Journal of Energy and Natural Resources
2023; 12(1): 1-6
http://www.sciencepublishinggroup.com/j/jenr
doi: 10.11648/j.jenr.20231201.11
ISSN: 2330-7366 (Print); ISSN: 2330-7404 (Online)

Stress Analysis of Large Diameter Pipe Interface Structure of Boiler Main Steam Pipe

Chen Ye*, Zheng Nenghong, Yang Rui, Xia Fengbing, Liu Tao, Huang Renjie, Chai Lijun

College of Mechanical Engineering, Sichuan University of Science and Engineering, Yibing, People's Republic of China

Email address:
cy32428yr@suse.edu.cn (Chen Ye), 995873807@qq.com (Zheng Nenghong), 501772641@qq.com (Yang Rui), 207095863@qq.com (Xia Fengbing), 2242991685@qq.com (Liu Tao), 812348492@qq.com (Huang Renjie), 1158656497@qq.com (Chai Lijun)

*Corresponding author

To cite this article:
Chen Ye, Zheng Nenghong, Yang Rui, Xia Fengbing, Liu Tao, Huang Renjie, Chai Lijun. Stress Analysis of Large Diameter Pipe Interface Structure of Boiler Main Steam Pipe. *Journal of Energy and Natural Resources*. Vol. 12, No. 1, 2023, pp. 1-6. doi: 10.11648/j.jenr.20231201.11

Received: March 17, 2023; **Accepted**: May 6, 2023; **Published**: May 18, 2023

Abstract: The "four tubes" safety of the boiler is related to the economic and safe operation of the boiler. In order to ensure the safety of boiler steam pipe large diameter nozzle welding place, the structural stress analysis of the special large-aperture nozzle pipeline designed in a boiler enterprise's design process is studied in this paper. In the case, the allowable angle of the pipeline exceeds the direct calculation range of JB4732-1995, therefore, the ANSYS finite element analysis method is used to calculate the structural stress of the pipeline, and the stress evaluation is carried out. The results show that the finite element method can effectively calculate the pipeline structure's stress calculation. In order to meet the requirements of JB 4732-1995 for the evaluation of various types of stress intensity step by step, the film bending stress is treated as S_{III}, and the conservative treatment is controlled by 1.5Sm, therefore, the finite element analysis results of the pipeline show that the maximum equivalent stress is 176.22 MPa, which is located at the connection between the large nozzle and the main pipe, and the larger the diameter of the pipe nozzle, the higher the equivalent stress. The stress evaluation results of the pipeline are evaluated according to the third stress intensity, and the strength of the analyzed parts meets the standard requirements.

Keywords: Large Aperture Takeover, Finite Element Method, Stress Analysis, Stress Evaluation

1. Introduction

The "Four-tube" safety of the boiler is related to the safe and economic operation of the boiler, so it has been wide a concern by domestic and foreign boiler enterprises and research institutions. The boiler's main steam pipes are used to transport high-temperature and high-pressure steam. The pressure and temperature of the working medium in the pipe are high. Once the leakage or even fracture occurs, it will seriously endanger the safety of the operator and the normal operation of the boiler unit.

The material used in the steam pipeline of the boiler is mainly structural steel. This material is sensitive to temperature changes. When the temperature rises to a certain extent, the performance of structural steel will change greatly. Excessive steam temperature is easy to cause high-temperature failure. In addition, the steam pipeline will cause fatigue failure of the pipeline under the alternating load of repeated pressurization and pressure relief [1, 2]. Different pipeline structures and stress conditions are very complicated, and there are certain requirements for the primary stress and secondary stress generated by the pipeline. Therefore, scholars at home and abroad have conducted extensive and in-depth research, mainly focusing on creep life analysis [3-6], pipeline performance evaluation monitoring [7, 8] stress analysis [9-11], and failure analysis [12]. To ensure the safety of boiler steam pipes, after long-term research, China has formulated JB4732-1995 "steel pressure vessel-analysis and design standard" [13], GB150-2011 "steel pressure vessel" [14] and GB/T 16507-2022 "water pipe boiler" [15] and other standards suitable for safety assessment and design of boiler pipes. However, in the design process of boiler enterprises, there are still special pipelines that exceed the existing standards. For example, it is very difficult to calculate the stress at the interface of large-diameter nozzle pipes with an allowable angle of more than 45° according to the published JB4732-1995 standard. There is a certain deviation between the results of primary stress, secondary stress, and the actual situation. Therefore, in the existing situation, the strength analysis and evaluation of the pipeline with a large diameter nozzle cannot be completed by simple calculation, and the strength evaluation can only be carried out by the finite element method.

In this paper, the problem of structural stress and stress assessment of special large-diameter nozzle pipes are discussed which needed to be designed in the design process of a boiler plant, the ANSYS finite element analysis software is used to establish the stress calculation model of steam pipes according to the design scheme. The stress linearization method is used to carry out the strength assessment of steam pipes at the interface of large-diameter pipes to ensure that the strength of boiler steam pipes meets the requirements.

2. Finite Element Model

2.1. Steam Pipe Geometry and Performance Parameters

According to the design requirements of a boiler plant, the size of the steam pipe is Φ219mm×16mm. There is a large aperture nozzle of Φ133mm×10mm in the horizontal 270° direction of the cylinder and the horizontal 192° direction. There are three small nozzles of Φ60mm×5mm in the direction of 90°. The distance between the small nozzles is 90mm, and the nearest distance between the small nozzle and the center point of the main nozzle is 10mm in the vertical direction. The basic design parameters and material parameters of the pipeline are shown in Table 1 and Table 2.

Table 1. *Basic design parameters of the pipeline.*

Projects	Unit	Parameter
Design pressure P_c	MPa	5.09
Design temperature	°C	275
corrosion addition	mm	0.5

Table 2. *Main material parameters of the pipeline.*

Projects	Unit	Parameter
material type		20G (steel tube)
Elastic modulus Et	MPa	181000
Poisson ratio μ		0.3
Thermal expansion coefficient α	(10-6/°C)	12.78
Allowable stress [σ]t	MPa	110

2.2. Finite Element Model

Stress assessment of the model is an important means to ensure the normal operation of the boiler. According to the design parameters, the allowable angle of the steam pipeline studied in this paper is 58°, which is beyond the scope of direct use of the JB4732-1995 steel pressure vessel-analysis design standard. Therefore, the finite element method is used to evaluate the stress of the model.

The finite element model shown in Figure 1 was established by ANSYS software. The following simplifications were made in the process of establishing the model: local details such as welds and fillets were ignored. The corrosion margin was considered in the model, and the wall thickness was subtracted by 0.5mm on the original basis. The 20-node hexahedral element solid186 of ANSYS software is used to divide the grid. The header cylinder is divided into 6 layers along the thickness direction, the wall thickness of the large nozzle is divided into 5 layers, and the wall thickness of the two small nozzles is divided into 4 layers. The whole finite element model is divided into 541139 nodes and 128,970 elements. The mesh discretization is shown in Figure 2.

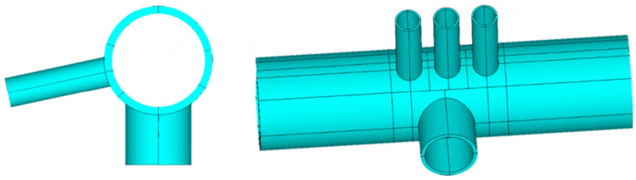


Figure 1. *Finite element model of steam pipeline.*

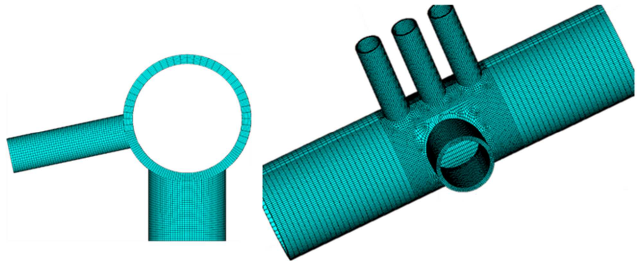


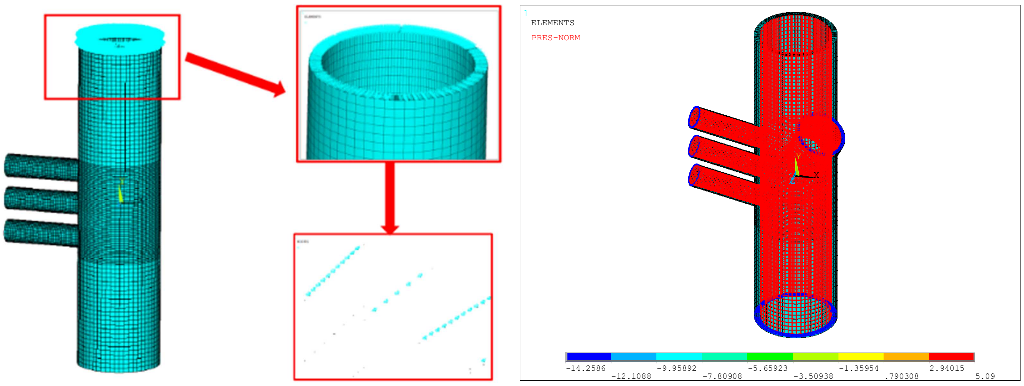
Figure 2. *Mesh generation of finite element model.*

2.3. Boundary Condition

Because the working medium in the boiler steam pipeline has pressure, the load applied in the finite element analysis is mainly divided into pressure load and temperature load. The design pressure is applied to the inner surface in contact with the medium, that is, internal pressure is applied to all inner surfaces, and the design pressure Pc=5.09MPa. The operating temperature of the whole model is 275°C and the ambient temperature is 20°C. According to the operation of the boiler, the constraints on the steam pipe are as follows:

1) the vertical displacement and rotation will be limited at the end (X=0);
2) The nodes of the end face are taken for circumferential displacement and axial restraint, that is, UY = 0, UZ = 0 in cylindrical coordinates;
3) In another end face and nozzle end face, apply the equivalent surface force generated by internal pressure.

The application of the boundary conditions and the loading are shown in Figure 3.



(a) (b)

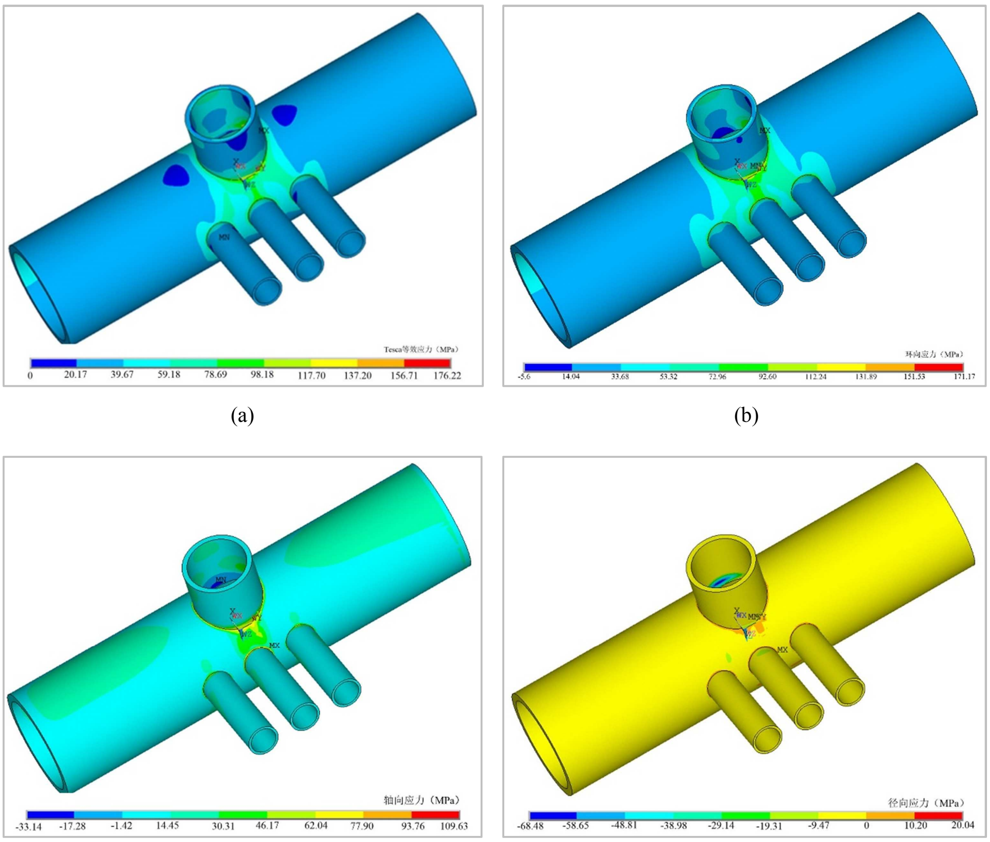
Figure 3. *Calculation model boundary conditions and load application ((a) computing model constraints; (b) applying load to the calculation model).*

3. Results and Analysis

3.1. Stress Cloud

Under a load of each design condition, the third strength cloud diagram of the pipeline is shown in Figure 4. It can be seen from the diagram that the bending moment caused by the opening of the nozzle makes the equivalent stress of the nozzle attachment lower, which is determined by the three stresses. The stress concentration is mainly in the main pipe and Dalian nozzle and small connecting pipe opening welding position. Figure 5 is the equivalent stress cloud diagram at the opening of the connecting pipe of two sizes. It can be seen from Figure 5 that the larger the opening size of the connecting pipe, the higher the equivalent stress, and the higher the possibility of danger at the opening. The maximum equivalent stress appears at the Dalian nozzle, with a maximum value of 176.22 MPa, while the maximum equivalent stress at the small nozzle interface is 142.29 MPa.

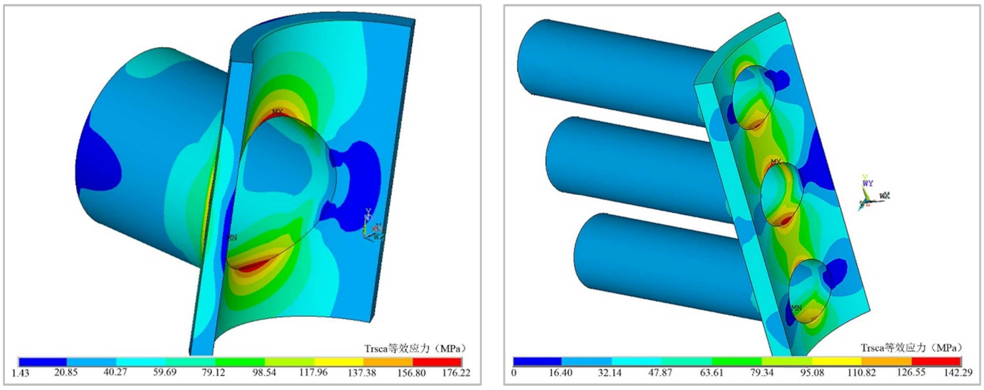
The figures should be clear and they should be numbered as Figure 1, Figure 2, Figure 3 etc. There should be annotations behind each figure as follows:



(a) (b)

(c) (d)

Figure 4. *The third strength cloud of the pipeline ((a) Overall Tesca equivalent stress cloud map; (b) Overall circumferential stress S1 cloud diagram; (c) Overall axial stress S2 program; (d) Overall radial stress S3 program).*



(a) (b)

Figure 5. *Local Trsca equivalent stress program of the large and small nozzle ((a) large nozzle; (b) small takeover).*

3.2. Stress Assess

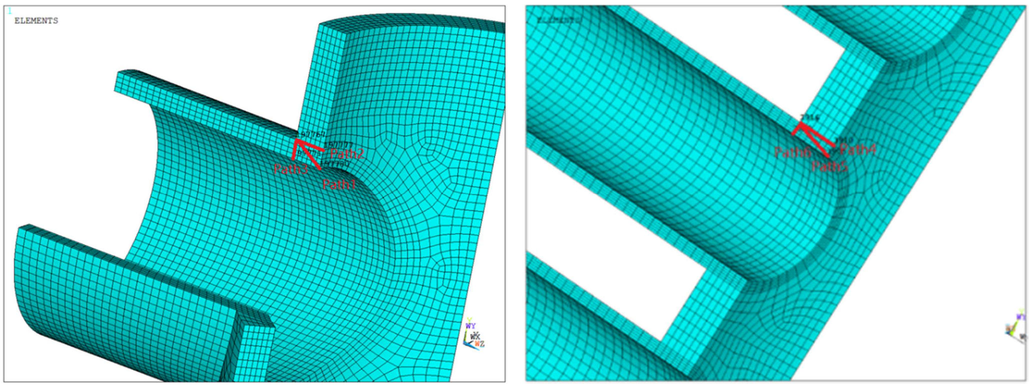
According to the stress distribution cloud diagram of the pipeline in Figure 4 and Figure 5, the maximum stress distribution surface is determined, then the most dangerous path is selected, and the calculation results are linearized to provide a basis for subsequent stress evaluation. According to JB4732-1995 ' steel pressure vessel-analysis design standard ' evaluation standard:

1) In the primary stress zone, the maximum average equivalent stress of the inner and outer walls or the average equivalent stress of the section is less than 1 times the allowable stress;
2) In the primary stress zone, the equivalent stress of the maximum equivalent stress or the sum of the section average stress and the bending stress is less than 1.5 times the allowable stress;
3) In the secondary stress zone, the maximum average stress of the inner and outer walls or the average equivalent stress of the section is less than 1.5 times the allowable stress;
4) In the secondary stress zone, the equivalent stress of the maximum equivalent stress or the sum of the section average stress and the bending stress is less than 3 times the allowable stress.

According to the selection principle of the stress linearization path, the linearization path is set by the maximum stress intensity node and the shortest distance along the wall thickness direction. The path is set along the wall thickness direction for the relatively high-stress intensity area. A total of 6 evaluation paths are selected for the large and small nozzles. The name and node number are shown in Table 3, and the specific path distribution is shown in Figure 6.

Table 3. *Evaluation Path.*

Path number	node number	node number
1	197767	197780
2	197767	197780
3	197767	197771
4	3916	3905
5	3916	3910
6	3916	3899



(a) (b)

Figure 6. *Assessment path of the stress assessment area ((a) large nozzle; (b) small pipe).*

According to Figure 6, the stress linearization path is set at the discontinuity of the structure. Through the function of stress linearization along the path provided by ANSYS software, various types of stress along the path of wall thickness can be distinguished. The results of stress linearization along each path are listed in Table 4 stress classification.

The bending stress near the nozzle or other openings contains both the primary stress component caused by static equilibrium and the secondary stress component caused by structural discontinuity.

Due to the finite element software, the total amount of membrane bending stress can only be given after linearization in ANSYS, and these two stresses cannot be further subdivided. Therefore, considering that the total bending stress at this place contains the primary bending stress required for static equilibrium and the secondary bending stress caused by deformation coordination since the two components cannot be distinguished, there is no additional bending stress in the external load in this paper. The bending stress in the path is used as the secondary stress, that is, it is reasonable and feasible to evaluate according to S_{IV}, and $3S_m$.

Further analysis, in order to meet the requirements of JB 4732-1995 for the evaluation of various types of stress intensity step by step, the film bending stress is treated as S_{III}, and the conservative treatment is controlled by $1.5S_m$, which is a very conservative way. Only Path1 is not rounded due to stress concentration, which exceeds the limit by 6%. Therefore, this paper still uses the method of JB4732-1995 to evaluate the third stress strength. According to the analysis and evaluation results, the strength of the analyzed part meets the standard requirements.

Table 4. *Stress evaluation results.*

Projects and Paths	$S_{II}(P_L)$ MPa		$S_{IV}(P_L+P_b+Q)$ MPa		Conclusion
	Calculated value	Allowable value $1.5KS_m$	Calculated value	Allowable value $3KS_m$	
Path1	114.42	165	175.75	330	Pass
Path1	94.27	165	130.09	330	
Path1	88.16	165	112.69	330	
Path1	100.51	165	144.44	330	
Path1	85.66	165	116.51	330	
Path1	78.44	165	95.27	330	

4. Conclusion

In this paper, the structural stress and stress assessment of the large opening nozzle pipeline in the design process of a boiler steam pipeline in a boiler enterprise are analyzed, and the following conclusions are obtained:

(1) The allowable angle of the pipeline interface is 58°, which is higher than the requirement of ≤ 45° which specified in the JB 4732 - 1995 design. The stress assessment of the pipeline can only be carried out by the finite element method.

(2) The finite element analysis results of the pipeline show that the maximum equivalent stress is 176.22MPa, which is located at the connection between the large nozzle and the main pipe.

(3) The results of the pipeline stress assessment show that the third stress intensity is evaluated. According to the analysis and evaluation results, the strength of the analyzed part meets the standard requirements.

Acknowledgements

This work is financially supported by the talent introduction project of Sichuan University of Science & Engineering (2019RC18) and the key laboratory of process equipment and control engineering in colleges and universities of Sichuan province (gk201910, gk202009).

References

[1] Bian Xiaoke. Failure analysis and life prediction of main steam pipelines [J]. Building materials and decoration, 2016, No. 419 (15): 165-6. (in Chinese)

[2] Wei Shengjun, Li Fufu, Qi Chang. Crack problems and prevention in inspection of boilers, pressure vessels and pressure pipelines [J]. Science and information technology, 2023, (2): 16-8. (in Chinese)

[3] Salifu S, Desai D, Kok S. Influence of Diverse Operating Cycles on the Useful Creep Life of P92 Steam Piping [J]. Journal of Failure Analysis and Prevention, 2021, 21 (3): 983-92.

[4] Murakami K, Komazaki S-I, Mitsueda T. Creep Remaining-Life Assessment of 2.25Cr-1Mo Steel Hot Reheat Steam Piping by Small Punch Test [J]. Tetsu-to-Hagane, 2022, 108 (1): 88-96.

[5] Chen Xingyang, Zhou Yang, Song Junjun, et al. Remaining Life Assessment of Main Steam Piping in Long-term High Temperature Service Power Plant [J]. Physical and Chemical Inspection-Physical Volume, 2023, 59 (01): 13-5+38. (in Chinese)

[6] SToresund J, Andersson D, Rantala J, et al. Creep analysis of a main steam pipe system [J]. Materials at High Temperatures, 2022, 39 (6): 678-88.

[7] GRIN' E A, STEPANOV V V, SARKISYAN V A, et al. Method for evaluating the technical state of boilers and piping in thermal power plants [J]. Power Technology and Engineering, 2012, 45 (5): 369-75.

[8] Choi W, Han J. Health-Monitoring Methodology for High-Temperature Steam Pipes of Power Plants Using Real-Time Displacement Data [J]. Applied Sciences, 2021, 11 (5).

[9] Pástor M, Lengvarský P, TREBUŇA F, et al. Prediction of failures in steam boiler using quantification of residual stresses [J]. Engineering Failure Analysis, 2020, 118.

[10] Qi Haifeng. Causes and prevention of cracks in boiler main steam pipeline [J]. Labor protection, 2021, (8): 80-1. (in Chinese)

[11] Liu Cong, Shen Liang, Lu He. Steam pipe stress calculation and support selection of 220t / h boiler [J]. Power plant system engineering, 2021, 37 (04): 47-8 + 50. (in Chinese)

[12] Lyu Y, Lian w, Sun z, et al. Failure Analysis of Abnormal Bulging and Cracking for High-Pressure Steam Pipe [J]. Journal of Materials Engineering and Performance, 2022, 31 (9): 7277-89.

[13] JB4732-1995, Steel Pressure Vessels, Design by Analysis, 2005. (in Chinese)

[14] GB 150.3-2011, Pressure Vessel Part 3: Design. (in Chinese)

[15] GB/T 16507-2022, Water Pipe Boiler. (in Chinese)