Short Research Article

Finite element analysis of steel truss bridge structure based on ANSYS

Abstract: With the rapid development of bridge engineering in China, steel truss bridge is widely used with many advantages such as clear force of the structure itself, easy to analyze, and low requirement for foundation. In this paper, the finite element analysis method is adopted to analyze the stress and deformation distribution of steel truss bridge on the platform of ANSYS, a large-scale finite element analysis software. In the static analysis, by applying the vehicle load, the operation mode of the structure is simulated, and the cloud diagrams of its displacement and stress are derived. The dangerous areas of the bridge are identified to provide a reference basis for the design of steel truss bridges.

Keywords: Steel truss bridge;Finite element method;Static analysis;Change rule;ANSYS

1.Introduction

A truss is a structure made up of multiple rods connected in a regular and rational way at their ends. The truss structure derives many different compositions from the original triangle, and finally returns to the original design of the triangle once again[1]. The joist beam form is a more specific expression of steel joist structure. The internal force of the structure can be balanced in its horizontal direction, which also avoids the effect of the horizontal force produced by the normal structure in the support, which is also a unique advantage of the joist structure, material properties are fully utilized in the truss structure[2-3]. The structural form of steel truss bridge is the combination of deck plate and main truss, and its main stressing method is to take the main truss as the core and bear all the loads together with the deck plate[4]. At present, the connection methods of steel truss bridge nodes are mainly riveted connection, welded connection and bolted connection[5]. At present, according to the different shear state of high-strength bolts, high-strength bolt connection is divided into compression high-strength bolt connection and friction high-strength boltconnection[6]. After 1970, the construction of steel bridges has become common in the form of welded connections, which means that the nodes are likewise welded. Integral nodes have now replaced assembled nodes[7-8].

The traditional bridge design method is more dependent on experience, and the need to meet the current situation, there is a need to change the traditional design approach, the use of active thinking for structural optimization[9-10]. The earliest theory of optimal structural design appeared in the mid-19th century, and the research related to the application of this optimization theory to bridge structural design appeared later[11-17]. In the early stage of structural optimization basically empirical and intuitive methods are used, optimization methods such as optimization criterion method and mathematical planning method appear gradually in the later stage. In 1960, the first use of mathematical planning methods in the process of structural optimization was achieved, which was a historic point in time as it represented the modernization phase of structural optimization[18]. There are also many scholars who have conducted in-depth studies on topology optimization aspects, and changing the structural material distribution is the topology optimization's The change of structural material distribution is the main purpose of topology optimization. When the load is fixed, the structural arrangement of truss bridge is optimized by limiting a certain structural range, and the cross-section of the bridge is optimized by geometrical planning. design, and optimize the cross-section of the bridge by means of geometric planning[19].

Truss is usually used to increase the strength of the structure. In truss structure, chord is the rod that forms the periphery of the truss, including upper chord and lower chord; the rod that connects the upper and lower chord is called web, which is divided into diagonal rod and vertical rod according to the different directions; the plane where the chord and web are located is called the plane of the main truss. A truss bridge has many advantages, such as clear structural force, easy to analyze, and low requirement for foundation, with truss as the main load-bearing member. In this paper, ANSYS software is used to analyze the static force of steel truss bridge in the way of command flow, and study the change rule of stress and displacement.

2.Features of ANSYS

ANSYS software is a large-scale general-purpose finite element analysis software integrating structural, fluid, electromagnetic field, acoustic field and thermal field analysis. It can be widely used in civil engineering, geology, mining, materials, machinery, instrumentation, thermal engineering, water conservancy, biomedical and atomic energy and other engineering analysis and scientific applications. In recent years, ANSYS software has developed rapidly and its functions have been continuously enhanced. Compared with other analysis software, ANSYS has the following main advantages:

1. Data unification: ANSYS uses a unified database to store geometric models, finite element models, material parameters, external loads and result data, thus ensuring data unification of pre- and post-processing, analytical solving and multi-field coupling analysis.

2. Powerful solver: ANSYS provides a variety of solvers, users can choose the appropriate solver according to the specific analysis problem.

3. Powerful nonlinear analysis function: ANSYS has a powerful nonlinear analysis function, can be geometric nonlinear, material

nonlinear and contact nonlinear analysis.

4. Multiple meshing methods: ANSYS provides Free meshing, Map meshing, Sweep meshing and other meshing methods, you can choose the appropriate network division according to the characteristics of the model.

5. Unique optimization function: Using the optimization design module of ANSYS, the topology, appearance and material of the structure can be optimized to determine the optimal design scheme.

6. Multi-field coupling function: ANSYS can realize multi-field coupling analysis and study the mutual influence among physical fields.

7. Friendly program interface: ANSYS provides an interface program with mainstream CAD software and other finite element analysis software, which can realize the import and export of data, such as Pro/Engineer, NASTRAN, UniGraphics, I-DEAS and so on.

8. Good user development environment: ANSYS provides a convenient secondary development platform, users can utilize APDL, UIDL and UPFS for secondary development.

3. Finite Element Modeling of Steel Truss Bridge Structures

3.1Structural background

It is known that a simple steel truss bridge consists of steel sections, the top and side girders, the bridge chords and the bottom girders are made of 3 different types of I-beam sections respectively. The length of the bridge is L=32m, the length of each section is 4m, and it consists of 8 sections of trusses, and the height of the bridge is H=5.5m. The thickness of the bridge deck is 0.3m concrete slab, and the geometrical performance parameters of each member of the bridge structure are as shown in Table 1, and the properties of the materials are as shown in Table 2.

The bridge can pass the truck, if only consider the truck is located in the middle position of the bridge, assuming that the mass of the truck is 4000kg, if take half of the model, the force of the truck on the bridge can be simplified as P1, P2 and P3, where P1= P3=5000 N, P2=10000N, and the steel truss bridge is shown in Fig. 1.

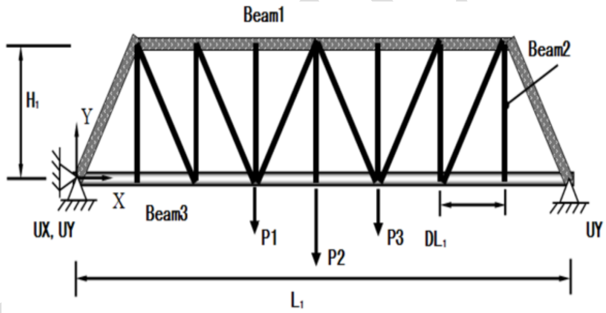

Fig.1 steel truss bridge

Table 1.Geometric performance parameters

member	section number	norm
Top and side beams (Beam1)	1	400*400*20*20
Bridge Stringers (Beam2)	2	400*400*16*16
floor beam (Beam3)	3	300*300*16*16

Table 2.Structural material properties

parameters	steels	concrete
modulus of elasticityEX	2.1E11	3.5E10
Poisson's ratioPRXY	0.30	0.18

DensitiesDENSE	7800	2500

3.2 Finite element modeling

ANSYS analysis process generally includes three steps: pre-processing, solving and post-processing.

ANSYS provides two operation modes: command flow and GUI, compared with the GUI operation mode, the command flow can reduce a lot of repetitive work, and it is convenient for modification and communication, so we choose the command flow mode for modeling and analysis here.

3.2.1 Pretreatment

The pre-processing mainly includes creating the finite element model, defining the cells, defining the material properties, and defining the cell divisions.

```
/prep7!Access to pre-processing
!Setting up units and materials
ET,1,BEAM188!Define the unit type
Keyopt,1,3,2
ET,2,SHELL181,
keyopt,1,3,2
MP,EX,1,2.1E11 !Define material modulus of elasticity
MP,PRXY,1,0.30 !Define material Poisson's ratio
MP,DENS,1,7800 !Defining material density
MP,EX,2,3.5E10 !Define material modulus of elasticity
MP,PRXY,2,0.18 !Define material Poisson's ratio
MP,DENS,2,2500!Defining material density
!Defining Cross-Section Characteristics
sectype, 1, beam, I,, 0
secoffset, cent
secdata, 0.4, 0.4, 0.4, 0.02, 0.02, 0.02
sectype, 2, beam, i,, 0
secoffset, cent
secdata, 0.4, 0.4, 0.4, 0.016, 0.016, 0.016,0,
sectype, 3, beam, i,,0
secoffset, cent
secdata, 0.3, 0.3, 0.3, 0.016, 0.016, 0.016, 0,
sect,4,shell
secdata,0.3,2,,1
!----- Define geometric keypoints
K,1,0,0,,
$ K,2,4,0,,$ K,3,8,0,,$K,4,12,0,,$K,5,16,0,,$K,6,20,0,,$K,7,24,0,,$K,8,28,0,,
$K,9,32,0,,$K,10,4,5.5,,$K,11,8,5.5,,$K,12,12,5.5,,$K,13,16,5.5,,$K,14,20,5.5,,$K,15,24,5.5,,$K,16,28,5.5,,
!-----Generating lines for bridge bottom girders through geometric points
L,1,2 $L,2,3 $L,3,4 $L,4,5 $L,5,6 $L,6,7 $L,7,8 $L,8,9
!------Generate lines for bridge roof and side girders
L,9,16 $L,15,16 $L,14,15 $L,13,14 $L,12,13 $L,11,12 $L,10,11 $L,1,10
!------Generating lines for bridge stringers
L,2,10 $L,3,10 $L,3,11 $L,4,11 $L,4,12 $L,4,13 $L,5,13 $L,6,13 $L,6,14 $L,6,15 $L,7,15 $L,7,16 $L,8,1
!------Select the top and side girders to specify the unit properties.
```

LSEL,S,,,9,16,1,

LATT,1,,1,,,, 2

!-----Select the bridge chord bar to specify the unit properties

LSEL,S,,,17,29,1,

LATT,1,,1,,,,3

!-----Select the underbridge girder to specify the unit properties

LSEL,S,,,1,8,1,

LATT,1,,1,,,,1

The generated single-row geometric model is shown in Fig.2.

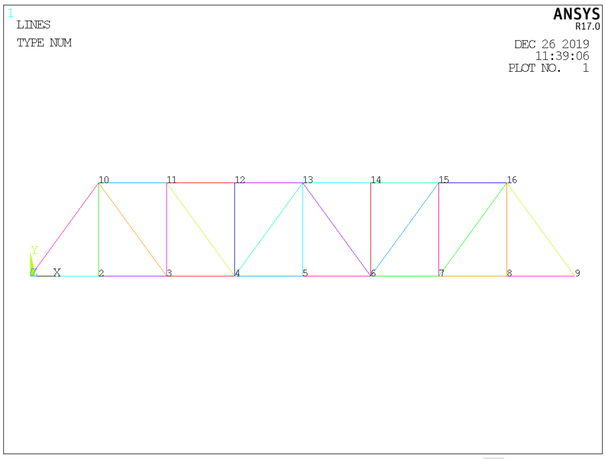

Fig. 2 Single-row geometric model

!------meshing

AllSEL,all !Resume selection of all objects

!Segmentation settings before cell division for all objects

LESIZE,all,,,1,,,,,1

!Unitizing all geometric lines

LMESH,all

The finite element analysis model is shown in Fig.3.

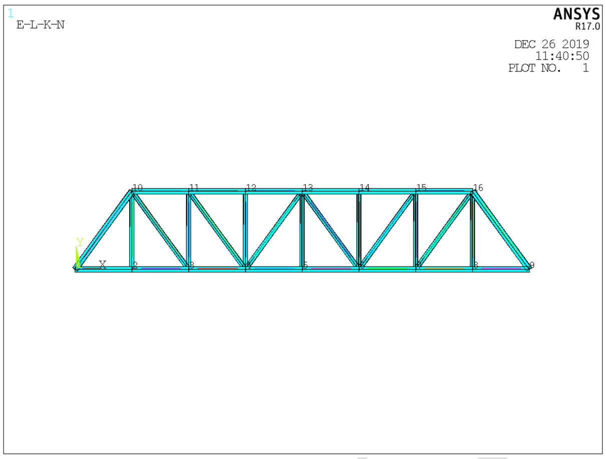

Fig. 3 Finite element analysis model

Lgen,2,all,,,,,5,, !copy command

Secnum,2 !Define the current section type as type 2

*Do,I,1,16 !Cyclic generation of transverse bars

E,I,I+16

*END DO

*DO,I,11,16 !Cyclic generation of top diagonal support

E,I,I+15

*END DO

E,11,26

Theoverall model of truss bridgeis shown inFig. 4.

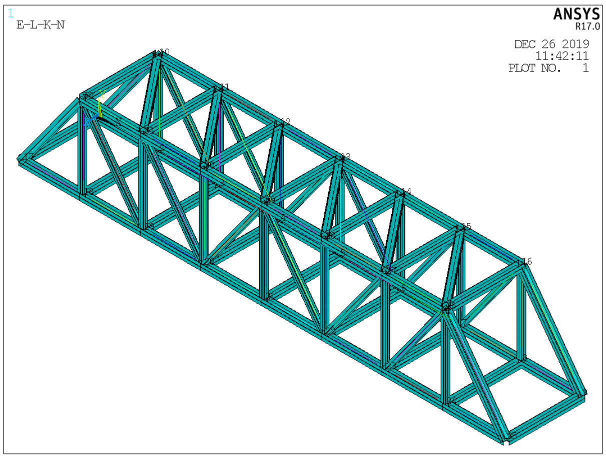

Fig. 4 Overall model of truss bridge

! Generation of bridge deck units

Type,2 !Specify unit types and materials for bridge decks

Mat,2

Secnum,4

E,1,17,18,2 !generator board

E,2,18,19,3

E,3,19,20,4

E,4,20,21,5

E,5,21,22,6

E,6,22,23,7

E,7,23,24,8

E,8,24,25,9

The integral finite element modeling of truss structure bridges is shown in Fig. 5.

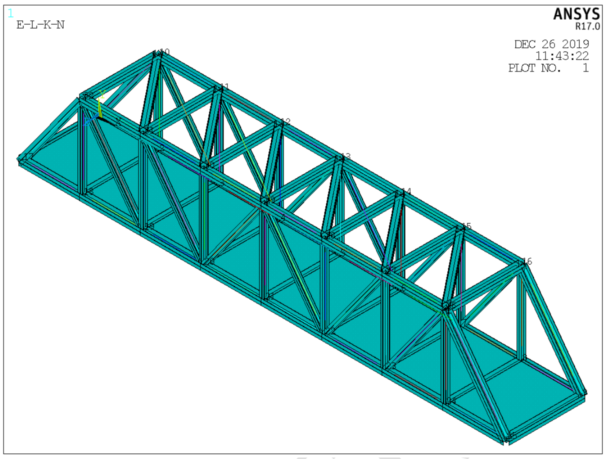

Fig. 5 Integral Finite Element Modeling of Truss Structure Bridges

3.2.2 Solving and Load Application

The solution process mainly consists of applying loads and boundary conditions, defining the solution type, defining the solver and the solution method.

!In the solution module, apply displacement constraints, external forces, and perform the solution.

/solu

NSEL,S,LOC,X,0 !Select the left node

D,All,,,,,,ALL,,,,,!Constrain all degrees of freedom

AllSEL,All !Restore the selection of all objects again

NSEL,S,LOC,X,32 !Select the right end node

D,All,,,,,,,UY,,,,!Binding UY

ALLSEL,al !estore the selection of all objects again

ACEL,0,10,0 !gravitational load

F,4,FY,-5000 !external load

F,6,FY,-5000

F,5,FY,-1000

F,20,FY,-5000

F,22,FY,-5000

F,21,FY,-1000

/replot !redraw

!Select all information (including all nodes, units, loads, etc.)

ALLSEL,ALL

Solve

The model after applying constraints and loads is shown in Fig.6.

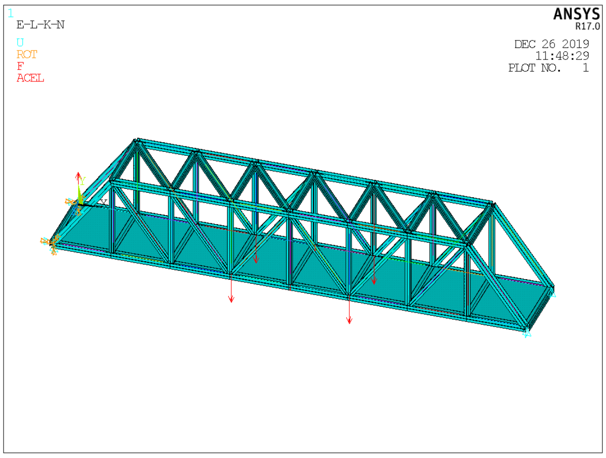

Fig. 6 Model after applying constraints and loads

4. Computational Architecture and Analysis

Post-processing is mainly used to view analysis results, result calculation and analysis, etc..

!=====Access to general post-processing modules

/post1 ! reprocess

Pldisp,1 ! Fig. 7 shows the structural deformation cloud map.

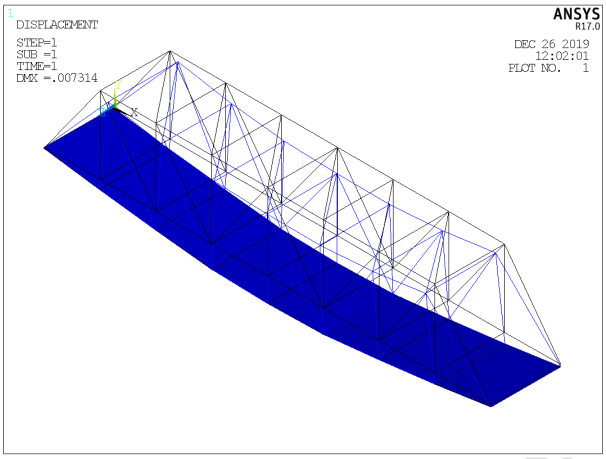

Fig. 7 Structural deformation cloud map

The maximum structural deformation is located in the middle of the span with a maximum value of 7.31mm, which meets the requirements for use.

Plnsol,u,sum,0,1.0 ! Displays a total displacement map

The total displacement deformation cloud map is shown in Fig.8.

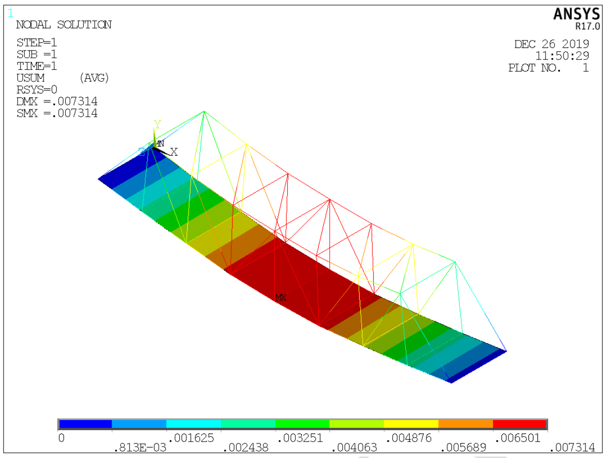

Fig. 8 Total displacement deformation cloud map

Plvect,u,,,,vect,elem,on,0 ! Show vector plot of total displacement of nodes.

The node displacement diagram is shown in Fig.9.

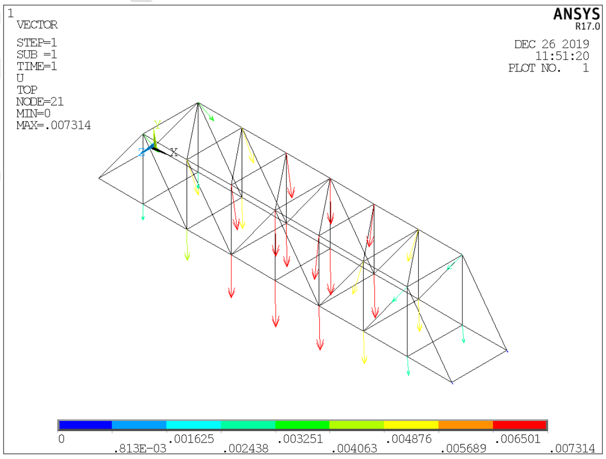

Fig. 9 Node displacement vector diagram

!------ Display of line unit axial force

Etable,zhou-i,smisc,1 !Defining the unit table

Etable,zhou-j,smisc,14

PLLS, zhou-i, zhou-j,1,0 !The axial force diagram as shown in Fig. 10.

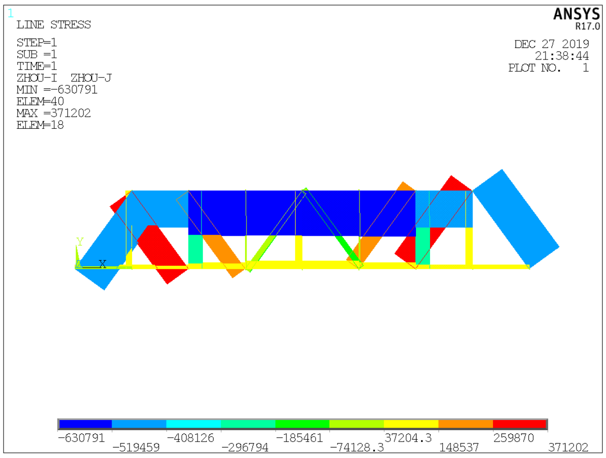

Fig. 10 Axial force diagram

Etable,jian-i,smisc,6 ! Define unit table, shear

Etable,jian-j,smisc,1 ! Define unit table, shear

plls,jian-i,jian-j,1,0 !The shear diagram as shown in Fig. 11.

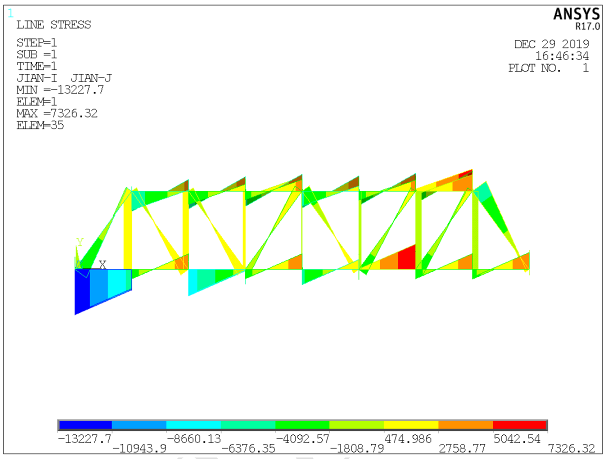

Fig. 11 Shear diagram

Etable,wan-i,smisc,3!Define unit table, bending moment

Etable,wan-j,smisc,16 !Define unit table, bending moment

plls,wan-I,wan-j,1,0!The bendingmoment diagram is shown in Fig. 12.

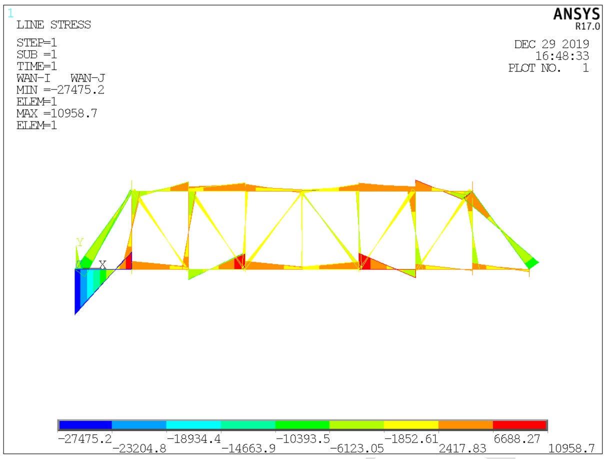

Fig. 12 Bending moment diagram

prnsol,u,comp ! List node displacements

finish

/exit,all

5. Conclude

In this paper, the static analysis of the steel truss bridge structure is carried out by using finite element analysis method on the platform of ANSYS, a large-scale general-purpose finite element analysis software. Boundary constraints, loads such as concentrated force and gravity are applied to the finite element model of the truss bridge, and the following conclusions are drawn:

(1) From the resultant deformation cloud diagram, it can be seen that the total maximum deformation of the bridge structure is 7.31 mm, which occurs in the middle region of the bridge.

(2) The maximum axial force, shear force and bending moment values of the structure are 371.20 kN, 10.96 kN and 7.33 kN·m. The maximum value of axial force occurs at the diagonal web bars at both ends of the structure, and the maximum values of shear force and bending moment both occur at the junction of

lower chord bar and the web bar in the middle section of the structure.

By analyzing each figure, it is obtained that the middle position of the bridge is its dangerous area. Therefore, the middle position should be given more consideration in the bridge design and construction.

Reference

[1] Gao Ge. Theoretical and Applied Research on Topology Optimization of Truss Structures [D]. Changchun. University of Chinese Academy of Sciences (Changchun Institute of Optical Precision Machinery and Physics, Chinese Academy of Sciences). 2017.

[2] Ma Zhenfu. Morphological creation of planar solid structures based on rod topology [D]. Harbin. Harbin Institute of Technology. 2018.

[3] Shen Tao. Research on Morphological Characterization of Building Based on Giant Truss Structure Technology [D]. Tianjin. Tianjin University. 2017.

[4] Zhao Tingting. Research on optimization design of a steel truss bridge structure [D]. Hebei. Hebei University of Engineering. 2019.

[5] Rui Yi. Comparative study of Chinese and European steel bridge specifications [D]. Chengdu. Southwest Jiaotong University. 2018.

[6] Li Yikuan. Introduction to the design of steel structure stability[J]. Building Materials and Decoration, 2018(29):124.

[7] Qiang Bin. Research on the spatial distribution of welding residual stresses and regulation technology of butt welding of thick plate of bridge steel [D]. Chengdu. Southwest Jiaotong University. 2018.

[8] Zhao Tingheng, Lin Yinyue. Integral node steel truss girder of Sunkou Yellow River Bridge on Beijing-Kowloon Line[J]. Journal of Railway Engineering. 1996(03):53-61.

[9] Yin Shiping, Lu Xiaoyang, Li Hongfu. Overview of the development of bridge structure optimization design [J]. Journal of Shandong Architectural Engineering Institute. 2004(04):65-68+76.

[10] Rozvany G I N. Aims, Scope, Methods, History and Unified Terminology of computer-Aided Topology Optimization in Structural Mechanics [J].Struct Multidisc Optim. 2001(35):1020-1048.

[11] Fu Qiang, Zhang Zepeng, Yan Xuezhai. A preliminary study on structural optimization design of cable-stayed bridges[J]. Journal of Hunan University (Natural Science Edition). 2001(S1):109-115.

[12]Svanberg K. Method of moving asymptotes-a new method for structural optimization. Int. J. Num.Meth.Eng. 1987,24,359-373.

[13] Moses F. Optimum structural design with failure probability constraints. AIAA. Journal. 1967,5(6):1152-1158.

[14] Wang Guangyuan, Dong Mingyao. Optimization of structural design. Beijing: Higher Education Press. 1987.

[15] Li Zhiwen. Fundamentals of optimal design of bridge structures. Beijing: People's Transportation Press. 1982.

[16] Qian Lingxi. Fundamentals of optimal design of engineering structures. Beijing. Water Conservancy and Electric Power Press. 1983.

[17] Yi Yunkun, Li Jianrun, Q Erren. Optimal

design of bridge structures. 2005,19(4):248-250.
[18] Dong Guixi, Wang Zangzhu. Current status and prospect of structural optimization design [J]. Electric Power Intelligence, 2000(01):5-7.
[19] Zhang Weisheng. Research on stress-related topology optimization problem based on level set method [D]. Dalian. Dalian University of Technology. 2013.