A FEM/DEM COUPLED AND EVOLVED MODEL AND ITS APPLICATION IN FAILURE SIMULATION OF GEOLOGICAL BODY

Chun Feng[1,2], Shi-hai Li[1], E. Oñate[2], Miquel Santasusana[2]

(1. Institute of Mechanics, Chinese Academy of Sciences, Beijing, China)

(2. International Centre for Numerical Methods in Engineering, Universitat Politècnica de Catalunya, Barcelona, Spain)

To simulate the failure process of geological body, a FEM/DEM coupled and evolved model is introduced. Traditional FEM and DEM algorithm is used, and explicit time integration method is adopted. To transfer the contact force between FEM elements and DEM elements, a point-edge contact model (2D) and point-face contact model (3D) are introduced in; normal and tangential springs are adopted. Strain softening Mohr-Coulomb model and maximum tensile model are selected for FEM. Once one FEM element reaches the critical plastic strain state, the FEM element will be deleted, and a particle will be generated at the same place with all information inherited from the deleted FEM. Some numerical cases show the accuracy and rationality of the model.

INTRODUCTION

The progressive failure of geological body is the main reason to cause different geological hazards. To simulate thus process precisely, the evolvement from continuous media to discontinuous media should be pay attention to. FEM and DEM are two important numerical approaches to simulate the failure process.

FEM is good at simulating continuous problems, such as elastic and plastic deformation of soil and rock mass under static or dynamic loads. For simulating the failure process of geological body, some skills should be introduced in, such as death element method (W.C. Zhu, et al, 2006), XFEM (Xiao Q Z, et al, 2005) and element cutting approach (Cottrell M G, et al, 2003). In death element method, once the element reaches the failure state, the element will be killed immediately, and then the initiation and propagation of crack could be simulated. Although death element method is simple to implement, it isn’t physical and the energy dissipation couldn’t be explained well. In XFEM, a jump function is used to represents the crack in an element, by which the discontinuous deformation could be gotten without cutting element. Although XFEM is a precise method to simulate the propagation of the crack, it could only solve some simple cases with only few cracks. In element cutting approach, when an element reaches the critical point, the element will be cut into two, and cutting direction is obtained according to the strength criteria. Although the cutting approach could form explicit crack, the cutting position is artificial defined, and element quality after cutting may be poor.

DEM is expert in solving discontinuous problems, such as collision and motion of granular media. Block DEM and particle DEM are two typical methods in DEM. When use block DEM to simulate the failure process, the crack only could occur on the boundaries of block, so the propagation direction of crack will be depends on the block shape entirely. Besides, the physical significance and affection of contact stiffness between two blocks should be carefully studied. In particle DEM, the particle is a rigid body, so the contact is used to represent the deformation and crack. Due to the random distribution of particles, the crack direction could be arbitrary (D M Yang, et al, 2011). However, the relationship between micro parameters (such as stiffness, damp, strength) and macro parameters (such as elastic modulus, Poisson ratio, and strength) are difficult to established, sometimes hundreds of numerical cases are needed.

For simulating the progressive failure process of geological body, a FEM/DEM coupled and evolved model is presented in this paper, and the advantages of FEM and DEM are integrated together.

MAIN IDEA

At very beginning, the numerical model is filled by FEM elements, and Hooke's law is used to calculate the elastic deformation, then softening Mohr-Coulomb model and maximum tensile model is introduced to obtain the equivalent shear plastic strain and maximum tensile plastic strain. If any plastic strain mentioned above exceeds the limit value, the FEM element will be killed, and the particles will be generated at the same place. The particle information will be inherited from the deleted FEM, and then the FEM/DEM contact force will be calculated. By the deletion of FEM element and generation of DEM element, the initiation and propagation of crack could be simulated, and Fig 1 shows the process from FEM to DEM. However, for simplifying the calculation, only one particle is created in an FEM element in this paper. The center of particle coincides with the centroid of FEM element, and the radius is the shortest distance from particle center to the sides of FEM element.

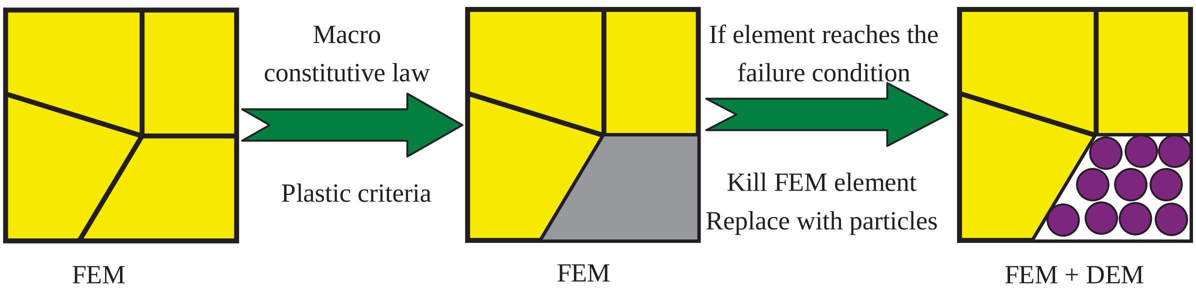


Fig 1. Process from FEM to DEM

ELEMENT STRAIN AND STRESS CALCULATION

The explicit time integration method on finite difference principle is adopted to calculate the progressive process through a time marching scheme. During calculation, the dynamic relaxation method is used to achieve convergence in a reasonable period time with small time steps.

To solve nonlinear problems (such as large rotation and plastic flow), incremental method is adopted and strain matrix [B] is used to calculate deformation force instead of stiffness matrix [K]. Besides, strain matrix [B] should be renewed at each time step. Eq. 1 shows the main steps to calculate node force by strain matrix with incremental method. Where $[B]_i$, $\{\Delta\varepsilon\}_i$, $\{\Delta\sigma\}_i$, w_i , J_i are strain matrix, incremental strain, incremental stress, integral coefficient and Jacobi determinant in Gaussian point i; $\{\sigma^n\}_i$ and $\{\sigma^o\}_i$ represent new stress and old stress in Gaussian point i; $[D]$, $\{\Delta u\}_e$, $\{F^n\}_e$ means the elastic matrix, incremental displacement vector and new node force vector of element ; N is the total number of Gaussian point.

$$\begin{cases} \{\Delta\varepsilon\}_i = [B]_i\{\Delta u\}_e \\ \{\Delta\sigma\}_i = [D]\{\Delta\varepsilon\}_i \\ \{\sigma^n\}_i = \{\sigma^o\}_i + \{\Delta\sigma\}_i \\ \{F^n\}_e = \sum_{i=1}^{N}[B]_i^T\{\sigma^n\}_i w_i J_i \end{cases} \tag{1}$$

To obtain the equivalent shear plastic strain and maximum tensile plastic strain, softening Mohr-Coulomb model and maximum tensile model is adopted (Eq. 2). Where σ_1 and σ_3 are minimum and maximum principle stress; C, ϕ, T present cohesion; inner friction angle and tensile strength; N_ϕ, α^p and σ^p are constant. From Eq. 2, if $f^s \le 0$ and $h \le 0$, shear failure will happen, and if $f^t \ge 0$ and $h > 0$, tensile failure will happen.

$$\begin{cases} f^s = \sigma_1 - \sigma_3 N_\varphi + 2C\sqrt{N_\varphi} \\ f^t = \sigma_3 - \min(T, C/\tan\varphi) \\ h = f^t + \alpha^P(\sigma_1 - \sigma^P) \end{cases} \tag{2}$$

With the increase of equivalent shear plastic strain and maximum tensile plastic strain, the cohesion and tensile strength of each element will be reduced linearly until to 0 (Eq. 3), where C_0 and T_0 are initial cohesion and tensile strength, ε_{sp} and ε_{tp} are equivalent shear plastic strain and maximum tensile plastic strain at current state, $\bar{\varepsilon}_{sp}$ and $\bar{\varepsilon}_{tp}$ mean limit value of thus two plastic strains.

$$\begin{cases} C = C_0(1 - \varepsilon_{sp}/\bar{\varepsilon}_{sp}) \\ T = T_0(1 - \varepsilon_{tp}/\bar{\varepsilon}_{tp}) \end{cases} \tag{3}$$

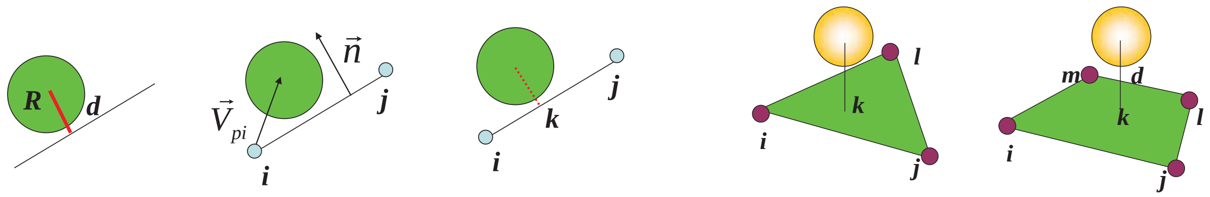
When any plastic strain mentioned above reaches the limit value, which means the cohesion or tensile strength reaches 0, the element will be deleted, and the crack appears.

CONTACT DETECTION ALGORITHM

For preliminary contact detection, static cell is used to improve the efficiency, and then particles and blocks are located in certain cells. For traditional cell approach, the main loop should be cells, so it’s very expensive for large simulations where the spatial distribution of objects is sparse and irregular (large number of empty cells). In this paper, the main loop is particle, so it will avoid looping the cells without particle or block in.

For accurate contact searching, two contact types should be considered, which are particle-particle contact and particle-block contact. Due to it’s easy to check the contact state for particle-particle, the emphasis of this paper will focus on the particle-block contact searching. To search particle-block contact and calculate contact force, a point-edge contact model in 2D and point-face contact model in 3D are introduced in, and incremental method with Mohr-Coulomb model and maximum tensile model is adopted to calculate the contact force.

For point-edge contact model in 2D (Fig 2 (a)), if the distance ($d = \left|\vec{V}_{pi} \bullet \vec{n}\right|$) between particle and one edge of block is smaller than radius of particle ($d < R$) and the projection point of particle lies in the edge ($d_{ik} \le d_{ij}$, $d_{jk} \le d_{ij}$), the point-edge contact is created immediately (one normal spring and one tangential spring), Where $\vec{V}_{pi}$ is the position vector, $\vec{n}$ is the outer normal vector of one edge. The weighted coefficient of point i and j could be calculated by the equation $w_i = d_{jk} / d_{ij}$ and $w_j = d_{ik} / d_{ij}$.



(a) Point-edge contact model in 2D (b) Point-face contact model in 3D

Fig 2. Two contact model between particle and block

Similar to point-edge contact model, for point-face contact model in 3D (Fig 2 (b)), the local coordinate system of the face should be obtained and the distance between particle and face should be calculated. If the distance is smaller than radius of particle and the projection point of particle lies in the face (for rectangle, $\left|(A_{ijk} + A_{jlk} + A_{lmk} + A_{mik} - A_{ijlm}) / A_{ijlm}\right| < tol$, where A means area), the point-face contact will be created (one normal spring and two tangential springs).

CASES STUDY

Particle Block Interaction. For testing particle block contact algorithm, the particle impact model is designed. The size of U-shape groove is 1m × 0.3m × 0.05m, which is filled by 461 triangle FEM elements.181 particles are used in this model, and the radius is from 1cm to 1.5cm with homogeneous distribution. Linear elastic model is adopted for the groove, with elastic module 10Mpa, Poisson ratio 0.3, and density 2000 kg/m^3. Rotation is allowed for particles, with the density 2000 kg/m^3. Mohr-Coulomb model and maximum tensile model are adopted for each contact, with normal stiffness 2 GPa/m, tangential stiffness 0.83Gpa/m, friction angle 26 degree, cohesion and tensile strength 0 Mpa. There is no gravity load on groove, with the top totally fixed. The particles move downward under gravity and then impact FEM elements, which will lead deformation of groove. Fig 3 shows the movement of the particles and the deformation of groove, which shows correctness of contact algorithm.

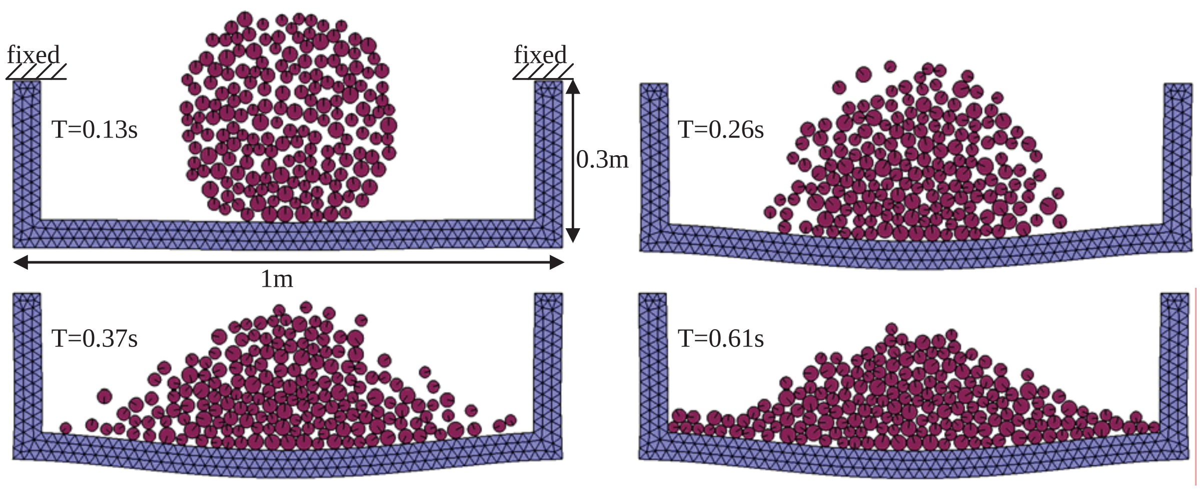
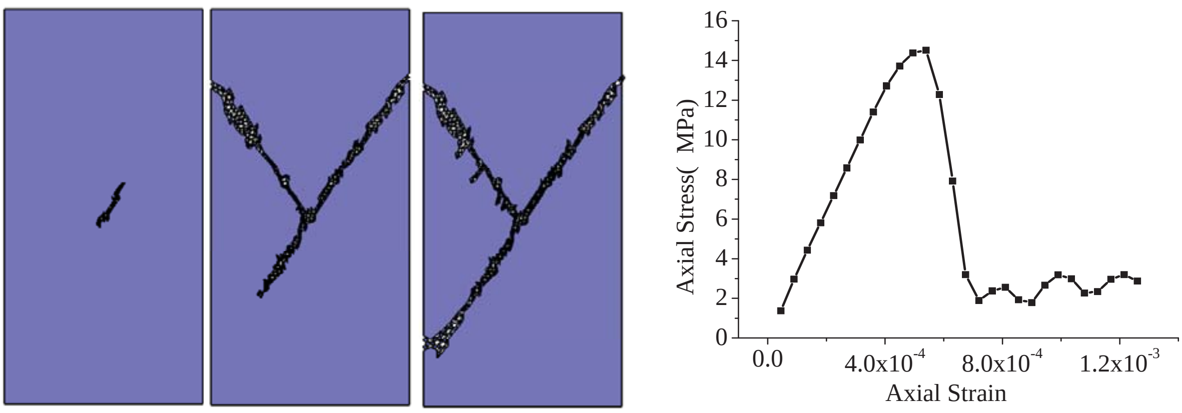


Fig 3. Interaction between FEM elements and particles

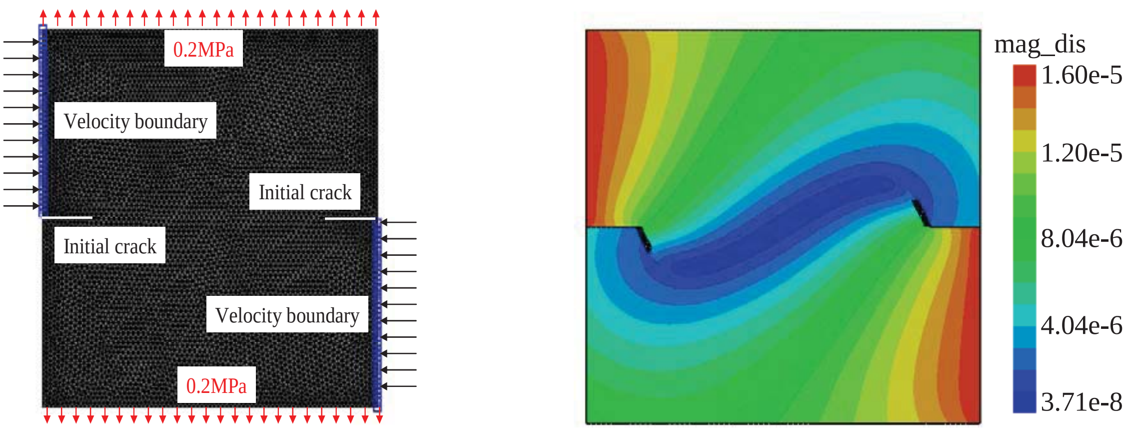
Rock Uniaxial Compression. The size of rock sample is 0.1m × 0.2m, which is formed by 7580 triangle FEM elements. Softening Mohr-Coulomb model and maximum tensile model is used, with elastic modulus 30GPa, Poisson ratio 0.25, initial cohesion 3Mpa, initial tensile strength 1Mpa, friction angle 40 degree, and dilation angle 10 degree. Limit value of equivalent shear plastic strain and maximum tensile plastic strain are 1% and 0.1% respectively. Numerical results in Fig 4 shows that, Y-type shear failure (Fig 4(a)) could be clearly observed and softening characteristics (Fig 4(b)) could be simulated well based on FEM/DEM coupled and evolved model.



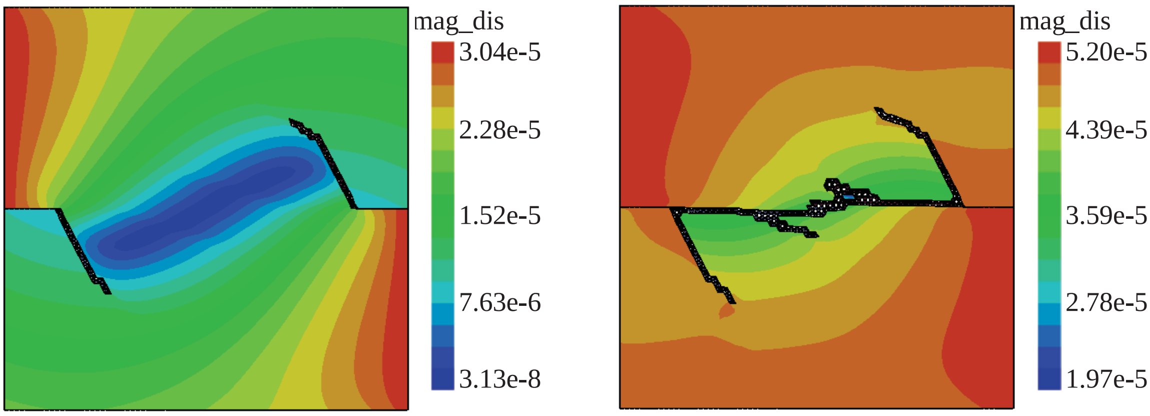
(a) Crack propagation under axial load (b) Stress VS strain under axial load

Fig 4. Numerical results on uniaxial compression

Rock Tensile & Shear Test. The size of the model is 0.2m × 0.2m, with 11100 triangle elements (Fig 5(a)). Material parameters are the same as the rock in uniaxial compression test. Axial tensile stress is applied on top and bottom boundary, with the value 0.2 Mpa. Initial cracks are set at middle layer, with normal direction vertical and length 2.5cm (both in left and right side). Quasi-static velocity boundaries are set at left (above middle layer) and right (below middle layer) boundary. With the increase of load step, new crack appears at initial crack tip first, then extends along dip direction and approaches to horizontal. Finally the horizontal crack on middle layer is completely through, and then rock slippage occurs. Fig 5(b)-5(d) (magnitude displacement contour) show the failure process of the test, the failure mode of numerical simulation agrees with the physical test well.



(a) Numerical model (b) Failure process 1



(c) Failure process 2 (d) Failure process 3

Fig 5. Numerical model and failure process of tensile & shear test

CONCLUSION

The FEM/DEM coupled and evolved model combines the advantage of FEM and DEM together. The deformation and plastic flow are simulated by FEM, friction and movement after failure is simulated by DEM. According to the deletion of FEM element and creation of DEM particle, the initiation and propagation of crack in geological body could be simulated. Some simple numerical cases are demonstrated to show the validity and accuracy of the model.

However, the model requires further more study, such as when the FEM changes to DEM, how to define the contact forces of the new created particles, and how to generate reasonable particle clusters in one FEM element.

ACKNOWLEDGMENTS

a. National Basic Research Program of the Ministry of Science and Technology of China under grant No 2010CB731506.

b. National Key Technology Research and Development Program of the Ministry of Science and Technology of China under grant No 2012BAK10B01.

REFERENCES

1.W.C. Zhu, C.A. Tang. 'Numerical simulation of Brazilian disk rock failure under static and dynamic loading', *International Journal of Rock Mechanics & Mining Sciences*, **43** , No.2, 236-252, 2006.

2.Xiao Q Z, Karihaloo B L. 'Improving the accuracy of XFEM crack tip fields using higher order quadrature and statically admissible stress recovery', *International Journal for Numerical Methods in Engineering*, **66**, No.9, 1378-1410, 2005.

3.Cottrell M G, Yu J, Wei Z J, et al. 'The numerical modelling of ceramics subject to impact using adaptive discrete element techniques', *Engineering Computations*, **20**, No.1, 82-106, 2003.

4.D M Yang, Y Sheng, J Q Ye, et al. 'Dynamic simulation of crack initiation and propagation in cross-ply laminates by DEM', *Composites Science and Technology*, **71**, No.11, 1410-1418, 2011.