The Transformation of Nonlinear Structure Analysis Model From NosaCAD to ABAQUS and PERFORM-3D

X.H. Wu & B. Zhang
Tongji University, College of Civil Engineering, China

SUMMARY:
NosaCAD, ABAQUS and PERFORM-3D are widely applied for nonlinear structure analysis. NosaCAD is developed with ObjectARX, second development tool of AutoCAD, and runs in the AutoCAD environment, so that NosaCAD possesses the powerful geometric processing function of AutoCAD. NosaCAD also has a set of modules to deal with the parameters for nonlinear analysis. ABAQUS and PERFORM-3D have powerful nonlinear analysis functions. However, it would be time-consuming to establish building structure model in the CAE of ABAQUS. PERFORM-3D can only transform the elastic parameters for analysis model from EATBS or SAP2000. In this paper, two transformation modules are developed for transforming the nonlinear analysis model from NosaCAD to ABAQUS and PERFORM-3D. The generated models can be calculated in ABAQUS or PERFORM-3D directly. An example model is transformed and the nonlinear time history analysis of the model is carried out in these three pieces of software.

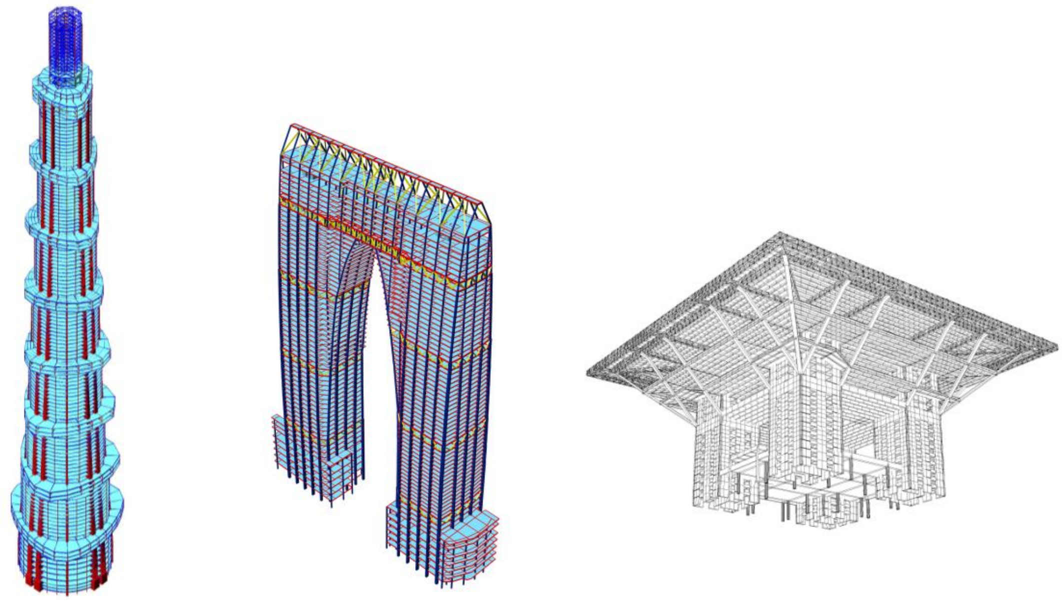
Keywords: Elasto-plastic time history analysis, Finite element method, Seismic performance

1. GENERAL INSTRUCTIONS

In the recent two decades a large amount of tall buildings have been constructed over the world and especially in Mainland China. In order to make the design unique and add beauty to cities, many new buildings adopt novel architectural styles, such as the up-down reversal number 4 shaped Shanghai International Design Centre (Lu *et al*., 2009) and the wheel-shaped Icon Hotel in Dubai (Berahman, 2010). However, irregularity and complexity of structures are inevitable for these special buildings. This requires structural engineers to entirely understand how these structures behave, especially in future earthquakes. Besides theoretical and experimental investigation, with the development of software and personal computer, nonlinear analysis for complex irregular buildings has gradually come to maturity. Earthquake engineering is relying more and more on nonlinear analysis as a tool for evaluating structural performance.

NosaCAD, ABAQUS and PERFORM-3D are widely applied for nonlinear structure analysis (Lu *et al*., 2009, Chen *et al*., 2011) (Fig. 1.1). NosaCAD is developed with ObjectARX, second development tool of AutoCAD, and runs in the AutoCAD environment. The powerful geometric processing function of AutoCAD can be used to established and edited structure analysis model. ABAQUS has powerful nonlinear analysis functions. However, CAE, the pre-processor of ABAQUS, is not developed for building structure model establishment especially, it would be time-consuming to establish analysis model in CAE. The analysis model of PERFORM-3D can be transformed from EATBS or SAP 2000, however, only the parameters for elastic analysis are transformed from EATBS or SAP 2000 and the parameters for nonlinear analysis would be calculated and input in further step.

In this paper, two transformation modules are developed for transforming the nonlinear analysis model from NosaCAD to ABAQUS and PERFORM-3D. Though NosaCAD has the function for nonlinear static and dynamic analysis, in some circumstances two or more analysis results from different software are needed for complement or comparison. For NosaCAD can form the parameters for nonlinear analysis, such as, moment-curvature parameter, fiber section parameter and shear wall parameter, these parameters also can be used for the model of ABAQUS and PERFORM-3D. The transformation module transforms the data of NosaCAD model, including geometry, material, load and nonlinear parameter into the data form of ABAQUS or PERFORM-3D and forms the analysis models which can be calculated in ABAQUS and PERFORM-3D directly. Finally, a high-rise building model of NosaCAD is transformed to that of ABAQUS and PERFORM-3D and the nonlinear time history analysis for this building models are carried out in these three pieces of software. A comparison of the results obtained from NosaCAD, ABAQUS and PERFORM-3D is provided and shows agreement to a certain degree.



(a) Shanghai Tower (b) the Gate of the Orient (c) China pavilion for Expo 2010

Figure 1.1. Analysis model of structure

2. NONLINEAR FINITE ELEMENT MODEL

2.1 Nonlinear Finite Element Model in NosaCAD

Frame element model is consisted of three different stiffness segments, one of which is a linear elastic segment in the middle part of element, and other two are elasto-plastic segments at both ends of element (Fig. 2.1).

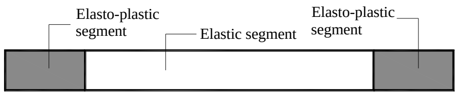


Figure 2.1. The frame element composed of three stiffness segments.

For the frame members mainly suffer bending moment, such as beam, bilinear and trilinear moment-curvature hysteretic models are adopted for the elasto-plastic segments of steel beams, concrete beams and steel reinforced concrete beams, respectively. The trilinear moment-curvature hysteretic curve is shown as Fig. 2.2. Considering columns, including inclined columns, bear bending moments in two directions as well as dynamic axial force, fiber model is employed to describe the nonlinear behavior in the elasto-plastic segments of column. The constitutive model of concrete in fiber model is shown as Fig. 2.3. The ideal elastic-plastic constitutive model, yield hardening taken into account, is adopted for steel and rebar. Fiber model is also adopted to simulate the elasto-plastic behavior of trusses. The relevant parameters of elastic-plastic analysis of the elements, such as cross-section parameters of fiber model and bending moment-curvature curve parameters are generated by NosaCAD according to sectional geometric parameters, material parameters and reinforcement.

The flat shell finite element, which is composed of diaphragm and plate, is used for shear wall and slab. The flat shell finite element model possesses rotational degrees of freedom in diaphragm, so that coupling beam element can be connected to shear wall with compatibility of deformation. In the nonlinear shell element, only the nonlinear property of diaphragm is taken into account and the plate is regarded as linear. For the nonlinear diaphragm, an orthogonal anisotropic concrete model based on equivalent uniaxial stress and strain relationship is adopted (Darwin and Pecknold, 1976) together with a biaxial strength envelope (Kupfer and Gerstle, 1973), meanwhile rebar is supposed to disperse in the element in certain directions according to the reinforcement. The concrete hysteresis curve of equivalent uniaxial stress-strain relationship is the same as that for fiber model (Fig. 2.3), and furthermore, the influence of the stress in orthogonal direction is considered. The ideal elasto-plastic model, taking into account the yield hardening, is still adopted for the reinforcement. It is assumed the smeared cracking is initiated once the tensile strength is reached and the second crack is permitted to appear orthogonal to the first one.

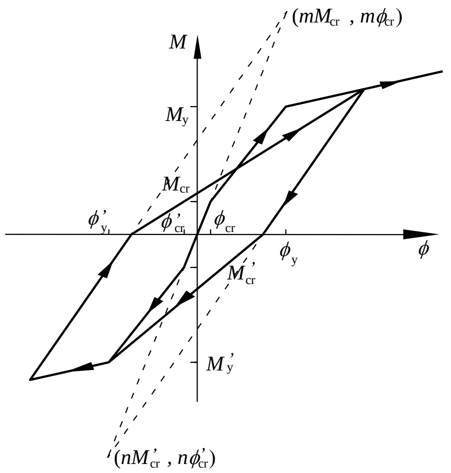


Figure 2.2. Trilinear moment-curvature hysteretic model

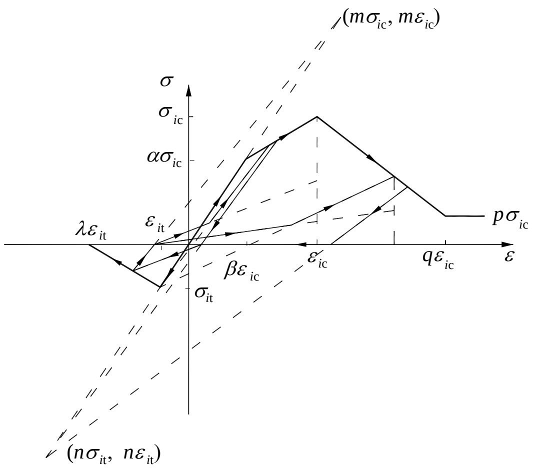


Figure 2.3. Concrete constitutive model

2.2 Nonlinear Finite Element Model in ABAQUS

Frame element is modelled with a first-order 3D Timoshenko beam element (B31), in which the transverse shear deformation is allowed. The beam section is divided into an array of fibers or section points, at which beam elements response are calculated and outputted. For space beam element with rectangular profile, 25 section points are considered by default in ABAQUS. The linear, reduced-integration, quadrilateral shell element (S4R) is used to model shear wall and slab, in which in-plane bending and shear and out-of-plane bending can be simulated simultaneously. Numerical integration is performed at a number of section points through the shell thickness to calculate the stress and strain independently at each section point. By default, ABAQUS uses five section points through the thickness of a homogeneous shell, which is sufficient for most nonlinear design problems.

The simulation of reinforcements can be achieved by two methods. One is to model each reinforcement bar separately. In this way, reinforcements should be embedded into concrete elements through node coupling to make them work together with concrete. The other is to treat the rebar as a layer with a constant thickness equal to the area of reinforcing bar. Here, the former one is used to model reinforcements in wall and slab, and the latter one is used to model reinforcement in beam and

column. The isotropic bilinear kinematics hardening model is used for reinforcement and steel, in which Bauschinger effect is considered. During the cyclic loading, no degradation is developed.
The damaged plasticity model is used to simulate the behaviour of concrete in the core wall, which used concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behaviour of concrete. Fig. 2.4 shows the mechanical response of concrete under cyclic loading (tension–compression–tension), where w_c and w_t are the compressive and tensile stiffness recovery factors respectively to control the recovery of compressive and tensile stiffness upon load reversal.

Since there are no proper concrete and reinforcement bar material model available for B31 element in current ABAQUS edition, user material subroutines were developed and incorporated into ABAQUS via VUMAT subroutine. This user material uses the uniaxial concrete and steel constitutive model same as that of NosaCAD (Fig. 2.3).

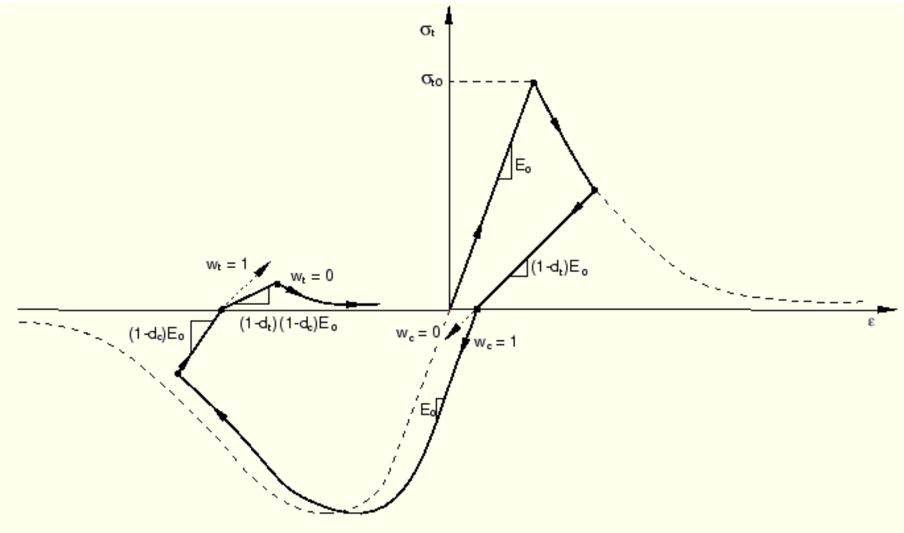


Figure 2.4. Uniaxial load cycle (tension–compression–tension) assuming w_c=1 and w_t=0.

2.3 Nonlinear Finite Element Model in PERFORM-3D

Same as that in NosaCAD, the moment-curvature hysteretic models are adopted for the frame members, which mainly suffer bending moment, and the fiber model is employed to describe the nonlinear behavior for the frame members, which bear bending moments in two directions as well as dynamic axial force. Unlike the elasto-plastic frame member in NosaCAD, which is consisted of two elasto-plastic segments and one elastic segment in the middle regularly, the elasto-plastic frame member in PERFORM-3D can be compounded of elastic segments and elasto-plastic segments in arbitrary formation. Beam-column element is usually end plastic region model or multi-segment plastic region model based on different segments assembly. The bilinear and trilinear moment-curvature hysteretic models are adopted for the elasto-plastic segments of steel beams, concrete beams and steel reinforced concrete beams, respectively. Macro layered element is adopted in PERFORM-3D to simulate shear wall component. One dimensional fiber element is used for simulating the compression-bending effect, while nonlinear or linear shear model is used for the shear effect in plane and elastic model is used for the bending, shear and torsion effect out of plane.

PERFORM-3D provides a uniform action-deformation relationship and hysteresis loop for almost all the nonlinear components in PERFORM-3D, such as concrete material constitutive for fiber, steel material constitutive for fiber, moment-curvature hysteretic relationship for frame element sections, etc. The intent of the PERFORM-3D action-deformation relationship, with points Y, U, L and R, is to capture the main aspects of the behavior, namely the initial stiffness, strain hardening, ultimate strength and strength loss (Fig. 2.5). The main intent of the PERFORM-3D hysteresis loop is to capture the dissipated energy (the area of the loop). This area is affected by stiffness degradation under cyclic loading. By the different parameters set, the certain action-deformation relationship and hysteresis loop are defined for actual inelastic behavior.

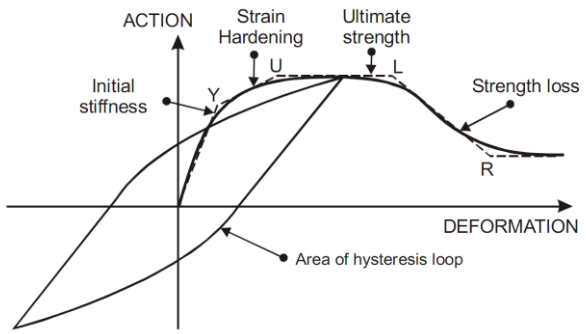


Figure 2.5. Main aspects of inelastic behaviour in PERFORM-3D

Steel constitutive model with bulking or non-bulking are available in PERFORM-3D. Non-bulking steel model is applied for reinforcement. Concrete constitutive model with Mander stress-strain relationship should be transferred in the action-deformation relationship of PERFORM-3D which can be determinate by 5 parameters and the strength loss is taken into account. The moment-curvature hysteretic relationship for frame element section should also be defined by the action-deformation relationship of PERFORM-3D, which can be determinate by 3 or 5 parameters.

As known that energy can be dissipated by nonlinear component under cyclic loading, and amount of the dissipated energy can be represented by the area of hysteretic loop. In PERFORM-3D, parameters of energy degradation are determinate by the maximum deformation and can be specified optionally (Fig. 2.6). PERFORM-3D gives the required energy degradation through adjusting the unloading-load stiffness, and the coefficient of energy degradation is taken as the ratio of the area of degraded and non-degraded hysteretic loop, which can be obtained from experiment and numerical simulation. In this paper, parameters of energy degradation are defined according to the degradation rule of unloading-load stiffness in Mander model, see Fig. 2.7.

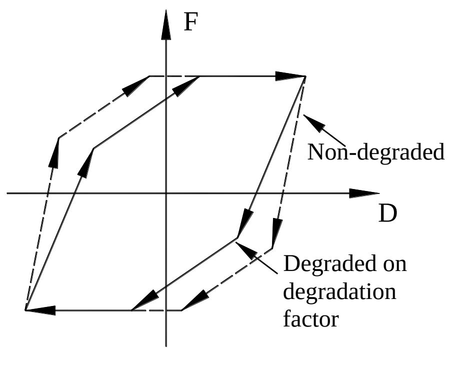


Figure 2.6. Hysteretic loop of energy degradation

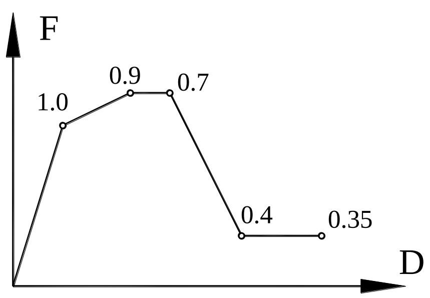


Figure 2.7. Degradation coefficients of concrete

3. MODEL TRANSFORMATION

The structural analysis model of different software has different data format, however, the data for describing structure properties, not only for elastic properties but also for nonlinear properties, has the same content in certain extent. NosaCAD is developed with ObjectARX, second development tool of AutoCAD, and runs in the AutoCAD environment. The powerful geometric processing function of AutoCAD can be used to established and edited structure analysis model. NosaCAD also has the function to form the parameters for linear and nonlinear analysis. If the data can be shared by other software, the efficiency of elasto-plastic analysis in those pieces of software can be improved remarkably.

3.1 Model Transformation from NosaCAD to ABAQUS

ABAQUS has text input file, the INP file. INP file has fixed text format. The data of finite element model, such as node coordinate, material properties and element properties, can be transformed from NosaCAD and written in the format of INP file directly. Because of the difference of nonlinear models between NosaCAD and ABAQUS, some changes and modifications of model parameters are needed.
Frame members are modelled using a first-order 3D Timoshenko beam element (B31), and has only one integral section along the element. In order to reflect the different deformation and nonlinear stiffness along the element, one frame member should be divided into several B31 elements, normally 6 to 10 segments. It is difficult to deal with the line load on frame member in ABAQUS and the line load will be converted into node loads, which are added on the nodes at the segment ends.

The reinforcement in wall and slab is modelled with each reinforcement bar separately, which is embedded into concrete elements to make them work together with concrete. Since ABAQUS/Explicit does not support the method for reinforcement bar embedded in frame element, the reinforcement in beams and columns is treated as a steel box, of which each edge has equivalent area of the reinforcement bar, see Fig. 3.1.

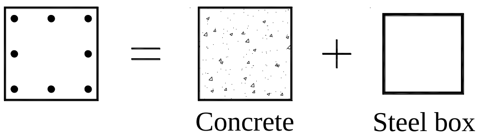


Figure 3.1. Reinforcement model for frame element

3.2 Model Transformation from NosaCAD to PERFORM-3D

Unlike ABAQUS, there is no text file for data input in PERFORM-3D. The binary data input file has to be formed in transforming from NosaCAD to PERFORM-3D.
The elastic parameters of finite element model can be transformed directly from NosaCAD and written in the binary format of PERFORM-3D input file. The nonlinear parameters, such as the parameters for moment-curvature, the parameters for fiber section, which are generated by NosaCAD, also can be shared by PERFORM-3D and transformed into the data format of PERFORM-3D directly. NosaCAD has the arbitrary cross-section editor and can generate fiber model of irregular section conveniently, see Fig. 3.2.

Since the difference of wall type between NosaCAD to PERFORM-3D, the former one belongs to micro model and the latter belongs to macro model, some parameters should be regenerated. The parameter for fiber section of wall can be generated according to the wall thickness and reinforcement of NosaCAD model. In NosaCAD and ABAQUS, the wall element has rotational stiffness in plane at node, and PERFORM-3D wall element has no such stiffness, the embedded rigid beam should be added in NosaCAD environment, so as to coordinate the deformation between the wall and coupling beam in PERFORM-3D model, see Fig. 3.3.

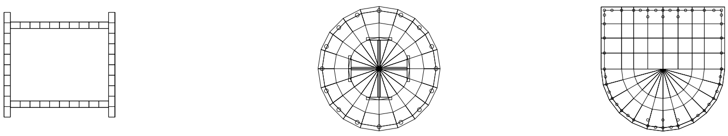


Figure 3.2. Fiber model of irregular section

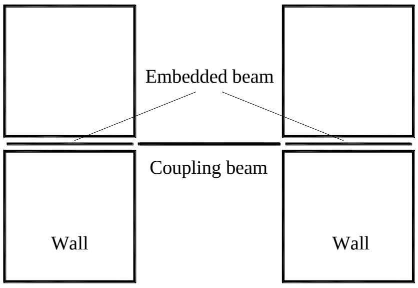


Figure 3.3. Connection between wall and coupling beam in PERFORM-3D

4. MODEL TRANSFORMATION AND ANALYSIS OF EXAMPLE

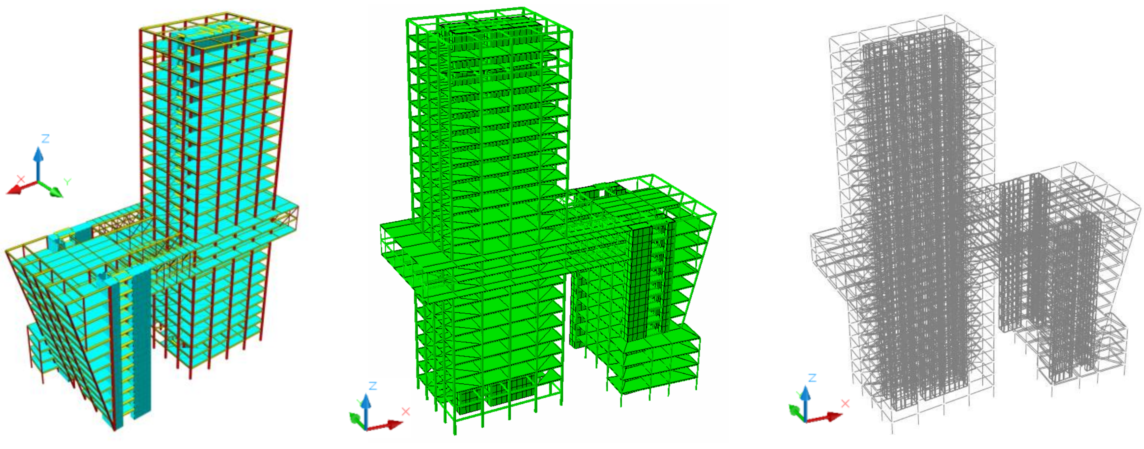
A high-rise multi-tower connected hybrid structure, the Shanghai International Design Centre (SHIDC), is used as the model transformation example. SHIDC is an office building which was constructed for the centennial anniversary of Tongji University in 2007. For its significant purpose, prestigious Japanese Architect Tadao Ando was invited to design the SHIDC. He decided to apply an overturned Arabic number 4 as the main elevation, see Fig. 4.1., which created a challenge for structural engineers to realize the design. A reinforced concrete core wall-steel frame (RCC-SF) hybrid structure was employed.

The structure consists of a Main tower, an Annex tower and a 4-story podium adjacent to the Annex tower. The Main tower is 25 stories with a height of 99 m, while the Annex tower is 12 stories with a height of 48 m. Both towers are RCC-SF systems. There are 7.5 m-span cantilever floors at the middle height of the Main tower from story 11 to story 13. Five inclined columns (the inclined angle is 15 degree) in the east of the Annex tower support cantilever beams at each story. The connecting corridor consists of 17.5 m steel truss spans, which are used to rigidly join the two towers between story 11 and story 12.

Figure 4.1. Architectural visualization of SHIDC building

On the benefit of the graphical editing function of AutoCAD, the complex analytical model of the structure can be efficiently established. Analytical model of the structure is composed of nonlinear elements of beam, column, truss and shear wall. Microscopic element model, flat shell element, is adopted for shear walls. The structural model is comprised of 6401 nodes and 9564 elements in all (Fig. 4.2(a)). The elements includes 23 elements of truss, 3453 elements of frame, 5949 quadrilateral flat shells and 139 triangular flat shells. By the transformation modules, the models for ABAQUS and PERFORM-3D analysis are generated, which can be used for analysis in ABAQUS and

PERFORM-3D directly. Fig. 4.2(b) and Fig. 4.2(c) show the model in ABAQUS and the model in PERFORM-3D respectively. Detailed shake table model testing of this building was also performed by a working group at the State Key Laboratory of Disaster Reduction in Civil Engineering at Tongji University, China (Lu *et al*., 2009).



(a) NosaCAD model (b) ABAQUS model (c) PERFORM-3D model

Figure 4.2. The model of structure

4.1. Natural Vibration Characteristics

The results of modal analysis calculated by different software are listed in Table 4.1. Because of the difference between the two towers, the fundamental vibration mode has torsion composition.

Table 4.1. Natural vibration period of structure

Order	Period (s)				Description
	NosaCAD	ABAQUS	PERFORM-3D	Test	
1	2.29	1.95	2.34	2.57	Translation in Y with torsion
2	1.30	1.14	1.44	1.71	Translation in X
3	1.12	0.98	1.24	1.21	Torsion
4	0.71	0.62	0.78	0.73	Second translation in Y with torsion
5	0.58	0.50	0.64	0.59	Second translation in X
6	0.40	0.36	0.51	0.47	Second Torsion

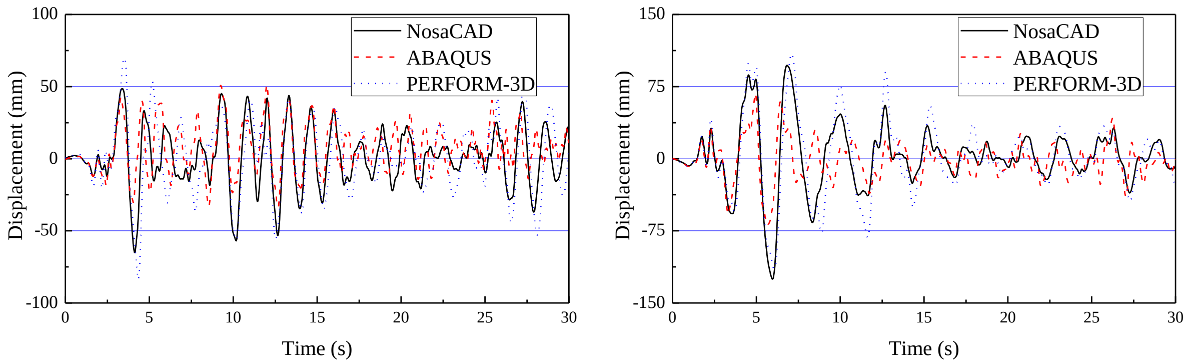
4.2 Nonlinear Time History Analysis

The El Centro record from the California Imperial Valley earthquake on 18 May 1940 was selected for elasto-plastic time history analysis. As Shanghai belongs to the seismic zone of intensity 7 (roughly equivalent to a modified Mercalli intensity of 7), the peak ground acceleration (PGA) corresponding to earthquake of major level is specified to be 0.200g. During the analysis, the earthquake records were inputted in three principal directions simultaneously (the N–S record is inputted in the X direction) with the PGA ratio of 1:0.85:0.65 (X:Y:Z). A damping ratio of 0.04 for SRC frame–RC core wall structural system was adopted.

There are two methods for integration of the equations of motion in common, one is the implicit method and the other is the explicit method. In an implicit dynamic analysis, a larger scale nonlinear equilibrium equation must be solved at each time increment, while in an explicit dynamic analysis, displacements and velocities are calculated in terms of quantities that are known at the beginning of an increment, therefore, the no larger scale nonlinear equilibrium equations need to be solved, which means that each increment is relatively inexpensive compared with the increments in an implicit integration scheme. However, iteration routine is not allowed in explicit dynamic analysis, and much smaller time step is required to guarantee the accuracy and convergence. With the development of the technique of parallel computation and the application of multiple cores CPU in personal computer, a very larger scale equilibrium equation can be solved within seconds. Therefore, implicit analysis method is also attractive for very large problems. In this paper, the implicit analysis method is adopted for time history in NosaCAD and PERFORM-3D, while the explicit analysis method is adopted for time history in ABAQUS.

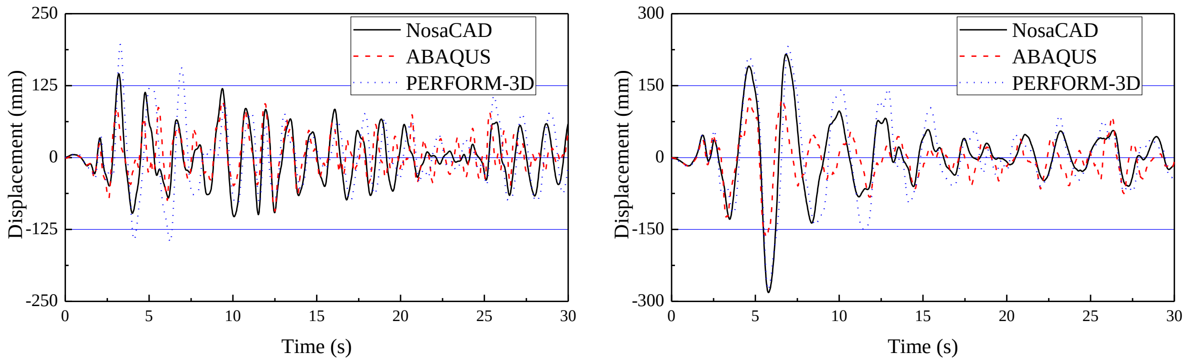
Normally, when the structure encounters the earthquake, static load such as gravity and service load has already acted on the structure. So, prior to the nonlinear dynamic time history analysis, a nonlinear static analysis was performed to obtain the initial stress state in structure members, which serves as the initial state of nonlinear dynamic analysis. According to the static analysis results, the structure totally weights 2.81×10^{5}kN.

Fig. 4.4 and Fig. 4.5 show comparison of the displacement response histories of two towers, which are calculated by NosaCAD, ABAQUS and PERFORM-3D respectively. It can be found that the displacement responses of NosaCAD and PERFORM-3D are closer than that of ABAQUS either in amplitude or in step.



(a) *x* direction (b) *y* direction

Figure 4.3. Displacement time history of the Annex tower.



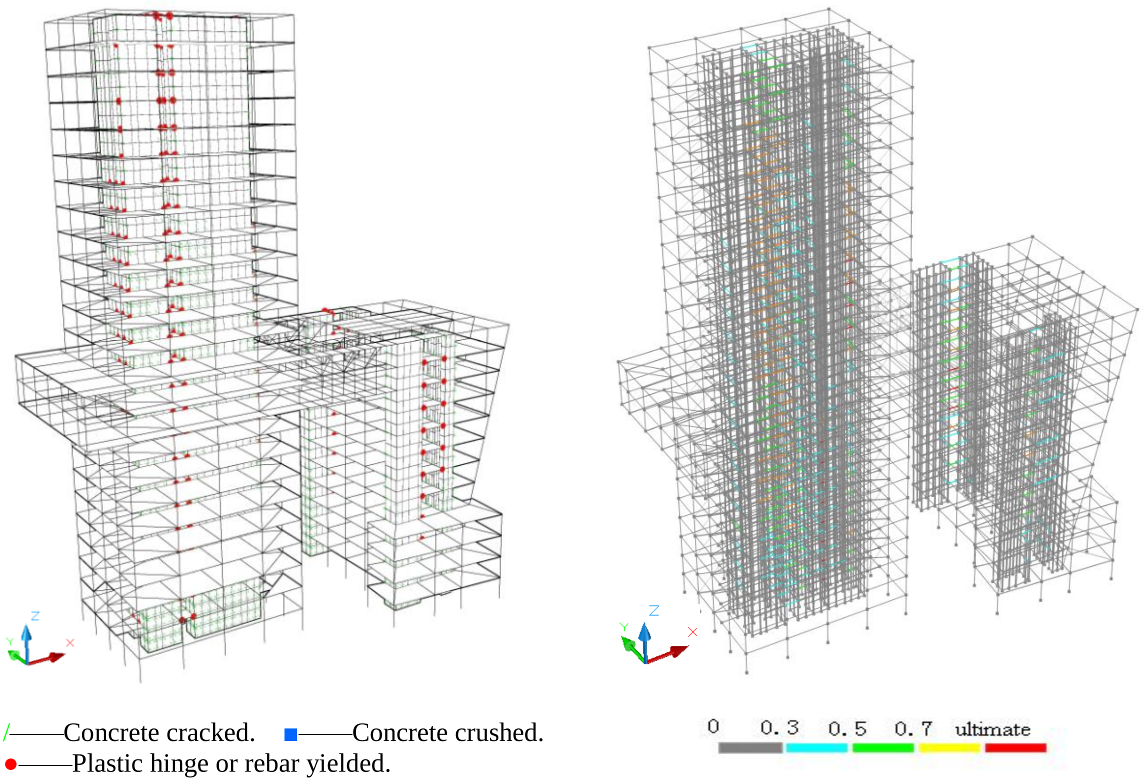
(a) *x* direction (b) *y* direction

Figure 4.4. Displacement time history of the Main tower.

It can be found from damage patterns of the structure (Fig. 4.5) and the animation of structure response in NosaCAD (http://www.nosacad.com/fxsl4.htm) that the plastic deformation mainly generated in the coupling beams of core wall in both towers. Some beams connecting the outer frame and core wall yielded and cracks occurred at the lower part of the core wall. The damage patterns obtained from ABAQUS and PERFORM-3D are nearly same as that of NosaCAD.

5. CONCLUSIONS

With more and more complex irregular buildings being constructed, structure engineering is relying more and more on nonlinear analysis as a tool for evaluating seismic performance of structures. For the complex response of the irregular building, in some circumstances more than one piece of software should be employed to carry out the nonlinear time history analysis. The NosaCAD has sophisticated function to establish nonlinear model. By the transformation, the elastic and plastic parameters generated by NosaCAD also can be shared by structural analysis software ABAQUS and PERFORM-3D, so that the efficiency of nonlinear analysis is improved markedly. The example of elasto-plastic model for a high-rise multi-tower connected hybrid structure showed the feasibility of the model transformation from NosaCAD to ABAQUS and PERFORM-3D. The analysis results from these models are agreement to a certain degree.



(a) NosaCAD (b) PERFORM-3D

Figure 4.5. Damage patterns of the structure

REFERENCES

Berahman, F. (2010). Performance-based Seismic Valuation of the Icon Hotel in Dubai United Arab Emirates. *The Structural Design of Tall and Special Buildings*. Published online: 7 Dec 2010. DOI:10.1002/tal.688

Chen, X.W., Han, X.L., Luo, F.N. and Wu, S.G. (2011). Fiber Element Based Elastic-Plastic Analysis Procedure and Engineering Application. *The Twelfth East Asia-Pacific Conference on Structural Engineering and Construction. Procedia Engineering.* **14:** 1807–1815

Darwin, D. and Pecknold D.A. (1976). Analysis of RC Shear Panels under Cyclic Loading. *Journal of the Structural Division* ASCE **102:2,**355-369.

Kupfer, H.B. and Gerstle, K.H. (1973). Behaviour of Concrete under Biaxial Stresses. *Journal of Engineering Mechanics Division* ASCE **99:4** 853-866.

Lu, X.L., Chen, L.Z., Zhou, Y. and Huang, Z.H. (2009). Shaking Table Model Tests on a Complex High-rise Building with Two Towers of Different Height Connected by Trusses. *The Structural Design of Tall and Special Buildings* **18**: 765–788. DOI:10.1002/tal.460

Lu, X.L., Su, N.F. and Zhou, Y. (2011). Nonlinear Time History Analysis of a Super-Tall Building with Setbacks in Elevation. *The Structural Design of Tall and Special Buildings*. Published online: 21 June 2011. DOI:10.1002/tal.717