Nonlinear Stress Analysis of Key Joints of Steel Truss Bridge

Zhenlei Liu* and Hao Sun
Department of civil engineering, University of Chang'an, School of Highway, Xi'an, Shaanxi, China.
*Email: 1372459151@qq.com

Abstract. Steel truss bridges are widely used in bridge engineering for the advantages of good ability of spanning capacity, construction and light self-weight. Main trusses are the main stressed component of steel truss bridge. And the main truss are made of truss members connected by integral joints. So, the safety of integral joints are very important for normal operation of steel truss bridge. The superstructure of the continuous steel truss bridge with double decks was selected as the engineering example. The software, MIDAS/CIVIL is used to establish the full finite element model of the continuous steel truss bridge. Based on the results of the full bridge model, the 3D finite element model integral joint considering material nonlinearity was established by software Abaqus. The stress distribution of the integral joint under the unfavorable external force were analyzed and compared. The results showed that the most parts of the integral joint are in elastic stage, and the stress distribution is inhomogeneous. The stresses of integral joint are greater than that of truss members. Except for individual stress concentration areas, the stresses in the center area of the integral joint are greater than the stress at the edge. All in all, the safety of integral joints for the engineering example can be guaranteed.

1. Introduction
Steel truss bridges are widely used in bridge engineering for the advantages of good ability of spanning capacity, construction and light self-weight. [1]. In recent years, the requirements for river crossing and lake crossing in cities have increased rapidly because of the rapid development of urban economy and transportation. Coupled with tight urban construction land and limited choice for bridge site, it is quite a good choice to construct a double-deck steel truss girder bridge for it can meet the demand of rapid passage of motor vehicles, as well as the slow travel of the both sides of the river. Besides, the special sculpt of double-deck steel truss girder bridge can show people a beautiful scenery [2][3].

Two examples are shown in figures 1,2. The superstructure of steel truss girder bridge is comprised of deck system, main truss and connection system [4]. The live load is directly supported by the main deck. while the connection can support horizontal load as well as provide lateral support. Main trusses
are the main stressed component of steel truss bridge. According to the connection types, the nodes are classified to externally attached nodes, integral nodes and interpolated nodes. The rods of externally attached joints are all welded, and gusset plates are placed on both sides of the rods, and then the rods are connected with rivets or high-strength bolts. For interpolated nodes, the web of chord near the codes is removed and then the plate is inserted into the position of web of chord. The connection between node plate and chord member is achieved by connection cover plate. The gusset plate of the integral joint and the adjacent chord vertical plate are welded into a whole in advance in the factory. The adjacent chords are spliced outside the node range with high-strength bolts on the construction site, and the diagonal rods and vertical rods of the frame are spliced within the node range [5] [6].

The node state can directly affect the serviceability performance of steel truss girder bridge. Therefore, it is of great significance for bridge working to have analysis research on nodes. Zhang Qiang et al. utilized finite element software ABAQUS to have analysis on nodes to get the conclusion that the increase in the thickness and size of the middle node caused the stress concentration to appear in the off-center variable section node [7]. Ma Hongbo et al. made use of finite element software MIDAS/CIVIL to analyze anchor nodes. And with the comparison and analysis of different stiffening girders, rational node styles can be worked out [8]. Li Ying et al. used finite element software ABAQUS to have elastic analysis and elastic-plastic ultimate static analysis on top nodes and then worked out that improving vertical stiffening plate on nodes have no effect on node ultimate carrying capacity [9]. Huang Pingming et al. analyzed the stress distribution of nodes with finite element software. The analysis shows that the stress distribution can be uniform and the stress of concrete can be within the limit state as long as the structure is rational [10].

The full bridge model of a double-deck highway steel truss girder bridge is established by MIDAS/CIVIL. Combining with the internal force distribution of each member of nodes, a node entity model is established by ABAQUS software. Applying Actual Internal Force on the Joint. And then the stress distribution is obtained by Analyzing the node stress cloud diagram.

2. Engineering Situations
The total length of the steel structure bridge is 605m, The main bridge is double deck system of variable height continuous steel truss girder. The upper and lower decks supported by oblique truss connection. The main bridge is symmetrically arranged with a total of 6 spans. The main span is 120m, the secondary side span is 70m, and the side span is 57.5m. The lower chords of main span and secondary span change parabolically. The arrangement of bridge is shown in figure 3.

![Figure 3. Arrangement of bridge.](image)

3. Finite Element Simulation Model of Whole Bridge
The MIDAS/CIVIL model can correctly reflect the internal force distribution of the member, but it cannot reflect the detailed stress information of the node. Therefore, MIDAS/CIVIL model for the overall truss is established to extract the internal force of the member. In order to obtain the stress change at each position of the node more clearly, the most dangerous combination of load is selected. The selected load combination is 1.2 times dead load + 1.4 times live load. The overall bridge model is shown in figures 4 and 5.
The Fx, Fy, Fz, Mx, My and Mz of each member of the node are calculated by MIDAS/CIVIL. The specific data are shown in table 1. The bar number starts from the right ventral bar and is numbered clockwise.

Table 1. Internal force table of each member.

|       | axial force (kN) | shearing force-y (kN) | shearing force-z (kN) | moment of torsion (kN*m) | bending moment-y (kN*m) | bending moment-z (kN*m) |
|-------|------------------|-----------------------|-----------------------|--------------------------|-------------------------|-------------------------|
| 1# bar| -7514.77         | -28.86                | 130.92                | 63.32                    | -642.55                 | 248.63                  |
| 2# bar| -23787.22        | 293.94                | -196.96               | 157.42                   | -2969.11                | 1279.94                 |
| 3# bar| -23809.56        | -293.86               | 194.45                | -163.15                  | -2984.38                | 1278.93                 |
| 4# bar| -7445.07         | 20.06                 | 129.51                | -68.85                   | -636.95                 | -222.65                 |

4. Finite Element Simulation Model of Node

In order to ensure the accuracy of the calculation results, the local model is intercepted from the whole bridge model according to certain principles. According to the Saint-Venant principle and the principle of static equivalence, the application of boundary conditions at the truncation has only a significant effect on the region near the truncation, but have no significant effect on the stress performance of the region far from the truncation. Therefore, the position of applying boundary conditions should be as far away from the concerned area as possible, that is, the selected range should be as large as possible. However, due to computer hardware constraints, taking into account the number of units and computational efficiency, the intercepted model range cannot be too large. Therefore, the rod length is 4m. The Node selection q345 steel, The material characteristic value: density is 7850 kg/m³, Young's modulus is 2×10^5 MPa, poisson ratio is 0.3.

4.1. Modelling

4.1.1. External Force Boundary. The key of the local model is the simulation of boundary conditions. According to the principle of static equivalence, the section internal force in the overall analysis is given to the corresponding position of the local model. The internal force of the truncated member is taken from the force results extracted from the MIDAS/CIVIL integral bridge model.

4.1.2. Displacement Boundary. The local model extracted from the overall model is in a static equilibrium state, which requires constraints to eliminate rigid body displacement. The displacement boundary condition of the local model is consistent with that of the whole model by applying surface constraint on the plane of the bearing position of the local model, so that the rigid body displacement in the local model can be removed. The boundary and local coordinate systems are shown in figure 6.
4.1.3. Mesh Subdivision. The node model components are simplified to facilitate the loading and calculation. Each component of the node is a solid element. In order to ensure the calculation accuracy, the eight-node linear hexahedral element (C3D8R) is used to mesh the steel nodes and each member. The spacing of the grid nodes is controlled to 0.01~0.1m, and refinement the grid in the structure connected parts. The grid division of nodes is shown in the figure 7.

5. Data Analysis
The calculation of steel structure is based on the fourth strength theory (shape change energy density theory), Using Mises equivalent stress as equivalent expression of one-dimensional yield stress under multiaxial stress state, The following is the expression:

$$\sigma_s = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$ (1)

In the formula: $\sigma_1$, $\sigma_2$, $\sigma_3$ are the three principal stresses at the dangerous point of the component.

Based on the fourth strength theory, $\sigma_s$ can be regarded as the standard of whether the steel reaches the yield strength. The Mise stress diagram of partial model is shown in Figure 8:

It can be found from the equivalent stress diagram of the whole node, the most parts of the integral joint are in elastic stage, and the stress distribution is inhomogeneous. Stress centralization exists in the Intersection position of abdomen rod, joint plate and chord rod. Other than that, Stress centralization also exists in the joint position of support and the inferior part of chord rod. It is seen that the maximum stress is 220Mpa, Which is less than the design value of steel strength.

According to the results of the local solid model and the overall bridge model, the stress distribution of each member section at the same position is obtained in the table 2. The following conclusions are drawn from the results, The gap between the two is controlled within 10 %, so the data of the two models are close, Which confirms that the selection of boundary conditions and simplification methods for the solid model are correct. In addition, The results also confirm the feasibility of using the ABAQUS to simulate the solid model.
In order to better reflect the stress distribution of joint, Three path lines are established in the joint area as shown in the figure 9. Mise stress on the path line is extracted, and the stress curve is drawn to observe the stress distribution on the path line.

Path 1 passes through the two upper chords and the joint, and the stress distribution is shown in the figure 10. We can see that the stress increases gradually from left to right, but the increasing rate in the range of chords is slow, and the magnitude of stress is between 100 and 120 MPa in this area. The stress growth rate accelerates when the path enters the joint area, and the stress reaches the maximum value of 170 MPa in the middle area of the joint. Because the joint is a symmetrical structure, the change trend of the right side of the curve is symmetrical with the left side.

Path 2 and 3 pass through the abdomen rod and joint, because the variation of vertical stress of path 2 and 3 is the same, only the variation of vertical stress of path 2 is given, and the stress distribution rule is shown in the figure 11. We can see that the stress increases gradually from left to right, but the increasing rate in the range of chords is slow, and the magnitude of stress is between 50 and 70 MPa in this area. The stress growth rate accelerates when the path enters the joint area, and the stress reaches 185 MPa, However, there is stress concentration at the junction of joint and chord, so the stress curve continues to increase after the stress decreases in this area.

6. Conclusion

The superstructure of the continuous steel truss bridge with double decks was selected as the engineering example. The software, Midas/Civil is used to establish the full finite element model of the continuous steel truss bridge. Based on the results of the full bridge model, the 3D finite element model integral joint considering material nonlinearity was established by software Abaqus. The stress distribution of the integral joint under the unfavorable external force were analyzed and compared.

The most parts of the integral joint are in elastic stage, and the stress distribution is inhomogeneous. Stress centralization exists in the Intersection position of abdomen rod ,joint plate and chord rod. Other than that, Stress centralization also exists in the joint position of support and the inferior part of chord rod,and the maximum stress is 220Mpa.

The stress distribution of each member section at the same position of the two model is approach, Which confirms the selection of boundary conditions and simplification methods for the solid model are correct.

Through path stress analysis, it is concluded that the variation speed of section stress in the rod is less than that in the joint range, and the maximum value is 170-180 MPa in the middle area of the joint, The stresses of integral joint are greater than that of truss members. Except for individual stress concentration areas, the stresses in the center area of the integral joint are greater than the stress at the

|   | whole bridge model | local solid model | Gap between the two |
|---|--------------------|-------------------|---------------------|
| 1# bar | 54.7MPa            | 53.2MPa           | 6.1%                |
| 2# bar  | 98.1MPa            | 92.18MPa          | 9.2%                |
| 3# bar  | 100.1MPa           | 99.23MPa          | 1%                  |
| 4# bar  | 54.7MPa            | 54.5MPa           | 3.7%                |

Figure 10. The variation stresses charts of path1.  Figure 11. The variation stresses charts of path2.
References
[1] Shang X C. Mechanical properties analysis of steel truss bridge joints [D]. Southeast University, 2020.
[2] Chen X. Static performance analysis of long-span stiffened steel truss bridge joints [D]. Chongqing Jiaotong University, 2018.
[3] Lv Y B. Research on the mechanical performance of highway steel beam of steel truss bridge [D]. Southwest Jiaotong University, 2018.
[4] Zhou X H, Liu Y J, Steel Bridges[M]. Beijing: People's publishing house transportation, 2019.
[5] Xing Y. Yang C 2017 Research on the Manufacturing Process of the Lower Chord with Integral Joints of Hutong Yangtze RiverBridge Steel Construction 32(4): 74-77.
[6] Li Z F, Niu Z G, Fang J, Ding S H, Li W J 2021 Mechanical Analysis of Key Joints in Jacking Construction of Large Continuous Steel Truss Bridge Journal of Railways 43 (04): 158-165.
[7] Zhang Q, Gao Q J and L P 2020 Finite Element Analysis of Steel Joint of Steel Truss Bridge Highway 65(07): 135-140.
[8] Ma H B and Ren Z Y 2012 Finite Element Analysis of Anchor Joints of Long Span Prestressed Steel Trestle. Engineering Journal of Wuhan University 45(S1): 95-97.
[9] Li Y, Huang K, Li J H, Xie Y H and Zhang J B 2013 Element Static Analysis of Steel Structure Joint Based on ABAQUS Building Structure 43(S2): 487-492.
[10] Huang P M, Zhang Y J and Shao H Y 2007 Local stress analysis of inclined truss bridge joints Journal of Wuhan University of Technology (Transportation Science & Engineering) (02): 228-231.