Numerical simulation study of quench tower in flue gas purification system

Peng Long¹, Zhiwei Li and Mengdi Li

School of Energy and Building Environment, Guilin University of Aerospace Technology, No. 2 Jinji Road, Guilin, Guangxi, China.

¹Email: longpeng@guat.edu.cn

Abstract. By establishing a numerical simulation model of the high-temperature flue gas spray evaporation process, the heat transfer and flow processes in the quench tower using 4 different numbers of atomizing nozzles were simulated. Research and analysis were conducted under specific conditions. Different number of atomizing nozzles, the flow field above the quench tower is different, the greater the number of atomizing nozzles, the smaller the area occupied by the air flow, the smaller the conical volume formed by the droplets, the less time it takes to fully evaporate. A single atomizing nozzle cools down faster within 2m from the top of the tower, have the longest trajectory length of the droplet movement, but the time for the droplet to completely evaporate is the longest.

1. Introduction

In the waste incineration process, a large amount of harmful substances will be generated during the incineration process, and its main components include dioxins, SO₂ and NOx [1]. In order to control dioxin emissions, on the one hand, the waste must be fully burned in the incinerator combustion chamber, on the other hand, it is necessary to suppress the generation of dioxin in the low-temperature flue [2]. In a pilot scale experiment of baking municipal solid waste at low temperatures (150-250°C), Kuo [3] et al. found that the efficiency of removing chlorine and mercury was 20.1% and 82.4%, respectively. Zhao [4] et al. investigated the formation mechanism of dioxin during waste incineration and found that the temperature range of dioxin regeneration is about 200-500°C, and the main activity range is 300-400°C. In order to reduce the production of dioxins, it is necessary to quickly cool the flue gas to below 200°C in a very short time.

Generally, the quenching process is completed by droplet atomization and heat absorption. As the core equipment of high temperature flue gas purification system containing dioxin, quench tower is very important to realize the rational design and optimized operation of quench tower system by studying the heat and mass transfer process between gas-liquid two phases in the spray evaporation process. Guo [5] et al. studied the effect of droplet size, droplet velocity and spray angle on the efficiency of deacidification by numerical simulation and experimental methods. Li [6] et al. simulated the flue gas-droplet mass transfer and chemical reaction in the deacidification tower of a waste incineration plant, and analyzed the influence of flue gas inlet velocity, stoichiometric ratio, droplet size and water spray rate on deacidification performance. Fabien [7] et al proposed a CFD water spray model to study the influence of relative humidity, spray angle, water mass flow rate and droplet size distribution on water droplet evaporation. Existing research mainly focuses on the influence of the size of atomized droplets, spray speed, spray angle, etc. on cooling with a single atomization nozzle. The
flow rate and speed change of a single atomizing nozzle is limited, and the performance changes in the quench tower may be different when multiple nozzles are used.

Therefore, the CFD calculation technology was used to establish the numerical model of the high-temperature flue gas spray evaporation process, and the heat transfer and flow process in a quench cooling tower with different numbers of nozzles were simulated. By studying the heat transfer and flow laws in the quench tower, it is of great significance to the design, production, operation and maintenance of the quench tower.

2. Numerical simulation method

2.1. Physical model

Because the research focuses on the spray evaporative cooling process of flue gas in the tower, ignoring the influence of the nozzle, waterway system and other small internal components on the flue gas flow field, the quench tower is simplified as shown in Figure 1. Flue gas flows in from the upper inlet and flows out from the bottom. The upper part of the tower is a truncated cone shape with a height of 1000mm; the cylindrical main tower of the quench tower is 10000mm high and the diameter is 2500mm; the flue gas inlet is a circular channel with a diameter of 1200mm, and an atomizing nozzle is arranged on the upper truncated cone. According to the calculation requirements, respectively there are 1, 2, 3 and 4 atomizing nozzles evenly arranged. The diameter of the ring of the cross section where the atomizing nozzle is located is 1850mm, and the distance from the flue gas inlet is 500mm. The atomizing nozzle is a conical nozzle with a spray angle of 30° and the spray direction is the same as the flue gas flow direction. The quench tower uses a structured grid. To ensure accuracy, the total number of grids is 497925.

![Figure 1. Schematic diagram of quench tower.](image)

2.2. Mathematical model

In the numerical simulation of the quench tower, the flue gas phase is solved using the Euler model. When the gas phase model converges, the discrete phase model (DPM) is used to simulate the droplet evaporation.

The control equations for the numerical solution of gas flow include: mass conservation equation, momentum conservation equation, energy conservation equation, component transfer equation, etc [8].

The particle trajectory considers the interaction of temperature, fluid action, energy, etc. through the particle trajectory movement under the Lagrangian system [9].
\[ m_p \frac{du_p}{dt} = \sum F_p + m_p g \]  

(1)

Where \( m_p \frac{du_p}{dt} \) is the inertial force, \( \sum F_p \) the combined external force including buoyancy, drag force, gravity gradient force, etc.; \( m_p g \) is the gravity of the particles.

The concentration of the particles is calculated as follows:

\[
\frac{d\rho_p}{dt} + \frac{\partial}{\partial x_j} \left( \rho_p u_p u_{p_j} \right) = N_i A_p M_w
\]  

(2)

\[
N_i = h_m \left( C_{i, w} - C_{i, w}^\infty \right)
\]  

(3)

Where \( \rho_p \) is the droplet density, kg/m\(^3\); \( t \) is the time, s; \( N_i \) is the molar diffusion rate of water vapor, kmol/(m\(^2\)s); \( A_p \) is the surface area of the droplet, m\(^2\); \( M_w \) is the molar mass of water, kg/kmol; \( h_m \) is the convective mass transfer coefficient, m/s; \( C_{i, w} \) is the molar concentration of water vapor on the droplet surface, kmol/m\(^3\); \( C_{i, w}^\infty \) is the molar concentration of water vapor in the flue gas, kmol/m\(^3\).

The particle momentum equation and particle energy equation are [10]:

\[
\frac{\partial}{\partial t} \left( \rho_p u_p \right) + \frac{\partial}{\partial x_j} \left( \rho_p u_p u_{p_j} \right) = \frac{1}{\tau_{rk}} \left( u_{vi} - u_{pi} \right) + \rho_p g_i + F_{k, M_0} + u_p S_m
\]  

(4)

\[
m_p C_p \frac{dT_p}{dt} = h A_p \left( T_w - T_p \right) + \frac{dm_p}{dt} h_{fg}
\]  

(5)

Where \( u_p \) is the particle velocity, m/s; \( u_{vi} \) is the gas velocity, m/s; \( \rho_p \) is the particle density, kg/m\(^3\); \( S_m \) is the mass flow rate of the particle source term, kg/s; \( C_p \) is the particle specific heat capacity, kJ/(kg·K); \( T_p \) is the particle temperature, K; \( h \) is Convection heat transfer coefficient, W/(m\(^2\)·K); \( T_w \) is gas temperature, K; \( \frac{dm_p}{dt} \) is the particle evaporation rate, kg/s; \( h_{fg} \) is the latent.

In the numerical simulation of the mass and heat transfer process of discrete phase droplet particles and flue gas, the following simplifications and assumptions are made: the discrete phase droplet particles are assumed to be spherical particles and there is no temperature gradient inside the particles; the wall surface is adiabatic wall, regardless of heat dissipation from the environment; does not consider radiative heat transfer. The boundary condition flue gas inlet adopts velocity inlet.

The inlet velocity of the flue gas is 2m/s, and the calculated Re is far greater than 2320, according to the literature [11], the standard \( \kappa-\varepsilon \) model is used for the turbulence model.

In order to explore the working performance in the quench tower when using different numbers of atomizing nozzles, four cases were designed and calculated, ensure the total mass flow of atomized water, the specific boundary conditions are set as shown in Table 1.

| Case | Water drop diameter (\( \mu m \)) | Number of nozzles | Flow rate of single nozzle (Kg/s) | Outlet (Pa) |
|------|---------------------------------|------------------|-------------------------------|------------|
| 1    | 100                             | 1                | 0.18                          | -200       |
| 2    | 100                             | 2                | 0.09                          | -200       |
| 3    | 100                             | 3                | 0.06                          | -200       |
| 4    | 100                             | 4                | 0.045                         | -200       |
3. Results and discussion

3.1. Velocity distribution

It can be seen from the velocity distribution on the central surface of the quench tower (Figure 2) that the high-temperature flue gas enters the tower from the top of the tower. The initial velocity is 2m/s. Since the upper part of the quench tower is a truncated cone, its cross-sectional area gradually increases, so the flue gas velocity is slightly reduced, and the surrounding airflow is driven back. At a distance of 0.5m from the top of the tower, the atomizing nozzle began to spray droplets at a speed of 5m/s, and the high-temperature flue gas and droplets moved downward in the same direction. The droplets spread outward in a conical shape, and there is an airflow rewinding outside, which is conducive to the heat transfer of the airflow in the tower, but there is a threat of erosion and erosion on the tower wall. In the area 5m below the top of the tower, the air flow is more uniform and flows vertically downward.

![Figure 2. Velocity distribution of central surface in case 1.](image)

It can be found from Figure 3 that the velocity distribution of the section at a distance of 1m from the top of the tower in four cases. The position directly below the atomizing nozzle is a high-speed area due to spray. The area surrounding the nozzle position has a high velocity due to the high temperature flue gas flowing above, and the outermost side is the low temperature air flow rewinding. The greater the number of atomizing nozzles, the smaller the area occupied by the air flow.

![Figure 3. Velocity distribution of the cross section at a position 1 meter away from the top of the tower in different cases.](image)

3.2. Temperature distribution

The temperature distribution on the central section of the quench tower is shown in Figure 4. It can be seen from the figure that after spraying, a large amount of heat is absorbed in the spray area at the
upper part of the quench tower due to droplet evaporation to form a typical conical low-temperature area below the atomizing nozzle. After the high-temperature flue gas passes through the atomizing nozzle, it comes into contact with the atomized droplets, and the temperature of the flue gas near the atomized droplets is lower; during the downward movement of the flue gas, the atomized droplets continue to flow from the flue gas. The heat is absorbed in the middle, so that the temperature of the flue gas is continuously reduced, and the droplets gradually evaporate into water vapor due to the heat absorption. At a position 4m away from the top of the tower, the temperature of flue gas and water vapor are basically the same.

![Temperature Contour](image1)

**Figure 4.** Central surface temperature distribution in different cases.

It can be found from the temperature distribution of the cross section at a position 1m away from the top of the tower (Figure 5) that during the downward movement of the flue gas, affected by the low-temperature droplets sprayed by the atomizing nozzle, the temperature at the position below the nozzle drops rapidly. At this time, the evaporation rate of the atomized droplets is small. High-temperature gas is concentrated in the central area of the tower.

![Temperature Distribution](image2)

**Figure 5.** Temperature distribution of the cross section at a position 1 meter away from the top of the tower in different cases.

In order to conveniently describe the temperature change along the height of the tower, the calculation formula for defining the temperature change rate is shown in formula (6).

\[
\text{Temperature change rate} = \frac{T_{i+1} - T_i}{T_0}
\]

(6)

Where \(T_0\) is the inlet flue gas temperature, 827k, and \(i\) is the section number of the distance from the top of the tower, \(T_i\) is the average temperature of the cross section.
As can be seen from Figure 6, case 1 has the fastest temperature drop at a position 2m away from the top of the tower. The temperature drop rate in the area of 2m to 3m remains at a relatively high level, and decreases sharply from 3m to 4m, and there is basically unchanged in the temperature below 4m. Case 2, case 3 and case 4 have similar changes. In the area 1-2m from the top of the tower, the temperature change is smaller than that of case 1, indicating that the number of nozzles has increased, and the evaporation heat absorption under the nozzle has not increased equally. In the area of 2-3m, the temperature change is greater than that in case 1, indicating that the vaporization of the atomized droplets in this area absorbs more heat than case 1. In the area of 3-4m, the temperature change is the same as case 1, and the change is rapidly reduced. Similarly, the temperature is basically unchanged in the area below 4m away from the top of the tower.

![Figure 6. Temperature changes along the height of the tower in different cases.](image)

It can be seen from the droplet particle trajectory that the droplets are ejected from the atomizing nozzle and are distributed in a cone (Figure 7). The smaller the number of atomizing nozzles, the greater the conical volume formed by the droplets. When a single atomizing nozzle, the longest trajectory length of the droplet movement. In the four cases, the atomized droplets did not move to the wall of the tower or to the bottom. They were completely evaporated in the area about 4m from the top of the tower.

![Figure 7. Trajectory diagram of droplets in different cases.](image)
The time for the atomized droplets to fully evaporate is shown in Table 2. It can be found that the more atomizing nozzles, the less time it takes to fully evaporate. Using two atomizing nozzles reduces the time to complete evaporation by 8.18% compared to using a single atomizing nozzle. Using 3 atomizing nozzles reduces the time to complete evaporation by 6.18% compared to using 2 atomizing nozzles. The use of 4 atomizing nozzles reduces the time to complete evaporation by 4.51% compared to the use of 3 atomizing nozzles. The greater the number of atomizing nozzles, the less the degree of complete evaporation time decreases.

Table 2. Total evaporation time in different cases.

| case | 1    | 2    | 3    | 4    |
|------|------|------|------|------|
| Total evaporation time (s) | 1.674 | 1.537 | 1.442 | 1.377 |

4. Conclusions

Through the numerical simulation of the working process of the quench tower using different numbers of atomizing nozzles, the following conclusions are obtained.

1. Around the cone-shaped distribution of atomized droplets, a local backflow will be formed, which is conducive to the heat transfer between the two phases of gas and particles. Different number of atomizing nozzles, the flow field above the quench tower is different, the greater the number of atomizing nozzles, the smaller the area occupied by the air flow.

2. The area within 4m from the top of the tower is the range of rapid temperature change in the quench tower. A single atomizing nozzle cools down faster within 2m from the top of the tower. From 2m to 3m, the change is slower than in other cases.

3. The smaller the number of atomizing nozzles, the greater the conical volume formed by the droplets. The single atomizing nozzle have the longest trajectory length of the droplet movement. The more atomizing nozzles, the less time it takes to fully evaporate. The greater the number of atomizing nozzles, the less the degree of complete evaporation time decreases.

Reference

[1] Jürgen V 2012 Reduction of dioxin emissions from thermal waste treatment plants: a brief survey Reviews in Environmental Science and Biotechnology 11 393-405
[2] Lin X Q, Li X D, Lu S Y, et al 2015 Dioxin emissions characteristics of a waste incinerator and prospects of the emissions reduction technology Journal of Engineering for Thermal Energy and Power 30(3) 329-332
[3] Wei-Chieh K, Janusz L, Krzysztof S, et al 2019 Low-temperature pre-treatment of municipal solid waste for efficient application in combustion systems Energy Conversion and Management 196 525-535
[4] Zhao Jirui, Zhang Ji, Liu Jiahong 2014 Synthesis and reduction of dioxin emissions in waste incineration process Environmental Science & Technology 37(120) 101-104
[5] Guo X T, Yin L J, Hu L H, et al 2020 Numerical simulation of wet deacidification process of sludge incineration flue gas Fuel 280(118480)
[6] Li C X, Jia Z N, Ye X M, et al 2018 Simulation on deacidification performance of waste incinerator flue gas by rotating spray drying Energy 152 652-665
[7] Fabien R, Stéphanie L, Bertrand C, et al 2019 CFD water spray model development and physical parameter study on the evaporative cooling Applied Thermal Engineering 149 960-974
[8] Tao W Q 2011 Numerical Heat Transfer Xi'an Jiaotong University Press 1-5
[9] Zhou L X 1994 Turbulent Flow and Two-phase Combustion Theory and Numerical Simulation Beijing: Science Press 1994
[10] Zheng H, Zheng C H, Yang Z D, et al 2018 Numerical simulation study on spray evaporation of desulfurization wastewater in flue duct China Environmental Science 4476-4483
[11] Montazeri H, Blocken B, Hensen J L M 2015 Building & Environment 83 129-141