Nuclear Operating Company
South Texas Project Electric Generating Station P.O. Box 289 Wadsworth, Texas 77483

January 18, 2011
U7-C-STP-NRC-110009

U. S. Nuclear Regulatory Commission
Attention: Document Control Desk
One White Flint North
11555 Rockville Pike
Rockville, MD 20852-2738

South Texas Project
Units 3 and 4
Docket Nos. 52-012 and 52-013
Response to Request for Additional Information

Attached are responses to NRC staff questions included in Request for Additional Information (RAI) letter numbers 366 and 369 related to Combined License Application (COLA) Part 2, Tier 2, Section 3.7. This completes the response to these NRC letters. The attachments address the responses to the RAI questions listed below:

03.07.02-29 03.07.02-30

There are no commitments in this response.

Where there are COLA markups, they will be made at the first routine COLA update following NRC acceptance of the RAI response.

If you have any questions regarding these responses, please contact me at (361) 972-7136, or Bill Mookhoek at (361) 972-7274.

I declare under penalty of perjury that the foregoing is true and correct.

Executed on 1/18/11

Scott Head
Manager, Regulatory Affairs
South Texas Project Units 3 & 4

jep

Attachments:
1. RAI 03.07.02-29
2. RAI 03.07.02-30

D091
NRO

STI 32809049

cc: w/o attachment except*
(paper copy)

Director, Office of New Reactors
U. S. Nuclear Regulatory Commission
One White Flint North
11555 Rockville Pike
Rockville, MD 20852-2738

Regional Administrator, Region IV
U. S. Nuclear Regulatory Commission
611 Ryan Plaza Drive, Suite 400
Arlington, Texas 76011-8064

Kathy C. Perkins, RN, MBA
Assistant Commissioner
Division for Regulatory Services
Texas Department of State Health Services
P. O. Box 149347
Austin, Texas 78714-9347

Alice Hamilton Rogers, P.E.
Inspection Unit Manager
Texas Department of State Health Services
P. O. Box 149347
Austin, Texas 78714-9347

*Steven P. Frantz, Esquire
A. H. Gutterman, Esquire
Morgan, Lewis & Bockius LLP
1111 Pennsylvania Ave. NW
Washington D.C. 20004

*Tom Tai
Two White Flint North
11545 Rockville Pike
Rockville, MD 20852

(electronic copy)

*George F. Wunder
*Tom Tai
Loren R. Plisco
U. S. Nuclear Regulatory Commission

Steve Winn
Joseph Kiwak
Eli Smith
Nuclear Innovation North America

Peter G. Nemeth
Crain, Caton & James, P.C.

Richard Peña
Kevin Pollo
L. D. Blaylock
CPS Energy

RAI 03.07.02-29

QUESTION:

RAI from Section 3.7 Audit, October 2010

For SSE ground motions, 10 CFR Part 50, Appendix S requires that SSCs will remain functional and within applicable stress, strain, and deformation limits and the evaluation must take into account soil-structure interaction (SSI) effects. Criterion III, "Design Control," of Appendix B to 10 CFR Part 50 states, in part, that "measures shall also be established for the selection and review for suitability of application of materials, parts, equipment, and processes that are essential to the safety related functions of the structures, systems and components." Additionally, Criterion III states in part that, "the design control measures shall provide for verifying or checking the adequacy of design,..." SRP Review guideline 3.8.1.II.4.F specifies that computer programs used in the design and analysis should be described and validated.

During the STP audit of Section 3.7, the verification and validation (V&V) documents of computer programs SASSI2000, SAP2000, and SHAKE2000 used in the seismic analysis of Category I structures were reviewed. The following issues were identified regarding these V&V documents:

SASSI2000 Version 3.0:

The SSI analysis performed with SASSI2000 is used to obtain the maximum accelerations, acceleration response spectra, and dynamic soil pressures that are used for seismic evaluation and design of the RB, CB, UHS Basin, RSW Pump House, and other seismic Category I structures. The dynamic forces, moments, and stresses are also calculated from the SASSI2000 analysis but are not used as design basis.

The V&V of three SASSI codes were reviewed. These codes are S&L SASSI2000-v3.0, SGH SASSI2000-v3.0 and SGH SASSI2000-v3.0-SGH. All three program V&V documentations do not adequately address all the program features that are used to calculate and obtain maximum accelerations, acceleration response spectra, and dynamic soil pressures. In particular, the scope of the test problems does not address the adequacy of the following program features that may be used in STP applications:

- General direction of load application in the model
- General orientation of elements in the model
- Accuracy of triangular elements (solid, shell and plane-strain) that may be used
- Acceptable aspect ratio of rectangular elements (solid, shell and plane-strain) to obtain accurate results, as used in the models
- Required mesh refinement to output out-of-plane responses in shell element
- Accuracy of the subtraction method for calculating foundation impedance

In addition, potential numerical instabilities with the use of high Poisson's ratio for modeling the saturated soil behavior in SASSI2000 may be of concern, as the Poisson's ratio approaches 0.5. As a result, the SASSI2000-v3.0 limitations with respect to capping the Poisson's ratio to avoid possible stability problems should be validated and stated.

Significant differences in the out-of-plane acceleration response of thick versus thin shell element models have also been observed in the analysis results with the thick shell model producing lower responses. This also needs to be further evaluated for SASSI2000-v3.0 as to the adequacy and limitations of the specific shell element type.

Without further demonstration that encompasses validation of the program features discussed above for STP applications, the staff cannot make a determination in the SER that the programs used in the seismic analysis will not adversely affect the SSI analysis result and meet the applicable regulations. As such, the applicant is requested to further demonstrate acceptability of SASSI2000 with additional test problems addressing the issues discussed above.

SAP2000 Version 10.1 and 14.1:

SAP2000 is used to calculate forces, moments and stresses for design of the site-specific seismic Category I structures such as UHS Basin, RSW Pump House, and RSW tunnel. The forces and moments are calculated by integrating stresses across design sections. It also appears that the thick shell element is used for modeling and design of slabs. Mesh sensitivity studies are also performed using time-history modal superposition method of fixed-base structure to assess the adequacy of the structural mesh refinement for calculation of accelerations and acceleration response spectra. To that extent, the SAP2000 V&V does not provide adequate validations for the following items:

- Accuracy of forces and moments calculated from section cuts in shell models
- Accuracy of thick shell element for calculating out-of-plane dynamic responses
- Accuracy of time-history modal analysis of fixed-base structures modeled using shell elements

As such, the applicant is requested to supplement the SAP2000 V&V with additional test problems to address the items discussed above. The staff needs this information to be able to conclude in the SER that the use of SAP2000 in STP applications will not adversely affect calculation of seismic forces and moments and the evaluation of SSI effects for Category I structures.

SHAKE2000 Version 3.5:

SHAKE2000 is used to calculate SSE-based foundation motions for SSI analysis of UHS Basin, RSW Pump House, and other Seismic Category I structures. The SHAKE2000 V&V has only tested soil models with up to 8 soil layers while the STP profile is a deep soil site that is modeled using large number of soil layers.

As such, the applicant is requested to further demonstrate acceptability of SHAKE2000 with additional test problems that check the use of large number of soil layers to encompass STP soil site. The staff needs this information to be able to conclude that the SSE-based foundation motion determined using SHAKE2000 computer program is adequate for STP application and meets the requirement of Appendix S to 10 CFR Part 50.

RESPONSE:

SASSI2000 Version 3.0:

The SASSI2000 Version 3.0 was procured from Isatis (agent for the Regents of the University of California). The program was installed at S&L computer system and named S&L SASSI2000-v3.0. Similarly, the program installed by SGH at their Boston office is named SGH SASSI2000-v3.0. These programs have been verified and validated (V&V) and the V&V documents for S&L SASSI2000-v3.0 and SGH SASSI2000-v3.0 are available in S&L office and SGH office, respectively.

All STP 3&4 soil-structure interaction (SSI) and structure-soil-structure interaction (SSSI) analyses have been performed using SGH SASSI2000-v3.0 and/or S&L SASSI2000-v3.0 except for the SSI analysis of the Ultimate Heat Sink (UHS)/Reactor Service Water (RSW) Pump House using a more refined mesh (i.e. model with 23.5 Hz passing frequency in the horizontal direction) as described in the response to RAI 03.07.02-24, Supplement 2, submitted with STPNOC letter U7-C-STP-NRC-100268, dated December 14, 2010, where the program SGH SASSI2000-v3.0-SGH was used. When the UHS/RSW Pump House SSI model was refined to accommodate higher passing frequency, the refined model exceeded the size capability of SGH SASSI2000-v3.0. The SGH SASSI2000-v3.0-SGH is a modified version of the SGH SASSI2000-v3.0 which allows handling of larger file sizes and reduces run time by using a more efficient solver. The validation of SGH SASSI2000-v3.0-SGH was performed by comparing the SSI analysis results for the UHS/RSW Pump House with the original SSI model (i.e. model with 15.6 Hz passing frequency in the horizontal direction) that were obtained using SGH SASSI2000-v.3 and SGH SASSI2000-v3.0-SGH. The results from the two programs matched within 2%. Furthermore, SGH SASSI2000-v3.0 and S&L SASSI2000-v3.0 are identical. Therefore, for ease of discussion, in the remainder of this response they will be referred to as "SASSI2000".

Prior to addressing the individual items noted in this RAI, the following should be noted in regards to the SSI and SSSI analyses for the STP 3&4 project:

- In all SSI and SSSI analyses, load applications are only in the global directions.
- In all SSI and SSSI analyses, elements are oriented along the global planes
- With the exception of a few transition elements for the refined model of the UHS/RSW Pump House, all shell elements in the SSI and SSSI analyses are rectangular elements.

- With the exception of a few transition elements of the 2D SSSI analyses, all plane-strain elements are rectangular elements. No triangular solid elements have been used in any of the SSI and SSSI analyses.
- With the exception of the original SSI model for UHS/RSW Pump House with maximum aspect ratio of 5.25, for all the remaining SSI and SSSI models including the refined model for the UHS/RSW Pump House, the maximum aspect ratio for the shell, solid and plain-strain elements is less than 5.
- All shell elements in the SSI analyses for spectra generation are thick shell elements.
- All shell elements in the SAP2000 models used for design of the structures are thick shell elements
- None of the designs are based on the element stresses from the SSI and/or SSSI analyses.

There are twenty five (25) existing problems, which have been used in V&V of various features of S&L SASSI2000-v3.0. The matrix of these 25 problems with the options validated by each problem is available in Table 5.1 of Calculation SVVR03.7.316-3.0 and was included as Table 1 in the program release memorandum. Similarly, there are 26 existing problems used in V&V of various features of SGH SASSI2000-v3.0. The matrix of these 26 problems with options validated by each problem is available in Table 1 of SGH document SAS-V3-0 Rev. 0.

The following provides the response to NRC Staff questions on specific program features. Please note that the V&V documentation of the computer programs are proprietary documents. Thus, only limited details are provided for the additional test problems described in this response. Full details and documentation for these test problems are available for review at S&L and SGH offices.

- **General direction of load application in the model**

  As noted earlier, in all STP 3&4 SSI and SSSI analyses the load applications are only along the global coordinates. Both S&L and SGH V&V documentation for SASSI2000 already have several problems with load applications along the global coordinates. Thus no additional test problems are performed.

- **General orientation of elements in the model**

  As noted earlier, in all STP 3&4 SSI and SSSI analyses the elements are oriented along the global planes. Both S&L and SGH V&V documentation for SASSI2000 already have several problems with load applications along the global coordinates. Thus no additional test problems are performed.

- **Accuracy of triangular elements (solid, shell and plane-strain) to obtain accurate results, as used in the models**

  Triangular elements have not been used in the STP 3&4 SSI analyses performed by S&L SASSI2000-v3.0. However, the existing S&L validation problems include triangular

solid elements. These problems have been validated for obtaining the transfer functions and acceleration responses in the models.

The SSI analyses performed for STP using the SGH SASSI2000-v3.0 use rectangular elements, except for the refined mesh SSI analysis of the UHS/RSW Pump House as described in the response to RAI 03.07.02-24, Supplement 2, submitted with STPNOC letter U7-C-STP-NRC-100268, dated December 14, 2010, where at isolated locations triangular elements are used for mesh transition. No stress results are calculated or used from the triangular elements. A review of the V&V documentation found that triangular elements were used in some problems but validation of the dynamic properties of triangular elements was not specifically addressed. The following additional validation problems have been included to further validate the accuracy of solid, shell, and plane-strain triangular elements.

Validation problem for thick shell in-plane:

A 6 in. long, 1 in. tall and 0.01 in. thick massless cantilever shear wall model aligned in the Y-Z plane with total mass of 0.2 lb-s^2/in^2 distributed as mass elements along the top of the wall is used. The wall has interaction nodes at the base and is connected to rigid soil. The nodes at the top of the wall are restrained in the vertical direction in order to model a pure shear case. The rectangular element finite element model consists of 150 (5 x 30) 3-D thick shell elements and the triangular element finite element model consists of 300 3-D thick shell elements as shown in Figure 03.07.02-29.1. Element and material properties are shown in Figure 03.07.02-29.1. The material damping is 4%.

In this model, the soil is assigned to behave rigidly. The shear wave velocity equal to 20,000 in./sec and compression wave velocity equal to 40,000 in./sec. The soil density is 0.130 lb/in^3 and damping ratio is 0%.

The seismic time history analysis is performed in the horizontal Y-direction (in-plane of the wall) for vertically propagating shear waves. The input motion is applied at the surface.

Table 03.07.02-29.1 shows the natural frequency comparison and Table 03.07.02-29.2 shows comparison for peak response spectra. Figure 03.07.02-29.2 through Figure 03.07.02-29.4: show comparisons of response spectra using the rectangular element model and the triangular element model. As can be seen from these comparisons, there is good agreement between the results from the triangular element model and the rectangular element model.

Validation problem for thick-shell out-of-plane:

The cantilever shear wall model is developed for two cases: 10 ft x 5 ft x 6 in. thick and 10 ft x 5 ft x 4 ft thick consisting of 50 (5 x 10) rectangular shell elements or 100

triangular elements. Model geometry and element and material properties are shown in Figure 03.07.02-29.5. Material damping is 4%.

The soil properties are assigned to behave rigidly. The shear wave velocity equal to 20,000 ft/sec and compression wave velocity equal to 40,000 ft/sec. The soil density is 0.130 kcf and damping ratio is 0%.

The seismic time history analysis is performed in the horizontal X-direction (out-of-plane of the wall) for vertically propagating shear waves. The input motion is applied at the surface.

Table 03.07.02-29.3 shows the natural frequency comparison and Table 03.07.02-29.4 shows comparison for peak response spectra. Figures 03.07.02-29.6 through 03.07.02-29.9 show comparisons of response spectra using the rectangular element model and the triangular element model. As can be seen from these comparisons, there is good agreement between the results from the triangular element model and the rectangular element model.

Validation problem for thick-shell axial:

The cantilever shear wall model is developed for a 10 ft x 5 ft x 4 ft structure consisting of 50 (5 x 10) rectangular shell elements or 100 triangular elements. Model geometry and element and material properties are shown in Figure 03.07.02-29.10. The shell elements are massless, mass elements are placed at the top of the wall, and material damping is 4%.

The soil properties are assigned to behave rigidly. The shear wave velocity equal to 20,000 ft/sec and compression wave velocity equal to 40,000 ft/sec. The soil density is 0.130 kcf and damping ratio is 0%.

The seismic time history analysis is performed in the vertical Z-direction for vertically propagating compression waves. The input motion is applied at the surface.

Table 03.07.02-29.5 shows the natural frequency comparison and Table 03.07.02-29.6 shows comparison for peak response spectra. Figures 03.07.02-29.11 and 03.07.02-29.12 show comparisons of response spectra using the rectangular element model and the triangular element model. As can be seen from these comparisons, there is good agreement between the results from the triangular element model and the rectangular element model.

Validation problem for plane-strain in-plane:

The cantilever shear wall model is developed for a 20 ft x 4 ft x 1 ft thick wall consisting of 5 2-D plane strain rectangular shell elements or 10 triangular elements modeled inside

an excavated area. Model geometry is shown in Figure 03.07.02-29.13. Material damping is 4%.

The soil properties are assigned to behave rigidly. The shear wave velocity equal to 20,000 ft/sec and compression wave velocity equal to 40,000 ft/sec. The soil density is 0.130 kcf and damping ratio is 0%.

The seismic time history analysis is performed in the horizontal X-direction for vertically propagating shear waves. The input motion is applied at the surface.

Table 03.07.02-29.7 shows the natural frequency comparison and Table 03.07.02-29.8 shows comparison for peak response spectra. Figure 03.07.02-29.14 shows comparison of response spectra using the rectangular element model and the triangular element model. As can be seen from these comparisons, there is good agreement between the results from the triangular element model and the rectangular element model.

Validation problem for plane-strain axial:

The cantilever shear wall model is developed for a 20 ft x 4 ft x 1 ft thick wall consisting of 5 2-D plane strain rectangular shell elements or 10 triangular elements. Model geometry is shown in Figure 03.07.02-29.13. Additional elements, 1 through 5, are included in the model, but are inactive since structural nodes connected to these elements are fixed. Material damping is 4%.

The soil properties are assigned to behave rigidly. The shear wave velocity equal to 20,000 ft/sec and compression wave velocity equal to 40,000 ft/sec. The soil density is 0.130 kcf and damping ratio is 0%.

The seismic time history analysis is performed in the vertical Z-direction for vertically propagating compression waves. The input motion is applied at the surface.

Table 03.07.02-29.9 shows the natural frequency comparison and Table 03.07.02-29.10 shows comparison for peak response spectra. Figure 03.07.02-29.15 shows comparison of response spectra using the rectangular element model and the triangular element model. As can be seen from these comparisons, there is good agreement between the results from the triangular element model and the rectangular element model.

- **Acceptable aspect ratio of rectangular elements (solid, shell and plane-strain) to obtain accurate results, as used in the models**

Both S&L and SGH V&V documentation for SASSI2000 have been expanded to include additional problems for aspect ratio of rectangular elements. Provided below are the details of the test problems and results added to the V&V documentation of S&L SASSI2000-v3.0 followed by details of the SGH test problems.

A parametric study with aspect ratios of 1:1, 1:2 , 1:3, 1:4 and 1:5 (models with solid elements, thick shell elements, thin shell elements and plane-strain elements) has been performed by S&L and is being added in the V&V documents. A similar parametric study with aspect ratios of 1:1, 1:2 , 1:3, 1:4, 1:5 and 1:10 has been performed by SGH. The results show that rectangular elements with aspect ratio up to 1:5 provide results within 5% accuracy with respect to elements with aspect ratio of 1:1. For STP SSI analyses the aspect ratio of rectangular elements is well within 1:5 except for a few elements with aspect ration of 5.25 which are used in the original SSI model for UHS/RSW Pump House. The following summarizes the details of each validation problem:

Validation Problem with Thin Shell Elements (S&L)

A 100 ft width, 80 ft long, and 4 ft thick concrete foundation supporting a 80 ft long, 1000 ft tall and 4 ft thick concrete wall at the middle strip of the foundation is used for the study. Five models are constructed, which have rectangular elements with aspect ratios of 1:1, 1:2, 1:3, 1:4 and 1:5. In the models, the foundation and excavated soil volume are modeled by solid elements and the concrete wall is modeled by thin plate/shell elements. The input motions are specified at the ground surface, in the global axes (two horizontal X, Y and vertical Z directions). The results compared are;
(a) maximum accelerations, (b) forces/moments and (c) In-structure response spectra at various elevations.

Table 03.07.02-29.11 shows the results of comparisons of maximum accelerations for various aspect ratios. The maximum difference between the accelerations is 4.0%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Table 03.07.02-29.12 shows the results of comparisons of maximum forces/moments for various aspect ratios. The maximum difference between the maximum forces/moments is 1.0%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Figures 03.07.02-29.16 through 03.07.02-29.18 show some typical comparisons of 5% damped in-structure response spectra. The comparisons show that the in-structure response spectra compare well for aspect ratios up to 1:5.

Based on above results it is concluded that the thin shell elements can be used with aspect ratios up to 1:5.

Validation Problem with Thick Shell Elements (S&L)

Models similar to above (thin plate element models) with 3-D thick plate/shell elements are analyzed and results compared.

Table 03.07.02-29.13 shows the results of comparisons of maximum accelerations for various aspect ratios. The maximum difference between the accelerations is 4.2%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Table 03.07.02-29.14 shows the results of comparisons of maximum forces/moments for various aspect ratios. The maximum difference between the maximum forces/moments is 2.1%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Figures 03.07.02-29.19 through 03.07.02-29.21 show the comparisons of some typical 5% damped in-structure response spectra. The comparisons show that the in-structure response spectra compare well for aspect ratios up to 1:5.

Based on above results it is concluded that the 3-D thick plate elements can be used with aspect ratios up to 1:5.

Validation Problem with Solid Elements (S&L)

Models similar to above with 3-D solids are analyzed and results compared.

Table 03.07.02-29.15 shows the results of comparisons of maximum accelerations for various aspect ratios. The maximum difference between the accelerations is 3.5%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Table 03.07.02-29.16 shows the results of comparisons of maximum forces for various aspect ratios. The maximum difference between the maximum forces is 2.9%, for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Figures 03.07.02-29.22 through 03.07.02-29.24 show the comparisons of some typical 5% damped in-structure response spectra. The comparisons show that the in-structure response spectra compare well for aspect ratios up to 1:5.

Based on above results it is concluded that the 3-D solid elements can be used with aspect ratios up to 1:5.

Validation Problem with Plane Strain Elements (S&L)

A unit width (1 ft) of 4 ft thick, 100 ft width concrete foundation supporting 10 ft thick wall (1 ft width) at the middle strip of foundation is analyzed. Five models are constructed, which have rectangular elements with aspect ratios of 1:1, 1:2, 1:3, 1:4 and 1:5. The input motions are specified in horizontal and vertical directions. The results compared are: (a) maximum accelerations, (b) In-structure response spectra at various elevations and (c) Maximum shear and axial forces.

Table 03.07.02-29.17 shows the results of comparisons of maximum accelerations for various aspect ratios. The maximum difference between the accelerations is 7.8% (less than 10%), for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Table 03.07.02-29.18 shows the results of comparisons of maximum forces for various aspect ratios. The maximum difference between the maximum forces is 5.9% (less than 10%), for model with aspect ratio of 1:5 (as compared to the aspect ratio of 1:1).

Figures 03.07.02-29.25 and 03.07.02-29.26 show the comparisons of some typical 5% damped in-structure response spectra. The comparisons show that the in-structure response spectra compare well for aspect ratios up to 1:5.

Based on above results it is concluded that the plane strain plate elements can be used with aspect ratios up to 1:5.

Validation Problem with Thick Shell Elements (SGH)

Two structures are used to study the effect of element aspect ratio:

- A 10 ft wide by 1.0 ft thick cantilever beam
- A 20 ft tall by 1.0 ft thick shear wall

For each structure, five models using rectangular 3-D thick plate/shell structural elements with aspect ratios of 1:1, 2:1, 4:1, 5:1, and 10:1 are used. The height of the cantilever in each model varies with aspect ratio of the elements used, with the total height equal to 100 ft multiplied by the aspect ratio of the elements in the model. The length of the shear wall in each model also varies with the aspect ratio of the elements used, with length equal to 50 ft multiplied by the aspect ratio of the element. Both structures are modeled on rigid soil. The models are subjected to harmonic and seismic loadings.

For harmonic loadings a force is applied to nodes at the top of the model corresponding to vertical (z) force applied to the cantilever model, horizontal out-of-plane force applied to the cantilever model, and horizontal in-plane (Y) force applied to the shear wall model. The harmonic force loading analyses are used to calculate element stresses and model stiffness which are compared to calculated theoretical values.

For seismic excitations, concentrated inertias are placed at the top of the model, and material damping is 4.0%. Seismic excitation is applied at the ground surface model corresponding to vertical (z) excitation applied to the cantilever model, horizontal out-of-plane excitation applied to the cantilever model, and horizontal in-plane (Y) excitation applied to the shear wall model. The nodal acceleration transfer functions are compared to theoretical calculated natural frequencies.

Table 03.07.02-29.19 shows the results of the comparisons of element stresses SASSI2000 with various aspect ratios output from SASS2000 from to theoretical values.

The maximum difference between the element stresses calculated with SASSI2000 and theoretical values is 2.5%.

Table 03.07.02-29.20 shows the results of the comparisons of structural stiffnesses with various aspect ratios calculated using output from SASSI2000 to theoretical values. The maximum difference between the structural stiffnesses calculated using output from SASSI2000 and theoretical values is 3.0%.

Table 03.07.02-29.21 shows the results of the comparisons of the first natural frequency of each structure with various aspect ratios output from SASSI2000 to theoretical values. The maximum difference between the first natural structural frequency calculated using output from SASSI2000 and theoretical values is 1.3%.

Based on above results it is concluded that the 3-D thick plate/shell elements can be used with aspect ratios up to 10:1.

Validation Problem with 3-D Solid Elements (SGH)

Models similar to above with 3-D solid structural elements used to model the cantilever and shear wall.

Table 03.07.02-29.22 shows the results of the comparisons of element stresses SASSI2000 with various aspect ratios output from SASS2000 from to theoretical values. The maximum difference between the element stresses calculated with SASSI2000 and theoretical values is 0.0%.

Table 03.07.02-29.23 shows the results of the comparisons of structural stiffnesses with various aspect ratios calculated using output from SASSI2000 to theoretical values. The maximum difference between the structural stiffnesses calculated using output from SASSI2000 and theoretical values is 2.9%.

Table 03.07.02-29.24 shows the results of the comparisons of the first natural frequency of each structure with various aspect ratios output from SASSI2000 to theoretical values. The maximum difference between the first natural structural frequency calculated using output from SASSI2000 and theoretical values is 3.0%.

Based on above results it is concluded that the 3-D solid elements can be used with aspect ratios up to 10:1.

Validation Problem with 2-D Plane Strain Elements (SGH)

Models similar to the validation of the thick shell elements (SGH) are used for validation of 2D plane strain elements. The models are excavated, and the 2D plane strain elements are placed within the excavation but not connected to the excavation sides, and the

models are not loaded in the out-of-plane direction because out-of-plane motion is incompatible with 2D analyses.

Table 03.07.02-29.25 shows the results of the comparisons of element stresses SASSI2000 with various aspect ratios output from SASS2000 from to theoretical values. The maximum difference between the element stresses calculated with SASSI2000 and theoretical values is 0.0%.

Table 03.07.02-29.26 shows the results of the comparisons of structural stiffnesses with various aspect ratios calculated using output from SASSI2000 to theoretical values. The maximum difference between the structural stiffnesses calculated using output from SASSI2000 and theoretical values is 3.1%.

Table 03.07.02-29.27 shows the results of the comparisons of the first natural frequency of each structure with various aspect ratios output from SASSI2000 to theoretical values. The maximum difference between the first natural structural frequency calculated using output from SASSI2000 and theoretical values is 3.6%.

Based on above results it is concluded that the 2-D plane strain elements can be used with aspect ratios up to 10:1.

- **Required mesh refinement to output out-of-plane responses in shell elements**

  The required mesh refinement to output out-of-plane responses in shell elements is not considered part of V&V process. Mesh refinements for out-of-plane responses depends on structural configuration and boundary conditions, hence is problem dependent. The user, who is experienced in finite element modeling, uses refined model for getting accurate final responses. Per NRC Staff request a mesh refinement study was performed for the SSI analysis of UHS/RSW Pump House and the design of UHS/RSW Pump House and in-structure response spectra cover the results obtained from the refined analysis. For more detailed information, see the response to RAI 03.07.02-24, Supplement 2, submitted with STPNOC letter U7-C-STP-NRC-100268, dated December 14, 2010.

- **Accuracy of subtraction method for calculating foundation impedance**

  Accuracy of subtraction method for calculating foundation impedance is covered in a number of existing V&V problems (as shown in Table 5.1 of Calculation S&L SVVR03.7.316-3.0, there are nine problems validating accuracy of subtraction method. There is also a problem, which compares results from subtraction method and direct method. Similarly, there are V&V problems demonstrating the accuracy of subtraction method, in the V&V document for SGH SASSI2000 v-3.

- **Limitations with respect to capping the Poisson's ratio to avoid possible stability problems should be validated and stated**

The limitation with respect to capping the Poisson's ratio to avoid possible stability problems is not considered a part of the V&V process. The significance of the limitation with respect to capping the Poisson's ratio to avoid stability problems in SASS2000 is problem dependent, i.e. depending on the soil properties. In general Poisson's ratio is capped in the range of 0.47 to 0.49 for determining the compression wave velocity for soil layers below groundwater level. The issue of numerical stability is part of the program user responsibilities. For STP 3&4 this issue has been studied in detail as described in the response to RAI 03.07.01-25, Supplement 1, submitted with STPNOC letter U7-C-STP-NRC-100253, dated November 29, 2010.

- **Adequacy and limitations of the specific shell element type (i.e. thin or thick shell elements)**

  For shell elements, two thickness formulations are available, thin or thick, with the difference being in the consideration of transverse shear deformations as noted below:

  - Thick shell formulation includes the effects of transverse shear deformation
  - Thin shell formulation neglects transverse shear deformation

  For situations where shear deformations are rather negligible and, therefore, use of thin shell elements may be justified, use of thick shell elements will not introduce any inaccuracy and the results using thick shell elements will be nearly identical (yet more accurate) to those using thin shell elements. For more information, see the response to RAI 03.07.02-25 submitted with STPNOC letter U7-C-STP-NRC-100268, dated December 14, 2010.

  The adequacy and limitations of the specific shell element type (i.e. thin or thick shell elements) is not considered part of the V&V process, because selection of appropriate shell element type is problem dependent and thus these issues are part of the program user responsibilities.

Update of SASSI 2000 V&V Reports

To further clarify/expand the V&V documentation, the SASSI2000 V&V Reports (both S&L and SGH) are updated as follows:

- Purpose and scope of the V&V document is updated and expanded to clearly specify how V&V of SASSI2000 program is performed.
- Purpose of each test problem is expanded to clearly state which options are being validated.
- Conclusion of each test problem is expanded to clearly specify what options are considered validated.

- The summary and conclusion section of the document is expanded to show in detail features of the program that are validated along with identification of the supporting test problem.

SAP2000 Version 10.1 and 14.1:

The following provides the response to NRC staff questions on specific SAP2000 program features:

- **Accuracy of forces and moments calculated from section cuts in shell models**

  Please note that this SAP2000 program feature has not yet been used for the STP 3&4 project. However, as requested by the NRC staff the following additional test problem has been added to the SAP2000 V&V documentation.

  A 300" long x 24" wide x 12" deep beam fixed on one end and simply supported on the other which supports a mid-span load of 60 kips along with an axial load of 60 kips axial load is modeled and the accuracy of the forces and moments from various section cuts are compared against the values determined by manual calculations. The SAP2000 finite element (FE) model consists of a 6 by 24 mesh with thick shell elements. The transverse load is 60 kips total at center and the axial load is 60 kips total at x=300".
  The model has the following boundary conditions.

  - $\delta x = \delta z = \theta y = 0$ @ x = 0"
  - $\delta x = 0$ @ x = 300"

  Comparison of shears and moments for 12 inches wide (half of the beam width) section at 5 locations are provided in Table 03.07.02-29.28. This comparison shows good agreement between the results from manual calculations and the SAP2000 section cuts with a maximum difference of 2%.

- **Accuracy of thick shell element for calculating out-of-plane dynamic responses**

  The following additional test problem has been added to the SAP2000 V&V documentation.

  A 300" x 400", 9" thick concrete plate supported on all 4 sides ($\delta x = \delta y = \delta z = 0$) is subjected to a concentrated harmonic load at the center. The maximum amplitude of the load is 1000 lbs, and the sine loading history consists of 5 cycles with period of 0.05 sec. The total analysis time is 1 sec. The problem was solved in SAP2000 using the modal time history analysis method (5 modes) and the results were compared against those from a similar ANSYS model. Table 03.07.02-29.29 provides a comparison of the results for modal frequencies, plate center displacement and support reactions. As can be seen from

these comparisons, there is good agreement between the results from SAP2000 and ANSYS with a maximum difference of 8.6%.

- **Accuracy of time-history modal analysis of fixed-base structures modeled using shell elements**

  The following additional test problem has been added to the SAP2000 V&V documentation.

  The problem consists of a 720" tall 200" x 300" tube (shell elements) with fixed base. The thickness of the shell element is 12". The tube is subjected to a 22 second long horizontal ground acceleration (parallel to the short side) history used in the STP 3&4 project. Modal time history analysis is to be used. All modes up to mass participation of 90% are included (28 modes). The results from SAP2000 analysis are compared to the results from ANSYS analysis. Tables 03.07.02-29.30 and 03.07.02-29.31 provide comparisons for modal frequencies, displacements, and support reactions. As can be seen from these comparisons, there is good agreement between the results from SAP2000 and ANSYS with a maximum difference of 8.1%.

SHAKE2000 Version 3.5

The theory and analysis technique used in SHAKE2000 Version 3.5 is essentially the same as original SHAKE, "A computer Program for Earthquake Response Analysis of Horizontally Layered Sites"; by Schnabel, P.B; Lysmer, J; and Seed, H.B Report No. 72-12, Earthquake Engineering Research Center, College of Engineering, University of California, Berkeley, December 1972.

The program is based on the continuous solution to wave equation adopted for use with transient motions through Fast Fourier Transform algorithm. The nonlinearity of the shear modulus and damping is accounted for by the use of equivalent linear soil properties using an iterative procedure to obtain values for modulus and damping compatible with effective strains in each layer. In the SHAKE2000 program, the maximum number of soil layer limitation of 200 for analysis of a soil profile is a program dimension assignment, and any anomaly caused by use of large number of soil layers can be detected by a qualified SSI analyst.

As requested by the NRC Staff, V&V documentation of the SHAKE2000 program has been expanded to include six (6) additional test problems with 116 soil layers to demonstrate acceptability of the SHAKE2000 program when using large number of soil layers. All STP 3&4 SHAKE2000 analyses have been performed using 100 or fewer soil layers. These 6 problems cover both in-profile (within) and Outcrop input-output options of the program. The analyses and results from these six problems demonstrate that the program is validated for large number of layers up to 116 layers.

Validation Problem 1

This validation problem is used to validate the iterated strain compatible shear modulus and damping and accelerations for various soil layers, for input motion defined at the base of the model. The soil profile property for validation problem 1 is the same as 8-layer problem used in the original V&V. Total depth of the soil profile is 116 ft. The soil profile is divided into 116 layers. The input motion is specified at the base of the soil profile and iterated strain compatible shear modulus and damping ratios and maximum accelerations of soil layers obtained from the 116 layer and 8 layer models are compared. The maximum difference between shear modulus values is less than 0.5% (see Table 03.07.02-29.32). The maximum difference between damping values is less than 2.3% (see Table 03.07.02-29.32). The maximum difference between acceleration values is less than 0.8% (see Table 03.07.02-29.33). Thus, the option to calculate the iterated strain compatible shear modulus and damping values, and accelerations for various soil layers due to seismic input at the base of the soil profile, for large number of soil layers, up to 116 layers is validated.

Validation Problem 2

This validation problem is used to validate the iterated strain dependent shear modulus and damping and accelerations for various soil layers, for input motion defined at the ground surface. This problem is the same as validation problem 1, except that the input is specified as outcrop at the ground surface. Iterated strain compatible shear modulus and damping ratios and maximum accelerations of soil layers obtained from the 116 layer and 8 layer models are compared. The maximum difference between shear modulus values is less than 0.3% (see Table 03.07.02-29.34). The maximum difference between damping values is less than 0.8% (see Table 03.07.02-29.34). The maximum difference between acceleration values is less than 3.3% (see Table 03.07.02-29.35). Thus the option to calculate the iterated strain compatible shear modulus and damping values, and accelerations for various soil layers due to seismic input defined at the ground surface, for large number of soil layers, up to 116 layers is validated.

Validation Problem 3

This validation problem is used to validate outcrop option of the program for within (in-profile) motion applied at the base of the soil profile and determining outcrop motion at the ground surface. The 116 layers problem is used with seismic input motion at the base of the model and in-profile motion (response spectra) and outcrop motion at 1 foot below the ground surface are obtained. The input motion is applied at the base as in-profile. For this problem, the in-profile and outcrop motion at 1 ft below the ground surface are compared. It is a technical fact that the in-profile and outcrop motion at the top layer (ground surface) are the same. The in-profile and outcrop motion are selected at 1 foot below the ground surface, since in SHAKE2000, the motion at the ground surface is always defined as outcrop. The selected 1 foot below the ground surface is used and is acceptable for comparison, because the frequency of the first layer is well above 50 Hz (much higher than the frequency content of input motion). For such a case the in-profile and outcrop motions

at 1 foot depth should be the same. Figure 03.07.02-29.27 shows the comparison between the in-profile and outcrop 5% damped spectra at 1 foot below the ground surface. The two spectra are the same (lay over on each other). Thus the option of the program for in-profile motion applied at the base of the soil profile and resulting outcrop motion at the ground surface is validated for large number of soil layers, up to 116 layers.

Validation Problem 4

This validation problem is used to validate outcrop option of the program for outcrop motion applied at the base of the soil profile and determining outcrop motion at the ground surface. The 116 layers problem is used with seismic outcrop input motion at the base of the model and in-profile motion (response spectra) and outcrop motion at 1 foot below the ground surface are obtained. For this problem, the in-profile and outcrop motion at 1 foot below the ground surface are compared. It is a technical fact that the in-profile and outcrop motion at the top layer (ground surface) are the same. The in-profile and outcrop motion are selected at 1 foot below the ground surface, since in SHAKE2000, the motion at the ground surface is always defined as outcrop. The selected 1 foot below the ground surface is used and is acceptable for comparison, because the frequency of the first layer is well above 50 Hz (much higher than the frequency content of input motion). For such a case the in-profile and outcrop motions at 1 foot depth should be the same. Figure 03.07.02-29.28 shows the comparison between the in-profile and outcrop 5% damped spectra at 1 foot below the ground surface. The two spectra are the same (lay over on each other). Thus the option of the program for outcrop motion applied at the base of the soil profile and resulting outcrop motion at the ground surface is validated for large number of soil layers, up to 116 layers.

Validation Problem 5

This validation problem is used to validate outcrop option of the program for outcrop motion applied at the ground surface and determining the outcrop motion at a certain depth of the soil profile. The 116 layer problem, with very stiff soil modulus is analyzed with outcrop input at the ground surface. The stiffness of the soil profile is such that the soil column has a frequency greater than 50 Hz. The input motion is the STP 3&4 site-specific SSE motion, which has highest frequency of 33 Hz. The in-profile motion and outcrop motion at 42 feet below the ground surface are obtained. For this problem, the in-profile and out-crop motion at a location below the ground surface should be the same. Figure 03.07.02-29.29 shows the comparison between the in-profile and outcrop 5% damped spectra at 42 feet below the ground surface. The two spectra are the same (lay over on each other). Thus the option to obtain outcrop motion at certain depth of the soil profile with outcrop input motion applied at the ground is validated for large number of soil layers, up to 116 layers.

Validation Problem 6

This validation problem is used to validate outcrop motion applied at a depth of the soil profile and determining outcrop motion at the ground surface. The 116 layer problem, with very stiff soil modulus is analyzed with outcrop motion input 42 feet below the ground surface. The stiffness of the soil profile is such that the soil column has a frequency greater than 50 Hz. The input motion is the STP 3&4 site-specific SSE motion, which has highest frequency of 33 Hz. In-profile and outcrop motion at 1 foot below the ground surface are obtained. For this problem, the in-profile and outcrop motion at 1 foot below the ground surface should be the same. Figure 03.07.02-29.30 shows the comparison between the in-profile and outcrop 5% damped spectra at 1 foot below the ground surface. The two spectra are the same (lay over on each other). Thus the option to obtain outcrop motion at the ground surface with outcrop input motion applied at a depth of the soil profile is validated for large number of soil layers, up to 116 layers.

COLA will be revised as shown in Enclosure 1 as a result of this response.

Table 03.07.02-29.1: Natural Frequency for Thick Shell Element Model with In-Plane Load

Element Model	Natural Frequency (Hz)		
	SASSI	Hand Calculation	Difference
Rectangular	2.88	2.96	2.70%
Triangle	2.88	2.96	2.70%

Table 03.07.02-29.2: Peak of Response Spectra for Thick Shell Element Model with In-Plane Load

Node	Acceleration (g)		
	Rectangular El. Model	Triangular El. Model	Difference
158	3.90	3.79	2.84%
171	3.03	3.09	-1.73%
184	3.90	3.75	3.92%

Table 03.07.02-29.3: First Mode Natural Frequencies for Thick Shell Element Model with Out-of-Plane Load

Element Model/ Node	Natural Frequency (Hz)				
	Rectangular El. Model	Triangular El. Model	Difference	Thin Plate Hand Calculation	Difference
10'x5'x6" - 31	0.27	0.27	0.00%	0.27	0%
10'x5'x4'- 31	1.98	2.00	-1.01%	2.16	8%
10'x5'x6" - 61	0.27	0.27	0.00%	0.27	0%
10'x5'x4'- 61	1.98	2.00	-1.01%	2.16	8%

Table 03.07.02-29.4: Peak of Response Spectra for Thick Shell Element Model with Out-of-Plane Load

Model/Node	Acceleration (g)		
	Rectangular El. Model	Triangular El. Model	Difference
10'x5'x6" - 31	1.93	2.00	-3.63%
10'x5'x4'- 31	2.10	2.15	-2.38%
10'x5'x6" - 61	1.86	1.77	4.84%
10'x5'x4'- 61	5.47	5.52	-0.91%

Table 03.07.02-29.5: First Mode Natural Frequencies for Thick Shell Element Model with Vertical Axial Load

Element Model/ Node	Natural Frequency (Hz)				
	Rectangular El. Model	Triangular El. Model	Difference	Thin Plate Hand Calculation	Maximum. Difference
10'x5'x4'- 31	8.11	8.13	-0.25%	8.12	0.12%
10'x5'x4'- 61	8.13	8.15	-0.25%	8.12	0.37%

Table 03.07.02-29.6: Peak of Response Spectra for Thick Shell Element Model with Vertical Axial Load

Model/Node	Acceleration (g)		
	Rectangular El. Model	Triangular El. Model	Difference
10'x5'x4'- 31	1.22	1.20	1.84%
10'x5'x4'- 61	2.28	2.24	1.65%

Table 03.07.02-29.7: First Mode Natural Frequencies for 2-D Plane Strain Element Model with In-Plane Load

Node	Natural Frequency (Hz)				
	Rectangular El. Model	Triangular El. Model	Difference	Hand Calculation	Difference
52	7.10	7.10	0.00%	7.04	-0.85%

Table 03.07.02-29.8: Peak of Response Spectra 2-D Plane Strain Element Model with In-Plane Load

Node	Acceleration (g)		
	Rectangular El. Model	Triangular El. Model	Difference
52	2.49	2.49	0.00%

Table 03.07.02-29.9: First Mode Natural Frequencies 2-D Plane Strain Element Model with Vertical Axial Load

Node	Natural Frequency (Hz)				
	Rectangular El. Model	Triangular El. Model	Difference	Hand Calculation	Difference
52	11.47	11.47	0.00%	11.48	0.09%

Table 03.07.02-29.10: Peak of Response Spectra 2-D Plane Strain Element Model with Vertical Axial Load

Node	Acceleration (g)		
	Rectangular El. Model	Triangular El. Model	Difference
52	1.89	1.89	0.00%

Table 03.07.02-29.11

Comparison of Maximum Acceleration (g) – Thin Shell Model									
Aspect Ratio	X Responses Due to X Direction Input Motion			Y Responses Due to Y Direction Input Motion			Z Responses Due to Z Direction Input Motion		
	Bottom of Wall	Middle of Wall	Top of Wall	Bottom of Wall	Middle of Wall	Top of Wall	Bottom of Wall	Middle of Wall	Top of Wall
1:1	0.1462	0.3110	0.6320	0.1356	0.2133	0.3056	0.1343	0.1426	0.1460
1:2	0.1462	0.3117	0.6381	0.1364	0.2178	0.3127	0.1348	0.1436	0.1471
1:3	0.1463	0.3110	0.6375	0.1366	0.2193	0.3149	0.1348	0.1437	0.1472
1:4	0.1462	0.3106	0.6372	0.1367	0.2201	0.3163	0.1349	0.1438	0.1473
1:5	0.1462	0.3105	0.6369	0.1367	0.2206	0.3173	0.1349	0.1439	0.1474
Maximum Difference (%)									
	0.07	0.4	1.0	0.8	3.4	4.0	0.4	0.9	1.0

Table 03.07.02-29.12

Comparison of Maximum Forces/Moment – Thin Shell Model			
Aspect Ratio	X –Dir. Input Motion Induced Bending Moment (ft-kips/ft)	Y-Dir. Input Motion Induced Total In-Plane Shear Force (kips)	Z-Dir. Input Motion Induced Total Axial Force (kips)
1:1	595.8	1017.3	648.2
1:2	596.3	1019.3	653.9
1:3	596.3	1021.7	654.2
1:4	596.3	1025.3	654.9
1:5	596.3	1027.7	655.3
Maximum Difference (%)	0.1	1.0	1.1

Table 03.07.02-29.13

Maximum Acceleration (g) – Thick Shell Model									
Aspect Ratio	X Responses Due to X Direction Input Motion			Y Responses Due to Y Direction Input Motion			Z Responses Due to Z Direction Input Motion		
	Bottom of Wall	Mid-Height of Wall	Top of Wall	Bottom of Wall	Mid-Height of Wall	Top of Wall	Bottom of Wall	Mid-Height of Wall	Top of Wall
1:1	0.1461	0.3168	0.6449	0.1357	0.2132	0.3058	0.1343	0.1426	0.1460
1:2	0.1462	0.3192	0.6500	0.1364	0.2186	0.3139	0.1348	0.1436	0.1471
1:3	0.1462	0.3194	0.6506	0.1366	0.2200	0.3162	0.1348	0.1437	0.1472
1:4	0.1462	0.3194	0.6507	0.1367	0.2208	0.3179	0.1349	0.1438	0.1473
1:5	0.1462	0.3195	0.6508	0.1368	0.2212	0.3189	0.1349	0.1439	0.1474
Maximum Difference (%)									
	0.07	0.9	0.9	0.8	3.8	4.2	0.4	0.9	1.0

Table 03.07.02-29.14

Comparison of Maximum Forces/Moment – Thick Shell Model			
Aspect Ratio	X –Dir. Input Motion Induced Bending Moment (ft-kips/ft)	Y-Dir. Input Motion Induced Total In-Plane Shear Force (kips)	Z-Dir. Input Motion Induced Total Axial Force (kips)
1:1	594.2	1013.3	644.3
1:2	594.1	1012.7	653.9
1:3	594.1	1017.5	655.9
1:4	594.1	1021.1	657.4
1:5	594.1	1023.4	658.1
Maximum Difference (%)	0.0	1.0	2.1

Table 03.07.02-29.15

Comparison of Maximum Acceleration (g) – 8-Node Solid Element Model									
Aspect Ratio	X Response Due to X Direction Input Motion			Y Response Due to Y Direction Input Motion			Z Response Due to Z Direction Input Motion		
	Bottom of Wall	Mid-Height of Wall	Top of Wall	Bottom of Wall	Mid-Height of Wall	Top of Wall	Bottom of Wall	Mid-Height of Wall	Top of Wall
1:1	0.1455	0.3179	0.6573	0.1359	0.2150	0.3084	0.1349	0.1432	0.1466
1:2	0.1455	0.3179	0.6578	0.1365	0.2195	0.3158	0.1351	0.1435	0.1469
1:3	0.1455	0.3183	0.6583	0.1366	0.2204	0.3173	0.1351	0.1436	0.1470
1:4	0.1455	0.3182	0.6584	0.1367	0.2208	0.3179	0.1351	0.1436	0.1470
1:5	0.1455	0.3180	0.6581	0.1368	0.2214	0.3192	0.1351	0.1437	0.1471
Maximum Difference (%)	< 1%	< 1%	< 1%	< 1%	2.9%	3.5%	< 1%	< 1%	< 1%

Table 03.07.02-29.16

Comparison of Wall Section Forces (kips) – 8-Node Solid Element Model			
Aspect Ratio	X Direction Input Motion Induced Total Shear Force V_{zx} (kips)	Y Direction Input Motion Induced Total Shear Force V_{zy} (kips)	Z Direction Input Motion Induced Total Axial Force P_z (kips)
1:1	953.3	1007.4	660.5
1:2	950.7	1027.8	662.1
1:3	951.0	1032.2	662.2
1:4	950.7	1034.2	662.3
1:5	951.0	1037.1	662.8
Maximum Difference (%)	< 1%	2.9%	< 1%

Table 03.07.02-29.17

Maximum Zero Period Accelerations (ZPA)						
Case	X-Dir Model			Z-Dir Model		
	Bottom of Wall (Node 21) - a_x (g)	Middle of Wall (Node 35) - a_x (g)	Top of Wall (Node 45) - a_x (g)	Bottom of Wall (Node 21) - a_z (g)	Middle of Wall (Node 35) - a_z (g)	Top of Wall (Node 45) - a_z (g)
1 (Aspect Ratio 1:1)	0.1501	0.2965	0.6442	0.1370	0.1396	0.1405
2 (Aspect Ratio 1:2)	0.1526	0.3039	0.6557	0.1374	0.1403	0.1412
3 (Aspect Ratio 1:3)	0.1533	0.3135	0.6594	0.1375	0.1404	0.1413
4 (Aspect Ratio 1:4)	0.1534	0.3176	0.6605	0.1375	0.1405	0.1414
5 (Aspect Ratio 1:5)	0.1535	0.3197	0.6609	0.1375	0.1405	0.1414
Maximum Difference	2.3%	7.8%	2.6%	0.4%	0.6%	0.6%

Table 03.07.02-29.18

Maximum Forces at the Bottom Wall Section		
Case	X-Dir Model	Z-Dir Model
	Shear Force Vx (kip)	Axial Force Pz (kip)
1 (Aspect Ratio 1:1)	39.130	19.869
2 (Aspect Ratio 1:2)	39.547	19.963
3 (Aspect Ratio 1:3)	40.724	19.981
4 (Aspect Ratio 1:4)	41.232	19.991
5 (Aspect Ratio 1:5)	41.458	19.995
Maximum Difference	5.9%	0.6%

Table 03.07.02-29.19

Maximum Element Stresses - Thick Plate/Shell Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Vertical Stress at Bottom of Cantilever (kip/ft)	SASSI2000	0.100	0.100	0.100	0.100	0.100
		Theoretical	0.100	0.100	0.100	0.100	0.100
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
	Horizontal Stress at Bottom of Cantilever (kip/ft)	SASSI2000	0.017	0.017	0.017	0.017	0.017
		Theoretical	0.017	0.017	0.017	0.017	0.017
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	Shear Stress at Bottom of Cantilever (kip/ft)	SASSI2000	0.100	0.100	0.100	0.100	0.100
		Theoretical	0.100	0.100	0.100	0.100	0.100
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
	Bending Stress at Bottom of Cantilever (kip-ft/ft)	SASSI2000	9.502	19.005	38.009	47.512	95.025
		Theoretical	9.500	19.000	38.000	47.500	95.000
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Shear Stress at Bottom of Wall (kip/ft)	SASSI2000	0.0200	0.0098	0.0049	0.0039	0.0020
		Theoretical	0.0200	0.0100	0.0050	0.0040	0.0020
		Percent Difference	0.0%	-2.0%	-2.0%	-2.5%	0.0%
Maximum Percent Difference			0.0%	2.0%	2.0%	2.5%	0.0%

Table 03.07.02-29.20

Structural Stiffnesses - Thick Plate/Shell Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Structural Stiffness at Top of Cantilever (kip/ft)	SASSI2000	3.218	1.609	0.805	0.644	0.322
		Theoretical	3.125	1.563	0.781	0.625	0.313
		Percent Difference	3.0%	3.0%	3.0%	3.0%	3.0%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	Structural Stiffness at Top of Cantilever (kip/ft)	SASSI2000	10.053	1.256	0.157	0.080	0.010
		Theoretical	10.000	1.250	0.156	0.080	0.010
		Percent Difference	0.5%	0.5%	0.5%	0.5%	0.5%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Structural Stiffness at Top of Wall (kip/ft)	SASSI2000	1.0755	2.1473	4.2863	5.3533	10.6952
		Theoretical	1.0684	2.1368	4.2735	5.3419	10.6838
		Percent Difference	0.7%	0.5%	0.3%	0.2%	0.1%
Maximum Percent Difference			3.0%	3.0%	3.0%	3.0%	3.0%

Table 03.07.02-29.21

First Natural Frequency - Thick Plate/Shell Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	First Natural Frequency at Top of Cantilever (Hz)	SASSI2000	16.162	11.426	8.081	7.227	5.103
		Theoretical	15.959	11.285	7.979	7.137	5.047
		Percent Difference	1.3%	1.3%	1.3%	1.3%	1.1%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	First Natural Frequency (Hz)	SASSI2000	28.564	10.107	3.564	2.563	0.903
		Theoretical	28.548	10.093	3.568	2.553	0.903
		Percent Difference	0.1%	0.1%	-0.1%	0.4%	0.1%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	First Natural Frequency (Hz)	SASSI2000	9.351	9.351	9.326	9.326	9.326
		Theoretical	9.331	9.331	9.331	9.331	9.331
		Percent Difference	0.2%	0.2%	-0.1%	-0.1%	-0.1%
Maximum Percent Difference			1.3%	1.3%	1.3%	1.3%	1.1%

Table 03.07.02-29.22

Maximum Element Stresses - 3-D Solid Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Vertical Stress at Bottom of Cantilever (kip/ft^2)	SASSI2000	0.010	0.010	0.010	0.010	0.010
		Theoretical	0.010	0.010	0.010	0.010	0.010
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	Shear Stress at Bottom of Cantilever (kip/ft^2)	SASSI2000	0.010	0.010	0.010	0.010	0.010
		Theoretical	0.010	0.010	0.010	0.010	0.010
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Shear Stress at Bottom of Wall (kip/ft^2)	SASSI2000	0.004	0.002	0.001	0.001	0.000
		Theoretical	0.004	0.002	0.001	0.001	0.000
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Maximum Percent Difference			0.0%	0.0%	0.0%	0.0%	0.0%

Table 03.07.02-29.23

Structural Stiffnesses - 3-D Solid Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Structural Stiffness at Top of Cantilever (kip/ft)	SASSI2000	400.8	200.4	100.2	80.19	40.08
		Theoretical	400.0	200.0	100.0	80.00	40.00
		Percent Difference	0.2%	0.2%	0.2%	0.2%	0.2%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	Structural Stiffness at Top of Cantilever (kip/ft)	SASSI2000	9709	1236	155.8	79.87	10.01
		Theoretical	10000	1250	156.3	80.00	10.00
		Percent Difference	-2.9%	-1.1%	-0.3%	-0.2%	0.1%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Structural Stiffness at Top of Wall (kip/ft)	SASSI2000	5.342	10.6838	21.3675	26.7094	53.4188
		Theoretical	5.342	10.6838	21.3675	26.7094	53.4188
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Maximum Percent Difference			2.9%	1.1%	0.3%	0.2%	0.2%

Table 03.07.02-29.24

First Natural Frequency - 3-D Solid Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	First Natural Frequency at Top of Cantilever (Hz)	SASSI2000	12.769	9.033	6.372	5.713	4.028
		Theoretical	12.767	9.028	6.383	5.710	4.037
		Percent Difference	0.0%	0.1%	-0.2%	0.1%	-0.2%
Cantilever Subjected to Harmonic Out-of-Plane Horizontal (Y) Force	First Natural Frequency (Hz)	SASSI2000	31.299	11.206	3.979	2.856	1.001
		Theoretical	31.917	11.285	3.990	2.855	1.009
		Percent Difference	-1.9%	-0.7%	-0.3%	0.1%	-0.8%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	First Natural Frequency (Hz)	SASSI2000	2.124	2.075	2.148	2.100	2.051
		Theoretical	2.087	2.087	2.087	2.087	2.087
		Percent Difference	1.8%	-0.5%	3.0%	0.6%	-1.7%
Maximum Percent Difference			1.9%	0.7%	3.0%	0.6%	1.7%

Table 03.07.02-29.25

Maximum Element Stresses – 2-D Plane Strain Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Vertical Stress at Bottom of Cantilever (kip/ft^2)	SASSI2000	0.100	0.100	0.100	0.100	0.100
		Theoretical	0.100	0.100	0.100	0.100	0.100
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
	Horizontal Stress at Bottom of Cantilever (kip/ft^2)	SASSI2000	0.020	0.020	0.020	0.020	0.020
		Theoretical	0.020	0.020	0.020	0.020	0.020
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Shear Stress at Bottom of Wall (kip/ft^2)	SASSI2000	0.020	0.010	0.005	0.004	0.002
		Theoretical	0.020	0.010	0.005	0.004	0.002
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Maximum Percent Difference			0.0%	0.0%	0.0%	0.0%	0.0%

Table 03.07.02-29.26

Structural Stiffnesses - 2-D Plane Strain Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	Structural Stiffness at Top of Cantilever (kip/ft)	SASSI2000	3.223	1.611	0.806	0.644	0.322
		Theoretical	3.125	1.563	0.781	0.625	0.313
		Percent Difference	3.1%	3.1%	3.1%	3.1%	3.1%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	Structural Stiffness at Top of Wall (kip/ft)	SASSI2000	1.068	2.137	4.274	5.342	10.684
		Theoretical	1.068	2.137	4.274	5.342	10.684
		Percent Difference	0.0%	0.0%	0.0%	0.0%	0.0%
Maximum Percent Difference			3.1%	3.1%	3.1%	3.1%	3.1%

Table 03.07.02-29.27

First Natural Frequency - Thick Shell Model							
			Aspect Ratio				
			1:1	2:1	4:1	5:1	10:1
Cantilever Subjected to Harmonic Axial Vertical (Z) Force	First Natural Frequency at Top of Cantilever (Hz)	SASSI2000	16.528	11.670	8.252	7.373	5.225
		Theoretical	15.959	11.285	7.979	7.137	5.047
		Percent Difference	3.6%	3.4%	3.4%	3.3%	3.5%
Shear Wall Subjected to Harmonic In-Plane Horizontal (X) Force	First Natural Frequency (Hz)	SASSI2000	9.326	9.326	9.326	9.326	9.326
		Theoretical	9.331	9.331	9.331	9.331	9.331
		Percent Difference	-0.1%	-0.1%	-0.1%	-0.1%	-0.1%
Maximum Percent Difference			3.6%	3.4%	3.4%	3.3%	3.5%

Table 03.07.02-29.28

Comparison of section cut shear results:

	Expected Results	SAP2000 Results	Difference (%)
From fixed end	Shear (kips)	Shear (kips)	
0"	20.63	20.64	< 1
75"	20.63	20.64	< 1
150"	-9.38	-9.36	< 1
225"	-9.38	-9.36	< 1
300"	9.38	9.36	< 1

Comparison of section cut moment results:

	Expected Results	SAP2000 Results	Difference (%)
From fixed end	Moment (kips-in)	Moment (kips-in)	
0"	-1688	-1691	< 1
75"	-140.6	-143.0	2
150"	1406	1405	< 1
225"	703.1	702.3	< 1
300"	0	0	0

In addition, the section axial force at every section is 30 kips, as expected.

Table 03.07.02-29.29

Comparison of modal frequency results:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Mode No.	Frequency (Hz)	Frequency (Hz)	
1	9.06037	9.071	< 1
2	18.8171	18.803	< 1
3	26.5053	26.427	< 1
4	35.1965	34.980	< 1
5	35.9126	35.853	< 1

Comparison of plate center displacement results:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Minimum Displacement (in)	-0.003120	-0.003118	< 1
Maximum Displacement (in)	0.002915	0.002873	1.5

Comparison of support reaction results:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Minimum Reaction at Corner (lbs)	-132.348	-123.72	7.0
Maximum Reaction at Corner (lbs)	87.208	80.30	8.6
Minimum Reaction at Short side Center (lbs)	-41.40	-41.21	< 1
Maximum Reaction at Short side Center (lbs)	62.62	63.26	1.0
Minimum Reaction at Long side Center (lbs)	-35.79	-36.28	1.4
Maximum Reaction at Long side Center (lbs)	44.40	45.31	2.1

Table 03.07.02-29.30

Comparison of modal frequency results:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Mode No.	Frequency (Hz)	Frequency (Hz)	
1	8.35560	8.332	< 1
2	9.62216	9.604	< 1
3	10.5209	10.46	< 1
4	11.9669	11.96	< 1
5	13.1260	13.08	< 1
6	15.0105	14.86	1
7	18.2422	18.19	< 1
8	19.8132	19.65	< 1
9	20.3556	20.17	< 1
10	24.3296	24.39	< 1
11	24.7822	24.66	< 1
12	25.9027	25.78	< 1
13	26.9772	26.76	< 1
14	27.5210	27.30	< 1
15	29.9460	29.43	2
16	30.5292	30.46	< 1
17	31.7276	31.26	2
18	31.8323	31.61	< 1
19	35.5686	35.16	1
20	36.1959	35.91	< 1
21	36.9213	36.56	1
22	37.9936	37.93	< 1
23	41.8152	41.45	1
24	43.6486	42.83	2
25	43.7251	42.84	2
26	43.9963	43.40	1
27	44.3450	43.98	< 1
28	44.5701	44.55	< 1

Tube displacement results at top center of long side:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Minimum Displacement (in)	-0.09197	-0.092618	< 1
Maximum Displacement (in)	0.1051	0.106377	1

Table 03.07.02-29.31

Support reaction results at base center of long side:

	Expected Results (ANSYS)	SAP2000 Results	Difference (%)
Minimum Reaction (lbs)	-582.7	-623.79	7.1
Maximum Reaction (lbs)	551.8	593.46	7.6
Minimum Moment (lbs-in)	-40280	-43545.49	8.1
Maximum Moment (lbs-in)	41830	41992.80	< 1

Table 03.07.02-29.32
Comparison of Iterated Strain Dependent Shear Modulus and Damping Values
(SHAKE2000 – 116 Layer Validation Problem 1)

validation-problem1					validation-problem1-116 layers						
Layer No.	Layer Thickness (ft)	Mid Layer Depth (ft)	Shear Modulus (ksf)	Damping	Layer No.	Layer Thickness (ft)	Mid Layer Depth (ft)	Shear Modulus (ksf)	Damping	Difference Shear Modulus (%)	Difference Damping (%)
1	7	3.5	4470.4	0.045	4	1	3.5	4471	0.044	0.013%	2.22%
2	13	13.5	813.7	0.112	14	1	13.5	816.8	0.111	0.381%	0.89%
3	10	25	3058.9	0.089	25	1	24.5	3067.3	0.088	0.054%	0.56%
					See Note 1		25	3060.55	0.0885		
					26	1	25.5	3053.8	0.089		
4	12	36	393.4	0.191	36	1	35.5	401.5	0.189	0.483%	0.26%
					See Note 1		36	395.3	0.1905		
					37	1	36.5	389.1	0.192		
5	15	49.5	648.4	0.177	50	1	49.5	649.1	0.177	0.108%	0.00%
6	14	64	3007.3	0.114	64	1	63.5	3013	0.113	0.035%	0.44%
					See Note 1		64	3008.35	0.1135		
					65	1	64.5	3003.7	0.114		
7	20	81	3521.8	0.117	81	1	80.5	3530.5	0.116	0.104%	0.43%
					See Note 1		81	3525.45	0.1165		
					82	1	81.5	3520.4	0.117		
8	25	103.5	842.6	0.193	104	1	103.5	845.1	0.193	0.297%	0.00%

Note 1) The values reported are the average of the values for the above and below layers.

Table 03.07.02-29.33
Comparison of Maximum Accelerations
(SHAKE2000 – 116 Layer Validation Problem 1)

validation-problem1			validation-problem1-116 layers			
Layer No.	Top Layer Depth (ft)	Max. Accel. (g)	Layer No.	Top Layer Depth (ft)	Max. Accel. (g)	Difference Max. Accel. (%)
1	0	0.2626	1	0	0.26067	0.735%
2	7	0.26178	8	7	0.25992	0.711%
3	20	0.25081	21	20	0.24969	0.447%
4	30	0.24765	31	30	0.24666	0.400%
5	42	0.23005	43	42	0.22978	0.117%
6	57	0.23926	58	57	0.23811	0.481%
7	71	0.23325	72	71	0.2323	0.407%
8	91	0.20981	92	91	0.20865	0.553%
9	116	0.178	117	116	0.178	0.000%

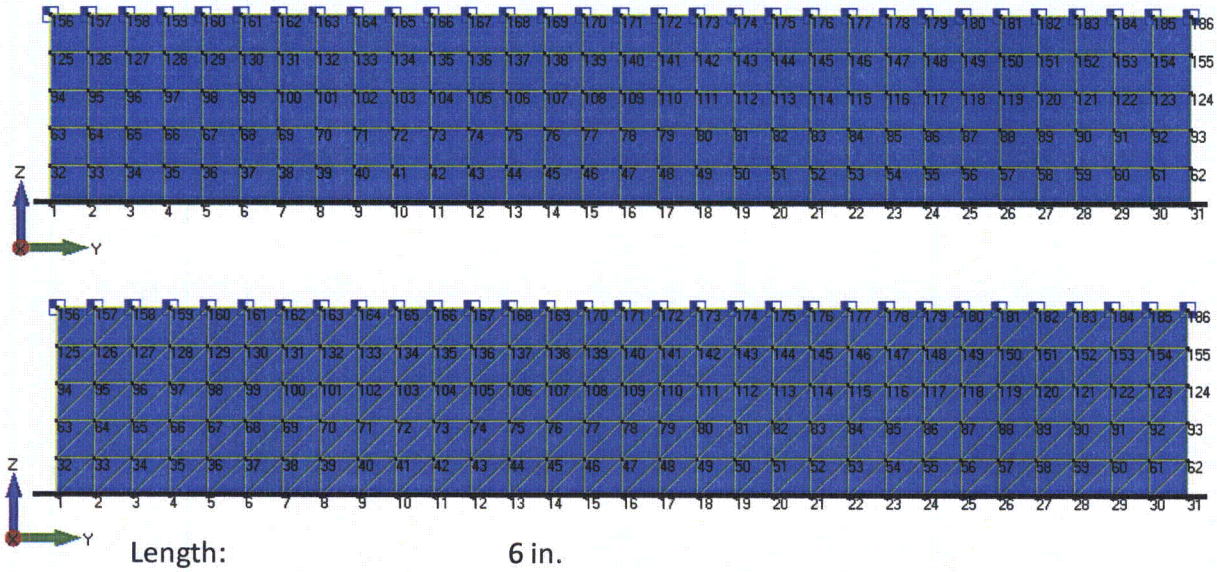
Table 03.07.02-29.34
Comparison of Iterated Strain Dependent Shear Modulus and Damping Values
(SHAKE2000 – 116 Layer Validation Problem 2)

validation-problem2					validation-problem2-116 layers						
Layer No.	Layer Thickness (ft)	Mid Layer Depth (ft)	Shear Modulus (ksf)	Damping	Layer No.	Layer Thickness (ft)	Mid Layer Depth (ft)	Shear Modulus (ksf)	Damping	Difference Shear Modulus (%)	Difference Damping (%)
1	7	3.5	4509.3	0.04	4	1	3.5	4509.3	0.04	0.000%	0.00%
2	13	13.5	1157	0.073	14	1	13.5	1154.4	0.073	0.225%	0.00%
3	10	25	3229.5	0.067	25	1	24.5	3231.7	0.066	0.002%	0.75%
					See Note 1		25	3229.55	0.0665		
					26	1	25.5	3227.4	0.067		
4	12	36	952.9	0.108	36	1	35.5	957.4	0.108	0.037%	0.46%
					See Note 1		36	952.55	0.1085		
					37	1	36.5	947.7	0.109		
5	15	49.5	1479.7	0.099	50	1	49.5	1476.8	0.099	0.196%	0.00%
6	14	64	3380.3	0.077	64	1	63.5	3385.1	0.076	0.065%	0.65%
					See Note 1		64	3382.5	0.0765		
					65	1	64.5	3377.9	0.077		
7	20	81	4016.3	0.076	81	1	80.5	4019	0.076	0.021%	0.00%
					See Note 1		81	4017.15	0.076		
					82	1	81.5	4015.3	0.076		
8	25	103.5	2528.2	0.088	104	1	103.5	2530.6	0.088	0.095%	0.00%

Note 1) The values reported are the average of the values for the above and below layers.

Table 03.07.02-29.35
Comparison of Maximum Accelerations
(SHAKE2000 – 116 Layer Validation Problem 2)

validation-problem2			validation-problem2-116 layers			
Layer No.	Top Layer Depth (ft)	Max. Accel. (g)	Layer No.	Top Layer Depth (ft)	Max. Accel. (g)	Difference Max. Accel. (%)
1	0	0.16147	1	0	0.16147	0.000%
2	7	0.15306	8	7	0.15304	0.013%
3	20	0.1652	21	20	0.16875	2.149%
4	30	0.14368	31	30	0.14749	2.652%
5	42	0.2446	43	42	0.2474	1.145%
6	57	0.36329	58	57	0.35371	2.637%
7	71	0.32167	72	71	0.3112	3.255%
8	91	0.41379	92	91	0.40235	2.765%
9	116	0.83661	117	116	0.83035	0.748%



Length:	6 in.
Height:	1 in.
Thickness:	0.1 in.
Young's Modulus:	3,000 psi
Poisson's Ratio:	0.3
Density of Wall:	0 (lb-s^2/in)/in^3
Total Mass at Top of Wall:	0.2 lbf-sec^2/in

Figure 03.07.02-29.1: Cantilever Thick Shell Finite Element Models for In-Plane Loading, Node Numbers are Shown, Rectangular Element Model (top) and Triangular Element Model (bottom)

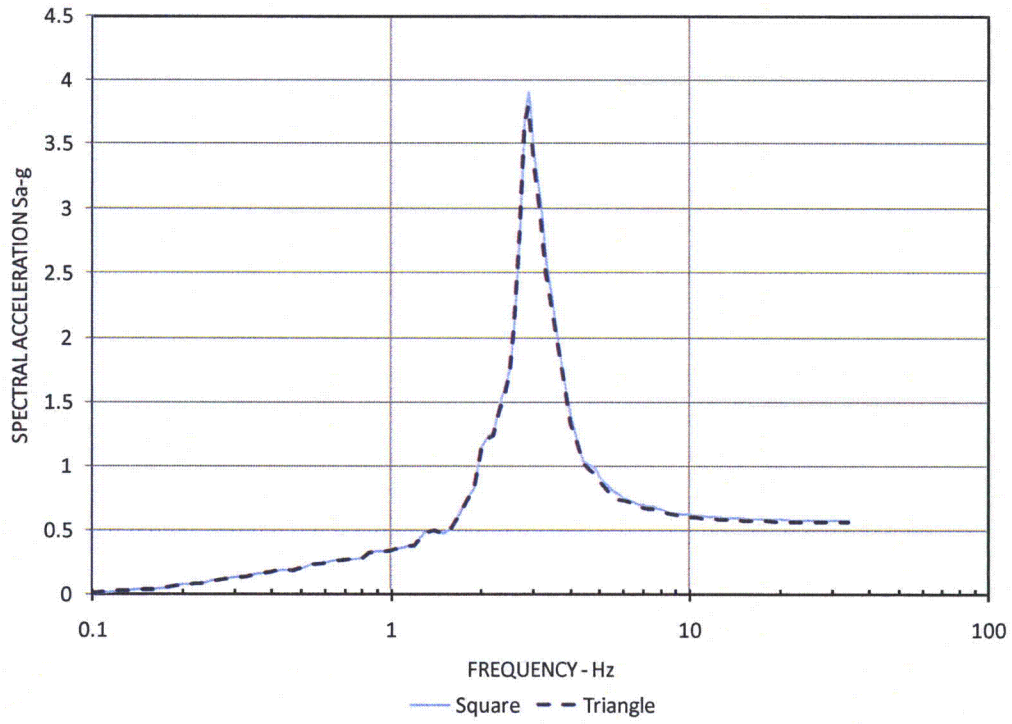


Figure 03.07.02-29.2: Acceleration Response Spectrum for Thick Shell Element Model for In-Plane Loading – Node 158

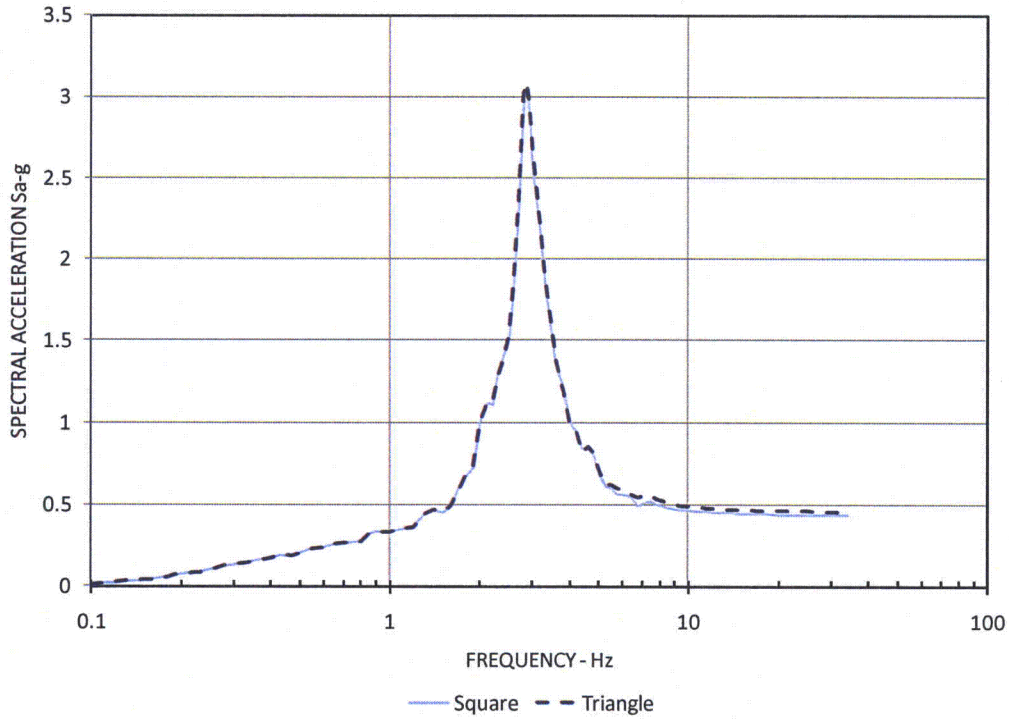


Figure 03.07.02-29.3: Acceleration Response Spectrum for Thick Shell Element Model for In-Plane Loading – Node 171

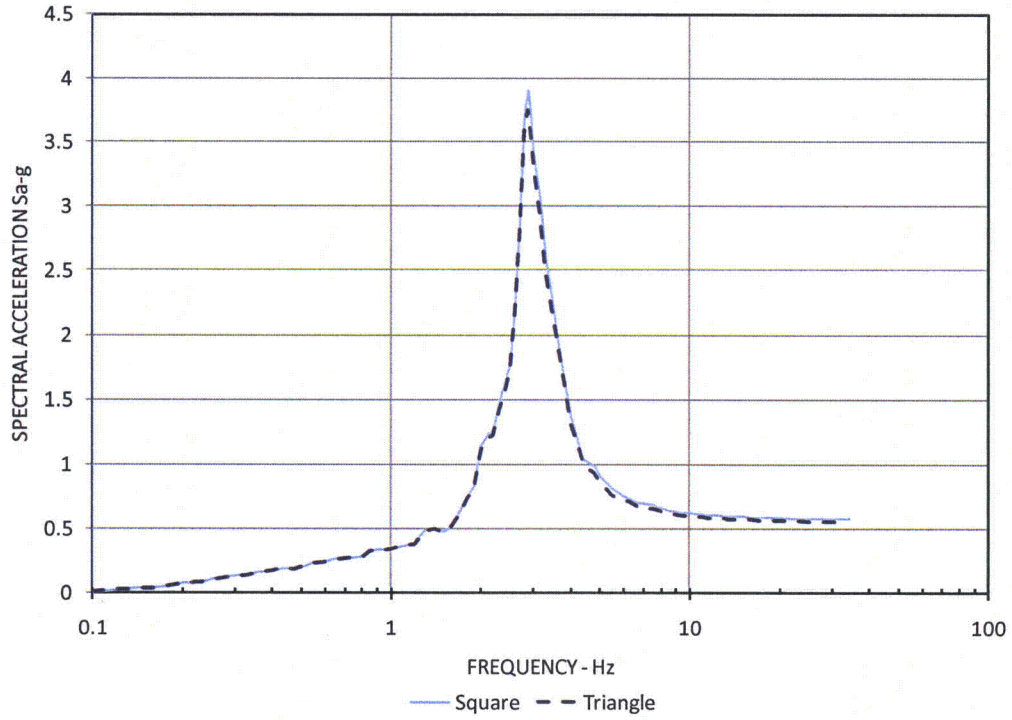


Figure 03.07.02-29.4: Acceleration Response Spectrum for Thick Shell Element Model for In-Plane Loading – Node 184

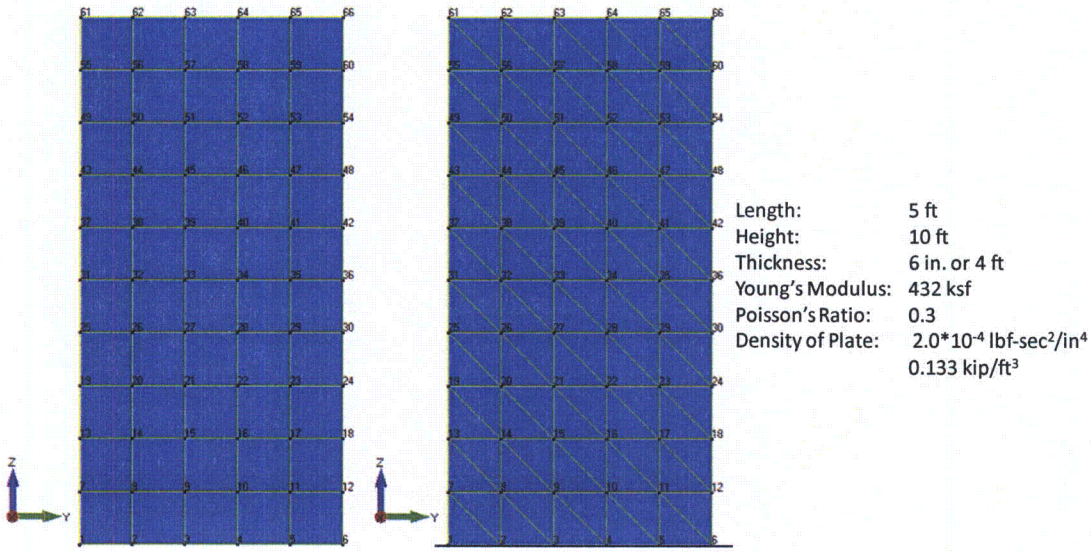


Figure 03.07.02-29.5: Cantilever Thick Shell Finite Element Models for Out-of-Plane Loading, Node Numbers are Shown, Rectangular Element Model (left) and Triangular Element Model (right)

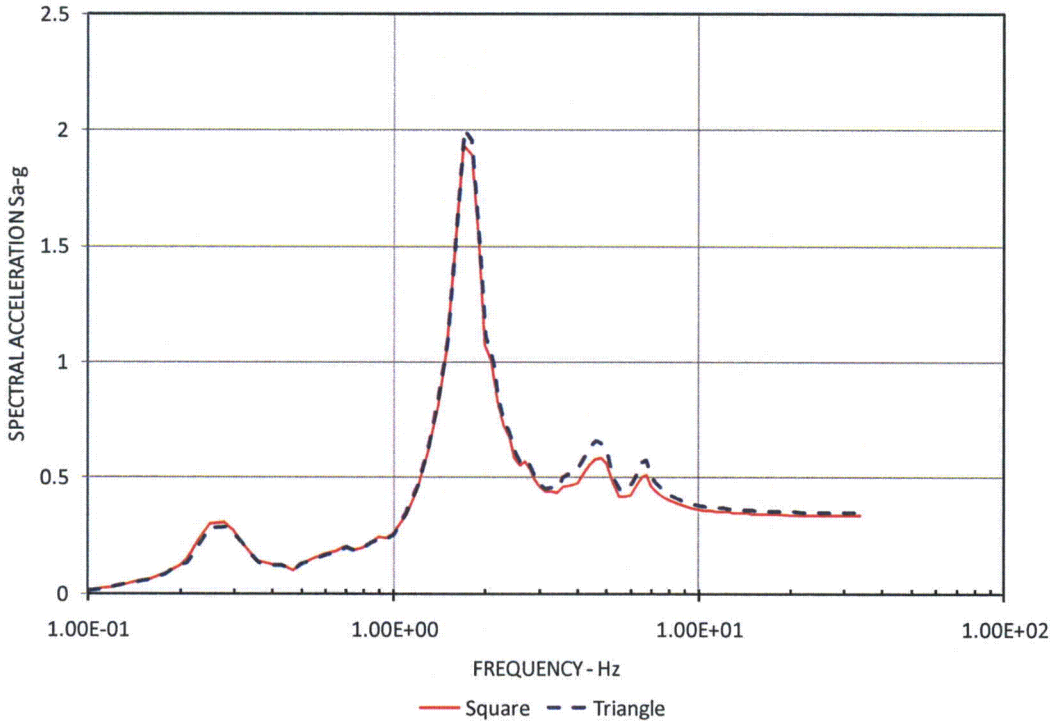


Figure 03.07.02-29.6: Acceleration Response Spectrum for Thick Shell Element Model for Out-of-Plane Loading – Node 31 – 6 in. Wall

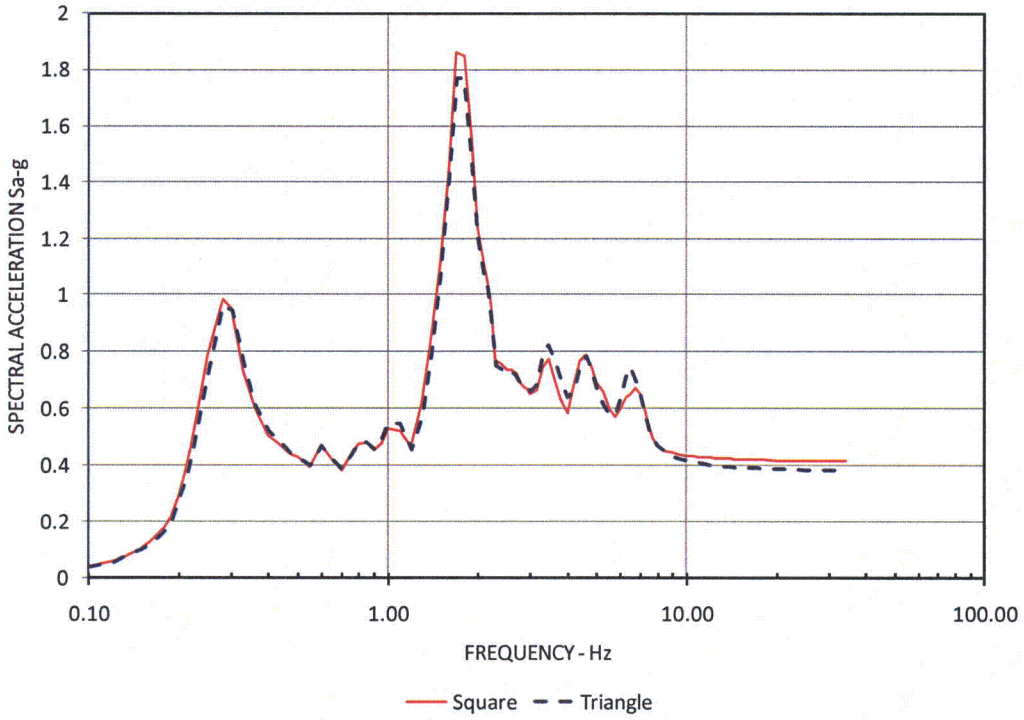


Figure 03.07.02-29.7: Acceleration Response Spectrum for Thick Shell Element Model for Out-of-Plane Loading – Node 61 – 6 in. Wall

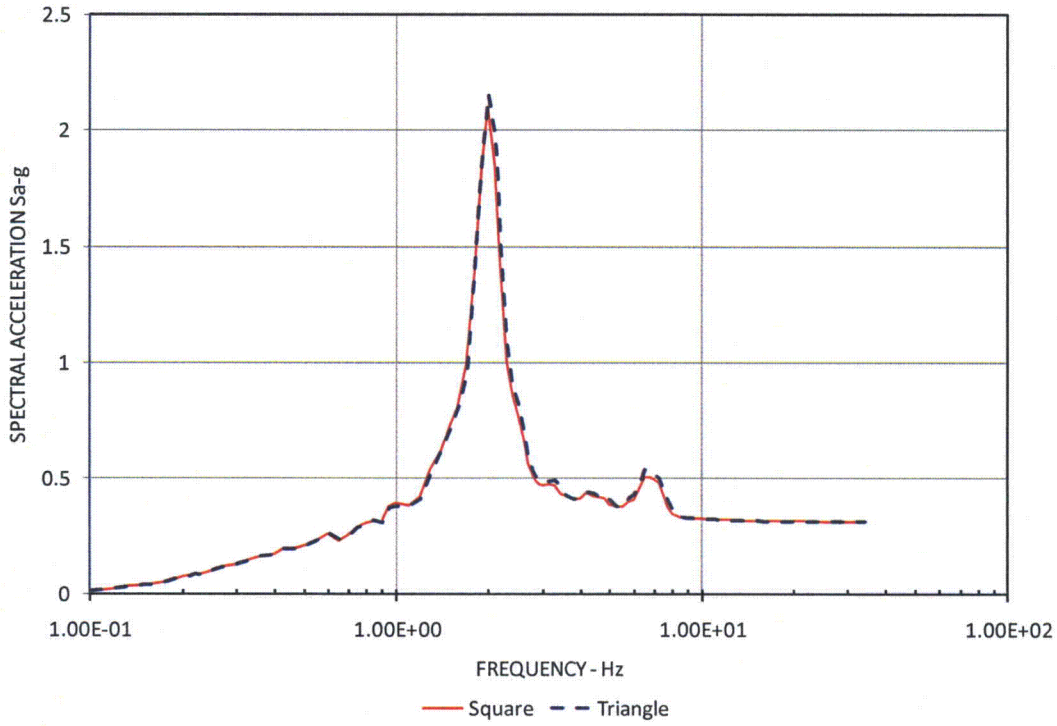


Figure 03.07.02-29.8: Acceleration Response Spectrum for Thick Shell Element Model for Out-of-Plane Loading – Node 31 –4 ft Wall

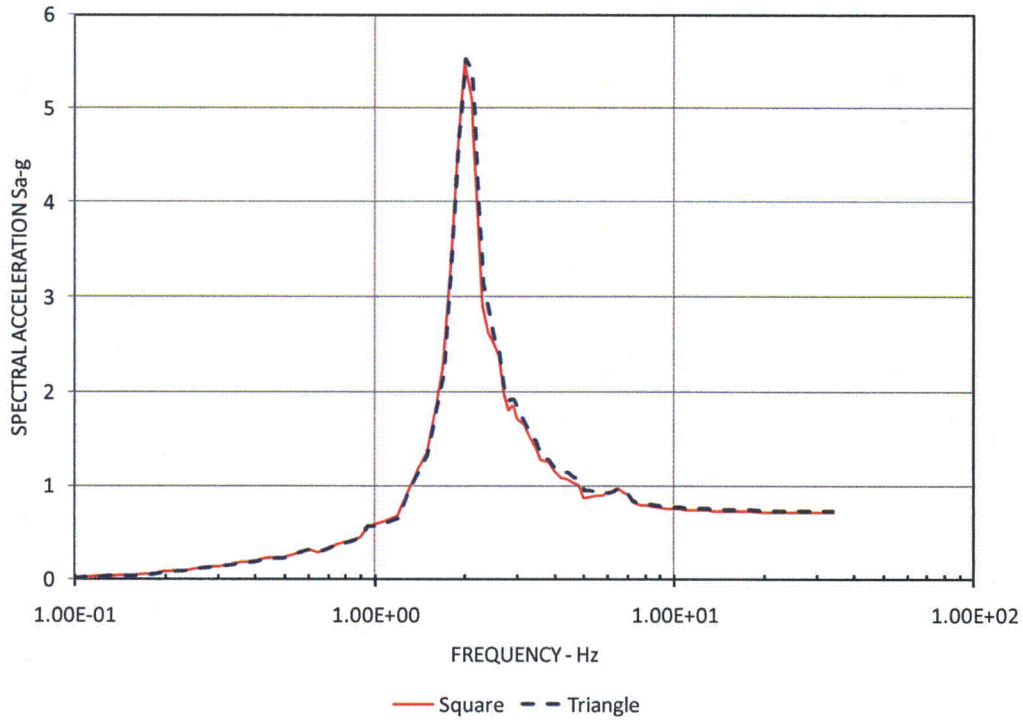


Figure 03.07.02-29.9: Acceleration Response Spectrum for Thick Shell Element Model for Out-of-Plane Loading – Node 61 –4 ft Wall

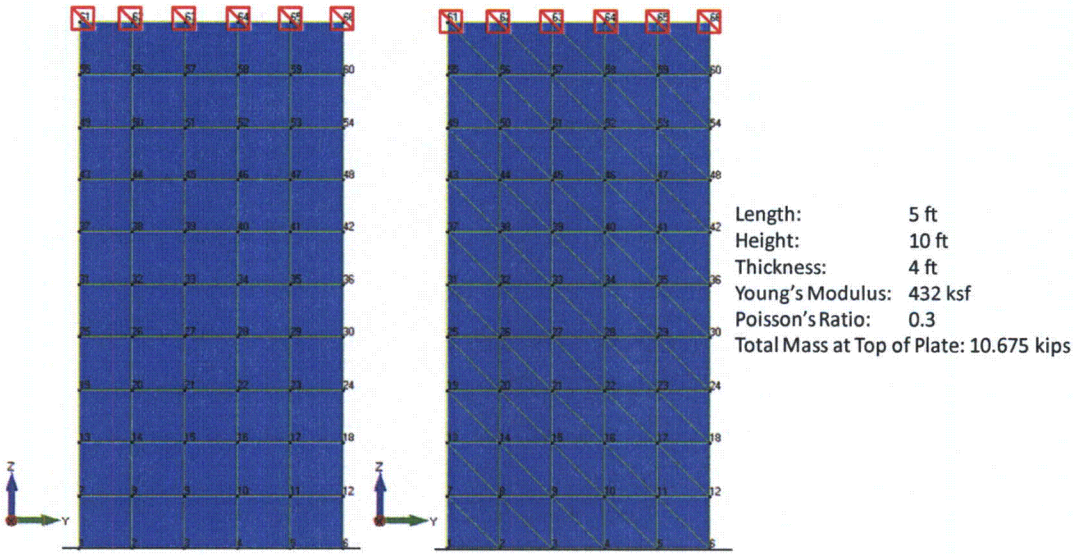


Figure 03.07.02-29.10: Cantilever Thick Shell Finite Element Models for Axial Loading, Node Numbers are Shown, Rectangular Element Model (left) and Triangular Element Model (right)

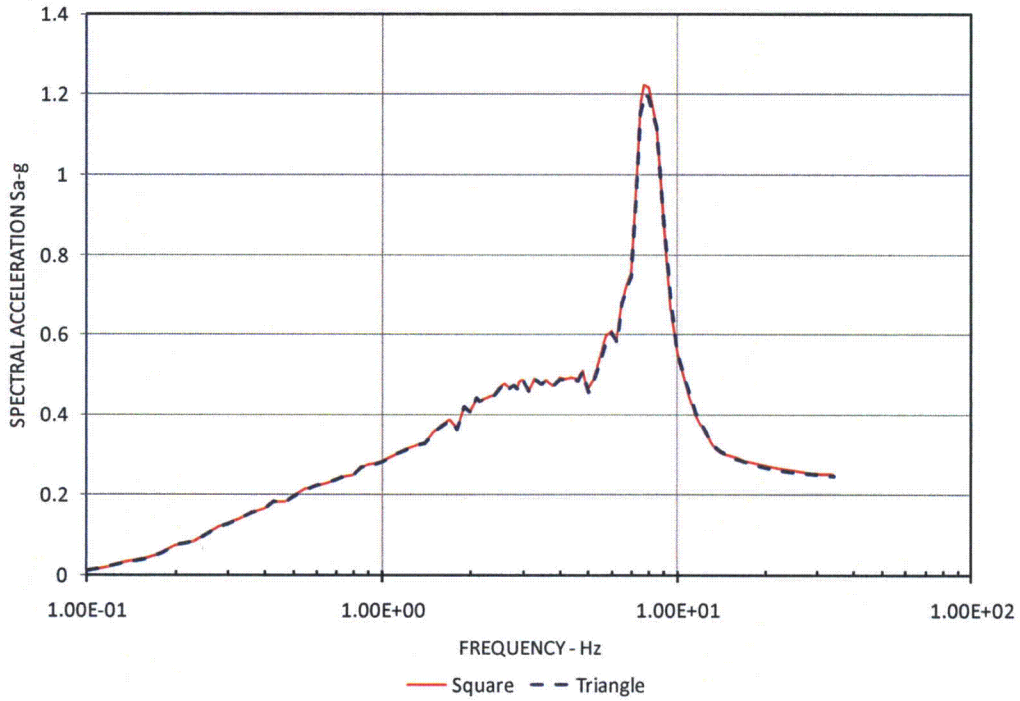


Figure 03.07.02-29.11: Acceleration Response Spectrum for Thick Shell Element Model with Vertical Axial Load – Node 31 –4 ft Wall

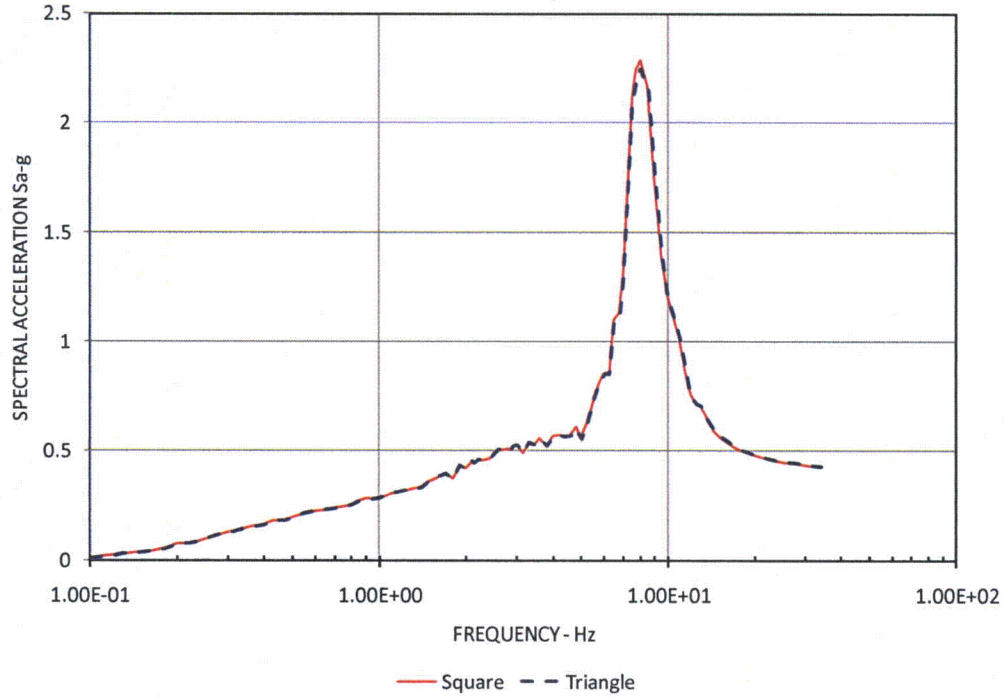


Figure 03.07.02-29.12: Acceleration Response Spectrum for Thick Shell Element Model with Vertical Axial Load – Node 61 –4 ft Wall

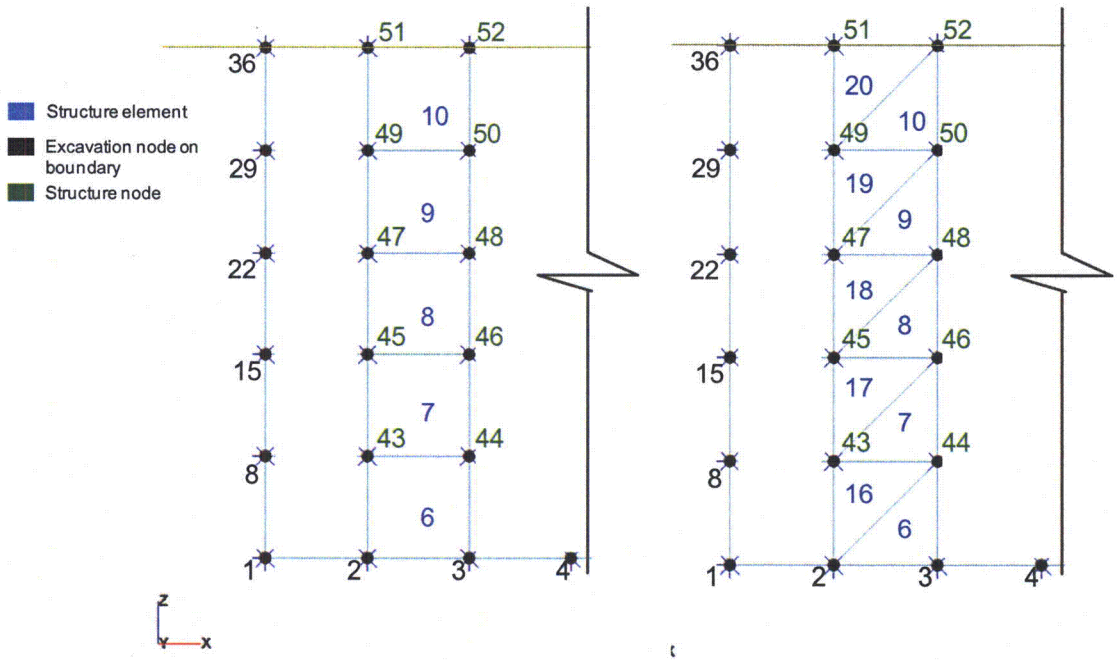


Figure 03.07.02-29.13: SASSI Model - Rectangular Element Structure shown on Left and Triangular Element Structure Shown on Right (Right Edge of Excavation Not Shown)

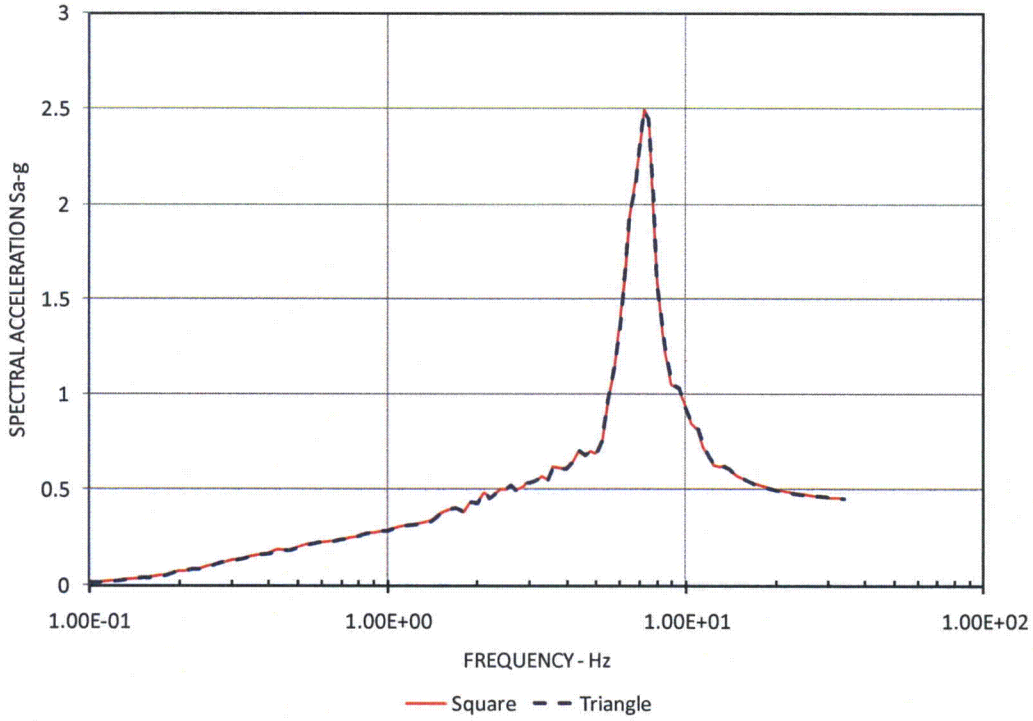


Figure 03.07.02-29.14: Acceleration Response Spectrum for 2-D Plane Strain Element Model with In-Plane Load – Node 52

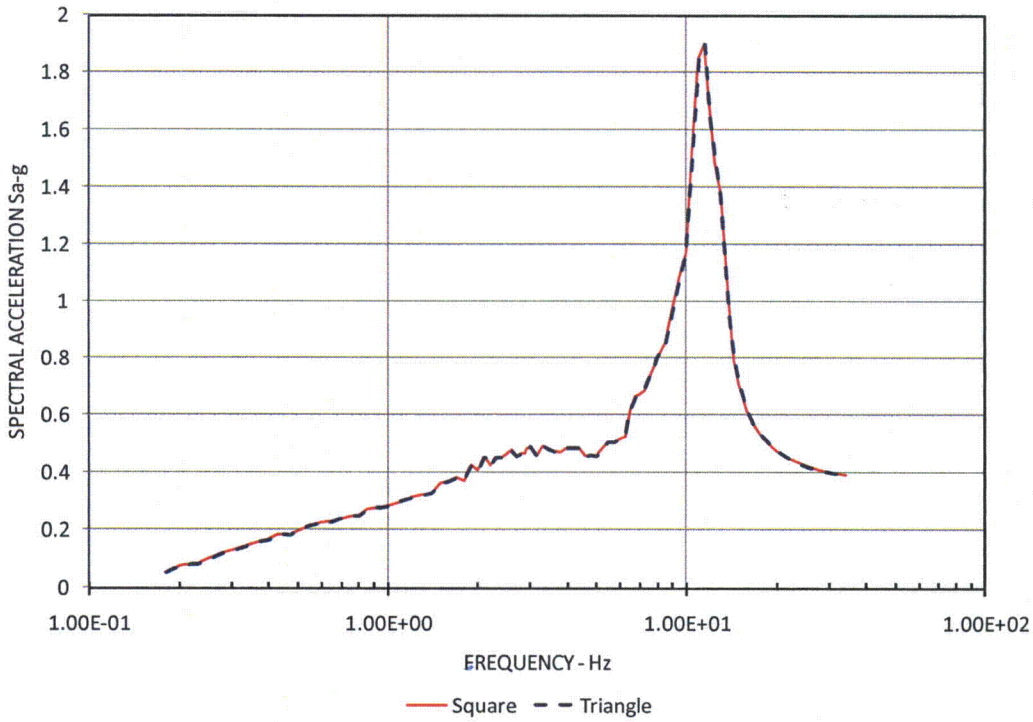


Figure 03.07.02-29.15: Acceleration Response Spectrum for the 2-D Plane Strain Element Model with Vertical Axial Load – Node 52

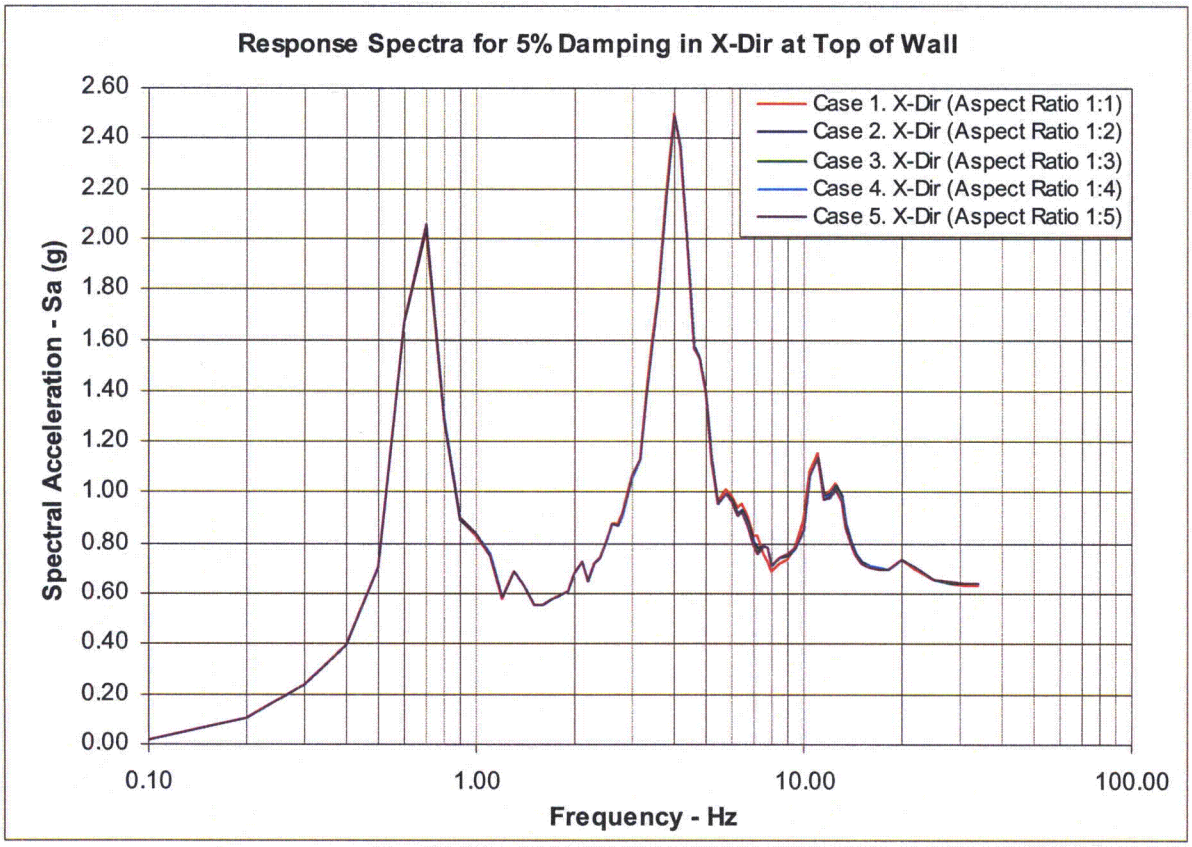


Figure 03.07.02-29.16: Comparison of 5% damped X-Direction Response Spectra at the Top of the Wall for Various aspect Ratios of Thin Plate Elements

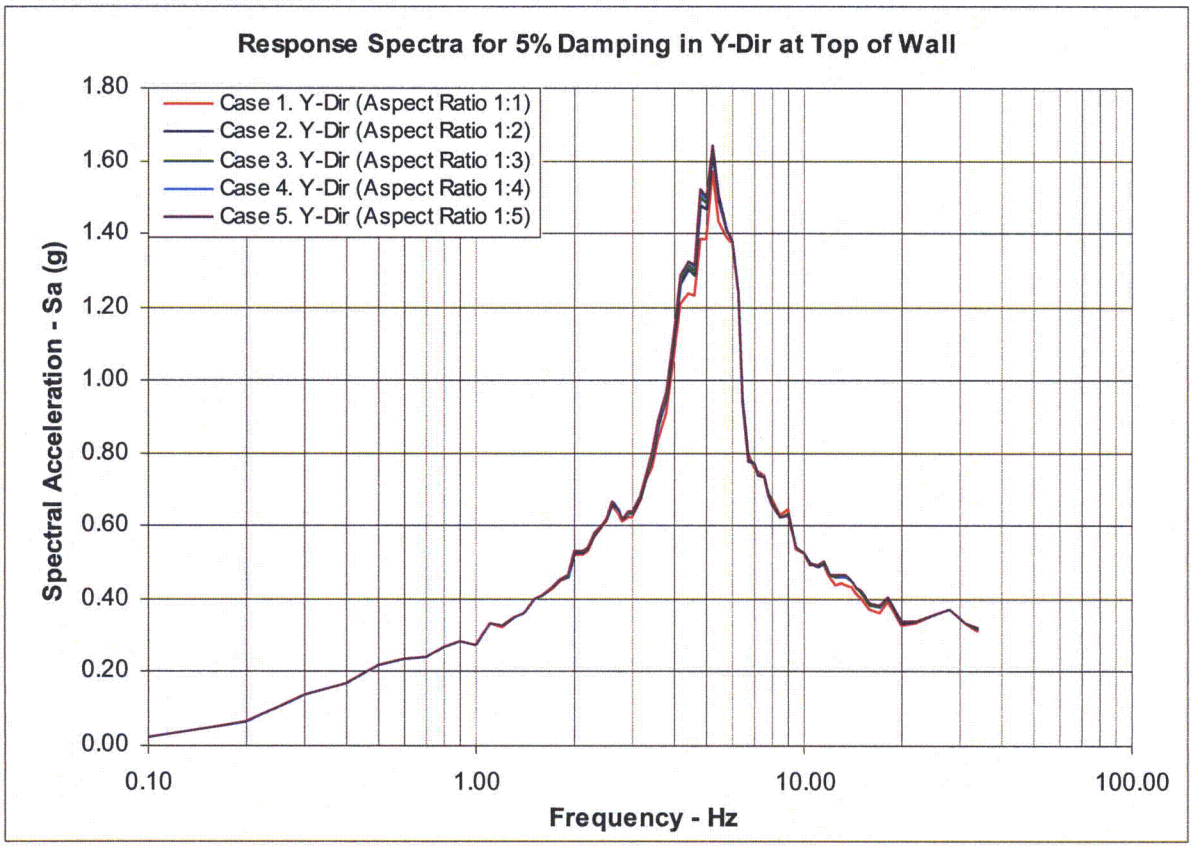


Figure 03.07.02-29.17: Comparison of 5% damped Y-Direction Response Spectra at the Top of the Wall for Various aspect Ratios of Thin Plate Elements

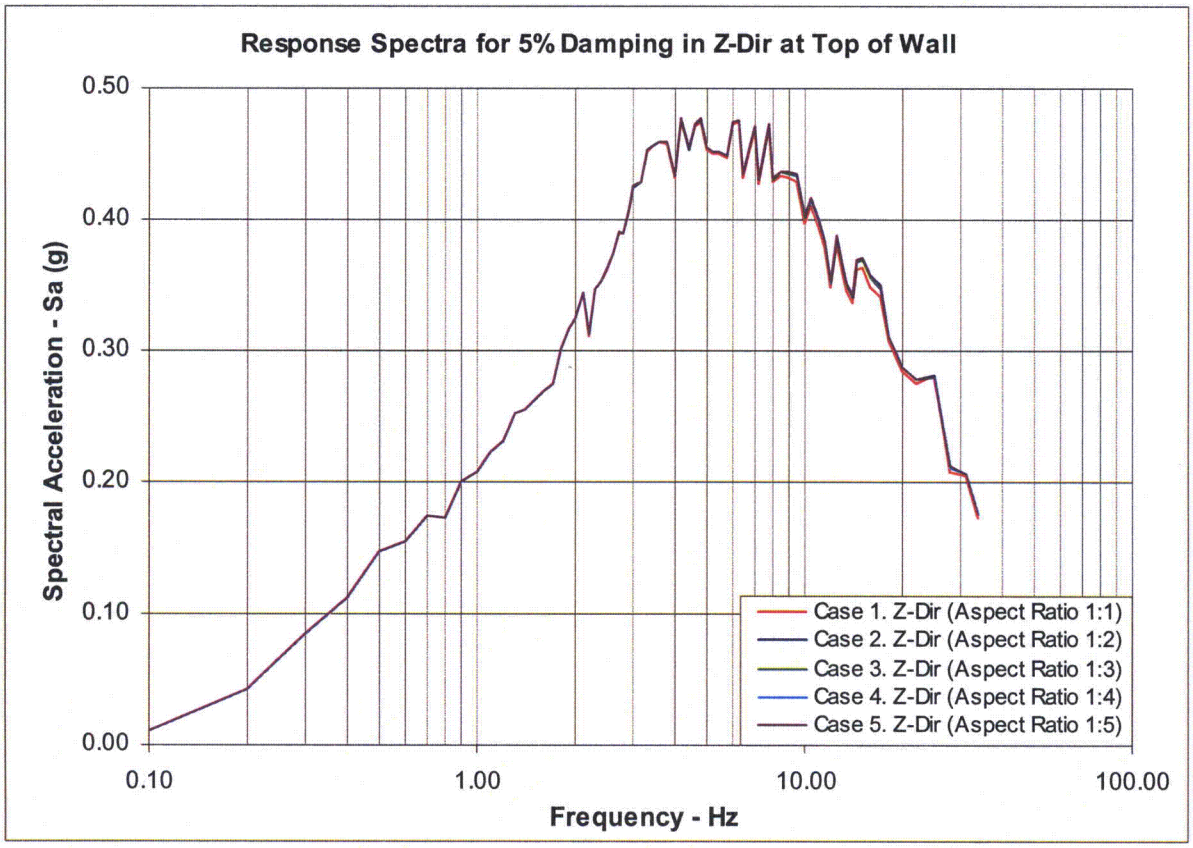


Figure 03.07.02-29.18: Comparison of 5% damped Z-Direction Response Spectra at the Top of the Wall for Various aspect Ratios of Thin Plate Elements

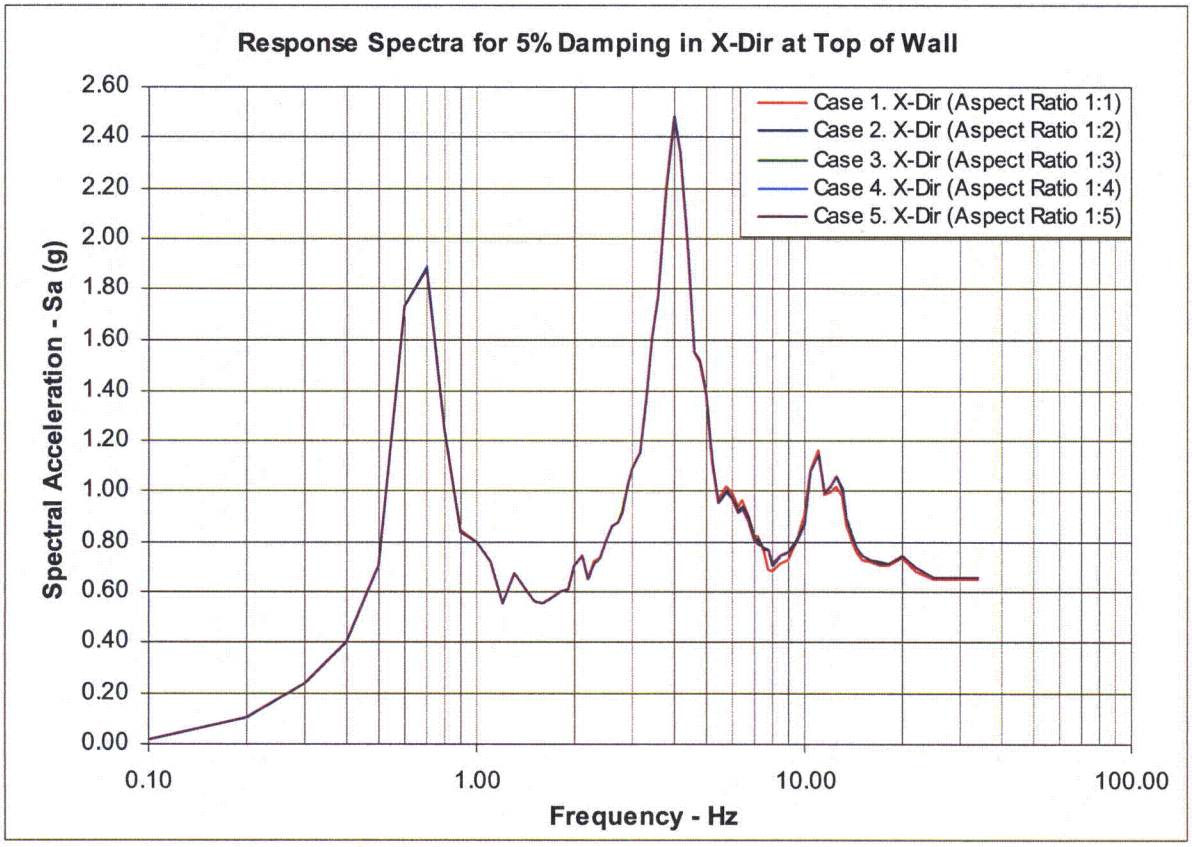


Figure 03.07.02-29.19: Comparison of 5% damped X-Direction Response Spectra at the Top of Wall for Various aspect Ratios of Thick Plate Elements

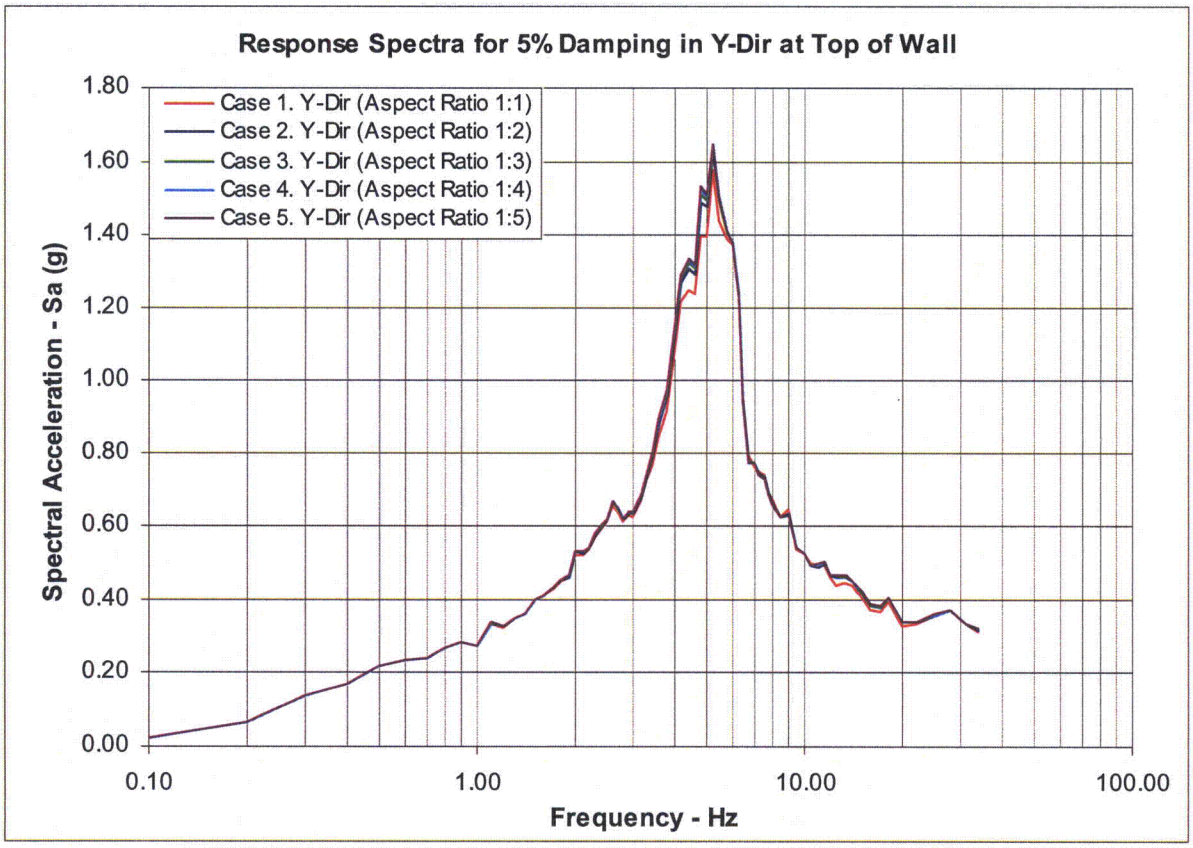


Figure 03.07.02-29.20: Comparison of 5% damped Y-Direction Response Spectra at the Top of Wall for Various aspect Ratios of Thick Plate Elements

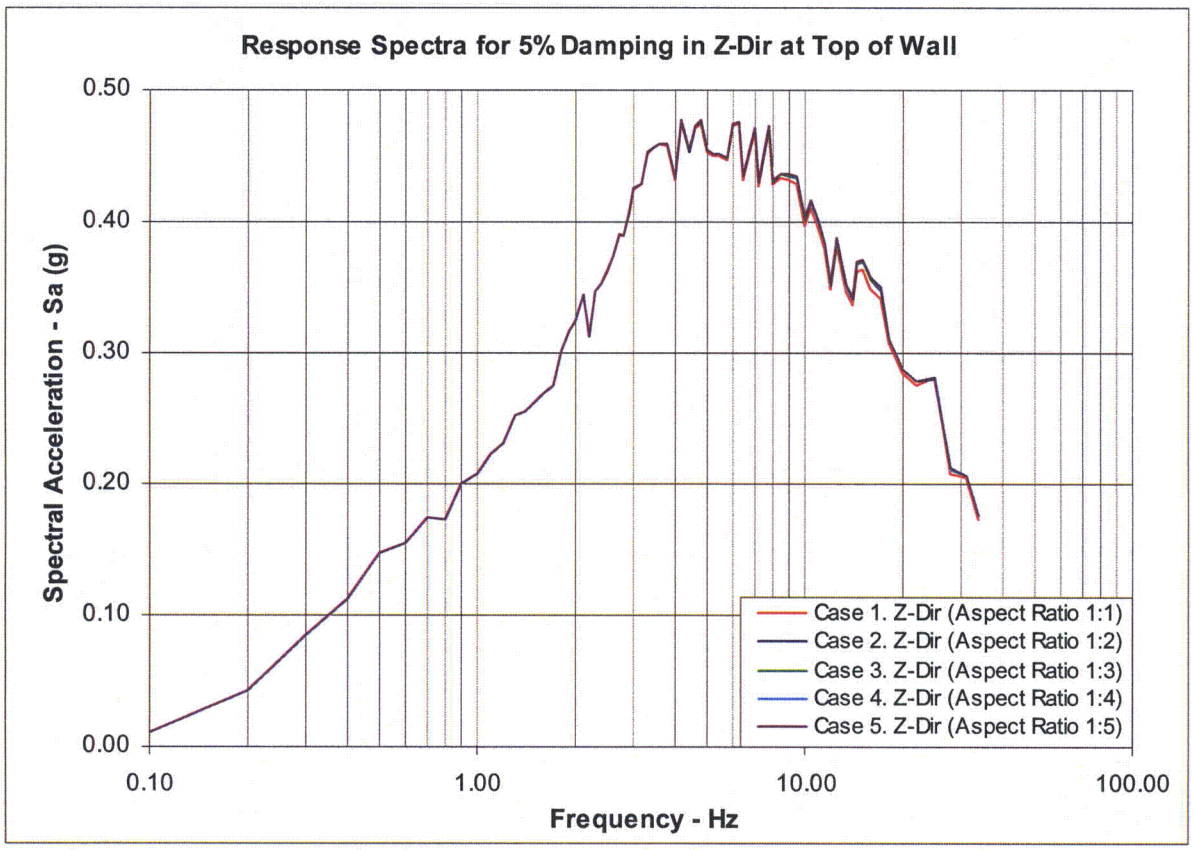


Figure 03.07.02-29.21: Comparison of 5% damped Z-Direction Response Spectra at the Top of Wall for Various aspect Ratios of Thick Plate Elements

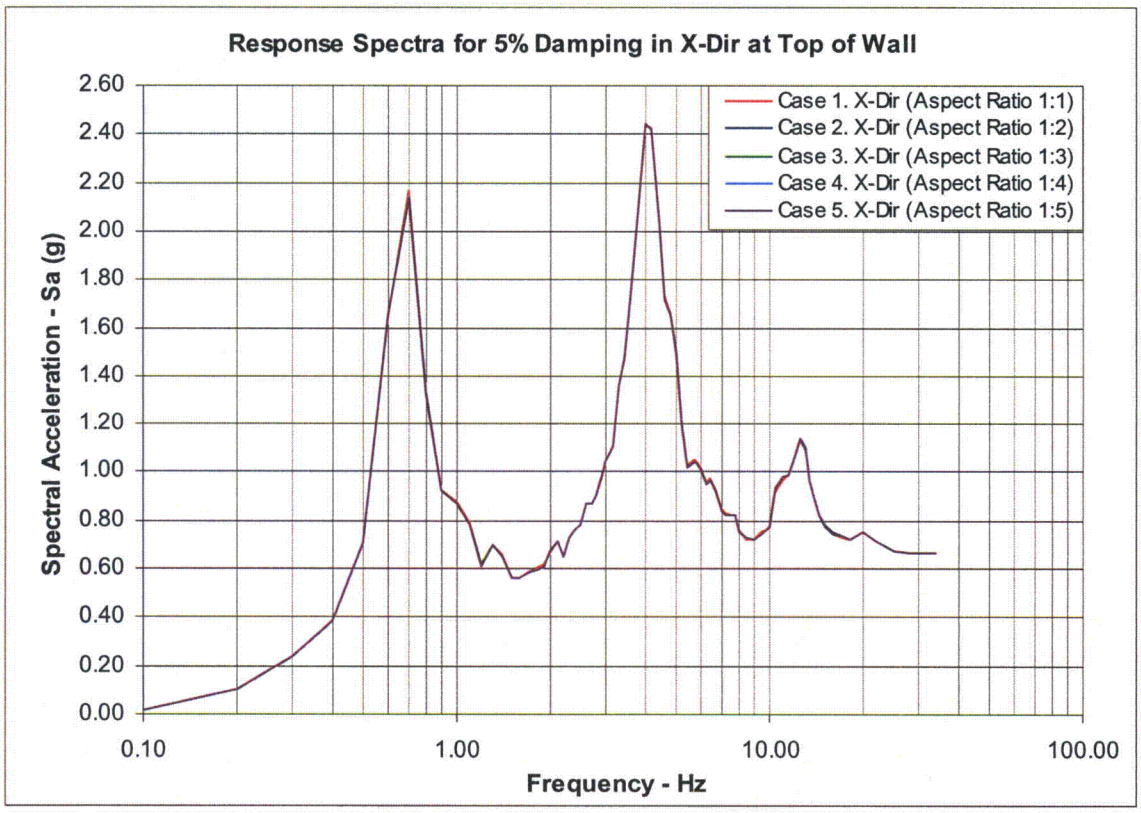


Figure 03.07.02-29.22: Comparison of 5% damped X-Direction Response Spectra at the Top of Wall for Various aspect Ratios of 8-Node Solid Elements

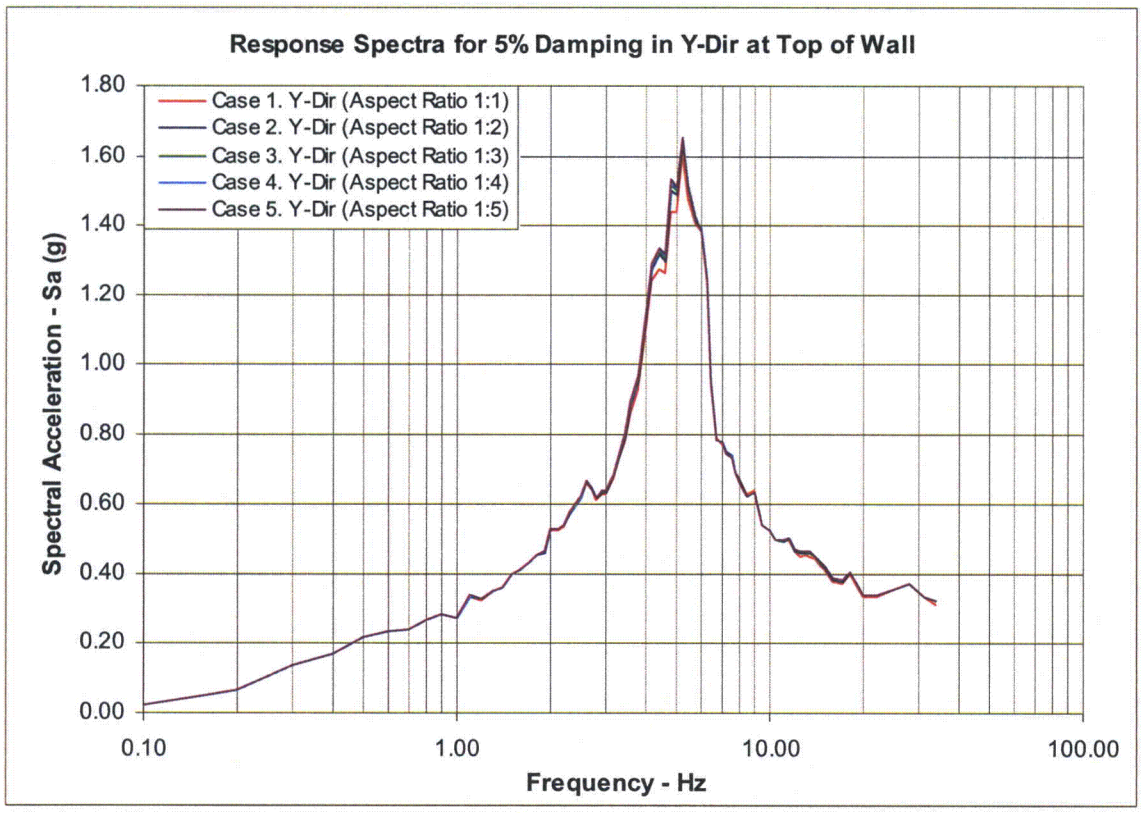


Figure 03.07.02-29.23: Comparison of 5% damped Y-Direction Response Spectra at the Top of Wall for Various aspect Ratios of 8-Node Solid Elements

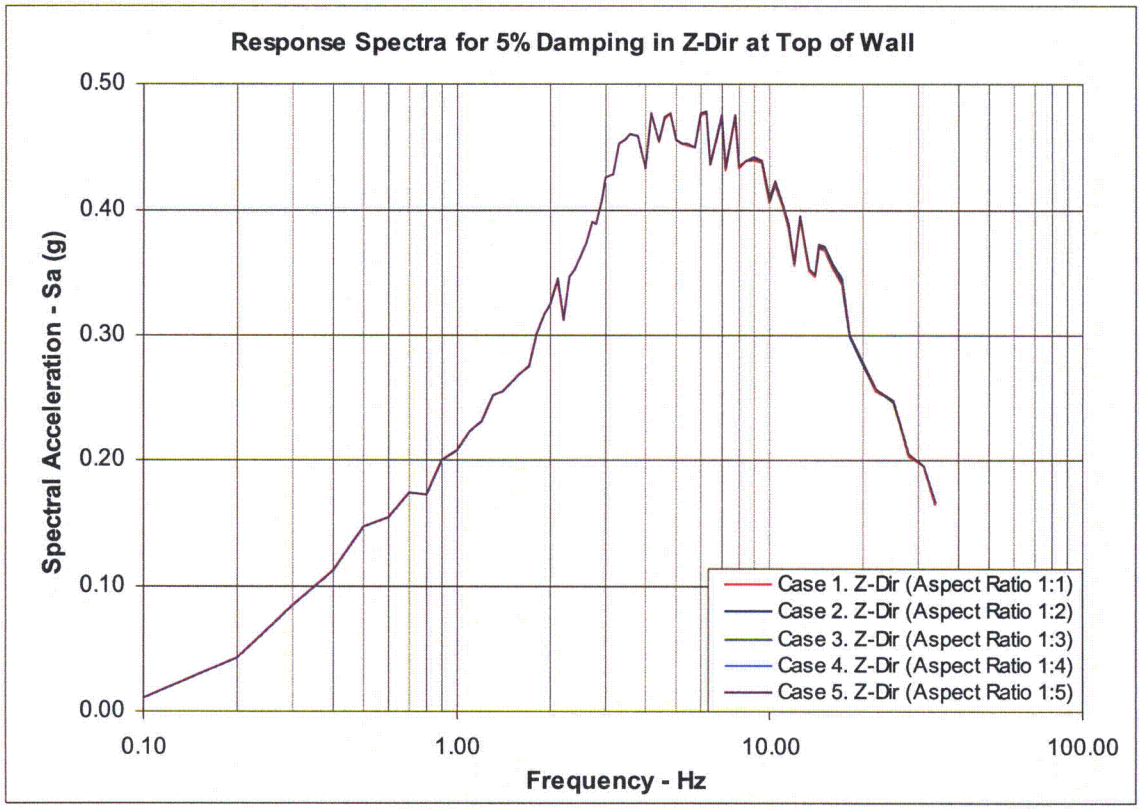


Figure 03.07.02-29.24: Comparison of 5% damped Z-Direction Response Spectra at the Top of Wall for Various aspect Ratios of 8-Node Solid Elements

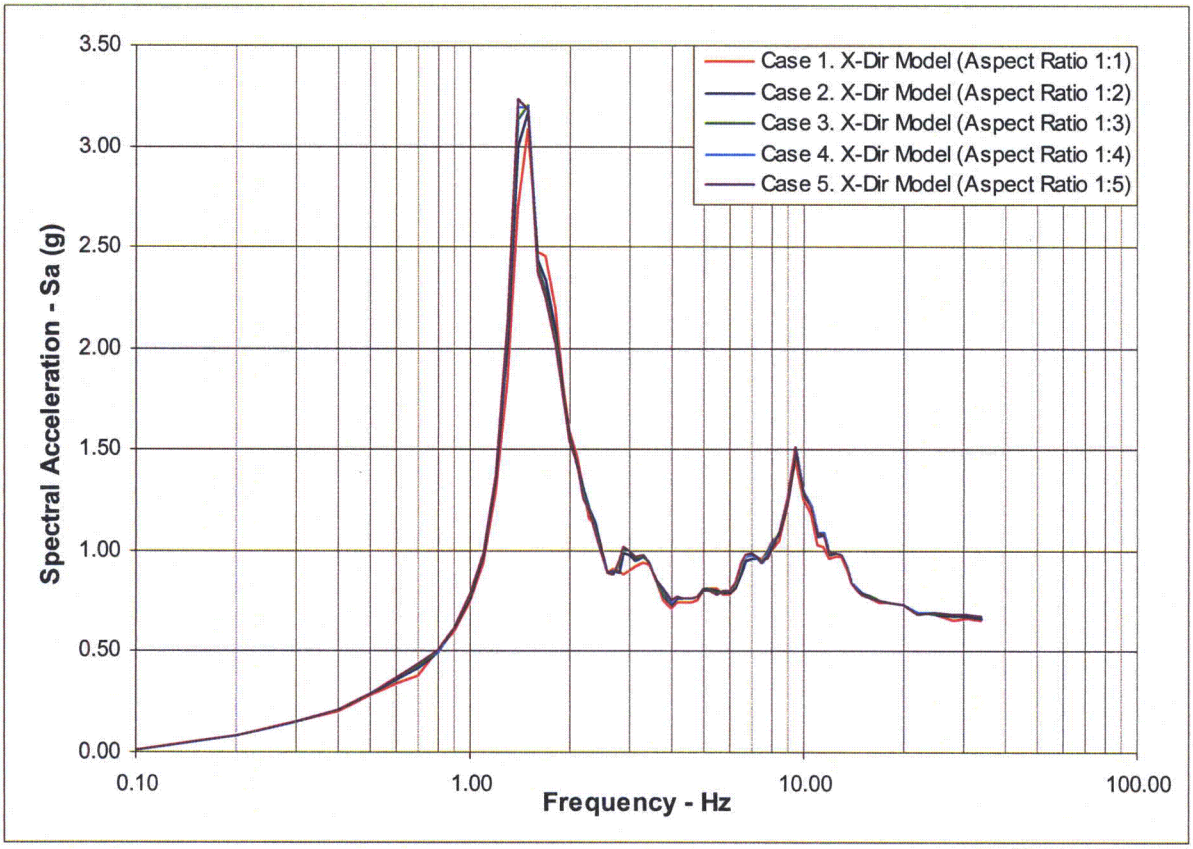


Figure 03.07.02-29.25: Comparison of 5% damped X-Direction Response Spectra at the Top of Wall for Various aspect Ratios of 2-D plane-Strain Elements

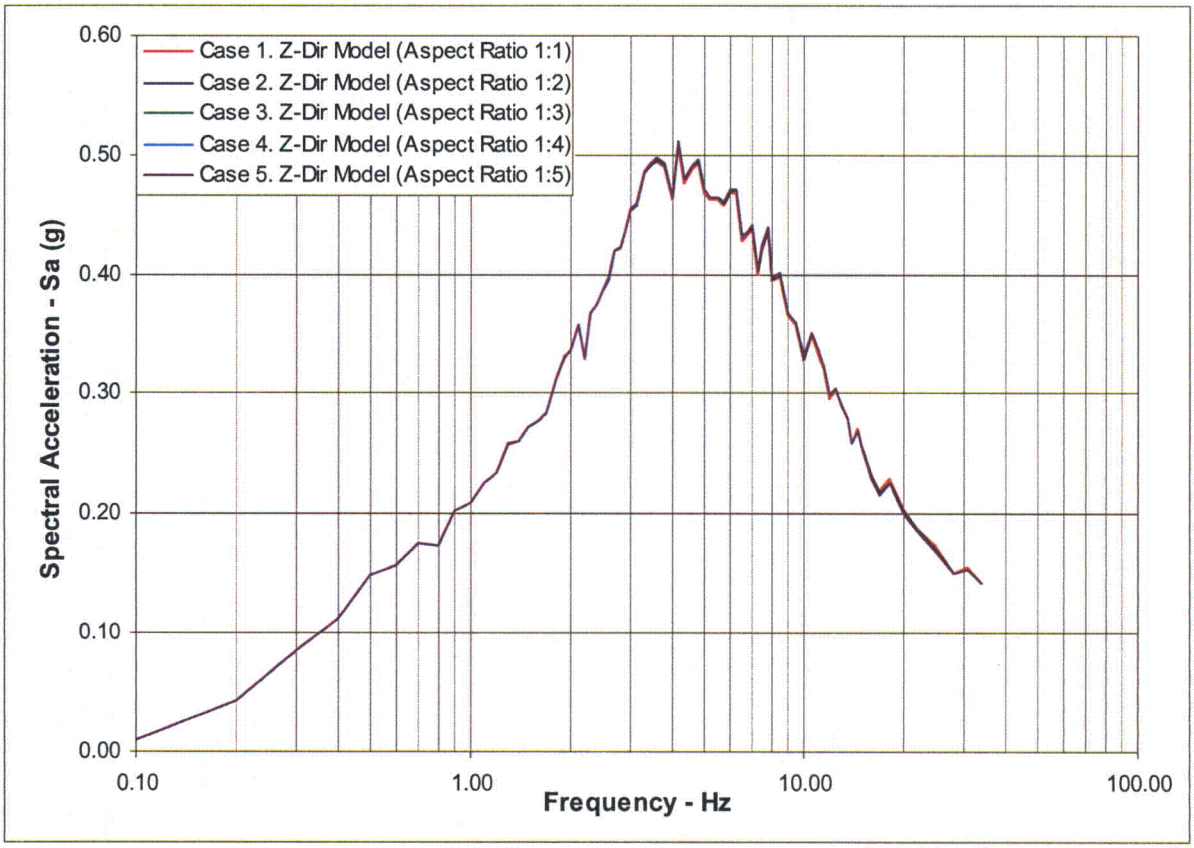


Figure 03.07.02-29.26: Comparison of 5% damped Y-Direction Response Spectra at the Top of Wall for Various aspect Ratios of 2-D plane-Strain Elements

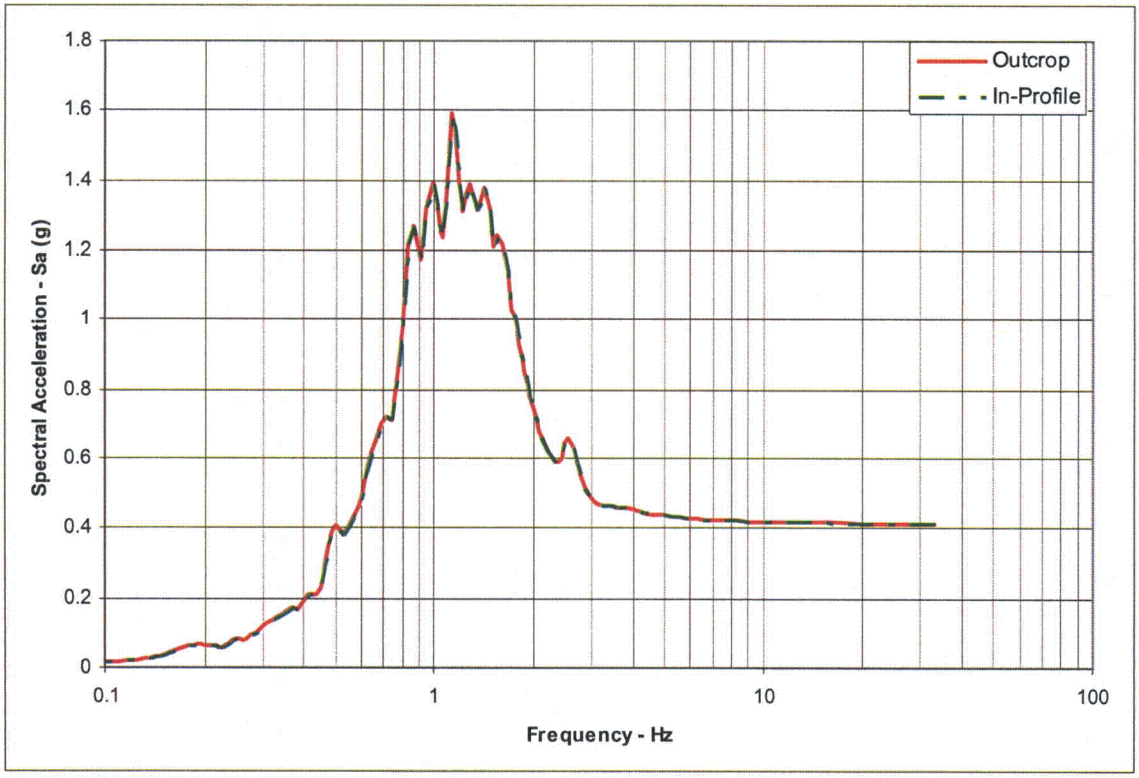


Figure 03.07.02-29.27: Comparison of 5% damped Spectra at 1 ft Below Ground Surface (SHAKE2000 116 Layer Validation Problem 3)

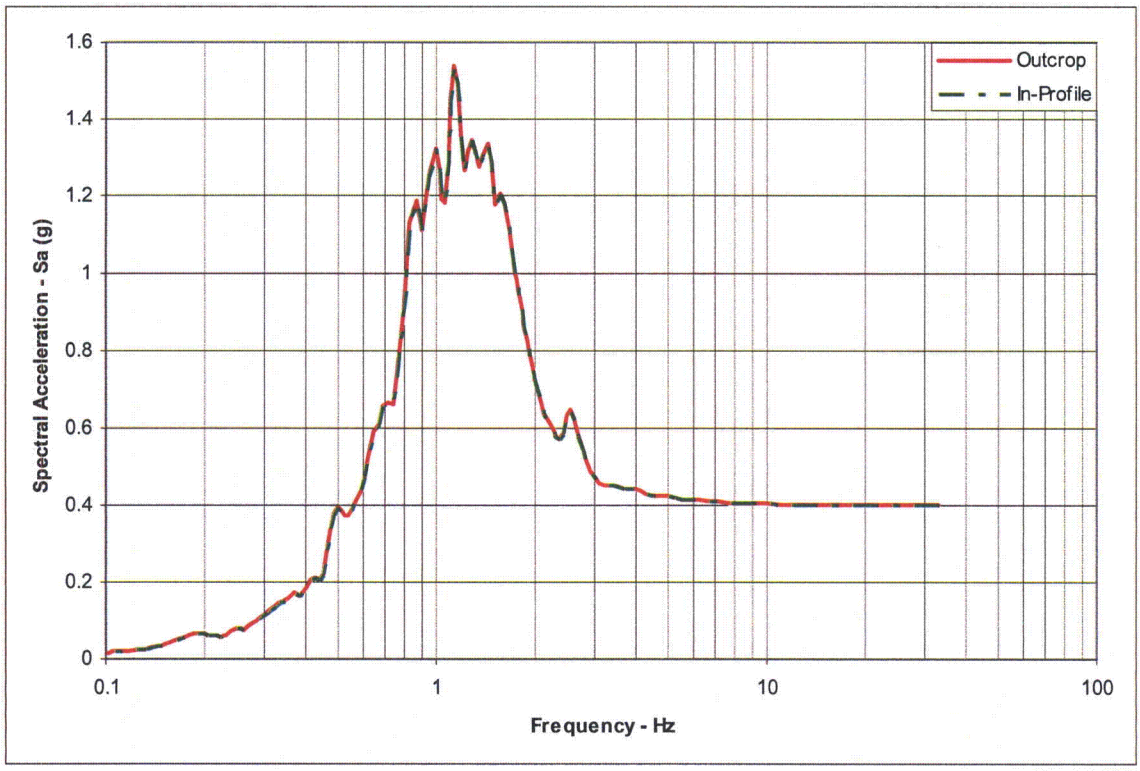


Figure 03.07.02-29.28: Comparison of 5% damped Spectra at 1 ft Below Ground Surface (SHAKE2000 116 Layer Validation Problem 4)

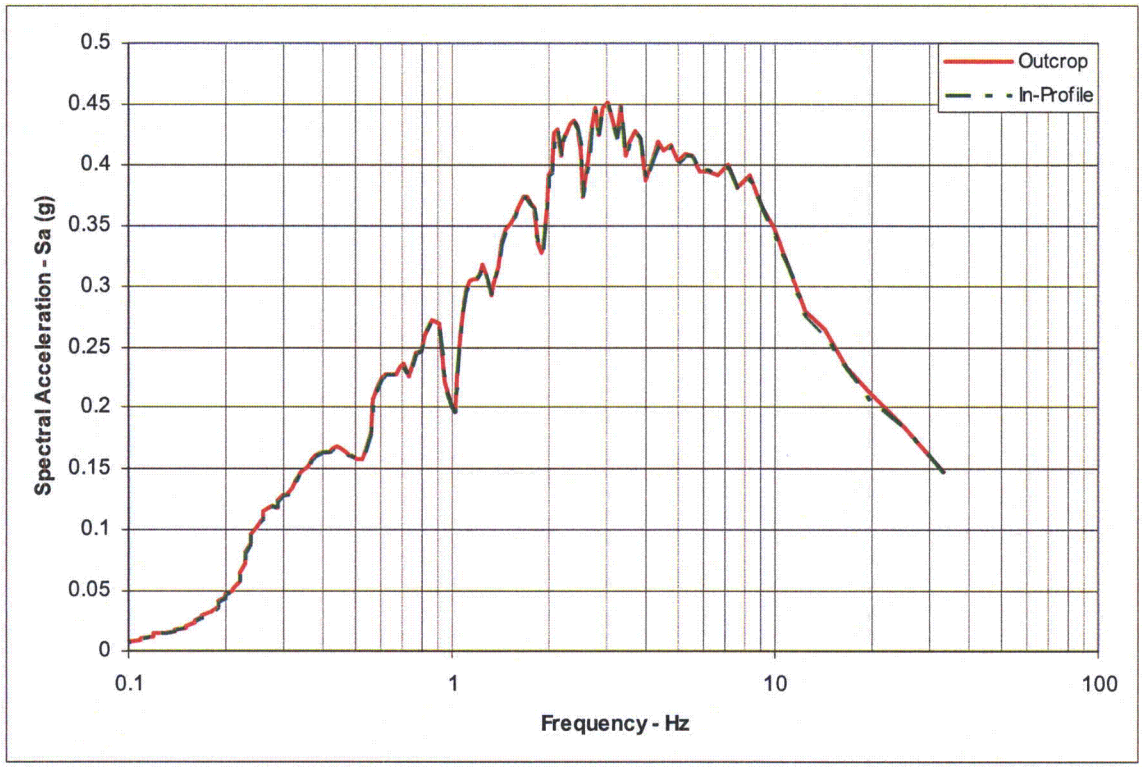


Figure 03.07.02-29.29: Comparison of 5% damped Spectra at 42 ft Below Ground Surface (SHAKE2000 116 Layer Validation Problem 5)

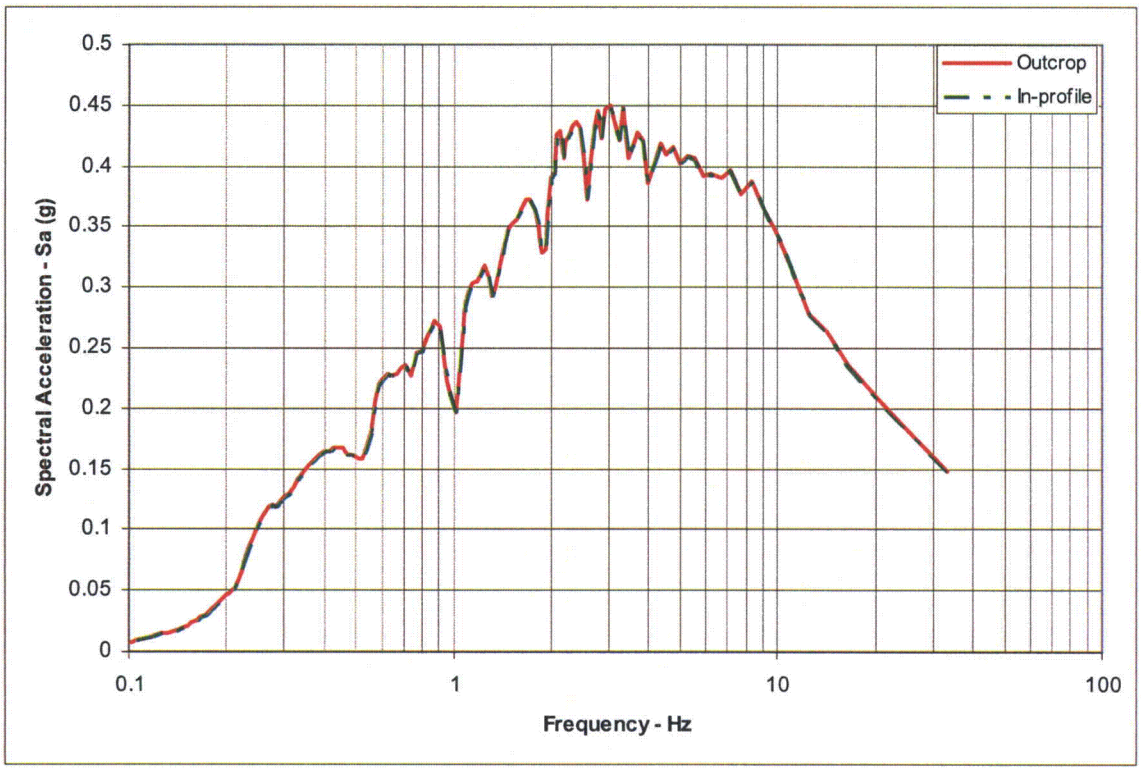


Figure 03.07.02-29.30: Comparison of 5% damped Spectra at 1 ft Below Ground Surface (SHAKE2000 116 Layer Validation Problem 6)

Enclosure 1
Revision to COLA Appendix 3C

3C.9 Free-Field Site Response Analysis (SHAKE2000 and P-SHAKE)

3C.9.1 Description

SHAKE2000 is used to perform the free-field site response analysis to generate the design- earthquake-induced strain-compatible free-field soil properties and site response motions required in the seismic SSI analysis. SHAKE2000 is a software application that integrates SHAKE, SHAKE91 and ShakEdit. SHAKE is a computer program for earthquake response analysis of horizontally layered sites developed at the University of California, Berkeley, by B. Schnabel, John Lysmer and H. B. Seed in 1972. SHAKE91 is a modified version of SHAKE for conducting equivalent linear seismic response analyses of horizontally layered soil deposits developed at the University of California, Davis, by I. M. Idriss and J. I. Sun. ShakEdit is a pre and postprocessor for SHAKE and SHAKE91 developed by Gustavo A. Ordonez.

P-SHAKE is a Bechtel proprietary modified version of SHAKE. P-SHAKE generates the same design earthquake-induced strain-compatible soil properties and site response motions as SHAKE does, and the input files of the two programs for the most part are compatible. However, P-SHAKE is built on a different program logic that allows the site response analysis to be performed with acceleration response spectrum as input instead of acceleration time histories used by SHAKE.

3C.9.2 Validation

SHAKE2000 was purchased from Gustavo A. Ordonez and validated by Sargent & Lundydeveloped by UC Berkeley. The program validation documentations is available at Sargent & Lundyare located at UC Berkeley.

P-SHAKE was developed by Bechtel. The program validation documents are located in Bechtel's Computation Service Library.

3C.9.3 Extent of Application

SHAKE2000 is used to generate free-field site response motions for use in seismic analysis of Category I structures, e.g., Reactor Building, Control Building and Ultimate Heat Sink.

P-SHAKE is used to provide site-specific earthquake-induced design ground motions and the associated strain-compatible soil properties for Category I structures, e.g., Reactor Building, Control Building, and Ultimate Heat Sink.

RAI 03.07.02-30

QUESTION:

RAI for STP Due to Fluor Part 21 Evaluation:

ABWR DCD, Tier 2, Section 3.2.5.3 states that *"the main steam piping, bypass line, and condenser are used to mitigate the consequences of an accident and are required to remain functional during and after an SSE."* ABWR DCD Section 3.2.5.3 further requires that *"Dynamic input loads for the design of the main steam lines in the turbine building are derived as follows: For locations on the basemat, the ARS shall be based upon the Regulatory Guide 1.60 Response spectra normalized to 0.6g (i.e., 2 times ARS of the site envelope). For locations at the operating deck level (either operating deck or turbine deck), the ARS used shall be the same as used at the reactor building end of the main steam tunnel. Seismic Anchor motions shall be similarly calculated."*

In a letter dated August 30, 2010, Fluor Enterprises, notified the NRC regarding exceedance of the ABWR DCD seismic design input requirements for main steam line (MSL) seismic analysis in the turbine building (TB) for STP 3 and 4 in accordance with 10 CFR Part 21.

As noted in the STP COLA, STP has taken a departure from the TB design as described in the ABWR DCD (STP DEP 1.2-2). In the course of detailed design of the STP 3 and 4 TBs, Fluor performed a dynamic analysis and generated floor response spectra (FRS) for the STP TBs. Fluor compared the STP TB FRS with the FRS specified in the DCD Tier 2, Section 3.2.5.3 for input to the Main Steam Line (MSL) seismic analysis. The comparison revealed that the FRS generated for the STP TB exceeded the FRS required by DCD Section 3.2.5.3 for MSLs. Fluor indicated that this issue is being addressed in the STP 3 and 4 design by use of conservative FRS generated during detailed design.

In view of the above, the applicant is requested to provide the following additional information in the FSAR regarding the seismic input used for the MSL and other important to safety SSCs in the TB:

1. The dynamic input loads (such as floor response spectra, anchor motions, etc) for the design of the MSLs in the TB including a description of the site-specific TB dynamic analysis model, the corresponding SSE input, and the computer programs used in the analysis; and,

2. An update to appropriate sections of the STP COL application including the applicable Departure Report in Part 7, the design descriptions or commitments identified in DCD Tier 1, Section 2.15.11 and the corresponding ITAAC for the TB as a result of STP specific FRS being higher than the FRS specified in DCD Tier 2, Section 3.2.5.3 for input to the MSL seismic analysis.

The staff needs this information to confirm that design of the MSLs and other important to safety SSCs in the TB appropriately consider the SSE design loads in combination with other

appropriate loads as required by the ABWR DCD, and to ensure the FSAR reflects the corresponding design basis.

RESPONSE:

The DCD does not provide a detailed design or design analysis for the Turbine Building (TB). Instead, the DCD includes a general arrangement for the TB, which utilizes reinforced concrete construction from the basemat to the operating deck and uses shear walls to provide lateral support. Additionally, DCD Section 3.2.5.3 provides requirements for the dynamic input to the analysis of the main steam line (MSL) in the TB.

In accepting the requirements included in the DCD, the FSER (NUREG-1503), on Pages 3-45 and 3-46 states:

> "... The staff concludes that the dynamic input loads for the design of the MSLs inside the turbine building are acceptable because (1) a comparison of the response spectra at the RB foundation level with the RG 1.60 response spectra anchored to 0.6g ZPA shows that the RG 1.60 response spectra anchored to the same 0.6g ZPA envelop the response spectra at the RB foundation level and (2) the turbine operating deck is located at approximately the same elevation as the anchor point of the main steamline at the RB side and the response spectra at the RB end were generated using an acceptable analysis approach as discussed..."

The NRC's acceptance, as noted above, is not based on a detailed design of the TB, and simply indicates that the turbine operating deck should be located at approximately the same elevation as the anchor point of the MSL at the Reactor Building (RB) side.

The critical features of the TB seismic design as described above in the ABWR DCD are not altered by STP DEP 1.2-2. With the departure, the TB still utilizes reinforced concrete construction from the basemat to the operating deck and uses shear walls to provide lateral support. The turbine operating deck is located at approximately the same elevation as the anchor point of the main steamline at the RB side. Therefore, the basis for the DCD requirements of dynamic input to the MSL design is still valid for the TB design while accounting for STP DEP 1.2-2.

Detailed design of the STP 3&4 TB is in progress. The analysis performed by Fluor was based on a design work in progress. Neither the TB design used by Fluor nor the seismic analyses performed by them was accepted by STPNOC. STPNOC continues to believe that the dynamic input for the MLS analysis as specified in DCD Section 3.2.5.3 is valid, and that the detailed design of the TB can be developed to be consistent with those provisions without any inconsistency or deficiency.

The DCD Tier 1 Section 2.15.11 does not include requirements related to seismic input for MSL analysis and design, and, therefore, this section is not affected by the Part 21 report by Fluor; i.e., even if the Part 21 report were accepted, the ITAAC would continue to be appropriate as currently written. In implementation of ITAAC 2.15.11 and ITAAC 2.10.1 (related to the Main Steam System design), a dynamic analysis of the TB will be performed to confirm that the DCD dynamic input requirements for the MSL are satisfied for the final design of the TB. If STPNOC were ever to determine that DCD Tier 2 Section 3.2.5.3 is not appropriate as applied to the STP

3&4 TB, it would take a departure in accordance with Section VIII.B.5 of the ABWR design certification rule.

No COLA revision is required as a result of this response.