NON-LINEAR ANALYSIS UP TO RUPTURE OF A MODEL OF A MULTI-CAVITY PRESTRESSED CONCRETE PRESSURE VESSEL*

B. REBORA, F. UFFER, T. ZIMMERMANN

IPEN, Institut de Production d'Energie, Swiss Federal Institute of Technology in Lausanne,
17, Avenue de Dapples, CH-1006 Lausanne, Switzerland

SUMMARY

Within the frame of a German-Swiss agreement concerning the project of a high-temperature nuclear plant (HHT), the Swiss Federal Institute for Reactor Research (EIR, in Würenlingen) has developed an integrated variant of an helium-cooled high temperature reactor of 3 × 500 Mwe.

A test on a model (1 : 20) of this prestressed concrete nuclear vessel with multiple cavities has been carried out under the supervision of "Bonnard et Gardel ingénieurs-conseils SA (BG). The model was built by the "Centre d'Etude du Béton Armé et Précontraint (CEBAP)" and the "Institut de Production d'Energie" has accomplished the numerical analysis of the model.

The aim of this analysis is to determine the mechanism of ruin and ultimate load of the structure. In addition, comparison with the results of the test emphasizes the mathematical model with a view to its utilisation for the analysis of any prestressed concrete nuclear vessel.

The model built by the CEBAP consists of a concrete cylinder of 2.4 m in diameter and 1.95 m high. 28 main cavities simulate the central reactor, the turbines, the cooler and so on. The model is three-dimensionally prestressed by vertical, radial and annular steel wires. The cavities were put under increasing water pressure until rupture occured.

The mathematical model used has been developed at the "Institut de Production d'Energie". The calculation of the prestressed concrete nuclear vessel model has been made possible thanks to the most recent developments and results obtained in the domain of non-linear analysis of massive reinforced and prestressed concrete structures. The method is based on three-dimensional isoparametric finite elements. Liner elements (membrane elements) and steel wire elements with a yield criterion for stell have been introduced. A good knowledge of the test parameters and of the material laws makes it possible to approach the results of the test reasonably well. The calculated mecanism of ruin corresponds to the one of the test.

The principal interest of this paper is to show the accuracy of non-linear analysis of a complex massive structure with the test results and the evolution of the behaviour of the structure from the apparition of the first crack up to the ruin by rupture of the steel wires.

---

* The trial is presented at this congress (see paper H 4/2).

1. Introduction

The reactor project upon which our article is based has been effected by the following firms :

- Brown Boveri & Cie A.G. ;
- Hochtemperatur-Reaktorbau GmbH ; Kernforschungsanlage Jülich GmbH ; NUKEM GmbH ;
- EIR (Eidgenössisches Institut für Reaktorforschung), and other Swiss enterprises ;

in the area of a programme of development for a nuclear power plant with a high temperature reactor and high power helium turbines ; this programme is encouraged by West Germany, its state of Nordrhein-Westfalen, and Switzerland.

One of the important component of this power plant consists of a reactor vessel of prestressed concrete. The reactor vessel's project has been elaborated by Bonnard and Gardel Consulting Engineers Ltd, Lausanne, Switzerland. This is presented in the form of a concrete cylinder 48 metres in diameter and 39 metres in height, comprising 28 main internal cavities in which is put together the nuclear and thermic equipment connected with the main helium circuit. It concerns a very massive structure, with complex geometry submitted to regulated three dimensional forces, of which the behaviour cannot be seen properly by manual methods of analysis.

A trial on a scaled down model has been analysed by CEBAP (Centre d'Etude du Béton Armé et Précontraint). The prestressed model scale 1/20 had been submitted to an increasing interior pressure until rupture occured. Parallel with this trial was a simulation by calculation of the behaviour of the structure. The computer programme used for calculation was made under the supervision of Professor A. Gardel by the "Institut de Production d'Energie" with the asistance of the Swiss National Science Foundation [1,2,3].

The aims of the calculation that we present are :

- the analysis of the behaviour in a service state and the evolution of non-linear phenomenon such as the cracking and the non-linearities in compressed concrete, yielding of prestressed steels and of the liner.
- to determine the break up mechanism and the ultimate load of the structure.
- to adapt the mathematical model with a view to its use for an factual vessel, allowing for all the elements which constitute the reduced model. A comparison with the trial results will show the very good agreement between the calculations and the measurements.

2. Mathematical model

The basic numeric model uses elastic isoparametric elements developed at the university of Swansea by the research group of Professor O.C. Zienkiewicz [4,5]. The process of frontal elimination developed by B. Irons [6] is used for resolving the system of linear equations. The non-linear behaviour of the materials is taken into account by a cycle of iterations of the pseudo elastic model without modifying the stiffness matrix [1,2,3].

3. Trial upon a reduced model of the reactor vessel

3.1 The concrete

The reduced model, made of finegrain-concrete, consists of a solid cylindrical block, 2.4 metres in diameter and 1.95 in height (fig. 1). For a gross volume of about 8.8 cubic metres, the reduced model has 40 cavities with a total volume equal to 1.4 cubic metres distributed on 28 cavities as follows : the reactor core, 3 horizontal turbines, 24 vertical cavities simulating the cooling elements and heat recuperator, etc.

3.2 The prestressing

The reduced model is prestressed in 3 dimensions by : 28 annular cables distributed over the total height by pairs of consecutive cables, 36 horizontal cables at turbine level, 159 vertical cables fixed upon the plugs of the 24 vertical cavities. Non of the cables are injected with concrete. An inox liner assures a perfect seal of the cavities.

4. Discretisation

4.1 Elements used

The discretisation of the elements constituting the structure is done with the aid of various finite elements. The reinforced concrete is modeled by the isoparametric finite elements of 20 nodes. The cavity liner is represented by membrane elements of 8 nodes. The assembly of these elements with these of the concrete is made in a manner to satisfy the fundamental hypothesis of reinforced concrete, i. e. $\varepsilon_s = \varepsilon_c$ (ε = strain, s = steel, c = concrete). The basic functions used are described in refs. [4,5,7].

For the annular prestressing, the membrane elements of two dimensions are degenerated to give only a stiffness of the steel in the direction of the cables. These elements represent cables that are solidified in concrete (injected) which is not the case in the test. It is shown that this difference is of little consequence.

The prestressed vertical cables are simulated with the aid of elements of 20 nodes (60 D) of which the rigidity is unidirectional according to the direction of the cables. As preceding, these elements are assembled to the elements of concrete so that the cables are solid with the concrete. It is for reasons of computor memory locations that we have not been able to simulate the non-injection of these two types of prestressed cables. The prestressed radial cables, however,correctly simulate the non-injected ones by unidirectional bar elements.

4.2 Choice of the finite element mesh

The trial vessel has 6 radial planes of symmetry (fig. 1). For reasons of computer memory and to have good discretisation, we had to choose a geometry with 12 radial symmetry planes. In the case of the vertical cavities, this symmetry represents quite well reality, but we have a dummy turbine in addition to the real turbine. Conversely, two horizontal cavities in the trial minimise the inconvenience of the dummy turbine; the pressure in the horizontal cavities and the turbine is being always identical.

The mesh chosen represents in fact a slice 1/12 of the trial model (fig. 2). There are 111 concrete elements and 2481 degrees of freedom (3 per node). The choice of this mesh is the result of the comparison of four calculated meshes in a linear elastic state ; this mesh representing the best behaviour of all the structure.

4.3 Definition of loads and boundaries

The forces of initial prestress are introduced as forces of pressure upon the outside faces of the vessel,or as point forces in the case of radial prestress. In the case of vertical prestress,the pressure is uniform by element,taking into account the over all distribution of the prestressed cables in the trial . The sum of the forces of prestress is in all cases identical to this of the trial. The initial annular prestress is uniform over all the height and is interrupted at turbine level where the point forces simulate the initial radial prestress. The inside pressure of the elements is introduced as pressure upon the faces of the elements. In the case of the reduced model the pressure on the vertical cavities and the turbine is transmitted to the cables by the metal plugs. In the case of the calculations, the plugs don't exist, and it was necessary to replace the components of pressure directed towards the outside, by equivalent pressures upon the neighbouring exterior elements. This equivalent pressure has the same effect upon the mesh as the plugs in the trial.

The internal pressures are multiplied by a load factor that one varies by increments until rupture occurs.

The mesh is bound in a manner to satisfy the conditions of symmetry that we have chosen (fig. 1). A vertical support under the mesh determines the structure statically.

4.4 Constitutive laws

The constitutive laws introduced into the calculation take into account all types of nonlinearities of the concrete and of the steel. The behaviour of the materials is considered as being independent of time [1,7 ; p. 273-278].

An elasto-plastic relationship of stress-strain is introduced into the calculations for all the steels. The yielding of the membrane element is determined by the criteria of von Mises [8, 9].

5. Analysis

5.1 Computer

The computer used was a CDC CYBER 7326 at the Swiss Federal Institute of Technology in Lausanne. The mesh chosen had 2481 degrees of freedom with a front of elimination of 83 nodes (249 variables).

It is necessary to reserve in the central computer site $170\ 000_8$ words for the first resolution. The time necessary for this last operation by the CPU is 1400_{10} seconds. For the following resolutions,$77\ 000_8$ words are necessary and about 50_{10} seconds of CPU time.

5.2 Linear analysis

A linear analysis was done at first for the case of the prestressed load only,with a view to examining if the stresses and strains were admissible. The mean pressure representing the vertical prestress upon the upper and lower surfaces,including the material between the core and the outside of the vessel is 51 $^{kg}/_{cm^2}$. The mean pressure of the annular prestress is 61.8 $^{kg}/_{cm^2}$. Under these loads the vessel is submitted only to a state of compression. The greatest stresses in this state are in the order of - 250 $^{kg}/_{cm^2}$,and one finds it in the centre of the upper and lower plates, also between the vertical cavities. This analysis shows that the prestress does not give rise to excessive stresses.

The analysis of the calculated results for an internal pressure of 90 $^{kg}/_{cm^2}$, shows that no tension, nor any non-linearity in compression is apparent. For this load the structure has a purely linear-elastic behaviour. The prestress is efficient. One notices the greatest stresses in compression (- 220 $^{kg}/_{cm^2}$) around the axis of the vessel in the two plates.

Around the core one notices a decrease of the vertical compression due to the prestress. Finally, using two calculations at different internal pressures (60 and 90 $^{kg}/_{cm}2$), one may determine the probable appearance of the first crack at turbine level under the core, for an internal pressure of about 129 $^{kg}/_{cm}2$.

5.3 Non-linear analysis up to rupture

At an internal load of 150 $^{kg}/_{cm}2$ (fig. 3), the first cracks were detected and one sees that the state of stress has evolued much. The stresses of compression in the plates have increased slightly, in addition,their principal directions have changed. The pressure in the core becomes preponderant with respect to the annular prestress, the upper plate and to a lesser degree,the lower plate are bent. In the corners of the core, some cracks appear ; one can also see in the horizontal cross section No 5 (fig. 3) that at 150 $^{kg}/_{cm}2$, the zone of concrete situated at turbine level is cracked. Notice that the dummy turbines and the form of the finite elements weaken this zone in the mathematical model. The peripheral cavities have provoked a vertical cracking at a level equal to that of the core in the thinnest zone. This cracking extends from cavity to cavity. One may indeed observe that the zone of neighbouring concrete is not compressed any more, and that this cracking is going to propagate itself by a relatively weak increase of load.

The increasing of the pressure in the cavities brings about a field of triaxial stresses which propagates itself in a manner similar to lines of force around magnets. In our case these magnets are represented by the cavities and the exterior surface of the vessel. At this stage of calculation one doesn't yet observe the yielding of the steel which only participates to a small proportion to the over-all stiffness of the structure.

When the internal pressure attains 180 $^{kg}/_{cm}2$ (fig. 4),the cracking previously detected propagates itself. The bending of the plates is accentuated. The cracks in the corner of the core propagate more. The new cracks appear near the exterior vertical cavity,in the direction of the neighbouring cavity,which is by symmetry the most distant.

At turbine level the concrete is cracked over a larger zone. If we observe the same cracks on the vertical cross section No 6 (fig. 4),we notice that the cracking is vertical and becomes horizontal along the turbine toward the outside. A similar phenomenon may be noticed in this which concerns the detected cracks near the external vertical cavity, (vertical cross section 6 on the right, horizontal cross sections 1, 2, 3, 4 fig. 4). The cracking due to the corner phenomenon described above,is directed toward the outside, it becoming vertical along the peripheral cavity and directing itself again toward the outside before reaching the top of the vessel. It is equally necessary to note the appearance of radial vertical cracks which direct themselves from the peripheral cavities toward the outside.

At this stage one may notice that if the structure holds together well, there is the appearance of three local break up schemes (fig. 4 and 6) :

- the upper plate is made to bend under the internal pressure of the core and under the vertical prestress (fig. 4 and 6a). The crack at the corner of the core appears at the place of the maximum negative moment of bend. It becomes horizontal and climbs between the internal cavities. The centre of the plate (axis of the core) displaces itself toward the top where one sees appearing a strong positive moment of bend and already a crack (vertical cross section No 7 (fig. 4) top left). The state of stress which is seen here is typical of a circular built-in bending plate.

- A similar phenomenon, although more complex, is observable for the lower plate (fig. 6 b). It has the same cracks going from the corners of the core. The turbine affects the state of stress and diverts the cracks horizontally ; one can see that a horizontal separation is about to occur on the lower plate at turbine level.
- The part of the concrete delimited by the exterior vertical cavity, by the plan of symmetry at 30°, and the exterior face of the vessel (cross sections 1 to 4 fig. 4 and 6c), has a tendancy to come apart from the structure. The vertical cracking joins the peripheral vertical

At the load of 210 $^{kg}/_{cm}2$ (fig. 5) the schemes of break up appear well defined and one notices that the structure only holds together due to the presence of the steel. The predicted cracks at 180 $^{kg}/_{cm}2$ have occured. The upper plate is cracked in the zone of the positive moment of bending and the lower plate is thus totally separated horizontally from the vessel, this being at the turbine level. In the horizontal cross sections one may see that the cracking appears between all the vertical cavities, the direction of the cracks going from one cavity to another. One notices at this load that perpendicular cracks appear where the first cracks were detected. It is necessary to notice the appearance at this stage, of a horizontal crack on the outside of the vessel at core height (vertical cross section 6 and 7). This crack is part of the break up scheme described in fig. 6c.

5.4 Comparison with the trial (fig. 7, 8)

The trial consisted of 7 load cycles, of which the first five served to analyse the behaviour of the vessel in a state of service, and the last two to determine the ultimate load and the mechanism of break up.

One may verify that the results of the elastic calculation correspond well to the behaviour of the trial vessel; the comparison of the vertical and horizontal deformations for an internal pressure of 90 $^{kg}/_{cm}2$ shows that in the linear domain, the calculation of the structure is horizontally a little more rigid than the trial model ($\varepsilon_{trial} = 0.000133$, $\varepsilon_{calculation} = 0.000108$) ; vertically the mean of the rigidity is the same.

In order to show the agreement of the results in the non-linear domain one may compare the values of stress increase in the annular prestressed steel in both cases at 210 $^{kg}/_{cm}2$ of internal pressure. The integral of the stresses at core height is almost the same in the two cases.

In the trial case the maximum pressure reached at rupture was equal to 240 $^{kg}/_{cm}2$. The rupture appeared outside by big deformations and large open cracks, and the rupture of an annular cable and serious leaks due to a rupture of the liner of the two peripheral vertical cavities (ruptures caused by cracking and important movements of the concrete).

The cutting up of the model permits the showing of the break up schemes, the superposition of these schemes with those furnished by the calculation, demonstrates the excellent agreement of the results of these two methods of investigation. The trial has shown on the other hand that the determining mechanism for the definition of the ultimate load (240 $^{kg}/_{cm^2}$) was this defined in fig. 6 (fig. 6c). At this time we are following our calculations for loads superior to 210 $^{kg}/_{cm^2}$, with a view to finding, with our mathematical model, what is the determining mechanism. These calculations will be finished in the next few weeks, and the results ultimately published.

6. Conclusions

For a massive structure such as the reactor vessel HHT,with numerous vertical and horizontal cavities,it is difficult if not impossible,to predict what will be the break up scheme,and to determine the safety coefficient with respect to the internal pressure,with a classic analysis of rupture. The paper that we have presented here shows that the mathematical model used permits a non-linear three-dimensional analysis. The comparison of the results, and in particular the break up schemes,demonstrate the excellent agreement of the calculations compared with the trial. The undeniable advantage of calculations over a destructive test, is to permit the variation of parameters, the modification of geometry, the prestress, etc. It is also possible to take into account thermic gradients as well as the laws of time dependent behaviour such as flowing, relaxation, etc. [1], phenomena which are very difficult to simulate in a trial.

References

[1] B. Saugy, Th. Zimmermann, M. Hussain - "Three Dimensional Rupture Analysis of a prestressed Concrete Pressure Vessel Including Creep Effects". Nucl. Eng. Design, 28, 1974

[2] B. Saugy - "Contribution à l'étude théorique du comportement des structures massives en béton armé sous charges rapides". Bull. Tech. de la Suisse Romande (1969) 22.

[3] M. Hussain, B. Saugy, Th. Zimmermann - "Analyse tridimensionnelle du comportement non linéaire d'un caisson de réacteur nucléaire en béton précontraint". (Proceedings 1 SMIRT ; CID-Publications, Luxembourg, EUR-4820 d-f-e (1972).

[4] J. Ergatoudis - "Isoparametric Finite Element in Two-and Three- dimensional Stress Analysis". Ph. D. Thesis, University of Wales, Swansea, 1968.

[5] O.C. Zienkiewicz - "The Finite Element Method in Engineering". Mc Graw-Hill, New-York (1971).

[6] B. Irons - "A Frontal Solution Program for Finite Element Analysis". Int. J. Numer Method Eng., 2 (1) (1970).

[7] B. Rebora, Th. Zimmermann, J.-P. Wolf - "Dynamic Rupture Analysis of Reinforced Concrete Shells". Nucl. Eng. Design, Vol. 37, No 2 (1976).

[8] R. Hill - "The Mathematical Theory of Plasticity". Oxford at the Clarenton Press, London (1950).

[9] F. Frey - "Le calcul élasto-plastique des structures par la méthode des éléments finis et son application à l'état plan de contrainte". Communication No 33, Laboratoire de mécanique des matériaux et statique des constructions, Université de Liège (1973).

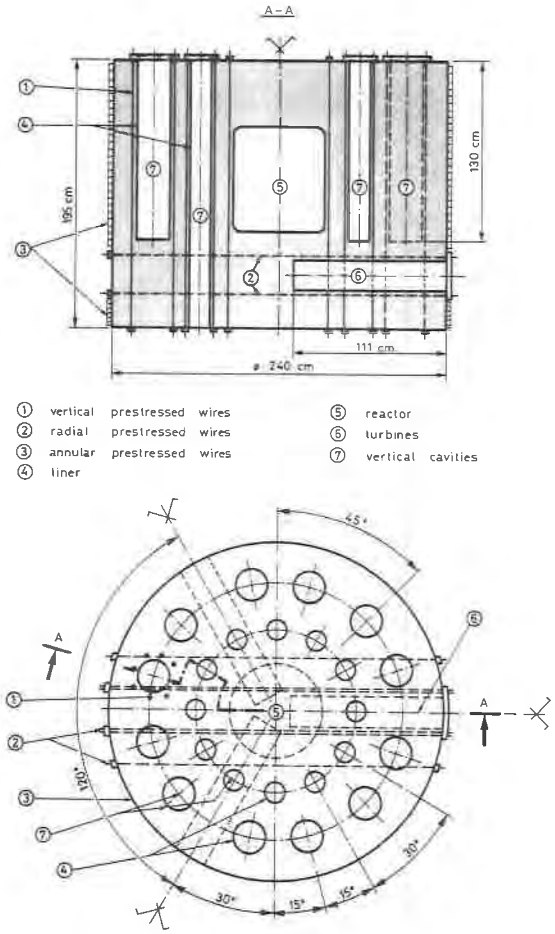


Fig. 1 TRIAL MODEL SCHEME

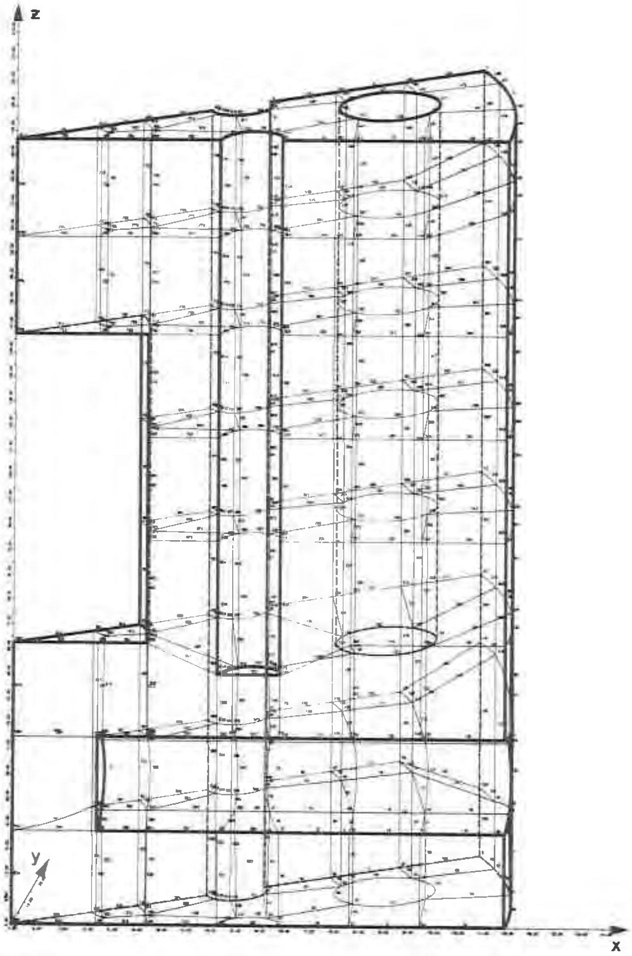


Fig. 2 FINITE ELEMENT MESH

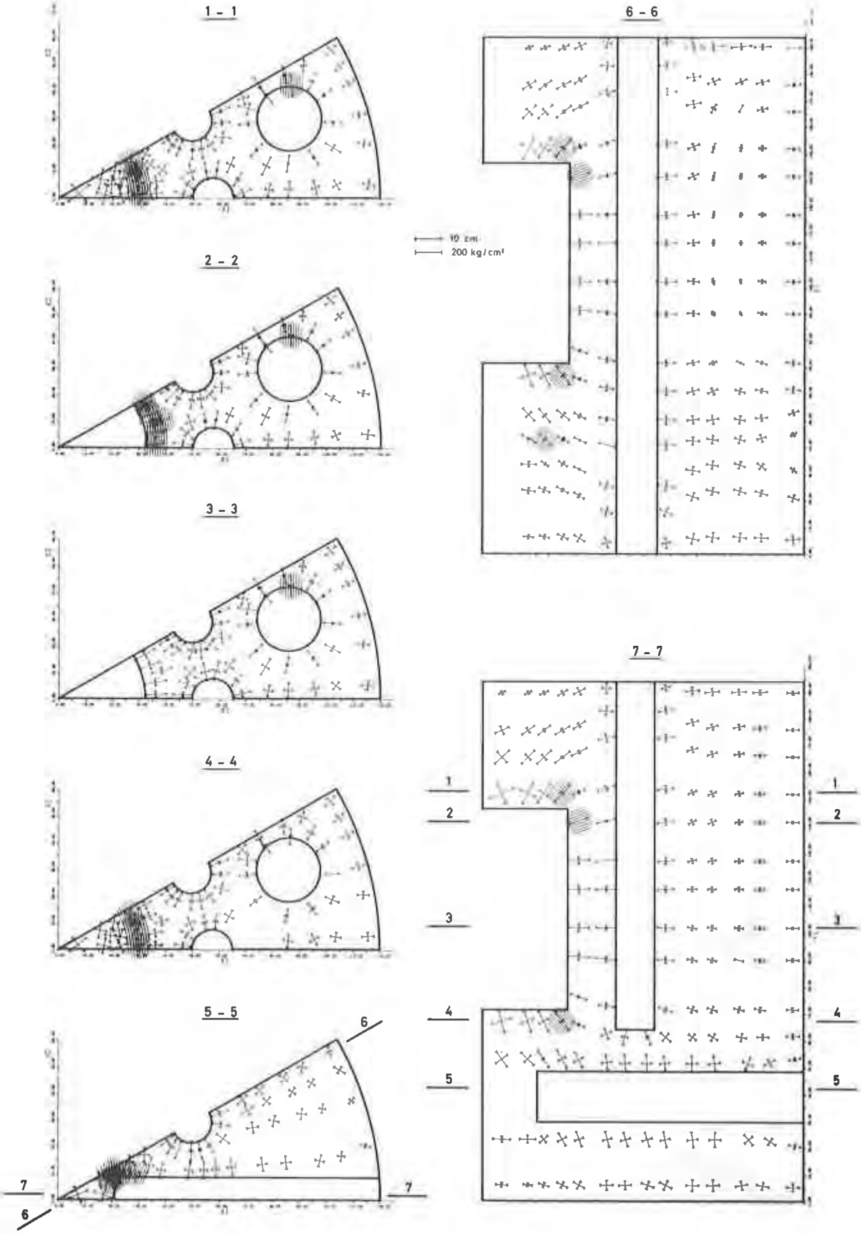


Fig. 3 CRACK PROPAGATION AT 150 kg/cm^2

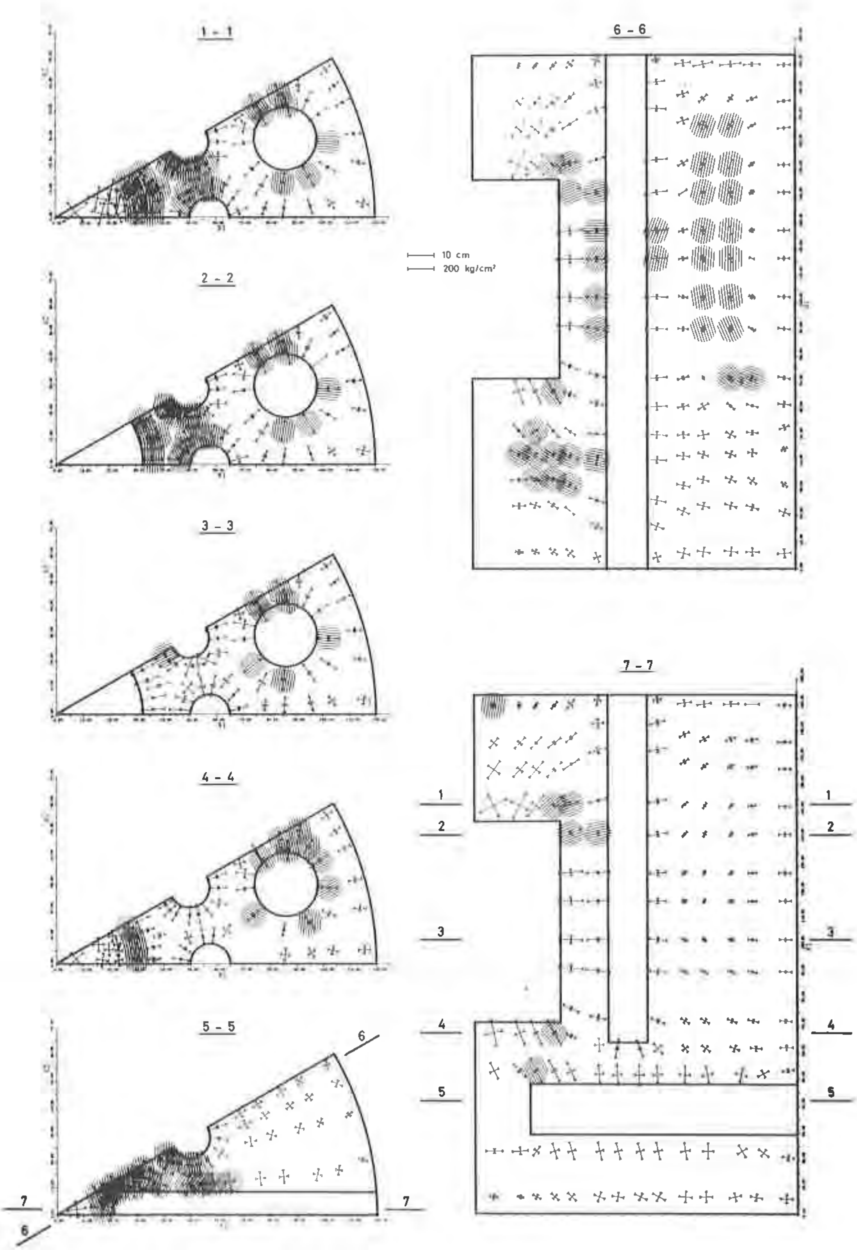


Fig. 4 CRACK PROPAGATION AT 180 kg/cm^2

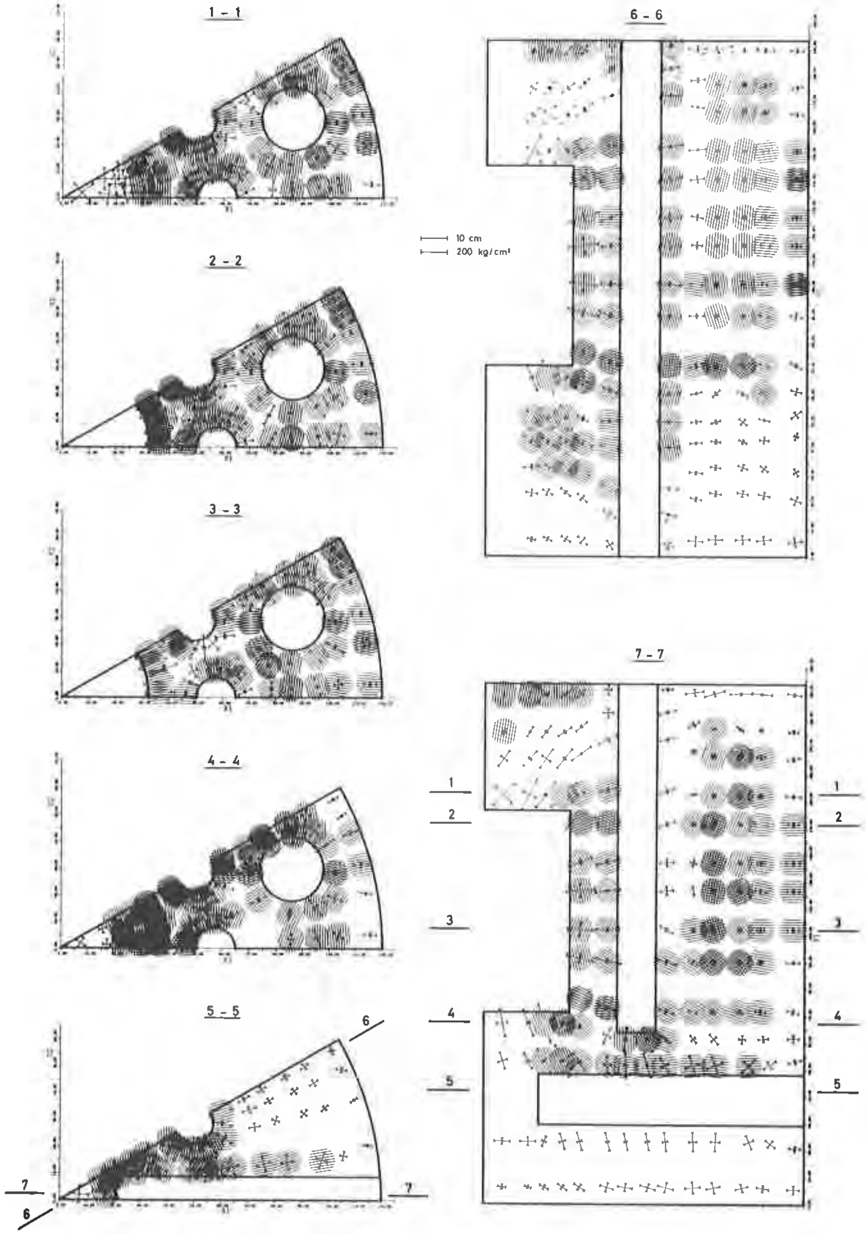
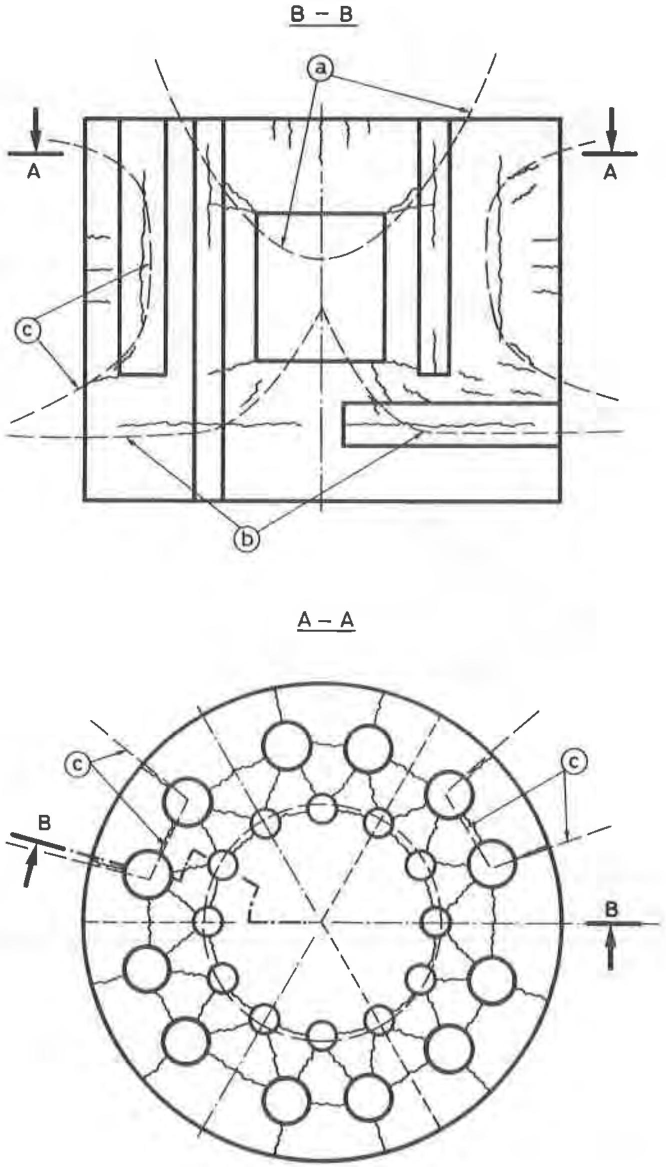


Fig. 5 CRACK PROPAGATION AT 210 kg/cm^2



B – B

A – A

Fig. 6   RUPTURE SCHEME

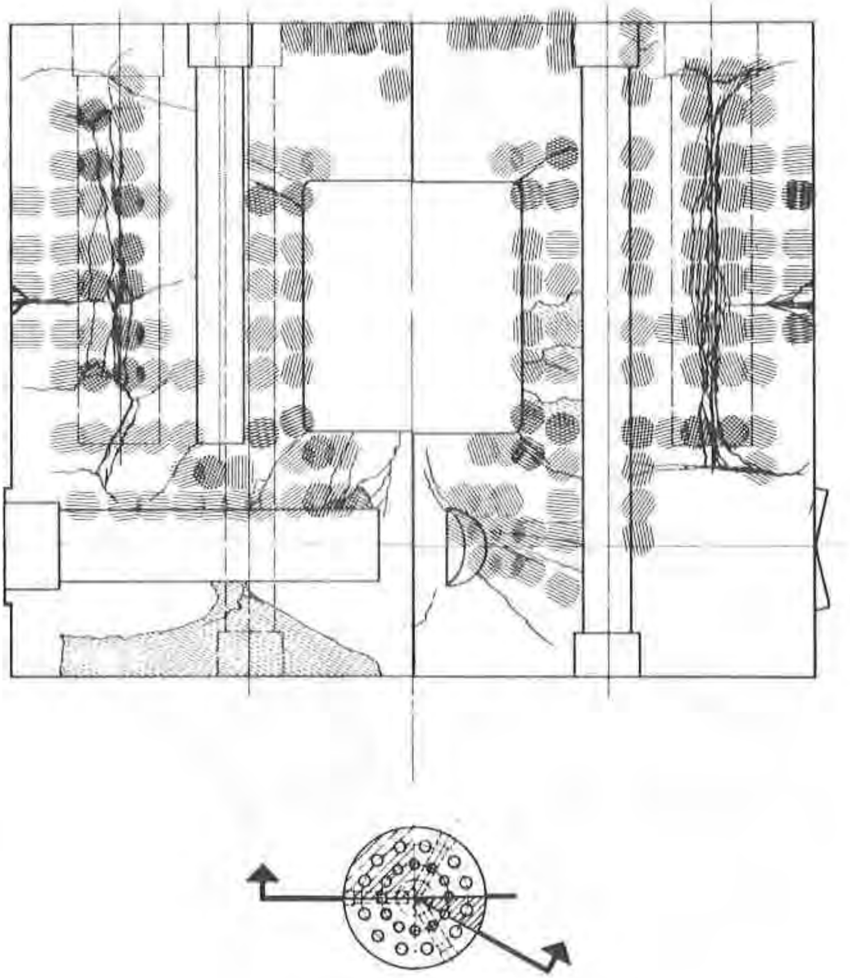


Fig. 7 COMPARISON OF RUPTURE SCHEME (VERTICAL CROSS-SECTION)

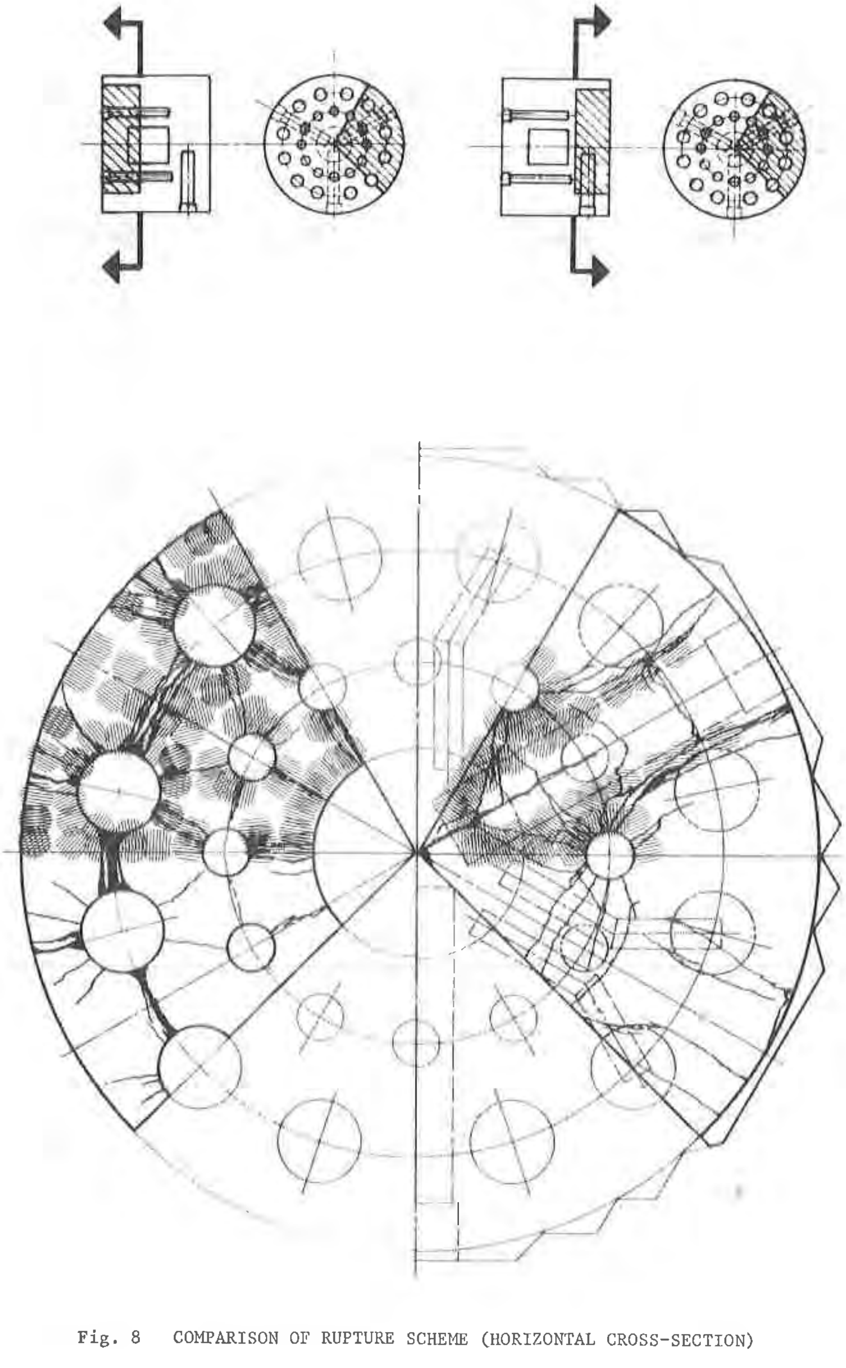


Fig. 8 COMPARISON OF RUPTURE SCHEME (HORIZONTAL CROSS-SECTION)