

Some Key Issues of Elasto-plastic Analysis for RC Frame Structure Retrofitted by FRP

Wei LIU^{1,a}, Ya-Ping PENG^{2,b,*}, Xin-Sheng XU^{1,c}

¹School of Civil Engineering and Architecture, University of Jinan, Jinan, China

²College of Urban Construction and Safety Engineering, Shanghai Institute of Technology, Shanghai, China

^aangle030488@126.com, ^bpengyp517@126.com, ^cxinsheng_xu@163.com

*Corresponding author

Keywords: FRP, Frame Structure, Elasto-plastic Analysis, Restoring Force Model.

Abstract. Based on summary and analysis of research results for RC frame structure retrofitted by FRP, the significance and necessity of elasto-plastic time history analysis was pointed out. Some key issues of elasto-plastic analysis using ABAQUS for RC frame structure retrofitted by FRP were discussed in the following aspects: the selection of modeling method and element type, restoring force model, and the definitions of steps and load, etc. Then several feasible suggestions were put forward.

Introduction

Fiber reinforced polymer (FRP) is widely used in the area of seismic retrofitting of reinforced concrete structure because of its light weight, high strength, corrosion resistance and convenient construction process, etc. Numerous experiments and engineering applications have been carried out for the theory and technology of concrete structure retrofitted by FRP in the member (beam, column and joint) level and has achieved remarkable results. The focus of research is gradually transferred into the whole structure.

Under strong earthquake, with some structure components entering into plastic state and structure stiffness changing, plastic internal force redistribution and large deformation of structure will take place. In order to meet seismic requirements, it's necessary to carry on elasto-plastic analysis of structure. At the same time, structure types which "must and should be carried out the calculation of elasto-plastic deformation" have been defined in article 5.5.2 of Code for Seismic Design of Buildings (GB 50010-2010) [1]. Therefore, elasto-plastic analysis plays an important role in the study of seismic performance of structures.

At present, the research of concrete structure retrofitted by FRP is mostly based on structural testing, and inevitably there are some limitations in practical application. On the other hand, existing theory of structure retrofitted by FRP is usually analyzed by static or quasi-static method, and accuracy of results is hard to be guaranteed. Under heavy load (especially the earthquake action), stress state of RC frame structure retrofitted by FRP is more complex than ordinary structure, and nonlinear properties have great influence on structures. Therefore, it's necessary to carry out dynamic elasto-plastic time history analysis for RC frame structure retrofitted by FRP.

Research Progress of Elasto-plastic Analysis on RC Frame Structure Retrofitted by FRP

Elasto-plastic Analysis Based on Tests

At present, the research on seismic behavior of concrete structures is generally carried out by mutual authentication of experiment and numerical simulation. A series of valuable results have been achieved.

The research of Reyes Garcia etc. combining the study of shaking table test and numerical simulation analysis using DRAIN-3DX [2], shows that failure mode of frame partly retrofitted by CFRP can be converted into the column hinge mechanism from the beam hinge mechanism. The overall damage degree of RC framework structure retrofitted by CFRP decreases by an average of 65% and seismic performance

of the whole structure improves significantly. Through shaking table test on the RC frame structure retrofitted by CFRP of Wang Daiyu etc. [3], it is found that the method of CFRP retrofitting has little effect on initial stiffness of structure, but the maximum displacement reaction decreases obviously and seismic performance of the overall structure retrofitted improves. Then the test results and analysis results are verified by Open Sees software, and the concept design principle for FRP seismic retrofitting nonductile RC frames is proposed.

Numerical Simulation Analysis

Research method combining structural tests and numerical simulation analysis has the advantages of strong pertinence and high credibility, but it also has the shortcomings of high cost, long periods, and most experiments are based on reduced-scale models with differences from actual structures. With the improvement of computer and successive development of all kinds of FEM software, pure studies on numerical simulation are paid more attentions.

Research results from Wang Zhenyu etc. based on Perform-3D [4] indicate that bearing capacity and ductility of structure are improved after retrofitting, and the behavior of structure under earthquake action changes little compared with the previous one. But the maximum seismic peak acceleration is greatly improved, and the weak layer may transfer after retrofitting. Time history analysis based on the ANSYS software of Song Ke etc. shows that the maximum floor displacement of frame structures after retrofitting improves significantly than ordinary structure (the top layer displacement is increased about 35%) and ductility of reinforced structure improves noticeably (as is shown in Table 1) [5]. The study of Liu Qiang shows that the method of CFRP retrofitting can effectively improve natural frequency and overall stiffness of framework, but has no significant effect on the vibration mode [6]. By comparing a series of indicators, such as the acceleration time history curve, the displacement time history curve, the maximum inter-story drift, bottom shear and development of plastic hinge, Li Peng put forward seismic retrofitting scheme of RC frame structure retrofitted by FRP based on joints [7].

Tab.1 Elasto-plastic Displacement of Structure [MPa]

Floor	The maximal story displacement [mm]		The maximal story drift [mm]		The maximal story drift ratio [rad]		Allowable interstorey drift index
	ordinary	retrofitted	ordinary	retrofitted	ordinary	retrofitted	
1	14.55	21.27	14.55	21.27	1/143	1/100	1/50
2	27.86	38.14	13.31	16.78	1/147	1/116	
3	35.43	47.88	7.56	9.74	1/256	1/200	

These researches indicate that seismic performance analysis of RC frame structure retrofitted by FRP is effective, especially the advantages of dynamic elasto-plastic analysis are more obvious. This method takes three elements of earthquake ground motion and dynamic characteristics into consideration. And it can reflect the whole process of stress and deformation of structure under earthquake action accurately. According to elasto-plastic behavior of structure after yielding, it also can evaluate seismic performance of structure under earthquake preliminarily and determine the weak part of structure. At last, modified opinions and suggestions for designing will be put forward.

At present, professional FEM software for dynamic elasto-plastic analysis mainly includes PERFORM-3D, MIDAS, ETABS and SAP2000, etc. But the definition of beam and column element in such software is achieved by plastic hinge model. It is difficult to preset reasonable length of plastic hinge because the reaction of structure after yielding is sensitive to the length of plastic hinge [8]. And it's hard for complex structures to determine failure mechanism and failure process of bars. With great advantages in elasto-plastic analysis, experts and scholars tend to use large general FEM software ABAQUS to establish models recently. It provides abundant element types and material models, especially the ability of solving, analyzing of nonlinear mechanics, and further developing, which shows strong vitality in elasto-plastic time history analysis.

Some Key Issues Worth of Attention on Elasto-plastic Analysis Based on ABAQUS

Selection of Modeling Method

An accurate and rational model plays an important role in both elastic and elasto-plastic analysis. For retrofitted concrete structure, there are usually two kinds of modeling methods based on ABAQUS: integral modeling and separated modeling, while the latter can be divided into interface element-separated modeling and displacement coordination-separated modeling.

Modeling method should be selected according to research objects and focus. For instance, there's only concrete element reinforced by **rebar Command* in the RC plane frame model using integral modeling in literature [9]. This method not only limits the number of elements, but considers the actual distribution of steel bar. However, it's not suitable with the integral analysis of large structures and explicit dynamic analysis of beam element.

Separated modeling method used in literature [10] can deal with the mechanical relationship between steel bar and concrete, where two different elements are respectively established, as well as the use of *embedded technology* to realize the degree of freedom coupling. But this method is more suitable for component level, and the slip of steel bar cannot be simulated without the aid of other measures, such as spring element or cohesive element.

Another attempt from Zhu Lili [11] is to use different elements whose nodes are shared with steel and concrete elements to ensure displacement compatibility. The concrete cover and core concrete are respectively treated as box section (without considering the effect of stirrups) and rectangular cross-section, and longitudinal reinforcement is replaced with box section according to the principles of equal area and position. Without special application condition, this method can be used for the analysis of overall framework.

Selection of Element Type

There are a wide range of elements available for solving different problems in ABAQUS. An important issue of simulation analysis is how to select the most suitable element according to different types of problems and solving requirements.

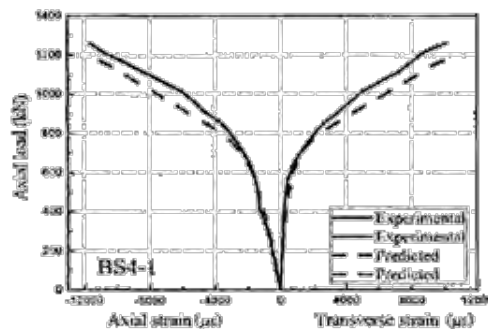


Fig. 1 Contrast between the Predicted and Experimental Axial Load-strain Curves

At present, the elements used for concrete structure analysis mainly include solid element, truss element, shell element and beam element, etc. Solid element can be used to simulate any structure. In the literature [12], simulation results of single component models are accurate, where C3D8R (3D solid element) is used for concrete and T3D2 (3D truss element) is for steel. Due to huge degree of freedom and high requirement for computer performance, these elements are seldom used for the whole structure. Instead, the bar-shell hybrid finite element method is widely used for this condition. B31 and B32 (3D beam element) considering shear effect are usually used for beam and column, and S4R (shell element) is for floor. As B31 element needs less computational cost when the quantity of grid is equivalent, it is suggested to simulate beam and column. Furthermore, this element can be subdivided to raise efficiency if it cannot meet demands of the precision [13]. With stable performance, S4R element is effective for thin and thick shell, and can be used for simulating large deformation and strain.

For the simulation of FRP, a general solution in ABAQUS is to use M3D4 (membrane element) [14, 15] and S4R (shell element) [16, 17]. The two methods can both well simulate the three-dimensional

stress state of concrete provided by variable constraint from FRP, and general trend of the stress-strain relationship from FEA result is broadly consistent with test results (Fig. 1). The problems are the following: (1) mainly for axial compression and small eccentric-compression columns; (2) not considering adhesive failure between FRP and concrete so that cannot be applied to cyclic loading; (3) simulation of the FRP layer only by changing its thickness.

Restoring Force Model

As restoring force model directly affects the FEA result, the definition of restoring force curve is the most basic for elasto-plastic analysis. There are two main methods to establish the restoring force model of FRP-confined concrete:

One method commonly used in ABAQUS is customized or software-based constitutive model. Simulation of FRP can be divided into two categories: one is the use of *skin enhancement layer*. Interaction between membrane and concrete element is simplified by the *tie constraint* [18]; the other is the S4R shell element using elastic model with *contact element* setting on their surface to consider the influence of friction force [19].

The other is to directly use the existing constitutive model of FRP-confined concrete. Some relevant studies have been carried out, especially WU Gang research group. Based on summary of a large number of experimental curves on FRP-confined rectangular concrete columns and the next theoretical analysis [20], the restoring force model considering stiffness and strength degradation is established (Fig. 2). Then the model is verified and can be used for elasto-plastic seismic response analysis of FRP-confined rectangular concrete structure. Though this method has been used for other FEM software successfully, application of elasto-plastic analysis based on ABAQUS is rare and needs further validation.

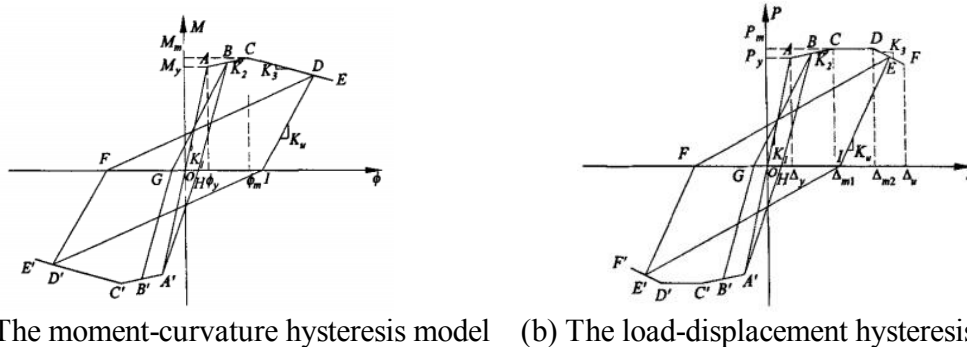


Fig. 2 Hysteresis Model for FRP Rectangular Concrete Columns

Definition of Steps and Load

Another key problem existing in elasto-plastic dynamic analysis is the simulation of earthquake action, and suitable earthquake wave is an important factor affecting the results of time history analysis. As most of the seismic wave preliminarily selected does not meet the three essential requirements of ground motion (ground motion intensity, spectrum characteristics, and duration), it can't be used until corresponding adjustment. The next task is to realize elasto-plastic analysis of finite element model effectively, including selection of algorithms and definition of steps and load. The increments, mainly including initial increment, maximum or minimum time increment and maximum number of increments, affect smooth operation and computation speed to a great extent. If initial increment is set too small, it will become time-consuming, and on the contrary will not converge.

Conclusion and Suggestion

Elasto-plastic analysis of RC frame structure based on ABAQUS has demonstrated its strong ability of nonlinear analysis, but seismic analysis of RC frame structure retrofitted by FRP needs further research.

1) Separated modeling method with additional steel element inside concrete is suitable for the analysis of integral structures, and is easy to operate.

2) Appropriate element types should be based on the requirements of precision and accuracy to achieve preferable results in short time. The bar-shell hybrid element method is used for RC frame structure mostly.

3) Currently, membrane element and shell element are mainly used to simulate the function of FRP. This method can better simulate three-dimensional stress state of concrete. However, it ignores adhesive failure between FRP and concrete and makes the simulation of FRP layer simplified. Present constitutive relationship of FRP-confined concrete has been well used in other FEA software, but still needs further verification in ABAQUS to simplify the modeling.

4) Seismic wave should be selected combining with relative standards [1] and literatures [21]. And it can't be used until corresponding adjustment. Rational steps and load setting will make simulation results approach the practical situation.

Acknowledgement

This research was financially supported by Natural Science Foundation of Shandong Province (ZR2012EEL25) and Introduction of Talent Fund Project of Shanghai Institute of Technology (10120K146001-YJ2014-01).

References

- [1]GB 5011-2010, Code for Seismic Design of Buildings, China Architecture & Building Press, Beijing, 2010.
- [2]R. Garcia, I. Hajirasouliha, K. Pilakoutas, Seismic behavior of deficient RC frames strengthened with CFRP composites, *Engineering Structure*. 32(2010) 3075-3085.
- [3]Wang Daiyu, Experimental and Analytical Investigation of Seismic Performance of Nonductile RC Frames Retrofitted with FRP, Harbin Institute of Technology, Harbin, 2012.
- [4]Wang Zhenyu, Wang Zhicong, Pan Xiaolan, Analytical investigation of seismic performance of low-rise non-ductile RC frames retrofitted by CFRP, *Academic Discussion Proceedings of the 7th National Construction Project on the Use of FRP*, Industrial Construction Magazine Agency. (2011) 148-151.
- [5]Song Ke, Xu Qian, Hao Zejing, Research on seismic performance for reinforced concrete frame structure with hybridized CFRP, *Journal of Hebei United University (Natural Science Edition)*. 34(2012) 126-131.
- [6]Liu Qiang, Performance Analysis on RC Frames Strengthened with CFRP, Tianjin University, Tianjin, 2012.
- [7]Li Peng, Dynamic Time History Analysis of RC Frame Seismic Retrofitted by FRP, University of Jinan, Jinan, 2013.
- [8]Du Ke, The Application and Compare of Three Dimensional Structure Elastoplastic Analysis Softwares in Common Use, Institute of Engineering Mechanics, China Earthquake Administration, Harbin, 2010.
- [9]Gao Xiangling, Zhang Yeshu, Li Jie, Numerical simulation of a RC frame based on ABAQUS beam elements, *Structural Engineers*. 29(2013) 19-26.
- [10]He Liusheng, Jiang Huanjun, Comparative study between elasto-plastic time-history analysis and shaking table test results for a frame structure, *Journal of Civil, Architectural & Environmental Engineering*. 32(2010) 568-570.
- [11]Zhu Lili, Study on the constitutive model of concrete and steel for explicit dynamic beam elements of ABAQUS, Shenyang Jianzhu University, Shenyang, 2012.

- [12]Wang Yuzhuo, Fu Chuanguo, Structural Analysis and Detailed Examples Based on ABAQUS, China Architecture & Building Press, Beijing, 2010.
- [13]Technical Center of Shanghai Xian Dai Architectural Design (Group) Co., Ltd., Dynamic Elasto-plastic Time History Analysis Applied in Structural Seismic Design, Shanghai science and Technology Press, Shanghai, 2013.
- [14]Deng Libang, Analysis on the Property of Circular Concrete Columns Restricted with FRP Subjected to Secondary Axial Compressive Load, Harbin Engineering University, Harbin, 2011.
- [15]Huang Can, FEA of Load Capacity for BFRP Jacketed Circular RC Columns under Eccentric Loading, Guangdong University of Technology, Guangzhou, 2014.
- [16]Huang Yan, Qi Lukuan, Nonlinear FEM analysis of circular concrete column confined with FRP under axial compression, China Railway Science. 29(2008) 46-50.
- [17]Jin Xirong, Experimental Studies on Behavior of FRP Confined Concrete Columns under Rapid Loading, Harbin Institute of Technology, Harbin, 2007.
- [18]Zhou Changdong, Li Chunlong, Strain evolution of concrete square columns confined by GFRP, Journal of Basic Science and Engineering. 21(2013) 137-146.
- [19]Xu Tian, Li Yan, Pan Jinglong, FEM analysis of stub column confined with FRP under axial compression, Low Temperature Architecture Technology, 8(2009) 48-49.
- [20]Wei Yang, Study on Mechanical Properties and Seismic Performance of FRP-Confined Rectangular Concrete Columns, Southeast University, Nanjing, 2007.
- [21]Wang Lijuan, Research on Ground Motions Record Input of Structure Time-history Analysis, Xinjiang University, Xinjiang, 2013.