Optimal Device for Waste Heat Recovery Process Based on Convective Drying Technology

Yusen Liao*, Yujing Chen and Xinyuan Yu
School of Mechanical and Electrical Engineering, Wuhan University of Technology, Wuhan, China

*Corresponding author: 283361@whut.edu.cn

Abstract. With the development of chemical industry, medicine, food and other industrial production in China, the amount of drying to wet materials is increasing, and the energy consumption of drying is also increasing year by year. Among them, the hot and humid waste gas produced by convective drying technology contains a large amount of waste heat, and the recovery and utilization of waste heat through the two-stage combination of gas-gas heat exchange and gas-liquid heat exchange is a method to reduce energy consumption, which has good economic value and energy conservation and emission reduction value.

1. Introduction
At present, there is still a high energy consumption in convective drying technology in Chinese industry, and most industrial equipment exhaust outlets do not have the problem of subsequent treatment. Therefore, it is of certain research significance to effectively reduce the energy consumption of convective drying by recycling hot and humid exhaust gas and preheating new air to obtain hot water from industrial raw materials at the same time.

2. Study Background
2.1. Residual heat
Residual heat refers to the heat discarded or discharged from the energy utilization system or equipment, including the heat released from the discharged heat carrier that is higher than the ambient temperature and the low caloric value of flammable wastes. Heat carriers emitting waste heat can be solid, liquid, or gaseous substances, and waste heat is distributed in a variety of energy consuming systems and equipment. According to statistics, the total industrial waste heat resources in China account for about 17% ~ 67% of the total fuel consumption, the recoverable waste heat resources are about 60% of the total waste heat resources, and the industrial waste heat recovery rate in China is 34.9% [1,2,3]. The low grade waste heat emitted in energy consumption has the characteristics of large amount and wide range, while the recovery and utilization of this part of energy faces a series of problems such as high investment and difficulty. Gas boiler exhaust gas that contains huge waste heat resources, with a huge amount of energy, energy level and great development potential and other significant characteristics.

Convective drying is a drying method that uses hot gas as a heat source to remove the steam produced by dehumidifying materials in the process of wet material drying. Convection dryer is widely used
because of its simple structure, convenient operation and strong adaptability. It is the most widely used drying equipment in production. It is widely used in chemical industry, medicine, food and other industrial production, and is an essential unit operation for many technological processes. Convective drying, that is, part of the heat carried by the drying medium is used to evaporate the moisture in the material, the other part is used to heat the material, and the rest is wasted for the heat loss of the dryer and that of the exhaust. Drying operation is a large energy consuming household, and its energy consumption accounts for 12% of the total energy consumption in the national economy, while the average thermal efficiency of drying process is about 3% [4]. Therefore, improving the drying process and improving the drying thermal efficiency are of great significance for reducing energy consumption and saving energy. Pictures 1 is the convection drying device for some industrial application.

![Industrial convection drying equipment](image)

**Figures 1.** Industrial convection drying equipment.

2.2. *Study significance*

The work of convective drying technology is shown in Figure 2. The inlet air of convection dryer is heated after passing through the heating furnace to become dry high-temperature gas; then, it enters the convection drying pipeline through the air inlet, and the materials in the drying pipeline are transmitted through the conveyor belt and move with the gas phase to form convection and carry out sufficient heat exchange, so as to take away the moisture on the product; then, the hot and humid air is discharged from the moisture outlet to complete the drying process.

![Flow chart of convective drying technology](image)

**Figure 2.** Flow chart of convective drying technology.

This project takes energy conservation and emission reduction as the starting point, through the analysis of the research status quo of industrial convective drying in China, through investigation and other research, combined with the current industrial development prospects, it is found that there is
unsufficient application aspects, energy loss and other conditions in industrial convective drying technology; and at present, the air waste heat recovery methods are widely classified and there are many solutions. After theoretical analysis, it is found that most industrial equipment exhaust outlets do not have subsequent treatment. Through process sequence adjustment and structure update, it has good realizability in terms of feasibility, and has good energy conservation as well as emission reduction benefits.

3. Study Content and Objectives

3.1. Study content
(1) To optimize the original residual heat recovery process based on convection drying technology.
(2) To test a new waste heat recovery process and to maximize the remaining heat recovery efficiency.

3.2. Study objectives
(1) In this project, hot and humid air was heat exchanged with external air to realize low-grade waste heat recovery;
(2) The secondary heat exchange was carried out by means of spraying cooling water through the access spray module at the outlet of the exhaust air exhaust, so that the temperature of the spraying water was increased, and was transmitted to the water storage tank through the insulation pipeline for hot water reserve of industrial raw materials.

4. Implementation Plan, Proposed Research Methods and Technical Routes

4.1. Implementation scheme
The overall idea of the device was modular design, which was divided into two modules: heat exchange module and spraying module. The heat exchange module was installed at the air inlet and functions before new air enters; the spray module was installed at the air outlet and mainly recycled the low grade heat energy of the lack of air.

The workflow diagram is shown in Figure 3.

![Figure 3. Overall flow chart of design.](image)

Before the new air was heated in the heating furnace, the first level of heat exchange was performed with the hot and humid boring gas in the heat exchange module, so that the new air temperature rises and then passed into the heating furnace for material drying; the boring gas after heat exchange entered the spray module for water-gas heat exchange, so that the positive enthalpy of the water changes and the temperature rose, therefore, the second level of low grade heat energy utilization was realized.
As shown in the overall design diagram in Figure 4, the heat exchange module and the spray module were installed on the original convection drying equipment. It was composed of convection working area, heat exchanger, sprayer, hot water storage tank and water pump.

4.1.1. Heat exchange module. Heat exchange adopted the form of indirect heat transfer in heat transfer mode. Heat exchanger was composed of inlet A and B, exhaust A and B, heat transfer area and diversion fan. Figure 5 shows the interior of the heat exchanger.

The workflow was as follows: hot and humid air entered from inlet A and exhaust A; new air entered from inlet B and exhaust port B. Moist heat air entered from inlet A and new air from inlet B, both of which were introduced into the heat transfer zone via the diversion fan. The heat transfer zone was composed of a series of metal sheets with certain corrugation shape. A thin rectangular channel was formed between various plates, and there were many corrugated thin plates stamped at a certain interval, which were sealed by gaskets around and compressed with overlapping frame and compression screw.

Figure 4. Schematic diagram of overall design.

Figure 5. Heat exchanger internal diagram.
The four angular holes of the plates and gaskets formed the distribution tube and collection tube of fluid, and at the same time, the cold and hot fluids were reasonably separated, so that they flowed in the flow channel on both sides of each plate, respectively, and heat exchange was performed through the plates. The heat exchanger adopted the cross design of air inlet and air outlet to increase the air flow distance, prolonged the air flow heat transfer time and promoted heat transfer; the plate heat exchanger was applied as the core in the middle of the heat exchanger. Its main advantages were as follows:

(1) High heat transfer coefficient. The flow channel spacing of the plate heat transfer core in the middle of the heat exchanger was narrow, the plate was corrugated, and the change of the flow channel section was very complex, which made the flow direction and velocity of the fluid continue to change, and the flow disturbance of the fluid increased, therefore, the fluid with a smaller flow rate could reach the turbulent state. In addition, compared with the shell and tube heat exchanger, the plate thickness of the plate heat exchanger was only half or a fraction of the latter, and the thermal resistance was greatly reduced, therefore, the plate heat exchanger had a higher heat transfer coefficient.

(2) It had less investment and high economy. Plate heat exchanger plates were mostly metal plates, and their raw materials were cheaper than the same metal pipe materials. In petrochemical enterprises, heat exchanger investment accounted for 30% ~ 50%, in refrigerators, the weight of evaporator and condenser accounted for about 30% ~ 40% of the total weight of the unit, and the power consumption of heat exchanger accounted for 30% ~ 45% of the total power consumption. It could be seen that plate heat exchangers had absolute comparative advantages in both heat transfer efficiency and investment economy.

4.1.2. Spray module. The spraying module adopted water-gas spraying heat transfer, which essentially belonged to direct contact heat transfer. This way greatly increased the gas-liquid two-phase contact area and could quickly complete heat transfer and mass transfer. Flue gas and water could realize stable contact heat transfer under a very small temperature difference without the need for metal heat transfer surface, moreover, it could reduce the flue gas side resistance, the volume of heat exchanger, and greatly reduce the cost of heat exchanger. The exhaust temperature of flue gas could reach below 20°C at the lowest. At the same time, through the deep recovery of condensation heat, the water temperature could also be increased to obtain industrial available raw material hot water.

![Figure 6. Schematic diagram of spray device.](image-url)
water outlet, water storage tank and water pump. The specific distribution inside the device is shown in Figure 7.

![Figure 7](image1.png)

Figure 7. Internal schematic diagram of spray device.

The spray head adopted the large area chassis support design with adhesion and support force, which could fix the spray head on the device and make the spray more stable; at the same time, the three-way spray head design could cover the device spray, the device shell was smooth, which was conducive to hot water collection. The spray head is shown in Figure 9.

![Figure 8](image2.png)

Figure 8. Connection diagram of spray head and pipeline.
Figure 9. Schematic diagram of spray head.

The workflow was mainly as follows: the cooling water was stored in the water storage tank, and the three spray nozzles at the top were pumped upwards by the water pump for realizing the spray function; the flue gas entered from the flue inlet and the flue outlet as removed, when the flue gas passed through the inside of the device, the spray nozzle sprinkled the cooling water around the bottom, and when the hot and humid flue gas passed through, it was in full contact with the sprayed condensate water for adequate heat transfer, the flue gas temperature further decreased, while the water temperature increased and was exported from the drain, so as to obtain the raw material hot water for reuse.

4.2. Study methods and technical routes

The project research method is shown in Figure 10, (1) went to the factory with appropriate waste heat for investigation; (2) through personal feelings and practical communication with factory workers, understood the waste heat waste problems existing in the drying process of the existing factory, and the existing waste heat recovery system; (3) integrated and summarized the information obtained from their
own investigation and the data queried, and obtained the conditions that the device shall meet and the functions we could achieve; (4) combined with the relevant mechanical design knowledge, designed and built the device we need; (5) used the software to carry out the simulation experiment, obtained the existing problems of the device, after continuously modification, ensured that it was correct before carrying out the processing; (6) carried out the physical experiment of the processed device on the factory, sorted out the existing problems, carried out the inspection and modification, carried out the calibration experiment again, and obtained the required device.

5. Study Basis and Feasibility Analysis of the Project

5.1. Project research basis
The temperature of the exhaust outlet after drying was 120℃ ~ 130℃. This part of hot and humid air entered the heat exchanger in the heat exchange module to perform indirect contact heat transfer with the air under temperature of 20℃ ~ 25℃, slightly heated the new air, and then the hot and humid air entered the spray module. The spray head in this module performed water spraying. According to the relevant literature, the air flow rate and the temperature of the sprayed water had an effect on this heat transfer process. The relationship between heating capacity and water temperature, air inlet temperature and air flow rate was obtained by experimental determination [5].

![Figure 11. Heating capacity at different water temperatures.](image)

![Figure 12. Heating capacity at different air inlet temperatures.](image)
Figure 13. Heating capacity at different air flow rates.

It could be seen from this experiment that for the spray module, the faster the air flow rate and the lower the spray water temperature, therefore, the heat transfer efficiency was limited.

In the spray module, the nozzle water spray and hot and humid air direct contact heat transfer mode was used. This heat transfer mode was based on that the lack of gas carried a large amount of water vapor, and the temperature dropped, while the heat transfer coefficient of water was in the range of 10 ~ 100, which was a more efficient heat transfer mode of specific gas-gas heat transfer type.

5.2. Feasibility analysis

5.2.1. Working state analysis. The bearing capacity of plate heat exchanger gasket determined its working temperature. For example, the maximum working temperature of rubber elastic gasket was generally lower than 200℃; while the maximum working temperature of compressed asbestos lint gasket was between 250℃ and 260℃. However, compared with rubber elastic gasket, the elasticity of compressed asbestos lint was obviously insufficient, and the bearing capacity was low. Therefore, the working temperature of plate heat exchanger was generally lower than 250℃. In this project, the exhaust port temperature after drying was 120 ~ 130℃, which was within the working temperature range of heat exchanger; at the same time, the pressure of dry air was not large and closed to normal pressure, and no additional working pressure was generated for the heat exchanger. In terms of temperature and pressure parameters, the normal operation of the equipment was ensured.

5.2.2. Feasibility analysis of equipment installation. The main body of the equipment was still a convection drying working area, which was installed open to facilitate cleaning and maintenance; the working area was separated from the recovery area and attached to its installation to facilitate disassembly and replacement, while reducing the difficulty of installation. It was suitable for the production requirements of the factory and had high feasibility in installation.

5.2.3. Numerical analysis of flow and heat transfer characteristics in heat transfer zone. At present, heat exchanger was mainly used to increase heat exchange by strengthening heat transfer method to increase heat transfer coefficient, and fluid software was commonly used for numerical analysis of it. According to the data, it was found that the numerical analysis of the fluid characteristics of the herringbone plate heat exchanger with the RNG K − ε model and wall function was mostly carried out, but this method was not accurate enough for the treatment near the wall. With the development of computer technology, meshing technology was more and more mature, especially the rapid
improvement of computer computing ability, which provided space for the use of low Reynolds number \( k - \varepsilon \) model. In summary, we used the low Reynolds number \( k - \varepsilon \) model and used the fluid software Fluent to complete the heat transfer field simulation on the basis of fully considering the heat transfer in the thermal boundary layer, and gave the parameters affecting the heat transfer performance. The following is the modeling and analysis process:

Step 1 Mathematical model assumptions. (1) The working medium was an incompressible Newtonian fluid; (2) Gravity and buoyancy due to density differences were ignore; (3) Due to the low fluid flow rate in the heat exchanger, the thermal effect produced by the viscous dissipation during fluid flow was ignored.

Step 2 Model selection. The calculation of flow near the wall was divided into two ways: 1 wall function method, 2 low Reynolds number turbulence model [6]. For the wall function method, it was required that the first node be set within the turbulent area, and the numerical value in the turbulent area was obtained by the wall function. This method did not calculate the viscous bottom layer, so the accurate value of the viscous bottom layer failed to be obtained; the low Reynolds number \( k - \varepsilon \) model calculated the viscous bottom layer, so it was required to arrange a sufficiently dense grid in the area close to the wall (due to the large calculation amount, it also put forward certain requirements for the computer), and at the same time, the low Reynolds number \( k - \varepsilon \) model and the standard model in the complete turbulent area maintained high consistency in the calculation results. Therefore, the low Reynolds number \( k - \varepsilon \) model was selected in this paper, and the equation \( k \) and equation \( \varepsilon \) are:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_t} \frac{\partial k}{\partial x_j} \right) \right] + G_k - \rho \varepsilon - \left[ 2 \mu \left( \frac{\partial k}{\partial n} \right)^2 \right] \tag{1}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_t} \frac{\partial \varepsilon}{\partial x_j} \right) \right] + C_{\varepsilon k} \frac{G_k}{k} f_2 - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} f_2 + \left[ 2 \mu \frac{\varepsilon}{\rho} \left( \frac{\partial u_i}{\partial n} \right)^2 \right] \tag{2}
\]

Where:

\[
\mu_t = C_{\mu} |f_{\mu}| \rho k^2 \varepsilon \tag{3}
\]

Where \( k \) is the fluid turbulent kinetic energy; \( \varepsilon \) represents the turbulent dissipation rate; \( G_k \) is defined as the generation term of the turbulent kinetic energy \( k \) (related to the velocity vector); \( n \) is the wall normal vector coordinate; and \( \mu \) is the velocity parallel to the wall. In the actual calculation, the normal coordinate \( n \) can be approximately taken as \( x \) or any of \( z \); the coefficient \( C_{1\varepsilon}, C_{2\varepsilon} \) and \( C_{\mu} \) are empirical constants; and, \( \sigma_{t}, \sigma_{\varepsilon} \) are the Planck (Pr) numbers corresponding to the turbulent kinetic energy \( k \) and dissipation rate \( \varepsilon \), respectively. \( C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.90, C_{\mu} = 0.99, \sigma_{t} = 1.0 \) and \( \sigma_{\varepsilon} = 1.2 \). In the above Equation (2), the part surrounded by the right-most symbol "\( \mid \)" of medium size is the part of low \( Re \) number model different from high \( Re \) number model. The introduction of damping function \( f_1, f_2 \) and \( f_{\mu} \) is actually to modify the coefficients \( C_{1\varepsilon}, C_{2\varepsilon} \) and \( C_{\mu} \) in the standard \( k - \varepsilon \) model. The calculation formula of each coefficient is:

\[
f_1 \approx 1.0 \tag{4}
\]

\[
f_2 = 1.0 - 0.3 \exp(-Re^2) \tag{5}
\]

\[
f_{\mu} = \exp \left( \frac{-2.5}{Re^{0.5}} \right) \tag{6}
\]
\[ Re = \frac{\rho k^2}{\eta e} \]  

(7)

Where \( Re \) is the turbulent Reynolds number.

Step 3 Heat transfer zone model. The heat transfer area was the main part of heat exchange of heat exchanger, which played a decisive role in the improvement of the overall heat exchange efficiency of heat exchanger. According to the solid structure of the heat transfer area of the herringbone plate heat exchanger, the area of 80 mm \( \times \) 100 mm was intercepted, and its specific parameters were: ripple height \( h = 3.5mm \), ripple spacing \( \lambda = 13.5mm \), and ripple inclination angle \( \beta = 60^\circ \).

Main analysis software: FLUENT module in ANSYS software;
Solution method: implicit separation variable method;
Speed and pressure coupling: SIMPLE method;
Discrete mode: second-order accuracy upwind format.

Step 4 Boundary conditions. A velocity inlet was used at the inlet, and the inlet temperature was set at 350 K; the outlet boundary condition was the pressure outlet, and the outlet pressure value was set at 0.1013 MPa. The wall region was set to the no-slip velocity boundary condition \( u = 0 \), and the temperature distribution obeyed the adiabatic boundary condition, \( \frac{\partial t}{\partial n} \bigg|_{wall} = 0 \).

Step 5 Numerical simulation as well as analysis.

Figure 14 shows the pressure distribution at the cross-section of the flow channel. The pressure drop distribution from the upper right inlet to the lower left outlet showed a step-like distribution, and the pressure gradient distribution was relatively uniform. It can be seen in the figure that the inlet end is orange-red and the pressure is significantly larger; the outlet end is green and the pressure gradient distribution is smaller, which indicates that the fluid pressure loss of herringbone heat exchanger is greater. However, there is an unequal pressure distribution at the left and right ends of the inlet segment, which is related to the different inlet areas.
Figure 15 shows the temperature profile in the heat exchanger flow channel, assuming an inlet temperature of 350 K and an outlet temperature of 300 K. In the figure, the upper right side is the thermal fluid inlet and the lower left side is the outlet. It can be seen that the temperature changes sharply along the fluid flow direction in the inlet area. The high temperature region appears behind the plate contact point, which is caused by the fierce bypass movement of the fluid around the plate contact point. The blocking of the plate contact point will force the fluid flow morphology to be more confusing and the heat transfer performance to be enhanced. In addition, the temperature difference at the inlet is related to the uneven flow distribution of inlet fluid.

Combining the above analysis:
(1) In this paper, the ICEM module in ANSYS software was used to draw the tetrahedral mesh containing the boundary layer, and the low Reynolds number model was selected to analyze the near wall area in detail, which especially focused on the area where the development of fluid turbulence in the viscous substrate was insufficient.

(2) The distribution of fluid pressure field, velocity field and temperature field in the heat transfer area was obtained by FLUENT software analysis. It was found that the distribution of fluid pressure and velocity was relatively uniform, and its value decreased with the flow in turn. The Reynolds number of fluid was linearly distributed with the Nusselt number and pressure drop, while the Nusselt number and pressure drop increased with the increase of Reynolds number. The distance between the extreme point of fluid turbulent kinetic energy and the contact point of corrugated plate was also numerically consistent. Fluid velocity changed obviously near the wall, and the flow pattern showed cross-flow turbulent kinetic energy pulse value spacing and plate contact point distance value was equal.

(3) In terms of installation, the feasibility was high, the change was small, and it was convenient for maintenance and affordable; the parameters were obtained through thermodynamic analysis, so as to better analyze the thermal energy utilization rate. Besides, the feasibility was high, therefore, the feasibility of the project could be verified and easy to implement.

References
[1] Zhe Lu. Analysis on the present situation of industrial waste heat recovery and utilization technology in China [J]. Equipment Manufacturing Technology, 2019, (12): 204-206.
[2] Longqing Yu, Feng Ma, Xuewei Hu. Comprehensive utilization of waste heat in low-temperature industry [J]. Resource Conservation and Environmental Protection, 2018, (4): 13, 17.
[3] Yun Zhou, Kang Wang, Siming Chen. Current situation and technical prospect of industrial waste heat utilization [J]. Science and Technology Intelligence Development and Economics, 2010, 20 (23): 162-164. DOI: 10.3969/j.issn .1005-6033.2010.23.070.

[4] Dengying Liu, Chongwen Cao. Exploration on the new development path of drying technology in China [J]. General Machinery, 2006, (7): 15-17. DOI: 10.3969/j.issn .1671-7139.2006.07.004.

[5] Chaoxing Yan, Yating Yang, Hanqing Lin, Weian Du, Iridium Liu. Study on heat transfer characteristics of spraying droplets in saturated steam with medium and low pressure [J/OL]. Atomic Energy Technology: 1-7 [2020-11-22].http://kns.cnki.net/kcms/detail/11.2044.TL.20200428.1517.006.html.

[6] Haifeng Tang. Numerical simulation and comparative analysis of offset jet and adherent jet by six low Reynolds number k-ε models [D]. Hangzhou University of Electronic Science and Technology, 2019.