Finite Element Analysis on Core-Assembled Lightweight Aggregate Concrete Slab

Kailin Yang¹ and Guangxiu Fang¹,∗
¹ Department of Structural Engineering, College of Engineering, Yanbian University, Yanji, Jilin, 133002, China
∗ Corresponding author’s e-mail: gxfang@ybu.edu.cn

Abstract. In this paper, the structural form of the core-assembled lightweight aggregate concrete slab has been changed, and the non-linear finite element analysis is carried out using ABAQUS software, which is compared with the experimental value. The damage model of ABAQUS found that the cracks in the simulation process are almost the same as the cracks in the test. The ABAQUS software is suitable for the flexural performance test of the core-assembled lightweight aggregate concrete slab. Finite element analysis provides a certain reference.

1. Introduction
Filled core assembled concrete slab (GXPCS) is different from general assembled concrete laminated slab (the cast-in-place part is laminated on the precast slab, the two meet the shear requirements of the laminated surface through truss or shear key), such The thickness of the prefabricated slab is small (small stiffness and low bearing capacity), which is easy to cause damage to the prefabricated slab during factory production or transportation. The prefabricated slab of GXPCS is a hollow slab, and the final slab height has been the same as the prefabricated slab. In the continuous structure, where the negative bending moment steel bar needs to be arranged in the lateral groove on the precast slab, and the post-cast concrete only needs to be poured in the hollow and groove to form an assembly. In this paper, under the interlocking mechanism proposed by Wu Fangbo, the groove structure is changed, and the three lateral grooves of the fabricated slab are changed into a rectangular groove to reduce the construction difficulty of the concrete slab.

2. Finite element simulation process
2.1. Element type selections
The concrete element used in this experiment is reduced integration, which is one integration point less than complete integration in each direction compared to full integration. It has the advantage of accurate displacement calculation and analysis when the grid is distorted. Advantages less impact on accuracy. For the solid element, the reduced integral element is generally used in the hexahedral element and the four-deformed element. In this chapter, the C3D8R model is selected for the concrete model. and Reinforced elements are truss elements in ABAQUS, which is generally divided into two categories, one is a three-node curved truss, such as T2D3 and T3D3, and the other is a two-node linear truss, such as T2D2 and T3D2. In this chapter, T3D2 is selected for the reinforcement model.
2.2. Constitutive choice

2.2.1. Concrete constitutive relationship
For the constitutive model of concrete, ABAQUS software can generally use three models, namely Concrete Smeared Cracking, Concrete Damaged Plasticity, and Cracking model for concrete. Concrete is used in this chapter. The plastic damage model, whose constitutive relationship is defined as the failure of concrete materials, which is divided into crushing and pulling cracks.

2.2.2. Reinforcement constitutive relationship
There are three kinds of constitutive relations for steel bars, which are double straight line model, double oblique line model, and trifold line model. In this chapter, the three-fold line model is used.

2.3. Modeling and meshing
The two prefabricated slab forms are modeled according to the size of test piece. The concrete modeling is shown in figure 1 to 2, and the reinforcement model is shown in figure 3. Then meshing is carried out, taking into account the special characteristics of the test piece with openings and side grooves. The meshing in this chapter uses "Sweep" meshing in order to allow the finite element to converge the calculation and improve the simulation accuracy. The grid size of the prefabricated slab model in this paper is 15mm. The specimen after meshing is shown in figure 4 to 5.

3. Test results

3.1. Stress analysis of slabs
As shown in figure 6 to 11. It is the stress cloud diagram of slabs LC1 and LC2. From the plot, it can be seen that the failure rules of the specimens are basically similar, and the area of maximum stress is concentrated at the upper flange of the hole. As the load increases, it is found that the stress of the hole develops with the width of the slab, and the stress on the top of the slab gradually increases, and developing to both sides. When the steel bar reaches yield, the concrete stress in the compression area increased rapidly until the simulation fails. It is found from the plot that changing the form of the structural plate on the side of the prefabricated plate has little effect on the mechanical performance of the prefabricated slab. (NC and LC stress are similar, in this paper only uses LC1 and LC2 as examples).

Fig. 6 Stress diagram of specimen LC1 before cracking

Fig. 7 Stress diagram of specimen LC2 after cracking

Fig. 8 Stress cloud model of specimen LC1 at yield

Fig. 9 Stress diagram of specimen LC2 before cracking

Fig. 10 Stress diagram of specimen LC2 after cracking
3.2. Crack analysis of slabs
The figures 12 to 15 show the distribution of cracks in slabs. It can be found from the figure that the crack development law of slabs obtained by finite element simulation is basically consistent with the laws during the test. The first crack is generated at the lower flange of the center hole, and with the load continues to increase, there are cracks in the flanges of other holes, few cracks occur between the two holes. When the load is added to a certain value, the side of the center hole oblique cracks begin to form on the flanges of the two holes and gradually developed upward until the simulation ended. The first crack in the bottom of the slab also occurs in the span, and there is a tendency for cracks to develop on both sides. Gradually, the cracks in the bottom of the slab under other holes occur and extend toward both sides.
3.3. Comparison experimental values and finite element

As shown in table 1, it is the comparison between the simulated value and the calculated value of the ABAQUS simulation calculation when the slabs cracking and yielding. From the table, it can be found that the simulated value under the characteristic load is often less than the measured value, which is because the test using manual loading, there will be certain mistakes in the control load of each level, there is friction between the loading slab and the steel slab, and the displacement gauge shakes slightly with the test piece, but the test value is not much different from the calculated value. ABAQUS finite element verification is applicable for GXPCS.

| Test piece | Test piece Cracking load | Test piece Yield load |
|------------|--------------------------|----------------------|
| Test value/kN | Analog value/kN | Test /Analog | Test value/kN | Analog value/kN | Test /Analog |
| LC1 | 8 | 6.7 | 0.81 | 17 | 15.6 | 0.92 |
| LC2 | 8 | 6.9 | 0.85 | 17 | 16.0 | 0.94 |
| NC1 | 9 | 7.0 | 0.72 | 19 | 17.6 | 0.93 |
| NC2 | 9 | 7.1 | 0.74 | 19 | 17.2 | 0.90 |

4. Conclusions

The ABAQUS software was used to verify and analyze the core-assembled light-weight aggregate concrete slabs and reached the following conclusions: The stress map and crack distribution map of the prefabricated plate obtained by the ABAQUS software are basically the same as the test results, and the crack development trend is basically consistent with the test. ABAQUS software is suitable for this experiment.

references

[1] Meng Fanlin Fabricated concrete hollow composite floor slab and its making method.
[2] Kailin Yang and Guangiu Fang 2018 IOP Conf. Ser.: Mater. Sci. Eng. 237 052069
[3] Kailin Yang and Guangiu Fang 2019 IOP Conf. Ser.: Mater. Sci. Eng. 446 022074
[4] Zhenhai Guo and Xudong Shi (2014) Principle and analysis of reinforced concrete. Tsinghua University Press.
[5] Guoli Zhang and Jun Su (2008) Nonlinear analysis of reinforced concrete based on ABAQUS. Science Technology and Engineering: 5620-5624.
[6] Jiang Qian and Kuo Lei (2008) Research on application of concrete damage plastic model. Structural Engineer. 24:22-23.
[7] Fangbo Wu and Biao Wu (2014) Analysis of Chain Occlusal Effect of Superimposed Surface of New Type Superimposed Floor. Journal of Hunan University (Natural Science Edition) 41:1-7
[8] Bing Sun and Dexin Ding (2008) Numerical analysis of secondary stress process of reinforced concrete superimposed structure. Journal of Huazhong University of Science and Technology (Urban Science Edition), 04: 260-263
[9] Study on Cored Assembled Lightweight Aggregate Concrete Plates, Proceedings of the 12th International Symposium Performance Improvement of Concrete for Long Life Span Structure, PICLS 2018.8: 171-175
[10] Analysis of bearing capacity of grouted assembled lightweight aggregate concrete plates, The international academic conference about modern construction technology between China and Korea in 2018, 2018 china korean International Symposium 2018.7: 27-31