

Failure of a Prestressed Concrete Containment Vessel

Summary

Finite element modeling of prestressed concrete containment vessels (PCCVs, Ref. 1) for nuclear power plants poses special challenges. PCCVs, which are heavily reinforced structures, are designed to deform beyond the cracking limits of the concrete. Abaqus has been used extensively for analyzing such structures in the nuclear utility industry (Ref. 2) and can be used to assess and improve the performance of these and other similar reinforced concrete structures.

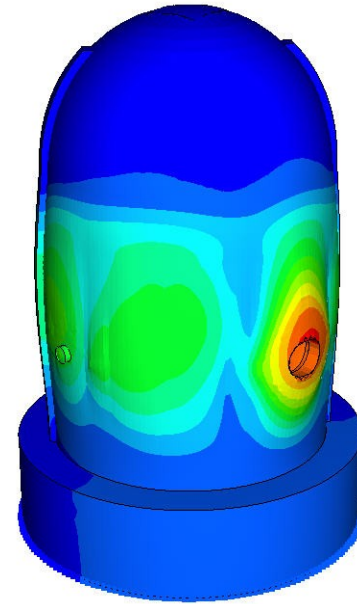
Background

Nuclear power plant containment structures are used for safeguarding against accidental release of radiation from nuclear reactors. These containment structures are composed of heavily reinforced and prestressed concrete. Accurate modeling of such structures is essential to ensure that they remain safe and reliable if an accident occurs. To validate the analytical models and finite element codes that are used to predict how much internal pressure is tolerated by such structures, a 1:4 scale model of an actual PCCV was constructed and loaded to failure at the U. S. Department of Energy's Sandia National Laboratories (Ref. 1).

Two tests were performed on the scale model: a Limit State Test (LST) and a Structural Failure Mode Test (SFMT). In both tests the structure was loaded beyond its design pressure. The LST was carried out to determine the internal pressure at which cracks begin to develop in the concrete leading to leakage of gases from inside the PCCV. The SFMT was carried out to determine the internal pressure at which complete structural collapse occurs. The structural response of the model was measured during both tests, and information on the failure mechanisms was collected. These data were used as the benchmark in a round robin test between design engineers for testing their material models and finite element analysis codes.

Dimensions of the PCCV scale model are shown in Figure 3. The structure is cylindrical, with a hemispherical dome. The overall height is 16.4 meters, and the radius of the cylinder is 5.3 meters. The cylinder's wall thickness is 0.32 meters, reducing to 0.275 meters in the dome after a short transition region. The bottom of the cylinder is joined to a base that is 3.5 meters thick.

The concrete has passive reinforcement as well as prestressed tendons in the hoop and longitudinal (vertical



Key Abaqus Features and Benefits

- Constitutive models for accurate modeling of the behavior of concrete structures, including damage due to cracking and associated loss of stiffness.
- The ability to model rebar accurately, including prestress effects. Abaqus also provides the option of specifying "prestress hold" to enable modeling of post-tensioned prestressed concrete structures.

in the cylinder, meridian in the dome) directions. The passive reinforcement is of mild steel, whereas the prestressed tendons use high-strength steel. The interior of the PCCV is lined with mild steel, 1.6 mm thick. The design pressure for the PCCV is 0.39 MPa.

Finite element modeling of a PCCV poses special analysis challenges. A PCCV is a heavily reinforced and prestressed concrete structure and is designed primarily for preventing any gas leakage from the inside of the PCCV to the outside environment. In some situations the PCCV also may be subjected to a much higher internal pressure leading to a full structural collapse wherein the concrete may deform well beyond its cracking limit. Abaqus has been used extensively for analyzing such structures in the nuclear utility industry; a representative example can be found in Ref. 2.

Finite Element Analysis Approach

As part of the pre- and post-test analyses of the containment vessel, a finite element model of the prototype PCCV was constructed by NNC Limited, UK (details of this work can be found in Ref. 3). In this technology brief the NNC model has been upgraded to take advantage of more recent Abaqus capabilities and now uses the concrete damaged plasticity model.

Abaqus provides two different constitutive models for the analysis of concrete. The smeared crack concrete model is capable of modeling compressive crushing as well as tensile cracking and postcracking behavior. The concrete damaged plasticity model provides for the degradation of the elastic stiffness induced by plastic straining, both in tension and compression.



Figure 1: 1:4 scale PCCV prototype (Ref. 1).

The concrete material is modeled using first-order, reduced-integration, continuum brick elements (Figure 4). The prestressed tendons in the hoop direction are modeled with prestressed rebar. The vertical and meridional tendons are modeled with prestressed trusses.

The liner of the PCCV is thin; hence, it was modeled with membrane elements.

The PCCV model consists of approximately 150,000 elements and 0.5 million variables and is loaded by an internal pressure of 2.0 MPa.



Figure 2: 1:4 scale PCCV prototype at the instant of structural failure (Ref. 1).

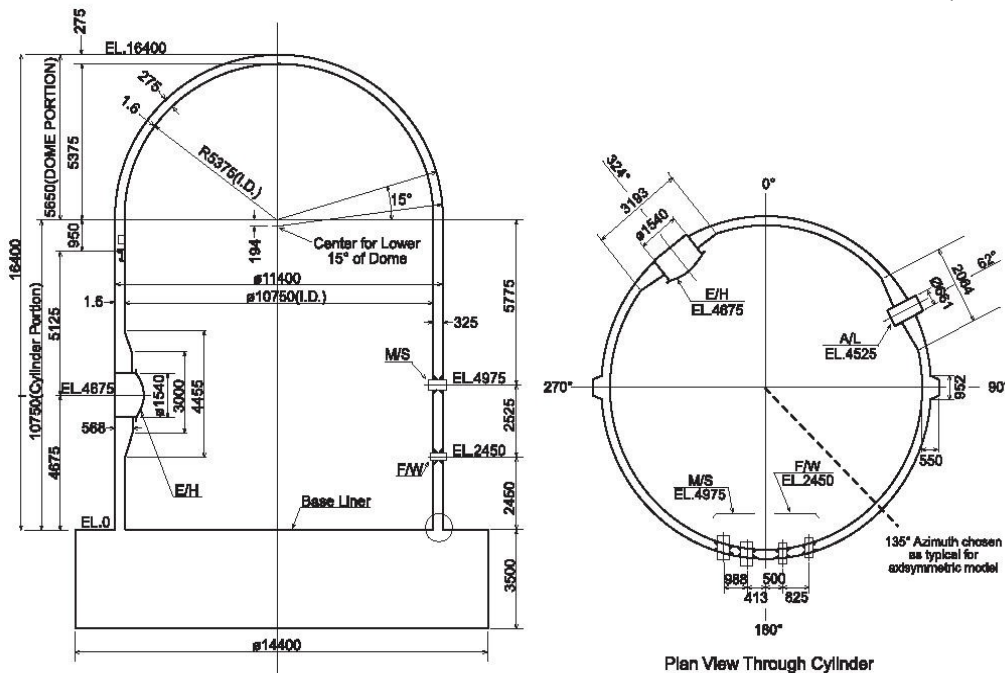


Figure 3: 1:4 scale PCCV model geometry (dimensions in mm), from Ref. 1.

Results and Conclusions

Abaqus analysis predicts the following:

Some of the rebar in the PCCV begin yielding at a load level of about 0.88 MPa. Some cracking in the concrete also takes place at this load. At a load level of about 0.997 MPa, rebar yielding and associated concrete cracking are extensive for the rebar in the hoop direction at the base of the PCCV buttresses at azimuths 270° and 90°. Azimuths and locations of the buttresses are shown in Figure 3. Results from the central cylinder of the model are shown in Figure 5 and Figure 6. Figure 5 shows a plot of output variable AC YIELD, which is a scalar quantity denoting the onset of yielding. A value of 0 indicates that the rebar material has not yielded, and a value of 1 indicates that the rebar material has yielded. Figure 6 shows a contour plot of output variable DAMAGET, which is a scalar degradation measure used to express the reduced tensile elastic modulus of concrete after it has sustained cracking damage. The regions where the rebar has yielded, as shown in Figure 5, also exhibit associated concrete cracking. In the LST it was observed experimentally that functional failure of the PCCV occurred due to liner tearing in a region containing repaired weld seams. This leakage associated with the liner tearing was first detected at a pressure of about 2.4 to 2.5 times the design pressure of 0.39 MPa; that is, at about 0.96 MPa. A subsequent increase in pressure during the LST resulted in further tearing at many strain concentration locations and further leakage (Ref. 1).

Structural collapse of the PCCV prototype occurs at an internal pressure of about 1.4 MPa, as inferred from the increase in the rate of change of displacements and the small time increments needed to run the analysis further beyond this load level. The experimentally measured internal pressure value for structural collapse is about 1.424 MPa (Ref. 1).

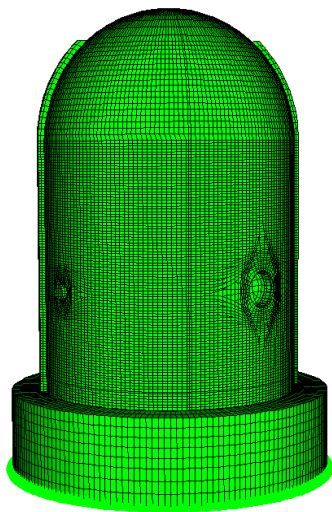


Figure 4: Undeformed finite element mesh of the PCCV prototype.

Contour plots of the displacement magnitude and the tensile damage near the complete collapse load of the PCCV are shown in Figure 7 and Figure 8, respectively. Figure 8 shows a contour of output variable DAMAGET. In Figure 9 and Figure 10 the load-displacement curves at two locations as predicted by Abaqus are compared with the experimental results.

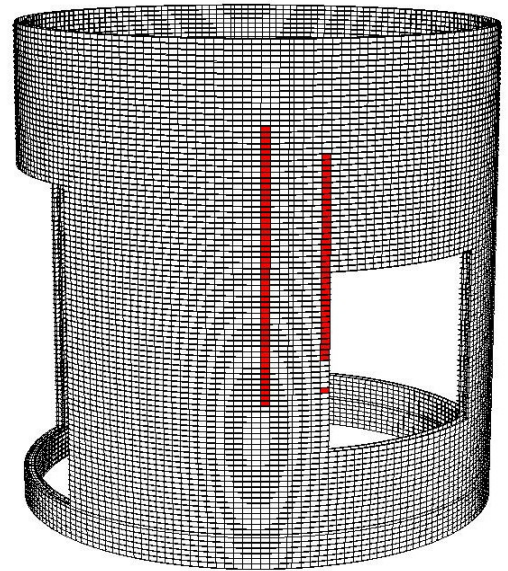
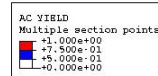


Figure 5: Contour plot of output variable AC YIELD at an internal pressure of 0.997 MPa.

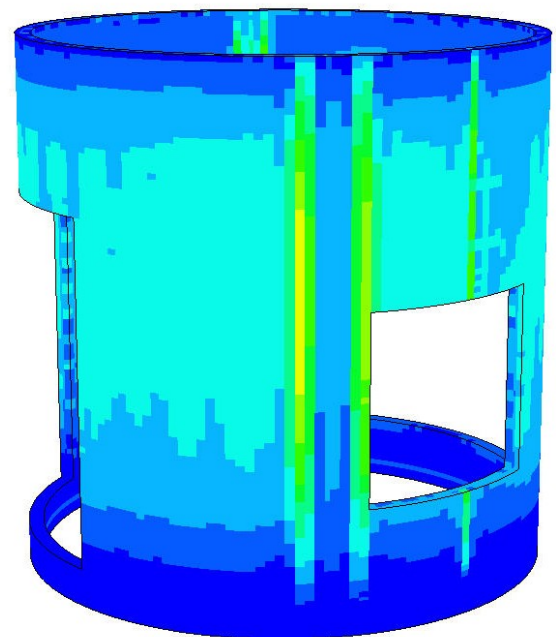
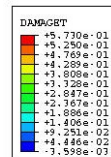


Figure 6: Contour plot of tensile damage in concrete at an internal pressure of 0.997 MPa.

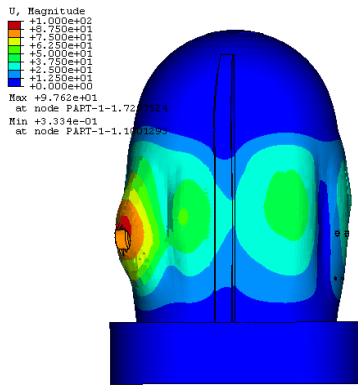


Figure 7: Deformed finite element mesh of the PCCV prototype at a load level of 1.52 MPa. Deformations have been magnified 8'.

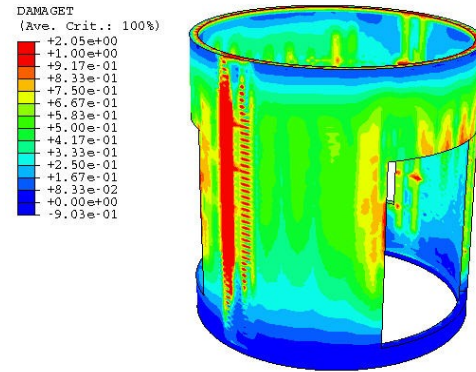


Figure 8: Contour plot of tensile damage in the central cylinder of the model PCCV at a load level of 1.52 MPa. Red contours show higher damage

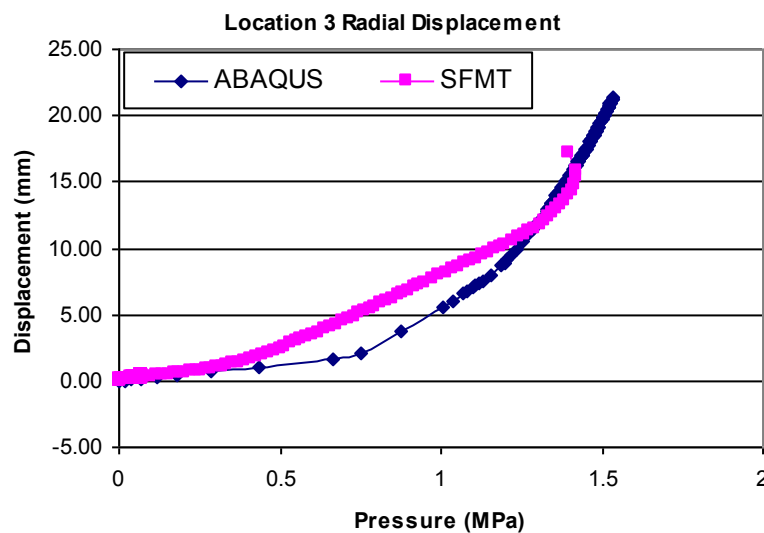


Figure 9: Values of radial displacement at Location 3 computed using Abaqus and measured experimentally in the SFMT.

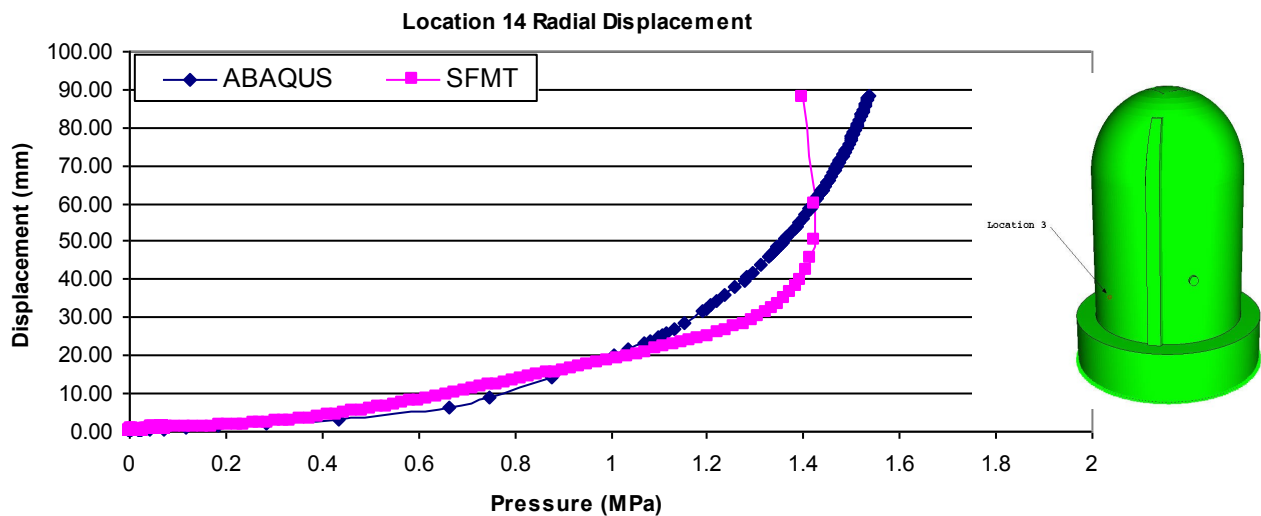


Figure 10: Values of radial displacement at Location 14 computed using Abaqus and measured experimentally in the SFMT.

Acknowledgements

Dassault Systèmes SIMULIA Corp. would like to acknowledge Dr. Nawal K. Prinja, Technical Manager, NNC Limited, UK, for providing NNC's original finite element model of the PCCV prototype.

References

1. Dameron, R. A., B. E. Hansen, D. R. Parker, Y. R. Rashid, M. Hessheimer, and J. F. Costello, *Posttest Analysis of the NUPEC/NRC 1:4 Scale Prestressed Concrete Containment Vessel Model*, NUREG/CR-6809, U. S. Nuclear Regulatory Commission; SAND2003-0839P, Sandia National Laboratories; ANA-01-0330, ANATECH Corporation.
2. Tang, H. T., "Abaqus Applications in Light Water Reactor Safety Analysis," Abaqus Users' Conference Proceedings, 1988.
3. Prinja, N. K., and D. Shepherd, "Numerical Simulation of Limit Load Testing of a 1/4 Scale Pre-Stressed Concrete Containment Vessel, Pressure Equipment Technology: Theory and Practice," Professional Engineering Publishing, Institute of Mechanical Engineers, London, 2003.
4. Prinja, N. K., and D. Shepherd, "Simulating Structural Collapse of a PWR Containment," Proceedings of the 17th International Conference on Structural Mechanics in Reactor Technology, Prague, Czech Republic, Paper H04-6, Aug 17–22, 2003.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
 - "Defining reinforcement," Section 2.2.3
 - "Concrete damaged plasticity," Section 22.6.3
- Example Problems Manual
 - "Seismic analysis of a concrete gravity dam," Section 2.1.15

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

