

Numerical Simulation Analysis of Seismic Performance for Framework Structure Based on Rayleigh Damping Algorithm

Zixia Kang^{1, #}, Weida Zhou^{2, #}, Luobu Bianba^{3, *, #}

¹ School of Civil Engineering, Southeast University, Nanjing, China, 211189

² College of Civil Engineering, Hefei University of Technology, Hefei, China, 230009

³ College of Civil Engineering, Central South University, Changsha, China, 410083

* Corresponding author: 18008902090@163.com

#These authors contributed equally.

Abstract. Seismic analysis of engineering structures can effectively ensure social and public safety, and in engineering practice. The elastic-plastic time history analysis of reinforced concrete frame structures is a more accurate analysis method for evaluating the seismic performance of the structure. According to relevant regulations, the seismic fortification intensity of a certain frame structure is 6 degrees. To analyze its seismic performance under earthquake action, this paper uses ABAQUS numerical simulation software combined with Ruili damping algorithm to establish a seismic model of the frame structure. The research results indicate that: (1) By calculating the interlayer displacement angle, the interlayer displacement angle of each layer meets the requirements of the specification, and the structure has good seismic performance. (2) The peak acceleration difference between the top layers of the structure decreases first and then increases from the bottom to the top. (3) There is a coupling relationship between structural deformation and the acceleration of the structure itself, and a coupling relationship between interlayer deformation and the peak stress at the top of the interlayer.

Keywords: Seismic analysis, ABAQUS, Rayleigh Damping, Seismic model of a frame structure.

1. Introduction

With the accelerated pace of urban development and the gradual increase in urban population, the issue of land resource scarcity in the process of urbanization construction in China has become increasingly severe, with a growing number of urban buildings. During the construction of modern architectural projects, due to the inherent characteristics of construction projects and the relatively high residential density, urban buildings impose higher requirements on seismic design levels.

Research on seismic performance of architectural projects, both domestically and internationally, has a long history. Wu et al. [1] studied the Guazhou Pump Station project, established a three-dimensional finite element model, simulated seismic dynamic response using the mode-type dynamic response spectrum method based on the "Seismic Design Code for Hydraulic Structures," and conducted numerical simulation analysis using ABAQUS software. By studying and analyzing the deformation and stress of the pump station, a seismic safety evaluation was conducted for the Guazhou Pump Station structure, providing valuable reference for seismic safety analysis of similar projects. Guo et al. [2] studied the South Cai Regulating Gate in Shunyi District, Beijing, conducted a finite element re-evaluation analysis of the overall seismic safety of the gate chamber structure using ABAQUS three-dimensional finite element analysis software and the mode-type decomposition response spectrum method. The results showed that the structural anti-sliding stability safety factor of the gate chamber exceeded the requirements of the water gate design code, meeting safety requirements. Zhang et al. [3] simulated and analyzed the seismic performance of an assembled integral concrete frame structure based on the general finite element software ABAQUS using multiscale modeling. The results indicated that multiscale modeling could effectively improve computational accuracy and reduce costs, well simulating the failure characteristics of the assembled integral concrete frame structure and its overall seismic performance. Dai [4] conducted a study on

the seismic performance of polyvinyl alcohol fiber-cement composite (PVA-ECC) columns. Using ABAQUS as the main research tool and combining the measured data of two specimens, an extended analysis was performed with variations in parameters such as hoop spacing, hoop diameter, and longitudinal diameter for 13 specimens, exploring the impact of various influencing parameters on the seismic performance of PVA-ECC column. Liang [5] investigated the seismic performance of concrete cylinders under pre-stressed CFRP reinforcement. Horizontal low-cycle reciprocating load tests were conducted on four specimens, and differences in failure modes, displacement ductility, and energy dissipation capacity were analyzed. Numerical simulations of different CFRP heights and different layers of reinforcement were performed using Abaqus. Zhang et al. [6] deepened the study on the seismic performance of reinforced concrete hollow shear wall systems. Three different opening rates of reinforced concrete hollow shear walls and one solid shear wall model were established using ABAQUS finite element software. Cheng et al. [7] studied the applicability of solid elements and fiber beam elements in ABAQUS software for numerical simulation of the hysteretic performance of (RC) columns under different failure modes. Nine reinforced concrete rectangular section columns' pseudo-static test data were collected from the U.S. PEER database, where the columns experienced different modes of failure such as flexural, flexural-shear, or shear. Huang et al. [8] used ABAQUS finite element software to numerically simulate the six existing low-cycle combination node tests under repetitive loads. The results showed that the adopted finite element model provided good technical support for the practical application of the square middle hollow layer steel tube concrete column-steel beam node. Wang [9] used ABAQUS finite element software to establish a three-dimensional refined finite element model based on a practical engineering case. The results showed that seismic response was significant when the comprehensive pipeline crossed the boundary of the soil layer, and seismic design should be strengthened at this location. Huang [10] studied the seismic performance and ductility of beam-column joints with flange openings. Using dynamic finite element software ABAQUS, a finite element model of the beam-column joint was established, analyzing the deformation of the joint under different loads and dynamically comparing it with the joint without a flange opening. The results showed that the beam-column joint with a flange opening exhibited good seismic performance and energy dissipation capacity, and opening the flange at the joint could alleviate stress concentration issues, contributing to the extension of the joint's fatigue life. Therefore, using ABAQUS for numerical simulation analysis has become quite common and accurate in engineering; ABAQUS can be used to analyze the overall characteristics and local performance of structures; It provides strong support in the structural analysis of composite materials; Horizontal low cycle reciprocating load tests can be used to simulate seismic effects and analyze the performance of specimens in terms of failure morphology, displacement ductility, stress distribution, and other aspects.

To ensure the safety of the frame structure, it is imperative to conduct seismic performance research on the structure. In the above case studies, the use of ABAQUS software in the field of structural seismic analysis has matured. Given this, this paper utilizes ABAQUS software combined with the Rayleigh damping algorithm to establish a seismic calculation model for framework structures. This aims to analyze the seismic performance of the structure under a seismic fortification intensity of 6 degrees. This numerical simulation method provides a reference for similar engineering projects.

2. Model Establishment

2.1. Model Establishment:

A seismic model of the frame structure is established based on the ABAQUS software platform (Figure 1). The building has a height of 4.5m for the first floor and 3.6m for the second to sixth floors. In the length direction, there are a total of 5 frames with a spacing of 6m between each frame, covering a total length of 24m. In the width direction, there are 4 frames with a spacing of 6m between each frame, covering a total width of 18m. The concrete grade is C30, with longitudinal reinforcement using

HRB400 steel bars with a diameter of 25mm, and stirrups using HRB335 steel bars with a diameter of 8mm (Table 1 and Table 2).

Table 1. Concrete type and its parameters

	Model	Elastic Modulus	f_c	f_t
Concrete	C30	30GPa	30MPa	1.43MPa

Table 2. Steel reinforcement model and its parameters

	Model	Elastic Modulus	f_y	$f_{y'}$
Steel	HRB335	200GPa	335MPa	335MPa
	HRB400	200GPa	400MPa	400MPa

Structured meshing is employed, dividing a total of 36,839 rock mesh elements. The mesh type is C3D8R, and the model's bottom is constrained with fixed constraints (Figure 2). In the interaction module, an embedded region is used to tightly bound the reinforcement and concrete together to share forces.

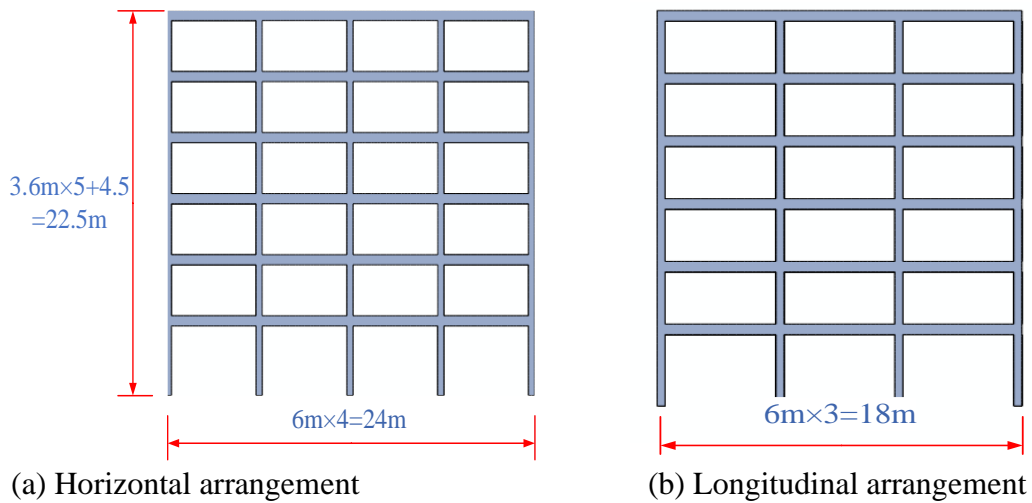


Figure 1. Model Dimensions

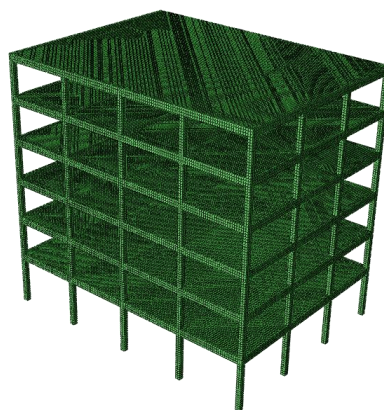


Figure 2. Mesh Model

2.2. Seismic Wave Input

In seismic analysis, Rayleigh damping is commonly employed for simulation calculations [11].

Rayleigh Damping:

$$[C] = \alpha[M] + \beta[K] \quad (1)$$

In the equation, α represents the mass damping coefficient, β represents the stiffness damping coefficient, $[M]$ is the mass matrix, and $[K]$ is the stiffness matrix.

Calculation formula for α :

$$\alpha = \frac{2\omega_i\omega_j(\xi_i\omega_j - \xi_j\omega_i)}{\omega_j^2 - \omega_i^2} \quad (2)$$

In the equation, ω_i represents the i-th natural frequency of the structure, ξ_i represents the damping ratio of the structure, typically 2% for concrete and 5% for steel.

Calculation formula for β :

$$\beta = \frac{2(\xi_i\omega_j - \xi_j\omega_i)}{\omega_j^2 - \omega_i^2} \quad (3)$$

Simplified Calculation:

$$\alpha = \frac{4\pi f_i f_j \xi}{f_i + f_j} \quad (4)$$

$$\beta = \frac{\xi}{\pi(f_i + f_j)} \quad (5)$$

In the equation, f_i represents the i-th natural frequency of the structure.

In this simulation, the Rayleigh damping parameters are calculated using the first two frequencies [12], and the results are shown in Table 3:

Table 3. Calculate the result parameter

	f_1	f_2	ξ	α	β
CONCRETE	0.63611	0.64375	0.02	0.0804	0.005
REBAR			0.05	0.201	0.0124

This paper uses artificial waves[13], and the peak acceleration values of the seismic acceleration are adjusted according to regulations to match the maximum value of the seismic acceleration time history used in the time history analysis (cm/s^2).

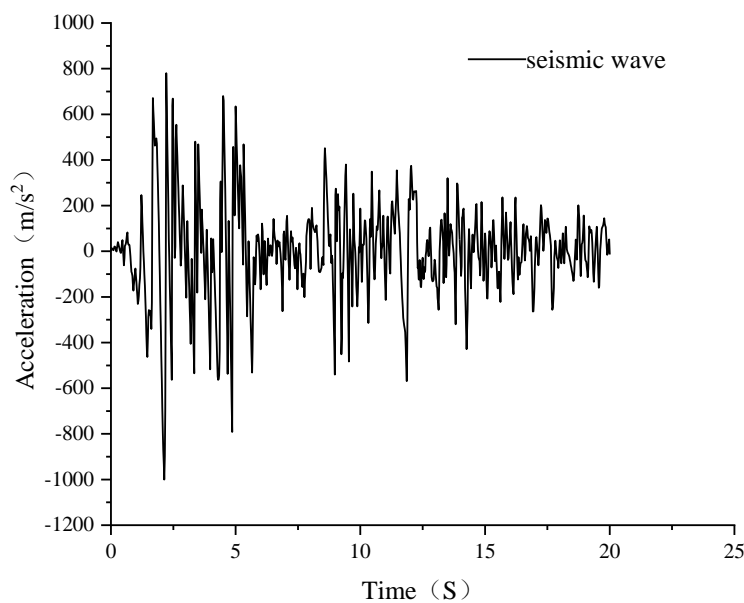


Figure 3. Raw seismic acceleration time history curve

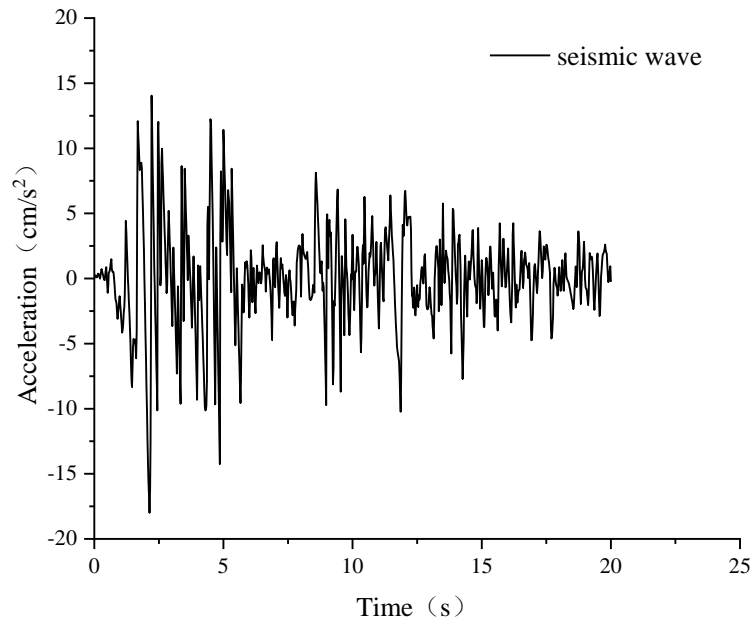


Figure 4. Adjusted seismic acceleration time history curve

According to the specifications, the peak acceleration value of seismic acceleration under a frequently encountered seismic intensity of level 6 is set at 18 cm/s².

3. Results Analysis:

3.1. Time History Analysis

Analysis of the output results of the finite element model reveals the inter-story displacement angles for each floor. By comparing these angles with the specified displacement angle limit of 1/550 according to the code, it can be determined whether the model meets the requirements. Through comparison, it is found that under the input seismic wave, the maximum inter-story displacement angle is 1/1062, far less than the displacement angle limit. Therefore, the inter-story displacement angles of the structure meet the corresponding code requirements, indicating that the model established within the elastic range is reasonable (Table 4) [12-15].

Table 4. Inter-story Displacement Angles of the Frame Model under Seismic Waves

Floors	Inter-story Displacement	Inter-story Displacement Angle	Displacement Angle Limit	Whether Satisfied
6	0.58	1/6207	1/550	Satisfied
5	0.99	1/3636	1/550	Satisfied
4	1.37	1/2624	1/550	Satisfied
3	1.65	1/2183	1/550	Satisfied
2	1.92	1/1876	1/550	Satisfied
1	3.39	1/1062	1/550	Satisfied

3.2. Structural Acceleration-Stress Response Analysis

The acceleration time history curves for structures on the 1st to 6th floors (F1 to F6) are separately extracted (Table 5, Figure 5). Under the same seismic action, the time at which the peak structural acceleration occurs is the same for different inter-story top structure accelerations. Additionally, the acceleration peak values of the structure increase with the height of the floor, with the maximum floor acceleration peak value being 471.54 mm/s². When comparing the differences in inter-story accelerations, it is observed that the largest difference in acceleration peak values occurs between the 4th and 5th floors, with an inter-story acceleration peak difference of 63.85 mm/s². As the floor height

increases from the 1st to the 4th floor, the difference in inter-story acceleration peak values gradually decreases. However, between the 4th and 6th floors, the variation in inter-story acceleration peak value differences is not significant [12] [14].

Table 5. Analysis of Inter-story Top Structure Acceleration Peak Values

Table: Analysis of Inter-story Top Structure Acceleration Peak Values		
Floors	Acceleration Peak Value /(mm/s ²)	Inter-story Acceleration Peak Difference/(mm/s ²)
6	470.54	52.73
5	417.81	63.85
4	353.96	26.75
3	327.21	50.82
2	276.39	60.89
1	215.50	/

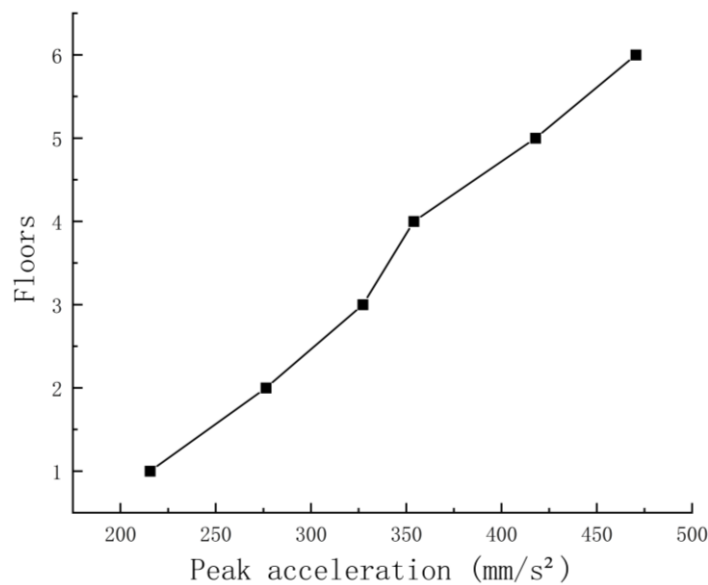


Figure 5. Peak acceleration changes along the peak floor acceleration

3.3. Structural Displacement-Stress Response Analysis

The displacement time history curves and stress time history curves for structures on the 1st to 6th floors (F1 to F6) are separately extracted (Table 6, Figure 6). Under the same seismic action, the time at which the peak displacement and the peak stress occur between different floors is the same. For structures on the 1st to 5th floors, the stress peak values gradually decrease with the increase in floor height. However, for the structure on the 6th floor, the stress peak value increases compared to the 5th floor. This suggests that the top of the structure lacks constraint conditions, leading to an increase in stress peak values. The maximum value of the inter-story top stress peak value is 5.01 Pa. It can be concluded that during construction, it is necessary to strengthen the control of the inter-story top position of the sixth floor. When comparing the differences in inter-story stress values, it is observed that for structures on the 1st to 5th floors, the inter-story top stress peak values gradually decrease with the increase in floor height. However, for the structure on the 6th floor, the inter-story top stress peak value increases compared to the 5th floor [13].

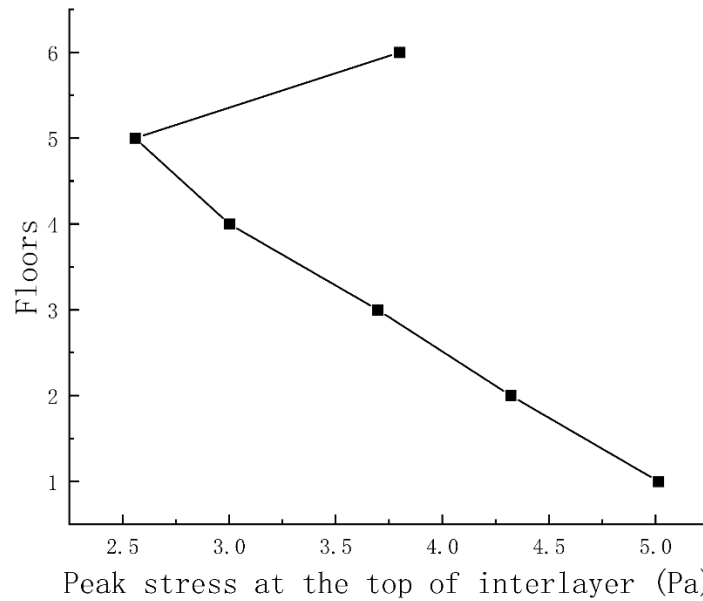


Figure 6. Peak stress variation at the top of the interlayer along the floor

Table 6. A analysis of Inter-story Top Structure Acceleration Peak Values

Floors	Stress Peak Values /Pa	Inter-story Stress Peak Difference /Pa
6	3.800	1.242
5	2.558	-0.443
4	3.001	-0.696
3	3.697	-0.624
2	4.321	-0.693
1	5.014	/

4. Conclusion

Through finite element modeling and analysis of a multi-story, multi-span reinforced concrete frame structure under seismic waves, the seismic responses under different seismic waves are obtained. The inter-story displacement angles are calculated and compared with the specified values, revealing that the structure meets the requirements.

(1) The difference in top inter-story acceleration peak values is smallest for floors 3-4, indicating a turning point in the variation pattern of inter-story top acceleration peak differences. Acceleration changes should be monitored for structural components. The maximum stress value occurs at the 6th floor, reaching 5.01 Pa.

(2) The variation in acceleration peak values increases with the height of the structure. Comparing inter-story acceleration differences reveals a coupling phenomenon between structural deformation and structural acceleration.

(3) Combined with displacement time history curves, it is observed that there is a coupling relationship between inter-story deformation and inter-story top stress peak values. Different inter-story constraint conditions affect the inter-story top stress. Further research is needed to understand the coupling patterns between inter-story deformation and inter-story top stress peak values.

References

- [1] Wu Hao, Li Baochun, Yuan Cong, et al. Seismic Safety Analysis of Guazhou Pump Station [J/OL]. Journal of Nantong University (Natural Science Edition), 1-11 [2023-11-11].
- [2] Dong Bowen, Li Dongqiao, Liang Jianwen, et al. Longitudinal Seismic Analysis of Cross-shaped Underground Pipe Gallery [J]. Journal of Natural Disasters, 2023, 32 (05): 90 - 99.

- [3] Zhang Wangxi, Wang Guan jie, Pang Bo, et al. Seismic Performance Analysis of Assembled Integral Concrete Frame Structure Based on Multiscale Model [J]. Journal of Chongqing University, 2023, 46 (10): 61 - 70.
- [4] Dai Tianjie. Finite Element Analysis of Seismic Performance of PVA-ECC Piers Based on ABAQUS [J]. Plateau Earthquake, 2023, 35 (02): 63 - 69.
- [5] Liang Xuhua. Analysis of Seismic Performance of Prestressed CFRP Reinforced Concrete Columns [J]. Journal of Sanming University, 2023, 40 (03): 91 - 100.
- [6] Zhang Purui, Guan Qun. Seismic Performance Analysis of Reinforced Concrete Hollow Shear Wall System [J]. Anhui Architecture, 2021, 28 (02): 67 - 68+73.
- [7] Cheng Xuebin, Ma Ying, Yuan Ziqi. Numerical Simulation Analysis of Seismic Performance of Reinforced Concrete Columns Based on ABAQUS [J]. Journal of Water Conservancy and Architecture Engineering, 2020, 1 8(06): 146 - 152.
- [8] Huang Chunxiao, Wei Zihan. Finite Element Analysis of Seismic Performance of Square Hollow Layered Steel Tube Concrete Column-Beam Joint Based on ABAQUS [J]. Jiangxi Building Materials, 2020, (07): 77 - 78+80.
- [9] Wang Taotao. Numerical Analysis of Seismic Resistance of Comprehensive Pipe Gallery Based on ABAQUS Finite Element Simulation [J]. Jiangsu Construction, 2020, (02): 96 - 99.
- [10] Huang Kang. Dynamic Analysis of Beam-Column Joint with Slit Flange Plate Based on ABAQUS [J]. Sichuan Building Materials, 2020, 46 (02): 78 - 79.
- [11] Li Zheng, Zhu Bingyin, Li Ning. Seismic Damage Analysis of Reinforced Concrete Frame Structure Based on ABAQUS [C]. In: Proceedings of the 3rd National Building Structure Technology Exchange Conference. Beijing: Editorial Department of Building Structure, 2011: 272 - 275.
- [12] Wang Weitao. Seismic Response Analysis of Reinforced Concrete Frame Structure [D]. Heilongjiang: Northeast Forestry University, 2014.
- [13] Xie Jiangmiao. Seismic Performance Study of Multi-story Multi-span Steel-Concrete Composite Beam-Steel Frame [D]. Heilongjiang: Harbin Institute of Technology, 2011.
- [14] Shao Chuandong, Ni Haohao, Zhuang Hui. Seismic Response Analysis of a Multi-story Frame Structure [J]. Low Temperature Building Technology, 2016, 38 (11): 52 - 53.
- [15] Zu Qingzhi. Damage Analysis of RC Frame Structure under Different Types of Earthquake Actions [J]. Journal of Changchun University of Engineering (Natural Science Edition), 2022, 23 (2): 43 - 50.