Numerical study of combustion and heat transfer in a composite heat carrier generator

Cong Yu², Ling Shi¹, Jiaying Hu² and Hongjiao Liu¹,*

¹Jianghan University, Hubei Key Laboratory of Industrial Fume & Dust Pollution Control, 430056 Wuhan, P.R. China
²Jianghan University, School of Intelligent Manufacturing, 430056 Wuhan, P.R. China
* Corresponding author: mxw1996@jhun.edu.cn

Abstract. To investigate the operational problems of the composite heat carrier generator (CHCG) in actual industrial applications such as overheating and poor safety performance, an integrated analytical model was established. For this model, the commercial software Fluent was first applied to simulate the gas-liquid turbulent flow, diesel vaporization and combustion, and the mixing process between the flue gas and the preheated water. Taking the parameters obtained from the Fluent model as the boundary condition, an indirect contact heat transfer model considering the heat transfer between the hot flue gas and the cold water has been solved. Based on this model, the areas where the phenomena of overheating and high thermal stress are prone to occur have been determined, and the size of the water sleeve has been redesigned.

1 Introduction
Nowadays, due to the global economic growth and the threat of the energy crisis, the continuous supply of energy has become very important. To address this problem, on the one hand, the byproduct fuels have been utilized in some fields such as steel mills and chemical plants [1]. However, most of these fuels are non-standard fuel gases with low calorific value, posing a great challenge for reliable industrial operation [2]. On the other hand, increasing attention has been paid to exploring unconventional fossil fuels such as heavy oil, bitumen, and shale oil.

The heavy oil resource deposited underground is enormous, which is more than twice as great as conventional light crude oil [3]. As the globally largest developing country, China has an abundant heavy oil resource with an explored reserve of 16×10⁸ tons. However, heavy oil exploitation is difficult because of its high viscosity, high density, and poor flow characteristics. Therefore, it is necessary to investigate the effective and efficient methods for heavy oil recovery.

As one of the thermal recovery methods, surface steam generation and injection is currently the most widely-used approach for heavy oil recovery. This technology applies boiler to generate steam, which is then transported to the targeted area in the formation and used to reduce the viscosity of hydrocarbons by temperature [4]. In this way, the mobility of oil can be increased. However, the steam injection technology is restricted by many factors, including the high cost of the facilities, the inefficient heat transfer, and the emission of greenhouse gases.

To overcome the drawbacks of traditional steam injection, a new method named thermochemical fluid (TCF) injection was proposed [5]. TCF is a composite heat carrier (CHC), mainly composed of
hot steam, flue gas, and chemical agents. TCF injection technology uses TCF to replace steam for the thermal production of heavy oil. It reduces viscosity and improves sweep volume through the combined effects of heat, gas and chemistry. The combustion products are prevented from emitting into the atmosphere as well. Therefore, this technology is regarded as a promising technical direction to improve oil displacement efficiency in the future.

TCF is generated by a combustion system named composite heat carrier generator (CHCG) [6]. In the recent application, this device presents a relatively poor safety performance and sometimes cannot reach the technical indicators such as gas flow rate and heat. However, there were very few studies on the design and optimization of CHCG. Therefore, an integrated numerical model combining gas-liquid flow, diesel combustion, and direct and indirect contact heat transfer was established in the present work. An optimized design method was then proposed based on this model.

2 Facility description

The composite heat carrier generator considered in this study can be divided into two parts. As shown in Fig. 1, the first part is a combustion zone located upstream of the device, consisting of a burner, a combustion chamber, and a water-preheated system. It can be seen that the diesel and the primary air (PA) enter the combustion chamber through the nozzle at the centre of the burner. Around this nozzle, eight nozzles are arranged evenly along the circular direction, in which seven of them are the inlets of the secondary air (SA), and one is the connector of the spark plug. Fuel and air are mixed and burnt inside the combustion chamber, and a ceramic layer is designed close to the fire to reduce the heat transfer rate. Outside the combustion chamber, a water sleeve made of metal is installed around the ceramic layer. Cold water flows in the water sleeve and absorbs the heat from the flue-gas side. The flow directions of the water and the flue gas are the same. At the outlet of the water sleeve, the preheated water is injected into the combustion chamber and evaporated. The mixture of the hot flue gas and the steam is called CHC, which enters the mixing zone through a gasification cup and a static mixer. At last, the CHC with a specific temperature and pressure is obtained.

Fig. 1. The schematic of the composite heat carrier generator.

3 Numerical Modelling

3.1 Modelling method

Fig. 2 shows the method of the iterative calculation to design the area of the heating surface of the water sleeve. The parameters of the preheated water at the outlet of the water sleeve such as temperature T0 and dryness fraction X0 was first assumed. According to these parameters, the inlet boundary condition of the preheated water and the thermal boundary condition on the inner wall of the ceramic layer can be determined, which were used to simulate the process inside the combustion chamber using Fluent. Based on the Fluent Modelling, if the overheating occurs on the gasification cup and its downstream metal walls, go back to reassume the parameters of the preheated water. If not,
the temperature and heat flux on the inner wall of the ceramic layer can be determined, which is the thermal boundary condition of fluid in the water sleeve. Assuming the area of the heating surface of the water sleeve and combining the information of the heat absorption, the outlet temperature of the water sleeve can be calculated. If this value cannot match the assumed parameters of the preheated water, go back to reassume the area of the heating surface. If these two values were close enough, continue to check if overheating occurs on the metal walls of the water sleeve. If yes, go back to reassume the parameters of the preheated water. If not, the loop is over.

![Diagram](image)

**Fig. 2.** Method of the iterative calculation.

### 3.2 Geometric model and mesh system

Fig. 3(a) shows the primary geometry size of the CHCG, in which the combustion zone is 1070 mm and the mixing zone is 980 mm. According to the original design, the 3-dimensional assembly model was established using SolidWorks, as shown in Fig. 3(b). To simplify the geometry and to extract the fluid domain, the assembly model was further modified with Space Claim Design Modeler (SCDM), as shown in Fig. 3(c) and Fig. 3(d). At last, the geometry model of the fluid domain was discretized, and the computational mesh has 1,483,632 cells, as shown in Fig. 3(e).
3.3 Mathematical model
The Modelling processes in Fluent can be described as follows: (1) The continuity equation, the momentum conservation equation and the turbulence model were solved. Due to the advantage of capturing strongly swirling flow, the realizable k- model considering curvature correction was applied to simulate the turbulent flow. (2) After the cold flow field achieved a convergent solution, the energy conservation equation, the discrete phase model (DPM) and the partially premixed combustion model (PPCM) were solved to consider the heat transfer, the trajectory and evaporation of the oil droplets, and the combustion of the volatile components of the diesel, respectively. For PPCM, the C equation considered the premixed combustion, and the non-premixed combustion was described using a probability density function (PDF) table. The diesel’s vaporization temperature and boiling point were initially set as 290 K, which were changed to 341 K and 447 K after ignition, respectively. (3) After the combustion process was convergent, the preheated water was injected into the combustion chamber. DPM was used to calculate the flow and heat transfer processes of the water droplets. (4) At the temperature field in the combustion chamber was stable, discrete ordinates (Do) model was applied to consider the radiant heat transfer.

3.4 Boundary condition
The inlet boundary conditions of the diesel, the PA and the SA, and the cold water's inlet parameters can be seen in Table 1.

Table 1. Inlet boundary conditions of the fuel, air and water.

| Parameters          | Value   | Parameters    | Value   |
|---------------------|---------|---------------|---------|
| Flow rate of the    | 0.008 kg/s | Flow of the PA | 0.095 kg/s |
| diesel              |         |               |         |
| Density of the      | 840 kg/m³ | Flow of the SA | 0.024 kg/s |
| diesel              |         |               |         |
Results and discussion

**4.1 Temperature of the gasification cup and its downstream metal walls**

Through the iterative calculation, the water temperature at the outlet of the water sleeve is determined as 373 K. Numerical Modelling was conducted using this parameter. Fig. 4(a) shows the temperature distribution of the CHC. It can be seen that there is a non-uniform temperature distribution of the gas close to the entrance of the combustion chamber. The reason is that the nozzles of the SA are not perfectly symmetric. One corner is occupied by the igniter nozzle, which cannot supply the air as well as the other SA nozzles during the operation. Therefore, the ignition on this side will delay. After ignition, the temperature of the CHC reaches the maximum value of about 2000 K. In the combustion chamber, the area directly in contact with the fire is the ceramic layer, which has an excellent performance on heat resistance. However, the overheating possibility of the metal wall of the water sleeve getting in touch with the ceramic layer should be considered, which will be discussed in the next section. At the outlet of the combustion chamber, the temperature of the CHC rapidly decreased to about 500 K due to the mixing effect with the preheated water. Therefore, for the gasification cup and its downstream facilities, approximate 99.6% of the metal wall has a temperature below 688 K, which meets the requirements of the allowable temperature 973 K, as shown in Fig. 4(b). However, it can be seen that the areas between two groups of the water nozzles present local high temperatures because these areas are blind spots that the water droplets cannot cover the high-temperature flue gas in time. Therefore, these areas should be protected by refractory materials as well.

![Fig. 4. Temperature distributions of the CHC and the CHCG.](image-url)
4.2 Temperature of the metal walls of the water sleeve

Fig. 5 shows the maximum temperature, the average temperature, and the minimum temperature of the ceramic walls along with the height of the combustion chamber, which is the result of the Fluent model. It can be seen that the maximum temperature increases rapidly, yet the minimum temperature stays unchanged at the initial segment of the combustion chamber, which is attributed to the uneven distribution of the SA nozzles. Thus, the temperature deviation of the wall at the same cross-section increases, leading to more considerable thermal stress at this segment. In the vicinity of the water injection points, the drop in maximum temperature is more significant than that of the minimum temperature, which results in a decline of the temperature deviation and the thermal stress.

![Temperature distribution of the inner wall of the ceramic layer along with the combustion chamber’s height.](image)

Taking the heat flux and the maximum wall temperature of the ceramic layer’s inner wall as the boundary condition, the temperature distribution of the metal wall of the water sleeve can be determined. In the calculation, the ceramic layer is divided into two layers. The first layer is 10 mm with a heat conductivity coefficient of 1.5 W/(m·K) and the second layer is 10 mm with a heat conductivity coefficient of 1.2 W/(m·K). The metal layer has a thickness of 12.5 mm and a 37 W/(m·K) heat conductivity coefficient. The calculation result shows that when the water sleeve has a circumference (see Fig. 6) of 8 mm and a pitch (see Fig. 6) of 10 mm, the maximum temperature of the metal wall of the water sleeve is about 780 K, which satisfies the design requirement.

![Structure of the water sleeve.](image)

5 Conclusion

In this study, an integrated model was established for a CHCG, which considered the diesel combustion and the direct and indirect contact heat transfer. Based on this model, the areas where the phenomena of overheating and high thermal stress are prone to occur have been determined, and the size of the water sleeve has been redesigned.

Acknowledgements

Funded by the National Natural Science Foundation of China (Grant No. 52006090) and the Open Foundation of Hubei Key Laboratory of Industrial Fume & Dust Pollution Control (Grant NO. HBIK2020—06).
References
[1] L. Zhang, W.W. Xie, Z.Y. Ren, Fuel 278, 118216 (2020)
[2] L. Zhang, Y. Xue, Q. Xie, Z.Y. Ren, Fuel 287, 119507 (2021)
[3] O.S. Alade, M. Hamdy, M. Mahmoud, D.A.A. Shehri, E. Mokheimer, S. Patil, A.A. Nakhli, J. Petrol. Sci. Eng. 186, 106702 (2020)
[4] L.X. Shi, D.S. Ma, P.C. Liu, X.L. Li, C.F. Xi, C.Wang, J. Petrol. Sci. Eng. 173, 146 (2019)
[5] M. Mahmoud, O.S. Alade, M. Hamdy, S. Patil, E.M.A. Mokheimer, Energ. Convers. Manage. 202, 112203 (2019)
[6] X. Li, Z. Yi, P.F. Gao, X.R. Li, J.Z. Zhang, S.F. Gu, G.A. Cui, U.S. Patent Application 14, 0076344 (2016)