The modal and harmonic response analysis of final superheater based on workbench

Yu Wan
Jiangsu Frontier Electric Technology CO., Ltd. Nanjing 211102, China
Email: wanyu_ft@163.com

Abstract. In order to avoid pipe failures affecting the operation of thermal power plants, the modal and harmonic response analysis method were carried out through the finite element software ANSYS Workbench to solve the long-term vibration of the final superheater at high temperature. Modal analysis can predict the natural frequencies of each stage of the structure and provide the frequency range for the harmonic response analysis. Then the response curve of stress to the frequency under the forced vibration was obtained on the base of harmonic response analysis, and observed the stress corresponding to the peak frequency. Furthermore, the fatigue life of the system was analyzed by the fatigue tool module in ANSYS Workbench. The results show that the fatigue safety factor of the final superheater pipe is greater than the allowed safety factor, and the fatigue life could meet the requirements for safe operation. The research results have certain reference value for further research on vibration fatigue of superheater pipelines.

1. Introduction
Coal-fired boiler superheater is a device that uses the heat of the products burned in the furnace to further heat saturated steam into superheated steam [1]. Multiple tubes are connected in parallel to form many tube screens in the final superheater, which are evenly distributed on the flow interface inside the boiler. The working temperature of this heating surface is highest among the pressure-bearing parts inside the boiler in the actual operation, the final superheater is disturbed by the external flue gas and the internal steam medium, forcing the tube panel to oscillate or even leak, causing the thermal power unit to be shut down. Grid peak shaving requires higher flexibility of thermal power units [2]. Therefore, the boiler needs to be operated under low load while ensuring a fast lifting load, which will further aggravate the vibration of the tube panel in the furnace. Therefore, it is very important to study the stress and fatigue life of the final superheater tube. Aditya et. al [3] performed CFD method to analyze flow induced vibration in superheater tube bundles, predicted vortex shedding and acoustic resonance. Aliakbar [4] used methods of metallography to measure oxide layer thickness and used CFD to identify the causes of failures of superheater tubes of a power plant’s boiler, and proved that the temperature difference between tubes is the cause of failure at the critical area. Felkowski [5] thought thermal expansion, high pressure and temperature lead to the shortening of superheaters lifetime. Few people use modal analysis and harmonic response analysis to analyze the strength and life of superheater tube vibration. And there are few researches on stress analysis and life prediction of superheater tube by methods of modal analysis and harmonic response analysis. It is convenient to use modal analysis and harmonic response analysis to learn the vibration of the structure.
and its weak parts [6]. It can provide reference for the optimization of structure and prediction of usage time.

In this paper, the finite element analysis software ANSYS Workbench is used to simulate and analyze the final superheater tube of a boiler plant. First, the three-dimensional solid model was created by CREO and meshed by ICEM. Then the modal analysis of the tube was carried out, and its natural frequency and mode are determined, which provides a frequency range for harmonic response analysis. Then, to observe the stress corresponding to the peak frequency, the curve of displacement versus frequency under forced vibration is obtained by harmonic response analysis which is based on the complete method. And the fatigue life is computed by the Fatigue Tool module.

2. Model and boundary condition

The final superheater consists of 20 tube coils. Because there are two structures of straight pipe and elbow at the inlet and outlet of the tube coil, the two different types of pipe w analyzed respectively. And piping specification for $\Phi$44.5×8mm in diameter. Figure 1 is the schematic diagram of final superheater structure. To simplify the calculation, take the outermost straight pipe and elbow for analysis, and its three-dimensional model is shown in Figure 2 and Figure 3.

Material properties: The materials of the pipeline are mainly TP347H and T91, and the welding material is Inconel 82. Referring to The practical manual of pressure vessel materials-carbon steel and alloy steel [7], their mechanical properties have been listed in Table 1.

Meshing: In the pipeline model, the solid element was generated by the sweep method, and to ensure the accuracy of calculation, girth welds, elbow, and other important parts were treated with mesh encryption. The number of grids has a great impact on the calculation accuracy, so as shown in Figure 4, the grid independence is verified, and the final number of grids is determined to be 431,752, with grid quality above 0.25, and the encrypted grids at the local weld are shown in Figure 5.

Boundary conditions: According to the actual working conditions of the superheater, fixed constraints should be applied to the end face and circumference of the sleeve.

Loads: There is a normal pressure of 24MPa on the inner wall of the pipe, and the ambient temperature is 600℃. Besides, the simple harmonic vibration frequency of the pipeline is 0.1-0.2Hz, and the node displacement load at the pipeline bottom is 250mm.

---

Figure 1. Schematic diagram of the final superheater structure (unit: mm).
Table 1. Mechanical properties of relevant materials.

| Materials | Density (Kg/m³) | Elasticity modulus E 10^5MPa | Poisson’s ratio μ | Linear expansion coefficient αs 10^-6K^-1 | Thermal conductivity λ W/(m/K) | Yield strength σ0.2 Mpa | Tensile strength σb Mpa |
|-----------|-----------------|-----------------------------|------------------|-----------------------------------------|-------------------------------|----------------------|-----------------------|
| T91       | 7780            | 1.7                         | 0.29             | 12.4                                    | 29.3                          | 326                  | 358                   |
| TP34 7H   | 8000            | 1.52                        | 0.29             | 18.7                                    | 23.4                          | 190                  | 390                   |
| Inconel 82| 7990            | 1.99                        | 0.3              | 1.99                                    | 21.4                          | 290                  | 290                   |

Figure 2. Straight tube.  
Figure 3. Elbow pipe.  
Figure 4. Grid independence verification  
Figure 5. Local grid refinement.
3. Modal analysis

Modal analysis is the basis of dynamic analysis, which is used to determine the vibration characteristics of the structure. The vibration characteristics of the structure are composed of the natural frequency and the mode. In order to avoid the vibration damage caused by poor structural design, modal analysis can be used to predict the natural frequencies, modes and the possibility of resonance [8].

When the influence of damping in the structure is ignored, the free vibration equation is as follows:

$$[M]\ddot{\{X\}} + [K]\{X\} = 0$$  \hspace{1cm} (1)

In equation (1), $[M]$ is the mass matrix in system, $[K]$ is the stiffness matrix in system, $\{X\}$ is the displacement response vectors at each point of the system.

Since the structure is free vibration in the form of simple harmonic motion, let:

$$\{X\} = \{\phi\}_i \sin(\omega_i t)$$  \hspace{1cm} (2)

In equation (2), $\{\phi\}_i$ is the eigenvector of the mode of the phrase $i$ natural frequency, $\omega_i$ is the natural angular frequency of phrase $i$.

By substituting the simple harmonic motion equation into the kinetic motion equation, the equation can be expressed as:

$$([K] - \omega^2[M])\{X\} = \{0\}$$  \hspace{1cm} (3)

3.1. Solution and result analysis

In typical modal analysis, because the only effective load is the zero displacement constraint, only boundary conditions is need ed to be set in the finite element model. The numerical analysis and simulation are carried out and show the first six modes in Figure 6 and 7.

**Figure 6.** The six modal shapes of the elbow pipeline.
Table 2. Inherent frequency of pipeline.

| Elbow pipe | Inherent frequency(Hz) | maximum amplitude(mm) | Straight pipe | Inherent frequency(Hz) | maximum amplitude(mm) |
|------------|------------------------|------------------------|---------------|------------------------|------------------------|
| 1          | 0.15741                | 4.0561                 | 1             | 0.14014                | 4.065                  |
| 2          | 0.26456                | 3.2321                 | 2             | 0.2357                 | 3.2417                 |
| 3          | 0.64316                | 3.1093                 | 3             | 0.56795                | 3.122                  |
| 4          | 1.0186                 | 3.6586                 | 4             | 0.90658                | 3.6638                 |
| 5          | 1.0309                 | 3.5001                 | 5             | 0.91857                | 3.4911                 |
| 6          | 1.4868                 | 3.4301                 | 6             | 1.3234                 | 3.4148                 |

As shown in Table 2, with the increase of the modal order, the natural frequency of the bent pipe increase from 0.15741 to 1.4868, and that of the straight pipe increase from 0.14014 to 1.3234. And the modal analysis of the first six orders shows that the natural frequency of the bent pipe and the straight pipe are close to each other, and the modes are roughly the same. The first-order modal shape mainly shows that the pipeline swings longitudinally to the left, and with the amplitude gradually decreasing from bottom to top, the maximum amplitude located at the bottom of the pipeline. The second-order modal shape is mainly manifested as the right lateral vibration of the pipeline, with the amplitude gradually decreasing from bottom to top, and the largest part was located at the bottom of the pipeline. The third-order modal shape is mainly manifested as alternating longitudinal vibration on both sides of the pipeline, and the maximum amplitude is located in the middle of the pipeline. The fourth-order modal shape is mainly manifested as the longitudinal vibration of the pipeline, and its larger displacement appears in the middle and the bottom of the pipeline. The fifth-order modal shape is mainly manifested as alternating lateral vibration of both sides of the pipeline and inward vibration, with the maximum amplitude located in the middle of the pipeline. The sixth-order modal shape is

Figure 7. The six modal shapes of the straight pipeline.

As shown in Table 2, with the increase of the modal order, the natural frequency of the bent pipe increase from 0.15741 to 1.4868, and that of the straight pipe increase from 0.14014 to 1.3234. And the modal analysis of the first six orders shows that the natural frequency of the bent pipe and the straight pipe are close to each other, and the modes are roughly the same. The first-order modal shape mainly shows that the pipeline swings longitudinally to the left, and with the amplitude gradually decreasing from bottom to top, the maximum amplitude located at the bottom of the pipeline. The second-order modal shape is mainly manifested as the right lateral vibration of the pipeline, with the amplitude gradually decreasing from bottom to top, and the largest part was located at the bottom of the pipeline. The third-order modal shape is mainly manifested as alternating longitudinal vibration on both sides of the pipeline, and the maximum amplitude is located in the middle of the pipeline. The fourth-order modal shape is mainly manifested as the longitudinal vibration of the pipeline, and its larger displacement appears in the middle and the bottom of the pipeline. The fifth-order modal shape is mainly manifested as alternating lateral vibration of both sides of the pipeline and inward vibration, with the maximum amplitude located in the middle of the pipeline. The sixth-order modal shape is
mainly manifested as the transverse vibration of the pipeline, and its large displacement occurs in the middle and the bottom of the pipeline. The maximum amplitude of each mode is between 3.4-4.1mm, and there is a maximum amplitude in the first mode.

4. Harmonic response analysis
Harmonic response analysis can be used to obtain the steady-state forced response of the system under sinusoidal load. Through the harmonic response analysis, the curve relationship between the response of the structure under excitation and frequency can be obtained, the “peak” response can be seen from these curves, and the amplitudes of stress and displacement corresponding to the peak frequency can be further observed. Furthermore, by analyzing the dynamic characteristics of the structure, we can predict the continuous dynamic characteristics of the structure and determine whether the design of the structure can overcome the harmful results caused by resonance and other forced vibration. Minimize the failures and losses that will occur in the design phase as much as possible. [9].

The motion equation of forced vibration of structure:

\[
[M] \dddot{\{X\}} + [C] \ddot{\{X\}} + [K] \{X\} = \{F\} \sin(\theta t) \tag{4}
\]

In equation (4), \([C]\) is the damping matrix in system, \([F]\) is the exciting force vector, \(\{X\}\) is the displacement response vector, \(\ddot{X}\) is the velocity vector, \(\dddot{X}\) is the acceleration vector.

The displacement response:

\[
\{X\} = \{A\} \sin(\theta t + \varphi) \tag{5}
\]

In equation (5), \(\varphi\) is the phase Angle of the displacement response hysteresis excitation, \(A\) is the displacement amplitude vector.

4.1. Solution and result analysis
In this paper, harmonic response analysis based on the complete method is used to solve the problem. A node displacement load with amplitude of 250mm in the z direction was added to the bottom position of the pipe. The frequency range was set from 0Hz to 0.5Hz, and the phase Angle is set as 90, and the ambient temperature is set as 600°C, and the solution intervals is set as 10. Figure 8-9 show the maximum displacement and maximum equivalent stress response curves.

**Figure 8.** Maximum displacement response curve (0-2.5Hz).

**Figure 9.** Maximum equivalent stress response curve (0-2.5Hz).
As shown in Figure 8 and 9, in the range of 0-2.5Hz, elbow and straight pipe have similar trend of displacement and maximum equivalent stress. The peak values of maximum displacement and maximum equivalent stress appear at 0.75Hz and 2Hz, and the maximum equivalent stress far exceeds the yield stress of the material. As shown in Figure 10 and 11, because the pipe is too long, when the pipe swings at the bottom, it is easy to produce a large displacement in the middle, and because the top casing limits the displacement of the pipe, there will be greater stress at the casing, which is prone to fracture. Therefore, it is recommended to apply drivepipe in the middle and bottom of the pipe to avoid large displacement.

At the same time, in order to determine whether the pipe is safe in the frequency range of given equipment operation, the harmonic response is simulated in the range of 0-0.5Hz. Figure 12 and 13 show the maximum displacement and maximum equivalent stress response curves in this frequency range. It can be seen that the maximum stress of the tube does not exceed the yield strength of the material 190 MPa, so the superheater can operate safely in this range.

![Figure 10. Deformation of straight pipe.](image1)

![Figure 11. Deformation of elbow pipe.](image2)

![Figure 12. Maximum displacement response curve(0-0.5Hz).](image3)
4.2. Fatigue life analysis
In the current study, the simple harmonic vibration fatigue of pipelines belongs to low stress and high cycle fatigue. On the base of harmonic response calculation, the Fatigue Tool module was added, and the Fatigue strength factor was set to 0.8. Symmetrical cyclic loads were selected to simulate the “positive - negative” fatigue. In Fatigue Tool, Insert/life, Safety Factor and Damage options were selected to set the Fatigue life, Safety Factor and cumulative Fatigue Damage Factor of the pipelines, and the design life of the pipeline was defined as cycle times. Table 3 is the minimum number of cycles calculated at different frequencies [10].

Figure 13. Maximum displacement response curve(0-0.5Hz).

Figure 14. Damage.
Figure 15. Safety factor.

a. Straight pipe

b. Elbow pipe

Figure 16. Life figure.

a. Straight pipe

b. Elbow pipe
Table 3. The minimum cycles at different frequencies.

| Frequency (Hz) | Minimum number of cycles |
|---------------|--------------------------|
| 5.00E-02      | 2.00E+07                 |
| 0.1           | 1.00E+07                 |
| 0.15          | 6.67E+06                 |
| 0.2           | 5.00E+06                 |
| 0.25          | 4.00E+06                 |
| 0.3           | 3.33E+06                 |
| 0.35          | 2.86E+06                 |
| 0.4           | 2.50E+06                 |
| 0.45          | 2.22E+06                 |
| 0.5           | 7.45E+05                 |

By analyzing the above results, it can be seen that with the increase of stress frequency, the minimum cycle number gradually decreases, so the pipe fatigue characteristics only need to be analyzed at 0.2Hz:

1) The damage result can be understood as the ratio of design life to usable life. When the damage value is greater than 1, fatigue failure could happen. As shown in Figure 14, the maximum cumulative fatigue damage coefficient is 2E-7<1, indicating that the design of the pipeline is qualified.

2) Safety factor is the ratio between the failure stress of the material used and the design stress defined in the parts or components. As shown in Figure 15, it can be seen that the contours of the pipeline are all red, and the fatigue safety factor is 15>1, indicating that the pipeline analyzed and designed is safe.

3) As shown in Figure 16, the minimum life is 5E+6 cycles. According to the mechanical fatigue theory, when the number of stress cycles the welded parts endure reaches 2E+6 without being destroyed, it is considered that they can withstand infinite stress cycles.

5. Conclusions

Due to the long-term vibration of the superheater pipeline, the failure of the welded joint is easy to occur due to fatigue damage, and even lead to the explosion of the pipe and other dangerous accidents. In the paper, a three-dimensional pipe model was established by CREO and then imported into ANSYS Workbench to be analyzed by finite element method. The analysis results are as follows. 1) Calculate the first six orders of natural frequency by modal analysis, get the amplitude and modal shape, and conduct a complete method of harmonic response analysis: when the pipeline is doing simple harmonic vibration, the frequency is within 0.1-0.2Hz. By obtaining the stress-frequency response curve and the equivalent stress cloud map at the peak, it can be known that the stress is less than the yield stress. 2) On the basis of harmonic response analysis, Fatigue Tool module was added to carry out Fatigue analysis of the pipeline. The results show that the pipeline fatigue safety factor is larger than the allowable safety factor, and the Fatigue cumulative damage factor is reasonable, so the final superheater could operate safely. The above results can be used to estimate the fatigue life of the superheater.

Acknowledgement

This study was supported by the Key Research and Development Foundation of Jiangsu Province (Grant No. 2018085).

References

[1] Othman, H, J. Purbolaksono, and B Ahmad 2009 Failure Investigation on Deformed Superheater Tubes Engineering Failure Analysis 16(1) 329-39

[2] Kim, Beom-Soo, Jung-Chel Chang, Yong-keun Jung 2010 Failure Analysis of the Super Heater
Tubes of S-Combined Power Plant *Engineering Failure Analysis* 17(4) 864-72.

[3] Aditya Kumar Pandey. L. A. Kumaraswamidhas. C. Kathirvelu. 2012 Analysis Of Flow Induced Vibration In Superheater Tube Bundles In Utility Boilers Using Computational Method *International Journal of Computational Engineering Research* 2(2) 5

[4] Aliakbar Shami. Seyed Ebrahim Moussavi Torshizi. Ali Jahangiri. Failure Analysis and Remedial Solution Suggestion for Superheater Tubes of a Power Plant Boiler *Transactions of the Indian Institute of Metals*

[5] Felkowski Łukasz. Mędrala Jakub 2019 Analysis of superheater tubes failure *E3S Web of Conferences* 82 1002

[6] Krishnasamy, Ram Kumar 2012 A Computational Approach to Thermomechanical Fatigue Life Predictions of Dissimilarly Welded Superheater Tubes

[7] Pan jiazhen 2000 The practical manual of pressure vessel materials-carbon steel and alloy steel.

[8] Long-ping, Zhang 2012 Finite Element Modal Analysis of V8 Engine Crankshaft Based on ANSYS Workbench *Machinery Design and Manufacture*

[9] Mathan, G, and N. Siva Prasad 2012 Study of Dynamic Response of Piping System with Gasketed Flanged Joints Using Finite Element Analysis *International Journal of Pressure Vessels and Piping* 89 28-32

[10] Wang Guoqiang 1999 Practical engineering numerical simulation technology and its application in ANSYS *Xi’an:Northwestern Polytechnical University Press* 86-93