



Analysis on Bearing Capacity and Stiffness of Ancient Wooden Architectural Components Under Various Vertical Loads

Yuhang Li^{1*}, Lixin Shen², Linxi Wang²

¹School of Mechanical and Electrical Engineering, Beijing Polytechnic, Beijing 100176, China

²School of Civil and Transportation Engineering, Beijing University of Civil Engineering and Architecture, Beijing 100044, China

Email: Yuhang Li: *liyuhang@bpi.edu.cn,
Lixin Shen: 2108590022082@stu.bucea.edu.cn,
Linxi Wang: wanglinxi0425@163.com

Abstract. The study investigates the load-bearing capacity and stiffness of ancient wooden architectural components under various vertical loads, with a focus on dougong and mortise-tenon joints, which are vital for the stability of ancient wooden structures. Using ANSYS software, finite element models were developed to simulate the effects of various vertical loads on these components. The study confirms that the performance of dougong and mortise-tenon joints improves with increased vertical load, enhancing their stiffness and load-bearing capacity. Particularly, the column head dougong shows a faster rate of load increase. The research provides valuable insights for the preservation and restoration of historical wooden buildings, ensuring their long-term maintenance and cultural heritage continuity.

Keywords: Ancient wooden buildings, Finite element analysis, Dougong, mortise-tenon joint, vertical load impact

1 Introduction

Dougong and mortise-tenon joints are among the most important components in large, towering ancient wooden architectural structures and are also the main energy-dissipating elements. In the analysis of the bearing capacity and stiffness of ancient wooden components, vertical loads are a key factor, directly affecting the stability and durability of the wooden components [1,2]. Over time, wooden components may be subject to environmental erosion, material aging, and fatigue corrosion, leading to a decrease in bearing capacity and a reduction in stiffness [3]. Therefore, systematic research on the performance of these components is crucial for ensuring the safety and stability of ancient buildings [4].

Studies have shown that the column foot joints and mortise-tenon joints, as the weak links in the wooden structure, play a decisive role in the strength and stiffness of the

entire structure [5]. By analyzing the performance of these key joints, simplified models can be established to accurately assess the overall performance and condition of the structure. For example, through full-scale experiments and numerical simulation, the correctness of theoretical models can be verified, and the deformation characteristics and stress states of joints under horizontal repeated loads can be studied [2,6]. In addition, the bending performance of mortise-tenon joints has an important impact on the lateral stiffness and bearing capacity of the structure. Through experimental research and finite element simulation of mortise-tenon joints, a deeper understanding of their moment-rotation relationship, ultimate bearing capacity, and failure modes can be achieved. These research results help to establish more accurate mechanical models to predict and evaluate the performance of wooden structures under different vertical loads. In practical applications, by reinforcing and repairing wooden components, their bearing capacity and stiffness can be improved, thus meeting the principle of "restoring the old as it was." For instance, the embedded wall can significantly enhance the stiffness and strength of the columnar structure, with transverse walls contributing more to the initial elastic stiffness and overall shear stiffness [7].

This paper uses the ANSYS software to establish a finite element model and discusses the changes in load and stiffness of Dou Gong and mortise-tenon under different vertical loads. It can provide a scientific basis for the protection and restoration of ancient buildings, ensuring the long-term preservation and inheritance of these precious cultural heritages.

2 Model Verification

2.1 Experimental Overview

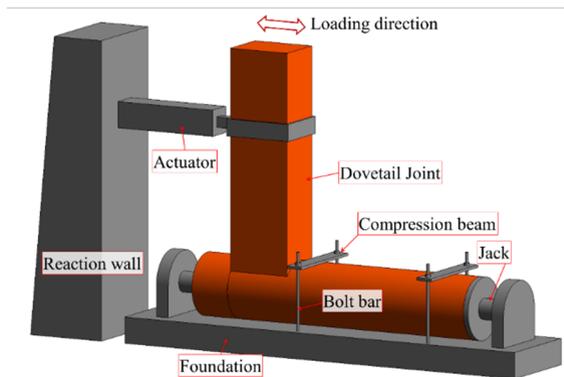


Fig. 1. Schematic diagram of tenon joint loading

When determining the parameters of Dou Gong and mortise-tenon joints, if an experimental method is used to construct the relationship between experimental parameters and overall model parameters, it will greatly increase costs. Therefore, a numerical simulation method based on ANSYS Workbench is used to unify the construction of Dou

Gong and mortise-tenon joints. To verify the accuracy of the numerical simulation method, the finite element model of the DJ1 mortise-tenon in the literature [8] is established using ANSYS, and numerical simulation calculations are carried out and compared with the experiment.

According to the literature [8], the swallow-tail tenon is selected as the research object for the loading test, with a scale ratio of 1:3.2. During the test, the column is laid flat, both ends are fixed, and the beam is placed vertically. The loading schematic is shown in Figure 1.

2.2 Finite Element Model

The finite element model of the swallow-tail tenon test is established using ANSYS Workbench, with dimensions and materials completely consistent with the test [8]. The wood used in the test is Chinese fir, and the elastic and plastic parameters of the material are shown in Table 1. In the numerical simulation, the wood is considered as an orthotropic material, with equal tensile and compressive modulus of elasticity, and the constitutive model for both tension and compression of the wood is a bilinear model. To define the plastic properties of the wood, the Hill yield criterion is called in the software, and the yield stress ratios in various directions of the wood are set. Solid186 elements are used to mesh the mortise-tenon joints. Due to the simplicity of the model, only a global element size of 0.01m is divided, and the finite element model mesh is shown in Figure 2. During the simulation, to be consistent with the test, both ends of the column are set to be fixed, with a steel pad at one end and a wooden padboard pressing the swallow-tail tenon at the other end. Additionally, a steel pad is set at the loading point, and the steel pads at both locations are constrained with bonded constraints. A vertical force is applied at the wooden padboard, and a horizontal displacement is applied at the steel pad at the loading point, with the loading system consistent with the test [9]. The contact between the mortise-tenon joints is frictional contact, and the contact formula uses the Augmented Lagrange method, with the friction coefficient taken as 0.4 [10] according to the test results.

Table 1. Elastic-Plastic Parameters of *Pinus sylvestris* Material

Elastic	Property	E_L (MPa)	E_R (MPa)	E_T (MPa)	G_{LT} (MPa)	G_{RT} (MPa)	G_{LR} (MPa)	μ_{LT}	μ_{RT}	μ_{LR}
	Value	3805	268	154	268	154	268	0.1	0.35	0.5
Plastic	Compressive strength (MPa)			Tensile strength (MPa)			Shear strength (MPa)			
	Property	Along grain	Transverse grain	Along grain	Transverse grain	Along grain	Transverse grain	Along grain	Transverse grain	
	Value	69.37	3.12	23.36	3.12	7.1	3.5			

Note: E is the elastic modulus, G is the shear modulus, μ is Poisson's ratio, L is longitudinal, R is radial, T is chord, RT is transverse, LR is radial, LT is chord.

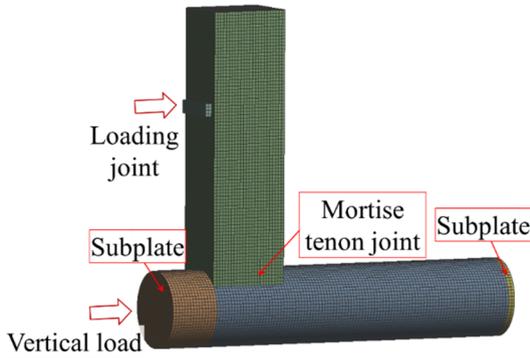


Fig. 2. Finite element model mesh division

2.3 Simulation and Test Comparison

The force-displacement curve obtained from the simulation can be converted into an $M-\theta$ curve, where M is derived by multiplying the force by the lever arm L , and θ is obtained by dividing the displacement by the lever arm L . The comparison between the skeleton curve of the numerical simulation and the test is shown in Figure 3. It can be seen that the shapes and trends of the two are basically the same, with the forward and reverse bending bearing capacities in the test differing by 7.33% and 3.28%, respectively, which is a small difference. This indicates that the finite element simulation method based on ANSYS Workbench is reliable, which also lays the foundation for subsequent unified research on Dou Gong and mortise-tenon.

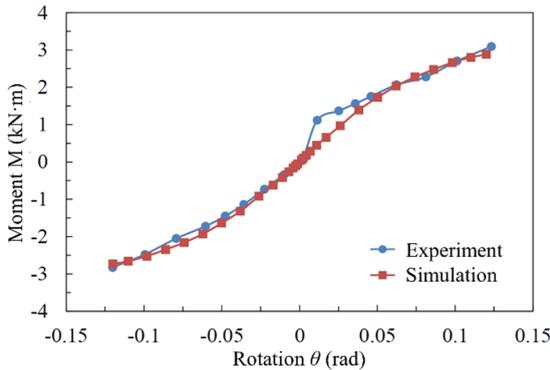


Fig. 3. Comparison of skeleton curves between numerical simulation and experiment

3 Model Establishment

3.1 Dou Gong

Using the aforementioned finite element simulation method, a unified study of Dou Gong is conducted, with main parameters including the geometric scale ratio of Dou Gong, material properties, and axial force. The main indicators are the initial stiffness of Dou Gong and the load at corresponding displacements.

Since the Ming layer Dou Gong and the flat seat layer's fork column Dou Gong are different, two types of models, the two-step column head Dou Gong and the three-step fork column head Dou Gong, are taken as full-scale models for the study, using ANSYS Workbench to establish the finite element models of the Ming layer column head and fork column Dou Gong, as shown in Figure 4. These two models also serve as the benchmark models for subsequent unified research. The bottom of the Dou of the Ming layer column head Dou Gong is fixed, with a steel pad set on the loading beam, and a vertical force (10kN) is applied on the top. A unidirectional horizontal displacement load is applied at one end of the beam, and the force-displacement curve is extracted. The fork column Dou Gong is subjected to a vertical force (10kN) on the fork column, and a horizontal load is applied on the fork column, extracting the force-displacement curve. The material properties of Dou Gong are the same as the parameters of the verification test, as shown in Table 1. The plastic constitutive model still uses the Hill yield criterion [11]. For each layer of the Dou and Gong in Dou Gong, tetrahedral meshes are used, with Solid187 elements and a mesh size of 0.04m. For the hidden tenons of each layer of the Dou and Gong, hexahedral meshes are used, with Solid186 elements and a mesh size of 0.0075m. The friction parameters between the Dou Gong and other parameters not mentioned are the same as the parameters in the verification simulation.

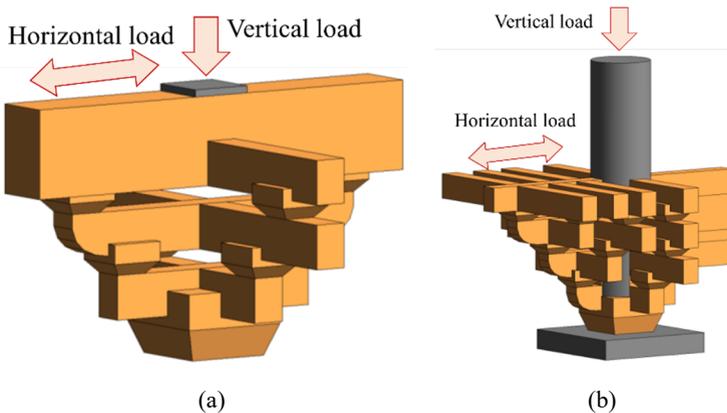


Fig. 4. DouGong model of: (a) Ming layer column head and (b) fork column

3.2 Mortise-Tenon Joints

Similarly, a unified study of the mortise-tenon joints is conducted, mainly including the geometric scale ratio of the mortise-tenon, material properties, and axial force. The main extracted indicators are the initial stiffness of the mortise-tenon and the load at the corresponding displacement. Since the swallow-tail tenon is more common in ancient wooden architectural structures, the swallow-tail tenon in the literature (Xue et al., 2016) is chosen as the object of the unified study.

Based on ANSYS Workbench, a finite element model is established. One end of the column is fixed with an iron plate, and the other end is compacted with a local general beam (as shown in Figure 1). An axial force is applied to the column, and a horizontal displacement load is applied to the end of the beam, and the force-displacement curve of the mortise-tenon is extracted. The material properties of the mortise-tenon joint are the same as those of the verification model, as shown in Table 1, and the yield criterion still uses the Hill criterion. Since the swallow-tail tenon is relatively simple, a hexahedral mesh can be used, with Solid186 elements and a mesh size of 0.04m, and other parameters are completely the same as the verification model.

4 Results

4.1 Dou Gong

Existing studies have shown that the greater the vertical load on Dou Gong, the better its hysteresis performance. Therefore, a unified study on the impact of vertical load on the hysteresis performance of Dou Gong is carried out. Taking the full-scale model as the research object, four models of each type of Dou Gong are established, each subjected to vertical loads of 10, 20, 30, and 40kN. The load-displacement curves of each Dou Gong are extracted, and the initial stiffness and the load values at corresponding displacements are calculated, with the 10kN component as the benchmark (vertical load ratio of 1), and the initial stiffness ratio and load ratio under various vertical loads are calculated, as shown in Figure 5. It can be seen from the figure that the load coefficient increases rapidly with the increase of the vertical force, and the stiffness coefficient also increases slowly with the increase of the vertical force. Among them, the column head Dou Gong has a faster load increase rate compared to the fork column head Dou Gong. Comparing the peak load under different vertical forces in the literature (Wang, 2007), it is known that when the vertical force is about doubled, the load is about 1.7 times the original load, which is similar to the calculation in Figure 5, indicating that this method can calculate the hysteresis parameters of Dou Gong under different vertical loads.

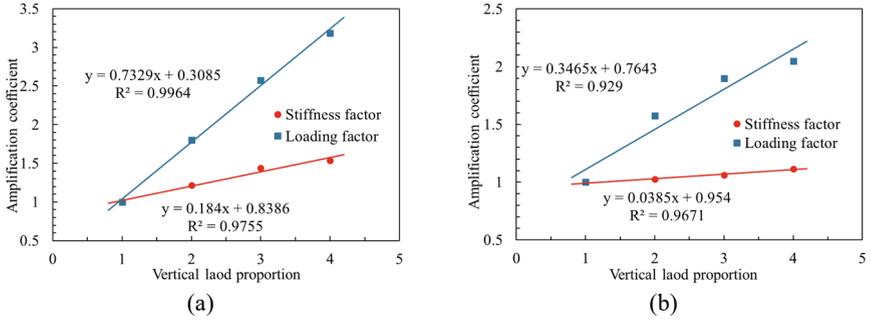


Fig. 5. The relationship between vertical load and the stiffness and strength of Dougong: (a) Ming layer column head and (b) fork column

4.2 Mortise-Tenon Joints

Four models are established to perform finite element simulation under different vertical force loads, with each model subjected to vertical loads of 10, 20, 30, and 40kN. The load-displacement curves of each mortise-tenon are extracted, and the initial stiffness ratio and load ratio at corresponding displacements are calculated with 10kN as the calculation benchmark (vertical load ratio of 1), as shown in Figure 6. It can be seen from the figure that the stiffness increases significantly with the increase of the vertical force, mainly because the vertical force received by the local general beam will increase the initial stiffness of the mortise-tenon joint. However, for the embedded swallow-tail tenon, the load coefficient remains almost unchanged.

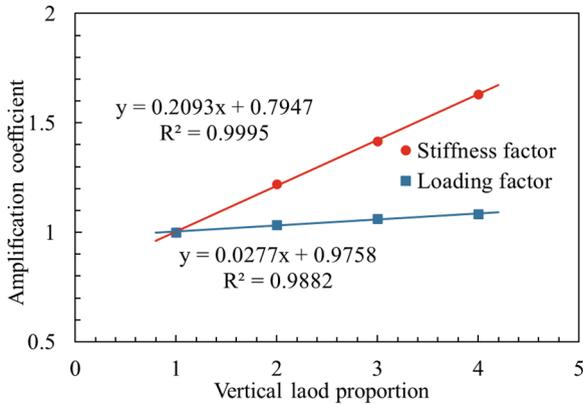


Fig. 6. The relationship between vertical load and the stiffness and strength of mortise-tenon.

5 Conclusions

This study used ANSYS software to establish finite element models and discussed the changes in load and stiffness of Dou Gong and mortise-tenon under different vertical loads. The conclusions are as follows:

This research used ANSYS software to establish finite element models of Dou Gong and mortise-tenon, analyzing the changes in load and stiffness under different vertical loads. The results show that the hysteresis performance of Dou Gong and mortise-tenon increases with the increase of vertical load, and the load coefficient and stiffness coefficient also increase accordingly. In particular, the column head Dou Gong has a faster load increase rate compared to the fork column head Dou Gong. The initial stiffness of the mortise-tenon joint increases significantly with the increase of vertical force, especially the vertical force received by the local general beam has a significant effect on improving the initial stiffness of the mortise-tenon joint. These findings provide a scientific basis for the protection and restoration of ancient buildings, helping to ensure the long-term preservation and inheritance of these precious cultural heritages.

Acknowledgments

This work was supported by the Beijing Polytechnic Key Program [grant numbers 2024X014-KXZ].

References

1. Li, Y., Deng, Y., Li, A. (2021). Simplified Simulation Method for Hysteretic Behavior of Wood Brackets. *Journal of Performance of Constructed Facilities*, 35(6): 04021085.
2. Xie, Q., Wang, L., Zhang, L., et al. (2020). Rotational behaviors of fork column dou gong: Experimental tests and hysteresis model. *Journal of Performance of Constructed Facilities*, 34(3), 04020032. [https://doi.org/10.1061/\(ASCE\)CF.1943-5189.0001432](https://doi.org/10.1061/(ASCE)CF.1943-5189.0001432)
3. Zhang, G., Sun, Z., Ding, M., et al. (2020). Field test and emergency reinforcement evaluation for an ancient wood pagoda. *The Structural Design of Tall and Special Buildings*, 29, e1686. <https://doi.org/10.1002/tal.1686>
4. Deng, Y., Li, Y., Li, A. (2023). Seismic safety assessments of historical timber buildings using updated finite element models: Case study of Yingxian wooden pagoda, China. *Journal of Building Engineering*, 63, 105454.
5. Chen, Z. Y. (2011). Research on the typical node and structural force performance of the Ying County wooden pagoda [doctoral dissertation]. Harbin Institute of Technology.
6. Yan, Z. (2019). Seismic performance test research and numerical analysis of a wooden pagoda's dougong [master dissertation]. Beijing University of Civil Engineering and Architecture.
7. Standoli, G., Salachoris, G. P., Masciotta, M. G., et al. (2021). Modal based FE model updating via genetic algorithms: Exploiting artificial intelligence to build realistic numerical models of historical structures. *Construction and Building Materials*, 303, 124393. <https://doi.org/10.1016/j.conbuildmat.2021.124393>

8. Xue, J. Y., Li, Y. Z., Xia, H. L., et al. (2016). Seismic performance test study on ancient building tenon joints with different looseness degrees. *Journal of Building Structures*, 37(4), 73-79.
9. Sha, B., Xie, L., Yong, X. et al. (2021). Hysteretic behavior of an ancient Chinese multi-layer timber substructure: A full-scale experimental test and analytical model. *Journal of Building Engineering*, 43, 103163.
10. Pan, Y., Zhang, Q., Wang, X. Y., et al. (2021). Mechanical model research on tenon joints of ancient building wood structures. *Journal of Building Structures*, 42(08), 151-159.
11. Wang, X. M. (2007). *ANSYS Engineering Structural Numerical Analysis*. Beijing: People's Communications Press.

Open Access This chapter is licensed under the terms of the Creative Commons Attribution-NonCommercial 4.0 International License (<http://creativecommons.org/licenses/by-nc/4.0/>), which permits any noncommercial use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license and indicate if changes were made.

The images or other third party material in this chapter are included in the chapter's Creative Commons license, unless indicated otherwise in a credit line to the material. If material is not included in the chapter's Creative Commons license and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder.

