# Development of Sophisticated Simulation Analysis Method of Actual Reinforced Concrete Building by Shaking Table Test–I

Project Representative
Yoshiyuki Kasai Department of Urban Environment and Information Sciences, Graduate School, Maebashi Institute of Technology

Authors
Yoshiyuki Kasai Department of Urban Environment and Information Sciences, Graduate School, Maebashi Institute of Technology.
Haruji Tsubota Department of Architecture, Graduate School, Maebashi Institute of Technology.
Yoshikazu Kanai Department of Urban Environment and Information Sciences, Graduate School, Maebashi Institute of Technology.
Shigefumi Takeda Department of Architecture, Graduate School, Maebashi Institute of Technology.
Kazukuni Niwa Terrabyte Co, Ltd.
Yoshio Manabe Terrabyte Co, Ltd.

A study[1] for establishing a sophisticated simulation analysis method utilizing an explicit finite element impact analysis method was conducted for the model comprising about 2.08 million elements of the main wall-frame of the actual scale six-story reinforced concrete building (total mass about 1,000 tons), which was tested on the shaking table with the input of seismic waves equivalent to those recorded in the 1995 Hanshin-Awaji earthquake. The results of the analysis show that the displacement response subjected to the actually measured waves (input acceleration factor of 100%) corresponding to a seismic intensity scale of 6 upper is smaller than that measured in the experiment. In the experiment, the intensity of the input waves increased in steps to 100%. The authors considered that cumulative damage to the structure caused by shaking before 100% intensity input waves is one of causes that affected such a difference and, therefore, conducted an analysis that considered the effects of the cumulative damage. The displacement response from such an analysis corresponded more closely to the results of the experiment. Analyses were also conducted for a fresh model free of cumulative damage by applying the input acceleration factor of 120%, 150%, and 200%, respectively, and the results were compared with the results of the experiment. The results of the analyses with an input acceleration factor between 120% and 150% corresponded to the results of the experiment. The analyses also indicated that the building would collapse when 200% input waves were applied. The authors intend to increase the number of analyses for various cases and then compare and verify such analyses with the results of experiments so that numerical shaking experiments can be conducted with this simulation analysis program for assessment of seismic safety under severe seismic conditions.

**Keywords**: Seismic response, Shaking table test, RC frame, Earth simulator, FEM simulation

## 1. Introduction

A study[1] for establishing a simulation analysis method using the explicit finite element impact analysis code LS-DYNA[2] was conducted on the shaking table test of the full-scale six-story reinforced concrete (RC) building, which can analyze the behavior of RC buildings under strong seismic loading close to the near collapse of the building structure.

An analysis of the seismic response was conducted for a sophisticated model of the main wall-frame of the six-story RC building in a damage-free fresh condition, based on the experimental data of full-scale building structure tested on the shaking table at the Hyogo Earthquake Engineering Research Center (E-Defense) with input seismic waves (input acceleration factor of 100%) equivalent to those recorded during the 1995 Hanshin-Awaji earthquake. Displacement response of the analytical result was smaller than that recorded in the experiment. The authors considered that one of the causes that affected such a difference was the cumulative damage of the building under the test loads, which occurred by gradually increased shaking intensity (prior shaking) before application of the actually measured waves (100%). Accordingly, analyses that consider the cumulative damage caused by such prior shaking were conducted, and the results were in comparatively good agreement with the experimental results.

## 2. Outline of shaking table test of an full-scale six-story RC building

The experiment that was analyzed was the shaking table test of the full-scale six-story RC building conducted in E-Defense. The data for the test conditions and the building used for the analysis were taken from the published report[3]. The structure of the building used for the analysis was the six-story, three-dimensional wall-frame consisting of two spans in the x-direction and three spans in the y-direction, and each span had a dimension of 5,000 mm, a floor-to-floor height of 2,500 mm, and overall building height of 15,000 mm. The test was conducted with seismic waves equivalent to those recorded at the Kobe Marine Observatory of the Japan Meteorological Agency during the 1995 Kobe-Awaji earthquake (corresponding to the seismic intensity of 6 upper) increasing the input acceleration factor in steps of 5%, 10%, 25%, 50%, and 100%, respectively, and finally 60%. Shaking was applied in three directions horizontally, the x- and y-directions and in the vertical direction, with the original seismic waves rotated 45 degrees, the N45W direction in the y-direction of the building under test, and the N45E direction in the x-direction. Based on such an application, the intention was that the ultimate fracture of the building would take place in the y-direction.

## 3. Summary of analysis

### 3.1 Analytical model

Figures 1 through 4 show the outline of the model used in the FEM analysis. In the model, concrete was represented as solid elements, and reinforcing bars were represented as beam elements as they were in the actual state; the concrete and reinforcing bar elements have common nodes assuming full adhesion between them. The foundation of the building was not represented in the model but represented as rigid shell elements where the bases of the columns were anchored. Input of the seismic waves was applied at the rigid shell elements in the analysis of the seismic response. The size of the analysis model was about 1.48 million elements for concrete, about 0.57 million elements for reinforcing bar, and about 30,000 elements for the rigid shell for total of about 2.08 million elements, and the total number of nodes was about 1.79 million. The material model installed in LS-DYNA[4] was used. Figure 5 shows the stress (σ)

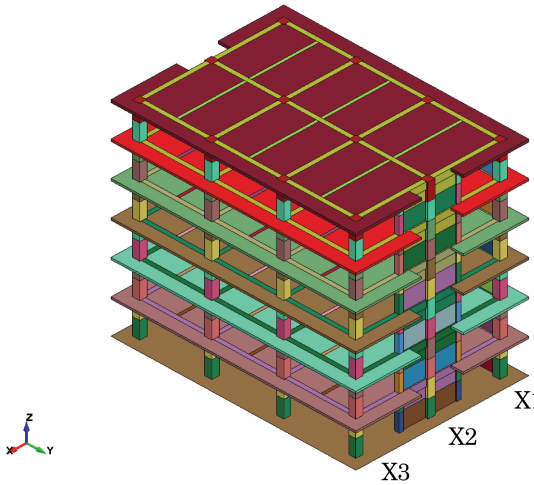



Fig. 1 View of the entire analytical model (Color-coded for input data layer recognition category).

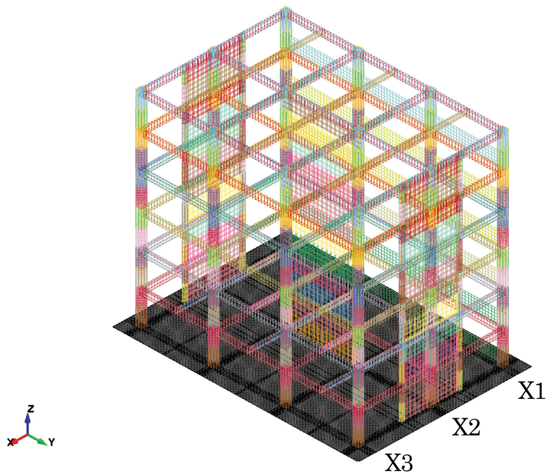



Fig. 2 Reinforcing bar model of the main frame.

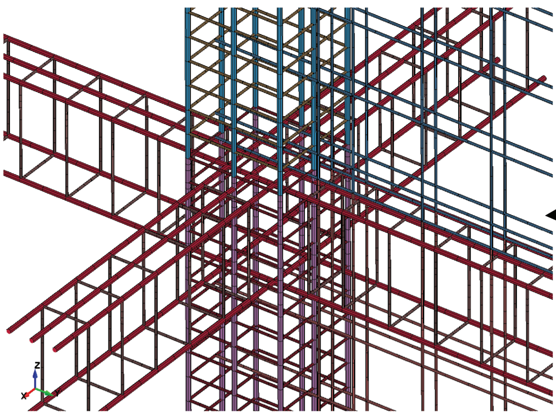


Fig. 3 Enlarged view of the reinforcing bar model of the main frame.

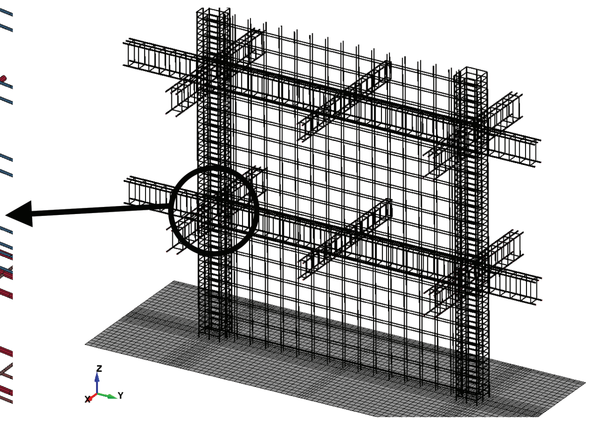
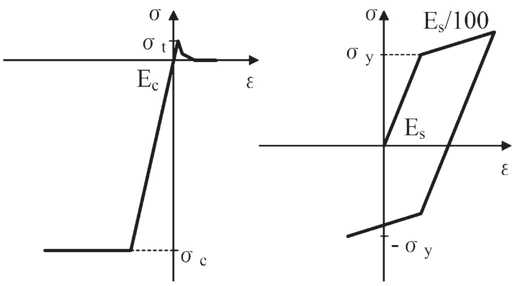


Fig. 4 Reinforcing bar model of the earthquake resistant wall.

and strain (ε) relationship of the material model used.

For the concrete element, the material model[5,6] was used with characteristics of Ottosen's fracture criterion[7], smeared cracks, etc. in consideration of strain rate effect stress relaxation in tension was dependent on the fracture energy and the crack width. For the reinforcing bar element, an isotropic elastic-plastic model in consideration of kinetic hardening was used, which is a bi-linear type where the plastic hardening coefficient after the yield is 1/100 of the elastic modulus.




Concrete model　　Reinforcing bar model

Fig. 5 Material model.

## 3.2 Conditions of seismic response analysis

In this analysis, an explicit dynamic finite element method was used. Consideration was given wherein the application of the load due to gravitational acceleration was increased gradually from 0 m/s$^2$ to 9.8 m/s$^2$ during the 0 to 0.6 seconds before the application of the seismic waves, which started at 0.6 seconds. Because of the large volume of data in the analysis of the six-story RC building, it took about 2 hours using 16 nodes (128 CPUs) of the Earth Simulator for calculation of the initial 1.0 seconds. After 1.0 seconds, it took about 3 hours for calculation of the next 1.0-second possibly due to the increased computing task load in treating the plastic region and fracture of the materials. Because use of the Earth Simulator for one operation is restricted to 12 hours, analysis for about 4 seconds was possible with 16 nodes (128 CPUs) used in one operation (12 hours) in the case of the analysis of the six-story RC building. Restarting the analysis was made up to 4.6 seconds in the case of no prior shaking and up to 13.6 seconds in the case of application of prior shaking, which was the remaining computing task. Damping characteristics in proportion to the mass with damping coefficient of 3% was considered. Central difference time integration in the explicit finite element method was used, and the time interval of about 3.8 microseconds ($3.8\times10^{-6}$s) with the data output interval of 1.0 milliseconds ($1.0\times10^{-3}$s) was used.

## 4. Results of seismic response analysis

Figure 6 shows the results of the analysis of the time-history waveform of the story drift of the first floor in the y-direction with the input acceleration and the experimental results[1]. While the results of the analysis with seismic waves with 100% and 120% input acceleration factors are smaller than the results of the experiment, the result of the analysis with the 150% input acceleration factor is larger than the results of the experiment, which mean that for seismic waves input into the fresh model that does not take cumulative damage into consideration, analysis with the input acceleration factor between 120% and 150% would correspond to the results of the experiment. When the results of the analysis for the fresh model (Case in Fig. 6) are compared with the results from the model taking cumulative damages into consideration (Case e) with input acceleration factor of 100%, the story drift for Case e is considerably greater than that of Case a and is close to the story drift measured in the experiment. By the way, cumulative damages occurred in the prior shaking were reproduced by the response due to seismic wave with 100% assumed to be equivalent to the total input effect due to 5%, 10%, 25%, and 50%, before the actually measured wave 100% and therefore Case e is subjected to 100%-100% inputs.

The stress conditions of the short columns with the spandrel walls and the foot of the earthquake resistant wall where damage occurred in the experiment was severe, the deformation condition of the concrete skeleton, and the deformation condition of the reinforcing bar are shown in Fig. 7 in the magnified view of the deformation. Figures 7 (a), (b), and (d) through (f) are contour maps showing von Mises equivalent stress, where the stress increases from the cold colored area to the warm colored area. In Fig. 7 (c), the main reinforcing bar of the short columns are resisting the seismic loads and swelling out a little under the constraints of the shear reinforcing bar. As shown in these diagrams, this analysis method allows flexible indication of conditions in detail of the building structure, such as the conditions of the reinforcements, stress conditions at any section of the structural elements, etc.

## 5 Conclusion

The time history seismic response analysis of the sophisticated FEM analysis model precisely representing concrete and reinforcing bar of the full-scale six-story RC building using the explicit finite element impact analysis method was conducted. The results of the simulation were consistent with the results of the experiment. The analysis method employed provides excellent features where the dynamic characteristics of the structure are automatically created by the material characteristics of the concrete and reinforcing bar, and by the arrangements, the dimensions, etc., of each structural element. The evaluation of the elastic-plastic characteristics up to large deformation caused by large input acceleration is possible, and the conditions for damage or fracture can be visually presented as the computer animation. Because of the analysis method using explicit algorithms, verification of computational accuracy and analysis results are required, and the method can possibly be used for analysis of the large-scale model and large input acceleration. The authors consider the

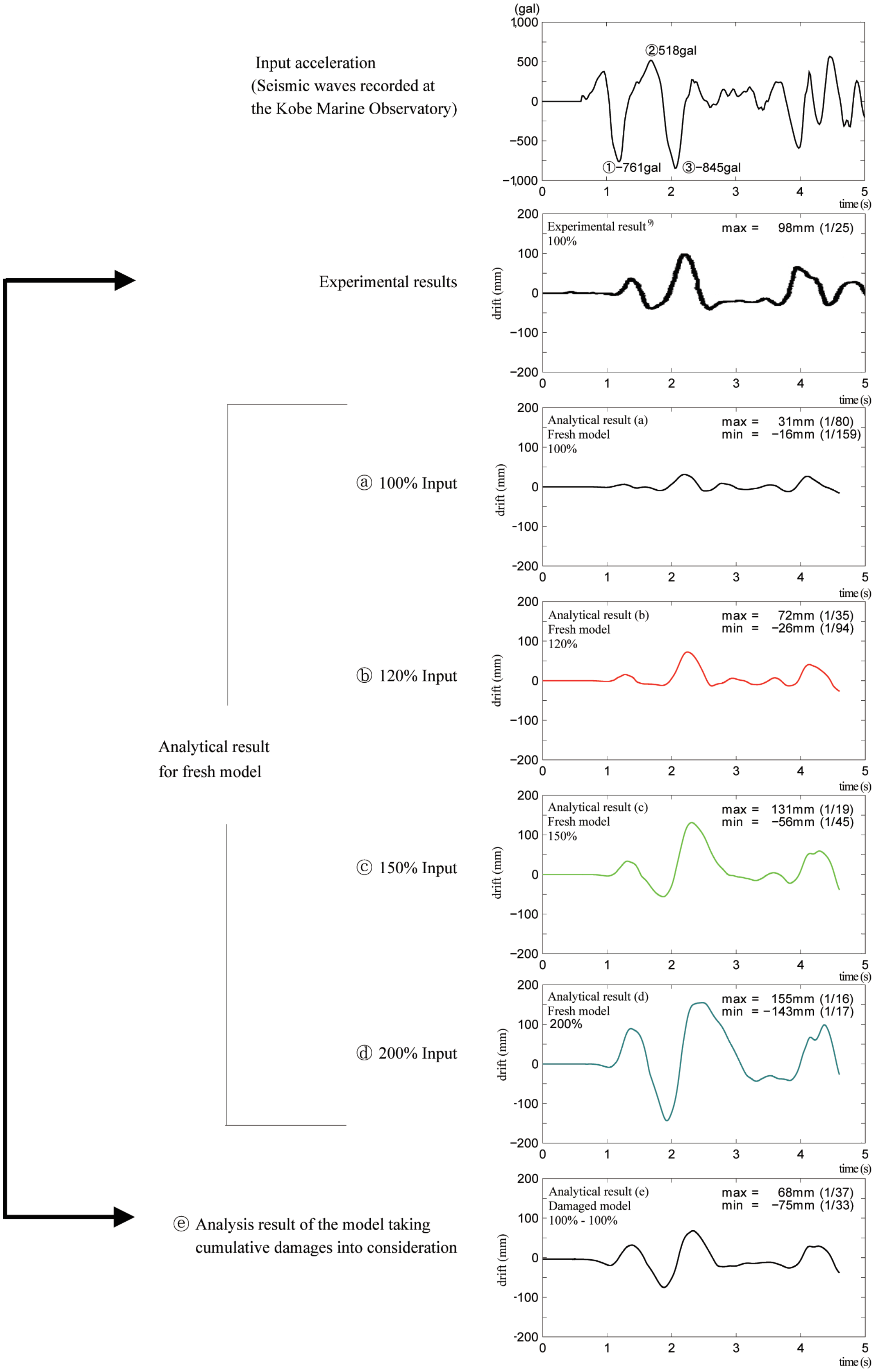
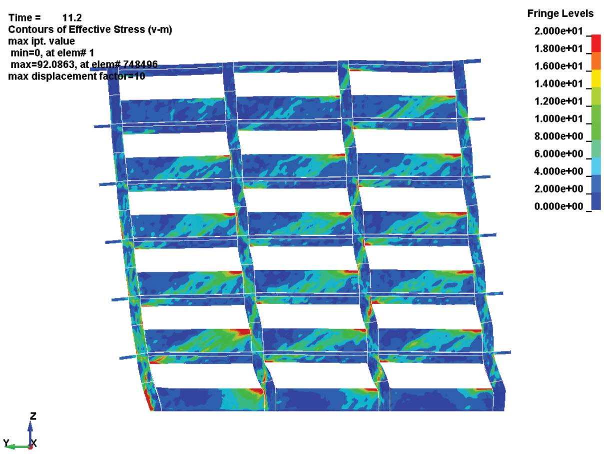
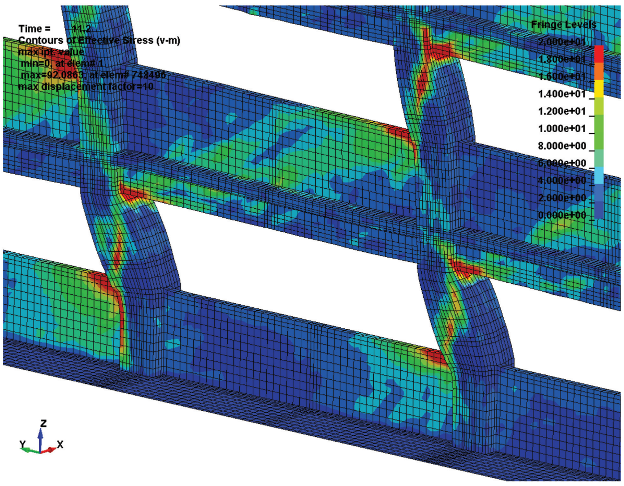



Fig. 6 Story drift time history.

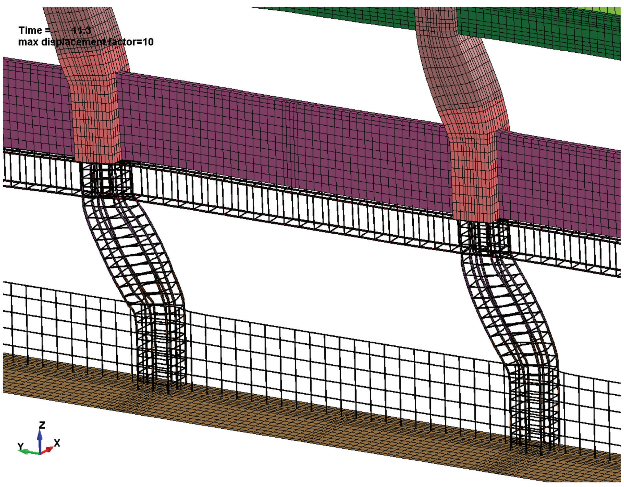
collection of analysis data for increasing the number of examples and verification of such analyses with the results of experiments so that shaking tests can be conducted in a simulation analysis program. When this is possible, the evaluation of shaking under extremely large input acceleration, which is impossible in a shaking table test, will become possible, and evaluation of seismic safety under severe seismic conditions as in the 2011 Great East Japan earthquake will become possible.



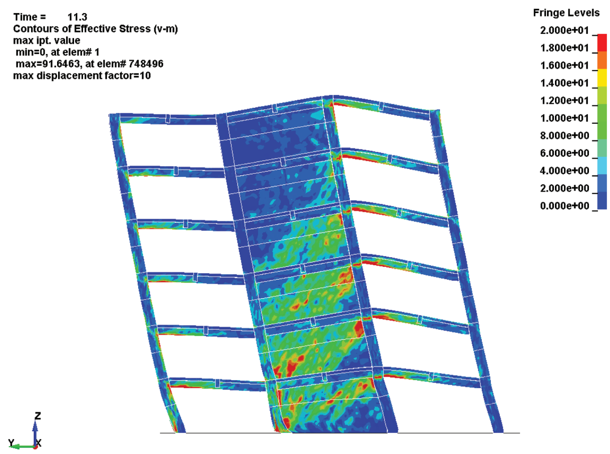

(a) Mises stress contour at $X_1$ frame



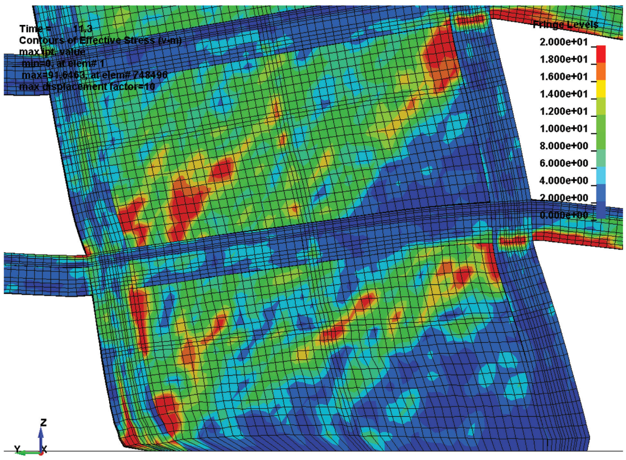
(b) Mises stress contour at $X_1$ frame (enlargement)



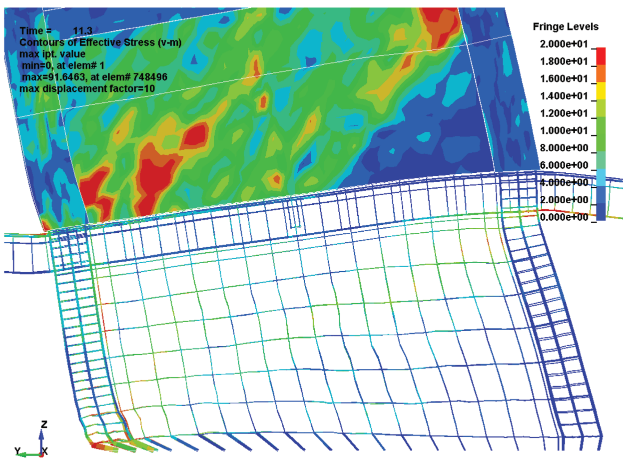
(c) Displacement at $X_1$ frame reinforcement bar




(d) Mises stress contour at $X_2$ frame



(e) Mises stress contour at $X_2$ frame (enlargement)




(f) Mises stress contour at $X_2$ frame (enlargement)

Fig. 7 FEM simulation analytical result (displacement is enlarged by 10 times).

## References


[1] Shigefumi Takeda, Yoshikazu Kanai, Yoshio Manabe, Kazukuni Niwa, Yu Narita, Haruji Tsubota, and Yoshiyuki Kasai, "Application of Impact Analysis Code based on the Explicit Time Integration to the Shaking Table Tests of Full-Scale Six Story RC Building, Part 2 ", Bulletin Maebashi Institute of Technology, No.14, pp.17-24, March 2011, (in Japanese ).

[2] LS-DYNA (general purpose transient dynamic finite element program), Livermore Software Technology Corporation (LSTC).

[3] Y. Kim, T. Kabeyasawa, T. Matsumori, and T. Kabeyasawa, "Analytical Studies on A 6-Story Full-Scale Reinforced Concrete Wall-Frame Structure to Collapse ", The 14th World Conference on Earthquake Engineering, October 12-17, 2008, Beijing, China.

[4] Livermore Software Technology Corporation (LSTC), "LS-DYNA KEYWORD USER'S MANUAL VOLUME I, II".

[5] B.J. Broadhouse, "The Winfrith Concrete Model in LS-DYNA3D", Report: SPD/D(95)363, Structural Performance Department, AEA Technology, Winfrith Technology Centre, U.K. (1995).

[6] B.J. Broadhouse and A.J. Neilson, "Modelling Reinforced Concrete Structures in DYNA3D", Safety and Engineering Division, United Kingdom Atomic Energy Authority, Winfrith, AEEW-M 2465, 1987.

[7] N.S. Ottosen, "Failure and Elasticity of Concrete", RISO-M1801, July 1975.

# 実大鉄筋コンクリート造建物の振動台実験の精密・詳細シミュレーション解析システムの開発　その 1

プロジェクト責任者
河西　良幸　　前橋工科大学　大学院工学研究科 環境・情報工学専攻

著者
河西　良幸　　前橋工科大学 大学院工学研究科 環境・情報工学専攻
坪田　張二　　前橋工科大学 大学院工学研究科 建築学専攻
金井　喜一　　前橋工科大学 大学院工学研究科 環境・情報工学専攻
武田　慈史　　前橋工科大学 大学院工学研究科 建築学専攻
丹羽　一邦　　株式会社 テラバイト
真鍋　慶生　　株式会社 テラバイト

実大鉄筋コンクリ－ト造 6 階建物（総質量約 1000t）の阪神淡路大震災での記録地震波を入力とした振動台実験での建物損傷状況の結果を例に、架構各部材をあるがままに精密・詳細な有限要素モデル化（約 208 万要素）し、陽解法の衝撃解析プログラムを適用して精密・詳細なシミュレ－ション解析システムの構築を試みた。震度 6 強相当の実測波形（100% 入力）による変位応答は実験に比べて小さい結果となった。実験では地震波の入力加速度倍率を順次増大させて与え、その後 100% の入力を行っている。この事前の加震による累積損傷が解析による変位応答が小さい要因のひとつと考え、累積損傷の影響を考慮した解析を行った。その結果、変位応答は比較的良く一致する結果となった。また、累積損傷の無いフレッシュな試験体に地震波の入力加速度倍率を、120%、150%、200% と変化させた解析も行い実験結果と比較した。振幅倍率 120% と 150% の中間程度の入力が実験結果と対応する結果となった。また、建物が倒壊するのは 200% の大きな入力の場合であることも分かった。

採用した解析法は、コンクリートと鉄筋の材料的特性を与え、各部材要素の配置、寸法等によって自ずと力学的な特性が取り込まれ、大入力に対する鉄筋コンクリ－ト建物の大変形までの弾塑性特性が評価でき、更に、損傷や破壊がコンピュータアニメーションとして表現できるという優れた面を持つ解析法である。 陽解法というアルゴリズムに基づく方法によるために、計算精度や解析結果の検証という過程が必要であるが、大規模モデル、大入力を扱い得るので、今後更に、解析事例の蓄積、実験結果との比較検証を進め、数値振動実験をシミュレ－ション解析システム上で行えるようにしたい。これにより、振動台実験では不可能な大入力加振の場合を評価し得ることができ、東日本大震災のような従来想定できなかったような過大な地震条件での耐震安全性の評価が可能になる。

キーワード : 地震応答 , 振動台実験 , 鉄筋コンクリート骨組 , 地球シミュレータ , FEM シミュレ－ション