FEM SIMULATION OF CONCRETE FRACTURE

V.Červenka[1], R.Pukl[1], R.Eligehausen[2]

Pull-out problem in plane stress state is analyzed by means of the finite element program SBETA. Fictitious crack approach is used to model tensile cracking. The softening modulus of concrete is adjusted according to the element size. A parameter study was conducted in connection with RILEM round-robin analysis of a pull-out test. The shape of the specimen as well as its size were varied. In total 24 analyses were performed. Each analysis represents simulation of an experiment and provide ample information on stress and strain state, crack development and failure mode. The results of the study show the following effects: span of supports, lateral constrain, size effect. For all analyses were used fixed and rotated crack models which provided different results in some cases.

INTRODUCTION

An anchoring bolt embeded in concrete mass represents a typical structural detail. The resisting pull-out force is a challenging design problem, which was typically solved experimentally. Recent advances in computational fracture mechanics make it possible to solve this problem numerically and to simulate the process of the failure of the anchoring zone. The authors developed a finite element program SBETA, which is determined for analysis of two-dimensional reinforced concrete structures and which includes also a non-linear fracture model. The program was used to perform a parameter study in connection with a round robin analysis organized by L.Elfgren on behalf of RILEM TC 90-FMA. Some results of this study are presented here.

PROGRAM SBETA

The computer program SBETA is based on the finite element method with two-dimensional quadrilateral 4-node finite element. The solution strategy for non-linear analysis is based on the arch length method, which enables the post-peak analysis. The crack development and failure modes can be stud-

[1]Czech Technical University Prague
[2]Stuttgart University

ied using the powerful graphical post-processor. The detail description of the constitutive model is given in report [1]. Here we shall describe only the fracture mechanics model, which is important for the following pull-out analysis.

Fictitious crack approach is used to model tensile cracking. Cracks are initiated when the principal tensile stress exceeds the tensile strength. Linear descending form of the stress-strain law in tension is used to model crack opening. The softening modulus E_t is adjusted to take into account the size effect using one of the following relations. If fracture energy G_f is known, the Bazant's [3] formula $E_t = -R_t^2 h/2G_f$ is used . Alternatively Vos [4] formula is used $E_t = -R_t h/w_0$, where w_0 is the crack width after complete stress release. R_t is the tensile strength, h is the crack band width and is approximately related to element size by $h = \sqrt{A}$, where A is the element area. Either rotated, or fixed crack model can be used. Another case of computer simulation of anchoring technique using program SBETA was treated in paper [2].

PULL-OUT TEST SIMULATION

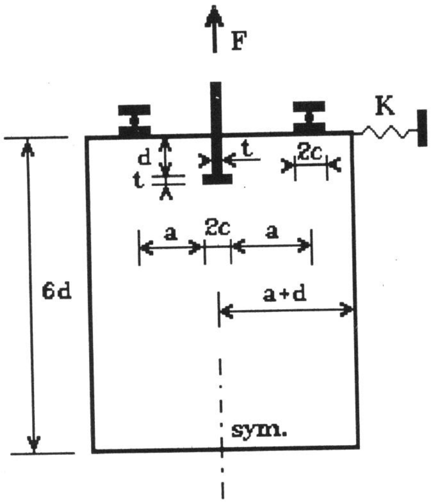
The geometry of the test specimen and material properties are shown in Fig.1. The specimen is in the plane stress state. In the parameter study the shape of the specimen (ratio a/d), as well as its size (embeded length of the bolt d) were varied. In the finite element analysis only symetrical half of the specimen was modelled. Three meshes for the specimen with shape ratio $a/d = 1$ and $d = 50$, 150, 450 mm are shown in Fig.2. The numerical simulation of the crack development and the strain localization for the case $a = d = 150$ mm are illustrated on Fig.3. The positions of the load stages are indicated by diamond marks on the load-displacement diagram on Fig.2.

The effect of the specimen shape on the load-displacement diagrams can be seen on Fig.4 ($K = 0$) and on Fig.5 ($K = \infty$). The lateral constrain significantly increase the maximum pull-out force. In cases without constrain the fixed crack model gave consistently higher maximum force than rotated crack model, while in cases with lateral constrain the maximum load was unefected for small spans and lower for high span $a = 2d$. The size effect is illustrated on Fig.6 for the shape $a = d$. The geometrical size factor is 3, but the load increase is in the range from 1.5 to 2.3.

The theoretical analysis should be supplemented by experiments in order to check its validity. This shall be done in the next phase of this research.

CONCLUSION

The nonlinear finite element analysis with program SBETA, which is based on smeared fictitious crack constitutive model, is capable to simulate the fracture process in the pull-out test. The numerical results should be supplemented by experimental investigation.



Geometry:
$d = 50, 150, 450$ mm
$a = d/2,\ d,\ 2d$
$2c = 3d/10$, $t = d/10$
thickness = 1 m

Lateral Constrain: $K = 0, \infty$

Material Properties:
$f_t = 3$ MPa, $f_c = 40$ MPa,
$E = 30$ GPa, $\nu = 0.2$
$G_f = 100$ N/m

Fig.1 Geometry and material parameters of the pull-out specimens.

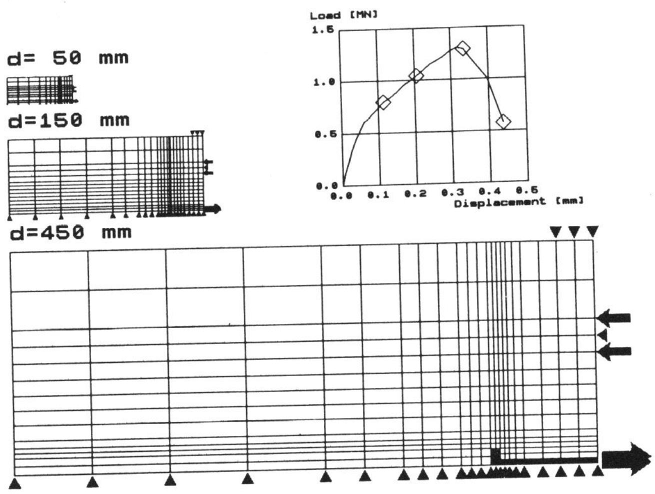


Fig.2 FEM meshes for the three sizes of specimens with shape $a/d = 1$. Load-displacement diagram for the specimen with $d = 150$ mm.

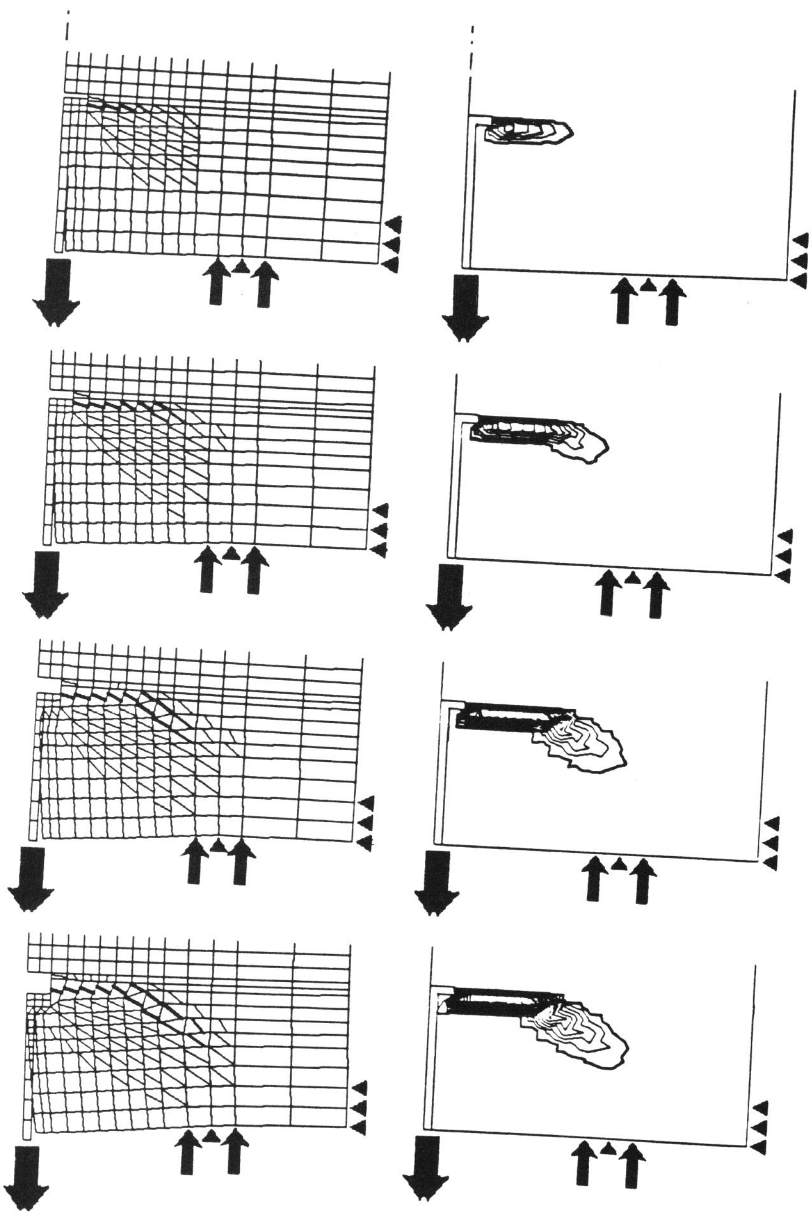


Fig.3 Simulation of crack development ($a = d = 150$ mm). Left: deformed shape with cracks; Right: isolines of maximum tensile strains (step=0.001).

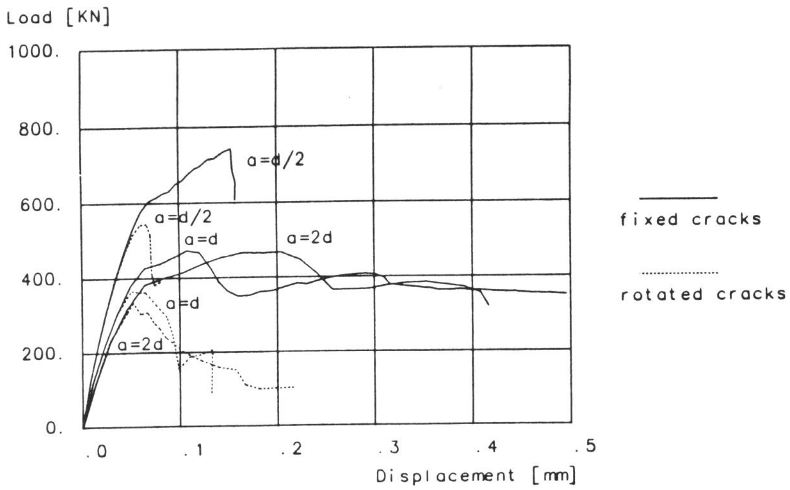


Fig.4 Load-displacement diagrams for specimens with embeded length d = 150 mm, without lateral constrain ($K = 0$).

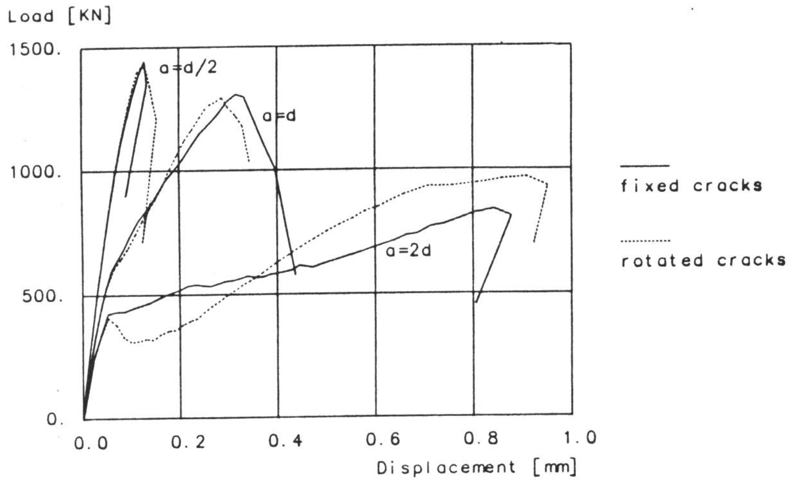


Fig.5 Load-displacement diagrams for specimens with embeded length d = 150 mm, with lateral constrain ($K = \infty$).

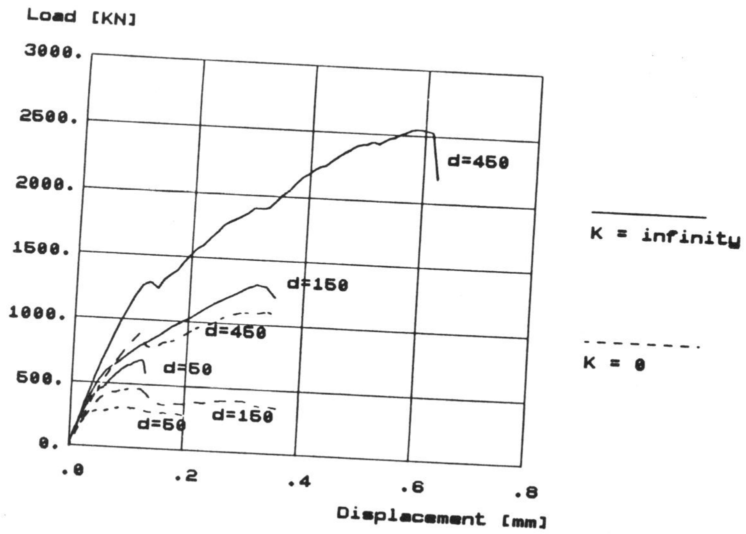


Fig.6 Load-displacement diagrams for different sizes of geometrically simillar specimens ($a/d = 1$). Fixed crack model.

References

[1] ČERVENKA, V., ELIGEHAUSEN, R., PUKL, R. - SBETA Computer Program For Nonlinear Finite Element Analysis of Concrete Structures, Part 1, Program Description, Part 2, User's Manual, Part 3, Examples of Applications, Institut für Werkstoffe im Bauwesen, Universität Stuttgart, 1990.

[2] ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - Computer Simulation Of Anchoring Technique In Reinforced Concrete Beams, Proc. Sec. Int. Conf. Computer Aided Analysis And Design Of Concrete Structures, Zell am See, Austria, 4th-6th April, 1990, Ed. N.Bicanic, H.Mang, Pineridge Press.

[3] BAŽANT, Z.P. - Mechanics of Distributed Cracking, Appl.Mech. Rev., ASME, 39 No. 5, 1986, pp. 675-705.

[4] VOS, E. - Influence of Loading Rate and Radial Pressure on Bond in Reinforecd Concrete, Dissertation, Delft University of Technology, 1983.