

The Engineering Practice of Elasto-plastic Dynamic Time-history Analysis on Complex Building Structures Using Abaqus

Dongya An, Chengming Li, Jiahua Zhang, Jiachun Cui, Wei Tian

(Shanghai Xian Dai Architectural Design (Group) Co.,Ltd. Shanghai, 200041)

Abstract: Compared with response spectrum method and the pushover method, Elasto-plastic dynamic time-history analysis method is considered to be a more accurate seismic analysis method. Because of Abaqus's strong non-linear calculation function, the software makes it possible for the method mentioned above to be applied. Elasto-plastic dynamic analysis on complex structures using Abaqus has a rapid development in China. In this paper, with CCTV Building, Shanghai World Expo Axis Sun Valley, Dalian Xiaoping island super-high-rise apartment building and many other cases of engineering practice, the problem-solving experience of seismic analysis on major projects is described, which will provide some reference to the analysis method's further development and engineering practice.

Keywords: Elasto-plastic dynamic time-history analysis, Abaqus, complicated building structure, seismic analysis

1. Introduction

China seismic code (GB 50011-2001) adopts applied method to prevent structures from destroy under minor earthquake and collapse under major earthquake, which is doing capacity design and elastic deformation checking under minor earthquake firstly then checking elastoplastic deformation under major earthquake.

Nowadays in the first phase, the elastic static analysis and dynamic time-history analysis method are relatively mature. And in domestic and abroad there is a lot of computing software which can provide good results. But in the second phase of the design the elasto-plastic analysis method is still under developing. Pushover method is a simplified elasto-plastic static analysis method, which is still being improved (Chopra, 2002). This method ignores the dynamic effect of the earthquake and is considered not to be an ideal method (FEMA 273&274, 1998). Elasto-plastic dynamic time-history analysis is to consider internal force and deformation of the structure at every moment during the whole process of earthquake. It can also point out structural cracking and yielding order and discover the concentration zone of stress and plastic deformation, thus help engineers to recognize the yielding mechanism, weak links and the possible failure modes, so the method is regarded as a relatively complete and accurate method.

Dynamic method requires high-speeded computer hardware, well-developed function of software and higher professional skills, which limits the scope of its use. However, the rapid development of high-rise buildings and complicated structures nowadays in China put pressing needs on the Elasto-plastic dynamic time-history analysis method. In particular, in the amending edition of

"Code for seismic design of buildings" and "Technical Specification for concrete structures of tall building ", the articles require Elasto-plastic dynamic time-history analysis to be compulsory method in some cases, therefore there is a group of excellent engineers is working on realization of the method, and Abaqus is regarded as one of the most popular powerful tools.

2. Reasons for selection Abaqus as a tool to do Elasto-plastic dynamic time-history analysis

As a general-purpose finite element program, Abaqus has a rapid development in many areas since entering the Chinese market. Especially in the last five years, Abaqus has experienced almost explosive development in the field of seismic analysis. It is not being recognized by the industry when it has just been introduced, however, with the continually self-development and redevelopment from the users, it is gradually accepted by the industry, and now Abaqus has accumulated much successful experiences in the field of seismic analysis which prompting itself as leading position in the area. Nevertheless, the selection of Abaqus to conduct elasto-plastic dynamic analysis is not the most convenient way, the reason is that the structural engineers need to pay a lot of intellectual effort during using Abaqus to complete their work. The reason why structural engineers prefer Abaqus as their tool is Abaqus has several very important advantages:

1. Powerful non-linear solution

Abaqus is a powerful finite element software of engineering simulation, and its solution to the problem is in the range from relatively simple linear analysis to many complex non-linear problems, related to geometry, material nonlinearity and material damage accumulation of other complex questions.

2. Explicit integration to solve two major problems

The core problem of Elasto-plastic dynamic time-history analysis is the numerical integration of dynamic equations. As the structure's non-linear nature, the modal superposition method is no longer applicable, and the direct integration method should be used. Implicit and explicit are the main direct integration method. Implicit algorithm is, obtain the next moment displacement according the balance equations, while the explicit algorithm is using the current balance to obtain the next moment displacement. In general, unconditionally stable implicit algorithm is usually used to solve dynamic problems, and NEWMARK method is the most widely used implicit algorithm. However, when the method is applied to Elasto-plastic dynamic time-history analysis, it encountered the following two problems: Firstly, implicit algorithm requires that every step to do matrix inversion, but with the structure's degrees of freedom increasing, the time spent for matrix inversion will have a geometric growth, and the computing time tends to be too more to be realistic; its second, when the structure come into a serious non-linear state, some steps need to be further divided into fine sub-step in order to ensure convergency. Even more serious is that when the structural stiffness have severe mutations or there is negative stiffness, the solution process can not be converged even when the step is re-segmented, and thus unable to obtain the result. Explicit algorithm requires that the stable step time is less than the minimum period of every element, usually an order of magnitude smaller than the implicit algorithm. But the explicit algorithm neither requires matrix inversion nor the formation of stiffness matrix, and thus much shorter time is enough for each step of the solution. And there is a linear relationship between the solving time and

element number, so that explicit algorithm has an advantage for the large-scale subject. Also because of using smaller calculation step, it can describe the seismic action more accurately, and also avoid divergence problem caused by serious non-linearity in structure as the implicit algorithm encountered. Based on the above analysis, explicit algorithm can solve the two major problems of time-consuming and convergence in large-scale computing.

3. Provide an open platform for secondary development

Abaqus provides the user with a relatively open secondary development platform. Despite Abaqus itself already has quite a powerful computing feature, including a rich element library and material library (Abaqus User Manual, 2006), it still can not automatically resolve all problems in building structure's Elasto-plastic dynamic time-history analysis. Reinforced concrete used in building structure is a kind of material with special properties, but also the most commonly used material. Whether the RC constitutive relation can be simulated accurately has a direct impact on the final results of the analysis. Abaqus itself provides a wide range of materials suitable for concrete constitutive model, in which damage model is considered a more accurate model, but this model can not be used directly in the beam-column model, and therefore to simulate the most commonly used element's material properties, users need to do some secondary development, and establish a suitable material constitutive model (Lee, 1998). Abaqus program provides such ways to do secondary development, and a variety of user material model can be developed, and ultimately applied to the entire structure of Elasto-plastic analysis.

4. Successive modeling

During the whole process of elasto-plastic time-history analysis, the structural engineers always spend much time and effort in modeling. As the first phase of the seismic design is flexible structural design and analysis, thus using some of the professional program to establish an elastic model is necessary. In the second stage of Elasto-plastic dynamic time-history analysis, more precise Elasto-plastic model including such element as reinforcement is required, and the re-establishment of a new model from scratch would cost engineers a large number of repetitive labor. Abaqus also provides open interfaces in this area, and engineers can find out a certain way to import the model from other professional software into Abaqus, and then do some necessary modification on this basis to obtain the final model for the Elasto-plastic analysis, thus saving a lot of time.

3. The cases that Abaqus provides assistance to structural engineers in the Elasto-plastic dynamic analysis

The structural analysis center of XianDai Group have conducted a lot of research on Abaqus' use in structural Elasto-plastic dynamic time-history analysis, and accumulated a lot of experience. A large number of engineering cases showed that Abaqus program provide a good help for engineers on different aspects and greatly improve the design efficiency and rationality. Here are some individual cases selected to show our work and derived inspiration.

3.1 Establishment of a full flow about calculation from the elastic to the elastoplastic

Because of designer's different habits they may use varied kinds of professional procedures to complete the elastic calculation and design, and usually they also need to have inter-checking process between different codes, and then do the Elasto-plastic dynamic time-history analysis ultimately. The model transformation between the various procedures is needed, and at the same time there are always much adjective job to complete the eventual establishment of a full fine model for Abaqus. The team of Xiandai Group accomplish the integrated flow (Shown in Figure 1)



Figure 1. Model transformation between the different procedures.

3.2 Checking the elastic analysis results from the professional software and finding out the oversight in modeling process

Compared with the elastic analysis, elasto-plastic analysis results are closer to the actual stress conditions. Generally, in Abaqus different static analytic cases and dynamic time-history under minor earthquake cases should be carried out before doing the rare-earthquake Elasto-plastic time-history analysis, and we can call them pre-calculation. Because materials' Elasto-plastic properties are involved in all cases, you can get more realistic results from the pre-calculation to check the elastic analysis, and find out some problems which can be not discover in the elastic process, for example, the major oversight in modeling due to the negligence of the operator.

Example 1: DongJiu Building

For DongJiu Building, when carrying out a minor earthquake check in Abaqus, we found layers of concrete columns in the lower floors of the tower appeared premature cracking damage, and there is a clear dividing line between the lower structure and the upper region. After the examination it

is found that these cracking columns have been specified a mistaken material number. This mistake had no warning in the elastic analysis but conducted explicit error information in the elasto-plastic process.



Figure 2. DongJiu Building.

3.3 Provide various help in the seismic performance evaluation of the overall structure

As the Elasto-plastic dynamic time-history analysis in seismic design process belongs to the second phase of the verification stage, the primary job after calculating is to assess the seismic performance of the structure. Abaqus has powerful post-processing functions, which can provide different aspects of the analysis results for the designer.

- 1.** Providing overall macro-indicators of the elasto-plastic deformation.

Example 2: CCTV

The total height of CCTV main building is 234 m, its construction area is about 400,000 square meters, the main building consists of two towers (vertical and horizontal direction are tilted six degrees), a 14-floor high-level cantilever structure at top, a 9-floor podiums at bottom and three-floor basement (as shown in Figure 6.1); the maximum length of cantilever is about 70 meters. The structure is in the form of steel-concrete frame, with the outer frame tube strengthened by diagonal steel braces. Part of the column with steel-concrete section, and the other with steel cross-section. Some RC beams and some pure steel ones. Aseismic design intensity is 8 degrees.

The initial elastic model is form Sap2000, and after it is transferred into Ansys software, there is a lot of modification work to do, in other words, we make Ansys software as the pre-processing tool, and then the final inp files are received which can be read by Abaqus.

Table1. Some key technical details.

Nodes	Elements	DOF	CPU number	Products used	CPU time	Hardware
218351	75849	711423	8	Abaqus/ Explicit	20 hours	HP7620 Minicomputer



Figure 3. CCTV mail building model.

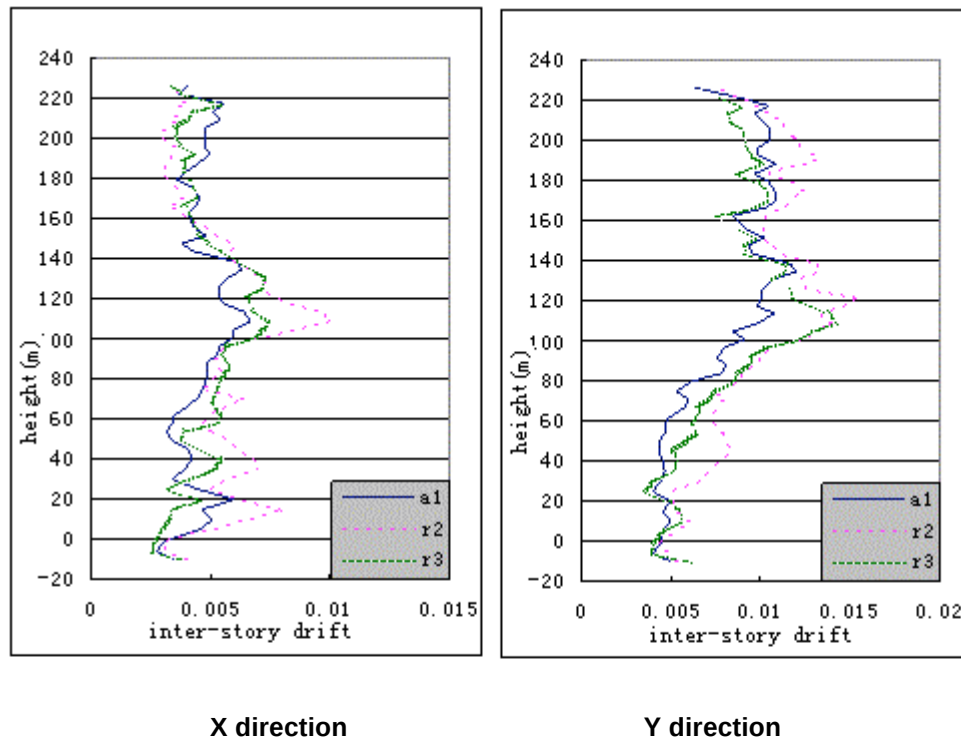


Figure 4. Inter-story drift curves.

Table 2. The maximum inter-story drift angle.

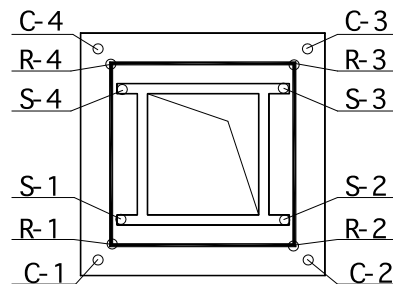
Tower	direction	wave A1	wave R2	wave R3
Tower A	X	1/150 (27th floor)	1/99 (27th floor)	1/135 (27th floor)

	Y	1/85 (34th floor)	1/64 (30th floor)	1/70 (27th floor)
Tower B	X	1/137 (9th floor)	1/121 (9th floor)	1/145 (27th floor)
	Y	1/73 (39th floor)	1/86 (29th floor)	1/82 (39th floor)

2. Providing stress and strain developing information

Abaqus can provide the stress and strain state in the section at any time, as well as the hysteresis curve of members to help engineers evaluate the actual stress conditions.

Example 2: CCTV



**Figure 5. Integration point location in SRC cross-section.
(C-concrete, R-rebar, S-structural steel)**

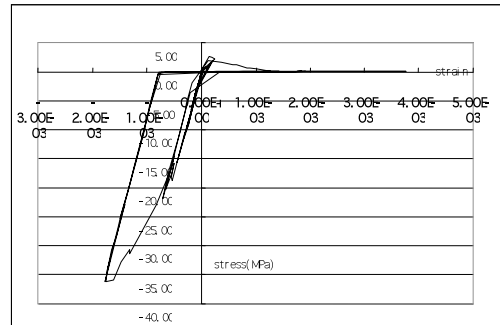


Figure 6. Hysteresis curve of concrete stress-strain relationship.

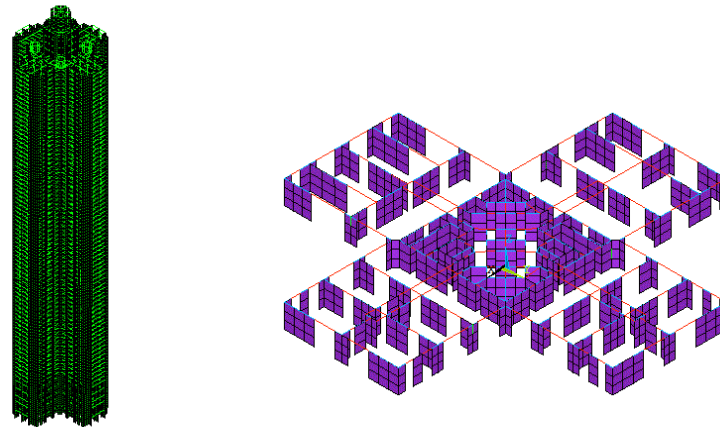
3. Providing the sequence of plastic development in the structure

Structural engineers expect great seismic safety of the designed structure. The first destruction of some components under major earthquake is acceptable, which can dissipate seismic energy and reduce damage to other members. In elasto-plastic dynamic time-history analysis, we need to look at the sequence of plasticity and damage to the structural members during the whole earthquake process and then evaluate whether the

structural program is reasonable, and we can achieve this purpose through elasto-plastic dynamic analysis by Abaqus.

Example 3: Super-high-rise apartment of Dalian Xiaoping Island

Ultra-high-rise apartment building, structural height is 253.7m, exceeding the limit height (230m) as prescriptive in related code to tube structure in 7 degree aseismic intensity. Irregular layout of cross shape. 5 tubes connected each other through connecting beams. The performance of these connecting beams under seismic need detailed investigation to verify whether they can play the role of a powerful seismic defense.



A. Whole model

B. Standard floor

Figure 7. Super-high-rise apartment model of Dalian Xiaoping Island.

4. Help designers to determine the weak layer or weak region of the structure

Example 4: ShenDu building

ShenDu building is located on South XiZang Road and closes to East XieTu Road. Its site area is about 6800m². This building is now being reconstructed by XianDai. The total height is 27.4m, plane shape is an L shape, side lengths are 40.6m and 33.8m respectively. The lower 5 floors are constructed into RC frame structure and top 2 floors into steel structure.

Elastic time-history analysis is performed in professional design software and then Elasto-plastic analysis process carried out in Abaqus. basing on the elastic results merely, we find no weak layer in the structure, however the Elasto-plastic analysis datum tell us there is obvious concentrative plastic deformation in the 4th floor in Y direction (as shown in figure 9). It is clear that some appropriate measures need to be taken to strengthen the weak layer.

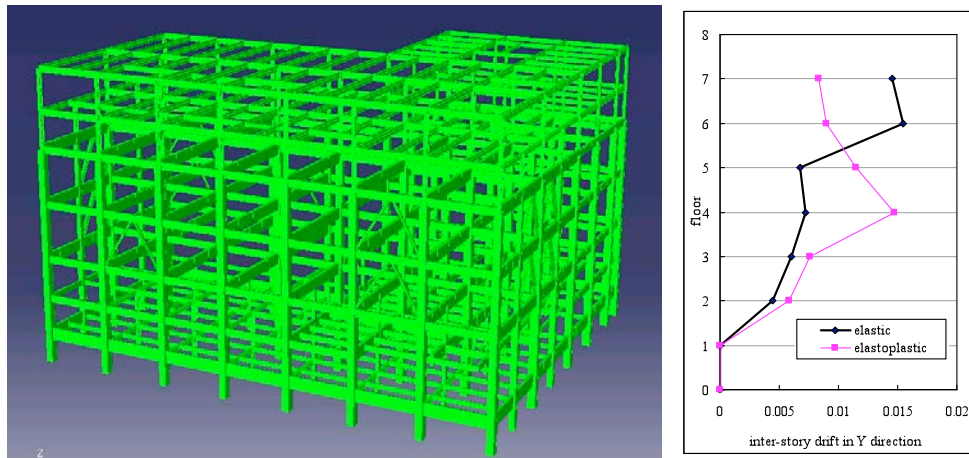


Figure 8. ShenDu building model in Abaqus. Figure 9. Inter-story drift curves.

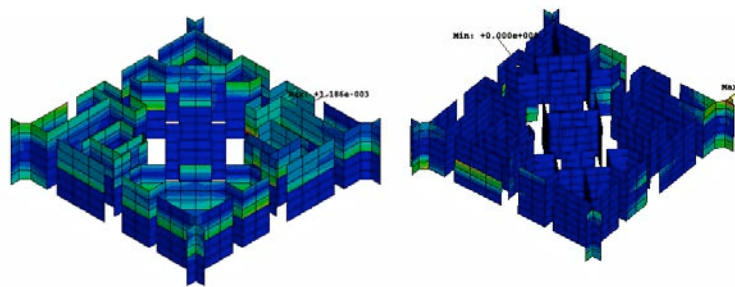
3.4 Provide a basis for structural optimization and design adjustment

Elasto-plastic dynamic time-history analysis is usually in the further design stage, in addition to evaluation of seismic performance, another important role of the analysis is to optimize the structure and adjust the program, and put forward appropriate measures to solve the problems found during the analysis, improve the structure's performance under earthquake. The following are several cases:

1. Adjustment of classification position along the tower height direction of cross-section and material strength of shear wall.

Example 5: Super-high-rise apartment of Dalian Xiaoping Island

In the case of Dalian Xiaoping island apartment building, after the calculation to the first program, it is found that there is serious concentrated plastic deformation at different height position of the shear wall tube. The examination revealed that in these floors the thickness and material grade of shear walls have simultaneous change, leading to stiffness mutations and concentrated plastic deformation. Based on this, adjustment measures is put forward to put the two changes at different location, which can weaken the vertical stiffness mutation of the structure and reduce the plastic deformation concentration, as shown in figure 10.



A. Before adjustment B. After adjustment

Figure 10. Plastic deformation concentration in 61th-62th floor.

2. Adjustment to new weak layer due to strengthening layers

Many super high-rise building structures use steel frame–RC core tube system, installed a few strengthening layers at different heights to improve cooperative working performance of the frame and the core tube. However, this arrangement often leads to new stress concentration and new weak layer.

Example 6: JiuLongCang building

The total construction area is 184,000 m², building height is 339m, a 72-floors tower with 3-floors basement, steel frame–RC core tube system. Two strengthening layers are located in 26-27 and 56-57 floor. Elasto-plastic dynamic time-history analysis is also conducted in Abaqus, which revealed that the concentrated plastic deformation appeared in the 28th floor and 58th floor (as shown in figure 12), the shear wall in that position also had severe damage. This is because the strengthening layers have great stiffness, while the layer just above it has a sudden decrease in stiffness, and this is detrimental to the structure's seismic performance. The suggestion is that change the stiffness in several continuous layers above the strengthening one through adjusting the structural arrangement. All of this is to avoid the emergence of new weak layer in the local floor.



A. Effect drawing



B. Structure model

Figure 11. JiuLongCang building.

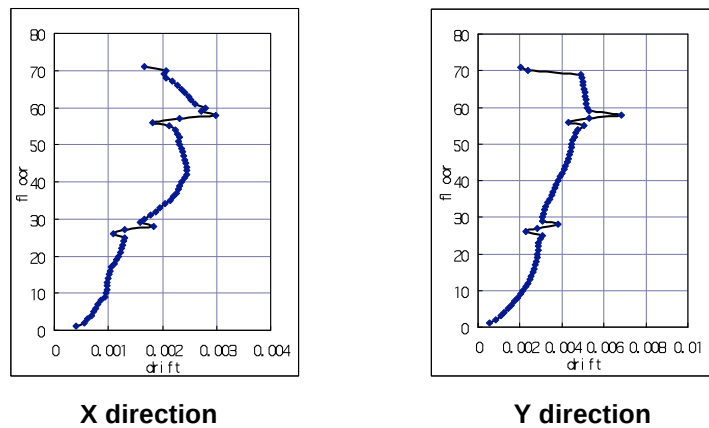


Figure 12. Inter-story drift curves.

3. Reinforced treatment for the premature cracking of the shear wall due to whiplash effect.

For super high-rise building, in general, the members' cross-section gradually reduced along the direction of the tower height and the material is also downgraded from bottom to top. This mode seems economic but not always on the safe side. In structural elasto-plastic time-history analysis, it is always found that the shear walls in the top layers have premature damage and the nearby columns bear tension crack. This is due to the reduction in pressure in the upper part of structure, and the components are more prone to bear tension under horizontal earthquake action. Especially if plane size is reduced at the top layers or having a high outstanding structure, the whiplash effect will lead to premature destruction of the surrounding components. Abaqus can always provide definite evidence in these questions for designers to modify their design.

Example 7: Changzhou TV building



Figure 13. Changzhou TV building.

Figure 14. The core tube has severe damage on top.

4. Optimization and adjustment of shear wall layout.

Excessive structural damage is not necessarily due to the deficient member, and blindly enlarge the cross-section will lead to greater seismic forces, thereby adding more reinforcement bar and make the design difficult in the end. Perhaps by adjusting the main components' (such as shear wall) layout, you can get twice the result with half the effort. Of course, such adjustments need to ensure that the overall stiffness of the structure meet all load conditions, not only the earthquake load requirements. In this case repeatedly adjustment is necessary, and then verify the results in Abaqus, go on doing this until meet the requirements.

3.5 Structural bars buckling and collapse simulation

Example 8: Sun Valley for EXPO Axis project

The project Expo Axis is the landmark building of Expo Shanghai 2010, and its structural safety depends largely on the security of the Sun Valley, which is the important component of the Expo Axis. Sun valley adopts a steel single-layer lattice structure system composed of triangular grids. The lengths of the top and bottom major axes of sun valley SV1-SV6 are about 90m、18m、60m、18m、60m、18m、70m、16m、60m、18m and 90m、21m. The bottom shape of SV4 is circular, while that of others is elliptical. The upper part of six sun valleys looks like an elliptic "horn". The height of six sun valleys is 42.00m (from -7.00m to 35.00m). The grid system has a complex shape and a large cantilever ranging from 21m to 40m. The grid system adopts welded steel tubular members with rectangular hollow sections. Rectangular members with solid sections are adopted at the top ring of the structure in order to enhance hoop effect. The lengths of structural members range from 1.50m to 3.50m, with heights of section ranging from 180mm to 500mm and width from 65mm to 120mm. Material of members and hollow joints is Q345B steel, while that of solid joints (joints of mast foot, pulling points of cables, membrane and top solid members) is G20Mn5 cast steel.

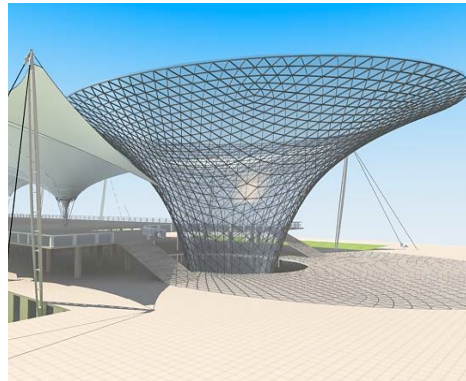


Figure 15. Effect drawing of Expo Axis. Figure 16. Effect drawing of Sun Valley.

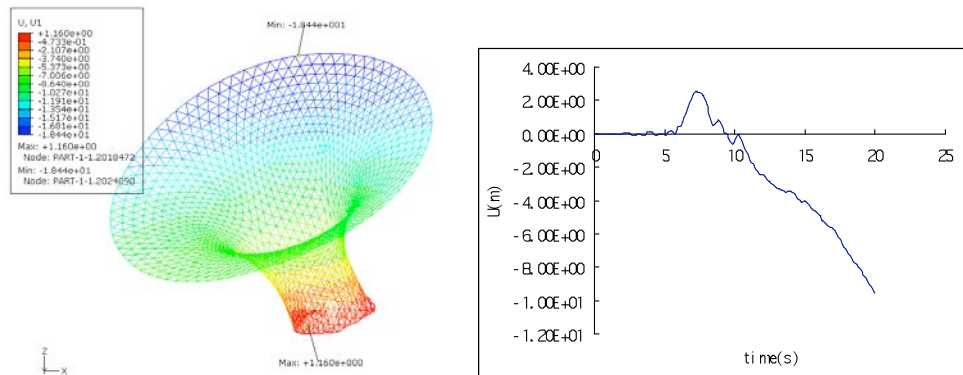


Figure 17. Moment of collapse immediately (under 12 times seismic action).
Figure 18. Displacement curve of peak node(under 12 times seismic action).

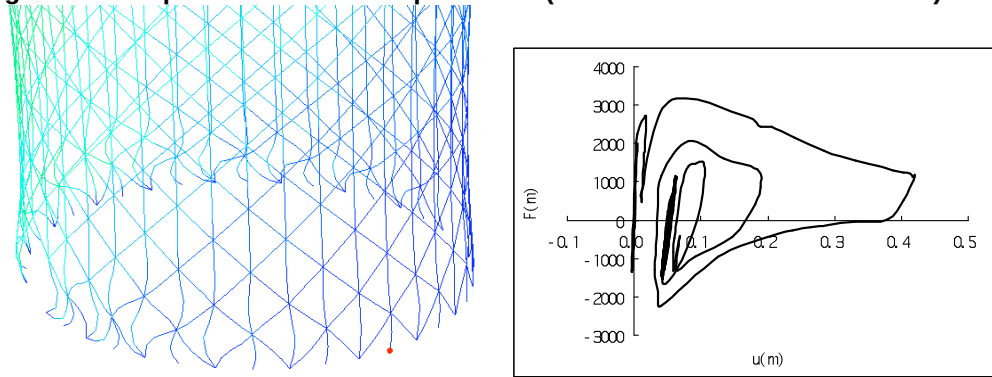


Figure 19. Bottom bars buckling.
Figure 20. Dynamic hysteresis curve of axial force and deformation

4. Brief summary

Abaqus has accumulated much successful experiences in the field of elasto-plastic dynamic time-history analysis. Using Abaqus the structural engineers can not only evaluate the building's performance under earthquake, but also do some appropriate optimization and adjustments to the program based on results of the analysis to improve the seismic performance of structures, and make the design more scientific and economical. Abaqus is also an open platform and the structural engineers' research work on this platform will make Abaqus software have better application in their field, prompting Abaqus' progress in the direction of simulating the real world.

5. References

1. Chopra, A. K and R. K. Goel, "A Model Pushover Analysis Procedure for Estimating Seismic Demands for Buildings," *Earthquake Engineering and Structural Dynamics*, 31(3),pp.561-582, 2002.

2. Lee, J., and G.L.Fenves, "Plastic-Damage Model for Cyclic Loading of Concrete Structures," Journal of Engineering Mechanics, 1998,124(8)
3. Manfredi G. "Evaluation of seismic energy demand, " Earthquake Engineering and Structural Dynamics, 30:485-499, 2001.
4. Abaqus, Inc, ABAQUS User Manual, V6.5.5, 2006.
5. Federal Emergency Management Agency. Guidelines and Commentary for the seismic rehabilitation of buildings. FEMA 273&274, 1998.
6. GB 50011-2001 ("Code for seismic design of buildings" of China)
7. JGJ 3-2002 ("Technical specification for concrete structures of tall buildings" of China, 2002.)
8. Seismic Evaluation Report of Shanghai ShenDu Building by Shanghai Xian Dai Architectural Design (Group) Co., Ltd; December, 2008.