NONLINEAR FINITE ELEMENT ANALYSIS TO PREDICT ULTIMATE PRESSURE CAPACITY OF A NUCLEAR POWER PLANT CONTAINMENT STRUCTURE

I. Tavakkoli[1], M.R. Kianoush[2], H. Abrishami[3], X. Han[3]

1) PhD Candidate, Civil Engineering Department, Ryerson University, Toronto, Canada
2) Professor, Civil Engineering Department, Ryerson University, Toronto, Canada
3) Candu Energy Inc., Mississauga, Canada

Abstract: Demand is increased in the nuclear power industry to investigate the response of containment structures to much higher internal pressures that could be encountered during a severe accident. The primary objective of this article is to calculate the Ultimate Pressure Capacity (UPC) of a nuclear power plant containment structure. In this study, for the containment structure, the most detailed model is obtained by a full 3D model. Nonlinear finite element method is employed to predict the response of the structure. For this purpose, finite element program ANSYS is used. The major challenges for modelling prestressing tendon element are to accommodate parameters affecting prestressing forces. To introduce the prestressing forces, initial strains are applied to the discrete tendon elements. This approach significantly increases the complexity of the problem; however, represent the most realistic modeling of the prestressing system at the dome of the containment structure. This method is more appropriate in comparison to the equivalent force approach in the previous studies. Both methods are discussed and the response of structure using different finite element approaches and the results of UPC are compared. It is observed that containment structure (CS) with subject of this study meets the design requirements of the current standards and behaves linearly up to 1.5 design basis pressure. The response of the structure using the discrete model for the prestressing tendon and the equivalent prestressing force approach vary by 14 percent.

1 INTRODUCTION

The containment system forms a continuous, pressure retaining envelope around the reactor core and the heat transport system. The containment structure protects the public and environment from all potential internal events, and is designed to withstand tornadoes, hurricanes, earthquakes and aircraft crashes; and to prevent the release of radioactive material to the environment. The containment structure contains of a cylindrical prestressed wall and a shallow dome. The perimeter wall is prestressed with a set of horizontal tendons and a set of vertical tendons. The shallow dome is prestressed with three sets of prestressing tendon located 120 degrees apart from each other. The entire structure, including concrete internal structures, is supported by a reinforced concrete base slab that ensures a fully enclosed boundary for environmental protection.

Demand is increased in the nuclear power industry to investigate the response of containment structures to much higher internal pressures that could be encountered during a severe accident. One of the design requirements for the nuclear power plant (NPP) containment structure is to have the ultimate pressure capacity (UPC) of at least twice the design pressure, and the structure is predicted to behave elastically to 1.5 times the design pressure (CSA N287.3-14, (2014). In abnormal conditions, such as Loss of Coolant Accident (LOCA), the containment structure is subjected to increasing internal pressure, and the concrete is under tension. Due to low tensile capacity of the concrete, concrete cracks easily in such condition. By increasing the internal pressure, the cracks penetrate through the thickness of the wall in the most weaken area. Then, it is followed by the plastic behavior of the reinforcing steel and prestressing tendons causing the loss of functions and finally the overall failure of the structure.

Finite Element Methods (FEM) for structural analysis has been widely used in nuclear power industry. Numerous generic and specific finite element programs have been developed in the last few decades. Among these programs, ANSYS is recommended as one of the most popular in both academic and commercial applications. ANSYS is widely used in nuclear industry.

The primary objective of this research study is to evaluate the response for the CS to much higher internal pressures that could be encountered during a severe accident. The objectives of this study includes introducing proper model for concrete material that is capable of presenting the mechanical behavior of concrete under compression and under tension as well as the behavior of concrete material after cracking. For the reinforcing steel, stress-strain curve for the material that includes the initial linear response, yielding and post yielding of the material as well as the thermal characteristics are the parameters that should be incorporated in the material model. For pre-stressing tendon system, mechanical behavior under tension and the thermal characteristics of the material should be captured in the finite element material model.

The major challenges for modelling prestressing tendon element are to accommodate parameters affecting prestressing forces. This includes: changes along the tendon profile, changes due to incremental pressure and changes with time. To introduce these critical changes, the discrete elements shall be attached to concrete elements at coinciding nodes and address initial prestressing by equivalent strain. This method is more appropriate in comparison to the equivalent force approach in the previous studies. Using the discrete model, the tendon layout is modelled as close as possible to the designed layout to capture the most accurate results of the prestressing system. This approach significantly increases the complexity of the problem; however, represent the most realistic modeling of the three layers of the prestressing system at the dome of the containment structure. In the current study, the results of the two approaches are compared.

This study is focused on nonlinear finite element analysis of the containment structure using finite element program ANSYS. However, the approaches and finding of this study can be implemented in other prestressed containment structures as well.

2 DEVELOPMENT OF FINITE ELEMENT MODEL

The first step in the finite element modelling of the containment structure is to produce the proper geometry model for the structure. For the containment structure, the most detailed model shall be obtained by a full 3D model. The containment structure has prestressed concrete cylindrical wall which is enclosed with a shallow prestressed dome. The perimeter wall (PW) is prestressed with a set of horizontal tendons and a set of vertical tendons. The thickness of PW in Candu containment structures is in range of 1000 mm to 1600 mm. The cylindrical wall has approximate radius of 22 m and height of 44 m. The shallow dome is prestressed with three sets of prestressing tendon located 120 degrees apart from each other. The entire structure, including concrete internal structures, is supported by a reinforced concrete base slab that ensures a fully enclosed boundary for environmental protection. The CS also contains of a steel liner at inner face of the PW and the dome to provide additional leakage protection. Note that the base mat and the steel liner are not model in the present FEM.

To compare different approach of modeling prestressing effects, two separate models are created. For both of the models, similar material properties, values, element types and dimensions are used to model concrete and reinforcing steel.

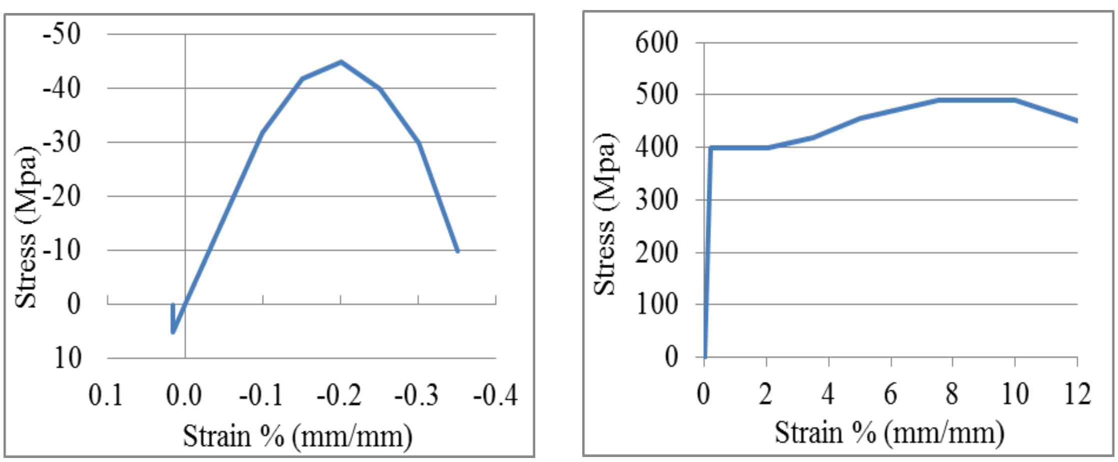
The CONCR model in ANSYS program enables the appropriate elements to account for both cracking and crushing parameters. Initially, concrete is treated as an incrementally linear elastic material. The isotropic properties to for concrete material are defined in the programs which are the modulus of elasticity and passion ratio.

For the CONCR model in ANSYS, the tri-axial failure surface model developed by William (1975) is employed. A total of five input strength parameters (each of which can be temperature dependent) are defined the failure surface as well as an ambient hydrostatic stress state. These strength parameters are the uni-axial tensile cracking stress based on the modulus of rupture of the concrete as well as the uni-axial and bi-axial crushing stress. The uni-axial crushing and the bi-axial crushing stresses are based on the uni-axial unconfined compressive strength and the ultimate bi-axial compressive strength of the concrete. More detailed information can be found in literature by Willam (1975) and ANYSY (2004). To model the plastic behavior and large displacement for concrete, kinematic hardening model with von Mises failure criterion is added.

If failure criterion defined for CONCR material is not satisfied, there is no crack or crushing in the concrete. Otherwise, material crushes if the stress is compression. If the concrete material crushes at an integration point, it is assumed that there is no contribution to stiffness of the element at that point. If the failure criterion is reached for all the planes in a tensile-tensile-tensile state, cracks are developed in the planes perpendicular to σ_1 and σ_2 and σ_3. In the tension-tension-compression state, if the failure criterion reaches for σ_1 and σ_2, cracking occurs in the planes perpendicular to those perpendicular directions.

In ANSYS program, the cracking is determined by the criterion of maximum tensile stress, called "tension cutoff". Additional information that are needed for introducing the Willam-Warnke failure model are; shear transfer coefficients for an open crack and closed crack. It should be noted that, shear strength reverses due to aggregate interlocking and should be accounted for by introducing retaining positive shear strength. The typical values for the shear transfer coefficients are 0 to 1.0, with value of 0 representing a smooth crack with complete loss of shear transfer and value of 1.0 representing a rough crack with no loss of shear transfer. William (1975) and ANYSY (2004).

In current study, a 3D reinforced concrete element with capability of modelling the behaviour of the concrete material is employed. The isotropic Solid-65 3D solid element is an 8-Node brick element that has three degrees of freedom (DOF) at each node, translation in X, Y and Z directions. 3D solid elements are capable of modelling the nonlinear behaviour through thickness of the wall compare to conventional shell modelling. It should be noted that only concrete element Solid65 support the concrete material model. This model is advantageous compared to the shell models employed in previous research studies, and is capable of introducing the contribution of the radial reinforcement. Figure 1(a) and 1(b) show the strain-stress response for concrete material and reinforcing steel material, respectively.



(a) (b)

Figure 1 Stress-strain curve for (a) concrete material, (b) reinforcing steel material

Two techniques are introduced in ANSYS program to models steel reinforcement for reinforced concrete. First is the smeared reinforcement approach, which assumes uniform distribution of reinforcing steel throughout the defined concrete elements. For the smeared reinforcement in solid65 element, the uni-axial stiffness of the reinforcement is added to concrete element. Smeared bars are introduced with strain compatibility with concrete.

The smeared reinforcement are introduced by Real Constant Set for Solid65 elements. Materiel number, volume ratio and orientation angel are the required input for this command. Where, material number refers to the type of the material for steel reinforcement. Volume ratio refers to the ratio the steel volume to concrete element. Moreover, the orientation angel refers to the orientation of the rebar in the smeared model. Each layer of reinforcing bars are assumed to be fully attached to the concrete element, providing displacement compatibility between bars and concrete. As a result, the DOF of the nodes at the rebar layer can easily be expressed in terms of the DOF of the nodes at the external layer through the tri-linear shape functions in the concrete element.

Second is the discrete model, where bar elements are used to model the reinforcement. In this model, bar elements are attached to concrete elements at coinciding nodes. Stiffness of the reinforcing bars is evaluated separately from that of the concrete element. These elements are produced with displacement compatibility between bars and concrete. The drawback of these models is that the concrete mesh is restricted by the location of the reinforcements. Compare to smeared model, discrete method is the most suitable for prestressing tendons. Link8 elements are used to model tendons in the currents study. For this approach, in the original finite element mesh, nodes should be located where the tendons are located. This coinciding nodes allows direct coupling between the concrete and tendon elements. The Link8 spar element is a uni-axial tension-compression element. This element has three degrees of freedom at each node; translation in X, Y and Z directions. For this element, plastic deformation capability is included. To consider the prestressing action an initial strain is added on the bar elements.

To model the plastic behavior and large displacement for reinforcing steel and prestressing tendons, kinematic hardening model with von-Mises failure criterion is added. The isotropic properties to for reinforcing steel and prestressing tendon material are defined in the programs which are the modulus of elasticity and passion ratio.

According to design of this containment structure, the height of the PW is divided into three sections along the height of the wall. The amount of meridional, hoop and transverse reinforcement in these sections vary from each other; however, constant among the height of each section. Also, the reinforcing detail varies at inside face and outside face of the PW. Accordingly, PW is divided into 3 volumes along the height of the wall and 3 volumes through the thickness of the wall. Wall is divided to 6 elements through the thickness and to 142 elements along the height. The amount of reinforcing steel at outer and inner volumes of the wall is calculated according to design of the containment structure. The vertical and meridional reinforcing steel are introduced as smeared reinforcement to the first two concrete elements at inner face and two concrete elements at outer face of the wall. The radial reinforcing steel are added as smeared reinforcement to all the elements for the PW.

The shallow dome of the containment structure is divided into six volumes in thickness to introduce the reinforcement at inner and outer faces of the dome. Meridional, hoop and shear reinforcement are modelled with the same approach as the reinforcing steel for the PW. Dome is divided to 74 elements along the arc of the dome. The smeared meridional and radial reinforcement are added to the first two elements at inner face of the dome and the two elements at outer face of the dome using the calculated reinforcement ratio and steel material properties.

For Model1, the amount of horizontal and vertical prestressing tendons for the wall beam is calculated. These elements are introduced by adding the reinforcement ratio for the smeared reinforcement to mid-section of the perimeter wall. For vertical tendons, the ratio is similar along the perimeter of the wall. For horizontal tendons, the ratio of the prestressing tendons is different along the height of the wall. The prestressing tendons are introduced starting 3000 mm from the base of the wall. To introduce the prestressing tendon for the shallow dome, the ratio of prestressing tendon is calculated and the value is assigned to mid-section of the dome. Schematic finite element of the concrete material for Model1 and Model2 of the CS. Using a wedge of the CS, variations of the RC sets along the height of the PW is also illustrated in this figure.

For Model1, the prestressing effects of wall horizontal tendons are introduced by applying a uniform external pressure acting normal to face of the PW. Moreover for the vertical tendons, forces are applied top of the PW at location of anchors. The effects of dome horizontal prestressing for Model1 is added by applying a uniform external pressure acting normal to face of the dome and tendon end forces at location of anchors. At location of anchorages, steel plates are added to finite element model to prevent premature failure. A 2D solid45 element is used to model the plate. Additional pressure is added to CS at the outer face of the ring beam to introduce the prestressing effects of the horizontal RB tendons. Figure 3 shows the equivalent external pressure and tendon end forces to model the effect of prestressing system in Model1.

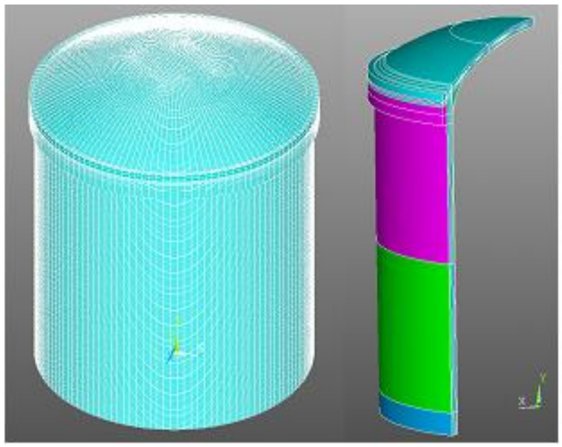


Figure 2- Finite element concrete elements and RC sets for Model1 and Model2

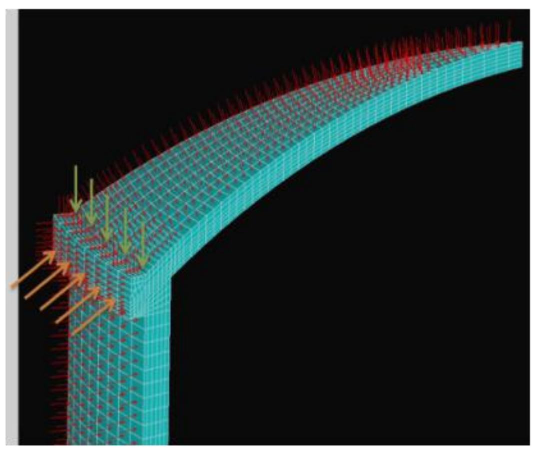
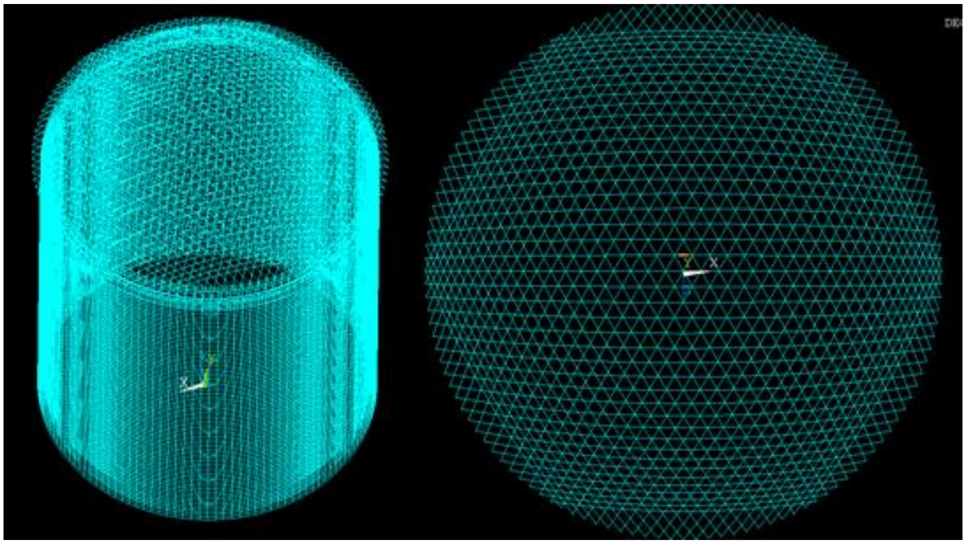


Figure 3- Prestressing equivalent forces for Model1

The concrete elements and real constant sets for reinforcing steel for the PW and the dome in Model2 is similar to Model1. However in Model2, the prestressing tendons in the model are represented by Link8 elements with uni-axial stiffness. The effect of prestressing is added as an initial strain on the elements. The real constant sets for prestressing tendons contains of the strain according to in-service tendon load and the area of the tendons.

It should be noted that to prevent any premature failure due to stress concentration at locations of anchorage, a thick steel plate is added to the finite element model to simulate the anchorage system. A 2D solid45 element is used to model the plate. Table 1 summarizes the Real Constant set assigned representing the smeared reinforcing steel at different location of the PW and the dome for Model2. Figure 4 (a) and (b) show the finite element Model2 for prestressing tendons using the Link8 elements for the perimeter wall, for the dome and for the ring beam tendons. As is shown in this figure, the prestressing system for the shallow dome consists of 3 layers of prestressing tendons that are located 120 degree apart from each other. This system is found to be more appropriate for the shallow dome. The Link elements are coupled with concrete element throughout the dome to assure that the prestressing forces are transferred to the dome structure. It should be noted that both models are fixed to the ground at the base.



(a) (b)

Figure 4- Finite element of prestressing tendons for Model2 (a) 3D model, (b) top view of the dome

Table 1- Real Constant set for Model2

Volume name	Location	RC set
Volume 1,2,3	Mid-Section	Set#1
Volume 1	Inner face	Set#3
	Outer face	Set#4
Volume 2	Inner face	Set#5
	Outer face	Set#6
Volume 3	Inner face	Set#7
	Outer face	Set#8
Dome	Inner face	Set#11
	Outer face	Set#12
	Mid-section	Set#13
PW Vertical tendon		Set#28
PW Horizontal tendon		Set#29
Dome Horizontal tendon		Set#30
Plate at anchorage		Set#32

3 LOADING AND ANALYSIS

The UPC analysis of the containment structure is conducted considering; dead loads, prestressing forces, and internal pressure.

Dead loads contain self-weight of the structure including weight of the cylindrical wall, ring beam and the shallow dome. The dead loads are applied as body forces by introducing proper mass densities to materials. Note that, the load factor of zero is used for live loads during LOCA. For first step of analysis, a static nonlinear finite element analysis is conducted to apply the self-weight of the structure.

For the next step, the prestressing effects are applied for both models. For Model1, the prestressing effects are added in a separate load step by applying the equivalent pressure and tendons end forces. For Model2, the prestressing effects are applied by adding an initial strain to the prestressing tendon elements as a separate load step.

Next step is to add uniform internal pressure acting on the entire internal surface of the structure. The transient nature of the internal pressure is ignored as its period is much longer than that of the structure. A static nonlinear finite element analysis is conducted to estimate ultimate pressure capacity of the CS. Because of small displacement in concrete compare to steel containment, geometry non-linearity or thinning of the wall need not to be accommodated in the model. It should be noted that the full Newton-Raphson method of analysis is used to compute the nonlinear response. The Newton-Raphson method is an iterative process of solving the nonlinear equations. This method updates the stiffness matrix at each iteration. As a result, the load is applied incrementally to the structure up to the failure.

4 RESULTS AND DISCUSSION

With application of prestressing effects on Model1, the CS has a 20mm downward deformation at dome apex and 5mm inward deformation at wall mid-height. Model1 behaves linearly under the explained loading condition. It should be noted that after application of prestressing equivalent force on Model1, the PW has inward deformation, consequently tensile stress forms at outer face of the PW at elevation of zero. Crack forms in this location in meridional direction when the tensile stress in vertical direction, reaches the tensile strength of the concrete. Theses cracks close after application of the internal pressure as the wall start to have outward deformation.

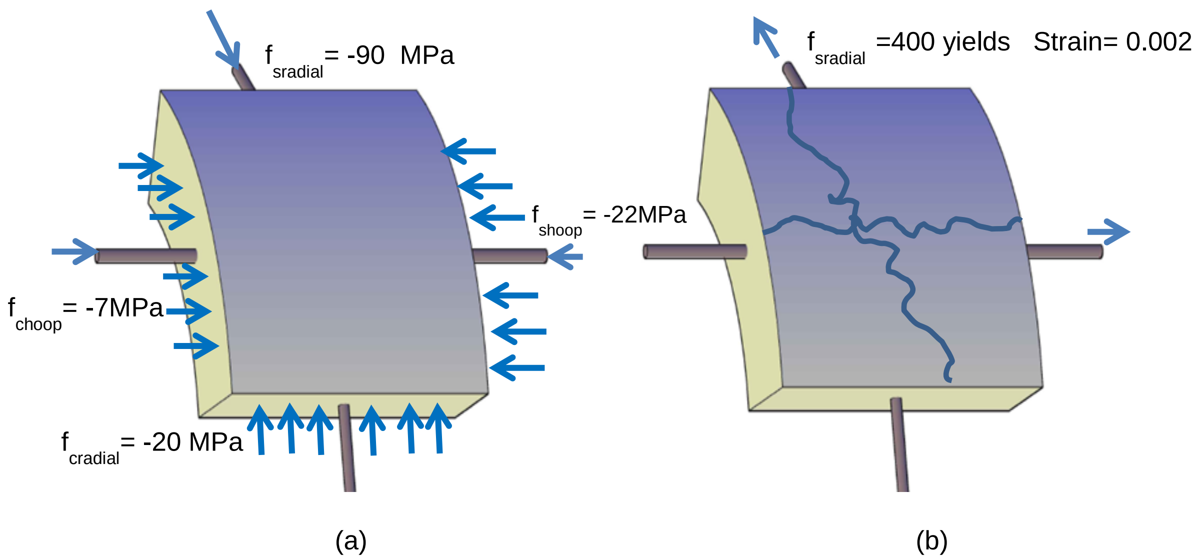


Figure 5- Stress state of a typical element at inner face of the dome for Model1 at internal pressure of: (a) Zero, (b) 875 kPa

Next loading step is to apply the internal pressure. The response of the structure in the form internal pressure against the normal displacement at top of the dome for Model1 is shown Figure 6. It is observed that containment structure (CS) subject of this study meets the design requirement of the currents standards and behaves linearly up to 1.5 design pressure of 400 kPa. The normal displacement at crown of the dome at the end of the linear response for Model1 is 15 mm.

First major crack at the wall outer face below the ring beam, and is due to meridional stresses at internal pressure of 680 kPa. At internal pressure of 780 kPa, cracks from at inner layer of the dome, which are due to radial stress. The response is followed by yielding of radial reinforcement at inner layer of the dome at internal pressure of 875 kPa.

The stress state of a typical element at inner face of the dome at internal pressure 875 kPa dome is shown in Figure 5. Finally, reinforcement yield in both radial and hoop direction throughout dome, which results in excessive deformation at internal pressure of 905 kPa. At internal pressure of 905 kPa, Model1 has outward deformation of 120 mm at crown of the dome. To reduce the computational efforts, the nonlinear analysis is not continued beyond this point.

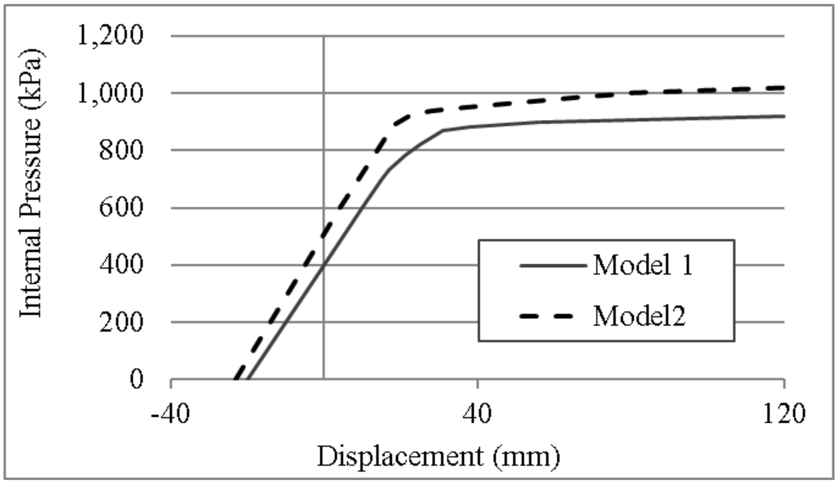


Figure 6- Internal pressure against normal displacement at dome apex

For Model2, the effect of prestressing is added by an initial strain on the prestressing tendons. The response of the structure in the form of the normal displacement at dome crown of the containment structure is shown in Figure 6. It can be observed that after application of all prestressing effects, the dome of the containment structure has a downward deformation of 25mm. With increase of internal pressure, the dome starts to have upward deformation and reaches the value of zero at internal pressure of 510 kPa.

Model2 also behaves linearly under prestressing effect with minor cracks at outer face of the wall at elevation of 0-1000 mm from the base. Those cracks close with application of internal pressure as the wall has outward deformation. The outward deformation of the wall induces compressive stress at outer face of the wall, but during this analysis the compressive stress never reaches the compressive strength of the concrete.

For Model2, there is an 18 mm outward deformation at the apex of the dome at the end of the linear response. Model2 as well behaves linearly beyond 1.5 times the design pressure and meets the design requirements.

For a typical concrete element at mid-height of the dome, the element is at compressive stress of 20 MPa in radial direction after transfer of prestressing effects. With increase of internal pressure, the concrete goes under tensile stress in radial direction and finally cracks at internal pressure of 830 kPa when the tensile stress reaches the value of 5 MPa in radial direction. According to Figure 6, the structure behaves linearly up to internal pressure of 830 kPa. A concrete element located at mid-height of the dome cracks due to radial stress. With formation of this crack, the dome loses its radial stiffness; there is a drastic change in stiffness of the structure. This element is located at inner face of the dome at mid-height of the dome. This element contributes to real constant set number 11.

After formation of this crack and with increase of internal pressure, the structure shows excessive vertical deformation. The radial reinforcement yields at internal pressure of 880 kPa and is followed by yielding of hoop reinforcement at internal pressure of 1030 kPa.

Figure 7shows top view of the prestressing tendons for the shallow dome. For the prestressing tendon located near the dome mid-height as shown in Figure 7, the tendon first reaches the mechanical strain of 0.008 at internal pressure of 890 kPa. Figure 8(a) and (b) show the response of the prestressing tendon element in the form of stress-strain relationship and internal pressure against the strain in the element, respectively.

One can conclude that the nonlinear response of the structure starts after formation of the cracks due to radial stresses and is followed by yielding of radial reinforcing steel. After yielding of the prestressing tendon, the response of the structure is followed by excessive deformation in both radial and vertical directions at internal pressure of 1030 kPa. To reduce the computational efforts, the analysis is not continued beyond this point. For model2, the vertical displacement of 120mm at dome apex happens at internal pressure of 1030kPa. In comparison with Model1, the internal pressure is 14% increased.

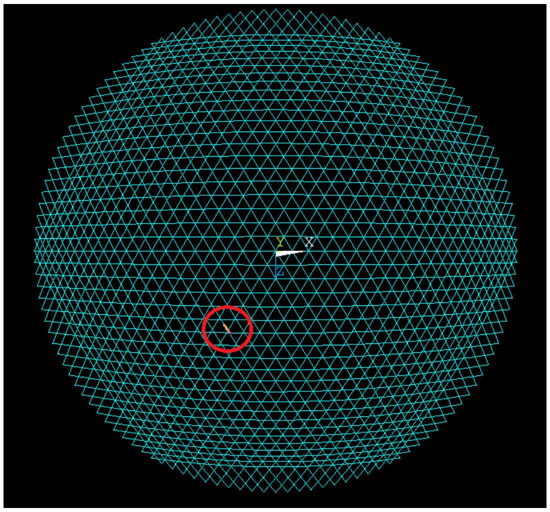
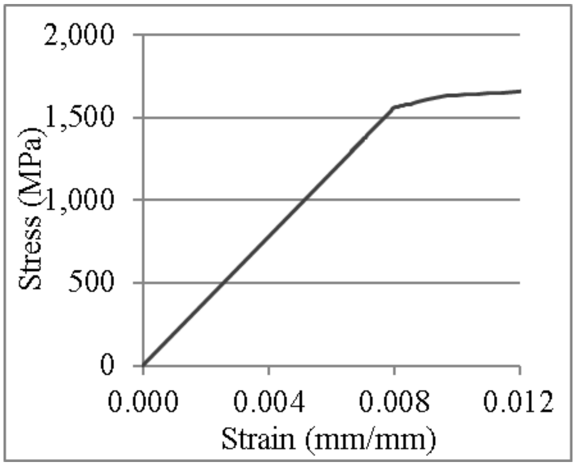
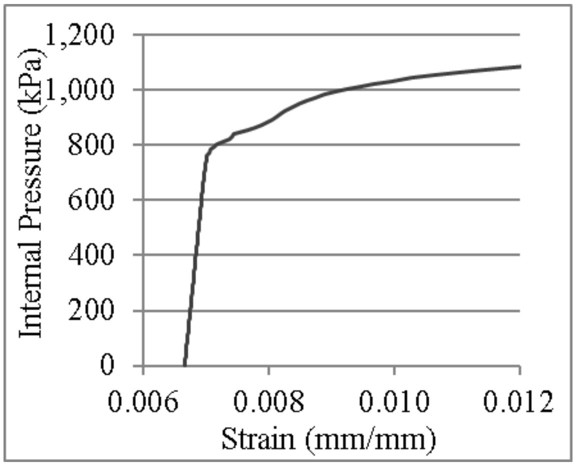


Figure 7- Location of the prestressing tendon that yields first





(a) (b)

Figure 8- Response of a prestressing tendon: (a) Stress-strain relationship for the bar element, (b) Internal pressure against strain in the bar element

5 CONCLUSIONS

Demand is increased in the nuclear power industry to investigate the response of containment structures to during a LOCA. This study is focused on nonlinear finite element analysis of a containment structure using finite element program ANSYS. However, the approaches and finding of this study can be implemented in other prestressed containment structures as well.

In this study, two different models are prepared to compare different approaches to introduce the prestressing effects. For Model1, the reinforcing steel and prestressing tendons are added as smeared reinforcement to concrete elements for both perimeter wall and the shallow dome. The effects of prestressing are added as equivalent pressure and tendon end forces. For Model2, bar elements are used for modelling the prestressing tendons. The major challenges for modelling prestressing tendon element are to accommodate parameters affecting prestressing forces. To introduce these critical changes, the discrete tendons elements are attached to concrete elements at coinciding nodes and address initial prestressing by equivalent strain in Model2. This method is more appropriate in comparison to the equivalent force approach in the previous studies. This approach significantly increases the complexity of the problem; however, represent the most realistic modeling of the three layers of the prestressing system at the dome of the containment structure.

Static non-linear finite element analysis is conduced to estimate the response of the CS to increasing internal pressure. The response of the structure in the form of the internal pressure against the normal displacement at dome apex is compared for the two approaches. It is concluded that using both models, the CS meets the design requirements for concrete containment structures and behaves linearly up to 1.5 the design pressure of 400 kPa. For Model2, the linear limit is 20 percent higher in comparison to Model1.

It is concluded that the discrete model represent a more realistic model of the prestressing system. Using discrete model, the changes in the prestressing forces due to long-term and short-term losses can be added according to geometry of the tendons. The value of the pressure capacity is increased by 14 percent in comparison with the Model1.

6 ACKNOWLEDGMENT

This project was initiated as a joint effort between Ryerson University and Candu Energy Inc. Authors would like to thank Candu Energy Inc. and Mitacs Accelerate PhD program for their financial support throughout this research study.

REFERENCES

ANSYS Inc. (2004). ANSYS help manual (Version 11.1), Global headquarters, Southpointe, 275 Technology drive, Canonsburg, PA.

CSA N287.3-14 (2014): Design Requirements for Concrete Containment Structures for CANDU Nuclear Power Plants: Revision (2009), Canadian Standard Association, Toronto, ON.

Candu Energy Inc. (2012). Technical report summary, CANDU Energy Inc., Mississauga, ON.

Hessheimer, M. F., and Dameron, R. A. (2006). Containment Integrity Research at Sandia National Laboratories. Division of Fuel, Engineering & Radiological Research, Office of Nuclear Regulatory Research, US Nuclear Regulatory Commission.

Radulescu, J. P., Pradolin, L., & Mamet, J. C. (1997). Ultimate pressure capacity of CANDU 6 containment structures.

Willam, K. J., and Warnke, E. P. (1975). Constitutive model for the triaxial behavior of concrete. In Proceedings, International Association for Bridge and Structural Engineering (Vol. 19, p. 174). ISMES, Bergamo, Italy.