Accurate inelastic structural analysis schemes for high-rise structures under earthquake excitations base on ABAQUS platform

J. Teng, Z.H. Li, A. Li, Y.B. Han & Z. Cai
Shenzhen Graduate School, Harbin Institute of Technology, P.R. China

SUMMARY:
With the further development of seismic design theory of building structure, the performance design and assessment of tall building require more detailed and reliable analytical results from the phase of the seismic structural analysis. As its important basis, the more elaborate and accurate finite element schemes in inelastic structural analysis have attracted great attention in the applications and researches. In this paper a conversion program, which can convert MIDAS (MIDAS, 2010) finite element model to ABAQUS (ABAQUS, 2010) finite element model, is developed by Python programming language based on parametric finite element modelling, thus the modelling process in ABAQUS is much simplified, and a new numerical strategy to model nonlinear damage behaviour of RC beam-column members based on level of material is presented by analysing section of fiber beam column element with the uniaxial damage constitutive relations of concrete. The nonlinear analysis efficiency based on ABAQUS platform for high-rise structures will be improved by using this analysis strategy. Converting several typical structural models with the program, the comparison of nonlinear analysis results shows that the program is effective and efficient.

Keywords: Conversion program, ABAQUS, MIDAS/Gen, Nonlinear analysis, Fiber beam-column element

1. INTRODUCTION

With the development of structural seismic design theory, the clear and quantitative index has been proposed to implement the seismic design and assessment of high-rise structures (Teng, 2006; Zhu, 2004). As the important basis for research applications, the precise elastic-plastic finite element analysis models and methods get attention of the engineering and scientific research field.

Dynamic elastoplastic analysis has become an important method to verify the seismic performance objectives of high-rise structures (Feito, 1995; Hatzigeorgiou, 2002). General-purpose finite element software ABAQUS, which has extensive material and element types, is becoming one of the main tool for structural dynamic elastoplastic analysis. But now the modelling efficiency for high-rise structures is low, which restricted its wide application in the field of structural engineering. Parameterized finite element modelling is a new concept, which containing the following three strategy, (a) modelling based on CAD platform, (b) parameterized modelling plug-in based on scripting language, and (c) from finite element model information to command-line file, directly face the solver.

In this paper, the conversion program of structural model is developed based on the third strategy above-mentioned, which the finite element parameters can be imported from MIDAS to ABAQUS. The process can guarantee that the conversion of nodes, elements, materials, cross sections, loading and other information are true, and then the efficiency of nonlinear time history analysis is improved.

To obtain elaborate nonlinear analytical results, namely to analysis macro member at a stress-strain level rather than do it with any common-used simplified macro inelastic model of member (Li, 2004; Li, 2006), the present paper adopts section integration scheme as the elaborate strategy for massive structural inelastic seismic analysis, and inelastic stress-strain constitutive are used at section integrating points. This strategy is implemented through secondary developments on ABAQUS, which breakthroughs the bottleneck of the extensive use of the software in massive structural inelastic seismic analysis.

2. BASIC STRATEGY OF PROGRAM DEVELOPMENT

This developed conversion program is completed based on the data exchange of the MIDAS/Gen mgt file and ABAQUS inp file, the workflow is shown in Figure 1. The structural finite element information is extracted from the mgt file, the information is set by the user through GUI and is written in inp file.

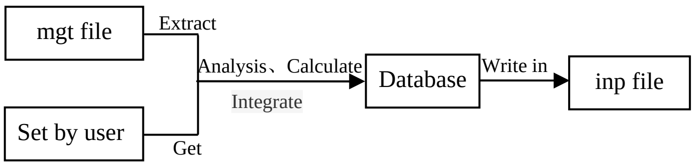


Figure 2.1. Workflow of conversion program

Finite element information, including elements, materials, sections, loads, floors and constrains is extracted from mgt file. The input of members reinforcement amount, analysis step and output are accomplished through using the GUI interface of this conversion program. The ABAQUS/CAE module is avoided, when submitting ABAQUS solver to calculate, which improve the efficiency of the pre process.

3. IMPLEMENTATION OF CONVERSION PROGRAM

3.1. Mgt File Data Structure

All the information of the structure of MIDAS model is contained in mgt file, which contains lots of internal data in clear arrangement. Part of the data shows that the general information of the structural model and the other part is specifically for MIDAS platform, which focused on the analysis in general information of structural model, such as nodes, elements, materials, cross sections, loads and so on. And the main data of general information is arranged and listed in Table 3.1. Because the data structure in Table 3.1 completely shows the characteristics of the finite element model, so the conversion process will extract data according to the Table below.

Table 3.1. Mgt File Data Structure

Data item	Group1	Group2	Group3	Group4	Group5
Node	Node No.	Coordinates			
Beam element	Element No.	Material No.	Section No.	Node	Degree
Wall element	Element No.	Material No.	Section No.	Node	
Material	Material No.	Material type	Grade		
Section	Section No.	Section type	Size		
Floor	Floor No.	Floor elevation			
Constraint	Constraint No.	Constraints			
Node load	Node No.	Load direction	Magnitude		
Line load	Element No.	Load type	Coordinate type	Location	Magnitude
Floor load	Load name	Floor No.	Load zone		
Pressure load	Element No.	Load type	Coordinate type	Magnitude	

3.2. Inp File Data Structure

The data composition of inp file is more flexible, its data structure is shown in Table 3.2. All information in structure model is reflected in the inp file, which formulate from this framework. And it can be directly submitted to the ABAQUS solver to model analysis or dynamic elastoplastic analysis.

Table 3.2. Inp File Data Structure

Data item	Group1	Group2	Group3	Group4	Group5
Node	Node No.	Coordinates			
Beam element	Element No.	Node			
Shell element	Element No.	Node			
Mass element	Element No.	Node	Magnitude		
Beam reinforcement	Element No.	Node			
Beam set	Set name	Element No.			
Shell set	Set name	Element No.			
Floor set	Floor set name	Element No.			
Beam section	Beam set name	Material name	Section type	Section size	1 axis vector
Shell section	Shell set name	Material name	Section size	Reinforcement	Arrangement
Material	Material name	Elastic	Plastic		
Constraints	Location	Type			
Analysis step	Definition	Output	Output set		

3.3. Software Programming

Python script language is used to write the program to realize the model data transfer and conversion. The conversion process is divided into eight parts in this menu: (1) To extract node and element information from mgt file to store in the database, to calculate the length of the beam element, area of the plate element to process the following analysis. (2) To read all the members reinforcement information stored in the database. (3) To extract material and cross section information from the mgt file to store in the database, according to the cross-section and reinforcement information to add reinforcement to the beam element, and to add rebar layer to shell element. (4) To extract beam loading, floor load and pressure load, dead load and live load from the mgt file. (5) To write the information of nodes, elements, materials, cross sections, reinforcement and transforming quality in inp file. (6) in Assemble Instance, to define the nodes and elements sets of output variable. (7) To create the ABAQUS analysis step, you can set the dynamic elastoplastic analysis or modal analysis. (8) To check the rules of writing. This conversion program menu is shown in Figure 2.

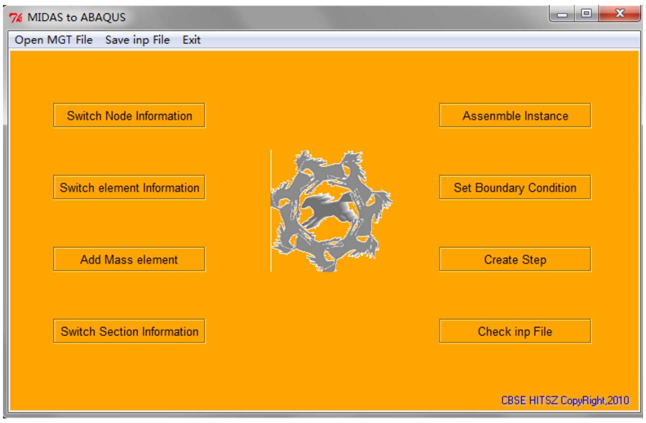


Figure 3.1. Conversion program menu

4. DEVELOPMENT AND APPLICATION KEY POINTS OF VUMAT

ABAQUS software support real-time cross-section integration of the beam-column element in dynamic analysis, which can capture real-time characteristics of the section material point. Simpson integral method is adopted, usually the integral points is 5.

The strain increment of the current and last increment coming from ABAQUS/Explicit analysis results is received in user material subroutine (VUMAT). The integration point stress values of the current step are returned and updated according to the user-defined stress-strain hysteresis algorithm. The section force and element internal force vector is obtained through integrating stress values, and the unbalanced node force is calculated automatically.

In theory, VUMAT is used in spatial beam-column element, including B31, B31OS, B32, B32OS, PIPE31 and PIPE32, which can consider the shear deformation and support the dynamic cross-section integration. The material parameters of concrete plastic damage constitutive model is inputted with command *User Material, including stress, strain and damage index. The solution-dependent state variables is inputted with command *Depvar, which is used in the corresponding field descriptions written to the output database.

5. CONVERSION PROGRAM VERIFICATION

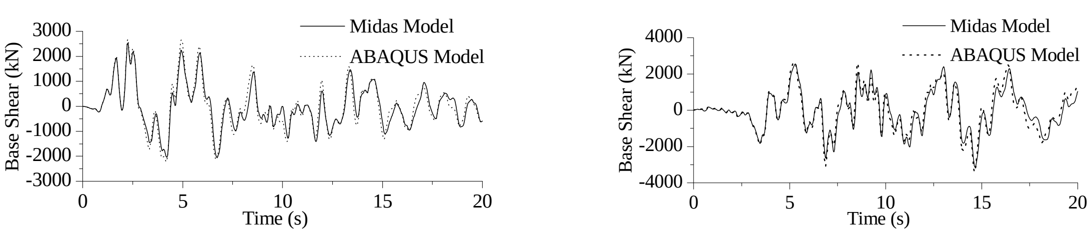
A 20-story frame structure model, with 3m story height, is built in MIDAS. Steel reinforced concretes are used for corner columns, and there are partial openings in slab. The MIDAS model of the structure is shown in Figure 3(a), and the ABAQUS model converted by conversion program is shown in Figure 3(b).

The comparison of mode analysis results of two models are listed in Table 5.1, which shows that the mass and period of each vibration mode is in good agreement. Then, dynamic elastic analysis is executed for two models. El Centro and Taft ground motions are elected as earthquake effects, with 35gal of PGA input in *X* direction. The comparisons of base shear force and top displacement time-history (see Figure 4 and Figure 5) show that the results of two models are relatively close.

The comparative results of modal analysis and elastic time history analysis used in this conversion program show that total mass and six-order vibration modes of ABAQUS model agree well with MIDAS model, and base shear and displacement is also more consistent. It verified that the conversion of loads, elements, materials and sections information is correct. Conversion process is stable and reliable.

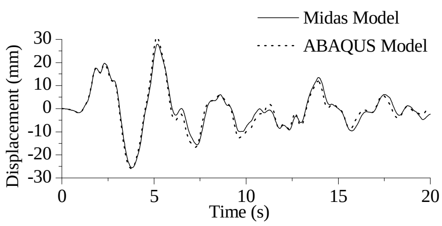
(a) MIDAS model (b) ABAQUS model

Figure 5.1. Frame structure model

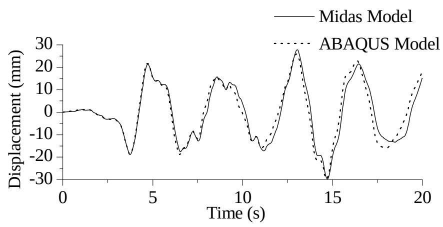


(a) El Centro wave (b) Taft wave

Figure 5.2. Comparison of base shear force time-history under earthquake



(a) El Centro wave



(b) Taft wave

Figure 5.3. Comparison of top displacement time-history under earthquake

Table 5.1. Comparison of Mode Analysis Results

	Mass(t)	1-st Period(s)	2-nd Period (s)	3-rd Period (s)
MIDAS	45285.9	3.8991	3.7968	3.3464
ABAQUS	45325.6	3.8957	3.7286	3.2622
Error	0.06%	0.10%	1.80%	2.52%

6. ELASTO-PLASTIC DYNAMIC ANALYSIS OF HIGH-RISE STRUCTURE

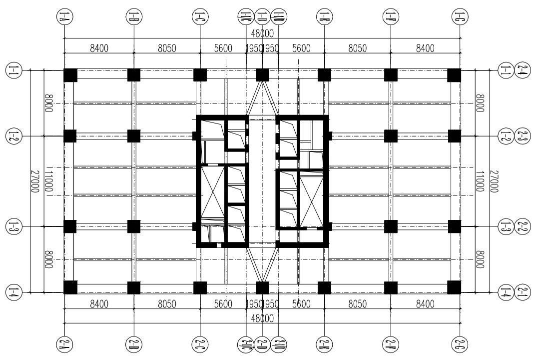
6.1. The Engineering Situation

A certain engineering structure is consisted of two super high-rise tower buildings, a two-floor basement and a spatial latticed structure, shown in Figure 6(a). The total height of each tower building is 154.5 meters (roof structure of 150.4 meters). Each tower has 37 floors that the 14th floor and 27th floor are equipment floors. The plane of the standard floor is a rectangle, which is shown in Figure 6(b). Since the architectural planes and function of the two tower buildings are identical (symmetrical), the form and arrangement of structure are also identical. What’s more, in this case, the towers and the spatial latticed structure are connected with sliding support connections or no connections (confirmed during the spatial structure designing); therefore the spatial structure’s potential influence to the towers is concerned in a form of load.

6.2. Analysis Model

The MIDAS TO ABAQUS model conversion program, which is developed by the authors, is used to convert the MIDAS model to ABAQUS model. With this software, the geometric modeling, the information of structure members, reinforcement, structure floors and the load could be converted quickly to ABAQUS model, which is shown in Figure 7. Fiber model is applied in the reinforced concrete beams, columns, and the steel reinforced concrete columns, while the shear walls and coupling beams are modeled by shell element in ABAQUS. In a comparing with other software, which is shown in Table 6.1, the results model analysis results from SATWE (CABR, 2010) and MIDAS agree well, which roughly verifies the validity of the ABAQUS model.


(a) Architectural effect drawing



(b) The plane of the tower standard floor

Figure 6.1. A Certain Engineering Structure

Table 6.1. Results of Model Analysis

	Period (s)					
	1^{st}	2^{nd}	3^{rd}	4^{th}	5^{th}	6^{th}
SATWE	3.1179	2.9754	1.8240	0.8975	0.7405	0.6306
MIDAS	3.0770	3.0485	1.5502	0.7899	0.7234	0.5359
ABAQUS	2.9771	2.9458	1.6351	0.8071	0.7267	0.5650

6.3. Results of Elesto-plastic Dynamic Analysis

6.3.1. Deformation and Base Shear Force

According to the result of Safety Evaluation, 3 earthquake waves, which are 1 artificial wave and 2 natural waves, are used in the analysis. When comparing to the Code (Mhuc P.R.C., 2002) Response Spectrum, which is shown in Figure 8, the result of the artificial one is more similar to the Code so that its result should be important. The peak acceleration of the earthquake wave is 220gal according to the Code and the Safety Evaluation.

Under rare earthquake excitations, the peak inter-story displacement angle, the top displacement and the base shear force are listed in Table 6.2, in which the peak displacement does not exceed the limit values (1/100) in the Code.

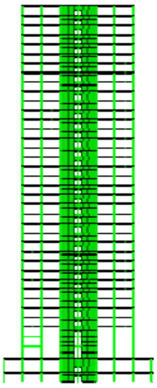


Figure 6.2. The whole model of the structure

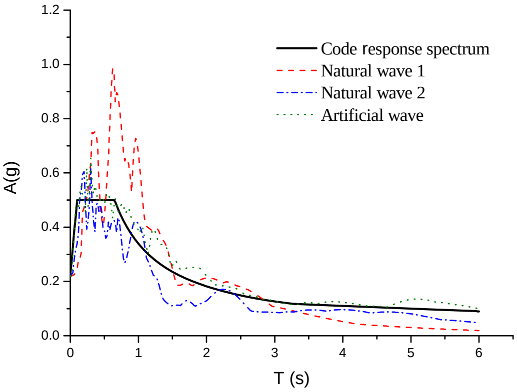


Figure 6.3. Response spectrum

Table 6.2. Displacement and Reaction Force

	Inter-story Displacement Angle	Maximum of Top Displacement (m)	Maximum of the Base Shear Force (N)
Artificial wave in X direction	1/165	0.476	8.785×10^{7}
Natural wave 1 in X direction	1/224	0.319	7.95×10^{7}
Natural wave 2 in X direction	1/200	0.485	8.768×10^{7}
Artificial wave in Y direction	1/235	0.443	7.821×10^{7}
Natural wave 1 in Y direction	1/292	0.297	6.27×10^{7}
Natural wave 2 in Y direction	1/225	0.463	9.35×10^{7}

6.3.2. Damage process under the earthquake in X direction

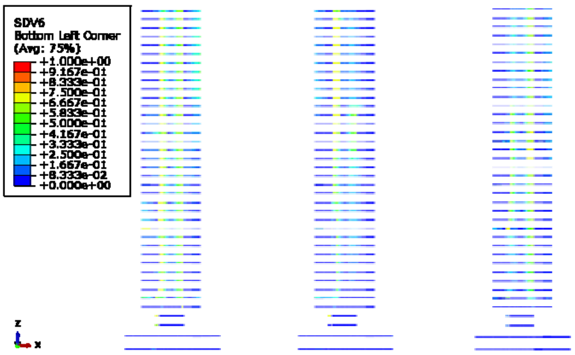
Under the earthquake excitations in *X* direction, damage of the frame beams in *X* direction showed up at the 10th second. At this time, damage mainly occurred near the structural transition layers in a form of bending failure in the end region. At the moment when the earthquake excitations stop, tension damage in the frame beams spread to the top of the structure and mainly in the end region of frame beams near the inner core. After all, the damage of the frame beams near the transition layers is more serious to that in the top of the structure.

Compression damage in the frame columns is negligible under earthquake, as shown in Figure 10. And the steel columns remain elastic with no plastic strain accumulated through the process, as shown in Figure 11. Therefore the frame columns show good work performance to ensure the vertical load transition in the structure during the earthquake action.

Under the artificial wave earthquake excitations, at the duration time of the 16th second, compression damage show up in the shear wall of the inner core in *X* direction near the transition layers while damage in the coupling beams in *X* direction is negligible until the duration time reaches 17 seconds. When the earthquake excitations stop, damage in the shear walls of inner core near the transition layer is relatively serious, which are shown in Figure 12 and Figure 14. Therefore, damage occurred in the coupling beams near the transition layers, but in the shear walls of the inner core, damage is limited, shown in Figure 13.

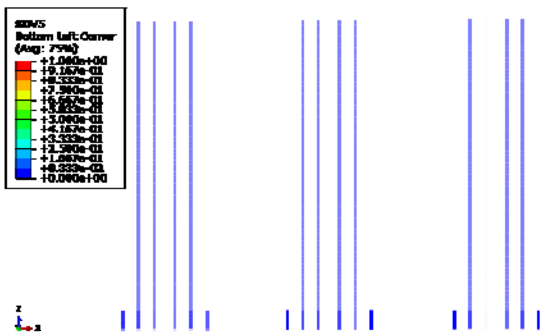
At the during time of the 12th second under the natural wave 1 earthquake excitations, damage partly show up in the outer shear walls in *X* direction near the transition layer while damage in the coupling beams of the inner core in *X* direction is negligible at that time. When the during time reaches 15 seconds, damage in the coupling beams occurred partly and damage in the shear walls started to spread. When the earthquake excitations stop, damage is found in the outer shear walls and coupling beams of the inner core near the transition layer, both in limited region, which are shown in Figure 12 and 14. Compression damage in the shear walls inside the inner core is negligible, shown in Figure 13.

At the during time of the 16.5th seconds under the natural wave 2 earthquake excitations, damage partly show up in the outer shear walls in *X* direction near the transition layer while damage in the coupling beams of the inner core in *X* direction is negligible at that time. When the during time reaches 19 seconds, damage in the coupling beams occurred in small region. When the earthquake excitations stop, damage is found in the outer shear walls and coupling beams of the inner core near the transition layer, both in limited region, seen in Figure 12 and 14. No remarkable compression damage is found in the shear walls inside the inner core, which is shown in Figure 13.



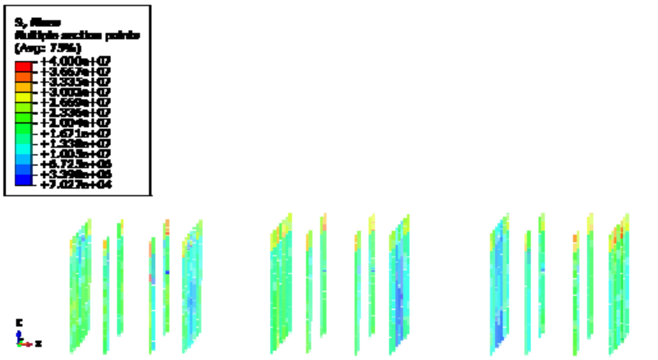
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.4. Tension damage in the frame beams in *X* direction



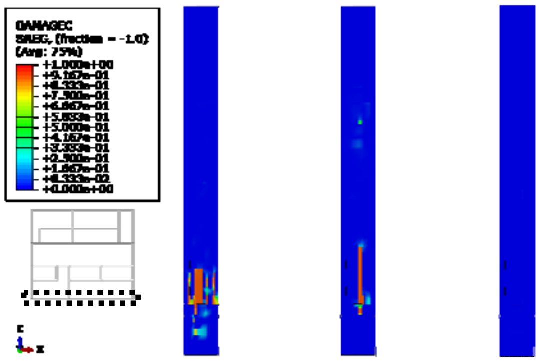
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.5. Compression damage in the frame columns in *X* direction



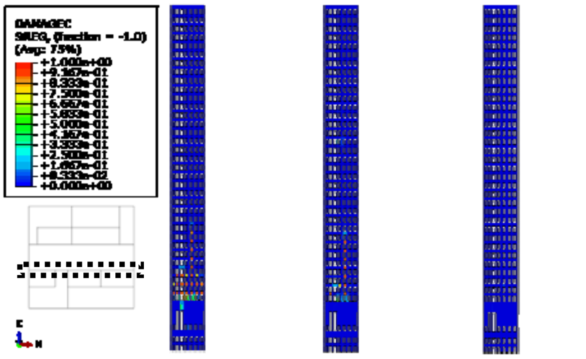
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.6. Plastic strain of the steel in floor 1-4 in *X* direction



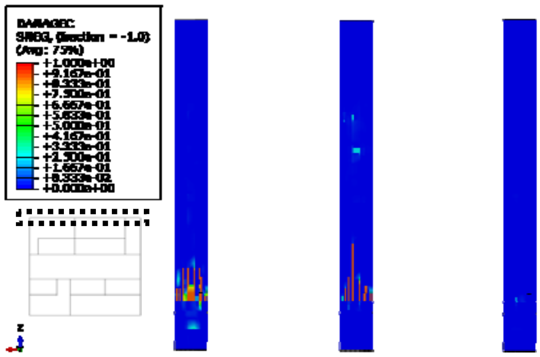
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.7. Compression damage of the shear walls in the dotted box in *X* direction



(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.8. Compression damage of the shear walls in the dotted box in *X* direction



(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.9. Compression damage of the shear walls in the dotted box in *X* direction

6.3.3. Damage Process under the Earthquake in Y Direction

Under the earthquake excitations in *Y* direction, damage of the frame beams in *Y* direction show up at the 10th second. At that time, damage mainly occurred near the structural transition layers in a form of bending failure in the end region. At the moment of the earthquake excitations stop, tension damage in the frame beams spread to the top of the structure and mainly in the end region of frame beams around the inner core of the upper structure. After all, the damage of the frame beams near the transition layers is more serious than that in the upper structure and the damage due to the artificial wave is more serious, which is shown in Figure 15.

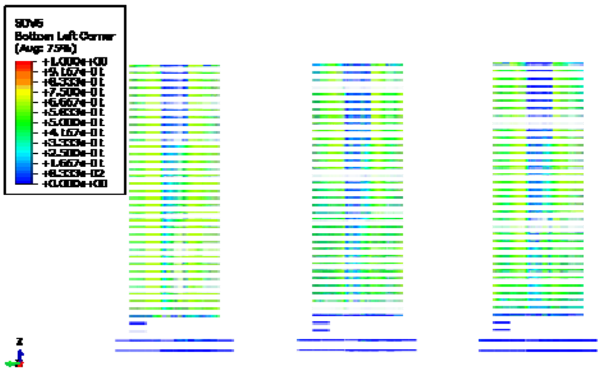
Compression damage occurred partly in a single frame column while most of the columns remained undamaged to ensure to the vertical load transition, which is shown in Figure 16. What's more, the steel column remained elastic, showing good performance, shown in Figure 17.

Under the artificial wave earthquake excitations, at the duration time of 7.5th second, compression damage show up in the coupling beams of the inner core in *Y* direction near the transition layers. When the earthquake stopped, the energy dissipation of coupling beams reach the limit and the compression damage spread to the shear walls near the transition layer, shown in Figure 18(a) and 19(a). Compression damage didn't occur in the shear wall of the inner core in *X* direction under the earthquake excitations.

At the during time of 8.5th second under the natural wave 1 earthquake excitations, compression damage show up in the coupling beams of the inner core in *Y* direction near the transition layers. When the earthquake excitations stop, the energy dissipation of coupling beams reach the limit and the compression damage spread to the shear walls near the transition layer, shown in Figure 18(b) and

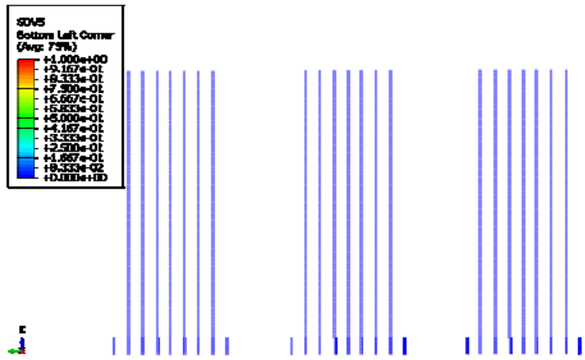
19(b). Compression damage didn't occur in the shear wall of the inner core in *X* direction during the earthquake.

Under the natural wave 2 earthquake excitations, when the during time reach 10.5 seconds, compression damage show up in the coupling beams of the inner core in *Y* direction near the transition layers. However compression damage is negligible in the shear walls and most of the coupling beams of the inner core in *Y* direction, shown in Figure 18(c) and 19(c). Therefore, during the whole earthquake, compression damage partly occurred in the shear walls only, which could transfer the vertical load efficiently.



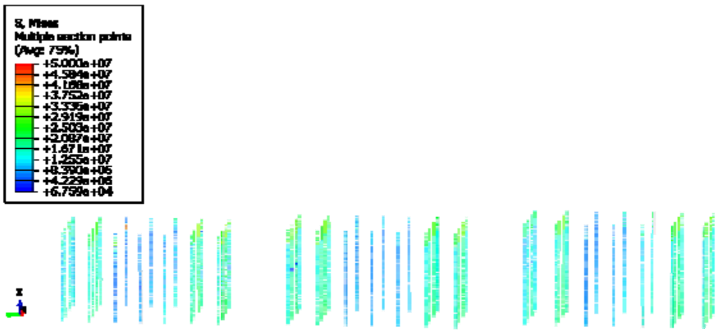
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.10. Tension damage in the frame beams in *Y* direction



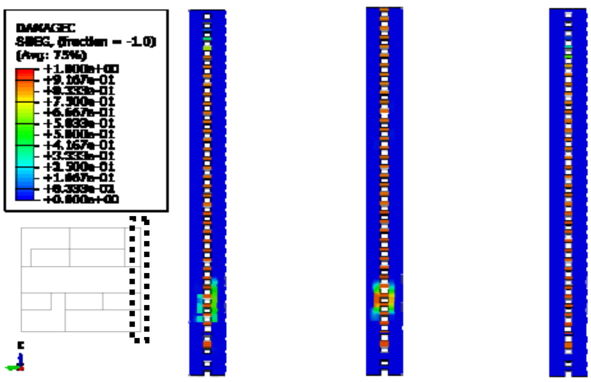
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.11. Compression damage in the frame columns in *Y* direction



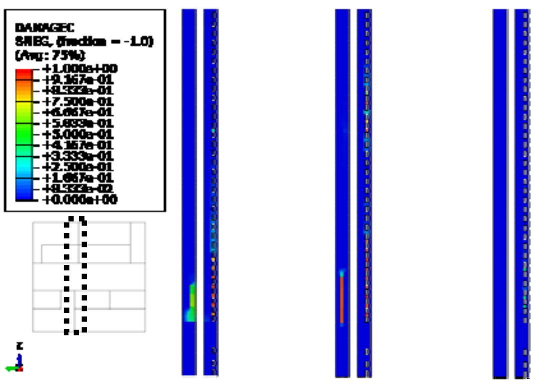
(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.12. Plastic strain of the steel in floor 1-4 in *Y* direction



(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.13. Compression damage of the shear walls in the dotted box in *Y* direction



(a) Artificial wave (b) Natural wave 1 (c) Natural wave 2

Figure 6.14. Compression damage of the shear walls in the dotted box in *Y* direction

7. CONCLUSIONS

With the rapid development of elastic-plastic analysis of structures, the efficient and accurate method of modeling and analysis strategy becomes more and more critical. Using this conversion program the structural model is converted perfectly from MIDAS/Gen to ABAQUS to achieve the rapid establishment of the structural ABAQUS model. Dynamic elastic-plastic time history analysis procedure is presented for engineering and designers. The comparative results show that the conversion process is practical and efficient. Using this analysis strategy, the efficiency of dynamic elastic-plastic analysis for the complicated building structures based on the ABAQUS platform is improved.

ACKNOWLEDGEMENT

The authors acknowledge the financial support from the National Natural Science Foundation of China under Grant No. 50938001 and 51008048.

REFERENCES

ABAQUS Inc. (2010). ABAQUS User Manual **v6.10**.

MIDAS IT Co., Ltd. (2010). MIDAS/Gen User Manual **v7.8**

CABR TECHNOLOGY Co., Ltd. (2010). SATWE User Manual **v2010**

Feito F., Torres J.C., Urena A. (1995). Orientation, simplicity, and inclusion test for planar polygons. *Computers & Graphics Art*. **19:4**,595-600.

Hatzigeorgiou G.D., Beskos D.E. (2002). Dynamic elastoplastic analysis of 3-D structures by the domain boundary element method. *Computers&Structures*. **80:3-4**,339-347.

Kilar V., Koren D. (2010). Simplified inelastic seismic analysis of base-isolated structures using the N2 method. *Earthquake Engineering and Structural Dynamic*. **39:9**,967-989.

Li B. Li H.N. (2004). Analytical method of elasto-plasticity for RC shear wall. *Earthquake Engineering and Engineering Vibration*, **24:1**, 76-81.

Li F.M. (2006). Parametrical Finite Element Method and Its Application. Chongqing University Press.

Mhuc P.R.C. (2002). Technical specification for concrete structures of tall buildings. China Architecture & Building Press.

Spacone E., Ciampi V., Filippou F.C. (1996). Mixed formulation of nonlinear beam finite element. *Computers & Structures*. **58:1**,71-83.

Teng J. and Li Z.H. (2006). Comparison of different elastic-plastic analysis method of complex high-rise structures under strong earthquake excitations. *4th International Conference on Earthquake Engineering*. Paper No. **118**.

Zhu J.J., Zhang P.J. et al. (2004). Pushover analysis for concrete structures of tall building. *Journal of Shangha University (English Edition)* , **8:3**,264-273.