UN-GROUTED TENDON MODELING IN PRESTRESSED CONCRETE CONTAINMENT VESSEL

J. Kähkönen[1], M. Vuorinen[1], P. Varpasuo[1]
[1]Fortum Ltd, POB 100, FI-00048 FORTUM, FINLAND
E-mail of corresponding author: jukka.kahkonen@fortum.com

ABSTRACT

A standard problem exercise (SPE) has been organized on the performance of containment vessels under severe accident conditions. The containment model in this SPE is the Sandia/NRC/NUPEC 1:4-Scale Pre-stressed Concrete Containment Vessel model (PCCV). The PCCV failure test was carried out in September 2000 at Sandia, USA. This paper describes the modeling method adopted to model the PCCV. The tendons in the PCCV were un-grouted. It was concluded that the usual way of tendon modeling, i.e. grouted truss element assumption, was not adequate for the problem at hand. To model the tendon slip an in-house Fortran -based preprocessor was developed. The preprocessor utilizes Abaqus FE -code connector elements. The connector element poses kinematic constraints to truss (tendon) element node so that the truss node can move only in tangential direction. The connector elements allow also friction definition. A small model of a horizontal tendon - concrete interface was first analyzed. Based on this small model results we concluded that the connector elements performed well. The tendon force distribution, including the seating losses, could be calculated accurately. The connector element modeling approach was applied to a large PCCV model. In the large model concrete containment was modeled with shell elements and the truss (tendon) elements were connected to the concrete with more 50 000 connector elements. The main results achieved in this analysis were: The ultimate capacity of PCCV was calculated accurately and the tendon force distributions could be estimated adequately.

INTRODUCTION

The standard problem exercise on the performance of containment vessels under severe accident conditions [1] studies an over-pressurization test of a pre-stressed concrete containment vessel [2]. The first phase of the exercise focuses on the local effects of the containment. Within phase 1 three models of the containment are examined. The local tendon behavior is studied with Model 1. The model 1 is presented in Figure 1. Model 2 is a local model of the equipment hatch that focuses on the steel-concrete interface and on the liner tears. Model 3 is a global model of the containment. With model 3 the focus is on the global response of the containment and on the liner tears. The model 3 is presented in Figure 2.

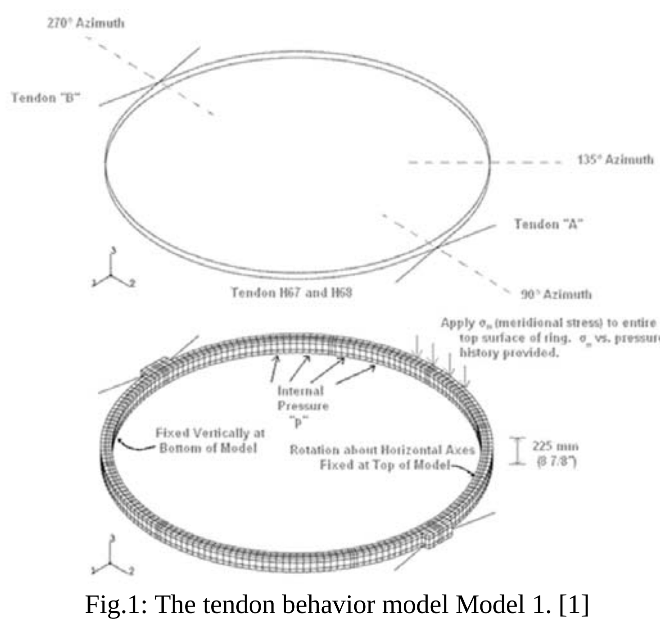

Fig.1: The tendon behavior model Model 1. [1]

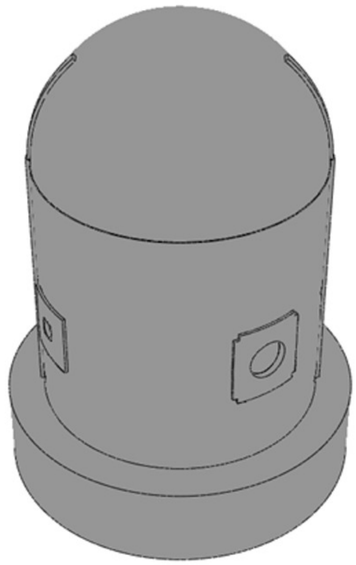

Fig.2: Global model.

Fortum has participated in the first phase of the exercise with simulations of models 1 and 3. The local behavior near the E/H is studied with a sub-model based on the global model 3. In this report the analyses of the models 1 and 3 are presented. In addition, some examinations on the liner fracture near the equipment hatch are presented.

TENDON BEHAVIOR MODEL

Structural model and loads

Model 1 is studied with Abaqus 6.10. Implicit dynamic code is used. The finite element model consists of 978 nodes and 1118 elements. Concrete is modeled with 420 shell elements (S4R). The concrete element thickness is 325 mm. In the azimuths 90 o and 270 o the shell thickness is 650 mm. Reinforcement and vertical tendons are modeled as rebar layers in the concrete section. The liner is modeled with 420 shell elements (S4R) that share the same nodes as the concrete elements. The liner elements are offset to the inner surface of the concrete elements. The liner element thickness is 1.6 mm. The horizontal ten-dons are modeled with 138 beam elements (T3D2). The connection between the concrete and the tendons is modeled with 140 connecter elements (CONN3D2). Bottom nodes of the concrete ring are constrained in vertical direction. The rotations around the horizontal axes are constrained in the upper nodes of the ring. The tendon-concrete interaction is modeled with slot connectors. The slots connect tendon nodes to the nearest concrete node. The slots allow tangential movement of the tendon nodes. The allowable movement direction is modeled with local coordinates. Friction value of 0.21 is used in the slot. Reinforcement and vertical tendons are modeled as rebar layers in the concrete elements. The pre-stressing of the vertical tendons is not modeled. The pre-stressing load is given as a connector displacement in the connectors at the end nodes of the tendons. The internal pressure is modeled as a pressure load on the liner shell elements. The meridional stress is given as point loads on the upper surface of the containment ring. The relation between the meridional stress and the internal pressure is calculated with the same method as presented in [3]. Concrete is modeled with the concrete damage plasticity model found in Abaqus. The material modeling for rebars and tendons follows von Mises plasticity model. For liner steel the fracture strain of 0.18 is used.

Results of ring model to study tendon behavior

Model behavior at the pressure milestones given in [1] is presented in Table 1.

Table 1: Pressure milestones

	Applied pressure
Concrete Hoop Stress (at 135^{o} azimuth) Equals Zero	1.4 x Pd (0.55 MPa)
Concrete Hoop Cracking Occurs (at 135^{o} azimuth)	1.9 x Pd (0.74 MPa)

Tendon A Reaches 1% Strain (at 135 ^{o} azimuth)	3.3 x Pd (1.29 MPa)
Tendon B Reaches 1% Strain (at 135 ^{o} azimuth)	3.3 x Pd (1.29 MPa)
Tendon A Reaches 2% Strain (at 135 ^{o} azimuth)	3.4 x Pd (1.33 MPa)
Tendon B Reaches 2% Strain (at 135 ^{o} azimuth)	3.4 x Pd (1.33 MPa)

The deformed shape of the ring at the failure pressure is presented in Figure 3. From Figure 3 it can be seen that at pressure 3.5 x Pd the ring has broken. In the test the final rupture occurred at azimuth 6^{o} with an effective pressure of 3.63 x Pd

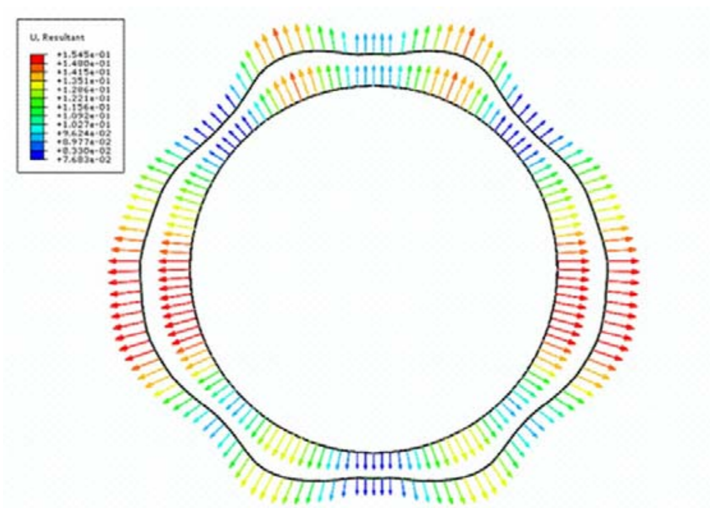

Fig.3: Deformation (x10) at applied pressure P = 3.5 x Pd

The tendon stress as the pressure increases is presented in Figure 4. The presented values are for tendon A. Tendon B behaves similarly. From the Figure 4 it can be seen that the model captures the tendon stress dependency on the internal pressure. As the pressure increases, the tendon stress rises up to the ultimate capacity of the tendon.

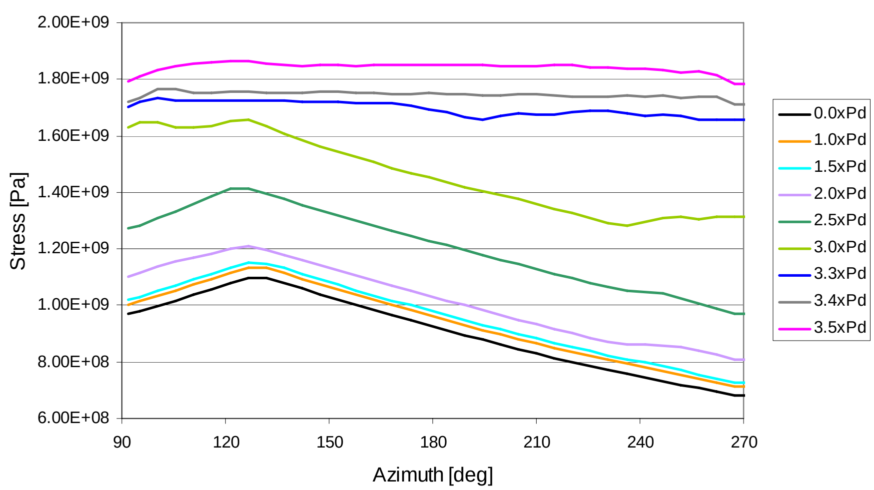

Fig.4: Stress distribution of tendon A as the pressure increases.

GLOBAL MODEL

The finite element model consists of 175603 nodes and 257580 elements. The concrete part of the containment is modeled with 22153 shell elements (S4R and S3R). The concrete element thickness is 325 mm at the cylinder part of the containment and 275 mm at the dome part of the containment. In the azimuths 90 o and 270 o the shell thickness is 650 mm at the cylinder part of the containment and 550 mm at the dome part of the containment. Near the equipment hatch (E/H) and the air lock (A/L) the element thickness is 550 mm. Reinforcements are modeled as rebar layers in the concrete section. The liner is modeled with 22153 shell elements (S4R and S3R) that share the same nodes as the concrete elements. The liner elements are offset to the inner surface of the concrete elements. The liner element thickness is 1.6 mm. The tendons are modeled with beam elements (T3D2). The horizontal tendons consist of 19370 elements, and the vertical tendons are modeled with 23280 elements. The connections between the concrete and the tendons are modeled with connecter elements (CONN3D2). The base of the containment is modeled with 42028 solid elements (C3D8R). The base is modeled as 3.5 m thick. The patches of the penetrations E/H and A/L are modeled with 52 1 mm thick shell elements (S3R). The FE-model is presented in Figure 5.

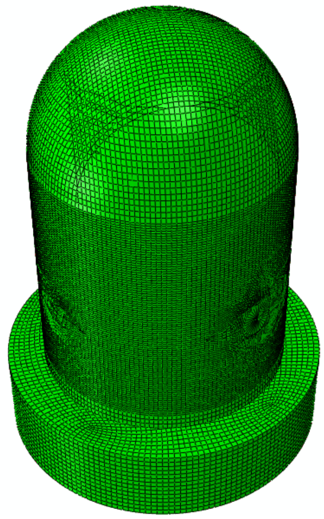

Fig.5: The global finite element model. Model 3

Applied load

The bottom nodes of the concrete base are constrained. The tendon-concrete interaction is modeled with slot connectors. The material modeling and the slot connectors are similar to the tendon behavior model. The load is applied to the model in four steps. In the first step, the gravity load is applied. Second, the pre-stressing load is added. The pre-stress is given as a connector load in the connectors at the end nodes of the tendons. For the horizontal tendons the load is 432 kN, and for the vertical tendons the load is 483 kN. In the third step, the anchorage loss is modeled by lowering the connector loads by 25 %. In the fourth step, the internal pressure load is applied. The internal pres-sure is modeled as a pressure load on the liner shell elements. The pressure is increased linearly over time (0.039 MN / s). The model is analyzed up to four times the design pressure Pd = 0.39 MN. Before the internal pressure is applied, the end nodes of the tendons are locked to the concrete shell elements.

RESULTS

In the following, the samples of response output from the standard output locations are presented. The standard output locations are presented in [2]. Figure 6 presents the measured and simulated displacement histories of the standard output location 4. The simulation results are in accordance with the test results.

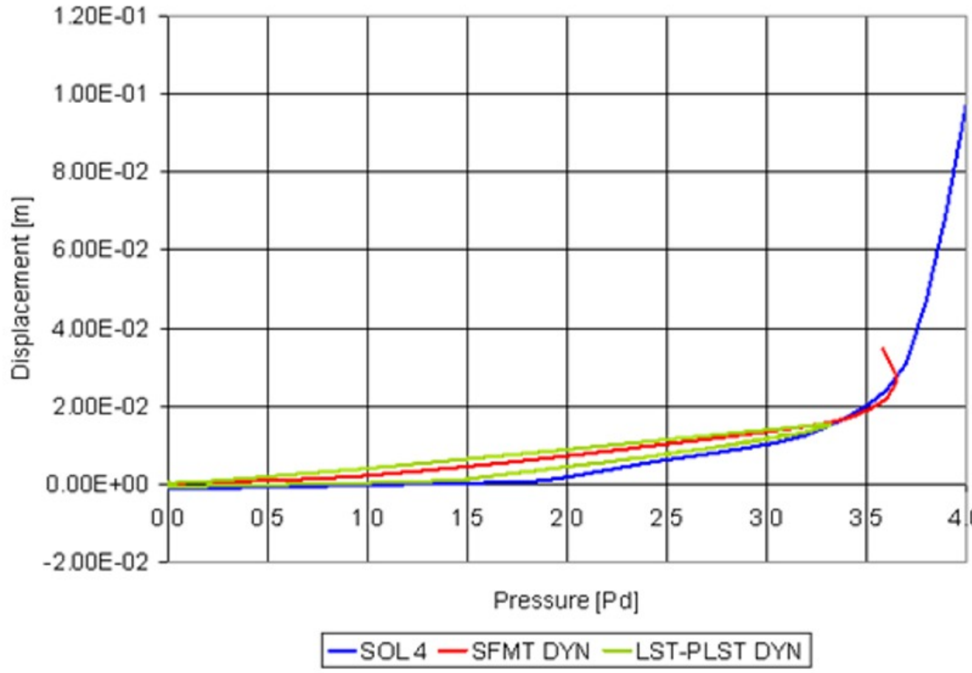

Fig.6: Radial displacement at Standard output location 4

Figure 7 present the measured and simulated tendon strains at the standard output locations 48 to 53. Figures 80 and 81 present the measured and simulated tendon forces at the standard output locations 54 and 55. The simula-tion results appear to be in accordance with the test results.

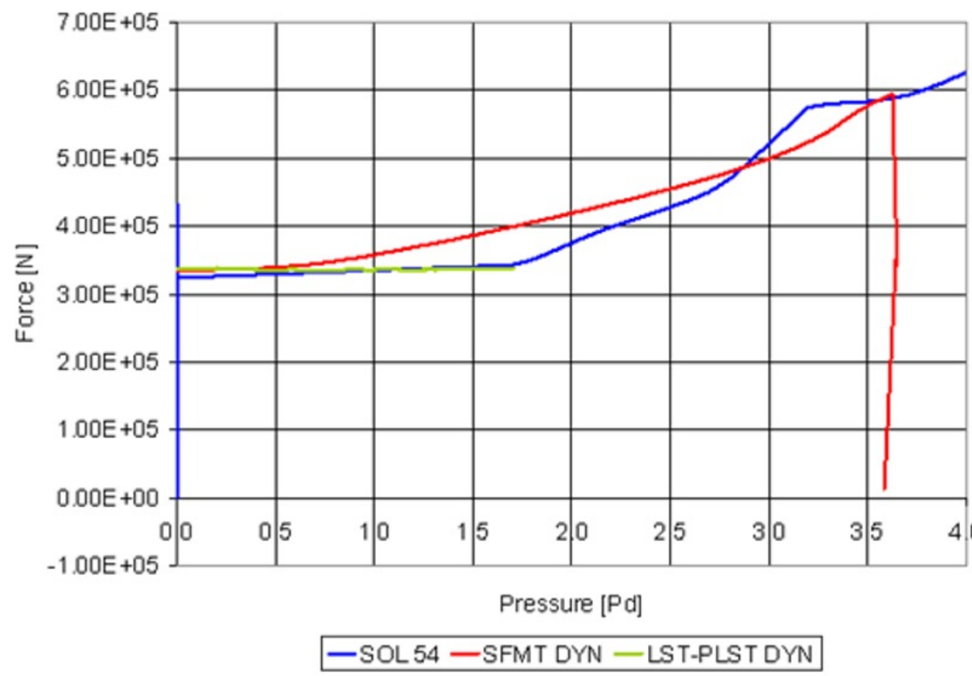

Fig.7: Tendon force at Standard output location 55

The containment deformation at failure is presented in Figure 8. From Figure 8 it is seen that rupture occurs between the two main openings equipment hatch and personnel air lock, which is in accordance with the test results.

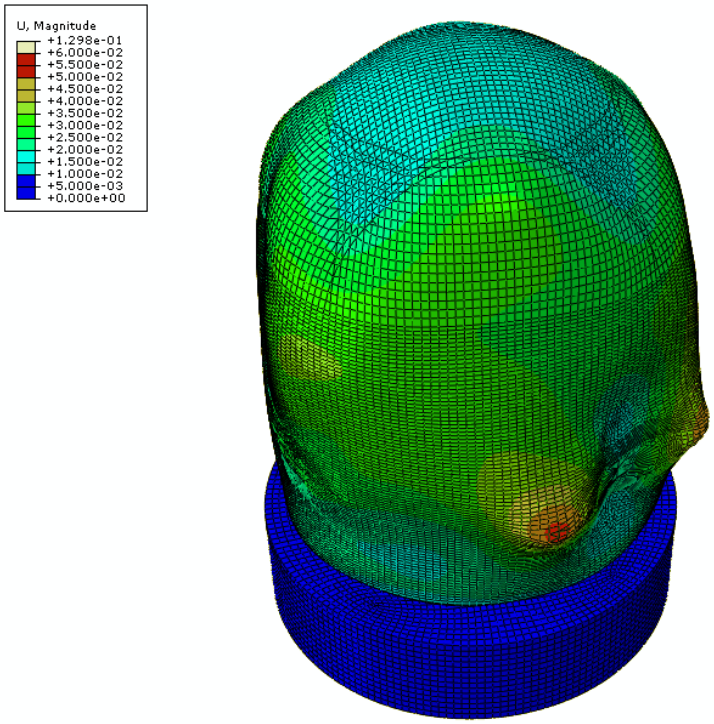

Fig.8: Global displacements (x 50) at applied pressure P = 3.5 x Pd

The displacement results show good agreement with the measurements. Calculated tendon forces are also satisfactory. The calculated strains, however, do not correspond to the measured values in all cases. This may be due to the used material models, the modelling of the rebar layers or the modelling of the liner-concrete interface. In addition, during the test there have been some difficulties with the strain measurements.

CONCLUSIONS

The Fortum participation to the first phase of the standard problem exercise on the performance of containment vessels under severe accident conditions is presented. The exercise consisted of three models of which model 1 and model 3 are presented. The used modeling techniques are presented. The major results are given and compared to the test data.

In the Fortum participation, finite element method and shell element meshes are used. The tendons are modeled with slot connectors, and the liner is modeled as a skin layer on the concrete elements. The modeling of the tendons with slot connectors gives satisfactory results. The modeling of the liner, however, has not been as satisfactory.

The tendon behavior, the displacements of the global model and the model behavior at rupture are in accordance with the test results. The strain results are not as satisfactory, but the orders of magnitude are reasonable.

REFERENCES

[1] Standard problem exercise on performance of containment vessel under severe accident conditions. Final Analysis Definition, White Paper to Define Analyses. 2010.

[2] NUREG/CR-6810. Over-pressurization Test of a 1:4-Scale Pre-stressed Concrete Containment Vessel Model. 2003.

[3] P. Anderson, Standard problem exercise on performance of containment vessel under severe accident conditions (SPE), Meridional stress for Tendon behavior model (model 1), Scanscot Technology, 2010.