RESEARCH ARTICLE OPEN ACCESS

Geometry optimization analysis of Casting Crane Spreader Beam based on Thermal-structural coupling

QU Kangjian，SHA Ling

(College of Mechanical Engineering, Shanghai University of Engineering Science, Shanghai, China)

Abstract:

For the problems of fatigue damage and shortened service life of the casting crane beam under heat radiation conditions，take225t casting crane spreader beam as an example, make thermal-structural coupling analysis of spreader beams based on Ansys finite element software, determined the area where the 225t spreader beam is most prone to fatigue failure under thermal radiation conditions, put forward local optimization of the area, and make stress analysis of the optimized beam, make a suggestion.

Keywords — **catalytic converter, canning, calibration, gap bulk density.**

1.INTRODUCTION

Casting cranes are mainly used for loading, unloading and handling molten steel in iron and steel smelting production, its working environment is bad, heat radiation is severe, work is frequent, loading rate is high, it's an essential heavy machinery equipment in the production process. In addition to the influence of mechanical load, the spreader beam is also affected by the thermal stress generated by the heat radiation of molten steel in the ladle, It is easy to produce fatigue damage in stress hot spots.

For the research of casting crane spreader, predecessors have done a lot of research: for the problems of high risk of heavy lifting, complicated process, difficulty in advance drill, and low efficiency of hoisting process planning, a 3D visualization simulation scheme for VR technology that can be used for hoisting process planning is put forward by Zhang Qinglei, Zhu Gongze etc[1].Zhao Qiuyuan, Wu Guanghe and others have developed a system that uses laser ranging technology to complete the rapid alignment of field bridge spreaders and container trailers[2].

Thermal-structure coupling analysis refers to the type of analysis of the influence of temperature field on physical quantities such as stress, strain, and displacement in a structure[3]. For thermal-structure coupling analysis, Sequential coupling analysis method is usually used in ANSYS[4], that is, thermal analysis is performed first to obtain the temperature field in the structure, then perform structural analysis, and adding the previously obtained temperature field as a body load to the structure, to solve the stress distribution of the structure, this method has been widely used in the company's production practices and product development.

On the basis of previous research, this paper uses 225t casting crane spreader beam as research object, using finite element analysis technology, through the application of thermal structure coupling analysis technology to study the stress distribution of spreader beam under heat radiation conditions, beams are structurally optimized to reduce stress concentrations and extend the life of the structure.

2 Thermal Analysis Fundamentals

Thermal analysis [5] (TA) refers to the method of analyzing the relationship between thermodynamic parameters or physical parameters as a function of temperature. Thermal analysis is used to calculate the temperature field distribution and other physical parameters of the spreader beam.

The collection of temperatures at various points in the material system is called the temperature field. The number of temperature field temperatures T is usually a function of the spatial coordinates (x, y, z) and the time variable t, which can be expressed as $T = (x, y, z, t)$.

If the system's net heat flow rate is 0, that is, the heat that flows into the system plus the heat generated by the system itself is equal to the heat that flows out of the system:

$$q_{in} + q_{generate} - q_{out} = 0 \tag{1}$$

The system is in thermal stability [6] , that is the temperature at any point in the steady state thermal analysis doesn’t change with time. The energy balance equation is (in matrix form):

$$[K]\{T\} = \{Q\} \tag{2}$$

In the formula, $[K]$ is the conduction matrix, including thermal conductivity, convection coefficient and emissivity and shape factor; $\{T\}$ is the node temperature vector; $\{Q\}$ is nodal heat flow rate vector, including heat generation.

ANSYS generates $[K]$, $\{T\}$ and $\{Q\}$ using model geometry parameters, material thermal performance parameters, and applied boundary conditions. From the boundary conditions and loads combined with thermal stress analysis [7, 8], it can be deduced that the overall stiffness matrix $[K]$ and load vector $\{P\}$ are:

$$[K]=\sum[K]^e \quad (3)$$

$$\{P\}=\sum\{P\}^e+\sum\{P\}_T^e \quad (4)$$

In the formula, $[K]^e$ is the unit stiffness matrix, $\{P\}^e$ is the unit node load, and $\{P\}_T^e$ is the unit node thermal load.

From displacement mode, from equilibrium conditions, variational principles and Hooke's law:

$$[K]\{\partial\}=\{P\} \qquad (5)$$

In the formula, $\{\partial\}$ is the node displacement vector.

Push to get the node displacement $\{\partial\}$, from the unit node displacement $\{\partial\}_T^e$ and temperature rise $\{\Delta T\}^e$, calculate the total strain $\{\varepsilon\}$ and thermal deformation $\{\varepsilon\}_T$, the final thermal stress expression is:

$$\{\sigma\}=[D](\{\varepsilon\}-\{\varepsilon\}_T) \qquad (6)$$

In the formula, $[D]$ is an elastic matrix.

Transient heat transfer process refers to a system heating or cooling process. During this process, the system temperature, heat flow rate, thermal boundary conditions, and system internal energy all change significantly over time.

3 Modeling and calculation

The 225t casting crane spreader beam is ax symmetric structure and symmetrical load, There are 4 pulley devices, 4 hanging fork holes, the hanging point distance is 7000mm and the hook spacing is 4400mm. A 3D solid model of the 225t casting crane spreader beam is established by using Solid works, See Figure 1 for details.

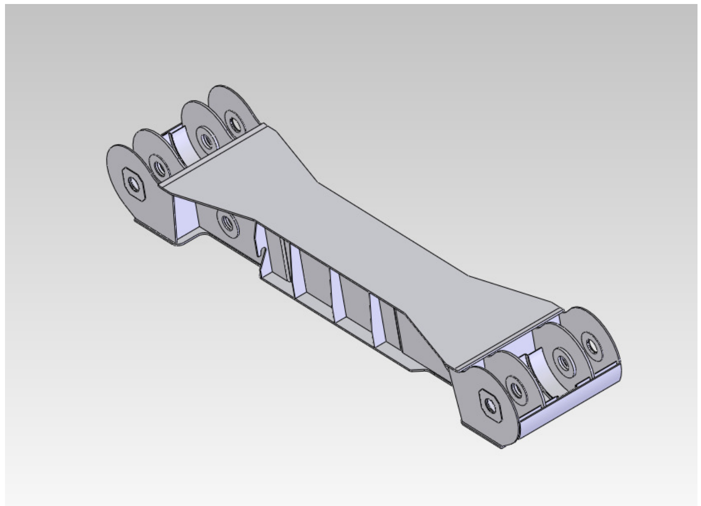

Figure 1 3D model of 225t casting crane spreader beam

Set up a seamless connection between Solid works and ANSYS Workbench. Open the finite element model of the completed beam in ANSYS Workbench. The grid division is automatically divided and locally refined to obtain the finite element model of the 225t beam. The crossbeam material was selected as Q345B steel, and the relevant parameters of the material are shown in Table 1.

Table □

Q345BMaterial properties

Density (t/m3)	Specific heat capacity (KJ/(kg*K))	Linear expansion coefficient ((C*10^6)^-1)	Thermal Conductivity (W/(m*K))	Elastic inertia (Gpa^-1)	Poisso n's ratio
7.85	0.46		53.2	2.06	0.3

3.1 Temperature field analysis

After completing the 3D modeling, the relevant physical parameters of the spreader beam are imported into the Workbench database，According to the principle of thermal structure coupling analysis, the Solid70 3-D three-dimensional thermal structure unit was selected. In order to more accurately simulate the actual condition of the beam transported by the beam, a molten steel temperature of 1600°C is applied in the area 3300mm vertically below the beam, and the temperature distribution of the beam is simulated. See Figure 2 and Figure 3 for details.

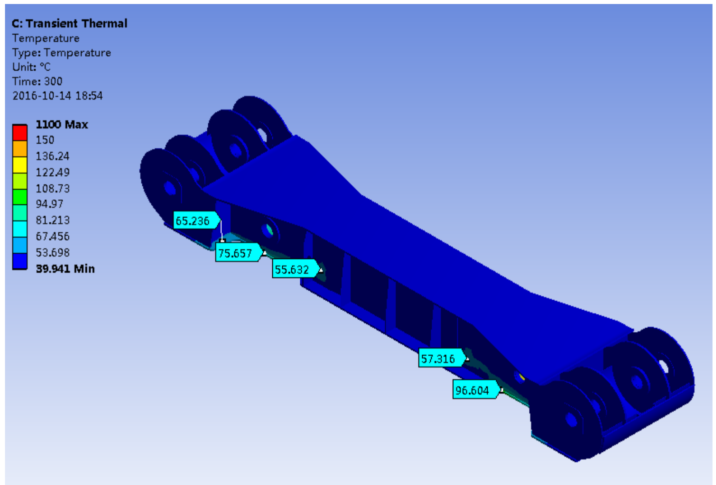

Figure 2Beam temperature field

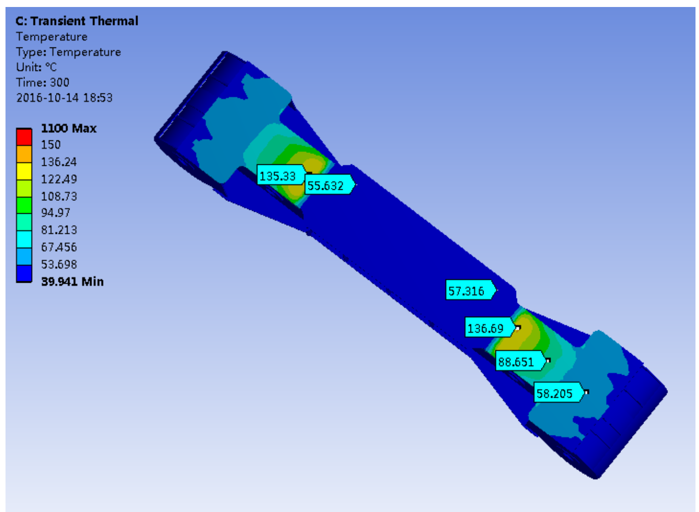

Figure 3The temperature field of the lower cover of the beam

It can be known from the calculation results that, due to the influence of the ladle radiation, the high-temperature area of the spreader beam is mainly distributed in the area of the lower cover plate of the beam, and the maximum temperature reaches about 136°C; At the same time, the temperature of the web area connected to it reaches about 75°C. Due to the lack of radiation protection in the middle of the web, the temperature in some areas also reaches about 56°C. The farther away from the ladle position is, the lower the influence of thermal radiation is.

3.1 Thermal-Structure Coupled Stress Analysis of Beams

The above-mentioned finite element software Ansys analyzed the temperature field of the 225t spreader beam. According to the thermal structure coupling method, we use indirect coupling analysis to convert the node temperature obtained from the temperature range of the spreader beam into a temperature load, that is the generated thermal stress. See Figure 4 for details.

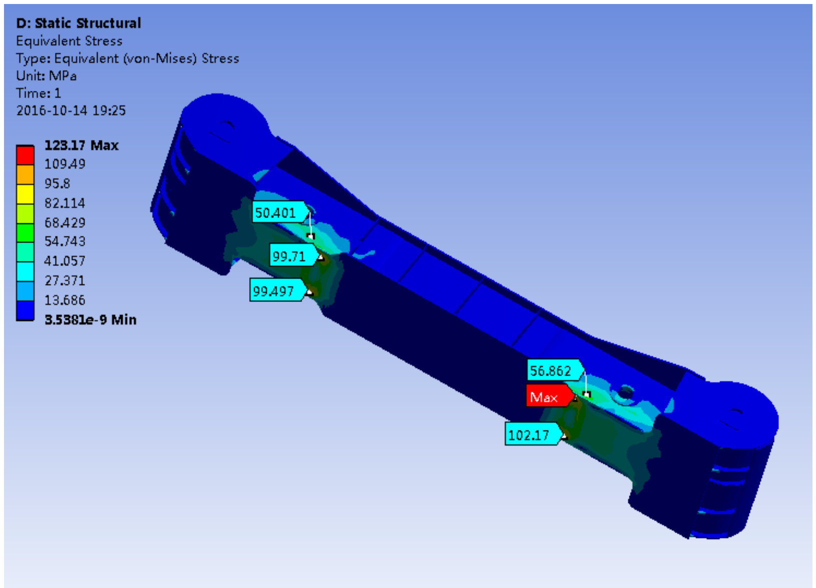

Figure 2Thermal stress of beams

From the calculation results, it can be known that the stress zone is mainly distributed in the area under the connecting shaft of the lower plate of the crossbeam. The maximum stress is 102Mpa, and from this area, the farther the stress is, the smaller the stress is; the stress concentration exists at the junction between the pulley shaft connecting plate and the cross beam upper cover plate, and the concentrated stress is 123.17 Mpa.

When the load is applied, since the weight of the spreader is much smaller than the lifting weight of 225t, it is ignored. In the four suspension fork holes, the vertical downward load bearing load 5625kg, in the eight pulley shaft hole each exerts two vertical load of 2812.5kg.Then the ‘body load’ obtained by the analysis is applied to the node of the structural analysis. After the simulation analysis, the thermal coupling stress distribution diagram of the 225t spreader beam is obtained. See Figure 5 and Figure 6 for details.

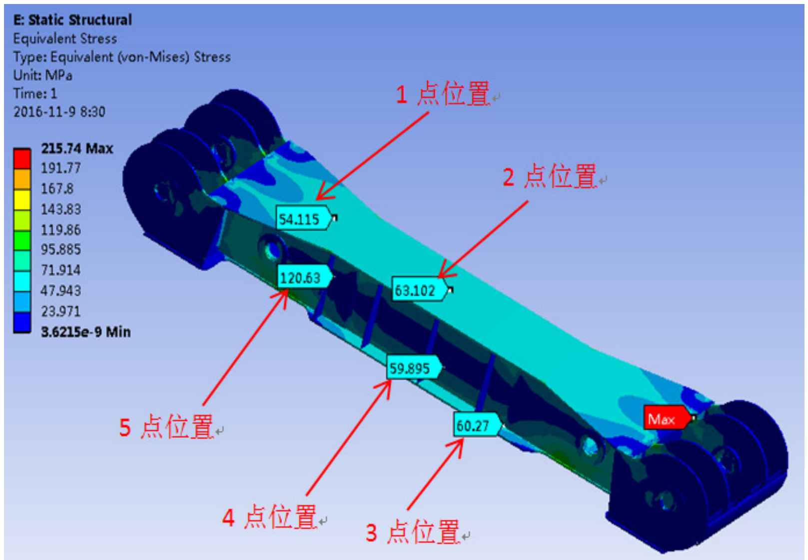

Figure 5Beam coupling stress

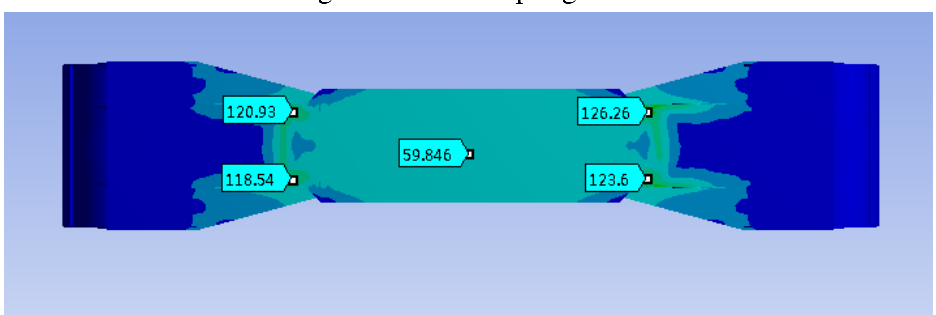

Figure 5Coupling stress of the lower deck of the beam

From the stress cloud diagram, it can be seen that the load of the beam is symmetrically distributed. Under the influence of thermal radiation, the stress distribution is mainly concentrated on the lower cover of the spreader beam(3, 5), especially in the area close to the suspension fork hole, the lower cover The maximum stress value of the board is 126Mpa, while the stress value of the middle part is about 60Mpa. The area of the upper cover plate of the crossbeam is farther from the ladle, and the influence of heat radiation is less. The stress in the upper cover (1, 2) is about 63Mpa. There is a stress concentration at the connection between the pulley connecting plate and the upper cover plate, the concentrated stress is 215 MPa, and the stress at the point where the web thin plate is connected (4) is 60 MPa. (In order to facilitate the calculation, the calculation result is rounded)

It can be seen that the areas affected by heat radiation are more severe and severe, the higher the coupled stress is, up to 126Mpa, and the stress at the connection between the pulley connecting plate and the upper cover plate is as high as 215Mpa, which seriously affects the load carrying capacity of the casting crane spreader beam，shorten the service life of the beam, so it is necessary to optimize the structure of the beam.

4Modeling and calculation

For the optimization of the 225t spreader beam structure, the influence of concentrated stress needs to be reduced firstly. The generation of concentrated stress is mainly related to the structure shape and component layout, and can be reduced by changing the layout of the components or changing the shape of the components. In the optimization design, there is a great stress due to the connection between the bent area of the upper cover plate and the web, up to 215Mpa. The upper cover plate of the 225t gantry spreader is modified to remove the bent part of the upper cover plate. See Figure 7 and Figure 8 for details.

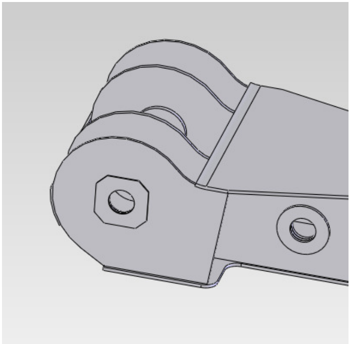

Figure 7Local 3D model before beam optimization

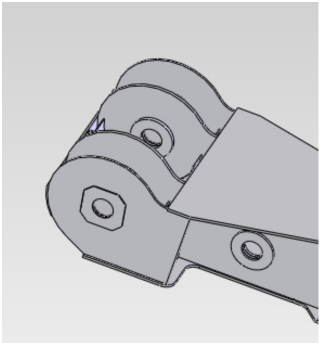

Figure 8Beam optimized local 3D model

Reload the model to get the new structure of the stress cloud，See Figure 9 for details.

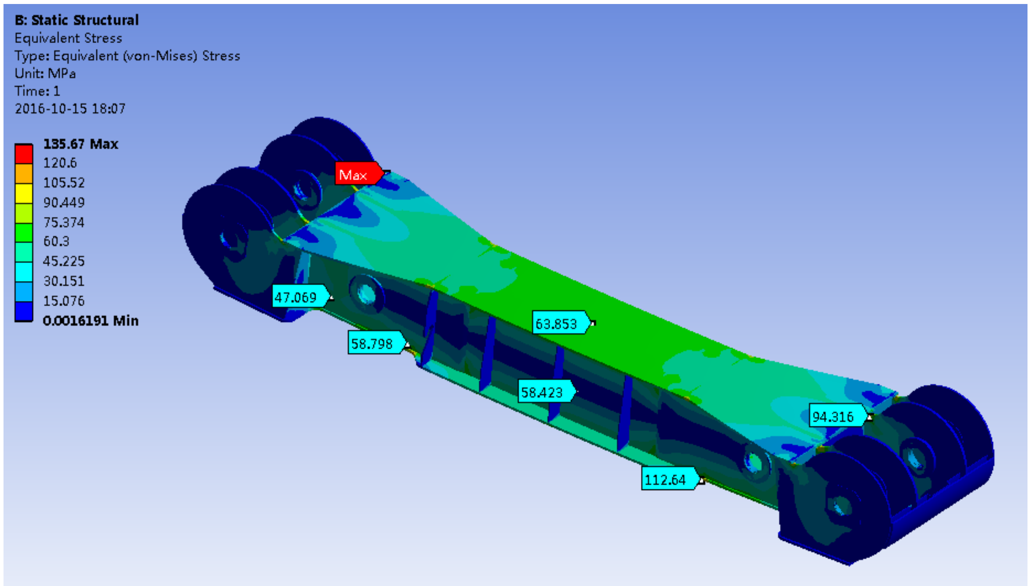

Figure 5Beam structure optimized stress cloud

From the stress cloud diagram, it can be seen that the stress concentration of the connection between the web and the upper cover after modification of the upper cover is significantly improved, from 215 MPa to 135 MPa, a decrease of 37%, and the lower cover plate area is also significantly reduced, and the maximum stress value is 113 MPa. However, the stress on the upper cover part of the beam remains basically unchanged. After the local structure of the beam was modified, under the conditions of heat radiation, the stress concentration was reduced and the service life of the beam was extended.

5Conclusion

This paper establishes a three-dimensional model of the spreader beam of a casting crane based on Solid works and Ansys finite element software，the results show：

（1）Under the influence of the ladle radiation, the maximum temperature of the spreader beam occurs in the area of the lower cover near the hanging fork hole. The farther away from the ladle position is, the lower the influence of thermal radiation is.

（2）The heat radiation has a significant influence on the stress of the lower cover plate of the spreader beam, especially in the area close to the hanging fork hole, stress concentrating occurs at the connection between the pulley connecting plate and the upper cover plate，it is of guiding significance to the areas that should be inspected during the process of design improvement.

（3）By modifying the upper cover plate of the 225t gantry spreader beam and removing the bent portion of the upper cover plate, the problem of stress concentration at the joint between the pulley connecting plate and the upper cover plate is effectively solved, and the force of the lower cover plate is also significantly reduced, and the cross beam is extended service life.

Reference：

[1] Zhang Qinglei,Zhu Gongze,Wu Dianliang,Hao Wenling.Dynamic Visual Modeling and Simulation of Hoisting Process [J].Agricultural Equipment &Vehicle Engineering,2017,(10):10-14.

[2] Zhao Qiuyuan, Ge Changqing, Wu Guanghe, Wu Weigang. Research on Rapid Positioning System of Container Gantry Crane Spreader and Container Trailer [J].Hoisting and Conveying Machinery,2017,(07):60-63.

[3] Xiao Jun Du,Xiao Qing Li,Ying Juan Yue,Fei Chen. Thermal-Structural Coupling Analysis of Planetary Roller Screw[J]. Applied Mechanics and Materials,2015,3864(757).

[4] Liping Pan,Zhu He,Baokuan Li,Kun Zhou,Ke Sun. Temperature Distribution and Thermal Deformation of the Crystallization Roller Based on the Direct Thermal-Structural Coupling Method[J]. JOM,2017,69(3).

[5] Qi Xu,Song Chen.Fast thermal analysis for fixed-outline 3D floorplanning[J].Integration,the VLSI Journal,2017.

[6] Levin P.A general solution of 3-D quasi-steady-state problem of a moving heat source on a semi-infinite solid[J]. Mechanics Research Communications,2008， 35(3):151-157.

[7] N. Laraqi,A. Bairi,L. Segui. Temperature and thermal resistance in frictional devices[J]. Applied Thermal Engineering,2004,24(17).

[8] Akbari M， Sinton D， Bahrami M． Geometrical effects on the temperature distribution in a half -space due to a moving heat source[J].Journal of Heat Transfer， 2011， 133(6) :064502.