International Journal of GEOMATE
(Geotechnique, Construction Materials and Environment)
ISSN: 2186-2982 (P), 2186-2990 (O), Japan
Vol. 31, Issue 145, September 2026, PP. 121-130; DOI: 10.21660/2026.145.5641

RESISTANCE ASSESSMENT OF BUILDINGS VIA MODEL CONVERSION FROM ETABS/SAP2000 TO ABAQUS WITH MATLAB INTERFACE

*Xie Chengxi[1], Chung Han Lim[2]

[1]Civil Engineering Faculty of Engineering, Universiti Malaysia Sabah, Kota Kinabalu, China
[2] Civil Engineering Faculty of Engineering, Universiti Malaysia Sabah, Kota Kinabalu, Malaysia

*Corresponding Author, Received: 06 May 2026, Revised: 17 June 2026, Accepted: 19 June 2026

ABSTRACT: The increasing complexity of high-rise and large-span structures requires reliable methods for evaluating seismic performance under rare earthquake events. Conventional full-scale experimental testing is costly, time-consuming, and impractical for routine engineering applications, creating a need for efficient computational approaches. This study proposes an integrated framework for seismic resistance assessment by combining ETABS/SAP2000, ABAQUS, and a MATLAB-based interface for automated model conversion. Enhanced concrete and steel constitutive models incorporating cyclic behavior and strain-rate effects were implemented and validated against experimental data from reinforced concrete column tests. The methodology was applied to an 8-story reinforced concrete frame–shear wall structure and a large-span steel truss gymnasium subjected to rare earthquake ground motions through nonlinear time-history analysis. Validation results showed that the converted ABAQUS models accurately preserved structural dynamic characteristics, with modal period deviations below 6.6% and material parameter deviations below 3% compared with experimental observations. For the frame–shear wall structure, maximum inter-story drift ratios were 1/136 and 1/161 in the X and Y directions, respectively, remaining within code limits. Damage analysis revealed progressive deterioration concentrated in coupling beams and wall bases, while most beams and columns remained elastic, confirming a desirable strong-column–weak-beam failure mechanism. The proposed framework provides an economical and accurate alternative to physical testing, enabling efficient seismic performance evaluation, damage prediction, and anti-collapse assessment of complex structures.

Keywords: Seismic resistance, Structural strength, Earthquake, Computer-aided analysis, Experimental verification, Seismic testing

1. INTRODUCTION

Earthquakes, resulting from the sudden release of energy within the Earth's lithosphere, pose a significant and persistent threat to densely populated urban environments, as tragically evidenced by the 2008 Wenchuan Earthquake (Mw 8.0), which resulted in nearly 70,000 fatalities and the collapse of numerous buildings, highlighting critical deficiencies in seismic design and construction quality across major infrastructure and residential sectors.Consequently, enhancing the seismic performance and resilience of building structures has become a paramount global research priority, particularly for complex high-rise and long-span constructions [1,3].As building structural systems grow increasingly sophisticated in both form and function, traditional linear analytical methods have proven inadequate for capturing the complex nonlinear responses activated during severe seismic events [2,4].When subjected to rare, intense ground motions, civil structures inevitably enter an elastoplastic state, characterized by large deformations, progressive yielding of components, stiffness degradation, and localized structural damage. Principal approaches for investigating this nonlinear behavior have traditionally been physical experiments, including full-scale structural tests and shaking table validations. However, these methods are prohibitively expensive, time-consuming, and inherently limited in the number of test scenarios they can cover, making them unfeasible for routine engineering practice and extensive parametric studies. To overcome these limitations, recent research has widely embraced high-fidelity numerical simulation techniques, with the ABAQUS finite element software platform emerging as a powerful tool due to its extensive library of concrete and steel constitutive models. For instance, comprehensive ABAQUS models incorporating nonlinear soil-pile-structure interaction have been successfully applied to study high-rise building responses on soft soil, providing reliable alternatives to physical testing [5].Concurrent studies have employed ABAQUS-based user material subroutines (UMAT/VUMAT) to model complex concrete damage and plasticity mechanisms under cyclic loading, demonstrating good agreement with experimental observations [6].Additionally, the software has been effectively integrated with design packages such as ETABS and SAP2000 to facilitate the necessary model transformations for elastoplastic dynamic time-history analyses, thus bridging the gap between professional architectural design and adv [7]. Despite these developments, significant research gaps remain, particularly regarding: (1) the efficient and automated transformation of design models to ABAQUS suitable for direct use in dynamic elastoplastic analysis, (2) the accurate simulation of nonlinear dynamic behavior in fiber-based beam element models using refined uniaxial concrete and steel constitutive relationships validated against experimental data, and (3) the practical evaluation of anti-collapse reserve capacity and localization of plastic damage in large-scale structures under rare seismic events.

The primary gap addressed in this study lies in the current technical bottleneck surrounding the widespread application of elastoplastic time-history analysis in complex structural engineering projects, where computational cost, user expertise, and reliable model conversion remain substantial barriers [8]. Specifically, while ABAQUS excels in material nonlinearity modeling, its direct linkage with professional design software has remained inefficient, often limiting its use to academic investigations rather than practical engineering consultation. Furthermore, while fiber-based beam element modeling is computationally efficient, the current ABAQUS native material models often lack the sufficient fidelity to capture complex concrete hysteretic behavior under reversed cyclic loads. This study introduces novelty by: (1) an interface program for automated model conversion from ETABS and SAP2000 to ABAQUS; (2) validating a user-defined concrete uniaxial constitutive model for fiber-section beam elements, which better represents concrete damage and plastic evolution under seismic loading; and (3) integrating these developments into a practical methodological framework for conducting incremental dynamic time-history analyses under rare-intensity earthquakes, validated against case studies.

The main objectives are: (1) to achieve efficient model conversion from ETABS/SAP2000 to ABAQUS, improving analysis efficiency; (2) to verify the validity of the developed material subroutine by comparing with experimental results and improve existing material models; (3) to obtain damage conditions of key components (e.g., shear walls, frames) and evaluate seismic performance; and (4) to clarify failure modes and weak parts of large-span steel truss structures to assess anti-collapse capacity. A step-by-step approach is adopted. First, interface programs are used to convert models from ETABS/SAP2000 to ABAQUS. Second, concrete and steel constitutive models are selected and compiled based on ABAQUS secondary development, incorporating strain-rate effects. Third, an 8-story concrete frame-shear wall structure is analyzed under rare seismic action. Finally, a nonlinear time-history analysis is conducted on a gymnasium to evaluate seismic response. The focus is on assessing seismic strength through computer simulation verified, studying displacement, stress distribution, and potential failure zones.

This paper Section 2 describes the methodology in detail, including the program development technology for model conversion (ETABS/SAP2000 to ABAQUS), the enhanced constitutive models with strain-rate effects for concrete and steel, and the seismic input analysis with multi-directional evaluation. Section 3 presents the analysis and results, beginning with multi-scale coupling verification using experimental data, followed by case studies of an 8-story frame-shear wall structure, including damage analysis and stress distribution in beams and columns under different seismic waves. Section 4 concludes the paper with a summary of key findings, practical implications for seismic design.

2. RESEARCH SIGNIFICANCE

This study aims to demonstrate that through appropriate verification, computer modeling can serve as a reliable method to assess structural failure mechanisms under seismic action, identify components entering the plastic state, verify anti-collapse capacity, and adjust structural schemes during the design stage. Such an approach contributes to establishing a safer urban environment in earthquake-prone areas. Due to the complexity of elastoplastic calculations and limitations of related tools, the analysis of rare seismic responses for complex structural engineering has not been widely adopted. This paper addresses this gap by using the ABAQUS software to conduct a systematic analysis of the elastic-plastic characteristics of buildings under seismic action.

3. METHODOLOGY

In this study, ETABS and SAP2000 were used as the source platforms for structural modeling, fully leveraging their efficient parametric design capabilities. The two software were respectively applied to different structural types: ETABS was used for an eight-story reinforced concrete frame-shear wall structure, while SAP2000 was used for a large-span steel truss building. Both platforms exported the structural models in the same ACCESS database format (.mdb), which were then read by an interface program based on MATLAB. The mapping rules between ETABS and SAP2000 followed the same keyword correspondence, as shown in Table 1. Therefore, the proposed interface program is also applicable to models from either software. This case study demonstrates the application of the interface program in different structural types and different source platforms.

In this study, the term “rare earthquake” is defined according to the seismic design codes adopted. For the 8-story frame-shear wall structure (Section 4.2), the rare earthquake action follows the Chinese Code for Seismic Design of Buildings (GB 50011-2010). For a site with Seismic Intensity 8, Site Class II, and Design Group 1, the rare earthquake level corresponds to a 2% probability of exceedance in a 50-year period. The peak ground acceleration (PGA) is set to 400 gal (0.4g) with a triaxial input ratio of X:Y:Z = 1:0.85:0.65 as specified in the code. For the large-span steel truss gymnasium in Malaysia (Section 5.1), the rare earthquake is defined based on Eurocode 8 and local seismic hazard studies, also using a 2% in 50 years probability level. The specific PGA for the gymnasium follows the values recommended by the applicable Malaysian seismic hazard maps, as cited in the model setup.

3.1 Program Development Technology

The interface program acts as a "translation bridge", converting the structural model in ETABS or SAP2000 into a format readable by ABAQUS. This process begins with modeling in SAP2000, exporting it as an ACCESS database, and using MATLAB to generate Python scripts for ABAQUS. Key steps include data conversion and identification. The so-called conversion: reconstructing the geometric elements such as points, lines, and surfaces in SAP2000 into geometric entities in ABAQUS. Identification: the program needs to assign a unique label to each converted point, line, and surface so that in the subsequent steps, the materials, sections, and loads and other attributes can be accurately assigned to them. Figure 1 below shows this process.

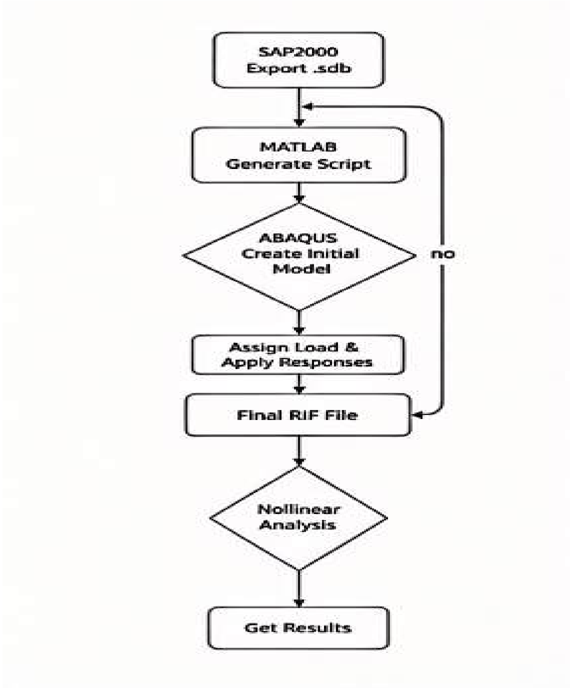


Fig.1 The Flowchart Of The Program

The conversion from SAP2000/ETABS to ABAQUS follows a systematic data-flow pipeline. First, the structural model is exported from SAP2000 as an ACCESS database (.mdb), which stores all geometric, material, section, load, and boundary information in tabular format. The MATLAB-based interface program then reads this database and maps each entity to corresponding ABAQUS keywords. Table 1 summarizes the key mapping rules.

Table.1 Mapping Of SAP2000/ETABS Entities To ABAQUS Keywords

SAP2000 MODEL	ETABS MODEL	ABAQUS KEYWORD	REMARK
"Joint Coordinates"	"Joint Coordinates"	"NODE, NODEL, ..."	Node Coordinates
	"Joint Assignments Summary"		
"Connectivity-Floor"	"Beam Connectivity Data"	"FLUENT, ELSET = ... TYP=ELSET"	Beam Element Definition
"Frame Section Assignments"	"Column Connectivity Data"		
"Connectivity-Area"	"Floor Connectivity Data"	"FLUENT, ELSET = ... TYP=CSR"	Shell Element Definition
"Area Section Assignments"	"Wall Connectivity Data"	"FLUENT, ELSET = ... TYP=CSR"	
	"Area Assignments Summary"		
"Material Properties02-Basic Mechanical Properties"	"Material Properties"	"Material, name..."	Material Definition
		"Elastic"	
		"Density"	
		"Thermal, alpha"	
"Frame Section Properties"	"Frame Section Properties"	"BEAM SECTION, SECTION = ... ELSET = ... MATERIAL = ..."	Frame Section Definition
"Frame Section Properties"	"Shell Section Properties"	"SHELL SECTION, ELSET = ... MATERIAL = ..."	Shell Section Definition
"Joint Restraint Assignments"	"Supports Restraints"	"BOUNDARY"	Boundary Conditions
"Frame Loads-Distributed"	"Frame Distributed Loads"	"MINISTRCIAL MASS, ELSET = ... INITIALS=PD LINE"	Line Loads (converted to mass)
"Joint Loads-Force"	"Joint Force Loads"	"FLUENT, ELSET = ... TYP=ELSET"	Joint Loads (converted to mass)
"Area Loads-Uniform"	"Area Uniform Loads"	"MINISTRCIAL MASS, ELSET = ... INITIALS=PD AREA"	Area Loads (converted to mass)

3.2 Enhanced Constitutive Models with Strain-Rate Effects

The concrete constitutive model is based on the Concrete02 model from OpenSEES, featuring a compressive backbone curve with parabolic ascending and linear descending segments, and a bilinear tensile model. The model accounts for stiffness degradation and damage accumulation under cyclic loading [9]. To simulate the nonlinear behavior under seismic motion more accurately, a viscoelastic-viscoplastic constitutive model (similar to the CDP model in ABAQUS) was incorporated, with strain-rate related parameters. Figure 2 shows the stress-strain curve of concrete under dynamic loading, which demonstrates the increase in peak stress and the changed softening behavior compared to static loading, indicating the effect of strain rate.

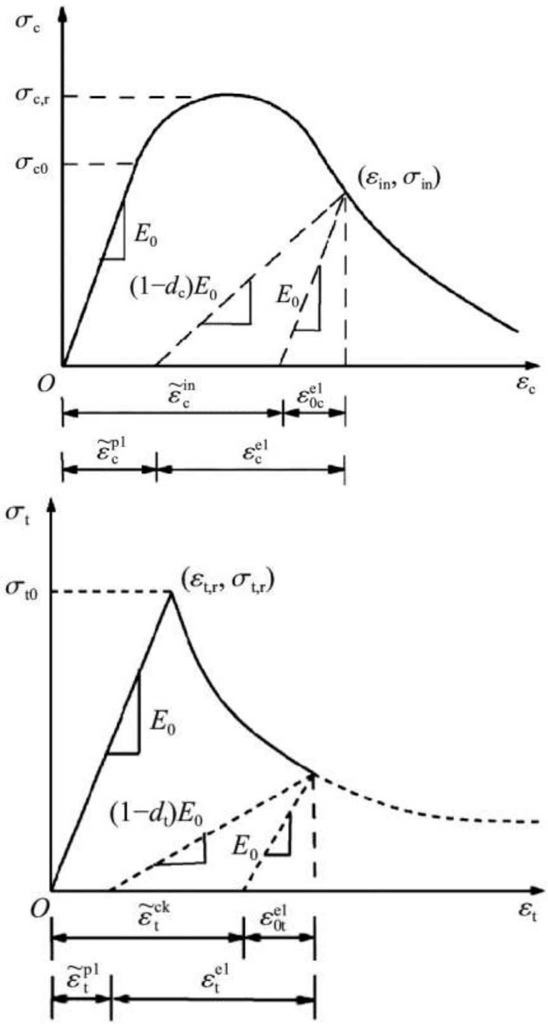
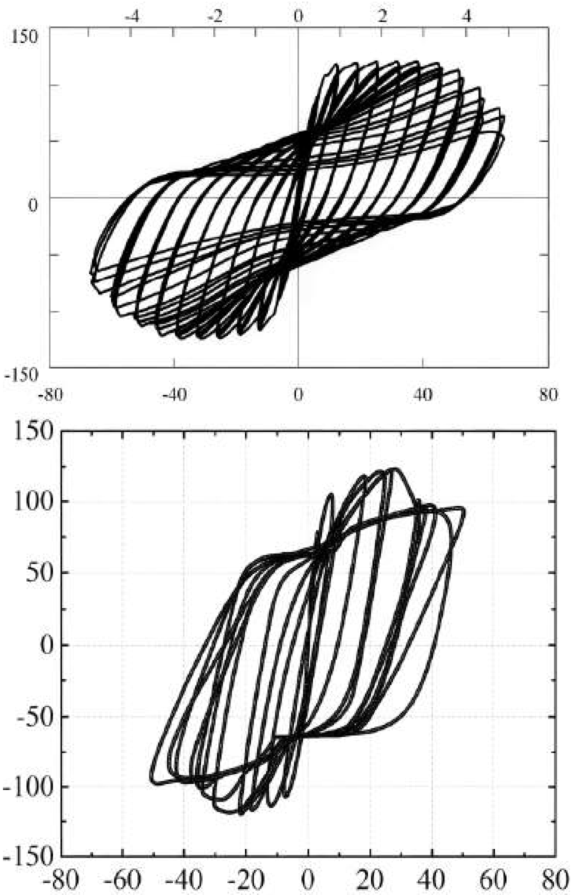


Fig.2 Concrete tensile σ-ε curve and Concrete compression σ-ε curve

Steel is modeled using a bilinear kinematic hardening model, considering the Bauschinger effect. Under different loading rates, the hysteresis curve exhibits variations in the Bauschinger effect, validating the superiority of the rate-dependent model for seismic simulations[11]. To validate the proposed steel constitutive model under cyclic loading, a numerical simulation was performed on the RC column specimen TP74 tested by Kawashima et al. (2003). The specimen was modeled using fiber-beam elements with the bilinear kinematic hardening steel model and the uniaxial concrete model described above. Figure 3 compares the simulated load-displacement hysteresis curve with the experimental result. The simulated response accurately reproduces the Bauschinger effect, the progressive stiffness degradation, and the energy dissipation capacity observed in the test. This validation demonstrates that the steel constitutive model is suitable for elastoplastic time-history analysis of reinforced concrete structures under seismic loading.

(a)



(b)

Fig. 3 Validation of the steel bilinear kinematic hardening model under cyclic loading

Fig. 3 Validation of the steel bilinear kinematic hardening model under cyclic loading: (a) experimental load‐displacement hysteresis curve of RC column specimen TP74 (Kawashima et al., 2003); (b) simulated hysteresis curve obtained using the proposed steel constitutive model implemented in ABAQUS. The close agreement between (a) and (b) confirms that the model captures the Bauschinger effect, cyclic hardening, and energy dissipation without artificial stiffness degradation.

3.3 Seismic Input Analysis with Multi-Directional Evaluation

Multiple sets of artificial seismic waves were generated based on the target response spectra adjusted from the Chinese Code for Seismic Design of Buildings (GB 50011-2010) for a rare earthquake level (2% probability of exceedance in 50 years). The peak ground acceleration (PGA) was uniformly scaled to 400 gal for all waves, and the triaxial input followed a PGA ratio of X:Y:Z = 1:0.85:0.65. For the gymnasium case (Section 5.1), the target spectrum was constructed according to Eurocode 8 for the same exceedance probability level, with site-specific PGA values taken from Malaysian seismic hazard studies[10]. Figures 4-12 present the acceleration time histories of the selected ground motions in the X, Y, and Z directions for the Taft, El-Centro, and artificial waves. The corresponding response spectra and Fourier amplitude spectra were computed and compared with the code target spectrum, as shown later in Figure 4-16.

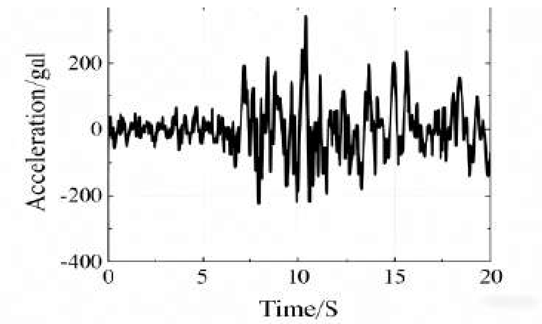


Fig. 4. TAFT (Y-direction)

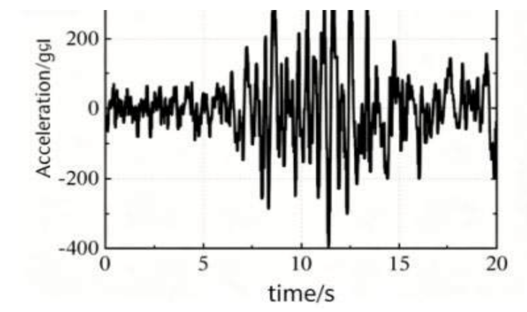


Fig. 5. TAFT (x-direction)

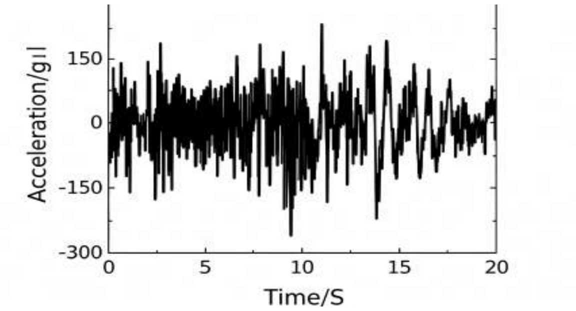


Fig. 6. TAFT (Z-direction)

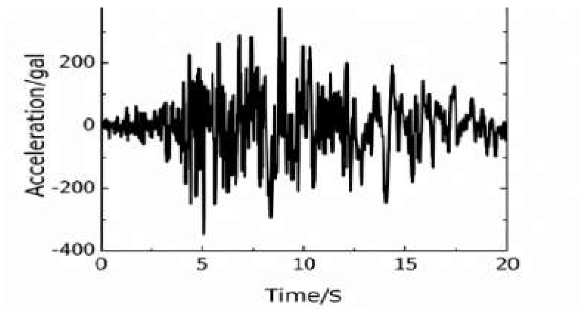


Fig. 7. EL-CENTRO (X-direction)

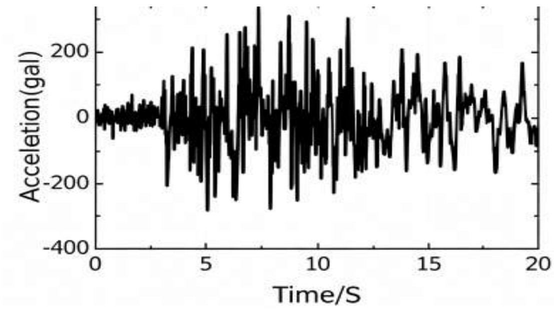


Fig. 8. EL-CENTRO (Y-direction)

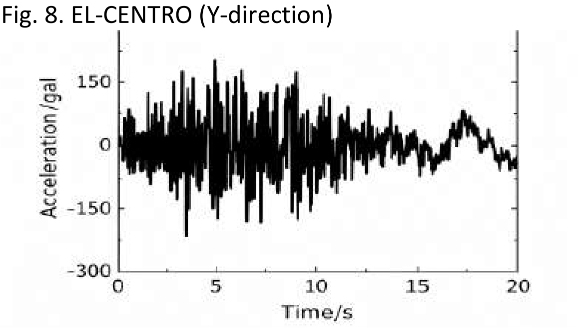


Fig. 9. EL-CENTRO (Z-direction)

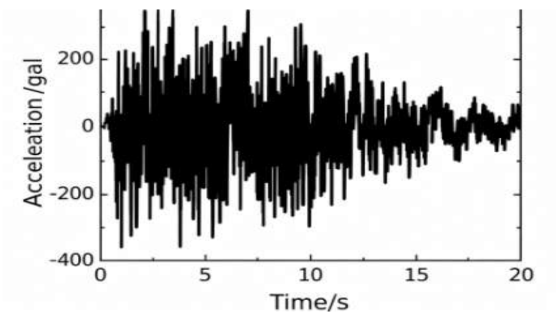


Fig. 10. Artificial wave (X-direction)

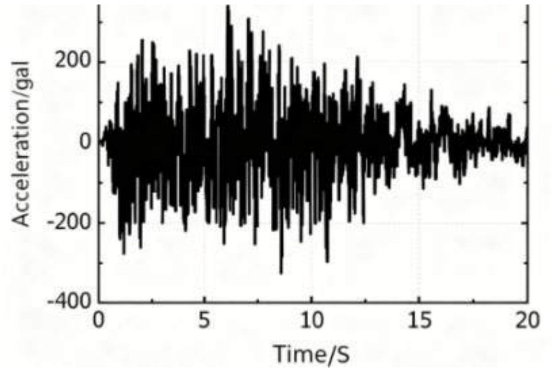


Fig. 11. Artificial wave (Y-direction)

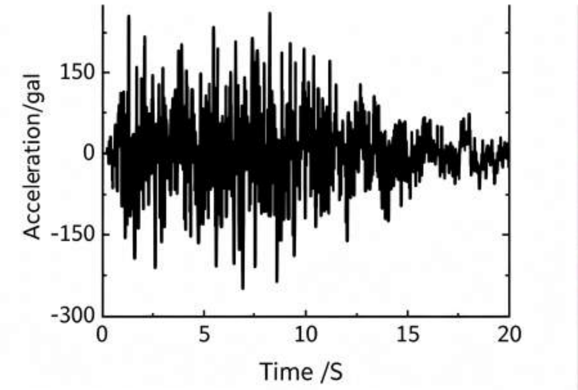


Fig. 12. Artificial wave (Z-direction)

The structural response under multi-directional inputs was analyzed, including inter-story displacement angle and base shear time-history curves, to quantify the impact of directional effects on structural damage.

4. ANALYSIS AND RESULTS

4.1 Multi-Scale Coupling for Fine Modeling

This numerical model was verified through experiments conducted by Professor Kawashima and Yashima [12]. The concrete specimen was 1750mm in height, with a base height of 400mm, and the concrete cross-section was 400mm. A load was applied 400mm from the top of the column. The reinforcement of the specimen is shown in Figure 13. The simulation results showed good agreement with the experimental load-displacement responses. As summarized in Table 2, the peak lateral load and corresponding displacement deviations between simulation and experiment were within 5% for specimens TP74 and TP77. The material parameter deviations (concrete compressive strength and steel yield strength) were below 3%. These quantitative comparisons confirm the effectiveness of the proposed model for nonlinear dynamic analysis[13].

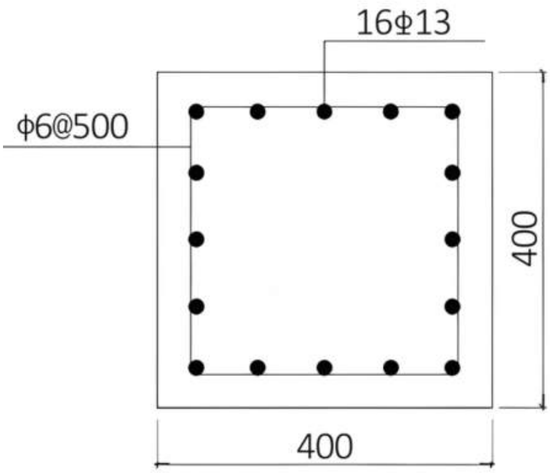


Fig. 13. Concrete Column Test Piece

Material parameter mapping was verified through comparative tables of concrete compressive strength and steel yield strength, with deviations below 3%, ensuring model reliability.

Table 2. Comparison of material parameters between experiment and simulation

Material property	Experimental (MPa)	Simulated (MPa)	Error (%)
Concrete compressive strength (fc)	29.66	29.1	-1.9
Steel yield strength (fy)	357	355	-0.6

4.2 Case Study: 8-Story Frame-Shear Wall Structure

The 8-story reinforced concrete frame-shear wall structure has a rectangular plan layout with six bays in the longitudinal direction (each 7.5 m, total 45 m) and three bays in the transverse direction (side spans 7.5 m, central corridor span 3.0 m). The first-story height is 4500 mm, and the second to eighth stories each have a height of 3600 mm, giving a total structural height of 29.7 m. The concrete strength is C40 for columns and shear walls on floors 1–4, and C30 for columns and walls on floors 5–8 as well as for all beams and slabs. Longitudinal reinforcement in beams and columns is HRB400 grade, and stirrups are HPB235 grade. Floor slabs are 100 mm thick with bidirectional distributed reinforcement ф10@150. Shear walls have a constant thickness of 200 mm. Column sections are 800×800 mm for floors 1–4 and 600×600 mm for floors 5–8. Beam sections are designated as GX (700×350 mm), GX1 (600×250 mm), and GX2 (400×250 mm) according to their layout positions. The structure is fixed at the base (all degrees of freedom restrained). The seismic design parameters follow the Chinese Code for Seismic Design of Buildings (GB 50011-2010): seismic fortification category B, safety grade II, design service life 50 years, basic seismic intensity 8 degrees, site class II, and design seismic group 1.

The view diagram of the model in ABAQUS after conversion is shown in Figure 14. The seismic input records included the Taft, El-Centro, and artificial waves. The model was transferred from ETABS to ABAQUS using the interface program, and its dynamic characteristics were verified.

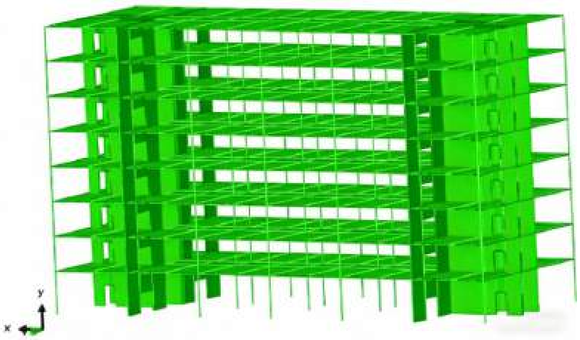


Fig. 14. Three views of the model in ABAQUS

Figure 15 shows the distribution curve of the maximum inter-story displacement angle of the structure floors under rare earthquakes. This curve reflects the lateral stiffness distribution of the structure along the height direction of the floors. In the X direction, the maximum inter-story displacement angle is 1/136, and in the Y direction, it is 1/161. According to GB 50011-2010, the allowable elastic-plastic inter-story drift ratio for reinforced concrete frame-shear wall structures under rare earthquakes is 1/100; both computed values are below this limit. In the X direction, the base shear force reaches 4541 kN, and in the Y direction, it reaches 3083 kN.

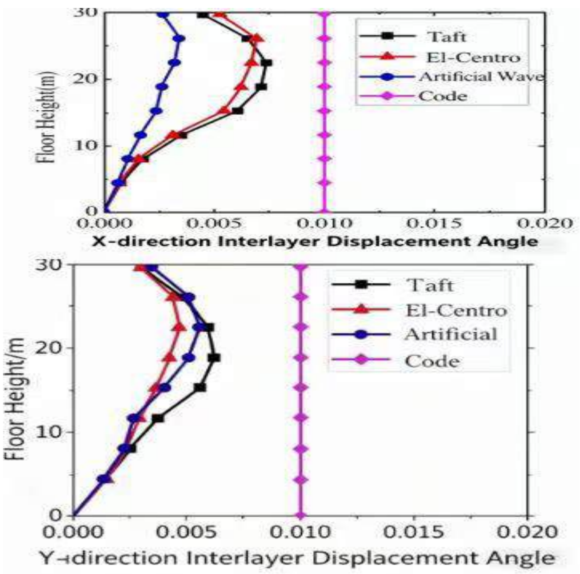


Fig. 15. The distribution curve of the maximum inter-floor displacement Angle of each floor.

4.3 Damage Analysis

As illustrated in Figure 16, progressive damage initiated at the base of the wall segments and the coupling beams under sustained seismic excitation. Among these components, the coupling beams exhibited more pronounced deterioration. By the seventh second of ground motion, the maximum damage factor reached 0.95, indicating near-total loss of load-carrying capacity. Concurrently, compressive damage became increasingly evident at the wall base and propagated upward, with a peak damage factor of approximately 0.5, as shown in Figure 17.

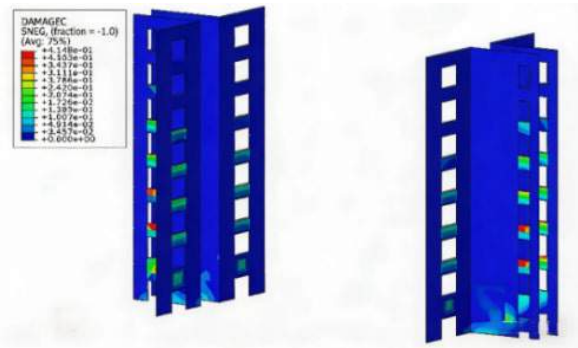


Fig. 16. Pressure diagram at 5s

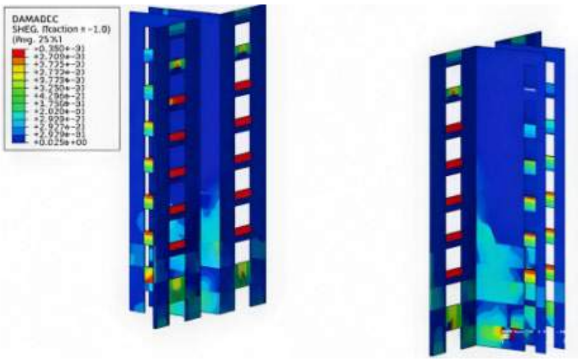


Fig. 17. Pressure diagram at 10s

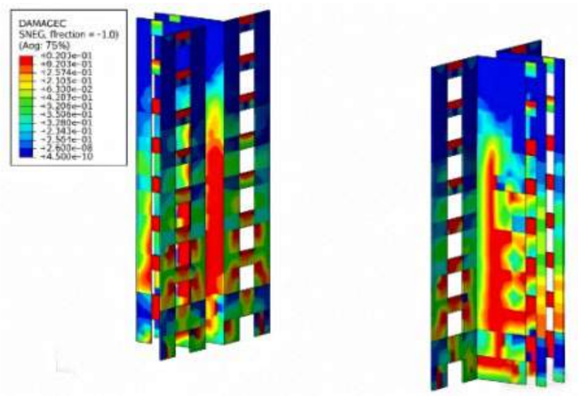


Fig. 18. Pressure diagram at 15s

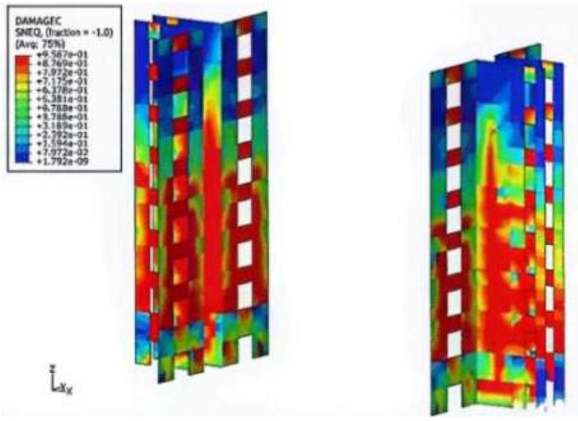


Fig. 19. Pressure diagram at 20s

At 13 seconds, nearly all coupling beams were severely compromised, while damage in the wall segments intensified and extended further—particularly in the X-direction wall segment, where the maximum damage factor attained 0.95; however, most wall regions registered damage factors ranging between 0.2 and 0.6 (Figure 18). Following 20 seconds of seismic excitation, damage in the wall segments continued to propagate, and damage factors in several localized regions increased further, with the global maximum again reaching 0.95 (Figure 19).

4.4 Damage in Beams and Columns: Stress Distribution Analysis

The stress mapping method has been extended to be applied in the structures of beams and columns. Under the action of Taft ground vibration, the stress of concrete beams and columns is shown in Figure 20. The stress distribution at the bottom of the beam and at the connection between the beam and the column reveals the plastic state. The plastic strain of the reinforcing bars in the concrete beams and columns is shown in Figure 21. From the figure, it can be seen that most of the reinforcing bars are still in the elastic working state, only in the intersection area of the beam and the column do some reinforcing bars enter the plastic state. The stress diagram at 20 seconds shows the peak stress in these areas, and the plastic strain of the reinforcing bars is as high as 3000 microns. The damage is controllable, and most of the components still remain elastic, which confirms the design concept of "strong columns and weak beams". The stress of concrete beams and columns under El-Centro ground vibration for 20 seconds is shown in Figure 22. The plastic strain of the reinforcing bars in the beams and columns is shown in Figure 23.

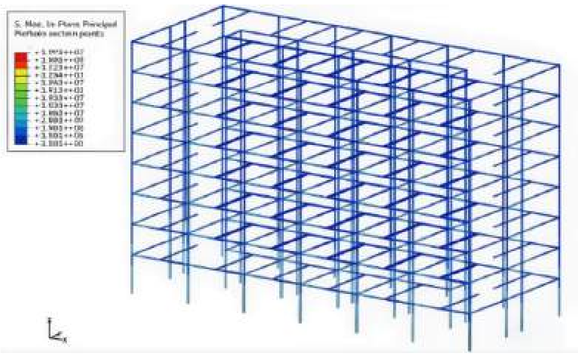


Fig. 20. Beam/column stress at 20s (TAFT)

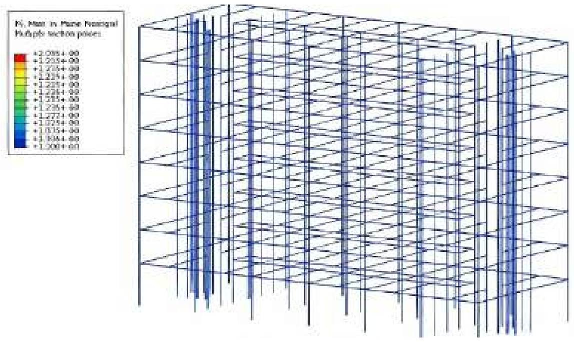


Fig. 21. Reinforcement strain at 20s (TAFT)

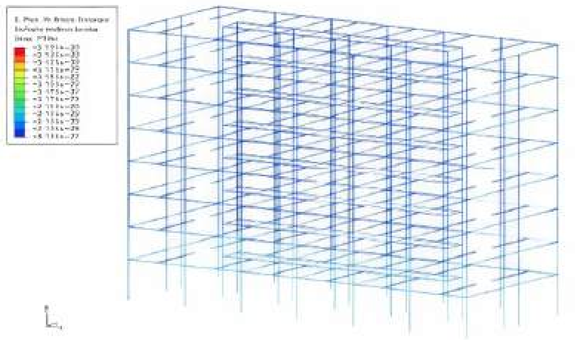


Fig. 22. Beam/column stress at 20s (EI)

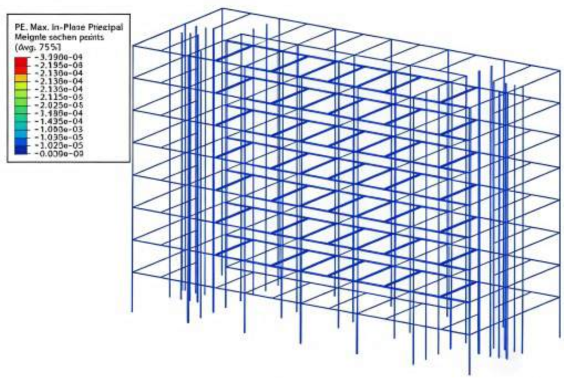


Fig. 23. Reinforcement strain at 20s (EI)

The stress of concrete beams and columns under artificial ground vibration for 20 seconds is shown in Figure 24, and the plastic strain of the reinforcing bars in the beams and columns is shown in Figure 25.

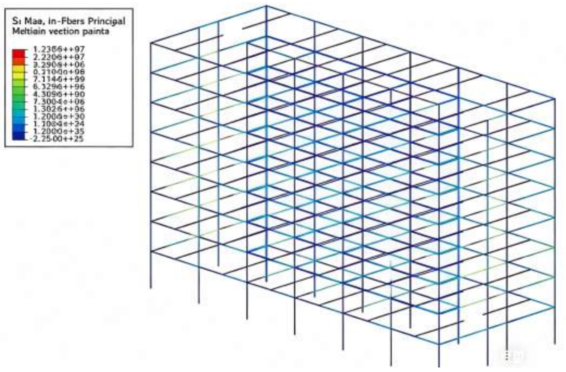


Fig. 24. Beam/column stress at 20s (Artificial)

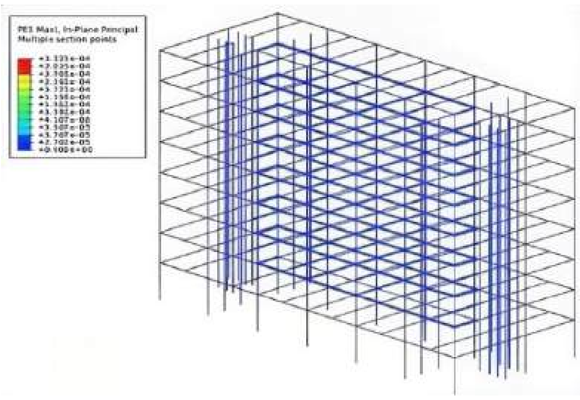


Fig. 25. Reinforcement strain at 20s (Artificial)

This case study demonstrates the efficacy of pressure diagrams in visualizing and quantifying seismic damage progression in an 8-story frame-shear wall structure. The analysis reveals a clear sequence of damage initiation, expansion, and stabilization, consistent across different seismic inputs. Key findings include the prioritized failure of connecting beams, controlled plastic development in beams and columns, and edge-dominated damage in slabs. The methods employed here, grounded in Chapter 4 of Document 1, provide a robust framework for seismic performance evaluation, emphasizing the importance of multi-directional analysis and detailed damage modeling for enhancing structural safety in earthquake-prone regions.

5. CONCLUSION

This study developed and validated a computational framework for seismic resistance evaluation of buildings by integrating ABAQUS with ETABS/SAP2000 via a MATLAB-based interface program and refined uniaxial constitutive models. The interface program automatically translates structural models from ETABS/SAP2000 to ABAQUS; modal analysis of the 8-story frame-shear wall structure showed that the first four vibration periods from ABAQUS deviated from those of ETABS by less than 5% (maximum error 6.6%), confirming that the conversion preserves dynamic characteristics without manual re-modelling. The user-defined uniaxial concrete subroutine was validated against the cyclic loading test of RC column TP74 (Kawashima et al., 2003), with key material parameters (concrete compressive strength and steel yield strength) showing deviations below 3%, and the simulated load-displacement hysteresis matching the experimental response, confirming the model's ability to capture stiffness degradation and damage accumulation under reversed cyclic loading. Under rare earthquakes (PGA = 400 gal, 2% probability of exceedance in 50 years), the 8-story RC frame-shear wall structure achieved maximum inter-story drift ratios of 1/136 in the X-direction and 1/161 in the Y-direction, both below the code limit of 1/100 (GB 50011-2010). Base shear forces reached 4541 kN (X) and 3083 kN (Y). Damage progression followed a clear sequence: coupling beams deteriorated first (damage factor up to 0.95 by 7 s), followed by wall bases (peak damage factor 0.5–0.6). Most beams and columns remained elastic except at beam-column joints (maximum bar plastic strain ≈ 3000 με), confirming a "strong column–weak beam" failure mode. For the large-span steel truss gymnasium under rare earthquake defined per Eurocode 8 (2% in 50 years), the maximum lateral displacement ratio of the concrete columns was 1/267 and the vertical deflection ratio of the steel roof was 1/298, both within code limits, demonstrating good energy redistribution capacity and no global collapse.

The proposed framework offers an economical alternative to full-scale physical testing. The automated model conversion eliminates manual errors (e.g., incorrect section orientation, missing shell thickness) and reduces modelling effort. The validated concrete subroutine improves ABAQUS native material models for fiber-beam elements, enabling more accurate simulation of concrete damage under cyclic seismic loading. The current study assumes rigid base conditions without soil-structure interaction (SSI). Future research should extend the methodology to include SSI via nonlinear soil springs, validate against real earthquake damage records (e.g., post-Wenchuan surveys), and integrate machine learning for rapid damage prediction. Such validation would further strengthen the credibility of the computational approach [14]. The proposed techniques effectively balance accuracy, cost, and practicality, advancing the state of the art in computational earthquake engineering [15].

6. REFERENCES

1. Gao C., Li Z., Li M., Zhang L., & Ma H. (2025). Dynamic time-history analysis of reinforced concrete structures considering damage in the non-principal axis direction. Magazine of Concrete Research, 77(11–12), 646–666. https://doi.org/10.1680/jmacr.24.00296
2. Zhang Y., Chen L., & Zhou D. (2022). Dynamic mechanical properties of hybrid fiber reinforced concrete. International Journal of Protective Structures, 13(3), 579–598. https://doi.org/10.1051/smdo/2023020
3. Harle S. M., Sagane S., Zanjad N., Bhadauria P. K. S., & Nistane H. P. (2024). Advancing seismic resilience: Focus on building design techniques. Structures, 66, 106432. https://doi.org/10.1016/j.istruc.2024.106432
4. Qin F. (2023). Simulation analysis of structural nonlinear seismic response. International Journal for Simulation and Multidisciplinary Design Optimization, 14, 18. https://doi.org/10.1051/smdo/2023020
5. Zhang W., Seylabi E. E., & Taciroglu E. (2019). An ABAQUS toolbox for soil-structure interaction analysis. Computers and Geotechnics, 114, 103143. https://doi.org/10.1016/j.compgeo.2019.103143
6. Hafezolghorani M., Hejazi F., Vaghei R., Jaafar M. S. B., & Karimzade K. (2017). Simplified damage plasticity model for concrete. Structural Engineering International, 27(1), 68–78. https://doi.org/10.2749/101686616X1081
7. Patel A. P. N., & Magar C. L. T. (2023). Design philosophy and seismic analysis of a multi-storey building (B+G+10) with flat slabs in seismic zone II using ETABS and SAFE software. International Journal of Scientific Research in Engineering and Management,7(10). https://doi.org/10.55041/ijsrem26198
8. Liang Q., Zhao C., & Hu J. (2020). A new elastoplastic time-history analysis method for frame structures. Advances in Civil Engineering, 2020, 1–8. https://doi.org/10.1155/2020/8818187
9. Babich D., Bastun V., & Dorodnykh T. (2019). Structural-probabilistic modeling of fatigue failure under elastic-plastic deformation. International Journal of Structural Integrity, 10(4), 484–496. https://doi.org/10.1108/IJSI-05-2018-0024
10. Miao Y. M., Ji H. W., & Shi Y. (2025). Bidirectional seismic response of assembled monolithic subway station-aboveground structure system under artificial bedrock ground motions. Underground Space, 21(2), 291–312. https://doi.org/10.1016/j.undsp.2024.08.008
11. Hosseini R., & Kumar K. (2024). Measuring the effective stress parameter using the multiphase lattice Boltzmann method and investigating the source of its hysteresis. arXiv preprint, arXiv:2501.00661. https://doi.org/10.48550/arXiv.2501.00661
12. Kawashima K., Watanabe G., & Hayakawa R. (2003). Seismic performance of RC bridge columns subjected to bilateral excitation. Journal of Structural Engineering, 129(11), 1469–1478. https://doi.org/10.1061/(ASCE)0733-9445(2003)129:11(1469)
13. Grande-Moran C., Fernandes B., Feltes D., Feltes J., Wu M., & Wells R. (2018). Case studies on dynamic load modeling. 2018 Clemson University Power Systems Conference (PSC), 1–8. https://doi.org/10.1109/PSC.2018.8664069
14. Liu R. (2023). Seismic design methods for concrete buildings. Applied and Computational Engineering, 24, 158–162. https://doi.org/10.54254/2755-2721/24/20230699
15. Suchocki C. (2023). Finite element implementation of a certain class of elasto-viscoplastic constitutive models. Acta Mechanica, 234, 4365–4389. https://doi.org/10.1007/s00707-023-03613-3