ABSTRACT

This paper presents the development of a three-dimensional (3D) Finite Element Model (FEM) of the reactor building Interior Concrete Structure (ICS) with an emphasis on the dynamic modeling requirements for performing seismic analyses subject to High-Frequency (HF) content typical of Central and Eastern United States (CEUS) hard rock sites. The proposed ICS FE model is comprised of different element types (i.e., Solid, Shell, Beam, Mass, and Matrix elements) to appropriately capture both global and local structural responses as well as any dynamic interaction between the ICS and the attached Nuclear Steam Supply System (NSSS) components. Mesh sensitivity studies are presented to illustrate the level of refinement needed to ensure fidelity of responses up to 50 Hz as prescribed by Nuclear Regulatory Commission (NRC) Interim Staff Guidance DC/COL-ISG-01. Comparisons to a conventional Lumped Mass Stick Model (LMSM) are also included to illustrate some of the local responses (such as “breathing” and “skewing” modes) that may not be properly accounted for by such simplified modeling approaches. It is concluded that the proposed ICS FE model is refined enough to sufficiently capture the dynamic responses of the ICS subject to the HF content input ground motion in seismic analyses and for the subsequent use of equipment qualification.

INTRODUCTION

Most of the technical literature on structural analysis and design of nuclear safety-related buildings have been devoted to Containment Buildings since the integrity of these structures is of upmost importance to ensure the safety and protection of the public against the potential risk of radiation release during a postulated nuclear reactor accident, e.g. Mello et al. (2009) & Qadeer et al. (2009). On the other hand, internal structures, or Interior Concrete Structure (ICS) as denoted in this paper, have not received a similar level of attention since the design of their massive concrete sections is usually accomplished by providing nominal steel reinforcement given the generally low level of stresses. However, it is the high stiffness provided by the ICS walls and compartments that requires special attention when performing seismic analysis in the Central Eastern US (CEUS) hard rock sites since the natural frequencies of the rigid structural components may now be tuned to the peak of the High Frequency (HF) ground motion spectra (as shown in Figure 1) with the potential for amplifying the input seismic accelerations.

In the past, Lumped Mass Stick Models (LMSMs) have been intensively used to capture the dynamic behaviors of ICS. Chen and Davenport (1977) proposed a way to calculate appropriate shear coefficients for an ICS LMSM. Saha et al. (1993) modeled a reactor building, ICS, and NSSS components as a LMSM and used to compare different soil conditions. These simplified models have served well the profession since they provide a good representation of the overall or global responses of the structure. Moreover, the frequencies associated with major building modes usually fell within the frequencies of the input ground motion provided in the form of spectral shapes by the U.S. Nuclear Regulatory Commission (NRC) Regulatory Guide 1.60, which normally consider the frequencies content of the input ground motion up to 33 Hz (see Figure 1). However, LMSMs may lack the refinement needed to sufficiently capture local responses and propagate the HF content of the input ground motion at a CEUS hard rock site. Therefore, there is a need to determine the analytical modeling and level of
refinement needed to properly predict both global and local responses of the ICS in the presence of input ground motion with HF content. This paper provides the development of a Finite Element Model (FEM) that serves the aforementioned purposes and presents the testing done on the model to ensure its adequacy for performing seismic analyses and qualification of equipments.

![Image of Ground Motion Response Spectra](image.png)

**Figure 1.** Ground motion response spectra (RG1.60 vs CEUS hard rock site)

**DESCRIPTION OF INTERIOR CONCRETE STRUCTURE**

Located inside the Primary Containment Structure (PCS), the main function of the ICS is to protect vital components from pipe rupture, missiles, and over-pressurization effects, and to provide supports and radiological shielding around the NSSS. As depicted in Figure 2, the ICS considered in this study consists of a primary shield wall surrounding the Reactor Vessel (RV) cavity, and secondary shield walls (also called as D-walls in this paper) surrounding the Steam Generators (SG), Reactor Coolant Pump (RCP), and Pressurizer (PRZ). The D-walls are approximately 88 ft tall with inner radius of 28 ft and thickness of 4 ft. Other structural components include the refueling canal, operating floors, and missile shielding panels.
FINITE ELEMENT MODEL

In this paper, a three-dimensional (3D) ICS FE model has been developed using structural analysis software, ANSYS Mechanical, Release 13.0 (2010). Figure 3 shows the isometric, plan, and elevation views of the proposed ICS FE model including the NSSS components. As shown in Figure 3, most of the concrete walls and floors including shield walls are modeled by using Shell181 element, which is a 4-node shell element having six degrees of freedom at each node and suitable for analyzing thin to moderate-thick shell per ANSYS Mechanical, Release 13.0 (2010). The massive RV cavity wall is modeled using Solid185, which is an 8-node for better accuracy. The NSSS components such as RV, SG, RCP, PRZ, and related connections (pipes) are modeled using Beam188 and Matrix27 elements. The effective masses of the NSSS components and miscellaneous equipment are modeled by using Mass21 element. Note that the FE meshes of the proposed FEM are automatically generated based on a selected typical mesh size. Based on the typical mesh size of 4ft, the proposed ICS FEM model is built with a total number of 7,500 finite elements (i.e., equivalent to 7,800 nodes). Such element size (4ft) is very-refined when required in areas of interest using free meshing in ANSYS and also when being under a high-frequency contents of earthquake inputs. The related mesh sensitivity study is followed in the next section.
EFFECTS OF MESH SIZE ON DYNAMIC ANALYSES AND RESPONSES

Accuracy and Size of Finite Element Model With Respect To Mesh Size

Finite element analysis results are known to be sensitive to the element type, shape, aspect ratio, and mesh size per Cook et al. (1989). Appropriate mesh size of a dynamic finite element models can be determined by several factors. For nuclear facility at a CEUS hard rock site, a structural model should be refined enough to capture appropriate dynamic responses for HF content (at least 50 Hz of the ground motion response spectrum) as per NRC Interim Staff Guidance Document No. 1 (2012). Also, the same FE model may be used for structural design purposes in a single step approach. Some guidelines and recommendations on mesh size selections are available in many references, e.g. ASCE 4 (1998) and Moreschi et al. (2012). As always, required-accuracy and efficiency of the dynamic FE model are key factors that shall be considered in the selection of appropriate mesh size and/or size of dynamic FE models. It is generally known as bigger FE models are (i.e., having many numbers of finite elements) more accurate FEA results are generated. In the following, the selection of an appropriate mesh size is shown as an example.
In order to select appropriate mesh size and to see its relations with accuracy and efficiency, the ICS-only FE model (i.e., without the NSSS components) has been examined by varying its typical mesh size from 2 to 16 ft increased by 2 ft, and taken as a ratio to the thickness of D-wall (4 ft). Note that this typical mesh size is not applicable to the entire finite elements of the ICS FEM since some elements like in the RV cavity walls are limited to a certain dimension, so they cannot be larger than a certain mesh size due to their geometry. The variance of this mesh size is mostly applied to D-walls. Figure 4(a) shows the size of the FEM in terms of the number of the generated finite elements along with their various mesh sizes. The ICS FE model with a mesh size ratio of 0.5 generates about 26,000 finite elements, while the FE model with a ratio of 2 needs only about 3,000 finite elements, which demonstrates that reducing the mesh size exponentially increases the size of FEM, directly related to the computational and post-processing cost. On the other hand, Figure 4(b) shows a relationship between the accuracy of the FEM and the mesh size as an example. Here, the first three fundamental natural frequencies (i.e., selected in the order of larger mass participation factors, MPFs) in the X direction are compared for the various mesh sizes. As expected, some accuracy is lost as the mesh size is increased. Based on these two results, the element size to thickness ratios ranged from 1.0 to 3.0 seems to provide a good compromise between accuracy of global solution and numerical computational effort. Further examination is performed next.

![Image of Figure 4(a)](image1)

![Image of Figure 4(b)](image2)

**Figure 4. Number of finite elements & maximum differences in natural frequencies for the FE models with different mesh size ratios (varies from 0.5–4.0)**

**Mesh Sensitivity on Dynamic Responses of Interior Concrete Structure**

The mesh size effect on the dynamic responses of the ICS FEM is also examined by using three different FE mesh refinements (i.e., very-refine, refine, and coarse) as depicted in Figure 5. Mentioned earlier, some finite elements like in the RV cavity walls and in certain elevations do not grow beyond a certain mesh size due to their inherent/particular geometric constraints. However, most finite elements like in the D-walls and internal walls are modified as per different mesh refinements.
In this paper, dynamic responses of the FE models including different mesh refinements are examined by comparing transfer functions (i.e., amplitude of steady-state response due to frequency-dependent unit dynamic loading) up to 100Hz at different nodal locations. Figure 6 shows transfer functions measured at the top of the D-wall of each FE model (i.e., very-refined, refined, and coarse) along the model X, Y, & Z directions. The results comparison per each direction is summarized as follows:

- **X-Dir:** All three FE models generate the same responses up to 8 Hz and then generate the highest peak amplification at 10/11Hz. However, their peak amplitudes are different per the models. The peak amplifications of both very-refined and refined FE models are calculated as about 10–11, while the coarse FE model generates a peak of 15 (i.e. over-estimating by about 40% than the others). Similarly, both very-refine and refine FE models calculate the same peak amplifications at 20Hz, but the coarse FE model calculates the peak at 22Hz (i.e., shifted to the right by 2Hz compared to the other two FE models). Overall, three FE models generate same/similar responses in the frequency range of 1~8Hz. However, different peak amplitudes and frequencies are distinctly calculated as frequencies are getting higher (particularly, after around 20Hz).

- **Y-Dir:** The highest peak amplification is observed at around 12 Hz for all three FE models. However, the calculated peak amplitudes are different by the FE models. The very-refined FE model generates the largest amplification of 11, and the refined FE model produces 10, a little bit less. Last, the coarse FE model generates the amplification of 8. Three FE models generate the same/similar responses in the frequencies up to 8Hz, but they generate different amplifications and their peak frequencies beyond 20 Hz.

- **Z-Dir:** Unlikely to the responses in X & Y directions, all three FE models generate same responses up to 50Hz. However, different amplifications and peak frequencies are calculated beyond 50Hz, and the differences are getting apparent over the higher frequency range.

Based on the above mesh sensitivity studies, both the very-refined and refined FE models generate the similar/same dynamic responses at least up to 50Hz. Thus, the mesh size ratio ranged from 1 to 2 seems to be appropriate and acceptable for the proposed ICS FEM.
Comparison with Conventional Lumped-Mass Stick Model

This section provides a comparison of the dynamic responses calculated from the proposed ICS FEM and a conventional LMSM. The conventional LMSM is constructed as a single-stick by idealizing the ICS as a system of beams and masses lumped at different elevations, not presented herein. Dynamic responses such as natural frequencies, mode shapes, and In-Structure Response Spectra (ISRS) of the considered ICS are calculated and compared in the following. Note that Mesh size ratio of 1.5 (i.e., average of very-refined and refine) is used in this comparison based on the mesh sensitivity study results.

Natural Frequencies and Mode Shapes

Natural frequencies and mode shapes of the ICS without NSS components have been calculated up to 100Hz by using both the proposed ICS FEM and the conventional LMSM. Tables 4 & 5 summarize the first three fundamental natural frequencies and the associated MPFs calculated from the FEM & the LMSM, respectively. The proposed ICS FEM calculates 12.69, 11.96, & 30.29Hz as the first natural frequencies, respectively in X, Y, & Z directions. The LMSM also generates the same/similar first natural frequencies (i.e., 13.44, 11.96, & 31.00Hz in X, Y, & Z directions). As shown, both FE models
calculate the same first fundamental natural frequencies in all three modeling directions. However, they generate different natural frequencies from the second mode, which may be attributed by the different fidelity of each FE model. It is well-understood that more refined FEM can calculate more local modes, most of which may be localized and have insignificant contributions to the global behavior, so they are considered as noise or ignored based on their MPFs. However, it is observed that some modes prior to 11.96Hz have significant effective mass contributions (larger than 10% MPF). Such modes at 8.02, 8.20, and 10.21Hz should not be considered as local modes anymore. Figure 7 provides the corresponding deform shapes of the D-walls (i.e., referred to as “breathing” mode herein). Similarly, some other local modes (e.g., “skewing” and “warping” deformed) are observed as shown in Figure 8. However, their MPFs are less than 10%. Note that the conventional LMSM cannot capture such “breathing” and “skewing” modes.

Table 4: Natural frequencies and MPFs calculated from the proposed ICS FEM

<table>
<thead>
<tr>
<th></th>
<th>North-South (X)</th>
<th></th>
<th>East-West (Y)</th>
<th></th>
<th>Vertical (Z)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode</td>
<td>MPF (%)</td>
<td>Freq. (Hz)</td>
<td>Mode</td>
<td>MPF (%)</td>
<td>Freq. (Hz)</td>
<td>Mode</td>
</tr>
<tr>
<td>7</td>
<td>36</td>
<td><strong>12.69</strong></td>
<td>6</td>
<td>20</td>
<td><strong>11.96</strong></td>
<td>27</td>
</tr>
<tr>
<td>5</td>
<td>16</td>
<td>10.21</td>
<td>3</td>
<td>17</td>
<td>8.20</td>
<td>26</td>
</tr>
<tr>
<td>25</td>
<td>12</td>
<td>28.65</td>
<td>2</td>
<td>15</td>
<td><strong>8.02</strong></td>
<td>29</td>
</tr>
</tbody>
</table>

Table 5: Natural frequencies and MPFs calculated from the conventional LMSM

<table>
<thead>
<tr>
<th></th>
<th>North-South (X)</th>
<th></th>
<th>East-West (Y)</th>
<th></th>
<th>Vertical (Z)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode</td>
<td>MPF (%)</td>
<td>Freq. (Hz)</td>
<td>Mode</td>
<td>MPF (%)</td>
<td>Freq. (Hz)</td>
<td>Mode</td>
</tr>
<tr>
<td>2</td>
<td>69</td>
<td><strong>13.44</strong></td>
<td>1</td>
<td>78</td>
<td><strong>11.96</strong></td>
<td>4</td>
</tr>
<tr>
<td>5</td>
<td>14</td>
<td>34.68</td>
<td>5</td>
<td>17</td>
<td>31.35</td>
<td>10</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>31.00</td>
<td>8</td>
<td>6</td>
<td>58.20</td>
<td>6</td>
</tr>
</tbody>
</table>

Figure 7. Breathing mode shapes of D-walls with MPFs larger than 10%

Figure 8. Other local modes (including skewing mode)
In-Structure Response Spectra

In-Structure Response Spectra (ISRS) of the proposed ICS FEM are compared with those generated by the LMSM. Fixed-base time history analyses are performed on both models with time history inputs matching the CEUS hard rock site spectra with HF content depicted in Figure 1. The ISRS responses are reported at EL 70 ft of the D-wall (i.e., near the top of the wall) and at the intersection between the internal and D-walls. The results are compared in Figure 9 and summarized as follows:

- **X-Dir:** Both the ICS FEM and the LMSM generate the overall similar responses. However, the LMSM under-estimates the ISRS from 8~10Hz, and it over-estimates the ISRS after 12Hz comparing to the ISRS of the ICS FEM. Such responses have been observed in the mesh sensitivity check (the coarse FEM over-estimates dynamic amplifications). Also the ICS FEM ISRS bumped up at 8~10Hz may be attributed to the “breathing” modes that cannot be captured by the conventional LMSM.

- **Y-Dir:** Overall, ISRS more similar than the X-direction ISRS are calculated from the ICS FEM and the LMSM. However, there are still noticeable differences observed for 6~10Hz and for 40~70Hz.

- **Z-Dir:** The same/similar ISRS are observed between the ICS FEM and the LMSM for the overall frequency range. However, slight differences can be observed after 50Hz.

![Figure 9. ISRS comparison between the ICS FEM and LMSM](image-url)
CONCLUSIONS AND DISCUSSIONS

In this paper, an FEM has been developed to perform seismic analyses and design of ICS subject to HF contents at CEUS hard rock sites. The effect of the mesh sensitivity to the dynamic analyses results (i.e., natural frequencies and dynamic amplifications) has been examined, and concluded that a mesh size ratio less ranged from 1 to 2 with respect to the D-wall thickness is appropriate. The appropriateness of the proposed ICS FEM was examined by comparing natural frequencies and mode shapes calculated from a conventional LMSM. It is observed that some local modes like “breathing” and “skewing” modes may not be captured in a simplified conventional LMSM, but the proposed ICS FEM can calculate such local modes, possibly having some effect on the attached NSSS components. It was also demonstrated that the conventional LMSM can effectively capture overall fundamental dynamic responses similar to ISRS of the ICS FEM. However, simplified conventional LMSM might not capture dynamic responses in the HF range. It is concluded that the proposed ICS FEM and modeling approaches are suitable to capture such undisclosed modes and sufficient for generating ISRS in the HF range and for subsequent use of the seismic analysis and qualification of NSSS components.

It has been demonstrated that the conventional LMSM can effectively capture the overall dynamic responses with less computational efforts and cost. However, such conventional single-stick LMSM has a lack of modeling fidelity capturing the local modes such as “breathing” and “skewing” modes due to its modeling assumption. Thus, it is suggested using a multiple-stick LMSM to properly capture the ICS cross sectional deformations in different directions like in the “skewing” mode. It is also anticipated that the use of an advanced beam element like Beam188, capable of exhibiting warping DOF in ANSYS Mechanical, Release 13.0 (2010), can account for local deformations such as D-wall “breathing”. Therefore, it is recommended to use advanced multiple stick FEM as a future study to properly capture such “breathing” and “skewing” modes when the stick modeling approach is used.

REFERENCES

American Society of Civil Engineers. (1998). *Seismic Analysis of Safety-Related Nuclear Structures and Commentary*. Reston, Virginia, USA

ANSYS Mechanical, Release 13.0, ANSYS, Inc., 2010


