Presented at the International Workshop on Infrastructure Systems for Nuclear Energy (IWISNE), December 15-17, 2010, Taipei, Taiwan

SASSI FE Program for Seismic Response Analysis of Nuclear Containment Structures

Mansour Tabatabaie[a]

Soil-structure interaction (SSI) analysis plays an important role in the seismic evaluation of nuclear power plants (NPPs). The results are used for both the structural design and the seismic qualification of components, equipments, and systems. Although numerous methods have been proposed in the last several decades, the SASSI program remains the preferred choice for performing SSI analyses of NPPs. This is due in large part to the manner in which the substructuring formulation is carried out. Essentially, the scattering problem and impedance problem are reduced to the site response solution and point-load solution, respectively, for a horizontally layered site. Despite this great advantage, however, computer processing and storage requirements limited the use of SASSI in nuclear projects to reduced structural models. But with the recent advancements in computer technology, SASSI is now able to solve large-scale models as well. A special version of the program incorporating large-core solutions is now available.

INTRODUCTION

Seismic response analysis of NPPs in the United States is often required for frequencies up to 33 Hz (NUREG 0800). In addition, NPPs founded on hard rock in the Eastern United States are now required to be analyzed for frequencies up to 50 Hz (ISG-1). Because the foundation soil media for typical NPPs and the side soil/backfill for NPPs founded on hard rock generally have low shear wave velocities ($V_s < 400$ m/s), the above passing frequency requirements often result in large-scale finite element soil and structural models that are too big to handle using the conventional SASSI program (Lysmer, et al., 1981).

To address this issue, the SASSI program has been modified to incorporate a large-core solution (LCS) model as well as free format and new data base structures. The new program ($^{\text{MTR/}}$**SASSI**) now makes it possible to analyze large-scale, deeply-embedded nuclear facilities. For instance, SSI models with over 100,000 nodes can now been analyzed

---

a) SC Solutions, 1261 Oakmead Parkway, Sunnyvale, CA 94085

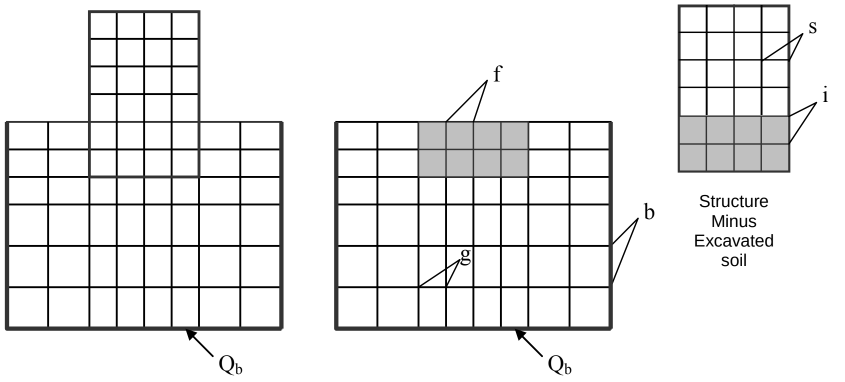
efficiently in $^{\mathrm{MTR/}}$**SASSI** using the LCS model. And since the structural nodes and elements can be numbered arbitrarily, the $^{\mathrm{MTR/}}$**SASSI** model may serve as a duplicate copy of the corresponding detailed FE model of the structure used for structural design. With the "glue" capability introduced in the new $^{\mathrm{MTR/}}$**SASSI** program, the structural model can be inserted into the soil model and analyzed without any further changes. This greatly facilitates model development, translation, calibration and maintenance.

This paper presents the theoretical basis of the program SASSI for 3-D seismic response analysis of structural systems embedded in a layered system over a uniform halfspace. This is followed by a description of the internal structure of the new $^{\mathrm{MTR/}}$**SASSI** program. To demonstrate the versatility of $^{\mathrm{MTR/}}$**SASSI** and its applicability to practical seismic problems, the results of the seismic response analysis of the US **EPR**™ nuclear island (NI), modeled using stick and detailed FE models, are presented for one generic soil case and EUR-based input motion.

METHODOLOGY

The basic method of analysis adopted by SASSI, referred to as the Flexible Volume Method (Tabatabaie, 1982), is based on the observation that solutions to scattering and impedance problems in the general substructuring approach can be greatly simplified if the interactions are considered over a volume rather than a boundary. The Flexible Volume Method is a substructuring procedure that uses finite element and complex frequency response methods to solve the dynamic response of SSI systems.

In the Flexible Volume Method, the complete soil-structure system [see Fig. 1(a)] is partitioned into two substructures, called the foundation and the structure [see Figs. 1(b) and 1(c), respectively]. In this partitioning, the structure consists of the structure model minus the excavated soil model (i.e., the soil to be excavated is retained within the foundation, leaving the halfspace as a horizontally layered system). Interaction between the structure and foundation occurs at all excavated soil nodes.



(a) Total System (b) Foundation (c) Structure

Figure 1. Substructuring of Interaction Model

The equations of motion for the Flexible Volume Method are developed by combining in the frequency domain the equations of motion for the structure and the soil using the concept of substructuring, thus leading to Eq. (1) below from which the total motions of the structure can be determined.

$$\begin{pmatrix} C_{ss} & C_{si} \\ C_{is} & C_{ii} - C_{ff} + X_f \end{pmatrix} \begin{Bmatrix} U_s \\ U_f \end{Bmatrix} = \begin{Bmatrix} 0 \\ X_f \, . \, U'_f \end{Bmatrix} \tag{1}$$

In Eq. (1), the subscripts “s”, “i” and “f” refer to DOFs associated with nodes on the structure, basement, and excavated soil, respectively. C is the complex-valued, frequency-dependent stiffness matrix, which is expressed in the form:

$$C(\omega) = K - \omega^2 M \tag{2}$$

M and K are the total mass and complex-valued stiffness matrices, respectively; ω is the circular frequency; U is the vector of complex-valued nodal point displacements; U’ is the vector of complex-valued free-field displacements; and X_f is a complex-valued, frequency-dependent matrix representing the dynamic stiffness of the foundation at the interaction nodes (X_f is referred to as the impedance matrix). The matrices M and K are assembled using standard finite element formulations.

Equation (1) considers only seismic forces. External loads at the structure nodes can also be considered simply by adding the amplitudes of these forces to the load vector [right-hand side of Eq. (1)] at each frequency. Equation (1) reduces the solution to the SSI problem for each frequency to three steps:

Step 1: Solve the site response problem to determine the free-field motion U'_f

Step 2: Solve the impedance problem to determine the impedance matrix X_f

Step 3: Solve the structural problem to determine the response $U = < U_s \; U_f >^T$

SITE RESPONSE ANALYSIS

The original site is assumed to consist of horizontal soil layers overlying a uniform halfspace. All material properties are assumed to be viscoelastic. However, the stiffness and damping of each layer are adjusted by the equivalent linear method to account for the material nonlinearities.

With the proposed system, only the free-field displacements at the layer interfaces where the structure is connected are of interest. Accordingly, the displacement amplitudes may be expressed in the following form for different wave types:

$$U'_f(x) = U_f \, e^{i(\omega t - kx)} \quad (3)$$

U_f is a vector (mode shape) containing the complex-valued interface amplitudes at and below the control point ($x = 0$); k is a complex-valued wave number expressing the speed at which the wave propagates and decays in the horizontal x-direction; t is the elapsed time; and $i = \sqrt{-1}$. Effective discrete methods have been developed (Chen, 1980) for determining appropriate mode shapes and wave numbers corresponding to the control motions at any layer interface for inclined P-, SV- and SH-waves, Rayleigh waves, and Love waves. Any combination of these waves can also be applied simultaneously.

IMPEDANCE ANALYSIS

As previously stated, the impedance matrix represents the dynamic stiffness of the foundation at the interaction nodes. It can thus be determined as the inverse of the dynamic flexibility matrix F_f calculated for the interaction nodes:

$$X_f = F_f^{-1} \quad (4)$$

The calculation of the dynamic flexibility matrix is based on a series of point load solutions for a layered system over uniform halfspace. This solution is obtained using a plane-strain FE model for 2-D problems (Tabatabaie, 1982) and an axi-symmetric FE model for 3-D problems (Tajirian, 1981). The result is a full complex-valued symmetric matrix, which is then inverted using an efficient in-place inversion algorithm to obtain the impedance matrix X_f. This method for computing the impedance matrix is called the Direct Model.

In order to make the above operation more cost-effective, an alternative method called the Skin Model was developed (Tabatabaie, 1982). With this approach, the interaction nodes are grouped into three different categories, referred to as interface, intermediate, and internal nodes. By definition, interface nodes are nodes that lie along the physical boundaries between the structure and the soil region (labeled by digit 1). Intermediate nodes are defined as those interaction nodes which are directly connected to interface nodes (labeled by digit 2). The remaining interaction nodes are internal nodes (labeled by digit 3).

From the above definitions, Eq. (4) can be partitioned into submatrices corresponding to interface, intermediate, and internal DOFs. When combined with the submatrices of the direct stiffness matrix of the excavated soil region, the entire impedance matrix for the interaction nodes can be written as seen below [Eq. (5)]. As shown in Eq. 5, this alternative model results in a much smaller flexibility matrix to be inverted (i.e., F_{11}^{-1}). Because the size of the interface nodes grows by square rather than by cube, as in the case of interaction nodes calculated via the Direct Model, the resulting computer run time and storage requirements reduce significantly.

$$- C_{ff} + X_f = \begin{pmatrix} F_{11}^{-1} (I - F_{12} . C_{12}^T) - C_{11} & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix} \tag{5}$$

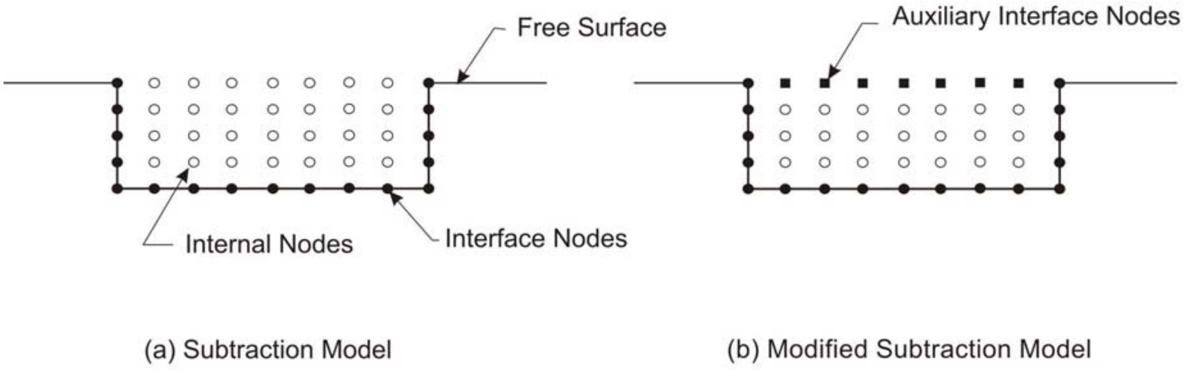
The Skin Model imposes the compatibility of displacements at the interface nodes, but at the internal nodes this compatibility is only inferred [see Eq. (5)]. Due to the numerical difference in deriving the impedance matrix and dynamic stiffness of the excavated soil model, the Skin Model only provides acceptable impedance solutions if the cut-off frequency is set very low (i.e., to $V_s/12h$ or even lower, where V_s is the shear wave velocity of the foundation media and h is the smallest element size in the excavated soil model).

The Subtraction Model is another alternative method for calculating the foundation impedance matrix. According to this model, the stiffness of the excavated soil model is condensed to that of the interface nodes (labeled by digit 1). This matrix is then subtracted from the inverse of the flexibility matrix of the same interface nodes [see Eq. (6)]. In theory, this would produce the impedance matrix of the interface nodes of a hole in the ground, which is then added to the stiffness of the structure on the left-hand side of Eq. (1).

$$-C_{ff} + X_f = \begin{pmatrix} F_{11}^{-1} - C_{11} & -C_{12} & 0 \\ -C_{21} & -C_{22} & -C_{23} \\ 0 & -C_{32} & -C_{33} \end{pmatrix} \tag{6}$$

Again, like the Skin Model, only the flexibility matrix associated with the interface nodes needs to be inverted (i.e., F_{11}^{-1}). For this reason, the Subtraction Model also offers greater savings in terms of computer run time and storage requirements than the Direct Model. Nonetheless, the Subtraction Model does not enforce compatibility of displacements at all interior nodes of the excavated soil volume, potentially causing accuracy to deteriorate at high frequencies. Recent studies (Tabatabaie, 2011) show that the transfer functions calculated using the Subtraction Model may contain erroneous peaks and valleys associated with the wave energy trapped within the excavated soil model. These anomalies are found to be more pronounced for structures with large shallow foundations.

The Modified Subtraction Model is a proposed improvement over the Subtraction Model. In this model, the compatibility of displacements – in addition to the skin nodes – is imposed at the internal nodes located on the free-field surface (referred to as auxiliary interface nodes) by specifying those nodes as interaction nodes (see Fig. 2).



(a) Subtraction Model

(b) Modified Subtraction Model

Figure 2. Illustration of Subtraction and Modified Subtraction Models

When the compatibility of displacements is also imposed at the internal nodes located at the free surface (as in the Modified Subtraction Model), the transfer functions become smoother, and the erroneous peaks and valleys in the response transfer functions disappear. The results of the Modified Subtraction Model are found to be closer to those of the Direct Model (Tabatabaie, 2011).

The size of the soil-structure systems with symmetric properties and loading may also be significantly reduced if only one-half or one-quarter of the model is analyzed. This will require the derivation of special impedance matrices (Tabatabaie, 1981).

STRUCTURAL ANALYSIS

The structural analysis involves forming the complex-valued coefficient matrix and load vector shown in Eq. (1), and solving for the response transfer functions. The structure and the excavated soil mass and stiffness are assembled using the conventional finite element method. An efficient out-of-core equation solver is used to solve the final assembled equations of motion shown in Eq. (1).

LAYOUT OF THE $^{\text{MTR/}}$SASSI PROGRAM

The layout of the $^{\text{MTR/}}$**SASSI** program is shown in Fig. 3. The program has a modular structure specifically designed for practical applications with the following characteristics:

- The site response analysis, impedance analysis, and formation of the basic stiffness and mass matrices for the structure can be performed separately. The results are stored on disk files.
- If the seismic wave field, external loads, soil properties, or arrangement of the superstructure are changed, then only part of the computations needs to be repeated.
- The final solution is stored (in the form of transfer functions) on a disk file from which the response quantities can be extracted without re-computing the entire solution.
- Both deterministic (time history) and probabilistic results can be obtained from the above files.

The function of each program module is briefly described below.

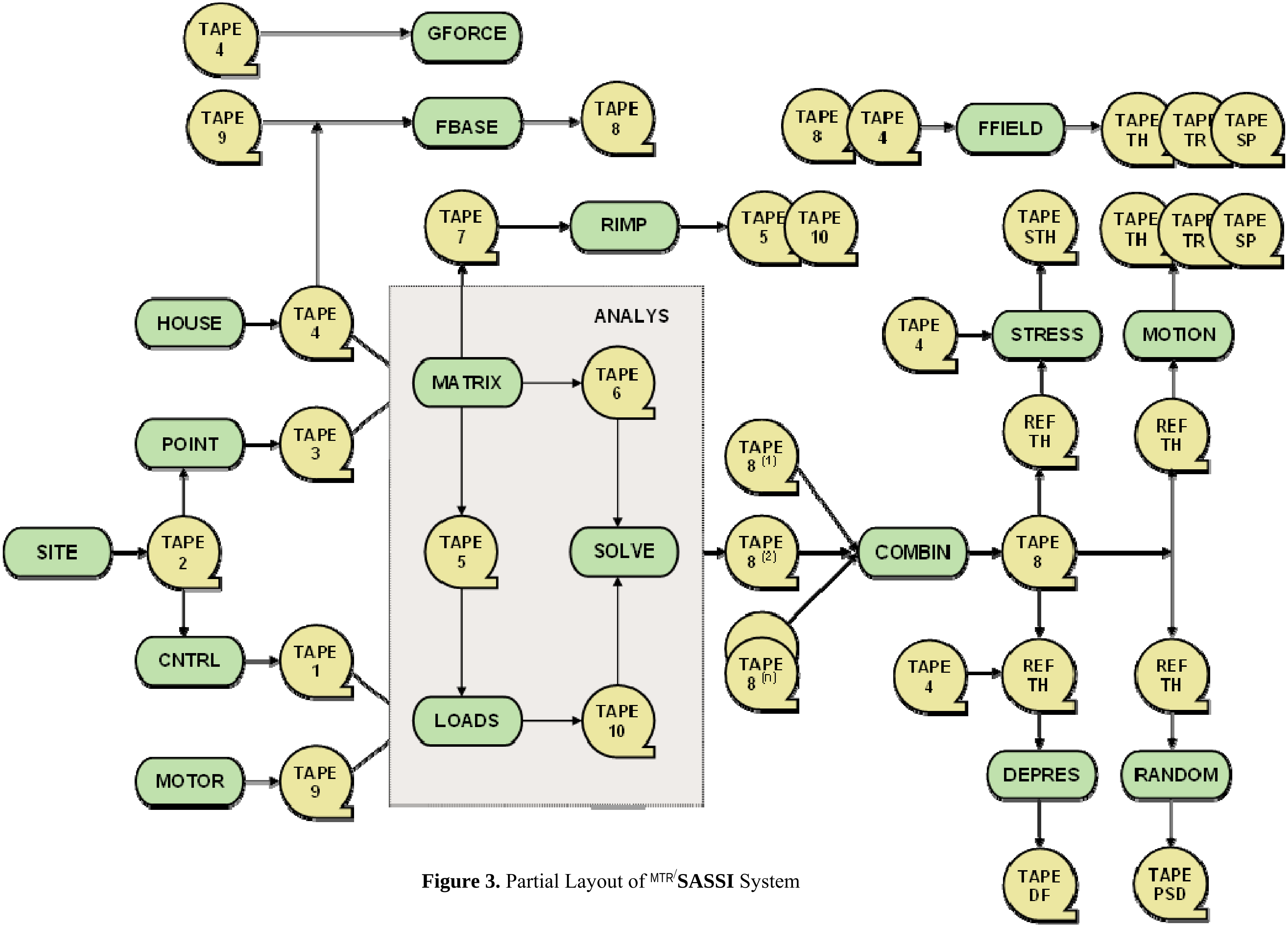


Figure 3. Partial Layout of $^{MTR/}$**SASSI** System

SITE

The program module SITE solves the eigenvalue problem of Rayleigh and Love wave cases for a horizontally layered site. The results of the eigenvalue solution are saved on Tape 2, which will later be used to 1) solve the site response problem in program module CNTRL and 2) compute the transmitting boundaries used in solving the impedance problem in program module POINT. Thus, the program module SITE must be executed before the program modules CNTRL and POINT.

CNTRL

The program module CNTRL solves the site response problem. This program reads the site properties and eigensolution via Tape 2, the nature of the control motion from the input data and, using this information, calculates the mode shapes and wave numbers. The results are then stored on Tape 1, which will later be used for seismic analysis. Thus, Tape 1 will not be generated for forced vibration problems. The program module CNTRL also has the capability to calculate incoherent ground motion input using coherence functions.

POINT

The program module POINT calculates the flexibility matrix of the interaction nodes for each frequency of interest. The program requires Tape 2 as input and stores the results on Tape 3. Thus, the program module SITE must be executed before the program module POINT.

HOUSE

The program module HOUSE is a standard finite element program. The element library includes 3-D solid, 3-D beam, 3-D flat plate/shell, 2-D plane-strain, 3-D spring, 2-D plane Love wave, 3-D stiffness/mass, and 3-D super elements. HOUSE forms the frequency-independent total mass and complex-valued stiffness matrices of the structure and excavated soil, and stores the results on Tape 4.

MOTOR

The program module MOTOR forms the elements of the load vector, which correspond to impact forces acting externally on the structure, or to forces acting within the structure. The generated information is stored on Tape 9.

ANALYS

The program module ANALYS is the heart of the $^{MTR/}$**SASSI** program. It drives the three subprograms MATRIX, LOADS and SOLVE, and thereby controls the restart modes of the program.

> MATRIX: Using the data from Tapes 3 and 4, MATRIX calculates the impedance matrix for each frequency and stores the results on Tape 5. MATRIX then reads the total mass and stiffness matrices of the structure and excavated soil from Tape 4, adds the impedance matrix to obtain the total stiffness of the system, and stores the results on a disk.
>
> LOADS: Using the data from Tape 1, LOADS calculates the load vector for each frequency and stores the results on a disk.
>
> SOLVE: The subprogram SOLVE reads the reduced total stiffness of the system and load vectors from the disk and performs back-substitution to obtain the total displacement response. The results in terms of uninterpolated transfer functions are stored on Tape 8.
>
> For typical problems, the transfer functions only need to be solved for 60 to 100 frequencies. The remaining values of the transfer functions are obtained by interpolation in the frequency domain. The actual interpolations are performed in the program modules MOTION, STRESS, RANDOM, DEPRES and FFIELD.

COMBN8

The program module COMBN8 makes it possible to consider new frequencies of analysis and to combine the results with the old transfer functions. This program reads the transfer functions stored on multiple input Tape 8s, combines them, and stores the results on a new Tape 8.

FBASE

The program module FBASE computes the response of a structural system on a fixed or flexible base. The analysis may be performed for external dynamic or static forces acting on the structure, or for a given seismic input motion at the base of the structure. FBASE reads the structural mass and complex-valued stiffness matrices from Tape 4, then reads either the input dynamic loads from Tape 9 or the seismic control motion from the input data file and calculates the response of the structure for each frequency of analysis. For seismic problems, the results are output in terms of absolute acceleration transfer functions (response acceleration/input acceleration). For forced vibration problems, the results are output in terms of displacement transfer functions (response displacement/input force). The resulting transfer functions are stored on Tape 8, which can then be input into the post-processors MOTION, STRESS and RANDOM to calculate the required responses at selected nodes.

GFORCE

The program module GFORCE computes the forces of a 1g acceleration applied in the global x-, y- and z-directions to all nodes in the structure. The program reads the structural information from Tape 4 and generates a list of forces based on the mass of the structural elements and nodes. The calculated forces in the z-direction may be input to the program module MOTOR to calculate gravity load information on Tape 9, which may then be input to the program modules ANALYS and/or FBASE for dead load analysis and the results combined with those of seismic load analysis.

RIMP

The program module RIMP may be used to obtain impedance matrices for multiple rigid foundations of arbitrary shape, founded in soil media or on pile foundations. Although these impedance functions may also be obtained through general flexible foundation analysis in $^{\text{MIT}}$/**SASSI**, the RIMP procedure greatly reduces the numerical effort and memory required to calculate the above impedances. It does this by taking advantage of the rigidity of the foundation and thus bypassing the inversion of the flexibility matrix in ANALYS. The program module RIMP reads the flexibility matrices from Tape 7 and the geometry of the rigid foundation system from the input data file. Then, for each specified frequency, it calculates the corresponding foundation impedance matrices on Tape 5. The selected impedance components may be further interpolated to obtain smooth foundation impedance functions. The results of the RIMP analysis are printed out as well as saved on output Tape 10 for later processing in the program module FBASE.

MOTION

The purpose of the (deterministic) post-processor MOTION is to produce maximum values and time histories of output response (accelerations, velocities, and displacements). It can also output transfer functions and response spectra. The program reads the time history of input motion (or force) from an input data file and transforms it into the frequency domain using the Fast Fourier Transform (FFT) algorithm. MOTION then reads the uninterpolated transfer functions from Tape 8 for the selected output responses, performs interpolation and convolution with the input motion (or force), and returns to the time domain using the inverse FFT algorithm. The resulting time histories of response may be output directly and saved on Tape TH, or converted to output response spectra and saved on Tape SP. The transfer functions can also be output directly and saved on Tape TF.

STRESS

The program module STRESS evaluates the maximum values and time histories of stresses and strains in the structural elements. The program can also compute the maximum octahedral shear strain at the center of each soil element. The time histories of stresses may be output directly and saved on Tape STH. Tapes 4 and 8 as well as the time history of input motion (or force) are part of the input for this program. Secondary nonlinearity of soils in the near-field may be accounted for by modifying the soil properties according to the computed values of maximum effective shear strain in each soil element.

RANDOM

The (probabilistic) post-processor RANDOM is in many respects similar to the program module MOTION. However, instead of inputting the time history of control motion, it reads a power spectral density (PSD) function of this motion. It then evaluates the root mean square (RMS) and PSD responses of the structure. The transfer functions can be output on Tape PTF.

DEPRES

The program module DEPRES calculates the dynamic soil reaction forces at interaction nodes due to dynamic soil pressures acting on the structural walls and basemat. Tapes 4 and 8 serve as input tapes for this program. The results of nodal forces may be output in terms of maximum values, time histories, and transfer functions. The time history of dynamic soil reaction forces may be saved on Tape DF.

FFIELD

The program module FFIELD computes the far-field soil responses. Although these responses may also be obtained by including additional far-field soil nodes in the SSI model, the FFIELD procedure greatly reduces the numerical effort and memory required to calculate them. In addition, if a far-field soil node was not originally included in the SSI model but later desired by the User, FFIELD will calculate the response without re-running the original SSI model.

The program reads the structural information from Tape 4, the point load solutions from Tape 3, and the un-interpolated transfer functions from Tape 8. It then calculates the response of the far-field soil nodes in terms of transfer functions at the same frequencies analyzed for the SSI model. The results are then output in a new Tape 8, which can then be input to the post-processor MOTION to calculate maximum values and time histories of computed acceleration, velocity and/or displacement responses as well as acceleration response spectra.

${}^{MTR/}$SASSI ANALYSIS OF US EPR^{TM} NUCLEAR ISLAND

The program ${}^{MTR/}$**SASSI** is used to analyze the seismic response of a large-scale structural model of the US **EPR**^{TM} Nuclear Island (NI). Figure 4 shows the plant layout.

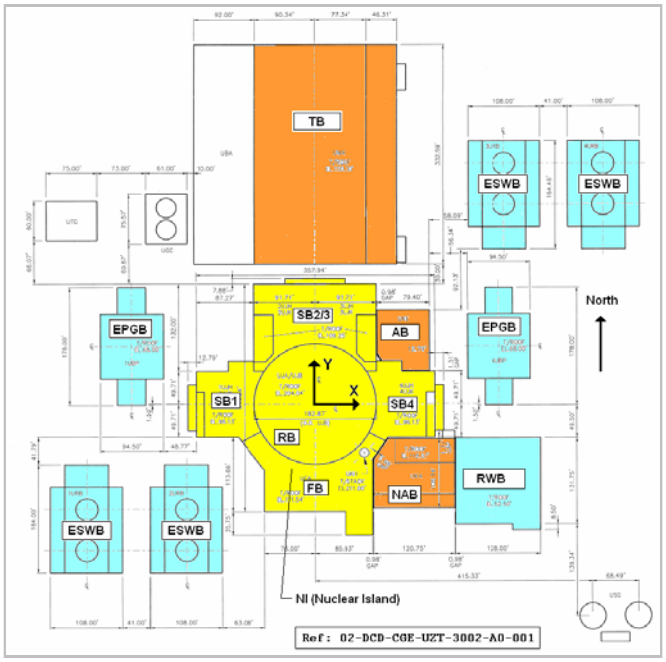


Figure 4. Layout of **EPR**^{TM} Plant

The plant consists primarily of a nuclear island (NI) and several other significant structures outside of and in close proximity to the NI. The NI structures consist of the Reactor Building Containment (RBC), Reactor Building Shield (RBS), Reactor Building Internal Structures (RBI), Fuel Building (FB), Safeguard Building 1 (SB1), Safeguard Building 2/3 (SB2/3), and Safeguard Building 4 (SB4) - all of which share a common foundation basemat. The NI is embedded approximately 11.6 meters below ground surface.

The plant is analyzed for 8 generic soil profiles as part of the standard design certification. The generic soil profiles used for the SSI analysis are shown in Fig. 5. The input motions consist of three-component, spectra–matched EUR Soft, EUR Medium and EUR Hard motions specified as free-field outcrop motions at the base of the NI basemat. Figure 6 shows the response spectra of the input motions.

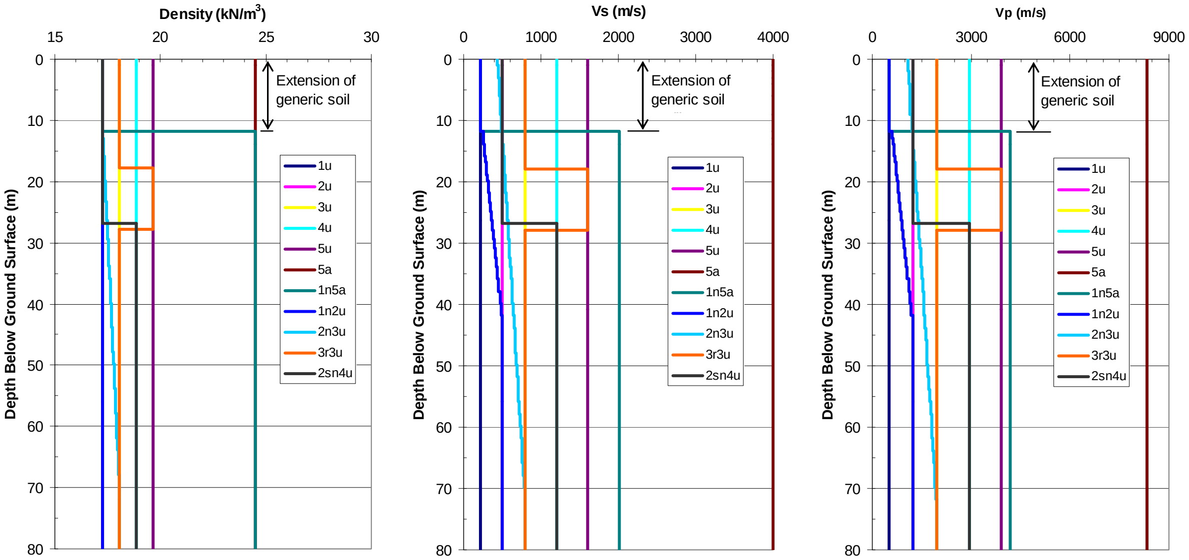


Figure 5. Generic Soil Profiles and Properties

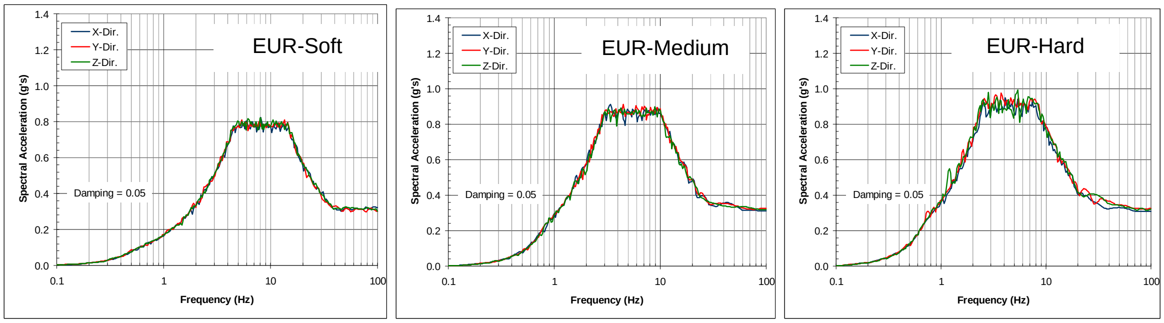


Figure 6. Acceleration Response Spectra of Reference Outcrop Motions

Two structural models were used for the SSI analysis: a stick model and a detailed finite element model, as shown in Figs. 7 and 8, respectively.

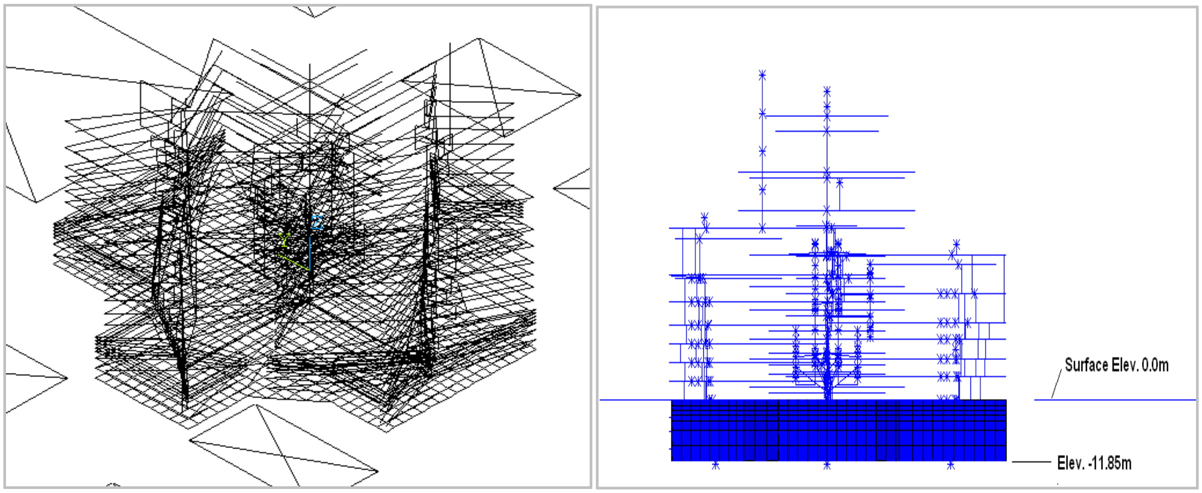


Figure 7. $^{\text{MTR/}}$**SASSI** Stick Model of NI Structures

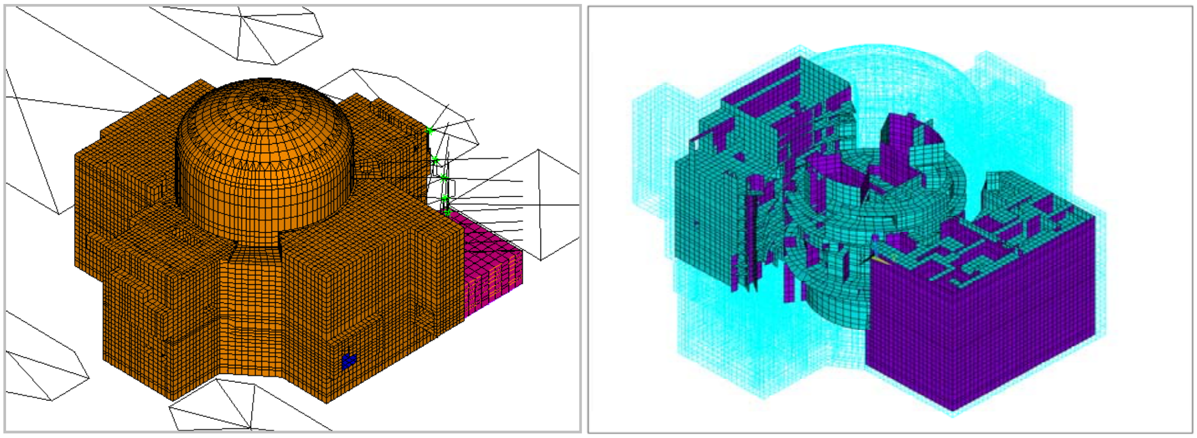


Figure 8. $^{\text{MTR/}}$**SASSI** Detailed FE Model of NI Structures

The stick model consists of multiple interconnected sticks representing the walls and columns between the principal floor elevations of the structures. To model embedment effects, horizontal rigid beams are added along the excavation face at soil layer interfaces where the NI walls bear against soil (these beams share common nodes with the soil interaction nodes). The beams are then connected to the FB shield stick, the SB2/3 shield stick, and the SB1 and SB4 sticks with rigid links to provide lateral support from side soils and to transfer forces from the side soils to the sticks. The detailed FE model incorporates all the major details of the NI structures. It consists mainly of shell elements representing the concrete floors, walls, and basemat - all of which are modeled as flexible members. The NSSS, major equipment supports, and polar crane are modeled by beam elements. The fixed-base modes of the stick model have been aligned against the global modes of the detailed FE model.

The FE model of the NI foundation is shown in Fig. 9. This model is the same for both the stick and detailed FE models, with one exception: in the stick model the basemat is assumed to be rigid. All the basement walls are connected to side soils, with the exception of the walls adjacent to the Nuclear Auxiliary Building (NAB) and Access Building (AB).

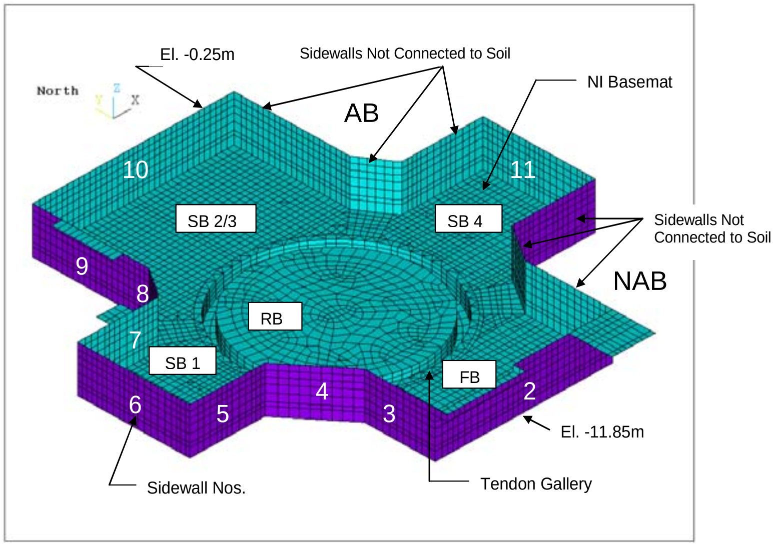


Figure 9. NI Foundation Model

Using a combination of soil profiles and input motions, a total of 13 SSI analysis cases of the embedded NI model were evaluated. Each analysis case consisted of three separate SASSI runs with three components of the input motion applied separately in the x-, y- and z-directions. The results of the three analyses (i.e., responses due to x-input, y-input and z-input) in terms of acceleration time history responses at output nodes were then algebraically summed, and the results were used to calculate the maximum accelerations and acceleration response spectra. The results of the SSI analyses of the US **EPR**^{TM} NI structures are used to address several aspects of the stick model used in SSI modeling.

ANALYSIS RESULTS

The $^{MTR/}$**SASSI** program calculated the SSI response of the US **EPR**^{TM} NI using both stick and detailed FE models. The analysis results are discussed for Soil Case 2sn4u and EUR Medium motion. The maximum average element size in the soil model is about 2.3 meters, which corresponds to a passing frequency of 500/5/2.3 = 44 Hz. Because the input motion lacks significant energy beyond 40 Hz, the frequency cutoff for the SSI model was set at 50 Hz. The analysis was performed for 42 and 66 computed frequencies of the stick and detailed finite element models, respectively, with the intermediate frequency response values of the transfer functions obtained by interpolation. The computed transfer functions at key structural locations were plotted and visually examined to ensure that adequate frequency responses were computed for later interpolation.

Typical results of the SSI analysis of the detailed FE model of the NI in terms of the computed maximum accelerations in the x-, y- and z-directions are shown in Figs. 10, 11 and 12, respectively. The maximum accelerations at several key locations in the major floor elevations of the NI structures are calculated from the stick and detailed FE model analyses and compared in Tables 1. Although the two models could not be compared at exactly the same locations, in general they indicate similar results.

The acceleration response spectra computed at the center of NI basemat and tops of the Reactor Building Internal Structures (RBIS), Reactor Building Containment (RBC), and Reactor Building Shield (RBS), together with the corresponding spectra of the reference foundation outcrop motions in the same direction, are shown in Figs. 13a through 13d, respectively. The results show comparable responses from the stick and detailed models at the center of NI basemat. However, the computed spectra in the structure can be quite different despite the similarity of the overall spectral shapes. The large amplification in the vertical spectral accelerations calculated from the stick model at frequencies above 10 Hz in the Reactor Containment Building is attributed to the differences in the rigidity of the basemat between the two models. In general, stick models are capable of determining global seismic responses, but they can lead to excessively conservative results in the vertical direction due to the limited number of modes that can be modeled.

Table 1. Comparison of Maximum Accelerations

Location	Stick Model				Detailed FE Model			
	Elev. (m)	Maximum Accel. (g's)			Elev. (m)	Maximum Accel. (g's)		
		X	Y	Z		X	Y	Z
Center of NI Basemat	-11.85	0.277	0.210	0.318	-11.85	0.262	0.224	0.319
Reactor Bldg Internal Structures	5.15	0.347	0.258	0.341	5.15	0.379	0.320	0.374
Reactor Bldg Internal Structures	19.50	0.421	0.391	0.366	19.50	0.513	0.419	0.388
Safeguard Building 1	29.30	0.564	0.502	0.501	21.0	0.478	0.356	0.399
Safeguard Building 2/3	12.00	0.411	0.409	0.446	16.30	0.400	0.433	0.413
Safeguard Building 4	29.30	0.580	0.621	0.556	21.0	0.340	0.335	0.394
Fuel Building	3.70	0.350	0.294	0.357	4.20	0.300	0.364	0.298
Reactor Bldg Containment	58.00	0.738	0.620	0.893	58.00	0.869	0.734	0.516
Reactor Bldg Shield	61.40	0.843	0.854	0.578	61.40	0.598	0.679	0.490

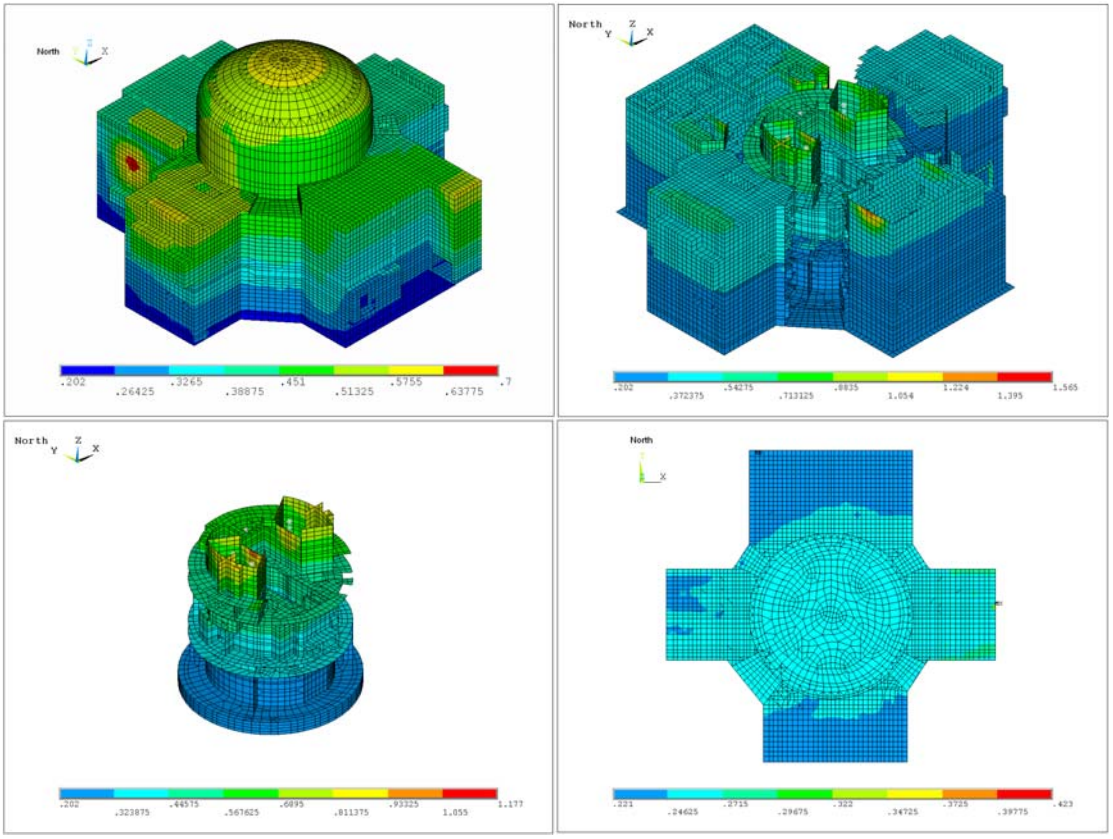


Figure 10. Contours of Maximum Accelerations in X-Direction

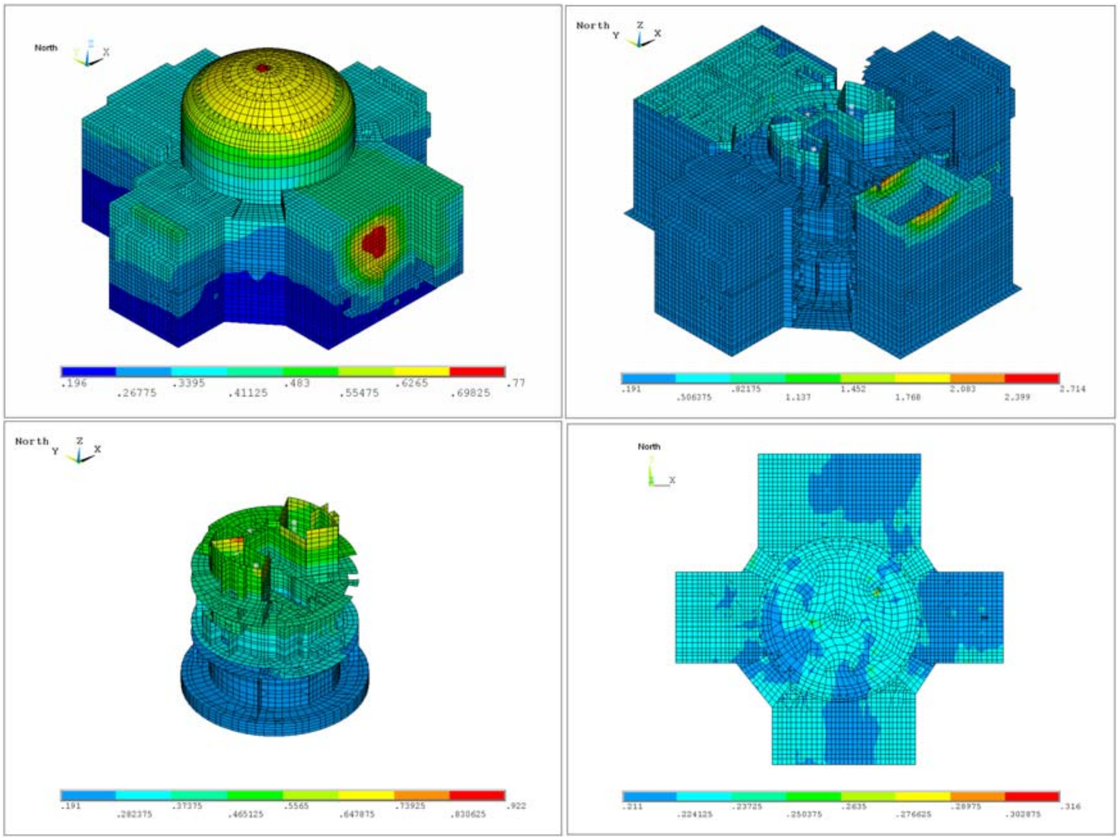


Figure 11. Contours of Maximum Accelerations in Y-Direction

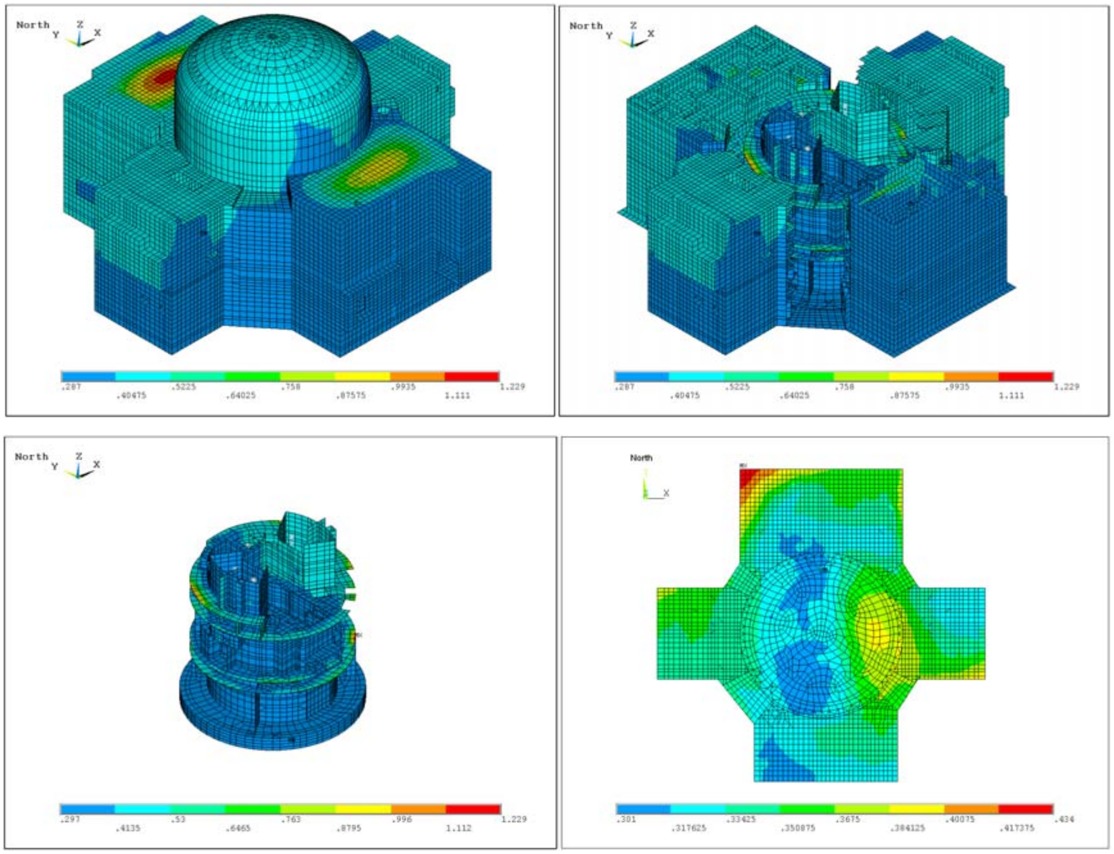


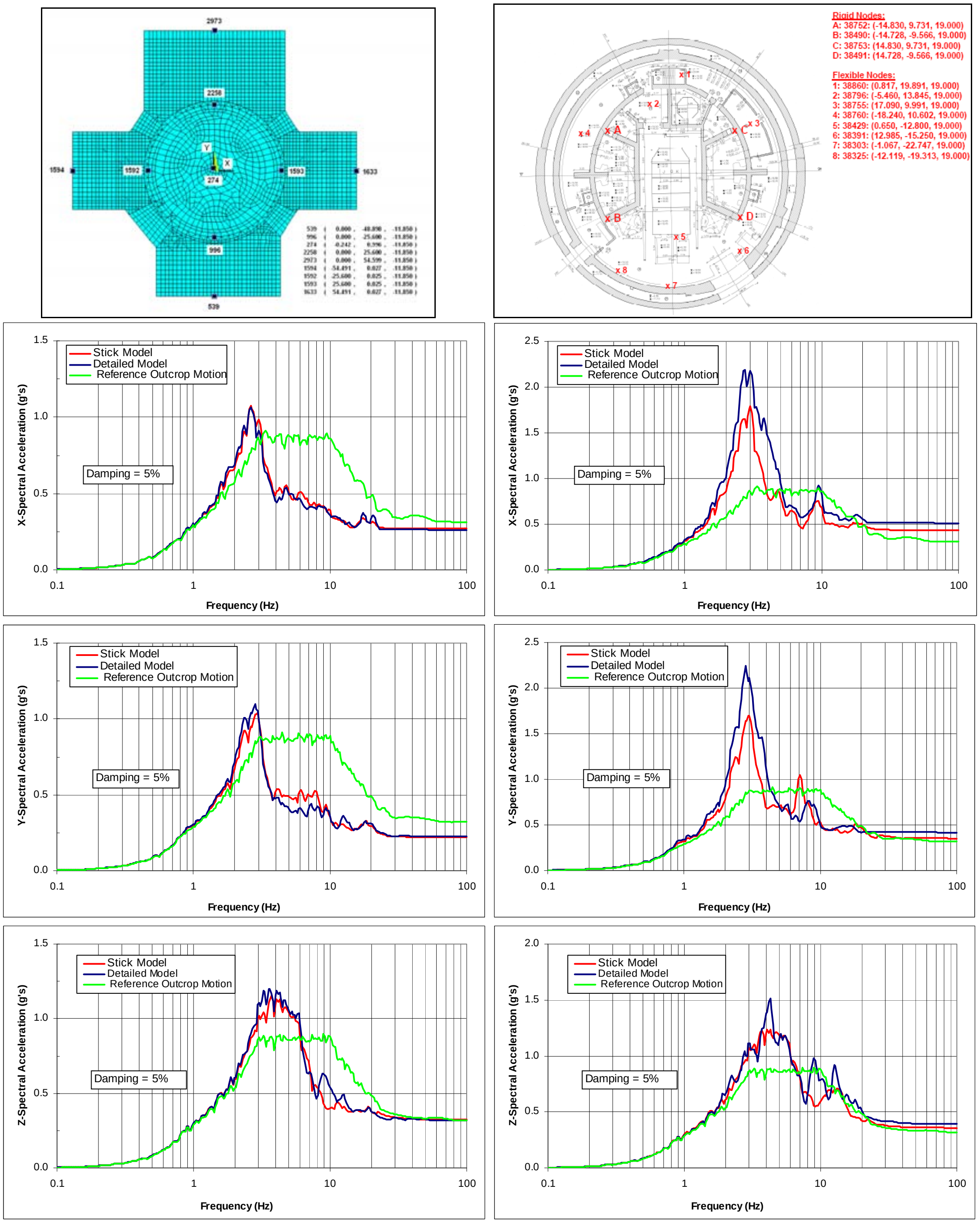
Figure 12. Contours of Maximum Accelerations in Z-Direction

Comparison of the results in terms of total inter-story shear forces and overturning moments indicates good agreement between the two models (see Fig. 14).

SUMMARY RESULTS

Based on the results of this study, the following conclusions can be drawn:

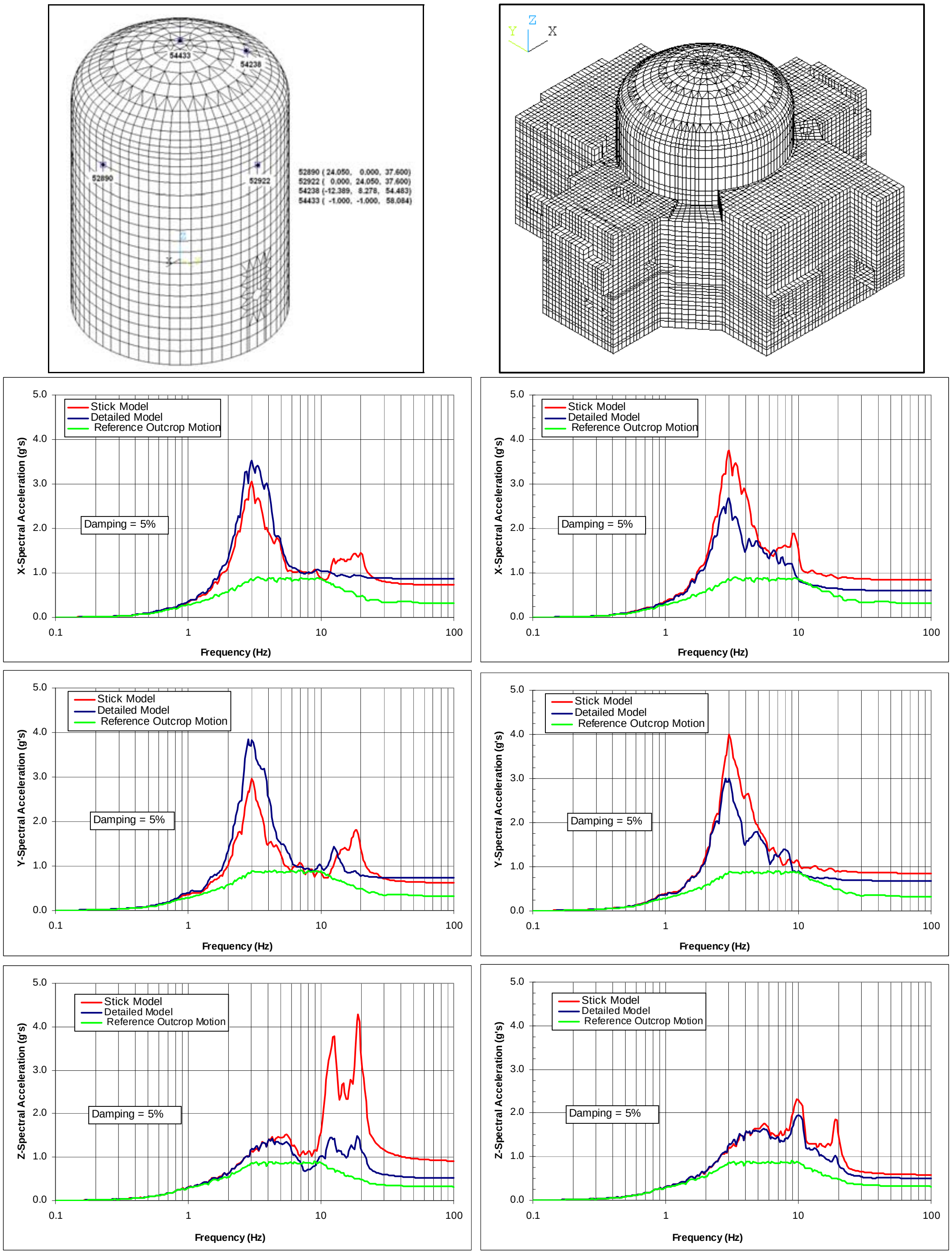
- Stick models are capable of determining global seismic responses, but they can lead to excessively conservative results in the vertical direction due to the limited number of modes that can be modeled.
- Detailed FE models capture local responses, thus eliminating the need for modeling single DOF oscillators.
- Effects of the basemat flexibilities can be considered in the detailed FE models.
- Meshing can be made sufficiently small in detailed FE models to capture the response due to high frequency input motions.



(a) Center of NI Basemat (El. -11.85 m)

(b) Top of RBIS, Node “B” (El. 19.5 m)

Figure 13. Comparison of Response Spectra



(c) Top Center of RBC (El. 58.0 m)

(d) Top Center of RBS (El. 58.0 m)

Figure 13. Comparison of Response Spectra (Cont’d)

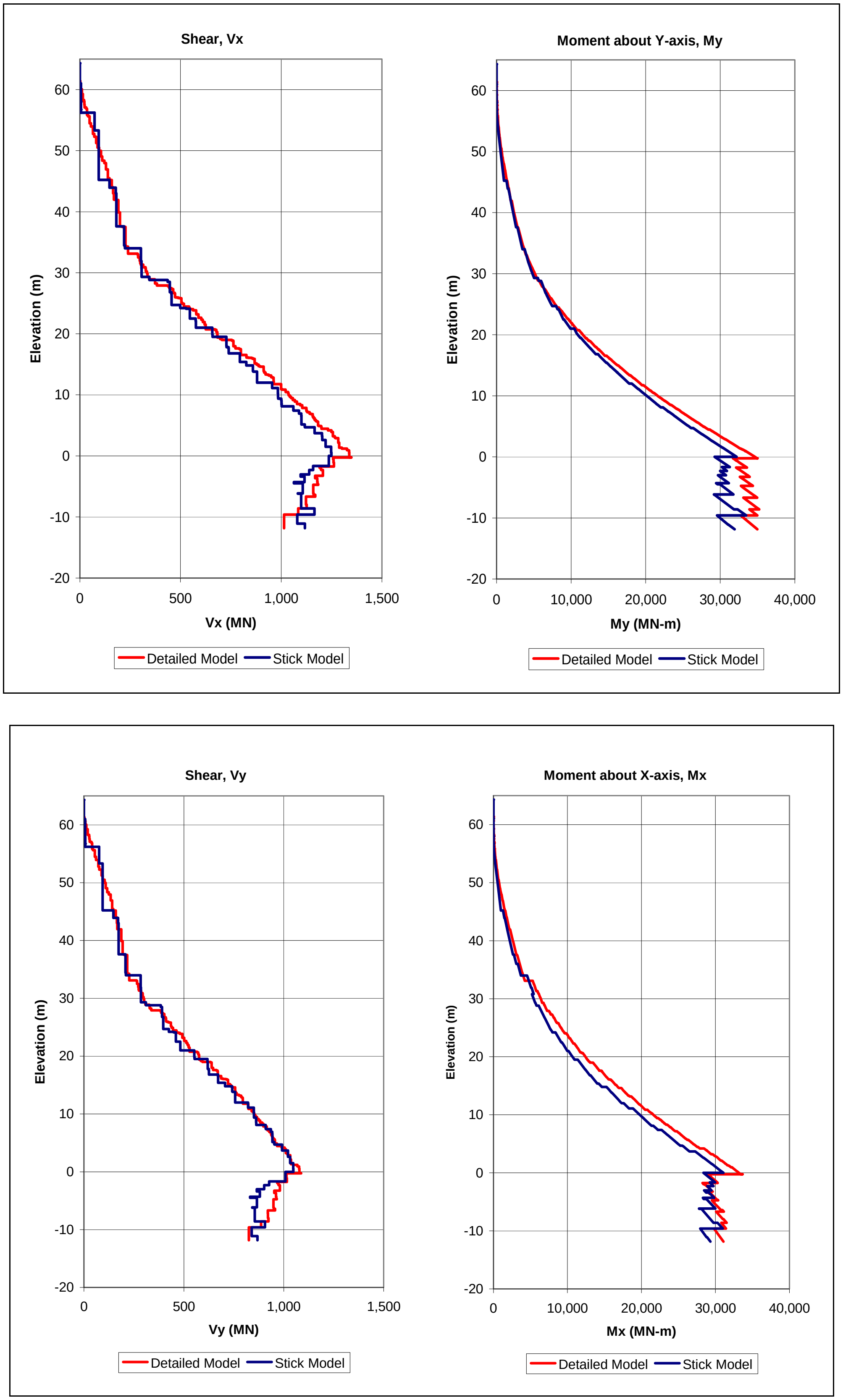


Figure 14. Comparison of Interstory Shear Forces and Overturning Moments

SUMMARY

In the past several decades, the SASSI program has been extensively used in the seismic response analysis of nuclear containment structures. This is due in large part to SASSI's innovative substructuring methodology, which is superior to the methodologies of other programs. Traditionally, SASSI has been limited to small-scale SSI models due to computer run time and storage requirements. But with the utilization of large-core memory, free-format and new data base structures, an enhanced version is now available ($^{MTR/}$**SASSI**) to handle large-scale SSI models efficiently. The results of the seismic response analyses of the US **EPR**^{TM} NI structures, modeled using stick and detailed FE models, demonstrate the versatility of $^{MTR/}$**SASSI** and its applicability to large-scale seismic problems.

ACKNOWLEDGEMENTS

AREVA NP has been instrumental in supporting the development of large-scale SSI modeling as part of its US **EPR**^{TM} standard design licensing. The support of J. Todd Oswald, Calvin Wong and Daniel B. Fisher of AREVA NP is hereby acknowledged.

REFERENCES

Chen, J.C. (1980), "Analysis of Local Vibrations in Free-field Seismic Ground Motions," Ph.D. Dissertation, University of California, Berkeley.

ISG-1, "Interim Staff Guidance on Seismic Issues of High Frequency Ground Motion," US Nuclear Regulatory Commission, Washington DC.

Lysmer, J., M. Tabatabaie, F. Tajirian, S. Vahdani and F. Ostadan (1981), "SASSI – A System for Analysis of Soil-Structure Interaction," Report No. UCB/GT/81-02, Geotechnical Engineering, Department of Civil Engineering, University of California, Berkeley.

$^{MTR/}$**SASSI** (2010), "System for Analysis of Soil-Structure Interaction," Version 8.3, MTR & Associates, Inc., Lafayette, California,

NUREG-0800, "Standard Review Plan for the Review of Safety Analysis Reports for Nuclear Power Plants: LWR Edition," Design of Structures, Components, Equipment, and Systems, Section 3.7, US Nuclear Regulatory Commission.

Tabatabaie, M. (1982), "The Flexible Volume Method for Dynamic Soil-Structure Interaction Analysis," Ph.D. Dissertation, University of California, Berkeley.

Tabatabaie, M. (2011), “The Accuracy of The Subtraction Model Examined in $^{\text{MTR/}}$SASSI,” Part I, White Paper, SC Solutions, Walnut Creek, California.

Tajirian, F. (1981), “Impedance Matrices and Interpolation Techniques for 3-D Interaction Analysis by the Flexible Volume Method,” Ph.D. Dissertation, University of California, Berkeley.