Vibration Analysis of the Crystal River Unit 3 Reactor Containment Structure

November 17, 2009
Rev. 2

Summary:

The analysis presented here is part of the root cause investigation by Performance Improvement International of the Reactor Building Containment Wall failure during the Steam Generator Replacement (SGR).

This analysis considers the entire Reactor Building and does not include the detail modeling that may predict local stress concentrations. Further, the wall at the SGR opening location is assumed to be intact in this analysis.

The vibration of the Reactor Building due to the following loads are modeled and analyzed:

- Hydro Cutting of the SGR opening
- Shock Load of Cutting Post-Tensioned Wires

The result of this analysis is:

1. The resonant frequencies of the structure are:
   a. 4.43 Hz – Swaying of Reactor Building
   b. 6.42, 7.48 & 8.37 Hz – Vibration frequencies of Wall Panels between buttresses
2. The modal analysis compares favorably to physical testing (7.4 Hz calculated vs. 7.3 Hz measured).
3. The vibration induced by Hydro Cutting for the SGR opening produces displacements less than 3.5×10^{-4} inch and tensile stresses less than 0.55 psi in the concrete.
4. The shock load of cutting 20 individual Wire Strands of one Hoop Tendon at one time induces vibrations with displacements less than 7×10^{-3} inch and tensile stresses less than 11 psi in the concrete.

P/62

Analysis Objective

The objective of the presented analysis is to study the following two loads for vibrations of the Reactor Containment Structure:

1. Pulsating load of the Hydro Cutting
2. Shock load of cutting of 20 individual Wire Strands of one Hoop Tendon at one time

Modeling Approach and Properties

The geometry of the Crystal River Plant Unit No. 3 Containment structure was modeled based on references 1 though 10. Abaqus version 6.9-1 Finite Element Analysis software was used to model and analyze the structure.

The following components of the structure are included in the model:

1. Concrete base (cylindrical geometry)
2. Concrete wall panels and buttresses (6 panels and 6 buttresses)
3. Concrete dome
4. Interior steel liner (3/8 inch thick wall and roof, 1/4 inch floor)
5. 144 Vertical Tendons (24 ea × 6 Bays) equally spaced around 360 degrees
6. 282 Hoop Tendons (47 ea × 6 Pairs of Bays)

The concrete structure is modeled using the 8-node linear brick elements with incompatible modes, C3D8I, for accurate bending representation. The steel liner is modeled using the 4-node linear shell element S4. The vertical and horizontal tendons are modeled using the 2-node truss element T2D2. The truss elements are embedded in the solid concrete elements with a prescribed initial stress.

The Tendons are made up of 163 individual wire strands, each with a diameter of 7 mm). The Tendons are modeled as being tensioned to 1,400 kip force which results in a prescribed stress of roughly 144,000 lb/in^2.

The Modulus of Elasticity and Mass Density of the concrete is based on Reference 11. The Poisson's Ratio of the concrete is based on Reference 12:

- $E_C = 4.287 \times 10^6$ lb/in^2
- $\nu_C = 0.2$
- $\rho_C = 150$ lb/ft^3

The steel Tendon and Liner properties are taken as:

- $E_S = 30 \times 10^6$ lb/in^2
- $\nu_S = 0.29$
- $\rho_S = 0.282$ lbm/in^3

The bottom of the concrete base is fixed in all translational degrees of freedom.

Finite Element Model Description

The model consists of three types of Finite Elements:

1. Continuum Solid Elements (C3D8I) representing the concrete structure
2. Structural Shell Elements (S4) representing the steel Liner Interior
3. Structural Truss Elements (T2D2) for the representation of the Vertical and Hoop Tendons

Figure 1 depicts the mesh of the concrete structure and the steel liner.

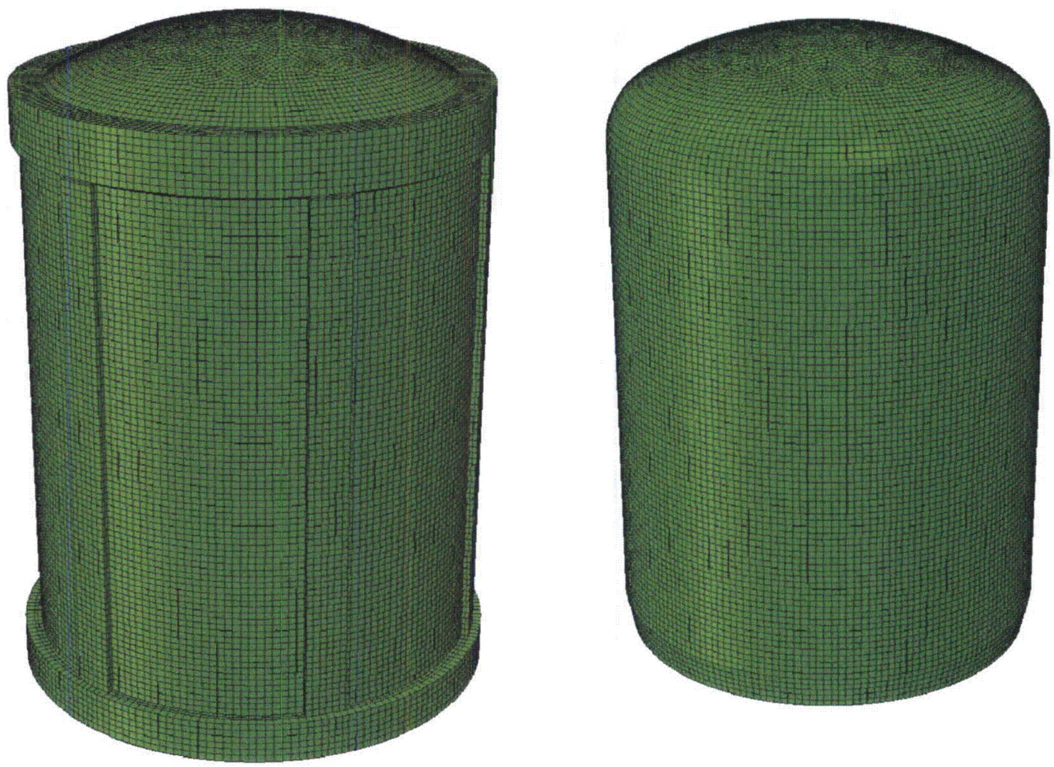

Figure 1: Finite Element Mesh of the Reactor Building. The exterior shown on left and the interior liner mesh shown on the right.

Figure 2 shows the mesh of the Vertical and Hoop Tendons and the location on the solid structure.

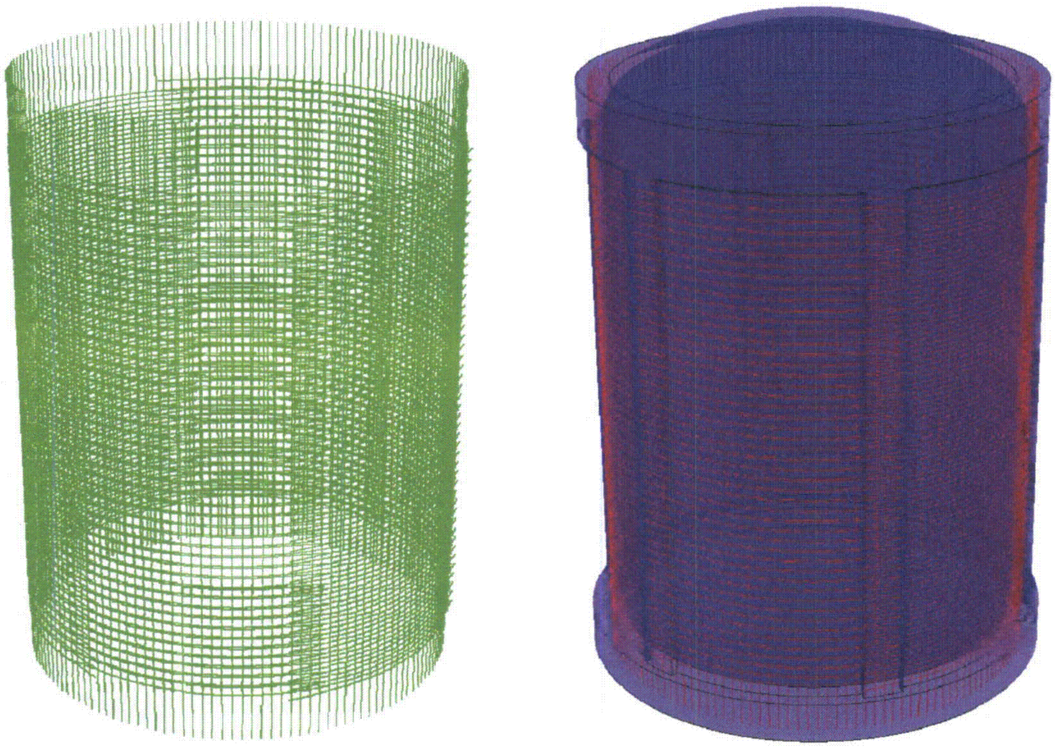

Figure 2: Vertical and Hoop Tendons shown separately on the left and where they are embedded in the concrete solid structure on the right.

Frequency Analysis

The frequency analysis is performed after a static baseline analysis which includes the tensioning of the Tendons and the application of gravity acceleration.

The following vibration modes and associated natural frequencies are found:

- Mode 1 & 2 (4.43 Hz): Swaying of the structure (increasing vibration amplitude with height)
- Mode 3 (6.42 Hz): Radial vibration of the panels of the cylindrical structure. The mode is three panels deflecting in as the other three are deflecting out (every other in and every other out)
- Mode 4 (6.57 Hz): Radial vibration of the cylindrical structure - similar to Mode 3 but rotated 30 degrees such that the buttresses are mostly vibrating instead of the panels
- Mode 5 & 6 (7.48 Hz): Radial vibration of the cylindrical structure with two peaks and two valleys in the circumferential direction
- Mode 7 & 8 (8.37 Hz): Radial vibration of the cylindrical structure with four peaks and four valleys in the circumferential direction
- Mode 9 (8.88 Hz): Twisting of the structure around the vertical centerline.
- Mode 10 & 11 (12.1 Hz): vibration of the cylindrical structure with five peaks and five valleys in the circumferential direction.

Experimental Validation of Frequency Analysis

The natural frequencies of the Containment Structure were measured by Crystal River Unit 3 Personnel (Reference 13). Accelerometers and two types of excitations were employed: (1) hammer impact, and (2) movement of the Polar Crane and lifting equipment during the installation of the Replacement Steam Generator.

The hammer test mostly excites the following frequencies: 7.3 Hz and 15 Hz where the 15 Hz is likely the first harmonic of the 7.3 Hz vibration. However, careful inspection of the spectrum plot of the wall between Buttress 2 and 3 (in Ref. 13) indicates a small peak around 4 Hz. Typically, hammer impact tests may excite more high frequencies than low frequencies. It is likely that the hammer impact test is not exciting the very low frequency associated with the swaying of the structure. In order to excite the tower swaying modes (Mode 1 & 2) a higher amount of energy that a hammer can produce may be needed.

The inspection of the spectrum that was collected during the movement of the crane clearly shows a peak at around 4 Hz. The movement of the crane will likely produce much higher energies and able to excite the bending vibration modes (Mode 1 & 2).

The frequency analysis of the Finite Element Model indicates that the following set of vibration modes exists of the radial vibration of the wall panels (between buttresses):

- Mode 3 (6.42 Hz)
- Mode 5 & 6 (7.48 Hz)
- Mode 7 & 8 (8.37 Hz)
- Mode 10 & 11 (12.1 Hz)

The three closely spaced natural frequencies of the model (6.42, 7.48 and 8.37 Hz) produce the average frequency 7.4 Hz. Depending on what the resolution of the test instrument is and the amount of averaging used, it is likely that the test was unable to resolve the closely spaced frequencies in this range.

It is concluded that the FEA model is likely a good approximation of reality (7.4 Hz calculated vs. 7.3 Hz measured).

Transient Dynamic Analysis of Hydro Demolition

A Modal Dynamic Analysis including the first 50 modes of free vibration along with the natural frequencies and the associated participation factors is utilized to study the response of the Reactor Building due to Hydro Demolition loads. Modal damping equal to 2% of the critical damping for each mode is assumed.

According to Reference 14, the Hydro Demolition equipment has the following characteristics:

- High pressure water is ejected from two heads, each containing three nozzles
- The water is pumped using individual positive displacement pumps operating at 500 rpm (8.33 Hz or 52.36 rad/s)
- The average pressure of the water is 17,000 – 17,500 psi
- The nozzles are 1/8 inch in diameter (0.012272 in^2 opening area)

For this analysis, it is assumed that the pressure pulsation is 25% of the average pressure. It is conservatively assumed that all nozzles are pumping water in phase. The average pressure was converted to a force as follows:

$F_{average}$ = (3 nozzles) × (2 heads) × (17000 psi pressure) × (0.012272 in^2 nozzle opening area) = 1252 lbf

The equation of the time dependent force used in the model is:

(1252 lbf) × (1 + 0.25 × Sin[52.36 *t*]), where *t* is time in seconds.

The force was conservatively concentrated at the most critical point at the midpoint of the bottom SGR opening.

In addition to the abovementioned Hydro Cutting load, the dynamic response to the following loads are also analyzed:

1. Higher pressure (22,000 psi) at 600 rpm
2. Higher pressure (22,000 psi) at 500 rpm
3. Higher pressure (22,000 psi) at 385 rpm (= 6.42 Hz, resonant frequency of Mode 3)

The third additional load listed above is the worst case hypothetical load where the forcing frequency exactly matches the lowest resonant frequency of the wall panels.

Figure 3 shows the maximum Principal Stress of the concrete structure due to load 17,000 psi @ 500 rpm, and 22,000 psi @ 600 rpm.

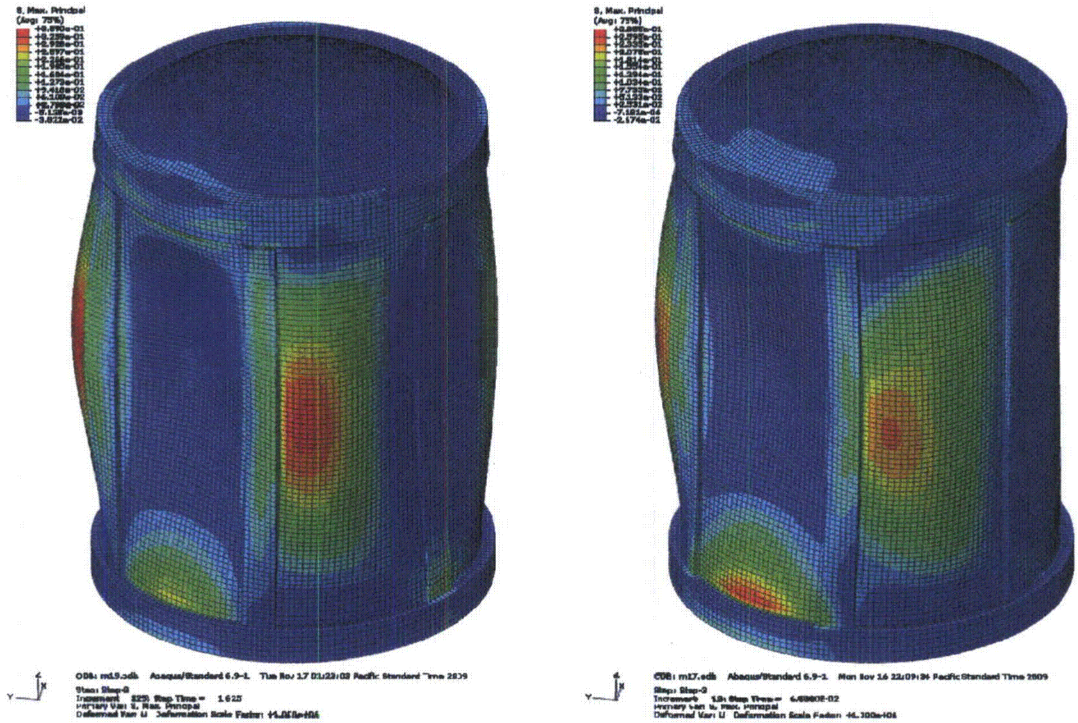

Figure 3: Maximum Principal Stress contours (psi) due to the Hydro Blasting load (17,000 psi @ 500 rpm on the left, and 22,000 psi @ 600 rpm on the right)

Figure 4 shows the maximum displacement of the structure due to the Hydro Blasting loads for the first 2 seconds for loads 17,000 psi @ 500 rpm; 22,000 psi @ 500 rpm; and 22,000 psi @ 600 rpm.

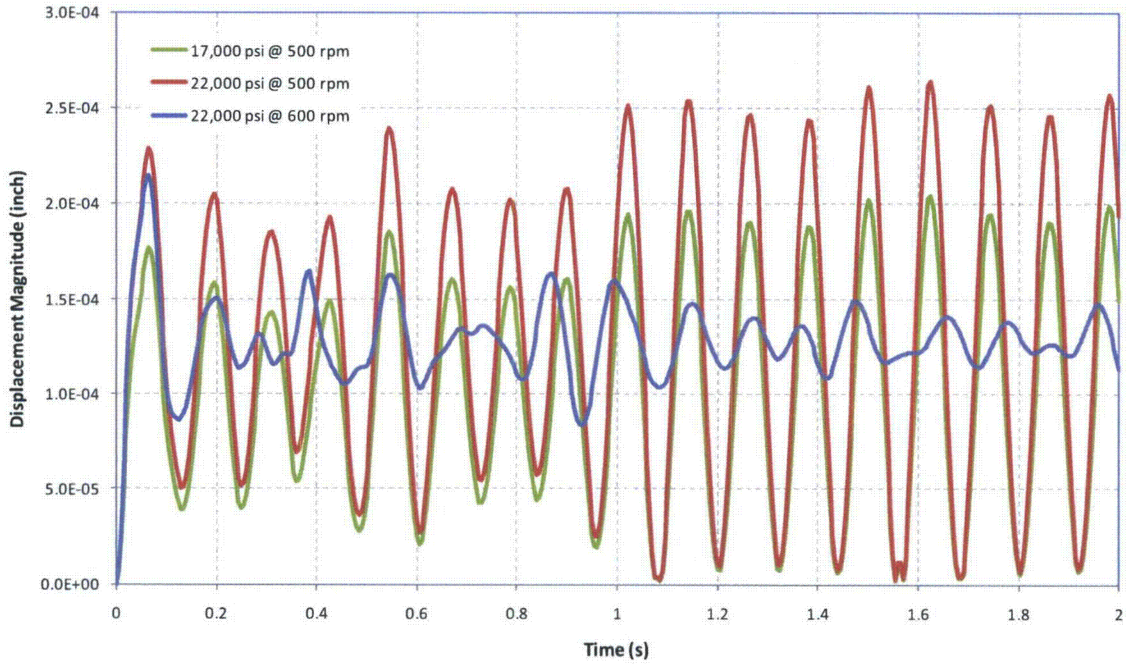

Figure 4: Maximum Displacement Magnitude (inch) of the Reactor Building due to three Hydro Cutting loads for the first two seconds.

Figure 5 shows the dynamic response due to the hypothetical worst case load where the forcing frequency matches the lowest natural frequency of the wall panel. As seen in the figure, the vibration of the wall panel reaches steady state within 4 seconds. After four seconds the amplitude of vibration is not increasing as the damping energy equals the applied energy due to the Hydro Cutting load.

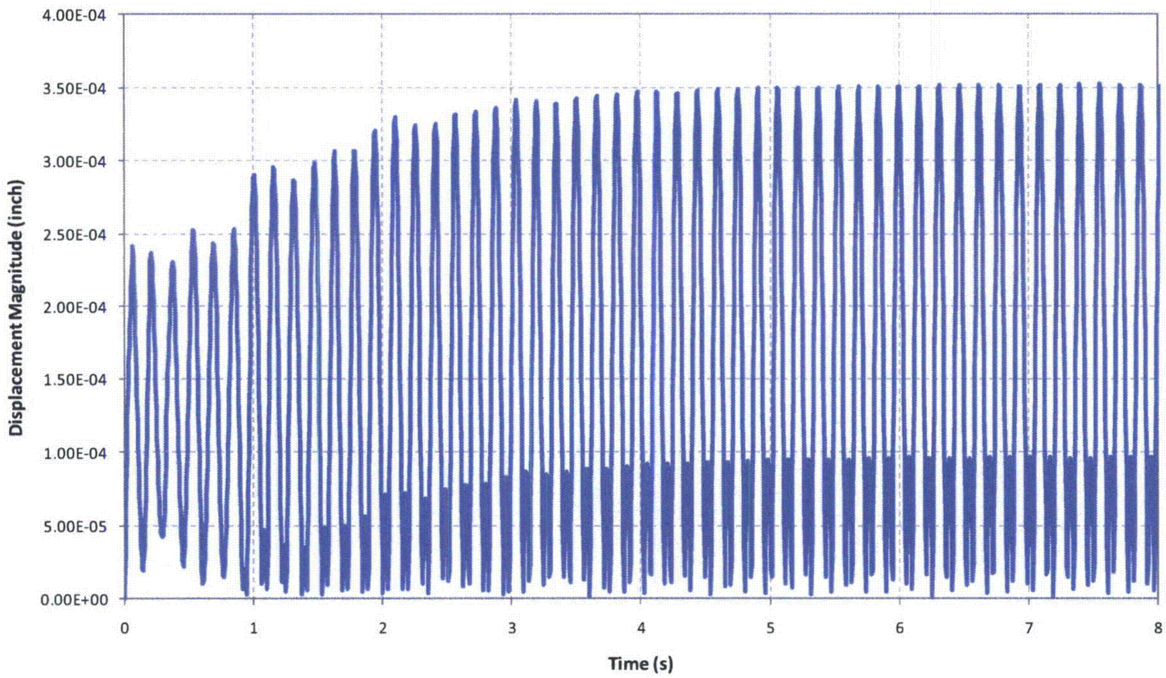

Figure 5: Maximum Displacement Magnitude (inch) of the Reactor Building due to the Hydro Cutting load 22,000 psi @ 385 rpm for the first eight seconds.

The summary of the Maximum Displacement and Maximum Principal Stress due to the Hydro Cutting loads analyzed are listed in Table 1.

Table 1: Displacement and stress results of the concrete structure due to various Hydro Blasting loads.

Hydro Cutting Pressure (psi)	Hydro Cutting Frequency (rpm)	Max Displacement Magnitude (in)	Max Principal Tensile Stress (psi)
17,000	500	2.0×10^{-4}	0.36
22,000	500	2.6×10^{-4}	0.46
22,000	600	2.2×10^{-4}	0.29
22,000	385 (wall resonance)	3.5×10^{-4}	0.55

References

1. Gilbert Associates, Inc., 1970, drawing "Reactor Building Exterior Wall – Concrete Outline"
2. Florida Power, Final Safety Analysis Report, Revision 31.3, Figure 5-2 "Reactor Building Typical Details"
3. Florida Power, Final Safety Analysis Report, Revision 31.3, Figure 5-18 "Base – Cylinder Junction Detail"
4. Florida Power, Final Safety Analysis Report, Revision 31.3, Figure 5-21 "Reactor Building Segments"
5. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 13 Layout"
6. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 42 Layout"
7. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 53 Layout"
8. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 64 Layout"
9. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 51 Layout"
10. Florida Power, 1998, drawing "IWE/IWL Inspection Hoop Tendon 62 Layout"
11. MPR Associates, Inc., 11/3/2009, "Radial Pressure at Hoop Tendons", Calculation No. 0102-0906-0135
12. Progress Energy, 11/8/1985, "Design Basis Document for the Containment", Rev. 6
13. Memorandum from Virgil W. Gunter, CR3 Systems Engineering, 11/14/2009, "Report of CR3 Containment Structure Vibration Monitoring and Impact Testing"
14. Telephone Interview with Dave McNeill (Mac&Mac, Vancouver), 10/28/2009. Documented in "10 28 interview Dave MacNeil.pdf"