Transactions of the **13th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 13), Escola de Engenharia - Universidade Federal do Rio Grande do Sul, Porto Alegre, Brazil, August 13-18, 1995

The strength of the reactor cavity of VVER-1000 NPP against steam explosion

Varpasuo, P.
IVO International Ltd., Ivo, Finland

ABSTRACT: The reactor cavity of VVER-1000 NPP is a thick-walled, cylindrical reinforced concrete structure. In case of molten core-water reaction during the severe reactor accident the load carrying capacity of the cavity structure is of interest against the short impulse type loading caused by the steam explosion phenomenon. The assumed size of the impulse was 20 kPa-s and the duration was 10 ms.

The static analysis of the structure was done with the aid of ABAQUS/STANDARD and ANSYS codes and the obtained results agreed reasonably with each other.

In order to obtain a qualitative picture of the behaviour of the structure under the impulse load a simplified single degree of freedom model was developed.

The dynamic analysis of the structure was carried out using the ABAQUS/EXPLICIT and ANSYS codes. The material properties in both runs were specified to be elasto-plastic and the cracking of concrete was accounted for.

1 INTRODUCTION

The reactor cavity is the key structure in containing the effects of a severe reactor accident with the molten core-water reaction. The loading is a very fast impulse type load and the dynamic behaviour of the structure and the material modelling are essential in estimating the strength of the structure for this loading situation. In this study the strength estimation of the cavity was made with the aid of available general purpose finite-element programs. The study was made in phases and the problem was approached with different solution strategies. First, the static strength of the structure was estimated in order to gain insight. Next, the simple, one degree of freedom model was investigated. In the last phase, the dynamic behaviour of structure was estimated as accurately as the available tools permitted.

2 THE STATIC STRENGTH EVALUATION

Reactor cavity is a reinforced concrete cylinder which supports the reactor vessel and surrounds the lower part of it. The inner radius of the cylinder varies from 2.90 m to 3.67 m and the wall thickness varies from 2.60 m to 1.83 m. The cubic strength of concrete in reactor cavity was assumed to be 35 MPa and the yield strength of reinforcement was taken to be 500 MPa. Elasto-plastic material properties were specified for both concrete and reinforcement and the concrete cracking was accounted for. The amount of reinforcement in hoop direction is 200 kg/m^3, 100 kg/m^3 in vertical direction and 50 kg/m^2 in radial direction, respectively. The element type for FEM model was chosen to be 8-node solid element. The size of the element was determined by selection of six layers in radial direction. The shape of the element was required to be so nearly cubic as possible. The hoop and vertical reinforcement in the model were described with layers on the surfaces of elements. The radial reinforcement was located at the middle surface of the element. The extent of the model was one quarter of the circumference and in the end surfaces the axisymmetric boundary conditions were applied. In radial direction there were no supports and the bottom surface was supported in vertical direction. The geometry of the model is depicted in Figure 1.

The analysis was carried out with the aid of ABAQUS/STANDARD and ANSYS programs. Both programs have capability to model reinforced concrete. In ABAQUS analysis several runs were performed in order to adjust the parameters for material modelling and for equilibrium iteration. The modified Riks method was chosen for equilibrium iteration and in this method the load increments are chosen automatically by the program within limits given by the user. The maximum value of internal pressure load reached in ABAQUS analysis before the solution convergence was lost was 10 MPa. However, at this point the hoop reinforcement pressure had not reached the yield limit. This fact decreases the reliability of the ABAQUS analysis results.

The static strength estimation for the cavity was also made by ANSYS. In ANSYS-analysis the maximum hoop reinforcement stress increased very regularly and reached the yield limit 500 MPa when the load incrementation process had reached the load value of 11.4 MPa. At this point the convergence of the solution was lost. The ANSYS strength value appears to be reliable because the evident cause of the solution divergence was the yield of hoop reinforcement.

3 THE ESTIMATION OF DYNAMIC BEHAVIOUR USING SINGLE DEGREE OF FREEDOM MODEL

The goal is to study the structure with an idealized model where the hoop reinforcement bears all the loading and concrete acts only as an inertial mass. Suitable model for this purpose is the single mass model with inelastic spring. The model is thought to consist of the whole, one meter high circular ring of the cylinder wall. The inner radius of the cylinder is 2.9 m and the wall thickness is 2.6 m. The dynamic pressure impulse acts onto the inner surface of the cylinder.

The size of the pressure impulse was assumed to be 100 kPa-s. The peak pressure of 20 MPa was selected and the triangular pulse shape with the rise time of 1 ms and the total duration of 10 ms was assumed. The pulse shape is depicted in Figure 2. The single degree of freedom system is governed by the equation:

$$M\ddot{u} + R(u) = F(t) \quad (1)$$

where M is the mass of the system, $\ddot{u}$ is the acceleration, u is the displacement, R(u) is the inelastic spring force and F(t) is the loading pulse. The computation was carried out using the finite difference method and EXCEL spreadsheet program. The properties of the inelastic spring were developed for the amounts of hoop reinforceent varying between 100 kg/m^3 and 300 kg/m^3. The results for the force histories in hoop reinforcement are given in Figure 3.

With the smallest amount of hoop reinforcement the bars yield for a time period which corresponds to the double duration of the load impulse. With the largest amount of hoop reinforcement the bars yield during 5 ms at the first half-cycle of the vibration. Further, it can be observed that at highest reinforcement amount the maximum value of hoop force is 50 MN and at smallest amount of reinforcement the maximum value of hoop-force is 15 MN. For the maximum value of radial velocity we have the range from 4 m/s to 7 m/s depending on the amount of hoop reinforcement. For maximum values of radial displacement we have the range from 20 mm to 80 mm which correspond to the hoop deformation values of 0.7% and 2.8% at the inner surface of the cylinder. The natural frequency of the single degree of freedom system vary between 38 Hz and 57 Hz depending on the reinforcement amount. Based on the results of the above maximum deformation values it can be deduced that the suitable amount of hoop reinforcement for the cavity cylinder is between 200 and 300 kg/m^3.

4 THE DYNAMIC RESPONSE OF THE STRUCTURE

The first attempt to investigate the dynamic response of the structure was made with the aid of ABAQUS/STANDARD program. However, this attempt was not successful. All attempted runs stopped at about one quarter point of the pressure pulse duration. The model characteristics were exactly the same as is case of chapter 2 where the static strength was evaluated. The definition for the impulse load was taken from chapter 3. At this point we found out that there existed a new version of ABAQUS program called ABAQUS/EXPLICIT. This version was suitable for dynamic response analyses of reinforced concrete structures. The reinforcement layers of the earlier model were replaced with truss element because this was the only way to model orthotropic reinforcement in ABAQUS/EXPLICIT. For the length of the analyzed time interval the ten fold length of the pulse load, namely, 100 ms was chosen. The results of the analysis are presented in the form of the stress time histories for the hoop reinforcement at the inner surface of the cylinder. In Figure 4 the elements from the model inner surface are presented with individual curves from lower boundary of the model up to the upper boundary. So there are 12 curves in this Figure. It can be observed from Figure 4 that reinforcement stress reaches the yield limit after 5 ms from the beginning of the pulse and the yield continues up to 25 ms from the start of the history. After this yield domain the elastic vibration begins and the frequency is about 29 Hz. The stress histories for hoop reinforcement correspond to the expectations on the basis of single degree of freedom model. Opposed to the stress histories the histories of tensile strains in hoop direction on the inner surface of the cylinder do not correspond to the expectations. The histories are still increasing at the end of the analyzed time interval and the values of strains are high reaching the level of 4%. This value is high compared to the specified strain capacity of the reinforcement bars. So the strain behaviour of the computed time history remains open question to be further clarified.

The dynamic response of the structure was also analyzed with the aid of ANSYS program. The load and structural modelling were kept the same as in ABAQUS analysis. The solution method in ANSYS is the Newton-Raphson equilibrium iteration combined with Newmark time integration method. The analyzed time interval in the ANSYS run was the duration of the loading pulse i.e. 10 ms. The rising part of the load pulse, namely, 1 ms was divided into 51 load steps and the decreasing part, 9 ms, was divided into 101 load steps. The results of the ANSYS analysis were also presented in the form of maximum stress history in hoop reinforcement. This history is given in Figure 5. The maximum value of this stress does not, however, reach the yield value which makes the ANSYS results not trustworthy and indicate that ANSYS cannot solve the problem correctly.

5 CONCLUSION

The static pressure capacity of the reactor cavity cylinder was determined in the first phase of the study. A simplified manual calculation gave the value of 8 MPa for the pressure capacity. The ABAQUS/STANDARD analysis gave the value of 10 MPa and the ANSYS analysis gave the value of 11.4 MPa. The result obtained by ANSYS seems to be reliable because at the end of the analysis the hoop reinforcement had reached its yield limit 500 MPa.

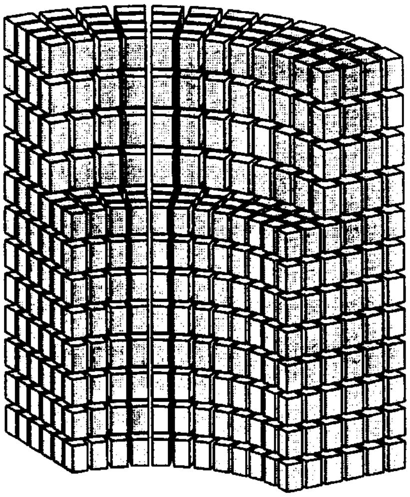
The second part of the task was the investigation of the transient dynamic response for the impulse load. The simplified analysis by single degree of freedom model gave a qualitative picture of the behaviour of the structure. This model was also useful for the estimation of the needed hoop reinforcement. From the displacement response histories it was deduced that suitable amount of reinforcement is 200 kg/m^3.

Both ABAQUS/EXPLICIT and ANSYS were capable to determine the dynamic response of the structure. The ABAQUS analysis gave good results for the stress histories of hoop reinforcement but the results for strain histories were controversial and not consistent with the stress histories. ANSYS analysis was done only for the duration of the pulse. The maximum value of the hoop reinforcement stress was about one half of the yield stress. This result is in strong variance with the results of ABAQUS analysis and with the result of single degree of freedom model.

It seems to be that the dynamic behaviour of the structure is not sufficiently explained by the results of this investigation and that more research is needed.

REFERENCES

ABAQUS User's manual, 1993, Hibbit, Karlsson & Sorensen Inc.
ANSYS User's manual, 1994, Swanson Analysis System Inc.



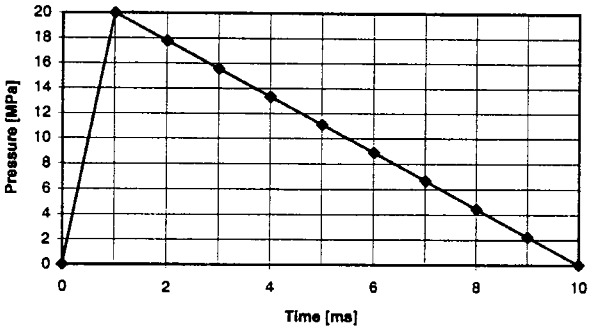


Figure 1. The geometry of the FEM model of reactor cavity

Figure 2. The impulse loading of the internal pressure

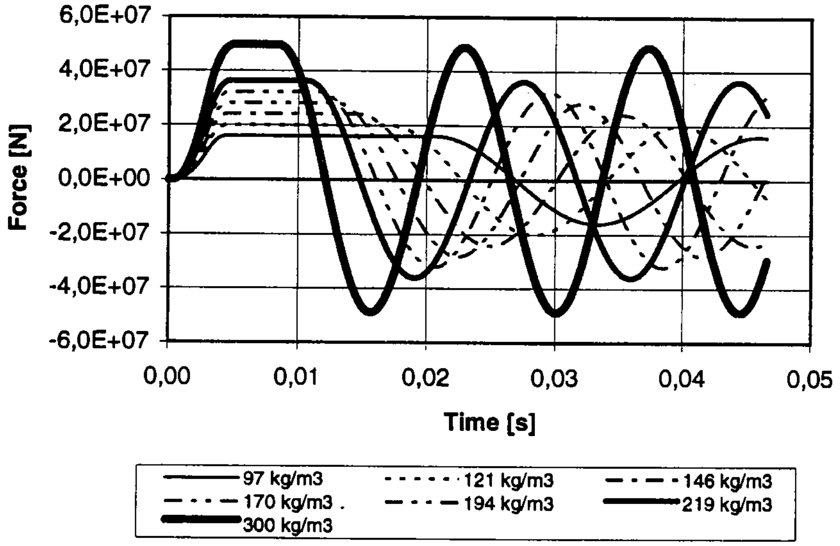


Figure 3. One DOF model. Force histories in hoop reinforcement

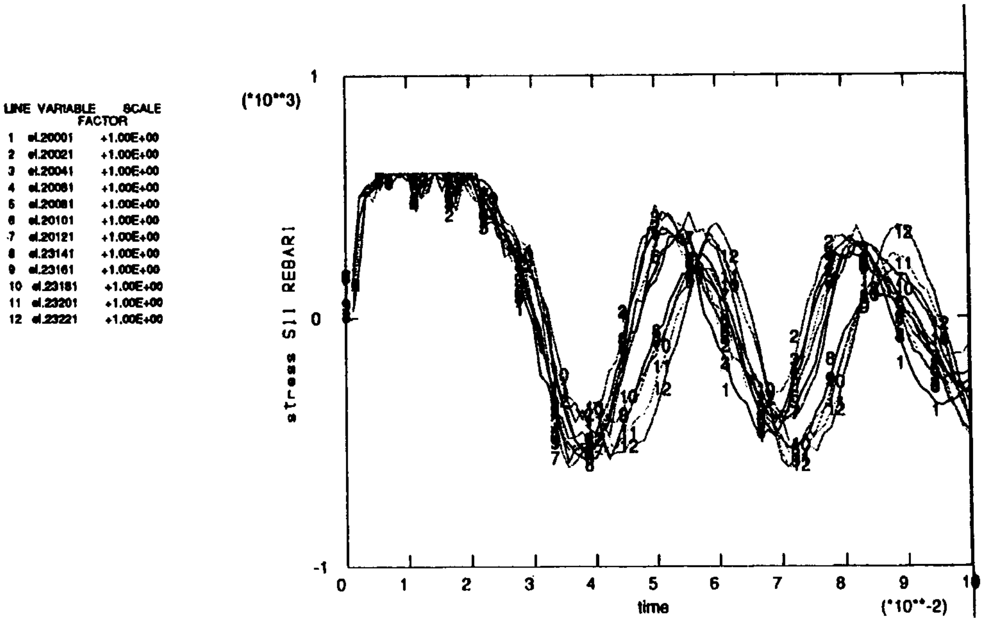


Figure 4. ABAQUS run. Stress histories in hoop reinforcement [MPa]

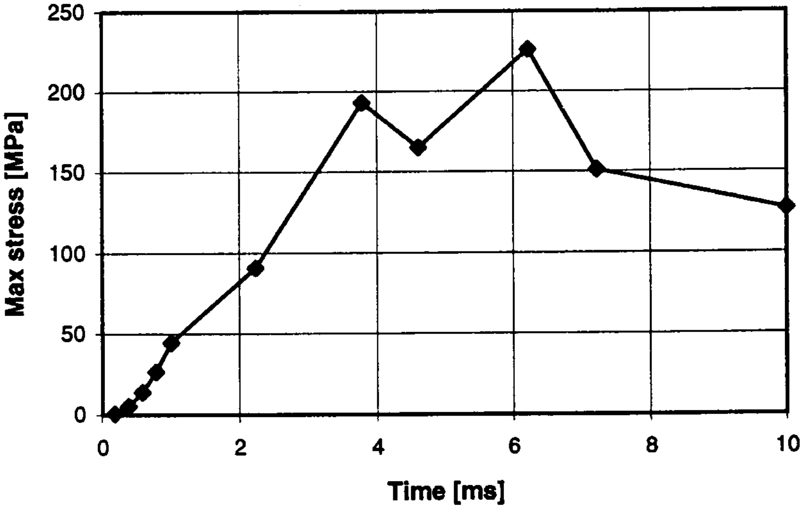


Figure 5. ANSYS run. Maximum stress in hoop reinforcement