

The Static Overpressurized Analysis of 1:6 Scale Reinforced Concrete Containment Using 3-D Unit Strip Model

Ing-Tzer Hwang, Kuen Ting

Institute of Nuclear Energy Research, Lung Tan, Taiwan

ABSTRACT

Nuclear power plant containment integrity has been one of focusing points on the safety research of US Nuclear Regulatory Commission. To predict the response of the Sandia 1/6 scale reinforced concrete containment model under severe overpressurization test, a 3-D strip model to deal with the global response of the structure using the advantages of an axisymmetric model is successfully developed and provided the boundary conditions for the local tearing analysis by ANSYS program. Good correlations between the computed results and the experimental data are shown.

INTRODUCTION

From the "Quick Look Report" for the model overpressurization-to-failure test sponsored by USNRC and conducted by Sandia Lab., the notion that concrete containment will leak before catastrophic burst under excess pressure was well proven[1]. The tearing of the liner plate has been shown to be one of the major causes of leakage. The objective of this study is an attempt to develop a methodology to simulate the local behavior of liner plate around the penetration area by using three dimensional analysis.

The nonlinear effects, such as geometric finite deformation and elastic-plastic material behaviors, are definitely required to be considered if realistic results are to be obtained. Due to the complexity of analyzing the nonlinear behavior, axisymmetric global models are usually used in these problems[2-4]. However, the homogenization approach of rebars, accounting their area, directions and elastic-plastic properties to meet the axisymmetric behavior, must be specially treated. The crack behavior of the concrete due to hoop stress will not be considered and make this analysis unrealistic. The complete three dimensional treatment instead of two dimension is imperative. The need for large computer memory and expense of vast computer time usually make the analysis impractical or impossible. The aim of this work is thus to develop a 3D strip model to deal with the global response of the containment using the advantages of an axisymmetric model and to provide the boundary conditions for the local model.

In the 3D strip model and local model developed herein, the computer code " ANSYS " version 4.3[5] is employed to analyze the 1/6 scale reinforced concrete containment model tested by Sandia National Lab.[1]. The concrete and rebar are modeled with the 8-node 3D reinforced concrete solid element (STIF 65). The liner plate is modeled with the 3-node plastic triangular shell element (STIF 48) considering the membrane stiffness only. The gap elements are used to model the basemat liftoff. The axisymmetric arrangement is achieved by setting the symmetric boundary conditions of circumferential displacements along the lateral

surface $u_\theta = 0$. The coupled relations are also applied to nodes at the same elevation. The local model analysis included a fine mesh representation of the penetration area around the elevation 13 feet. Special attention is placed on the discussion of liner plate behavior in the intersection of different plate thicknesses.

Monotonically increasing pressure loading is applied to the 3D strip model. The increments of the loading are 2-3 psig up to 50 psig internal pressure and 5 psi afterwards. The maximum applied pressure is 160 psig.

The convergence criteria of the bilinear elements, plasticity and large deflection are all considered. The incremental Newton-Raphson approach is employed for analyzing plasticity effects. Yielding of liner and rebars is based on the von Mises criterion, Prandtl-Reuss flow rules and isotropic hardening rule. The concrete will fracture if any principal stress reaches 500psi (yielding stress of concrete). Crushing is not considered here.

Using the developed analysis procedure, detailed analyses are performed for the 1/6 scale containment test. Good correlations between the computed results and experimental data exists up to 120 psig. Due to the convergence errors, the deviations of computed results are significant after 120 psig. However, this 3D strip finite element model is to provide an efficient realistic three dimensional analysis. The conclusions drawn from a detailed examination of the obtained results as to the usefulness of the containment overpressurization are also noted.

BRIEF DESCRIPTION OF THE TEST MODEL

The 1/6th scale reinforced concrete containment test was conducted by U.S. Sandia National Laboratory as shown in Fig.1. The model has a cylindrical body with capped equipment hatch and penetration opens, hemispherical dome and flat basemat. The model is 33'7" in dome area, and 3'7" in basemat. The inside surface of the model is lined with 1/16" and 1/12" steel plate in cylindrical body and dome area respectively. No.4 steel rebar is mostly used.

FINITE ELEMENT MODEL

The 3D strip finite element model used in the analysis is shown in Fig.2. Five and four layers of the elements are included in the basemat, and the cylindrical wall and dome, respectively. The concrete and rebars are modeled with 8-node, 3D reinforced concrete solid elements (STIF 65). The rebars are smeared throughout the concrete element considering three different directions to represent the hoop, meridional and seismic rebars. The liner plate is modeled with 3-node plastic triangular shell elements (STIF 48) considering the membrane stiffness only. The bottom of the basemat rest on the soil is modeled by 3D interface element (STIF 52) for supporting compression only to simulate the uplift behavior of basemat. A total of 288 concrete elements, 122 plastic triangular shell elements and 42 gap elements are used in the complete 3D strip model resulting in 670 nodes with 1367 degrees of freedom as shown in Fig.3.

To achieve the axisymmetric arrangement, symmetric boundary conditions of $u_\theta = 0$ (u_θ is the circumferential displacement) are applied along the lateral surfaces of the strip model, $\theta = 0$ and $\theta = \alpha$. The coupled relations are also considered for the nodes equal elevations.

To investigate the local effect of liner plate tearing that results in major leaks, a three dimensional local model with refined grid covering the area between the different thicknesses at the elevation 13 feet is generated. The model grid arrangement is shown in Fig.4. The liner plate is modeled with 3-node plastic triangular shell elements (STIF 48). 530 plastic triangular shell elements are used in this local model resulting in 288 nodes with 1728 degrees of freedom. The displacement boundary conditions are taken from the global strip model. The same pressurization condition as the global analysis is applied to the local model.

Monotonically increasing pressure loading is applied to this strip model. The increment of the loading is chosen as 2-3 psi up to 50 psig internal pressure and 5 psi afterwards. Maximum applied pressure is 150 psig.

The convergence criteria used in the ANSYS program are as follows:

A. Bilinear element convergence

For a bilinear element, since it has at least two different status conditions (open/close gap, sliding/no sliding interfaces), convergence occurs whenever the status of all the bilinear elements remains unchanged from the previous status.

B. Plasticity convergence

The plasticity convergence occurs when the "plasticity convergence ratio" for all elements is less than or equal to 10 percent.

C. Large deflection convergence

The convergence is reached whenever the change in the deflection between consecutive iterations, for each translational degree of freedom, is less than 5.0 percent. A maximum of 5 iterations per load step is allowed.

For STIF 48, the incremental Newton-Raphson approach is used for analyzing plastic effects; whereas for STIF 65, the direct iteration procedure, i.e. residual strain method, is employed.

Yielding of liner and rebars is based on the von Mises criterion, Prandtl-Reuss flow equations and isotropic hardening rule. The geometric nonlinearities are also included in this analysis. The material will fracture if any principal stress reaches the yield stress of the concrete. Crushing is not considered here.

RESULTS AND DISCUSSIONS

From the analysis results, it can be observed that up to 28psig the containment is still in elastic range. Over this value, the meridional cracks appear at the base of the cylindrical wall. When the pressure reaches 42 psig, the concrete hoop cracks abruptly appear and propagate upward and downward. In this load period, the option KUSE = -1 in the ANSYS program is chosen, i.e. the reformulation of the stiffness matrix is performed for each load step. However, the reformulation of the stiffness matrix will make the solution divergent after 42 psig. Thus, option KUSE = 0 is chosen over 42 psig to maintain convergence. The errors due to non-reformulation of stiffness matrix will not be significant, since most portions of the concrete have fractured and the loads are directly applied to the liner and rebars.

The meridional cracks start at the basemat and propagate outward to form major cracks through the cylindrical wall at around 40 to 70 psig. The yielding of the liner plates occur at 85 psig at mid-height of cylindrical wall and spread to the whole cylinder rapidly. Hoop rebars begin to yield also at mid-height of cylindrical wall about 130 psig.

The histories of radial and vertical (relative to the basemat) displacements at mid-height of cylindrical wall are displayed in Fig.5 and 6, respectively. Fig.7 and Fig.8 show the axial strain of meridional rebar at the base of cylindrical wall and hoop rebar at the mid-height of cylindrical wall, respectively. The maximum principal strain of liner at the base and mid-height of cylindrical wall are drawn in Fig.9 and Fig.10, respectively. The experimental data from the Sandia National Laboratory are also shown in Fig. 5-10. Good correlations between the computed results and experimental data are shown up to 120 psig. The significant deviations of computed results compared with experimental data are due to the accumulated errors of each load increment in material and geometric nonlinearities.

The effort of local model analysis is concentrated on the local stress and strain histories in the vicinity of different thickness area. From the stress contour plot shown in Fig.11, the

highest liner stress appears near the welding connection between 1/16 inch and 3/16 inch liner plate. This gives the potential of liner rupture at that area. The maximum principal strain distributions with the internal pressure at different regions are shown in Fig.12. At the region far away from the insert thicker plate area (as shown in Fig.4, region B), the values of the maximum principal strain is good correlations with the results of the strip global analysis. The maximum principal strain near the corner of the insert thicker plate approaches 2% higher than other areas due to the stress concentration effect.

CONCLUSION

An efficient 3D strip finite element model has been successfully developed herein instead of a complete three dimensional analysis to predict the ultimate capacity of reinforced concrete containment due to accidental overpressurization. Based on the results of the global strip model, the local effects of liner plate are also evaluated.

REFERENCES

- [1] Horschel,D.S., "Quick look report" for the Model Test, Sandia National Lab.,Aug. 1987.
- [2] Dunham,R.S.,et. al., "Methods for ultimate load analysis of concrete containments", EPRI Report NP-4046, June 1985.
- [3] Dameron,R.A.,et. al., "Methods for ultimate load analysis of concrete containments: second phase", EPRI Report NP-4869M, March 1987.
- [4] McGaughy,J.P.Jr.,Lin,F.T. and Sen,S.K., "Ultimate internal pressure capacity of a reinforced concrete Mark III containment", SMiRT 8th Conference, Vol.J3/9, pp.157-162.
- [4] Hsiau,H.N.,et. al., "Pretest predictions for the responses of the Sandia 1/6th scale reinforced concrete containment model under severe overpressurization", TPE Report, August 1987.
- [5] Desalvo,G. and Gorman,R., "User information manual and theoretical manual", ANSYS Engineering System.

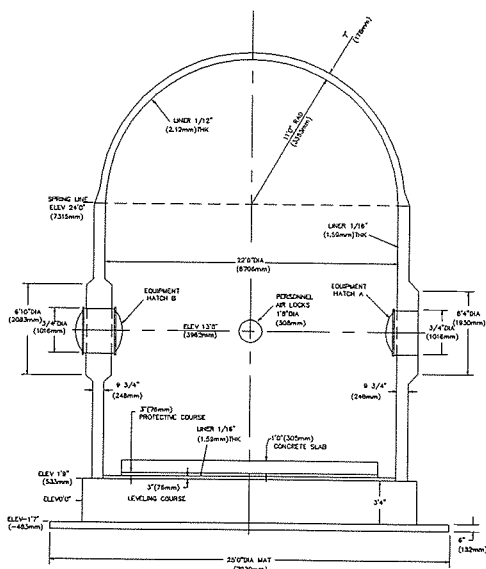


Fig.1 Sandia 1/6 scale reinforced concrete containment model

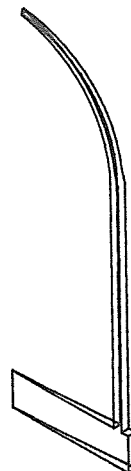
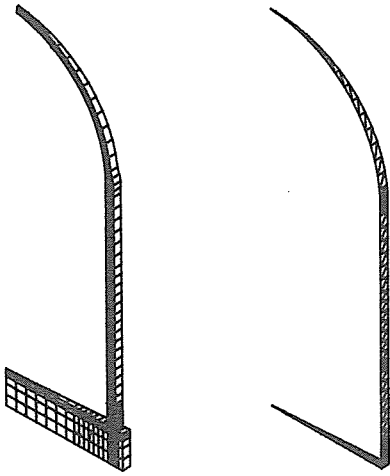


Fig.2 3D strip finite element model



(a) Concrete (b) Liner
Fig.3 Finite element model

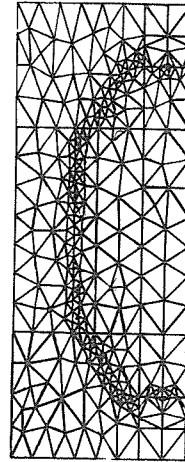


Fig.4 Finite elements arrangement of local model

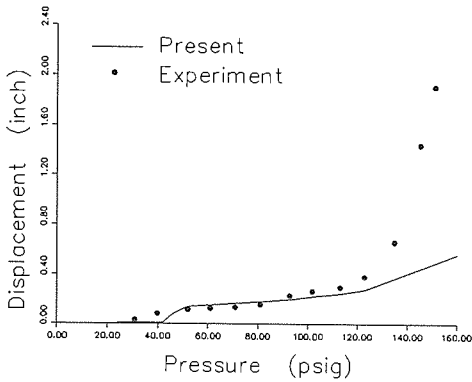


Fig.5 Radial displacement at cylinder mid-height

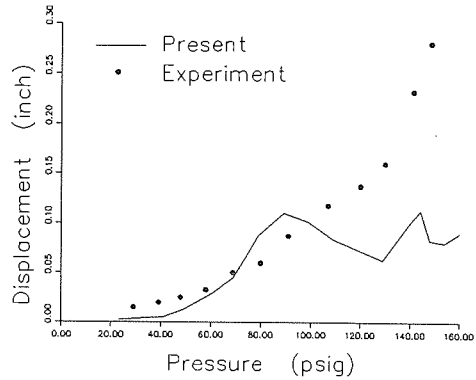


Fig.6 Vertical displacement at cylinder mid-height

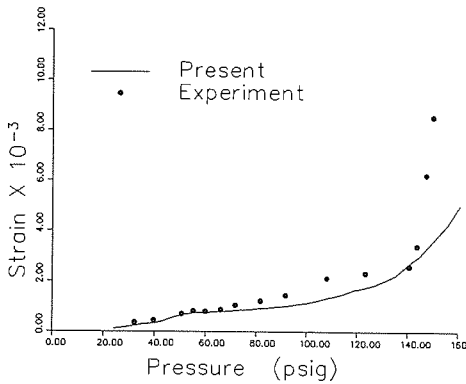


Fig.7 Axial strain of meridional rebar at the base of cylindrical wall

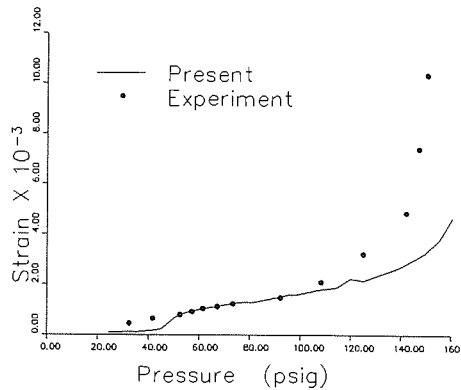


Fig.8 Axial strain of hoop rebar at cylinder mid-height

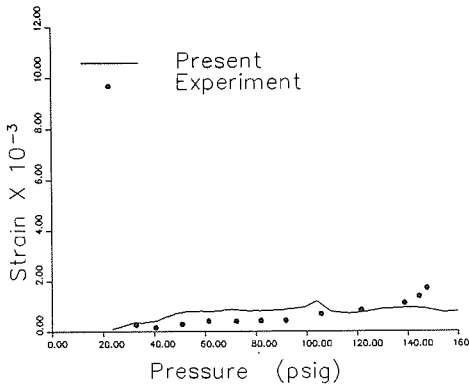


Fig.9 Maximum principal strain of liner at the base of cylindrical wall

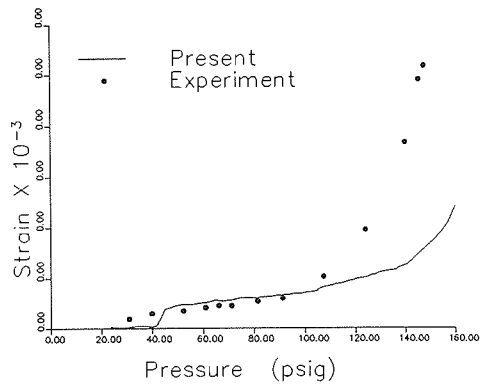


Fig.10 Maximum principal strain of liner at cylinder mid-height

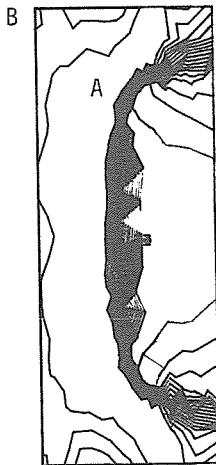


Fig.11 Stress contour in the vicinity of different thickness area

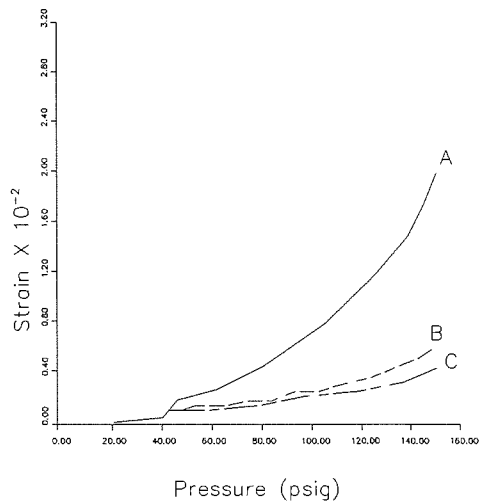


Fig.12 Maximum principal strain distribution

The positions of strain curve A,B see Fig.11

Strain curve C -- global analysis