SASSI 2000

A SYSTEM FOR ANALYSIS OF SOIL-STRUCTURE INTERACTION

VERSION 3

USER'S MANUAL

Farhang Ostadan

APRIL 2007

ACKNOWLEDGEMENT

Original SASSI Development Team

John Lysmer, Farhang Ostadan,
Mansour Tabatabaie, Fredrick Tajirian, and Shahriar Vahdani

Geotechnical Engineering Division
Civil Engineering Department
University of California, Berkeley, CA 94720

SASSI2000

VERSION 1

November 1999

John Lysmer, Farhang Ostadan, Chih Cheng Chin

SASSI2000

VERSION 2

January 2006
Farhang Ostadan

DISCLAIMER

Every reasonable effort was made to provide a comprehensive and flexible computer program. However, the computer program itself and associated documentation are supplied without representation of warranty, expressed or implied, as to its content, accuracy, or freedom from defects or errors.

RECORD OF REVISION

Version Number	Date of Revision	Description
3	April 2007	Original issue

LIST OF EFFECTIVE PAGES

Sheet	Latest Revision	Sheet	Latest Revision
Cover Sheet	3		
i	3		
ii	3	3-1	3
iii	3	3-2	3
iv	3	3-3	3
v	3	3-4	3
vi	3	3-5	3
vii	3	3-6	3
viii	3	3-7	3
ix	3		
x	3	4-1	3
xi	3	4-2	3
xii	3	4-3	3
xiii	3	4-4	3
xiv	3	4-5	3
xv	3	4-6	3
xvi	3	4-7	3
		4-8	3
		4-9	3
1-1	3	4-10	3
1-2	3	4-11	3
1-3	3	4-12	3
1-4	3	4-13	3
1-5	3	4-14	3
		4-15	3
2-1	3	4-16	3
2-2	3	4-17	3
2-3	3	4-18	3
2-4	3	4-19	3
2-5	3	4-20	3
2-6	3	4-21	3
2-7	3	4-22	3
2-8	3	4-23	3

Sheet	Latest Revision	Sheet	Latest Revision
4-24	3	4-57	3
4-25	3	4-58	3
4-26	3	4-59	3
4-27	3	4-60	3
4-28	3	4-61	3
4-29	3	4-62	3
4-30	3	4-63	3
4-31	3	4-64	3
4-32	3	4-65	3
4-33	3	4-66	3
4-34	3	4-67	3
4-35	3	4-68	3
4-36	3	4-69	3
4-37	3	4-70	3
4-38	3	4-71	3
4-39	3	4-72	3
4-40	3	4-73	3
4-41	3	4-74	3
4-42	3	4-75	3
4-43	3	4-76	3
4-44	3	4-77	3
4-45	3	4-78	3
4-46	3	4-79	3
4-47	3	4-80	3
4-48	3	4-81	3
4-49	3	4-82	3
4-50	3	4-83	3
4-51	3	4-84	3
4-52	3	4-85	3
4-53	3	4-86	3
4-54	3	4-87	3
4-55	3	4-88	3
4-56	3	4-89	3

Sheet	Latest Revision	Sheet	Latest Revision
4-90	3	4-123	3
4-91	3		
4-92	3	5-1	3
4-93	3	5-2	3
4-94	3	5-2	3
4-95	3	5-4	3
4-96	3	5-5	3
4-97	3	5-6	3
4-98	3	5-7	3
4-99	3	5-8	3
4-100	3	5-9	3
4-101	3	5-10	3
4-102	3	5-11	3
4-103	3	5-12	3
4-104	3	5-13	3
4-105	3	5-14	3
4-106	3	5-15	3
4-107	3	5-16	3
4-108	3	5-17	3
4-109	3	5-18	3
4-110	3	5-19	3
4-111	3	5-20	3
4-112	3	5-21	3
4-113	3	5-22	3
4-114	3	5-23	3
4-115	3	5-24	3
4-116	3	5-25	3
4-117	3	5-26	3
4-118	3	5-27	3
4-119	3	5-28	3
4-120	3	5-29	3
4-121	3	5-30	3
4-122	3	5-31	3

Sheet	Latest Revision	Sheet	Latest Revision
5-32	3	5-66	3
5-32	3	5-67	3
5-34	3	5-68	3
5-35	3	5-69	3
5-36	3	5-70	3
5-37	3	5-72	3
5-38	3	5-73	3
5-39	3	5-72	3
5-40	3	5-74	3
5-41	3	5-75	3
5-42	3	5-76	3
5-43	3	5-77	3
5-44	3	5-78	3
5-45	3	5-79	3
5-46	3	5-80	3
5-47	3	5-82	3
5-48	3	5-82	3
5-49	3	5-83	3
5-50	3	5-84	3
5-51	3	5-85	3
5-52	3	5-86	3
5-53	3	5-87	3
5-54	3	5-88	3
5-55	3	5-89	3
5-56	3	5-90	3
5-57	3	5-92	3
5-58	3	5-92	3
5-59	3	5-93	3
5-60	3	5-94	3
5-61	3	5-95	3
5-62	3	5-96	3
5-63	3	5-97	3
5-64	3	5-98	3
5-65	3	5-99	3

Sheet	Latest Revision	Sheet	Latest Revision
5-100	3	5-124	3
5-101	3	5-125	3
5-102	3	5-126	3
5-103	3	5-127	3
5-104	3	5-128	3
5-105	3	5-129	3
5-106	3	5-140	3
5-107	3	5-141	3
5-108	3	5-142	3
5-109	3	5-143	3
5-110	3	5-144	3
5-111	3	5-145	3
5-112	3	5-146	3
5-113	3	5-147	3
5-114	3	5-148	3
5-115	3	5-149	3
5-116	3	5-150	3
5-117	3	5-151	3
5-118	3	5-152	3
5-119	3	5-153	3
5-120	3	5-154	3
5-121	3	5-155	3
5-122	3	5-156	3
5-123	3	5-157	3
5-124	3	5-158	3
5-125	3	5-159	3
5-126	3	5-160	3
5-127	3	5-161	3
5-128	3	5-162	3
5-129	3	5-163	3
5-120	3	5-164	3
5-121	3	5-165	3
5-122	3	5-166	3
5-123	3	5-167	3

Sheet	Latest Revision	Sheet	Latest Revision
5-168	3	B-1	3
5-169	3	B-2	3
5-170	3	B-3	3
5-171	3		
5-172	3		
5-173	3		
5-174	3		
5-175	3		
5-176	3		
5-177	3		
5-178	3		
5-179	3		
5-180	3		
5-181	3		
5-182	3		
5-183	3		
5-184	3		
5-185	3		
5-186	3		
5-187	3		
5-188	3		
5-189	3		
5-190	3		
5-191	3		
5-192	3		
6-1	3		
6-2	3		
A-1	3		
A-2	3		
A-3	3		

FEATURES/OPTIONS CHECKLIST

Use Appendix B for Features/Options Checklist.

TABLE OF CONTENTS

TABLES

TABLE **PAGE**

FIGURES

FIGURE **PAGE**

CHAPTER 1
INTRODUCTION

1.1 PROGRAM DESCRIPTION

SASSI, a System for Analysis of Soil-Structure Interaction, consists of a number of interrelated computer program modules which can be used to solve a wide range of dynamic soil-structure interaction problems in two or three dimensions.

1.2 CAPABILITIES AND LIMITATIONS

The computer program SASSI2000 has the following capabilities and limitations:

1.2.1 Soil and Structure Idealization

1. The site consists of semi-infinite elastic or viscoelastic horizontal layers on a rigid base or a semi-infinite elastic or viscoelastic halfspace.

2. The structure(s) are idealized by standard two- or three-dimensional finite elements connected at nodal points.

3. Each nodal point on the structure may have up to six displacement degrees of freedom. The user has the option to delete one or more of the degrees of freedom thereby reducing the size of the problem accordingly.

4. The excavated soil zone(s) are idealized by standard plane strain or three-dimensional solid elements. The finite element models of the structure and excavated soil have common nodes at the boundary.

5. Depending on the method selected for impedance analysis, the interaction between the foundation and the structure occurs at all basement nodes, including those in the basement volume or occurs only at the common boundary nodes.

6. All the interaction nodes lie on the soil layer interfaces with translational degree-of-freedom. Rotations from the structure are transferred by translation by connecting at several interacting nodes.

7. The mass matrix is assumed to be 50% lumped and 50% consistent except for the structural beam elements and plate elements where consistent mass matrix and lump mass matrix are used, respectively.

8. Material damping is introduced by the use of complex moduli, which leads to effective damping ratios which are frequency-independent and may vary from element to element.

1.2.2 Dynamic Loadings

1. The seismic environment may consist of an arbitrary three-dimensional superposition of inclined body waves and surface waves.

2. Earthquake excitation is defined by a time history of acceleration called control motion. The control motion is assigned to one of the three global directions at the control point which lies on a soil layer interface.

3. In addition to seismic loads, it is possible to introduce external forces or moments such as impact loads, wave forces, or loads from rotating machinery acting directly on the structure. The external forces are applied at the nodal points and are assumed to have similar time histories. However, it is possible to assign different maximum amplitudes and arrival times to each dynamic load applied at a nodal point. This feature enables the program user to define moving dynamic loads on the structure.

4. Transient input time histories such as earthquake record or impact loads are handled by the Fast Fourier Transform technique. Therefore, the time histories must be specified at equal time intervals.

1.2.3 Finite Element Library

Currently, the SASSI finite element library consists of the following element types:

1. Three-dimensional solid element (eight-node brick) with three translational degrees of freedom per node. It is also possible to include nine incompatible displacement modes in this element when it is used to model the structure.

2. Three-dimensional beam element with three translational and three rotational degrees of freedom per node.

3. Four-node quadrilateral thin elements with three translational and two rotational degrees of freedom per node.

4. Four-node quadrilateral thick elements with three translational and three rotational degrees of freedom per node.

5. Two-dimensional four-node plane strain finite element with two translational degrees of freedom per node.

6. Three-dimensional spring element with three translational and three rotational degrees of freedom per node.

7. Three-dimensional inter-pile elements to model pile groups.

8. A new sub-structuring method to compute single and pile group impedance functions.

9. Three-dimensional stiffness/mass matrix element with three translational and three rotational degrees of freedom per node.

1.2.4 Nonlinear Analysis

1. The analytical method used in the SASSI program is restricted to linear analysis. However, approximate nonlinear analysis can be performed by an iterative scheme called "Equivalent Linear Method."

2. Primary nonlinear effects in the free-field and secondary nonlinear effects in a limited region near the structure can be considered.

1.2.5 Interpolation Scheme

An efficient interpolation scheme on complex response functions has been developed for the SASSI program. By the use of the new interpolation scheme it is sufficient to compute the response at, say, 15 to 20 frequencies, from which the intermediate solutions can be obtained by interpolation.

1.2.6 System of Units

Any system of units may be chosen to be used in the SASSI program as long as the units of the input data are consistent in all the program modules.

1.3 ORGANIZATION OF THE MANUAL

An introduction of the SASSI computer program was given in Chapter 1. This chapter also lists the capabilities and limitations of the SASSI program modules.

In Chapter 2, a brief description of the theoretical background is given to describe the analytical procedure used in the SASSI program.

In Chapter 3, the organization and operational features of the SASSI program are presented.

In Chapter 4, the application guide to SASSI analysis is presented. This chapter explains the logical approach to SASSI analysis of SSI problems and how to run a SASSI job. A sample problem to illustrate these steps has also been included in this chapter.

In Chapter 5, the SASSI program modules are individually described. The general description of each program module begins with an introduction to the program operation, followed by the input guide. The input guide also contains numerous comments which are helpful to the user in the day-to-day use of the program.

Appendix A is the location to include the features/options checklist after installation of the program and its verification on the host system. Appendix B includes the comment forms and error report forms.

CHAPTER 2
THEORETICAL BACKGROUND

2.1 THE FLEXIBLE VOLUME METHOD

The basic methods of analysis adopted by the computer program SASSI2000 are called the flexible volume and the recently developed subtraction methods (Refs. 1 and 3). These methods are formulated in the frequency domain using the complex response method and the finite element technique. A detailed description of each method is presented in the theoretical manual. For discussion on basic numerical operations in SASSI, only the flexible volume method is considered below.

In the flexible volume method, the complete soil-structure system, shown in Fig. 2.1-1(a), is partitioned into two substructures, namely, the foundation and the structure, as shown in Figs. 2.1-1(b) and 2.1-1(c), respectively. In this partitioning, the structure consists of the superstructure plus the basement minus the excavated soil; i.e., the soil to be excavated is retained with the foundation. Interaction between the structure and the foundation occurs at all basement nodes.

The equations of motion for the flexible volume method are developed by combining the equation of motion for the structure with those of the soil in the frequency domain using the concepts of substructuring, thus leading to:

$$\begin{bmatrix} C_{ss} & C_{si} \\ C_{is} & \left(C_{ii} - C_{ff} + X_{ff}\right) \end{bmatrix} \begin{Bmatrix} u_s \\ u'_f \end{Bmatrix} = \begin{Bmatrix} 0 \\ X_{ff} \cdot u'_f \end{Bmatrix} \tag{2.1-1}$$

From which the final total motions of the structure can be determined. In these equations, the subscripts s, i, and f refer to degrees of freedom associated with the nodes on superstructure, basement, and excavated soil, respectively. C is the complex frequency-dependent stiffness matrix:

$$C(\omega) = K - \omega^2 M \tag{2.1-2}$$

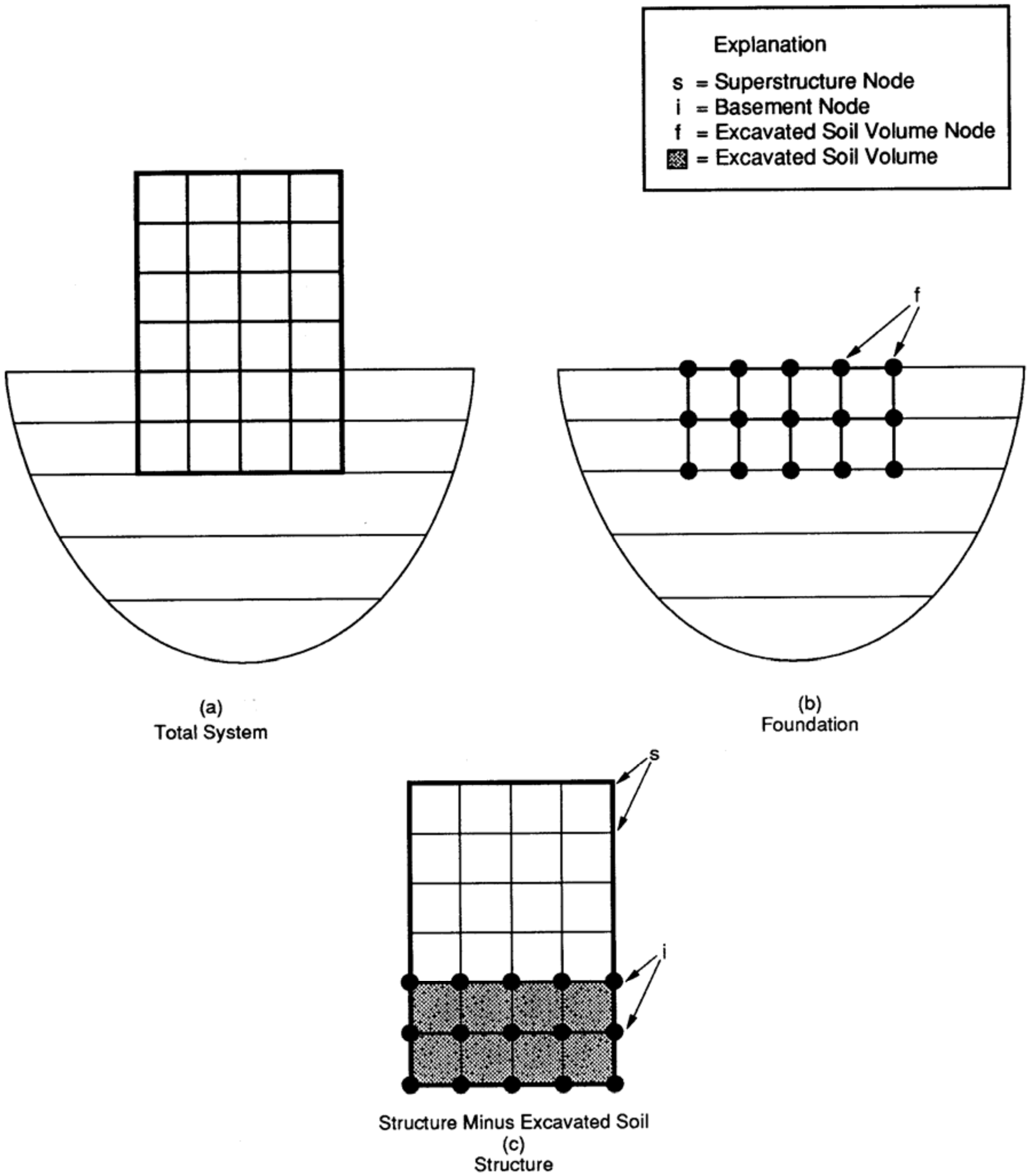


Figure 2.1-1. Substructuring in the Flexible Volume Method

Where M and K are the total mass and complex stiffness matrices, respectively, assembled as described in Ref. 1, and ω is the frequency of vibration; u is the vector of complex nodal point displacements; X_{ff} is a frequency-dependent matrix, which represents the dynamic stiffness of the foundation at the interaction nodes. X_{ff} will be referred to as the impedance matrix.

Equation (2.1-1) considers only seismic forces. External loads at the superstructure and basement nodes can be considered simply by adding the amplitudes of these forces to the load vector (right-hand side of Eq. (2.1-1) at each frequency. Thus the final motions of the structure can be determined from the following equations:

$$\begin{bmatrix} C_{ss} & C_{si} \\ C_{is} & \left(C_{ii} - C_{ff} + X_{ff}\right) \end{bmatrix} \begin{Bmatrix} U_s \\ U_f \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix} \tag{2.1-3}$$

Where P_s and P_f are the amplitudes of external forces at the superstructure and basement nodes, respectively.

According to this formulation, the solution of the soil-structure interaction problem reduces to three main steps (for each frequency):

1. <u>Solve the site response problem</u> to determine the free-field motions U_f' within the embedded part of the structure.

2. <u>Solve the impedance problem</u> to determine the matrix X_{ff}.

3. <u>Solve the structural problem.</u> This involves forming the complex stiffness matrices and load vectors shown in Eqs. (2.1-1) and (2.1-3) and solving these equations for the final displacements.

2.2 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 as described in Ref. 1.

Methods for solving the site response problem corresponding to inclined body waves and surface waves have also been described in Ref. 1.

In this formulation, only the free-field displacements of the layer interfaces where the structure is connected are of interest. Accordingly, and this is possible for all of the above-mentioned wave types, displacement amplitudes will be expressed in the form:

$$u_f'(x) = U_f' \cdot e^{i(\omega t - kx)} \tag{2.2-1}$$

where U_f' is a vector (mode shapes) which contains the interface amplitudes at and below the control point (x=0) and k is a complex wave number which expresses how fast the wave propagates and decays in the horizontal x-direction. Effective discrete methods have been developed (Ref. 1) for determining appropriate mode shapes and wave numbers corresponding to control motions at any layer interface for inclined P-, SV-, and SH-waves, Rayleigh waves, and Love waves. Any combination of such waves can be applied.

2.3 IMPEDANCE ANALYSIS

As previously stated, the impedance matrix represents the dynamic stiffness of the foundation at the interaction nodes. Thus, it can be determined from the inverse of the dynamic flexibility matrix F_{ff} for these nodes:

$$X_{ff} = F_{ff}^{-1} \tag{2.3-1}$$

$[F_{ff}]$ is a full symmetric complex matrix, and an efficient in-place inversion subroutine is currently used for such operation. This method is called "the direct method" for computing the impedance matrix. The other two methods, the skin method and the more efficient subtraction method are described in the theoretical manual.

2.4 STRUCTURAL ANALYSIS

The superstructure plus the basement minus the excavated soil (Fig. 2.l-1[c]), will be collectively called the near-field zone. This entire zone may be modeled in two or three dimensions using the finite element method described in Ref. 1.

Effective methods of modeling this zone and solving the resulting equations of motion, Eq. (2.1-1) or Eq. (2.1-3), are described in detail in Ref. 1.

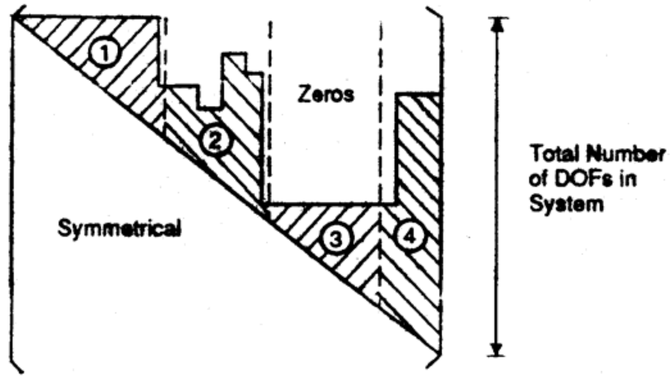
2.5 SUMMARY OF COMPUTATIONAL STEPS

The steady-state equations of motion for the near-field zone are stated in Eq. (2.1-1) for earthquake loading and in Eq. (2.1-3) for direct forces. As indicated by these equations, for each frequency, ω, the impedance matrix, the load vector, and the dynamic stiffness matrices ($C = K - \omega^2 M$) for the structure (including the basement and any irregular zone) and the excavated soil are formed. After forming the equations of motion, they must be solved. The matrices involved are often very large, especially for three-dimensional problems, and must be stored in blocks on low-speed storage devices even when large electronic computers are used. An efficient scheme for performing the necessary operations of SASSI analysis is described in the next chapter.

Below are given some details of the operations which must be performed for each frequency of the analysis:

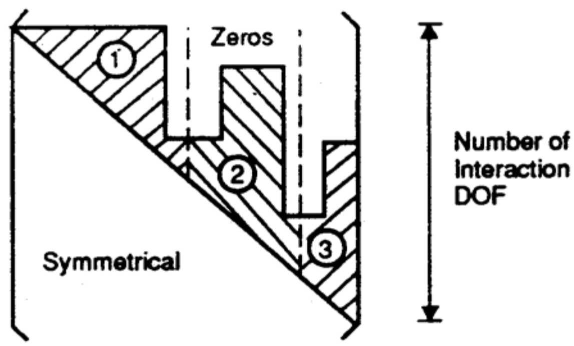
1. <u>Form dynamic stiffness of structure</u>: The total frequency- dependent complex stiffness of the structure is computed from Eq. (2.1-2) using the total stiffness and mass matrices. This matrix is stored in blocks as shown below in preparation for solution by the active column method.

$$\begin{bmatrix} C_{ss} & C_{si} \\ C_{is} & C_{ii} \end{bmatrix} =$$



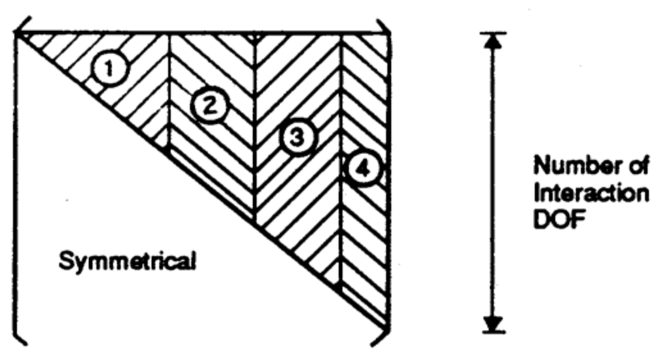
2. Form dynamic stiffness of excavated soil: The total frequency-dependent complex stiffness of the excavated soil is computed from Eq. (2.1-2) using the total stiffness and mass matrices. This matrix is also stored in blocks as shown below.

$C_{ff} =$



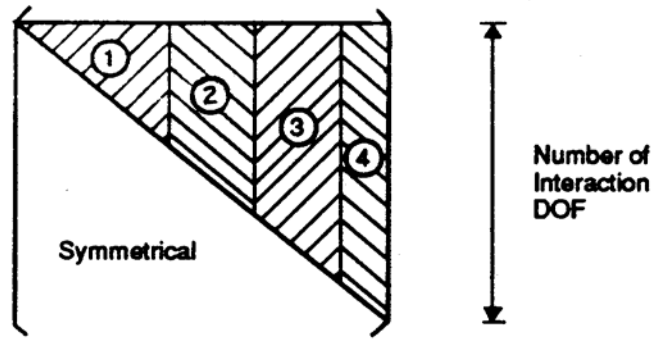
3. Form impedance matrix: If the direct method is used, then the entire flexibility matrix for the interaction nodes is constructed using the method described in Ref. 1. It is stored in blocks as follows:

$F_{ff} =$



The matrix is then inverted in place, using a special subroutine for symmetric matrices, to obtain the impedance matrix, which is stored in the same form:

X_{ff} =



Finally, using the direct stiffness matrix of the excavated soil obtained in Step 2, the entire impedance matrix is computed from Eq. (2.3-2) and stored as for the direct method.

4. Form the total stiffness of the system: The total stiffness of the system can now be obtained by adding X_{ff} to and subtracting C_{ff} from the total stiffness of the structure:

$$\text{coefficient matrix} = \begin{bmatrix} C_{ss} & C_{si} \\ C_{is} & C_{ii} - C_{ff} + X_{ff} \end{bmatrix}$$

5. Triangularization: The coefficient matrix obtained above is then triangularized and stored again in the same form as for the stiffness of the structure.

6. Form load vector: For seismic analysis, the load vector is computed by multiplying the impedance matrix, X_{ff}, by a vector which contains the free-field motions at all the interaction degrees of freedom, u_f',

$$\text{load vector} = \begin{Bmatrix} 0 \\ X_{ff} \cdot u_f' \end{Bmatrix}$$

And for foundation vibration analysis, the load vector is formed from the given external forces,

$$\text{load vector} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

7. Solution of the equations: Finally, the acceleration (or displacement) amplitudes are obtained by forward reduction and back-substitution of the load vector using the reduced coefficient matrix obtained in Step 5.

CHAPTER 3
THE COMPUTER CODE SASSI

A modular computer code, SASSI (Ref. 3), has been developed to perform the operations described in Chapter 2. The code has been arranged specifically for practical applications and has the following characteristics:

a. The site response analysis, the impedance analysis, and the formation of the basic stiffness and mass matrices for the structure can be performed separately and the results stored on magnetic tapes or disks.

b. Thus, if the seismic environment, the external loads, the soil properties, or the arrangement of the superstructure are changed, only part of the computation needs to be repeated.

c. The final solution is stored (in the form of transfer functions) on a magnetic tape from which specific information can be extracted when required without recomputation of the entire solution.

d. Both deterministic (time history) and probabilistic results can be obtained from the above tape.

3.1 LAYOUT

The general layout of the system is shown in Fig. 3.1-1.

a. HOUSE

In this program the mass and stiffness matrices of all the elements used in the model are computed and stored on Tape 4. These properties are frequency independent and the computation is performed only once.

b. MOTOR

This program forms the load vector in Eq. (2.2-1) or (2.2-3). The loads may correspond to impact forces, rotating machinery, or simple unit forces to be used to determine the impedance of a foundation. It is possible to allow for loads acting out of phase. The results are stored on Tape 9.

c. SITE

This program solves the site response problem. The control point and wave composition of the control motion are defined. The information needed to compute vector $\{U'_f\}$ used in Eq. (2.2-1) is computed at this stage and is saved on Tape 1. The program also stores information required for the transmitting boundary calculation on Tape 2. The actual time history of the control motion is not required in this program.

d. POINT

This program consists of two subprograms, namely, P0INT2 and P0INT3 for two- and three-dimensional problems, respectively. The modules described in Sections 4.1 and 4.2 are solved and the results, which provide the information required to form the flexibility matrix, are saved on Tape 3. Tape 2 created by program SITE is used as input.

e. INCOH (**not available**)

This program is used to develop the free field vector following the formulation of incoherence models described in Chapter 2 of the theoretical manual using Tape 7 from HOUSE. The results are saved on Tape 11 to be used in ANALYS.

f. SPILE

This program is used to solve for single pile and pile groups using pile impedance method described in Chapter 5 of the theoretical manual. The input tape is Tape 2 from POINT and the results are saved on Tape 31 for use in ANALYS.

g. MATRIX

In this program, Tapes 3 and 4 are used as input to form the impedance matrix [X_{ff}], for each frequency. The impedance matrix is saved on Tape 5. The coefficient matrix in Eqs. (2.1-1), (2.1-37) is also formed, triangularized, and stored on Tape 6.

h. LOADS

Using the data of Tapes 1 and 9, this program computes the load vectors in Eqs. (2.1-1), (2.1-3) for each frequency and stores them on the internal tape.

i. SOLVE

In this program the reduced stiffness matrices are read from Tape 6. The program then performs the back-substitution using the load vectors on the internal tape. If Eq. (2.1-1) is being solved, the solution is the transfer functions from the control motion to the final motions. If Eq. (2.1-3) is being solved, the solution is a set of transfer functions from external loads to total displacements. In either case, the results are stored on Tape 8.

The subprograms MATRIX, LOAD, and SOLVE are combined into a controlling program called ANALYS.

Interpolation of the transfer functions in the frequency domain and further output requirements are handled by the following subprograms.

j. MOTION

This program is a post-processor. It reads the transfer functions from Tape 8, performs the interpolations described in Section 6.3, and computes the final response at a specified node selected by the user. Tape 4 is also used as input to provide the information on master and slave nodes. Acceleration, velocity, or displacement of the response in terms of time history, peak value, or the response spectrum may be requested.

k. STRESS

This program reads Tapes 8 and 4 and computes requested stress, strain, and forces time histories and peak values in structural members.

l. COMBINE

If after interpolation it is found that some additional frequencies need to be included, this program combines the corresponding Tapes 8 and produces a new Tape 8.

m. PLOT

This program reads Tape 7 from HOUSE and generates screen and printer plots of the model in HOUSE. Selected element and node groups may be used for plotting and importing to WORD or EXCEL files.

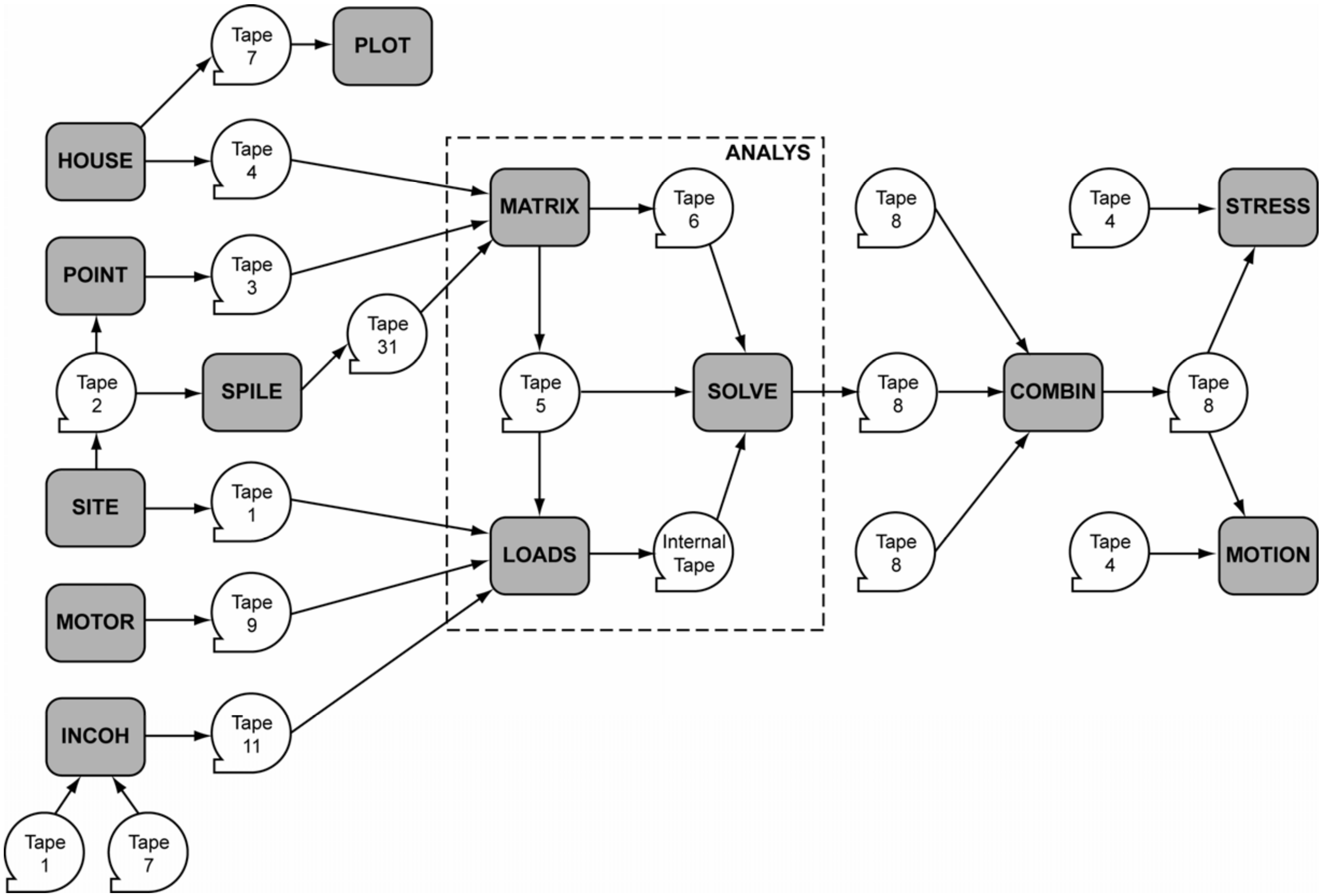


Figure 3.1-1. Layout of Computer Program SASSI

3.2 OPERATIONAL FEATURES

The system shown in Fig. 3.1-1 has been specifically designed to provide maximum flexibility and economy for practical applications.

Clearly, on any given project most of the programs have to be executed. However, if changes occur in the design parameters, only parts of the system have to be re-executed. Consider the following changes:

a. Change in control motion

Suppose results are required for a different time history (or response spectrum) of the control motion. Then, as long as the nature of the seismic environment (i.e., the type of wave field) is not changed, only the program module MOTION or STRESS has to be re-executed.

b. Change in seismic environment

Suppose the structure was originally analyzed for the effects of vertically propagating body waves and that results are required for the case of incident Rayleigh waves causing the same motion at the control point as in the free-field. In this case only part of the program module SITE and all of the program modules, LOAD, SOLVE, and MOTION have to be re-executed. This is so because the information on Tapes 2, 5, and 6 remains unchanged.

c. Change in dynamic loading

If changes are made in dynamic loads applied directly on the structure, only the program modules MOTOR, LOAD, SOLVE, and MOTION have to be re-executed. However, if only the time history of dynamic loads is changed while the loading pattern is not changed, only the program module MOTION has to be re-executed.

d. Changes is superstructure

If changes are made in the superstructure, only HOUSE and part of MATRIX, SOLVE, and MOTION have to be re-executed.

e. Postprocessors

Since the data on Tape 8 are independent of the manner in which the frequency content of the control motion is specified, the user has at any time the choice of deterministic or probabilistic analysis.

Also, since the postprocessors can be restarted from Tape 8, the user has at any time the option of requesting additional responses when needed. This is especially important for three-dimensional analysis for which the volume of the complete solution output easily becomes unmanageable.

CHAPTER 4
APPLICATION GUIDE

4.1 INTRODUCTION

Performing a three-dimensional soil-structure interaction (SSI) analysis by the computer program SASSI may become lengthy, especially when the number of interaction nodes exceeds several hundred nodes. Thus, it is important that the program users understand the capabilities of the program in order to efficiently perform the SASSI analysis. This requires knowledge of the analytical procedures used in this computer program. A set of guidelines with some examples describing step by step on how to proceed with SASSI analysis is considered to be very helpful for the users to utilize the program capabilities. It is, therefore, the purpose of this chapter to provide such guidelines with examples.

Section 4.2 describes the steps which are usually taken during a SASSI analysis. In the second part of this section, these steps are further elaborated and the important parameters of the analysis are discussed in more detail.

Section 4.3 describes the SASSI program modules which are executed to perform the initiation, post-processing and restarting analyses.

Section 4.4 describes the means of organizing and saving the output tapes generated by the program.

Finally, an example problems is presented in Section 4.5 to show the applications of the program to seismic soil-structure interaction problems.

4.2 SASSI SSI ANALYSIS PROCEDURE

4.2.1 Steps Involved in SASSI Analysis

The seismic soil-structure interaction analysis by the SASSI program usually involves the following steps:

Step 1 Select the time history of control motion and compute its response spectrum to highlight the dominant frequencies contained in the input motion.

Step 2 Determine the possible important frequency ranges of the soil-structure interaction response by examining the dominant frequencies of the structure(s) with the fixed-base condition. These frequencies can be obtained either by SASSI or another standard finite element program such as SAP, STARDYNE, etc.

Step 3 Based on the results of Steps 1 and 2, determine the cut-off frequency of the analysis (see item 7 of Section 4.2.2).

Step 4 Based on the assumption of vertically propagating shear wave and specified location of the control motion, compute the strain-compatible free-field soil properties by using, for example, the computer program SHAKE (see item 14 of Section 4.2.2).

Step 5 Based on the cutoff frequency and the iterated soil properties obtained from the SHAKE analysis performed in Step 4, select the soil profile for the SASSI analysis (see item 8 of Section 4.2.2).

Step 6 Select the discrete structural model as explained in item 9 of Section 4.2.2.

Step 7 Select the discrete excavated soil model as described in item 10 of Section 4.2.2.

Step 8 Select the method of computing the impedance matrix as described in item 11 of Section 4.2.2.

Step 9 Select the frequencies for which the site response and point load problems are to be solved. It is recommended to choose at least 15 to 20 frequencies, possibly at equal intervals. Note, however, that the impedance problem and the final response of the system are usually solved at fewer frequencies than those specified above. These frequencies are selected according to the information obtained from Steps 1 and 2 and later on can be increased, if necessary, to improve the accuracy of the interpolated transfer functions.

Step 10 Perform SASSI initiation analysis to compute the uninterpolated transfer functions at all the nodes of the system. This analysis, as outlined in Section 4.4.1, requires five computer runs:

a. Execute program module SITE in Mode 1 based on the information of Steps 5 and 9 and the specified location of the control motion. This analysis yields the information needed to form the transmitting boundary in the program module POINT and to solve for the site response problem in the program module SITE in Mode 2. This information is saved on Tape 2 (item A of Section 4.4.1).

b. Execute the program module POINT (POINT2 for two-dimensional and POINT3 for three-dimensional analysis) by using Tape 2 as input and specifying the maximum embedment of the structure and also the radius of the point load as given by the formula in the POINT User's Manual. This radius ensures the compatibility of the point load solution and discretized geometry of the basement of the structure. This analysis yields information on point load solution, saved on

Tape 3, to be used to form the flexibility matrix of the basement of the structure (item B of Section 4.4.1).

c. Execute the program module HOUSE based on the information of Steps 6, 7, and 8. This analysis yields the complex stiffness and mass matrices of the structure and excavated soil, saved on Tape 4 (item C of Section 4.4.1).

d. Execute the program module SITE in Mode 2 to perform the site response analysis by using Tape 2 as input and specifying the nature of the seismic wave field. This analysis yields a set of free-field motions which are saved on Tape 1 (item D of Section 4.4.1).

e. If incoherent ground motion is considered, execute the module INCOH by using Tape 7 from HOUSE and Tape 1 from SITE. The results are saved on Tape 11 to be used by ANALYS. **This module is not available yet.**

f. Execute the program module ANALYS by using Tapes 1, 3, and 4 as input. Tape 11 should be used if ground motion incoherency is considered. This analysis yields the impedance matrices on Tape 5, the triangularized stiffness of the total system on Tape 6, and the final uninterpolated transfer functions at all the nodal points on Tape 8 (item E of Section 4.4.1). The frequencies selected at this stage for interactive analysis are based on the information obtained from Steps 1 and 2. However, in order to expedite the execution of the program module ANALYS as well as reduce the size of the files generated by this program, it is recommended to break the frequency array into several sub-arrays whereby the analysis for each sub-array is performed separately, as discussed in item 12 of Section 4.2.2.

Step 11 Based on the results of Step 10, perform SASSI post-processing analyses to compute the response of the system. This analysis, as outlined in Section 4.4.2, usually consists of the following computer runs:

a. Execute the program module COMBIN, if necessary, to combine the transfer functions (on Tape 8's) obtained for different frequency arrays in Step 10 (item A of Section 4.4.2).

b. Execute the program modules MOTION and STRESS, based on the results of Step 10 and item 1 of this step, to compute the response of the system (item B of Section 4.4.2). MOTION and STRESS use Tapes 4 and 8 as input.

Step 12 It is also possible at this stage to add new frequencies to the response based on the results obtained in Step 11. The procedure involved is described in item 12 of Section 4.2.2.

Step 13 Perform SASSI restarting analysis if changes occur in either the superstructure or the seismic environment of the problem analyzed above. The procedures involved are described in Sections 4.4.3.1 and 4.4.3.2, respectively.

In the case of foundation vibration analysis by the SASSI program, the above steps described for the seismic analysis are still applicable except that:

1. The time history of control motion in Step 1 is replaced by the reference time history of the external dynamic forces.

2. Step 4 is unnecessary.

3. The iterated soil properties in Step 5 are replaced by the initial soil properties

4. The site response problem is eliminated from Step 9 and second part of item a of Step 10.

5. Item d of Step 10 is replaced by a different analysis performed by the program module MOTOR to obtain the load vector on Tape 9, which replaces Tape 1 in item e of Step 10.

6. The dynamic environment in Step 13 is replaced by the external dynamic forces.

4.2.2 Considerations Prior to SASSI Analysis

In order to make effective use of the program SASSI, the user should consider the following items:

1. Rigid vs Flexible Basement

   Currently, the program does not take advantage of the rigid basement assumption since a flexible basement is always assumed in the program. As a result, no saving is obtained by the rigid basement assumption. It is, therefore, recommended to carry out the analysis for actual properties of the basement. If it is necessary to evaluate the effect of rigid basement assumption, a restart analysis can be performed by selecting an elastic modulus for basement to be 10^4 to 10^5 times the elastic modulus of the soil. The restart analysis can be performed at a fraction of the initial cost.

2. Surface vs Embedded Structure (ZSRFCE)

   It is possible to treat the embedded structures as surface structures if the embedment effect is negligible. By doing so, a big saving can be achieved in the SASSI analysis since the number of interaction nodes is largely reduced.

The parameter ZSRFCE in the program HOUSE controls the embedment condition of the structure. For surface structures, this parameter is equal to the Z-coordinate of the base of the structure.

3. Symmetry of the System (NSYMPL)

SASSI has the capability to take advantage of the geometrical symmetry of structures subjected to symmetric or anti-symmetric loading. Therefore, the cost of the analysis can be drastically reduced by utilizing this capability of the program.

The parameter NSYMPL is used to specify the number of symmetric planes of the system.

4. Rigid Base Rock vs Halfspace Condition (LSUB)

SASSI has the capability to simulate the existence of a uniformly damped, or undamped, halfspace below the top soil layers. Therefore, it can avoid using very deep soil models with many sub layers and leads to additional savings when the soft soil extends to relatively large depth or when the rock boundary can not be established.

In case of halfspace simulation, the program automatically adds an additional soil layer below the top layers with the thickness of $(1.5*V_s)/f$, where V_s is the shear wave velocity of the halfspace and f is the frequency of analysis. This added soil layer is further subdivided into LSUB sub layers specified by the user in program module SITE. In addition to this, the viscous dashpots are added to the base of the new soil model.

The input parameter LSUB is recommended to be set at 10. If LSUB=0, there is no halfspace simulation.

5. Cutoff Frequency (f_{NF})

The cutoff frequency is an important parameter since it not only sets an upper limit on the number of frequencies to be analyzed but also controls the maximum allowable element sizes, and thus, the dimension of the stiffness and mass matrix of the problem. The input form of the cutoff frequency f_{NF} to SASSI is described in item 7. The factors governing the choice of the cut-off frequency are:

a. The frequency content of the input motion.

b. The dominant frequencies of the entire system.

c. The time increment of the input time history.

6. Selection of frequency points (f_i)

The frequencies to be selected for the SASSI analysis depend on the number of peaks in the transfer function and how close these peaks are located relative to each other. This information can be obtained from the fixed-base analysis of the structure. The fixed-base natural frequencies will show the approximate location of the peaks in the structure, and the importance of each peak can be seen from either the mode participation factors or the fixed-base transfer functions. Since an efficient interpolation scheme on complex response functions has been incorporated into SASSI, and since the effect of the soil-structure interaction is to flatten the sharp peaks and sometimes eliminate some of the structural peaks, it is usually sufficient to solve for 10 to 20 frequencies and the intermediate solutions can then be obtained by interpolation.

Also, SASSI enables the user to add new solved frequency responses to the old solved frequency set. Therefore, it is possible to start the analysis with few frequencies, and then examine the transfer functions and add new frequencies as needed.

7. Input of Frequency Points [$NFREQ_i$)

In the SASSI program, the transfer functions are computed at discrete frequency points which are integer multiples of the frequency step, DF. For general deterministic analysis, the frequency step is calculated from

$$DF = 1/(DT * NFFT)$$

where the input parameters DT and NFFT are the time step and number of points to be used in the Fourier transform of the time history, respectively. For probabilistic analysis of single harmonic forced vibration analysis, the time history input is not required; therefore, the user can directly specify DF.

Once the frequency step is defined, the frequency points f_i are input through the use of integer frequency numbers, $NFREQ_i$, defined as follows:

$$NFREQ_i = f_i/DF \qquad i = 1, 2, ... NF$$

where NF is the total number of frequency points selected for the analysis according to item 6 of this section.

The maximum frequency number to be specified in the SASSI program is controlled by the cutoff frequency and can be obtained as follows:

$$NFREQ_{NF} = f_{NF}/DF$$

where f_{NF} is the cutoff frequency.

8. Modeling of Soil Profile

The soil supporting the structure must consist of semi-infinite elastic or viscoelastic horizontal layers resting on a rigid base rock or semi-infinite elastic or viscoelastic halfspace.

The allowable layer thickness for the SASSI analysis is determined using the simple rule that the layer thickness must not exceed one fifth the wavelength at the highest frequency of analysis. Based on this, the soil profile is selected by subdividing the soil layers into several sub layers.

9. Modeling of Structure

The structure is model led by the two- or three-dimensional finite elements available in the SASSI program. The selection of elements and nodal points follows the same procedure as in the other standard finite element programs. The only limitations are:

a. The structure must contain the interaction nodes inside the basement volume(s) even if there is no structural element to connect such nodes.

b. All the interaction nodes of the structure(s), which are below the ground surface, must lie on the soil layer interfaces.

c. The maximum horizontal distance between two adjacent interaction nodes in an excavated volume must be smaller than $V_s/(5*f_{NF})$, where V_s is the smaller shear wave velocity of the top and bottom soil layers connected to the two interaction nodes and f_{NF} is the highest frequency of analysis.

10. Modeling of Excavated Soil

In the SASSI analysis of the embedded structures, the excavated soil zone(s) must be modeled by the two dimensional plane strain or three-dimensional solid elements connecting the interaction nodes of the structure. The element sizes for the excavated soil elements are controlled by the distance between the interaction nodes obtained in item 9 in this section.

11. Direct vs Skin vs Substraction Method (NIMP)

Currently, there are three methods available for calculation of the impedance matrix in SASSI, namely, the direct (NIMP=1), the skin (NIMP=2) and the subtraction (NIMP=3) methods. In the direct method, the impedance matrix of the interaction nodes is computed by the direct inversion of the corresponding dynamic flexibility matrix. A less rigorous but more cost-effective solution can be obtained by using the skin method, where the impedance matrix is computed through a special formulation by combining sub matrices of the dynamic flexibility matrix and the direct stiffness matrix of the excavated soil. The most efficient method is the subtraction method, where the impedance matrix is only needed for the basement boundary nodes.

12. Addition of new frequencies to the transfer function on Tape 8

The program SASSI has an additional provision which enables the user to solve for the transfer function at any additional frequencies by the program module ANALYS. The results are later combined by the program module COMBIN, as long as the specified frequencies reside on the input tapes. For example, suppose the program modules SITE and POINT were executed for 10 frequencies (0.98, 2.93, 4.88, 6.84, 8.79, 10.74, 12.70, 14.65, 15.6662, and 17.58 Hz) and Tapes 1 and 3 were created. Furthermore, it is assumed that the program module ANALYS was executed only for 5 frequencies which reside on the above Tapes 1 and 3. Let us now assume that the analysis is to be repeated for two new frequencies (2.93 and 15.66 Hz) and the results are to be combined with those of the old frequencies. Since the new frequencies reside on the above Tapes 1 and 3, and Tape 4 is frequency independent, program module ANALYS is executed to solve for the new 2 frequencies. Subsequently, program module COMBIN can be used to add new frequency responses to the old ones on Tape 8.

13. Impedance Matrices of a Rigid Foundation

The impedance matrices, K and C, of a rigid foundation are computed from the foundation compliance matrices, f and g, using the formula

$$K + i\omega D = (f + ig)^{-1} \qquad (4.2\text{-}1)$$

where ω is the frequency of analysis and $i = \sqrt{-1}$.

In the program SASSI, the columns of the foundation compliance matrices can be obtained by applying a unit amplitude force or moment in the desired direction at a specified point on the foundation, and computing the resulting real and imaginary parts of displacements or rotations of that point (i.e., f and g). By inverting the compliance matrices using the above formula, the corresponding impedance matrices (i.e., K and D) are computed.

14. Nonlinear Soil Behavior

SASSI is a frequency domain analysis program which uses the principle of superposition, and therefore is restricted to linear analysis. However, approximate nonlinear analyses can be performed using an iterative scheme called the Equivalent Linear Method. In applying this method to seismic soil-structure interaction problems, it is useful to consider the nonlinear effect in two parts. The effect due to free-field motion is called the primary nonlinear effect, and the effect due to interaction is called the secondary nonlinear effect. The latter one is confined to a limited region near the structure (irregular zone) and has only a minor influence on the motions of deeply embedded structures, such as nuclear power plants. Thus, in many cases, it is sufficient to consider only the primary nonlinear effect, i.e., the iteration on soil properties involved in the equivalent linear method needs to be performed only for the free-field analysis.

In view of the above assumption, an efficient way of performing a seismic soil-structure interaction analysis is to separate it into two stages. The first stage is a site response analysis with iteration on soil properties, say, by the computer program SHAKE. This analysis takes care of all primary nonlinear effects and yields a horizontally layered site profile with properties that are compatible with the levels of strain in the free-field. The second stage is an interaction analysis by SASSI. In this stage, the above strain-compatible soil profile is used for the site profile and no iterations are performed on the soil properties; i.e., the secondary nonlinear effect is neglected.

The secondary effect in a limited region encompassing the basement of the structure can be considered by including in the model an extended near-field zone. The soil strains within the irregular part of this zone can be computed by the program, and the properties of the irregular zone can be changed iteratively according to the equivalent linear method. The procedure for the secondary effect iterations has not yet been implemented in the computer program SASSI. If need be, the iterations can be performed by hand calculations of the new soil properties and input to SASSI for reanalysis.

The secondary nonlinear effect may be important for surface structures subjected to high-intensity earthquakes. They may also have some influence on the distribution of dynamic earth pressures for embedded structures.

4.3 ADDITIONAL NUMERICAL AND MODELLING CRITERIA

4.3.1 Finite Element Discretization

In order to accurately transmit the waves, the finite element model should be discretized so that the largest side of each element does not exceed $\lambda/8$; where λ is the shortest wavelength of interest in the analysis. The wavelength criterion can be relaxed to $\lambda/5$ if the mass matrix used in the analysis is constructed from the combination of consistent and lump mass matrices (usually 50% each). Since the mass matrix computation in SASSI is automated to consist of 50% lump mass and 50% consistent mass, the $\lambda/5$ criterion can be used in constructing the models.

4.3.2 Halfspace Simulation

In order to simulate the halfspace condition at the bottom boundary, two techniques of variable depth method and viscous boundary at the base are used (Ref. I). In the variable depth method, n extra layers with total thickness of 1.5λ and with the properties of halfspace are added to the soil profile. The wavelength, λ, is the shear wavelength in halfspace and is a function of frequency. Thus, the added soil layer thickness varies with frequency. The choice of 1.5λ arose from the observation that fundamental modes of Rayleigh wave in halfspace decay with depth and essentially vanish at a depth corresponding to 1.5λ. Furthermore, the 1.5λ layer thickness is subdivided into n layers with increasing thickness with depth. The choice of n=10 is sufficient for many practical cases. With this technique, the layer thickness will increase with depth and decreasing frequency. This is the desired characteristic of the model since surface wave mode shapes decrease exponentially with depth and since their depth of penetration increases with decreasing frequency.

The soil model with added extra layers is further improved by replacing the rigid boundary at the base of the extended layer system with viscous boundary by placing dashpots in horizontal and vertical directions. The halfspace simulation is specified in program module SITE.

The effectiveness and adequacy of this halfspace simulation technique have been demonstrated in benchmark problems which were used as validation test problems for SASSI (Ref. 2).

4.4 HOW TO RUN A SASSI JOB

The first step in running a SASSI job is to determine to which of the following three groups the job belongs:

1. Initiation (or basic runs)

2. Post processing

3. Restarting

Each of the above groups requires one or several interrelated SASSI program modules, shown in Fig. 4.4-1, to be executed in a specified order. Therefore, the next step is to perform the operations of the corresponding group described in the following sections.

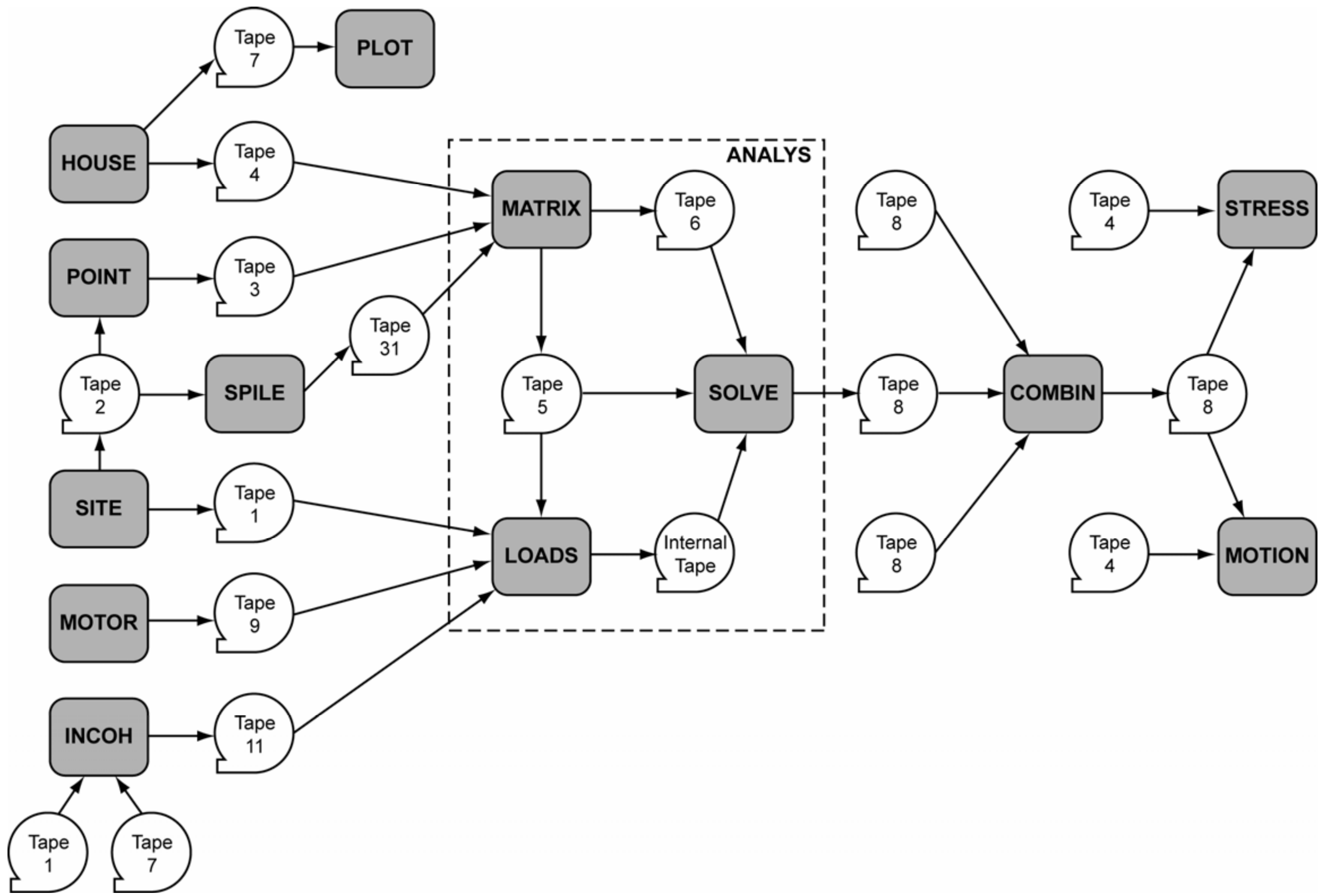


Figure 4.4–1. SASSI Program Modules

4.4.1 Initiation

The initiation of SASSI for interaction analysis basically consists of 5 major computer runs:

A. RUN I - SITE RUN (Mode I)

B. RUN 2 - POINT RUN

C. RUN 3 - HOUSE RUN

D. RUN 4 - SITE RUN (Mode 2) and INCOH RUN or MOTOR RUN

E. RUN 5 - ANALYS RUN (Mode I)

These runs generate the output Tapes 1 through 9 which are then saved on either the disk or the magnetic tapes. Tape 8 contains the un-interpolated transfer functions computed for the specified frequencies at all the nodal points of the system and is used in the postprocessors to compute the final response of the system. The remaining tapes contain the information which may be used later to restart the program module ANALYS to analyze a new problem.

The 5 basic runs listed above are illustrated in Figs. 4.4-2 for seismic coherent motion analysis, Fig. 4.4-3 for seismic incoherent analysis and Fig. 4.4-4 for forced vibrations problem, respectively. For seismic problems RUN 1 and RUN 4 can be combined into one run at the user's option. It is also possible to change the order of the runs such that RUN 3 is performed before RUN 1 is executed. However, since the foundation of the structure (in HOUSE) must be compatible with the soil layer system (in SITE) and the point load solution (in POINT), it is recommended that the first runs be performed after the models for the structure and the site have been established.

To compute for the impedance function of single piles and pile groups when using the pile impedance method, the runs shown in Fig. 4.4-5 are executed. In these runs, SITE, SPILE are executed to get the impedance functions for single piles. For pile groups, HOUSE, MOTOR and ANALYS are executed to get the compliance from the ANALY results.

4.4.2 Post processing

Once Tape 8 is obtained, it is used to compute the required response(s) of the system through one or more of the following runs:

A. RUN 1 - COMBIN RUN

B. RUN $(2)_1$ - MOTION RUN

C. RUN $(2)_2$ - STRESS RUN

RUN 1 is necessary only if new frequencies are to be added to the old Tape 8. These runs are illustrated in Figure 4.4-6.

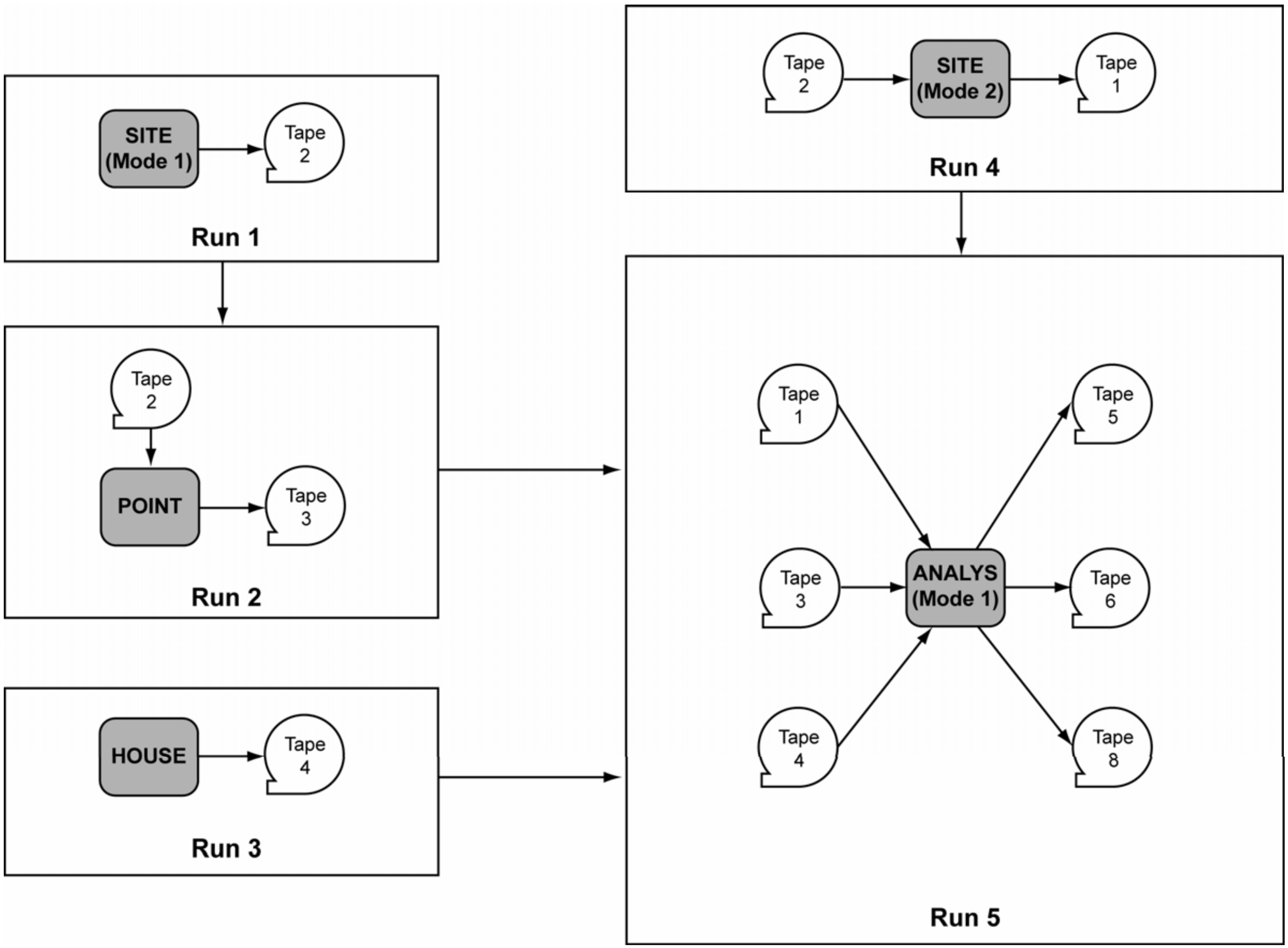


Figure 4.4–2. SASSI Initiation Runs for Seismic Interaction Analysis (Coherent Motion)

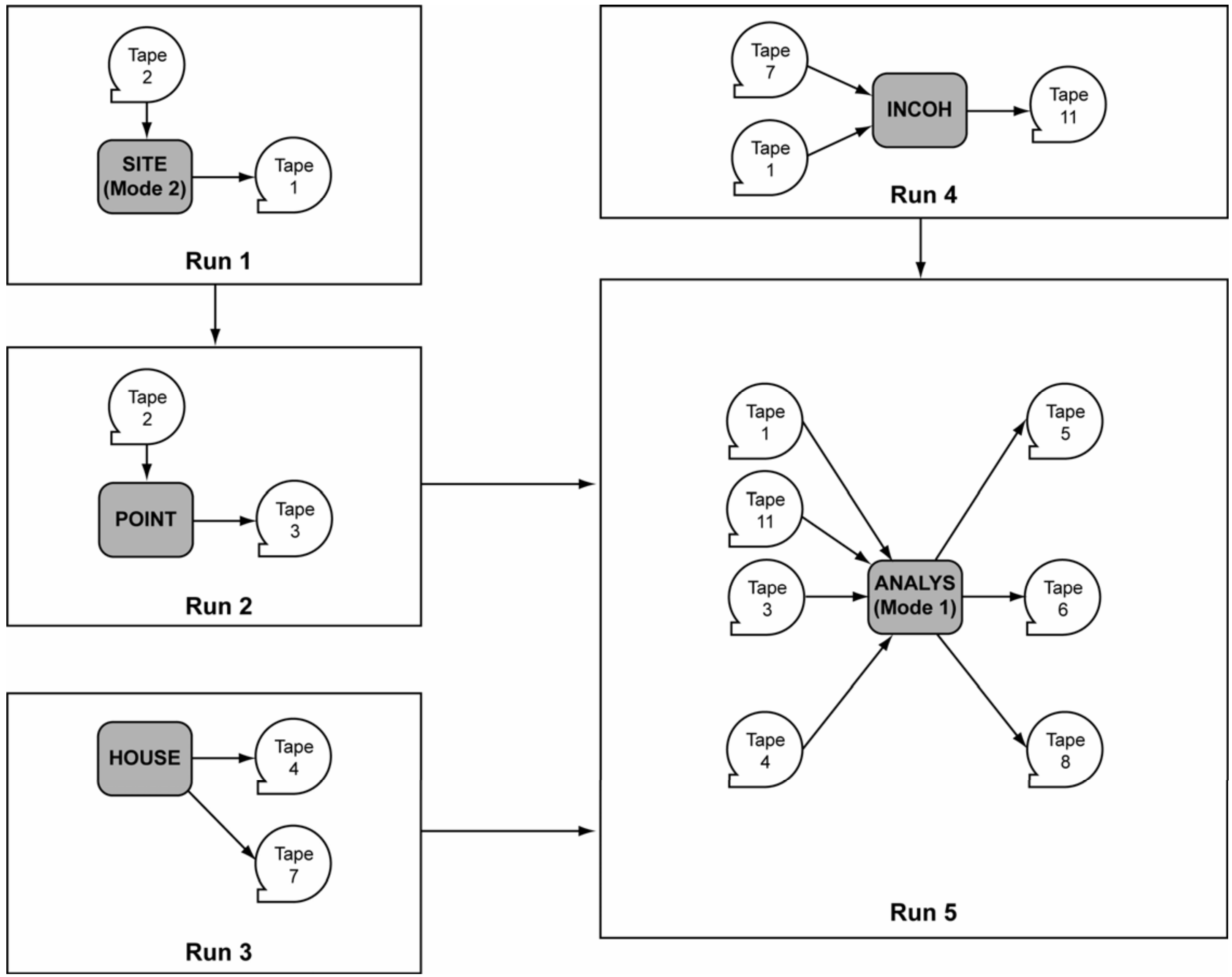


Figure 4.4–3. SASSI Initiation Runs for Force Vibration Interaction Analysis (Incoherent Motion, module INCOH not ready yet)

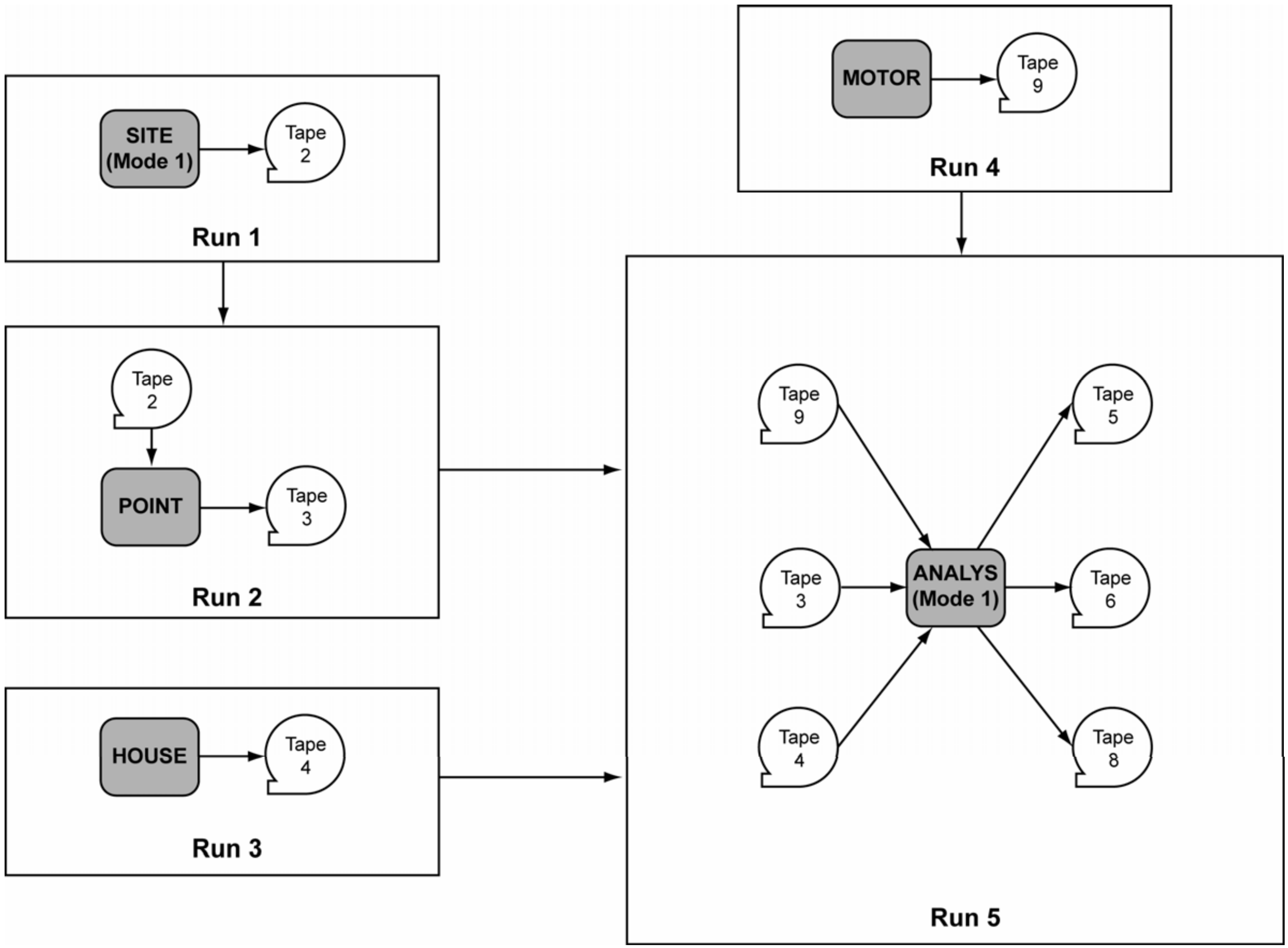


Figure 4.4–4. SASSI Post processing Runs

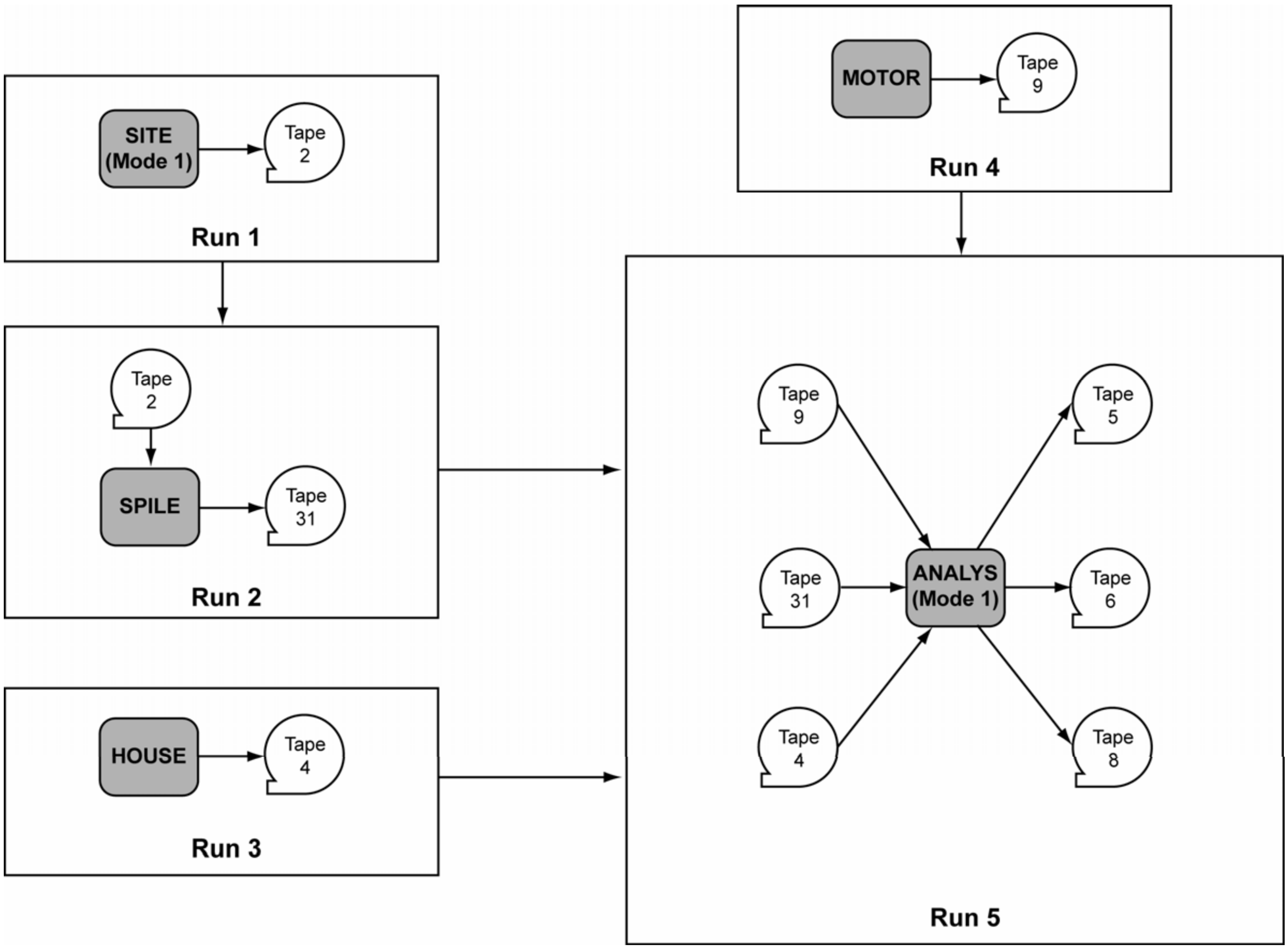


Figure 4.4–5. SASSI Initiation Runs for Pile Impedance Method

4.4.3 Restarting

The basic restart modes of SASSI are:

1. Restart with new superstructure

2. Restart with new seismic environment

3. Restart with new dynamic loading

Each of the above modes involves only two computer runs as described below.

4.4.3.1 Restart with New Superstructure

This mode which can be performed for both the seismic and foundation vibration problems consists of the following two runs:

A. RUN 1 - HOUSE RUN

B. RUN 2 - ANALYS RUN (Mode 2)

The new Tape 4 obtained from RUN 1 is used with the old Tape 5 and Tape 1 (or Tape 9) as input to RUN 2. RUN 2 will then create Tape 6 and Tape 8. Figures 4.4-7(a) and 4.4-7(b) illustrate these runs for the seismic and foundation vibration analyses, respectively.

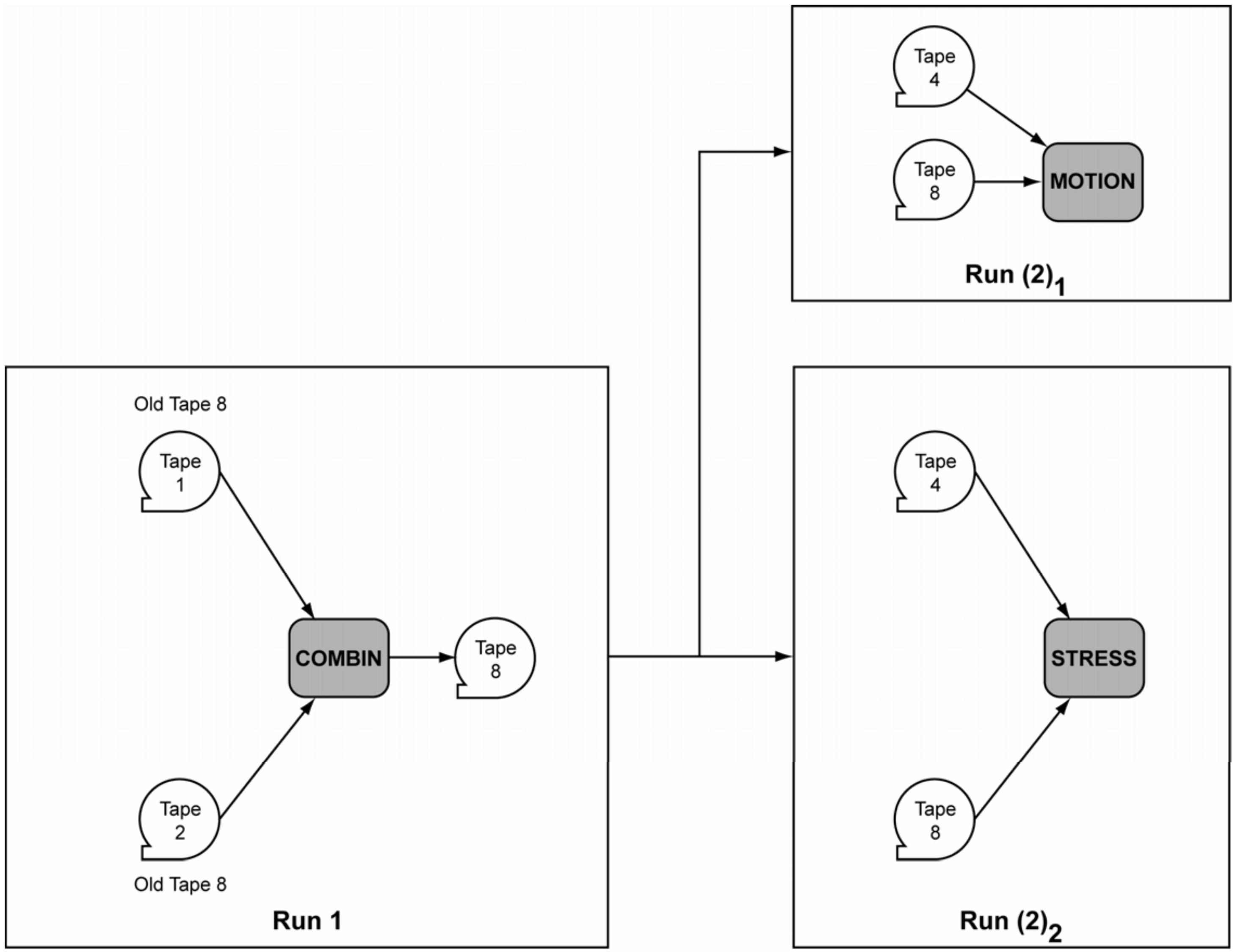
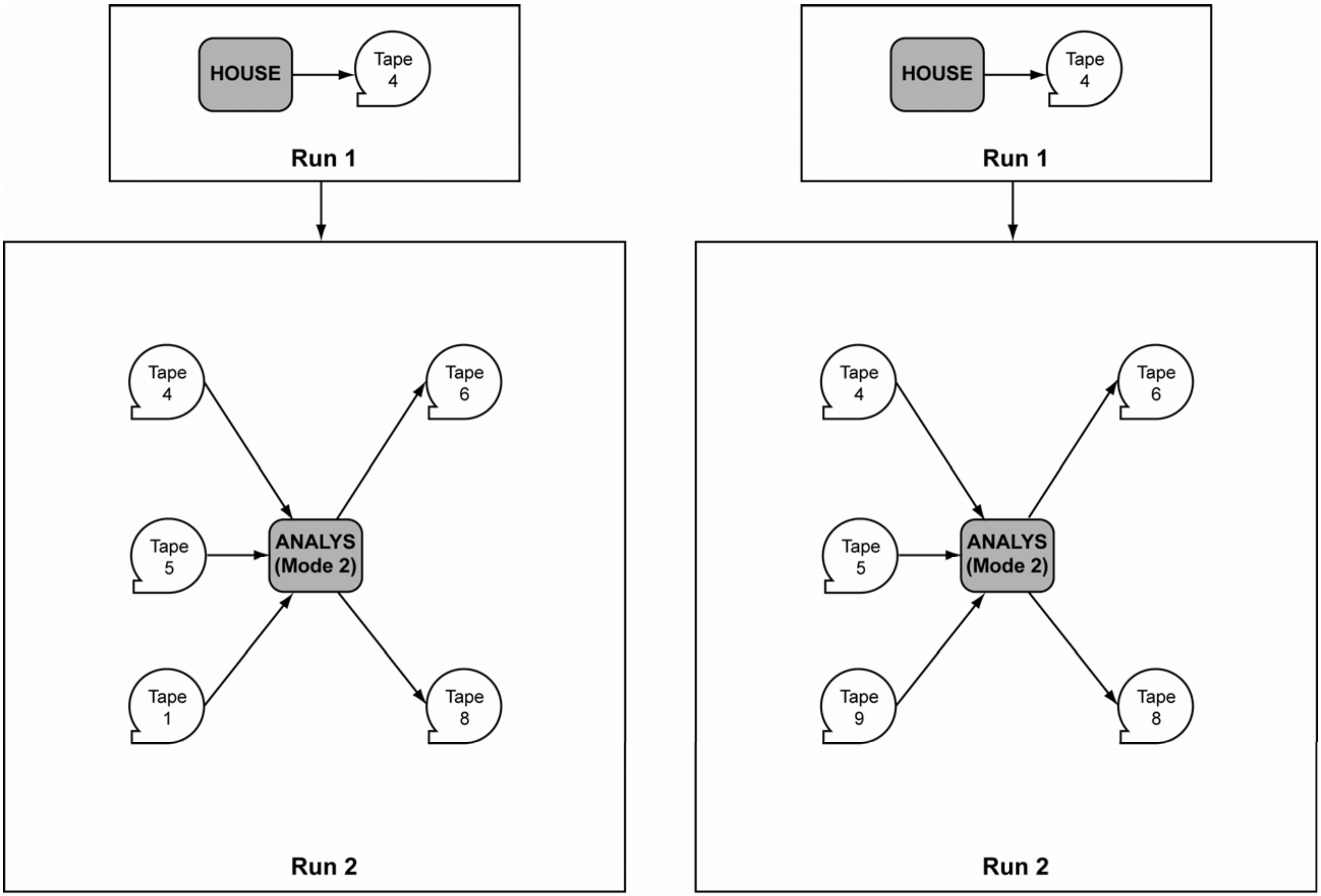


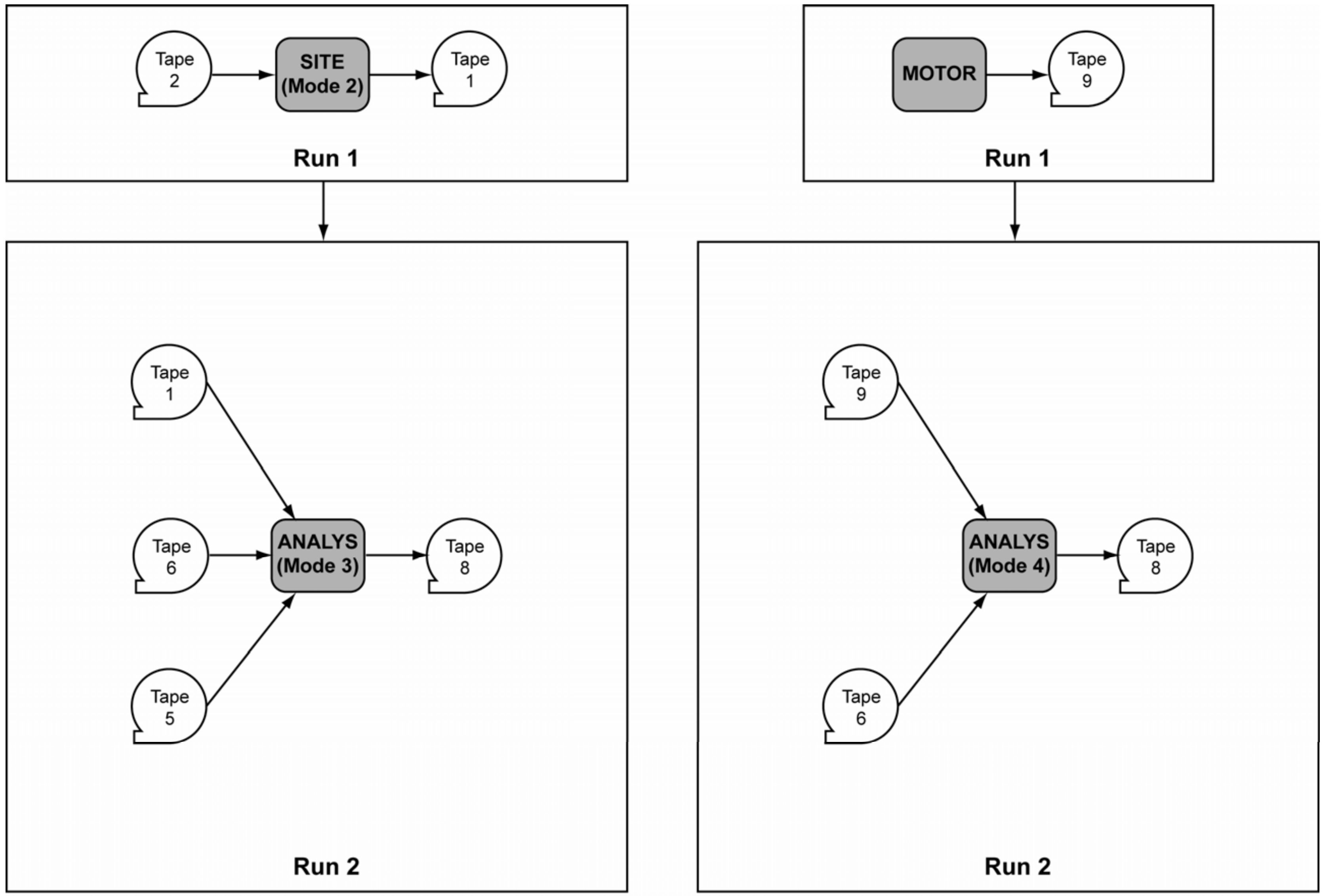
Figure 4.4–6. SASSI Post processing Runs



(a) Seismic Analysis

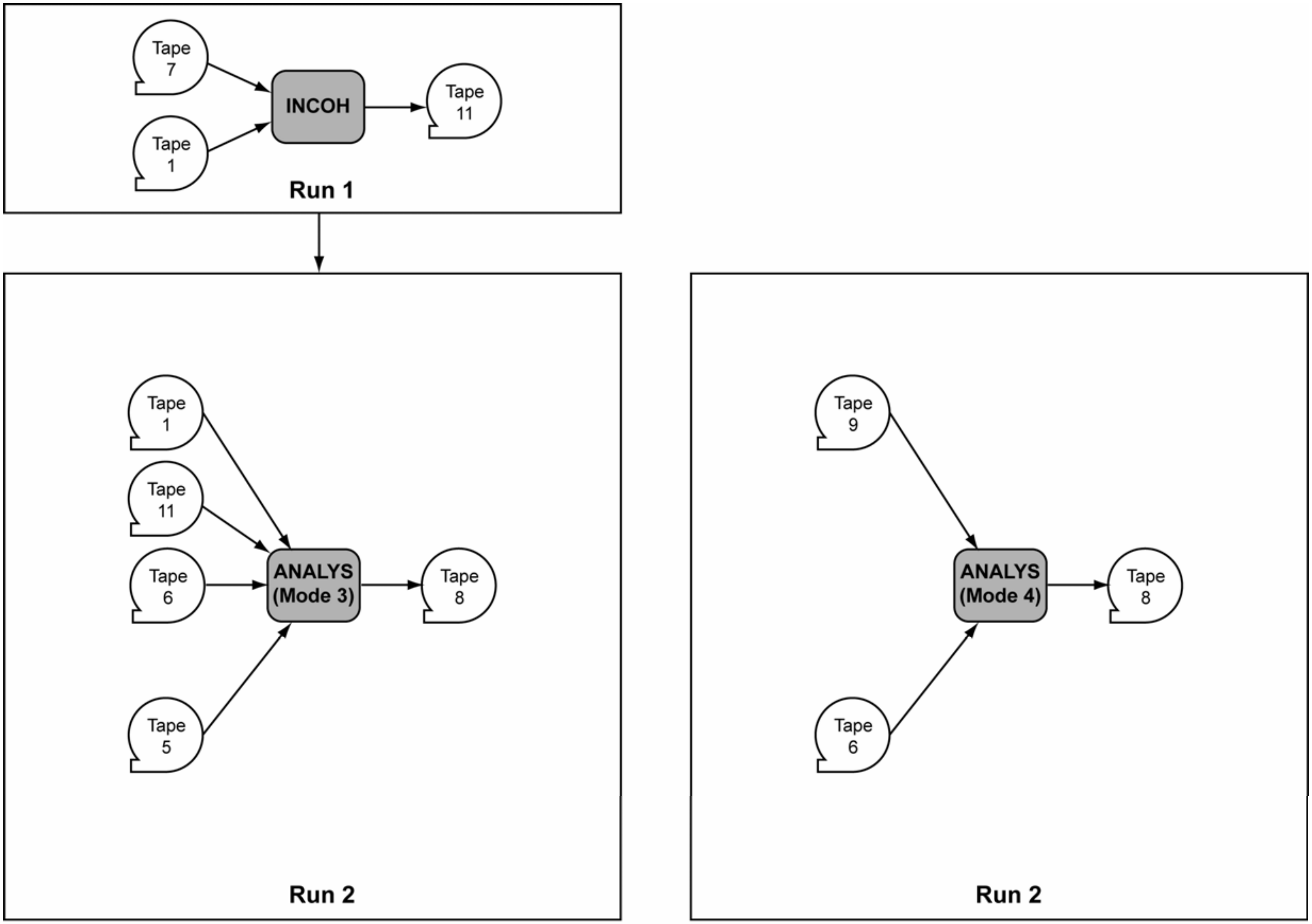
(b) Force Vibration Analysis

Figure 4.4–7. SASSI Restart Runs for New Superstructure Case



(a) New Seismic Loading (b) New Direct Loading

Figure 4.4–8. SASSI Restart Runs for New Loading Case



(a) New Seismic Loading (b) New Direct Loading

Figure 4.4–9. SASSI Restart Runs for New Seismic Loading Case (Incoherent Motion, INCOH not available)

4.4.3.2 Restart with New Seismic Environment (Coherent Motion)

This mode, which can be performed only for seismic problems, consists of the following two runs:

A. RUN 1 - SITE RUN (Mode 2)

B. RUN 2 - ANALYS RUN (Mode 3)

The new Tape 1 obtained from RUN 1 is used with the old Tape 5 and Tape 6 as input to RUN 2. RUN 2 will then create a new Tape 8. These runs are illustrated in Figure 4.4-8(a).

4.4.3.3 Restart with New Dynamic Loading

This mode, which can be performed only for the foundation vibration problems, consists of the following two runs:

A. RUN 1 - MOTOR RUN

B. RUN 2 - ANALYS RUN (Mode 4)

The new Tape 9 obtained from RUN 1 is used with the old Tape 6 as input to RUN 2. RUN 2 will then create a new Tape 8. The runs are illustrated in Figure 4.4-8(b).

4.5 SAVING SASSI OUTPUT FILES

The basic SASSI runs create 8 files, as shown in Figures 4.4-2, 4.4-3 and 4.4-4. However, the number of files created in a SASSI analysis usually exceed 8 because of:

a. Repeating the entire analysis for new frequencies

b. Restarting the program with new superstructures

c. Restarting the program with a new seismic environment (or dynamic loading)

Therefore, it becomes very important to organize these files in such a way that the later access to the files can be done easily.

There are basically two types of device which can be used on any computer system to save the data in the output files: disks and magnetic tapes.

4.5.1 Disk Storage

Even though it is more convenient and faster to save or access the data on disk, the permanent disk storage is not recommended due to its high cost especially for the case in which the amount of data generated on some of the output files is very large. However, if the disk is selected as the primary storing device, then the user can choose an arbitrary name for each output file, thus making organization and access very easy.

4.5.2 Magnetic Tape Storage

There are two types of magnetic tape storage:

4.5.2.1 Mass Storage on Magnetic Tapes (also called File Manager)

In this type of storage, the output files are first saved on disk and subsequently copied from disk onto magnetic tapes by running a separate job. In some computer systems, these two tasks are performed simultaneously. Therefore, there is no need to submit the second job.

If the files on the mass storage are assigned names and accessed randomly like the permanent files on disk, this type of storage is probably the most effective way of saving and accessing SASSI output files.

In case this type of storage is not available on a computer system, the standard storage described below can be used.

4.5.2.2 Standard/Sequential Storage on Magnetic Tapes

In this type of storage, the output files are saved sequentially on a magnetic tape separated by a file mark. Since the system cannot identify each individual file by the name, the user has to keep track of the order in which the files are saved on the magnetic tapes. This will make it possible to add new files to the existing files on a tape or to skip certain number of files before reading the target file.

4.6 EXAMPLE PROBLEMS

Two example problems are presented to illustrate the application of SASSI to impedance and soil-structure interaction (SSI) analysis. These problems also describe how to select the finite element model for the soil profile, the structure, and the number of frequencies to be used for the analysis.

4.6.1 Example Problem 1, Reactor Building with Surface Foundation

4.6.1.1 Description

The problem considered is a pressurized water reactor (PWR) building supporting on a uniform damped halfspace (see BC-TOP-4A, Ref. 5) subjected to vertically propagating shear waves with the control motion specified at the ground surface. The total soil-structure system is shown in Fig. 4.6-1. The building, which consists of a containment structure and internal walls, is modeled by stick models.

The problem is analyzed for two cases. In the first case, the horizontal and rocking impedance functions of the rigid massless foundation are computed and the results are compared with the reference solution of the problem. In the second case, the SSI response of the containment is computed.

The material properties of the soil and geometry of the structure are given in Fig. 4.6-1. The sectional properties of the structure are given in Table 4.6-1. A uniform critical damping of 2% was used for the superstructure and basemat. The control motion selected for the analysis is a scaled version of the El Centro 1940 NS Component. The acceleration time history of the motion is shown in Fig. 4.6-2. The motion is scaled to maximum acceleration of 0.1g with a duration of 10.24 seconds digitized at time intervals of 0.005 second. The corresponding acceleration response spectra at 2% damping are shown in Fig. 4.6-3. The cut-off frequency used for the analysis is 25 Hz.

4.6.1.2 Modeling

4.6.1.2.1 Soil Model

The maximum allowable layer thickness was computed according to the criterion described in item 8 of Section 4.2.2:

Max. allowable thickness = 2000/(5x25) = 16 ft

Based on this value, the soil profile for the SASSI analysis was selected as shown in Fig. 4.6-4. This profile consists of 4 top layers and 10 extra layers with variable thickness plus viscous dashpots. The extra 10 layers and viscous dashpots are added by the program at the user's request to simulate the halfspace condition.

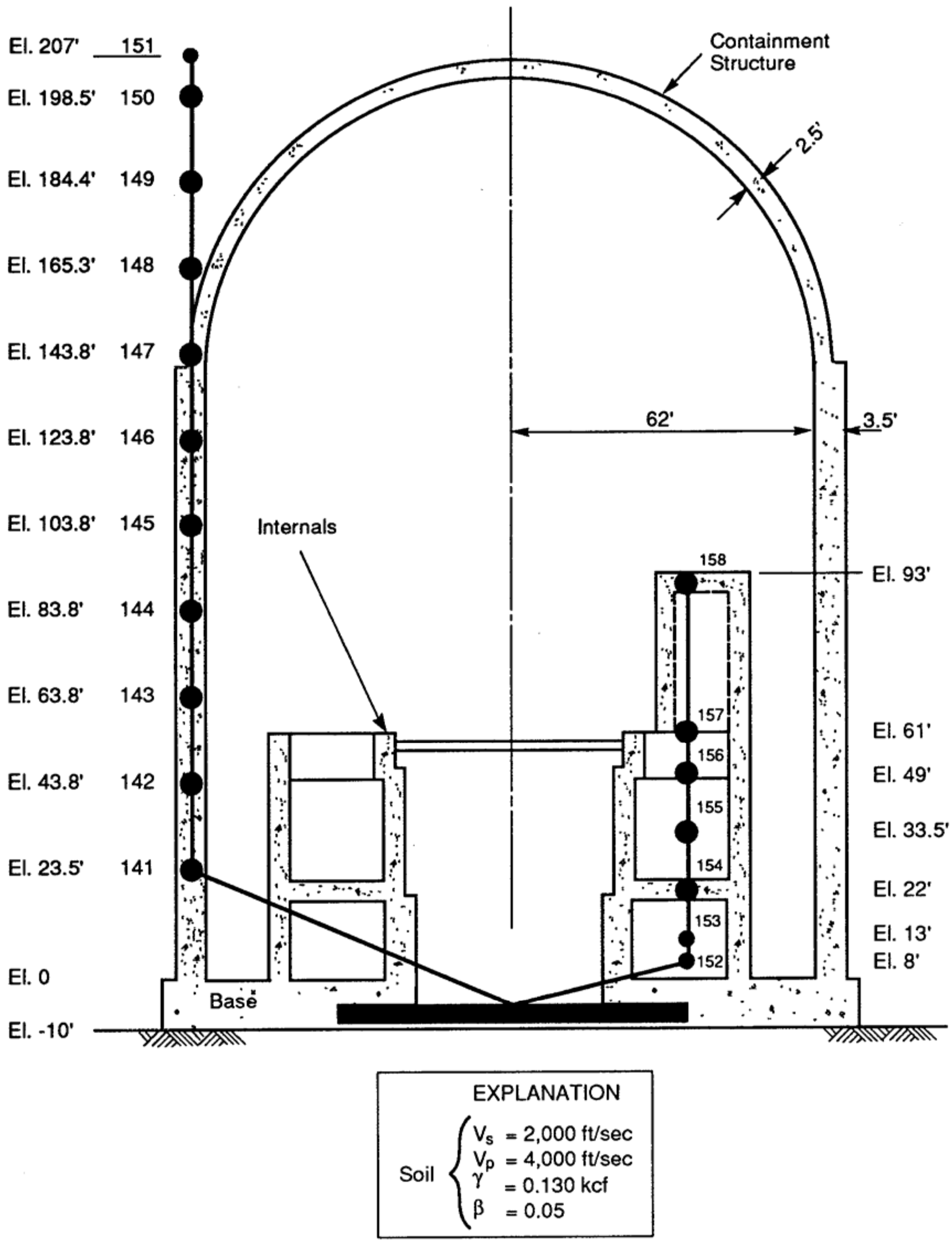


Figure 4.6–1. Lumped-Mass Stick Models of the Containment and Internal Structures

Table 4.6-1. Properties of the Structural Models of the Containment Building and Internals (Concrete Modulus E = 6.9 x 10^5 ksf, G = 2.7 x 10^5 ksf)

Joint Properties			Member Properties			
Mass No.	$M_i g$ (kips)		Location between Joint No.	Area (ft^2)	Shear Area (ft^2)	Moment of Inertia x 10^{-6} (ft^4)
base	20000	C				
1	46000	O	base to 1	1400	700	2.8
3	4200	N	1 to 2	1400	700	2.8
4	4200	T	3 to 4	1400	700	2.8
5	4200	A	4 to 5	1400	700	2.8
6	4200	I	5 to 6	1400	700	2.8
7	4610	N	6 to 7	1400	700	2.8
8	3020	M	7 to 8	990	500	1.9
9	2470	E	8 to 9	990	500	1.5
10	2120	N	9 to 10	990	500	0.8
11	190	T	10 to 11	990	500	0.2
12	2800	I	base to 12	2000	1320	1.1
13	2510	N	12 to 13	2560	1560	1.2
14	6290	T	13 to 14	2210	1460	1.2
15	3760	E	14 to 15	1960	730	1.3
16	8540	R	15 to 16	1740	600	0.9
17	1220	N	16 to 17	780	360	0.2
18	820	A	17 to 18	190	70	0.004

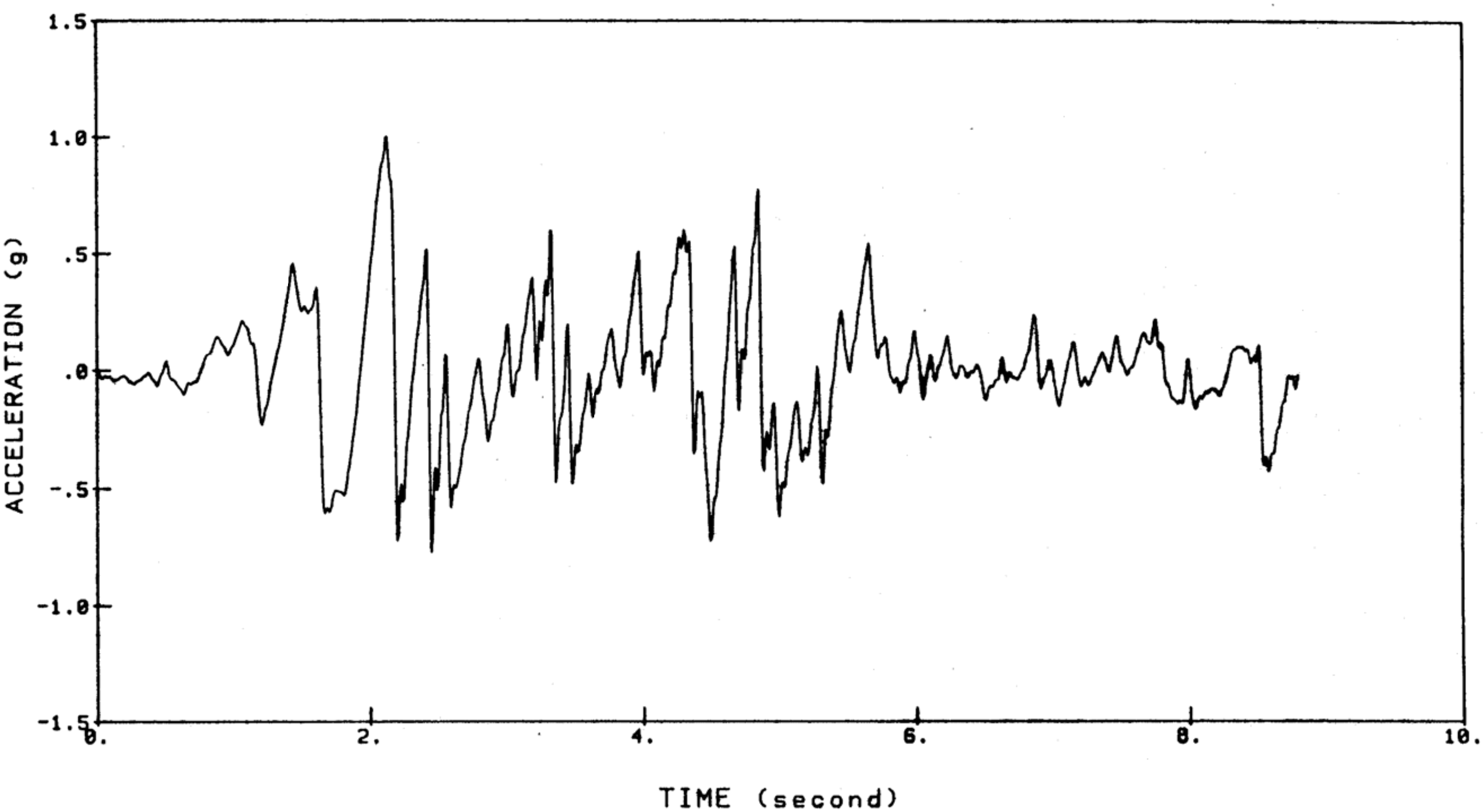


Figure 4.6–2. Acc. Time History of El Centro 1940 (N-S)

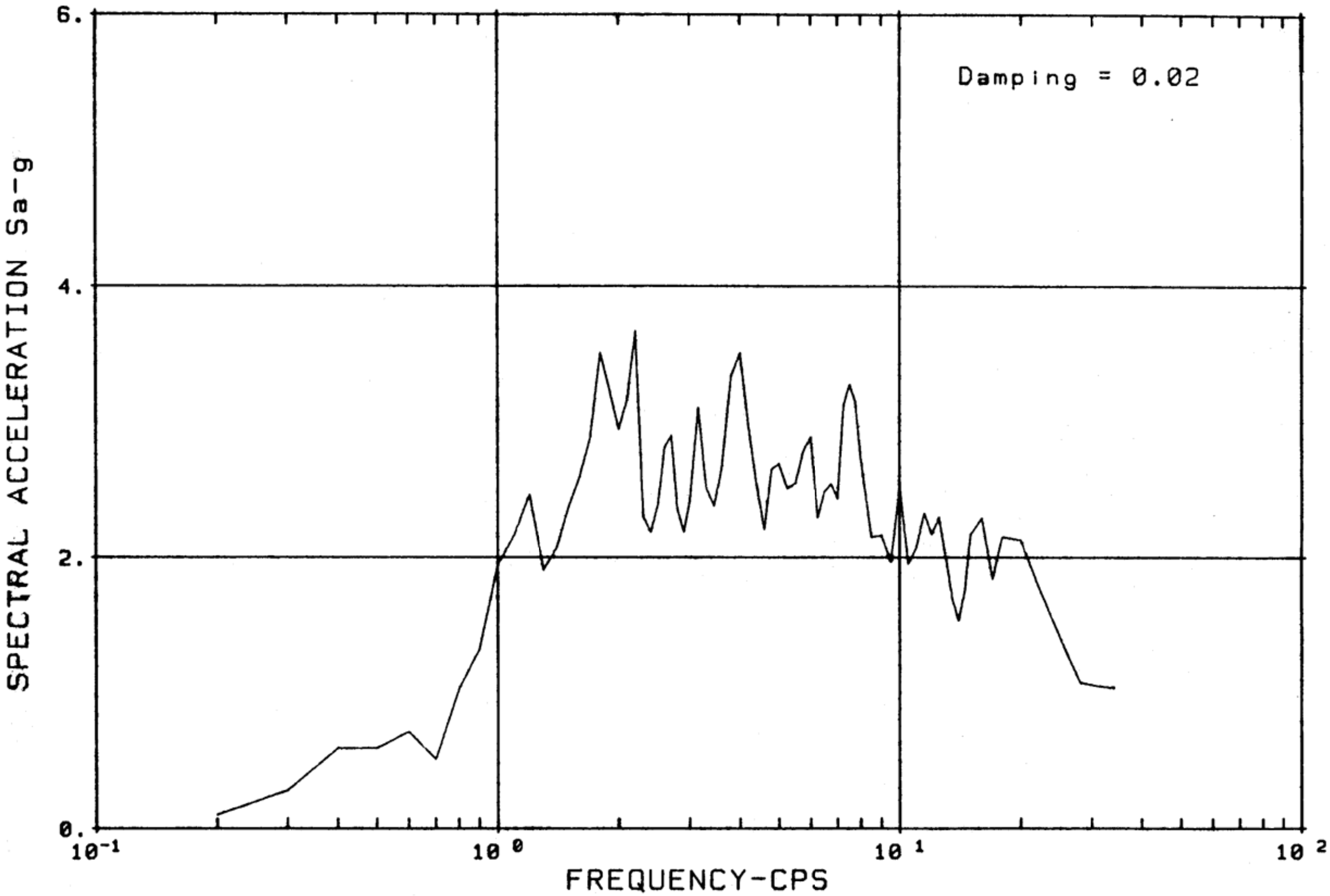


Figure 4.6–3. Acc. Response Spectrum of El Centro (N-S)

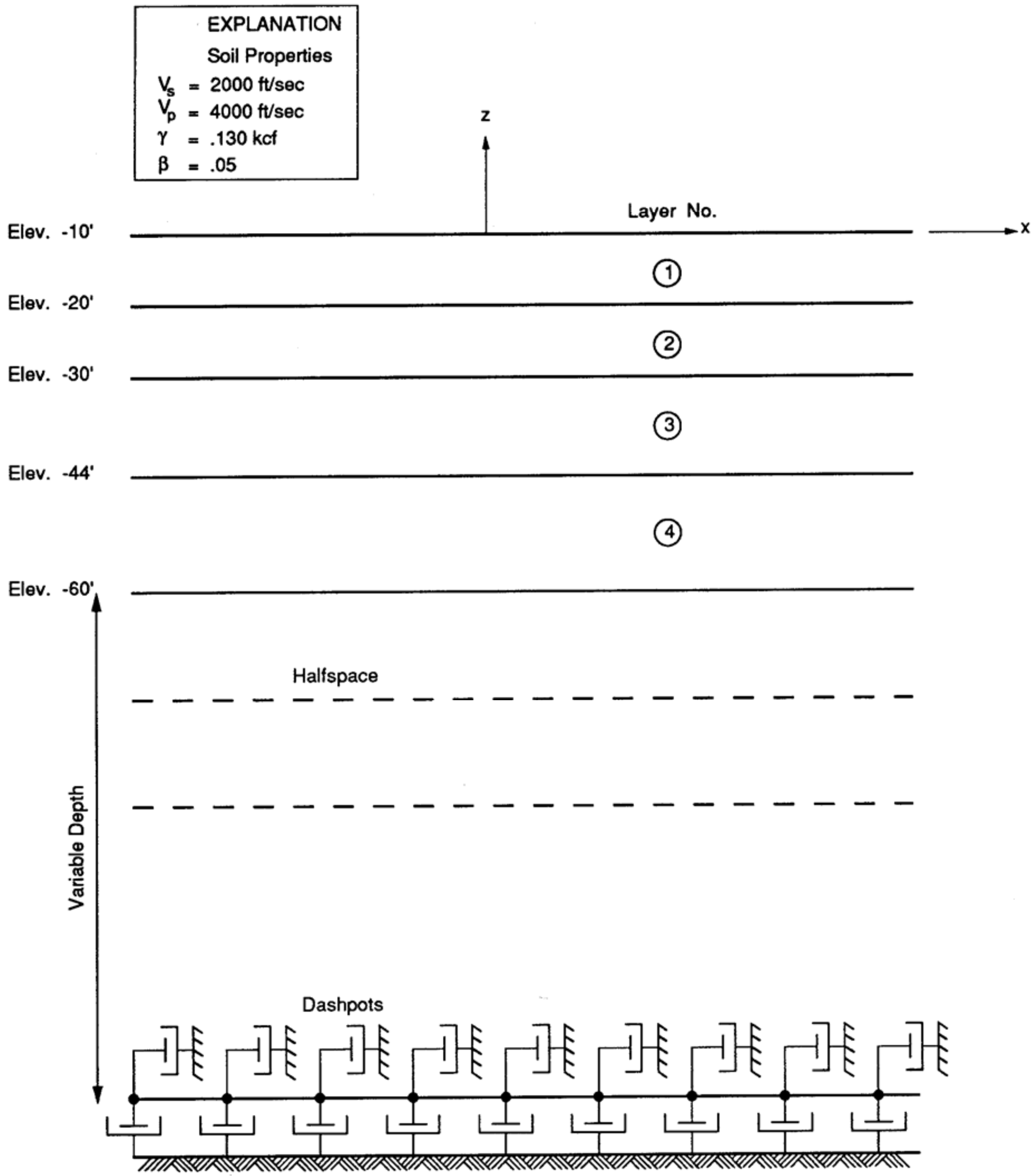


Figure 4.6–4. Soil Profile and Properties

4.6.1.2.2 Structural Model

The structural finite element model used in the interaction analysis is shown in Fig. 4.6-5. It includes the superstructure and the basemat. The superstructure (the containment and internals) are modeled by 18 beam elements as shown in Fig. 4.6-6, and the basemat is modeled by 88 solid elements connected to the underlying soil at 69 nodes as shown in Figs. 4.6-7, 4.6-8, and 4.6-9. Rigid links represented by beams of large flexural and axial rigidities are used to connect the stick models to the basemat. Fig. 4.6-10 shows the rigid link model for the containment structure and internal walls, respectively.

The wavelength criterion described in item 9 of Section 4.2.2 was used to select the element sizes of the basemat in the horizontal direction.

$$\text{Max. allowable element size} = \frac{2000}{5 \times 25} = 16 \text{ ft}$$

The selected element sizes for the basemat slightly violates the 16-foot length limit obtained above, but its effect is considered to be negligible.

Rotational mass of the superstructures was ignored during these analyses.

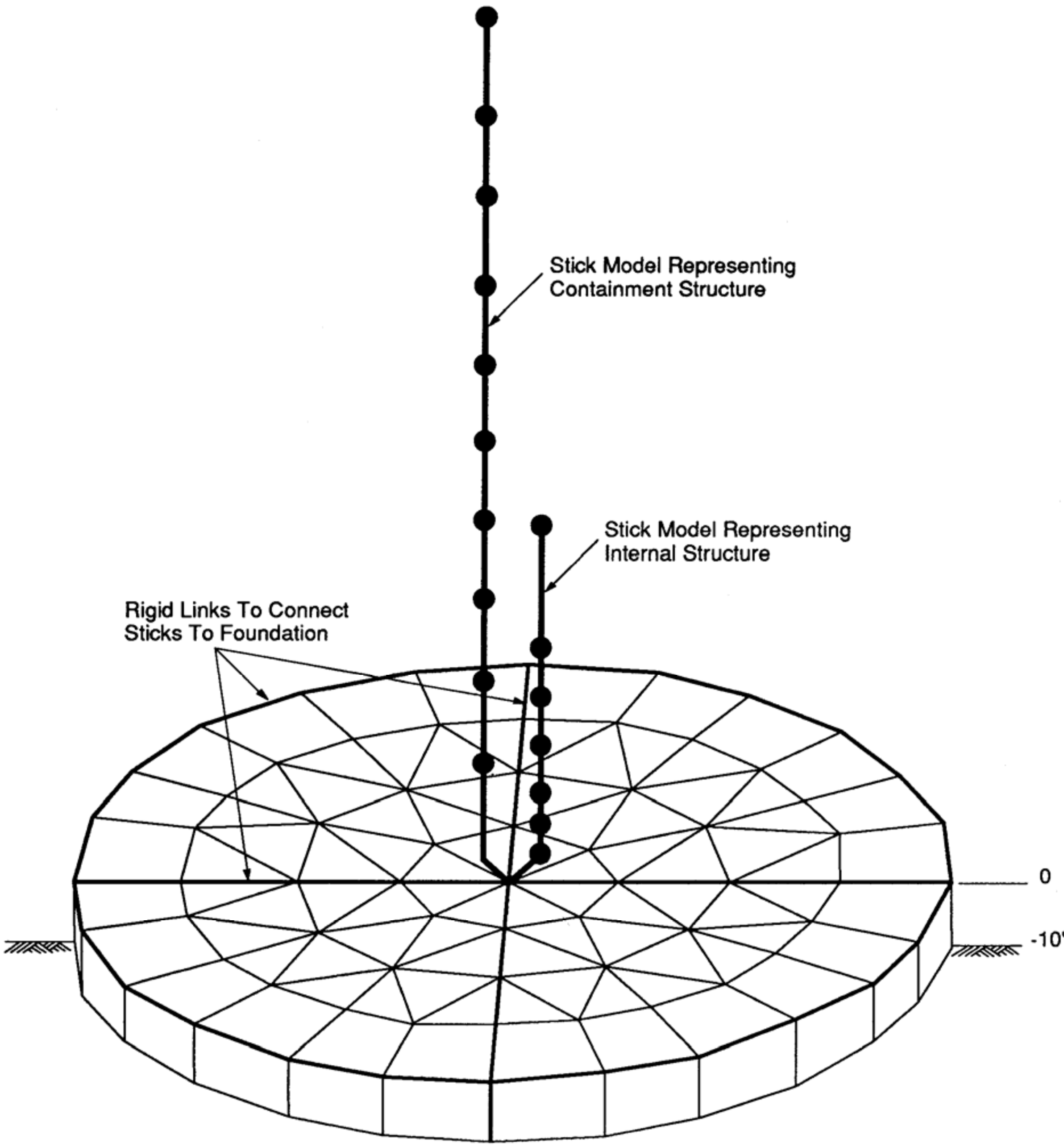


Figure 4.6–5. Finite Element Model

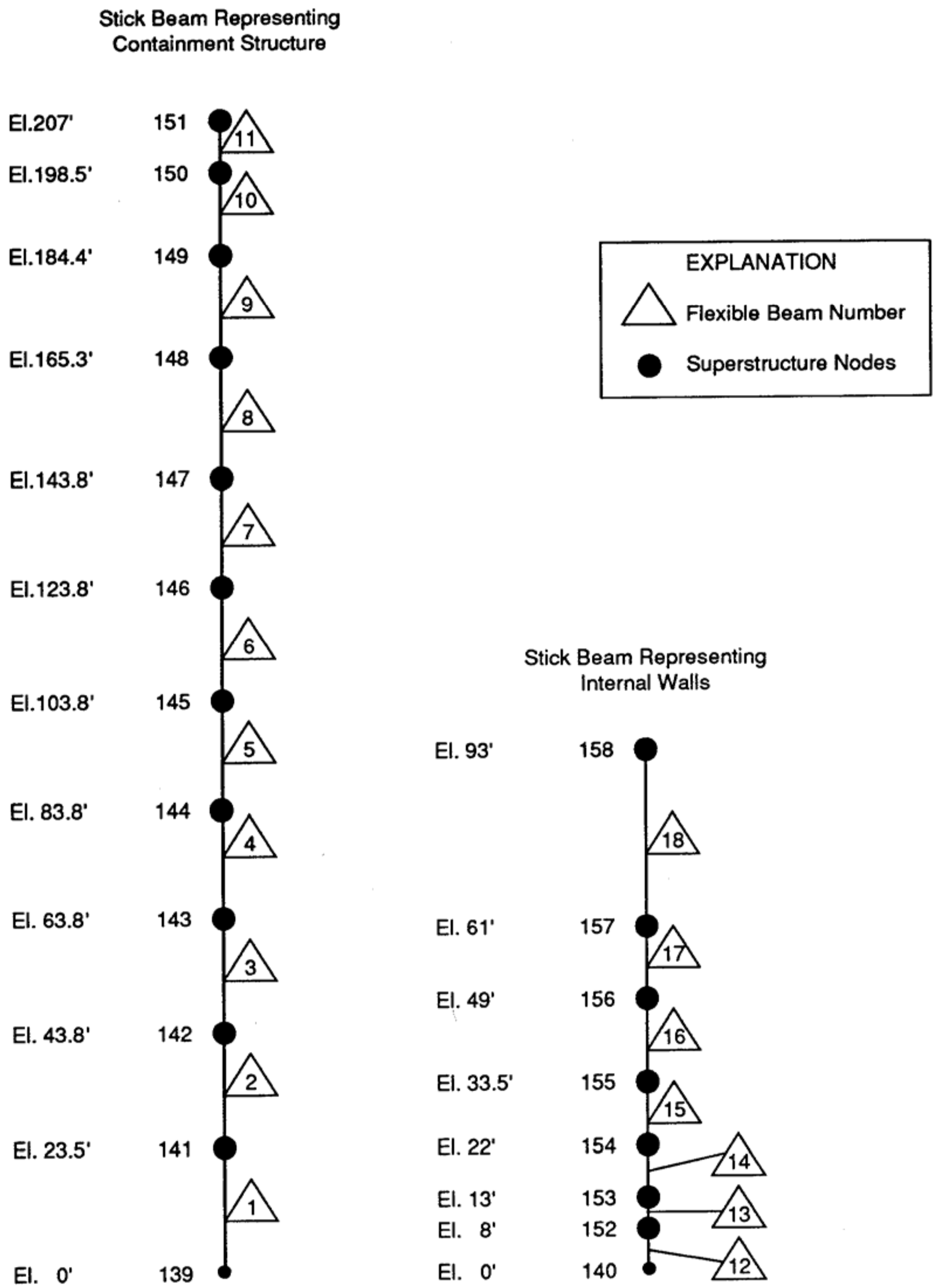


Figure 4.6–6. Finite Element Stick Models Representing Containment and Internal Structures

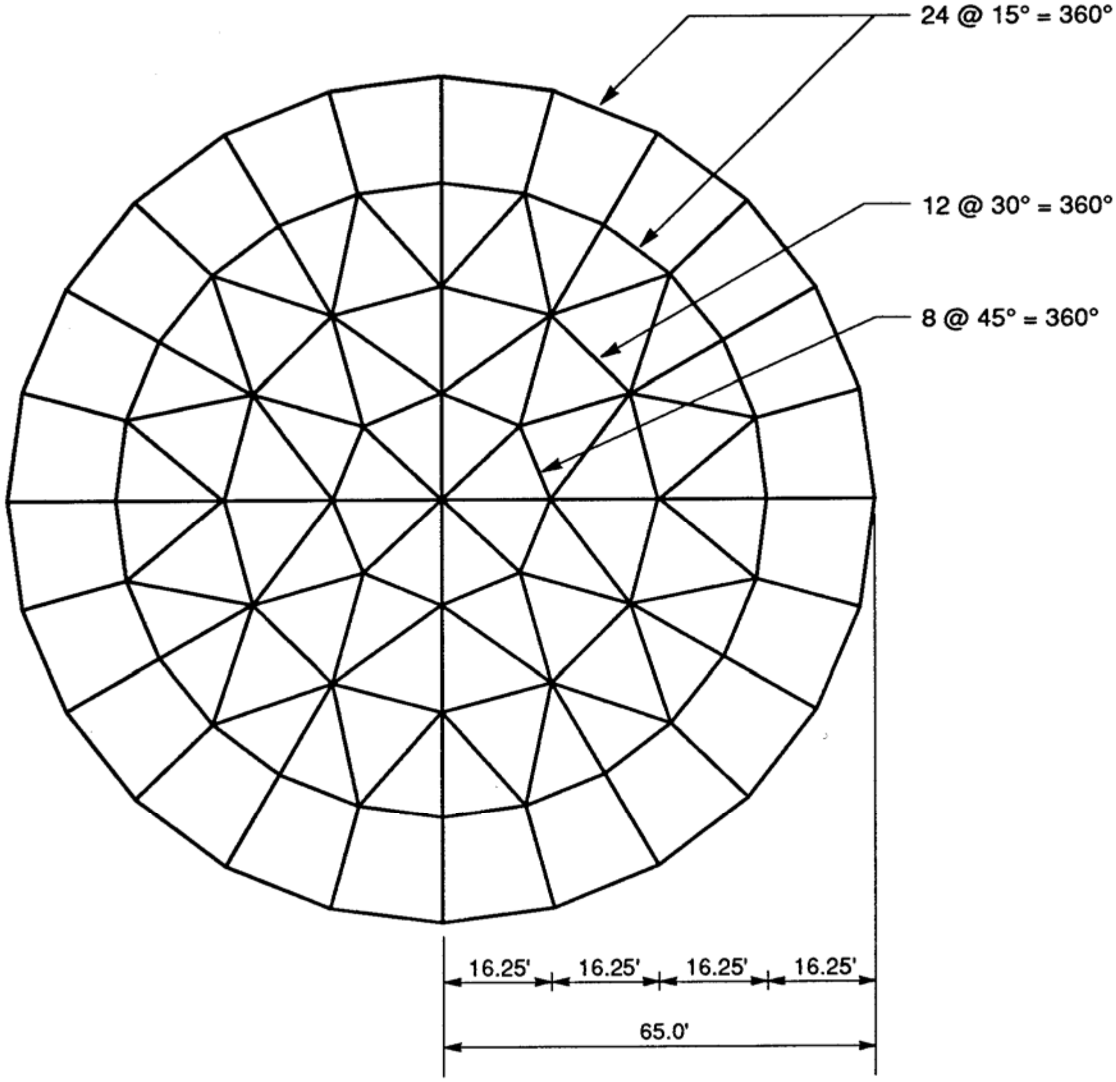


Figure 4.6–7. Geometry of Basemat Discretization

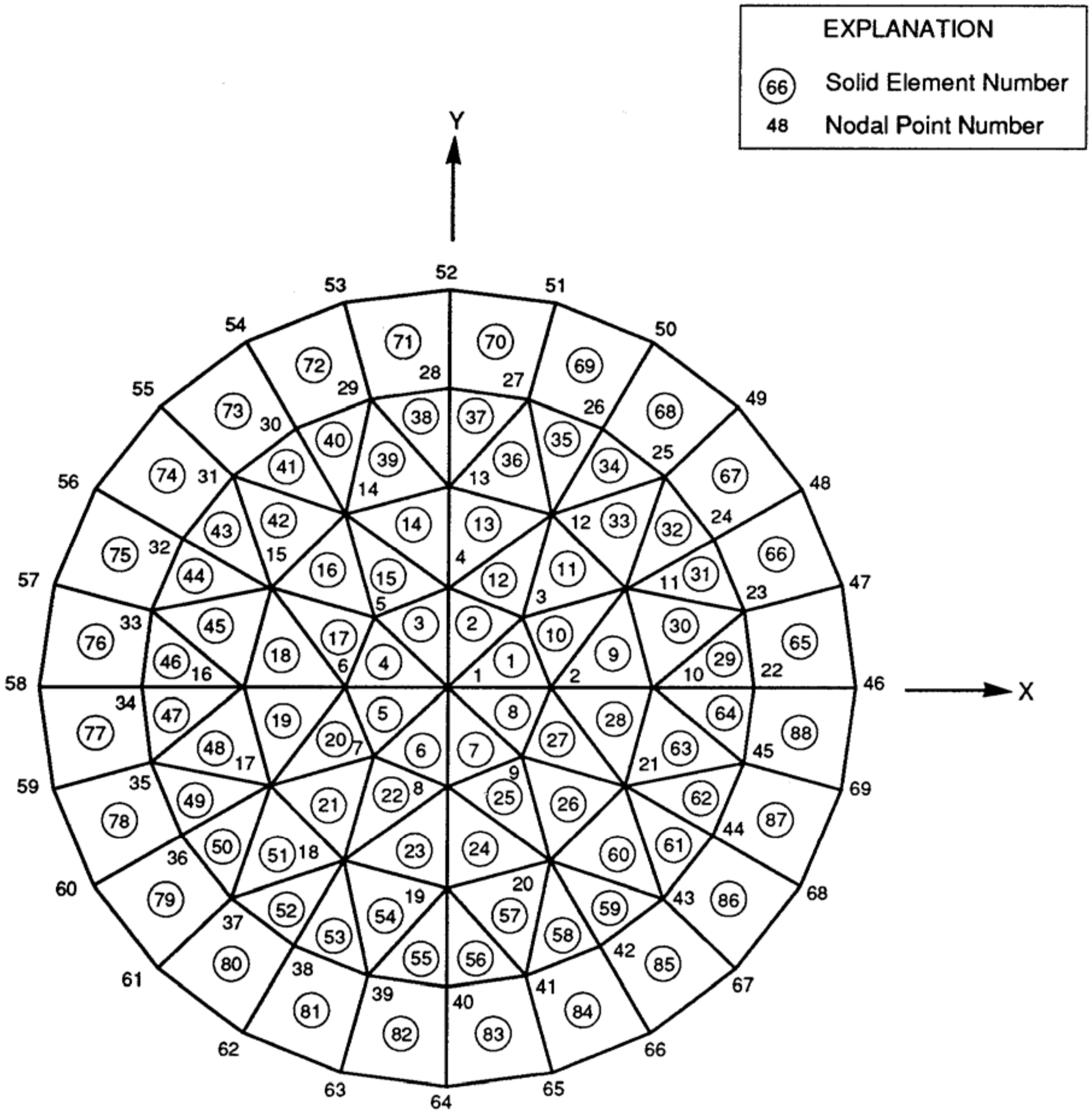


Figure 4.6–8. Nodal Point and Element Numbers for Basemat (Elevation – 10 ft)

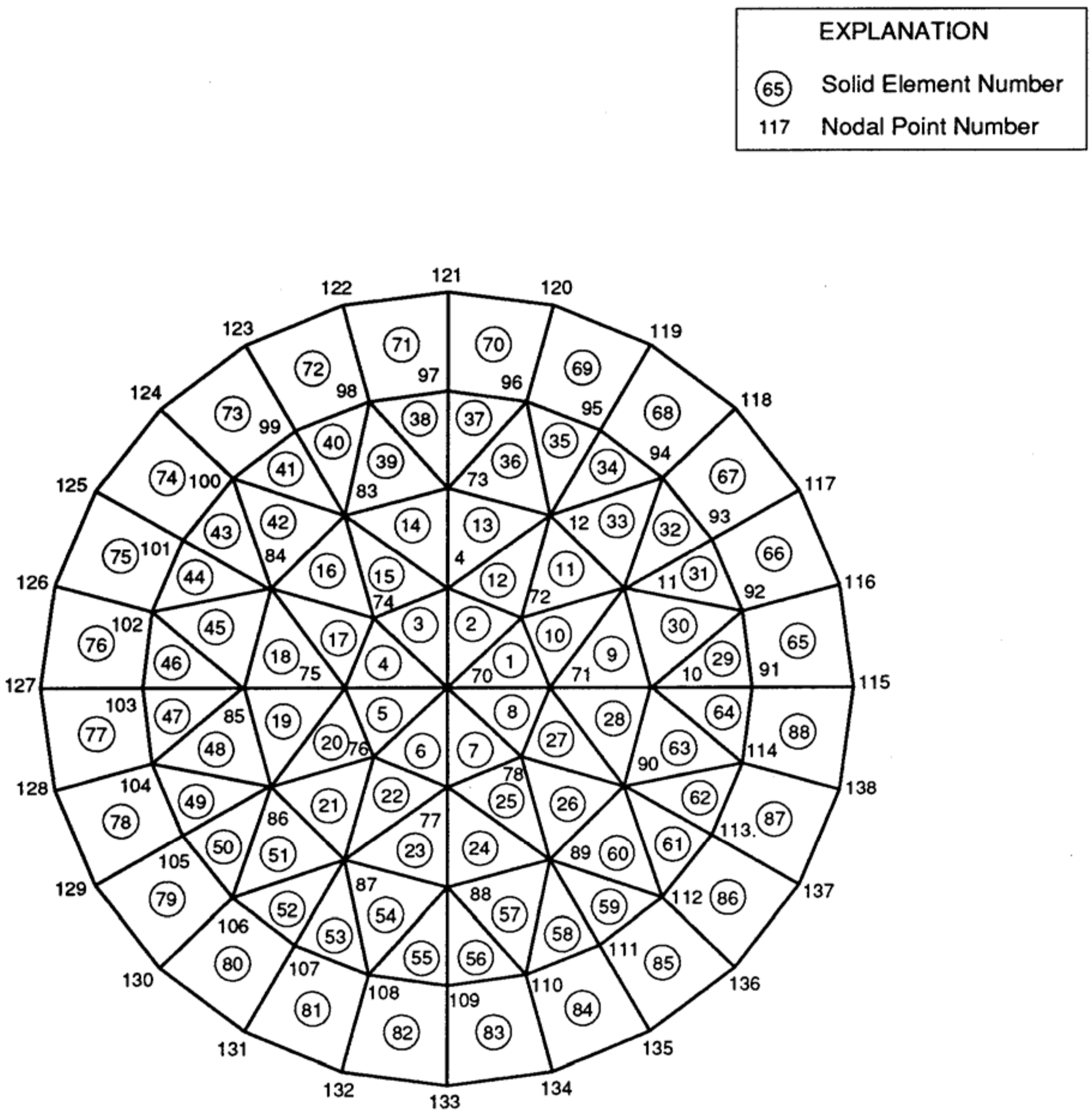


Figure 4.6–9. Nodal Point and Element Numbers for Basemat (Elevation – 0 ft)

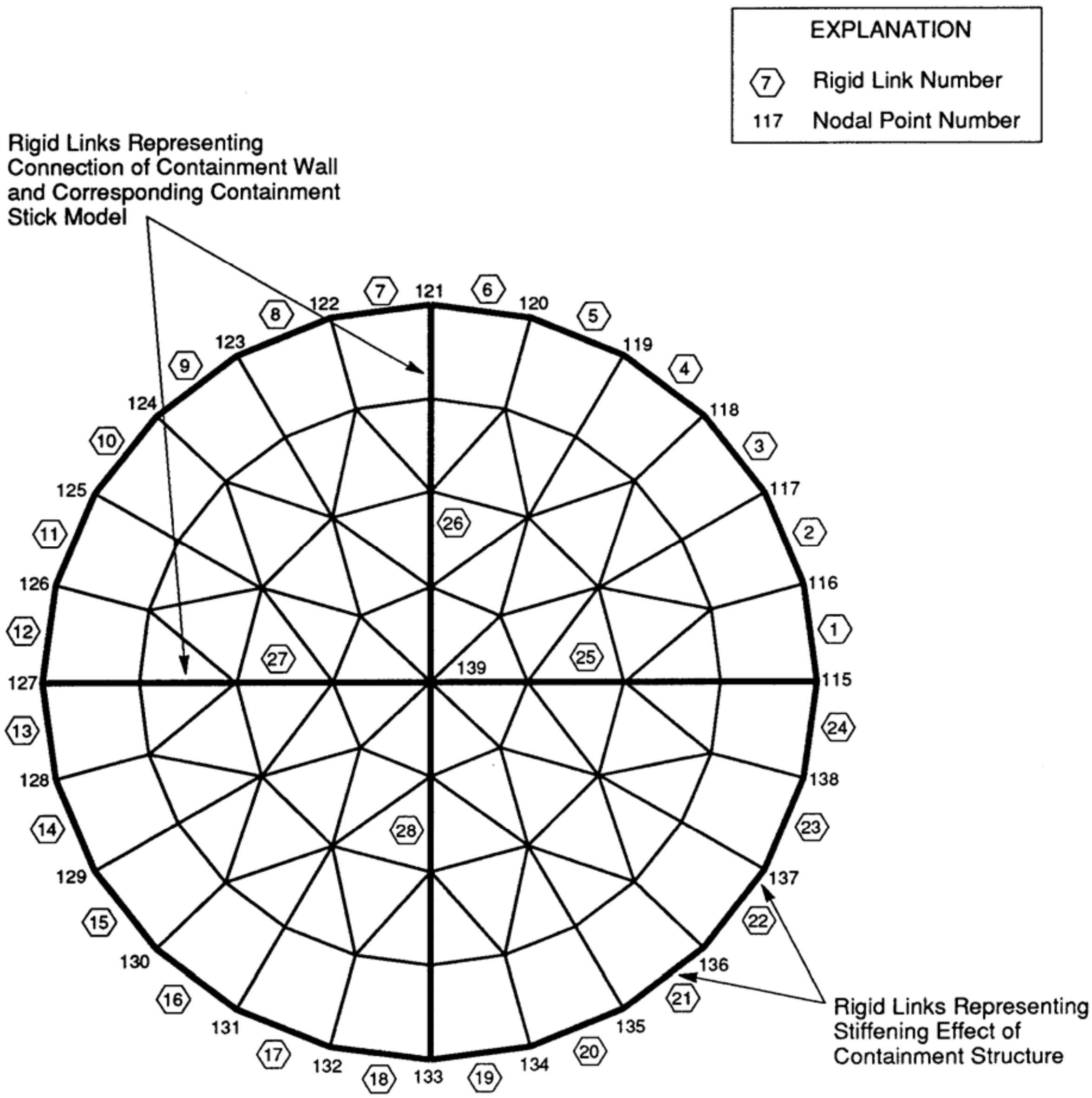


Figure 4.6–10. Rigid Link Model for Containment Structure (Elevation 0 ft)

4.6.1.2.3 Excavated Soil Model

Since this is a surface structure, there is no excavated soil model.

4.6.1.3 Description of Analyses

The analyses performed are described below.

4.6.1.3.1 Foundation Impedance Analysis

The foundation impedance matrices are computed and saved by the computer program SASSI. These matrices are later used in the interaction analysis. To illustrate the accuracy of these matrices, the impedance coefficients of the rigid massless basemat are calculated and compared with available solutions.

4.6.1.3.2 Interaction Analysis

The foundation impedance matrices obtained above are used to perform the complete interaction analysis by the computer program SASSI.

4.6.1.4 Analysis Cases Performed

4.6.1.4.1 Foundation Impedance Analysis

The frequency step and the highest frequency number used in the SASSI analysis are calculated according to item 5 of Section 4.2.2 as follows:

DF = 1/(2048x0.005) = 0.0976 Hz

$NFREQ_{max}$ = 25/.0976 = 256

The highest frequency number selected is 260. The frequencies used in the analysis are listed in Table 4.6-2.

The horizontal and rocking foundation impedance coefficients at the base of the rigid massless basemat were then computed at the above frequencies following the procedures described in item 13 of Section 4.2.2. This required the following SASSI program modules to be executed in order to compute the horizontal/rocking compliance coefficients:

d. SITE (Mode 1)

e. POINT3

f. HOUSE

g. MOTOR

h. ANALYS (Mode 1)

and the following restart runs as described in Section 4.4.3.3 to compute the rocking/coupling compliance coefficients:

j. MOTOR

k. ANALYS (Mode 4)

Table 4.6-2. Frequencies Selected for Impedance and Interaction Analysis by SASSI

Frequency Number	Frequency (Hz)	Circular Frequency (Rad/sec)
10	.9766E + 00	.6136E + 01
30	.2940E + 01	.1841E + 02
50	.4883E + 01	.3068E + 02
60	.5859E + 01	.3682E + 02
80	.7813E + 01	.4909E + 02
100	.9766e + 01	.6136E + 02
120	.1172E + 02	.7363E + 02
140	.1367E + 02	.8590E + 02
160	.1563E + 02	.9817E + 02
180	.1758E + 02	.1104E + 03
210	.2051E + 02	.1289E + 03
230	.2246E + 02	.1411E + 03
260	.2539E + 02	.1595E + 03

The structural model used in this analysis consists of only the basemat with the nodal point numbering shown in Figs. 4.6-8 and 4.6-9. The basemat is modeled by massless elements with very large elastic modulus so that it behaves like a rigid massless circular footing.

The triangularized stiffness of the total system (on Tape 6) obtained from the ANALYS run in item h above is saved to restart the program ANALYS in item k above. The impedance matrix (on Tape 5) is also saved to be used in the interaction analysis described in the next section. The analysis results are discussed in Section 4.6.1.5.2

4.6.1.4.2 Interaction Analysis

The interaction analysis was performed by running for SASSI program modules, as described in Section 4.4.3.1, and by computing the transfer function at the frequencies listed in Table 4.6-2. The modules are:

n. HOUSE

o. SITE (Mode 2)

p. ANALYS (Mode 2)

q. MOTION

Since the impedance matrices on Tape 5 can be recovered from the analysis performed in Section 4.6.1.4.1, only part of the program module ANALYS in item p above was re-executed.

4.6.1.5 Analysis Results

4.6.1.5.1 Input/Output Files

The input/output files corresponding to the impedance and SSI analysis are listed in Table 4.6-3. This table also includes the names that are assigned to the tapes for conveniences of tapein/tapeout activity.

4.6.1.5.2 Foundation Impedance Analysis

The SASSI analysis yields the horizontal and rocking stiffness and damping coefficients of the foundation as shown in Figs. 4.6-11 through 4.6-14. The impedance is computed by inverting the resultant displacements obtained from SASSI analysis. The horizontal impedance coefficients are obtained using Eq. (4.2-1) (see Section 4.2.2) by using the horizontal displacement of Node 1 (see Fig. 4.6-8) from output E2C1HAO (see Table 4.6-3). Similarly, the rocking impedances are computed from the rocking compliance coefficients obtained from the vertical displacement of Node 46 from output E2C1RAO (see Table 4.6-3) divided by the radius of the footing.

The figures also show the corresponding foundation impedances reported by Luco (Ref. 6). The comparison of these results shows good agreement over the frequency range of analysis.

Table 4.6-3. Input/Output Files of Example Problem 1

MODULE USED	HORIZONTAL IMPEDANCE				ROCKING IMPEDANCE				SSI			
	INPUT FILE	OUTPUT FILE	TAPE IN	TAPE OUT	INPUT FILE	OUTPUT FILE	TAPE IN	TAPE OUT	INPUT FILE	OUTPUT FILE	TAPE IN	TAPE OUT
SITE	E1C1SD*	E1C1SO	—	E1C1T2**	—	—	—	—	—	—	—	—
POINT 3	E1C1PD	E1C1PO	E1C1T2	E1C1T3	—	—	—	—	—	—	—	—
HOUSE	E1C1HD	E1CH1HO	—	E1C1T4	—	—	—	—	E1C2HD	E1C2HO	—	E1C2T4
MOTOR	E1C1HMD	E1C1HMO	—	E1C1HT9	E1C1RMD	E1C1RMO	—	E1C1RT9	—		—	—
ANALYS	E1C1HAD	E1C1HAO	E1C1T3 E1C1T4 E1C1HT9	E1C1T5 E1C1T6 E1C1HT8	E1C1RAD	E1C1RAO	E1C1RT9 E1C1T6	E1C1RT8	E1C2AD	E1C2AO	E1C1T1 E1C1T5 E1C2T4	E1C2T6 E1C2T8
MOTION	—	—	—	—	—	—	—	—	E1C2OD	E1C2OO	E1C2T8	E1C2T12

* E1C1SD - Stands for Example 1, Case 1, Site Data

** E1C1T2 - Stands for Example 1, Case 1, Tape 2

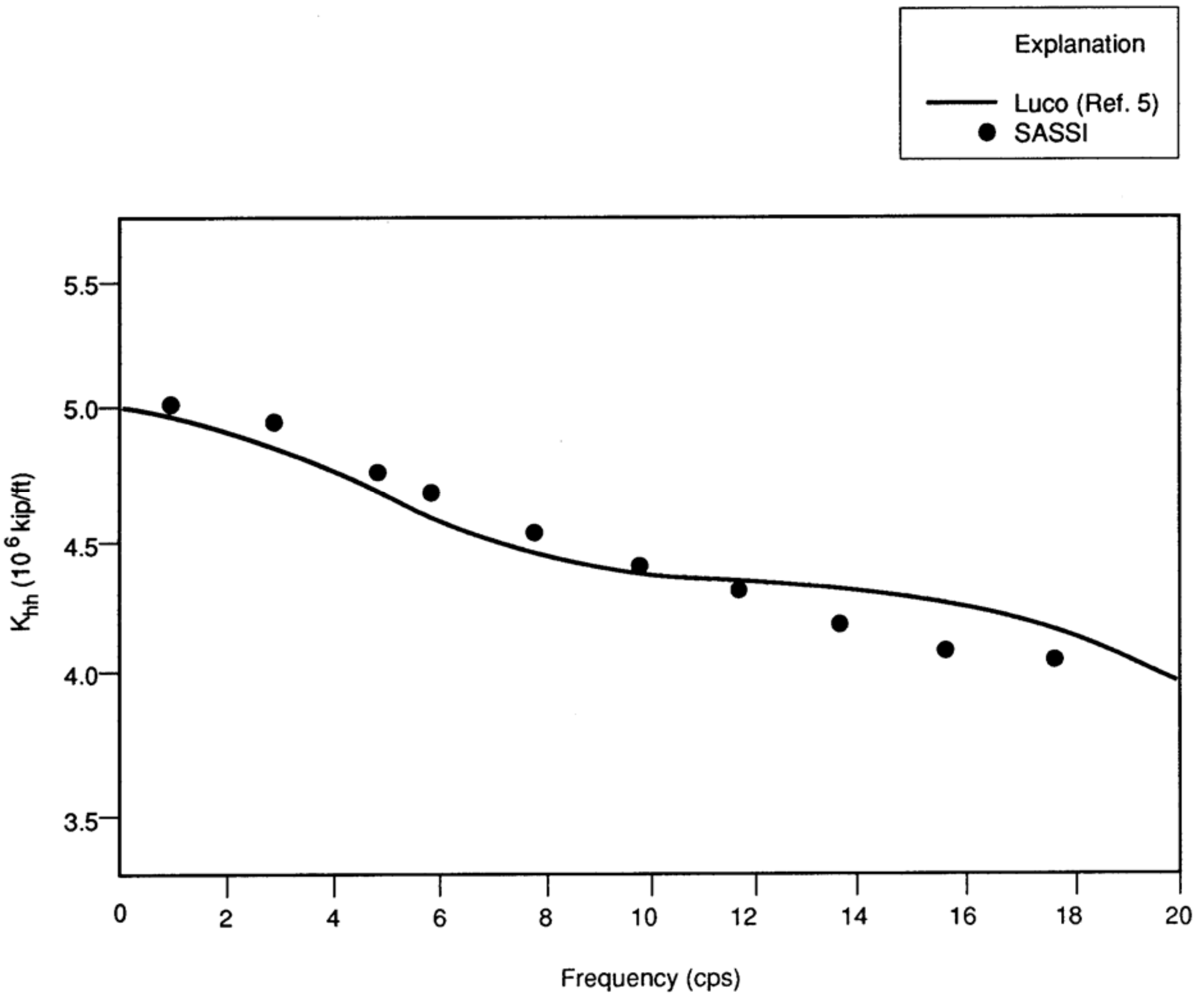


Figure 4.6–11. Horizontal Stiffness Coefficients

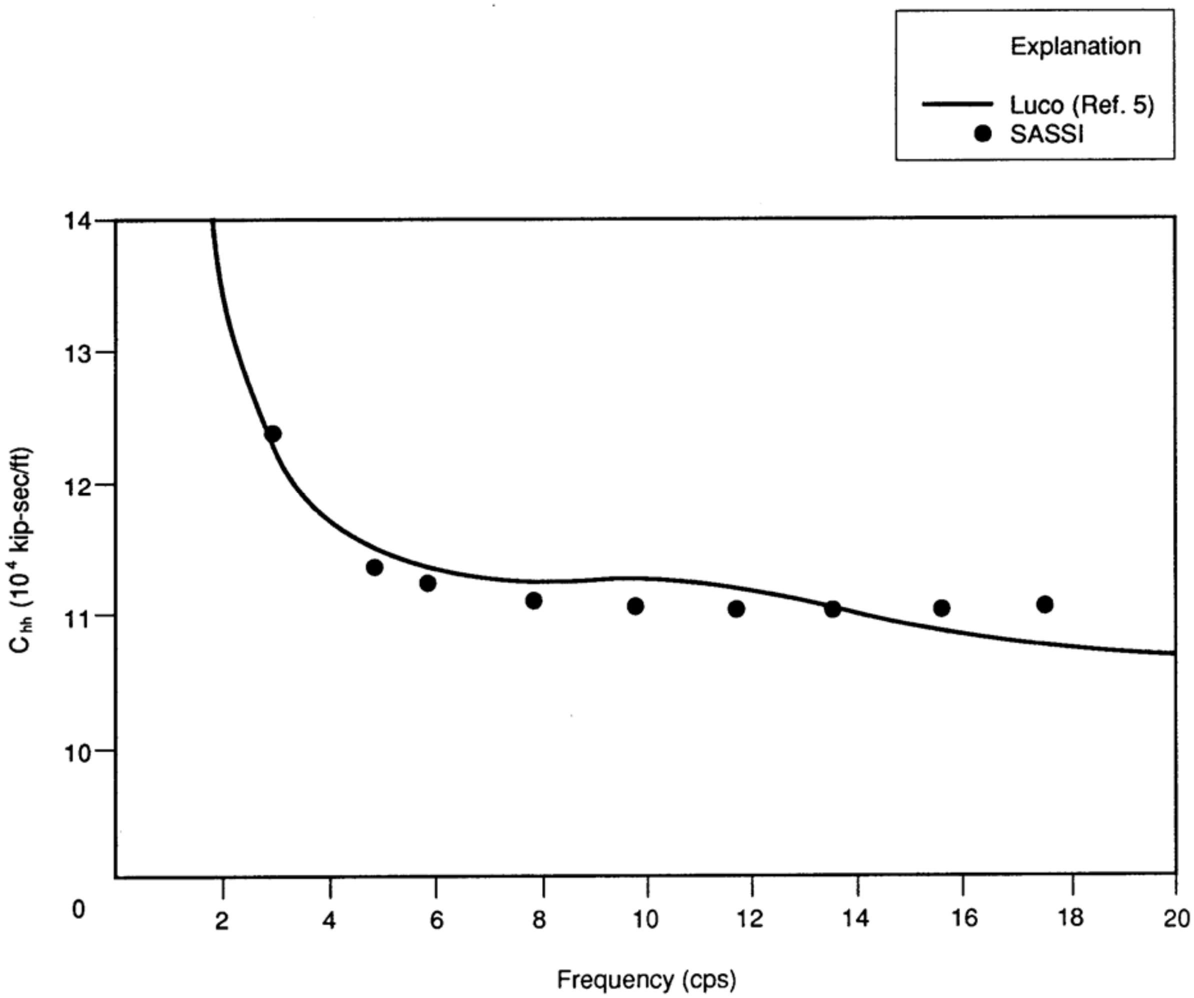


Figure 4.6–12. Horizontal Damping Coefficients

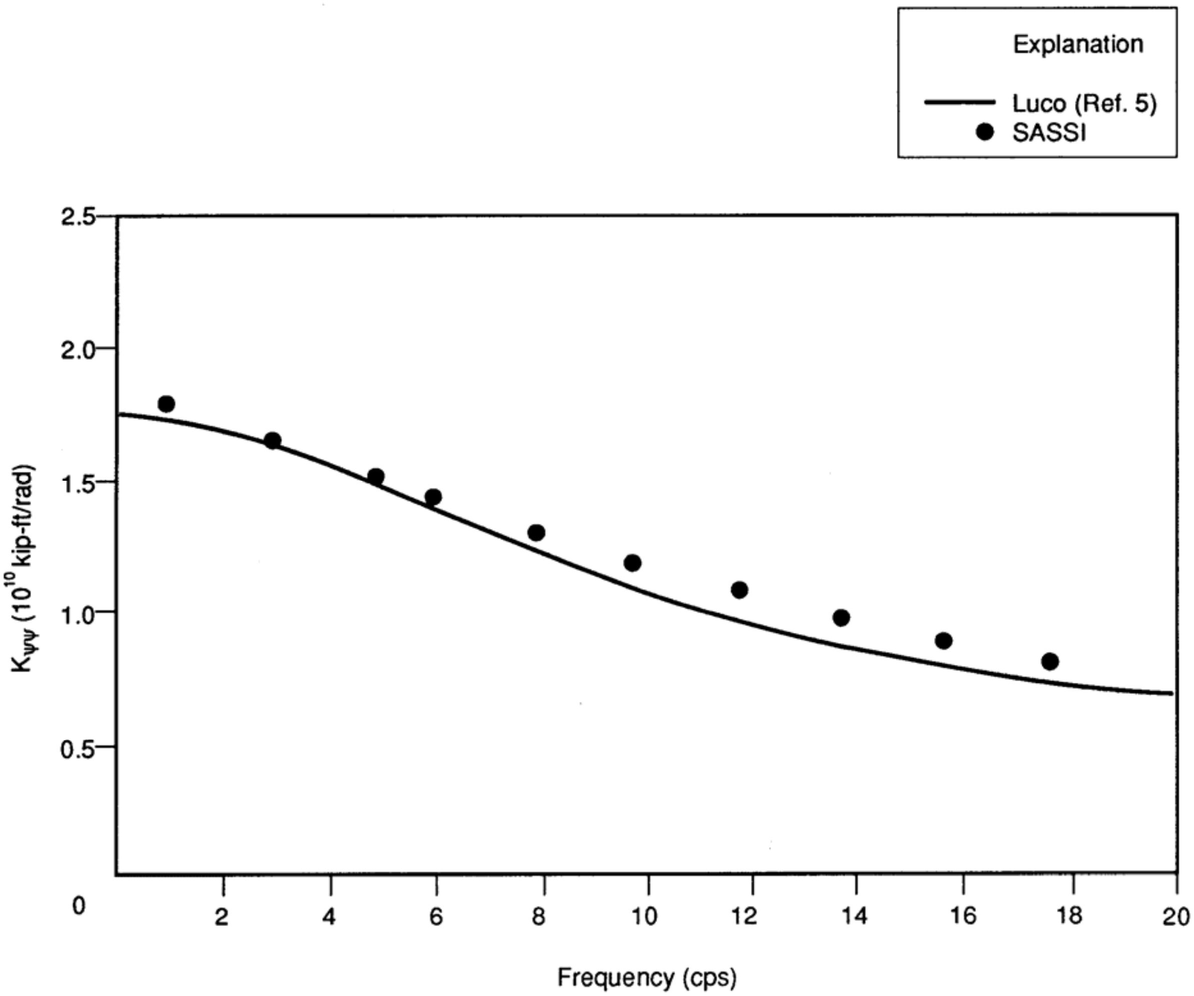


Figure 4.6–13. Rocking Stiffness Coefficients

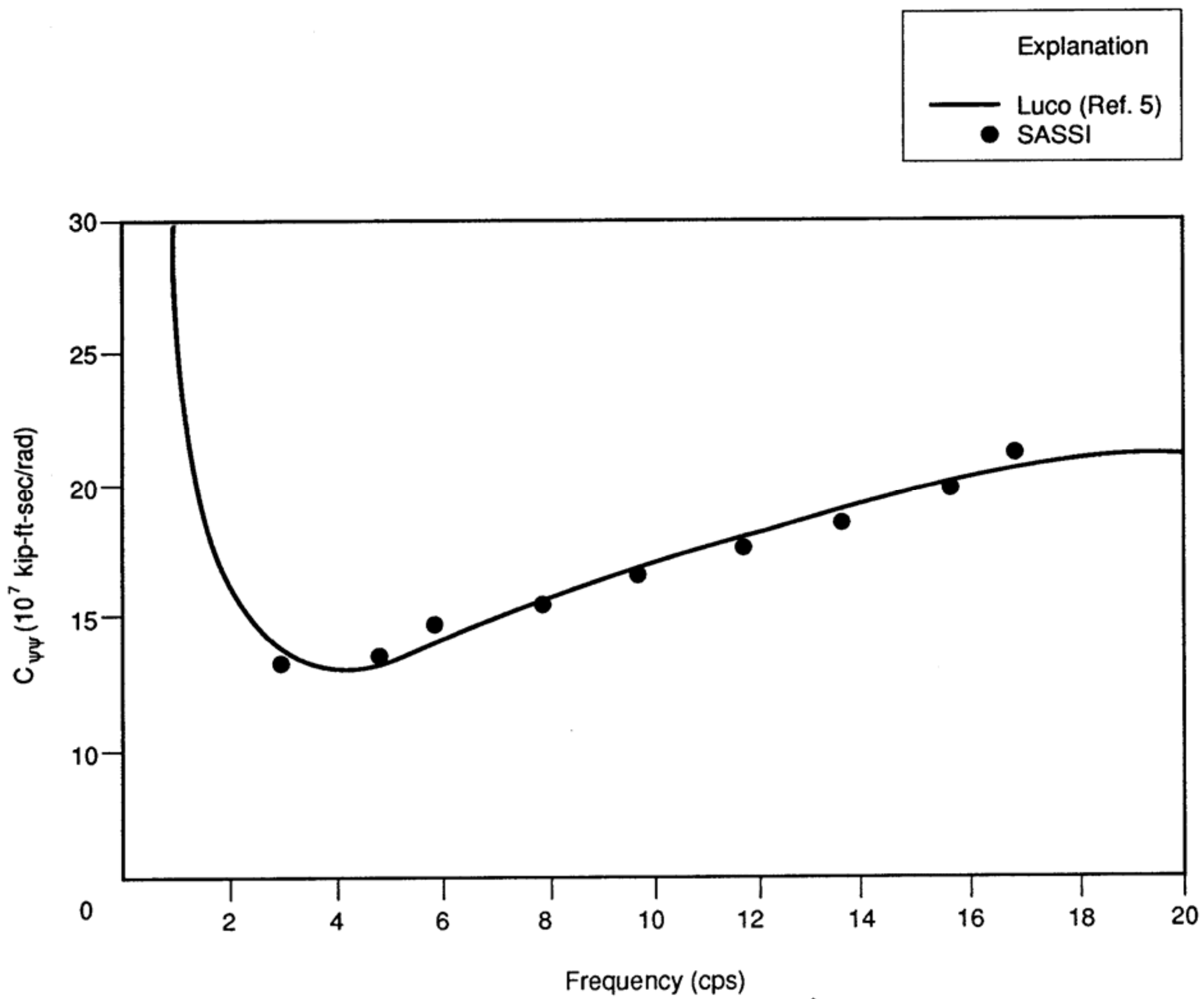


Figure 4.6–14. Rocking Damping Coefficients

4.6.1.5.3 Interaction Analysis

Table 4.6-4 shows the maximum absolute acceleration of the containment building and internal walls obtained from the interaction analysis by SASSI.

Additional results include the 2% acceleration spectrum computed for the horizontal motions at the top of the internal structure and shown in Figs. 4.6-15. The results are compared with the results obtained from computer program FASS (Ref. 7) using the impedance functions obtained from SASSI. The comparison of the results shows good agreement between the two solutions.

Table 4.6-4. Maximum Absolute Accelerations (g's)
Obtained from SASSI Analysis

Node No.	Interaction
141	0.11
142	0.13
143	0.15
144	0.16
145	0.17
146	0.18
147	0.22
148	0.28
149	0.33
150	0.36
151	0.38
152	0.10
153	0.10
154	0.11
155	0.12
156	0.14
157	0.15
158	0.26

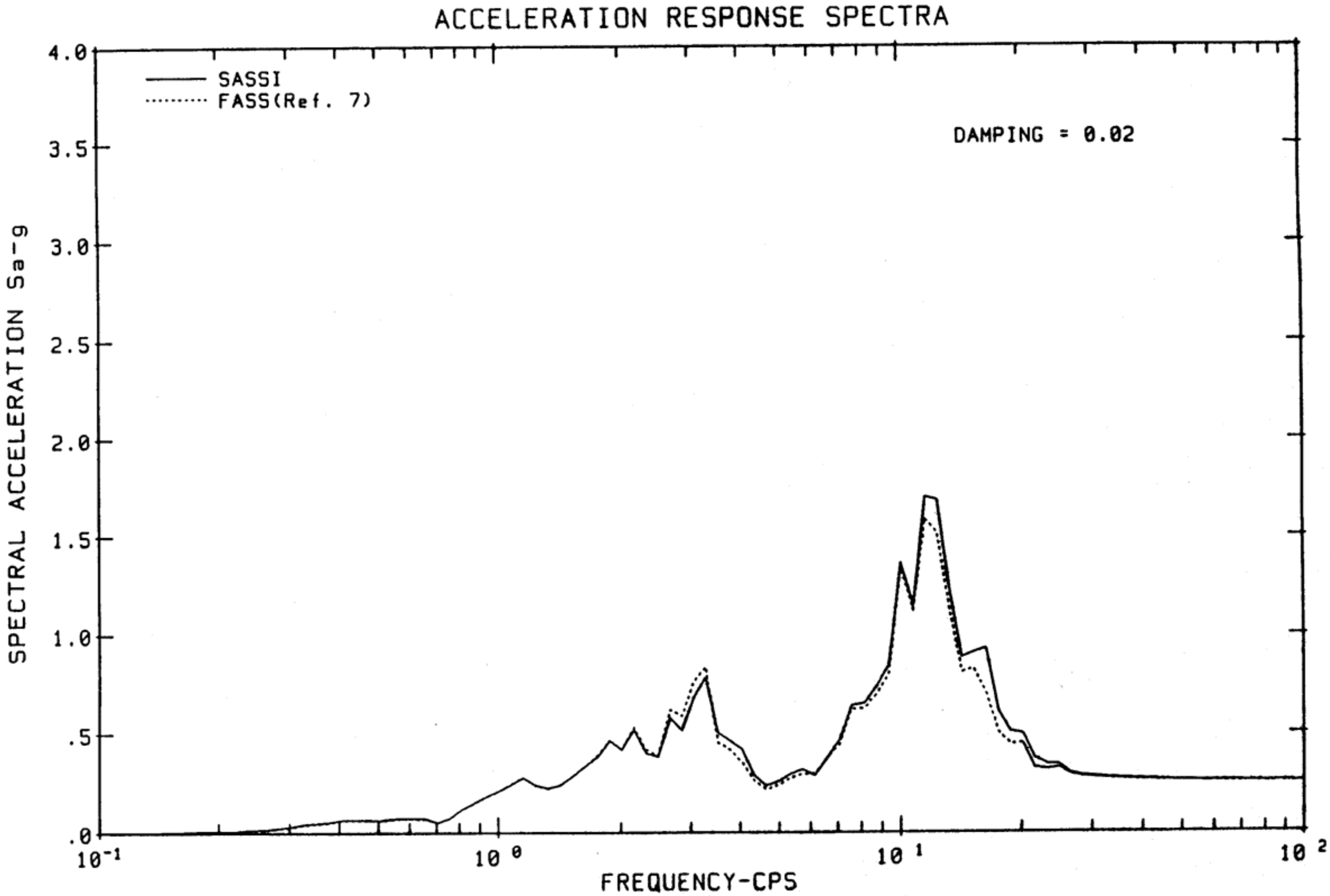


Figure 4.6–15. Absolute Acceleration Response at Top of Internal Structure

4.6.2 Example Problem 2, Lotung SSI Experiment

4.6.2.1 Description

In the late 80's, Electric Power Research Institute (EPRI) with the cooperation from Taiwan Power Company (TPC) and US NRC conducted a large-scale experiment in the earthquake active area of Lotung, Taiwan with the objective of validating SSI analysis methodologies and reducing uncertainties in design. In this experiment, a 1/4scale containment model was constructed and instrumented in order to record the SSI motions of the containment at several locations in the model, and at the free-field at stations adjacent to the model. The recorded data were subsequently used in a series of round-robin blind-prediction studies using the currently available SSI methodologies. The results of these studies were published in Ref. 8.

In this problem, seismic SSI responses of the containment model during one of the strong seismic excitations are computed using SASSI, and the results are compared with the recorded motions in the model.

The containment model is approximately a 1/4-scale model of a prototype containment in configuration. The model is a cylinder-shaped, reinforced concrete shell structure with a flat roof slab and a flat basemat (see Fig. 4.6-16). The structure is 50 ft high from which 15 ft is below the ground. The cylindrical shell has a constant wall thickness of 1 ft and an outside diameter of 34.5 ft. The roof slab is 4 ft thick and the basemat is 3 ft thick. Inside the containment, a steel shell structure simulating a prototype steam generator is supported on the basemat at about 7.1 ft away from the center of the basemat and 22.5 degrees south of the west axis of the containment (see Fig. 4.6-17). A 4-leg, 4-bend, 6-inch diameter, schedule 40 piping is connected from the top of the model steam generator to the containment shell at 1 ft below the grade (see Figs. 4.6-16 and 4.6-17). The model steam generator is supported by a steel frame which also serves as the lateral support to the steam generator. The four H-shape columns of the steel frame are welded to a 13 mm thick baseplate which is, in turn, anchored by four 22 mm diameter 650 mm long anchor bolts to the top of the basemat at about 5 ft above the base. The entire model steam generator with its supporting frame and piping is referred to as the internal structure.

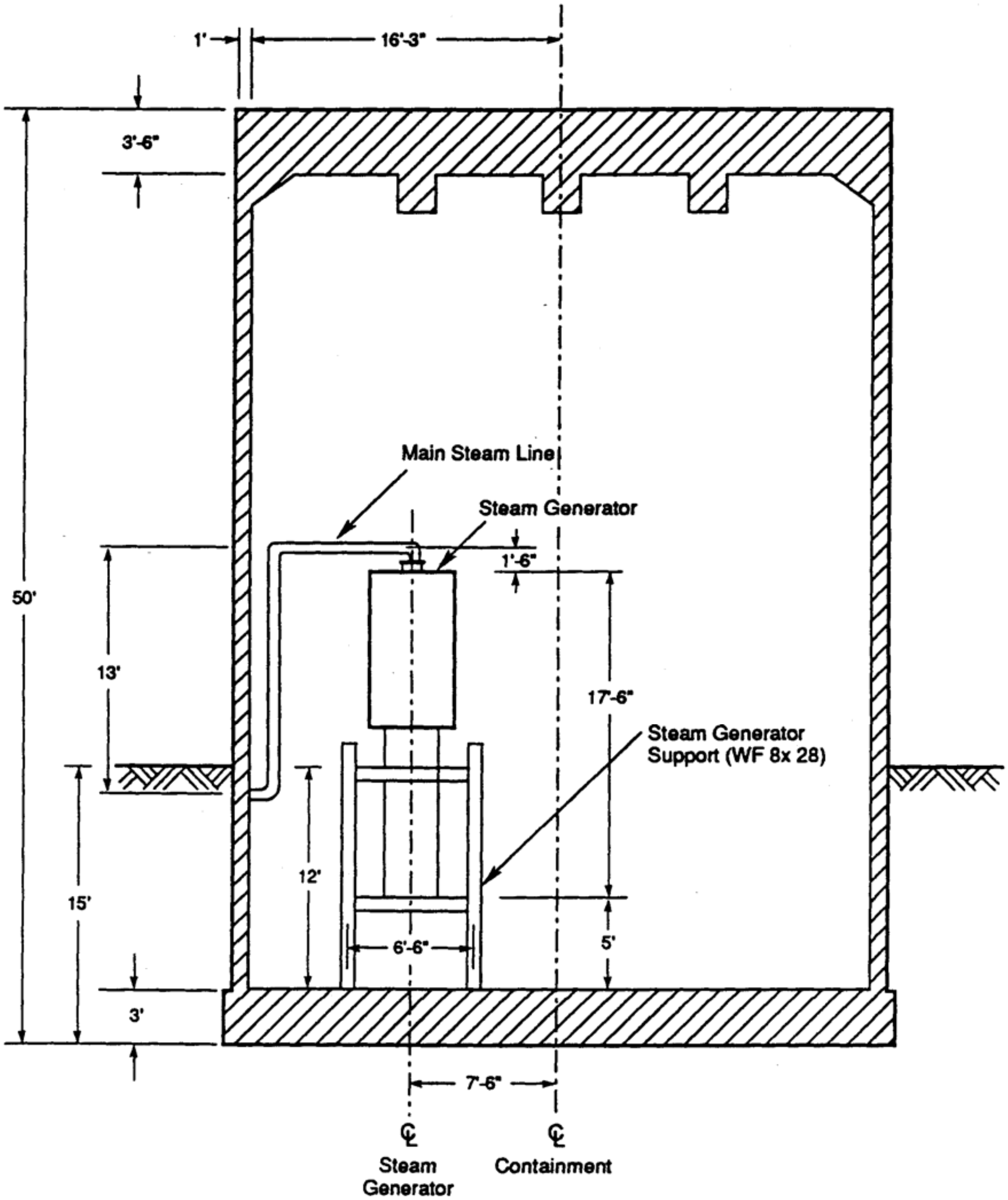


Figure 4.6–16. Vertical Cross-Section View of the 1/4-Scale Containment Model

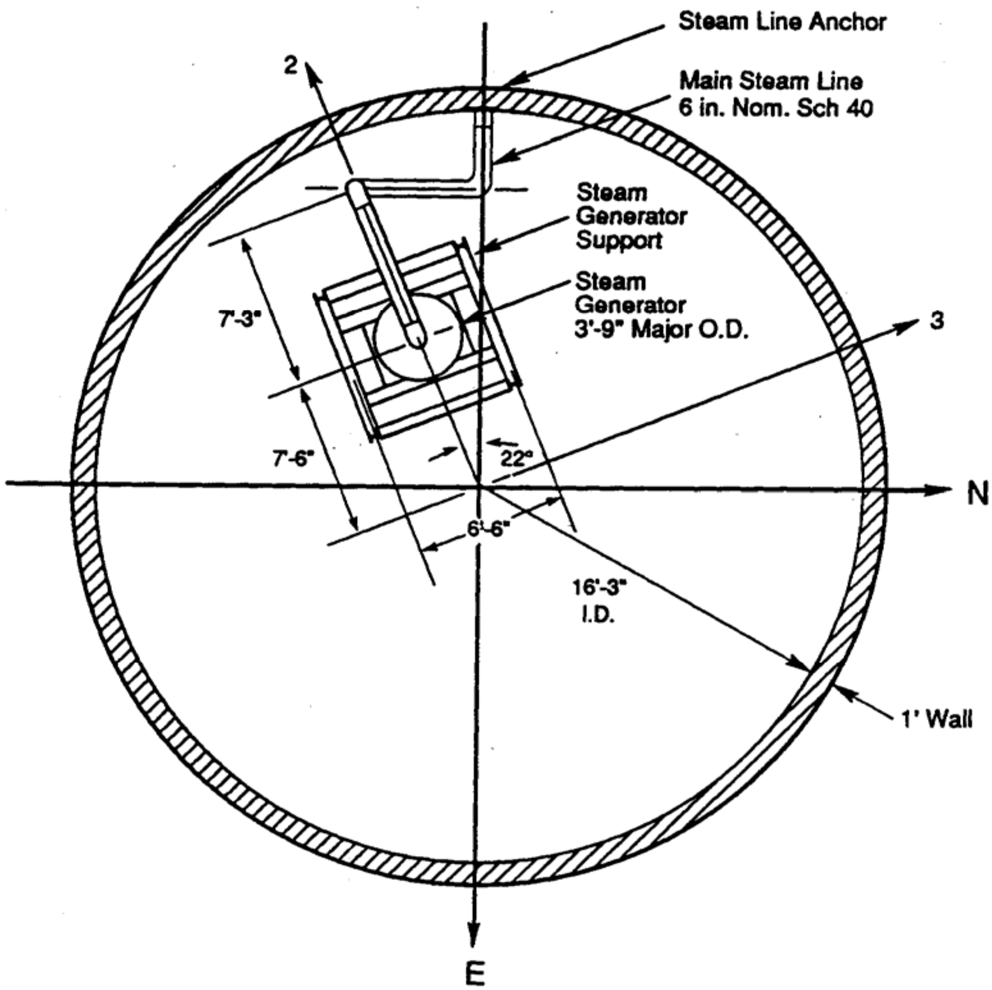


Figure 4.6–17. Horizontal Cross-Section of the 1/4-Scale Containment Model

The site of the 1/4-scale containment model is located in Lotung, a rural township in the northeast corner of Taiwan, where the strong motion array Taiwan No. 1 (SMART-1 array) is located. The site is in a relatively flat ground with a relatively soft surface soil layer of thickness about 200 ft, overlaying deep alluvium strata. The bedrock is approximately 1,500 ft below the ground surface.

SSI analysis results of this experiment is used to validate seismic SSI analysis capability of SASSI against recorded motions for embedded structures. Since the containment is embedded with a significant embedment ratio (15' over 50'), both the direct and the substructure methods are used to show the accuracy of the methods and the efficiency of the subtraction method.

4.6.2.2 Modeling

4.6.2.2.1 Soil Model

The SASSI site model is constructed from the average strain-compatible soil properties obtained from the free-field SHAKE deconvolution analysis using the horizontal components of the recorded motion of the earthquake event LSST07 of May 20, 1986 (see Ref. 8). In the free-field analysis reported in Ref. 8; following the common industry practice, the initial low strain soil properties obtained from the field geophysical survey along with the strain-dependent shear modulus and damping ratios obtained from dynamic triaxial and resonant column test results were used in the SHAKE analysis. The SASSI site model developed from the free-field SHAKE analysis of Ref. 8 is shown in Fig. 4.6-18. During the Lotung experiment, several earthquakes with low to moderately high intensities were recorded at the site and in the containment model. However, for the purpose of this validation problem, SASSI SSI analysis has been performed for one case of horizontal East-West shaking (see Fig. 4.6-17 for the EW direction) during one of the more severe shaking events. The input motion is the East-West component of the recorded motion in the free-field at the grade level (transverse component of LSST07 event at Station FA1-5 in Ref. 8). The acceleration time history of input motion and its response spectrum at five percent damping are shown in Figures 4.6-19 and 4.6-20, respectively. The recorded motion has maximum acceleration of 0.157 g.

Z

Elevation (ft)	Layer No.	Vs (ft/sec)	Vp (ft/sec)	γ (kcf)
0				
	1	253	1290	.134
-4				
	2	253	1290	.134
-8				
	3	253	1290	.134
-12				
	4	253	1290	.134
-15				
	5	253	1290	.134
-19.6				
	6	253	1290	.134
-24.6				
	7	301	1534	.123
-30.0				
	8	301	1534	.123
-36.0				
	9	301	1534	.123
-41.0				
	10	391	1994	.128
-46.0				
	11	391	1994	.128
-51.0				
	12	391	1994	.128
-56.0				
	13	391	1994	.128
-65.0				
	14	391	1994	.128
-74.0				
	15	391	1994	.128
-82.0				
	16	391	1994	.128
-90.0				
	17	391	1994	.128
-98.0				
	18	391	1994	.128
-106.				
	19	529	2697	.124
-114.				
		536	2733	.117
	Halfspace			

Y

Material damping of 17% was used for all layers

Figure 4.6–18. SASSI Soil Profile Model

4.6.2.2.2 Structural Model

SASSI model was constructed using one plane of symmetry as shown in Fig. 4.6-21. Using one plane of symmetry implies the existence of another internal structure with respect to the plane of symmetry. This assumption, however, was considered satisfactory since the internal structure weight is only one percent of the total structure weight.

In the SASSI model, the containment shell structure above the grade was modeled using 3-D lump mass stick model. The model and its properties are shown in Fig. 4.6-22 and Table 4.6-5. The shell structure below the ground surface was modeled using plate elements. The basemat and excavated soil volume were modeled using brick elements (see Fig. 4.6-21). The basement model discretized at Elevation -15, -12, -8, -4 and 0 ft are shown in Figs. 4.6-23 through 4-6-27. The shell stick model above the grade is connected to the basement model by a series of rigid links as shown in Fig. 4.6-28. A simplified 4-lump-mass stick model for the steam generator and its supporting frame was developed from the detail finite element model of the system (Ref. 8). The simplified steam generator with its support stick model and their properties are shown in Fig. 4.6-29 and Table 4.6-6.

The steam generator was connected to the top of the basemat at Elevation -12 ft by a series of rigid links as shown in Fig. 4.6-24b. The piping was modeled using the lump mass beam model as shown in Fig. 4.6.30. The piping is connected to the top of the steam generator at one end and to the side wall of the shell structure below ground surface (see Fig. 4.6-21). The properties of the piping model is shown in Table 4.6-7. The fixed base frequencies of the containment shell, steam generator and piping system are shown in Table 4.6-8 (see Ref. 8).

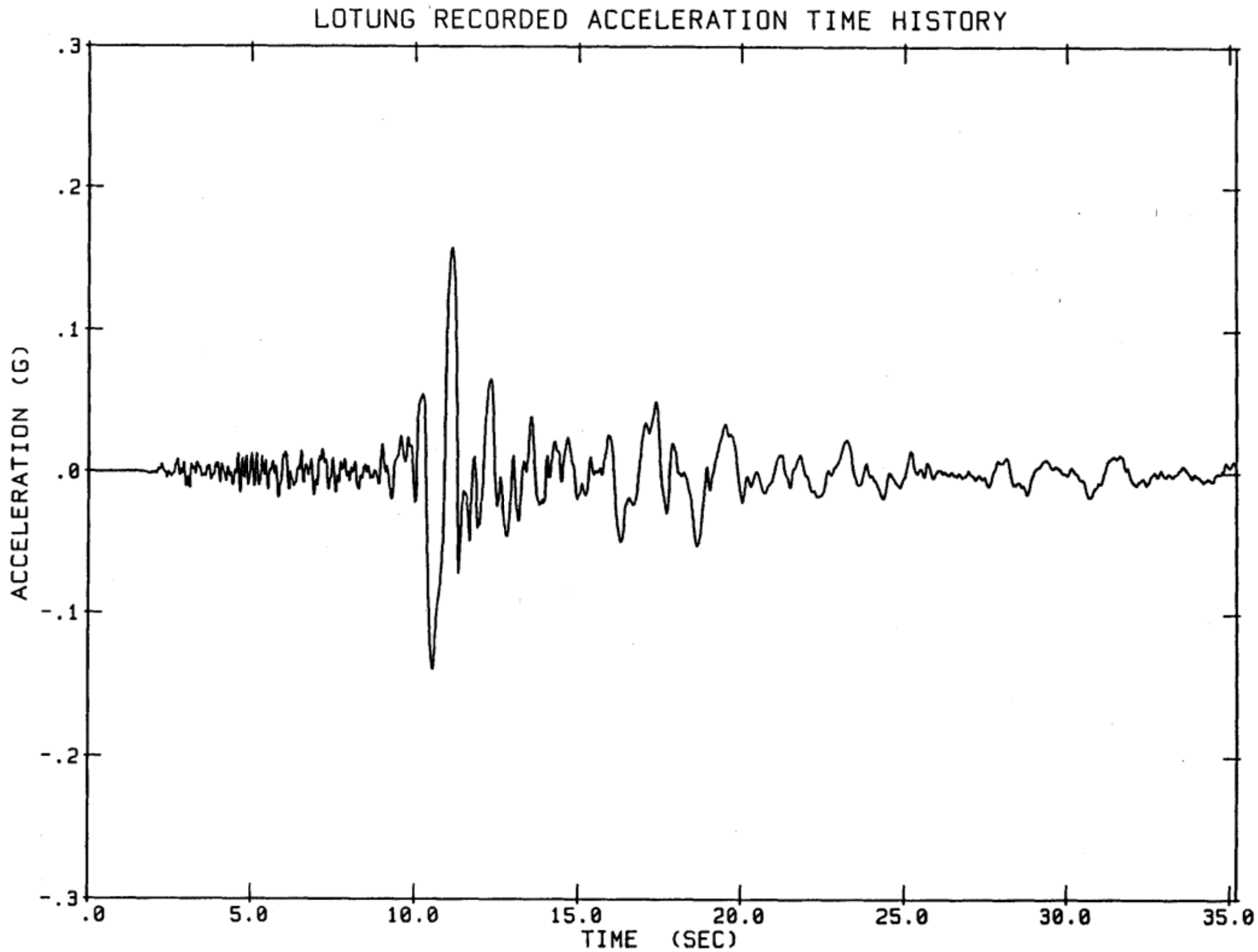


Figure 4.6–19. Acceleration Time History of Input Model

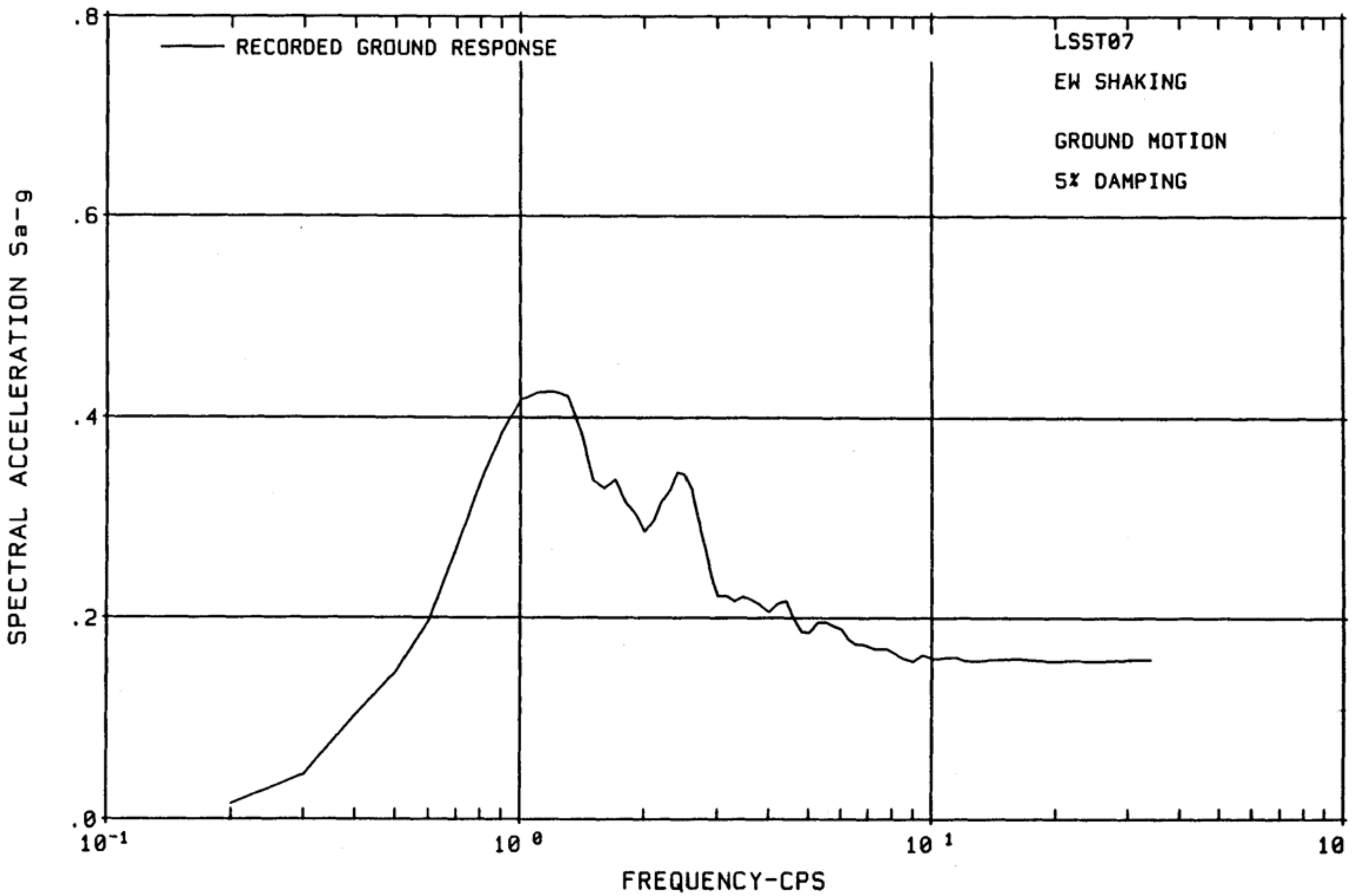


Figure 4.6–20. Acceleration Response Spectrum of Input Model

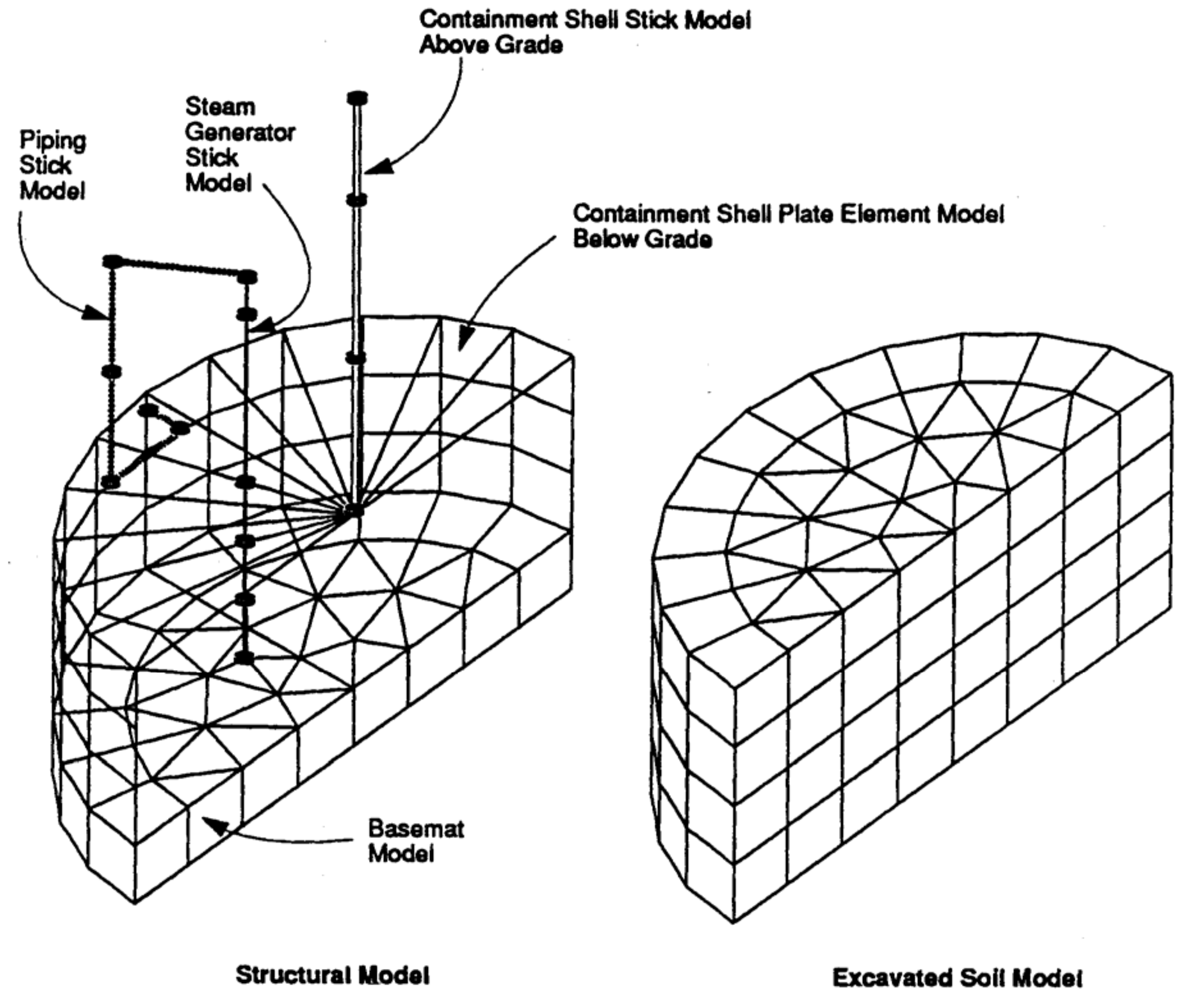


Figure 4.6–21. Configuration of SASSI Finite Element Model

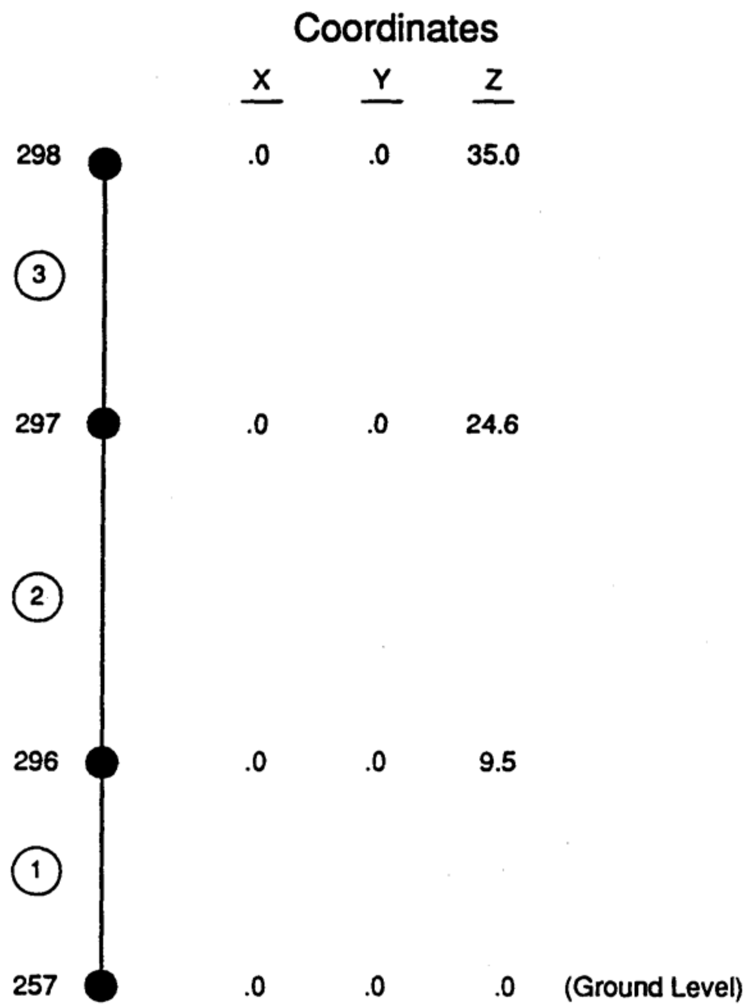


Figure 4.6–22. Containment Stick Model Above Grade

Table 4.6-5. Properties of the Structural Models of the Steam Generator

(Steel Modulus E = 2.09 x 10^6 ksf, G = 0.80 x 10^6 ksf, β = 2%)

Joint Properties			Member Properties					
Mass No.	M_j x g (kips)	I_j x g (kip-ft^2)	Location Between Joint Noh	Area (ft^2)	Shear Area (ft^2)*		Moment of Inertia x (ft^4)*	
					A2	A3	I2	I3
299	—							
300	0.0673	—	299 to 300	0.0341	.00495	.00312	0.1225	0.1042
301	0.0519	—	300 to 301	0.572	.30500	.30500	0.6120	0.6120
313	0.0352	—	301 to 313	0.572	.30500	.30500	0.6120	0.6120
302	0.0287	—	313 to 302	0.832	.44200	.44200	1.3996	1.3990
			300 to 301	0.0341	.00495	.00312	0.1225	0.1042

Note: See Fig. 4.6-17 for the orientation of the local 2 and 3 axes.

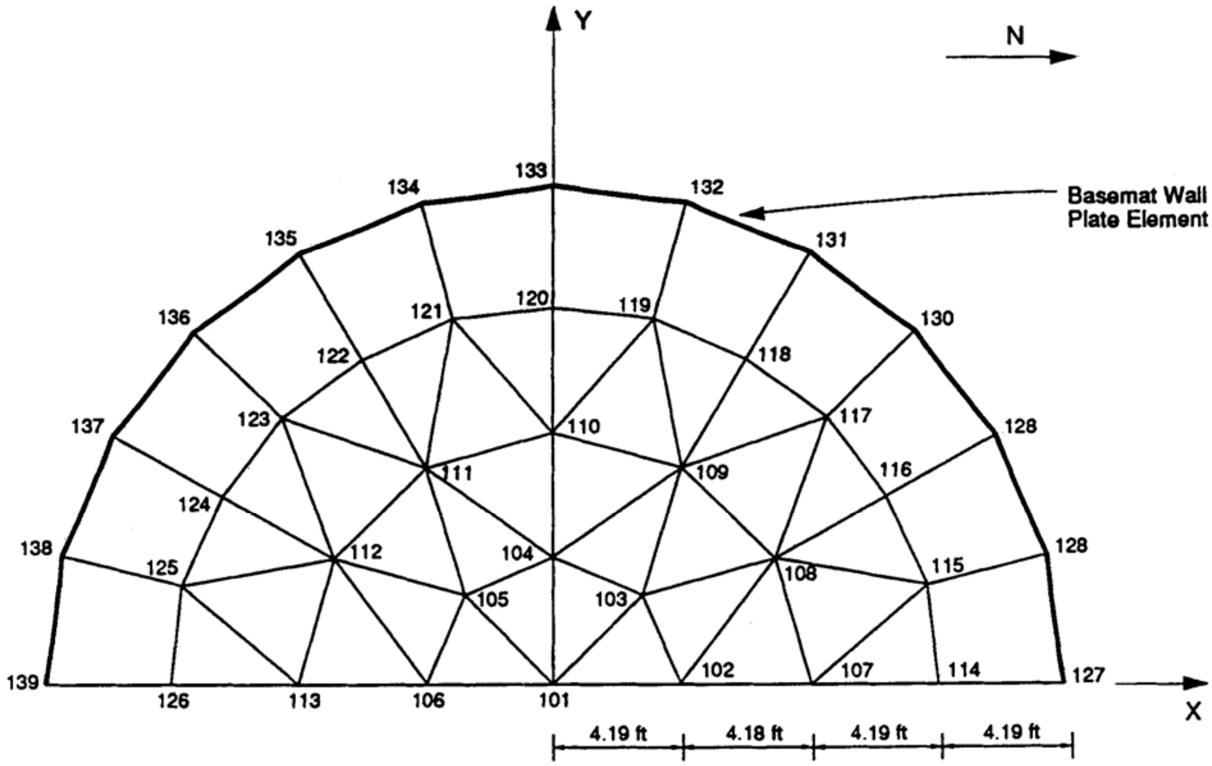
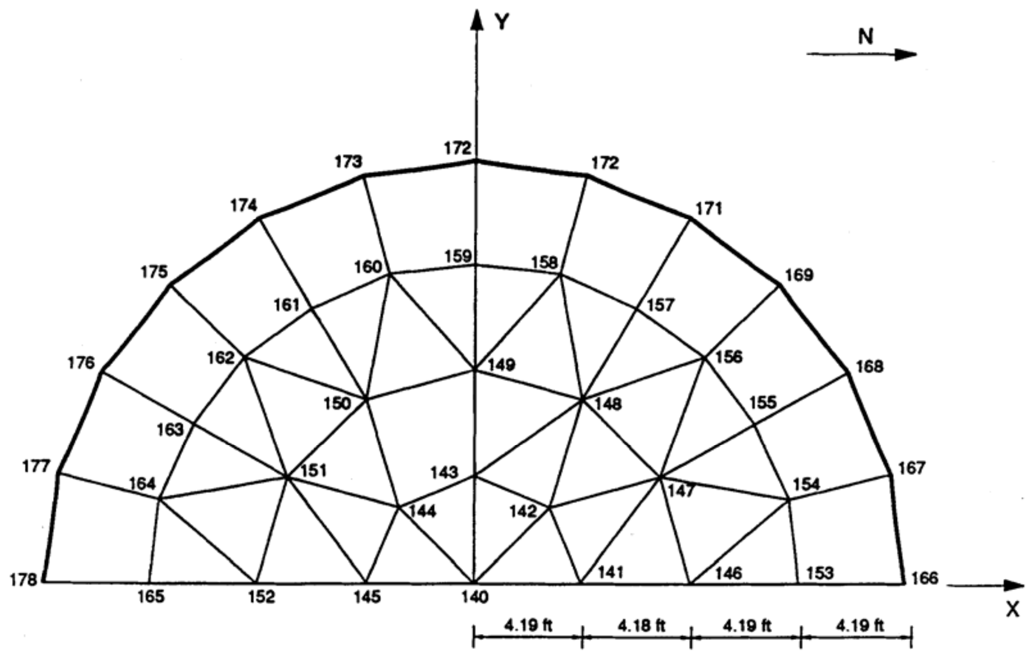
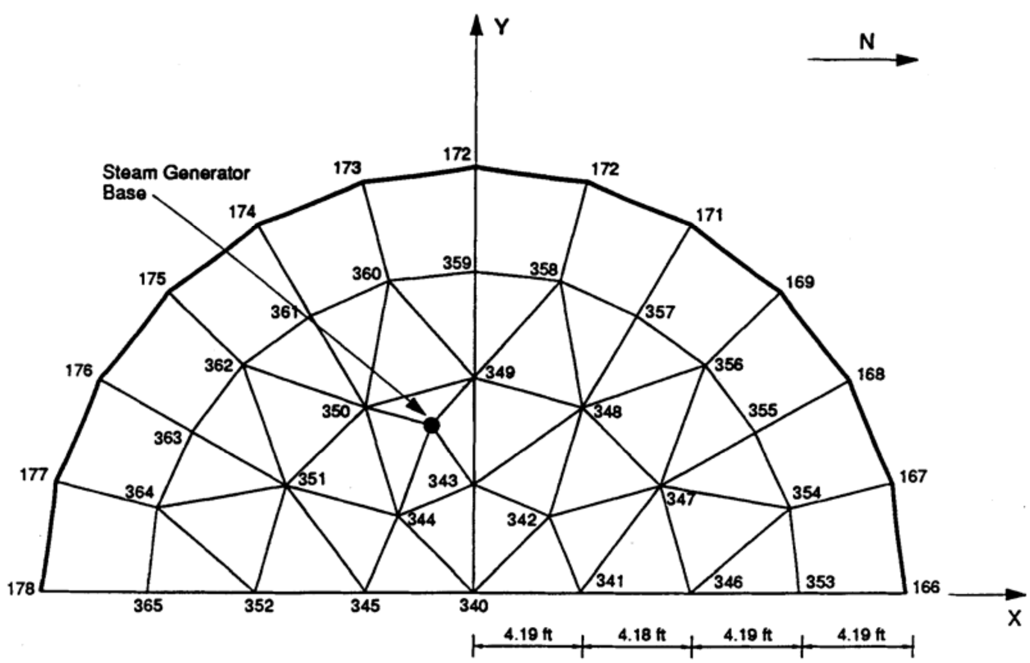


Figure 4.6–23. Foundation Model at Elevation -15 ft



(a)



(b)

Figure 4.6–24. Foundation Model at Elevation -12 ft

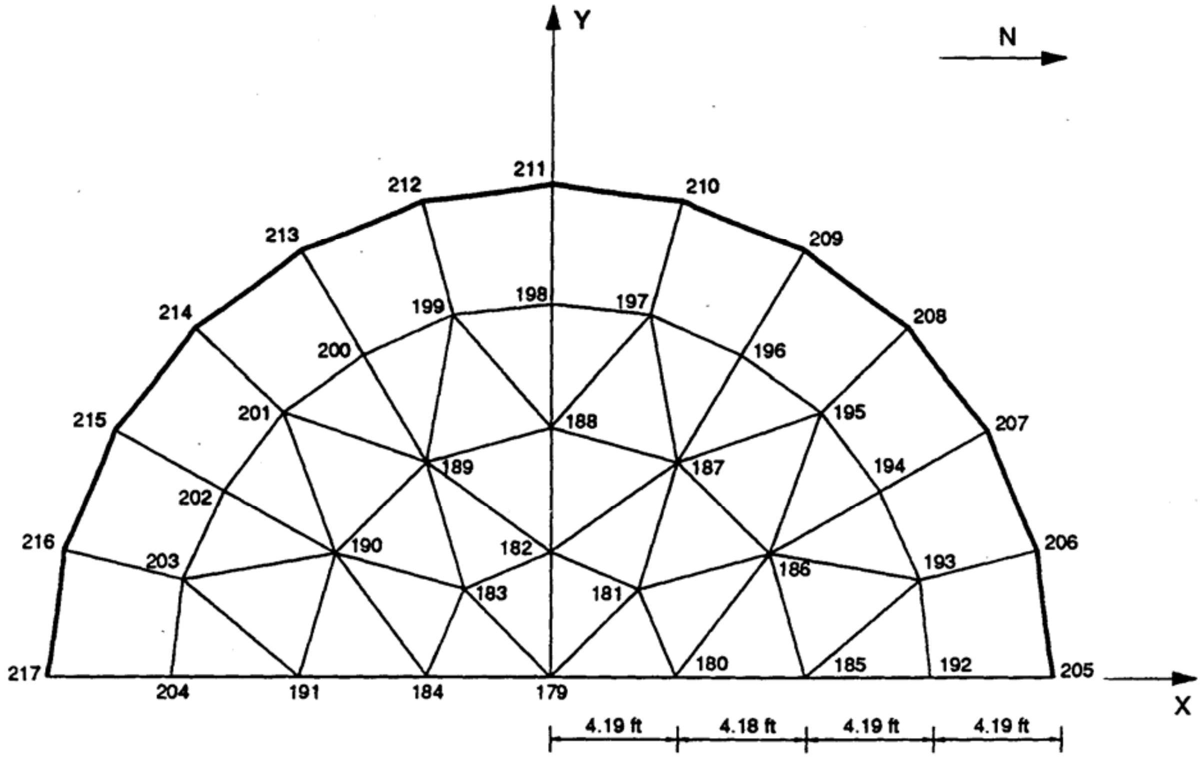


Figure 4.6–25. Foundation Model at Elevation -8 ft

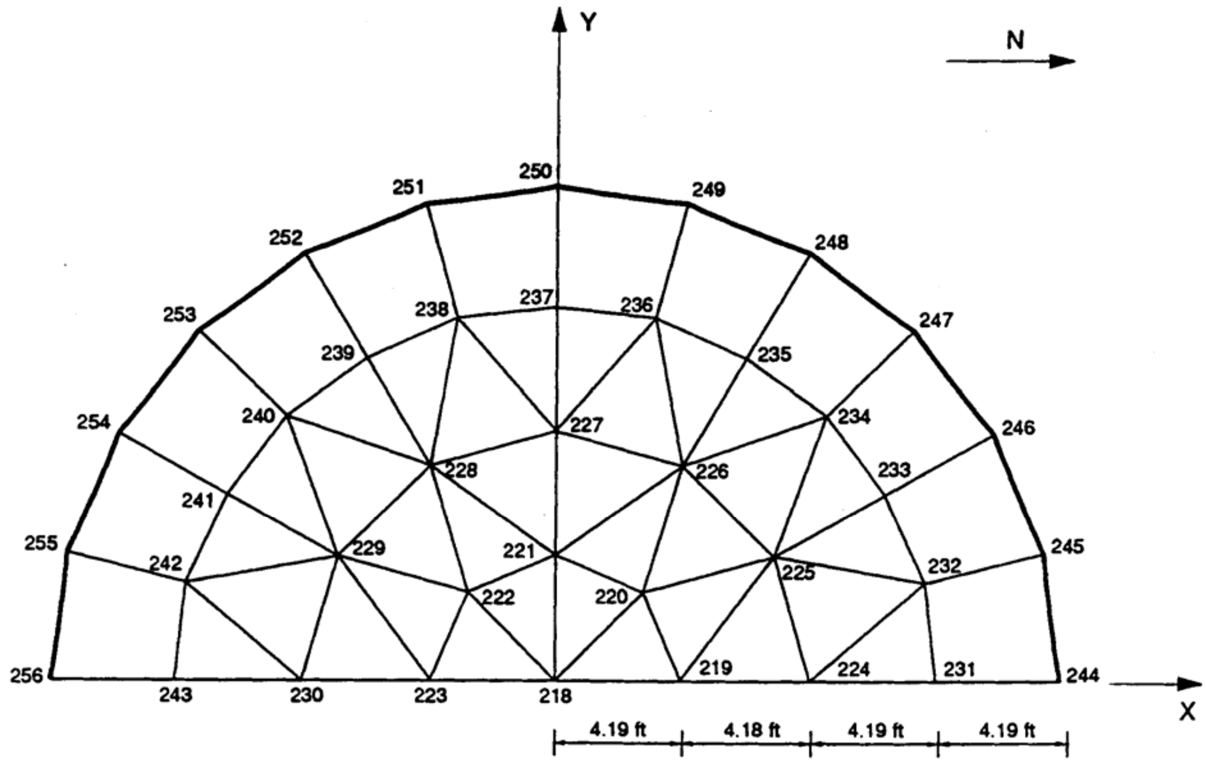


Figure 4.6–26. Foundation Model at Elevation -4 ft

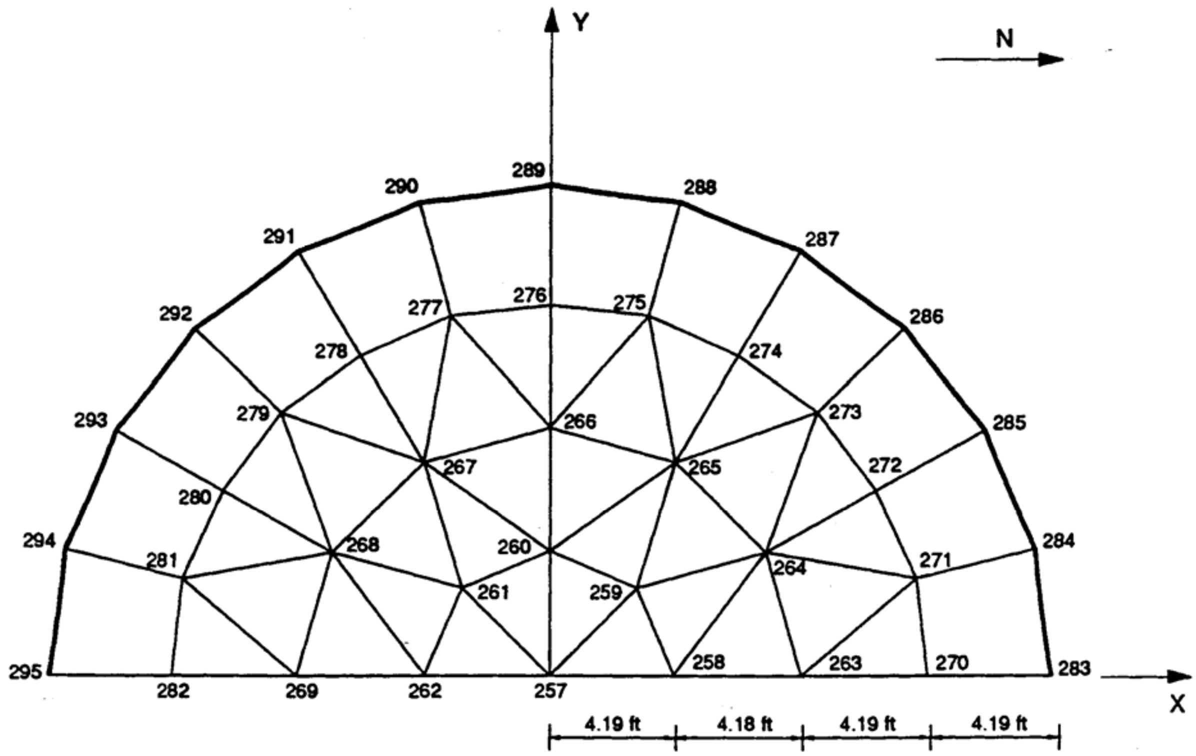


Figure 4.6–27. Foundation Model at Elevation 0 ft

4.6.2.2.3 SSI Frequencies and Wave Field

The SASSI analysis for this problem is performed for frequencies shown in Table 4.6-9. The frequency step (DF) used in the analysis is computed from

$$DF = \frac{1}{NFFTXDT}$$

where NFFT is 4096 and time step, DT, is 0.01 second. Frequencies of analysis, f, are the product of the frequency step and the selected frequency number (NF).

The control motion is specified to be vertically propagating SV-wave with a control point defined at the grade level where control motion was recorded in the free-field.

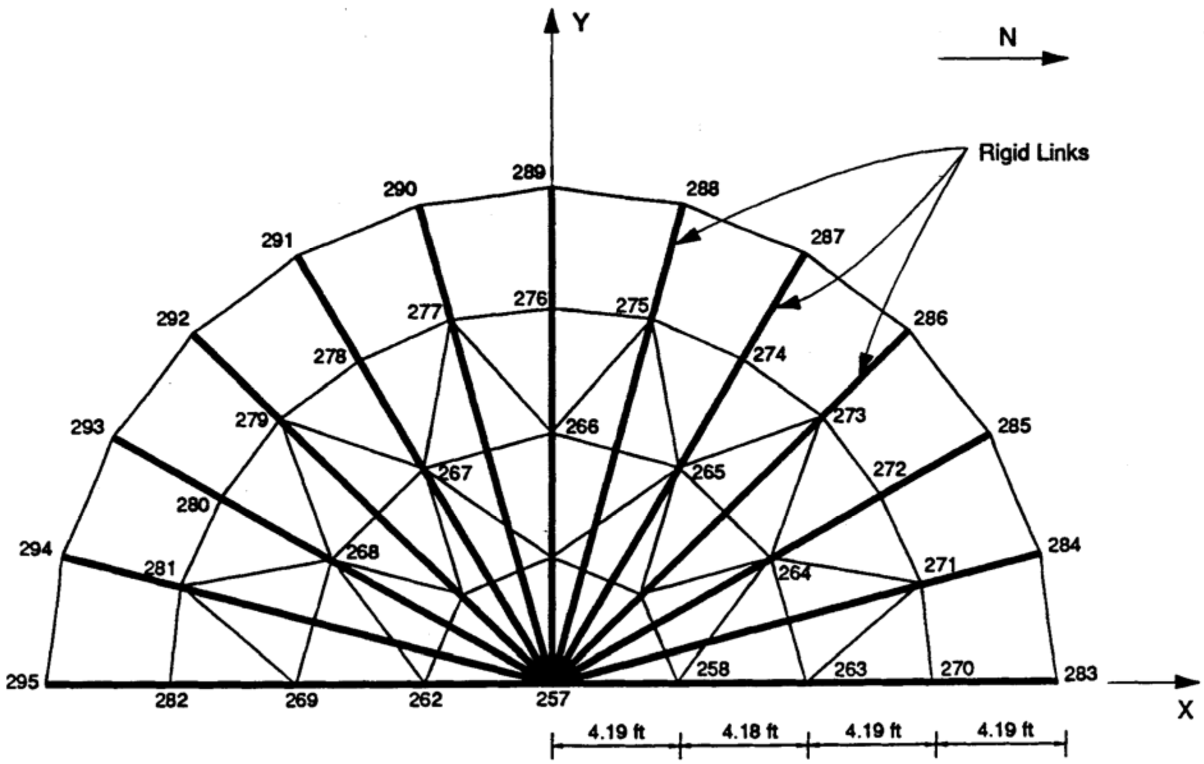


Figure 4.6–28. Rigid Links to Connect Shell Stick Model to Foundation Basement Model at Elevation 0 ft

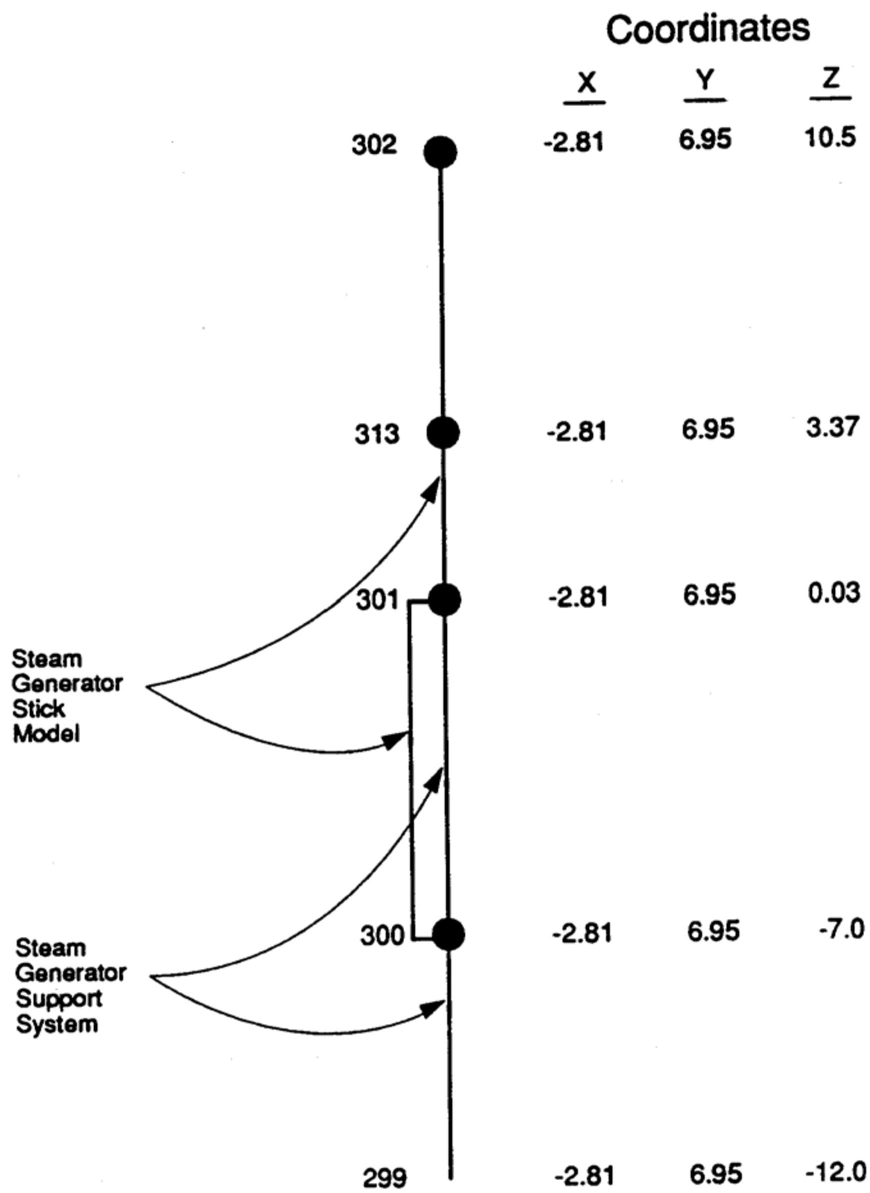


Figure 4.6–29. Steam Generator and Support System Stick Models

Table 4.6-6. Properties of the Structural Models of the Containment Shell Above Grade

(Concrete Modulus E = 5.18 x 10^5 ksf, G = 2.16 x 10^5 ksf, β = 4%)

Joint Properties			Member Properties			
Mass No.	M_j x g (kips)	I_j x g (kip-ft^2)	Location Between Joint No.	Area (ft^2)	Shear Area (ft^2)	Moment of Inertia (ft^4)
257	1.192	1952				
296	3.156	—	257 to 296	52.62	27.7	14780
297	3.266	—	29,6 to 297	52.62	27.7	14780
298	10.296	—	297 to 298	52.62	27.7	14780

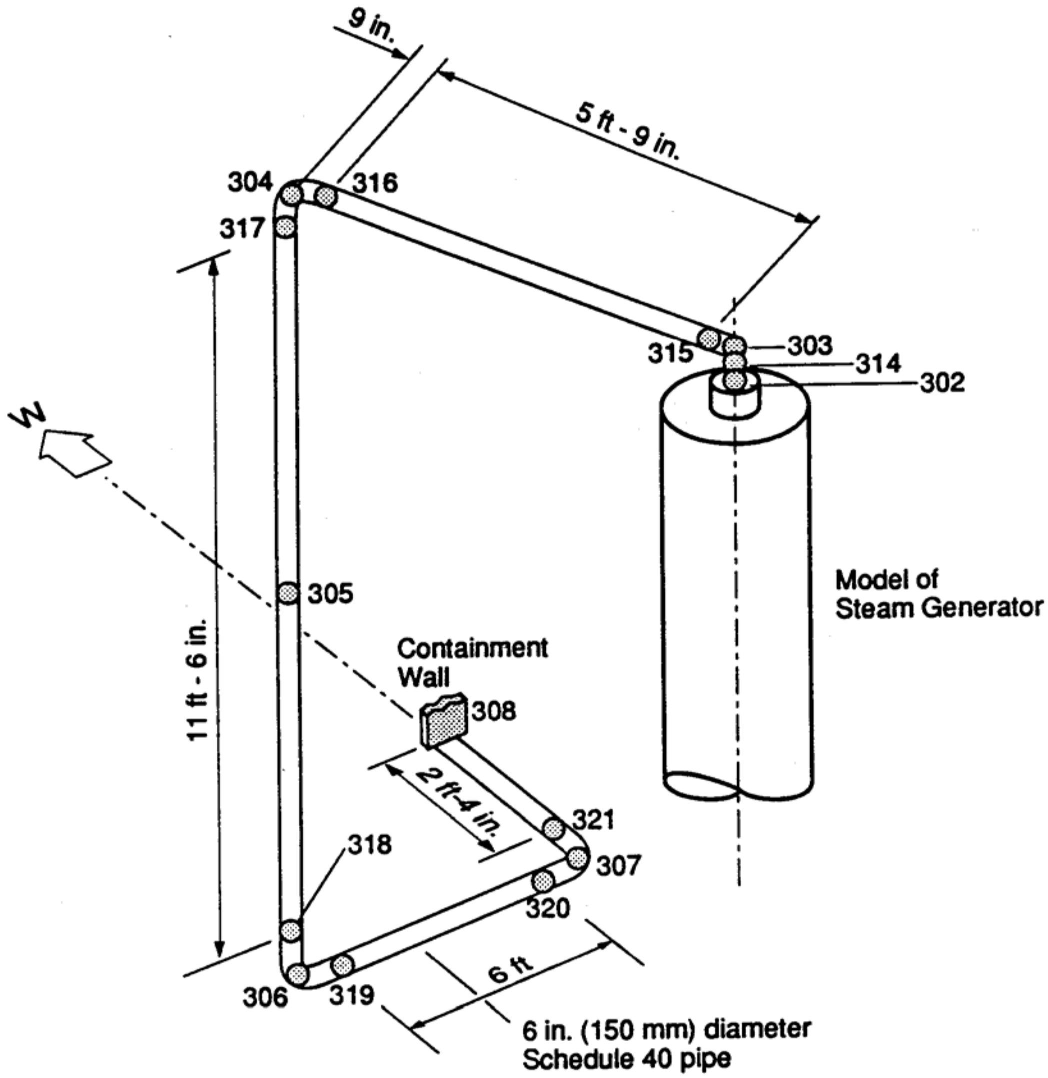


Figure 4.6–30. Piping System Model

Table 4.6-7. Properties of the Structural Models of the Piping System

(Steel Modulus E = 2.09 x 10^6 ksf, G = 0.80 x 10^6 ksf, β = 1%)

Joint Properties			Member Properties			
Mass No.	M_j x g (kips)	I_j x g (kip-ft^2)	Location Between Joint No.	Area (ft^2)	Shear Area (ft^2) A2 and A3*	Moment of Inertia (ft^4) I2 and I3
314	0.000554					
303	—	—	302 to 314	0.0956	0.0442	0.01226
315	0.00131	—	314 to 303	0.0388	0.0204	0.00027
316	0.00206	—	303 to 315	0.0388	0.0204	0.00027
304	—	—	315 to 316	0.0388	0.0204	0.00135
317	—	—	316 to 304	0.0388	0.0204	0.00027
305	—	—	304 to 317	0.0388	0.0204	0.00027
318	0.00180	—	317 to 305	0.0388	0.0204	0.00135
306	—	—	305 to 318	0.0388	0.0204	0.00135
319	—	—	318 to 306	0.0388	0.0204	0.00027
320	0.00120	—	306 to 319	0.0388	0.0204	0.00027
307	—	—	319 to 320	0.0388	0.0204	0.00135
321	—	—	320 to 307	0.0388	0.0204	0.00027
308	0.00038	—	307 to 321	0.0388	0.0204	0.00027
			321 to 308	0.0388	0.0204	0.00135

Note: See Fig 4.6-17 for the orientation of the local 2 and 3 axes.

Table 4.6-8. Fixed-Base Modal Properties of the Containment Model Structures

Mode No.	Frequencies f (HZ)	Modal Masses (In Percent) of Total		
		(N-S)	(E-W)	(Vertical)
1*	5.51	0.8	0.05	0.
2*	5.67	0.04	0.76	0.
3*	10.49	0.	0.01	0.01
4#	10.82	0.92	83.08	0.
5#	10.82	83.1	0.92	0.01
6*	15.41	0.	0.01	0.03
7*	16.22	0.03	0.	1.0
8*	18.29	0.07	0.02	0.
9*	19.95	0.03	0.15	0.
10#	32.88	0.	0.	89.19
11*	38.46	0.01	0.	0.
12*	42.88	0.	0.	0.82
13*	43.71	0.	0.	0.12
14#	48.46	12.89	.37	0.
15#	48.47	0.37	2.9	0.
Cumulative Ratios:		98.25	98.27	90.19

Total Weight = 1.4 x 10^3 kips

----
* = Frequencies of the internal structures (steam generator and its supporting steel frame coupled with piping.)
\# = Frequencies of the containment shell.

Table 4.6-9. Frequencies of Analysis

Frequency Number (NF)	Frequency f (HZ)
1	0.024
41	1.0
82	2.0
103	2.51
111	2.71
122	2.98
144	3.51
184	4.49
204	4.98
213	5.20
225	5.49
234	5.71
266	6.49
287	7.01
397	9.69
442	10.79
532	12.99

4.6.2.2.4 Analysis Cases

The SASSI analysis has been performed for one case of East-West horizontal shaking. The SASSI sequence of runs and input/output files for this validation problem are shown in Table 4.6-10.

4.6.2.2.5 Results and Comparison

The results of the SASSI analysis in terms of 5 percent acceleration response spectrum at 4 locations in the structure are computed and compared with the corresponding recorded motions. The 4 locations in the containment are shown in Fig. 4.6-31 which consists of 2 points at the top (Node 311 in SASSI model) and base of the containment (southern end, Node 178) and 2 points at the top and lower platform support of steam generator (Nodes 302 and 300, respectively). Acceleration response spectra of the recorded motions at these 4 points are compared with the SASSI results in Figs. 4.6-32 through 4.6-35. As shown, SASSI results are in good agreement with the recorded SSI motions. Both the direct method and the substance method predict accurate results in comparison with the recorded motions.

The CPU time obtained from the ANALYS output using the direct and the subtraction methods are shown on Table 4.6-11. The platform used is IBM PC 166MHz with Windows 95 operating system. This comparison shows a significant reduction in CPU time when the subtraction method is used. Also, the size of Tapes 5 and 6 are much smaller when the subtraction method is used.

Table 4.6-10. Input/Output Files of Example Problem 2

MODULE USED	CASE 1 OF ANALYSIS			
	INPUT FILE	OUTPUT FILE	TAPE IN	TAPE OUT
SITE	E2C1SD*	E2C1SO	—	E2C1T1 E2C1T2**
POINT 3	E2C1PD	E2C1PO	E2C1T2	E2C1T3
HOUSE	E2C1HD Direct Or Subtraction		E2C1HO	E2C1T4
ANALYS	E2C1AD	E2C1AO	E2C1T 1E2C1T3 E2C1T4	E2C1T5 E2C1T6 E2C1T8
MOTION	E2C1OD	E2C1OO	E2C1T8	—

* E2C1SD - Stands for Example 2, Case 1, Site Data

** E2C1T2 - Stands for Example 2, Case 1, Tape 2

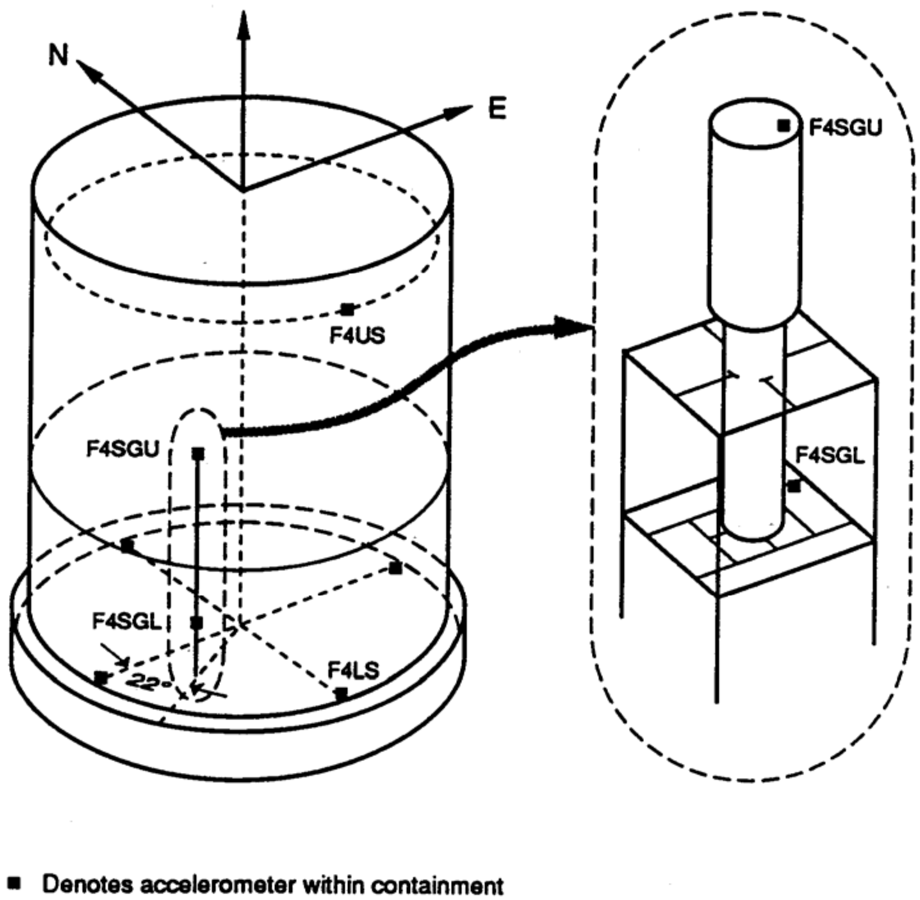


Figure 4.6–31. Locations of Accelerometers

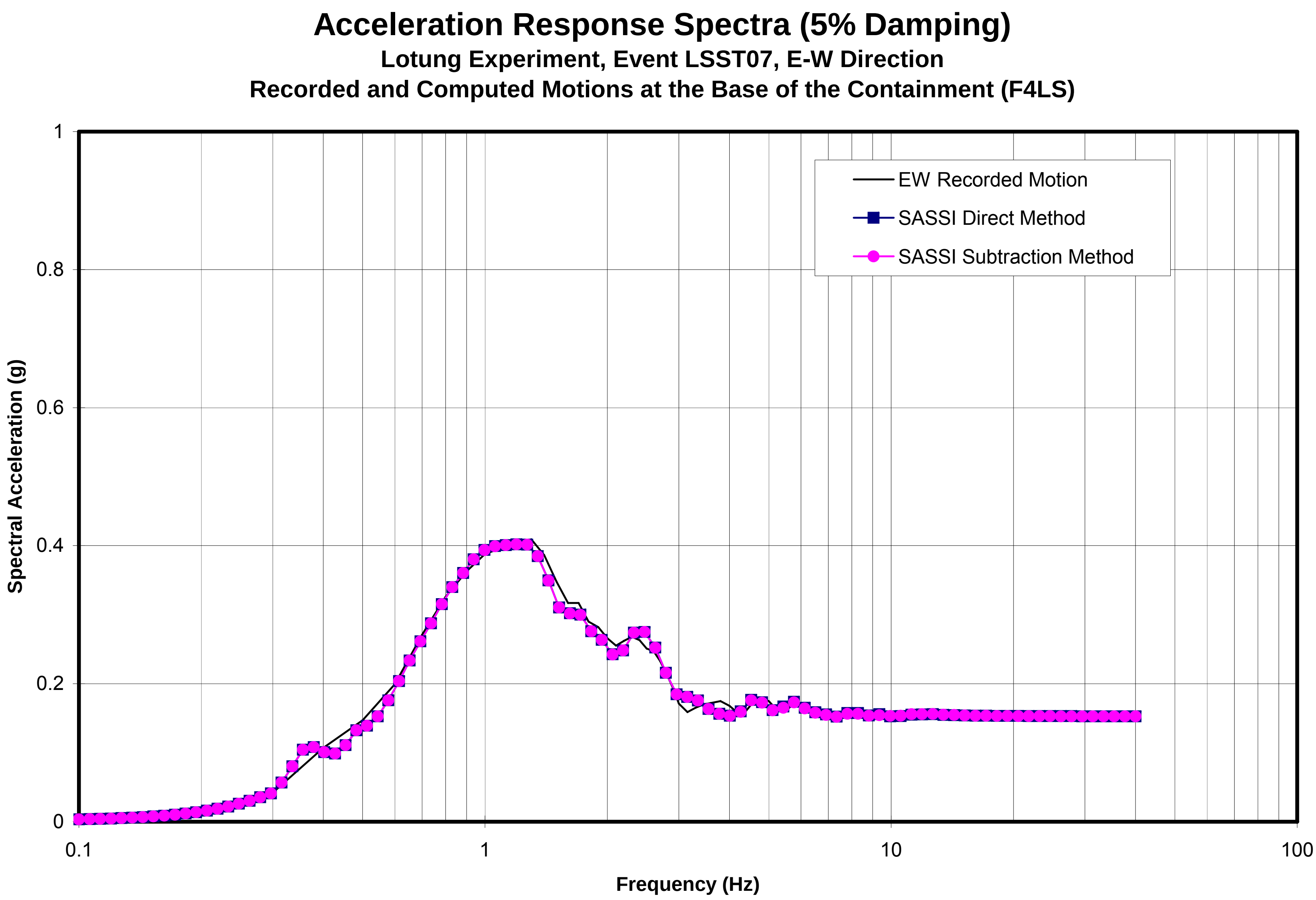


Figure 4.6–32. Comparison of Responses at the Top of the Basemat

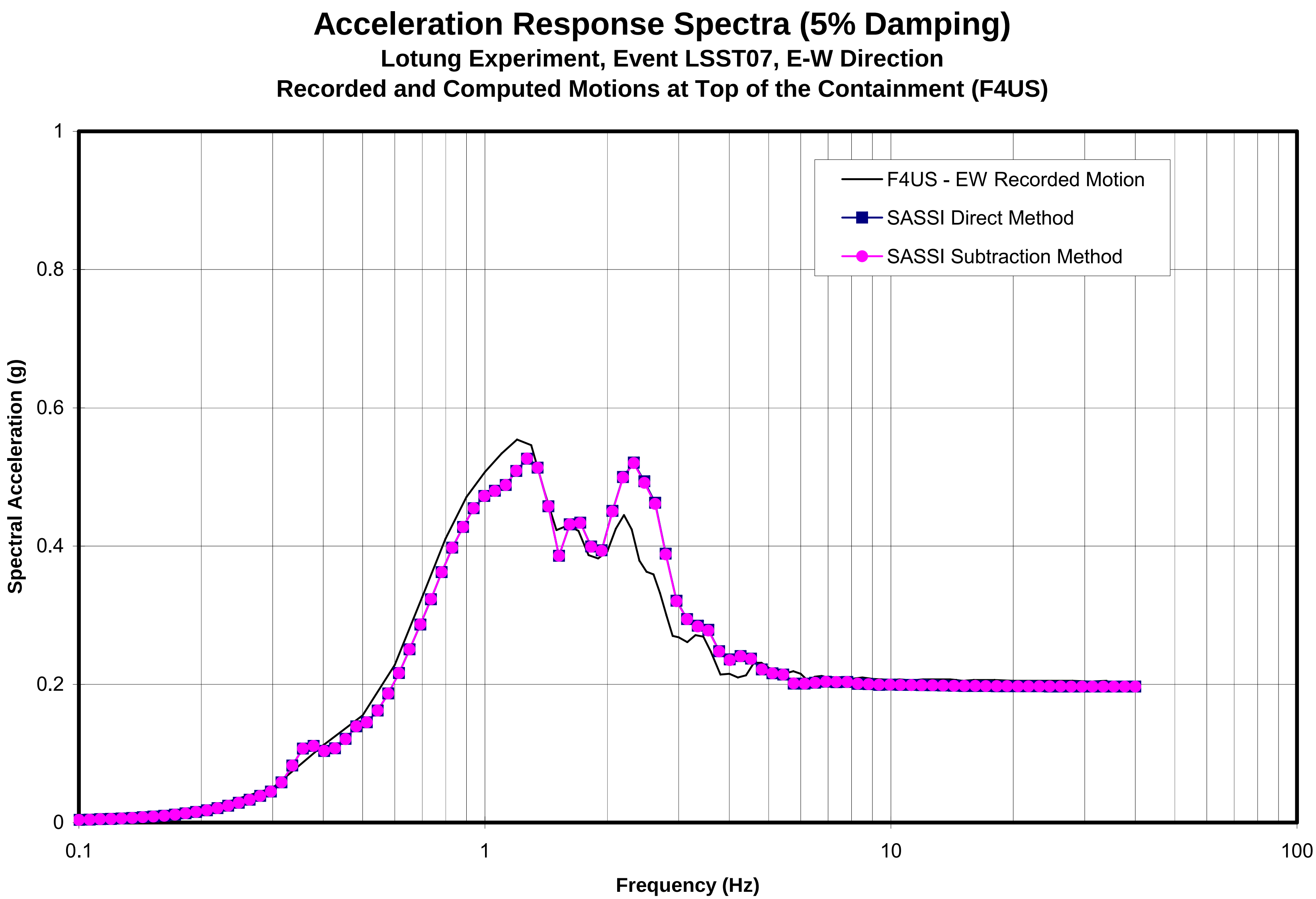


Figure 4.6–33. Comparison of the Responses at the Top of the Containment

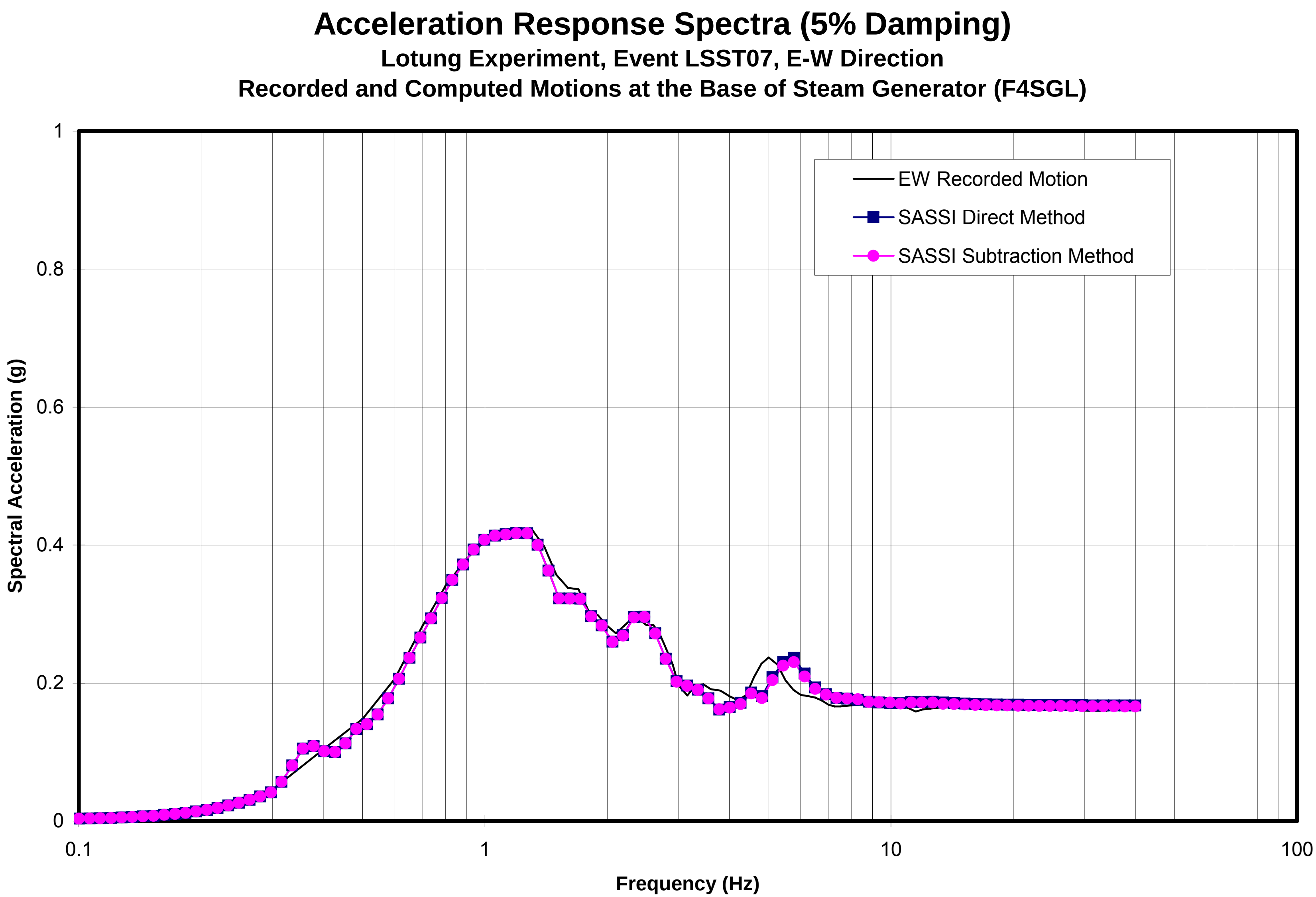


Figure 4.6–34. Comparison of the Response at the Base of Steam Generator

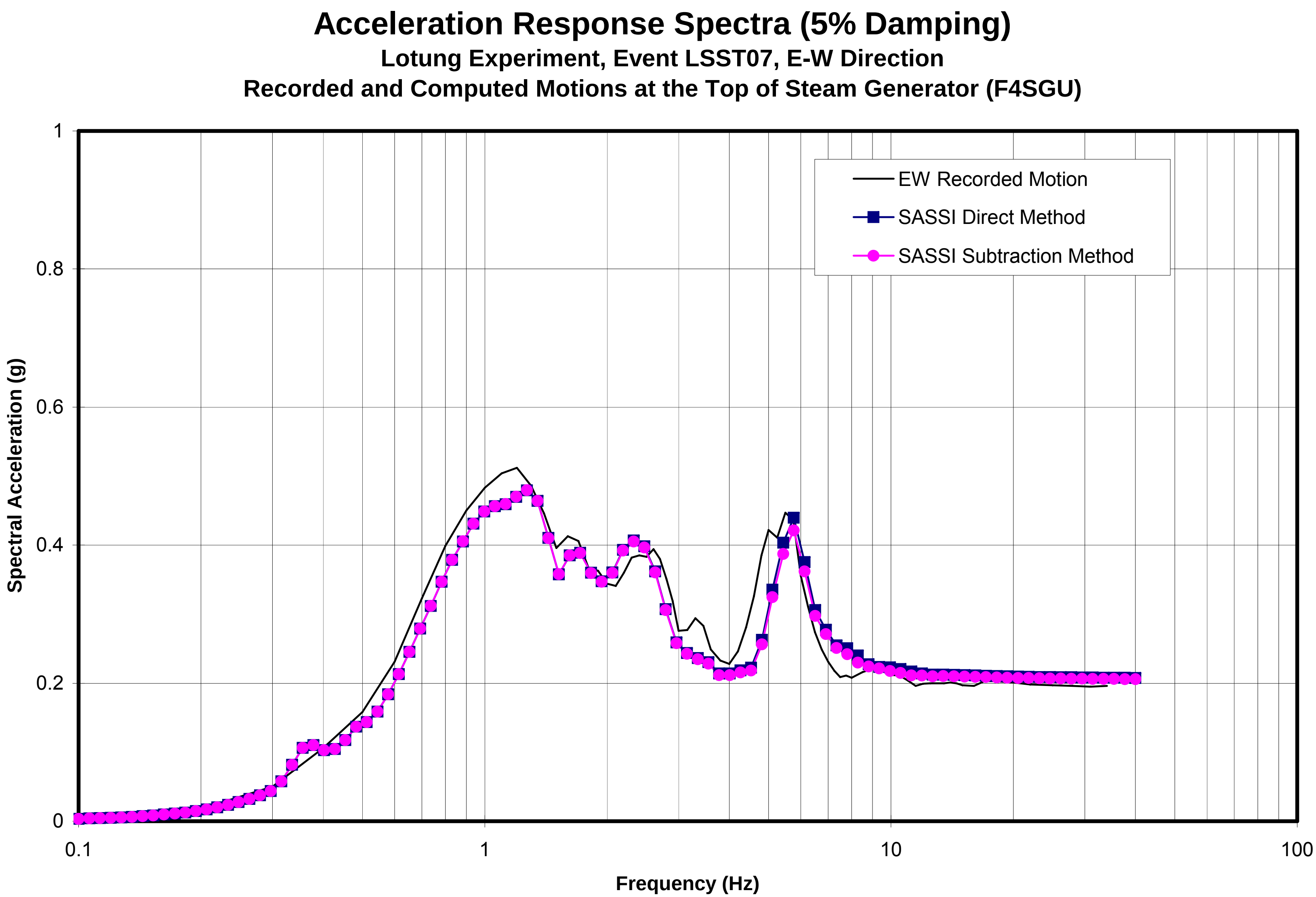


Figure 4.6–35. Comparison of the Responses at the Top of Steam Generator

Table 4.6-11. CPU Time from ANALYS Output of Example 2 (Seconds)

	Form Flexibility Matrix	Form Impedance Matrix	Total Solution Time
Direct Method	1423	760	2845
Subtraction Method	289	197	841

4.6.3 Example Problem 3, Dynamic Responses Of Pile Groups In A Homogeneous Halfspace Using Inter-Pile Elements

4.6.3.1 Description

Horizontal and vertical responses at the pile head of a single pile and three pile groups of different configurations in a homogeneous halfspace are computed in reaction to harmonic unit loads at the pile head. The pile-soil-pile interaction effects are simulated using the 3-dimensional inter-pile elements. SASSI responses are used to compute the normalized dynamic stiffness and damping coefficients of the pile groups and are compared with the analytical results reported by Kaynia et al. (10) to validate the analysis capability of SASSI for dynamic responses of 3-dimensional pile groups.

4.6.3.2 Analysis Models

This problem consists of 4 cases. In Case 1, the horizontal and vertical dynamic responses at pile head of a single pile are computed. In Case 2, the responses of a 2x2 pile group with S/D = 2 are computed, where D is the diameter of the pile and S is the center-to-center spacing between two piles. Case 3 is for a 2x2 pile group with S/D = 5. And Case 4 is for a 3x3 pile group with S/D = 2.

The SASSI site model considered is shown in Figure 4.6-36. The model consists of 15 top layers of equal thickness of 2 ft. and additional 10 layers with varying thicknesses with the viscous boundary at the base to simulate the halfspace. All layers and the underlying halfspace are of the following material properties:

Shear Wave Velocity	V_s	=	300 ft/sec
Poisson's Ratio	v	=	0.40
Material Damping	β	=	0.05
Unit Weight	γ	=	112.8 pcf

The SASSI analysis models for the 4 cases are shown on Figs. 4.6-37, through 4.6-40, respectively. Each model consists of 3 parts: the excavated soil model using 8-node brick elements, the inter-pile element model, representing the interacting volume within the pile group, using the 3-D inter-pile element models, and the 3-D beam element model to model the stiffness and mass properties of each pile. All piles modeled in the analysis cases are circular piles with constant diameter D = 2' and a total length L = 30 ft. The material properties of the piles are:

Young's Modulus	E_p	=	8.28×10^8 psf
Poisson's ratio	ν_p	=	0.25
Weight density	γ_p	=	161.1 pcf
Material damping	β_p	=	0.0

In the single pile case (Case 1) the rotational degrees-of-freedom (DOFs) are fixed, representing the "fixed-head" condition. In Cases 2, 3 and 4, the individual piles are linked together with rigid beams, representing a rigid pile cap restricting rotational movement of individual piles.

The frequencies selected for the analysis are shown in Table 4.6-12. The dimensionless frequency ratio, a_o, shown in this table is defined by

$$a_o = \frac{2\pi f D}{V_s} \qquad (4.6\text{-}1)$$

where D is the diameter of the pile (2 ft.); V_s is the shear wave velocity of the halfspace (300 ft./sec.) and f is the frequency of analysis. Frequencies (f) are the product of frequency step (DF = 0.01 Hz) and the frequency numbers (NF).

Table 4.6-12. Frequencies of SASSI Analysis

Frequency Number NF	Frequency (Hz)	Dimensionless Frequency a_0
1	0.01	0.0004
10	0.10	0.004
239	2.39	0.100
477	4.77	0.200
716	7.16	0.300
954	9.54	0.400
1193	11.93	0.500
1431	14.31	0.599
1670	16.70	0.700
1908	19.08	0.799
2147	21.47	0.899
2385	23.85	0.999

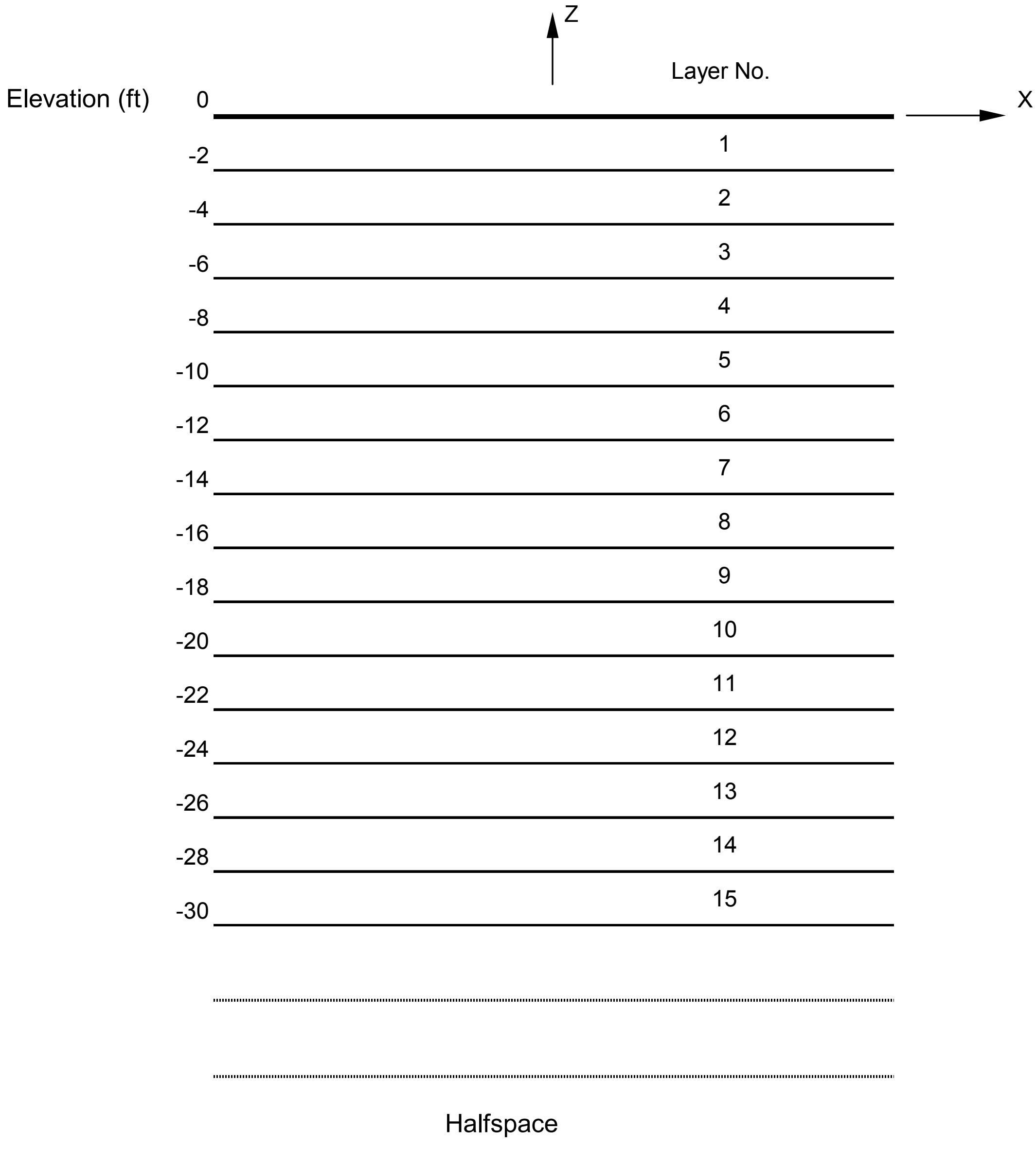


Figure 4.6–36. SASSI Soil Profile Model

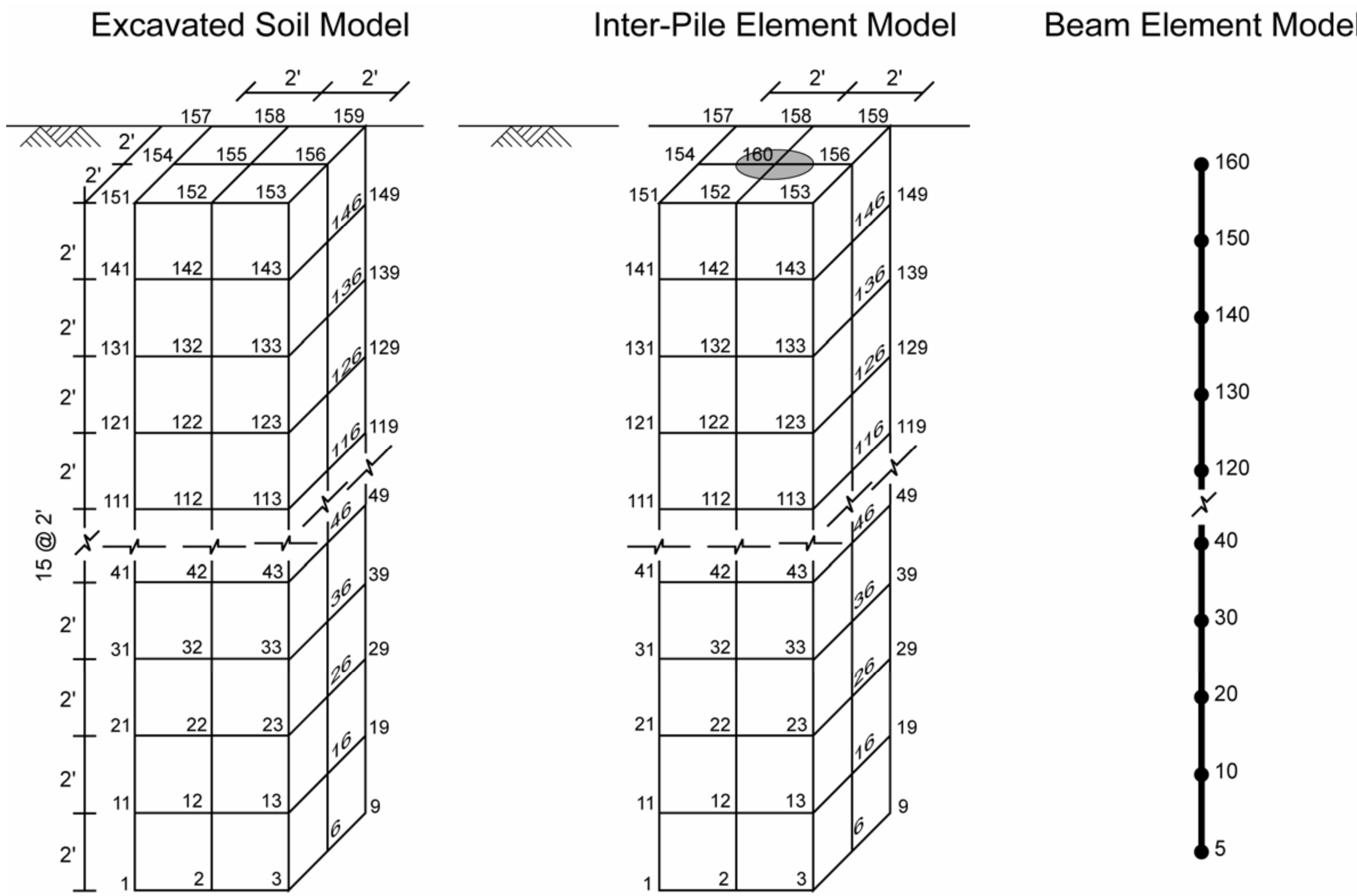


Figure 4.6–37. SASSI Analysis Model for a Single Pile

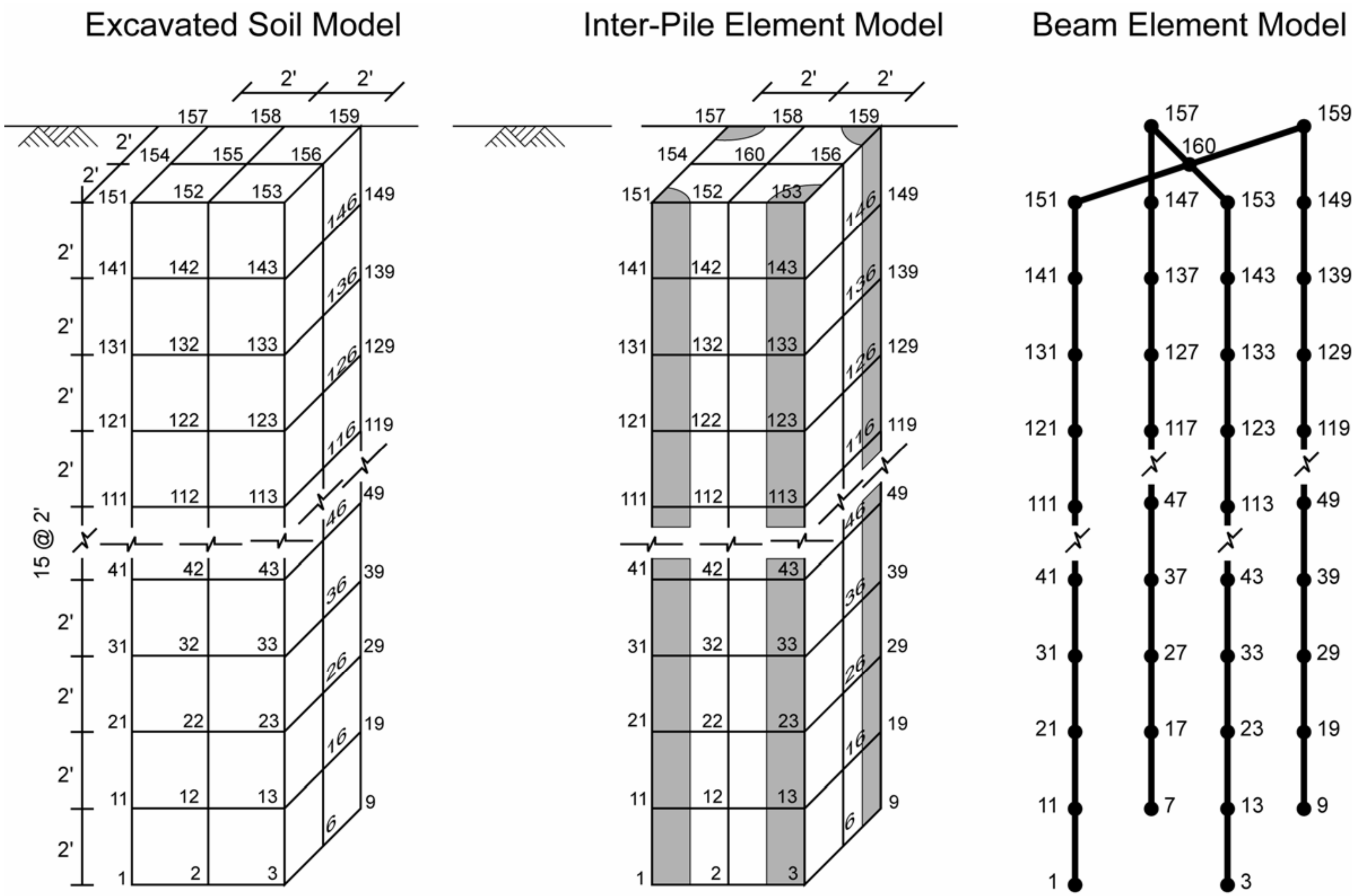


Figure 4.6–38. SASSI Analysis Model for a 2x2, *S*/D=2 Pile

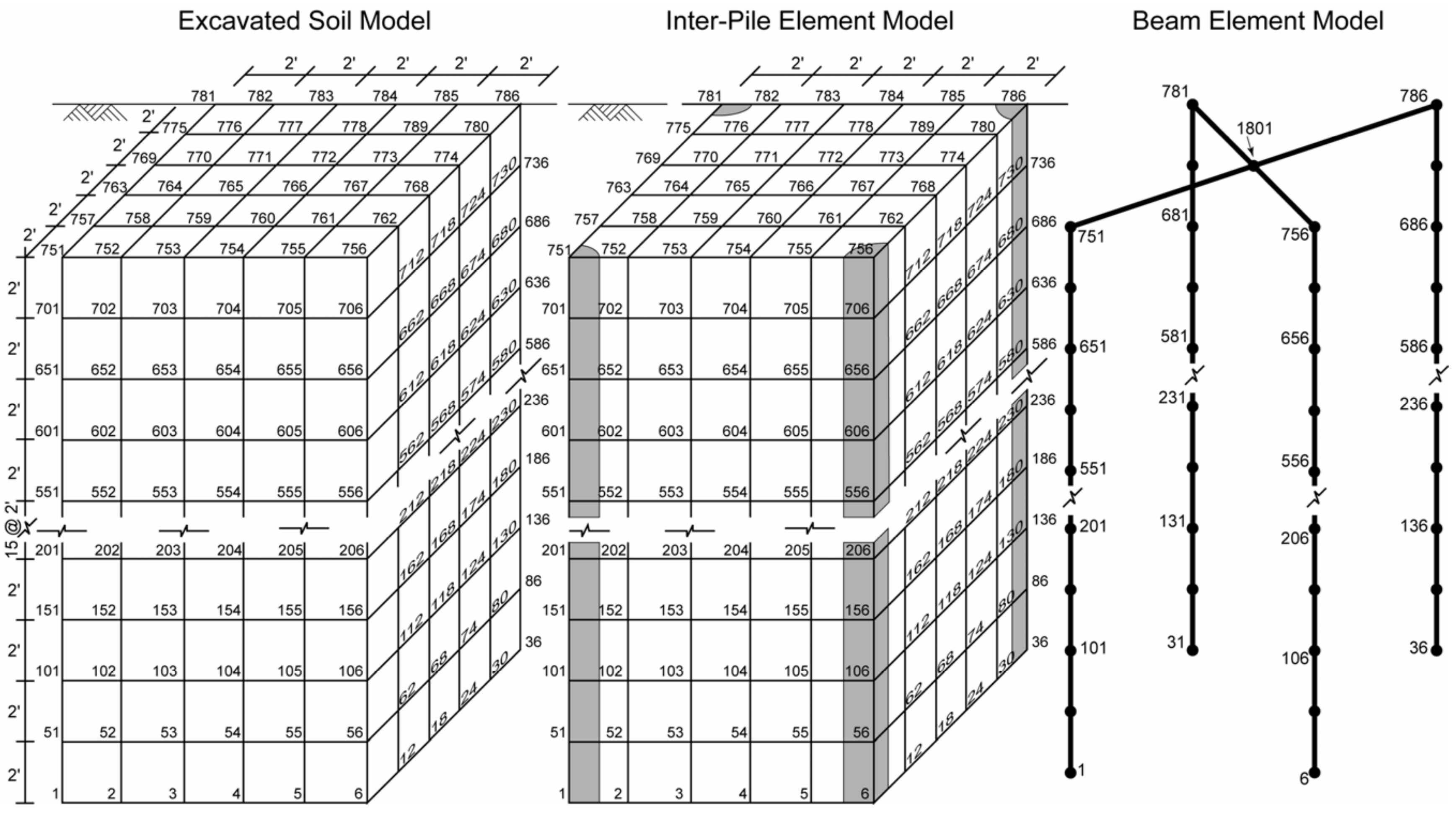


Figure 4.6–39. SASSI Analysis Model for a 2x2, S/D=5 Pile Group

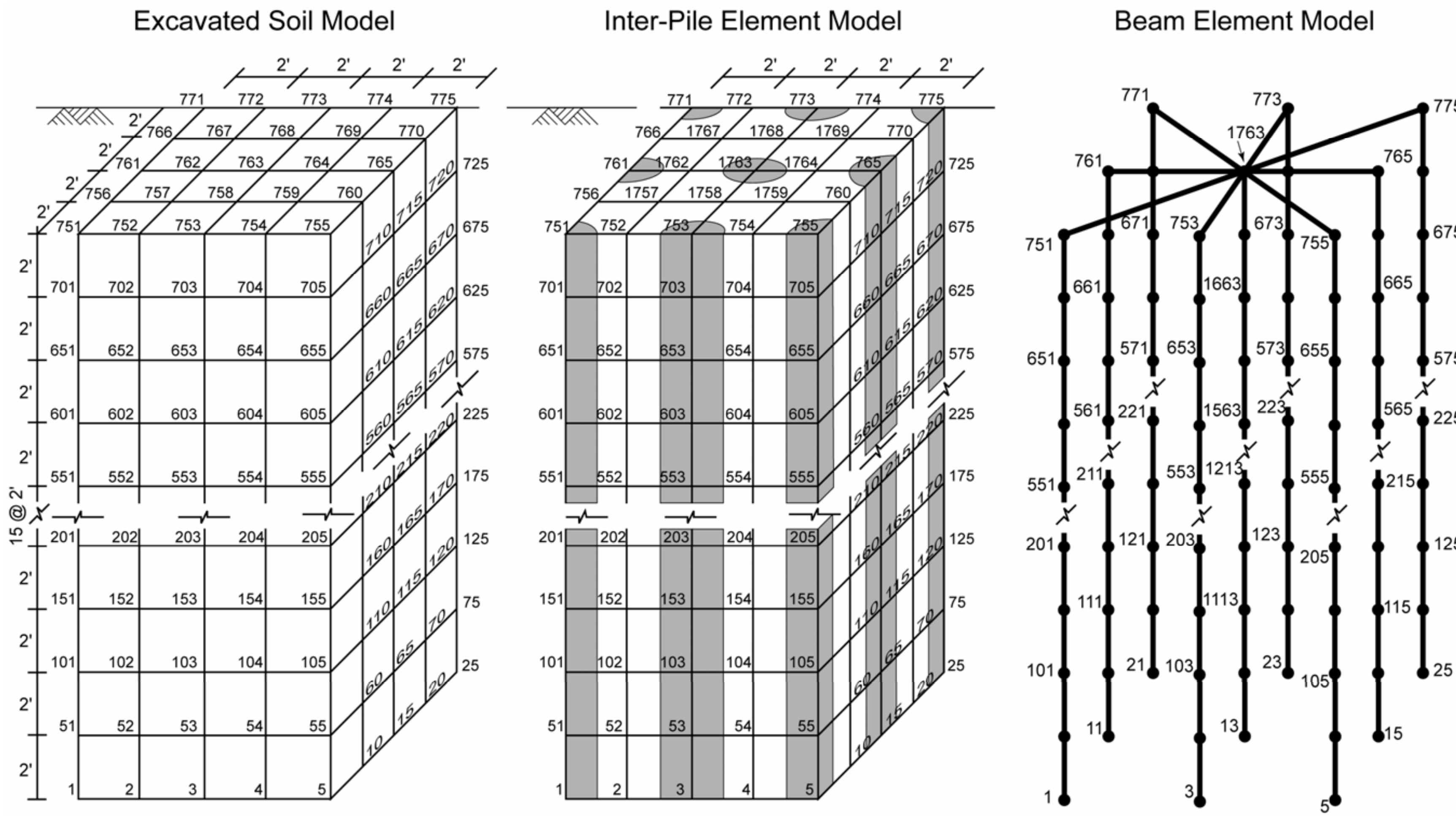


Figure 4.6–40. SASSI Analysis Model for a 3x3, S/D=2 Pile Group

4.6.3.3 Analysis Cases

As described above, the SASSI runs for this problem consist of four (4) cases: Case 1 through Case 4 consist of a single pile, a 2x2 group with S/D = 2, a 2x2 group with S/D = 5, and a 3x3 group with S/D = 2, respectively.

In Case 1, the uniform harmonic load, in separately horizontal and vertical directions, is applied at the pile head, Node 160. Responses at the same node are computed.

In Cases 2, 3, and 4, the unit harmonic load, in separately horizontal and vertical directions, is applied at the center of the pile cap, Node 160, 1763 and 1801, and the responses at the same node are computed, respectively.

The SASSI run sequences and input/output files for this problem are shown in Table 4.6-13.

Table 4.6-13. Input/Output Files of Example Problem No. 3

	CASE 1 OF ANALYSIS				CASE 2 OF ANALYSIS			
SASSI MODULE	**INPUT**	**OUTPUT**	**TAPEIN**	**TAPE OUT**	**INPUT**	**OUTPUT**	**TAPEIN**	**TAPE OUT**
SITE	E3C1.SD	E3C1.SO		E3C1.T2	E3C2.SD	E3C2.SO		E3C2.T2
POINT3	E3C1.PD	E3C1.PO	E3C1.T2	E3C1.T3	E3C2.PD	E3C2.PO	E3C2.T2	E3C2.T3
HOUSE	E3C1.HD	E3C1.HO		E3C1.T4	E3C2.HD	E3C2.HO		E3C2.T4
MOTOR	E3C1H.MD	E3C1H.MO		E3C1H.T9	E3C2H.MD	E3C2H.MO		E3C2H.T9
ANALYS	E3C1.AD	E3C1H.AO	E3C1.T3 E3C1.T4 E3C1H.T9	E3C1H.T8	E3C2.AD	E3C2H.AO	E3C2.T3 E3C2.T4 E3C2H.T9	E3C2H.T8
MOTOR	E3C1V.MD	E3C1V.MO		E3C1V.T9	E3C2V.MD	E3C2V.MO		E3C2V.T9
ANALYS	E3C1.AD	E3C1V.AO	E3C1.T3 E3C1.T4 E3C1V.T9	E3C1V.T8	E3C2.AD	E3C2V.AO	E3C2.T3 E3C2.T4 E3C2V.T9	E3C2V.T8

	CASE 3 OF ANALYSIS				CASE 4 OF ANALYSIS			
SASSI MODULE	**INPUT**	**OUTPUT**	**TAPEIN**	**TAPE OUT**	**INPUT**	**OUTPUT**	**TAPEIN**	**TAPE OUT**
SITE	E3C3.SD	E3C3.SO		E3C3.T2	E3C4.SD	E3C4.SO		E3C4.T2
POINT3	E3C3.PD	E3C3.PO	E3C3.T2	E3C3.T3	E3C4.PD	E3C4.PO	E3C4.T2	E3C4.T3
HOUSE	E3C3.HD	E3C3.HO		E3C3.T4	E3C4.HD	E3C4.HO		E3C4.T4
MOTOR	E3C3H.MD	E3C3H.MO		E3C3H.T9	E3C4H.MD	E3C4H.MO		E3C4H.T9
ANALYS	E3C3.AD	E3C3H.AO	E3C3.T3 E3C3.T4 E3C3H.T9	E3C3H.T8	E3C4.AD	E3C4H.AO	E3C4.T3 E3C4.T4 E3C4H.T9	E3C4H.T8
MOTOR	E3C3V.MD	E3C3V.MO		E3C3V.T9	E3C4V.MD	E3C4V.MO		E3C4V.T9
ANALYS	E3C3.AD	E3C3V.AO	E3C3.T3 E3C3.T4 E3C3V.T9	E3C3V.T8	E3C4.AD	E3C4V.AO	E3C4.T3 E3C4.T4 E3C4V.T9	E3C4V.T8

4.6.3.4 Results and Comparison

The force-displacement relationship at the pile head can in general term be expressed in the following form:

$$P_x = \mathrm{K}_{\mathrm{xx}}^{\mathrm{st}}\left(K_{xx} + \mathrm{ia_o}C_{xx}\right)\left(\mathrm{ReU_{xx}} + \mathrm{iImU_{xx}}\right) \tag{4.6-2}$$

$$P_z = \mathrm{K}_{\mathrm{zz}}^{\mathrm{st}}\left(K_{zz} + \mathrm{ia_o}C_{zz}\right)\left(\mathrm{ReU_{zz}} + \mathrm{iImU_{zz}}\right) \tag{4.6-3}$$

where P_x, P_z is the force acting on the pile head in x- and z-directions, and $\mathrm{K}_{\mathrm{xx}}^{\mathrm{st}}$, $\mathrm{K}_{\mathrm{zz}}^{\mathrm{st}}$ are the static stiffness in the xx- and zz-directions, respectively; a_0 is the dimensionless frequency as defined in Eq (4.6-1), K_{xx}, K_{zz} are the normalized dynamic stiffness terms, and C_{xx}, C_{zz} are the normalized damping terms. U_{xx} and U_{zz} are the computed responses at the pile head. All the terms in the above equations except the static stiffness terms are functions of frequency.

For the particular case of unit load, $P_x = 1$ and $P_z = 1$, the normalized stiffness and damping terms from the above expressions can be reduced to

$$K_{ii} = \frac{1}{\mathrm{K}_{\mathrm{ii}}^{\mathrm{st}}}\frac{\mathrm{ReU_{ii}}}{|\mathrm{U_{ii}}|^2}, \qquad \mathrm{ii = xx,\ zz} \tag{4.6-4}$$

$$C_{ii} = \frac{-1}{\mathrm{a_0K_{ii}^{st}}}\frac{\mathrm{ImU_{ii}}}{|\mathrm{U_{ii}}|^2}, \qquad \mathrm{ii = xx,\ zz} \tag{4.6-5}$$

where $\mathrm{K}_{\mathrm{xx}}^{\mathrm{st}}$, $\mathrm{K}_{\mathrm{zz}}^{\mathrm{st}}$ are the stiffness of the foundation calculated at $a_0 = 0$.

For pile groups, the responses of K_{ii}, C_{ii}, ii = xx, zz, are normalized by the response of single pile:

$$K_{ii}^{g} = \frac{1}{NK_{ii}^{st,s}} \frac{ReU_{ii}^{g}}{|U_{ii}^{g}|^2}, \qquad \text{ii = xx, zz} \qquad (4.6\text{-}6)$$

$$C_{ii}^{g} = \frac{-1}{a_0 NK_{ii}^{st,s}} \frac{ImU_{ii}^{g}}{|U_{ii}^{g}|^2}, \qquad \text{ii = xx, zz} \qquad (4.6\text{-}7)$$

where the superscript g indicates the term for the pile group, and the superscript s indicates the term for a single pile. N is the total number of piles in the group.

The computed results in all four cases are computed and normalized according to Eqs (4.6-2) through (4.6-7) and are compared with the results reported by Kaynia (10). Figure 4.6-41 shows the comparison for the single pile case. Figure 4.6-42 shows the comparison for the 2x2 group, S/D=2 case. Figure 4.6-43 shows the comparison for the 2x2 group, S/D=5 case, and Figure 4.6-44 shows the comparison for the 3x3 group, S/D=2 case. It can be observed from the comparisons that in all cases good to very good agreement between the corresponding results are reached.

4.6.3.5 Conclusion

The good agreement between the SASSI results and the results reported by Kaynia validates the analysis capability of SASSI for single piles and pile groups with the inter-pile elements.

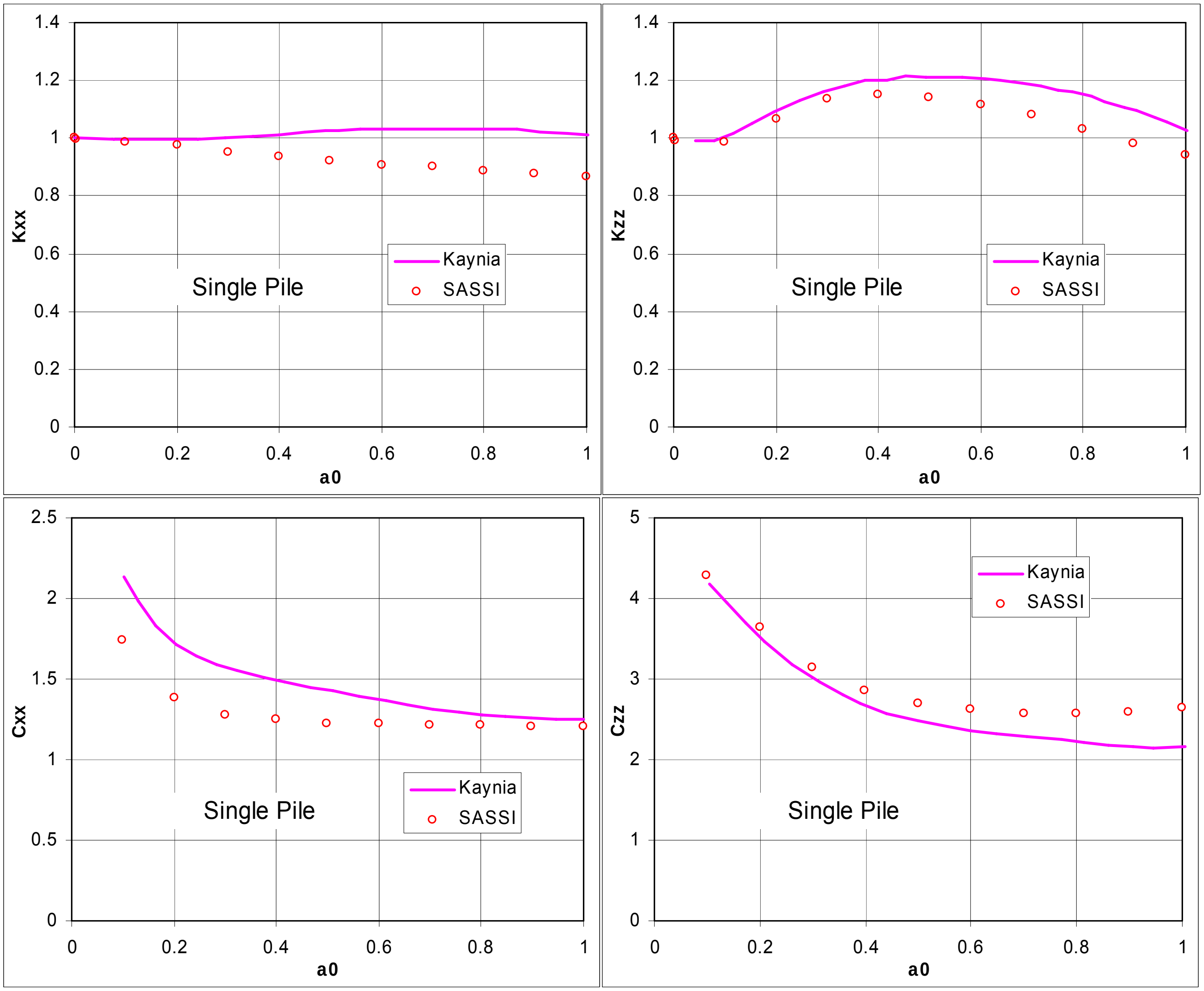


Figure 4.6–41. SASSI Analysis Results for a Single Pile

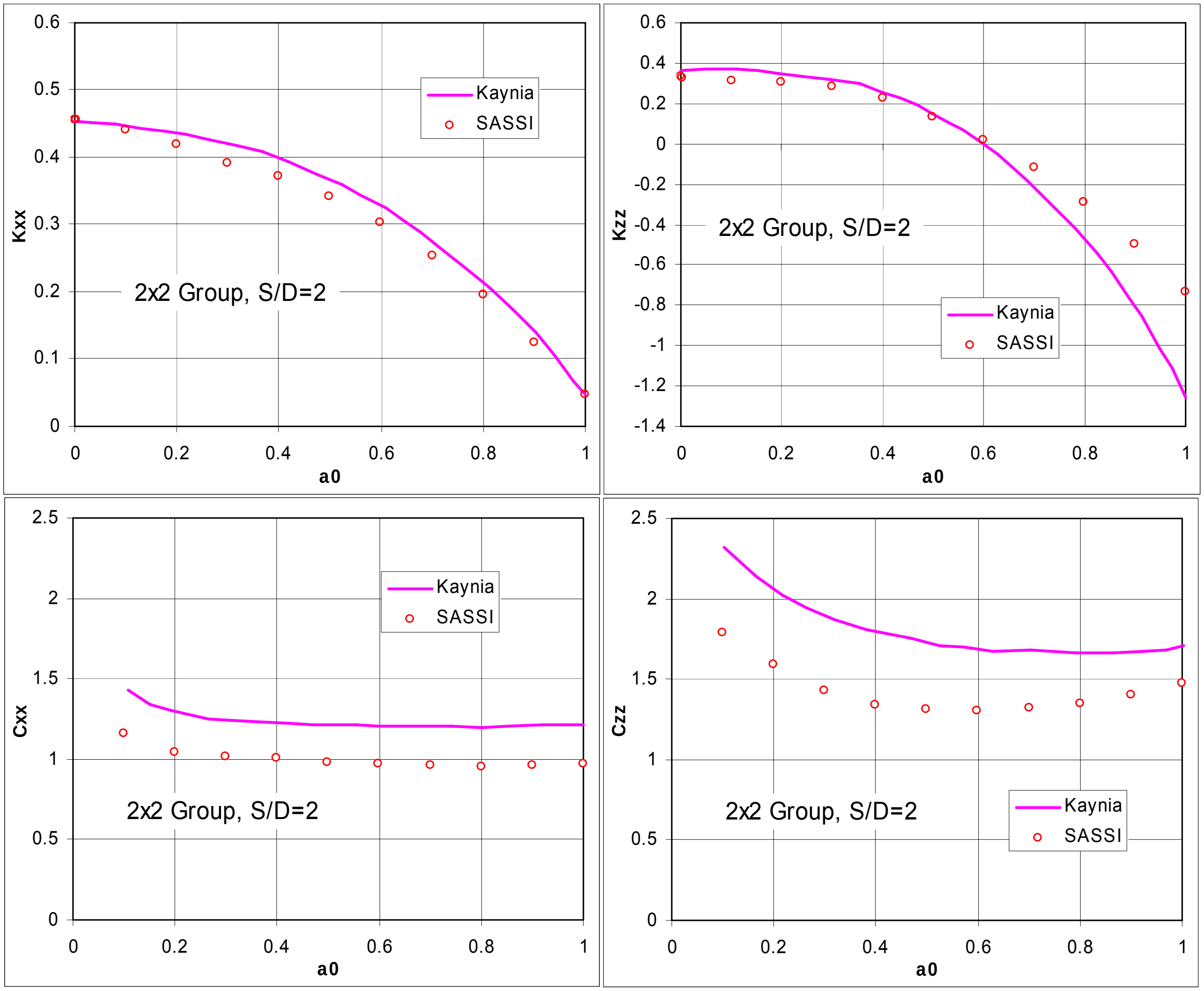


Figure 4.6–42. SASSI Analysis Results for a 2x2, S/D = 2 Pile Group

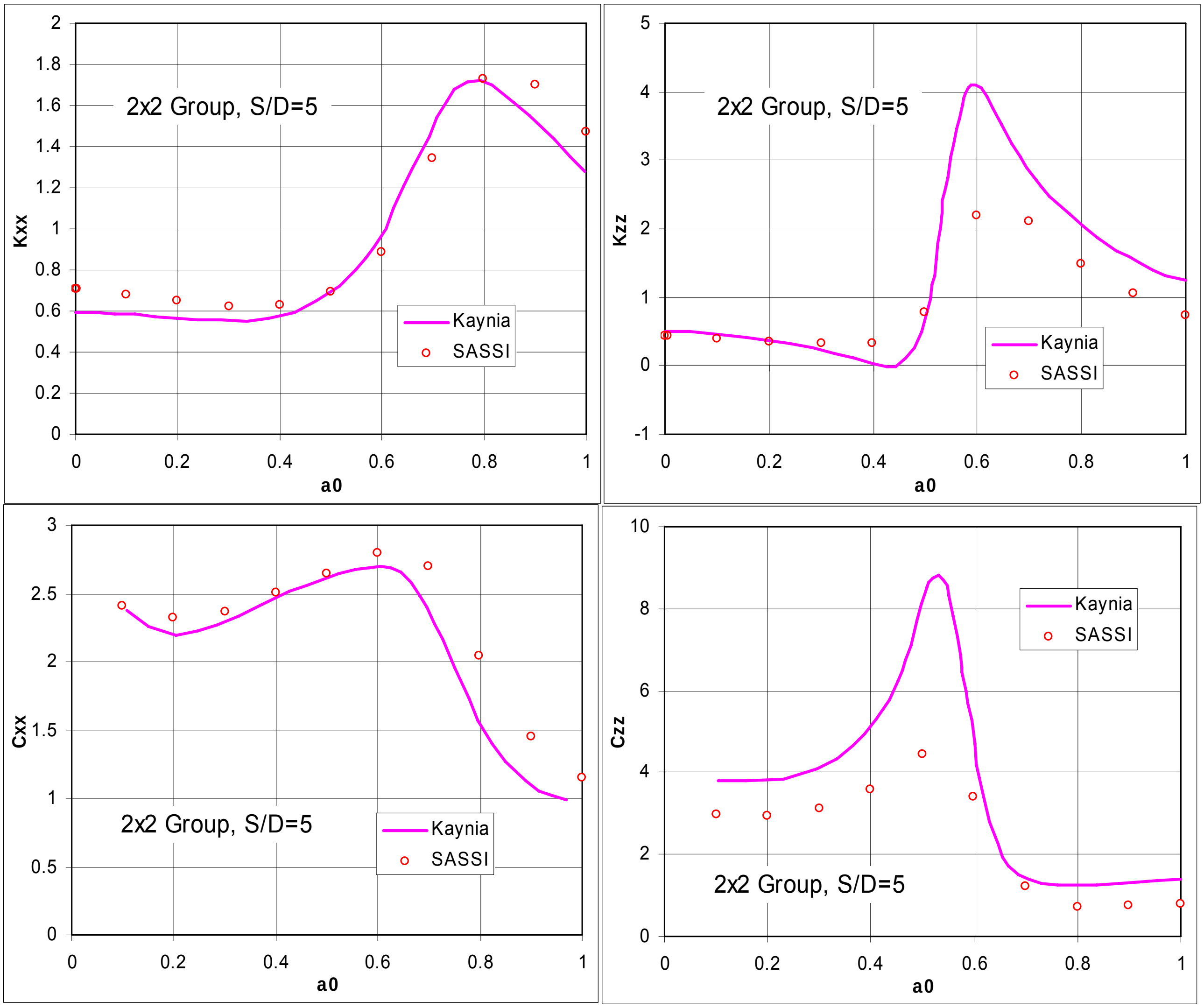


Figure 4.6–43. SASSI Analysis Results for a 2x2, S/D = 5 Pile Group

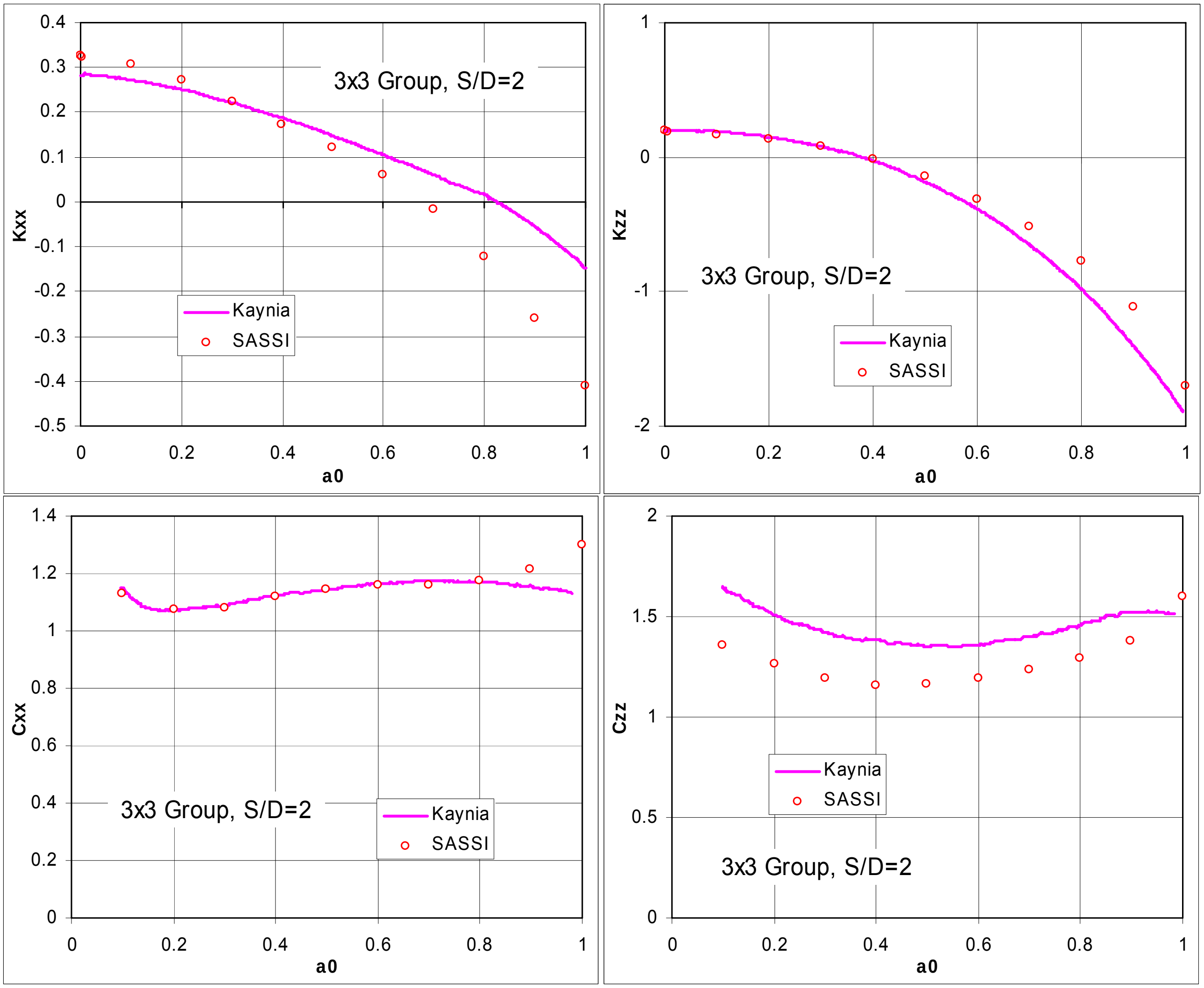


Figure 4.6–44. SASSI Analysis Results for a 3x3, S/D = 2 Pile Group

4.6.4 Example Problem 4, Dynamic Responses Of Pile Groups In A Homogeneous Halfspace Using The Pile Impedance Method

4.6.4.1 Description

Horizontal and vertical responses at center of the pile cap of three pile groups of different configurations in a homogeneous halfspace are computed in reaction to harmonic unit loads at the pile head. The pile-soil-pile interaction effects are simulated using the semi-analytical pile impedance method. SASSI responses are used to compute the dynamic stiffness and damping coefficients of the pile group. The results are compared with the analytical results reported by Kaynia et al (10) to validate the analysis capability of SASSI for dynamic responses of 3-dimensional pile groups.

4.6.4.2 Analysis Models

This problem consists of three (3) cases. In Case 1, the horizontal and vertical dynamic responses at center of pile cap of 2x2 pile groups with S/D = 2, 5 and 10 are computed, where D is the diameter of the pile and S is the center-to-center spacing between two piles. Case 2 is for 3x3 pile groups with S/D = 2, 5, and 10. And Case 3 is for 4x4 pile groups with S/D = 2, 5 and 10.

The SASSI site model considered is shown in Figure 4.6-45. The model consists of 12 top layers of equal thickness of 3 ft. and additional 10 layers with varying thicknesses with the viscous boundary at the base to simulate the halfspace. All layers and the underlying halfspace are of the following material properties:

Shear Wave Velocity	V_s	=	300 ft/sec
Poisson's Ratio	v	=	0.40
Material Damping	β	=	0.05
Unit Weight	γ	=	112.7 pcf

The SASSI analysis models for the 3 cases are shown on Figs. 4.6-46, through 4.6-48, respectively. In each case, the pile impedance of a single pile, or the complex frequency-dependent stiffness of the pile, is first computed with respect to the unit load on pile head using the program module **SPILE**. All piles considered in the analysis cases are circular piles with constant diameter D = 2' and a total length L = 30 ft. The material properties of the piles are:

Young's Modulus	E_p	=	$8.28\text{x}10^8$ psf
Poisson's ratio	ν_p	=	0.25
Weight density	γ_p	=	161.1 pcf
Material damping	β_p	=	0.0

In all cases, the pile head is fixed at the rotational degrees-of-freedom (DOFs) but free to move at the translational DOFs. Since all piles considered in the model cases are of one type, there is only one type of pile impedance needs to be computed. The pile impedance information is written in Tape 31.

In the **HOUSE** model, the nodal points where the pile heads are located are identified with the number of the type(s) of the pile impedance. Rigid beams are used to connect the points to form a pile cap.

The frequencies selected for the analysis are shown in Table 4.6-14. The dimensionless frequency ratio, a_o, shown in this table is defined by

$$a_o = \frac{2\pi fD}{V_s} \tag*{4.6-8)}$$

where D is the diameter of the pile (2 ft.); V_s is the shear wave velocity of the halfspace (300 ft./sec.) and f is the frequency of analysis. Frequencies (f) are the product of frequency step (DF = 0.01 Hz) and the frequency numbers (NF).

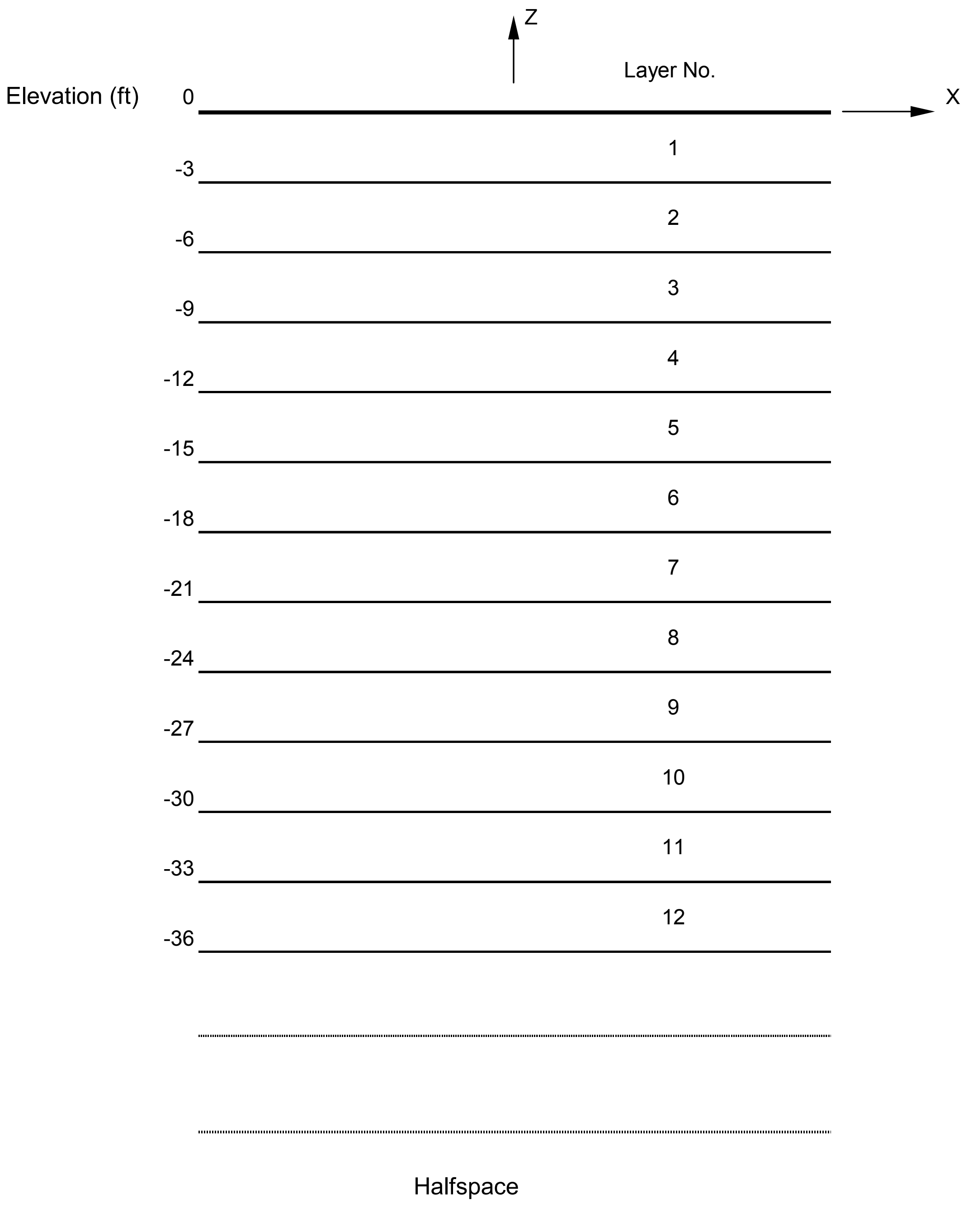
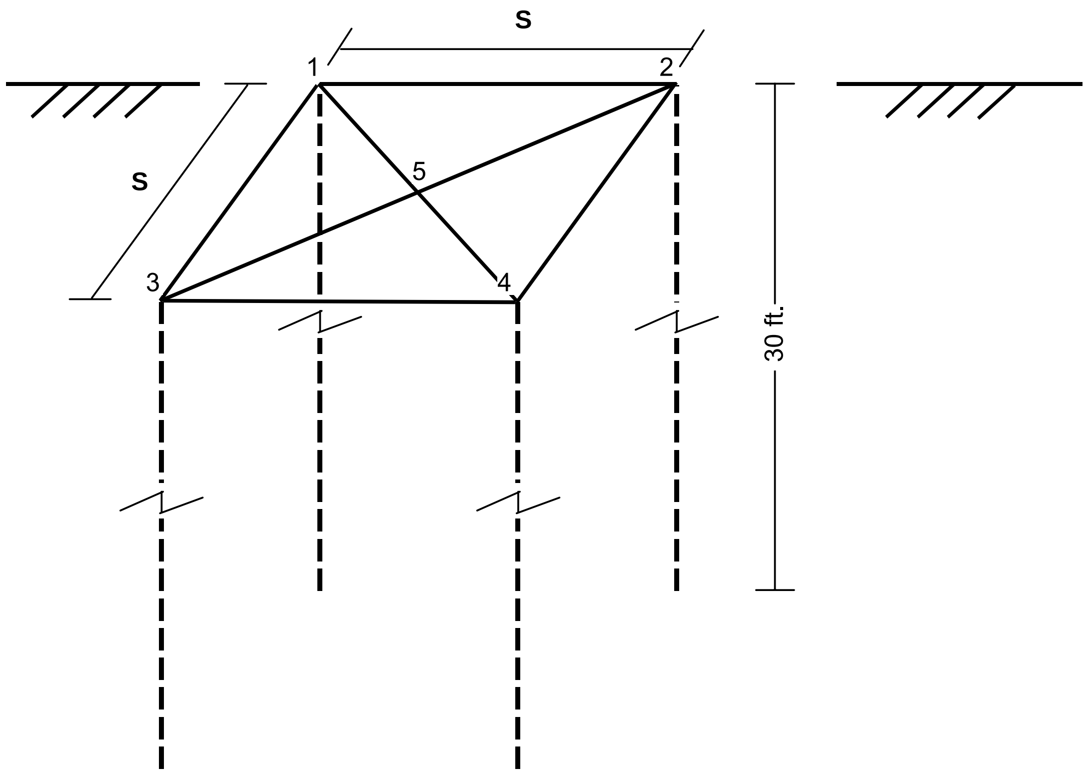


Figure 4.6–45. SASSI Soil Profile Model



D = 2 ft.

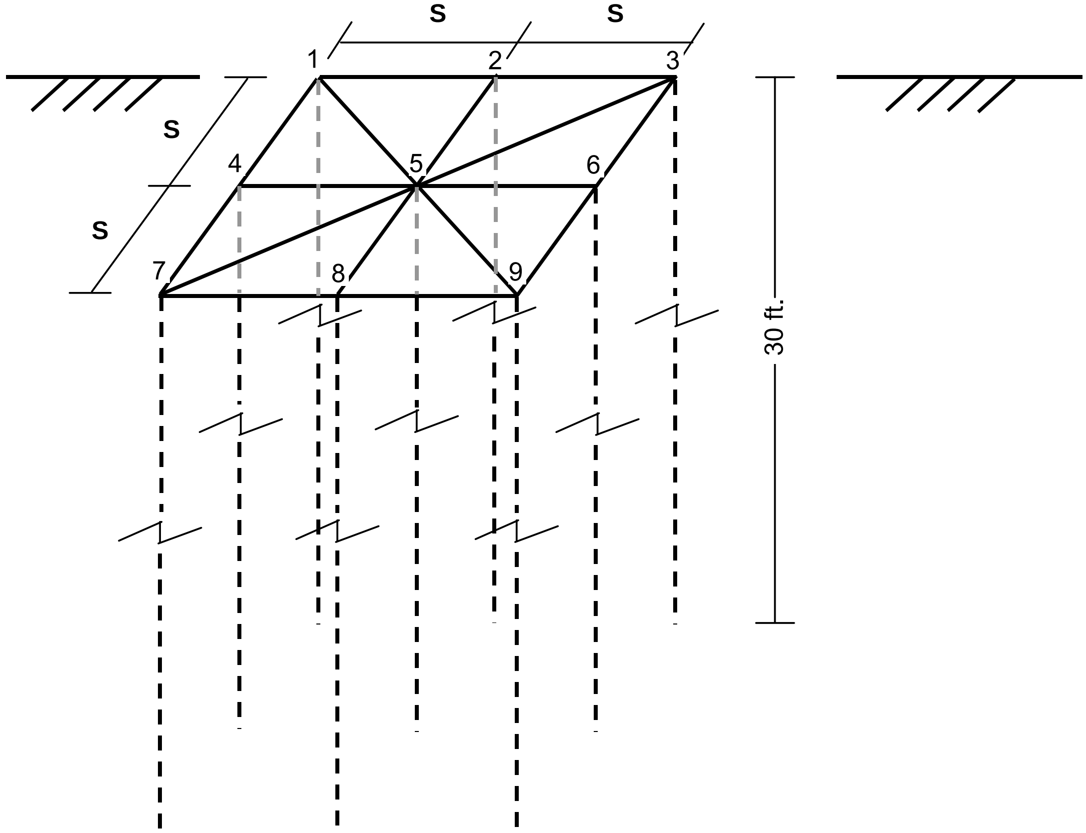
S = 4, 10, 20 ft.

The characteristics of piles are modeled in pile-impedance

All nodes on the ground surface are connected with rigid beams to model the pile cap

Unit horizontal and vertical harmonic forces are applied on Node 5

Figure 4.6–46. SASSI Analysis Model for Case 1: 2x2 Pile Groups



D = 2 ft.

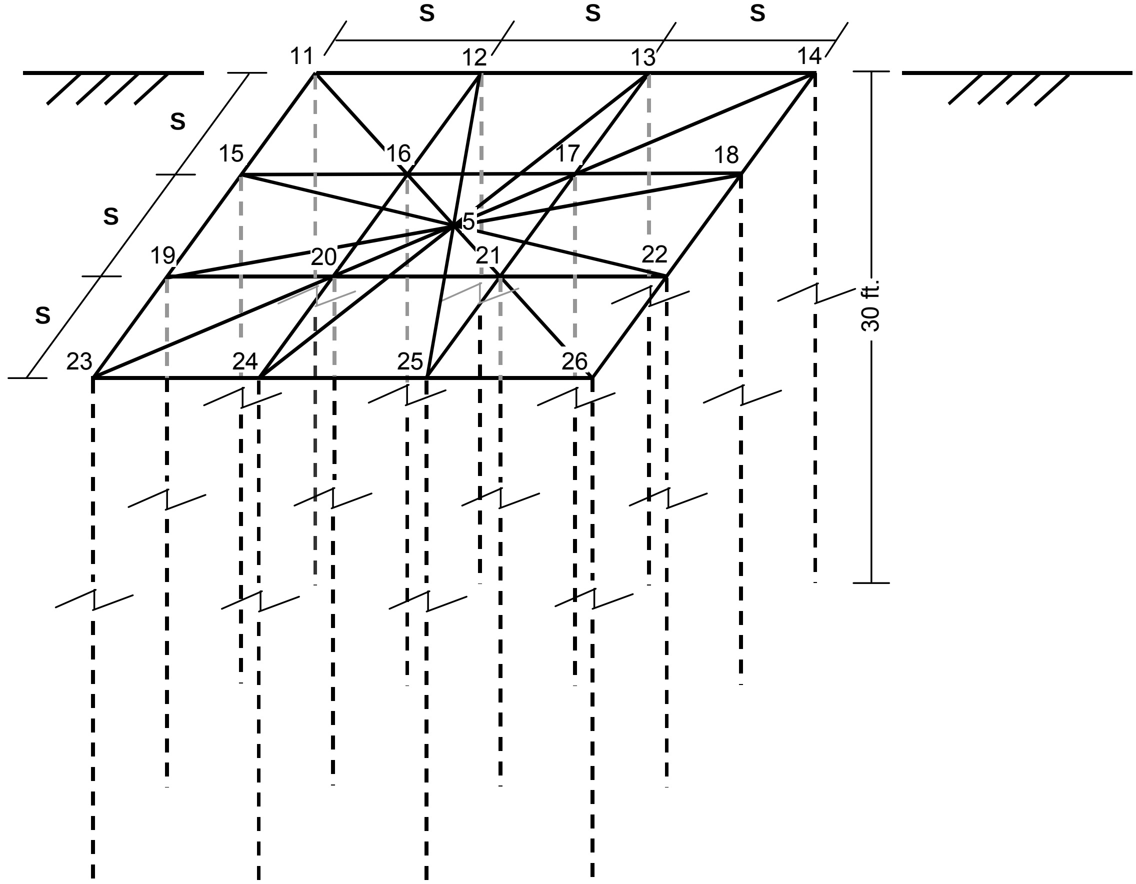
S = 4, 10, 20 ft.

The characteristics of piles are modeled in pile-impedance

All nodes on the ground surface are connected with rigid beams to model the pile cap

Unit horizontal and vertical harmonic forces are applied on Node 5

Figure 4.6–47. SASSI Analysis Model for Case 2: 3x3 Pile Groups



D = 2 ft.

S = 4, 10, 20 ft.

The characteristics of piles are modeled in pile-impedance

All nodes on the ground surface are connected with rigid beams to model the pile cap

Unit horizontal and vertical harmonic forces are applied on Node 5

Figure 4.6–48. SASSI Analysis Model for Case 3: 4x4 Pile Groups

Table 4.6-14. Frequencies of SASSI Analysis

Frequency Number NF	Frequency (Hz)	Dimensionless Frequency a_0
10	0.10	0.004
477	4.77	0.200
955	9.55	0.400
1432	14.32	0.600
1910	19.10	0.800
2387	23.87	1.000

4.6.4.3 Analysis Cases

As described above, the SASSI runs for this problem consist of three (3) cases: Case 1 is for 2x2 pile groups, with pile spacing S/D = 2, 5, and 10. Case 2 is for 3x3 pile groups, with pile spacing S/D = 2, 5, and 10. And Case 3 is for 4x4 pile groups, with pile spacing S/D = 2, 5 and 10, respectively.

In each case, a unit harmonic load, in separately horizontal and vertical directions, is applied at the center of the pile cap, Node 5. Responses at the same node are computed and then processed.

The SASSI run sequences and input/output files for this problem are shown in Table 4.6-15.

Table 4.6-15. Input/Output Files of Example Problem No. 4

SASSI MODULE	CASE 1 OF ANALYSIS				CASE 2 OF ANALYSIS				CASE 3 OF ANALYSIS			
	INPUT	OUTPUT	TAPEIN	TAPEOUT	INPUT	OUTPUT	TAPEIN	TAPEOUT	INPUT	OUTPUT	TAPEIN	TAPEOUT
SITE	E4C1.SD	E4C1.SO		E4C1.T2	E4C2.SD	E4C2.SO		E4C2.T2	E4C3.SD	E4C3.SO		E4C3.T2
SPILE	E4C1.SPD	E4C1.SPO	E4C1.T2	E4C1.T31	E4C2.SPD	E4C2.SPO	E4C2.T2	E4C2.T31	E4C3.SPD	E4C3.SPO	E4C3.T2	E4C3.T31
HOUSE	E4C1A.HD	E4C1A.HO		E4C1A.T4	E4C2A.HD	E4C2A.HO		E4C2A.T4	E4C3A.HD	E4C3A.HO		E4C3A.T4
HOUSE	E4C1B.HD	E4C1B.HO		E4C1B.T4	E4C2B.HD	E4C2B.HO		E4C2B.T4	E4C3B.HD	E4C3B.HO		E4C3B.T4
HOUSE	E4C1C.HD	E4C1C.HO		E4C1C.T4	E4C2C.HD	E4C2C.HO		E4C2C.T4	E4C3C.HD	E4C3C.HO		E4C3C.T4
MOTOR	E4C1H.MD	E4C1H.MO		E4C1H.T9	E4C2H.MD	E4C2H.MO		E4C2H.T9	E4C3H.MD	E4C3H.MO		E4C3H.T9
MOTOR	E4C1V.MD	E4C1V.MO		E4C1V.T9	E4C2V.MD	E4C2V.MO		E4C2V.T9	E4C3V.MD	E4C3V.MO		E4C3V.T9
ANALYS	E4C1.AD	E4C1AH.AO	E4C1.T31 E4C1A.T4 E4C1H.T9	E4C1AH.T8	E4C2.AD	E4C2AH.AO	E4C2.T31 E4C2A.T4 E4C2H.T9	E4C2AH.T8	E4C3.AD	E4C3AH.AO	E4C3.T31 E4C3A.T4 E4C3H.T9	E4C3AH.T8
ANALYS	E4C1.AD	E4C1AV.AO	E4C1A.T4 E4C1V.T9	E4C1AV.T8	E4C2.AD	E4C2AV.AO	E4C2A.T4 E4C2V.T9	E4C2AV.T8	E4C3.AD	E4C3AV.AO	E4C3A.T4 E4C3V.T9	E4C3AV.T8
ANALYS	E4C1.AD	E4C1BH.AO	E4C1B.T4 E4C1H.T9	E4C1BH.T8	E4C2.AD	E4C2BH.AO	E4C2B.T4 E4C2H.T9	E4C2BH.T8	E4C3.AD	E4C3BH.AO	E4C3B.T4 E4C3H.T9	E4C3BH.T8
ANALSY	E4C1.AD	E4C1BV.AO	E4C1B.T4 E4C1V.T9	E4C1BV.T8	E4C2.AD	E4C2BV.AO	E4C2B.T4 E4C2V.T9	E4C2BV.T8	E4C3.AD	E4C3BV.AO	E4C3B.T4 E4C3V.T9	E4C3BV.T8
ANALYS	E4C1.AD	E4C1CH.AO	E4C1C.T4 E4C1H.T9	E4C1CH.T8	E4C2.AD	E4C2CH.AO	E4C2C.T4 E4C2H.T9	E4C2CH.T8	E4C3.AD	E4C3CH.AO	E4C3C.T4 E4C3H.T9	E4C3CH.T8
ANALYS	E4C1.AD	E4C1CV.AO	E4C1C.T4 E4C1V.T9	E4C1CV.T8	E4C2.AD	E4C2CV.AO	E4C2C.T4 E4C2V.T9	E4C2CV.T8	E4C3.AD	E4C3CV.AO	E4C3C.T4 E4C3V.T9	E4C3CV.T8

4.6.4.4 Results and Comparison

The force-displacement relationship at the pile head can in general term be expressed in the following form:

$$P_x = K_{xx}^{st}\left(K_{xx} + ia_o C_{xx}\right)\left(\mathrm{Re}U_{xx} + i\mathrm{Im}U_{xx}\right) \quad (4.6\text{-}9)$$

$$P_z = K_{zz}^{st}\left(K_{zz} + ia_o C_{zz}\right)\left(\mathrm{Re}U_{zz} + i\mathrm{Im}U_{zz}\right) \quad (4.6\text{-}10)$$

where P_x, P_z is the force acting on the pile head in x- and z-directions, and K_{xx}^{st}, K_{zz}^{st} are the static stiffness in the xx- and zz-directions, respectively; a_0 is the dimensionless frequency as defined in Eq (4.6-8), K_{xx}, K_{zz} are the normalized dynamic stiffness terms, and C_{xx}, C_{zz} are the normalized damping terms. U_{xx} and U_{zz} are the computed responses at the pile head. All the terms in the above equations except the static stiffness terms are functions of frequency.

For the particular case of unit load, $P_x = 1$ and $P_z = 1$, the normalized stiffness and damping terms from the above expressions can be reduced to

$$K_{ii} = \frac{1}{K_{ii}^{st}}\frac{\mathrm{Re}U_{ii}}{|U_{ii}|^2}, \qquad ii = xx,\ zz \quad (4.6\text{-}11)$$

$$C_{ii} = \frac{-1}{a_0 K_{ii}^{st}}\frac{\mathrm{Im}U_{ii}}{|U_{ii}|^2}, \qquad ii = xx,\ zz \quad (4.6\text{-}12)$$

where K_{xx}^{st}, K_{zz}^{st} are the stiffness of the foundation calculated at a_0 = 0.

For pile groups, the responses of K_{ii}, C_{ii}, ii = xx, zz, are normalized by the response of single pile:

$$K_{ii}^{g} = \frac{1}{NK_{ii}^{st,s}} \frac{\mathrm{Re} U_{ii}^{g}}{|U_{ii}^{g}|^2}, \qquad \text{ii = xx, zz} \qquad (4.6\text{-}13)$$

$$C_{ii}^{g} = \frac{-1}{a_0 NK_{ii}^{st,s}} \frac{\mathrm{Im} U_{ii}^{g}}{|U_{ii}^{g}|^2}, \qquad \text{ii = xx, zz} \qquad (4.6\text{-}14)$$

where the superscript g indicates the term for the pile group, and the superscript s indicates the term for a single pile. N is the total number of piles in the group.

The computed results in all four cases are computed and normalized according to Eqs (4.6-9) through (4.6-14) and are compared with the results reported by Kaynia (10). Figure 4.6-49 shows the comparison for Case 1, the 2x2 pile groups. Figure 4.6-50 shows the comparison for Case 2, the 3x3 pile groups. Figure 4.6-51 shows the comparison for Case 3, the 4x4 pile groups. It can be observed from the comparisons that in all cases good to very good agreement between the corresponding results are reached.

4.6.4.5 Conclusion

The good agreement between the SASSI results and the results reported by Kaynia validates the analysis capability of SASSI for pile groups with the pile impedance method.

2 x 2 Pile Groups, Fixed-Head Piles, Soft Soil Medium
Horizontal and Vertical Dynamic Stiffnesses

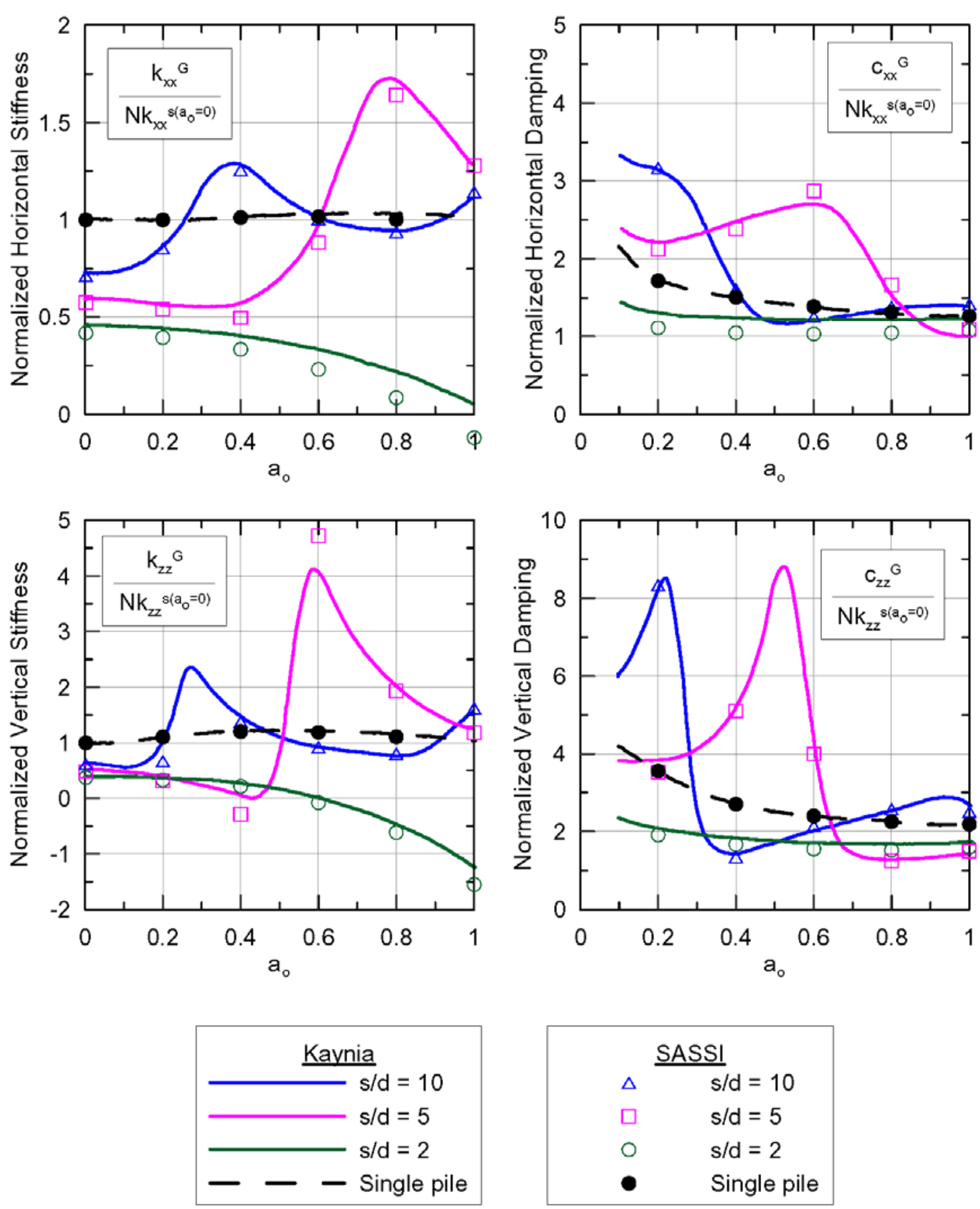


Figure 4.6–49. SASSI Analysis Results for 2x2 Pile Groups

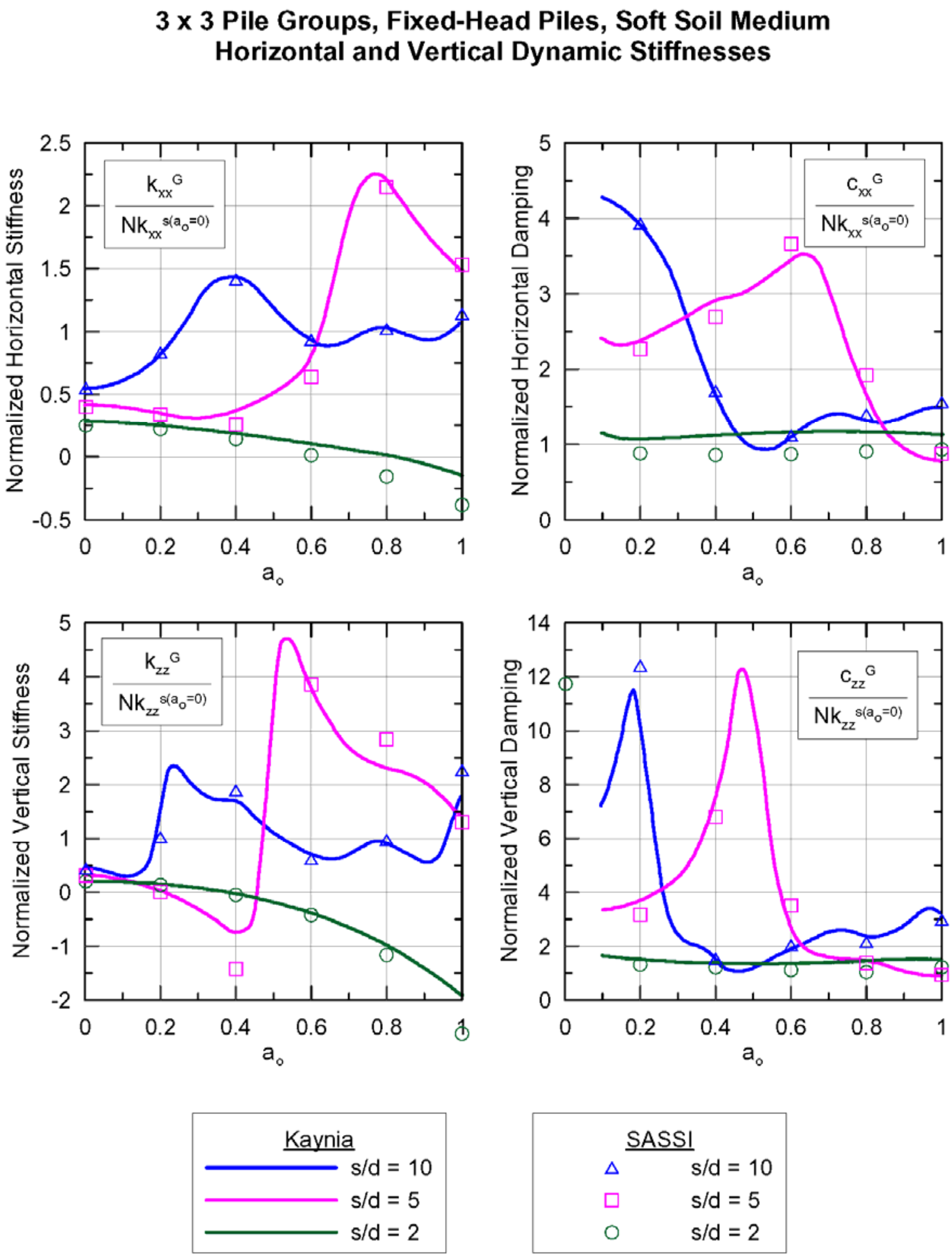


Figure 4.6–50. SASSI Analysis Results for 3x3 Pile Groups

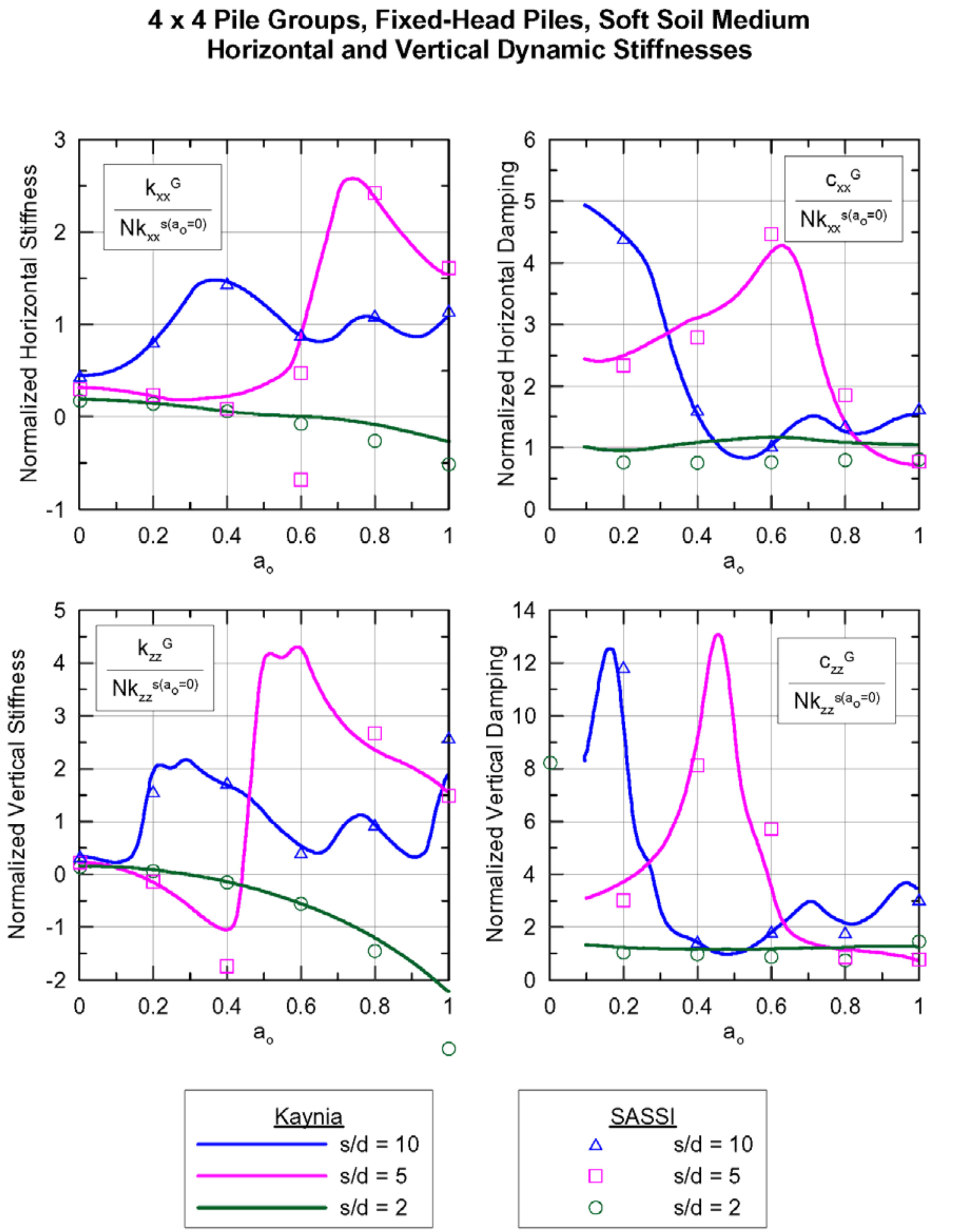


Figure 4.6–51. SASSI Analysis Results for 4x4 Pile Groups

CHAPTER 5
INPUT GUIDE TO SASSI PROGRAM MODULES

5.1 COMPUTER PROGRAM FOR SASSI MODULES

The current version of the SASSI program consists of the following computer programs:

A. SITE
B. POINT2 and POINT3
C. SPILE
D. HOUSE
E. INCOH
F. MOTOR
G. ANALYS
H. COMBIN
I. MOTION
J. STRESS
K. PLOT

5.2 PROGRAM MODULES

5.2.1 SITE

The program module SITE has two basic functions:

1. Form and solve transmitting boundary eigenvalue problem - Mode 1

The program reads the soil layer properties and for each specified frequency forms the transmitting boundary submatrices for Rayleigh and Love wave cases. Then it solves the two eigenvalue problems:

$$k^2A_2 + kA_1 + A_0 - \omega^2M = 0 \qquad (5.2\text{-}1)$$

$$k^2A_2 + A_0 - \omega^2 = 0 \;, \qquad (5.2\text{-}2)$$

from which the eigenvalues (k) and eigenvectors are obtained. The results are then written on Tape 2.

The halfspace condition is also simulated at this stage. The program automatically generates a specified number of sublayers whose thicknesses vary with frequency attached to viscous dashpots at the base. The generated sublayers and dashpots are then added to the fixed top layers. Such provision requires some modifications in the above equations which are carried out in this program.

Tape 2 provides the information needed to run Mode 2 in the program module SITE as well as to compute for the transmitting boundary In the program module POINT. Thus, this tape always has to be generated.

Since the eigenvalue problems to be solved for an arbitrary three-dimensional horizontally layered site are the same as those to be solved for a plane strain model, the information on Tape 2 can be used for both two- and three-dimensional cases as well.

In order to run this mode, all the input cards from A.1 to A.6 of this section must be supplied. If the execution is to be stopped after Mode 1, the card A.6 must be entered as a blank card.

2. Solve the site response problem - Mode 2

The program recovers the soil layer properties and the eigensolutions for Rayleigh and Love wave cases from Tape 2. Then, according to the existence of each wave type, the program computes the mode shapes and wave numbers for each wave type in the coordinate system defined in the program module SITE. Then, once the composition of the wave types causing the seismic environment and the nature of the control motion is known, the program will scale and superimpose the results of all the wave types. These results are then stored on Tape 1, which is used later for seismic analysis. Thus, this tape will not be generated for foundation vibration analysis.

If the seismic environment is the same for a two- and three-dimensional case, the information on Tape 1 can be used for both problems.

In order to run Mode I and Mode 2 together, all the input cards from A.1 to A.10 of this section must be supplied. However, to restart the program in Mode 2 alone, only the Input cards from A.6 to A.10 of this section are needed.

A.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
		(1)	= 1, complete solution for MODE 1
		(2)	= -1, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If the program is to be restarted in Mode 2 using Tape 2 as input, then skip all the input cards A.1 through A.5 and start from Section **A.6**.

(2) If **NOPT** < 0, most of the calculation required during execution of Mode 1 will be bypassed.

A.2 MASTER CONTROL CARD (3I5)

Columns	Variable	Notes	Entry
1-5	**NTL**	(1)	Number of soil layers (maximum = 100)
6-10	**NF**		Total number of frequencies of analysis (maximum = 100)
15	**LSUB**	(2)	= 0, no simulation of halfspace
			> 3. number of layers generated to simulate halfspace (maximum = 20)

Notes:

(1) All the soil layers must be labeled with integer numbers starting from I at the surface to **NTL**. It does not include halfspace or layers generated to simulate halfspace.

(2) If **LSUB** = 0, the soil profile will be assumed on rigid base. Otherwise, **LSUB** sublayers whose thicknesses vary with frequency are generated to simulate halfspace. Also, the program will add viscous boundary to account for radiation damping in the halfspace through the lower boundary. Using **LSUB** = 10 is recommended for the case of halfspace.

A.3 SYSTEM OF UNITS CARD (F10.O)

Columns	Variable	Notes	Entry
1-10	**GRAV**	(1)	Acceleration of gravity

Notes:

(1) To be used for computation of mass matrix.

A.4 SOIL LAYER DATA CARDS

A.4.1 TOP LAYERS (I5, 6F10.0)

Columns	Variable	Notes	Entry
1-5	**N**	(1)	Layer number
6-15	**H**		Layer thickness
16-25	**W**		Unit weight
26-35	**VS**		S-wave velocity
36-45	**VP**		P-wave velocity
46-55	**DS**		S-wave associated damping ratio
56-65	**DP**		P-wave associated damping ratio

A.4.2 HALFSPACE (15X, 5F10.0)

Columns	Variable	Notes	Entry
1-15			Blank
16-25	**WH**	(2)	Unit weight of halfspace
26-35	**VSH**		S-wave velocity
36-45	**VPH**		P-wave velocity
46-55	**DSH**		S-wave associated damping ratio
56-65	**DPH**		P-wave associated damping ratio

Notes:

(1) Total of **NTL** cards must be given and all soil layer data must be defined in soil layer number sequence.

(2) One card must be given to define properties of halfspace. For example, if a hard layer exists at depth or **LSUB**=0, then properties of competent rock should be supplied; otherwise, properties of existing materials at depth must be provided.

A.5 FREQUENCY DATA CARDS

A.5.1 FREQUENCY CONTROL CARD (2F10.0, I15)

Columns	Variable	Notes	Entry
1-10	**DF**	(1)	Frequency step (HZ)
11-20	**DT**	(1)	Time step of control motion (sec)
21-25	**NFFT**	(1)	Number of values to be used in Fourier transform of control motion. Must be power of 2.

A.5.2 FREQUENCY NUMBER CARDS (16I5)

Columns	Variable	Notes	Entry
1-5	**NFR(1)**	(2)	Frequency no. 1
6-10	**NFR(2)**		Frequency no. 2
11-15	**NFR(3)**		Frequency no. 3
•	•		•
•	•		•
•	•		•

Notes:

(1) Case A - Deterministic Analysis

DT and **NFFT** for the selected time history of the control motion must be given, and the frequency step may be left blank. The program will compute the frequency step as follows:

$$DF = \frac{1}{(NFFT * DT)}$$

This frequency step may then be used to set up frequency numbers in Section **A.5.2**.

Case B - Probabilistic (or Single Harmonic) Analysis

DF must be given and may be directly used to set up frequency numbers in Section **A.5.2**. In this case, **DT** and **NFFT** are never used and therefore may be left blank.

(2) The total of **NF** frequency numbers must be given. All the frequency numbers must be positive nonzero integer numbers. The program will automatically reorder the input frequency numbers in ascending order and will stop if two or more equal-frequency numbers are detected. Frequencies f_i, for which solutions are obtained, are defined as follows:

$$f_i = NFR(i) * DF$$

The highest frequency of analysis is then equal to the highest frequency number multiplied by the frequency step.

A.6 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
		(1)	= 0, stop, no more data
		(2)	= 2, complete solution for MODE 2
		(3)	= -1, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) **NOPT** = 0 terminates the execution of the program at the end of Mode 1, and no data are required after this card.

(2) If **NOPT** = 2, continue providing the remaining data cards. However, if this is the restart of Mode 2 with Tape 2 as input, then the input cards A.1 to A.5 are not required.

(3) If **NOPT** < 0, most of the calculations required during normal execution of Mode 2 are bypassed. Mode I must be executed before the data check can be done in Mode 2.

A.7 WAVE FIELD DATA CARDS

A.7.1 WAVE FIELD TYPE CARD (2I5)

Columns	Variable	Notes	Entry
5	**IWTYP**		= 1, combination of P-, SV-, and R-waves
			= 2, combination of SH- and L-waves

A.7.2 WAVE FIELD TYPE 1 CARD (3I5, 2F10.0)

Skip this section if **IWTYP** = 2

Columns	Variable	Notes	Entry
5	**IRWAVE**	(1)	= 0, no R-wave field
			= 1, R-wave field (shortest wavelength method)
			= 2, R-wave field (least decay method)
10	**IVWAVE**	(1)	= 0, no SV-wave field
			= 1, SV-wave field
15	**IPWAVE**	(1)	= 0, no P-wave field
			= 1, P-wave field
16-25	**ANGS**	(2)	Incident angle of SV-wave (degree)
25-35	**ANGP**	(2)	Incident angle of P-wave (degree)

A.7.3 WAVE FIELD TYPE I CARD (2I5, F10.0)

Skip this section if **IWTYP** = 1

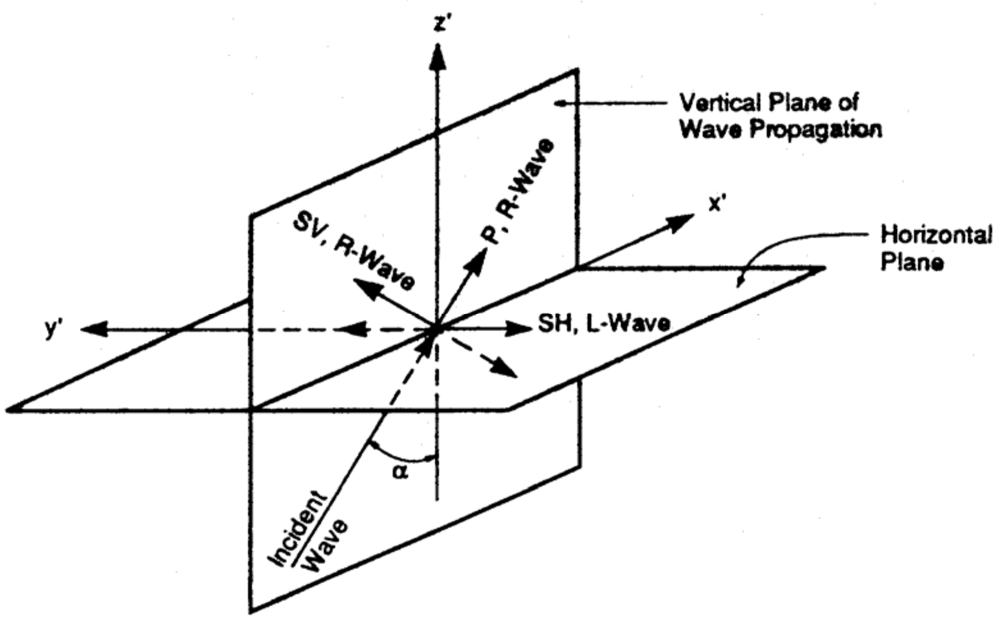
Columns	Variable	Notes	Entry
5	**IRWAVE**	(1)	= 0, no L-wave field
			= 1, L-wave field
10	**IHWAVE**	(1)	= 0, no SH-wave field
			= 1, SH-wave field
11-20	**ANGH**	(2)	Incident angle of SH-wave (degree)

Notes:

(1) The seismic environment may be assumed to consist of one single wave type or several wave types. The basic wave types are P-waves and S-waves, which are also called body waves. When these waves impinge on the ground surface or layer interfaces, surface waves may be generated which include R-waves and L-waves.

P-waves involve particle motions in the direction of wave propagation. S-waves involve particle motions perpendicular to the direction of wave propagation. S-wave motions in the vertical plane are called SV-waves. Horizontal S-waves are called SH-waves. R-waves involve horizontally propagating elliptical motions in the vertical plane and L-waves consist of horizontal motions perpendicular to the horizontal direction of wave propagation.

With the above definitions, a coordinate system in program SITE has been set up such that P-waves, SV-waves, and R-waves involve particle displacements in x'z'-plane while SH-waves and L-waves involve particle displacements along y'-axis. Therefore z' is always vertical up, x' is in the vertical plane of wave propagation, and y' is perpendicular to x' and z' and following the right-hand rule.



(2) Incident angle is defined as the angle between direction of propagation and z' axis shown as α in the above figure. For vertically propagating body waves, this angle is zero.

A.8 CONTROL MOTION CARD (4X, A1, 2I5)

Columns	Variable	Notes	Entry
5	**KCOMP**	(1)	= X, control motion in X' direction
			= Y, control motion in Y' direction
			= Z, control motion in Z' direction
6-10	**NLCP**	(2)	Layer number of control point
11-15	**NFCP**	(3)	Number of frequencies used to define ratio curve of wave participations in control motion (≥2)

Notes:

(1) Transformation of x' y' z' coordinate to final xyz coordinate of soil structure system will be done in the program module ANALYS.

(2) Control point is defined as the point where the control motion is specified. It will be located at the top of the given layer number, e.g., NLXZ = 1 for control point at the surface.

(3) In the case of seismic environment composed of two or more wave types, the ratio of participation of each wave type must be given. This ratio in general may be frequency-dependent and is defined at a number of selected discrete frequencies for each wave type. These frequencies must cover the frequency range of analysis. The ratio values for intermediate frequencies will be obtained by simple interpolation and therefore need not be given at exact frequencies for which complete solution is required.

In the case of seismic environments consisting of one simple wave type, two frequencies (one in the beginning and the other at the end of the frequency range of analysis) with assigned ratio value of 1 are enough to define the ratio curve.

Also note that all the ratio values are positive decimal numbers less than or equal to 1, and summation of the ratio values of all the participating wave types at any frequencies must be 1.

A.9 WAVE COMPOSITION OF CONTROL MOTION ON X' Z' -PLANE

Skip this section if **IWTYP** = 2.

A.9.1 FREQUENCY CARDS (16I5)

Columns	Variable	Notes	Entry
1-5	**NFXZ(1)**	(1)	Frequency no. 1 to define ratio curve
6-10	**NFXZ(2)**		Frequency no. 2 to define ratio curve
•	•		•
•	•		•
•	•		•

A.9.2 R-WAVE RATIO CARDS (8F10.0)

Skip if no R-wave field (**IRWAVE** = 0).

Columns	Variable	Notes	Entry
1-10	**XZR(1)**	(1)	R-wave ratio at frequency no. 1
11-20	**XZR(2)**		R-wave ratio at frequency no. 2
•	•		•
•	•		•
•	•		•

A.9.3 SV-WAVE RATIO CARDS (8F10.0)

Skip if no SV-wave field (**IVWAVE** = 0).

Columns	Variable	Notes	Entry
1-10	**XZS(1)**	(1)	SV-wave ratio at frequency no. 1
11-20	**XZS(2)**		SV-wave ratio at frequency no. 2
•	•		•
•	•		•
•	•		•

A.9.4 P-WAVE RATIO CARDS (8F10.0)

Skip if no P-wave field (**IVWAVE** = 0).

Columns	Variable	Notes	Entry
1-10	**XZP(1)**	(1)	P-wave ratio at frequency no. 1
11-20	**XZP(2)**		P-wave ratio at frequency no. 2
•	•		•
•	•		•
•	•		•

Notes:

(1) Refer to note 3 in Section **A.8**.

A.10 WAVE COMPOSITION OF CONTROL MOTION ALONG Y'-AXIS (8F10.0)

Skip this section if **IWTYP** = 1.

Columns	Variable	Notes	Entry
1-5	**NFY(1)**	(1)	Frequency no. 1 to define ratio curve
6-10	**NFY(2)**		Frequency no. 2 to define ratio curve
•	•		•
•	•		•
•	•		•

A.10.2 L-WAVE RATIO CARDS (8F10.0)

Skip if no L-wave field (**ILWAVE** = 0).

Columns	Variable	Notes	Entry
1-10	**YL(1)**	(1)	L-wave ratio at frequency no. 1
11-20	**YL(2)**		L-wave ratio at frequency no. 2
•	•		•
•	•		•
•	•		•

A.10.3 SH-WAVE RATIO CARDS (8F10.0)

Skip if no SH-wave field (**IHWAVE** = 0).

Columns	Variable	Notes	Entry
1-10	**YS(1)**	(1)	SH-wave ratio at frequency no. 1
11-20	**YS(2)**		SH-wave ratio at frequency no. 2
•	•		•
•	•		•
•	•		•

Notes:

(1) Refer to note 3 in Section **A.8**.

A.11 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop, no more data

Notes:

(1) Must be the last data card

5.2.2 POINT

The program module **POINT** recovers the soil layer properties and the eigen solutions for the Rayleigh and Love wave cases from Tape 2. Then, for each frequency specified in the program module **SITE** and for given radius of the central zone, the program solves for the point loads applied at the surface of the layered system and on the layer interfaces below the ground surface. The maximum number of layers that the structure is embedded into the ground determines how deep the point loads are applied below the ground surface.

The results which are obtained in the local coordinate system of the program module POINT are saved on Tape 3 and are later used to compute the flexibility matrix of the interaction nodes in the program module ANALYS.

For two-dimensional problems the program module POINT2 and for three-dimensional problems the program module POINT3 must be executed.

B.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	<0, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If NOPT < 0, most of the calculations required during normal execution are bypassed.

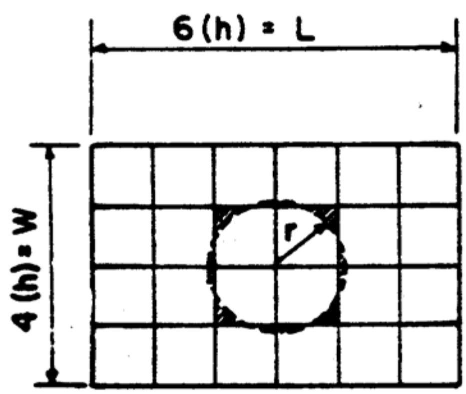
B.2 GENERAL INFORMATION CARD (I5, F10.0)

Columns	Variable	Notes	Entry
1-5	**LSTFCE**	(1)	Last layer number in the near field zone
6-15	**RADIUS**	(2)	Radius of the central zone in the point load solution. Must be positive and nonzero.

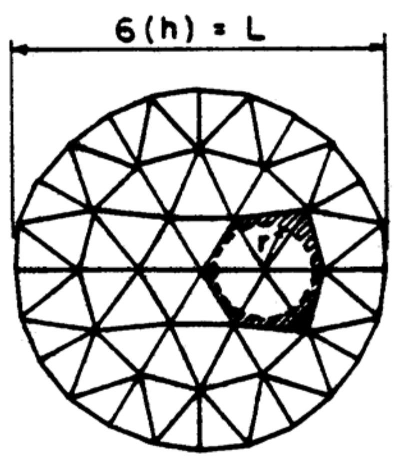
Notes:

(1) This parameter is the maximum number of layers in the ground that the structure (including the irregular soil zone) is embedded into. The smaller this number, the less information need be passed on to Tape 3 and therefore less computation and storage will be needed to form the flexibility matrix in the program module **ANALYS**. However, this number must be large enough to ensure that the excavated soil region will not extend deeper than the specified layer number. For surface structures with no assumed irregular soil zone, let **LSTFCE** = 0.

(2) The value of this parameter depends on the geometry of foundation discretization by finite element method. For two relatively uniform meshes, this value is given below.

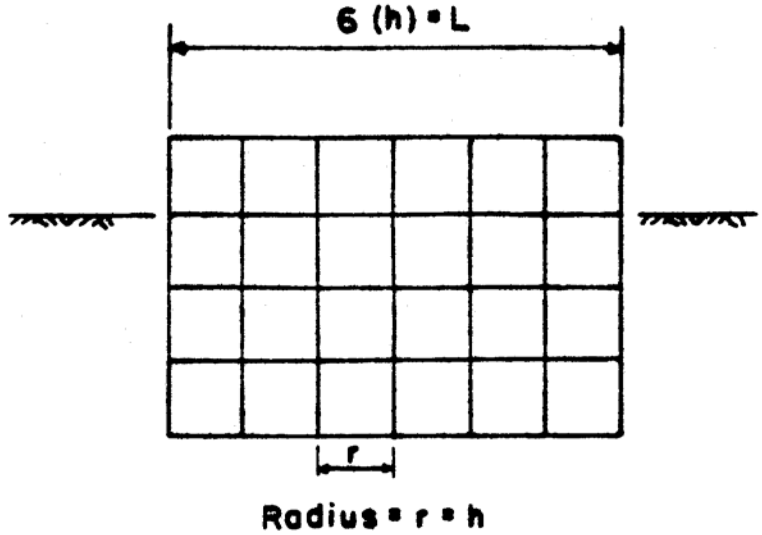


Radius = r = 0.90 h



Radius = r = 0.85 h

An average value can be obtained for uniform meshes, but using uniform meshes as much as possible is recommended. For 2-D cases, the radius of the central zone shall be selected as follows:



For 2-D cases, the results are not sensitive to changes in the assumed radius for the central zone as long as this radius is of the same order as the dimension of the finite elements in the interaction volume (refer to verification manual).

B.3 OPERATION MODE CARD (I5)

Columns	**Variable**	**Notes**	**Entry**
5	**NOPT**	(1)	Operation mode
			= 0, stop, no more data.

Notes:

(1) Must be the last data card.

5.2.3 SPILE

The program module **SPILE** computes for the vertical and horizontal impedance functions for one or several vertical single piles. These functions are used in the program **ANALYS** to compute the pile group impedance functions considering pile-soil-pile interaction effects. In the module **SPILE**, the soil layer properties and the eigensolutions for the Rayleigh and Love wave cases saved on Tape 2 by module **SITE** are first recovered. Then, for each frequency specified in the module **SITE** and for given properties and radii of the single pile(s), the program solves for the impedance functions (displacements due to horizontal and vertical unit load applied on the pile head) in terms of stiffness and damping coefficients for each single pile specified. The pile head may be assumed to be fixed or free to rotate, and may be located at or below the ground surface. If pile group impedance functions are not needed, there is no need to run **ANALYSIS**.

The computed impedance functions in local coordinate system are written on Tape 31 (the file **SASSI.T31**) and will be used in the program module **ANALYS** for computing the impedance of the pile groups.

The pile impedance functions calculated in this module are applicable to three-dimensional problems only. It should be noted that this approach develops only the impedance functions for single piles and pile groups and can not be used for seismic analysis. The direct impact loading may be considered in this method.

C.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If NOPT < 0, most of the calculations required during normal execution are bypassed.

C.2 GENERAL INFORMATION CARD (2I5)

Columns	Variable	Notes	Entry
1-5	**LSTFCE**	(1)	last layer number in the near field zone
6-0	**NTPILE**	(2)	Total number of single piles to be considered.

Notes:

(1) This parameter is the maximum number of layers in the ground that the structure (including the irregular soil zone) is embedded into. The smaller is the number, the less information needs be passed on to Tape 31. However, this number must be large enough to ensure that the excavated soil region will not extend deeper than the specified layer number. For surface structures on pile foundation, let **LSTFCE** = 0.

(2) The value of this parameter depends on the different types of piles in terms of length, properties, radius and pile head boundary condition in the foundation. For foundations with one type of pile only, use 1 for this parameter.

C.3 PILE PROPERTY CARD (7I5, F10.0)

Repeat the following cards for **NTPILE** number of piles

Columns	Variable	Notes	Entry
1-5	**IPILE**	(1)	Pile type number
6-10	**LPILET**	(2)	Layer number at the pile head
11-15	**LPILEB**	(2)	Layer number at the pile tip
16-20	**IPROP**	(3)	Total number of pile properties
21-25	**ICODE**	(4)	Code to assign pile material properties =-1, input elastic modulus and Poisson's ratio =0, input constrained and shear moduli =1, input P- and S- wave velocities
26-30	**IROT**		= 0, free head pile = 1, fixed head pile
31-35	**KPRINT**	(4)	= 0, do not print displacement = 1, print displacement at pile head
36-40	**VSAO**	(5)	shear wave velocity to compute for normalized frequency ratio

Notes:

(1) This parameter identifies the pile type number and it varies from 1 to NTPILE

(2) These two variables determine the length of the pile

(3) Each pile may have several different material properties along its length. This parameter defines the number of different properties in one pile.

(4) Displacement results at the pile head used to compute the impedance functions are printed. It is suggested to suppress the print out to reduce output size.

(5) This parameter is used to compute the dimensionless frequency ratio Ao in order to report the impedance results and is not used in computing the impedance functions in **SPILE**. The ratio is Ao = 2*π*r*f/VSAO where r is the radius of the pile, f is the frequency of the analysis.

C.4 PILE PROPERTY CARD (F10.0)

Columns	Variable	Notes	Entry
1-10	**RADIUS**	(1)	Radius of the pile

Notes:

(1) For non-circular pile an equivalent radius should be provided

C.5 PILE PROPERTY CARD (16I5)

Columns	Variable	Notes	Entry
Repeat this card for IPROP properties			
1-5	**IDPROP(1)**	(1)	Property ID for element 1
6-10	**IDPROP(2)**	(1)	Property ID for element 2
			Repeat as many as needed

Notes:

(1) Each pile is modeled internally by a stack of axisymmetric elements between the two layers of LPILET and LPILEB. Total number of elements is LPILEB-LPILET. Each element may have its own elastic properties. This option is useful for piles that have varying properties with depth. Skip this card if IPROP = 1.

C.6 PILE MATERIAL PROPERTY CARD (I5, 6F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material identification number
6-15	**M(N)**		Elastic Modulus/constrained modulus/P-wave velocity
16-25	**G(N)**		Poisson’s ratio/shear Modulus/S-wave velocity
26-35	**W(N)**		Unit weigh of material
36-45	**DP(N)**		P-wave associated damping ratio
46-55	**DS(N)**		S-wave associated damping ratio
56-65	**ALPHA(N)**	(1)	Correction factor for vertical stiffness of the pile

Notes:

(1) In modeling the piles, the actual radius of the pile should be used. Since each pile is modeled by axisymmetric solid elements, the sectional properties of the pile, e.g., shear area, moment of inertia, etc. should be used to compute the equivalent elastic properties of the pile in horizontal direction. For vertical direction, the coefficient ALPHA is used to multiply by the stiffness in vertical direction when adjustment of vertical stiffness is needed. This ratio may be needed for pipe piles.

C.7 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop, no more data.

Notes:

(1) Must be the last data card.

5.2.4 HOUSE

The program module **HOUSE** is a standard finite element program that computes the basic frequency independent global mass and stiffness matrices, M and K, for the structure and excavated soil.

Two separate finite element models are constructed, one for the structure and the other for the excavated soil. The models share the same nodal points at all interaction nodes but are different elsewhere.

Each nodal point on the structure may have up to six displacement degrees of freedom. The program has the capability to delete or constrain to another node any number of the degrees of freedom specified by the user. Therefore, the user can control and optimize the size of the equations to be solved.

The element types available to model the structure include:

1. 3-D eight-node solid element (with option for inclusion of nine incompatible displacement modes)
2. 3-D beam element
3. Three to four-node quadrilateral thin shell element
4. 2-D four-node plane strain finite element
5. 3-D inter-pile element
6. 3-D spring element
7. 3-D stiffness/mass element
8. Three to four-node quadrilateral thick shell element

Other element types to be added to the program in the future to model the structure are:

1. 2-D inter-pile element
2. 2-D fluid element
3. 3-D fluid element

4. 1-D plane love wave element

The excavated soil zones are modeled by the following element types:

1. 3-D eight-node solid elements (without incompatible modes)
2. 2-D four-node plane strain finite element

The finite element models of the structure and the excavated soil must be selected in such a way that every interaction node below the ground should lie on a soil layer interface.

The program reads the nodal point input data, nodal types, soil layer properties, and element data for the structural and excavated soil elements, then forms the element mass and stiffness matrices for these elements which are later assembled into the corresponding global mass and stiffness matrices. These matrices are stored in compacted blocks in preparation for solution by the active column method later in the program module **ANALYS**. The results are written on Tape 4 (The file **SASSI.T4**). Certain portion of model data are also written on Tape 7 (**SASSI.T7**) for optional on-screen plotting, and for optional ground incoherency motion computation.

D.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If **NOPT** < 0, most of the calculations required during normal execution are bypassed.

A comment line can be inserted anywhere in the HOUSE input file by starting with the $ sign in column 1. The comment card is intended to provide additional information describing the various parts of input file. Comments will not printed in the HOUSE output.

D.2 CORE AND BLOCK STORAGE CONTROL CARD (3I10)

Columns	Variable	Notes	Entry
1-10	**MAXC**	(1)	Maximum number of columns to be assigned to each block (can be ignored if initiation run)
11-20	**MAXT**	(1)	Maximum number of terms to be assigned to each block (can be ignored if initiation run)
21-30	**MUSE**	(2)	Maximum decimal field length to be used for blank common (modified by program so that it will be integer multiple of 512); leave blank if program does not use dynamic core storage allocation

Notes:

(1) **MAXC** and **MAXT** are used to set block sizes for out-of-core operations. If either one is left blank during initiation runs, both are automatically set by the program. Enter **MAXC** and **MAXT** values which were used in the initiation runs to set the block sizes of stiffness matrices during restart runs with new superstructure. This will enable the program to recover the impedance matrices with compatible block storage. **MAXC** and **MAXT** are printed in the table labeled "BLOCK STORAGE INFORMATION" in the output.

(2) This parameter is used to set the blank common size in the program. If left blank, maximum available field length is used.

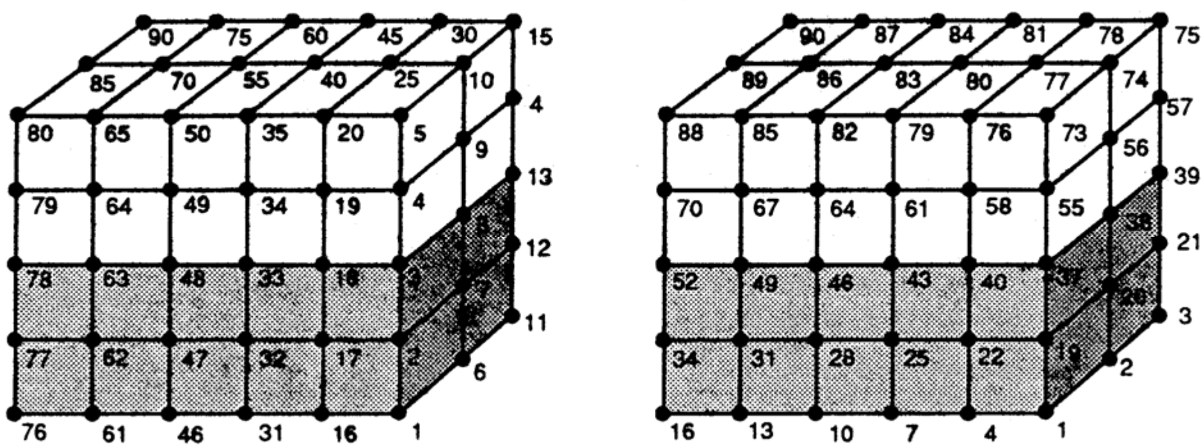
D.3 MASTER CONTROL CARD (9I5)

Columns	Variable	Notes	Entry
1-5	**NUMNP**	(1)	Total number of nodes in the system
6-10	**NUMGP**	(2)	Total number of nodes at/below ground surface which act as interaction nodes
11-15	**NUMEG**	(3)	Total number of different element groups
16-20	**NUML**	(4)	Total number of soil layers
21-25	**NUMLM**		Total number of nodes with lumped mass or inertia
30	**NSYMPL**	(5)	Total number of planes/line or symmetry or anti-symmetry (maximum of 2)
35	**NIMP**	(6)	Method of computing impedance matrix
			= 1, direct flexible volume method
			= 2, skin flexible volume method
			= 3, subtraction method (recommended)
40	**NDIM**		Dimension of analysis
			= 1, 1-D plane love wave (not available)
			= 2, 2-D plane strain

			= 3, general 3-D
45	**NTPILE**	(7)	= blank, pile impedance method is **<u>not</u>** used
			> 0, total number of pile types modeled in **SPILE** for pile impedance method

Notes:

(1) All the nodes in the structure must be labeled with integer numbers starting from 1 to **NUMNP**. The numbering is arbitrary. However, in order to minimize storage, computation, and block operations as well as to provide flexibility to restart the program with a new superstructure, it is recommended to number the nodes at or below the ground surface first, preferably layer by layer starting from the bottom. The figures shown below are examples of good and bad numbering systems.



(2) Count the total number of nodal points which reside below the ground elevation, including ground surfaces. In the above example **NUMGP** = 54 if direct or skin method is used. **NUMGP** = 46 if subtraction method is used.

(3) All the elements in the system must be grouped separately according to their type, e.g., beam, pile, etc. The elements in each group must be labeled sequentially from 1 to total number of elements in that group.

It is also possible to use more than one group for an element type. For example, all structural brick elements may be considered as one group and all excavated soil elements as another group.

(4) Only soil layers which reside in the basement of the structure need be specified. They will be used to compute properties of excavated soil elements. All the soil layers must be labeled sequentially from 1 at the top to **NUML** at the bottom. For surface structures with no irregular soil zone, let **NUML** = 0.

(5) Any combination of a maximum of two structural planes/line of symmetry which are symmetric or anti-symmetric relative to the loading can be considered in the program. In case of 3-D analysis, the planes of symmetry or anti-symmetry must be parallel to the xz or yz planes. In case of 1-D or 2-D analysis, the line of symmetry/anti-symmetry must be parallel to the z-axis. Also note that the name symmetry or anti-symmetry is used in relation to the loading. Leave blank if no advantage is to be taken of symmetry or anti-symmetry.

(6) Currently three methods are used for the computation of the impedance matrix, namely, the direct, the skin and the subtraction methods. If the direct method is to be used (**NIMP** = 1), all the nodes used to model excavated soil are interaction nodes. These nodes are entered in Section **D.8.1.2**. The direct method involves inversion of a full complex symmetric matrix as big as 3* (total number of interaction nodes).

If the skin method is to be used (**NIMP** = 2), all the interaction nodes are divided into three different types, namely, interface (nodes by boundary), intermediate (nodes connected directly to interface nodes), and internal (remaining nodes). These nodes are entered in Sections **D.8.2.2** through **D.8.2.4**. This method involves inversion of a full complex symmetric matrix only as big as 3* (total number of interface nodes) and therefore is more economical.

If the subtraction method is used (**NIMP** = 3), all the nodes on the boundary of the embedded part of the model (covering the skin of the embedded part) are the interaction nodes. Use of the subtraction method results in a significantly smaller set of interaction nodes without loss of any accuracy. This method is recommended as a primary method of impedance analysis. The interaction nodes are entered in Section **D.8.1.2**.

It should also be noted that all the nodes in the superstructure (or above the ground surface) are not connected to the soil are therefore not interaction nodes.

(7) Pile impedance is calculated in the program module **SPILE** and written on Tape 31 for use in **HOUSE**. If the pile impedance method is not used in the analysis, leave NTPILE blank.

D.4 SYSTEM OF UNITS CARD (F10.0)

Columns	Variable	Notes	Entry
1-10	**GRAV**	(1)	Acceleration of gravity

Notes:

(1) To be used for computation of mass matrix.

D.5 GROUND ELEVATION CARD (F10.0)

Columns	Variable	Notes	Entry
1-10	**ZSRFCE**	(1)	Z-coordinate of ground level

Notes:

(1) All the nodes below this elevation are assumed to be connected to the ground unless interaction nodes are specified (see Section **D.8**).

D.6 PLANE(S)/LINE OF SYMMETRY/ANTI-SYMMETRY CONTROL CARD (5I5)

Skip this section if **NSYMPL** = 0; otherwise provide **NSYMPL** cards.

Columns	Variable	Notes	Entry
1-5	**N**	(1)	Plane/line of symmetry/anti-symmetry number
5-10	**NPLTYP (N)**		Type of plane/line = 1 for symmetry = -1 for anti-symmetry
11-15	**NPT(1,N)**	(2)	First reference nodal point number on this plane/line
16-20	**NPT(2,N)**	(2)	Second reference nodal point number on this plane/line
21-25	**NPT(3,N)**	(2)	Third reference nodal point number on this plane (ignore for 1-D or 2- analysis)

Notes:

(1) Plane(s)/line of symmetry or anti-symmetry can be labeled in any order starting from 1 to **NSYMPL** (**NSYMPL** ≤2).

(2) Each plane (or line) of symmetry/anti-symmetry is defined by three (or two) reference nodal points. The three nodes which define a plane must not lie on a straight line.

D.7 NODAL POINT CARDS (I5, A1, I4, 5I5, 3F10.0, 2I5)

Columns	Variable	Notes	Entry
1-5	**N**	(1)	Nodal point-number
6	**NC**	(2)	Symbol describing coordinate system for this node
			= (blank), Cartesian (x,y,z)
			= C, cylindrical (R,θ,Z)
			= S, spherical (R,θ,φ)
7-10	**ID(N,1)**	(3)	x-translation B.C. and C.C code
11-15	**ID(N,2)**		y-translation B.C. and C.C code
16-20	**ID(N,3)**		z-translation B.C. and C.C code
21-25	**ID(N,4)**		xx-rotation B.C. and C.C code
26-30	**ID(N,5)**		yy-rotation B.C. and C.C code
31-35	**ID(N,6)**		zz-rotation B.C. and C.C code
36-45	**XORD(N)**		x-ordinate (R if cylindrical or spherical)
46-55	**YORD(N)**	(4)	y-ordinate (θ-degree if cylindrical or spherical); leave blank or 2-D analysis
56-65	**ZORD(N)**		z-translation (Z if cylindrical, φ-degree if spherical); Z-axis must always point upward

66-70	**KN**	(5)	Node number increment
71-75	**NPILE**	(6)	Pile impedance type specified at the node

Notes:

(1) A special cylindrical or spherical coordinate system is allowed for the global description of nodal point locations. If a "c" or "s" is entered in column 1, then the entries given in columns 36 through 65 are taken to be referenced to a global (R,θ,Z) or (R,θ,φ) system rather than to be the standard (X,Y,Z) system (see Figure in page 5-39). The program converts the cylindrical or spherical coordinate system to cartesian coordinates using the formulas:

Cylindrical Systems		**Spherical Systems**	
X =	$R\cos\theta$	X =	$R\cos\theta \sin\varphi$
Y =	$R\sin\theta$	Y =	$R\sin\theta \sin\varphi$
Z =	Z	Z =	$R\cos\varphi$

Cylindrical or spherical coordinate input is merely a user convenience for locating nodes in the standard (X,Y,Z) system, and no other references to the cylindrical or spherical system is implied; i.e., boundary condition specifications, output displacement components, etc., are referenced to the (X,Y,Z) system. Cylindrical or spherical coordinates are echoed in the printout labeled "NODAL INPUT DATA" (indicated by a "c" or "s" preceding the node number). The corresponding cartesian coordinates are printed in the GENERATED NODAL POINT DATA.

(2) Nodal data must be defined for all NUMNP nodes. Nodal cards need not be in sequence but the last one must be the last node number in the system. If nodal cards are defined more than once, the last definition will be used. All the nodal points at or below the ground surface must reside on the soil layer Interfaces.

(3) Six boundary and constrain conditions (B.C. and C.C.) codes must be assigned to each nodal point number. They can have the following numbers:

ID(N,M) = 0free translation (or rotation)
= 1 fixed translation (or rotation)
= Nm, constrained translation (or rotation) to node number Nm.

In general, any nodal point can have six degrees of freedom (three translations and three rotations). Therefore, the maximum number of degrees of freedom in a system is six times the total number of nodal points. A free degree of freedom will allow the node to translate (or rotate) in that direction as the solution dictates. Concentrated masses (or forces) then may be applied at this degree of freedom. One system equilibrium equation is required for each free degree of freedom.

A fixed degree of freedom will not allow the node to translate or rotate in that direction. Therefore, it will be removed from the final set of equilibrium equations. Any concentrated masses (or forces) assigned to this degree of freedom will be ignored by the program.

The motion of node in constrained (enslaved) degree of freedom is defined by the motion of the master node. The equations for the constrained degrees of freedom are eliminated from the global equation of motion since the motion of the slave node is entirely dependant on the motion of the master node. Equations of a rigid body motion define the motion of the constrained degree of freedom as function of the master node motion. The salve-master constrains are used to transform the stiffness and mass properties to the degrees of freedom associated with the master node and is only applicable to **beam**, **thin shell, and thick shell** elements. **No interaction node** should be considered as slave node.

Nodes that are used for geometric references only (i.e., nodes not assigned to any elements) must have all six degrees of freedom fixed. These nodes are sometimes needed to define the geometry of beam sections. Also, nodal degrees of freedom having undefined stiffness (such as rotations in all brick elements, out-of-plane components in a two-dimensional plane strain model) should be deleted.

Removing unwanted degrees of freedom has the advantage of reducing the size of the set of equations that must be solved. The following table lists the degrees of freedom that are defined by each different element type.

ELEMENT TYPE	DOF WITH DEFINED STIFFNESS/MASS					
	X	**Y**	**Z**	**XX**	**YY**	**ZZ**
1. 3-D/Brick	ŏ	ŏ	ŏ			
2. 3-D/Beam	ŏ	ŏ	ŏ	ŏ	ŏ	ŏ
3. 3-D/Thin Shell	ŏ	ŏ	ŏ	ŏ	ŏ	
4. 2-D/Plane Strain	ŏ		ŏ			
5. 3-D/Pile	ŏ	ŏ	ŏ	ŏ	ŏ	ŏ
6. 2-D/Pile	ŏ		ŏ		ŏ	
7. 3-D/Spring	ŏ	ŏ	ŏ	ŏ	ŏ	ŏ
8. 1-D/Plane Love Wave		ŏ				
10. 3-D/Thick Shell	ŏ	ŏ	ŏ	ŏ	ŏ	ŏ

Note, for example, that for all-3-D/brick model, only the X, Y, Z, translations are defined at the nodes and the number of equations can be cut in half by deleting the three rotational components at every node. If a node is common to two or more element types, then the non-trivial degrees of freedom are found by combination. For example, all six components are possible at a node common to both beam and solid elements; i.e., beam governs. Symmetric structures (with symmetric loading only) may also be analyzed by modeling only one-half or one quarter of the structure and constraining appropriate degrees of freedom on the planes of symmetry.

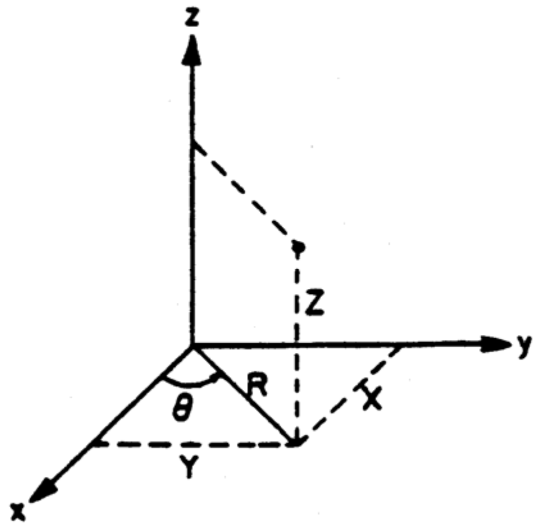
(4) The Z-coordinate must always be chosen vertical upward and the right-hand rule must be used to set X and Y coordinates. For 1-D or 2-D analysis, the program will ignore the Y-coordinate of the nodal points (see the following figure).

(5) The nodal data for a series of nodal points which are sequentially numbered between the beginning and end of any straight line, and which are equally spaced along that line, may be generated from information given on the first and last nodal card in the series. If (N_1,...,KN_1) and (N_2,...KN_2) are the information on the first and

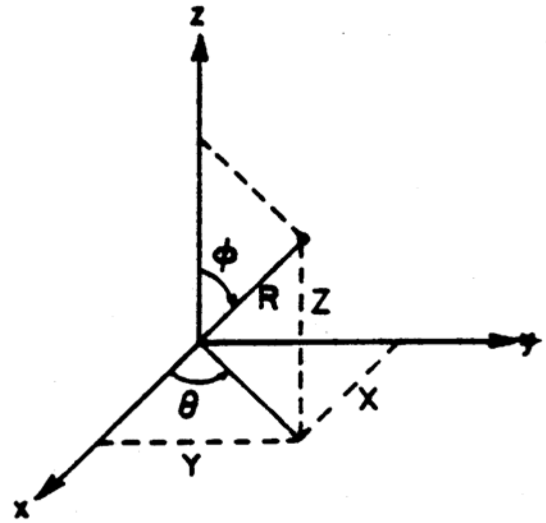
last nodal cards in a series, the first generated node will be N_1 + KN_2 and the second generated node will be N_1 + 2*KN_2, etc. Generation will continue until node number N_2 - KN_2 is established. Therefore N_2 - N_1 must be evenly divisible by KN_2. The boundary condition codes and nodal point type code for the generated nodal points are set equal to the values given on the first card.

Coordinate generation is done in cartesian, cylindrical, or spherical coordinates if the first card in the series is in cartesian, cylindrical, or spherical coordinates, respectively.

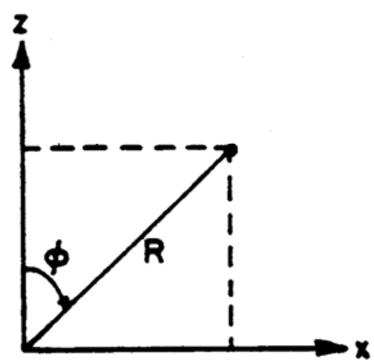
(6) The number of pile impedance associated with the node should be less or equal to total number of pile impedance, **NTPILE** (see Section **D.3**)



3-D Cartesian/Cylindrical



3-D Cartesian/Spherical



Plane Cartesian/Spherical

D.8 INTERACTION NODES CARDS

Provide the required data cards only for the section which corresponds to the method to be used for computing impedance matrix (**NIMP** = 1 or 2 or 3).

D.8.1 DIRECT OR SUBTRACTION METHOD (complete only if NIMP = 1 or NIMP = 3)

D.8.1.1 INTERACTION NODES CONTROL CARD (I5)

Columns	Variable	Notes	Entry
1-5	**INTACT**	(1)	Total number of interaction nodes to be entered

D.8.1.2 INTERACTION NODES DATA CARDS (16I5)

Skip this section if **INTACT** = 0; otherwise provide INTACT node numbers.

Columns	Variable	Notes	Entry
1-5	**N(1)**	(2)	First interaction node number
6-10	**N(2)**		Second interaction node number
•	•		•
•	•		•
•	•		•

Terminate by a zero node number.

D.8.2 SKIN METHOD (complete only if NIMP = 2)

D.8.2.1 INTERACTION NODES CONTROL CARD (3I5)

Columns	Variable	Notes	Entry
1-5	**INTFCE**		Total number of interaction nodes
6-10	**INTMED**		Total number of intermediate nodes
11-15	**INTRNL**		Total number of internal nodes

D.8.2.2 INTERFACE NODES DATA CARDS (16I5)

Provide **INTFCE** node numbers.

Columns	Variable	Notes	Entry
1-5	**NE(1)**	(2)	First interface node number
6-10	**NE(2)**		Second interface node number
•	•		•
•	•		•
•	•		•

Terminate by a zero node number.

D.8.2.3 INTERMEDIATE NODES DATA CARDS (16I5)

Provide **INTMED** node numbers.

Columns	Variable	Notes	Entry
1-5	**ND(1)**	(2)	First intermediate node number
6-10	**ND(2)**		Second intermediate node number
•	•		•
•	•		•
•	•		•

Terminate by a zero node number.

D.8.2.4 INTERNAL NODES DATA CARDS (16I5)

Skip this section if **INTRNL** = 0; otherwise provide **INTRNL** node numbers.

Columns	Variable	Notes	Entry
1-5	**NL(1)**	(2)	First internal node number
6-10	**NL(2)**		Second internal node number
•	•		•
•	•		•
•	•		•

Terminated by a zero node number.

Notes:

(1) if this parameter is zero, the program automatically assumes that every node at or below the ground surface is an interaction node.

(2) A series of nodes can be generated by entering the first node in the series, the last node in the series, and the generation code as a negative number; e.g.:

(5 11 -2 4 15 19 17 -1 0) is equivalent to
(5 7 9 11 4 15 19 18 17 0).

D.9 SOIL LAYER DATA (I5, 6F10.0)

This section must be skipped if **NUML** = 0 (e.g., in the case of surface structures)

Columns	Variable	Notes	Entry
1-5	**N**	(1)	Layer Number
6-15	**G**		Thickness
16-25	**W**		Unit weight
26-35	**VS**		S-wave velocity
36-45	**VP**		P-wave velocity
46-55	**DS**		S-wave associated damping ratio
56-55	**DP**		P-wave associated damping ratio

Notes:

(1) Layer data cards need not be input in layer-order sequence; eventually, however, all layers in the set (1 to **NUML**) must be defined.

D.10 ELEMENT LIBRARY

This element library consists of the following nine element types:

D.10.1 Three-dimensional solid element (eight-node brick) with three translational degrees of freedom per node. This element is identified with number 1 in the element library.

D.10.2 Three-dimensional beam element with three translational and three rotational degrees of freedom per node. This element is identified with number 2 in the element library.

D.10.3 Three-dimensional quadrilateral thin shell element with three translational and two rotational degrees of freedom per node. The element stiffness matrix includes five terms, two translations defining the in-plane stiffness of the structural member and two rotations defining the out of plane bending stiffness of the structural member. The out of plane translational degree of freedom is also included but it neglects transverse shear stiffness. The drilling rotational stiffness of the member is also neglected. This element is identified with number 3 in the element library.

D.10.4 Two-dimensional four-node plane strain finite element with two translational degrees of freedom per node. This element is identified with number 4 in the element library.

D.10.5 Three-dimensional pile element with three translational and three rotational degrees of freedom per node. This element is identified with number 5 in the element library

D.10.6 Two-dimensional pile element with two translational and one rotational degrees of freedom per node. This element is identified with number 6 in the element library (not available).

D.10.7 Three-dimensional spring element with three translational and three rotational degrees of freedom per node. This element is identified with number 7 in the element library.

D.10.8 One-dimensional plane love wave element with one out-of-plane translational degree of freedom per node. This element is identified with number 8 in the element library (not available).

D.10.9 Three-dimensional stiffness/mass matrix element with three translational and three rotational degrees of freedom per node. This element is identified with number 9 in the element library.

D.10.10 Three-dimensional quadrilateral thick shell element with three translational and three rotational degrees of freedom per node. The element stiffness matrix includes six terms, two translations and one rotation defining the in-plane stiffness of the structural member and one translation and two rotations defining the out of plane bending stiffness of the structural member. The element formulation includes the out-of-plane shear stiffness. This element is identified with number 10 in the element library.

D.10.1 THREE-DIMENSIONAL SOLID ELEMENTS (EIGHT-NODE BRICK)

Groups of solid elements are described by the following sequence of cards.

D.10.1.1 CONTROL INFORMATION (5I5,5X,A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 1
6-10	**NPAR(2)**		Total number of 8-node solid elements
11-15	**NPAR(3)**	(1)	Number of material types
19-20	**NPAR(4)**	(2)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear modulus
			= 1, input P- and S-wave velocities
25	**NPAR(5)**	(3)	Incompatible mode code
			= 0, include incompatible modes
			≠ 0, suppress incompatible modes
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.1.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Skip this section if **NPAR(3)** = 0.

Columns	Variable	Notes	Entry
1-5	**N**		Material-type number
6-15	**M(N)**	(2)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(2)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight of material
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DN(N)**		S-wave-associated damping ratio

D.10.1.3 EIGHT-NODE SOLID ELEMENT CARDS (13I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(3)	Element number
6-10	**INP(1)**	(4)	Nodal point 1
11-15	**INP(2)**		Nodal point 2
16-2O	**INP(3)**		Nodal point 3

Columns	Variable	Notes	Entry
21-25	**INP(4)**		Nodal point 4
26-30	**INP(5)**		Nodal point 5
31-35	**INP(6)**		Nodal point 6
36-40	**INP(7)**		Nodal point 7
41-45	**INP(8)**		Nodal point 8
50	**ININT**	(5)	Integration order
54-55	**INTYP**	(6)	Element type
			= 1, structural element
			= -1, excavated soil element
56-60	**IMAT**	(8)	Material-type number for structural elements/soil layer number for soil elements
61-65	**IINC**	(9)	Element generator code

Notes:

(1) The value of this parameter is selected as zero if all the elements in this group are excavated soil elements.

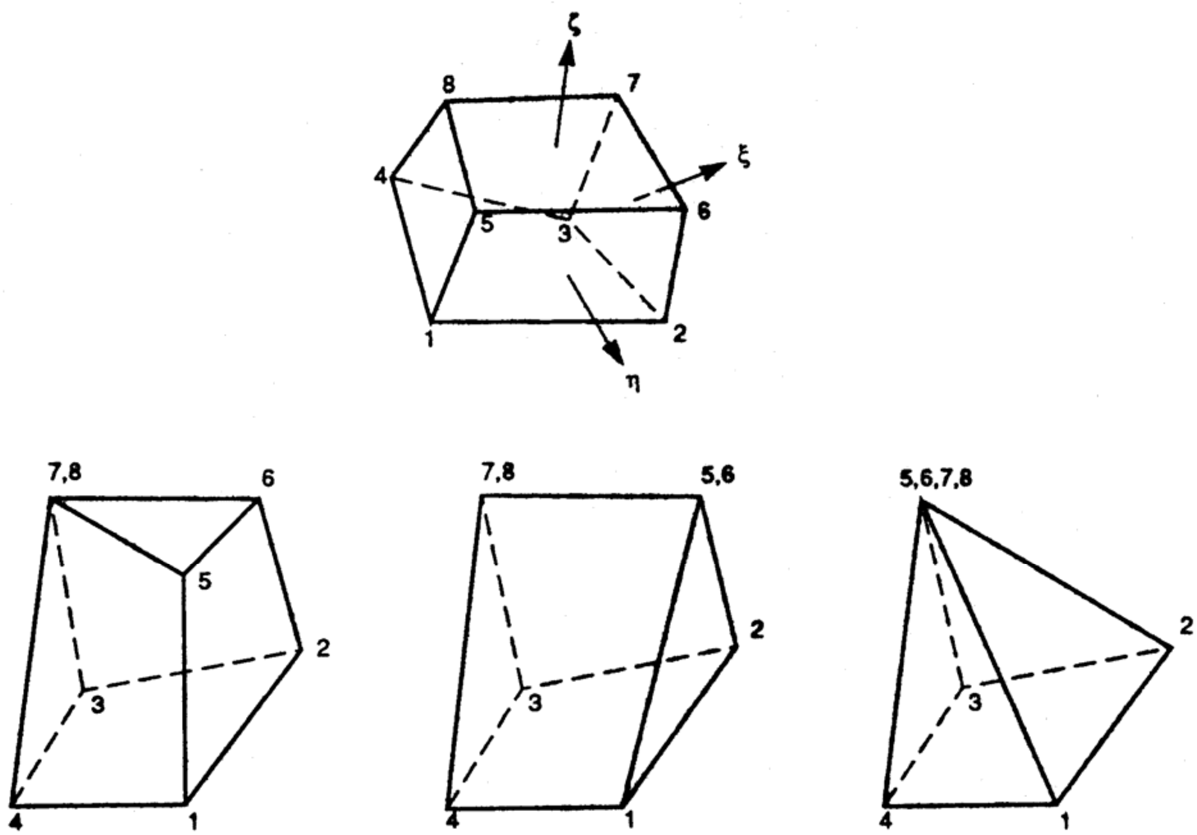
(2) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

MATERIAL PROPERTY CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(3) Nine incompatible displacement modes are included in the formation of the stiffness matrix and can be suppressed at user's option by assigning a nonzero integer number to **NPAR(5)**. The program will automatically suppress all incompatible modes for excavated soil elements. Also, incompatible modes must not be used for solid elements with fewer than eight nodes.

(4) Element cards must be in ascending order.

(5) Element numbering for a general eight-node solid element and some other configurations is shown below.



(6) The following integration orders are recommended for solid elements.

ININT	ELEMENT
2	Rectangular
3	Skewed
4	Extremely distorted in shape

However, using very distorted elements should be avoided as much as possible.

(7) **INTYP** = 1 (or -1) will cause the element mass and stiffness to be eventually added or subtracted from total mass and stiffness of the system.

(8) The material properties of structural elements (**INTYP** = 1) are obtained from Section **D.10.1.2**, where **IMAT** refers to the material type number. For excavated soil elements (**INTYP** = **-**1), these properties are obtained from Section **D.9**, where **IMAT** refers to the layer number in the free-field from which the excavated soil element is obtained.

(9) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

(b) Nodal point numbers are generated by adding **IINC** to those of the preceding element. (If omitted, **IINC** is set to 1.)

(c) The same material properties, integration order, and element type are used as for the preceding element.

(d) An element card for the last element must be supplied.

Element Stiffness Generation

If **ININT** = 0: A new element stiffness is not formed. Element stiffness is assumed to be identical to that of the preceding element.

D.10.2 THREE-DIMENSIONAL BEAM ELEMENTS

Groups of beam elements are described by the following sequence of cards.

D.10.2.1 C0NTR0L INFORMATION (5I5,5X,A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 2
6-10	**NPAR(2)**		Total number of beam elements
11-15	**NPAR(3)**		Number of material types
16-20	**NPAR(4)**		Number of geometric property types
24-25	**NPAR(5)**	(1)	Material property code = -1 , input elastic modulus and Poisson's ratio 0, input constrained and shear moduli = 1, input P- and S-wave velocities
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.2.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material identification number
6-15	**M(N)**	(1)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(1)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight of material
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DS(N)**		S-wave-associated damping ratio

D.10.2.3 ELEMENT GEOMETRIC PROPERTY CARDS (I5, 6F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Geometric property identification number
6-15	**ELP(1,N)**	(2)	Axial area
16-25	**ELP(2,N)**	(2)	Shear area associated with shear forces in local 2-direction
26-35	**ELP(3,N)**	(2)	Shear area associated with shear forces in local 3-direction
36-45	**ELP(4,N)**	(2)	Torsional inertia

46-55	**ELP(5,N)**	(2)	Flextural inertia about local 2-axis
56-65	**ELP(6,N)**	(2)	Flextural inertia about local 3-axis

D.10.2.4 BEAM ELEMENT CARDS (7I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(3)	Element number
6-10	**INI**		Node number I
11-15	**INJ**		Node number J
16-20	**INK**	(4)	Node number K
21-25	**IMAT**		Material property number
26-30	**IMEL**		Element geometric property number
31-35	**IINC**	(5)	Element generator code
40-45	**IB1**	(6)	End release code at node number I
50-55	**IB2**	(6)	End release code at node number J

Notes:

(1) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

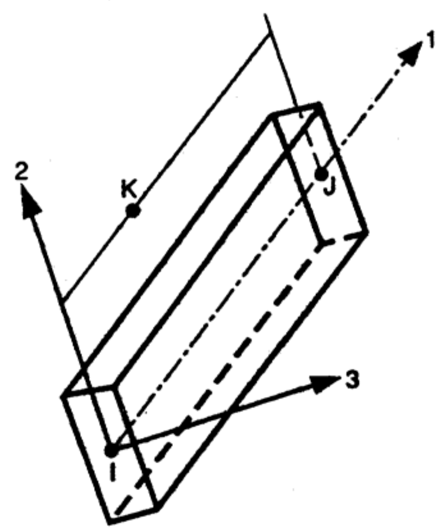
MATERIAL PROPERTY

CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(2) Section properties of some mostly used cross sections are given below. One card is required for each unit set of properties. If shear deformations are not going to be included in the analysis, let **ELP(2,N)** and **ELP(3,N)** be zero.

(3) Element cards must be in ascending order.

(4) Node **K** is a geometric reference point which is used to define local 1, 2, and 3 axes of the beam element. Node **K**, which can also be any other nodal point in the system, must not lie on the local I axis. See figure below.



(5) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

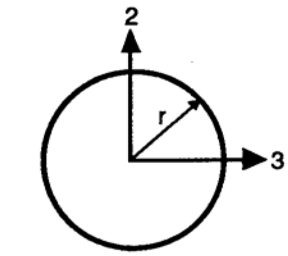
(b) Nodal points **I** and **J** are generated by adding **IINC** to those of the preceding element. (If omitted, **IINC** is set to 1.)

(c) Nodal point **K** Is duplicated from the preceding element.

(d) The same material properties, and section properties are used as for the preceding element.

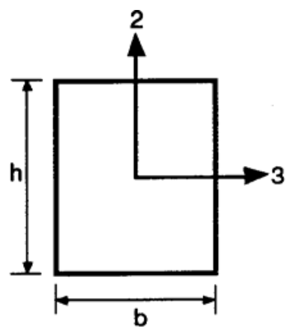
(e) An element card for the last element must be supplied.

(6) The end release code at each node is a six digit number to be zero and/or one. The 1st, 2nd,--------6th digits, respectively correspond to the force components P1, P2, P3, M1, M2, M3 at each node. If one of the element end forces is known to be zero (hinge or roller), the digit corresponding to that component is one.



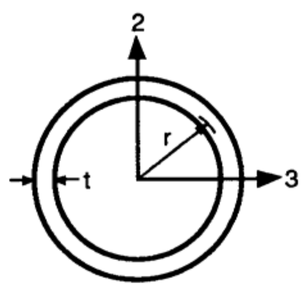
$I_2 = \pi r^4/4$
$I_3 = \pi r^4/4$
$J = \pi r^4/2$

$f_2 = f_3 = 10/9$
$A = \pi r^2$



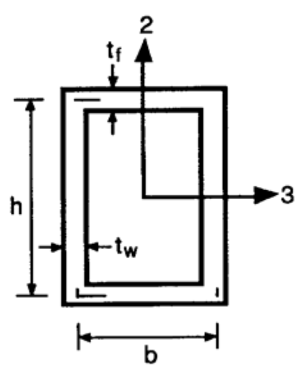
$I_2 = hb^3/12$
$I_3 = hb^3/12$
$J \cong [1/3 - 0.21(b/h)(1 - b^4/12h^4)]hb^3$

$f_2 = f_3 = 6/5$
$A = bh$



$I_2 \cong \pi r^3 t$
$I_3 \cong \pi r^3 t$
$J \cong 2\pi r^3 t$

$f_2 = f_3 = 2$
$A = 2\pi rt$



$I_2 \cong (h^2/6)\ (ht_w + 3bt_f)$
$I_3 \cong (b^2/6)\ (bt_f + 3ht_w)$
$J \cong 2\ b^2 h^2 (t_f\, t_w)/(bt_w + ht_f)$

$f_2 = A/(2ht_w)$
$f_3 = A/[2(b + 2t_w)t_f]$
$A = 2(bt_f + ht_w)$

Note: Ratio of Shear Area / Axial Area = 1/f

D.10.3 PLATE/SHELL ELEMENTS

Groups of plate/thin shell elements are described by the following sequence of cards.

D.10.3.1 CONTROL INFORMATION (4I5,10X,A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 3
6-10	**NPAR(2)**		Total number of plate/shell elements
11-15	**NPAR(3)**		Number of material types
19-20	**NPAR(4)**	(1)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear moduli
			= 1, input P- and S-wave velocities
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.3.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material -type number
6-15	**M(N)**	(1)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(1)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**	(2)	Unit weight
36-45	**DP(N)**	(3)	P-wave-associated damping ratio
46-55	**DS(N)**	(3)	S-wave-associated damping ratio

D.10.3.3 PLATE/SHELL ELEMENT CARDS (8I5, F10.0)

Columns	Variable	Notes	Entry
1-5	**INEL**	(4)	Element number
6-10	**INP(1)**	(5)	Nodal point **I**
11-15	**INP(2)**		Nodal point **J**
16-20	**INP(3)**		Nodal point **K**
21-25	**INP(4)**	(6)	Nodal point **L**
26-30	**INP(5)**	(7)	Nodal point **0**
31-35	**IMAT**		Material-type number
36-40	**IINC**	(8)	Element generator code
41-50	**TH**		Element thickness

Notes:

(1) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

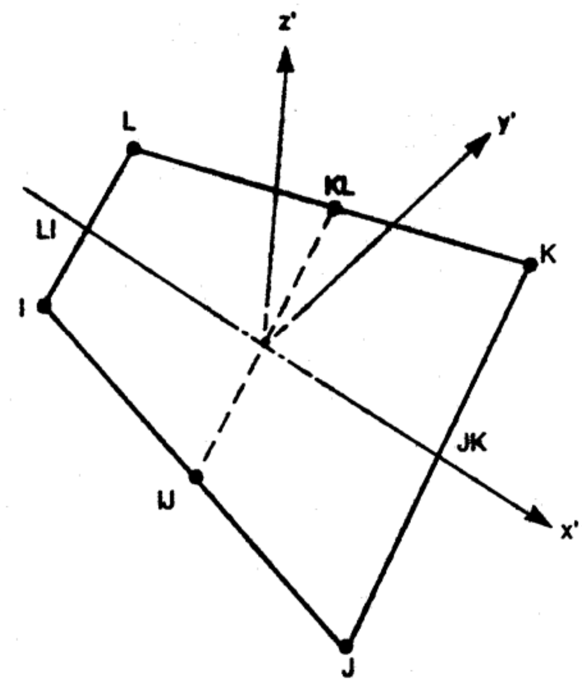
MATERIAL PROPERTY CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(2) Mass matrix is lumped mass matrix for plate elements.

(3) P-wave and S-wave damping ratios must be equal.

(4) Element cards must be in ascending order.

(5) The nodal point numbers I, J, K, and L must be in sequence in a counterclockwise direction around the element. Local coordinates In a four-node shell element are shown below.



X' specified by **LI-JK**, where **LI** and **JK** are midpoints of sides **L-I** and **J-K**.

Z' normal to X' and to the line adjoining midpoints **IJ** and **KL** (line **IJ-KL**).

Y' normal to X' and Z' to complete the right handed systems the local system is used to compute the resultant forces.

(6) Leave blank for triangular elements.

(7) When left blank, mid-node properties are computed by averaging the four modes.

(8) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

(b) Nodal points I, J, K, and L are generated by adding IINC to those of the preceding element. (If omitted, IINC is set to 1.)

(c) The same material properties and element thickness are used as for the preceding element.

(d) An element card for the last element must be supplied.

D.10.4 TWO-DIMENSIONAL FINITE ELEMENTS (QUAD-2D)

Groups of 2-D finite elements are described by the following sequence of cards.

D.10.4.1 CONTROL INFORMATION (4I5,10X,A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 4
6-10	**NPAR(2)**		Total number of 2-D finite elements
11-15	**NPAR(3)**	(1)	Number of material types
19-20	**NPAR(4)**	(2)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear moduli
			= 1, input P- and S-wave velocities
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.4.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Skip this section if **NPAR(3)** = 0.

Columns	Variable	Notes	Entry
1-5	**N**		Material-type number
6-15	**M(N)**	(2)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(2)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DS(N)**		S-wave-associated damping ratio

D.10.4.3 2-D FINITE ELEMENT CARDS (8I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(3)	Element number
6-10	**INP(1)**	(4)	Nodal point 1
11-15	**INP(2)**		Nodal point 2
16-20	**INP(3)**		Nodal point 3
21-25	**INP(4)**		Nodal point 4
26-30	**INTYP**	(5)	Element type
			= 1 , structural element
			= -1, soil element
31-35	**IMAT**	(6)	Material-type number for structural element/soil layer number for soil elements
36-40	**IINC**	(7)	Element generator code

Notes:

(1) The value of this parameter is selected as zero if all the elements in this group are excavated soil elements.

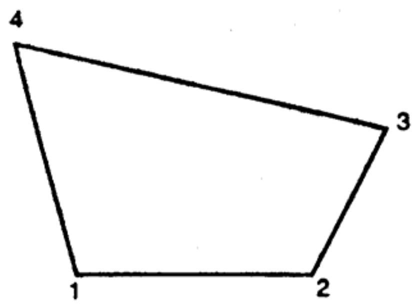
(2) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

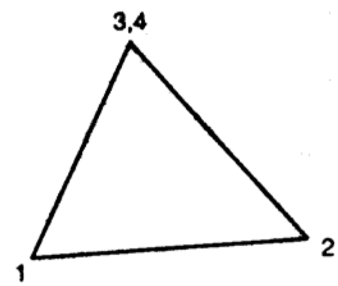
MATERIAL PROPERTY

CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(3) Element cards must be in ascending order.

(4) The nodal point numbers 1, 2, 3, and 4 must be in sequence in a counterclockwise direction around the element.





For triangular elements, the number of nodal point 3 must be equal to that of nodal point 4.

(5) INTYP = 1 (or -1) will cause the element mass and stiffness to be eventually added or subtracted from total mass and stiffness of the system.

(6) The material properties of structural elements (INTYP = 1) are obtained from Section **D.10.4.2**, where IMAT refers to the material type number. For excavated soil elements (INTYP = -1), these properties are obtained from Section **D.9**, where IMAT refers to the layer number in the free-field from which the excavated soil element is obtained.

(7) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

(b) Nodal point numbers are generated by adding **IINC** to those of the preceding element. (If omitted, **IINC** is set to l.)

(c) The same material properties and element type are used as for the preceding element.

(d) An element card for the last element must be supplied.

D.10.5 THREE-DIMENSIONAL INTER-PILE ELEMENTS

This element is capable of considering compatibility between soil and pile by taking into account the effect of pile diameter. The compatibility is considered at the edges where the piles are connected, and therefore, if there is no pile, the common linear displacement variation along the boundary is assumed. The existence of pile at each corner is recognized by the program if one assigns area to the pile at that corner (see Section **D.10.5.3.2**). The individual piles must be modeled by standard beam elements when considering soil-pile interaction.

Groups of 3-D pile elements are described by the following sequence of cards.

D.10.5.1 CONTROL INFORMATION (4I5,10X,A50)

Columns	Variable	Notes	Entry
1-5	**NPAR(1)**		The number 5
6-10	**NPAR(2)**		Total number of 3-0 pile elements
11-15	**NPAR(3)**		Number of soil material types
19-20	**NPAR(4)**	(1)	Material property code = -1, input elastic modulus and Poisson's ratio = 0, input constrained and shear moduli = 1, input P- and S-wave velocities
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.5.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material-type number
6-15	**M(N)**	(1)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(1)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DS(N)**		S-wave-associated damping ratio

D.10.5.3 3-D INTER-PILE ELEMENT CARDS

D.10.5.3.1 NODAL CARDS (16I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(2)	Element number
6-10	**INP(1)**	(3)	Nodal point 1
11-15	**INP(2)**		Nodal point 2
16-20	**INP(3)**		Nodal point 3
21-25	**INP(4)**		Nodal point 4

26-30	**INP(5)**		Nodal point 5
31-35	**INP(6)**		Nodal point 6
36-40	**INP(7)**		Nodal point 7
41-45	**INP(8)**		Nodal point 8
46-50	**ININT**	(4)	Integration order
51-55	**IMAT**		Material number for soil element
56-60	**IINC**	(5)	Element generator code

D.10.5.3.2 AREAS OF THE PILE SECTIONS INCLUDED IN THE ELEMENT (4F10.0)

Columns	Variable	Notes	Entry
1-10	**AX(1)**	(6)	Part of area of pile 1 inside the element
11-20	**AX(2)**		Part of area of pile 2 inside the element
21-30	**AX(3)**		Part of area of pile 3 inside the element
31-40	**AX(4)**		Part of area of pile 4 inside the element

Notes:

(1) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

MATERIAL PROPERTY

CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(2) Element cards must be in ascending order.

(3) See the following figure for numbering of element connectivities.

(4) The value of ININT is of no significance; however, the following are possible:

ININT = 0 A new element stiffness is not formed; element stiffness is assumed to be identical to that of the preceding element.

ININT > 0 Stiffness of the element is formed.

ININT < 0 A new element stiffness is formed only for the first element in the series and the same element stiffness is used for succeeding elements that are to be generated.

(5) If a series of cards is omitted, then generation is possible as follows:

(a) Element cards must be in ascending order.

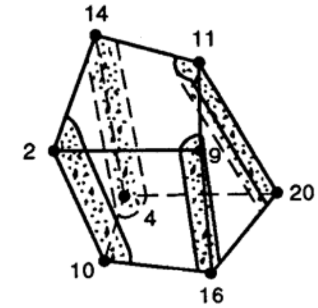
(b) Nodal point numbers are generated by adding **IINC** to those of the preceding element (if omitted, **IINC** is set to 1).

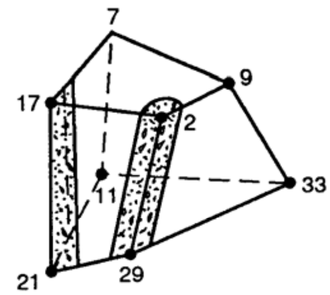
(c) Same material properties are used as for the preceding element.

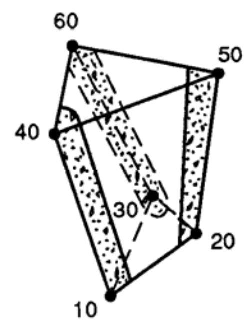
(d) Element card for the last element must be supplied.

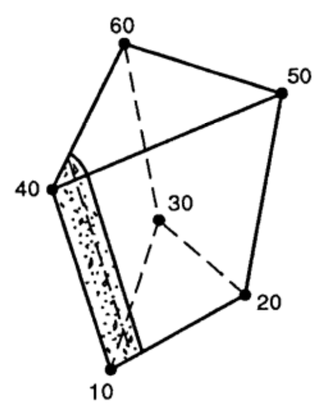
(6) See the following figure for areas of the pile.

N_1	N_2	N_3	N_4	N_5	N_6	N_7	N_8	AX_1	AX_2	AX_3	AX_4
10	16	20	4	2	9	11	14	a_1	a_2	a_3	a_4
21	29	33	11	17	2	9	7	a_1	a_2	0	0
10	20	30	10	40	50	60	40	a_1	a_2	a_3	a_1
10	20	30	10	40	50	60	40	a_1	0	0	a_1









D.10.6 TWO-DIMENSIONAL INTER-PILE ELEMENTS (not available)

This element is capable of considering compatibility between soil and pile by taking into account the effect of pile diameter. The compatibility is considered at the edges where the piles are connected, and therefore, if there is no pile, the common linear displacement variation along the boundary is assumed. The existence of pile at each corner is recognized by the program if one assigns area to the pile at that corner (see Section **D.10.6.3.2**). The individual piles must be modeled by standard beam elements when considering soil-pile interaction.

Groups of 2-D inter-pile elements are described by the following sequence of cards.

D.10.6.1 CONTROL INFORMATION (4I5)

Columns	Variable	Notes	Entry
1-5	**NPAR(1)**		The number 6
6-10	**NPAR(2)**		Total number of 2-D pile elements
11-15	**NPAR(3)**		Number of soil material types
19-20	**NPAR(4)**	(1)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear moduli
			= 1, input P- and S-wave velocities

D.10.6.2 MATERIAL PROPERTY CARDS (I5, 5F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material type number
6-15	**M(N)**	(1)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(1)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DS(N)**		S-wave-associated damping ratio

D.10.6.3 2-D INTER-PILE ELEMENT CARDS

D.10.6.3.1 N0DAL CARDS (16I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(2)	Element number
6-10	**INP(1)**	(3)	Nodal point l
11-15	**INP(2)**		Nodal point 2
16-20	**INP(3)**		Nodal point 3
21-25	**INP(4)**		Nodal point 4
26-30	**ININT**	(4)	Integration order
31-35	**IMAT**		Material number for soil element
36-40	**IINC**	(5)	Element generator code

D.10.6.3.2 AREAS OF THE PILE SECTIONS INCLUDED IN THE ELEMENT (2F10.0)

Columns	Variable	Notes	Entry
1-10	**AX(1)**	(6)	Part of area of pile 1 inside the element
11-20	**AX(2)**		Part of area of pile 2 inside the element

Notes:

(1) The following table shows how M and G are defined on material property cards of an element group by choosing a material property code:

MATERIAL PROPERTY CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(2) Element cards must be in ascending order.

(3) See the following figure for numbering of element connectivities.

(4) The value of ININT is of no significance; however, the following are possible:

ININT = 0	A new element stiffness is not formed; element stiffness is assumed to be identical to that of the preceding element.
ININT > 0	Stiffness of the element is formed.
ININT < 0	A new element stiffness is formed only for the first element in the series and the same element stiffness is used for succeeding elements that are to be generated.

(5) If a series of cards is omitted, then generation is possible as follows:

(a) Element cards must be in ascending order.

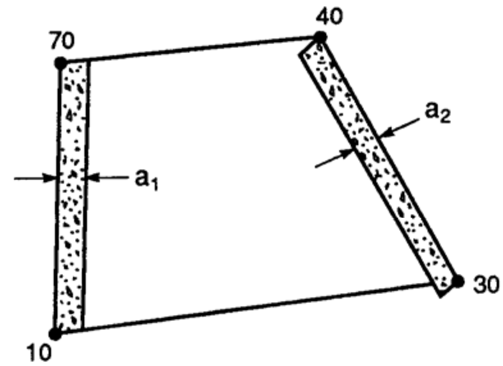
(b) Nodal point numbers are generated by adding IINC to those of the preceding element (if omitted, IINC is set to 1).

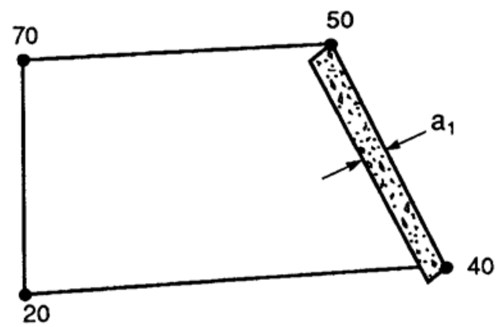
(c) Same material properties are used as for the preceding element.

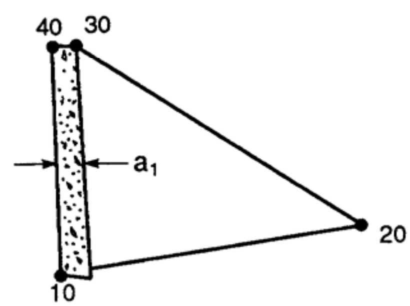
(d) Element card for the last element must be supplied.

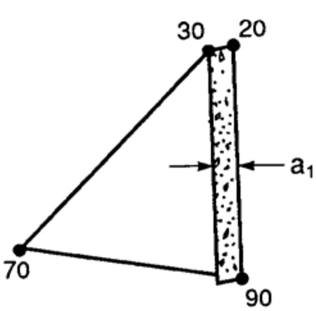
(6) See the following figure for areas of the pile.

N_1	N_2	N_3	N_4	AX_1	AX_2
10	30	40	70	a_1	a_2
20	40	50	70	0	a_1
10	20	30	40	a_1	0
70	90	20	30	0	a_1









D.10.7 THREE-DIMENSIONAL SPRING ELEMENTS

Grow of spring elements are described by the following sequence of cards.

D.10.7.1 CONTROL INFORMATION (3I5, 15X, A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 7
6-10	**NPAR(2)**		Total number of 3-D spring elements
11-15	**NPAR(3)**		Total number of different element types
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.7.2 ELEMENT TYPE CARDS (I5, 6F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Element-type number
6-15	**SPR(1,N)**	(1)	Translational spring constant in global x-direction
16-25	**SPR(2,N)**		Translational spring constant in global y-direction
26-35	**SPR(3,N)**		Translational spring constant in global z-direction

36-45	**SPR(4,N)**		Rotational spring constant in global xx-direction
46-55	**SPR(5,N)**		Rotational spring constant in global yy-direction
56-65	**SPR(6,N)**		Rotational spring constant in global zz-direction
66-75	**SPR(7,N)**	(2)	Damping ratio

D.10.7.3 3-D SPRING ELEMENT CARDS (5I5)

Columns	Variable	Note	Entry
1-5	**INEL**		Element number
6-10	**INI**	(2)	Node number I
11-15	**INJ**		Node number J
16-20	**IMAT**		Element type number
21-25	**IINC**	(3)	Generation code

Notes:

(1) Spring constants are directly added to the global stiffness matrix and thus these constants must be given in global xyz directions. Note that the spring constants in the six global directions are uncoupled.

(2) **I** is the node number at one end of the spring element and **J** is the node number at the other end.

(3) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

(b) Nodal points **I** and **J** are generated by adding **IINC** to those of the preceding element. (If omitted, **IINC** is set to 1).

(c) The same spring constants are used as for the preceding element.

(d) An element card for the last element must be supplied.

D.10.8 ONE-DIMENSIONAL PLANE LOVE-WAVE ELEMENTS (not available)

Group of love-wave elements are described by the following sequence of cards.

D.10.8.1 CONTROL INFORMATION (4I5)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 8
6-10	**NPAR(2)**		Total number of love-wave elements
11-15	**NPAR(3)**	(1)	Number of material types
16-20	**NPAR(4)**	(2)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear moduli
			= 1, input P- and S-wave velocities

D.10.8.2 MATERIAL PROPERTY CARDS (I5, F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		Material-type number
6-15	**M(H)**	(2)	Elastic modulus/constrained modulus/P-wave velocity
16-25	**G(N)**	(2)	Poisson's ratio/shear modulus/S-wave velocity
26-35	**W(N)**		Unit weight
36-45	**DP(N)**		P-wave-associated damping ratio
46-55	**DS(N)**		S-wave-associated damping ratio

D.10.8.3 LOVE-WAVE ELEMENT CARDS (8I5)

Columns	Variable	Notes	Entry
1-5	**INEL**	(3)	Element number
6-10	**INP(1)**	(4)	Nodal point 1
11-15	**INP(2)**		Nodal point 2
16-20	**INP(3)**		Nodal point 3
21-25	**INP(4)**		Nodal point 4
26-30	**INTYP**	(5)	Element type
			= 1, structural element
			= -1, soil element
31-35	**IMAT**	(6)	Material type number for structural elements/soil layer number for soil elements
36-40	**IINC**	(7)	Element generator code

Notes:

(1) Refer to note 1 of Section **D.10.1.**

(2) Refer to note 2 of Section **D.10.1.**

(3) Element cards must be in ascending order.

(4) Refer to note 4 of Section **D.10.4.**

(5) Refer to note 7 of Section **D.10.1.**

(6) Refer to note 8 of Section **D.10.1.**

(7) Refer to note 7 of Section **D.10.4.**

D.10.9 STIFFNESS/MASS MATRIX ELEMENTS

Groups of stiffness/mass matrix elements are described by the following sequence of cards.

D.10.9.1 CONTROL INFORMATION (3I5, 15X, A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 9
6-10	**NPAR(2)**		Total number of matrix elements
15	**NPAR(3)**	(1)	Mass type code
			= 0 (or blank), enter mass
			≠ 0, enter weight
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.9.2.1 MASS MATRIX ELEMENT CARDS (4I5)

Columns	Variable	Notes	Entry
1-5	**NEL**		Element number
6-10	**NI**		Node number **I**
11-15	**NJ**		Node number **J**
16-20	**NK**	(2)	Node number **K**

D.10.9.2.2 ELEMENT STIFFNESS/MASS MATRIX CARDS (2I5, 3F10.0)

Columns	Variable	Notes	Entry
1-5	**NR(I)**	(3)	Row number
6-10	**NC(I)**	(3)	Column number
11-20	**ZSR(I)**	(4)	Real part of stiffness term
21-30	**ZSI(I)**	(4)	Imaginary part of stiffness term
31-40	**ZM(I)**		Mass/weight value

Notes:

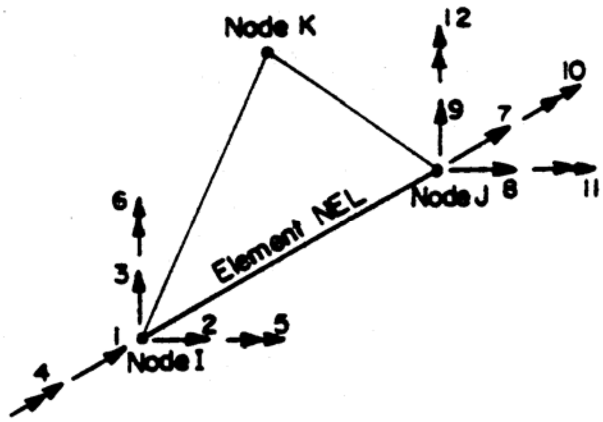
(1) If this parameter is non-zero, the values entered in columns 31-40 in Section **D.10.9.2.2** are considered in weight unit and therefore are divided by the acceleration of gravity to be converted to mass unit.

(2) This node is used to set up the element local coordinate system in which the element stiffness/mass matrix is defined. Item 4 of Section **D.10.2** describes how this local coordinate system is set up. If this node is left blank, the stiffness/mass entries for this element are considered in the global coordinate system.

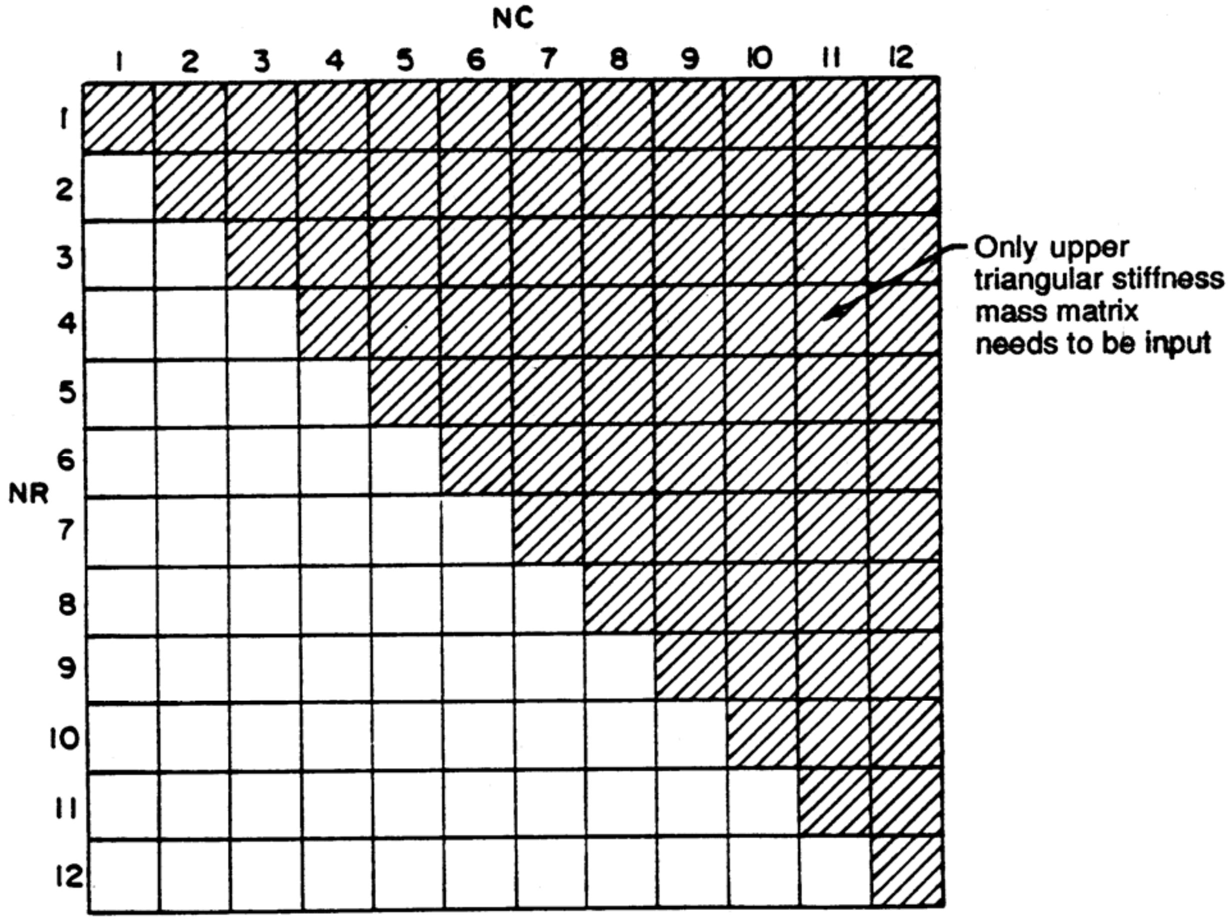
(3) The parameters **NR** and **NC** refer to the rows and columns of the element stiffness/mass matrix, respectively, as shown in figure. Because of the symmetry, only the upper triangular matrix needs to be input.

(4) The elements of the complex stiffness-matrix are computed in the program from the following formula:

$$K(I) = ZSR(I) + i * ZSI(I)$$



Node K
12
10
9
7
Node J
8
11
6
3
Element NEL
1
2
5
4
Node I



NC
1 2 3 4 5 6 7 8 9 10 11 12
NR
1 2 3 4 5 6 7 8 9 10 11 12
Only upper triangular stiffness mass matrix needs to be input

D.10.10 THICK SHELL ELEMENTS

Groups of thick shell elements are described by the following sequence of cards.

D.10.10.1 CONTROL INFORMATION (4I5, 10X, A50)

Columns	Variable	Notes	Entry
5	**NPAR(1)**		The number 10
6-10	**NPAR(2)**		Total number of shell elements
11-15	**NPAR(3)**		Number of material types
19-20	**NPAR(4)**	(1)	Material property code
			= -1, input elastic modulus and Poisson's ratio
			= 0, input constrained and shear moduli
			= 1, input P- and S-wave velocities
31-80	**ELGRPID**		Any character string to define the element group, will be printed in the HOUSE output

D.10.10.2 MATERIAL PROPERTY CARDS (A5,I5, 5F10.0)

Columns	Variable	Notes	Entry
1-5	**MI**	(2)	Material type index
			= ISO Isotropic material
			= ORTHO orthotropic material
6-11	**N**	(3)	Material identification number
11-20	**M(N)**	(1)	Elastic modulus/constrained modulus/P-wave velocity (blank for orthotropic material)
21-30	**G(N)**	(1)	Poisson's ratio/shear modulus/S-wave velocity (blank for orthotropic material)
31-40	**W(N)**	(4)	Unit weight
41-50	**DP(N)**	(5)	P-wave-associated damping ratio
51-60	**DS(N)**	(5)	S-wave-associated damping ratio

D.10.10.3 ORTOTROPIC MATERIAL PROPERTY CARDS (6F10.0)

Skip this section if **MI = ISO**

Columns	Variable	Notes	Entry
1-11	EX	(6)	Elastic modulus in element x-direction
11-20	EY		Elastic modulus in element y-direction
21-30	NXY		Poisson's ratio of xy plane
31-40	GXY		Shear modulus of xy plane
41-50	GXZ		Shear modulus of xz plane
51-60	GYZ	(6)	Shear modulus of yz plane

D.10.10.4 THICK SHELL ELEMENT CARDS (I5, A5,4I5,5x,I5,F10.0)

Columns	Variable	Notes	Entry
1-5	**INEL**	(7)	Element number
6-10	**INEL**	(8)	Element indentification tag
11-15	**INP(1)**	(9)	Nodal point **I**
16-20	**INP(2)**		Nodal point **J**
21-25	**INP(3)**		Nodal point **K**

26-30	**INP(4)**	(9)	Nodal point **L**
36-40	**IMAT**		Material-type number
41-45	**IINC**	(10)	Element generator code
46-55	**TH**		Element thickness

D.10.10.5 THICK SHELL ELEMENT END CARD (A9)

Columns	Variable	Notes	Entry
1-9	**IEND**	(11)	Text to end thick shell element input =END SHELL17

Notes:

(1) The following table shows how M and G are defined on isotropic material property cards of an element group by choosing a material property code:

MATERIAL PROPERTY CODE	M	G
-1	Elastic modulus	Poisson's ratio
0	Constrained modulus	Shear modulus
1	P-wave velocity	S-wave velocity

(2) Enter the word “ISO” in columns 1-3 for isotropic material. Enter the word “ ORTHO” in columns 1-5 for orthotropic material.

(3) Material property identification number must be input sequentially starting with the number one (1).

(4) Consistent mass matrix is formed for thick shell element.

(5) P-wave and S-wave damping ratios must be equal.

(6) Orthotropic material properties are specified in reference to the element local coordinates (x,y and z) (see Figure in Note 8).

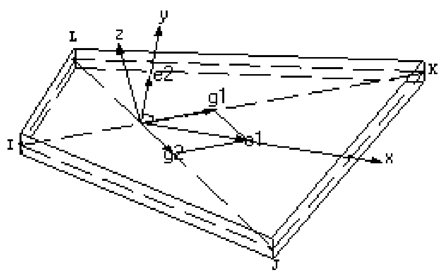
(7) Set of cards should be arranged in element number increasing sequence; it starts with one and ends with specified total number of shell elements NPAR(2)). If any two consecutive Element Data Cards have element numbers N_1 and N_2 respectively, where the number difference is greater than one, $N_2 > N_1 + 1$, the omitted element data is generated as follows:

   A. The element number is incremented by one.

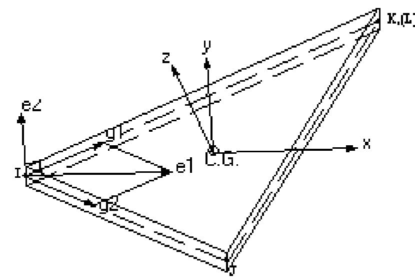
   B. Nodes 1 through 4 are incremented by node number increment for automatic generation, KN, specified in the preceding card (i.e., N_1 element).

   C. Element identification tag, material properties, element thickness, and stress reference plane identifier are set equal to the value specified in the first Element Data Card in the generated series (i.e., N_1 element).

(8) The sequence of nodal numbers **I**, **J**, **K**, and **L** and right-hand rule define the positive direction of local z axis. The figure below shows the definition of local coordinate system for quadrilateral and triangular element.



Quadrilateral Element



Triangular Element

For triangular element, number of node L must be equal to number of node K. If an interior angle of a triangular element is < 10° or > 170° (see figure above), a warning message is printed.

(9) If a series of cards is omitted, then generation will be possible as follows:

(a) Element cards must be in ascending order.

(b) Nodal points **I**, **J**, **K**, and **L** are generated by adding **IINC** to those of the preceding element. (If omitted, **IINC** is set to 1.)

(c) The same material properties and element thickness are used as for the preceding element.

(d) An element card for the last element must be supplied.

(10) This card with the text "END SHELL17" must be provided to end the input for the group of thick shell elements.

D.11 CONCENTRATED MASS CARDS (LUMP MASS)

If no concentrated mass (**NUMLM** = 0), skip this section.

D.11.1 FIRST AND CONSECUTIVE CARDS (2I5, 6F10.0)

Columns	Variable	Notes	Entry
1-5	**N**		(1) Nodal point number
6-10	**MTYP**		(2) Type of mass entry = 0, entry in mass units > 0, entry in weight units
11-20	**XMASS (N,1)**		Translational mass acting in x- direction
21-30	**XMASS (N,2)**		Translational mass acting in y-direction
31-40	**XMASS (N,3)**		Translational mass acting in z-direction
41-50	**XMASS (N,4)**		Rotational mass acting in xx-direction
51-60	**XMASS (N,5)**		Rotational mass acting in yy-direction
61-70	**XMASS (N,6)**		Rotational mass acting in zz-direction

Notes:

(1) Total of **NUMLM** cards must be provided for this section.

(2) If a non-zero value is assigned by **MTYP**, then the entries given in columns 11 through 70 are taken to be in weight units rather than mass units. Thus, in order to convert these values to mass units, the program divides these values by the acceleration of gravity given in Section **D.4**.

D.12 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop no more data

Notes:

(1) Must be the last data card.

5.2.5 INCOH (Not Available)

The program module **INCOH** computes the incoherency characteristics of the input free-field ground motion with respect to the foundation size and the frequency range of analysis. Two incoherent models are built in the program module: one is based on the paper by Mita and Luco (1986), and the other is based on the paper published by Abrahamson (1993). The program first computes the coherency matrix for the given foundation model in the frequency range to be analyzed, and then decomposes the matrix into generalized eigenvalues and eigenvectors, and then recombine all the generalized modes to form influence factors for the ground motion.

The program module needs to read Tape 1 (The file **SASSI.T1** as generated by **SITE**) for information of frequency range, and read Tape 7 (The file **SASSI.T7** as generated by **HOUSE**) for information of the structural model.

The computed influence factors are written in Tape 11 (File **SASSI.T11**) to be used in the program module **ANALYS**.

E.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If **NOPT** < 0, most of the calculations required during normal execution are bypassed.

E.2 OPERATION CONTROL CARD (2I5)

Columns	Variable	Notes	Entry
1-5	**ICOH**		= 1, Incoherency is computed using Luco's model = 2, Incoherency is computed using Abrahamson's model
6-10	**NPRTX**		= 1, Coherency values at each projected node for each frequency are printed in the optional file SASSI.ICH, this is a large output

and is only needed for detailed checking purposes.

= 0, No value is printed.

E.3 MODEL CONTROL CARD (6F10.0)

Columns	Variable	Notes	Entry
1-10	γ_x	(1)	Luco's model incoherency parameter γ_x
11-20	γ_y	(1)	Luco's model incoherency parameter γ_y
21-30	γ_z	(1)	Luco's model incoherency parameter γ_z
31-40	**Cs**	(2)	Luco's model shear wave velocity $\mathbf{V_s}$ or Apparent Wave velocity in Abrahamson's Model
41-50	**NCSI**	(3)	Number of incoherency modes to be considered, =0 all modes are used
51-60	α	(1)	Luco's model parameter α

Notes:

(1) See Luco's model for definition of parameters, Applicable when ICOH = 1 is selected, leave blank for ICOH =2.

(2) The parameter Cs defines the soil shear wave velocity when Luco's model is used. It also defines the apparent wave velocity when Abrahamson's model is used.

(3) Typically first few modes are sufficient to characterize the ground motion incoherency. However if a value of "0" is used all modes will be used.

E.4 INCOHERENCY MODEL PARAMETERS

Applicable for both **ICOH** =1 and 2.

E.4.1 NUMBER OF EMBEDMENT LAYERS (I5)

Columns	Variable	Notes	Entry
1-5	**NEMBD**	(1)	Number of layers in which the structure foundation is embedded, use "0" for surface foundation. Maximum value is 20

E.4.2 MODEL EMBEDMENT ELEVATION (F10.0)

Provide **NEMBD** cards. Skip if **NEMBD = 0**

Columns	Variable	Notes	Entry
First card			
1-10	**ELV1**	(2)	Z-coordinate of bottom of the first soil layer for the model embedment
Second card			
1-10	**ELV2**	(2)	Z-coordinate of bottom of the second soil layer for the model embedment

Notes:

(1) **NEMBD** is the number of soil layers where structure is embedded in the soil.

(2) **ELV1, ELV2,** … are the Z-coordinates of the bottom layer interface of soil layers in which the structure is embedded.

E.5 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode = 0, stop no more data

Notes:

(1) Must be the last data card.

5.2.6 MOTOR

For each specified frequency, the program module **MOTOR** forms the elements of the load vector in Eq. (2.1-3) which corresponds to external forces such as impact loads and rotating machinery acting directly on the structure. The results are stored on Tape 9.

F.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			Blank
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If **NOPT** < 0, most of the calculations required during normal execution are bypassed.

F.2 MASTER CONTROL CARD (2I5)

Columns	Variable	Notes	Entry
1-5	**NLP**		Total number of loaded points
6-10	**NF**		Total number of frequencies of analysis

F.3 SYSTEM OF UNITS CARD (F10.0)

Columns	Variable	Notes	Entry
1-10	**GRAV**		Acceleration of gravity

F.4 FREQUENCY DATA CARDS

F.4.1 FREQUENCY CONTROL CARD (2F10.0, I5)

Columns	Variable	Notes	Entry
1-10	**DF**	(1)	Frequency step (HZ)
11-20	**DT**	(1)	Time step of control motion (sec)
21-25	**NFFT**	(1)	Number of values to be used in Fourier Transform of control motion; must be a power of 2

F.4.2 FREQUENCY CONTROL CARDS (16I5)

Columns	Variable	Notes	Entry
1-5	**NFR(1)**	(2)	Frequency no. I
6-10	**NFR(2)**		Frequency no. 2
11-15	**NFR(3)**		Frequency no. 3
•	•		•
•	•		•
•	•		•

Notes:

(1) Case A - Deterministic Analysis

DT and **NFFT** for the dynamic input motion must be given, and the frequency step may be left blank. The program will compute the frequency step as follows:

$$DF = \frac{1}{(INFFT * DT)}$$

Then this frequency step may be used to set up frequency numbers in Section **D.4.2.**

Case B - Probabilistic Analysis/Harmonic Machine Vibration Analysis

DF must be given and may be directly used to set up frequency numbers in Section **D.4.2** in which case **DT** and **NFFT** are never used and may be left blank.

(2) The total of **NF** frequency numbers must be given. All the frequency numbers must be positive nonzero integer numbers. The program will automatically reorder the input frequency numbers in ascending order and will stop if two or more equal frequency numbers are detected. Frequencies f_i, for which solutions are obtained, are defined as follows:

$$f_i = NFR(i) * DF$$

The highest frequency of analysis is then equal to the highest frequency number multiplied by the frequency step.

F.5 CONCENTRATED DYNAMIC LOAD DATA CARDS

F.5.1 FIRST CARD - LOAD FACTORS (I5, 5X, 6F10.0, 2I5)

Columns	Variable	Notes	Entry
1-5	**NODE**	(1)	Nodal point number
11-20	**AMPL(1)**	(2)	Force factor in x-direction
21-30	**AMPL(2)**		Force factor in y-direction
31-40	**AMPL(3)**		Force factor in z-direction
41-50	**AMPL(4)**		Moment factor in x-direction
51-60	**AMPL(5)**		Moment factor in y-direction
61-70	**AMPL(6)**		Moment factor in z-direction
71-75	**KN**	(3)	Node number increment
80	**KT**	(2)	Arrival time code
			= 0, zero arrival time
			= -1, nonzero arrival time

F.5.2 SECOND CARD - LOAD ARRIVAL TIMES (6F10.0)

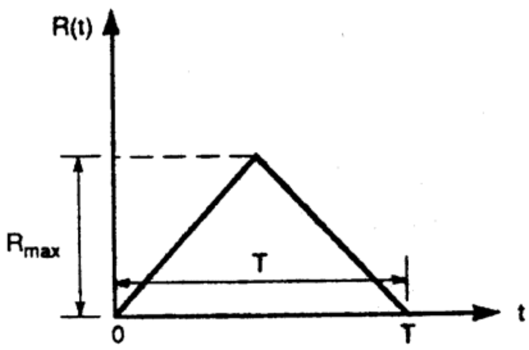
Skip this card if no arrival time at this node (**KT** = 0).

Columns	Variable	Notes	Entry
1-10	**DTX**	(2)	Force arrival time in x-direction (sec)
11-20	**DTY**		Force arrival time in y-direction (sec)
21-30	**DTZ**		Force arrival time in z-direction (sec)
31-40	**DTXX**		Moment arrival time in x-direction (sec)
41-50	**DTYY**		Moment arrival time in y-direction (sec)
51-60	**DDTZZ**		Moment arrival time in z-direction (sec)

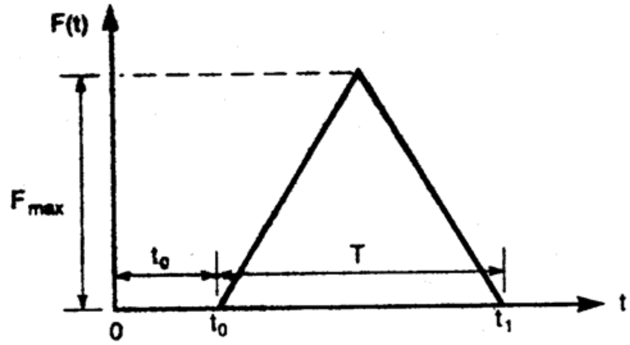
Notes:

(1) One card must be given for every nodal point with nonzero applied dynamic force or moment. Nodal cards need not be in node order sequence. If nodal cards are defined more than once, the new force or moment will be added up to current values at that node.

(2) Dynamic loads with similar time history but different maximum amplitude and arrival time may be applied at nodal points. Reference time history is defined as having maximum reference amplitude and zero arrival time, which means the load starts acting on the nodal point at time zero. All other time histories must be given relative to reference time history by defining load factor and arrival time. Load factor is defined as the ratio between maximum amplitude of applied load to maximum reference amplitude. See figure below.



Reference Time History



Arbitrary Time History

where

Load factor = $\frac{F_{max}}{R_{max}}$

Arrival time = t_o

(3) Generation is possible for a number of nodal points omitted in a series. If $NODE_1$..., KN_1, KT_2 are the information on the first and last cards in a series, the first generated node will be $NODE_1 + KN_2$ and second generated node will be $NODE_1 + 2 * KN_2$ etc. Generation will continue until node number $NODE_2 - KN_2$ is established. Therefore, $NODE_2 - NODE_1$ must be evenly divisible by KN_2. Load factors and arrival times of forces and moments applied at generated nodes are set equal to the values given on the first card in the series. If no generation is required, leave KN blank for all nodal point cards.

F.6 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop no more data

Notes:

(1) Must be the last data card.

5.2.7 ANALYS

The program module **ANALYS** is the heart of the program SASSI. It drives the three subprograms MATRIX, LOAD, and SOLVE and therefore controls the restart modes of the program.

The program basically functions in four different modes. The first mode is the initiation mode and the other three are the restart modes of the program.

5.2.7.1 Mode 1 - Initiation

This is the first mode to be executed for a new problem. In this mode the program basically reads the three input tapes - Tape 1 (or Tape 9), Tape 3, and Tape 4 - and generates the output tapes - Tape 5, Tape 6, and Tape 8. Tape 5, Tape 6, and Tape 8 contain the impedance matrices of the interaction nodes, the reduced modified stiffnesses of the total system, and the acceleration (or displacement) transfer functions relative to the control motion (input dynamic force), respectively, computed for the specified frequencies which must reside on the input tapes. This mode involves performing all the operations given in Step 1 through Step 7 in Section **2.5**. In addition to the three output tapes, it is also possible to request the printout of the uninterpolated transfer functions.

It should be noted that the control motion defined by the user (on Tape 1) for seismic problems is in the **SITE** coordinate system x'y'z' and must be transformed to the global structural coordinate system xyz in **ANALYS**. Therefore, the user must enter the angle between two coordinate systems in program module **ANALYS**. In addition to this, the user also enters the location of the control point on the horizontal plane at this stage.

5.2.7.2 Mode 2 - new superstructure

If the physical properties of the structures are changed or the geometry of the structure is altered, then, as long as other data remain intact, program module **ANALYS** can be restarted using Mode 2. This mode requires only part of program **ANALYS** to be re-executed.

In order to use this mode, a new Tape 4 has to be generated by re-executing program module **HOUSE**. Since the geometry and numbering of the nodal points below the ground have not changed, the impedance matrices can be recovered from Tape 5. The information on the two input tapes is then used to compute the new reduced modified stiffness of the structure to be saved on a new Tape 6. Tape I (or Tape 9) is also input so that the new transfer functions of the response can be computed and then saved on a new Tape 8.

This mode involves the same operations as Mode 1 except that Step 3 is skipped in Section **2.5**.

5.2.7.3 Mode 3 - new seismic environment

If the seismic environment is changed, the information on Tape 1 will change while Tapes 5 and 6 remain intact. Therefore, if Tape 5 and Tape 6 had been created and saved from some previous analysis, program **ANALYS** could be restarted in Mode 3 by inputting a new Tape 1 and recovering the information on Tapes 5 and 6. The results are saved on a new Tape 8.

This mode involves only the operations given in Steps 6 and 7 of Section **2.5**.

5.2.7.4 Mode 4 - new dynamic loading

If the problem is to be analyzed for a new set of external forces, then the program module **ANALYS** can be restarted by using Mode 4. This mode is very similar to Mode 3 except that it is for foundation vibration rather than seismic problems. Therefore, Tape 5 is not required as input. Only Tape 6 and a new Tape 9 are required. The results are also stored on a new Tape 8.

The mode involves operations similar to those in Mode 3.

G.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			
9-80	**HED**		Contain information to be printed with output

Notes:

(1) If **NOPT** < 0, most of the calculations required during normal execution are bypassed.

G.2 MASTER CONTROL CARD (6I5)

Columns	Variable	Notes	Entry
5	**MEOF**		Type of analysis
			= 1, seismic analysis
			= 2, foundation vibration analysis
10	**MODE**	(I)	Mode of analysis
			= 1, Mode I
			= 2, Mode 2
			= 3, Mode 3
			= 4, Mode 4
15	**MSAVE**	(2)	Tape save option
			= 0, do not save Tape 6
			= 1, save Tape 6
16-20	**NPRINT**	(3)	Print option
			= 0, don't print transfer functions < 0, print transfer functions for all non-fixed nodes
			= n, print transfer function for n nodes only

21-25	**NUMFR**	(4)	> 0, total number of frequencies of analysis
			= 0; NUMFR and frequency numbers are taken from Tape 1 (or Tape 9)
26-30	**ICOH**	(5)	= 0, no ground motion incoherency is considered
			= 1, ground motion incoherency is considered

Notes:

(1) The program may be run in one of the following modes shown in the following table.

Mode of Analysis	Case	Tapes to be Mounted	Tapes to be Created
1	Initiation	1 (or 9), 3, 4	5, 6, 8
2	Restart - new superstructure (new Tape 4)	1 (or 9), 4, 5	6, 8
3	Restart - new seismic environment (new Tape 1)	1, 5, 6	8
4	Restart - new dynamic loading (new Tape 9)	9, 6	8

As seen from the preceding table, Mode 3 will run only for the case of seismic analysis (**MEOF** = 1) while Mode 4 will run only for the case of foundation vibration analysis (**MEOF** = 2).

(2) This option will allow the user to save Tape 6 (reduced complex stiffness of the system) in order to be able to restart Mode 3 and Mode 4 at a later time.

(3) If **NPRINT** < 0, the program will print out the response of all the nodal points to all frequencies for which the solution has been obtained (all non-fixed nodes). If **NPRINT** = 0, no response will be printed. If **NPRINT** = n, only the response for n nodes will be printed.

(4) Frequencies for which a complete solution is desired must be specified at this stage. The program will automatically survey these frequencies to make sure that they reside on the tapes that are mounted before starting the execution. If one or more frequencies are not found on the mounted tapes, the program will stop.

It is also possible to break the complete frequency array into smaller groups and then run each group separately. Later results of these separate runs can be combined into the complete solution. This has the advantage that the runs will be smaller and the created files will occupy smaller space in the computer during the program execution.

This option also makes it possible to solve the problem for new frequencies and combine the results with those of old frequencies if the analysis so demands at a later time.

Since more than 90% of the cost of the complete analysis goes to this program, assessment of the final cost of the job can be made in advance by estimating the cost of doing one single frequency and multiplying it by the number of frequencies for which a solution is desired.

If NUMFR = 0, then Section **G.3** must be skipped and the program will take NUMFR and frequency numbers of analysis from Tape 1 (or Tape 9).

(5) If ICOH = 1, the free-filed motion will be modified based on the incoherency model selected in program module INCOH. Tape SASSI.T11 generated by INCOH will be used as input to ANALYSIS.

G.3 NODAL POINT CARDS (16I5)

Skip this card if **NPRINT** ≤ 0, otherwise provide total number of **NPRINT** node numbers (limited to 1000 nodes).

Columns	Variable	Notes	Entry
1-5	**NODALP(1)**	(1)	Node 1
6-10	**NODALP(2)**		Node 2
•	•		•
•	•		•
•	•		•

G.4 FREQUENCY NUMBER CARDS (16I5)

Skip this section if **NUMFR** = 0.

Columns	Variable	Notes	Entry
1-5	**NFR(1)**	(1)	Frequency no. 1.
6-10	**NFR(2)**		Frequency no. 2
11-15	**NFR(3)**		Frequency no. 3
•	•		•
•	•		•
•	•		•

Notes:

(1) Total of **NUMFR** frequency numbers must be given. All the frequency numbers must be positive nonzero integer numbers. The program will automatically reorder the input frequency numbers in ascending order and will stop if two or more equal frequency numbers are detected.

If **NUMFR** = 0, skip this section and the program will obtain **NUMFR** and frequency numbers from Tape 1 (or 9).

Frequency step, time step, and number of FFT points are obtained from Tape 1 (or 9) and are checked against other input tapes for consistency. This information, along with frequency numbers at this section, will pass on to every tape created.

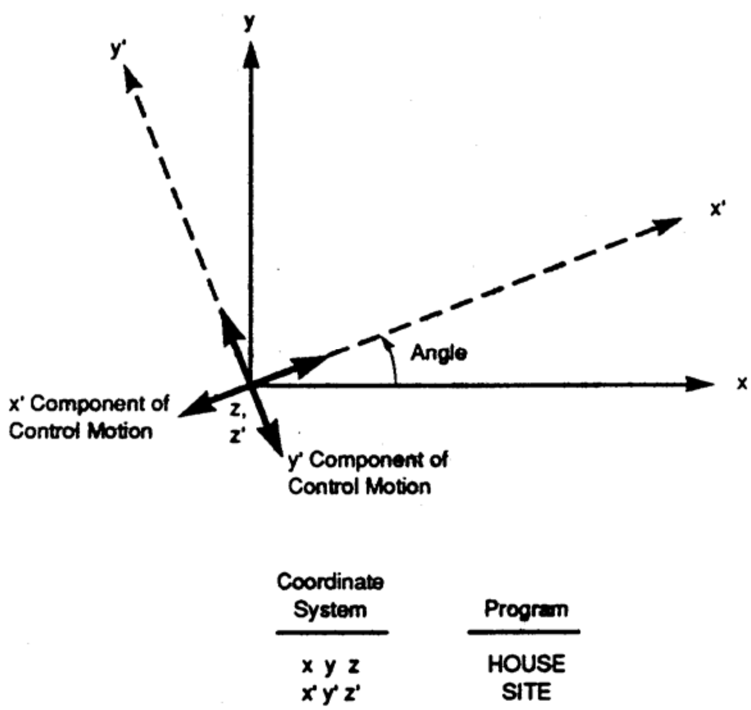
G.5 CONTROL MOTION CARD FOR SEISMIC ANALYSIS (3F10.0)

Skip this section if foundation vibration analysis (**ME0F** = 2).

Columns	Variable	Notes	Entry
1-10	**XCNTR0L**		x-coordinate of control point
11-20	**YCNTROL**		y-coordinate of control point
21-30	**ANG**	(1)	Coordinate transformation angle (degrees)

Note:

(1) This is the angle between x' component of the control motion (as defined in program **SITE**) and x coordinate of the system.



5.2.8 COMBIN

The program module **C0MBIN** combines the frequencies on two Tape 8's created in the program **ANALYS** and creates a new Tape 8 which now contains the frequencies from both tapes.

The old Tape 8's must be mounted as Tape 21 and Tape 22 and the results will be written on Tape 8. No other input data are required by this program.

If two equal frequencies are encountered on the mounted tapes, the program will select the one which resides on Tape 21 and discard the other one.

H.1 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop no more data

Notes:

(1) Must be the last data card.

5.2.9 MOTION

The main function of the program module **MOTION** is to compute acceleration, velocity, and displacement (only for foundation vibration) time histories of the response. It may also output transfer functions and response spectra. The program also provides punched output or printer plots of the response at user's option.

In case of seismic problems, the program reads the acceleration time history of the control motion from cards and transforms it to the frequency domain using Fast Fourier Transform techniques. It then reads the uninterpolated transfer functions from Tape 8 for selected output points, performs the interpolation and the convolution with the control motion, and returns to the time domain using the inverse Fast Fourier Transformer algorithm. The resulting time histories of acceleration or computed velocity may be output directly or converted to output response spectra. The module reads the geometry of the structural model from Tape 4 to perform the transformations needed to calculate the motion of the nodes with constrained degrees of freedom.

In addition to the above capabilities, if Option 2 is selected, MOTION computes the relative displacements between the reference node and any other sets of nodes in the model in the direction specified by the user. The reference node may also be the free-field input motion (control motion at control point location). The relative displacement is calculated from the difference in the response of the 2 nodes in the frequency domain and converted to time domain. Both maximum relative displacements and the time history of relative displacements may be requested. If Option 2 is selected, other responses such as maximum acceleration, acceleration response spectra, etc. associated with option 1 will not be calculated. Relative displacement calculation is available to seismic problems only.

I.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**NOPT**		Operation mode
			= 1, complete solution
			= 2, relative displacement only
		(1)	< 0, data check only
2-8			Blank
9-80	**HED**		Contain information to be print with output

Notes:

(1) If **NOPT** <0, most of the calculations required during execution are bypassed.

I.2 OUTPUT CONTROL CARD (6I5, F10.0, I5, 4X, A1)

Columns	Variable	Notes	Entry
5	**NTIME**	(1)	Analysis type code = 0, only transfer functions to be output = 1, otherwise
6-10	**NOUT**		Total number of nodal points where output is required
11-15	**ND**	(2)	Number of constant damping values for response spectrum analysis; skip if **NTIME** = 0
16-20	**NSKIP**		Output code for all time histories; skip if **NTIME** = 0 = 0, only table to be printed > 1, plot every **NSKIP**-th point
21-25	**N411**	(3)	Pairs of frequency-dependent damping values for acceleration response spectra computation
26-30	**N75**	(4)	= 0, do not include 75 standard frequencies in response spectra computation = 1, include 75 standard frequencies in response spectra computation
31-40	**DUR**		Total duration of time histories to be plotted; skip if **NTIME** = 0

41-45	**NREF**	(5)	skip if **NOPT** is 0 or 1 > 0, the reference node number = 0, free-field input motion is used as reference motion
50	**IEQM**	(5)	skip if **NOPT** is 0 or 1 “X”, or “Y”, or “Z” for direction of relative displacement calculation

Notes:

(1) If **NTIME** = 0, only transfer function plots may be requested at nodal points and time history of input motion need not be supplied.

(2) A set of constant damping values can be specified for response spectra computation. The same damping values will be used for all spectral frequencies.

(3) For certain applications, such as piping analysis, a set of frequency-dependent damping values can be specified for response spectra computation. The most common set is ASME N-411 damping values for piping analysis. The parameter **N411** specifies the number of pairs of damping versus frequency points to be entered in Section **G.4.3**. If **N411** is greater than 0, **ND** should be at least 1 or greater in order to have frequency points selected.

(4) Both ASCE 4-98 and US NRC SRP 3.7.1 specify 75 standard frequencies in addition to structural frequencies for response spectra computation. With **N75** = 1, the 75 frequencies will be included in the set of frequencies for response spectra computation.

(5) **NREF** and **IEQM** define the reference node number and the direction of interest for calculation of relative displacement. If **NREF** is specified to be 0, the input motion at control point will be used as reference motion. The **IEQM** specifies the direction of the response for both the reference node and other nodal points for which

relative displacement are computed. For example if “X” is specified, the relative displacement in the X-direction between the selected node and the reference node will be computed.

I.3 OUTPUT REQUEST CARDS

I.3.1 OUTPUT CONTROL CARD (I5)

Skip this card if seismic analysis (**MEOF** = 1).

Columns	Variable	Notes	Entry
1-5	**NTYPE**	(1)	Type of response
			= 1, output only displacements
			= 2, output only velocities
			= 3, output only accelerations

I.3.2 NODAL OUTPUT CONTROL CARDS (I5, 4X, 6I1, 4X, 6I1, 4X, 6I1)

Columns	Variable	Notes	Entry
1-5	**NODE(I)**	(2)	Nodal point number where output is required
10-15	**KEY(I,J)**	(3)	Output control key for response in x-direction
20-25	**KEY(I,J)**		Output control key for response in y-direction
30-35	**KEY(I,J)**		Output control key for response in z-direction
40-45	**KEY(I,J)**		Output control key for response in xx-direction
50-55	**KEY(I,J)**		Output control key for response in yy-direction

60-65	**KEY(I,J)**	Output control key for response in zz-direction

Notes:

(1) Only accelerations can be requested for seismic problems. **NTYPE** controls the type of response to be output for the foundation vibration problem.

(2) Total of **NOUT** cards must be, given in this section. Nodal point cards where output is requested need not be in node order sequence; the program will reorder them automatically. Also, output control keys in directions which are constrained will be ignored.

(3) Output control keys are six-digit integer numbers (ijklmn) which are defined as follows:

i = 1--- plot transfer function* , transfer function values will be saved on Tape 13, (SASSI.T13)
= 0--- otherwise

j = 1--- save time history of requested response on Tape 12** , (SASSI.T12)
= 0--- otherwise

k = 1---plot time history of requested response**
= 0--- otherwise

l = 1--- plot acceleration and velocity response spectra***
= 0--- otherwise

m = 1---save acceleration and velocity response spectra on Tape 16*** , (SASSI.T16)
= 0---otherwise

n = 1---print maximum requested response**
= 0---otherwise

If **NOPT** = 2 (relative displacement calculation), the input for i, l, m should be left blank.

If J = 1, time histories of the requested response are computed and stored on Tape 12. This tape can later be used to recover time histories for plotting purposes. The following example shows how the time histories are obtained from Tape 12:

```
DIMENSION A(NFFT)
DO 1000 N=1, NN

      READ (12,100) DIR, NODE, NFFT, (A(I),I=1,NFFT)
100   FORMAT (////29X,A2,29X,I4/17X,I5//(8F9.6))
      •
      •
      •
1000 CONTINUE
```

where

NN = total number of time histories
DIR = direction of motion
NODE = nodal point number to which the time history belongs
NFFT = total number of points in the time history
A = vector containing the time history of the response

---

* Transfer function for seismic problems is defined for total acceleration response while for foundation vibration problems it is defined for total displacement response.

** Requested response for seismic problems is acceleration; for foundation vibration problems it is determined by NTYPE.

*** Response spectra is computed independent of the value of NTYPE. Therefore, displacement response spectra cannot be requested.

I.4 RESPONSE SPECTRA DATA CARDS

Skip this section if **ND** = 0 or **NOPT** = 2.

I.4.1 FIRST CARD (2F10.4, I5)

Columns	Variable	Notes	Entry
1-10	**FSTRT**	(1)	First frequency used in response spectrum analysis--HZ
11-20	**FLAST**	(1)	Last frequency used in response spectrum analysis--HZ
21-25	**NINT**	(2)	Total number of frequency steps for response spectra. NINT + I spectral values are computed

I.4.2 SECOND CARD (8F10.4)

Columns	Variable	Notes	Entry
1-10	**DAMP(1)**	(3)	Damping ratio used in response spectra
•	•		•
•	•		•
•	•		•

Notes:

(1) First and last frequency points for response spectra computation are specified. Leave blank if **N75** = 1 on card I.2. If **N75** = 0 and card I.4.1 is left blank, standard values (**FSTRT** = 0.4, **FLAST** = 40, **NINT** = 40) are assumed. This will lead to a plot which fills one page.

(2) If **N75** = 0, **NINT** +1 frequencies between **FSTRT** and **FLAST** frequencies will be generated and used in the response spectra computation. If **N75** = 1, additional frequencies beyond 75 standard frequencies for the total of **NINT** frequencies should be specified. The additional frequency points are read from Tape 15 (SASSI.T15) with the format of 8F10.0. For example, if **N75** = 1 and **NINT** = 90, additional 15 frequency points should be specified on Tape 15.

(3) Total of **ND** damping ratios must be given.

I.5 VARIABLE DAMPING FOR RESPONSE SPECTRA

Skip this section if **NOPT** = 2.

Skip this section if **N411** = 0, otherwise provide **N411** frequency and damping values.

I.5.1 FREQUENCY CARD (8F10.0)

Columns	Variable	Notes	Entry
1-10	**FRQ(1)**		First frequency point
11-20	**FRQ(2)**		Second frequency point
•	•		•
•	•		•
•	•		•

I.5.2 DAMPING CARD (8F10.0)

Columns	Variable	Notes	Entry
1-10	**DN411(1)**		Damping value for first point
11-20	**DN411(2)**		Damping value for second point
•	•		•
•	•		•
•	•		•

I.6 INPUT MOTION DATA CARDS

Skip this section if **NTIME** = 0.

I.6.1 CONTROL CARD (2I5, 3F10.0)

Columns	Variable	Notes	Entry
1-5	**NFFT**	(1)	Number of values to be used in Fourier Transform
6-10	**NEQZ**		Number of acceleration (or force) values to be read from cards
11-20	**DT**	(2)	Time step -- sec
21-30	**EQMUL**		Multiplication factor for scaling time history. Use only if **UGMAX** = 0; leave blank otherwise
31-40	**UGMAX**		Maximum value of time history to be used. The values of time history will be scaled to give maximum value = **UGMAX**. Use only if **EQMUL** = 0; leave blank otherwise

Notes:

(1) **NFFT** must be a power of 2 and includes the trailing zeroes. If this number does not agree with the **NFFT** value in **SITE** (or **MOTOR**), the program will stop.

(2) The program will compute the frequency step from the following formula:

$$DF = \frac{1}{(NFFT * DT)}$$

If **DF** and **DT** do not agree with those in **SITE** (or **MOTOR**), the program will stop.

I.7 CONTROL TIME HISTORY

The input for this section should be provided on Tape 14 (SASSI.T14).

I.7.1 CONTROL TIME HISTORY ID CARD (18A4)

Columns	Variable	Notes	Entry
1-72	**ID(I)**		Identification for control time history

I.7.2 CONTROL TIME HISTORY VALUES (8F9.6/8E10.3)

Total of (NEQZ + 7)/8 cards.

Columns	Variable	Notes	Entry
1-72	**A(I)**	(1)	8 values

Notes:

(3) All time history values must be given at time steps **DT**. For seismic problems, provide acceleration values in (G) in (8F9.6) format; for foundation vibration problems, provide force values in (8EI0.3) format.

I.8 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode
			= 0, stop no more data

Notes:

(1) Must be the last data card.

5.2.10 STRESS

The main function of the program module STRESS is to compute and output maximum stress, forces, or moments in the elements. The user may also request time histories of these components to be printed out and also saved on a magnetic tape.

The program reads acceleration (displacement) transfer functions from Tape 8 and information about elements from Tape 4. Then, for each requested element, the program calculates the stress, force, or moment components at each frequency, performs interpolation and convolution with the control motion, and finds the corresponding time histories by returning to the time domain using the inverse Fast Fourier Transform algorithm.

Time history of octahedral shear stress (strain), which is a measure of maximum shear stress (strain), is calculated in time domain for each time interval DT. From this, the effective shear strain (0.65 * maximum shear strain) for the soil elements can be estimated and used to find new strain-compatible soil properties.

J.1 OPERATION MODE CARD AND TITLE (I5, 3X, 18A4)

Columns	Variable	Notes	Entry
1-5	**N0PT**		Operation mode
			= 1, complete solution
		(1)	< 0, data check only
6-8			Blank
9-80	**NED**		Contain information to be printed with output

Notes:

(1) If **N0PT** < 0, most of the calculations required during normal execution are bypassed.

J.2 MASTER CONTROL CARD (2I5, 4X, I1)

Columns	Variable	Notes	Entry
1-5	**NGOUT**	(1)	Total number of element groups
10	**ITER**	(2)	Iteration control key = 1, automatic computation of strains in all soil elements (not available) = 0, otherwise
15	**IFPU**		Output control key = 1, save stress time histories on Tape 12 = 0, otherwise

Notes:

(1) This is the total number of element groups for which output is requested.

(2) If secondary nonlinear effects are going to be considered, then let **ITER** = 1; otherwise leave blank.

J.3 ELEMENT GROUP CARDS

One set of cards for each element group requested in Section **J.2** must be provided as follows:

J.3.1 THREE-DIMENSIONAL SOLID ELEMENTS

Skip this section if no request for stresses or strains in 3-D solid elements.

J.3.1.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE(1)**		The number 1
6-10	**ICODE(2)**	(1)	The order number
11-15	**ICODE(3)**	(2)	Total number of elements in this group for which output is requested

J.3.1.2 OUTPUT REQUEST CARDS (I5, 4X, 7I1, 13X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**Key(1)**	(3)	Output control key for stress (strain in xx-direction
11	**Key(2)**		Output control key for stress (strain in yy-direction
12	**Key(3)**		Output control key for stress (strain in zz-direction
13	**Key(4)**		Output control key for stress (strain in xy-direction
14	**Key(5)**		Output control key for stress (strain in xz-direction
15	**Key(6)**		Output control key for stress (strain in yz-direction
16	**Key(7)**		Output control key for octahedral shear stress (strain)
30	**KN**	(4)	Generation control key = 0, no generation = 1, otherwise

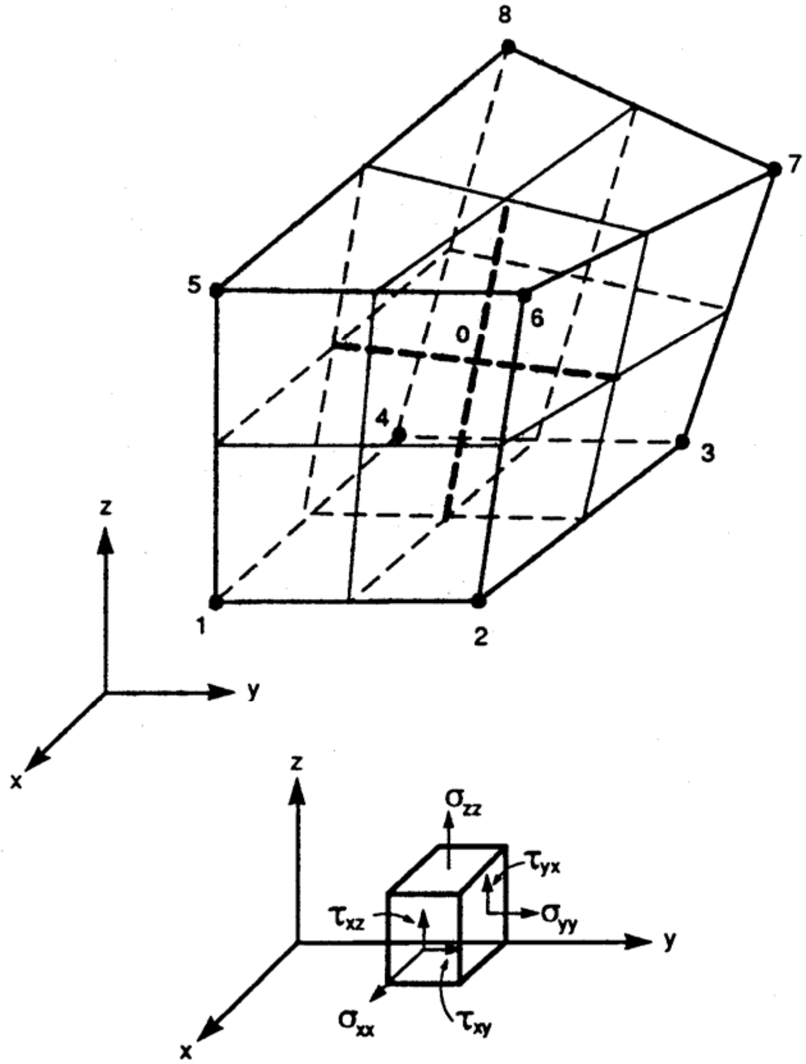
Notes:

(1) If there is more than one group of 3-D solid elements, specify the order of the requested group, e.g., **ICODE**(2) = 2 will request the second set of 3-D solid elements. The order number for each element group is printed in the **HOUSE** output. If left blank, the program will select the first group of solid elements it encountered.

(2) This number must be less than or equal to the total number of elements in this group. Otherwise the program will stop.

(3) KEY = 0, no request for this component
= 1, print only maximum response and save maximum values on Tape 18, (SASSI.T18)
= 2, print maximum response and save response time history on Tape 12, (SASSI.T12)

The stresses in 3-D solid elements are computed at the centroid of the element and are referred to in global axes. These stresses are shown in the following figure.



If KEY=2, time histories of the requested response are computed and stored on Tape I2. This tape can later be used to recover the time histories for plotting purposes. The following example shows how the time histories are obtained from Tape I2:

```
        DIMENSION A(NFFT)
        DO 1000 N=1,NN
        READ (12,100) N1,N2,DIR, (A(I),I=1,NFFT)
100     FORMAT (2II0,A4/(8EIO.3))
          •
```

•

•

1000 CONTINUE

where

NN = total number of requested time histories

N1 = element number

N2 = element group number

NFFT = total number of points in the time history

A = array containing the time history of the response

In addition to the user-requested response, the program nay also output other components of the response if calculation of these components is necessary in order to determine the requested response. For example, in order to output maximum octahedral shear stress for 3-D solid elements, all six components of stress (σ_{xx}, σ_{yy}, σ_{zz}, τ_{xy}, τ_{xz}, τ_{zx}) must be calculated. Therefore, the program will also output the maximum response of these components. However, the corresponding time histories will not be saved on Tape 12 unless they are specifically requested by the user.

(4) Generation is possible for a number of elements omitted in a series. Suppose (N_1,...,KN_1) and (N_2,...,KN_2) are the information on two consecutive cards--if KN_1 is equal to 1, all the elements between N_1 and N_2 will be generated and the output control keys for the generated elements will be set equal to the values given on the first card. On the other hand, if KN_1 is equal to 0, no generation between N_1 and N_2 will take place.

J.3.2 THREE-DIMENSIONAL BEAM ELEMENTS

Skip this section if no request for forces or moments in 3-D beam elements.

J.3.2.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE(1)**		The number 2
6-10	**ICODE(2)**	(1)	The order number
11-15	**ICODE(3)**	(2)	Total number of elements in this group for which output is requested

J.3.2.2 OUTPUT REQUEST CARDS (I5, 4X, 12I1, 8X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**KEY(1)**	(3)	Output control key for force in 1-direction (node I)
11	**KEY(2)**		Output control key for force in 2-direction (node I)
12	**KEY(3)**		Output control key for force in 3-direction (node I)
13	**KEY(4)**		Output control key for moment in 1-direction (node I)

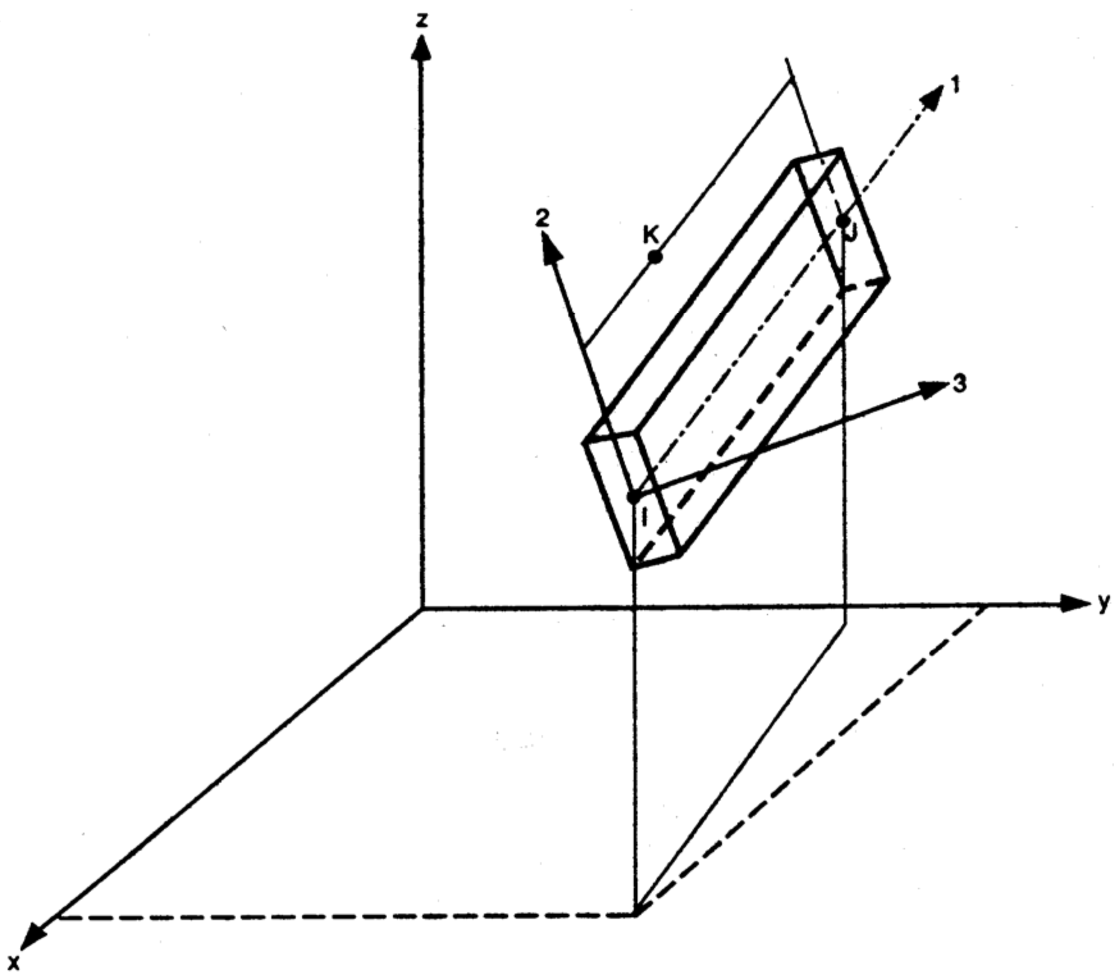
14	**KEY(5)**		Output control key for moment in 2-direction (node I)
15	**KEY(6)**		Output control key for moment in 3-direction (node I)
16	**KEY(7)**		Output control key for force in 1-direction (node J)
17	**KEY(8)**		Output control key for force in 2-direction (node J)
18	**KEY(9)**		Output control key for force in 3-direction (node J)
19	**KEY(10)**		Output control key for moment in 1-direction (node J)
20	**KEY(11)**		Output control key for moment in 2-direction (node J)
21	**KEY(12)**		Output control key for moment in 3-direction (node J)
30	**KN**	(4)	Generation control key = 0, no generation = 1, otherwise

Notes:

(1) See note I of Section **J.3.1**

(2) See note 2 of Section **J.3.1**

(3) See note 3 of Section **J.3.1**. The forces and moments in beam elements are computed at the end and are referenced in local beam axes, in the following figure.



(4) See note 4 of Section **J.3.1**.

J.3.3 PLATE/THIN SHELL ELEMENTS

Skip this section if no request for forces or moments in plate elements.

J.3.3.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE(1)**		The number 3
6-10	**ICODE(2)**	(1)	The order number
11-15	**ICODE(3)**	(2)	Total number of elements in this group for which output is requested

J.3.3.2 OUTPUT REQUEST CARDS (I5, 4X, 6I1, 14X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**KEY(1)**	(3)	Output control key for force component resultant $S_{x'x'}$
11	**KEY(2)**	(3)	Output control key for force component $S_{y'y'}$
12	**KEY(3)**	(3)	Output control key for force component $S_{x'y'}$
13	**KEY(4)**	(3)	Output control key for moment component $M_{x'x'}$

14	**KEY(5)**	(3)	Output control key for moment component $M_{y'y'}$
15	**KEY(6)**	(3)	Output control key for moment component $M_{x'y'}$
30	**KN**		(4) Generation control key
			= 0, no generation
			= 1, otherwise

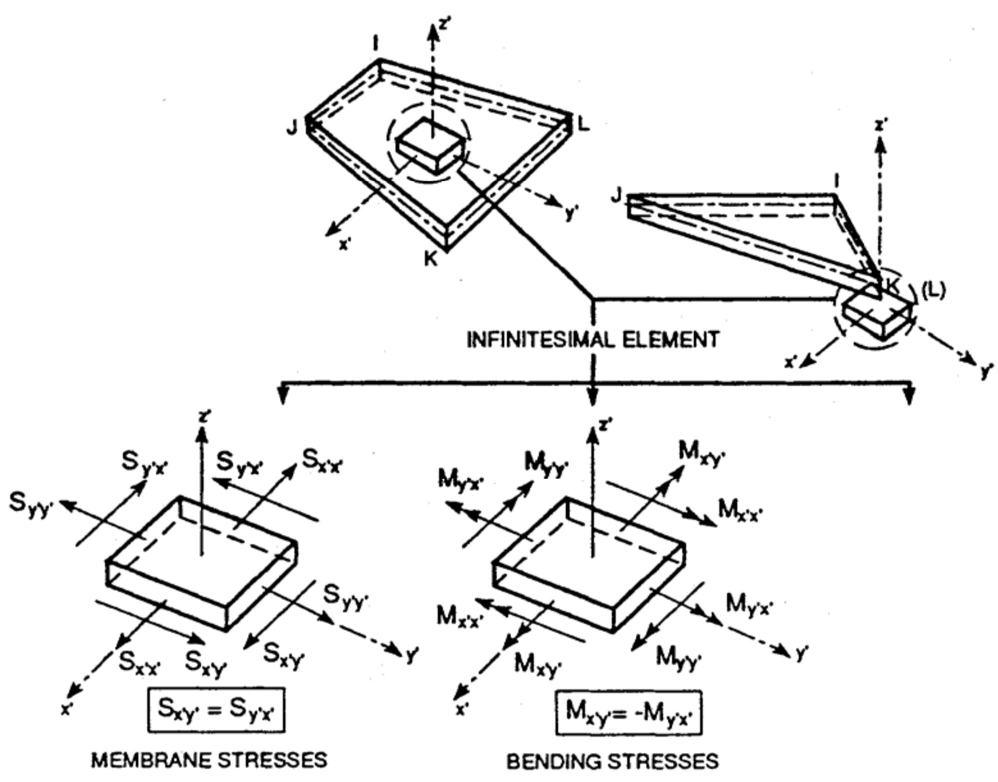
Notes:

(1) See note I of Section **J.3.1**.

(2) See note 2 of Section **J.3.1**.

(3) The membrane force and bending moments are computed with respect to local element coordinates. The forces are in units of force/area (F/L^2) and the moments are in moment/length (F.L/L). The location of the infinitesimal elements where the forces and moments are computed and the positive definitions of each component are shown in the figure.

POSITIVE PLATE ELEMENT OUTPUT FORCES



(1) See note 4 of Section **J.3.1**.

J.3.4 TWO-DIMENSIONAL FINITE ELEMENTS

Skip this section if no request for stresses or strains in 2-D finite elements.

J.3.4.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE(1)**		The number 4
6-10	**ICODE(2)**	(1)	The order number
11-15	**ICODE(3)**	(2)	Total number of elements in this group for which output is requested

J.3.4.2 OUTPUT REQUEST CARDS (I5, 4X, 3I1, I7X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**KEY(1)**	(3)	Output control key for stress (strain) in yy-direction
11	**KEY(2)**		Output control key for stress (strain) in zz-direction
12	**KEY(3)**		Output control key for stress (strain) in yz-direction
30	**KN**	(4)	Generation control key

= 0, no generation

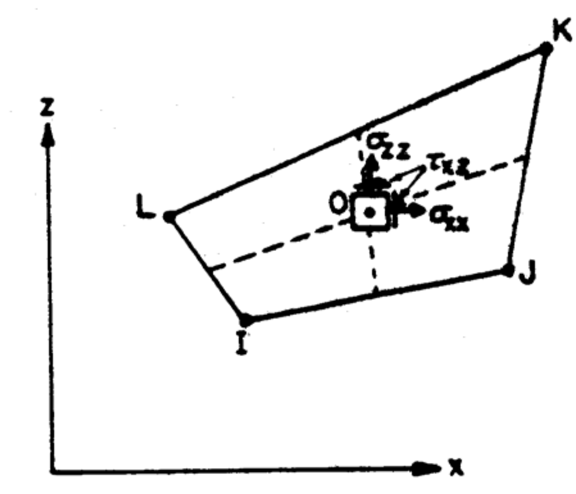
= 1, otherwise

Notes:

(1) See note 1 of Section **J.3.1**.

(2) See note 2 of Section **J.3.1**.

(3) See note 3 of Section **J.3.1**. The stresses in 2-D finite elements are computed at the center of the element and are referred to in global axes. These stresses are shown in the next figure.



(4) See note 4 of Section **J.3.1**.

J.3.5 THREE-DIMENSIONAL PILE ELEMENTS

Not available.

J.3.6 TWO-DIMENSIONAL PILE ELEMENTS

Not available.

J.3.7 THREE-DIMENSIONAL SPRING ELEMENTS

Skip this section if no request for forces or moments in spring elements.

J.3.7.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE (1)**		The number 7
6-10	**ICODE (2)**	(1)	The order number
11-15	**ICODE (3)**	(2)	Total number of elements in this group for which output is requested

J.3.7.2 OUTPUT REQUEST CARDS (I5, 4X, 6I1, 14X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**KEY(1)**	(3)	Output control key for force in x-direction
11	**KEY(2)**		Output control key for force in y-direction
12	**KEY(3)**		Output control key for force in z-direction
13	**KEY(4)**		Output control key for moment in xx-direction
14	**KEY(5)**		Output control key for moment in yy-direction
15	**KEY(6)**		Output control key for moment in zz-direction

30	**KN**	(4)	Generation control key
			= 0, no generation
			= 1, otherwise

Notes:

(1) See note 1 of Section **J.3.1**.
(2) See note 2 of Section **J.3.1**.
(3) See note 3 of Section **J.3.1**.
(4) See note 4 of Section **J.3.1**.

J.3.8 ONE-DIMENSIONAL PLANE LOVE-WAVE ELEMENTS

Not available.

J.3.9 THICK SHELL ELEMENT

Skip this section if no request for forces or moments in four node thick shell elements. The stress results for triangular elements are inaccurate.

J.3.9.1 CONTROL INFORMATION (4X, I1, 2I5)

Columns	Variable	Notes	Entry
5	**ICODE(1)**		The number 10
6-10	**ICODE(2)**	(1)	The order number
11-15	**ICODE(3)**	(2)	Total number of elements in this group for which output is requested

J.3.9.2 OUTPUT REQUEST CARDS (I5, 4X, 8I1, 12X, I1)

Columns	Variable	Notes	Entry
1-5	**N**		Element number
10	**KEY(1)**	(3)	Output control key for force component resultant $S_{x'x'}$
11	**KEY(2)**	(3)	Output control key for force component $S_{y'y'}$
12	**KEY(3)**	(3)	Output control key for force component $S_{x'y'}$
13	**KEY(4)**	(3)	Output control key for moment component $M_{x'x'}$

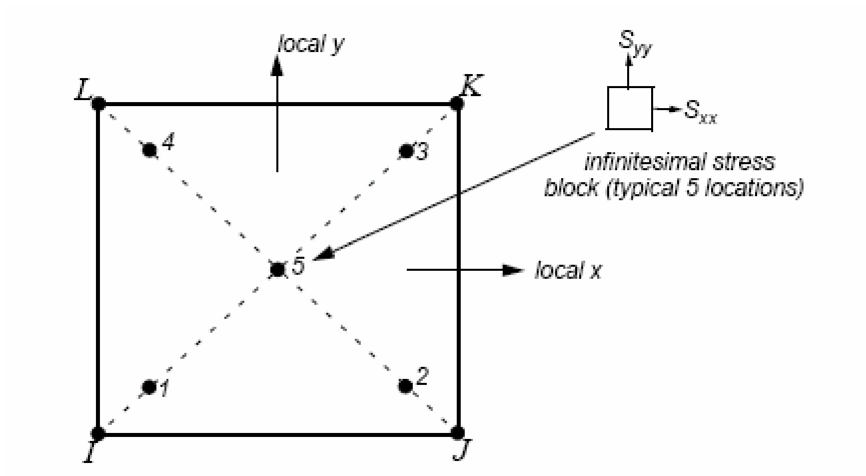
14	**KEY(5)**	(3)	Output control key for moment component $M_{y'y'}$
15	**KEY(6)**	(3)	Output control key for moment component $M_{x'y'}$
16	**KEY(7)**	(3)	Output control key for moment component $V_{x'z'}$
17	**KEY(8)**	(3)	Output control key for moment component $V_{y'z'}$
30	**KN**	(4)	Generation control key = 0, no generation = 1, otherwise

Notes:

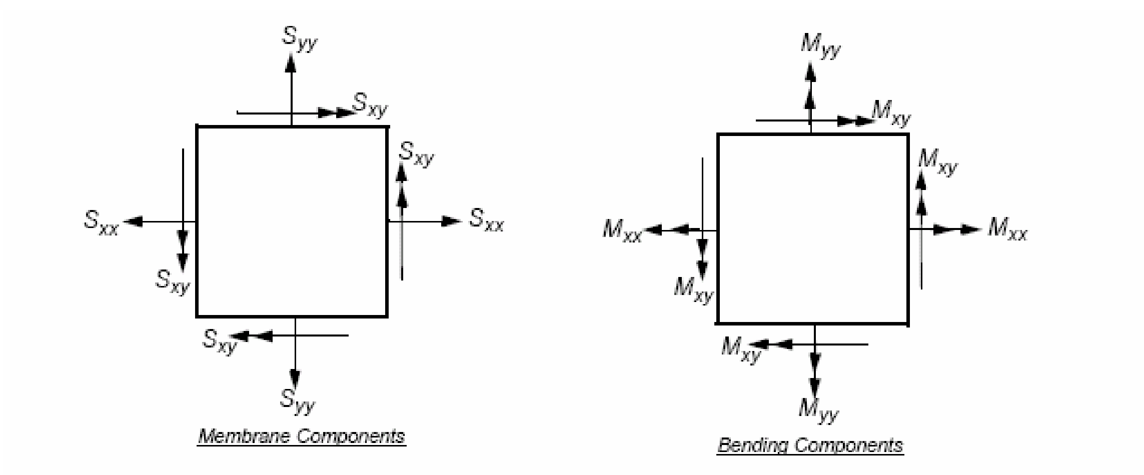
(1) See note I of Section **J.3.1**.

(2) See note 2 of Section **J.3.1**.

(4) The element force and moments are computed with respect to local element coordinates in five locations, four corner nodes (I, J, K and L) and the element center of gravity (C.G.). The figure below shows the location of the infinitesimal elements where the forces and moments are computed and the positive definitions of each component.



Positive local z-axis is oriented upward. Positive face of shell element (viewed nodes numbered counter-clockwise). Points where stresses are computed are numbered 1 through 5. Points 1 through 4 are located approximately 80% from the element CG to the corner nodes. Point 5 is located at the element CG.



The output key can have the following values:

KEY = 0, print only maximum responses on Tape 18 (SASSI.T18) at all five locations where stresses are computed.

= 1, print maximum response at all five locations and save time history on Tape 12, (SASSI.T12) of stress in corner Node I.

= 2, print maximum response at all five locations and save time history on Tape 12, (SASSI.T12) of stress in corner Node J.

= 3, print maximum response at all five locations and save time history on Tape 12, (SASSI.T12) of stress in corner Node K.

= 4, print maximum response at all five locations and save time history on Tape 12, (SASSI.T12) of stress in corner Node L.

= 5, print maximum response at all five locations and save time history on Tape 12, (SASSI.T12) of stress in center of gravity (C.G.)

= 6, print maximum response and save time history on Tape 12, (SASSI.T12) at all five locations

The forces are in units of force per length of the element (F/L) and the moments are in moment/length (F.L/L). It should be noted that for thick shell elements forces are computed in units of force per length (F/L) whereas for thin shell elements, the forces are computed in terms of stresses (F/(L*L)).

(4) See note 4 of Section **J.3.1**.

J.4 INPUT MOTION DATA CARDS (2I5, 3F10.0)

Columns	Variable	Notes	Entry
1-5	**NFFT**	(1)	Number of values to be used in Fourier Transform
6-10	**NEQZ**		Number of acceleration (or force) values to be read from cards
11-20	**DT**	(2)	Time step -- sec.
21-30	**EQMUL**		Multiplication factor for scaling time history. Use only if UGMAX = 0; leave blank otherwise
31-40	**UGMAX**		Maximum value of time history to be used. The values of time history will be scaled to give maximum value = UGMAX. Use only if EQMUL = 0 ; leave blank otherwise

Notes:

(1) **NFFT** must be a power of 2 and includes the trailing zeroes. If this number does not agree with the **NFFT** value in **SITE** (or **MOTOR**), the program will stop.

(2) The program will compute the frequency step from the following formula:

$$DF = \frac{1}{(NFFT * DT)}$$

If **DF** and **DT** do not agree with those in **SITE** (or **MOTOR**), the program will stop.

J.5 TIME HISTORY DATA

Data for time history should be provided on Tape 14, (SASSI.T14).

J.5.1 CONTROL TIME HISTORY ID CARD (18A4)

Columns	Variable	Notes	Entry
1-72	**ID(I)**		Identification for control time history

J.5.2 CONTROL TIME HISTORY VALUES (8F9.6/8E10.3)

Total of (NEQZ + 7)/8 cards.

Columns	Variable	Notes	Entry
1-72	**A(I)**	(1)	8 values

Notes:

(1) All time history values must be given at time steps DT. For seismic problems, provide acceleration values in (G) In (8F9.6) format; for foundation vibration problems, provide force values in (8E10.3) format.

J.6 OPERATION MODE CARD (I5)

Columns	Variable	Notes	Entry
5	**NOPT**	(1)	Operation mode = 0, stop no more data

Note:

(1) Must be the last data card.

5.2.11 PLOT

This program reads the nodal and element data from Tape 7 generated by the program HOUSE and at user's choice plots the entire or selected parts of the model. Selected groups of elements or nodes such as interaction nodes, nodes with nodal mass, etc. may be chosen for plotting. Both screen and printer plots are available. Plot files can be saved for importing to WORD or EXCEL files for report preparation. For this program to run, the file "COMDLG32.OCX" included on SASSI disk should be copied to C:\WINDOWS\SYSTEM32 on your computer.

K.1 FILE MENU

K.1.1 OPEN INPUT

Opens (with browsing capability) an *Input-File* (**SASSI.T7 from HOUSE**) and plots it with default attributes.

K.1.2 OPEN VIEW

Opens (with browsing capability) a previously saved *View-File* (**which is a *.viw file**) and plots it with its *saved attributes*.

K.1.3 SAVE IMAGE

Saves (with browsing capability) the current plot as a BMP file that can be retrieved by any appropriate program (e.g. MS-Word, MS-Excel, MS-Paint.)

K.1.4 SAVE VIEW

Saves (with browsing capability) the current plot as a *View-File* (***.viw**), containing the Input-File Name and all user-set attributes such as Group-Selection, Point-of-View Coordinates, Zoom-Factor, etc., that can be retrieved by SASSI_PLT (using *Open View* as described above) for viewing and/or modification later on.

K.1.5 NEW ARCHIVE

Creates (with browsing capability) a new Archive-Directory (***.arc**) to be used later for plot (BMP-File) archiving (see Save Archive below). The *Archive* would only keep BMP (i.e. Saved Image) files as its sole purpose is to PRINT them later.

K.1.6 SAVE ARCHIVE

Saves (with browsing capability) the current plot as a *BMP-file* in an Archive-Directory (which should have been previously created using *New Archive* as described above). All the images (i.e. *BMP-file*) in an *Archive* can be retrieved and printed one after the other by SASSI_PLT on request (see Print Archive below).

K.1.7 COPY TO CLIPPER-BOARD

Copies the current plot as a *BMP-image* to the Clipper-Board so that it can be pasted into any of the MS Applications.

K.1.8 PRINT IMAGE

Prints (to the selected/default printer) the current plot.

K.1.9 PRINT ARCHIVE

Prints (to the selected/default printer) all the Images saved in the selected Archive (via browsing) in the background. In doing this, the current plot is kept in tact.

K.1.01 PRINTER SETUP

Selects the Printer to be used in all printing requests.

K.1.11 VIEW DATA

Displays the current *Input-File* in a separate (text) window and provides the search capability. In doing this, if the *Input-File* is greater than 64K, it would recall upon *Internet Explorer* to do the job. In any case, the *Input-File* is read-in as Read-Only while providing a *Text-Search* capability.

K.1.12 EXIT

Terminates the PLOT program execution.

K.2 VIEW MODEL MENU

K.2.1 RENDERED MODEL

Sets the View-Mode to *“Solid with Edges”*. In this *mode*, no *hidden-line* (e.g. Brick-Edges) will be shown. Moreover, *Node-Numbers*, *Element-Numbers*, *Interaction-Nodes* and *Nodes with Lumped-Mass* will not be displayed if requested (using the *View Details* button as described below).

K.2.2 RENDERED MODEL WITH NODES

Sets the View-Mode to *“Solid with Edges & Nodes”*. In this *mode*, while showing the *Node-Symbols* too, no *hidden-line* (e.g. Brick-Edges) and/or *Hidden-Node* will be shown. Moreover, from the set of *Node-Numbers*, *Interaction-Nodes*, *Nodes with Lumped-Mass* and *Element-Numbers*, only the last one will **not** be displayed if requested (using the View Details button as described below).

K.2.3 TRANSPARENT MODEL

Sets the View-Mode to *“Edges with Nodes”*. In this *mode*, nothing is hidden and thus, all the *Edges and Node-Symbols* will be shown. Moreover, all of *Interaction-Nodes*, *Nodes with Lumped-Mass*, *Node- Numbers and Element -Numbers* can be displayed if requested (using the *View Details* button as described below).

K.3 VIEW RANGE MENU

K.3.1 GROUPS

Can be used for overall and/or individual *Element-Groups* selection/deselection as well as their coloring. In doing this, it would open up a *Group-Selection Window* in which the following **two sets** of selection criteria are presented plus an **OK** and a **Cancel** button:

Structural/Excavated *Selection-Buttons*

This set consists of the following three *Radio-Buttons*:

- Select Group Elements
  If selected (pushed), the *default selection mode* will be on. In this mode, all the Element-Group Selection *Check-Boxes* and their corresponding *items* are **activated** and can be used.

- All Structural Elements
  If selected, all the Element-Group Selection *Check-Boxes* and their corresponding *items* are **deactivated** while only the **Structural Elements** are selected to be shown in the plot.

- All Excavated Soil Elements
  If selected, all the Element-Group Selection *Check-Boxes* and their corresponding *items* are **deactivated** while only the **Excavated Soil Elements** are selected to be shown in the plot.

Element-Group Selection *Check-Boxes* and associated items

This set consists of seven rows, one for each *Element-Group* (from 1 to 7 and six being unused), in which the following items are presented:

- Check-Box with its associated *Element-Group's Name* which is used to select/deselect the entire *E.G.* and when deselected, all following items will be *deactivated*.

- Text-Box which is used to specify *Individual-Elements* to be present in the plot. The selected *Element-Numbers* can be typed in individually with a **comma** separating them or as a **range of no's.** in the form of **n-m**. Any mixture of the two forms (i.e. comma-separated or ranged E.N.'s) is also valid. (e.g.: 1,3,5 or 1-3,5 or 1,2,4-6). The default is **"1-max group-no."**.

- Color-Button that is used to modify the color associated with this *Element-Group*. Please note that the color selection and/or modification would **only apply** to the current plot.

K.3.2 RANGE

Can be used for specifying up to **three** *Viewing-Ranges*, one on each coordinate axis. In doing this, it would open up a *Range-Selection Window* in which three rows of *Range-Specification* (one for each coordinate) are presented each containing the following, plus an **OK** and a **Cancel** button:

- Full-Range *Check-Box* (for the coordinate) that can be selected/deselected to specify *Full-Range Viewing* on the coordinate (when selected) or *Partial Viewing* (when deselected). Obviously, on the latter case, there is a provision for specifying the *Partial-Range* as described below.

- From-To *Text-Boxes* (for the coordinate) that is **only** activated when the Full-Range *Check-Box* is deselected. Each value (i.e. From and To) can be given independently but the former should be less than the latter. The default values are obtained from the *Input-File* and displayed for each coordinate at the start or when the Full-Range is reselected.

K.4 SYMBOL SIZE MENU

K.4.1 LARGE

If selected, doubles the size of all symbols, numbers and their appropriate distances.

K.4.2 STANDARD

If selected, the size of all symbols, numbers and their appropriate distances are set to the default value.

K.4.3 SMALL

If selected, halves the size of all symbols, numbers and their appropriate distances.

K.5 TOOLBAR BUTTONS

- Zoom In

  Can be used for Zoom-In for an additional 10%.

- Zoom Out

  Can be used for Zoom-Out for an additional 10%.

- Zoom

  Can be used for Zoom-In or Zoom-Out by specifying the zoom-value > 0.

- Rotate

  Can be used to change the **direction** at which the plot is viewed by specifying a **Point-of-View (PoV) coordinate**. The program will then use the *direction* from this **PoV** to the **Origin** as the **Viewing-Direction**. In doing this, it would open up a *Rotation Window* in which the following **two sets** of rotation elements are presented plus an **OK** and a **Cancel** button:

  - Preset *View-Point* Buttons

    This set consists of the following four *Viewing-Buttons*:

    - X-Z Plane (the default **PoV**)

      This sets the **PoV** to be at (0,-1,0).
    - X-Y Plane

      This sets the **PoV** to be at (0,0,1).
    - Y-Z Plane

      This sets the **PoV** to be at (1,0,0).
    - Tilted View

      This sets the **PoV** to be at (1,1,1).

  - Viewing-Direction Specification
  - This set consists of three *Text-Boxes* (one for each coordinate) and can be used to specify an *arbitrary* **PoV** anywhere in the XYZ-space.

- Move

  Can be used to move the current plot by specifying the **new** XYZ coordinates of the it's center.

- View Details

  Can be used to add/remove details to the current plot. In doing this, it would open up a *View Details* Window in which six check-boxes are presented, plus an **OK** and a **Cancel** button, to select/deselect any one of the following details on the plot:

  1. Add Node Numbering
  2. Add Element Numbering
  3. Interaction-Nodes Display
  4. Nodes with Lumped Mass Display
  5. Symmetry Line/Plane Display
  6. Ground Surface Display

- Boundary
  - Can be used to add/remove any one of the six boundary conditions on the nodes of the current plot. In doing this, it would open up a *Boundary* Window in which six check-boxes are presented, plus an **OK** and a **Cancel** button, to select/deselect any one of the following Boundary-Condition Displays on the plot:

  1. X Boundary
  2. Y Boundary
  3. Z Boundary
  4. XX Boundary
  5. YY Boundary
  6. ZZ Boundary

- Reset

  Can be used to reset the current plot to the last setting that has been saved/retrieved into/from an *Input-File/View-File*.

CHAPTER 6
REFERENCES

1. Computer Program SASSI2000, Version 2 - Theoretical Manual, January 2006.

2. Computer Program SASSI2000, Version 2 - Validation Manual, 2006.

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

4. Schnabel, P. B., Lysmer, J. and Seed, H. B., "SHAKE - A Computer Program for Earthquake Response Analysis of Horizontally Layered Sites," Report No. EERC 72-12, Earthquake Engineering Research Center, University of California, Berkeley, CA, 1972.

5. BC-TOP-4A, "Seismic Analysis of Structures and Equipment for Nuclear Power Plants," Rev. 3, Bechtel Power Corporation, San Francisco, CA, November 1974.

6. Luco, J. E., "Impedance Functions for a Rigid Foundation on a Layered Medium," Nuclear Engineering and Design 31, 204-217, 1974.

7. Computer Program CE933 (FASS), (Version Al-1), Fourier Analysis of Soil-Structure System, Bechtel Power Corporation, San Francisco, CA.

8. Proceedings: EPRI/NRC/TPC Workshop on Seismic Soil-Structure Interaction Analysis Techniques Using Data from Lotung, Taiwan, NP-6145, prepared by Electric Power Research Institute, Palo Alto, California, December 1987.

9. Mita, A. and Luco J.E. (1986): “Response of Structures to Spatially Random Ground Motion,” 3rd U.S. Conference on Earthquake Engineering, Charlston, South Carolina.

10. Kaynia, A. M., (1982): “Dynamic stiffness and seismic response of flexible piles,” Research report R82-03, School of Civil Engineering, Constructed Facilities Division, Massachusetts Institute of Technology, Cambridge, Massachusetts.

APPENDIX A

VALIDATED FEATURES / OPTIONS CHECKLIST

(This page to be completed upon installation of the program and verification of the options on the host system)

APPENDIX B
COMMENT FORMS
AND
ERROR REPORTS

S A S S I

READER'S COMMENT FORM

Manual Title: ______________________ Revision No.:__________

Please complete this form and return to:

sassi@sassi2000.net

Your comments and suggestions about this manual are requested to improve the manual and the program itself. Your comments may include clarity, accuracy, completeness, organization, index, etc.

C0MMENTS: (Please give specific page and line reference where appropriate.)

FR0M Name or Title (incl. email): ____

Company Name:
____

Company Address:
____

City: ____________________ State: ___________________ Zip Code: ______________

Country: ____

S A S S I

SUSPECTED ERROR REPORT

Please complete this form and return to:

sassi@sassi2000.net

This form is for reporting a suspected error in the computer program.

Program Module Name: ____________________ Version No.: ________________________

Computer Systems: _____________________ System: ______________________________

REP0RTED BY: Name or Title (incl. email): ____

Company Name:
____

Company Address:
____

City: _____________________ State: _________________ Zip Code: _______________

Country: ____

DESCRIPTI0N (Please be complete. Append relevant files, listings, core dumps, etc.)