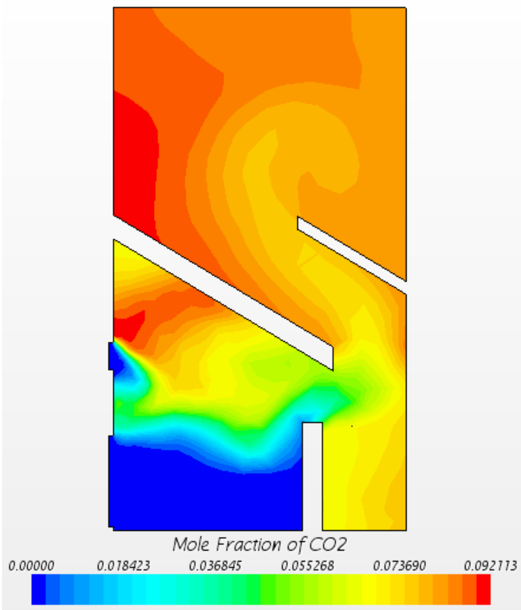

Figure 6. Distribution of CO_2 in combustion chamber.

By analyzing the distribution of gases in the combustion chamber, combustion can be optimized to burn as much of the fuel in the furnace as possible, reducing the production of carbon monoxide, and improving the thermal efficiency of combustion.

4.3. Analysis of combustion and heat transfer characteristics

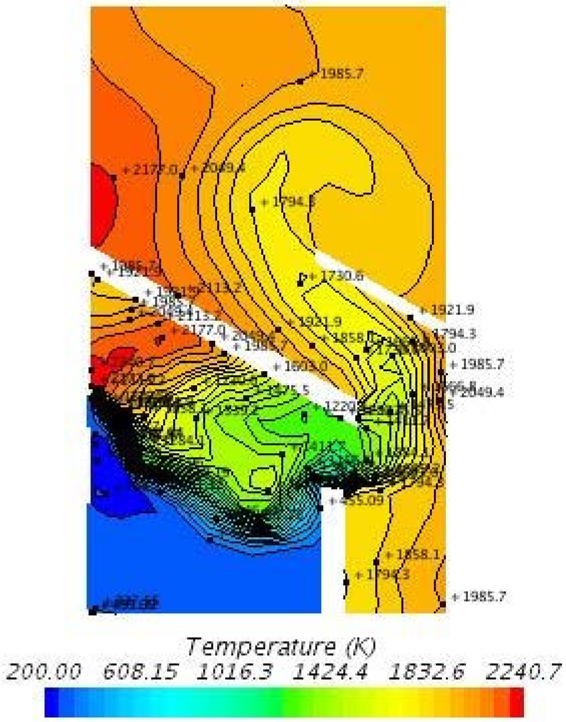

Figure 7. Temperature distribution of the combustion chamber.

From Figure 7 we can see that fuel and air enter the lower left side of the combustion chamber. The first combustion occurs after the maximum temperature has reached more than 1,000 K. The highest temperature in the first combustion chamber is found in the upper left corner. This is due to the application of fire-retardant material to the water channel, so that smoke travels upward along the S-shaped curve, resulting in the accumulation of high-temperature flue gas in the upper left corner of the chamber. As the smoke rises into the large space in the upper part of the chamber, more air can come in contact with the combustion fuel, and the temperature of the flue gas increases. The temperature distribution is S-shaped.

5. Conclusions

In this paper, the design specifications of a boiler were used to create a three-dimensional model of the boiler and set the specific boundary conditions. A numerical simulation using the equations of the turbulence model was conducted to simulate the cloud. Cluster cloud can be analyzed for any type of combustion chamber by changing the boundary conditions and other specifications in the model. This article will help to provide the basis for the research pertaining to boiler combustion and the improvement of the physical structure of the boiler [5].

Acknowledgement
The authors would like to thank the Project by Clean Development Mechanism (CDM) of China (1113050) and the Science and Technology Project of Jilin Province Development and Reform Commission (2013-729).

References

[1] Zhou Z J, Zhuo Y, Wang Z H, Hu X, Zhou J H and Cen K F 2014 Process design and optimization of state-of-the-art carbon capture technologies *Environ Prog. Sustainable Energy* **33**(3) 993-999
[2] Rajika J K A T and Narayana M 2016 Modelling and simulation of wood chip combustion in a hot air generator system *SpringerPlus* **5**(1) 1166
[3] Haseli Y, Oijen J A V and Goey L P H D 2011 A detailed one-dimensional model of combustion of a woody biomass particle *Bioresource Technology* **102**(20) 9772-9782
[4] Rajput P, Sarin M M, Sharma D and Singh D 2014 Atmospheric polycyclic aromatic hydrocarbons and isomer ratios as tracers of biomass burning emissions in Northern India *Environmental Science and Pollution Research* **21**(8) 5724-5729
[5] Wang X, Zhang H, Lin C J, Fu X W, Zhang Y P and Feng X B 2015 Transboundary transport and deposition of Hg emission from springtime biomass burning in the Indo-China Peninsula *J. Geophys. Res. Atmos.* **120**(18) 9758-9771