

Simulation Steel Fiber Concrete Composite Outer Wallboard under Vertical Load Test with ABAQUS

Yabo Wang ^{*1}, Chuanhao Xi², Gaofeng Dou³

¹.*School of Civil Engineering, Jilin JIANZHU University, Changchun, Jilin,130118, China.*

².*School of Civil Engineering, Jilin JIANZHU University, Changchun, 130118, China.*

Abstract — The paper describes the form of SFRC rib board in steel fiber concrete composite outer wallboard, and simulates stress performance of composite outer wallboard under vertical loads. We compare the practical test to analysis to determine the differences and similarities.

Keywords - composite outer wallboard, steel fiber concrete, ABAQUS simulation test.

I. INTRODUCTION

With the development of the construction industry, people pay more attention to the construction of environment protection and energy saving, the new building materials, process, component forms emerge continuously. As an important part of building about Outer wallboard, the researchers continue to study it in-depth. Composite outer wallboard is a kind of building outer wallboard to adapt the times. With the advantages of more forms, thin board, lightly weight, economic type, the prospects for development in construction industry is very well. Steel fiber concrete composite outer wallboard is one kind of wallboard.

Steel fiber concrete composite outer wallboard reduce the thickness of traditional outer wallboard about building. At the same time, it also optimizes the plane coefficient and increase the effective useful area of building. Specific to this text, the steel fiber concrete composite outer wallboard has four different board and they get together as a whole. In the premise of guaranteeing the stress of wallboard, it improving the effect of energy saving, so it is necessary to study the steel fiber concrete composite outer wallboard.

This text is mainly to simulate fiber concrete composite outer wallboard with the finite element analysis software ABAQUS, which is based on the investigation of vertical load bearing building system with SFRC composed wallboard, and compare the difference about simulation test and practical text [1].

II. THE COMPOSITE OF STEEL FIBER CONCRETE COMPOSITE OUTER WALLBOARD

Steel fiber concrete composite outer wallboard, Fig 1. can be divided into four parts. We can see SFRC rib board, rock wool board, asphalt perlite board, gypsum board. Among the SFRC rib board, it has two parts.

Steel concrete rib, the steel concrete rib cross in the composite outer wallboard as a whole, it' s section size is

$60 \times 105\text{mm}^2$. There are two layers about longitudinal tensile bar in every rib, and fixed with erect ribs. The ribs along the edge of wallboard are closed, ribs and ribs cross into steel mesh. In the steel concrete, concrete grade is C30, cement grade is 425 ordinary cement. It is necessary to mix water reducer and early strength agent to configure. Steel fiber concrete board, (the length) $d=40\text{mm}$, (the ratio of length to diameter) $l/d \leq 60$, (volume ratio of steel fiber) $V_f=1.2\%$. In the steel fiber concrete board, concrete grade is C30, cement grade is 425 cement.

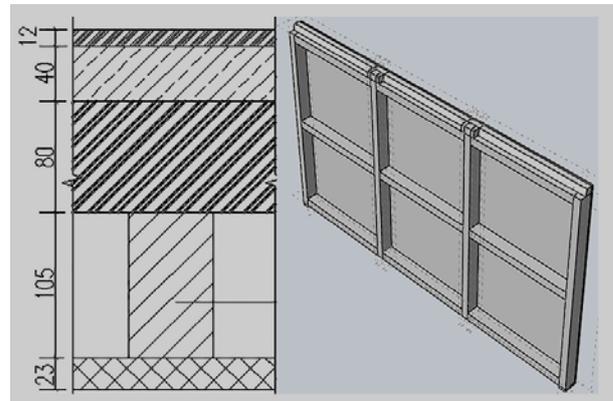


Fig 1. Composite wallboard Sectional View and Simulate SFRC rib board Test.

III. ABAQUS SIMULATE STEEL FIBER CONCRETE COMPOSITE OUTER WALLBOARD

In the steel fiber concrete composite outer wallboard, the mainly part to bear load is SFRC rib board. In order to simplify the simulated test with ABAQUS, the author only choose the SFRC rib for simulated target. The sectional size of simulated rib board is $3000 \times 1500 \times 128\text{mm}^3$ (without reinforcement area), as shown in Fig 1.

In order to avoid local destruction around action point under load application, widen the local concrete around the action point, and flush it to SFRC rib board, improve strength to ensure the test smoothly to load^[2].

At the simulate test with ABAQUS, concrete is 3D-variable-stretchable solid, steel bar is 3D-variable-plane-wire solid. Steel bar simulate in truss unit, and combine into steel mesh, then make it into concrete by embedded constraint, steel bar and concrete can work together^[3].

In ABAQUS, concentrated force can not be worked on the surface of concrete, so we can put a reference point on the strengthen local position by coupling constraint, then load the application on the reference point.

We simplify the material properties to confirm material parameter about 23mm thickness steel fiber concrete. Nowadays, the research on mechanical parameters of concrete mechanical parameters are very mature. But the mechanical properties are not exact when ordinary concrete materials mixed with other materials. According to the relevant steel fiber concrete briquettes test, compared to ordinary concrete, mixed with a small amount of steel fiber, the strength of steel fiber concrete won't be reduced, in contrast, it will improve a certain proportion of the strength. At the same time, the ductility and shear-bearing capacity of concrete are also enhanced. So, Only the young's modulus of elastic parameters is slightly higher than that of ordinary concrete in this simulation experiment, and Other material properties are similar to ordinary concrete^[4].

One end of the boundary for SFRC rib board with fixed hinge, $U1=0, U2=0, UR3=0$. the other end with movable hinge, $U2=0, UR3=0$. In order to avoid stress concentration, we can put a block to eliminate^[5].

The global size factor for concrete is 0.05 when we Divide the mesh, but we can reduce it to 0.03 when we divide the mesh for steel which bears main force. If we reduce size factor, the number of unit and the calculation time will increase, but iterative calculation is difficult to convergence^[6].

When we compute and analyze model, it is easy to convergence for elastic deformation before concrete cracking by using linear elastic theory to analysis stress of steel concrete structure. Because we have simplified the model very simple, the plastic deformation for concrete cracking and softened have not considered, the result will be error^[7].

According to the real test, loading method can be input 7 level, 14/15t, 20/15t, 26/15t, 32/15t(21.33KN), 38/15t, 44/15t, 50/15t(33.33KN)^[1]. The test has two jacks and two concentrated force points of action, so the load becomes to 38/15t~44/15t (50.7~58.7KN), there are normal section crack. When the load becomes to 44/15t~50/15t (58.7KN~66.7KN), concrete reaches to maximum compressive strength ($f_c=2.4e7N/mm^2$), and steel reaches to yield strength. In order to get ultimate load, we should adjust load to 58.7~66.7KN. If the load becomes 62KN, steel stress will reach $2.1e8N/mm^2$.

IV. THE COMPARISON BETWEEN SIMULATION TEST AND PRACTICAL TEST

The comparison between simulation test and practical test can be analysis from load paths, the destruction of the

component form, ultimate load and failure load and so on. From the comparison of them, the validity of practical was verified, and analysis the influence of test result about simulated test for simplified model.

A. Load paths

The load paths of practical test from load point to longitudinal steel and bearing. Slender web has the function of stabilizing supporting, and the layer of board is no layering and no separation. The destruction is overall damage, and there is no local failure and collapse, so the mechanical behavior of whole composite wallboard is well^[1].

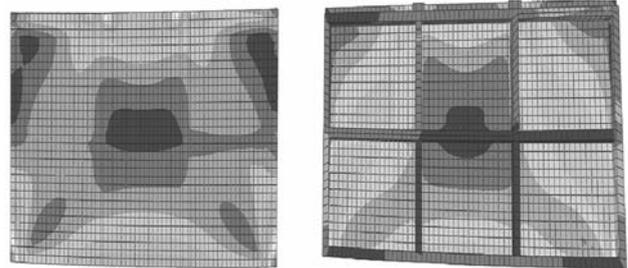


Fig 2. Concrete Cloud Atlas.

The result of ABAQUS simulation, Fig 2. , shows that the strained condition of concrete and steel mesh conform to practical test. The load paths form local strengthening regional to longitudinal steel and bearing diagonal. Steel fiber concrete rib(23mm thickness) don't have collapse, and develop the function of supporting wallboard. According to Fig 3. , we can see that the mechanical behavior of SFRC rib is well about ABAQUS simulation. Steel concrete rib play a framework role in all rib layers of composite wallboard, and the bearing confirms to practical test.

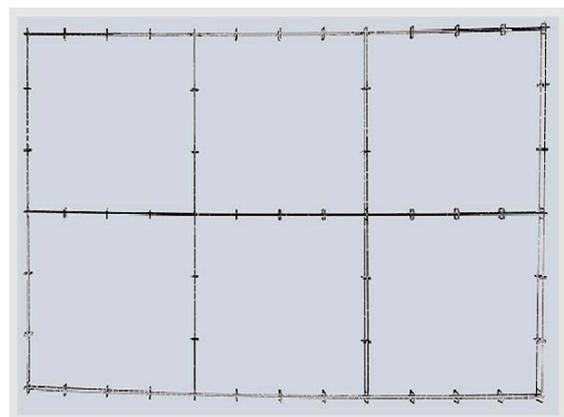


Fig 3. Concrete Cloud Atlas.

In practical test, we put two tripods in front and back of wallboard, and eight small steel plates, link together with wallboard along thickness direction. This is equal to put constrains around the board. Under vertical load, bending moment and displacement will be produced on board, lateral direction won't be collapse. In simulation test, we don't put constraint boundary around board. By means of

adjusting the thickness of steel fiber concrete rib layer, form Fig 4, we can see that if the thickness reduced , SFRC rib will be warping.

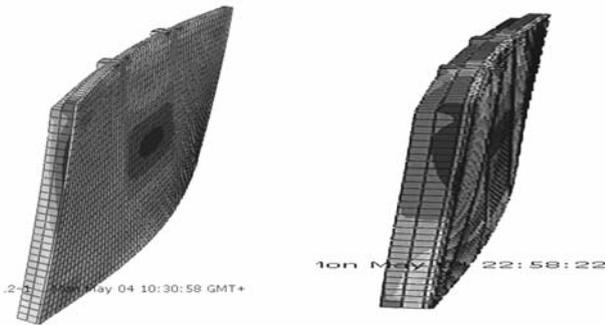


Fig 4. SFRC rib warping.

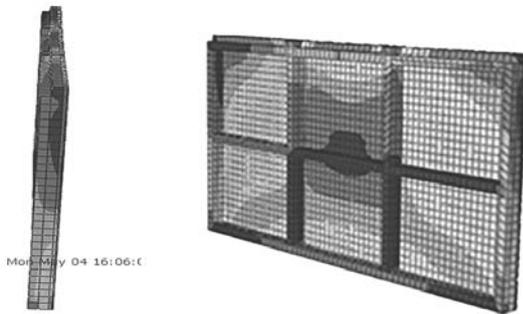


Fig 5. SFRC rib normal Stress.

Fig 6. SFRC rib Cloud Atlas of Stress.

Besides, the distance of hanger bars has great influence on rib simulation. If the distance is 200mm~450mm, in terms of this test wallboard, along the increasing load, there are lateral distortion, edge warping, local crushing at bearing at loading. We can avoid these by putting lateral strut and heel block with boundary condition, then the normal stress result of state of SFRC rib can be seen like Fig 5. And Fig 6.. And we can see that the color and depth of cloud chart about wallboard are different, in ABAQUS input visual interface cloud chart along increasing load..

B. The Destruction of the Component Form

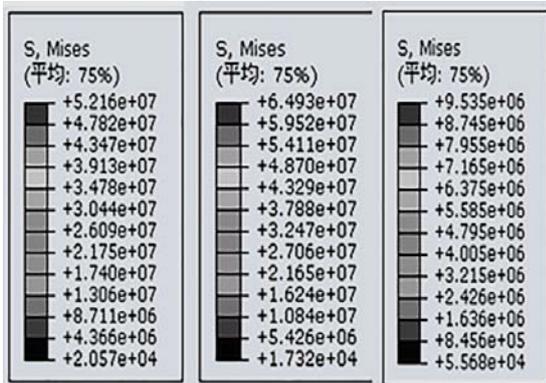


Fig 7. Steel Mises Stress.

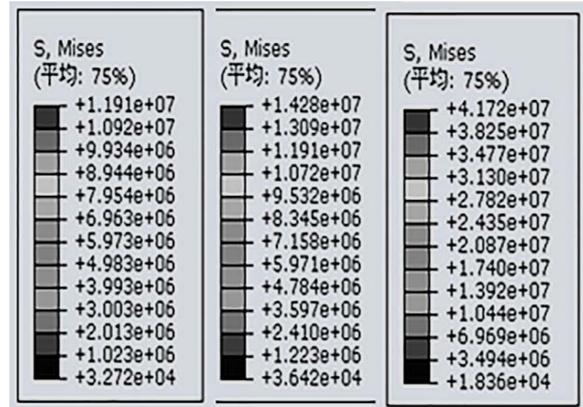


Fig 8. Concrete Mises Stress.

When the wallboard Under the floor under two concentrated force at the same time, along with the increased of the graded load, as shown on Fig 7. and Fig 8., we can see the maximum mises of steel bar and concrete is Nonlinear increased. But by analyzing the stress data of steel bar and concrete, the distribution of the stress in the wallboard is consistent with the measured data. The main stress is located in the support of the wall panels, stress concentration is the main reason causing the failure of the wallboard. Because the bending failure and shear failure are all reflected on wallboard, we can get a conclusion that the final failure on wallboard is arch-rib failure.

C Ultimate Load and Failure Load

In simulation SFRC rib under vertical load with ABAQUS, the ultimate load is 62KN. Compare to the practical test, the load reduces 7.01%.The error is relative small, and it is in line with the expected results. Limited to the present research on the steel fiber mechanical parameters which can not be a good quantitative, it confirm to the prediction of simulation test which simplified the model. Under design load, the wallboard is in elastic stage^[7]. The numerical value of stress and strain are not big. Along the increasing load, before reaching the ultimate load of SFRC rib, stress and strain of every point in plastic stage are non-linear. After the ultimate load, the stress are not increasing and strain continue to grow.

V. CONCLUSION

This paper simulates steel fiber concrete composite wallboard by general software ABAQUS, the main focus of this test is mechanical behavior of SFRC rib under vertical load. Compared to the simulation test and practical test, the mechanical behavior is the same result basically and expectation. In simplified model, the dispersion of concrete, cracking and fiber concrete material characteristics don't discuss in depth and no reliable data, so there are errors in simulation result. But it does not influence the validity of practical test.

REFERENCES

- [1] Y.B. Wang, "The Investigation of Vertical Load Bearing Building System with SFRC Composed Wallboard", pp. 67-85, 1992,
- [2] Z. H. Guo, X. D. Shi, "Reinforced Concrete Theory and Analyse", Tsinghua University press, 2003, pp. 121-183,
- [3] J. G. Nie, Y. H. Wang, "The development and application of steel-concrete composite structure fiber model with ABAQUS", 2012, pp. 15-37,
- [4] Y. H. Zhao, J. J. Zheng, "The boundary effect of the directional coefficient of concrete 3D elastic modulus", The newspaper of Beijing Jiaotong University, vol. 01, pp. 47-50, 2004,
- [5] Y. Z. Wang, C. G. Fu, "Structure Engineering Analysis and Example Explanation with ABAQUS", pp. 10-82, 2004,
- [6] Y. P. Shi, Y. R. Zhou, "Finite Element Analysis for Example of Explanation In Detail", China Building Industry Press, pp. 189-202, 2010,
- [7] G. L. Zhang, J. Su, "Analysis of Concrete Non-linear with ABAQUS", Machinery Industry Press, pp. 17-92, 2014,