Seismic analysis of multi-story frame office building based on SAP2000

Jiuyang Li $^{1,*}$, Fangqi Li $^{1,*}$, Zhiru Cai $^{2}$

$^{1}$ Changchun Institute of Technology, Jilin Province Key Laboratory for Earthquake Resistance & Hazard Mitigation of Civil Engineering, Jilin Province Research Center of Construction Technology & Engineering For Housing in Cold Regions, Changchun, China.

$^{2}$ Jilin Tongren Architectural Design Co., Ltd

$^{*}$ Corresponding author’s e-mail: li15037157044@163.com

Abstract: Taking a multi-storey reinforced concrete frame structure office building as an example, the model analysis, response spectrum analysis and time history analysis of the whole structure are carried out by using the finite element software SAP2000, and the output results of y-pillar in different scenarios are analyzed. The results show that the overall seismic performance of the frame meets the design specifications, and the y-pillar structure meets the elastic and non yield characteristics of the large earthquake.

1. Introduction

Earthquake is a kind of natural disaster with sudden and destructive nature, which poses a very serious threat to human life and property security. A sudden earthquake can make the whole city into a piece of ruins in a moment. In addition, earthquakes may also cause secondary disasters such as fire, tsunami and debris flow. China is one of the countries in the world in seismic zone. From the characteristics of earthquake distribution, China is at the intersection of the two major earthquake tectonic systems in the world [1-3]. With the rapid development of science and technology, people pay more and more attention to the safety problems brought by earthquake. In order to avoid the loss of personal property to the maximum extent, many scholars, in addition to innovation of disaster prevention technology, also carry on theoretical exploration in the seismic design of structures. Because the structure of the building itself is the key factor to determine the seismic performance, it is of great significance to choose a reasonable structure form to disperse the shock energy generated by the earthquake and slow down the delay process of earthquake disaster. This is of great significance to improve the seismic performance of the building structure. Therefore, based on the idea of the influence of building structure on seismic performance, the output results of the frame structure in different scenarios are analyzed in detail.

Based on SAP2000 finite element analysis software, this paper makes linear analysis on a multi-storey reinforced concrete frame structure office building, including modal analysis, time history analysis and response spectrum analysis. Through static elastoplastic analysis of various working conditions, the floor displacement, inter story displacement angle and natural vibration period of the whole structure are obtained, and the shear diagram under different seismic scenarios is analyzed. The
results of simulation show that the overall seismic performance of the frame meets the design specifications.

2. Model Overview

2.1. Project overview
The project in this paper is a reinforced concrete multi-storey frame office building structure, a total of 5 floors, the total height of the building is 22.5m (excluding the foundation buried depth). The seismic fortification intensity of 7 degrees is adopted. The seismic fortification category is class C, and the site belongs to class II. The design earthquake group is the first group, and the design basic seismic acceleration value is 0.1g. The whole frame structure is made of C30 concrete. The section of the second-floor beam is 1100mm×1000mm, and that of the third, fourth and fifth floor beams is 1100mm×500mm. The section size of Y-shaped column under axial compression is 1000mm×1500mm, and that of branch column is 1000mm × 1000mm. The beam column and column base of the model is rigid connection.

2.2. Load determination
The response spectrum function is defined and the seismic load is applied to the X direction, and the characteristic period is 0.35s. In the Z direction, the main load of the structure is the constant load representing the dead weight of the frame. Besides the frame deadweight, other gravity loads including personnel, equipment and so on are abstracted as live loads. The distribution of live loads is shown in Table 1. It is assumed that the wind load acts uniformly around the building, and the effect is mainly deformation along the thickness of the wall slab. The basic wind pressure value of 50 years return period in this area is taken as 0.65 kN/m², and the ground roughness is grade B, and the stress surface is drawn around the building and wind load is added.

| Table 1. Standard value of live load |
|-----------------------------------|
| Uniform live load (kN/m²) | Second floor | Three floor | Four floor | Fifth floor |
| 3.0 | 3.5 | 2.0 | 3.5 |

3. Simulation Analysis

3.1. Introduction to SAP2000
SAP2000 is the most powerful version of windows in SAP series, and it is the most integrated and efficient software for 3D static and dynamic structure analysis. SAP2000 integrates all the calculation and analysis functions of load calculation, dynamic seismic analysis, linear and nonlinear analysis, steady-state and functional spectral density analysis and static pushover analysis, and contains the latest linear, nonlinear, static and dynamic analysis technology, which is fast and reasonable and reliable. Since the birth of sap, it has been widely used in the field of structural engineering analysis.

The general steps of structure analysis and design with SAP2000 are as follows:
1) Create and modify models to define the material, geometry, load, and analysis parameters of the structure;
2) Model analysis
3) View analysis results
4) View and optimize structural design

This process is an iterative process, which may contain several cycles of the above steps, and all steps can be performed seamlessly with the SAP2000 graphical user interface.
3.2 Model establishment

The finite element analysis software SAP2000 is used to model the frame structure, and the three-dimensional model is shown in Figure 1. There are 350 nodes and 528 frame units. The length direction of the structure is x axis, the width direction is Y axis, and the vertical direction is Z axis.

![Image of three-dimensional model of multi-story frame structure SAP2000]

3.3 Modal analysis

Modal analysis, also known as modal superposition dynamic analysis, is the most commonly used and effective method in seismic analysis of linear structural systems [6,7]. And modal analysis can provide the basis of dynamic analysis for time history analysis and response spectrum analysis. Ritz vector method can avoid the error caused by high-order mode truncation and get more accurate calculation results. Therefore, Ritz vector method is used to decouple the mode of the model. The mass participation coefficient and natural vibration period of the first 12 orders of the structure are obtained by modal analysis, and the results are shown in Table 2.

![Table 2. Natural vibration period and mass participation coefficient]

| Mode number | cycle/s | U_x | U_y | U_x+U_y | sumU_x | sumU_y | R_z |
|-------------|---------|-----|-----|---------|--------|--------|-----|
| 1           | 0.933733| 0.96| 4.12E-07 | 0.96 | 0.96 | 4.12E-07 | 0.000236 |
| 2           | 0.575112| 1.81E-05 | 0.48 | 0.480018 | 0.96 | 0.48 | 0.82 |
| 3           | 0.525138| 1.98E-05 | 0.3 | 0.30002 | 0.96 | 0.78 | 0.005554 |
| 4           | 0.312576| 2.43E-05 | 0.009148 | 0.009172 | 0.96 | 0.79 | 0.003317 |
| 5           | 0.293072| 0.03109 | 6.27E-06 | 0.031096 | 0.99 | 0.79 | 2.14E-05 |
| 6           | 0.248274| 6.06E-05 | 0.00021 | 0.0000271 | 0.99 | 0.79 | 0.01755 |
| 7           | 0.23155 | 3.75E-07 | 0.12 | 0.12 | 0.99 | 0.91 | 0.05707 |
| 8           | 0.223629| 2.67E-05 | 4.87E-05 | 7.54E-05 | 0.99 | 0.91 | 7.58E-05 |
| 9           | 0.200455| 3.87E-09 | 0.003809 | 0.003809 | 0.99 | 0.92 | 0.005703 |
| 10          | 0.192939| 1.36E-07 | 0.003021 | 0.003021 | 0.99 | 0.92 | 0.000333 |
| 11          | 0.192545| 6.28E-07 | 0.000237 | 0.000238 | 0.99 | 0.92 | 0.005493 |
| 12          | 0.182408| 1.3E-07 | 0.01379 | 0.01379 | 0.99 | 0.93 | 0.00512 |

It can be concluded from Table 2 that the cumulative values of the mass participation coefficients of each mode in X and Y directions of the model are greater than 0.9, which meets the requirements of the lower limit of cumulative mass required by the seismic code for modal analysis interception. By comparing the values of U_x + U_y and R_z, it is found that in the first period, there is U_x + U_y > R_z, and
\( U_X > U_Y \), which indicates that the first mode of the model is mainly translation in X direction. The second period \( U_X + U_Y < R_Z \) indicates that the structure is mainly torsional. In addition, it can be seen from the table that the vibration period of the structure is short. In order to ensure that the predominant period of the site can be avoided in all kinds of seismic scenarios, it is necessary to add isolation bearings.

3.4 Time history analysis of structures under different seismic waves

According to the requirements of Chinese code, when using time history analysis method, no less than two groups of natural waves should be selected according to the site type and earthquake grouping. Therefore, El-Centro wave and Yermo wave, which are widely used in seismic design and experimental research, are selected in this paper. Table 3 shows the basic parameters of seismic wave, in which the peak acceleration is the actual peak acceleration of the selected seismic wave, and the waveform is shown in Figure 2. In the time history analysis, it is necessary to adjust the peak acceleration. According to the checking calculation requirements, the peak acceleration is adjusted to 80 cm/s², and the minimum duration in the analysis and calculation is 30 s.

| Schedule name | Acquisition interval /s | Peak acceleration | Duration /s |
|---------------|--------------------------|-------------------|-------------|
| EL-Centro     | 0.02                     | 259.776           | 39.21       |
| YERMO         | 0.02                     | 239.8             | 42.23       |

(a) Adjusted El-Centro wave acceleration time history curve

(b) Time history curve of Yermo wave acceleration after adjustment

Figure 2. Seismic waveform

| floor | Horizontal displacement / mm | Interlayer displacement angle / rad | Horizontal displacement / mm | Interlayer displacement angle / rad |
|-------|------------------------------|------------------------------------|------------------------------|------------------------------------|
| 2     | 6.02                         | 0.00035                            | 5.05                         | 0.00026                            |
| 3     | 8.47                         | 0.00062                            | 7.09                         | 0.00034                            |
| 4     | 9.98                         | 0.00021                            | 8.34                         | 0.0004                             |
| 5     | 10.79                        | 0.00013                            | 9.01                         | 0.00047                            |
As shown in Table 4, no matter the horizontal displacement of the floor or the inter story displacement angle, the model structure meets the seismic code, and the inter story displacement angle is less than 1/550, indicating that the structure can maintain the elastic stage work.

3.5. Analysis of the results of y-pillar

The El Centro seismic wave is loaded into the X direction of the model. Because the model structure is symmetrically distributed in the y-axis direction, the output results of the three Y-shaped columns distributed on the model side are analyzed.

![Figure 3. Shear diagram of axial compression column](image)

![Figure 4. A-column shear diagram](image)

**Table 5. Maximum shear force of axially compressed column**

|          | A-axis column | B-axis column | C-axis column |
|----------|---------------|---------------|---------------|
| Shear V2/N | -126692.19    | 24202.62      | 103646.48     |

It can be seen from Figure 3 that the shear force of the three axial compression columns is evenly distributed in the X direction, and the direction of shear force of column A is the negative half axis of X, while the direction of shear force of column B and C is opposite. Table 5 shows the specific shear force values of three axially compressed columns, from which it can be seen that although the sizes of the three columns are different, the resultant shear force of B and C columns is roughly equal to that of a column.
Table 6. Shear value of branch column

| A-branch column | B-branch column | C-branch column |
|-----------------|-----------------|-----------------|
| Shear V2/N      | -143358.8       | -170957.4       | -56589.1       | -345109.4       | -76365.7       | -210865.7       |

It can be seen from table 5 and table 6 that no matter the axial compression column or the branch column, the magnitude of the shear force is within the elastic range, and although the two branch columns of the Y-shaped column bear slightly different shear force, the whole structure is symmetrically distributed, and the shear force of the Y-shaped columns on both sides is in a balanced state.

4. Conclusions

Based on SAP2000 simulation platform, this paper analyzes the modal and time history of a reinforced concrete multi-storey frame office building. The results show that without any isolation measures, the vibration period of the structure is small, and the predominant period of the site may not be avoided in some scenarios, which leads to the occurrence of structural resonance. In addition, whether it is translation or torsion, the deformation degree of the structure is not large. Under the action of different seismic waves, the interlayer displacement angle of each floor is within the regulation range. In addition to the overall analysis of the structure, the local characteristics of the Y-shaped strut are also analyzed. The results show that both the whole structure and the Y-shaped strut can meet the requirements of the code under the predetermined seismic intensity scenario.

References

[1] GB 50352-2005. (2005) General principles for design of civil buildings. Beijing: China Construction Industry Press, Beijing.
[2] Laogan B T, Elnashai A S. (1999) Structural performance and economics of tall high strength RC buildings in seismic regions. Structural Design of Tall Buildings, 8(3): 171-204.
[3] GB 50011-2010. (2010) Code for seismic design of buildings. China Construction Industry Press, Beijing.
[4] Newmark N M, Rosenblueth E, Pao Y H. (1971) Fundamentals of Earthquake Engineering. Fundamentals of earthquake engineering. Prentice-Hall, London.
[5] Jiang Rui. (2004) application of SAP2000 in static elastoplastic analysis. Journal of Zhengzhou University, (4).
[6] Saidi M, Sozen M A. (1981) Simple nonlinear analysis of R/C structures. Journal of Structural Division, ASCE, 107(5): 937-952.
[7] Ahmed Ghobarah. (2001) Performance-Based Design in Earthquake Engineering: State of Development. Engineering Structures, 23(8): 878-884.