The Elastic-Plastic Dynamic Analysis Software GSEPA and Its Application in Seismic Analysis of Tall Buildings

Jiao Ke, Lai Hongli, Chen Xing, Huang Zhenkang

The Architectural Design and Research Institute of Guangdong Province, Guangzhou, China
Email: gscad@21cn.com

ABSTRACT: The elastic-plastic dynamic analysis of the high-rise buildings under strong earthquake will help engineers to evaluate the antiseismic performance, optimize the structural scheme and strengthen the weak parts of the structure. This paper introduces the elastic-plastic dynamic analysis software GSEPA based on ABAQUS, which is developed for seismic analysis of the high-rise structures by the authors. GSEPA can read data directly from structural design software, which includes the connectivity between the components, section sizes, material properties, rebar of concrete beams, etc., for generating ABAQUS model. Three practical projects analyzed by GSEPA are given, including a 331m high lighthouse-style structure, a twin-tower structure connected by space corridor with 160m high and a 210m high transfer layer structure. The results of the three-dimensional elastic-plastic analysis show the effectiveness of GSEPA, which can be referred by other practical engineering.

KEYWORDS: Elastic-plastic dynamic analysis, ABAQUS, software, high-rise building

1. INTRODUCE

With the accelerated process of urbanization, urban high-rise buildings are getting more and more higher and closer between each other. The risk due to the collapse of buildings under earthquake is more and more greater. Due to the unpredictability of earthquake, as well as the discreteness of seismic waves, the preferred method used by engineers in the structural design is to enhance the ability for reducing earthquake damage. In the codes for seismic design of different countries, both the safety and the economy of the structure are taken into account. In addition to the elastic analysis under "small earthquake", the elastic-plastic analysis under "strong earthquake" is necessary. When the response of the structure under strong earthquake is beyond the scope of the elasticity, the structure behavior will be changed from elastic status to elastic-plastic status. The relationship between deformation and load will be changed from linear to nonlinear, and the internal forces and deformation of structure will have significant changes.

In order to make clear the process from gradually cracking, damage until the collapse, understand its antiseismic performance for a concrete high-rise building, and further strengthen the weak parts intentionally to prevent the collapse of the structure, it is necessary to carry out the elastic-plastic seismic response analysis. Now the available software packages for elastic-plastic analysis are general purpose finite element analysis packages, which provide powerful analysis functions, but the operations are complicated. So the application of these packages in the design of building structures is difficult. This paper introduces an elastic-plastic analysis program GSEPA for high-rise building, which adopts ABAQUS as the core solver. The GSEPA can directly create the numerical model of ABAQUS from the design software, and therefore engineers can easy perform elastic-plastic analysis for tall buildings.

2. NONLINEAR STRUCTURAL ANALYSIS SOFTWARE FOR EARTHQUAKE RESPONSE

2.1. The commonly used elastic-plastic analysis software

LS-DYNA, MARC, ADINA, ANSYS, ABAQUS, NASTRAN, etc., are commonly used for the structure elastic-plastic analysis. Each package has its own feathers. For example, ABAQUS has powerful non-linear capacity which provides concrete elastic-plastic fracture/damage model and reinforced concrete model, and allows users to create their own material models. The concrete model is suitable for the elastic-plastic dynamic analysis of reinforced concrete structures. Its main advantages are:

(1) Can accurately simulate the response of concrete structure under monotonous loading, cyclic loading and dynamic loading, and can be defined to be sensitive to the rate of strain;

(2) Uses concepts of isotropic damage elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete;

(3) Consists of the combination of non-associated multi-hardening plasticity and scalar (isotropic) damage elasticity to describe the irreversible damage that occurs during the fracturing process;

Elastic-plastic dynamic analysis needs to consider two issues: the constitutive relationship of reinforced concrete materials and the solution efficiency of non-linear equations. Due to the rich concrete material models and powerful ability to solve nonlinear equations, we choose ABAQUS as the computational core for GSEPA, and the pre- and post-processing are completed by GSEPA.

2.2. Some issues of elastic-plastic dynamic analysis in high-rise buildings

Before the elastic-plastic time-history analysis, the seismic wave must be quantitated in accordance with period of time, and then imported to the differential equations of structural system. The step by step integration method is adopted to calculate the dynamic response of the structure in the whole time domain. The internal forces and deformation of all members in each time step, and the occurring order of the plastic hinge are outputted. According to the strength and deformation, we can judge the safety and reliability of the structure under earthquake and determine the structural failure mechanism. In the calculation of elastic-plastic dynamic analysis for high-rise building, the following issues must be paid attention.

2.2.1 Model simplification

As high-level model is very complex, especially including reinforced concrete members, the computational model must be simplified properly in order to improve the efficiency and ensure the convergence. For example, some secondary beams or small walls with small stiffness and little effect on the structure should be removed.

2.2.2 The model of restoring force

According to the properties of material, component type and state of the internal force, select the appropriate model of restoring force of the structure and identify the corresponding structure (or bar) cracking, yield and ultimate displacement, as well as the stiffness values of the polygonal line of the characteristic curve of restoring force.

2.2.3 The choice of seismic waves

The principle of choosing seismic waves should enable the three characteristics (seismic intensity, spectrum-shape parameters and the duration of strong earthquakes) of input seismic waves consistent with the specific conditions of the construction site. According to "Seismic Design of Building" proposal, the rare earthquake analysis adopts one artificial seismic wave and two natural strong earthquake waves of recorded acceleration of ground motion as the input of the nonlinear dynamic-history analysis.

2.2.4 Meshing and member generation

The meshing and member generation by different scale and system of structure should be considered. The density of element subdivision should be confirmed in accordance with computational requirements and hardware conditions. The quality of element subdivision will affect the convergence and the accuracy of calculation. Meshing of walls, columns, beams and panels should consider the effect of adjacent components. A large number of nodes of non-coincidence should be avoided.

2.2.5 Evaluation of seismic performance

The purpose of the elastic-plastic analysis of structure under earthquake is to evaluate the structural seismic performance, for high-rise buildings, mainly on the following issues:

1) The nonlinear behaviors of structure under strong earthquake, such as the deformation (e.g., largest top displacement, the largest story drifts), the plastic and damage situation of members (e.g., the largest base shear).

2) The important parts for special attention, including the strengthened parts at the bottom of structure, the beams and braces of transfer layer, floor with large opening, equipment floor and the strengthening floor, and so on.

3) Seismic performance of the overall structure, finding the weak parts of the structure.

3. THE ELASTIC-PLASTIC DYNAMIC ANALYSIS SOFTWARE GSEPA

GSEPA is the program which transforms the model from general purpose analysis and design software GSSAP to ABAQUS. The solver used in GSEPA is the large finite element software ABAQUS/STANDARD and ABAQUS/EXPLICIT. For the large scale of calculation data in high-level structures, we adopt the direct integration explicit approach to solve the large-scale non-linear equations. The main features of GSEPA are listed as follows.

3.1. Simplification of the structure model

1) The beam with cross section height less than 400mm and length less than 2m will be changed into the equivalent loading.
2) The shear wall with length less than 0.5m will be processed.
3) Shear wall openings will be aligned.
4) The short beam and wall due to eccentricity of members should be aligned with the lap point.
5) The deep beam with large ratio of height to width should be modeled as a wall with opening.

3.2. Generation of material model

ABAQUS provides a widely range of constitutive model. The shell element with concrete elastic-plastic damage model is very suitable for simulating shear wall in reinforced concrete structure. We choose the tension-compression stress-strain relations of Appendix C of the specific terms "Reinforced Concrete Design Specifications" as the skeleton of the concrete hysteretic curve, and add the damage factor to constitute an integrated concrete tension-compression hysteretic curve, as shown in Figure 1. The yield and hardening of the steel model used linear hardening two-segment model, with strengthening stage of the E'=0.01E, using Mises-yield criteria and linear hardening, as shown in Figure 2.

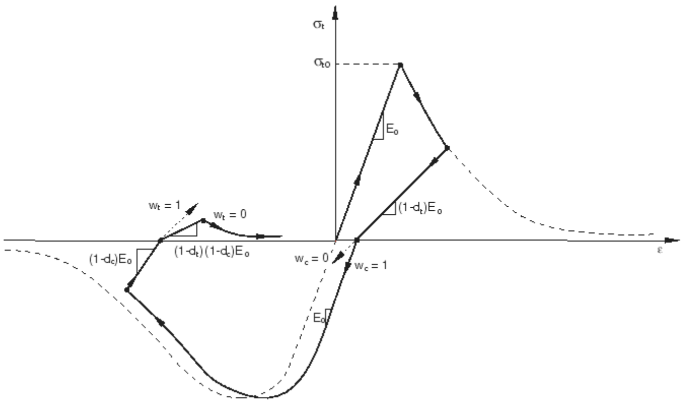

Figure 1 Concrete tension-compression stiffness restoration curve

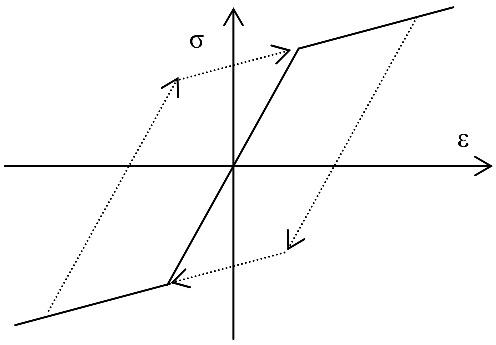

Figure 2 Steel tension-compression hysteretic curve

3.3. Meshing and member subdivision

GSEPA has four kinds of automatic subdivision forms:
1) Line element subdivision, such as beam with variable cross-section;
2) Rectangular element meshing for wall with opening;
3) Meshing of polygonal slabs;

4) Meshing of knaggy-polygon slabs;

3.4. Generation of the geometry and load data

It mainly includes nodes and elements data, point load, line and body load, reinforced implantation in concrete structures, the eccentric treatment between members, element-direction nodes generation, etc.

3.5. Boundary conditions and constraint information.

3.6. Generation of the information for construction sequential analysis

Each member can be assigned to an individual step number for construction sequential analysis, and each number is a load case. The stiffness and load is modified in each step by the element life-death function. For the high-rise connected structures, if the construction plan has not determined yet in preliminary design stage, it is better to perform of the one-time load calculation.

3.7. The results summary and chart generation

4. APPLICATIONS OF GSEPA

4.1. Lighthouse-style structure

A lighthouse-style structure, GuzhenDengdu (located in Zhongshan, Guangdong province), is a modern facilities building which integrates tour, hotel, office, exhibition, business, entertainment, and other functions. Its construction area is about 118,000 square meters with 66-story and two-story basement. The main structure is 318.8 m high, and the radius of the smallest plane is only 12 m.

The results of the elastic-plastic analysis show that the upper part of the "chimney" has a significant “whip” effect, and the floor which has the largest story drift angle is not the first failure position of the structure. As the starting point of the “whip” effect, despite of a smaller story drift angle, the 45th story becomes a weak story and would be damaged and lose capacity due to the large shear deformation in this story. Some information is shown in Figure 3.

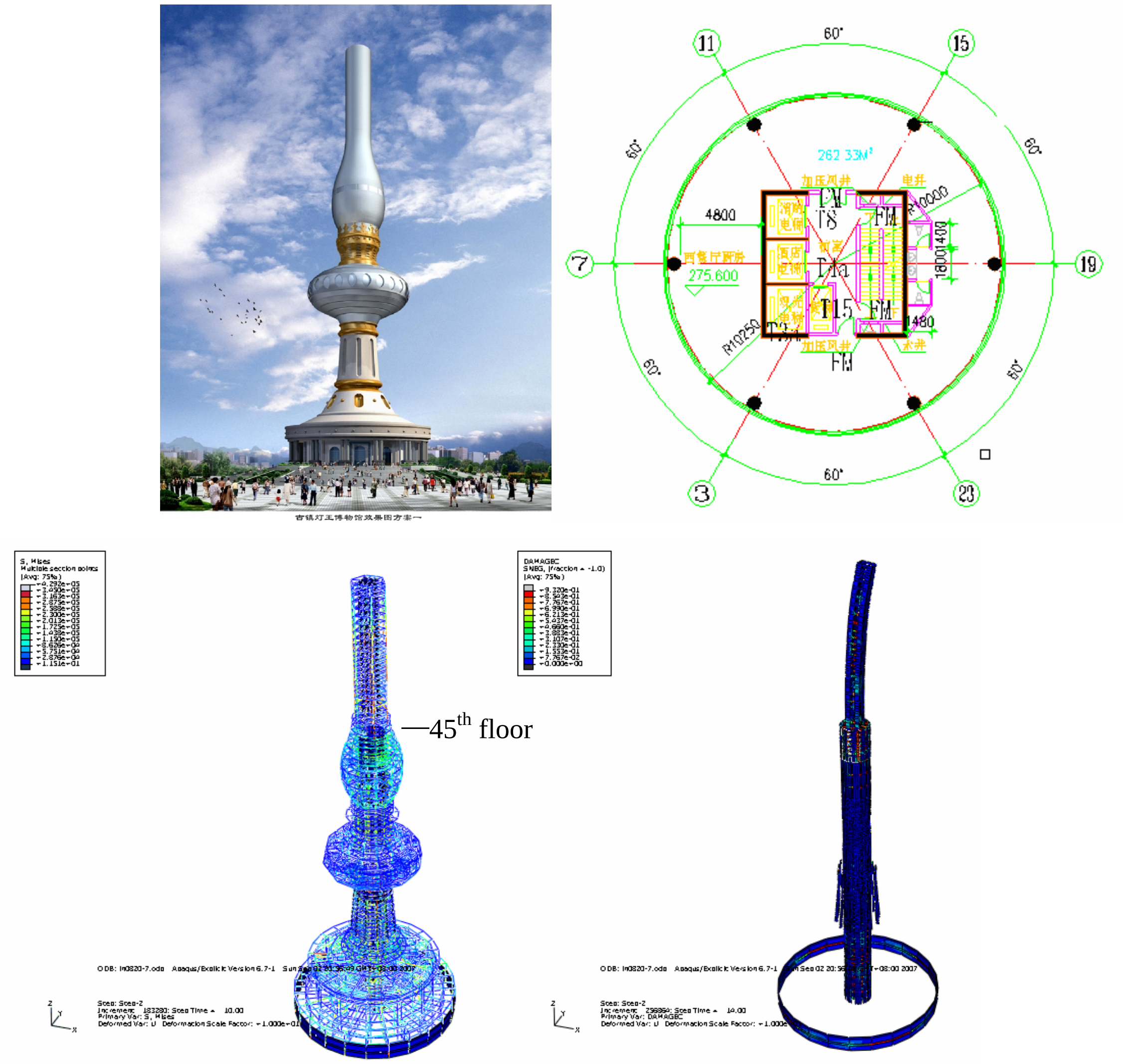

Figure 3 Three-dimensional effect, the floor plan, overall Mises stress and shear wall compression damage

4.2. Twin-tower structures connected by space corridor

The building is located in Dongguan, Guangdong Province, and will be a landmark of Dongguan. Its floor area is about 225,000 square meters, with 39-story and two-story basement. It is the structure of Siamese twin towers. The main structure is 161 meters high, which use frame-core tube structural system (see Figure 4).

The structure has the following features: structural rigidity and shear strength have a mutation between connection floors and the lower floors; the vibration mode of towers is coupled, and torsion is easily induced under earthquake. Because the connection and the two towers connected rigidly, torsion is easily cause brittle failure, and the connected part of the structure is sensitive to the vertical vibration.

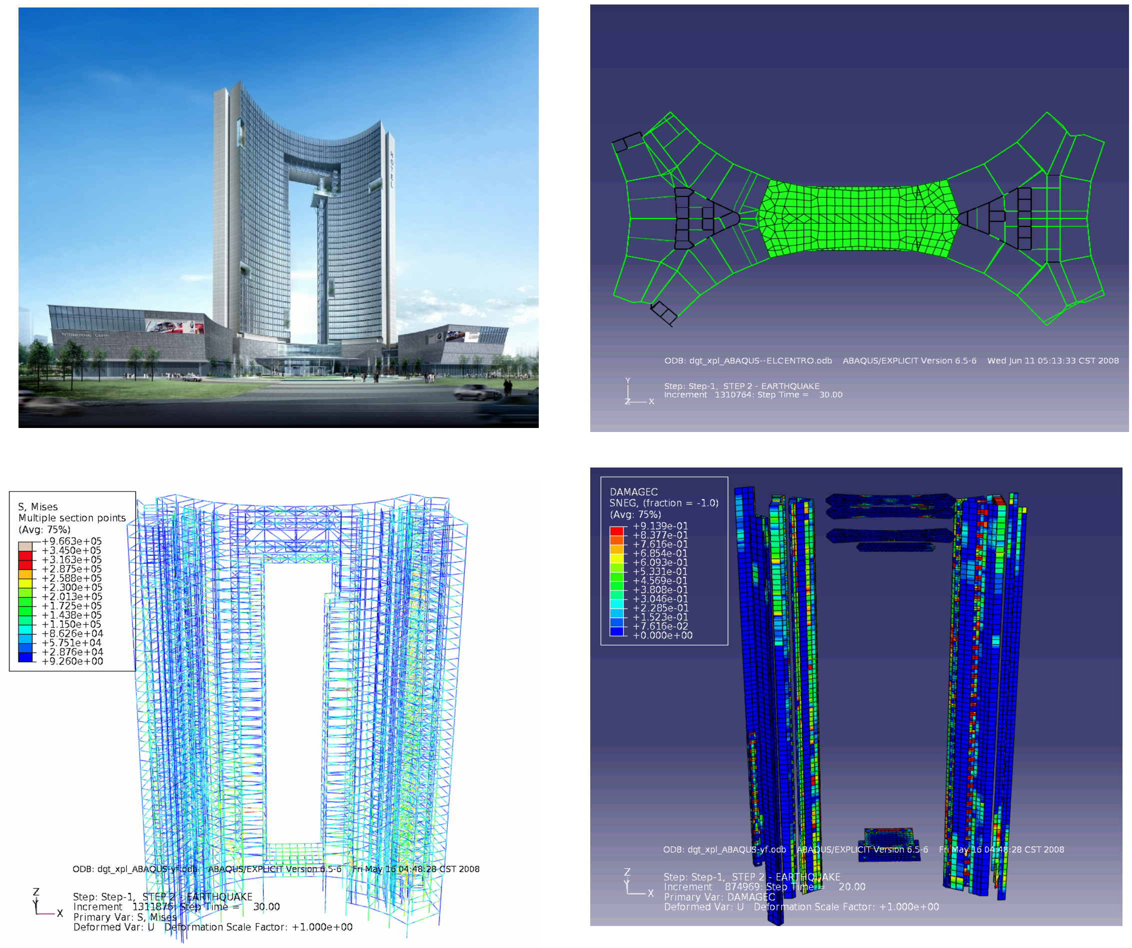

Figure 4 Three-dimensional effect, the floor plan, overall Mises stress and shear wall compression damage

4.3. Transfer layer structure

The 53-story 208 meters high building with 100,000 square meters construction area is a frame-tube structure (Figure 5). In the 6th, 7th, 28th and 45th story there are partial transfers.

In original design, because of the rigidity change, the upper part above 46th story shows significantly "whip" effect. Although the fist and the second periods are close to each other in elastic analysis, and the different of strength in the two directions is not very obvious, the elastic-plastic analysis shows that X-direction is the weak direction; the upper part above 46th floor obviously yields with more damaged structural components. The new design makes the "whip" effect significantly reduce by reducing the stiffness mutation of the upper structure.

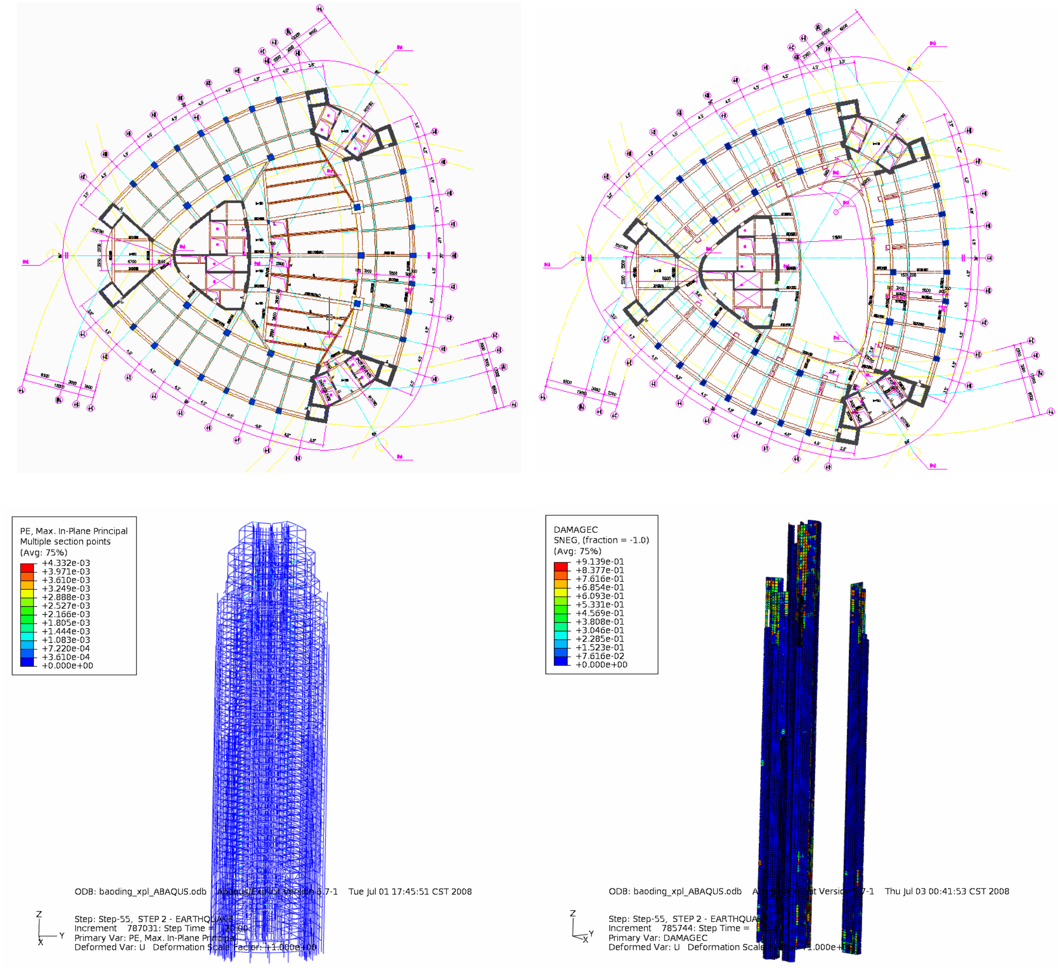

Figure 5 The 28th transfer layer, the standard floor plan, the plastic strain and shear wall compression damage

5. SUMMARY

This paper introduces the elastic-plastic dynamic analysis program GSEPA based on ABAQUS, which is developed for seismic analysis of the high-rise structures. GSEPA can directly read data from the design software and generate ABAQUS model, thus be greatly convenient for design of the structure under rare earthquake. This paper also gives the elastic-plastic analysis for three typical projects by GSEPA. The results show the effectiveness and feasibility of GSEPA for analyzing complex high-rise building structures.

REFERENCES

ABAQUS, Inc. (2007). ABAQUS User Manual (v6.7-1).
ABAQUS, Inc. (2007). ABAQUS Theory Manual (v6.7-1).