C. D. WU, D. L. GUO, M. W. HE, C. J. WANG, X. D. MENG, L. J. ZHANG, K. WANG, H. C. JI
ISSN 0543-5846
METABK 63(3-4) 435-438 (2024)
UDC – UDK 669.14.018:697.8:539.374:518.1:620.16-539.5=111

DEFORMATION ANALYSIS BEFORE AND AFTER HOISTING AND REINFORCEMENT OF LARGE FLUE

Received – Primljeno: 2024-01-29
Accepted – Prihvaćeno: 2024-03-30
Original Scientific Paper – Izvorni znanstveni rad

In the process of carrying out the installation of large flue, because the thickness of the flue is thin, it is easy to deform when hoisting with wire rope, and too much deformation will affect the subsequent use of the flue, resulting in waste, so in order to reduce the deformation of the flue, the existing conditions are utilized, and the bracket is added inside the flue to play the role of support, and the deformation of the flue is reduced. Using 3D modeling and numerical simulation software, the flue structure before and after adding supports was modeled and numerically analyzed, and the two simulation results were compared, and the deformation of the flue after adding supports was analyzed to be relatively obvious under the action of hoisting.

Keywords:Q235 steel; flue; numerical simulation; plastic deformation; stress-strain

INTRODUCTION

Hoisting is a type of construction work that uses lifting equipment to lift components and install them fixed in the design position. It is divided into sub-piece hoisting and overall hoisting. The former is to hoist the various components of buildings and structures in turn; The latter is to assemble each component into a whole structure on the ground and then hoist. This time, the former is used, and different types of flues are hoisted to the designated position in order of succession. Jiang Shan et al. used the rigid-plastic modal approximation method and ABAQUS finite element analysis to study the deformation mode and large deflection of concrete-filled steel pipe (CFST) structural components under lateral impact load. Using the yield criterion and the foundation beam model, a modal analytical solution of the plastic behavior of concrete-filled steel tubular components considering the interaction between bending and axial tensile bearing under lateral impact is obtained. The rigid-plastic mode solution captures the finite element results, and the deformation modes are consistent with the existing experimental results[1]. Wang et al. used the Finite Element ABAQUS to numerically simulate the bearing capacity of a circular concrete-filled steel tube column with local corrosion under eccentric load. The effects of corrosion site, corrosion volume rate, and corrosion area rate on the bearing capacity were also compared [2]. Cao Shuiliang et al. used the structural finite element analysis method to analyze and calculate the stress of the steam pipeline under complex working conditions, and through comparative analysis, it can be concluded that the maximum stress of the pipeline before the addition of pipeline support and hanger is 232,4 MPa, and the maximum stress of the pipeline after adding pipeline support and hanger is 212,1 MPa[3]. Saad et al. presented the experimental results of five full-size slender concrete-filled steel pipes with eccentric axial loads using high-strength concrete and high-strength steel as raw materials, and observed that the load-bearing capacity of the columns decreased significantly with the increase of eccentricity, which was consistent with the predictions based on the design criteria and the ABAQUS finite element simulation. The finite element validation study shows that the model established by ABAQUS can predict the structural behavior, strain and peak load reasonably and accurately [4]. Liu Yunfei used the ABAQUS finite element calculation software to carry out structural design and strength calculation, and it was necessary to comprehensively analyze and calculate the structural strength, modality, and instability characteristics of the vacuum pipeline, so as to reduce the weight on the basis of ensuring the structural safety to obtain the optimal scheme [5].

In this paper, through the application of Q235 steel as the flue in engineering, a model is established to simulate the finite elements, and the force and deformation before and after the bracket are analyzed in the actual hoisting operation.

FINITE ELEMENT MODEL ESTABLISHMENT

Q235 has good plasticity and welding performance, molding ability is very good, all many profiles (such as

C. D. Wu, M. W. He, C. J. Wang, X. D. Meng, L. J. Zhang, K. Wang CHINA MCC 22 GROUP CORPORATION LTD. D. L. Guo, H. C. Ji (E-mail: jihongchao@ncst.edu.cn) College of Mechanical Engineering, North China University of Science and Technology, Hebei, Tangshan, China.

Angle steel, round steel, I-steel, channel steel and other materials for many Q235), and has a certain strength, suitable for Bridges, buildings and other engineering structures, good practical performance, the price is relatively cheap, cost-effective [6]. Q235 steel is divided into four grades: A, B, C, and D, which are divided according to the different carbon content. The track used in this project is Q235A grade steel, and its main chemical composition is C 0,14~0,22 %, Mn 0,30~0,65 %, Si≤0,30 %, S≤0,050 %, P≤0,045 % [7], and the rest of the components are Fe. The inner diameter of the tubular flue is 6 000 mm, the outer diameter is 6 028 mm, and the diameter of the wire rope is 20 mm. The assembly drawing of the wire rope and the flue is shown in the figure. The inner diameter of the internal bracket steel pipe is 15 mm, the outer diameter is 25 mm, the length is 6 m, the end edge of the bracket is rounded, and it can fit perfectly with the flue when installed inside the flue, three brackets are added to one side of the flue, one is horizontal, and the remaining two are horizontally angled with the first one at an angle of 45°. The assembly drawing after adding the bracket is shown in Figure 1.

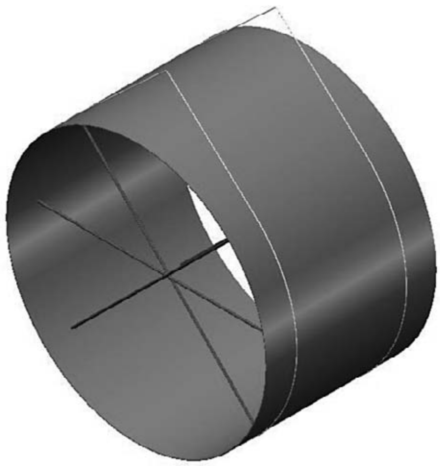

Figure 1 Three-dimensional view of assembly

NUMERICAL SIMULATION

Firstly, the three-dimensional structure of the electric hoist and the I-beam track is established and assembled with SolidWorks software, the conversion format is imported to the Abaqus software, the imported assembly is modeled with finite elements, the stress and strain of the track are observed, the material properties of Q235A are created, and the track material properties are given, the roller is set to a rigid body, and the inertial mass is set to it, and then the analysis step of a certain duration and frequency is created, the numerical simulation data to be output is selected, and the stress and strain situation need to be selected this time. Then create the interaction to set the friction coefficient and create constraints, select the top of the wire rope as the reference point, the position of the reference point and the hoisting hook contact is coupled, define the boundary conditions of the reference point, let it move in the forward direction, limit its movement and rotation in other directions, and then set the load, select the natural gravity, and then according to the running state to the reference point of the previously selected wire rope, give a certain muzzle velocity or acceleration, and the speed given should meet the actual requirements of the project, Too large is easy to produce potential safety hazards, too small will lead to too long engineering time and affect efficiency, so the speed should be appropriately selected.

Meshing is an important part of establishing a finite element model, which requires more problems to be considered, requires a large amount of work, and the divided mesh form will have a direct impact on the calculation accuracy and calculation scale. In order to ensure the accuracy of the numerical simulation, the size of the mesh should be selected according to the material properties of the 3D model and be as small as possible, and the number of meshes will affect the accuracy of the calculation results and the size of the calculation scale. Generally speaking, the number of grids increases, the calculation accuracy will improve, but at the same time, the calculation scale will also increase, so the two factors should be weighed when determining the number of grids. Because the flue, wire rope and steel pipe are very symmetrical and uniform structures as a whole, and their stress is relatively uniform, the hexahedral mesh with better mesh quality is used when dividing the mesh, and the mesh of the corresponding size is adopted according to the size of the parts, so that the time used for calculation can be shortened, and it will not have a greater impact on the results. The final grid is shown in Figure 2. Table 1 lists the number of meshes.

SIMULATION RESULTS AND ANALYSIS

In the process of hoisting the flue, according to the above modeling method, the finite elements are ana-

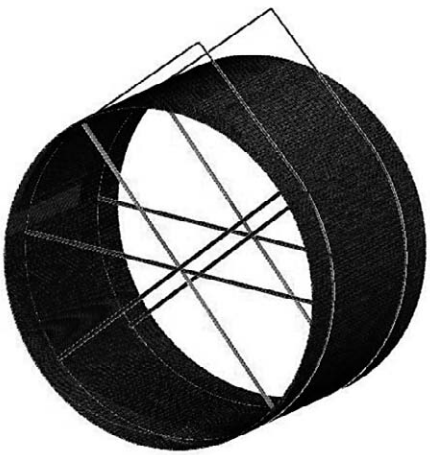

Figure 2 3D grid diagram

Table 1 **Number of grids**

Object	Volume/ mm/s	Number of grids
Flue	28	126 054
Wire rope	10	7 984
Bracket	10	9 600

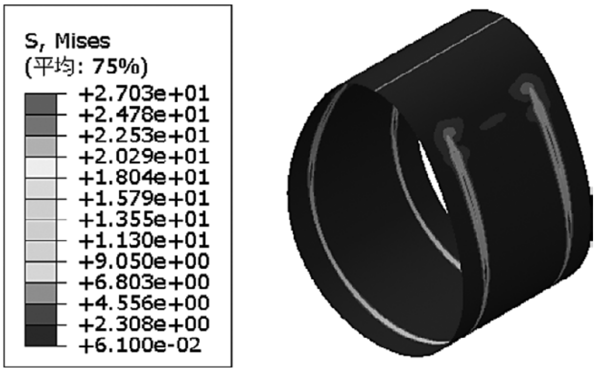

Figure 3 Stress in the unreinforced hoisting state

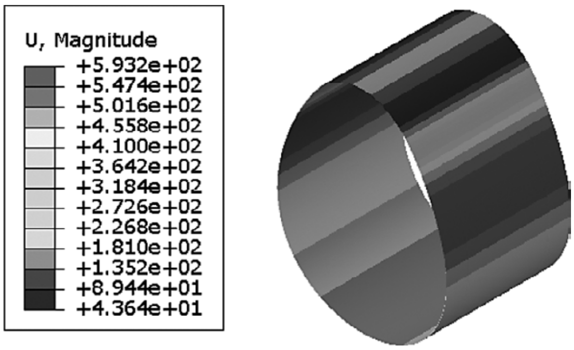

Figure 4 Strain displacement in the unreinforced hoisting state

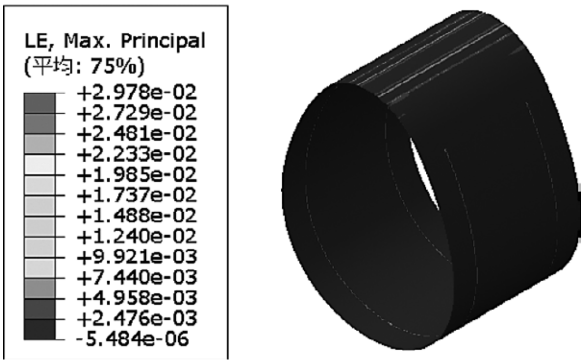

Figure 5 Plastic deformation in the unreinforced hoisting state

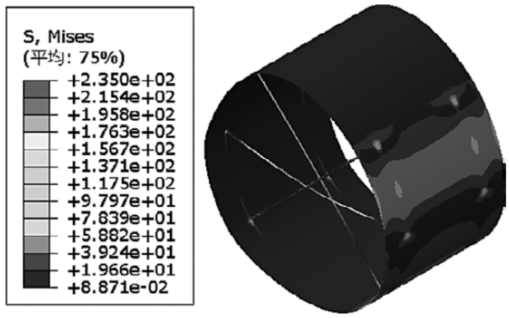

Figure 6 Stress in the hoisting state is reinforced

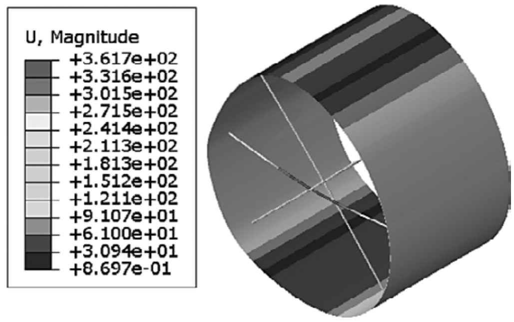

Figure 7 Reinforcement of strain displacement in hoisting state

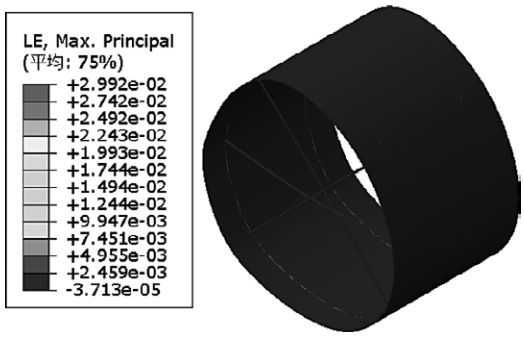

Figure 8 Plastic deformation in the state of reinforcement and hoisting

lyzed according to the selected process parameters, and the numerical simulation results of the flue and the bracket before and after reinforcement are obtained.

From the numerical simulation results of stress and strain, it can be seen that the flue before reinforcement has a very large deformation in the process of being hoisted, but the stress of the flue is not up to the yield stress of Q235 of 235 MPa. The maximum stress is 27,03 MPa, and the deformation that occurs is elastic deformation, but due to the limitation of installation space, the flue deformation is too large and will cause the problem of insufficient installation space, so the deformation of the flue is reduced as much as possible in the hoisting and installation process, as can be seen from the figure, the deformation displacement at the bottom of the flue has reached 593,2 mm, and it can be seen from the figure that the plastic deformation of the flue mainly occurs in the part in contact with the wire rope, and the maximum plastic deformation is $2{,}978\times10^{-2}$ mm, and the type variable is very small relative to the thickness of the flue, It is negligible, mainly because of the excessive elastic deformation of the flue. The diameter of the flue itself is very large, so it should be reinforced to reduce the deformation during the hoisting process.

As shown in the Figure 3-7, the stress-strain diagram of the flue and support after adding the bracket is added. As can be seen from the figure, the stress of the flue as a whole becomes larger after adding the bracket, and the maximum stress is 235 MPa on the bracket. The maximum stress of the flue itself is about 73,89 MPa, which has a more obvious improvement before the reinforcement, but it has not yet reached the yield strength of Q235, on the other hand, the bracket, the stress of the bracket has reached the maximum, and a large deformation has also occurred, the maximum displacement of the flue itself is 271,5 mm, and the displacement of the bracket has reached 361,7 mm, which is not much improved compared with the amount of phase change before reinforcement, and the plastic deformation of the flue is 2,992×10^{-2} mm, It's not much the same as before the reinforcement. On the whole, due to the installation space problem, the length change of the flue up and down is the main optimization and reduction part, so the displacement of the bottom of the flue before reinforcement is 593,2 mm, and the displacement of the bottom of the flue after reinforcement is 61mm, which is a very obvious improvement, so the reinforcement optimization is very important for the hoisting and transportation.

CONCLUSION

According to the numerical simulation results before and after adding the bracket, it can be seen that before adding the bracket, the maximum displacement at the bottom of the flue is 593,2 mm, and the maximum stress is 27,03 MPa, and after adding the bracket, the maximum displacement at the bottom of the flue is 61 mm, and the maximum stress is 73,89 MPa.

Acknowledgments

This work is supported by China Minmetals Group Co., Ltd. Science and Technology Special Plan Project (2021ZXA06) and funded by Science Research Project of Hebei Education Department (CXY2024054).

REFERENCES

[1] S. Jing, G. Y. Lu. Modal analysis of concrete filled steel tubular structure under lateral impact loading [J]. Structures, 56(2023),104932.

[2] K. X. Wang, M. C. Chen, R. Zhang, et al. Finite element simulation of load bearing capacity of circular CFST long columns with localized corrosion under eccentric load [J]. Structures, 43(2022),1629-1642.

[3] S. L. Cao, X. S. Shi. Stress analysis of steam pipeline based on Abaqus [J].Pipeline Technology and Equipment, 02(2017),7-9.

[4] Saad Younas, Ehab Hamed, D. X. Li, et al. Eccentrically loaded concrete-filled steel tubesmade with high strength materials [J]. Engineering Structures, (2023), 115246.

[5] Y. F. Liu. Strength analysis of vacuum pipeline based on ABAQUS [J]. China Equipment Engineering, 07(2021), 249-251.

[6] X. A. Zha. High Entropy Welding of 304/Q235 Stainless Steel Composite Plate [D]. East China Jiaotong University,2022.DOI:10.27147/d.cnki.ghdju.2022.000202.

[7] Y. P. Tian. Preparation and Properties of Wear Resistant Coating on Q235 Steel Surface [D]. Changzhou University,2022.DOI:10.27739/d.cnki.gjsgy2022.000380.

Note: The responsible translator for English language is W LIU-North China University of Science and Technology, China