Numerical Simulation Study on Radiation Section of Dehydrogenation Furnace

Linlin Duan¹, Cenfan Liu¹*, Sheng Chen¹, Fang Ji¹, Guide Deng¹

¹China Special Equipment Inspection and Research Institute, Heping Street, Chaoyang District, Beijing, 100029, China

*Corresponding author’s e-mail: liucenfan288@163.com

Abstract. The three-dimensional numerical simulation of the radiation section of a dehydrogenation furnace was carried out by CFD, the thermal-structure coupling analysis was performed. The full-scale meshing of the geometric model was used in this paper, the temperature field and flow field of the furnace and the furnace tube were simulated. The thermal radiation values and temperature values of the simulation results were compared with the design values, it shows that the simulated values are in good agreement with the designed values. The corresponding numerical calculation can be performed for half or 4/1 of the geometry under the condition with the geometric symmetry of the furnace and the furnace tube being good. The simulation results can provide a reference for the design and optimization of the dehydrogenation furnace.

1. Introduction
The furnace is the core device for propane dehydrogenation, and the radiant section is the main place for heat exchange and chemical reaction in the furnace. It is the most important component of the reformer. However, the furnace tube in the radiant section may not be able to long-term safe operation due to crack and creep with local overheating, so focus on the radiant section of the furnace [1]. The traditional research method for radiant section has limitations for the current large-scale dehydrogenation furnace, with the rapid development of computers, the CFD numerical simulation method can obtain a large amount of simulation data under less design conditions, it can provide the necessary information of the heat transfer situation and fluid flow in the furnace to optimize the furnace structure [2,3].

For the combustion and heat transfer problems in industrial furnaces [4], the computational fluid dynamics can be used to simulate the combustion and heat transfer conditions of the furnace under various working conditions in the case of determining structure of the dehydrogenation furnace and the amount of fuel intake. According to the position of the burner and the difference of reformer, many scholars have conducted corresponding research [5-7]. After establishing a three-dimensional model of geometry, generally, the simulation is divided into two parts: the numerical simulation of flame combustion and the fluid flow heat transfer simulation in the furnace tube, which will obtain accurate and reliable simulation results with less time and cost [8]. It needs the heat flux density or temperature, the heat flux distribution of the outer wall of the furnace tube in the combustion field, to calculate the coupling of steady-state combustion and thermal response of the fluid in the furnace tube. The heat flux density on the outer wall surface of the furnace tube is used as a coupling condition to further calculate the flow and heat transfer inside the furnace tube. In this paper, the calculation results are compared with the design values to adjust the parameters of the furnace and fuel to achieve long-term efficient and safe...
2. Setting of numerical simulation for dehydrogenation furnace

2.1 Control equation

1) Mass conservation equation

\[ \frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g \mathbf{v}_g) = 0 \]  

(1)

2) Momentum conservation equation

\[ \frac{\partial}{\partial t} (\rho_g \mathbf{v}_g) + \nabla \cdot (\rho_g \mathbf{v}_g \mathbf{v}_g) = -\nabla p + \nabla \cdot \mathbf{T}_g + \rho_g \mathbf{g} + \mathbf{F} \]  

(2)

3) Energy conservation equation

\[ \frac{\partial}{\partial t} (\rho_g h_g) + \nabla \cdot (\rho_g h_g \mathbf{v}_g) = \nabla \cdot \left[ (\mathbf{e}_g + \mathbf{e}_{g,t}) \nabla T_g \right] + S_g \]  

(3)

4) The calculation formula of the mixed fraction

\[ f = \frac{Z_{t,oxy} - Z_{t,fuel}}{Z_{t,fuel} - Z_{t,oxy}} \]  

(4)

In this study, the heat flux of the outer wall of the furnace tube is first obtained by combustion simulation. But the heat flux density at the interface of the solid domain fluid should be regarded as part of the calculation result, rather than the known condition. This is a typical conjugate heat transfer problem.

2.2 Geometric model

This paper mainly studies the radiant section of the dehydrogenation furnace, which includes 24 circular burners and 16 square burners. There are 24 tubes in the furnace radiant section. The radiant section is 11m long, 8m wide and 9m high, as shown in Figure 1. The burner is in the bottom, venting outlet is at the top, and the entry and exit method of mediums in the conversion tube are middle tubes to entry, other tubes to exit.

Figure 1. Three-dimensional geometric model and bottom of the radiant section

2.3. Meshing

The meshing method is poly-hexcore meshing, as shown in figure 2. The grid-independent verification is performed. The number of dehydrogenation furnace grids is 22.6 million for, and the conversion tubes is 18.2 million.
2.4 Boundary conditions and simulation settings

Under the design condition, the main component of the fuel of the dehydrogenation furnace is hydrogen. The air inlet flow rate of a single circular burner is 0.78kg/s, the gas inlet flow rate is 0.034kg/s; the air inlet flow rate of a single square burner is 0.39kg/s, gas inlet flow rate is 0.017k/s. The main component of the medium in the furnace tube is propane, the inlet flow rate is 89kg/s, the inlet and outlet temperatures are respectively 462°C and 620°C, the inlet and outlet pressures are respectively 0.34MPaA and 0.12MPaA.

The specific heat capacity, viscosity, and thermal conductivity of a fuel such as hydrogen are defined by a temperature equation, and a piecewise-polynomial is selected. The wall material is stainless steel.

Steady state calculation is adopted for the furnace combustion, the combustion model is non-premixed combustion, the radiation is discrete coordinate model, the turbulence model is Realisable $k$-$\varepsilon$, the velocity pressure coupling is coupled algorithm, and the discrete format pressure is PRESTO\cite{10}. The simulation in the furnace tube is the steady-state calculation of the single-phase flow and the component transport equation.

3. Simulation results

3.1 Comparison of simulation results and design values

Through the simulation calculation, the relevant numerical results are obtained. The design values of the dehydrogenation furnace are compared with the simulation values, as shown in table 1. Overall, the simulation results agree well with the design values.

| Category                              | Design Value | Simulation Value |
|---------------------------------------|--------------|------------------|
| The average temperature of the flue gas outlet | 812°C        | 841°C            |
| Average heat flux of the tube         | 32883 W/m²   | 32342 W/m²       |
| Average pressure of furnace tube inlet | 0.34MPaA     | 0.32MPaA         |
| The average inlet speed of the tube   | About 80m/s  | 93m/s            |
| Furnace tube outlet average temperature | 620°C        | 624°C            |
| The average outlet speed of the tube  | About 250m/s | 264m/s           |

3.2 Temperature and velocity distribution

The temperature distribution of the X-axis section of the center circular, side circular, and square burners are shown in Fig. 3. The flame shape of the circular burner is circular, and the center flame height is 1/3 of the total furnace height, that is highly uniform with design condition. The flame shape of square burners is square, and the flame height is about half of the circular flame height. The temperature distribution in the middle and upper sections of circular burner is relatively uniform and the temperature...
is high. The temperature of the side circular burner near the furnace wall is lower, and the temperature of intermediate furnace is relatively uniform. The square burners have uneven temperature distribution in the middle and upper regions, and the middle part is high temperature zone, and the temperature on both sides is low.

Figure 3. Temperature distribution of the X-axis section of the center circular, side circular and square burners

Temperature distribution of the inner wall of the furnace tube, and velocity distribution inside the conversion tubes are shown in Fig.4. The temperature distribution on the inner wall of conversion tubes is low on both sides, but in the middle tube wall is high, so the temperature gradient is large, and the high temperature area is concentrated in the middle of the furnace. The temperature and velocity distribution of the internal flow field of the conversion tubes are consistent. The flow velocity and temperature of the medium on the outlet side are significantly higher than that on the inlet side. The medium passes through the upper and lower tubes to reach the outlet, the temperature and velocity of the upper part of the outlet tube are also the largest. The temperature of reforming tubes in the middle of the furnace is higher than that of the outside.

Figure 4. Temperature distribution of the inner wall of the furnace tube, velocity distribution inside the conversion tubes
4. Conclusion
The thermo-mechanical coupling characteristics of the radiant section of the dehydrogenation furnace are numerically simulated based on the CFD method. The temperature and velocity distribution of the fluid in the furnace radiant section and the conversion tubes are obtained, and the simulated values and design values of the parameters are compared well.

Overall, considering the calculation amount, the model can be numerically simulated with half or 1/4 of the geometric model in case of good symmetry of the furnace and the conversion tubes.

The simulation results are consistent with the experimental results, and the simulation method can be applied to similar furnace combustion and conversion tube design.

Acknowledgements
The authors are supported financially by Internal Project of China Special Equipment Inspection and Research Institute (2019Youth12).

References
[1] Pei Chen. Numerical Simulation of Radiation Section of Natural Gas Steam Reforming Hydrogen Reforming Furnace [D]. China University of Petroleum (Beijing), 2016.
[2] CFD simulation of the sidewall fired tubular reforming furnace. Nielsen P, Christiansen L J. ASME 2002 Pressure Vessels and Piping Conference . 2002
[3] Development of a numerical model for natural gas steam reforming and coupling with a furnace model[J] . C. Ventura, J.L.T. Azevedo. International Journal of Hydrogen Energy . 2010 (18)
[4] Tubular heating furnace [M]. China Petrochemical Press, edited by Qian Jialin, 2003
[5] Xiaojun Zhang, Analysis of Temperature Field and Smoke Flow Field of Large Hydrogen Reforming Furnace[J]. Petrochemical Equipment Technology, 2010, 31(03):37-40+71.
[6] The progress in water gas shift and steam reforming hydrogen production technologies – A review[J] . Trevor L. LeValley, Anthony R. Richard, Maohong Fan. International Journal of Hydrogen Energy . 2014 (30)
[7] Yu Long, Xiangping Yang, Enjie Yin, Yanping Liu. Study on Comprehensive Numerical Simulation in Hydrogen Reforming Furnace[J]. Refining Technology & Engineering, 2010, 40(08):22-25.
[8] Xiaojiao Li. Simulation of temperature distribution in furnace and hydrogenation stress analysis of furnace tube [D]. Beijing University of Chemical Technology, 2012.
[9] Fujun Wang. Computational fluid dynamics analysis [M]. Beijing: Tsinghua University Press, 2004, 52
[10] Effects of geometry and operating conditions on hydrogen productivity of a fuel cell reformer[J]. Jae Seong Lee, Juhyeong Seo, Ho Young Kim, Jaewon Chung, Sam S. Yoon. International Journal of Heat and Mass Transfer . 2014