

Finite element analysis of bridge based on ANSYS

Abstract:With the rapidly development of bridge engineering, steel truss bridge is widely used because of its advantages such as clear structure stress, easy analysis and low foundation requirement. In this paper, the finite element method is used to analyze the stress distribution of steel truss bridge on the platform of ANSYS. In the static analysis, by loading the vehicle load, the cloud chart of displacement, stress and others are obtained, and the dangerous area of the bridge is found out, which provide the basis for the design of the steel truss bridge.

Keywords:Finite element analysis; Truss; Static analysis; ANSYS

1.Introduction

Truss is usually used to increase the strength of structure. Truss bridge generally consists of main frame, upper and lower horizontal longitudinal connection system, intermediate transverse bracing frame and deck system^[1]. In truss, chord is the outer member of truss, including upper chord and lower chord; the member connecting upper and lower chord is called web, which is divided into diagonal and vertical according to different directions; the plane where chord and web are located is called main truss plane. Truss bridges, with trusses as the main load-bearing members of the superstructure, have many advantages, such as clear structural forces, easy to analyse, and low requirements for foundations. In this paper, the static analysis of steel truss bridge is carried out by ANSYS software with the method of command flow^[2].

2.Features of ANSYS

ANSYS software is a large-scale general-purpose finite element analysis software integrating structural, fluid, electromagnetic, acoustic and thermal field analysis, which can be widely used in the analysis and scientific analysis of civil, geological, mining, materials, mechanical, instrumentation, thermal, hydraulic, biomedical and atomic energy engineering^[3]. In recent years, ANSYS software has developed rapidly and its functions have been enhanced. Compared with other analysis software, ANSYS mainly has the following advantages:

1.Data unity: ANSYS uses a unified database to store the geometric model, finite element model, material parameters, external loads and resultant data, thus ensuring the unity of data for pre- and post-processing, analysis and solution and multi-field coupling analysis.

2. Powerful solving function: 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^[4].

5. Unique optimisation function: Using the optimisation design module of ANSYS, the topology, appearance and material of the structure can be optimised to determine the optimal design scheme^[5].

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

7. Friendly program interface: ANSYS provides the interface program with mainstream CAD software and other finite element analysis software, which can realise 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 use APDL, UIDL and UPFS, etc. for its secondary development.

3. ANSYS modelling of steel truss bridge structures

3.1 Description of the problem

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 $L=32\text{m}$, the length of each section is 4m , and it consists of 8 sections of trusses, and the height of the bridge is $H=5.5\text{m}$. 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 P_1 , P_2 and P_3 , where $P_1=P_3=5000\text{ N}$, $P_2=10000\text{N}$, and the steel truss bridge is shown in Fig. 1.

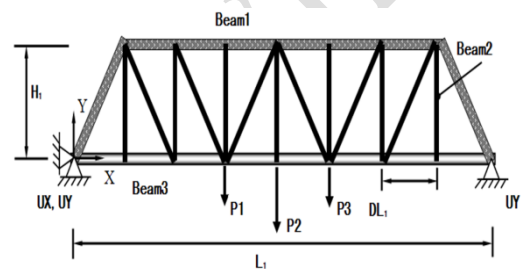


Fig.1 steel truss bridge

Table 1. Rod specifications for steel truss bridges

| 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. Material Properties of Steel Joist

| parameters | steels | concrete |
|-----------------------------|--------|----------|
| modulus of elasticity E_X | 2.1E11 | 3.5E10 |
| Poisson's | 0.30 | 0.18 |

| | | |
|----------------|------|------|
| ratioPRXY | | |
| DensitiesDENSE | 7800 | 2500 |

3.2 Model building

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,8,16
!----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.

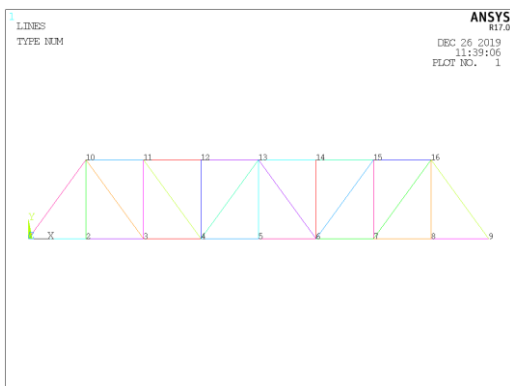


Fig. 2 Generated single-row geometric model

!-----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 model after meshing is shown in Fig.3.

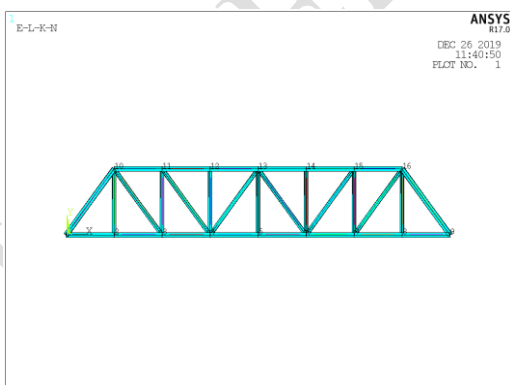


Fig. 3 Model after meshing

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

The overall truss model is shown in Fig. 4.

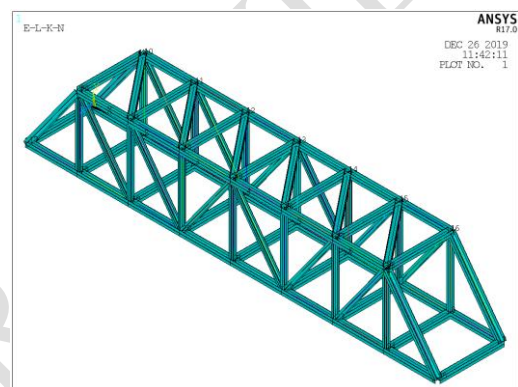


Fig. 4 Overall truss model

! 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 end of modeling is shown in Fig. 5.

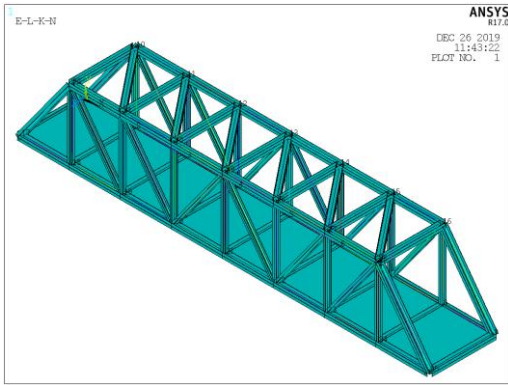


Fig. 5 Finite element model of bridge body

3.2.2 Loading and solving

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,all !Restore 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.

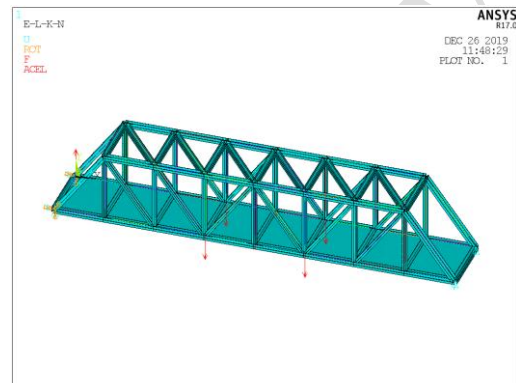


Fig. 6 Model after applying constraints and loads

4. Calculations from ANSYS static 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.

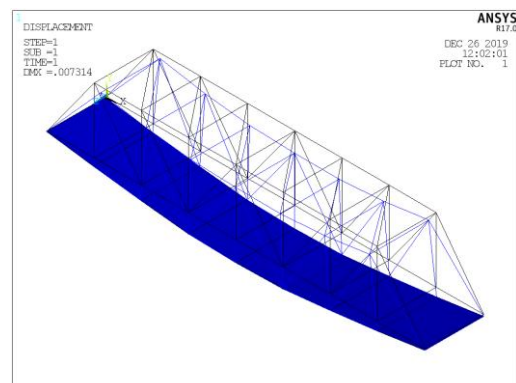


Fig. 7 Structural deformation diagram

The maximum structural deformation is located in the middle of the span with a maximum value of 0.007314m, which meets the requirements for use.

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

The total displacement cloud is shown in Fig.8.

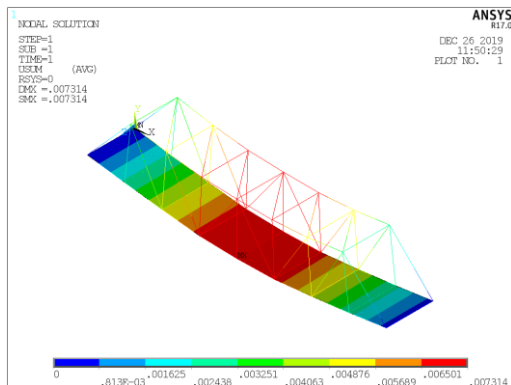


Fig. 8 Total displacement cloud

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

The node displacement diagram is shown in Fig.9.

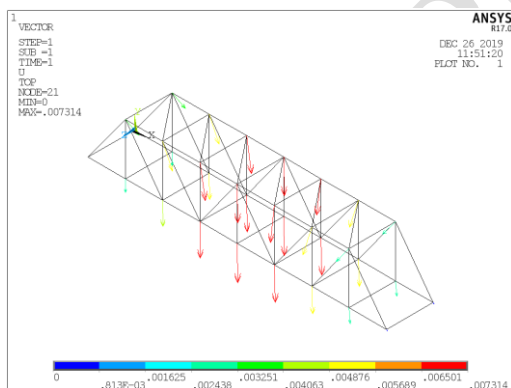


Fig. 9 Node displacement diagram

!-----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 !Draw the axial force diagram as shown in Fig. 10.

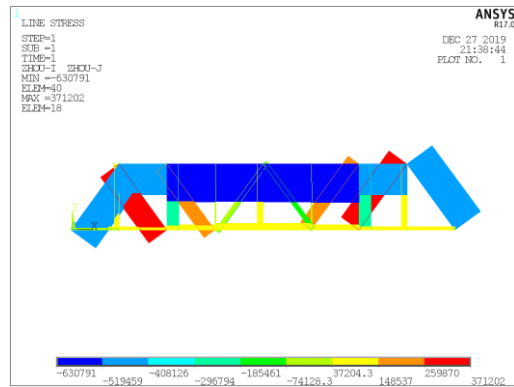


Fig. 10 Axial force diagram

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

Etable,jian-j,smisc,19 !Define unit table, shear

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

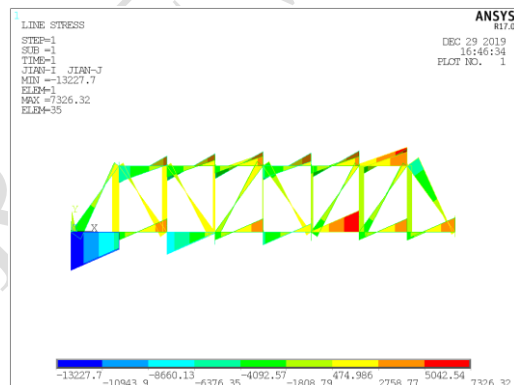


Fig. 11 Shear diagram

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.

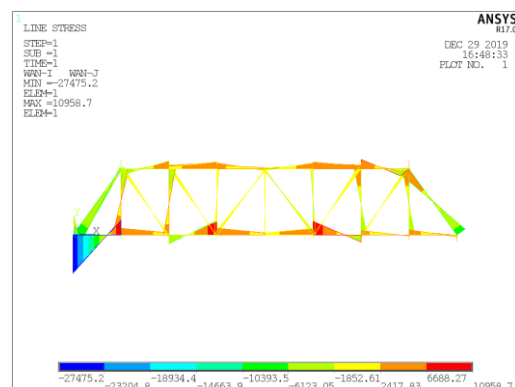


Fig. 12 Bending moment diagram

```
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. In the static analysis, displacement constraints are applied to the finite element model of the truss bridge and concentrated force and gravity are applied, and then the deformation diagram of the bridge structure, the total displacement cloud diagram, the node displacement vector display diagram and the axial force, shear force and bending moment diagrams are obtained by solving, through analyzing the diagrams, it is obtained that the middle position of the bridge is its dangerous area. Therefore, the middle position should be given more consideration in bridge design and construction.

Deficiencies: Due to the time relationship, the study of finite element method and ANSYS software is not very good, except for the ones used in the text, other functions of ANSYS, such as nonlinear structural analysis, thermodynamic analysis and so on, have not been studied. And the part involving modal analysis and optimization is not fully studied.

Reference

- [1] Wang X M. Numerical Analysis of Engineering Structures in ANSYS [M]. Beijing:People's Transportation Press,2012.
- [2] Zhang Y, Gu ZF, Song Y. Practical Tutorial

of ANSYS Finite Element Analysis [M]. Beijing:Tsinghua University Press,2004:151-174.

- [3] Zhang Z H, Li S Ki. ANSYS Finite Element Analysis Theory and Engineering Applications [M]. Beijing:Electronic Industry Press,2008:38-54.
- [4] Zhang Z H. ANSYS structural analysis of engineering applications [M]. Beijing:Machinery Industry Press,2010:509-537.
- [5] Lai Y B, Hu R X, Huang S Z. Typical examples of finite element analysis in civil engineering [M]. Beijing:Electronic Industry Press,2007:253-294.