V - 415

SIMULATION OF THE INTERNAL FAILURE MECHANISM OF CONCRETE UNDER COMPRESSION

Yokohama National University, Student Member, Mostafa A. M. ABDEEN
Yokohama National University, Member, Tatsuya TSUBAKI

1. INTRODUCTION

Concrete is a complex and heterogeneous material and it is difficult to analyze its failure properties by the finite element method (FEM) in which concrete is considered a homogeneous, continuous medium in general. The Distinct Element Method (DEM) was introduced by Cundall to analyze the granular assembly numerically assuming that each individual element satisfies the equation of motion and the law of action and reaction. The first model used two-dimensional polygonal elements and the second model used circular elements to reduce the complexity of the model and computational time. Later, the interface element method (IEM) was introduced by Zubelewicz and Bažant. This method modifies the DEM by considering the brittle aggregate composites as a system of perfectly rigid particles separated by interface layers whose normal stiffness deteriorates due to loading. In this research a numerical method is investigated to simulate the internal failure mechanism of concrete under compression using two-dimensional disc elements with statistical variation of local contact strength between the elements. Also the crack and deformation patterns at the global failure are presented.

2. FORMULATION OF TWO-DIMENSIONAL DISC ELEMENTS

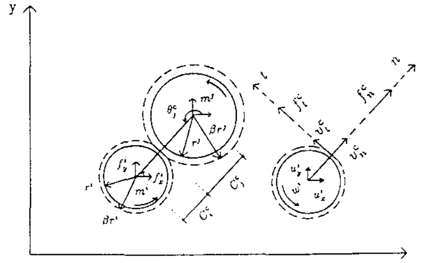

Fig.1 Two-Disc Elements with Their Interface Zones

Fig. 1 shows two general disc elements in contact with each other. It is assumed that each element is imagined to be surrounded by an annular influence zone indicated by dashed lines. In the figure, x,y are the global axes, n,t are the local axes defined at the center of the contact zone.

The relative normal and tangential displacements of element j with respect to element i at the center of the contact zone (v_n^c,v_t^c) can be obtained as follows.

$$V^c = -T^{jc}\,U^j - T^{ic}\,U^i \quad ; \quad V^c = [v_n^c, v_t^c]^T \quad ; \quad U^i = [u_x^i, u_y^i, w^i]^T \quad ;$$

$$T^{ic} = \begin{bmatrix} cos\theta^c & sin\theta^c & 0 \\ -sin\theta^c & cos\theta^c & r^i \end{bmatrix} \tag{1}$$

where V^c is the vector of relative displacement at the center of contact zone, T^{ic} is the transformation matrix needed to transform the global quantities of element i to the local quantities at the center of the contact zone, U^i is the global displacement vector for the centroid of element i, $\theta_i^c = \theta_j^c - \pi$ and u_x^i, u_y^i, w^i are the global displacements and rotation of the centroid of element i. Superscript c indicates the quantities related to the center of the contact zone.

The local contact forces at the contact zone (f_n^c,f_t^c) can be obtained by applying the force-displacement relation as follows.

$$F^c = K^c V^c \quad ; \quad F^c = [f_n^c, f_t^c]^T \quad ; \quad K^c = \begin{bmatrix} k_n^c & 0 \\ 0 & k_t^c \end{bmatrix} \tag{2}$$

where F^c is the local contact force vector at the center of the contact zone, K^c is the stiffness matrix of the contact zone, and k_n^c and k_t^c are the normal and tangential stiffnesses calculated from the stiffnesses of the interface zone.

Then, the force equilibrium equation for element i is expressed as follows.

$$F^i = \sum_{j=1}^{n} T^{icT} F^c \quad ; \quad F^i = [f_x^i, f_y^i, m^i]^T \tag{3}$$

where n is the number of elements which are in contact with element i, F^i is the vector of external global forces on element i. Eq. 3 will be made for every element in sequence. Then the number of final equations will be three times the number of elements. These equations will be solved directly to get the global displacement vector for the elements of the structure.

The reactions of the elements can be obtained by the following equation.

$$R = KU \tag{4}$$

where R is the global reaction vector for all the elements in sequence, K is the global stiffness matrix for the structure, and U is the global displacement vector for the elements in sequence.

3.NUMERICAL SIMULATION AND DISCUSSION

Fig. 2 shows the simulation result of concrete under uniaxial compression taking into account statistical variation of the local contact strength using the normal distribution. In the analysis the elements in the top and bottom rows are not constrained horizontally. It can be noticed from Fig. 2(d) that the stress-strain relationship for different coefficient of variation (w) is almost the same up to the peack but after that by increasing the coefficient of variation the curve becomes softer. Fig. 2(e) shows the stress-strain relationship for 10 different sets of random values for the local strength with w=20%.

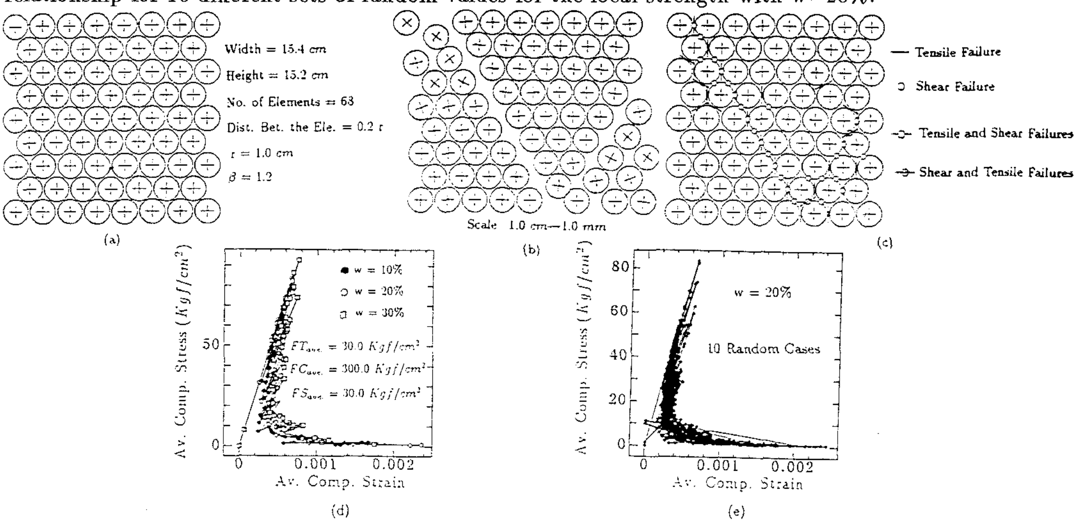

Fig. 2 Simulation of Concrete under Compression:
(a) Mesh Pattern ; (b) Final Deformation Pattern ; (c) Final Crack Pattern ;
(d) Stress-Strain Relationship for Different w ; (e) Stress-Strain Relationship for Const w

4. CONCLUSION

Through numerical simulations, it is confirmed that the disc element model is able to simulate the internal crack propagation and the stress-strain relationship of a material like concrete under compression up to the global failure.