# Remaining Structural Capacity of Power Plant Frame
Capacité portante résiduelle d'une fondation sous portiques de turbines
Resttragfåahigkeit eines Turbinen-Rahmenfundaments

Jana MARGOLDOVÁ
Dr. Eng.
Czech Techn. Univ.
Prague, Czech Rep.

Radomír PUKL
Dr. Eng.
Czech Techn. Univ.
Prague, Czech Rep.

Vladimír ĆERVENKA
Consult. Engineer
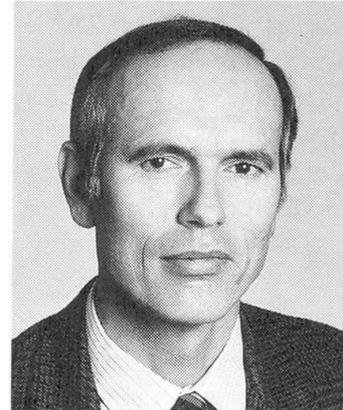Cervenka Consulting
Prague, Czech Rep.

J. Margoldová received her Pd.D. degree in 1987 from the Czech Technical University in Prague. She works as a research engineer at the Klokner Institute of the Czech Technical University.
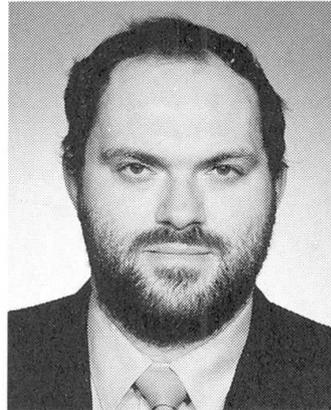
R. Pukl received his Ph.D. degree in 1985 from the Czech Technical University in Prague. He works in the field of computational mechanics at the Czech Technical University in Prague and at the Stuttgart University.

V. Cervenka received his Ph.D. degree in 1970 from the University of Colorado in Boulder. He is active in research and consulting of reinforced concrete structures and in software development.

SUMMARY

Remaining structural capacity of the turbine foundation frame in an old power station was assessed by means of the nonlinear finite element analysis performed by a computer programme. The effects of the quality of filler concrete of joints and of the boundary conditions were studied parametrically.

RÉSUMÉ

L'article traite de la détermination de la capacité portante résiduelle d'une fondation sous portiques, prévue pour des turbines dans une ancienne centrale électrique, au moyen de calcul par éléments finis non linéaires. Les vérifications ont porté sur l'influence de la qualité du béton des noeuds des portiques et des conditions aux limites.

ZUSAMMENFASSUNG

Die Resttragfähigkeit von Rahmenfundamenten für Turbinen in einem alten Kraftwerk ist mittels einer nichtlinearen Finite-Element-Analyse bestimmt worden. Die Einflüsse der Betonqualität in Rahmenknoten und der Randbedingungen sind untersucht worden.

INTRODUCTION

Heavy prefabricated reinforced concrete frames were typically used for construction of large thermal electric power plants in Czechoslovakia in the 50s and 60s within an extensive energy-production plan. These power plants burn low-quality brown coal and are the main source of energy in Czechoslovakia. In the course of time number of problems have emerged in connection with the servise of these plants. To mention only the most serious ones: uncontrolled environmental polution and structural damages due to heavy service loading. The most exposed structures are the reinforced concrete foundation frames of turbines. They are subjected to large static and dynamic loadings, chemical and thermal effects. Today in many cases these structures are also at the end of their designed life time. This life time is about half of that of similar unexpposed structures. It is in the interest of the electric power industry to extend the life service of these structures and thus to avoid building new plants. This tendency is also evident world-wide. In this context the technical diagnostic is becoming the important engineering branche. It is expoliting the reliability theory, the structural modeling and on site investigations. The last two mentioned categories were used in the present report for the diagnostic of the remaining structural capacity.

The concerned power plant is located in the North-West Bohemia and has been under reconstruction. It had been subjected to the long term monitoring to determine the extent of wearing after twenty years of service. In order to asses the remaining structural capacity of the turbine foundation the structure was analyzed by the finite element program SBETA. The damage and failure states of the structure were simulated. The goal of this analysis was to simulate the effects of the poorly manufactured joints of the precast members on the load carying capacity. The results served to design the measures for the necessary reconstruction.

2. PROGRAM SBETA

Program SBETA was recently developed at the Institute of Material Science of the University of Stuttgart in cooperation with the Klokner Institute of the Czech Technical University in Prague. It is a commercial program designed for the analysis of reinforced concrete structures in the plane stress state. It can predict the response of complex concrete structures, with or without reinforcement, in all stages of loading, including failure and post-failure. It can be used to analyze the remaining structural capacity of existing structures. Details about the program and its constitutive model can be found in papers [5,6] and documentation [8]. Here only a brief description is given. The other applications of the program are reported in ref. [2,3,4,7]

The constitutive model in SBETA is based on the smeared material approach with isotropic damage model in uncracked concrete and orthotropic damage after cracking. The behavior of concrete is described by the stress-strain diagram, which is composed of the four branches: non-linear loading in compression, linear loading in tension, and linear softening in both, tension and compression. The parameters of this diagram are adjusted according to the plane stress state using the biaxial failure function of Kupfer for compression. The mechanics of cracked reinforced concrete, which is relevant to this study case, includes: (a) reduction of compressive strength in direction parallel with cracks; (b) variable shear retention factor; (c) tension stiffening. All these properties are controlled by the tensile strain, which reflects the crack opening. The nonlinear fracture mechanics is introduced by means of the Bažant's crack band theory [1]. The tension softening modulus is adjusted for each element according to the fracture enegry. Both, fixed and rotated crack models are implemented. Reinforcement behavior is bi-linear.

A four-node quadrilateral finite element is used for the concrete. The reinforcement can be included either in a smeared form, as a part of the concrete element, or discrete, as a bar element passing through the quadrilateral element. The updated Lagrangean formulation is adopted allowing the modeling of a second order geometry effect. The non-linear solution is performed by means of a step-wise loading and by an equilibrium iteration within a load step. Newton-Raphson and arc-length methods are the options for the solution strategy in the equilibrium iteration.

The program system SBETA includes a pre-processor, a solution program, and an efficient post-processor. The finite element analysis can be interactively controlled and runs in several levels of real-time graphics. Thus, the solution process can be observed and solution parameters can be adjusted by a user if necessary. A restart option is available. The post-processor generates automatically deformed shapes and images of stress, strain and damage fields (cracking, crushing). All results of the analysis presented in this paper are produced by the SBETA post-processors.

3. ANALYSIS OF DAMAGED POWER PLANT FRAME

3.1 Girder Joint in Precast Foundation Frame

The schematic plan view of the frame is in Fig.1. The precast reinforced concrete girders A and B are supported by column. Each girder is formed by a couple of two identical girders, A_1, A_2 and B_1, B_2. The interaction of the coupling girders is assured by the slab, which is casted over the girders. The joint of girdes is located above the column. During the construction a space of 350 mm had been left between the vertical end faces of adjacent girders, the reinforcement was welded and the space was filled by concrete. The location of the joint is denoted in Fig.1. There were doubts about the quality and correct casting of the filler concrete and about full interaction of parallel longitudinal girders. The diagnosis of the frame was done by means of nondestructive testing methods combined with visual investigations supported by endoscop. However, a detailed investigation of this joint was impossible because of technological obstructions. The main purpose of the structural analysis was to make an estimate of the function of the joint under various assumptions of quality of the filler concrete.

3.2 Finite Element Model

In the linear analysis of the whole space frame, which was performed also for other purposes, the function of the structural detail of joint in the global structural system was studied. On the basis of this global analysis the region of damaged joint was identified. From the complex structure of the frame only a section adjacent to the joint was modeled. The surrounding structure was approximated by appropriate boundary conditions and artificial springs.

The analytical model, its geometry, boundary conditions and reinforcing are shown in Fig.2. The finite element mesh is shown in Fig.3. It has 387 quadrilateral elements. The vertical stirrups were modeled by smeared reinforcement and all main horizontal and inclined bars were modeled by discrete reinforcement. The concrete quality of girders identified from core-drilled samples was 28 MPA. The nominal quality of filler concrete in the joint was 34 MPA. However, there were doubts about the quality of its casting and its actual state could not be reliably verified. Therefore, variable properties of the filler concrete were considered in this study by

values of 1%, 10%, 60% and 100% of its nominal value.

Two alternatives of boundary conditions were considered for the modelling of the surrounding structure. In the first one the column under the joint of the longitudinal girders was modeled by springs and the continuity of the structure was modeled by conditions of the symetry in the middle of the girders. In the second case the support on the column was rigid and the continuity of the girders was modeled by the springs. This enabled the approximate simulation of the axial displacements of girders due to flexibility of the frame structure.

The loading is due to technological forces which are trasfered through numerous fastenings as indicated by vertical arrows in Fig.2. The force in the load-displacement diagram refers to the sum of the technological forces. Two loading cases were considered. In the first loading case the full interaction of girders A_1, A_2 and B_1, B_2 was assumed and the girders were loaded by the half of the total technological load. (The structure was designed under this assumption.) The second loading case assumed no interaction between the girders and the full technological loading was applied on the internal girders. In both cases the dead load of girders was included.

The study was performed on the personal computer 286 under MS-DOS operating system. Solution of one case on this computer took about 10 hours of computer time. (Of course in case of PC486 the time would be much shorter.) In addition to the ultimate load capacity each analysis provided ample of informations on stress and strain state, crack patterns and failure mode. The fixed crack approach has been used in all cases.

3.3 Discussion of Results

The behavior of analyzed system is illustrated in Fig.4., which shows the crack patterns in three load stages and the failure state with cracks, crushing and amplified deformations. The load displacement diagram with the load levels corresponding to the stages in Fig.4 is shown in Fig.5. The yielding of reinforcement is also graphically indicated (but apparent only from coloured output). Strong shear behavior is evident from inclined cracking. The failure mode was dependent on the degree of lateral constraint and the quality of the filler concrete in the joint. In case of a high lateral constraint and the 100% quality of the filler concrete the maximum load was 4.75 times of the admissible loading. The failure mode is of the concrete arch-type, with crushing of the cracked concrete in the web and in the bending compression zone.

The effect of the filler concrete quality on the frame behavior can be seen from the comparison of Figures 4(c) and 6. The load level is 2 times of the admissible load. The quality of the filler concrete described by the compressive strength was 100% in Fig.4(c), 60% in Fig.6(a) and 10% in Fig.6(b). In case of the lowest quality, Fig.6(b), the filler concrete in the joint fails in compression and after that the behavior is fairly ductile with all major reinforcement yielding. The ultimate load factor, (related to the admissible load) is in this case 3.5. The reduction of load capacity was also caused by partially releasing the lateral constraint. In the most unfavorable case, with the elastic springs modeling the lateral constraint and 1% of filler concrete quality the ultimate load was almost equal to the admissible load with no marginal safety.

The results of the numerical analysis were used to support the design of measures for extending the service life of the frame, which were based on the restoring the full interaction of girders and strengthening the space frame. They were also utilized in a reliability analysis. It was a valuable contribution to the safety and economy of the engineering solutuion.

4. CONCLUSIONS

The computer program SBETA was succesfully applied to the assesment of the remaining structural capacity of the turbine foundation frame. The nonlinear finite element analysis proved to be a rational method for determination of ultimate load capacity of this statically indetermined structure, whose behavior significantly deviates from the simple design models based on cross-sectional analysis. The FE analysis was used to design the economical and rational repair measures .

REFERENCES

[1]   BAŽANT, Z.P., OH, B.H. - Crack Band Theory for Fracture of Concrete. Mater. Struct. RILEM, Paris, France, 16, 1983, pp. 155-177

[2]   ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - Computer Simulation of Anchoring Technique in Reinforced Concrete Beams. Computer Aided Analysis and Design of Concrete Structures, Proceedings 2nd SCI-C International Conference, Eds. N. Bicanic, H. Mang, Zell am See, Austria, April 1990, Pineridge Press, Swabsea, U.K. 1990, ISBN 0-906674-74-3, pp. 1-20

[3]   ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - FEM Simulation of Concrete Fracture. ECF 8 Fracture Behavior and Design of Materials and Structures, Preprints of the 8th Biennial European Conference on Fracture, Torino, Italy, October 1990, pp. 728-733

[4]   ČERVENKA, V., ELIGEHAUSEN, R., PUKL, R. - SBETA Computer Program for Nonlinear Finite Element Analysis of Reinforced Concrete Structures. IWB Mitteilungen 1990/1, Stuttgart University, ISSN 0932-5921, ISBN 3-9801833-6-X

[5]   ČERVENKA, V., ELIGEHAUSEN, R., PUKL, R. - Computer Models Of Concrete Structures. IABSE Colloquium "Structural Concrete", Stuttgart, Germany, April 1991, ISBN 3-85748-063-7, pp. 311-320

[6]   ČERVENKA, V., PUKL, R. - Computer Models Of Concrete Structures. Structural Engineering Int., Vol. 2, No. 2, May 1992, IABSE Zurich, Switzerland, ISSN 1016-8664, pp. 103-107

[7]   ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - Fracture Analysis of Concrete Plane Stress Pull-Out Tests. Fracture Processes in Concrete, Rock and Ceramics, Noordwijk, Netherlands, June 1991, pp. 899-908

[8]   SBETA – Nonlinear Finite Element Analysis of Reinforced Concrete Structures, Software and Documentation, Peekel Instruments, Industrieweg 161, 3044 AS Rotterdam, The Netherland.
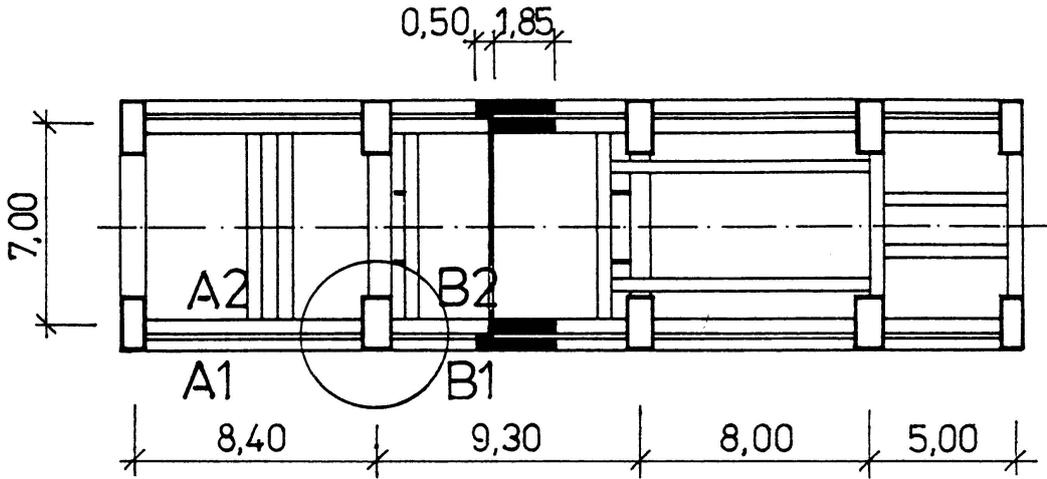
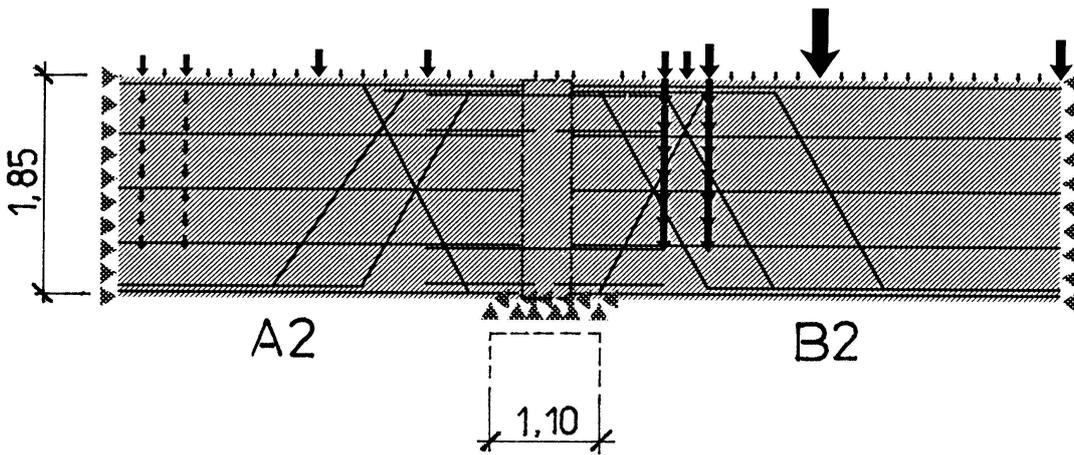Fig.1 Power station turbine frame: schematic plan view (dim. in meters).

Fig.2 Structural model of a detail with reinforcement, loading
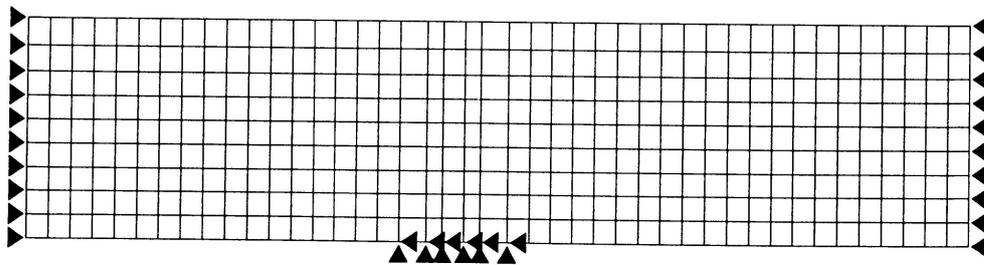and boundary conditions.

Fig.3 Finite element mesh of the structural model.

(a)

P=2.7 MN
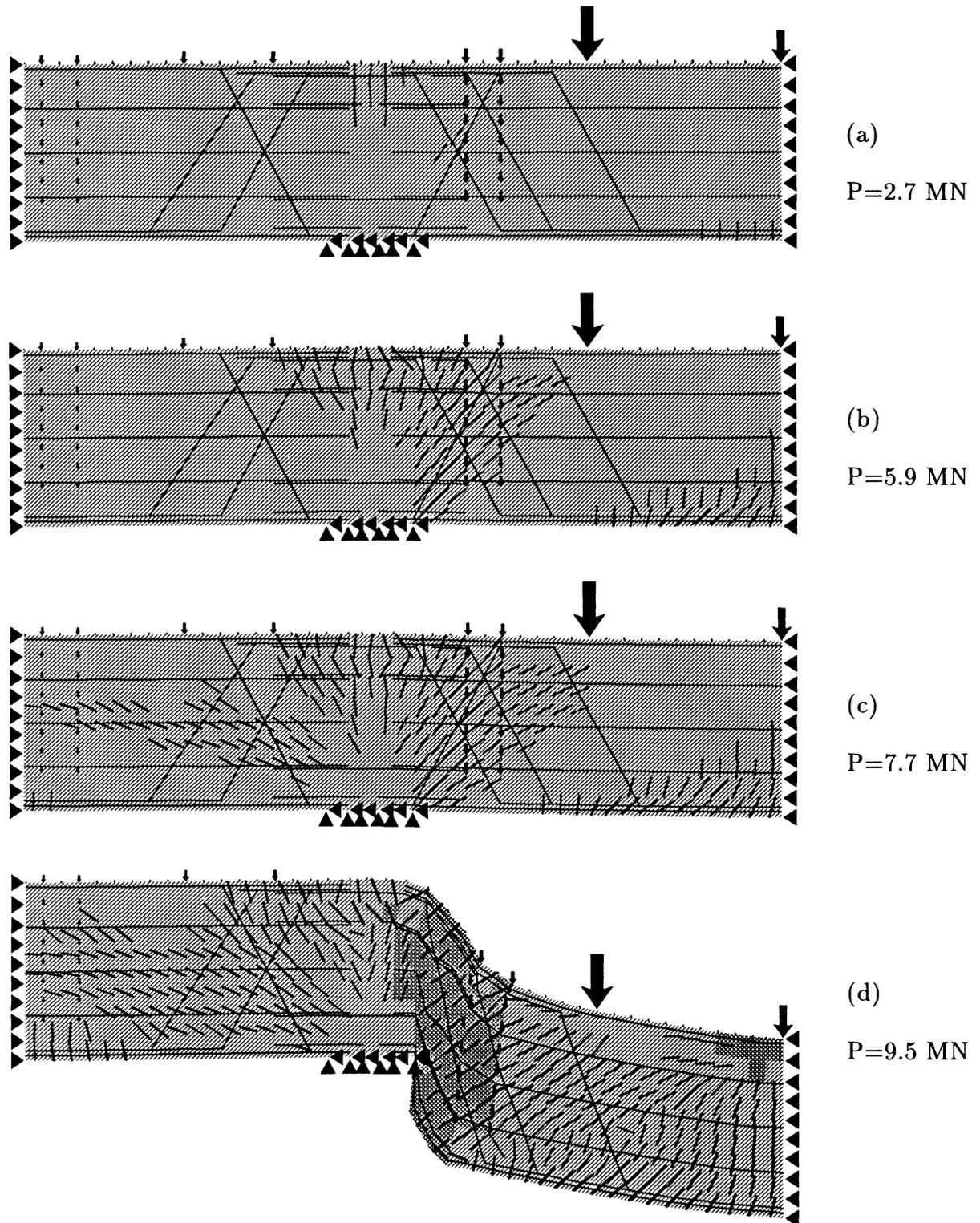
(b)

P=5.9 MN

(c)

P=7.7 MN
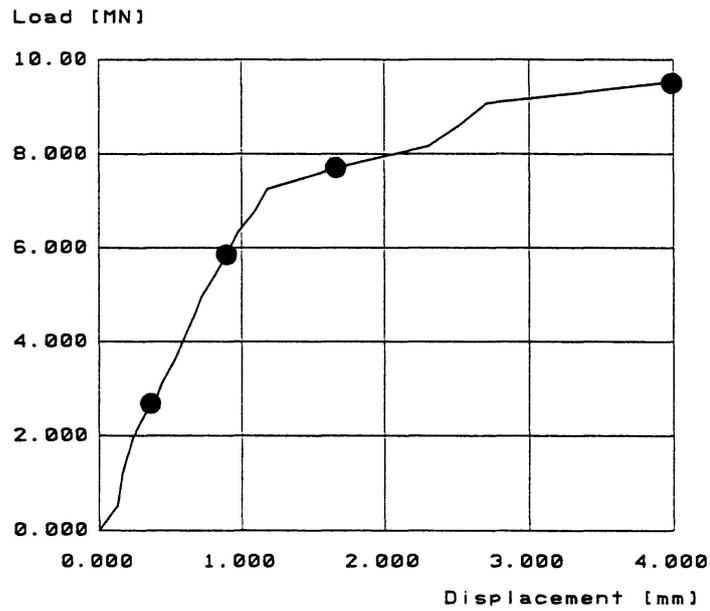
(d)

P=9.5 MN

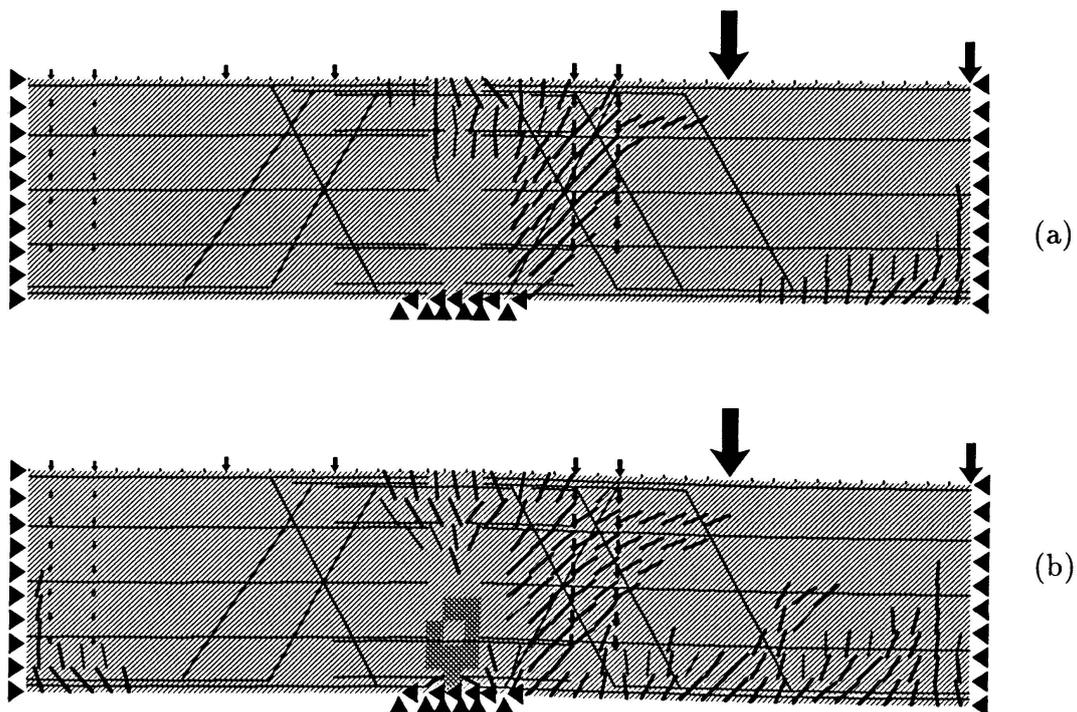Fig.4 Sequence of crack patterns.

Fig.5 Load-displacement diagram.

Fig.6 Comparison of damage states for cases of different joint concrete qualities.
Load factor = 2.