Damage Simulation of Large-Scale Concrete Structures Subjected to Earthquake Excitation Based on an Equivalent Damage Element Method

X. Q. Yin, J. B. Li, G. Lin & H. Zhong
School of Civil and Hydraulic Engineering, Dalian University of Technology, Dalian, China

SUMMARY:
Mesomechanics is widely applied to simulate the fracture process of concrete structures, while it's not capable of simulating a large-scale concrete structure owing to the huge computational capacity needed. For purpose of engineering practice, this paper presents an equivalent damage (ED) element for failure or damage analysis of large-scale concrete structures, and a practical external-interface with ANSYS is developed based on User Programmable Features (UPFs).The influence of mesoscale heterogeneity of concrete as approximately simulated by ED element, which is of relatively bigger size and assumed as homogeneous, is considered by randomly prescribed material properties according to the Weibull distribution law. The external-interface with ANSYS, which is portable and convenient to use, is developed as the basic computation tool. To justify the applicability and correctness of the proposed method for engineering purpose, seismic overload response analysis of Koyna Gravity concrete dam during the 1967 earthquake is presented. In addition, a group of meshing schemes with different sizes is addressed to investigate the effect of the sizes of ED elements and an optimized mesh size is determined based on the comparison among different mesh sizes.

Keywords: Mesomechanical model; Large-scale structure; Equivalent Damage element; UPFs; ANSYS

1. INTRODUCTION

It is commonly known that the large-scale concrete structure failures due to strong earthquakes will cause catastrophic consequences, such as the Koyna dam in India suffered severe damage cracking in 1967. To eliminate the potential risks brought by structure failures, seismic safety evaluation of such structures is well recognized. Among them, the simulation of the fracture process of concrete, which is a complex heterogeneous material widely used for the construction of structures, is a crucial issue.

For simplicity, it is convenient to take the concrete as a homogeneous material without considering the heterogeneity in engineering applications. In recent years, with the development of computer hardware and software techniques levels continual enhance, the mesoscale models have emerged as a powerful numerical procedure for simulating the fracture process and evaluating the macroscale response of concrete considering the heterogeneity, including the random particle model by Bazant et al. (1990), the lattice model by van Mier and his associates (1997), the mesoscale elastic-brittle model by Tang C. A.(2003), and M-H mesoscale mechanical model by Mahamed and Hansen (1999a, b). In all these models, the concrete is taken as heterogeneity material composed of three phases, i.e., mortar, aggregates, and interfaces between them at the meso-level and most work focus on static response of the laboratory specimens, while cases of practical engineering structures subjected to cyclic loading are not available. Then, instead of simulating the multiphase components of concrete, Zhong, H. et al. (2011) and Tang, X. W. et al. (2011) have extended to seismic failure modeling of high concrete dams, in which the structures are discretized by using mesoscale mesh of finite elements and the influence of heterogeneity of concrete is approximated by random distribution of material properties.

In this present paper, we developed the Equivalent Damage (ED) element based on the mesoscale model in the commercial finite element (FE) code ANSYS (2007) for fracture process simulation of large-scale structures subjected to earthquake excitation, with the main purpose of providing practical tools for researchers and engineering practitioners. In the seismic analysis, the meso heterogeneity of concrete simulated by ED elements, which are with refined mesh at relatively meso-level, i.e., the size of the element is small enough in contrast to the structures and assumed as homogeneous, is considered by randomly prescribing the material properties in each ED element according to the Weibull distribution law. Meanwhile, in strong earthquake shocks, although the structures may go through strong nonlinear process and the numerical model has the potential to incorporate various factors including complicated geometry and applied loaded, flexibility of foundation and large computation, the ANSYS capabilities for analysis of nonlinear problems together with the user interface options available in the code have proved to be useful for such a seismic analysis. In addition, the user can benefit from the many built-in features of ANSYS, such as pre-processing, post processing and powerful computational capability. Base on that, the damage evolution and failure pattern of the Koyna gravity dam are investigated as a numerical demonstration. Furthermore, the effect of ED element size on simulation result is explored and an optimized mesh size is determined.

2. SEISMIC ANALYSIS CONSIDERING HETEROGENEITY OF CONCRETE

2.1. Heterogeneous of concrete

As previously mentioned, the concrete is always divided into three phases at the meso-level in the simulation of failure process of concrete specimens in laboratories. However, it is unrealistic and unnecessary for large-scale structures to model exactly three phases of concrete in seismic analysis. Therefore, based on the characteristic of large-scale concrete, the structure can be idealized as a bigger sample space and discretized with finite elements at relatively meso-level, while each element referred to as ED element is assumed to be homogeneous and isotropic due to the influence of heterogeneous in such a small element for the structure is not relatively effective. To capture the heterogeneous characters of concrete, mechanical parameters of each ED element, including the elastic modulus, the Poisson's ratio and the strength, are assumed to conform to the Weibull distribution law, whose probability density function is shown as follows:

$$f(u) = \frac{m}{u_0}\left(\frac{u}{u_0}\right)^{m-1}\exp\left(-\frac{u}{u_0}\right)^{m} \tag{2.1}$$

where u is a given mechanical parameters of each element (such as the elastic modulus or strength); u_0 is average value of the corresponding parameter and m is the heterogeneity index which quantifies the degree of material heterogeneity in the sample space. According to the definition of the Weibull distribution, a greater m corresponds to a more homogeneous numerical model of structure, on the contrary, a more heterogonous model will be obtained. As a result, the concrete of large-scale structures can exhibit the feature of heterogeneous through refined discrete model with the ED elements whose mechanical properties are defined without uncertainty once the assignment process of properties has been completed.

2.2. Damage evolution and failure criterion

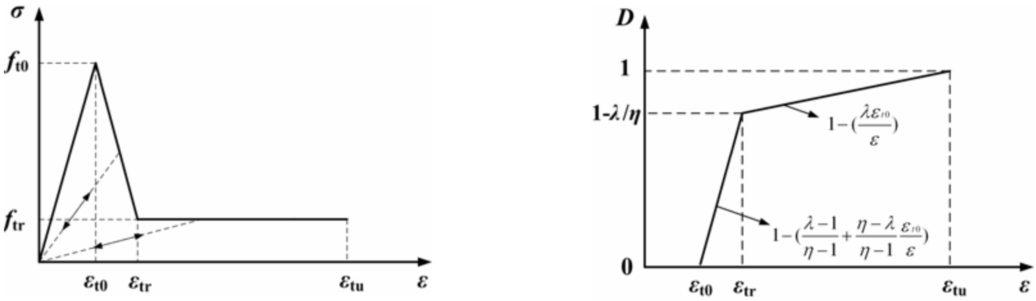
It is recognized that damage and nonlinear of concrete are leading mortality cause of the initiation and propagation of microcracks. In the proposed method, to simulate the behaviors of microcracks, all the ED elements are assumed to be with the elastic damage constitutive laws which are linear elastic until the damage occurs. Two damage evolution model for tensile and compression are applied for the definition of the constitutive relationship of the concrete and corresponding failure criterion, i.e. maximum tensile strain criterion and Mohr-Coulomb criterion are chosen as the damage threshold.

For each ED element, the material is assumed to be undamaged linear elastic and isotropic before the initiation of damage. Then the stiffness of the element is assumed to degrade gradually as damage progress and Poisson's ratio to be unaffected, with the elastic modulus of the damaged material given by

$$E = (1 - D)E_0 \tag{2.2}$$

where E_0 and E are the initial elastic modulus and damaged elastic modulus respectively and D represents the damage variable, which ranges from zero for the undamaged material to one for complete damaged state.

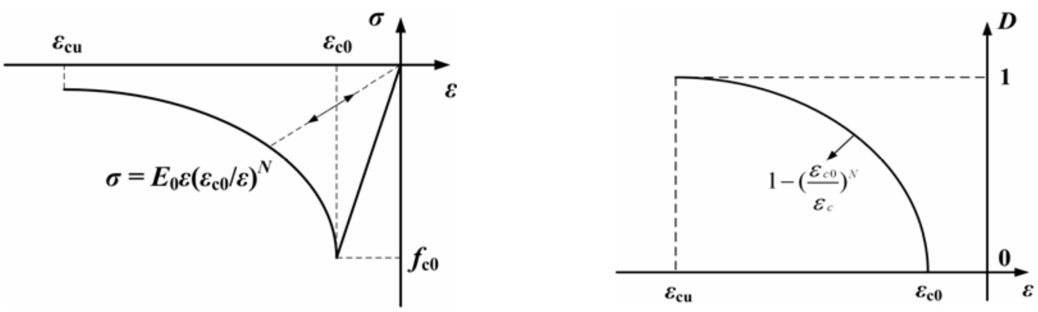
For the tension introduced damage, the constitutive relation and damage evolution model for an ED element are illustrated in the Fig. 2.1. Hereinto, the Fig. 2.1(a) shows the bilinear stress-strain curve with the softening branch by a simple linear function, while the Fig. 2.1(b) shows the damage variable D at any given strain to describe the damage evolution process. When the maximum tensile damage criterion is met, damage of the ED element occurs and the stress-strain curve descends linearly before reaching a specified residual strength. It is followed by serious damage for ED element with a constant residual strength until the element is completed damage when the tensile strain is larger than the ultimate strain. As is shown in the Fig. 2.1(b), ε_{t0}, ε_{tr}, ε_{tu} represent the elastic strain limit, threshold for the residual section and the ultimate strain, respectively. $\eta = \varepsilon_{tr}/\varepsilon_{t0}$ characterizes the range of the descending section. Parameter λ stands for the ratio between residual strength f_{tr} and f_{t0} and lager implies the stronger bearing capacity of the element after being damaged.



(a) stress-strain curve (b) damage evolution

Figure 2.1 Constitutive relation and damage evolution model for concrete in tension

Corresponding to the tensile damage, a constitutive law is given in Fig. 2.2(a) in compression with the softening section by a power function under which shear damage is assumed to occur according to the Mohr-Coulomb criterion and the damage variable D is described in Fig. 2.2(b), where ε_{c0} and ε_{cu} represent the elastic strain limit and threshold for the ultimate strain, respectively.



(a) stress-strain curve (b) damage evolution

Figure 2.2 Constitutive relation and damage evolution model for concrete in compression

When the structures are subjected to the external loading, considering the tensile failure is much easier to appear, the tensile damage is always checked first and the shear damage is omitted once the tensile damage occurs, while the shear damage is checked only when the tensile damage is not attained. In addition, according to the damage evolution damage model stated above, the cracking of concrete is taken to be coincident with only the completed damage ED elements, that is whose tensile strains attain or exceed the ultimate strain are taken to have cracked with a crack width equal to the ED element width. Furthermore, shear damage will lead only to degradation but not to the appearance of cracks. Based on these assumptions, the initiation, propagation and interaction of multiple cracks can be simulated easily.

2.3. Incremental seismic analysis with iterations

During the structures subjected to the earthquake excitation, once the strains of ED elements reach the damage threshold, the properties will weaken gradually and may complete damaged at last. Such a phenomenon is typical of a nonlinear problem, which can only be solved by incremental methods, with iterations for stress redistribution and removing the unbalance force. Consider a typical incremental step from time t to $t + \Delta t$, the equation of motion for structures at the time $t + \Delta t$ can be written as

$$[M]\{\ddot{u}\}_{t+\Delta t} + [C]_{t+\Delta t}\{\dot{u}\}_{t+\Delta t} + [K]_{t+\Delta t}\{u\}_{t+\Delta t} = \{F^a\}_{t+\Delta t} \tag{2.3}$$

where $[M]$, $[C]$ and $[K]$ are mass, damping and stiffness matrices of the structure discretized by ED elements, respectively; $\{u\}$, $\{\dot{u}\}$ and $\{\ddot{u}\}$ are nodal displacement, velocity and acceleration, respectiv-ely; and $\{F^a\}$ is the applied load vector. As shown in the Fig. 2.1(a) and Fig. 2.2(a), a damaged ED element is unload or reload elastically along the same path with its degraded elastic modulus. Therefore, based on the Newmark's β method with constant average acceleration, i.e., $\beta = 0.25$ and $\gamma = 0.5$, the Newmark assumptions should be modified to include the feature of iteration, that is

$$\{u\}^i_{t+\Delta t} = \{u\}_t + \{\Delta u\}^i \tag{2.4a}$$
$$\{\dot{u}\}^i_{t+\Delta t} = a_1\{\Delta u\}^i - a_4\{\dot{u}\}_t - a_5\{\ddot{u}\}_t \tag{2.4b}$$
$$\{\ddot{u}\}^i_{t+\Delta t} = a_0\{\Delta u\}^i - a_2\{\dot{u}\}_t - a_3\{\ddot{u}\}_t \tag{2.4c}$$

in which the right superscript i on each symbol indicates the number of iteration and the coefficients and those to appear are defined as

$$a_0 = \frac{1}{\beta\Delta t^2};\ a_1 = \frac{\gamma}{\beta\Delta t};\ a_2 = \frac{1}{\beta\Delta t};\ a_3 = \frac{1}{2\beta}\text{-}1;\ a_4 = \frac{\gamma}{\beta}\text{-}1;\ a_5 = \frac{\Delta t}{2}(\frac{\gamma}{\beta}\text{-}2);\ a_6 = \Delta t(1-\gamma);\ a_7 = \gamma\Delta t \tag{2.5}$$

By the finite difference Eqn. 2.4, the Eqn. 2.3 of motion can be manipulated to yield the following equivalent stiffness equations:

$$[\bar{K}]^i_{t+\Delta t}\{\Delta u\}^i = \{F^a\}_{t+\Delta t} - \{\bar{F}\}^i_{t+\Delta t} \tag{2.6}$$

where the effective stiffness $[\bar{K}]^i_{t+\Delta t} = a_0[M] + a_1[C]^i_{t+\Delta t} + [K]^i_{t+\Delta t}$ and effective resistant force vector $\{\bar{F}\}_t$ is

$$\{\bar{F}\}^i_{t+\Delta t} = \{F^{\text{int}}\}^i_{t+\Delta t} - [M](a_2\{\dot{u}\}_t + a_3\{\ddot{u}\}_t) - [C]^i_{t+\Delta t}(a_4\{\dot{u}\}_t + a_5\{\ddot{u}\}_t) \tag{2.7}$$

in which $\{F^{\text{int}}\}$ is the vector of restoring loads calculated from the element stresses.

In accordance with the above-state methods, for each time step, the displacement increments $\{\Delta u\}^i$ can be obtained from Eqn. 2.6 using the elastic modulus in the last iteration of previous time step and total motion vector from Eqn. 2.4 at first, and then check whether or not there are damaged elements by the failure criterion stated in section 2.2 If no new damage appears, go to next time step without iterations. Otherwise update elastic modulus from Eqn. 2.2 and recalculate displacements and stresses with iterations until no damage occurs or the convergence condition is satisfied.

3. ANSYS IMPLEMENTATION OF ED ELEMENT

The commercial FE software ANSYS provides three tools, i.e. UPFs, ANSYS Parametric Design Language (APDL) and User Interface Design Language (UIDL) for user to customize and expand its existing capabilities. In this section, we describe the main features related to the implementation of the ED element through the user subroutine interface *UserElem* based on UPFs in ANSYS Version 11.0. Although implementing a non-standard finite element in ANSYS does impose certain restrictions, it also provides access to many of the available features. In our implementation, we will also give a general overview of the pre- and post- processing for the sake of facility of understanding and grasping the whole process and extending its application to a broader class of problems.

3.1. Preprocessing: Model with ED element

For the user-defined element, the element type will not be embedded in element library of ANSYS even after compiling and linking the user element subroutine into standard program. According to the programmer's manual for ANSYS, *ET*, *USRELEM* and *USRDOF* commands are used to define the element type, element characteristics and nodal degree of freedoms (DOFs) of user element, respectiv-ely. It is worth noting that the name of the element must be *user300* and ANSYS does not have capabilities for user element to preprocess and post process when *KeyShape* option which is the item of *USRELEM* is *ANYSHAPE* (that is, no specified shape). Because no restrictions are placed on the shape of ED elements only if the size is small enough, the pre- and post- processing are supplied by the user directly through the Graphical User Interface (GUI) or APDL. Therefore, other steps for analysis such as defining real constants, creating FE models, applying boundary conditions and loads and specifying solution options are similar to standard element. Alternatively, the standard element that has the same element characteristics (such as the number of nodes, dimensions and DOFs etc.) as the ED element is used in the mesh generation and modified by ED element using *EMODIF* command in the process of analysis in order to exert the advantages of GUI and APDL for complex structures.

3.2. Equivalent Damage element definition

UPFs are capabilities for which you can write your own FORTRAN routines and allow you to customize the ANSYS program to your needs, which may be a user-defined material-behavior option, element, failure criterion (for composites), and so on. In present method, the user subroutine *UserElem* which is used to develop the ED element can provide an interface to ANSYS code above the element level. The subroutine passes all element information (such as element characteristics, properties and motion vector of nodes etc.) needed to create a user-defined element and returns all data (such as stiffness, mass and damping matrices etc.) and results from the element to update the ANSYS database and files. To run an analysis including the user-defined subroutine, what we should do first is to design and program the custom subroutine *UserElem* which constitute the core of the implementation, and then compiling and linking subroutine to ANSYS should be accomplished so that the user-defined element can be employed by ANSYS. Compiling and linking UPFs on different system are described in detail in the manual (2007), so only the development of subroutine *UserElem* is summarized here.

3.2.1. Modified effective force

ANSYS internal procedures for solving the nonlinear equations through Newton-Raphson method are not applicable to solve the Eqn. 2.6, since there is a difference in iteration process between ED element and standard element (as show in Fig. 3.1). Fig. 3.1(a) shows an iteration process for standard element in which the resistant force vector $\{F^{nr}\}$ are computed from configuration $\{u^i\}$ in last iteration and the difference between $\{F^a\}$ and $\{F^{nr}\}$ is the actual applied force in this iteration, whereas Fig. 3.1(b) shows an iteration process for ED element with the applied force remaining constant. Therefore, minor modification is required to accommodate the effective force. In the nonlinear seismic analysis, the Newton-Raphson method is employed along with the Newmark assumptions, the equation with the feature of iteration can be written as

$$[\bar{K}]^i_{t+\Delta t}\{\Delta u\}^i_{t+\Delta t} = \{F^a\}_{t+\Delta t} - \{F^{nr}\}^i_{t+\Delta t} \tag{3.1}$$

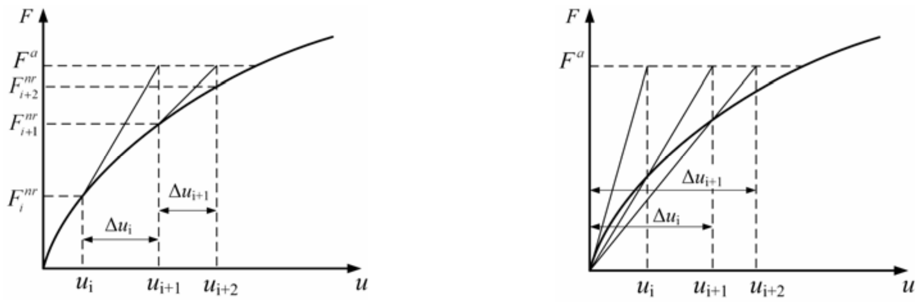
where $[\bar{K}]$ with the same expression as in the Eqn. 2.6 and the effective resistant force $\{F^{nr}\}$ is

$$\{F^{nr}\}^i_{t+\Delta t} = \{F^{\text{int}}\}^i_{t+\Delta t} + [M](a_0\{\Delta u\}^i_{t+\Delta t} - a_2\{\dot{u}\}_t - a_3\{\ddot{u}\}_t) + [C]^i_{t+\Delta t}(a_1\{\Delta u\}^i_{t+\Delta t} - a_4\{\dot{u}\}_t - a_5\{\ddot{u}\}_t) \tag{3.2}$$

where $\{F^{\text{int}}\}$ is determined in the subroutine above the element level, while the inertia and damping effects are defined in the standard ANSYS use run. As can be seen in the Fig. 3.1, although $[\bar{K}]$ and $\{\Delta u\}$ stand for different means in two iteration process, the same expressions can be obtained. Therefore, $[\bar{K}]$and $\{\Delta u\}$ for ED element instead of standard element returns to ANSYS database by taking advantage of two important displacement vector, that is total displacement $\{u^{\text{t}}\}$ and iteration displacement $\{\Delta u^{\text{it}}\}$ at the last iteration. The effective force vector in Eqn. 2.7 is modified as

$$\begin{aligned}\{F^M\}^i_{t+\Delta t} = \{F^{\text{int}}\}^i_{t+\Delta t} &- [M](a_0(\{u^t\}^i_{t+\Delta t} - \{u^t\}_t) - a_2(\{\dot{u}^t\}_t - \{\dot{u}^a\}_t) - a_3(\{\ddot{u}^t\}_t - \{\ddot{u}^a\}_t)) \\ &- [C]^i_{t+\Delta t}(a_1(\{u^t\}^i_{t+\Delta t} - \{u^t\}_t) - a_4(\{\dot{u}^t\}_t - \{\dot{u}^a\}_t) - a_5(\{\ddot{u}^t\}_t - \{\ddot{u}^a\}_t))\end{aligned} \tag{3.3}$$

and the $\{\Delta u\}^i$ in Eqn. 2.4 is modified as $\{\Delta u^{\text{it}}\}$, where the superscript a represents the actual motion vector in the last load step. However, as one of the output result, $\{u^{\text{t}}\}$ is not the actual response of structures due to the accumulation effect in each load step, while the actual response is the $\{\Delta u^{\text{it}}\}$ in the last iteration.



(a) Solution for standard element

(b) Solution for ED element

Figure 3.1 The difference in iteration process between standard element and ED element

3.2.2. ED element implementation in ANSYS

On the basis of theory mentioned in section 2, the procedures of seismic analysis for concrete structures using ED elements in ANSYS are detailed as follows, and the part (c)-(e) is the ED element definition.

(a) Creating FE model of the large-scale structure, applying boundary conditions and specifying solution options are done by using GUI and APDL (described in section 3.1).
(b) After going into the solution stage, the load vector $\{F^{\text{a}}\}$ is applied in each loadstep firstly, and then
(c) ANSYS program gives element information from database to subroutine of ED element through the interface *UserElem* and runs the following code. To generate the target mechanical properties conformed to Weibull distribution law, the Monte Carlo method that generates the random number above all and combines with the probability density function in Eqn. 2.1 is implemented when first time entering the element. Otherwise, the mechanical properties defined as global variable (FORTRAN COMMON) are stored in a specified array and referenced in next iteration or other load steps. The strain ε of element is then calculated through the actual displacement ($\{u\}_{\text{t}}+\{\Delta u^{\text{it}}\}$) in current iteration to judge whether the element has been damaged.
(d) Check if the damage variable D^i_l (D^i_{lt} for tension and D^i_{lc} for compression) is greater than D^0_l where the superscript i indicates the current number of iteration, the superscript 0 denotes the last iteration of the previous load step and the subscript l indicates the current load step. If yes, the ED

element has been damage and the element will be checked if damage continues according to the corresponding damage model. Otherwise, perform the damage evolution model and failure criterion described in section 2.2 to judge. After determining the damage variable D_l^i , the damaged elastic modulus will be update using the Eqn. 2.2 and D_l^i of each element is need to output in the external file for further post processing of crack simulation after convergence.

(e) Determine the element matrix ([*K*] for stiffness, [*M*] for mass and [*C*] for damping) and effective force $\{F^M\}$ using Eqn. 3.3 If [*C*] is assumed to be the Rayleigh damping and α and β damping multipliers (*ALPHA* and *BETA* command) are defined in the standard ANSYS use run, additional damping effects will be applied to the ED element so that [*C*] should be not determined in this case. Similarly, stiffness matrix returns to ANSYS database is [*K*] rather than equivalent stiffness $[\bar{K}]$ due to the mass and damping effects has been considered in the use run.

(f) Return all data and results to ANSYS database, the globe matrix is then assembled and response is calculated by ANSYS code. Then the convergence conditions are checked and determine to go to the next iteration step or the next load step.

3.3. Post processing

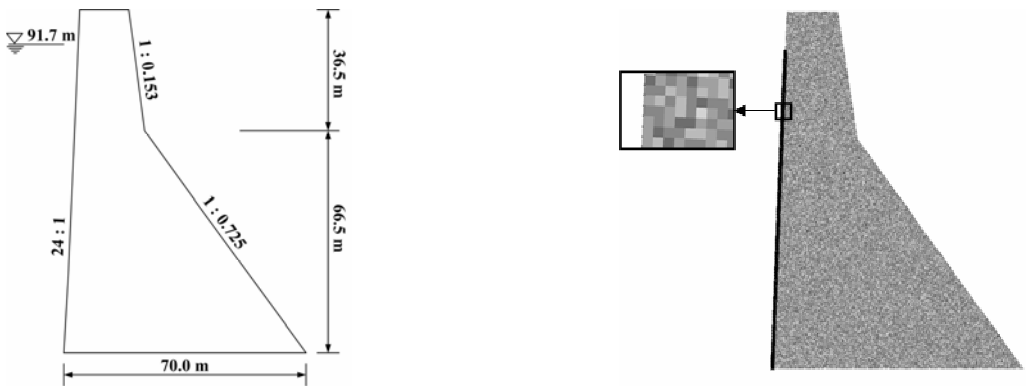
Due to the difference in the iteration process (described in section3.2.1) and total displacement of DOFs is not allowed to be modified, the response of structures stored in the .rst file is not the actual deformation. Therefore, we have output the actual motion vector into an external file in the specified format after convergence in each load step. After the solution is done, the external file contains all the relevant information of ED elements and *DNSOL* command is used to plot the deformation of structures through GUI. In addition, the ANSYS does not have capabilities to plot the process of initiation and propagation of multiple cracks because the code does not post-process the information generated by user-defended element. As stated in the section 2.2, the crack width of the concrete equal to the ED element width so that a simple approach which unselect the damaged ED elements according to damage information in each load step and the principal tensile stress is plotted simultaneously is present. We have used this approach for the numerical example that appears in section 4, this enables the multiple cracks to be seen clearly and forecast the trend to propagate.

4. FAILURE ANALYSIS OF KOYNA GRAVITY DAM

Based on the ED element embedded in standard ANSYS, the damage and fracture behavior of the Koyna gravity dam during the 1967 earthquake is provided to verify the mesoscale model as well as demonstrate the development of non-standard element for seismic analysis due to this problem has been extensively studied by other researchers. Also, the comparison is made with the failure pattern of the dam to explore the effect of sizes of the ED elements.

4.1. FE discretization and Material parameters

The values of material parameters for dam adopted in the analysis are similar to those by other investigators (Lee and Fenves 1998; Tang, X. W. et al. 2011): the dynamic elastic modulus E_0 = 31.0 GPa, the Poisson’s ratio v = 0.2, the mass density of concrete ρ = 2, 643 kg/m^3 and the dynamic tensile strength f_t = 2.9 MPa. Mechanical parameters of ED element are assumed to conform to the Weibull distribution and the heterogeneity index m is assumed to be 2 for elastic modulus and tensile strength and 100 for Poisson’s ratio. Fig. 4.1(b) presents the distribution of elastic modulus of dam and colors ranging from lighter to darker are used to describe the value of elastic modulus from lower to higher. Meanwhile, the parameters in the constitutive are listed in Table 4.1 (Hong Zhong et al. 2011). The dam-foundation interaction is neglected and the foundation is assumed to be rigid. The material damping is considered via the Rayleigh damping and the damping ratios for all modes of vibration are 5% in analyses.



(a) Geometry (b) Finite element mesh with ED element

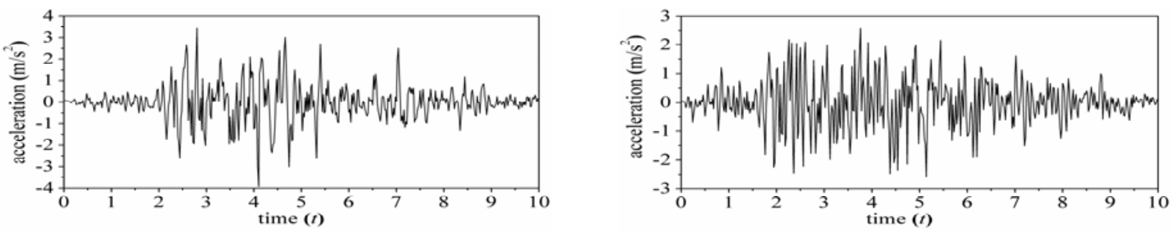
Figure 4.1 Geometry and FE model of Koyna Dam

Table 4.1 Parameters In The Constitutive

Type of constitutive relation	Parameters		
Tension	η = 2.0	ξ = 10	λ = 0.05
Compression	N = 4	ξ = 100	λ = 0.2

4.2. Loading conditions

The normal static loads including the dam gravity, hydrostatic pressure of reservoir on the upstream face and hydrodynamic pressures. The dynamic loads include earthquakes in two directions and the horizontal and vertical records of the Koyna earthquake are shown in Fig.4.2. The peak ground accelerations are 0.40g in the horizontal direction and 0.26g in the vertical direction. The hydrodynamic pressures are modeled via the Westergaard added-mass assumption and modeled with element MASS 21 in ANSYS (as shown in Fig.4.1 (b)).



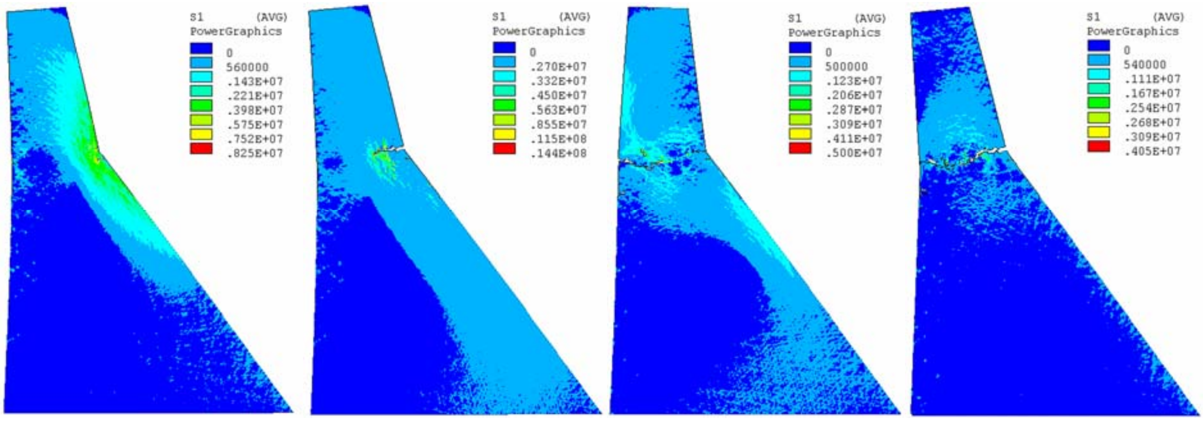
(a) Horizontal (b) Vertical

Figure 4.2 Koyna earthquake records of December 11, 1973

4.3. Failure process of the dam

During the dam subjected to acceleration time histories, four typical time points (2.62s, 4.28s, 4.78s and 4.98s) are chosen in the failure process of dam and the corresponding failure patterns with principal tensile stress (S1) are shown in Fig. 4.3.

As can be seen in Fig. 4.3, when the dam is loaded during the first two or three seconds, although only several element are damaged and no visible crack occurs, the slope of downstream surface changes are characterized by the high tensile stress which leads to further damage. With the on-going acceleration excitation, at 4.28s, a crack appears and propagates from the downstream surface with high tensile stress. Then due to the rigorously shocking in the downstream direction, at 4.78s a crack appears in the upstream surface and propagates towards downstream. During the next seconds, these two cracks propagate quickly and finally meet indicating the dam fails, and then the other local and short cracks appear until the end of calculation. As a result, one main horizontal crack initiates and propagates through the dam at the level where slop changes abruptly which is consistent with those reported by other researchers (Lee, J. and Fenves 1998; Pekau, O. A. et al. 1995), while the some other local damage also occurred due to considering the effect of the concrete heterogeneity.



(a) 2.62s (b) 4.28s (c) 4.78s (d) 4.98s

Figure 4.3 Failure process of dam with principal tensile stress (deformation scaled by 100)

4.3. Effect of sizes of ED elements

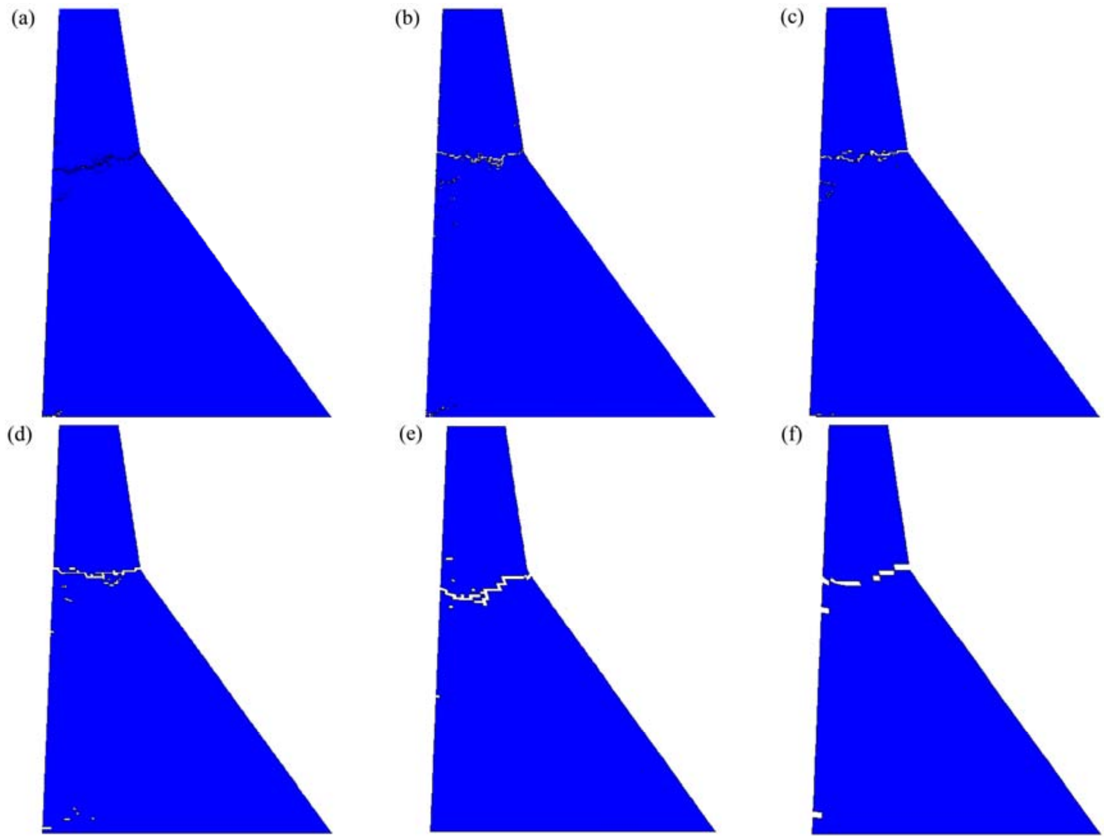


Figure 4.4 Effect of element sizes on failure patterns. (a) 0.1m. (b) 0.3m. (c) 0.5m. (d) 0.8m. (e) 1m. (f) 2m.

The size of the ED element which is at the relatively meso-level with no specific definition can influence the simulation results and this influence is considered here. The meso-level is generally considered to be of the scale of 10^{-3} *m* for the study of the macroscopic mechanical response of concrete. Compared with the large-scale structures at 100 *m* level, the referred relatively meso-level is suitable to be of the scale of 10^{-2} *m* or10^{-1} *m*. Fig. 4.4 shows a set of failure patterns of the Koyna dam as examined above using six element sizes (widths): 0.1 *m* (91318 elements), 0.3 *m* (43238 elements), 0.5 *m* (20006 elements), 0.8 *m* (10506 elements), 1.0 *m* (4238 elements), 2.0 *m* (1405 elements), with all other conditions being the same. It can be seen that the places which the main crack occurs for all results are basically accorded, although the widths of the crack are obvious difference due to the size of the elements. However, an increase in the size of elements leads to a more homogeneous character of the dam with local damage rarely occurring owing to the assumption of homogeneous for the ED element. The results for the smaller element size are much closer to each other and the short cracks may occur on the dam face. Therefore, the smaller the element size, the more heterogeneous the concrete will be. Meanwhile, the 0.5 $m \times 0.5\ m$ elements or smaller sizes are acceptable for the numerical simulation of structures at 100 m level. In order to strike a balance between the need to model the concrete heterogeneity and computational efficiency, a refined mesh is used for the potential parts of stress concentration alternatively.

5. CONCLUSIONS

In order to make the feasibility and effectiveness of the mesoscale model for damage simulation of large-scale concrete structures subjected to earthquake excitation, the ED element method is developed and a systematic description of its implementation within the commercial FE software ANSYS through employing the user element interface *UserElem* is presented. The proposed ED elements which are used to discretize the concrete structures are assumed to be homogeneous at the meso-level while the influence of heterogeneity of concrete is approximated by randomly prescribing the material properties in each ED element. The user element interface *UserElem* which enables a successful integration of the ED element with the ANSYS platform after compiling and linking has been systematically developed. The damage and fracture behavior of the Koyna Gravity Dam was analyzed to demonstrate the reliability and flexibility of the ED element in the seismic analysis for larger-scale concrete structures. Comparisons with earthquake damage investigation and the work of other researchers verify the correctness of the method as well as the corresponding programming. Moreover, the simulation result is influenced by the size of the ED element, indicating that the relatively meso-scale (0.5 $m \times 0.5\ m$ or smaller) are preferable for the simulation of larger-scale structures.

ACKNOWLEDGEMENTS
This research work was supported by the Science Fund for Creative Research Groups of the National Natural Science Foundation of China (Grant No. 51121005) , Open Research Fund Program of State key Laboratory of Hydro science and Engineering (Grant No. shlhse-2010-C-03) and the State Key Program of National Natural Science of China (Grant No. 51138001)

REFERENCES

Bazant, Z. P. and Tabbara, M. R. (1990). Random particle models for fracture of aggregate or fiber composites. *Journal of Engineering Mechanics* **116:8**, 1686-1705.

Van Mier, J. G. M. (1997). Fracture processes of concrete: Assessment of material parameters for fracture models, CRC Press, Boca Raton, FL.

Tang, C. A. and Zhu, W. C. (2003). Concrete damage and fracture numerical simulate, Science Press, Beijing.

Mohamed, A. R. and Hansen, W. (1999a). Micromechanical modeling of concrete response under static loading: Part I-Model development and validation. *ACI Structural Journal* **96:2**, 196-203.

Mohamed, A. R. and Hansen, W. (1999b). Micromechanical modeling of concrete response under static loading: Part II-Model predictions for shear and compressive loading. *ACI Structural Journal* **96:3**, 354-358.

Zhong, H. and Lin, G. (2011). Seismic failure modeling of concrete dams considering heterogeneity of concrete. *Soil Dynamics and Earthquake Engineering*. **31:12**, 1678-1689.

Tang, X. W. and Zhou, Y.D. (2011). Study on the Heterogeneity of Concrete and Its Failure Behavior Using the Equivalent Probabilistic Model. *Journal of materials in civil engineering*. **23:4**, 402-413

ANSYS Incorporated. (2007). Programmer's Manual for ANSYS.

Lee, J. and Fenves, G. L. (1998). Plastic-damage concrete model for earthquake analysis of dams. *Earthquake engineering and structural dynamics* **27:9**, 937-956.

Pekau, O. A., Feng, L. M. and Zhang, C. H. (1995). Seismic fracture of Koyna dam: Case study. *Earthquake engineering and structural dynamics* **24:1**, 15-33.