

Simulation of RC Structure Collapse Due to Earthquake Loading

Cen Zhou

Bridge Engineering Department, Tongji University, China

Limin Sun

Bridge Engineering Department, Tongji University, China

Wei Cheng

Bridge Engineering Department, Tongji University, China

Lichu Fan

State Key Lab for Disaster Reduction in Civil Engng at Tongji University, China

Abstract

Under the action of earthquakes, RC structures may undergo from small elastic deformation, cracking to propagation, local fracture and even collapse. Based on ANSYS UPFs, a new element is created to take cracks in concrete into account with Smeared Crack Method. A new material model is also defined to describe hysteresis characters of reinforced concrete under cyclic loading. In the discrete stage, LS-DYNA is used to analyze the structure collapse. By this procedure, the whole collapse process of RC structures can be simulated by using ANSYS.

Introduction

The recent large earthquakes, such as Northridge Earthquake in 1994, Kobe Earthquake in 1995 and Taiwan Ji-Ji Earthquake in 1999, all have made serious damages to modern reinforced concrete structures. Although damages of structures under earthquakes are inevitable, collapses of structures will result in tremendous losses of lives and properties. Therefore, it is important to comprehend not only seismic responses of structures in elastic and plastic response phase but also their collapse processes. The simulation of collapse due to earthquakes becomes more significant in seismic analysis of structures.

Dynamic finite element method is one of the most common methods for analyzing structural responses under earthquakes. Most structure analyses can be carried out by ANSYS, including simple seismic response analyses. However, it seems impossible to describe the whole collapse process of RC structures from small elastic deformations state to collapses using an existing commercial software. Attributed to the openness and extendibility of ANSYS with User Programmable Features (UPFs), both new element and material can be defined by users, so the complex responses of RC structures under earthquakes could be analyzed more accurately.

After local fractures, the structure may be regarded as several distinct bodies (elements), and adjacent elements may contact or separate from one another. LS-DYNA can treat contact and collision problem without the restriction of deformation continuity condition. So the combination of the UPFs and LS-DYNA is a powerful tool for simulation of RC structure collapse.

Strategies

Finite element method (FEM) and distinct element method (DEM) are the most common methods for damage and collapse analysis of RC structures. The advantages and disadvantages of these methods are compared (Table 1). FEM is valid only before structures collapse, while DEM is able to treat the whole response process from elastic phase to post-collapse, however, it is less accurate for elasto-plastic analysis and time-consuming for collapse analysis. Therefore the authors suggest two collapse analysis strategies according to study purposes. The first strategy is to adopt EDEM with a simple and approximate material

Table 1 - FEM and DEM for Damage and Collapse Analysis of RC Structures

Analysis Method		Modeling for RC structure			Applicable range
		Elastic	Plastic (post-crack)	Local failure & Re-contact	
FEM	Beam-hinge	conventional	plastic hinge	×	before collapse
	Smearred crack	conventional	smear crack	×	before collapse
	Discrete crack	conventional	joint element or re-mesh	×	before collapse
DEM	Ordinary	×	×	Assemble of discrete bodies	after collapse
	EDEM & AEM	small rigid bodies connected by springs	yield or partially failure of springs	+ Re-contact spring	whole process

“×” means invalid

model to obtain a qualitative collapse analysis result with short computational time, if the research aim is to investigate the performances of structures after collapse. The second one is to use a combined approach of FEM and DEM for the analysis before and after collapse, respectively, if the structural responses before and after collapse are all concerned. In this work, the second strategy is used due to the author’s research purpose.

The powerful standard tools and open architecture of ANSYS make the method realizable. In the first phase before the structure collapses, user’s element and material can be defined with UPFs to assure the accuracy of the seismic response analysis with FEM. LS-DYNA is suitable for the second phase after the structure collapses with DEM and can provide fast solutions for large deformations, multiple nonlinearities, and complex contact/impact problems. Just like boundary element method (BEM) and its coupled analysis with FEM can be carried out by ANSYS/LS-DYNA, the DEM can also be taken into account.

With this strategy, an integration of UPFs and LS-DYNA may simulate collapse with combined method of FEM and DEM.

User defined element

A user-defined reinforced concrete plate element, USER101 is developed based on UPFs. The main reason for RC structure collapse is that structure is severely damaged due to transverse earthquake, and then the structure is not able to support its self-weight to avoid collapse. A planar analysis is considered in this research and it can save a lot of CPU time, so USER101 is extended from PLANE42 in ANSYS element library. For the steel bar is smeared in concrete, USER101 should reflect the properties of both steel and concrete. The descriptions of USER101 are shown as below.

Element name: USER101

Nodes: I, J, K, L

Degree of freedom: UX, UY

Real constants: the thickness of element (thickness), the material number (MAT1, MAT2), the steel reinforcement volume ratio (VR1, VR2) and the orientation angles (THETA1, THETA2) of steel reinforcement in two directions.

The difference from PLANE42 is that USER101 contains steel reinforcement in two directions, which can be determined by the material number, the volume ratio and the orientation angle. All the parameters should be input as real constants.

Material properties: EX, ALPX, PRXY or NUXY, DENS, (for concrete). EX, ALPX, DENS (for each steel bar)

Supply DAMP and REFT only once for the element (use MAT command to assign material property set).

Special features: Plasticity, Cracking, Crushing, Large deflection, Large strain, Stress stiffening, Birth and death and etc.

KEYOPT(1)

- 0 Element coordinate system is parallel to the global coordinate system
- 1 Element coordinate system is based on the element I-J side

KEYOPT(2)

- 0 Include extra displacement shapes
- 1 Suppress extra displacement shapes

KEYOPT(3)

- 0 Plane stress
- 2 Plane strain (Z strain = 0.0)

KEYOPT(5)

- 0 Basic element solution
- 1 Repeat basic solution for all integration points
- 2 Nodal stress solution
- 3 Provide cracks in the element (different from PLANE42)

KEYOPT(6)

- 0 Concrete nonlinear solution only at centroid
- 3 Solution also at each integration point

KEYOPT(7)

- 0 Using average stress and average strain relations.
- 1 Using equivalent stress and equivalent strain relations.

In PLANE42 there isn't KEYOPT(7), as USER101 uses a rather different way to calculate the stress in element, so this key value provides relative computing option.

KEYOPT(9)

- 0 No user subroutine to provide initial stress (default)
- 1 Read initial stress data from user subroutine USTRESS

The fundamental data of concrete listed below should be entered in the data table with the TB commands. Data without input are assumed to be zero, except for defaults described below. The constant table is started by using the TB command (with Lab = CONCR). Up to eight constants may be defined with the TBDATA commands following a temperature definition on the TBTEMP command. Up to six temperatures (NTEMP = 6 maximum on the TB command) may be defined with the TBTEMP commands.

The constants (C1-C9) entered on the TBDATA commands (6 per command), after each TBTEMP command, are:

- C1 --- Shear transfer coefficients for an open crack.
- C2 --- Shear transfer coefficients for a closed crack.
- C3 --- Uniaxial tensile cracking stress.
- C4 --- Uniaxial crushing stress (positive).
- C5 --- Biaxial crushing stress (positive).
- C6 --- Ambient hydrostatic stress state for use with constants 7 and 8.
- C7 --- Biaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
- C8 --- Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
- C9 --- Not used.

Absence of the data table removes the cracking and crushing capability. A value of -1 for constant 3 or 4 also removes the cracking or crushing capability, respectively. If constants 1-4 are input and constants 5-8 are omitted, the latter constants default as discussed in Section 4.7 of the ANSYS Theory Reference . If any one of Constants 5-8 is input, there is no default and all 8 constants must be input.

There are some other variables which should be saved in the computing procedure: degenerating information, cracking or crushing status, loading history, and hysteresis characters variables of concrete.

During calculation, invalid element shape may lead to singularity matrix and calculating termination at last, so it should be avoided by geometry checks.

Standard element shape:

The element is defined by four nodes with two degrees of freedom at each node, translations in the nodal x and y directions. Nodes I, J, K, L are in counter clockwise direction (Figure 1), and the element must not be concave.

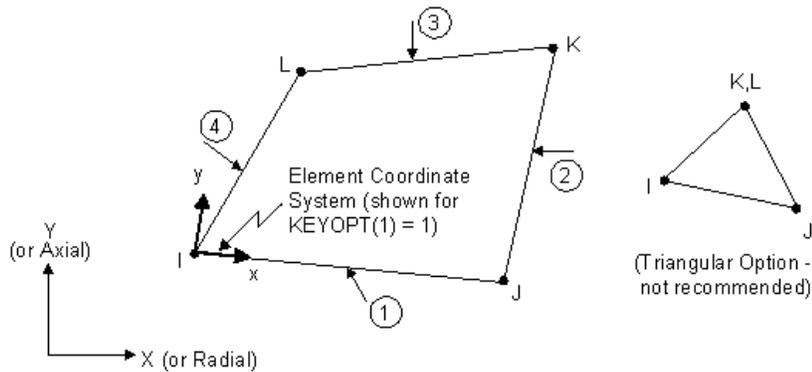


Figure 1 – Standard Element Shape

Several possible invalid shapes are as follows

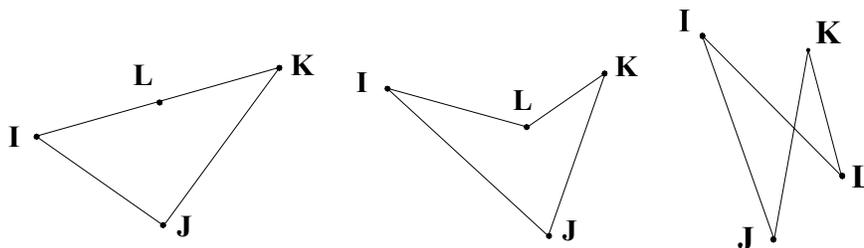


Figure 2 - Area of triangle Invalid element shapes

For the stability of computing process, the invalid element shapes should be avoided (Figure 2). In the subroutine, the area of each triangle in the element is computed via equation (1). If area of any triangle is negative, which means the invalid shape appears, the program will be terminated. The area of is calculated as below:

In the assumption that nodes P1, P2, P3 are in counter clockwise direction (Figure 3.), the area of triangle can be obtained from

$$S = \frac{1}{2} \begin{vmatrix} x_1 & y_1 & 1 \\ x_2 & y_2 & 1 \\ x_3 & y_3 & 1 \end{vmatrix} \quad (1)$$

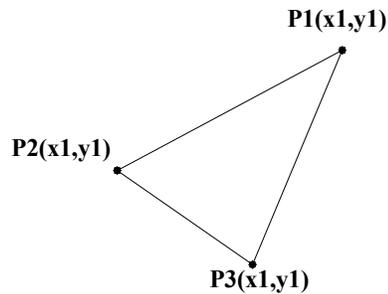


Figure 3 – Area of Triangle

User defined material

Proper constitutive model is the base of seismic analysis. Based on smeared crack assumption, the material model is constructed by combining the constitutive laws for concrete and steel (Okamura and Maekawa, 1991). In this model, the rules of cracked concrete under compression, tension and shearing are taken into consideration, as well as that of steel bar in cracked concrete. This model has been verified by a lot of tests. And it has the advantage to characterize concrete rather well with only a few parameters.

The user material routine, USERMAT, is used to define material stress-strain relation. For the model is considered to be used for the planar user-defined element introduced in the previous chapter, only the subroutine USERMATPS is modified to describe the model above. For every Newton-Raphson iteration, USERMAT is called at every integration point. Stresses, strains and state variables are passed in as the values at the beginning of the time increment, and current strain increment, then USERMAT updates the stresses and state variables to the values at the end of the time increment. In this research, several reserved arguments are used for further development. For example, var1 is used to note whether yielding stress of concrete has been exceeded before.

The model of cracked concrete under tension and compression is discussed as below. First of all, in this research, nonlinearity of concrete should be taken into account appropriately. In standard ANSYS, several material behavior models, such as BKIN, MKIN, BISO and MISO, have been provided to characterize the elasto-plastic response of material. However they are limited in dealing with the hysteresis property of concrete and interaction of steel. So an effective nonlinear stress-strain relation for concrete under dynamic load should be defined by users. Furthermore earthquake input base acceleration is rather irregular, so the stress in concrete is very complex and uncertain. Loading, unloading and reloading may occur alternately, and improper constitutive model will lead to unsteady calculation. At last, under the earthquake action, cracks will appear in most concrete structure, so the effect of concrete should be reflected in user-defined

constitutive model. The model of concrete under tension and compression is shown in Figure 4, ω is an affecting factor of the lateral crack to stress-strain relation in the vertical direction. When concrete cracks, concrete softening effect must be taken into account. The concrete strength decays in the descending branch and in the post-crack tension branch. This effect leads to negative stiffness and singular matrix. The solution may diverge and terminate unexpectedly. It is approximated using step-down rule that takes zero stiffness in computation and corrects the unbalance to meet the real stress-strain relation. As an option minus stiffness may be used for the descending branches if explicitly indicated, as shown in Figure 5. This rule is applied by default. An optional method is reloading-decay rule: reloading towards a point with lower strength on the descending curve, as shown in Figure 6.

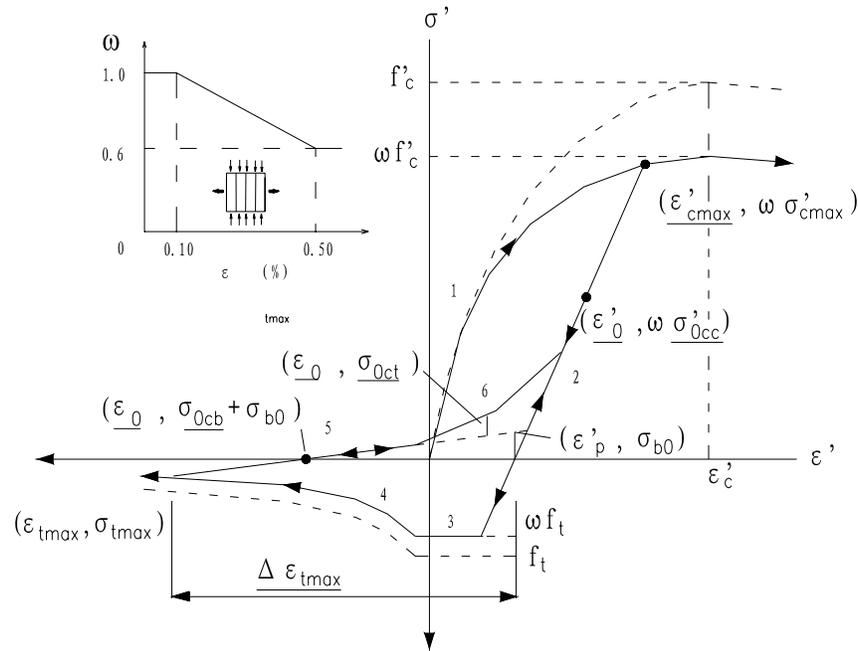


Figure 4 - Constitutive Model of Cracked Concrete Under Compression and Tensile

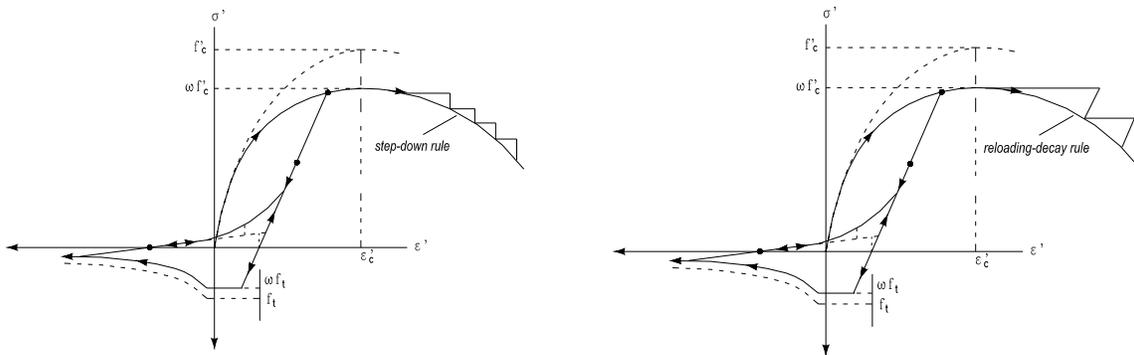


Figure 5 – Step-down Rule

Figure 6 – Reloading-decay Rule

Integration

With non-standard use of ANSYS, UPFs, several extended modules are built to complete the analysis before RC structure fracture. They are user-defined element module, user-defined material module, crack solution module, failure solution module and post-processing module. LS-DYNA is used to analyze the collision of RC structure after fracture. How to divide structure into several collision bodies in LS-DYNA is based on the result above, and its modeling is also completed with standard ANSYS. So integration (Figure 7.) of LS-DYNA and UPFs with standard ANSYS is a solution plan for simulation of RC structure collapse due to earthquake.

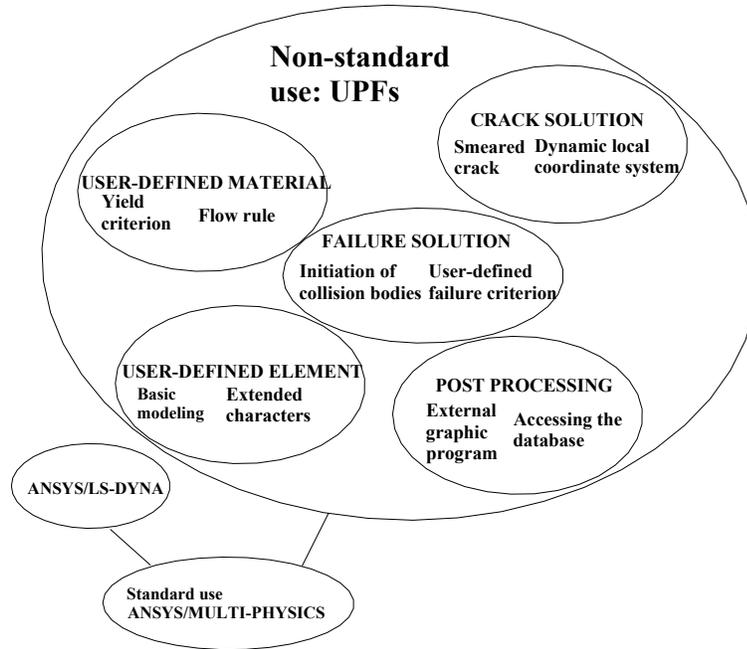


Figure 7 - Integration

Conclusion

Based on integration of UPFs and ANSYS/LS-DYNA, a solution plan is introduced to carry out the simulation of RC structure collapse due to earthquake. Both user-defined element and material are proposed to make the RC structure response under earthquake more accurately. By this means, the whole collapse process can be simulated quite well. And use of the open architecture of ANSYS is a highly efficient way to develop structure analysis program, and save structure engineers' time to deal with structure itself.

Acknowledgement: The research presented in this paper was performed with ANSYS Simulation Lab offered by ANSYS Inc. (CHINA). And it was also financially supported by Chinese Ministry of Education (Fund No. 99042).

References

- ANSYS. Inc., 2001, "ANSYS Theory Reference", Southpointe.
- ANSYS. Inc., 2001, "Guide to ANSYS User Programmable Features", Southpointe.
- Bathe, K. J., 1996, "Finite Element Procedures", Prentice Hall, Englewood Cliffs, New Jersey.
- Chopra, A. K., 1995, "Dynamics of Structures", Prentice Hall, Englewood Cliffs, New Jersey.
- Okamura, H., and Maekawa, K., 1991, "Nonlinear Analysis and Constitutive Models of Reinforced Concrete", Gihodo Co. Ltd., Tokyo, pp.90-109.