Large-Scale Simulation of the Dynamic Behavior of Rein-Forced Concrete Buildings

Project Leader
Eizaburo Tachibana Frontier Research Center, Graduate School of Engineering, Osaka University

Authors
Yasunori Mizushima Division of Global Architecture, Graduate School of Engineering, Osaka University
Noriyo Ichinose Engineering Technology Division, JRI Solutions Ltd.
Eizaburou Tachibana Frontier Research Center, Graduate School of Engineering, Osaka University

In order to prevent urban disaster, it is very important to simulate the actual dynamic behavior of building structure suffering big earthquake. For this aim, usually, buildings are considered as simple model in which mass and springs are connected in series. However, these simple models are not enough to evaluate actual behavior of buildings on earthquake. In this report, more accurate models are used for FEM analysis of RC buildings. Concrete and steel bar are modeled by cubic solid elements and by beam elements. Standard mesh sizes of those elements are 10 cm. In this simulation, concrete piles and surrounding ground are modeled by similar way.

Keywords: Multi-scale Analysis, FEM, reinforced concrete, soil-structure interaction

1. Introduction

Until 1980's, reinforced concrete structures were not applied for high rise buildings (taller than about 20 stories buildings). Most of them were designed as steel structures. But by virtue of the recent development of high strength cement, many high rise buildings have been constructing as reinforced concrete structure. However, those buildings have not experienced the big earthquake.

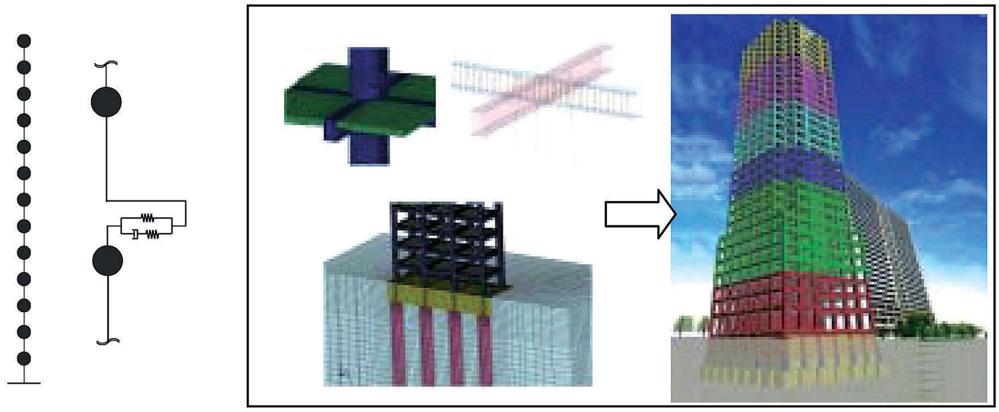
Therefore, it is very important to predict the actual dynamic behaviors of those buildings by numerical simulations. Usually, numerical simulations are executed by simple models. Typical frame models for static simulation are composed by 'linear elements'. One beam or one column is represented by one linear element. For dynamic simulations, 'mass and spring' models are used. (Shown as Fig. 1) One mass represents total mass of one floor. However, these simple models are not enough to explain the actual behavior of buildings on earthquake shown as Photo.1 or Photo.2.

In this report, more accurate models are used for FEM analysis.

Many civil engineers wanted to know the real behaviors of structures by using these accurate models. However, in this case the total number of variables (such as displacement of nodal points) will be larger than 10 millions. This number is out of scale for usual computers. So, until now, these desires were remain as only a dream.

Photo 1 Kobe earthquake (1995).

Photo 2 Chi-chi earthquake (1999).



(a) simple model

(b) accurate model

Fig. 1 Conceptual image of analysis model.

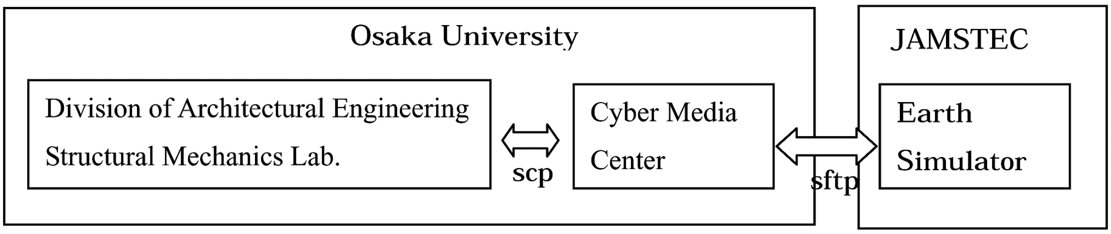


Fig. 2 Remote Use of Earth Simulator of JAMSTEC.

2. Remote use of Earth Simulator

In this report, numerical simulations were executed by using ES (Earth Simulator) of JAMSTEC (Japan Agency for Marine-Earth Science and Technology). The executing flow is shown as Fig. 2.

In order to use ES, high level of vectorizing and parallelizing were required from JAMSTEC. The finite element code of 'LS-DYNA', which is applied in this project, cleared well against this barrier. The original LS-DYNA was developed at Lawrence Livermore National Laboratory with presuming to use vector array processors named GRAY.

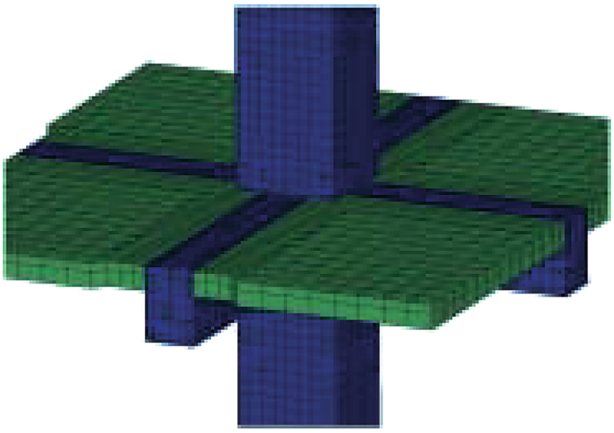


Fig. 3 Mesh division of concrete.

3. Material property

Cubic solid elements (10 cm × 10 cm × 10 cm) are used for concrete material. (Shown as Fig. 3)

General stress-strength relation of concrete under compression is expressed in Fig. 5(a). This curve is valid only for the case of 'uni-axial loading'. In other words, no constrains are given at side surface like as cylinder tests of plain concrete. However, actual concrete beams and columns is constrained or confined by steel bars. (Shown as Fig. 4)

Concrete behave, depending on intensity of confined compression, as plastic hardening or softening material shown as Fig. 5(b). Therefore 'PSEUDO_TENSOR model'[1], which is prepared in LS-DYNA is used for concrete model. This model is considered these three dimensional effects[2, 3].

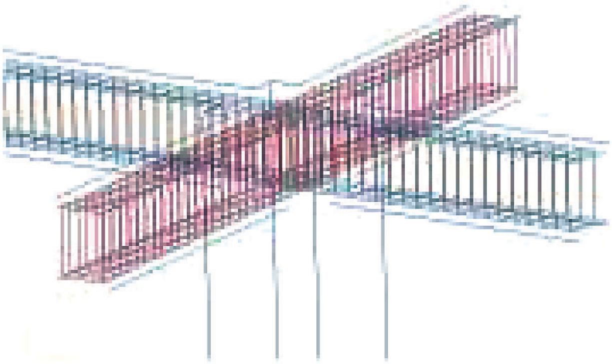
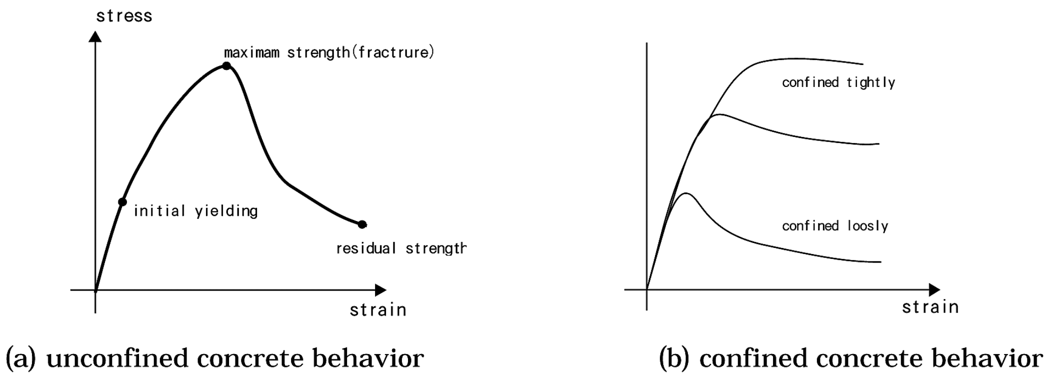


Fig. 4 Confined beam by steel bars.



(a) unconfined concrete behavior

(b) confined concrete behavior

Fig. 5 Stress strain curve of concrete material.

One-dimensional elements are used for steel bar. The stress strain curve is assumed as bi-linear type. the surrounding soil and the foundation including piles are modeled with elastic material.

4. Connecting steel bar with concrete

Mesh generating procedures are the most tedious works in the finite element method. And it may be the best way that the concrete element and the steel bar element can transfer their nodal forces at 'common nodal points'. However, it is not practical. How can we adjust the mesh size of concrete element to steel bar element which are so complicated like as Fig. 4? We need more elegant and automatic mesh generating way.

In this report, a conventional joining technique of following way is adopted.

<Simple joining technique>

Step-1: Mesh generation for both concrete element (solid model) and steel bar element (beam model).

Step-2: Find out 'nearest surface' of solid model for all nodal points of beam model.

Step-3: Connecting them with virtual beam like as shown in Fig. 6.

In this way, no adjusting treatments are needed in mesh generating procedure. But, if the length of the virtual beam is longer than solid scale (10 cm), proper changes of local mesh are needed.

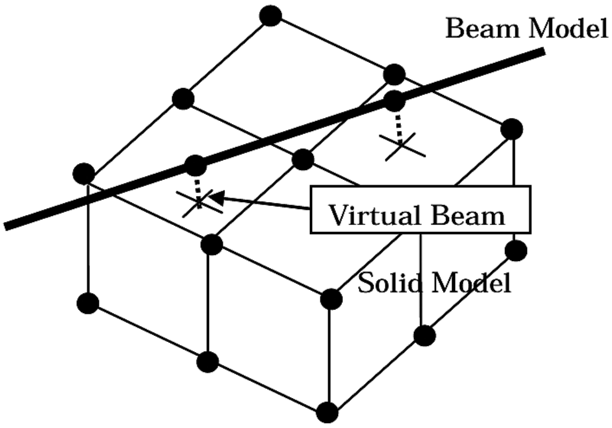


Fig. 6 Joining technique of concrete and steel bar.

5. Boundary Conditions of Soil and Structure

An image of full scale model is shown as Fig. 7. The outer surfaces of soil are assigned as no reflection boundaries against wave propagation. The boundaries between concrete piles and its surrounded soil are assigned as contact surfaces which allow slide and separation.

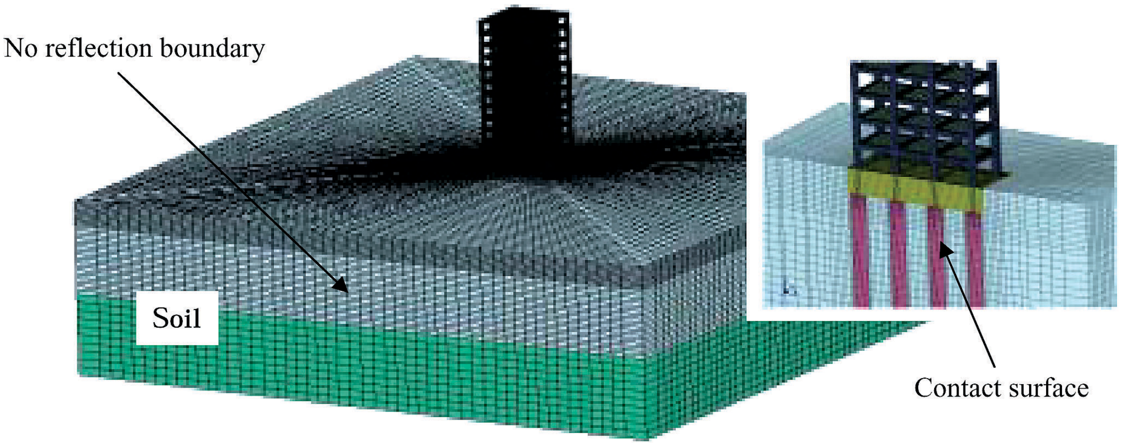


Fig. 7 Full scale model and boundary condition.

6. Configuration of upper structure

The outline of upper structure is shown as Fig. 8. It is three spans in the plan, and twelve floors in the elevation. All beams and columns are modeled as like as Fig. 3 and Fig. 4. Concrete slabs are modeled as homogeneous materials which have equivalent stiffness.

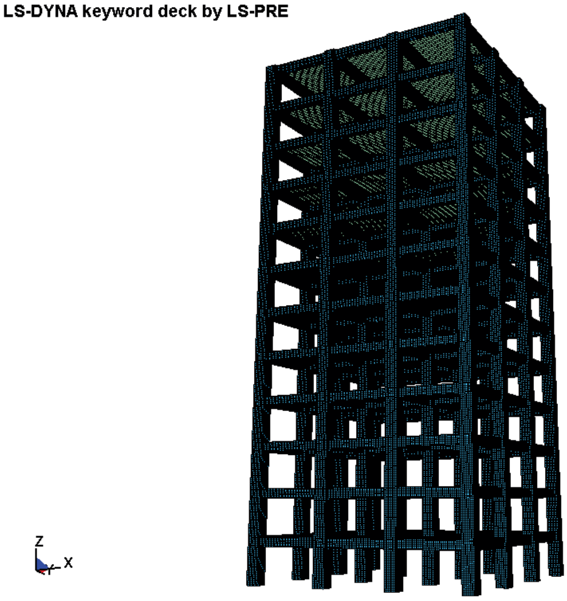
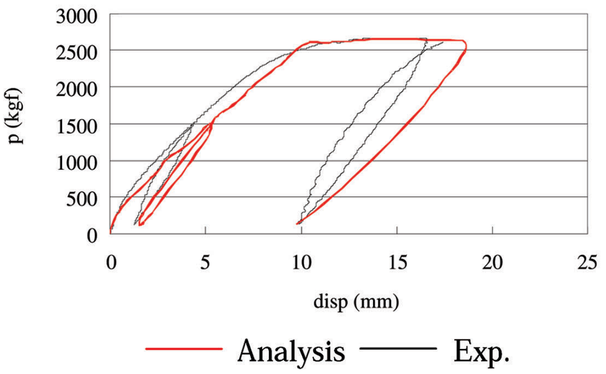
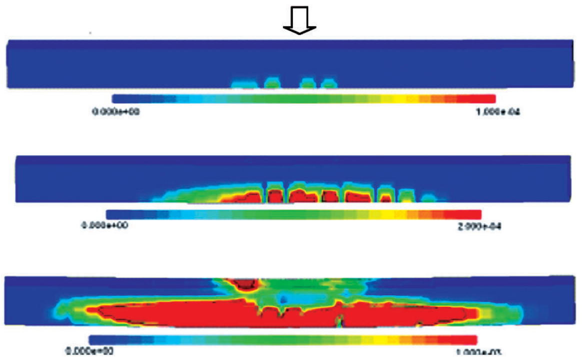


Fig. 8 Outline of upper structure.



(a) Load-Displacement curve



(b) Mises equivalent stress

Fig. 9 Test-1 (Simple beam test).

7. Pre-examination

Before earthquake response analysis, following three tests were executed.

Test-1: Comparison of simulated results and experimental results in case of simple beam. (Fig. 9)

Test-2: Two systems were simulated to know the effect of attenuation in free vibration caused by soil-structure interactions. (Fig.10) Those systems are a mass-spring-damper system, and an integrated mass-spring-damper system including three-dimensional soil model.

Test-3: Comparison of axial stress of column for two cases about joinning methods. 1) using the common node of concrete and steel bar. 2) Using virtual bar for connecting concrete and steel bar. (Fig.11)

As can be seen in Fig. 9-(a), the load-displacement curves of experimental result show good accordance with calculated curve. From these results, 'PSEUDO_TENSOR model' that is used for concrete is an adequate model. In Fig.10 the case of (One mass-spring-damper+3D soil), the amplitude of vibration are decreasing rapidly. This means the vibration energy of the upper structure is dispersed to the ground.

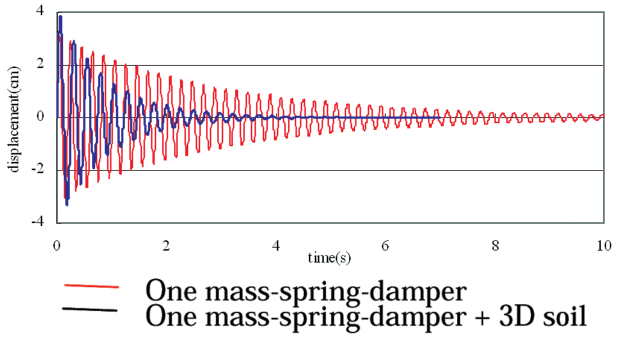
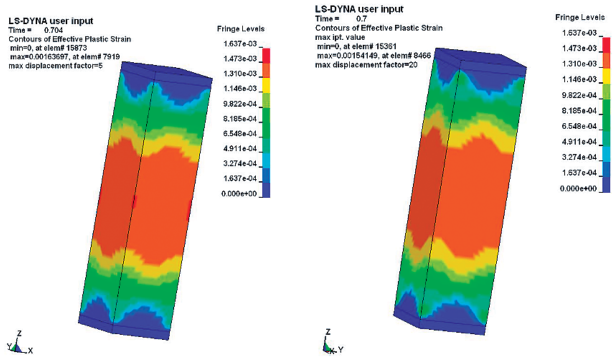


Fig.10 Test-2 (free vibration).



a) Using common nodal pont

b) Connected by virtual bar

Fig.11 Test-3 (Axial load for column).

8. Numerical simulation of upper structure and whole structure

Two cases were simulated.

Case-1: Only upper structure

Case-2: Whole structure including the upper structure, foundation, piles and surrounding soil.

In the Case-1, the acceleration of JMA-Kobe (NS) that was recorded at 1995 was used as input data. Those data were given at the bottom points of column of the first floor.

In the Case-2, an inputted underground wave was calculated from the JMA-Kobe (NS) wave by the one-dimensional fluctuation propagation theory.

These calculated data was given to the bottom nodes of the ground.

8.1 Stress and strain distribution for Case-1

Mises equivalent stress distributions and residual plastic strain distributions are shown in Fig.12 and Fig.13, respectively.

In Fig.12, the Mises equivalent stress grows larger in lower stories.

Obviously, this result may be caused by the gravity of the each structure. And, the residual plastic strain can be observed at joining parts of beams and columns.

8.2 Stress distribution for Case-2

The Mises equivalent stress distribution in Case-2 is shown as Fig.14. In this figure, all mesh lines of surrounding soil is neglected.

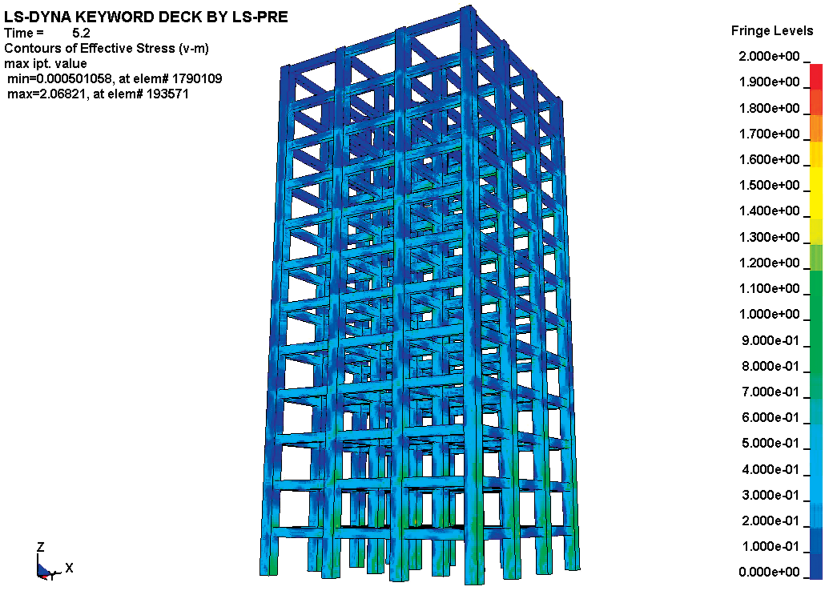


Fig.12 Mises equivalent stress distribution.

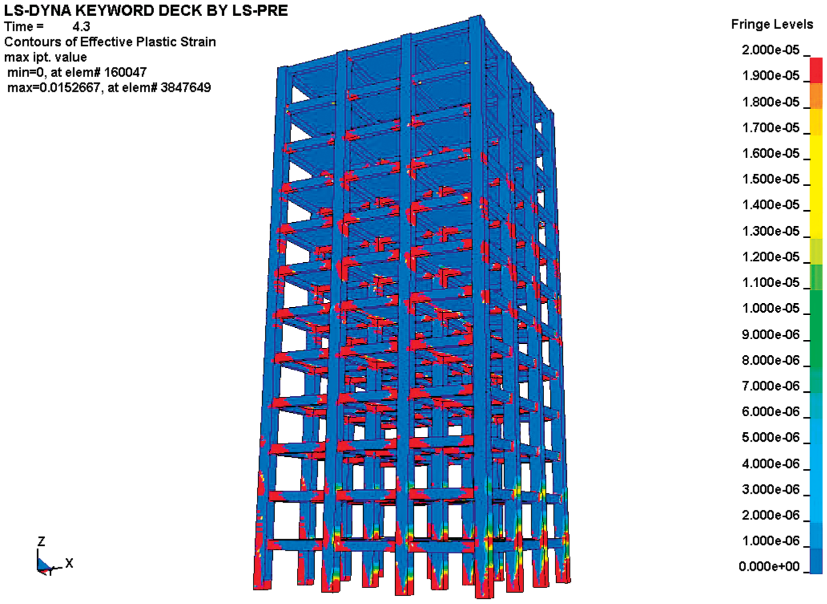


Fig.13 Residual strain distribution.

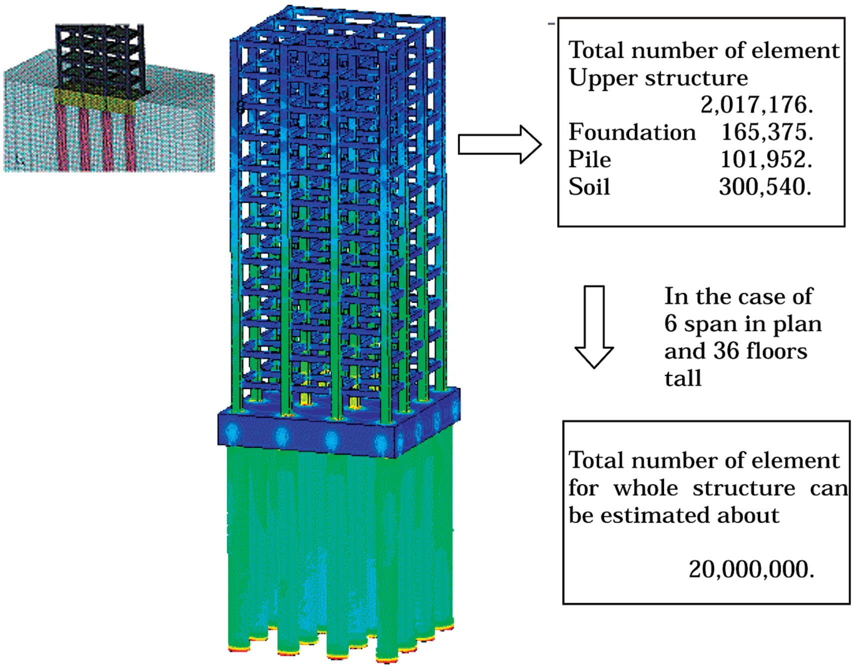


Fig.14 Mises equivalent stress distribution for whole structure.

The simulated time duration is stop at 0.4 second by the limitation of usable CPU time.

9. Conclutions

1) 'Remote using system' has developed. We can use Earth Simulator at Osaka University as if we are in the next room in ES center.
2) Validity of joining techniques of inserting virtual bar was examined, and it was proved that the method is very practical to generate mesh data.
3) Soil-structure interactions were checked by free vibration of two different models. One is the whole structural model including soil and foundation. Another is upper structure only which is fixed on rigid plate. As the results, the dominant natural period became longer, and amplitude of free vibration decreased rapidly. These results sustain the modelization.
4) Earthquake response analysis was carried out both for the upper structure and the whole structure. 'JMA-Kobe' wave was used as an input excitation. Stable results were gained. These results seem to be correct from the overall viewpoint.

However, in order to discuss detail, this research should be developed further in the next fiscal year. We have to establish large scale simulation system of tall buildings before big earthquake attack on metropolitan in Japan.

Acknowledgement

The authors acknowledge that many of cooperation by Dr. Tetsuya Sato (JAMSTEC), Dr. Kunihiko Watanabe (JAMSTEC), Dr. Shigenobu Hirose (JAMSTEC), Prof. Masao Toyoda (Osaka Univ.), Dr. Manabu Higashida (Osaka Univ.), and Mr. Heitaro Doi (JRI Sol.). Also the authors would like to thank all members of GASST PROJECT (Dr. Takehito Kitano (The Kansai Electric Power Co.), Mr. Shigeru Hikone (Arup Japan), Dr. Hiroaki Yokoyama (Konoike Construction Co.), Dr. Yoshinori Iwasaki (Geo-Research Institute), Dr. Naohiro Nakamura (Takenaka Co.)) for their valuable comments.

References

[1] LS-DYNA User's Manual Version 970 Vol.II (Livermore Software Technology Co.)
[2] L.E. Schwer, and Y.D. Murray, "A three-invariant smooth cap model with mixed hardening," Int. J. for Numerical and Analytical Method in Geomechanics, Vol.18, pp.657–688, 1994.
[3] K. Shibata, and E. Tachibana, "Numerical simulation for fracture of concrete filled circular steel tube subjected to the axial compression," J. Struct. Constr. Eng., AIJ, no.601, pp.167–174, March 2006.

鉄筋コンクリート造による高層建物の大規模3次元有限要素解析

プロジェクトリーダー
橘　英三郎　　大阪大学　フロンティア研究センター

著者
水島　靖典　　大阪大学大学院　工学研究科　地球総合工学専攻　建築工学部門
一ノ瀬規世　　日本総研ソリューションズ
橘　英三郎　　大阪大学　フロンティア研究センター

1. 共同研究の目的

超高強度コンクリートの普及により、50階程度の建物まで鉄筋コンクリート造（RC造）の設計が可能となり、マンションなどを中心に近年急速に普及しつつある。通常、地震時における動的挙動は、建物の層数と同じ数の質点が縦方向に繋がったモデル（串だんご型モデル）を用いて解析がなされる。しかし、局所的な破壊、それによる全体崩壊、さらには杭や地盤による3次元的な影響、などを知るうえでは、そうした串だんご型モデルの解析では不十分で、本来は、地盤や、鉄筋も含めた柱や梁、杭などを3次元的にモデル化して動的解析を行い、耐震安全性を評価すべきである。

そのためには1千万元程度以上の未知数からなるモデルを解く必要が生じ、通常のコンピュータでは解析不可能となる。

本共同研究では、そうした能力を有する地球シミュレータを用いて解析するとともに、共同研究の最終年度までには、高層RC造の耐震安全性検討のための新たな道筋の構築を目指す。

2. 解析環境

2.1 利用形式

阪大の研究室から阪大のサイバーメディアセンターを介して、地球シミュレーターをリモートバッジ形式で利用する。

2.2 解析ソフト

FEM陽解法のLS-DYNAを用いた。本コードはベクトルプロセッサーの使用を前提としており、地球シミュレータとの整合性を有している。

2.3 材料モデル

コンクリートは3軸応力下の弾塑性を扱えるモデルを用いる。鉄筋は、バイリニア型弾塑性モデル、地盤は弾性モデルとする。又、床スラブは鉄筋の効果を考慮した均質なコンクリートとする。

2.4 形状およびメッシュ分割

上部構造は幅3スパン奥行き3スパン、高さ12層のモデルとし、柱、梁の断面形状や鉄筋の主筋および横筋（あばら筋）などは、実例に沿ってモデル化した。上部構造のメッシュ分割は基本的に10cm角とし、基礎や地盤はやや大きめの分割とした。総要素数は2,585,043であった。

2.5 鉄筋とコンクリートの一体化

鉄筋とコンクリートの一体化については、鉄筋の節点と、それと最近傍のコンクリート要素の表面とを仮想ビームで繋ぐ方法を用いた。この方法により、メッシュ分割の手間が大幅に軽減され、また、すべりや離間なども簡便に扱うことが可能となる。

3. 解析結果

3.1 予備解析

解析に先立ち簡単なモデルを用いて、1）単純梁の弾塑性解析、2）1自由度系と3次元地盤の、自由振動による連成解析、3）軸圧縮を受ける柱の、鉄筋とコンクリートの結合度解析、をそれぞれ行い、モデル化がほぼ妥当であることを確認した。

3.2 本解析

12層の上部構造だけを対象としJMA-Kobeの地震波を1層の柱脚部に入力して動的弾塑性応力解析を行い、柱梁接合部付近に残留歪が生じること、低層部にいくほど応力が大きくなること、などがわかった。次に、基礎や地盤を含めた一体解析を行った。

演算時間は、128ノード、12時間で約3秒間の地震入力による応答解析が可能であることが分かった。

4. まとめ

共同研究の初年度として、阪大からの地球シミュレータ利用システムの確立と、鉄筋コンクリートのモデル化手法の検討を行った。

さらに、12層のRC造建物の基礎、杭、地盤を含めた一体解析を行った。実際には少なくとも地震発生後の9秒間程度の解析は必要であり、又、階数も、3倍の36層程度とすると、今回の結果から推定して、演算時間は延べ100時間程度を要することになる。

今後、ユニット分割を工夫することにより現在の並列化効率を高めるとともに、より安全な超高層RC建物を設計するための手法を提案していく。