Abstract. In this paper, the stress bearing behavior of three different frame structure groups (all-cast-in-place RC, all-cast-in-place steel cage concrete and fabricated steel cage concrete) which under horizontal low cyclic reciprocating force will be analyzed theoretically. Simulation results show that, in these three structures, all-cast-in-place steel cage concrete structure has the maximum bearing capacity, and fabricated steel cage concrete structure is next to it. However, fabricated structure will produce stress concentration phenomenon in its nodes. So, strengthening the connection parts has to be considered. Meanwhile, in these three structures, beam end is where maximum stress can be produced, which illustrates that the damage of concrete is first occurred at beam end and meets the needs of “strong column and weak beam” seismic design standards.

Introduction
Nowadays, urbanization in our country is being vigorously promoted. While the rapid development of prefabricated concrete structure and the realization of housing industrialization is the key to quickly complete the urbanization[1]. Prefabricated steel cage concrete structure system involved in this paper, is a new system of prefabricated concrete structures in recent years. Compared with the traditional reinforced concrete structure, the innovation of the steel cage structure is to replace the longitudinal bars in the original structure with the vertical bars in the steel cage and replace stirrups in the original structure with the horizontal bars. Due to the structural advantages of the steel cage, it can better constrain the core concrete after pouring concrete, and improve the brittleness of the structure, so as to increase the integrity of the structure, so that it has better strength and durability. This paper will compare and analyze whether the new type of structural system meets the seismic standard and the bearing capacity of concrete structure.

Introduction of Test Plan
Component Parameter Design
According to the relevant specification[2], the frame prototype was designed as a six-story two-bay frame. According to the actual conditions of the laboratory of Anhui Jianzhu University, scale frame prototype can be reduced to 1:2 and simplified it into two-story two-bay. The Layer height is 1.5m. The span is 2.0m. The column section size is 200mm *200mm and the beam section size is 125mm*200mm. Concrete strength grade uses C30, and longitudinal rebar adopts HRB400. Stirrup employsHPB300 and steel plate uses Q345. First, according to PKPM to calculate the reinforcement of frame prototype, then on the basis of the similarity theory, it calculates the reinforcement of frame model by the reinforcement of frame prototype[3].

Loading Design[4]
In this paper, the seismic loads are simulated by applying low cyclic loading to the three groups of the test model. According to the principle of equal axial compression ratio, the axial compression ratio of middle column is 0.48 and the border column is 0.36. Calculated by the relevant formula, the
load applied to the middle column is 274.56KN and the border column is 205.92KN. The loading system adopts load - displacement control method. Multi stage loading method is adopted before the specimen yielded. The load order is: ±5KN, ±10KN, ±15KN……and so on. After each step, you should observe the hysteresis curve. When there is a clear inflection point, displacement control method instated. Recording the maximum displacement of the frame when it reaches the yield point. The base of the displacement control method is the yield displacement. Such as 1.5 times, 2.0 times, 2.5 times, 3.0 times, 3.5 times, 4.0 times...... and so on. Each level is cycled three times. As the displacement loading increases step by step, when measured hysteresis curve reaches 85% of the peak load, it is considered that the component has been destroyed and the simulation stopped. As is shown in figure 1.

![Figure 1. Load control schematic.](image)

The Modeling of the Finite Element Software ANSYS\[5\][6]

**Modeling Steps**

This paper is based on the finite element software ANSYS. Firstly, the finite element model is constructed according to the designed component parameters, which is to complete the relevant parameters of the material properties, strengthening principle and the unique characteristics of the material itself. On the basis of this, it simplifies the designed structure model. Then it gives a grid division based on the resulting structure model. Secondly, it loads the structure model according to the loading scheme and solves the simulation results. In this section, we will first define the boundary conditions of the specimen, and then according to the load-displacement loading system of the loading scheme, specific loads are applied first, then displacement loads are applied after the structure yields. Finally, the simulation results are investigated.

**Constitutive Relations of Materials**

**Constitutive Relation of Concrete.** The uniaxial tensile failure criterion of concrete is used in this model. The constitutive relation of concrete in compression adopts Kent-Park calculation formula model. This model contains two curves -- ascending and descending segments.

**Constitutive Relation of Rebar and Steel Plate.** In this paper, the criterion of rebar and steel plate yield is based on Van-Merseys’s yield criterion. Specifically, an equivalent load is applied to the material, and material appears plastic deformation immediately when the yield condition is reached. Rebars in the model have longitudinal rebars and stirrups, and both of them adopt the elastic hardening model. The steel plate employs multilinear isotropic hardening model.

**Simulation Results and Analysis**

**Hysteresis Curve**

Simulation of the hysteresis curve of the three groups of frame is shown as in figure 2 to 4. It can be concluded from the figures, the hysteresis curves of the three groups are relatively full. The energy-consuming capacity and seismic performance are good.
Further analysis can be seen by comparison [7]. During the loading process, when in the vicinity of the yield load, comparison of fabricated steel cage frame and all-cast-in-place steel cage frame. The occurrence distinct inflection point, the former should be smaller. This is due to the use of the horizontal slats of the fabricated structure in the node, the concrete restraint is reduced at the node part and the integrity is reduced too. Also, it causes the strength of the site reduced. So how to deal with the problem of strengthening the site of the node is the key to improve the seismic performance of fabricated steel cage concrete structure.

Viewed from the area surrounded by the three sets of curves, the largest one is all-cast-in-place steel cage concrete structure, in which plastic deformation ability of this structure is the strongest and the residual deformation is minimal. So the seismic performance of this structure is the best.

Viewed from the slope of the three sets of curves, the slope of the three groups frames all varies with increasing load. Compared to RC structure, the slopes of the curves of the two steel cage frames are relatively small. It shows that the steel cage frame is more excellent in stiffness degradation under load [8].

Stress Nephogram

The stress nephograms of the three groups frame are shown as in figure 5 to 10.

The stress nephograms of all-cast-in-place RC structure
The stress nephograms of fabricated steel cage concrete structure

Analysis of the three groups stress nephograms above can be seen:
The location of the maximum stress is produced at the end of the beam and column feet. Three groups of structures are all meet the "strong column weak beam" seismic design standards\cite{9}.

Compared to the traditional RC frame structure, two sets of steel cage concrete frame structures can bear more load and the all cast-in-place steel cage concrete frame can bear the biggest load.

From the point of view of the stress at the node, fabricated structure produces a much greater stress than all-cast-in-place structure\cite{10}. The stiffness of the fabricated structure at the node is less than the other parts, so a stress concentration occurs at the node location. Fabricated structure shows a good level of capacity because of its good integrity. Therefore, how to deal with the connection problem at the nodes is the problem that development fabricated structure must be solved.

Conclusion
The simulation of three different structures is carried out. From the simulation results we can see:
Compared with all-cast-in-place RC frame structure and all-cast-in-place steel cage frame structure, the latter has a greater yield strength and greater capacity. It means that steel cage structure can meet the higher seismic requirements.

Compared with fabricated steel cage frame structure and all-cast-in-place steel cage frame structure, due to the changes of stiffness at the node of the former one caused by the stress concentration phenomenon, before applying this new structure, how to deal at the nodes must be fully considered, though the integrity of fabricated structure is good.

As fabricated structure has many advantages, while fabricated steel cage structure has a better seismic performance than traditional RC structure. So, in a convenient condition of resource allocation, this structure can be considered for using.

Acknowledgments
This project supported by Anhui Provincial Science and Technology Research Project Funding through grant No.1501041133, and Anhui Provincial Universities Natural Science Research Project No. KJ2015A046.
References


