V -91

Development of Three-Dimensional Interface Element Model For Simulation of Failure Behavior of Concrete

Yokohama National University Student Member George E. VULPE
Yokohama National University Member Tatsuya TSUBAKI

1. INTRODUCTION

A three-dimensional simulation model based on discontinuum mechanics theory is necessary in order to analyze concrete materials in general stress states more accurately. In this study the Interface Element Method is used for numerical simulation of the behavior of concrete. The advantage of this approach in the failure simulation of brittle materials is that the failure mechanism can be determined by following the movement of each particle and the internal mechanical state of the interfaces between particles.

2. THREE-DIMENSIONAL INTERFACE ELEMENT MODEL

For the three-dimensional analysis a new 3D Interface Element Model is developed. Concrete is modeled as an assembly of rigid spheres surrounded by spherical interaction zone. The spherical shape allows displacements and rotations between elements and permits a variety of meshes to be chosen. In the first stage a regular mesh is employed for simplicity. Interface elements are located between each two adjacent spheres that are interacting. An interface is divided into a number of layers and has a cylindrical shape with equal radius and length equal to the distance between the two spheres. The stresses are calculated for each interface layer. The interface is assumed to fail when a certain number of layers fail.

While the system is in the initial condition (uncracked state) it behaves elastically so that the global displacements **U** of each sphere are obtained by solving the following equation:

$$\mathbf{K}\,\mathbf{U} = \mathbf{F} \qquad (1)$$

where **K** is the assembled global stiffness matrix of the system and **F** is the global forces corresponding to displacements **U**.

Displacements and forces at the center of spherical particle *i* are represented by generalized displacement and force vectors:

$$\mathbf{U}^i = \{ u^i_x , u^i_y , u^i_z , \omega^i , \theta^i , \varphi^i \}^T \qquad (2)$$

$$\mathbf{F}^i = \{ f^i_x , f^i_y , f^i_z , m^i_x , m^i_y , m^i_z \}^T \qquad (3)$$

where u^i_x, u^i_y, u^i_z are displacement components, ω^i, θ^i, φ^i are the particle rotations, f^i_x , f^i_y , f^i_z are resultant forces at the center of particle *i*, and m^i_x , m^i_y , m^i_z are resultant moments. At the interface center *c* the relative normal and tangential displacements can be expressed in terms of displacements at the centers of adjacent particles *i* and *j*:

$$\mathbf{V}^c = -\mathbf{T}^{jc}\mathbf{U}^j - \mathbf{T}^{ic}\mathbf{U}^i \qquad (4)$$

$$\mathbf{V}^c = \{ v^c_n , v^c_{tx} , v^c_{ty} , \omega^c_n , \omega^c_{tx} , \omega^c_{ty} \}^T \qquad (5)$$

where T^{jc} is a 6x6 transformation matrix between particle *i* and contact point *c* while the normal and tangential force components and moments at the interface center *c* are:

$$\mathbf{F}^c = \{ f^c_n , f^c_{tx} , f^c_{ty}, m^c_n , m^c_{tx} , m^c_{ty} \}^T \qquad (6)$$

The interface center *c* is characterized by the force-displacement relation:

$$\mathbf{F}^c = \mathbf{k}^c\,\mathbf{V}^c \qquad (7)$$

where $\mathbf{k}^c$ is the stiffness matrix for the interface layer and $\mathbf{k}^c_f$, $\mathbf{k}^c_m$ are the diagonal 3x3 stiffness matrix of translation and rotation respectively.

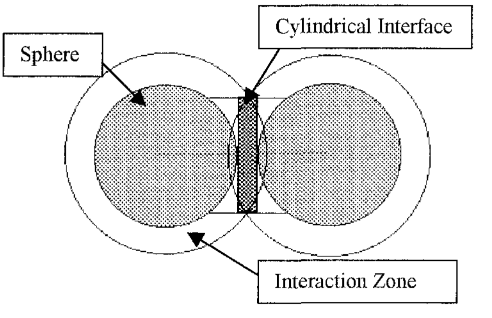


Fig.1 Interface Element Model

$$\mathbf{k}^c = \begin{bmatrix} k^c_f & 0 \\ 0 & k^c_m \end{bmatrix} \qquad k^c_f = \begin{bmatrix} k^c_{fx} & 0 & 0 \\ 0 & k^c_{fy} & 0 \\ 0 & 0 & k^c_{fz} \end{bmatrix} \qquad k^c_m = \begin{bmatrix} k^c_{mx} & 0 & 0 \\ 0 & k^c_{my} & 0 \\ 0 & 0 & k^c_{mz} \end{bmatrix} \qquad (8)$$

The brittle material stress-strain relationship is considered linear elastic up to the peak. The maximum stress failure criterion is used and it is examined independently in normal and tangential directions. Failure is assumed to be brittle in tension but ductile in compression and shear. Gradual degradation model is assumed in shear and compression by reducing the stiffness each time the failure condition is satisfied at one interface layer. The analysis procedure is based on the secant analysis method in order to obtain computational stability. It consists in imposing unit loads and determining from the equilibrium equations the stresses for all elements. Considering the ratio between stresses and strengths at each interface, the value of force that determines failure in only one interface layer is obtained. After the reduction of corresponding stiffness, the computational procedure is continued until the maximum allowable displacement is achieved.

Key words: Simulation, 3D Model, Concrete, Failure.
79-5 Tokiwadai, Hodogaya-ku, Yokohama 240-8501

3. NUMERICAL SIMULATIONS

Results of the numerical simulation are compared with corresponding uniaxial experimental test results by Kupfer on compression and tension (see Figs. 3-7). Different regular meshes are used to study the influence of mesh in the crack propagation. The specimen used in Kupfer's experiment is a rectangular plate (20x20x5cm) made by normal strength concrete (f_c=31.5 N/mm^2 and E=32500 N/mm^2). The homogeneous 3D mesh consists of 122 elements and the elements are arranged as in Fig. 2, making angles of 45^0 with surrounding elements situated in the same plane. Horizontal, vertical and diagonal interfaces connect spheres that are at a distance between centers smaller than 5cm.

In compression, after the initial linear elastic behavior, the stress-strain diagram (Fig. 5) shows a descending branch modeling the softening. The change of volumetric strain is shown in Fig. 6. The crack pattern is shown in Fig. 3 with tensile vertical and diagonal cracks propagating through the specimen, while at the base of the specimen, failure occurs due to shear and compression. In order to model this behavior, the compressive strength is reduced as a function of the lateral tensile strain. The width of the line showing the tensile crack is proportional to the crack opening.

Figure 7 shows the stress-strain diagram for uniaxial tension. The behavior is brittle and a softening tail follows. The crack propagates in a horizontal plane through the specimen due to tensile failure of interfaces (Fig. 4) in accordance with results from the experimental tests.

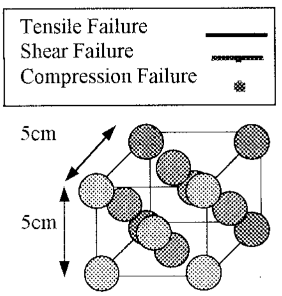
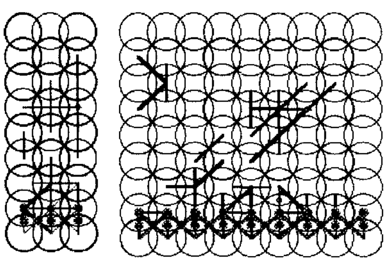
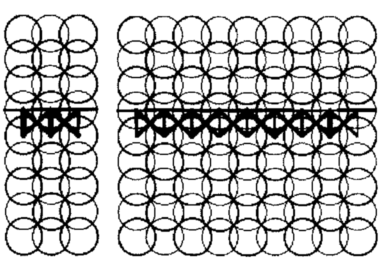


Fig. 2 Elements of the Mesh



side view front view

Fig. 3 Crack Pattern for Uniaxial Compression



side view front view

Fig. 4 Crack Pattern for Uniaxial Tension

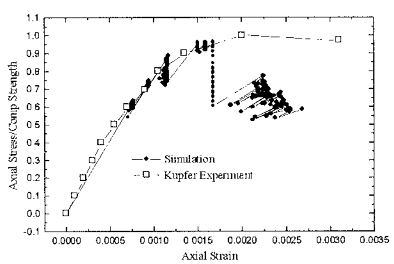


Fig. 5 Stress-strain Diagram for Uniaxial Compression

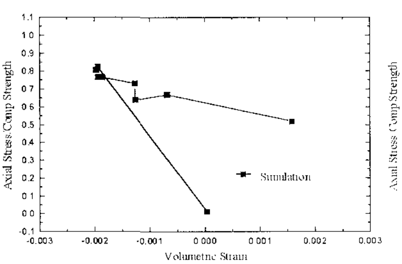


Fig. 6 Volumetric strain for Uniaxial Compression

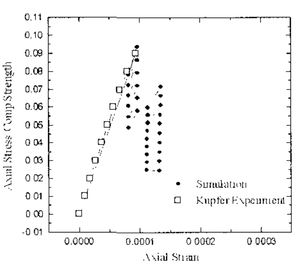


Fig. 7 Stress-strain Diagram for Uniaxial Tension

4. CONCLUSIONS

It is confirmed that the present model can simulate the behavior of the concrete material under uniaxial state of stress and that the deformation of the specimen and crack development can be followed up to the failure. The three-dimensional internal failure state is observed with the present model.

REFERENCES

1) Abdeen, M.A.M., Tsubaki, T., *Journal of Structural Eng.*, JSCE, Vol.42, pp.239-246, 1996.
2) Kupfer, H., Hilsdorf, H.K., Rusch, H., *ACI Journal*, Vol.66, No.8, pp.656-666, 1969.