20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20)
Espoo, Finland, August 9-14, 2009
SMiRT 20-Division V, Paper 1885

INELASTIC ANALYSIS OF BARC PRESTRESSED CONCRETE CONTAINMENT MODEL

Tarvinder Singh, R K Singh, A K Ghosh, and H S Kushwaha

Reactor safety Division, Health Safety and Environment Group
Bhabha Atomic Research Centre, Trombay, Mumbai, India, e-mail: tarvindersinghs@gmail.com

Keywords: PHWR Containment Structure, Ultimate Load Capacity, Concrete Structures, Finite Element, Inelastic Analysis, Axi-symmetric Model, Shell Model

1 ABSTRACT

Bhabha Atomic Research Centre (BARC) has initiated an experimental program at BARC Tarapur Containment Test Facility to evaluate the ultimate load capacity of Indian PHWR containment structures. For this study, BARC Containment Model (BARCOM), which is 1:4 scale representation of Tarapur Atomic Power Station (TAPS) unit-3&4 540 MWe PHWR Inner Containment of Pre-stressed Concrete has been constructed with the design pressure of 0.1413 MPa (1.44 kg/cm2 (g)).

The objectives of the present paper are to understand the behavior of containment model under internal over-pressurization, predict the response of BARCOM at specified sensor locations, obtain the various failure modes and identify the additional critical locations important for instrumentation during the experiment. The pre-test analysis has been carried for BARCOM, which is going to be tested shortly.

2 INTRODUCTION

Bhabha Atomic Research Centre (BARC) has initiated an experimental program at BARC Tarapur Containment Test Facility to evaluate the ultimate load capacity of Indian PHWR containment structures [1]. For this study, BARC Containment Model (BARCOM), which is 1:4 scale representation of Tarapur Atomic Power Station (TAPS) unit-3&4 540 MWe PHWR Inner Containment of Pre-stressed Concrete has been constructed. The BARCOM shown schematically in Fig 1 includes all the main features of the prototype containment structure such as the pre-stressed concrete cylindrical wall structure with a tori-spherical dome, two steam generator (SG) openings in the dome along with main air lock (MAL), fuelling machine airlock (FMAL) and emergency air lock barrel (EAL) openings in the cylindrical wall. BARCOM has been designed for an internal design pressure (Pd) of 0.1413 MPa (1.44 kg/cm^2 (g)), which is same as the prototype containment structure.

The objective of the present numerical simulation is to understand the behavior of containment model under internal pressure, predict the response of BARCOM at specified sensor locations and obtain the various failure modes and identify the additional critical locations important for instrumentation during the experiment. The structural response of the containment model is assessed in terms of wall and dome displacement, cracking of concrete, longitudinal and hoop stresses and strains. Another objective of the analysis was to predict the various failure modes of BARCOM with regard to the concrete cracking, reinforcement yielding and tendon inelastic behavior along with the estimation of the ultimate load capacity of the containment model.

Two models have been used for the present study as reported in the round robin pre-test proceedings [1], which have been evolved from the information provided in the design document and the associated drawings [2]. Initially a simplified axisymmetric model is used to predict the BARCOM response in the free field region up to the ultimate load capacity. Subsequently a detail 3D shell model is used, which simulates all the five major openings namely MAL, FMAL, EAL and the two SG openings. The axisymmetric model and the shell model predict the ultimate load capacity as 0.50 MPa (3.54Pd) and 0.3674 MPa (2.6 Pd) respectively. The preliminary simplified axisymmetric analysis was undertaken in the initial phase for understanding the general overall structural behavior of the BARCOM to identify the critical locations for instrumentation during the model construction. The 3D shell model predicts a factor of safety of 2.6Pd, which BARCOM test model is expected to reach. The failure location is predicted near MAL. However, further analysis is needed

to quantify the factor of safety with detail 3D solid elements, which should account for the local structural behavior due to various openings more accurately.

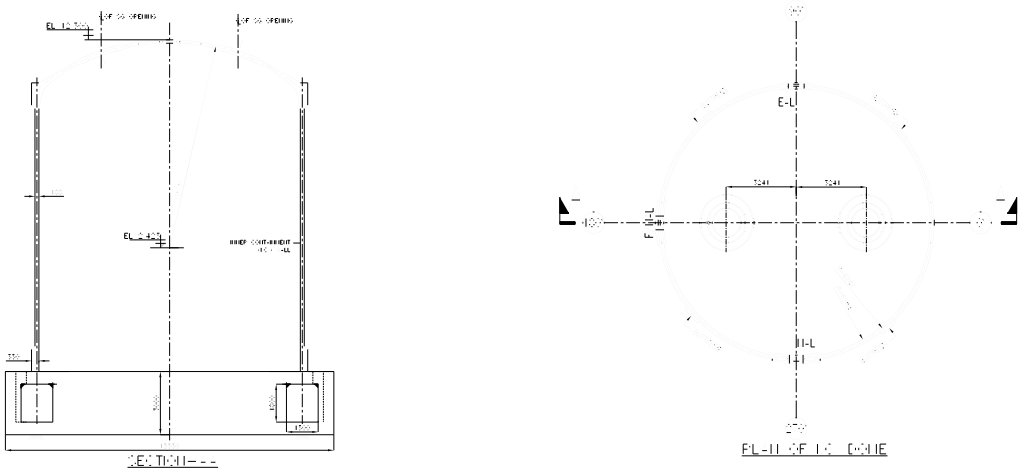


Fig.1. Outline sketch of BARC Containment Model (BARCOM)

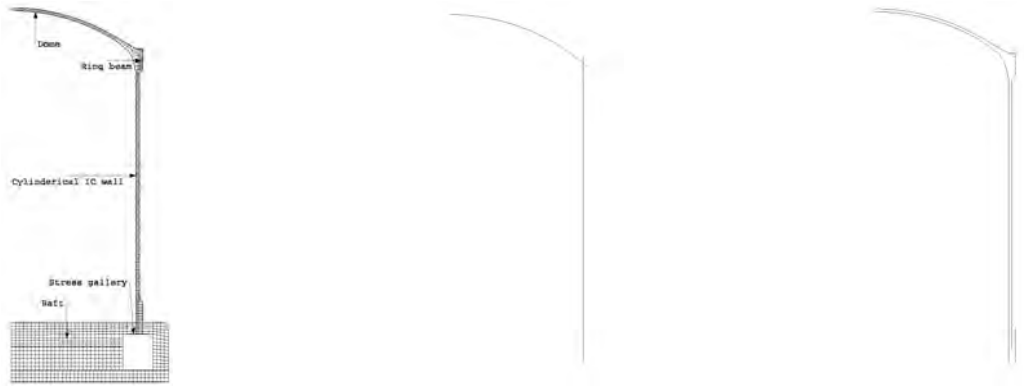
3 DESCRIPTION OF NUMERICAL MODELS:-

To evaluate the ultimate load capacity of BARCOM two finite element models have been used for the present pre-test analysis using the formulation available in the in-house code ULCA [3], the features of this code is described in Section 4 of this report.

3.1 Axisymmetric Finite Element Model: -

Initially a simplified numerical analysis procedure is selected for the analysis of BARCOM with axi-symmetric model [4]. With this assumption on the model geometry, the various openings are not considered for the present model. This numerical model thus predicts the average free field response of the BARCOM and the influence of the discontinuities due to the raft wall junction and the wall ring beam and dome junctions are also accounted in this model.

Fig 2 shows the FEM model details that have been used for the present numerical analysis. The concrete structure including the wall, dome and raft are modeled by quadratic 8-node continuum axi-symmetric elements. In IC wall and dome the element size is governed by spacing of pre-stressing cables and the cover around the reinforcement. In raft the element size is governed by the reinforcement spacing. Total 1810 element are used to model the concrete structure. The hoop reinforcements and tendons are modeled as rebar elements, which are represented as steel layers of equivalent smeared thickness in a particular continuum axi-symmetric element. These rebar elements have uni-axial behavior resisting only the axial force in the bar direction, which is the hoop direction in the present model. A total of 470 steel rebar elements for hoop reinforcement members and 235 number of rebar elements for hoop tendons have been included in the present model. The longitudinal reinforcements and tendons were modeled as embedded axi-symmetric membrane elements, which is average (single circumferential layer) representation of the discrete reinforcements and tendons. Using orthotropic material properties it is ensured that the bars carry stress only along their individual axial directions. The thicknesses of the steel layers have been calculated to represent the BARCOM reinforcement and pre-stressing tendons in the axi-symmetric model. The number of longitudinal embedded steel elements is 301 for the longitudinal reinforcement members and 147 for the longitudinal tendons. A constant pre-stress of 1478 MPa has been applied as initial stress in the hoop rebar elements and embedded longitudinal tendon elements. In present study, it is assumed that there is no slip between the concrete and steel rebar /embedded members. To consider the effect of the reinforcement, the tension stiffening is used in concrete material model. At the base of raft fix boundary conditions are applied (u_r=0 and u_z=0) and on the axis of symmetry the axisymmetric boundary conditions are applied (u_r=0).

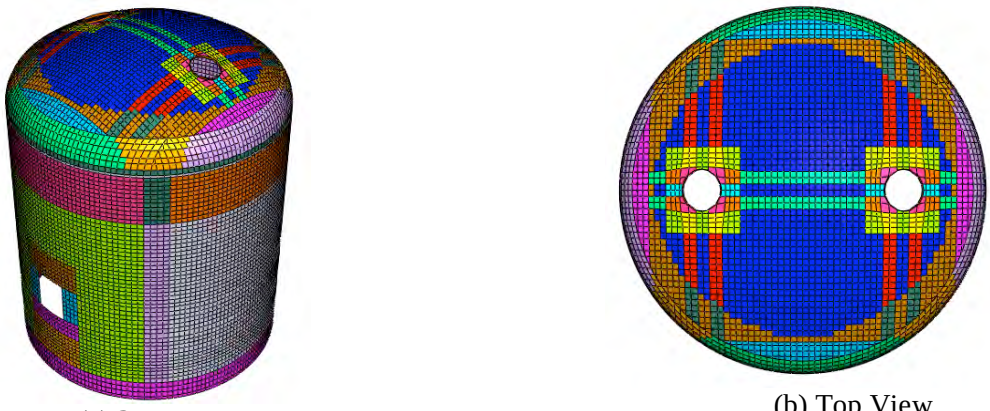


(a) (b) (c)

(a) Concrete FE model , (b) & (c) Longitudinal Tendon and Reinforcement FE Models
Fig.2. Axisymmetric Finite Element Model of BARCOM

3.2 Shell Finite Element Model: -

Subsequently a detail 3D shell model is evolved, which simulates all the five major openings namely MAL, FMAL, EAL and the two SG openings. Fig 3 shows the FEM model that has been used for the present numerical analysis. The FEM formulation is based on the bilinear layered shell element formulation for thin shell structures using the first order shear deformation theory for the composite structure. The analysis code adopts degenerate concept of formulating four nodded general isoparametric shell elements using selective integration scheme. In this code a layered approach with single midpoint integration scheme is adopted for each layer which takes into account the non-linear stress profile in the thickness direction and material properties can be modeled as a discontinuous function of thickness. This layered approach has been used to represent the different concrete layers thus it is possible to simulate progressive cracking through the shell thickness for bending loads. The reinforcement steel is represented as a smeared layer of equivalent thickness which has uniaxial strength and rigidity and is attached with concrete to simulate no slip condition, thus global anisotropic behavior of shell and material non linearity in steel rebars are accounted. For the present analysis bilinear four noded 3D shell elements having six degree of freedom at each node and seven number of concrete layers across the shell element has been used. All the hoop and longitudinal reinforcements and tendons were modeled as smeared layers of steel. The geometric features of the main penetrations and buttresses were modeled accurately to simulate the extra stiffening provided by the local thickening of the wall. Total 13748 shell elements are used for the present analysis. At the wall base fix boundary conditions are applied. A constant prestress of 1478 MPa has been applied as initial stress.



(a) 3D view (b) Top View

Fig.3. Three-Dimensional Shell Finite Element Model for BARCOM

3.3 Numerical Analysis Procedure:-

The nonlinear analysis is carried out with both the models using Newton Raphson Method of incremental load stepping technique [5]. In the initial first step the dead weight and pre-stressing forces are considered

for analysis. Subsequently the design pressure of 0.1413 MPa (1.44 Kg/sq cm) is applied, which is incremented in steps to arrive at the ultimate load capacity of the model. Displacements, stresses, strains and the inelastic response in terms of concrete cracking, reinforcement and tendon yielding are studied at different load steps.

4 MATERIAL TEST DATA FOR ANALYSIS:-

4.1 Concrete :-

For analysis parabolic equation (3.1) is used for curve fitting of given uniaxial Stress- Stain curve data (Table 1) of concrete.

$$\sigma = E_0\varepsilon - (0.5E_0\varepsilon^2)/\varepsilon_0 \tag{3.1}$$

Where
σ = Stress in MPa
ε = Stain
E_0 = Modulus of Elasticity
ε_0 = Stain with respect to Ultimate stress (0.002 form IS- 456)

Table 1. Concrete Material Data

Cubic Characteristic Compressive Strength	45 N/mm^2
Modulus of Elasticity	33540 N/mm^2
Ultimate tensile Strength	2.78 N/mm^2

4.2 Steel Rebar

For steel bar bilinear elastic-plastic stress strain curve are used. That is generated from given data in Table 2.

Table 2. Steel Material Data

Yield Stress	415 N/mm^2
Modulus of Elasticity	200000 N/mm^2

4.3 Tendon :-

For steel bar bilinear elastic-plastic stress strain curve are used. This is generated from given data in Table 3.

Table 3. Steel Material Data

0.2 % Proof Stress	1683 N/mm^2
1% Extension Stress	1649N/mm^2
Ultimate tensile strength	1848 N/mm^2
Modulus of Elasticity	189600 N/mm^2

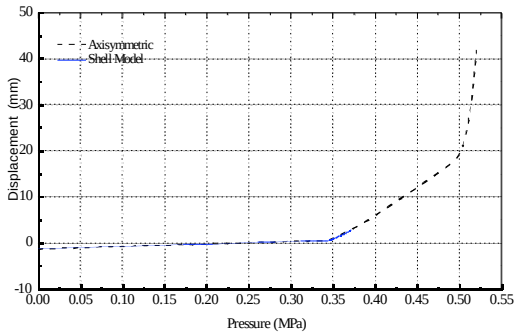
5 CONSTITUTIVE AND FAILURE MODELS

The analysis has been carried out with an in-house finite element code ULCA [3]. This code has been extensively used to predict the ultimate load capacity of Indian Pressurized Heavy Water Reactor (PHWR)

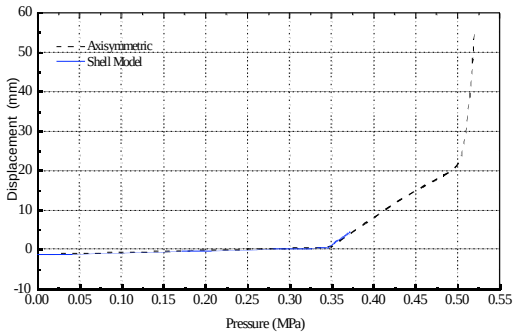
reinforced and pre-stressed concrete containments for postulated severe, beyond the design basis accidents. The material model includes orthotropic behaviour of concrete with stress and strain based failure criteria. The compressive yielding and crushing is predicted by Drucker Prager theory using the concrete compressive strength. In addition, the strain based failure criterion is also used which depends on the ultimate strain of concrete. For tensile load, in place of discrete crack simulation the smeared crack model predicts the concrete cracking. Tensile strength based criterion is used to detect the crack initiation in concrete. In the numerical simulation the crack initiation is assumed when the principle stress in concrete reaches the tensile strength of concrete in uniaxial tension at any integration point. Further, the through thickness crack is assumed when all the integration points across the thickness are cracked. The stress biaxiality is considered with the help of Kupfer's experimental curve [6]. The softening behaviour of concrete is predicted with the help of tension stiffening parameters that account for the bond effect between the steel members and concrete. The non-linear behaviour of the rebars and steel liners is simulated by Von Mises yield criterion. The detail theoretical formulation is available in [7].

6 PLOTS OF STRAIN AND DISPLACEMENT HISTORIES AS FUNCTION OF PRESSURE AT SPECIFIED SENSOR LOCATIONS (SSLs):-

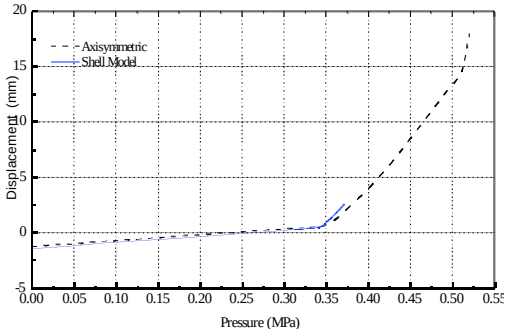
During the analysis 69 Standard Sensor Locations (SSL) were monitored and out of which response of some location were as shown in Fig4. The plots presented predict the linear and nonlinear responses of containment due to pre-stress with dead load, design pressure and further due to over-pressurization respectively. At few locations the average free field response obtained from the axisymmetric model are also included in addition to the results obtained with the detail 3D shell model for comparative study.



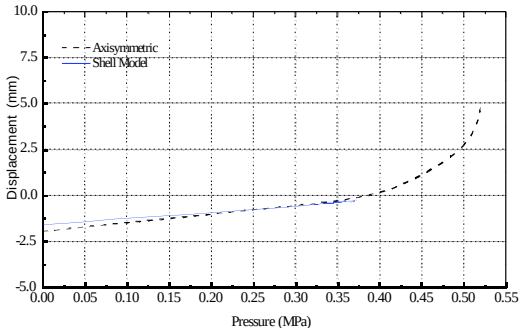
SSL1- Radial Displacement at El 3.00 m at free field



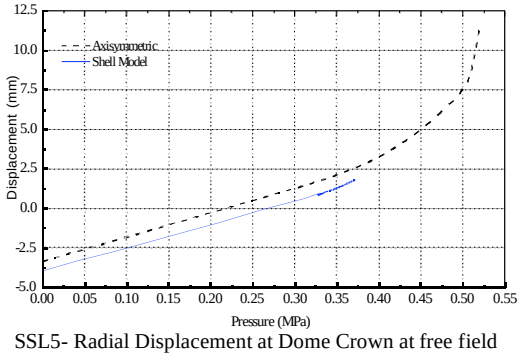
SSL2- Radial Displacement at El 6.80 m at free field



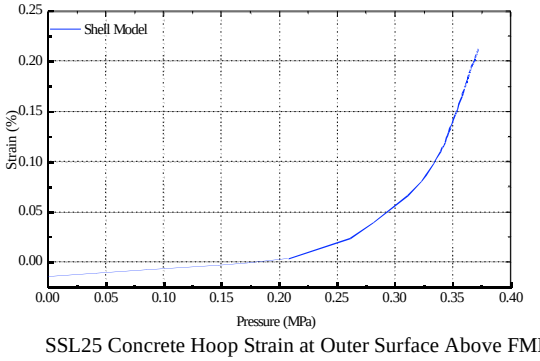
SSL3- Radial Displacement at El 10.35 m at free field



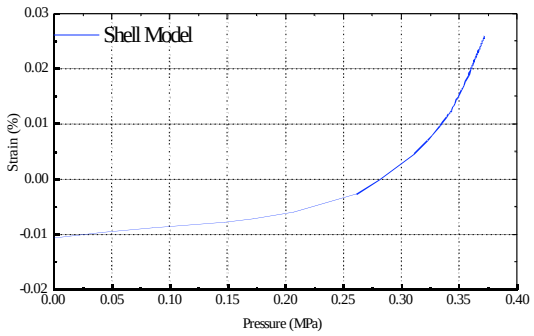
SSL4- Vertical Displacement at Top of ring Beam at free field



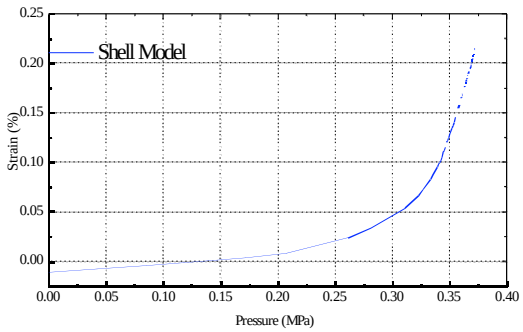
SSL5- Radial Displacement at Dome Crown at free field



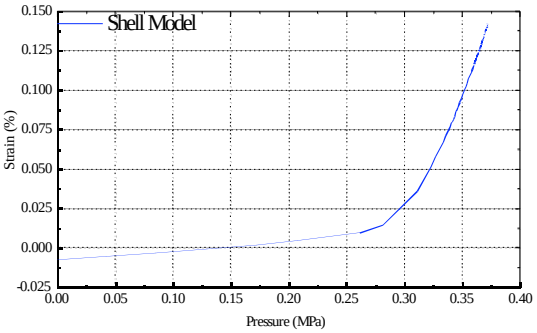
SSL25 Concrete Hoop Strain at Outer Surface Above FMLB



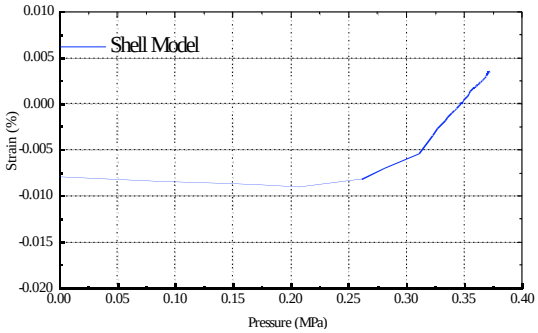
SSL26 Concrete Axial Strain at Outer Surface Above FMLB



SSL41 Concrete Hoop Strain at Outer Surface Above EAL



SSL45 Concrete Hoop Strain at Outer Surface Above MAL



SSL46 Concrete Axial Strain at Outer Surface Above MAL

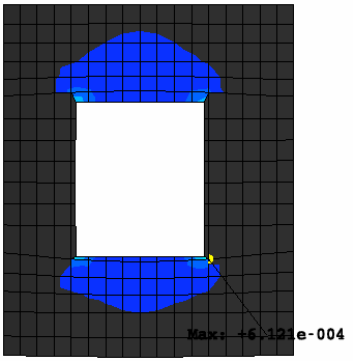
Fig4. Plots at different Standard Sensor Locations

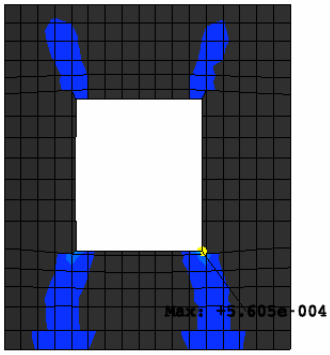
7 IDENTIFIED MILESTONES AND EXPLANATION

- **First appearance of concrete cracking in wall and dome in hoop and longitudinal directions**

Due to hoop stress the cracks in the longitudinal direction appear in IC Wall, near MAL and EAL at 0.185 MPa (1.31Pd).
Due to longitudinal stress the crack in the hoop direction appear in IC Wall near MAL at 0.296 MPa (2.09Pd).
In Dome at Pressure 0.233 MPa (1.649 Pd) the meridonial cracks appear at the location near the dome ring beam joint.

All the above are reported in Fig 5.


Longitudinal Cracks in IC wall


Hoop Crack in IC wall

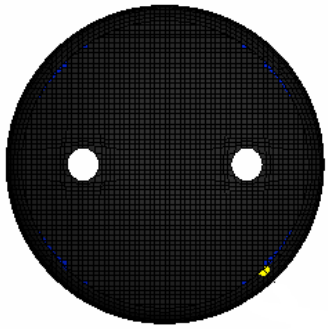

Cracking in Dome

Fig5: Different location of initial cracking

- **First through wall thickness cracking in the wall and dome**

In IC Wall at 0.20 MPa (1.42Pd) near MAL IC wall
In Dome at 0.317 MPa (2.24Pd) near SG opening

- **First appearance of crack at discontinuity regions such as base wall junction and near ring beam**

At base and IC wall surface crack during Prestressing
In dome near the ring beam 0.233 MPa (1.65Pd)

- **First cracking of dome at SG opening, MAL, EAL and FMLB**

SG opening 0.3 MPa (2.12Pd)
MAL and EAL opening 0.185 MPa (1.31Pd)
FMAL opening 0.233 MPa (1.65Pd)

- **First yielding of the reinforcement bars in hoop and longitudinal directions and subsequent strain levels at the various pressure levels**

Inner Hoop reinforcement near MAL at 0.34 MPa (2.4Pd)

- **First significant loss of leak tightness and the criteria used for the same**

At 1.42 Pd in wall and 2.24 Pd in dome

- **Maximum pressure sustained by the model before massive leakage signifying the functional failure of the containment structure**

At 2.60 Pd through thickness cracking in the entire cylindrical wall will develop, which will result in to massive leakage and further pressurization will not be possible. However, the shielding cover will be maintained.

- **Ultimate collapse pressure of the containment test model signifying the structural failure including the best estimate, the lower and upper bounds and the criteria used for arriving at the same.**

The Axisymmetric model predicts the upper bound ultimate pressure of 0.5MPa (3.54Pd), which will never be exceeded. While the 3D Shell model predicts the crack initiation and through thickness cracking with the ultimate pressure of 0.366MPa (2.6Pd) due to local failure around MAL. The analysis with refined 3D model has been undertaken, which will give the better estimate of the failure pressure and is likely to be in between the lower bound of 2.60 Pd and upper bound of 3.54 Pd. Ultimate deformed shapes of containment with these two models are shown in Fig 6(a)-6(b).

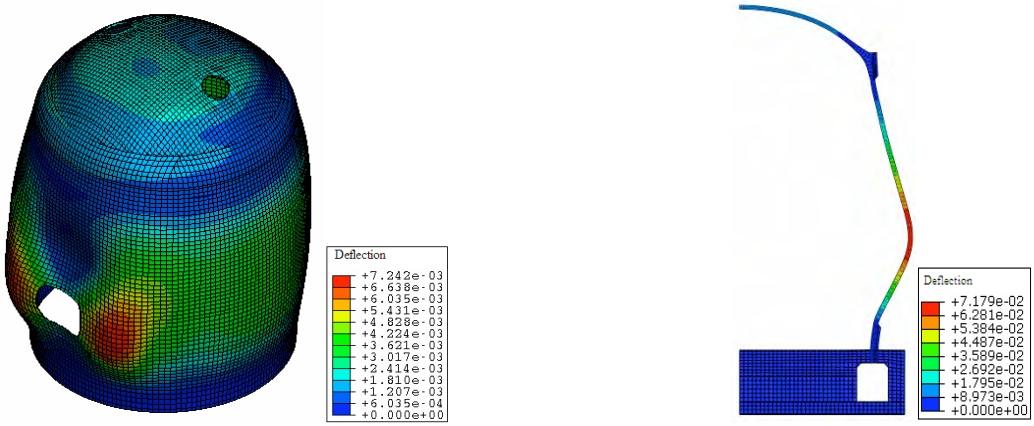


Fig6(a-b): Deformed shape with 3D Shell Model with openings and Axi-symmetric Model

8 DISCUSSION OF RESULTS

The displacement and strain plots obtained from axisymmetric and shell models show good comparison in the free filed (SSL1-4) and reasonable comparison in the discontinuity regions of base wall junction and ring beam However, due to simplified average tendon representation in the dome, the differences in BARCOM response are observed at few locations (SSL5). The comparative statement on the results obtained from the two models is presented in Table 4.

Table 4. Compersion of Axisymmetric and Shell Model (Free Field Region)

MILESTONES	AXISYMMETRIC PRESSURE	SHELL PRESSURE	LOCATION	REMARK
Surface Crack	During Pre-stressing	During Pre-stressing	At the IC wall base	Only cover crack
First appearance of crack with depth above 50%	2.0Pd	2.20Pd	Near the ring beam	Reasonable comparison
First appearance of through thickness crack	2.3Pd	2.42Pd	Near the mid height of IC wall	Reasonable comparison
Dome cracking	3.0Pd	****	At 25 degree angle form apex of dome	Presence of SG opening in 3D shell model results in different cracking pattern
Nonlinear deformation behavior	2.30Pd	2.35Pd	Near the mid height of IC wall	Reasonable comparison
Reinforcement yielding	2.83Pd	****	In Hoop Direction near the mid height of IC wall	Detail 3D model required for yielding simulation around MAL
Pre-stress Cable yielding	3.52Pd	****	In Hoop Direction near the mid height of IC wall	-do-
Ultimate Failure	3.54Pd	2.6Pd	Full IC wall cracked	Lower prediction in shell model due to opening

9 CONCLUSIONS

In the present study two finite element models have been used to predict the ultimate load capacity of BARCOM test model. Our upper bound estimate of failure pressure obtained from the axisymmetric model is 3.54 Pd, which is unlikely to be exceeded. However, this model ignores the influence of local openings and is found to be suitable for free filed response evaluation in an approximate average manner. The 3D shell model with all the major five openings predicts a lower bound failure pressure of 2.60 Pd, which needs to be further studied with detail modelling around the MAL and additional fracture mechanics criteria.

REFERENCES

1) Singh R K. 2009. Pre-Test Report on International Round Robin Analysis of BARC Containment (BARCOM) Test Model. Jan 28, 2009. ISBN: 97.8-81-8372-046-5.

2) BARC/X Plan/5.06A1/Round Robin Analysis/ Model Document/version r0 and DRG. NO. BARCOM - NPCIL-01 to BARCOM - NPCIL-36

3) "ULCA" BARC in-house analysis code for containment structural analysis

4) Singh Tarvinder, Singh R K and Ghosh A K, "Axisymmetric Global Structural Analysis of BARC Prestressed Concrete Containment Model for Beyond Design Pressure", BARC External Report, BARC/2008/E/020, October 2008.

5) Crisfield, M. A., "Non-linear Finite Element Analysis of Solids and Structures" VOLUME 1 / Volume 2

6) Kupfer, H.B., Gerstle, K.K., 1973. Behaviour of concrete under biaxial stresses. ASCE J. Eng. Mech.Div. 99 (EM4).

7) Basha S M, Singh R K, Patnaik R, Ramanujam S, Kushwaha H S, Venkat Raj V, " Predictions of Ultimate Load Capacity for Pre-Stressed Concrete Containment Vessel Model with BARC Finite Element Code ULCA", *Annals of Nuclear Energy*, pp 437-471, 30, 2003.