Study on Mechanical Performance of Connection joints In Novel Modular Steel Structure Buildings

Qi Hu¹, Bing Lin¹, Yonglan Xie¹* and Wenqiang Chen¹

¹China Construction Industrial Engineering and Technology Research Academy, Beijing, 100300, PRC
*Corresponding author’s e-mail: xieyonglan@cscec.com.cn

Abstract. The overall mechanical performance of modular steel structure buildings are largely determined by the mechanical performance of the connection joints between the adjacent modules. In this paper, a novel connection joints between the adjacent modules is taken as research object, which has the advantages of convenient construction and assembly. By means of FEA, three models including the joint model, the model of adding ribs and the model of increasing beam height are calculated, and the experimental results were compared. The finite element model uses solid elements to simulate the detail structure of joints and contact elements to the contact relationship between components. The research results show that the results of finite element analysis are in good agreement with the experimental results in terms of deformation state and bearing capacity, and the accuracy of the finite element model is verified. The finite element model established in this paper can simulate the stress state of various parts of the joint which can make up for the shortage of the experimental measurement.

1. Introduction

Modular steel structure building is an advanced stage of the development of building industrialization, and it is also a kind of highly integrated prefabricated buildings. The modular steel structure building takes functional modules as the basic unit, and only needs to connect adjacent modules into a whole on site. It has the characteristics of modular design, factory production, assembly construction, etc [1]. It can be seen that the whole mechanical properties of modular buildings largely depend on the mechanical properties of the connection joints between modules, so it is very important for the study on the connection joints between modules. At present, domestic and foreign research scholars have conducted research on different connection joints and achieved certain results [2-5]. However, the current connection joints have problems such as complex structure, unclear force transmission paths, and inability to accurately simulate calculations. This is not convenient for the further promotion and application of modular steel structure buildings [6-7].

In response to the above problems, this article takes connection joints between novel modules with simple structure and convenient assembly as the research object. The finite element analysis is used to simulate the joint model and the model of add ribs and increase beam height on the basis of the joint model, and the experimental results were compared.

2. Model design

2.1. The joint structure

The inter-module connection joint studied in this article is composed of three parts: the upper
component, the lower component and the positioning plate. The three parts are connected vertically by bolts to form a whole. The joint form is simple and easy to install. The structure is shown in Figure 1.

![Joint composition graph](image1)
![Joint assembly model](image2)

Figure 1 Structure form

2.2. Model design

According to the force state of the module structure under the action of seismic force, the beam-column members mainly bear bending moment and shear force, and the connecting bolts mainly bear tensile force and shear force. The calculation model design is shown in Figure 2, the upper beam column selects the position of the inverted bending point and the beam ends are supported by sliding bearings. This can accurately simulate the upper beam-column boundary conditions, and its force state is consistent with the actual structure. The bolt connection force between the upper and lower structures is consistent with the actual, which can verify the connection performance between joints. The length of the lower column has not been taken to the position of the reverse bending point, so its force state is different from the actual member. Force analysis is shown in Figure 3. Since the purpose of this article is to study the mechanical properties of the connecting joints between modules, the model selection is reasonable.

![Figure 2 Model design schematic](image3)
![Figure 3 Force analysis diagram](image4)

In view of the problems of connecting joints, including complex structure, unclear force transmission path, and inability to accurately simulate and calculate. Take the connection joint form described in this article as the research object. The finite element analysis is used to simulate the joint model and the model of add ribs and increase beam height on the basis of the joint model. Model information is shown in Table 1.
Table 1 Model information

| Number | Specimen Number | Column specifications | Beam specifications | Whether to add ribs | Remark |
|--------|-----------------|-----------------------|--------------------|---------------------|--------|
| 1      | JD-a-0          | □200×6                | H a×150×4.5×6      | No                  | a:200(bottom)+150(top) |
| 2      | JD-b-0          | □200×6                | H b×150×4.5×6      | No                  | b:250(bottom)+2000(top) |
| 3      | JD-a-1          | □200×6                | H a×150×4.5×6      | Yes                 | a:200(bottom)+150(top) |

3. Finite element model

3.1. Material constitutive

The material constitutive model reflects the characteristics of the material and directly relates to whether the finite element simulation can reflect the actual situation under load. The test piece material is steel Q235. The material constitutive is an elastic strengthening model, and the stress-strain relationship after yielding is an oblique straight line with a small slope.

\[ E' = \alpha E \]  

\( \alpha \) is taken as 0.01.

![Figure 4 Material constitutive model](image)

3.2. Interaction

In this article, the entity element in the ABAQUS general finite element analysis software is used to perform finite element analysis on the joint. The model size is consistent with the test piece size. Use "merge" to simulate welding connections between components. Set up "contact unit" to simulate the contact behavior of related components inside the test piece.

This article uses the "surface-to-surface" contact type. The set contact includes: nut and corner plate, bolt rod and bolt hole, upper and lower components and connecting plate, etc. The friction calculation adopts Lagrange multiplier algorithm, the tangential friction coefficient of contact is 0.15, the contact stiffness factor is 0.5, and the normal stiffness is 103000MPa.
3.3. Boundary and loading
According to the model design, the bottom of the column in the lower assembly test piece adopts a fixed constraint and a sliding support at the beam end. The loading method is unidirectional displacement loading, and the loading point is the position of the reverse bending moment point of the column in the upper assembly test piece. The specific setting is shown in Figure 6.

3.4. Meshing
In order to improve the calculation accuracy, the model adopts hexahedral meshing. The element adopts the 8-node hexahedral linear non-coordinating mode element C3D8I, and the meshing model is shown in Figure 7.

4. Comparison and analysis
4.1. Comparison of deformation state
The deformation state comparison between numerical simulation and test results is shown in Figure 8. Through comparison, it can be seen that the finite element calculation results and the experimental results have good consistency in the overall and local deformation states.
4.2. Comparison of bearing capacity

The finite element calculation results and the load-displacement curve measured by the test are shown in Figure 9; the comparison of joint stiffness and ultimate bearing capacity is shown in Table 2.
Table 2 Comparison table of stiffness and ultimate bearing capacity

| Specimen number | Stiffness (kN·m·rad⁻¹) | Er(%) | ultimate bearing capacity (kN) | Er(%) |
|-----------------|------------------------|-------|-------------------------------|-------|
| Test value T    | Calculated value C     |       | Test value T                  | Calculated value C |
| JD-a-0          | 5855.0                 | 0.1   | 85.6                          | 3.2   |
| JD-b-0          | 6065.0                 | 1.3   | 93.1                          | 4.1   |
| JD-a-1          | 5957.3                 | 0.8   | 89.1                          | 1.7   |

Remark: $E_r = 100\% \times |C - T| / C$

By comparison, it can be concluded that the finite element calculation results are in good agreement with the experimental results. Adding ribs and increasing beam height can both improve the mechanical performance of the joints, and the relative error between the calculated value and test value of stiffness and ultimate bearing capacity is within 5%, and the finite element modeling method which proposed in this paper is proved to be accurate.

5. Conclusion
The research results show that the results of finite element analysis are in good agreement with the experimental results in terms of deformation state and bearing capacity, and the accuracy of the finite element model is verified. The finite element model established in this paper can simulate the stress state of various parts of the joint which can make up for the shortage of the experimental measurement.

Acknowledgement
This research was supported by the Research and demonstration of standardization, sizing and industrialization of temporary facilities on construction site (Project No. 2016YFC0702102-02). The authors are grateful for this financial support.

References
[1] Qu KX. (2014) Study on Steel Structural System in Modular Buildings Tianjin University.
[2] Liu XC, Ren X, Zhan XX, et. (2018) Mechanical Property Analysis of Beam-to-Column Connection in a Box-Type Modular Prefabricated Steel Structure Building Industrial Construction, 5: 62-69.
[3] Deng EF Zong L, Ding Y. (2018) Mechanical Properties of Innovative Connection for Integrated Modular Steel Construction. Journal of Tianjin University (Science and Technology), 7: 6.
[4] Chen Z, Liu J, Yu Y. (2017) Experimental study on interior connections in modular steel buildings. Engineering Structures, 147: 625-638.
[5] Doh J H, Ho N M, Miller D, et al (2016). Steel bracket connection on modular buildings. J Steel Structure Construction, 121: 2472-0437.
[6] Ye JR, Y H. (2019) Review on Connection of Modular Steel Building. Guangdong Architecture Civil Engineering, 3: 9-12.
[7] Lee S S, Park K S, Hong S Y, et al.(2015) Behavior of C-Shaped Beam to Square Hollow Section Column Connection in Modular Frame. Journal of Korean Society of Steel Construction, 5: 471-481.