Analysis of Wind-Resistant Ability and Failure Mode of Prefabricated WPC Enclosure Based on ABAQUS

Tong Luo\textsuperscript{1*}, Shangwei Chen\textsuperscript{1}, Qiaoyan Wang\textsuperscript{2}

\textsuperscript{1}Guizhou construction science research & design institute limited company of CSCEC, Guiyang, 550006, China
\textsuperscript{2}School of Civil Engineering, Guizhou University, Guiyang, 550025, China
Email: 992355298@qq.com

Abstract. In this paper, a prefabricated wood-plastic enclosure finite element analysis model, and its stress distribution under wind load and the failure mode of the enclosure under the maximum wind load bearing capacity are analyzed with the ABAQUS finite element code. The results show that under the horizontally distributed load, the mid-span deflection, mid-span stress and strain of the baffle gradually increase from bottom to top. The column has greater constraints on the baffles 1-1 to 1-3, and less on the upper baffle, resulting in a gradual increase in the maximum stress of the upper baffle. When the horizontally distribution load reached the maximum load of 1.74 KPa, cracks began to appear at the support contacts of the columns, resulting in component failure.

1. Introduction
Wood-plastic composite (WPC) is a new type of energy-saving and environmentally-friendly composite material that is widely used in gardens, automobiles, and construction industries. With the enhancement of people's environmental consciousness, the output and market of WPC are continuously expanding [1-3]. WPC materials have excellent properties such as water resistance, acid and alkali resistance, easy molding and processing, and recyclability, which have been favored by more and more scientific researchers, manufacturers and consumers [4]. As a new type of high turnover, recyclable prefabricated enclosure, WPC enclosure is a high-quality alternative to traditional brick wall enclosure, and has been put into market applications [5-6]. The WPC enclosure is mainly subjected to wind loads during its service life and the main load-bearing components are baffles and columns. The baffles bear the wind load and transfer the force to the columns, and the columns ensure the stability of the overall structure. Therefore, in order to verify the stability of the wind resistance of WPC enclosure, this paper uses ABAQUS finite element analysis software to simulate and analyze the wind load resistance of WPC enclosure [7-8], which provides a basis for the on-site wind-resistant design and installation of WPC enclosure.

2. Structure Form and Model Establishment of Fabricated WPC Enclosure

2.1. Fabricated WPC Enclosure Structure
The height of the enclosure is 2.0 m, the column spacing is 2.0 m, and the upper and lower beams are fixed to the enclosure baffle surface through the slot. The structure of the enclosure unit is shown in figure 1 and figure 2.
The main component forms are as follows:

1- baffle; 2- column; 3- column cap; 4-upper beam; 5-connector; 6-lower beam; 7-bracket; 8-base

The main component forms are as follows:

(a) Column  (b) Top view of column  (c) Column cap  (d) Beam

(e) Column connector  (f) beams and connectors  (g) baffle

Figure 1. Structure of enclosure unit.

Figure 2. Component form of enclosure unit.

2.2. Finite Element Model and Material Constitutive

The WPC enclosure structure unit is modeled by finite element simulation analysis. The model was composed of baffles, columns, upper and lower beams, bases and diagonal supports. The model diagram is shown in figure 3. The enclosure members are numbered, the windward side is from left to right, column 1, column 2, and column 3. The bases and supports are numbered in the same way, and the baffles are numbered from left to right and from bottom to top number, for example, the bottom board of the left frame is board 1-1. In the actual situation, the baffle is inserted into the card slot of
the column, so the relative displacement between the baffle and the column is negligible. Tie contact is used in the simulation, and surface-to-surface contact is adopted between the baffles and between the base and the column [9-10].

![Figure 3. Enclosure model diagram.](image)

In the finite element analysis of the material constitutive relationship, the performance parameters of WPC are shown in table 1, and the performance parameters of the steel used are shown in table 2. The simulated loading method is staged loading. Before the pressure is 1000 Pa, the pressure of each stage is increased by 200 Pa, and the load is divided into 5 stages. After the pressure is 1000 Pa, the pressure of each stage is increased by 100 Pa until the model is broken.

**Table 1. PE wood plastic material parameters.**

| Density (g/mm$^3$) | Elastic Modulus (MPa) | Poisson's ratio | Yield stress (MPa) | Maximum stress (MPa) | Maximum absolute plastic strain |
|--------------------|-----------------------|-----------------|--------------------|----------------------|--------------------------------|
| 0.000917           | 4100.97               | 0.36            | 3.255              | 11.096               | 0.00381                        |

**Table 2. Steel material parameters.**

| Density (g/mm$^3$) | Elastic Modulus (MPa) | Poisson's ratio |
|--------------------|-----------------------|-----------------|
| 0.0078             | 2.06e5                | 0.31            |

### 3. Analysis of Overall Structural Performance of WPC Enclosure

#### 3.1. Finite Element Analysis Results of Baffle

Based on the results of finite element analysis, the 1-6 baffle components in the model were extracted, and the stress, strain cloud diagram and baffle deformation diagram along the length of the baffle were analysed.

Figure 4(a) is the stress cloud diagram of the leeward surface of the baffle, and figure 4(b) is the stress cloud diagram of the windward surface of the baffle. It can be seen from the figure that under the action of wind load (Uniformly distributed load), the leeward panel mainly bears tensile stress. The stress at the mid-span is greater than the rest, and the maximum stress is 7.33 MPa. The windward panel is mainly subjected to compressive stress, and the stress at the mid-span is also greater than the other positions, the maximum stress is 6.94 MPa, and the maximum tensile stress is almost equal to the maximum compressive stress. Figure 5 shows the strain cloud diagrams of the windward and leeward sides of the baffle 1-6, the maximum tensile strain is 0.0033, and the maximum compressive strain is 0.0019. Figure 6 is a deformation diagram of the baffles 1-6 along the Z-axis direction, it can be seen from that the maximum deformation of the baffle is at the middle position of the baffle, which is 93.37 mm. The finite element simulation results are shown in table 3.
Figure 4. Baffle 1-6# strain cloud diagram.

(a) Leeward side.  
(b) Windward side.

Figure 5. Baffle 1-6# strain cloud diagram.

(a) Leeward side.  
(b) Windward side.

Figure 6. Displacement cloud diagram of board 1-6# Z axis.

Table 3. Finite element simulation results of baffles.

| Mid-span winding (mm) | Maximum tensile stress (MPa) | Maximum compressive stress (MPa) | Maximum tensile strain | Maximum compressive strain |
|-----------------------|------------------------------|---------------------------------|------------------------|---------------------------|
| 1-1                   | 52.20                        | 5.81                            | 4.41                   | 0.0021                    | 0.0010                    |
| 1-2                   | 55.52                        | 5.84                            | 4.57                   | 0.0022                    | 0.0011                    |
| 1-3                   | 67.20                        | 6.12                            | 4.92                   | 0.0023                    | 0.0012                    |
| 1-4                   | 77.47                        | 6.45                            | 5.24                   | 0.0027                    | 0.0013                    |
| 1-5                   | 85.88                        | 6.94                            | 6.16                   | 0.0030                    | 0.0015                    |
| 1-6                   | 93.37                        | 7.33                            | 6.94                   | 0.0033                    | 0.0019                    |

It can be seen from table 3 that, from bottom to top, the mid-span winding, mid-span stress and strain of the baffle gradually increase. From baffle 1-1 to baffle 1-3, the stress growth trend is relatively slow, and the stress growth trend of baffle 1-3 to baffle 1-6 is relatively fast. The column is restrained by the bottom end and diagonal support, which restricts baffle 1-1 to baffle 1-3 is larger, and the restraining force of the column on the upper baffle is smaller, which causes the maximum stress of the upper baffle to gradually increase.
3.2. Finite Element Analysis Results of Column

Analyze the stress and strain of No. 2 column along the length. Figure 7(a) is the stress cloud diagram along the Y-axis of the side view of the No. 2 column and the diagonal support, and the left side is the windward side. It can be seen from the figure that the stress is mainly concentrated in the contact part between the column and the oblique support. The tensile stress is the main part in the windward surface of the column, while the compressive stress is the main part in the contact part of the oblique support, and the maximum tensile stress is 7.08 MPa.

![Stress cloud diagram.](image1)

![Strain cloud diagram.](image2)

**Figure 7.** Y-direction stress cloud diagram in side view of No. 2 column.

Figure 8 is a stress cloud diagram of the contact position between the column and the diagonal support. It can be seen from figure 8 that stress concentration occurs between the column wall and the supporting part, and the oblique support provides an inward force to the column, causing the column wall to receive an inward bending moment here, and the inner wall is first X-axis and Y-axis tensile stress in the direction. Comparing figure 8(a) with figure 8(b), it can be seen that the main direction of stress is the X-axis, when the stress in the X-axis direction reaches the maximum tensile strength, cracks appear on the column wall and the component fails. It can also be seen from the figure 8 that if the column is divided from the support, the upper structure can be regarded as a cantilever structure subjected to uniformly distributed loads, and the lower structure is a statically indeterminate structure with fixed ends at both ends, and the two parts transmit force to the oblique support. That’s provide a clockwise bending moment, resulting in greater reaction force at the upper part of the diagonal support and lower reaction force.

![X axis direction.](image3)

![Y axis direction.](image4)

**Figure 8.** Stress cloud diagram of the contact position between the column and the diagonal support.
3.3. Finite Element Analysis Results of the Retaining Structure

Figure 9 shows the curve of the stress change with load at the maximum stress of the baffle and column. It can be seen from the figure that the curve is divided into two stages: the first stage is the elastic stage, and its deformation will gradually recover as the load is removed. When the load reaches 0.565 kPa, the stress reaches 3.25 MPa, and the baffle enters the elastoplastic stage. The structure in the elastoplastic stage will have a part of residual deformation as the load is removed. When the maximum load is reached, cracks begin to appear at the support contact of the column and the component fails. The simulation results are shown in table 4.

![Figure 9. Stress-load curve.](image)

### Table 4. Finite element simulation results of enclosure structure.

| Bearing capacity (KPa) | Mid-span winding (mm) | Column top displacement (mm) | Maximum board stress (MPa) | Maximum column stress (MPa) |
|------------------------|-----------------------|-----------------------------|---------------------------|-----------------------------|
| 1.74                   | 96.37                 | 17.79                       | 7.26                      | 11.77                       |

It can be seen from table 4 that the maximum stress of the column is much greater than the maximum stress of the baffle. The maximum stress of the overall enclosure structure occurs at the contact part of the column and the supporting structure, and it is also the first place where local failure occurs. The failure form changes from the collapse of the column root to the partial collapse of the supporting part, which avoids the collapse of the entire enclosure structure.

4. Conclusion

The research in this article shows that under the horizontally distributed load, the mid-span deflection, mid-span stress and strain of the baffle gradually increase from bottom to top. The column has greater constraints on the baffles 1-1 to 1-3, and less on the upper baffle, resulting in a gradual increase in the maximum stress of the upper baffle. When the horizontally distribution load reached the maximum load of 1.74 KPa, cracks began to appear at the support contacts of the columns, resulting in component failure.

Acknowledgments

This research was supported by the National Key Research and Development Program of China (Project No. 2016YFC0702102).
References

[1] Wang Q W, et al. 2007 Wood-Plastic Composites and Products (Beijing: Chemical Industry Press).

[2] Li J 2014 Trends in China’s Wood Plastic Composite Market China Wood-based Panels 21(02) 1-3.

[3] Zhang Y, et al. 2016 Improvement of Mechanical properties of wood-plastic composite floors based on the optimum structural design Acta Mechanica Solida Sinica 29(4) 444-454.

[4] The China Plastics Industry Editorial Office 2016 China Plastics Industry 44(3) 1-46.

[5] Martins G, et al. 2017 Optimization of a wood plastic composite for architectural applications Procedia Manufacturing 12 203-220.

[6] Daniel F, et al. 2016 Investigations on ageing of wood-plastic composites for outdoor applications: A meta-analysis using empiric data derived from diverse weathering trials Construction and Building Materials 124 1142-1152.

[7] Pavan B, et al. 2020 Assay of moment resisting precast and monolithic frames using ABAQUS software Materials Today: Proceedings 33(1) 162-169.

[8] Balineni H, et al. 2020 Analysis of dry and wet connections in precast beam-column joint using ABAQUS software Materials Today: Proceedings 33(1) 287-295.

[9] Singiresu S, et al. 2018 The Finite Element Method in Engineering 6th edition pp 671-701.

[10] Zhang J H, et al. 2012 ABAQUS Basic Primer and Case Mastery (Beijing: Publishing House of Electronics Industry).