Thermal Stress Analysis of Tube Plate and Tube Bundle of Multi-Tube Pass Spirally Corrugated Tubes Heat Exchanger

Yao Zheng and Hehui Wang*

School of Mechanical and Power Engineering, East China University of Science and Technology, Shanghai, 200237, China

*Corresponding author e-mail: Y30180588@mail.ecust.edu.cn

Abstract. This paper analyses the thermal stress of multi-tube heat exchanger with spirally corrugated tubes using numerical simulations. The simulations are focused on the thermal stress of the tube bundle and tube plate, which have large temperature difference. In this paper, the heat exchanger design software HTRI is creatively used to calculated the temperature boundary conditions of the heat exchanger, which is difficult to obtain. Based on the indirect coupling method of ANSYS, the thermal stress intensity and thermal deformation between the tube bundle and the tube plate of the heat exchanger are calculated. And the calculation results can provide important references for the design and optimization of the heat exchanger.

1. Introduction

The tube plate and the shell of the fixed tube plate heat exchanger are generally connected by welding, and the tube plate and the tube must be joined by expansion welding. Due to the temperature difference between the fluid in the tube and the fluid outside the tube, there is thermal stress at the junction. And the heat exchanger studied in this paper is a four-tube-pass heat exchanger. The temperature difference of the fluid between each tube-pass also causes the tube plate to be heated unevenly and there is a thermal stress. Excessive thermal stress will cause the heat exchanger tube bundle and tube plate to be pulled apart at the junction, causing the tubes to fall off or liquid leakage, and affect the safe operation of the heat exchanger. Therefore, it is necessary to analyze and evaluate the thermal stress of the heat exchanger. [1]

2. Finite element model and mesh

2.1. Finite element model

The research object of this paper is a four-tube-pass heat exchanger of a chemical company, with water vapor in the shell side, an inlet temperature of 100°C and a pressure of 65 kPa, and a cold phenol solution inlet temperature of 48°C and a pressure of 200 kPa in the tube side. This paper focuses on the thermal stress of tube plate and tube bundle, so only the tube plate and tube bundle are used for analysis.

Figure 1 and Figure 2 are respectively the diagram of the heat exchanger and the diagram of a spirally corrugated tube. The total length of the heat exchanger is 4680mm, the total length of the heat exchange tube is 4000mm, and the diameter of the shell side cylinder is 400mm. The difference in size between the axial and circumferential dimensions of the heat exchanger is too large, so you can consider dividing the heat exchanger in the axial direction. According to the position of the baffle, the heat exchanger...
can be divided into five areas A、B、C、D and E. Then the heat exchanger can be calculated in sections.

The heat exchanger structure is symmetric in the circumferential direction, but the temperature load is asymmetric. Because it has four tube passes, the temperature boundary conditions in each tube pass are different. The tube-plate split diagrams are shown in Figures 3 and 4. So the heat exchanger cannot be simplified by axisymmetric. However, the tube plate and the corresponding tube bundle can be divided into four zones by using the tube pass split, as shown in Figure 4, combined with the axial segmentation, each zone has five shaft segments 1A、1B、1C、1D and 1E、2A、2B、2C、2D and 2E... and so on.

Since there is a large fluid temperature difference between the tube and the shell side at the fluid inlet, the connection between the tube plate and the tube bundle here will have a large temperature gradient. At the same time, due to the discontinuous structure of the tube bundle and the tube plate, the heat exchange tube is restrained at both ends, and the restraint effect of the tube plate on the heat exchange tube far away from the tube plate is gradually reduced, and the temperature change trend is relatively gentle, which is resulting in large thermal stress at both ends of the tube bundle and small thermal stress in the middle. Therefore, only 1A and 2A structural models need to be established, as shown in Figures 5 and 6.

2.2. Element selection and mesh
In theory, the shell element has the advantages of fewer nodes and saving calculation time, but the solid element can consider the change of the structure thickness, so this paper chooses the solid element Solid186. The meshing process was performed with ANSYS Workbench v. 17.0(ANSYS, Inc., Canonsburg, PA, USA) using an unstructured grid, and the mesh size of the tube plate and tube bundle are 0.004m and 0.005m, respectively. The number of nodes in the 1A mesh is 1486715 and the number of cells is 865625. The number of nodes in the 2A mesh is 1607074 and the number of cells is 870317. The grid division results are shown in Figures 7 and 8.
3. Boundary conditions and solutions

3.1. Thermal analysis loading and boundary conditions

According to the simulation calculation results of HTRI, the thermal analysis boundary conditions of the heat exchanger can be obtained. The convective heat transfer coefficients and temperature results of the shell side and tube side calculated by HTRI are summarized in Table 1. The schematic diagram of the temperature load application is shown in Figures 9 and 10. [2]

| Grouping | Project | 1A | 2A | 3A | 4A | 1E | 2E | 3E | 4E |
|----------|---------|----|----|----|----|----|----|----|----|
| 1        | Temperature°C | 48 | 71 | 75 | 83 | 60 | 63 | 79 | 80 |
|          | H W/(m²·K)    | 2000 | 1900 | 1850 | 700 | 2100 | 2000 | 1500 | 1100 |
| 2        | Temperature°C | 100 | 98 | 95 | 90 | 87 | 87 | 87 | 87 |
|          | H W/(m²·K)    | 1450 | 1450 | 1450 | 1450 | 4000 | 4000 | 4000 | 4000 |

Notes: 1 is the inner tube wall surface and the tube side tube sheet surface; 2 is the outer tube wall surface and the shell side tube sheet surface; H is convection heat transfer coefficient.

3.2. Structural model loading and boundary conditions

The HTRI heat exchanger calculation software calculates the pressure of the tube side and the shell side, and obtains the static boundary conditions, as shown in Table 2. The schematic diagram of the static load boundary conditions is shown in Figure 11 and Figure 12.

| Item | Group | 1A | 2A | 3A | 4A | 1E | 2E | 3E | 4E |
|------|-------|----|----|----|----|----|----|----|----|
| Pressure (KPa) | Inner tube wall | 200 | 198 | 198 | 197 | 198.7 | 198.6 | 197.7 | 197.6 |
|      | Outer tube wall | 65 | 65 | 65 | 65 | 61.5 | 61.5 | 61.5 | 61.5 |

Figure 7. 1A grid structure  
Figure 8. 2A grid structure  
Figure 9. 1A temperature boundary conditions  
Figure 10. 2A temperature boundary conditions  
Figure 11. Static boundary conditions of 1A  
Figure 12. Static boundary conditions of 2A
4. Analysis of finite element results

4.1. Temperature field analysis
(1) Observing the eight components, it can be seen that the part with the largest temperature difference is at the junction of the tube plant and the heat exchange tube. The highest temperature is on the end of the tube bundle and the side of the tube plate shell side, and the lowest temperature is on the tube side of the tube plate.

(2) In the inlet section of the shell side of the heat exchanger, the component with the largest temperature difference is 1A, as shown in Figure 13, the highest temperature reaches 88.8°C, and the lowest temperature is 62.6°C; the component with the smallest temperature difference is 4A, as shown in Figure 14. At the exit section of the shell side of the heat exchanger, the component with the largest temperature difference is 1E, as shown in Figure 15, the highest temperature reaches 86.1°C, and the lowest temperature is 70.3°C; the component with the smallest temperature difference is 4E, as shown in Figure 16.

4.2. Thermal stress analysis
(1) Observing the thermal stress cloud diagrams of the eight components, the point of maximum stress is on the side of the shell side of the tube plate, the minimum stress is on the tube bundle, and the area with the most complicated stress distribution is in the opening area of the tube plate. The reason is that the structure here is not continuous and the thermal deformation is greatly restricted, which leads to the concentration of thermal stress. [4]

(2) Among the parts at the inlet section of the shell side, the parts with higher stress are 1A and 4A, and the maximum stress are respectively 720MPa and 804MPa, the results are shown in figure 17 and 18. Among the components in the exit section of the shell side, the components with greater stress are also 1E and 4E, and the maximum stress are respectively 718MPa and 773MPa.
5. Stress linearization assessment

The finite element analysis of the tube plate and the local structure of the tube bundle under the coupled load of the temperature-static field is carried out, and the stress intensity of the analysis results is evaluated. The basis of the evaluation is ASME BPVC VIII-2. The selection principle of the stress linearization path is: (1) Set the linearization path through the node with maximum stress intensity and the shortest distance along the wall thickness direction; (2) For the relatively high stress intensity area, set the path along the wall thickness direction. According to the relevant regulations of ASME BPVC VIII-2, the bending stress in the discontinuous zone of the overall structure should be classified as secondary stress. [5] Select 4A to evaluate the stress intensity, and the path selection is shown in Figures 19 and 20.

The maximum temperature of the tube plate stress maximum area is about 90 ℃, and the allowable stress of the tube plate material at this temperature is 172.5 MPa. The lower the temperature, the higher the allowable stress of the material. To be conservative, the stress value at the highest temperature point is selected for stress linearization evaluation. The 4A stress linearization curve is shown in Figures 21 and 22, and the stress evaluation results are shown in Tables 3.

![Figure 19. Path setting of the opening part](image1)
![Figure 20. Boundary path of tube plate](image2)

![Figure 21. 4A path 1 stress linearization curve](image3)
![Figure 22. 4A path 2 stress linearization curve](image4)

| Stress assessment path | Stress intensity combination type | Calculated value of stress intensity /MPa | Allowable value of stress intensity /MPa | Evaluation Results |
|------------------------|-----------------------------------|------------------------------------------|---------------------------------------|--------------------|
| Path 1                 | PL                                | 205                                      | 258.8                                 | Pass               |
|                        | PL+Pb+Q                           | 237                                      | 517.5                                 | Pass               |
| Path 2                 | PL                                | 140                                      | 258.8                                 | Pass               |
|                        | PL+Pb+Q                           | 720                                      | 517.5                                 | Fail               |

By analysis, it can be known that the tube plate is subjected to greater thermal stress under the action of temperature load. Besides, the tube plate stress distribution is uneven, and there are many dangerous nodes.

6. Conclusion

(1) The largest temperature difference is at the junction of the tube plate and the tubes. The highest temperature is on the end of the tube bundle and the side of the tube plate shell side, and the lowest
temperature is on the tube side of the tube plate.

(2) The point of maximum stress is at the junction of the tube plate and the tube bundle, the minimum stress is at the tube bundle. And the area with the most complicated stress distribution is in the opening area of the tube plate, where is also the most dangerous area.

References
[1] Hongwei Cai. Thermal stress analysis and structural optimization of fixed tube-plate heat exchangers[D]. Nanchang University, 2018.
[2] Mei Wu. Research on segmented heat transfer algorithm of shell-and-tube heat exchanger[D]. Dalian University of Technology, 2019.
[3] Jiuyi Liu. Study on the enhanced heat transfer and tube plate strength of shell and tube heat exchangers[D]. Beijing University of Chemical Technology, 2017.
[4] Jie Zhao, Feng Li, Moujun Liu. Thermal structure coupling field analysis of fixed tube-plate heat exchanger based on ANSYS[J]. New Technology and New Process, 2012(07): 37-40.
[5] JB4732-95. Steel pressure vessel.