COMPUTER SIMULATION OF 3-DIMENSIONAL SEISMIC RESPONSES FOR RC FRAME STRUCTURES

Qiang Zhang¹, Xianglin Gu², and Qinghua Huang³

ABSTRACT
Based on multi-spring model, a simulation system for existing RC frame structures under 3-dimensional earthquake is developed, and it is one part of the REliability Assessment for Existing building Structures (REASES) integrated software. The program adopts proper hysteresis models of the steel springs and concrete springs in the beams and columns of the frame in an existing building structure. According to these hysteresis models, the plastic and elastic deformation of the beam and column elements can be considered and the stiffness matrixes of elements are calculated at every time step. The program not only can be modeled visually, but also has a visual post-processor which can show the real time damage states of the elements and the real time responses of whole frame structure during the earthquake. In order to examine the simulation effect, a comparison between the shaking table test responses of a 3-storey RC frame model and the simulation results of the model is carried out. It shows that the system could be used to assess the seismic behavior of an existing building structure.

KEY WORDS
simulation, RC frame, earthquake response, 3-dimension, multi-spring model, hysteresis model.

INTRODUCTION
The damage of reinforced concrete structures due to earthquakes, the environment, and other kinds of loadings is currently an important problem, and the evaluation of the residual capacity of existing RC structures subjected to earthquakes is one of the most important structural duties for many engineers.

Since the computing technology is highly developed, the nonlinear response analysis and visualized simulation of RC frame structures under earthquake have been widely researched in recent years. Earthquakes contribute damage to concrete and reinforcement reducing their strength, stiffness, and influencing the ductility and hysteretic energy of the section. All these factors will change the behavior of the RC structure. Many hysteretic models can be used to calculate the dynamic responses of RC structures(Roufaiel and Meyer 1987).

¹ Postgraduate Student, Dept. of Build. Engrg., Tongji Univ., Shanghai, China, zhangbuddy@hotmail.com
² Prof., Dept. of Build. Engrg., Tongji Univ., Shanghai, China, gxiangl@online.sh.cn
³ PhD. Student, Dept. of Build. Engrg., Tongji Univ., Shanghai, China, chadyellow@126.com
On the other hand, the computer simulations have been effectively used to simulate the seismic responses of structures, and some great progresses in this field have been achieved (Gu and Sun 2002, Lai et al. 1984, Li 1993). However, most researches focus on the plane problems, and there are few 3-dimensional analysis programs about nonlinear analysis for RC frame structures, which have both visualized pre-processor and post-processor.

In this paper, we developed a simulation system for existing RC frame building structures under 3-dimensional earthquake, which is one part of the experiment platform for integrated software. The system sets up a space frame model, and in the element model, a new concept of MacroSpring has been introduced. By adopting proper hysteresis models of the steel springs and concrete springs in the elements of the frame, the system could be used to simulate the responses of RC frame structures.

THE REASES INTEGRATED SOFTWARE

As a reliability assessment software under the guideline of correlative codes for existing structures, REASES focuses on scientificity and practicality for engineering. REASES consists of two sets of platform systems. The main one is an assessment platform for existing structures; the other one is an experiment platform including three modules currently shown in Figure 1. The detail of REASES will be discussed in other paper, the system discussed in this paper is the Module2 in Figure 1.

Experiment platform is an important platform of REASES. The main platform has three parts: pre-, pro-, and post-processor. The simulation system uses the visualized modeling and display processor of REASES. The results of the simulation will provide a powerful proof for the assessment for an existing building, especially under earthquake.

![Figure 1. Reliability assessment software for existing building structures](image-url)
DYNAMIC EQUATIONS OF THE STRUCTURE

All of the beams and columns are simplified as bar elements and the masses of elements and slabs are all converged on the nearest node in the analysis model, so dynamic equations of the structure are written as Eq.(1).

\[
[M] \{\Delta a\} + [C] \{\Delta v\} + [K] \{\Delta d\} = -[M] \{\Delta a_g\}
\]  

(1)

Where, \([M]\), \([C]\) and \([K]\) are the mass, damping and stiffness matrix of the structure respectively, \{\Delta a\}, \{\Delta v\} and \{\Delta d\} are the increments of accleration, velocity and displacement response of the structure respectively, \{\Delta a_g\} are the increments of 3-dimensional ground acclerations.

\([M]\) is the lumped mass matrix. \([C]\) is Rayleigh damping matrix as Eq.(2), and \(\alpha, \beta\) are two constants which are independent of frequences(Gu and Sun 2002).

\[
[C] = \alpha[M] + \beta[K]
\]  

(2)

Using Newmark-\(\beta\) method, the dynamic responses of the structure can be calculated step by step.

MULTI-SPRING MODEL FOR BAR ELEMENTS

Multi-spring model was proposed by many researchers(Lai et al. 1984, Li 1993), which represents a significant advance both in simplicity and accuracy. This model separates inelastic and elastic deformation and assembles the inelastic deformation to the two ends of a bar element, which are called inelastic and elastic unit as shown in Figure 2. The region undergoing inelastic deformation is represented by a set of springs representing concrete and steel (Figure 3). Inelastic behavior is controlled by the description of the stress-strain properties of steel and concrete. The spring force is calculated as a product of the tributary area of the spring and stress of the materiel at the central of the area. An imaginary spring length \(\eta h\) presenting plastic zone is assumed to calculate the deformation for a given strain.

![Figure 2. Model of bar element](image)

![Figure 3. Springs of inelastic unit and division of the section](image)

The model converts the stress and the strain of the material into force and displacement of the spring, using Eq.(3), it changes the material stress-strain relationship into force-displacement relationship of the spring as Figure 4.
Where, \( p \) is the force of the spring; \( \sigma \) is the average stress of the spring; \( A_{sp} \) is the area of the spring; \( d \) is the displacement of the spring; \( \eta \) is a coefficient, \( \eta=0.75 \) for RC element; \( h \) is the depth of the element section and \( \varepsilon \) is the average strain of the spring. The typical parameters in Figure 4 may be calculated using Eqs.4–6.

\[
k_{sed} = k_{cc} \left( \frac{d_{cy}}{d_{cu}} \right)^{0.2} ; k_{sed} = k_{sy} \left( \frac{d_{sy}}{d_{su}} \right)^{0.2}
\]

\[
d_{shelly} = d_{cy} ; \quad p_{shelly} = p_{cy} ; \quad d_{shelu} = d_{cu}
\]

\[
d_{core} = (1+10\rho')d_{cy} ; \quad p_{core} = (1+10\rho')p_{cy} ; \quad d_{coreu} = (2+600\rho')d_{cu}
\]

Where, \( d_{shelly}, d_{core} \) are the “yielding” deformations of shell and core concrete spring respectively; \( d_{cy}, d_{cu} \) are the “yielding” and ultimate deformations of concrete spring without confinement; \( p_{shelly}, p_{core} \) are the “yielding” forces of shell and core concrete spring respectively; \( p_{cy} \) is the “yielding” force of concrete spring without confinement; \( d_{shelu}, d_{coreu} \) are the ultimate deformations of shell and core concrete spring respectively; \( \rho' \) is the amount of confinement steel, \( \rho' = 2(b'' + h')A_{sv1}/(b'h's) \), in which \( b'' \) and \( h'' \) are the width and depth of the confined core concrete; \( A_{sv1} \) is the cross-sectional area of one leg of a stirrup; and \( s \) is the spacing of stirrups.

**STIFFNESS MATRIX OF ELEMENTS**

In the nonlinear analysis, how to set up the element model is a fatal problem, of which how to get and modify the stiffness matrix of elements is of great importance. Based on the multi-spring model, two MacroSpring units have been set at the two ends of the element, and each MacroSpring unit is formed by a set of concrete springs and steel springs. The MacroSpring unit is under biaxial bending and axial force, while concrete spring and steel spring are under axial tension or compression. Hence, the coupling problem of biaxial
bending and axial tension or compression has been properly solved. It is assumed that the torsional stiffness is a constant and the element would not suffer the damage caused by the shear force.

When an element is under axial force as Figure 5, the displacements of elastic unit are \( u_A \), \( u_B \), the inelastic displacements of MacroSprings are \( u_{RA} \), \( u_{RB} \). Hence, the absolute axial displacements of element are \( u_1 = u_A + u_{RA}, u_2 = u_B + u_{RB} \).

According to the physical behavior of the element, the equations between the axial force \( N_A, N_B \) and the displacement \( u_1, u_2 \) are

\[
N_A = \frac{EA}{L} \left( u_1 - u_2 \right) \left[ \frac{1}{k_A} + \frac{1}{k_B} \right] + 1 \tag{7}
\]

\[
N_B = \frac{EA}{L} (-u_1 + u_2) \left[ \frac{1}{k_A} + \frac{1}{k_B} \right] + 1 \tag{8}
\]

Where, \( k_A, k_B \) are axial stiffnesses of MacroSprings, \( A \) is the sectional area of the element, and \( L \) is the length of the element.

When an element is under bending force as Figure 6, the elastic rotation of elastic unit are \( \theta_A, \theta_B \), the inelastic rotation of MacroSprings are \( \theta_{RA}, \theta_{RB} \). Hence, the absolute rotational displacements of element are \( \theta_1 = \theta_A + \theta_{RA}, \theta_2 = \theta_B + \theta_{RB} \), and the bending stiffnesses of MacroSprings are \( r_A = M_A / \theta_{RA}, r_B = M_B / \theta_{RB} \). Where, \( M_A \) and \( M_B \) are bending forces in MacroSprings.

\[ M_A = \frac{EI}{L} (S_u \theta_1 + S_y \theta_y) + \frac{EI}{L^2} (S_{uu} + S_{yy}) (v_1 - v_2) \tag{9} \]

\[ M_B = \frac{EI}{L} (S_y \theta_1 + S_y \theta_y) + \frac{EI}{L^2} (S_{yy} + S_{yy}) (v_1 + v_2) \tag{10} \]

\[ V_A = \frac{EI}{L} (S_u + 2S_y + S_{yy}) (v_1 - v_2) + \frac{EI}{L^2} (S_u + S_{yy}) \theta_1 + \frac{EI}{L^2} (S_y + S_{yy}) \theta_2 \tag{11} \]

\[ V_B = \frac{EI}{L} (S_u + 2S_y + S_{yy})(-v_1 + v_2) - \frac{EI}{L^2} (S_y + S_{yy}) \theta_1 - \frac{EI}{L^2} (S_y + S_{yy}) \theta_2 \tag{12} \]

So the stiffness matrix of an element is
where, $\psi_{11} = 1 \left[ \frac{EA}{L} \left( \frac{1}{k_A} + \frac{1}{k_B} \right) + 1 \right]$, $\psi_{22} = S_{zii} + 2S_{zij} + S_{zjj}$, $\psi_{26} = S_{zii} + S_{zij}$, $\psi_{212} = S_{zjj} + S_{zij}$

$\psi_{33} = S_{yii} + 2S_{yij} + S_{yjj}$, $\psi_{35} = S_{yii} + S_{yij}$, $\psi_{311} = S_{yjj} + S_{yij}$, $\psi_{311} = S_{yjj}$, $\psi_{55} = S_{yii}$, $\psi_{511} = S_{yjj}$,

$\psi_{66} = S_{zii}$, $\psi_{612} = S_{zij}$, $\psi_{1111} = S_{yjj}$, $\psi_{1212} = S_{zjj}$,

$S_{zii} = (4 + 12EI_z/Lr_{zB})/r_z$, $S_{zjj} = 2/r_z$, $S_{zij} = (4 + 12EI_z/Lr_{zA})/r_z$,

$S_{yii} = (4 + 12EI_y/Lr_{yB})/r_y$, $S_{yjj} = 2/r_y$, $S_{yij} = (4 + 12EI_y/Lr_{yA})/r_y$,

$S_{r_{zB}} = (1 + 4EI_z/Lr_{zA})(1 + 4EI_z/Lr_{zB}) - 4(EI_z/L)^2/(r_{zA}r_{zB})$;

$S_{r_{yB}} = (1 + 4EI_y/Lr_{yA})(1 + 4EI_y/Lr_{yB}) - 4(EI_y/L)^2/(r_{yA}r_{yB})$.

**Development of the Simulation System**

The system is one part of the experiment platform of the integrated software REASES. Through visualized modeling, the simulation system could get the information of the structure.
and the earthquake wave, and prepare for calculating. After calculating by the kernel programme, the results could be displayed by the post-processor. As an example, the modeling and display of the results for a 3-storey frame structure is shown in Figure 7.

![Figure 7. The modeling and display of the system](image)

According to the information provided by pre-processor, the kernel programme sets up a 3-dimensional FEM frame model and get the elastic global stiffness matrix of structure. If the damage of an existing building is identified, the initial stiffness matrix of the structure needs to be modified using Eq. (14) (Gu and Sun 2002).

\[
[K_i] = \left( \frac{f_{01}}{f_{01}} \right)^2 [K_0]
\]  

(14)

\([K_0], [K_i]\) are stiffness matrixes before and after damage, and \(f_{01}, f_{i1}\) are the basic frequencies of the structure before and after damage. The kernel programme is organized as Figure 8.

**COMPARISON BETWEEN COMPUTING AND TEST RESULTS**

The shaking table test of a 3-storey frame is done in the State Key Laboratory of Tongji University, the information of the test model is shown in Figure 9. The earthquake wave is shown as Table 1.

The comparison of calculating and testing responses for the model is shown in Figure 10, 11 and 12, which shows that the simulation system is applicable. However, big error of displacement responses in X direction can be seen, which urges us to modify the simulation system in the future.

Through the computer simulation, it is found that all plastic hinges appear in the base floor of the model in case 2 (Figure 7). This is the same with what has been observed in the test. Unfortunately, in case 3, the system could not calculate the whole responses of the RC
Figure 8. The flowchart of the kernel programme

(a) Test model on the shaking table  (b) Details of the model

Figure 9. The information of the test model

Table 1: Earthquake wave used in the test

<table>
<thead>
<tr>
<th>Case</th>
<th>Earthquake wave</th>
<th>Peak value of acceleration(g)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>In X</td>
</tr>
<tr>
<td>1</td>
<td>3D EL-Centro wave</td>
<td>0.10</td>
</tr>
<tr>
<td>2</td>
<td>3D EL-Centro wave</td>
<td>0.36</td>
</tr>
<tr>
<td>3</td>
<td>3D EL-Centro wave</td>
<td>0.84</td>
</tr>
</tbody>
</table>

Compressive strength of concrete is 20.7Mpa; Yield strength of reinforcement is 440Mpa.
frame, after hundreds of timesteps, the calculation could not be continued. In the shaking

Figure 10. The comparison of computing and test responses on the top of the model in Case 1

Figure 11. The comparison of computing and test responses on the top of the model in Case 2
table test, the frame model collapsed in case 3. It is difficult to simulate the collapse responses based on FEM and it is very important to develop a special system which can be used to simulate the collapse responses of the structure.

CONCLUSIONS
As a part of the integrated software, REASES, the computer simulation system developed by the authors based on the multi-spring model can simulate the elasto-plastic responses of RC frame structures under 3-dimensional earthquake. And it can be used as an assistant tool for the assessment of existing buildings. Due to the shortcomings of FEM, the FEM based system in this paper can not be used to simulate the collapse reponses of RC structures.

REFERENCES