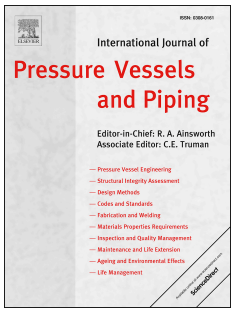
Accepted Manuscript



Finite element modelling of a nuclear containment structure subjected to high internal pressure

I. Tavakkoli, PhD Candidate, Structural Engineering Analyst, M.R. Kianoush, Professor, H. Abrishami, Lead Containment Structure Design, X. Han, Director

PII: S0308-0161(16)30301-5

DOI: 10.1016/j.ijpvp.2017.05.004

Reference: IPVP 3613

To appear in: *International Journal of Pressure Vessels and Piping*

Received Date: 29 August 2016

Accepted Date: 8 May 2017

Please cite this article as: Tavakkoli I, Kianoush MR, Abrishami H, Han X, Finite element modelling of a nuclear containment structure subjected to high internal pressure, *International Journal of Pressure Vessels and Piping* (2017), doi: 10.1016/j.ijpvp.2017.05.004.

International Journal of Pressure Vessels and Piping

Finite Element Modelling of a Nuclear Containment Structure Subjected to High Internal Pressure

I. Tavakkoli [1, 2], M.R. Kianoush [3], H. Abrishami [4], X. Han [5]

1) PhD Candidate, Civil Engineering Department, Ryerson University, Toronto, Canada
2) Structural Engineering Analyst, Civil/Design/Layout, SNC-Lavalin Inc., Mississauga, Canada
3) Professor, Civil Engineering Department, Ryerson University, Toronto, Canada
4) Lead Containment Structure Design, Civil/Design/Layout, SNC-Lavalin Inc., Mississauga, Canada
5) Director, Civil and Project Engineering, SNC-Lavalin Inc., Mississauga, Canada

Abstract: The objective of this article is to investigate the response of a nuclear power plant containment structure to a much higher internal pressure that could be encountered during a severe accident. To predict the nonlinear response of the prestressed containment structure under internal pressure, finite element program ANSYS is employed. The major challenges for modelling prestressing tendon element are to accommodate parameters affecting prestressing forces including prestressing losses. In this study, the prestressing tendons are modelled using discrete elements. This method is more appropriate approach to model the prestressing tendons when they are not located in a symmetrical fashion. The effects of prestressing system are added directly as initial strain on the prestressing tendon in the finite element model. The proposed approach enables capability of the finite element model to update the prestressing forces, accommodating the non-uniform losses in prestressing tendon forces. It is observed that the containment structure subject of this study meets the design requirement of the current standards. The containment structure behaves linearly up to 1.5 design pressure and the ultimate pressure capacity is estimated as 2.7 of the design pressure.

Keywords: Nuclear power plant, containment structure, finite element analysis, prestressed structure, ultimate pressure capacity

1 Introduction

The containment system forms a continuous, pressure retaining envelope around the reactor core and the heat transport system. The containment structure (CS) is designed to protect the public and environment from all potential internal events. In addition, it is designed to withstand all the external loads such as; 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 (PW) and a 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. The entire structure is supported by a thick reinforced concrete base slab that ensures a fully enclosed boundary for environmental protection.

There is a demand 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 at least 1.5 times the design pressure [1]. Under abnormal