Damage analysis of RC tall building structures under earthquake loading

Zheng Li*, Qingying Ren†

†China Architecture Design & Research Group, Beijing, 100044, China
*Corresponding author’s e-mail: zhenglitju@163.com

Abstract. A Practical 1D cyclic model for concrete is established. The model is a combination between the skeleton curves in Chinese Concrete Code (GB50010-2010) and the modified hysteretic rule in Yassin model. The cyclic model has the tensile and compress damage variables which are able to depict the damage of concrete. The present model is developed as a user material subroutine VUMAT in ABAQUS. Then, the damage evolution analysis of a RC tall building structure was performed by the proposed damage model and the built-in damage model for concrete in ABAQUS. The results show that the cyclic model is suitable for the damage evolution and pattern analysis of RC tall building structures under earthquake loading.

1. Introduction

RC structures are widely used in the world as a main kind of building structures. And the seismic performance of RC structures is paid attention to by the engineers and researchers. However, because the property of the concrete as a composite material is more complicated and the uncertainty of the earthquakes excitation, the severe damage and even the collapse of the RC structures take place sometimes under strong earthquakes. Thus simulation of the damage pattern and evolution of RC structures under earthquake loading by the computer directly becomes an interesting field. This is more useful for the structural designers to identify the weak positions of structures and take some enhanced measures beforehand. There are two practical methods in simulation of the seismic damage of RC structures basically, i.e. lumped damage model and the distributed damage model. The latter is more attractive nowadays because of the development of the capacity of computer. The uniaxial damage model under cyclic loading is the key part of the distributed (fiber) damage model method.

More accurate stress-strain curves of concrete under compression and tension are suggested in the Chinese Code: Code for Design of Concrete Structures (GB50010-2010) [1], where the tensile and compressive damage variables are adopted. They are assumed as skeleton curves in the study. And the modified version of the hysteretic rule in Yassin model [2] is proposed. The combination between them forms a practical 1D cyclic constitutive model for concrete which is applicable to the seismic nonlinear response or damage analysis of RC structures. On the other hand, there is a built-in concrete damage model [3] in the finite element software ABAQUS. However, the built-in model is not suited to the beam element in ABAQUS. In addition, the geometrically simplified beam-column element needs a relatively refined 1D concrete model for an accurate seismic performance evaluation of the RC frame related structures. Based on ABAQUS, the user material subroutine VUMAT is developed according to the proposed model. Then, the equivalent simulation method of the fiber section of the beam element [4] and the multi-layer section of the shell element are employed. With this modeling
approach. A RC frame-core wall structure was analyzed for the seismic damage process in the platform ABAQUS.

2. 1D cyclic model for concrete

2.1. Skeleton curves in Chinese Code
The tensile stress-strain skeleton curve is given by the following equations:

\[ \sigma = (1 - d_t)E_c \varepsilon \]

\[ d_t = \begin{cases} 1 - \rho_t \left[ 1.2 - 0.2x^5 \right] & x \leq 1 \\ 1 - \rho_t \left( x - 1 \right)^{1.7} + x & x > 1 \end{cases} \]

\[ x = \frac{\varepsilon}{\varepsilon_{c,t}} \]

\[ \rho_t = \frac{f_{t,r}}{E_c \varepsilon_{t,r}} \]

\[ \sigma = (1 - d_c)E_c \varepsilon \]

\[ d_c = \begin{cases} 1 - \frac{\rho_c n}{n - 1 + x^n} & x \leq 1 \\ 1 - \frac{\rho_c}{\alpha_c \left( x - 1 \right)^{1.7} + x} & x > 1 \end{cases} \]

\[ \rho_c = \frac{f_{c,r}}{E_c \varepsilon_{c,r}} \]

\[ n = \frac{E_c \varepsilon_{c,r} - f_{c,r}}{E_c \varepsilon_{c,r}} \]

\[ x = \frac{\varepsilon}{\varepsilon_{c,r}} \]

Where, \( E_c \) is the elastic modulus; \( \sigma \) and \( \varepsilon \) are the tensile stress and strain, respectively; \( d_t \) is the tensile damage variable; \( f_{t,r} \) is the tensile strength; \( \varepsilon_{t,r} \) is the strain corresponding to the \( f_{t,r} \).

The compressive stress-strain skeleton curve is given by the following equations:

\[ \sigma = (1 - d_c)E_c \varepsilon \]

\[ d_c = \begin{cases} 1 - \frac{\rho_c n}{n - 1 + x^n} & x \leq 1 \\ 1 - \frac{\rho_c}{\alpha_c \left( x - 1 \right)^{1.7} + x} & x > 1 \end{cases} \]

\[ \rho_c = \frac{f_{c,r}}{E_c \varepsilon_{c,r}} \]

\[ n = \frac{E_c \varepsilon_{c,r} - f_{c,r}}{E_c \varepsilon_{c,r}} \]

\[ x = \frac{\varepsilon}{\varepsilon_{c,r}} \]

Where, \( d_c \) is the compressive damage variable; \( f_{c,r} \) is the compressive strength; \( \varepsilon_{c,r} \) is the strain corresponding to \( f_{c,r} \).

Moreover, an expression is introduced as follows:

\[ E_c = 1.2 \frac{f_{t,r}}{\varepsilon_{t,r}} \]

2.2. Modified hysteretic rule in Yassin model
The compressive hysteretic behavior of the modified rule is shown in Figure 1. In the compressive skeleton, the damage variable is an elastic damage quantity. Considering the plastic effect, the degenerated stiffness is determined by the focus model. The coordinates of the focus are as follows:

\[ \varepsilon_r = \frac{0.2 f_{c,r} - E_{20} \varepsilon_{20}}{E_c - E_{20}} \]

\[ \sigma_r = E_c \varepsilon_r \]
Where, \(E_{20}\) is the reloading stiffness corresponding to \(0.2f_{c-r}\cdot e_{20}\) is the strain corresponding to \(0.2f_{c-r}\). Different from the original Yassin model, the Bezier-curve unloading path is defined as follows.

First of all, the coordinates of the reference point (control point) are as follows:

\[
e_a = \frac{E_t\varepsilon_{m} - E_p\varepsilon_{p} - \sigma_m}{E_t - E_p}
\]

\[
\sigma_a = E_p\left(\varepsilon_a - \varepsilon_p\right)
\]

Thus, the unloading path is given by the expression:

\[
\sigma_{\text{max}} = (\sigma_m - 2\gamma_a)t^2 + 2(\gamma_a - \sigma_m)t + \sigma_m
\]

\[
t = \frac{-a_1 \pm \sqrt{a_1^2 - 4a_2a_0}}{2a_2} \in [0, 1]
\]

\[
a_0 = \varepsilon_m - \varepsilon
\]

\[
a_1 = 2(x_a - \varepsilon_m)
\]

\[
a_2 = \varepsilon_m - 2x_a + \varepsilon_p
\]

The linear reloading path is given by the expression:

\[
\sigma_{\text{min}} = 0.5E_t\left(\varepsilon_c - \varepsilon_i\right)
\]

\[
E_r = \frac{\sigma_m - \sigma_i}{\varepsilon_m - \varepsilon_i}
\]

\[
\varepsilon_t = \varepsilon_m - \frac{\sigma_m}{E_r}
\]

Where, \(\sigma_m\) and \(\varepsilon_m\) are the unloading stress and strain, respectively in the compressive skeleton curve.

When partial unloading and reloading happen, the full unloading and reloading paths become two small envelopes.

Taking into account the compressive damage effect, the tensile stress expression is modified as follows.

\[
\sigma = E_{tp}\varepsilon
\]

\[
E_{tp} = (1 - d_t)(1 - s\varepsilon)E_t
\]

Where, \(s \in [0, 1]\). The tensile hysteretic loops are shown in figure 2.
3. Modeling approach in ABAQUS

ABAQUS has user-friendly pre-postprocessor, a lot of built-in material models including concrete damaged plasticity model and the function of multi-layer shell element [3]. However, the modeling work for complex building structures is difficult in ABAQUS preprocessor. In the present study, the structure modeling tool is YJK [5], which is a building structure design software with the model import function from YJK to ABAQUS. And the imported model contains the real reinforcement data. This makes the modeling very fast and convenient.

A material subroutine VUMAT is developed by FORTRAN for the proposed 1D cyclic constitutive model for concrete. Then the developed 1D model could be used in fiber beam elements in ABAQUS. On the other hand, the built-in 2D damaged plasticity model for concrete is applied to the shell element case. Additionally, the kinematic hardening plasticity model in ABAQUS is used as the constitutive model for rebars.

The beam and column members are simulated with the fiber beam model, and the reinforcement in the beam and the column are simulated by the equivalent modeling method [3], as shown in figure 3. And the slab and wall members are simulated with the multi-layer shell element which is a built-in function in ABAQUS through the rebar layer definition in the shell section, as shown in figure 4.

Figure 3. Discretization of beam and column sections.

Figure 4. Multi-layer shell element in ABAQUS.

4. Example study: a RC frame-core wall structure

4.1. Finite element model

The structure system of the RC tall building is the frame-core wall structure. The planar layout of the structure is shown in figure 5. The height of each floor is 3.3m. The section size of columns is 500 mm x 500 mm, the section size of beams is 300 mm x 500 mm, the thickness of slabs is 100mm, the thickness of walls is 300mm. The strength grade of concrete is C25 for beams, columns and walls and C20 for slabs. The strength grade of the longitudinal reinforcement is HRB335. And the reinforcement of the members is calculated automatically by YJK.

The finite element model is shown in figure 6. The proposed cyclic model for concrete is adopted with the beam element B32 for the beams and columns. The built-in damage model for concrete is used with the shell element S4R for the walls and slabs. And the confinement effect of the concrete is not taken into account in the example study. The explicit solver ABAQUS/Explicit is used.
For the sake of simplicity, the El-Centro earthquake recorder (NS component) is adopted as a uniaxial input. However, the acceleration amplitude of the earthquake recorder is scaled to make PGA equal 125gal. The waveform is shown in figure 7.

4.2. Damage progress under earthquake loading

In the analysis, tensile and compressive damage value of each concrete fiber for the beam element and layer for the shell element can be obtained and displayed by ABAQUS. The contour of damage in the typical concrete fiber/layer is selected to show the damage process of the whole structure. As shown in figure 8 (a), (b), (c), the tensile damage pattern of the frame part was different from the pure frame structure, because of the influence of the core wall and the degree of damage was not serious as a whole. And the tensile damage zone of the core wall part appeared in the bottom. And the coupling beam was damaged relatively severely under the tensile stress. On the other hand, the degree of the compressive damage of the frame part was slight, as shown in figure 9. However, the medium damage still occurred in the he coupling beam in the lower zone under the compressive stress. The maximum in-plane stress of the slabs was less than 0.4MPa, so there was no damage in it.
Figure 8. Tensile damage contour at 2.0s (a), 3.8s (b), 12.0s (c).

Figure 9. Compressive damage contour at 12.0s.
5. Conclusions

The practical 1D cyclic constitutive model for concrete is proposed and implemented in ABAQUS. And then the damage pattern and process of a RC tall building structure was analyzed under earthquake loading in ABAQUS. The damage contour of the structures illustrates the characteristics of the damage evolution of the RC frame-core wall structures directly, which is in good agreement with the actual seismic damage phenomenon.

References

[1] MOHURD of China. (2010) Code for Design of Concrete Structures (GB50010-2010). China Architecture & Building Press, Beijing.
[2] Mohd-Yassin, M.H. (1994) Nonlinear analysis of prestressed concrete structures under monotonic and cyclic loads. University of California, Berkeley.
[3] ABAQUS Inc. (2014) ABAQUS User’s Manual (Version 6.11). ABAQUS Inc., Rhode Island.
[4] Li, C., Li, Z., Wang, G. (2007) Study on elastic-plastic finite element analysis based on section fiber beam-column elements. Building Structures, 37:33-35.
[5] YJK Inc. (2019) YJK User’s Manual (Version 2.0.0). YJK Inc., Beijing.