CARES
Computer Analysis for Rapid Evaluation of Structures Version 1.2

Manuscript Completed: September 1995
Date Published:

Prepared by
Carl J. Costantino, Charles A. Miller,
Ernest Heymsfield, Anzhi Yang

Costantino, Miller and Associates
8 Hastings Road
Monsey, New York 10952

Prepared for
Division of Engineering
Office of Nuclear Reactor Regulation
U.S. Nuclear Regulatory Commission
Washington, DC 20555
NRC JCN J21484

ABSTRACT

This report presents a summary of the improvements and additions made to the CARES Computer Code to allow the user greater flexibility in analyzing seismic problems of current interest. This is the second upgrade to the CARES Code developed for NRC and is labeled as Version 1.2. The following primary additions have been made to extend the capabilities of the analysis from previous versions.

a. The program has been made operational on the Sun SPARC 20 work station available at NRC. To perform on-screen plotting of computed results, the IMSL Exponent Graphics routines appropriate for the SPARC workstation have been added to the Code.

b. A number of detailed improvements have been added to the various sections of the Code to make the operation better able to satisfy NRC Staff needs. These include:

General:

- Addition to the hard copy list of the input and output data the printing of the CARES Version No., the date of the computation, the titles of file name used and generated, etc., to assist the user in keeping track of computed data files.

Motion Fitting Section:

- Expand the time history record size to 8192 points to better capture high frequency spectral definitions;
- Provide the ability to select a portion of the input time record for computation of response;
- Include relating parameters of the time enveloping function to earthquake magnitude for consistent development of artificial time histories;
- Incorporate the RG 1.60 enveloping criteria in the motion fitting criteria;
- Allow for either random selection of phase angles of the Fourier components of the acceleration record or have them chosen from an existing earthquake record.

Convolution Section:

- List a complete summary of initial and computed iterated soil properties in the output files;
- Include the tau/sigma stress calculation in each soil layer for use in liquefaction evaluations;
- Let the maximum allowable error in strain computation be selected by the user.

Spectra Computation Section:

- Increase the number of frequency points and range of values at which response spectra are calculated.

Structural Response Section:

- Add an in-plane shear model to the element library;
- Include a correction to the SSI module to allow treatment of embedded rectangular foundations for use in the AP600 evaluations;
- Simplify input of nodal restraints in the structural stick models.

Plotting Section:

- Add IMSL plotting capability for on-screen plotting capability as described above;
- Allow multiple curves on any screen plot.

TABLE OF CONTENTS

APPENDICES:

A. Screen Dumps and Selected Plots for Sample Problem No. 1 and Sample Problem No. 2
B. File Contents for Sample Problem No. 1
C. Data for Structural Verification Problems
D. SSI.RUN File for AP 600 Model
E. PSD Plotting Samples

LIST OF FIGURES

LIST OF TABLES

ACKNOWLEDGEMENT

The authors wish to thank Dr. Raman Pichumani (NRR/ESGB), the NRC Project Manager, under whose direction this project was performed. His comments and suggestions made during the planning and development stages for the revised CARES System are especially acknowledged.

We also wish to thank Mr. Goutam Bagchi and Dr. Robert Rothman (NRR/EQBG) as well as Mr. Herman Graves (RES/SSEB) for their encouragement and suggestions regarding applications of CARES to licensing issues facing NRC.

1.0 INTRODUCTION

During the late 1980's, the CARES (Computer Analysis for Rapid Evaluation of Structures) computer program was developed for the Nuclear Regulatory Commission (NRC) and published at Brookhaven National Laboratory (Ref. 3) to enable treatment of seismic studies of structural systems of interest to the NRC. At the time, the CARES system was generated by combining two available computer programs which treated the free-field response calculation (the SLAVE Code, Ref. 1) and the soil-structure interaction analysis of simplified stick models (the SIM Code, Ref. 2). Desirable ancillary postprocessing capabilities were added (plotting, spectra and PSD computation, etc.) to enhance the usefulness of the Code. The CARES Code was geared for use on a small personal computer (either a DOS or Macintosh compatible system) and was intended to provide the NRC Staff with the capability to rapidly evaluate seismic response of relatively simplified structural stick models. Such a capability enables the Staff to relatively quickly check the validity and/or accuracy of the data received from various Applicants. These submittals are typically obtained from numerical studies performed with large state-of-the-art structural computer packages which are difficult to assess without spending significant time and effort in the review process. By performing simplified model studies, the sensitivity of computed responses to variations in a host of controlling parameters can often be evaluated and thereby assist the Staff in gaining confidence in the results obtained from the larger computer studies.

Since that original development, the usefulness of the CARES concept has been validated by its application to a number of problems investigated by the Staff. Having the capability to easily perform small analyses at one's desk directly on a PC has been shown to be a valuable asset for the Staff. As always, however, advances in the state-of-the-art of seismic response calculations made the extension of the capability incorporated within the CARES Code desirable. In addition, as long as improvements were being contemplated, the increase in the size of the problem that may be treated within the CARES framework was mentioned, since the new structural stick models of interest in the advanced reactor programs were significantly larger than the models associated with the original CARES development. The CARES Code has thus been extended and modified to satisfy these new objectives, leading to the development of a new version, which has been labeled as CARES v. 1.1. All the capabilities previously available in CARES 1.0 had been maintained in this updated version.

Following the development of CARES v. 1.1, a second enhancement to the Code was undertaken to enable it to run on the Sun SPARC workstation, located at NRC, which operates in the UNIX environment. This second update to the CARES Code has therefore been developed and is labeled as CARES v. 1.2. This third version which is described herein is limited to operate on the SPARC workstation although it can be made operational on other machines with some minor reprogramming effort. The primary advantage to this enhancement is to allow the Staff the ability

to run significantly larger problems at much shorter running times as compared to the previous versions of CARES which were PC oriented. In addition, with the larger computational space available on the workstation, some enhancements were added which are described in this report. To further utilize the capabilities of the workstation, the plotting capability of CARES v. 1.1 has also been modified and extended, using the IMSL plotting software appropriate to the Sun. On-line plots of computed results can now be obtained in this new version which can then be saved or printed as desired. Only the details of the enhancements are presented in this report.

1.1 Overview of the CARES Code

The CARES Code was originally designed to operate in a desktop PC environment (either DOS or Macintosh systems) and now operates on the Sun SPARC Station. It is geared to provide user friendly input/output with rapid turnaround. The Code is organized in a modular fashion so that each task or module can fit within the "confines" of the relatively small RAM available in either the typical desktop or the extended workstation environment. The modular format also allows for a number of differing capabilities to be accessed directly by the user in an interactive fashion. The primary functions contained in the CARES Code fall into three distinct categories which can be summarized as follows.

a. A free-field computational algorithm allows for analysis of the seismic response of a layered soil column subjected to upwardly propagating horizontal shear waves developed by a given input seismic motion. The input motion can be specified by means of a target response spectrum appropriate for a given earthquake magnitude at a given range from the source, or an actual accelerogram at a given location within the soil column. The output from this computation is the motion at other locations within the soil column which is compatible with this input motion as well as the final stress and strain conditions developed in each soil layer.

b. From the ground motions generated in the free-field module above, the seismic response of a structure embedded within or on the soil can be determined. This calculation includes the effects of depth of burial on the SSI impedance functions used in the structural response calculation.

c. A variety of postprocessing capabilities are provided to allow for generation of other parameters of interest in the dynamic response (e.g., in-structure response spectra, PSD calculation, Fourier decomposition, etc.) and plotting of results.

In the revisions to the CARES Code, a number of additions have been made to extend the capabilities of the analysis. In the structural module, several aspects have been modified. In

CARES v. 1.1, the primary change was associated with the expansion of the size of the structural model that could be treated. The original version of CARES (v. 1.0) was designed for analysis of relatively small lumped mass stick models having relatively few degrees of freedom. The structural models currently being evaluated in the various advanced reactor studies are significantly larger than previously envisioned. To enable treatment of these models within the CARES format, significant modifications were made in CARES v. 1.1 not only to lengths of vectors and matrices used in the structural analysis, but also to the methods of analysis to improve the speed of solution. An improved Fast Fourier Transform (FFT) method was incorporated to reduce run time required to convert between the time and frequency domains. In addition, the capability for treatment of rigid links within the structural model as well as an improved algorithm for handling composite damping have been added. Improvements added to CARES v. 1.2 include an in-plane shear wall module to the element library, an improved method of specifying nodal restraints of the structural stick(s) and the incorporation of an improved SSI model to allow treatment of rectangular foundations. As part of these improvements, an extensive stick model has been developed in the CARES format to represent the AP600 advanced reactor system.

In the free-field module, several major changes were incorporated in CARES v. 1.1 which extend the capability of the Code. The first change allowed for the inclusion of a rock outcrop model within the soil column formulation. This improvement which was long ago added to the SLAVE Code (Ref. 1) extends the capability of CARES to coincide with other free-field column analyses (Ref. 4). In addition, extended soil degradation modules were incorporated to keep pace with recent developments (Ref.6). In the new CARES v. 1.2, the computations have been extended to include calculation of tau/sigma stress ratios for liquefaction evaluations. Improvements have also been added to the soil degradation specification in the Code, as well as output of initial and final soil properties in the convolution calculations. The analytical bases for the modifications are discussed in Section 2 with sample problems validating the modifications presented in Section 4.

Other minor enhancements have also been added to improve the user friendliness of the Code. Other additions have been made in CARES v. 1.2 extending the motion description as well as the response spectra capabilities, allowing multiple plotting capability in any plot and adding the IMSL exponent graphics capability for online plotting on SPARC station.

1.2 Program Modules

As described above, the CARES Code performs three separate functions which can be labeled as the free-field analysis, the structural or SSI analysis and the ancillary or pre- and post-processing analyses of ground motions or spectra. In the original version of CARES v. 1.0, the PC-based program was essentially contained within the umbrella of a single "General Manager"

which allowed for transfer of data between any and all of the individual modules directly within one access of the Code. This was no longer possible with the large expansion of the Code capability incorporated within v. 1.1. Each of the free-field and structural modules were significantly expanded in size making the use of a single General Manager inappropriate. Thus, to allow the extensions desired from v. 1.0 to 1.1, the CARES Code was divided into separate modules. The further extensions made in v. 1.2 maintain the separate modules with common input/output format of files to allow easy communication between the modules. The modules are labeled as CRSSOIL, CRSSTRUC, CRSPOST and CRSPLOT. Each performs its indicated function and data transfer accomplished using by using data files generated by each module and stored on the disk.

1.3 Operation on Sun SPARC Station

As mentioned above, CARES version 1.2 is divided into four program segments or modules for use on the workstation. The executable files of four segments of CARES are indicated by the four icons. Simply double clicking on these icons invokes either CRSSOIL, CRSSTRUC, CRSPOST, or CRSPLOT of the CARES program, v 1.2. Data entry is described in Section 3 of this report and can be easily followed by the machine prompts for interactive operation of the Code. Version 1.2 of CARES can be used to generate the response of the large stick model of the AP600 nuclear island to seismic input. This material is discussed in Section 5.

2.0 ANALYTIC EXTENSIONS/MODIFICATIONS

This section of the report presents a summary of the additions and/or modifications made to the CARES program to provide the desired extended computational capabilities. As discussed previously, the additions to the computer program have centered on three areas which have to do with (a) the development of the seismic motions used as input to the response calculations and the Fast Fourier Transform (FFT) routine used to convert efficiently between the time and frequency domains, (b) the free-field convolution program incorporating the capability to include the outcrop as well as extended soil modulus degradation formulations, and (c) extended structural response capability to allow for treatment of the much larger structural stick models being considered for the study of seismic responses of stick models for the advanced reactor programs. The following paragraphs present a summary of the analytic formulations required for these modifications.

2.1 Seismic Motion Algorithm

This section of the report presents a summary of the algorithm implemented in this version of CARES to generate artificial time histories compatible with specified acceleration response spectra as well as a description of the Fast Fourier Transform (FFT) method used to rapidly transform between the time and frequency domains. All calculations within the CARES program are based on the property that any (periodic) function in the time domain can be expanded into the frequency domain using the discrete Fourier transform, which can be written as

$$X(t) = \sum_{j=0}^{K-1} D_j e^{i \omega_j t} \qquad (2.1.1)$$

where K is the total number of frequency components of the motion (including the zero or steady state term) spaced at a uniform frequency increment. The term ω_j is the circular frequency and D_j is the complex Fourier coefficient of the sinusoidal motion. By properly computing the array of coefficients D_j, the time history of an accelerogram can be generated which has the desired characteristics of observed seismic motions. The details of this formulation are summarized in Section 2.1.3.

2.1.1 Accelerogram Characteristics

The input data required to generate a given artificial accelerogram consists of the target response spectrum (specified at a given level of equipment damping), the desired total duration (t_{max}) of the accelerogram and the desired number of time records (N_t) spaced at a uniform time increment. The total number of records comprising the pulse is determined from the input

parameter M through the relation

$$N_t = 2^M \tag{2.1.2}$$

The time increment of the record is then computed as

$$\Delta t = t_{max}/N_t \tag{2.1.3}$$

Based on the requirements of the FFT algorithm used in CARES (and described in Section 2.1.3), the total number of Fourier components required in the expansion of equation 2.1.1 is the same as the number of time points. However, the particular formulation used requires only one half the number of coefficients, with the remaining determined as the complex conjugates. The number of independent coefficients to be specified is then given by

$$N_C = N_t / 2 + 1 \tag{2.1.4}$$

and the corresponding frequency increment determined from

$$\Delta f = 1 / t_{max} \tag{2.1.5}$$

As an example of these calculations, if the total pulse duration desired is 20 seconds and the parameter M selected as 11, the characteristics of the generated motion are determined as

t_{max}	= 20 seconds	N_t	= 2048 time records
Δt	= 0.00977 seconds	N_C	= 1025 frequency records
Δf	= 0.05 hz	f_{max}	= 51.20 hz

Notice that the maximum frequency content of the motion, f_{max}, is equivalent to the Nyquist frequency, or

$$f_{max} = 1 /(2 \Delta t) \tag{2.1.6}$$

2.1.2 Simulation Algorithm

Several modifications have been included in the motion generation algorithm used in CARES to try to improve the character of the generated accelerogram to make it have more of the known characteristics of actual records. The first of these modifications makes use of an

enveloping (or shape) function in order to incorporate the known nonstationarity of earthquake records. The ensemble of sinusoidal motions represented by equations 2.1.1 is therefore modified by means of a deterministic multiplying function E(t) shown as

$$X(t) = E(t) \sum_{j=-N_C+2}^{N_C-1} D_j e^{i\omega_j t} \qquad (2.1.7)$$

where the coefficients D_j are complex. The particular enveloping function that has been selected is a simple trilinear function consisting of a linear rise to a value of unity at a rise time of t_r , a constant value of unity for the strong motion duration, t_s , followed by a linear ramp down to zero for a decay time of t_d. The total duration of the enveloping function is then the sum of the rise time, the strong motion duration and the decay time, or

$$t'_{max}(M) = t_r(M) + t_s(M) + t_d(M) \qquad (2.1.8)$$

The time parameters of equation 2.1.8 have all been related to a specified earthquake magnitude estimate (M) by fitting empirical data obtained from Ref. 11. Thus, the CARES code only asks for an estimate of the earthquake magnitude (generally defined by the magnitude scale M_w) to obtain the controlling parameters of the enveloping function. The purpose of this approach is to arrive at reasonable or realistic estimates of artificial accelerograms for use in response studies. In the calculations, the total duration of the motion is taken as the larger of t'_{max} from equation 2.1.8 and the desired maximum duration input, t_{max}. If the magnitude dependent duration of equation 2.1.8 is less than the desired total duration of motion defined in equation 2.1.3, the discretized time record generated is packed with zeros from t'_{max} to t_{max}.

An additional computational feature included in the motion generation program is a base line correction algorithm as described in Ref. 12. The purpose of the correction is to let the ground velocity reach zero at the end of the motion. This is achieved with a parabolic correction to the baseline of the accelerogram, where the coefficients of the correction relation are selected to minimize the mean square value of the velocity. During any one trial record development, the generated accelerogram is integrated with zero initial conditions and then baseline corrected. The influence of the correction is generally small in terms of accelerations (high frequency content) but may be significant in the lower frequency range where velocity and displacement characteristics become important.

An additional computational feature incorporated in the motion algorithm is termed "cutting and flipping". If the trial record has accelerations which exceed the target peak acceleration, the trial

time record is modified by cutting the acceleration at the peak value and subtracting the difference between the original and target peaks from the record. The purpose of this computation is to obviously limit the peak accelerations to the desired value but also to keep the "look" of the time record realistic, with no flat spots which would be generated by simply cutting the peaks of the record.

The motion computation then follows in a relatively straight forward manner. The initial trial amplitudes of the Fourier components (magnitude of the terms D_i of equation 2.1.1) are initially set equal to the values determined from the input target response spectrum. A corresponding random array of phase angles is generated which has a uniform distribution in the range from 0 to 2π radians. From these trial components, a trial time history of the motion is generated using the inverse FFT algorithm. The acceleration record is then modified by the magnitude dependent enveloping function, baseline corrected, and any exceedances cut and flipped. A response spectrum of this motion is then calculated along with new Fourier components obtained from FFT algorithm. This trial response spectrum is then compared to the desired target response spectrum, and the Fourier components modified by the relation

$$C_i^{new} = C_i^{old} \frac{R_{target}}{R_{trial}} \qquad (2.1.9)$$

where the terms R refer to the values of the spectra at the frequency corresponding to Fourier frequency being modified. The phase angles ϕ_i originally generated at the first trial are maintained in succeeding trials. The new components are then used to form a new trial accelerogram and the process repeated. If after a number of trials, convergence of the process is not satisfactory, a new fitting procedure should be undertaken using new phase angles which will generally be different from the first set.

2.1.3 FAST FOURIER TRANSFORM Algorithm

The CARES program uses a Fast Fourier Transform (FFT) algorithm to determine the Fourier components of a time history or to generate the time history from the Fourier components (inverse FFT). The subroutine was generated from the analysis of Ref. 10 which is based on the original formulation of Cooley and Tukey (Ref. 9). The complex discrete Fourier series is defined as:

$$X(t) = \sum_{j=-N/2+1}^{N/2} A_j \, e^{i \omega_j t} \tag{2.1.10}$$

where X(t) is the discrete uniformly spaced time function, N equals the total number of time history records (N_t of equation 2.1.2) and t is time. The complex Fourier coefficients in the negative range are the complex conjugates of the Fourier coefficients in the corresponding positive range, or

$$\text{real}\,(A_{-j}) + \text{imag}\,(A_{-j}) = \text{real}\,(A_j) - \text{imag}\,(A_j) \tag{2.1.11}$$

Having the Fourier components from 0 to N/2 provides sufficient information to determine the remaining Fourier components. Since the Fourier components are cyclic, the components from j =-N/2 +1 to j =-1 are equal to the components from j= N/2 + 1 to j =N. Equation 2.1.10 can then be rewritten as:

$$X(t) = \sum_{j=0}^{N-1} A_j \, e^{i \omega_j t} \tag{2.1.12}$$

Considering the time record to be specified at a uniform spacing Δt, the parameter $\omega_j t$ of equation 2.1.12 can be written in terms of the record number, by considering both t and t_{max} to be multiples of the time increment, or

$$\omega_j t = 2 \pi j k / N \tag{2.1.13}$$

where N is the total number of records in the accelerogram, or $N = t_{max}/\Delta t$ and k is the time history record number, that is $t = k \Delta t$. Equation 2.1.12 can now be written as

$$X_k = \sum_{j=0}^{N-1} A_j e^{i\frac{2\pi j k}{N}} \qquad (2.1.14)$$

. Equation 2.1.14 represents the inverse discrete Fourier transform. The coefficients A_j of the transform are defined as

$$A_j = \frac{1}{N}\sum_{k=0}^{N-1} X_k e^{-i\frac{2\pi j k}{N}} \qquad (2.1.15)$$

A requirement of the computational algorithm is that the number of the time history records be set equal to the value of 2^M, where M is a positive integer. For cases where the total number of records in the time history is not equal to this value, the time history record is padded with zeros at the end of the record to attain the required number of records.

2.2 Free-Field Analysis

Free-Field analysis associated with the vertical propagation of horizontal earthquake ground motions is typically performed by using either a discrete method based on finite element procedures (such as performed in the FLUSH Code) or by using a continuous method based on the solution to the one dimensional wave equation (such as performed in the SHAKE Code). Both methods are based on the assumption of one dimensional vertical shear wave propagation through the soil deposit. The objective of this analysis is to determine the horizontal motion-time histories (accelerograms) developed at any depth of a horizontally bedded soil system, when a specified accelerogram is input at any of the soil layer interfaces or at the rock outcrop.

The method employed in the CARES system to obtain this solution is the continuous model. A given soil column is assumed to be composed of a number of uniform soil layers of arbitrary thickness, each with linear soil properties. Such a typical configuration involving four layers is shown in Figure 2.1. Vertical shear wave propagation through this layered system produces horizontal stresses and accelerations which satisfy the shear wave equation for each soil layer. The stress-strain relation for each layer is specified in the form:

$$\tau = G\gamma + \eta\dot{\gamma} \qquad (2.2.1)$$

where τ is the shear stress, γ the shear strain , G the shear modulus, and η the viscosity coefficient of a given soil layer. A dot over a character indicates differentiation with respect to time.

The shear wave equation for each layer is:

$$\rho \frac{\partial^2 u}{\partial t^2} = G \frac{\partial^2 u}{\partial x^2} + \eta \frac{\partial}{\partial t}\left[\frac{\partial^2 u}{\partial x^2}\right] \qquad (2.2.2)$$

where u is the total horizontal displacement of any point in the layer, x is the vertical distance (positive downward, see Figure 2.1) of a point in the ground measured from the soil surface, t represents time and ρ is the mass density of the soil in the layer. In the CARES computer code, each soil layer is assumed to possess constant properties G, η , and ρ throughout its thickness.

As indicated above, each soil material in the column is treated as a linear material with both shear stiffness and damping properties specified. This linear relation has often been used for seismic applications (e. g., Ref. 5), provided that the damping parameter η is interpreted as representing a mechanism to dissipate energy per strain cycle as found from tests for the actual nonlinear behavior of the soil under cyclic shear loadings, as for example in the resonant column shear test. The energy lost per cycle of loading due to the actual nonlinear behavior is measured by the hysteretic damping ratio, D, which has been found experimentally to be reasonably independent of frequency, particularly in the low frequency ranges of interest to structural response. If this energy loss per cycle is equated to the energy loss per cycle due to the linear viscosity term of equation (2.2.1), the hysteretic damping ratio is found to be related to the viscosity coefficient, η, by

$$D = \frac{\eta\,\omega}{2\,G} \tag{2.2.3}$$

where D is the hysteretic damping ratio, η is the viscosity coefficient, ω is the circular frequency of the motion component and G is the soil shear modulus.

Considering steady-state motions at a given forcing frequency ω and applying the Fourier transform to equation (2.2.2), the equation of motion becomes

$$(G + i\,\omega\,\eta)\,\frac{\partial^2 \bar{u}}{\partial\, x^2} + \rho\,\omega^2\,\bar{u} = 0 \tag{2.2.4}$$

where $\bar{u}$ is the Fourier transform of the horizontal displacement at any depth x. Considering hysteretic damping,the complex shear modulus of the material becomes:

$$G^* = G\,(\,1 + i\,2\,D\,) \tag{2.2.5}$$

and equation (2.2.4) simplifies to :

$$\frac{\partial^2 \bar{u}}{\partial x^2} + (k^*)^2\,\bar{u} = 0 \tag{2.2.6}$$

where

$$(k^*) = \frac{\omega}{\sqrt{G^*/\rho}}$$

is the complex wave number. The solution of equation (2.2.6) is:

$$\bar{u}_i = A_i \exp(+ik_i^* x) + B_i \exp(-ik_i^* x) \tag{2.2.7}$$

where A_i and B_i represent the magnitudes of the incident and reflected waves respectively. The boundary and interface conditions between soil layers are

$$\begin{aligned} &\bar{\tau}_i(x=0) = 0 \\ &\bar{u}_i(x= x_i) = \bar{u}_{i+1}(x= x_{i+1}) \\ &\bar{\tau}_i(x= x_i) = \bar{\tau}_{i+1}(x= x_{i+1}) \end{aligned} \tag{2.2.8}$$

Applying the conditions of equations 2.2.8 to equations 2.2.7, the coefficients A_{i+1} and B_{i+1} at a given layer interface are determined in terms of the coefficients at the previous layer by

$$A_{i+1} = 1/2\left[A_i \left(1. + \frac{k_i^* G_i^*}{k_{i+1}^* G_{i+1}^*}\right)\exp[ix_i(k_i^* - k_{i+1}^*)] + B_i \left(1. - \frac{k_i^* G_i^*}{k_{i+1}^* G_{i+1}^*}\right)\exp[-ix_i(k_i^* + k_{i+1}^*)]\right] \tag{2.2.9}$$

$$B_{i+1} = 1/2\left[A_i \left(1. - \frac{k_i^* G_i^*}{k_{i+1}^* G_{i+1}^*}\right)\exp[ix_i(k_i^* + k_{i+1}^*)] + B_i \left(1. + \frac{k_i^* G_i^*}{k_{i+1}^* G_{i+1}^*}\right)\exp[-ix_i(k_i^* - k_{i+1}^*)]\right]$$

Calculations begin by assuming the coefficient A_1 at the top of the first layer (ground surface) to be unity (that is, have real and imaginary components given by 1,0). Using the known shear stress condition at the surface yields the coefficient B_1 equal to A_1. All the remaining coefficients can then be computed from equation 2.2.9. At the layer interface where the input ground motion is specified (i = Layer), the acceleration magnitude from the calculation is compared with the known acceleration magnitude specified from the input motion. The remaining A_i's and

B_i's are then corrected by the ratio of the known acceleration magnitude to the calculated acceleration magnitude. If an outcrop is defined at the bottom of the soil column, the motion at the outcrop is determined by considering that the shear stress at the outcrop is equal to zero and the incident wave from the soil column solution is the same as the incident wave for the uniform half-space. Because of the zero stress condition at the outcrop, the reflected wave solution can be written in terms of the incident wave solution of the half-space.

CARES requires that the soil properties, low strain shear modulus (G_{max}), the soil unit weight (γ), and the thickness (H), be specified for each soil layer. Half-space properties are also required if the half-space is to be considered in the analysis. Options for soil modulus degradation models incorporated within CARES include:

a. Seed-Idriss 1970 (Ref. 5),
b. Idriss 1990 (Ref. 7),
c. GEI from Savannah River measurements (Ref. 6),
d. constant damping and no reduction in the shear modulus, or
e. a user specified degradation model.

The specific degradation properties for the three models contained in CARES are listed in Table 2.1.

The standard iterative procedure is used (a la SHAKE, Ref. 4) to account for nonlinear soil behavior within the context of this linear analysis. Calculations are performed by initially assuming an initial trial value of effective shear strain within each soil layer. Using the soil degradation data specified for the soil, the shear modulus and hysteretic damping for each soil layer is determined. At the end of each computational cycle, the calculated effective shear strain is computed and compared with the assumed trial value. When the assumed and calculated shear strain differ by less than 5% within each soil layer, the solution is considered to have converged.

Two methods are used to calculate maximum effective strain. In the first case, the maximum strain is computed in the time domain for each soil layer.The Fourier components of shear strain are combined using the inverse FFT algorithm to obtain the actual maximum shear strain developed in each soil layer during the course of the motion. The effective cyclic shear strain developed in the layer is then estimated from the maximum strain by

$$\gamma_{effective} = 0.65\ \gamma_{max} \qquad (2.2.10)$$

This effective cyclic shear strain is then used to obtain the effective shear modulus and damping ratio for the next iterative response computation from the available soil degradation data provided as input for each soil layer.

As an alternative approach available to approximate the effective cyclic shear strain, the maximum shear strain can be estimated from the RMS strain values, eliminating the need for conversion to the time domain from the frequency domain. The relationship used for this conversion is

$$\gamma_{max} = \gamma_{RMS} \left[\frac{a_{max}}{a_{RMS}}\right] \qquad (2.2.11)$$

where the term in parenthesis is the ratio of the peak to the RMS value of the input acceleration record. The effective strain is then calculated from equation 2.2.10 above. In the CARES program, strains are computed at the top and bottom of each soil layer and averaged to determine the effective strain for the layer. It should be noted that although the maximum strain calculated directly in the time domain is more accurate, it is a slower calculation and has been found to not differ significantly from the RMS strain calculated in the frequency domain.

TABLE 2.1
SOILDEGRADATIONMODELS

SHEARMODULUSRATIO-G/G_{max}

SHEAR STRAIN (%)	SEED-IDRISS 70 SAND	SEED-IDRISS 70 CLAY	IDRISS 1990 SAND	IDRISS 1990 CLAY	GEI 1991 0'-50'	GEI 1991 50'-100'	GEI 1991 100'-250'	GEI 1991 250'-500'	GEI 1991 500'+
0.000100	1.000	1.000	1.000	1.000	1.000	1.000	1.000	1.000	1.000
0.000316	0.984	0.913	1.000	1.000	0.990	0.990	0.990	0.995	1.000
0.00100	0.934	0.761	0.990	1.000	0.980	0.980	0.980	0.990	1.000
0.00316	0.826	0.565	0.955	0.979	0.910	0.935	0.947	0.965	0.985
0.0100	0.656	0.400	0.850	0.941	0.780	0.816	0.852	0.890	0.925
0.0316	0.443	0.261	0.628	0.839	0.560	0.610	0.670	0.725	0.775
0.100	0.246	0.152	0.370	0.656	0.330	0.360	0.425	0.495	0.565
0.316	0.115	0.076	0.176	0.429	0.160	0.175	0.200	0.240	0.300
1.00	0.049	0.037	0.080	0.238	0.065	0.065	0.065	0.080	0.100
3.16	0.049	0.013	0.080	0.238	0.040	0.040	0.040	0.040	0.040
10.00	0.049	0.004	0.080	0.238	0.040	0.040	0.040	0.040	0.040

HYSTERETIC DAMPING RATIO (%)

SHEAR STRAIN (%)	SEED-IDRISS 70 SAND	SEED-IDRISS 70 CLAY	IDRISS 1990 SAND	IDRISS 1990 CLAY	GEI 1991 0'-50'	GEI 1991 50'-100'	GEI 1991 100'-250'	GEI 1991 250'-500'	GEI 1991 500'+
0.000100	0.50	2.50	0.24	0.24	1.50	1.50	1.50	1.50	1.50
0.000316	0.80	2.50	0.44	0.44	1.50	1.50	1.50	1.50	1.50
0.00100	1.70	2.50	0.80	0.80	1.50	1.50	1.50	1.50	1.50
0.00316	3.20	3.50	1.46	1.46	1.75	1.60	1.50	1.50	1.50
0.0100	5.60	4.75	2.80	2.80	3.85	3.15	2.50	2.10	1.75
0.0316	10.00	6.50	5.31	5.31	7.75	6.50	5.00	3.75	2.50
0.100	15.50	9.25	9.80	9.80	13.10	11.75	10.00	8.25	6.25
0.316	21.00	13.80	15.74	15.74	18.75	17.75	16.50	14.75	13.00
1.00	24.60	20.00	21.00	21.00	23.00	22.50	22.00	20.50	19.00
3.16	24.60	26.00	21.00	21.00	26.00	25.60	25.40	24.25	23.10
10.00	24.60	29.00	21.00	21.00	26.00	25.60	25.40	24.25	23.10

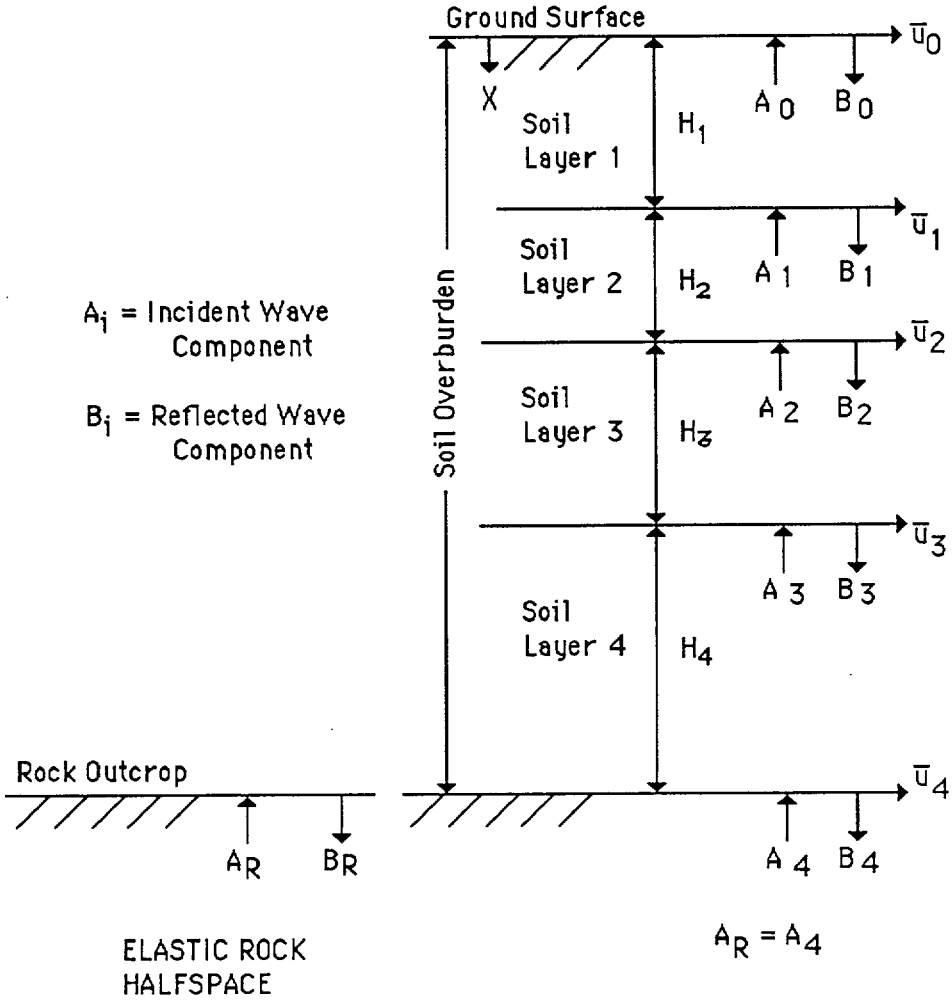

Figure 2.1. Definition of Rock Outcrop Motion in Soil Column Analysis

2.3 Modifications to Structural Analysis

The structural analysis portion of the CARES computer system determines the response of the structure to a seismic input. Several changes were made to this option of CARES to accommodate problems of the size of AP 600 stick model provided by the Staff. Changes were made both in support of Version 1.1 and 1.2 as indicated below. The sample problems were run with Version 1.2.

2.3.1 Data Storage (Version 1.1)

Solutions to the structural response problem are carried out in the frequency domain in CARES. In this approach, structural stiffness and mass matrices are first generated. The dynamic matrix is then formed at each frequency in the Fourier expansion by combining the inertial and damping matrices with the stiffness matrix. Solutions are then determined at each frequency of the expansion using a Gaussian elimination procedure (Ref. 15). In the original CARES Version 1.0 (Ref. 3), the program was written to store the entire structural stiffness and dynamic matrices in RAM and performed the elimination on all elements of the matrix. Although this approach simplifies the programming aspects of the solution process, it is inefficient for treating large problems since the matrices tend to be banded and operation on the zero elements beyond the bandwidth is not required. CARES Version 1.1 was therefore modified so that only the upper half of the stiffness and dynamic matrices are saved in core, and the elimination solution algorithm modified so that operations are not performed on the zero elements beyond the band width. This significantly reduces the memory requirements and solution time for large problems such as those associated with the System 80+ stick model.

In the frequency domain approach, all of the unknown structural deformations can be expanded into the sine and cosine components of the Fourier expansion. In CARES Version 1.0, the matrix storage was organized so that all of the coefficients associated with the sine coefficients were stored first, followed by all of those associated with the cosine coefficients. This storage strategy is again simple from a programming point of view, but leads to large computational bandwidths. In the modified CARES Version 1.1, this procedure was therefore modified so that the data associated with the both coefficients for a single degree of freedom were stored together, reducing computational bandwidths, but increasing programming difficulty. All of these changes are opaque to the user and do not effect the input/output requirements of CARES Version 1.1. To demonstrate that the programming modifications were properly made, Sample problem S.1, discussed in Section 4.2, demonstrates that CARES Version 1.1 yields the same results as obtained from Version 1.0. Similarly, Sample problem S.4 demonstrates that solutions obtained with Version 1.1 yield the same results as obtained from analytic solutions to simple structural models.

2.3.2 Rigid Links (Version 1.1)

In the development of many structural stick models, it has often been found necessary to use rigid connections between some nodes. An example of such a need arises in models where the floor element within the structure is modeled as a rigid element, and the connecting elements both above and below the floor are modeled with separate vertical flexible sticks. The location of the stick above the floor is also generally different from the location of the stick representing the structure below the floor. A rigid link, in the plane of the floor, is then used to connect the two sticks in the horizontal direction. An example of this application is shown in Fig. 4.3 with the results of the sample problem validating this modification. In many applications, the rigid link can often be effectively provided by means of a flexible element with a stiffness that is set artificially high. This can, however, cause numerical problems. It is also inefficient since the rigid link can be used to reduce the total number of degrees of freedom in the problem, reducing effective computational times.

A rigid link capability has therefore been provided in the updated CARES Version 1.1, by allowing the user to define "slave" nodes that are tied to "master" nodes. The deformations (D_s) of the slave nodes are then related to the master node deformations (D_m) by the transformation

$$D_s = T_{sm} D_m \tag{2.3.1}$$

where, T_{sm} is a transformation matrix defined by

$$T_{sm} = \begin{bmatrix} 1 & 0 & 0 & 0 & Z_s-Z_m & -Y_s+Y_m \\ 0 & 1 & 0 & -Z_s+Z_m & 0 & X_s-X_m \\ 0 & 0 & 1 & Y_s-Y_m & -X_s+X_m & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

where, X, Y, and Z refer to the coordinates of either the master (m) or slave (s) nodes.

This transformation is then used in the program to eliminate the slave degrees of freedom from the list of unknowns and replace them with the kinematically equivalent master degrees of freedom. There are two restrictions in the specification of slave nodes. First, one node cannot be slave to a master node which happens to be slave to a third node. This is actually not a true

restriction since the first slave node can be made to have the third node as its specified master. Secondly, an SSI node (node to which SSI loads are applied) cannot be a slave node.

2.3.3 Structural Damping (Version 1.1)

The structural damping matrix (C) in CARES is expressed as a linear combination of a the mass matrix and stiffness matrix through the relation

$$C=\alpha M+\beta K \tag{2.3.2}$$

where α and β are parameters to be determined. The value of these parameters are computed in CARES based on the first two structural fixed-base frequencies (ω_1 and ω_2) and the user specified ratio of critical damping (p) appropriate for the structure:

$$\alpha = 2\,p\,\omega_1\,\omega_2 / (\omega_1+\omega_2)$$
$$\beta = 2\,p\,/(\omega_1+\omega_2) \tag{2.3.3}$$

Two changes were made to this process in the updated Version of CARES. First, the original Version of CARES computed the first two structural frequencies and mode shapes by using the JACOBI method (Ref. 15) of analysis. This has two disadvantages for application to large problems. First, it requires a large storage area since the full stiffness and mass matrices must be saved. Secondly, the computational time can become excessive for large problems since the JACOBI method determines all modes, not just the two that are required to evaluate the damping parameters. The JACOBI method was therefore replaced with a POWER method (Ref. 15) in the current Version of CARES. This procedure only computes the required two lowest modes and no additional storage is required over that needed for the solution phase of CARES.

There may be a requirement to treat structural damping in a manner so that various parts of the structure may be specified to have different ratios of critical damping. This is done by using the composite damping model (Ref. 16). The damping matrix is still defined as a linear combination of the mass and stiffness matrices, but the damping in each of the first two modes is defined as a weighted average of the damping in each of the structural elements, with the weighting function based on the ratio of the strain energy stored in the element to the total energy stored in all of the elements. Therefore, if ϕ_{ij} is the ith modal displacement vector associated with element j, ζ_j is the desired ratio of critical damping desired for element j, and k_j is the jth element stiffness matrix, the

composite damping ratio (p_i) for mode i is defined as:

$$p_i = \sum_j \zeta_j \phi_{ij}' k_j \phi_{ij} / \sum_j \phi_{ij}' k_j \phi_{ij} \qquad (2.3.4)$$

The parameters used to define the damping matrix are then:

$$\alpha = 2 (p_1 \omega_1 \omega_2^2 - p_2 \omega_1^2 \omega_2) / (\omega_2^2 - \omega_1^2)$$
$$\beta = 2 (p_2 \omega_2 - p_1 \omega_1) / (\omega_2^2 - \omega_1^2) \qquad (2.3.5)$$

Sample problem S.3 (discussed in Section 4.2) is used to demonstrate this composite damping addition to CARES Version 1.1.

2.3.4 Addition of Shear Wall Element (Version 1.2)

A shear wall element was added to CARES so that the element library now contains: 3-D beams, 3-D springs, and shear walls. The shear wall element is a plane stress quadrilateral element. The active degrees of freedom are the two in-plane deformations at each of the four nodes of the element. Since the element stiffness matrices in CARES are stored in global coordinates however, the global stiffness matrix of the shear wall element includes the three deformations (in the global coordinate system) at each of the nodes. It is important to note that the element has no stiffness resisting out of plane deformations or any rotational deformations. These components must be restrained at those nodes where all stiffness results from shear wall elements.

The number of shear wall elements is specified with the parameter NSTIF3 which is input in the control card. The stiffness matrix for the shear wall elements is generated in the new subroutine "SHRWL." The upper half of the element stiffness matrices (in global coordinates) are stored on the scratch tape "SHEARW." The required input to define the shear wall elements is the material property data followed with the four nodes and the wall thickness. The material property data is the same as used for the 3-D beams: Young's modulus, Poisson's ratio, and damping.

The shear wall model is validated with problem S.5 discussed in Section 4.2.

2.3.5 SSI Coefficients for Rectangular Foundations (Version 1.2)

Version 1.1 of CARES contains SSI coefficients for circular embedded foundations. Rectangular foundations have been treated using an "equivalent" (based on area) circular foundation. This approximation is clearly restricted to foundations which are close to square. A rectangular foundation SSI model was therefore added to Version 1.2 of CARES.

A literature search was undertaken to find a SSI model that could be used to generate coefficients that were applicable to a rectangular foundation with specified lengths in each direction and a given depth of embedment. Of course, the fundamental approximation made in the lumped parameter SSI model of CARES remains that the foundation must be rigid. The work of Pais and Kausel (Reference 17) was selected. Pais and Kausel reviewed the available analytic solutions for rectangular foundations and generalized the results so that solutions for the full range of foundation sizes was made available. Static finite element solutions were used to fill in the gaps in the available data and to generalize the results.

The data in Reference 17 is restricted to the uniform soil case as are most of the available analytic solutions. These data are modified in the following manner to treat the case where the soil beneath the foundation has different properties from the soil along the sidewalls of the foundation. The SSI coefficients are first found for the case of a uniform soil having the properties of the soil along the sidewall. SSI coefficients for a surface foundation (zero embedment depth) are then found for the two cases with a soil having the properties of the soil beneath the foundation and the properties of the soil along the sidewall. The difference between these two solutions is then added to the initial solution for the foundation embedded in a soil with the properties of the sidewall soil. These manipulations are opaque to the user and one only needs to specify the two sets of soil properties.

2.3.6 Miscellaneous Changes (Version 1.2)

Several miscellaneous changes have been made to improve the operation of the code:

- The Veletsos SSI model for circular foundations has been eliminated from the code. It gives essentially the same results as obtained for the Beredugo - Novak and Kausel models, but is restricted to surface foundations. It therefore provided no additional capability.

- A parameter "NDIR" , input on the control card in Version 1.0 and 1.1, was used to specify the direction of the input motion and response. The purpose of this parameter was to simplify the specification of the restraints for single stick models. For example, all degrees of freedom of the single stick except for the x-displacement and rotation about the y axis were automatically

restrained when NDIR=1 (motion and response in X-direction). This of course cannot be used for multiple sticks. Since most of the recent CARES applications involve multiple sticks, this parameter has been eliminated. All degrees of freedom at all nodes are unrestrained Version 1.2 unless restraints are specified by the user. The specification of restraints has been simplified as discussed in the next item.

- The restraints are specified in three categories:

  - Individual restraints: the user specifies node and DOF restrained for each of NREST1 restraints.

  - Nodes with all DOF restrained: the user specifies node numbers for those nodes which have all DOF restrained. There are NREST2 such nodes.

  - DOF restrained at all nodes: the user specifies the DOF numbers which are restrained at all nodes. There are NREST3 such DOF.

- The diagonal elements of the global stiffness matrix are now reviewed within the code to identify all DOF which have zero stiffness. These DOF are identified by "Equation Number" printed out in the SSI.RUN file and the solution terminated because of the singularity in the stiffness matrix.The user can then review the listing of equation numbers in the SSI.RUN file to identify the node and DOF for which there is an improper specification of restraints.

3.0 USER'S MANUAL DESCRIPTION

The following pages present a description of the data input files needed to exercise the various options available in the current version of CARES. In general, the CARES program is menu driven; that is, data input is entered through the keyboard in response to prompts from the screen. In some cases, data is entered through preformed data input files which have been generated as output from previous sessions with the program, exercising other options available. Obviously, these preformed data input files can also be generated directly by the user, assuming that the file format is known. However, it should be noted that when generating these preformed data files, it is recommended that tabs not be used to space data, as difficulties may be encountered when reading the data file, particularly when using free format input. Rather, it is recommended that the spacer bar be used to obtain the desired spacing of the data records.

It should also be noted that in the UNIX or MAC versions of the Code described herein, file names are limited to 20 characters. When using the Code on a DOS machine, however, the file names are limited to 8 characters. In the descriptions provided on the following pages for the various options of the Code, the editorial format used consists of :

- CARES prompted questions in **BOLD CAPITALS**,
- Required input alphanumeric responses from the keyboard in ***BOLD ITALICS***,
- Discussion of the questions and responses in normal type.

As discussed previously, the revised CARES program has been separated into four separate applications. This does not cause significant inconvenience to the normal user, except for the need to start different applications for cases where the user wishes to transfer between options controlled by the different applications. The connection between the various operations is made through data files generated as output from each operation which is then used as input to the other operations.

For operation on the UNIX environment, the CARES application set has been separated into four modules:

CRSSOIL	- treats the free-field convolution problem,
CRSSTRUC	- treats the structural response problem,
CRSPOST	- prepares data for post processing,
CRSPLOT	- plots various data to the screen.

A number of different capabilities are included within each of the modules, in addition to the primary analysis areas listed above. The general capabilities available within each of the four CARES modules can be summarized as follows. The details of the input/output format for each

analysis option is described on the following pages.

CRSSOIL:

This module is composed of 7 analyses options.

Analysis 1: Converts an input acceleration time history data file into a modified file organized into the CARES format.

Analysis 2: Generates an artificial accelerogram to match a specified acceleration response spectrum at a given level of equipment damping. The time history so generated can then be used as input to the other modules and options in the CARES package.

Analysis 3: Generates the Fourier components appropriate for a given time history using the Fast Fourier Transform (FFT) routine. The Fourier components of the time history are then used as input to the structural response and free-field soil column response analyses.

Analysis 4: Performs the free-field convolution analysis of a given soil column using the Fourier components of a time history as input to the column. The properties of each layer of the soil column are defined in terms of the initial low strain shear modulus, the soil density and strain degradation data (shear modulus ratio and hysteretic damping). Amplification response of the soil column to unit frequency inputs can be performed by using as input Fourier components set to unit magnitude and constant degradation properties for the soil layers.

Analysis 5: Generates the time history of an accelerogram using as input the Fourier components of the motion. The inverse FFT is used in this calculation.

Analysis 6: Generates an acceleration response spectra for a given input acceleration time history.

Analysis 7: Generates the power spectral density function for a given input Fourier component file.

CRSSTRUC:

This module of the CARES revision consists of 3 separate Analysis options; namely:

Analysis 1: This option is the same as Analysis 3 in the CRSSOIL module, and was included in

this module for convenience only. The analysis option generates the Fourier components appropriate for a given time history using the Fast Fourier Transform routine. The Fourier components of the time history are then used as input to the structural response and free-field soil column response problems.

Analysis 2: This analysis option generates the input data file required for performing the structural response analysis. If an applicable data file already exists, and if it can be easily modified by changing a limited number of parameters, this option does not have to be accessed to perform the response calculation.

Analysis 3: This option performs the structural response calculations, including SSI effects, using the data input file generated in Analysis 2. The output file from this option consists of the Fourier components of the computed responses at all the output locations of the structural model specified previously. The postprocessing and plotting of this output is performed in CRSPOST.

CRSPOST:

CRSPOST is the module that is used to perform the postprocessing analysis of input or output files used in or obtained from either CRSSOIL or CRSSTRUC. The module is composed of four options.

Analysis 1: This option is used to compute the time history of a record from the Fourier components, using the inverse FFT analysis.

Analysis 2: This option is used to generate the acceleration response spectrum for a given time history record.

Analysis 3: This option is used to calculate the response spectra at output nodes of the structure from the output data file generated from CRSSTRUC.

Analysis 4: This option is used to prepare PSD related data files. The tasks are (1) generate an acceleration time history to satisfy a user-supplied target PSD function, (2) optional clipping of the acceleration time history and then calculate the PSD of the clipped or nonclipped time history, and (3) multiply the acceleration time history by an envelope function and compute the response spectra of the modified record.

CRSPLOT:

This module generates screen plots of a number of functions developed as output from the various options available from the code. For application on the SUN workstations, the CARES plotting module makes use of the IMSL Exponent Graphics plotting software.

Analysis 1: Converts an input acceleration time history data file into a modified file organized into the CARES format.

Analysis 2: Generates a screen plot of an acceleration time history that can be saved to a file.

Analysis 3: Generates a screen plot of a computed response spectrum at a given value of equipment damping which can be saved to a file.

Analysis 4: Generates a screen plot of the power spectral density function computed for a given time history that can be saved to a file.

Analysis 5: Generates an X-Y plot for any given set of data prepared in the proper input format that plots on the screen and can be saved to a file.

3.1 CRSSOIL:

Upon selection of this application, the user is given seven analysis options to proceed, which are defined by the screen prompt:

```
**********************************************************
**********************************************************
*****                  CARES (SOILS)                  *****
**********************************************************
**********************************************************
* ANALYSIS #                                             *
*                                                        *
* 1.  CONVERTS INPUT ACCELERATION TIME HISTORY           *
*     DATA FILE TO CARES FORMAT                          *
*                                                        *
* 2.  GENERATES ACCELEROGRAM TO MATCH A GIVEN            *
*     TARGET RESPONSE SPECTRUM                           *
*                                                        *
* 3.  GENERATES FOURIER COMPONENTS FOR GIVEN             *
*     ACCELEROGRAM; USED AS INPUT TO SOIL COLUMN         *
*     AND SSI ANALYSIS                                   *
*                                                        *
* 4.  PERFORMS CONVOLUTION ANALYSIS OF A GIVEN SOIL      *
*     COLUMN TO ACCELEROGRAM DEFINED IN TERMS OF         *
*     ITS FOURIER COMPONENTS                             *
*                                                        *
* 5.  GENERATES TIME HISTORY OF ACCELEROGRAM FROM        *
*     A FOURIER COMPONENT FILE                           *
*                                                        *
* 6.  GENERATES ACCELERATION RESPONSE SPECTRUM FOR       *
*     A GIVEN ACCELERATION TIME HISTORY                  *
*                                                        *
* 7.  GENERATES POWER SPECTRAL DENSITY FUNCTION          *
*     FOR A GIVEN FOURIER COMPONENT FILE                 *
*                                                        *
* 8.  EXITS FROM CRSSOIL                                 *
*                                                        *
**********************************************************
```

CRSSOIL.ANALYSIS 1:

After selecting ANALYSIS #1, CONVERTS TIME HISTORY FILES TO CARES FORMAT, the title for the subroutine appears on the screen:

TIME HISTORY FILE CONVERSION:
CONVERTS A GIVEN TIME HISTORY TO CARES FORMAT

The program is used to convert an existing time history file to the format used in CARES and to reduce the total number of time history records. The program asks for the name of the input time history file:

INPUT FILE NAME?

Input the name of the existing time history file using a maximum of 20 characters. Next, the number of header cards is requested.

NUMBER OF HEADER CARDS IN INPUT FILE?

Input the number of lines in the input file that precede the time history records. This determines the number of lines that are required to be skipped before reading the first time history record. Parameters which define the time history are then requested.

TIME INCREMENT FOR INPUT TIME HISTORY (secs)?

Input the time increment of the input time history file.

NUMBER OF RECORDS IN INPUT FILE (LE. 20,000)?

Input the total number of records in the input time history file. The program allows for any format to be used for the time history records.

ENTER THE FORMAT OF THE INPUT TIME HISTORY RECORDS

Enter the format that is used for the time history records, eg. 6F10.3. The program checks if this format is correct.

THE FORMAT OF THE TIME HISTORY RECORDS = *FORMAT*
IS THIS CORRECT (Y/N)?

If the format is correct, enter *Y*. If the format is wrong, enter *N* and the format will be requested again.

Next, enter the name to be used for the output file of the generated time history in CARES format, using less than 20 characters.

OUTPUT FILE NAME (LE.20 LETTERS)?

Input the name of the new time history file using a maximum of 20 characters.

HEADER CARD FOR OUTPUT FILE (UP TO 80 LTRS)?

Input the title card to be placed at the head of the new time history file. A maximum of 80 characters can be used. The program also allows for the option of selecting a portion of the total record, reducing the number of time steps in the newly generated time history record by skipping time points in the original time history.

ENTER RECORD INCREMENT TO BE USED?
eg. ENTER 1 TO USE EVERY TIME HISTORY RECORD
eg. ENTER 2 TO USE EVERY OTHER TIME HISTORY RECORD

Enter "1" if every time history record is to be used. Enter "2" if every other record is to be used. The final number of time history records, when using CARES, must be less than 8200.

The program also permits the user the ability to select a segment from the input time history record. The program asks:

IS ONLY A SEGMENT OF THE INPUT TIME HISTORY WANTED (Y/N)

If only a portion of the time history is required, enter *Y*. If the entire time history is required to be used, enter *N*. If *Y* is entered, the code further asks for the start and finish times to be used from the input time history:

ENTER STARTING TIME (sec)

Enter the time in which to begin the time history at.

ENTER STOPPING TIME (sec)

Enter the stopping time. The new time history will begin at the starting time established by the user and end at the stopping time. Therefore, allowing for a window of the input time history to be used. In order to scale the time history, a scale factor is also requested.

ENTER SCALE FACTOR TO CONVERT TO in/sec2 ?

Enter the scale factor that will be applied to each time history record. The program will then read the old data file, scale it by the scale factor and prepare the new output file in CARES format.

CRSSOIL.ANALYSIS 2:

After selecting ANALYSIS #2, GENERATES TIME HISTORY FOR A GIVEN TARGET RESPONSE SPECTRUM, the title for the subroutine appears on the screen:

SEISMOTION: ACCELERATION TIME HISTORY
FOR A GIVEN SPECTRA

The program can be used with an existing input file or allows the user to form an input file through prompts. The program asks:

DOES AN INPUT FILE FOR THE
CARES MOTION PROGRAM EXIST (Y/N)?

The user enters *Y* for yes or *N* for no If **YES**, the program requests the name of the input file:

INPUT FILE NAME (LE.20 LETTERS) ?

Input the name of the existing input file using a maximum of 20 characters. The program checks if the name of the input file is correct.

IS THIS NAME OKAY (Y/N) ?

The user enters *Y* for yes or *N* for no. If *Y*, the program runs without any further questions providing intermediate screen output. If user enters *N* , no input file is available and one will then be formed for future use. The file is formed by the user responding to a number of screen prompts. The title "**INPUT INFORMATION**" is first typed on the screen

The name for the new input file to be formed is requested (maximum of 20 characters):

NAME OF NEW INPUT FILE TO BE FORMED?

The title identifier card for the output files is requested. This card will be the header card on all these output files.

TITLE CARD FOR INPUT FILE (LE.80 LTRS) ?

User then inputs the title that will be used for the output files using a maximum of 80 characters. The program checks if the name of the input file and the title are correct.

INPUT FILE NAME: *AINPT*
TITLE: *ATITLE*
IS THIS CORRECT (Y/N)?

The user inputs *Y* if the input file and title are correct. If *N* , the questions are again asked.

Next, the code asks for the names of the output files to be formed. The user inputs the names for the various output files, using a maximum of 20 characters for the UNIX/MAC versions of the Code:

OUTPUT SPECTRA FILE NAME (A20).....?
Input the output spectra file name, *SPOUT*.

OUTPUT ACC TIME HISTORY FILE NAME.....?
Input the generated acceleration time history file name, *ACCOUT*.

OUTPUT VEL TIME HISTORY FILE NAME.....?
Input the generated velocity time history file name, *AVEL*

OUTPUT DISPL TIME HISTORY FILE NAME.....?
Input the generated displacement time history file name, *ADISPL*

The program echo prints the names of the output files for verification

TITLE CARD *MTITLE*
SPECTRA FILE NAME *SPOUT*
ACCELERATION FILE NAME *ACCOUT*
VELOCITY FILE NAME *AVEL*
DISPLACEMENT FILE NAME *ADISPL*

IS THIS CORRECT (Y/N)?

If the information is correct, input *Y*. If any of the information is incorrect, input *N* and the names of the files will be requested again.

For the generation of the time history, a target response spectrum is required. The number of frequency points of the target response spectrum is requested.

NO. OF SPECTRA POINTS (LE. 100) ?

Input the number of frequency points, *NOPT*, for the target response spectrum with a maximum

of 100 points. For each spectral point, ***I***, the frequency (cps) and the spectral acceleration (g's) are required.

TARGET SPECTRAL POINT *I*
FREQUENCY (HZ) ?

Input the frequency (cps), ***SFRT(I)***, at spectral point ***I***

SPECTRA VALUE (Gs) ?

Input the spectral acceleration, ***AAMPT(I)***, in g's at spectral point ***I*** . The requests for frequency and sperctral acceleration are repeated **NOPT** times.

The peak acceleration of the time history defines the maximum absolute acceleration value for the acceleration time history.

PEAK ACCELERATION (Gs) ?

Input the peak acceleration, ***PEAK***. The earthquake magnitude determines the shape of the enveloping function and the duration of the pulse.

EARTHQUAKE MAGNITUDE (FOR DURATION) ?

Input the earthquake magnitude, ***EQMAG,*** M_w. From this value. the program determines the shape of the enveloping function and prints ***EQMAG, TRISE, TSTRON, TDECAY, TDUR*** of the enveloping function. The duration, ***TDUR***, is the minimum duration of the generated function. The program then queries:

ARE THESE PARAMETERS ACCEPTABLE (Y/N)?

If ***N***, new envelope parameters can be input individually. The user can set the duration time to be longer than this minimum duration time so as to include a quiescent period following the pulse.The program queries:

DURATION TIME DESIRED (secs)?

Input the total length of the time history , ***TMAX*** , in seconds. If the duration time is entered as less than the minimum duration time, the length of the record will be set to the minimum duration time.

The motion subroutine uses the Fast Fourier Transform (FFT) algorithm. Therefore, the number of time history records must be 2^M, where M is a positive integer. To determine the total number of time history records, the user inputs the value of the parameter ***M*** :

M VALUE (# RECORDS = 2M, M< 14) ?**

Input the parameter **M** not to exceed the value of 13. The number of records is then equal to 2^M. Note that if M = 11, NRCDS = 2048; if M = 12, NRCDS = 4096; if M = 13, NRCDS = 8192.

The spectral damping requested next is the equipment damping associated with the target response spectrum.

DAMPING VALUE (%) ?

Input the value of damping in percent, ***BETA***.

The motion calculation then uses an iterative process to calculate the Fourier components of the time history. The more iterations used, generally the better the fit of the calculated response spectrum to the target response spectrum.

NUMBER OF ITERATIONS (MINIMUM = 5) ?

Enter the number of iterations, ***NITER***, to be used in the motion program. If a number less than 5 is entered, ***NITER*** will be set to 5. This input information is echo printed or.

MAXIMUM DURATION TIME = ***TMAX***
NUMBER OF TIME HISTORY RECORDS = ***NRCDS***
TARGET RESPONSE SPECTRUM DAMPING = ***BETA***
NUMBER OF ITERATIONS (5 min) = ***NITER***
ARE THESE VALUES CORRECT (Y/N)

If these values are correct, enter ***Y***. If the values are incorrect, enter ***N*** and the values will be requested again.

The phase angles used for the generation of the time history can be generated using three methods. The user is requested to select one of the three methods:

METHOD USED TO GENERATE PHASE ANGLES:
IF RANDOM NUMBERS ARE USED TO GENERATED PHASE ANGLES,

INPUT 1
IF FOURIER COMPONENTS OF AN ACTUAL EARTHQUAKE ARE USED, INPUT 2
IF A TIME HISTORY OF AN ACTUAL EARTHQUAKE IS USED, INPUT 3 (FILES ARE ASSUMED TO BE IN CARES FORMAT)

The user inputs either 1, 2, or 3 for the value of ***IPHASE.*** If either 2 or 3 is selected, the file name that includes the input information to develop the phase angles is requested. These files are assumed to be in CARES format. For ***IPHASE*** equal to 2:

INPUT THE FOURIER COMPONENT FILE NAME ?

Enter the name of the Fourier component file used to generate phase angles. After this value is entered, the value is checked:

FOURIER COMPONENT FILE NAME = *PHSFIL*
IS THIS NAME CORRECT (Y/N) ...?

If this value is correct, enter ***Y***. If the value is incorrect, enter ***N*** and question will be asked again. For ***IPHASE*** equal to 3, the time history file name is asked for.

INPUT THE TIME HISTORY FILE NAME

Enter the name of the time history file to be used to generate phase angles. The phase angles will then be determined by calculating the Fourier components of the time history and using these phase angles for the generated record. After this value is entered, the value is checked:

TIME HISTORY FILE NAME = *PHSFIL*
IS THIS NAME CORRECT (Y/N) ...?

If this value is correct, enter ***Y***. If the value is incorrect, enter ***N*** and the question will be requested again. The input information for the determination of phase angles used in the motion program are printed on the screen: For IPHASE equal to 1, the following is printed to the screen:

PHASE ANGLES ARE RANDOMLY GENERATED
IS THIS CORRECT (Y/N)

For IPHASE equal to 2:
PHASE ANGLES ARE GENERATED FROM THE FOURIER COMPONENT FILE, *PHSFIL*

IS THIS CORRECT (Y/N)

For IPHASE equal to 3:
PHASE ANGLES ARE GENERATED FROM THE FOURIER COMPONENT FILE GENERATED FROM THE TIME HISTORY, *PHSFIL*
IS THIS CORRECT (Y/N)

If this input information is correct, enter *Y* otherwise enter *N*. If *N* is entered the questions related to phase angles will be requested again.

Next, the screen prompts if the user would like to require the generated motion to satisfy the Reg Guide 1.60 enveloping criteria

> **DO YOU WANT TO APPLY REG GUIDE CRITERIA TO THE GENERATED MOTION (Y/N)**
> **A) MAXIMUM OF 5 POINTS BELOW THE TARGET**
> **B) NO POINT LESS THAN 90% OF THE TARGET**

Enter *Y* to apply the requirement of the reg guide criteria. Enter *N* to relax this requirement. The information is rechecked:

If ***AREG = Y*** is supplied by the user, the following will be printed to the screen:
REG GUIDE CRITERIA **WILL BE APPLIED TO THE GENERATED MOTION**

If ***AREG = N*** is supplied by the user, the following response is printed to the screen:
REG GUIDE CRITERIA **WILL NOT BE APPLIED TO THE GENERATED MOTION**

The input information, including the shape of the time history enveloping function, is printed on the screen for the user to check for acceptability.

> **NUMBER OF SPECTRA POINTS = *NOPT***

and for each spectra point, I, the values of frequency and spectral acceleration: ***SFRT(I)***, ***AAMPT(I)*** as well as:

PEAK ACCELERATION (Gs)	=	***PEAK***
EARTHQUAKE MAGNITUDE	=	***EQMAG***
DAMPING (%)	=	***BETA***
TIME INCREMENT (secs)	=	***DT***

NUMBER OF RECORDS	=	***NRCDS***
NUMBER OF ITERATIONS	=	***NITER***
MAXIMUM DURATION	=	***TMAX***
PULSE DURATION	=	***TDUR***
RISE TIME	=	***TRISE***
STRONG MOTION	=	***TSTRON***
DECAY TIME	=	***TDECAY***

and

for **IPHASE** =1: **PHASE ANGLES ARE RANDOMLY GENERATED**

for **IPHASE** =2: **PHASE ANGLES ARE GENERATED FROM THE FOURIER COMPONENT FILE, PHSFIL**

for **IPHASE** =3: **PHASE ANGLES ARE GENERATED FROM THE FOURIER COMPONENT FILE OBTAINED FROM THE TIME HISTORY, PHSFIL**

For **AREG** =Y: **REG GUIDE CRITERIA **WILL BE APPLIED** TO THE GENERATED MOTION**

For **AREG** =N: **REG GUIDE CRITERIA **WILL NOT BE APPLIED** TO THE GENERATED MOTION**

IS THIS DATA CORRECT (Y/N)?

If there are any errors in the information, input *N* and the questions will again be asked. If the information is correct, type *Y* and the program will proceed to generate the time history.

CRSSOIL.ANALYSIS 3:

ANALYSIS #3 calculates the Fourier Transform of a given discrete time history record using the Fast Fourier Transform (FFT), assuming that the records are uniformly spaced at a specified time increment. When Analysis #3 is selected from the CRSSOIL menu, the following screen comments appear:

PROGRAM TO COMPUTE FOURIER
COMPONENTS OF ACCELEROMETER RECORD

The input time history is assumed to have the following format:

DESCRIPTOR CARD 1
DESCRIPTOR CARD 2
DESCRIPTOR CARD 3
DESCRIPTOR CARD 4
NO. OF RECORDS, DELTA T (I5,F10.0)
ACCELERATION RECORDS, printed 7 to a card, and formatted as 7F10.0.

The units of the Fourier components will be the same as the units of the input time record.

The program next queries the user for the input time history file name.

INPUT TIME HISTORY FILE NAME?

Input the name of the input time history file, ***AFILE*** , using a maximum of 20 characters. The program checks if this input name is correct.

INPUT TIME HISTORY FILE NAME = *AFILE*
IS THIS FILE NAME CORRECT (Y/N)?

If ***Y*** is selected, the program continues to the next question. If ***N*** is selected, the input file name is again requested. A scaling factor is included in the Fourier Transform analysis to allow for scaling or redimensioning of the time history values.

SCALE TIME HISTORY UP OR DOWN (Y/N)?

Enter ***Y*** if the time history record is to be scaled by a factor. Enter ***N*** if there is no change required. If ***Y*** is selected, the factor is requested:

SCALE FACTOR.....?

Enter a scaling factor, ***SCALE*** . The program checks with the user if the scale factor input is correct.

SCALE FACTOR = *SCALE*

IS THIS SCALE FACTOR CORRECT (Y/N)?

If ***SCALE*** is correct, enter ***Y*** . If ***N*** is selected, the program again asks for the scale factor. The time history record is then read into the machine and adjusted to have 2^M records. The following summary is printed to the screen.

INPUT TIME HISTORY HAS *NNRCDS* RECORDS
HAS BEEN PADDED WITH *ICOUNT* ZEROS
NEW TIME HISTORY HAS *NRCDS* RECORDS

FIRST HEADER CARD	(Problem Title Card)	
SECOND HEADER CARD	(Peak Acceleration Card)	
NO. OF RECORDS	=	***NRCDS***
TIME INCREMENT (SEC.)	=	***DT***
PEAK POSITIVE ACCEL.	=	***XDDMAX***
AT RECORD NUMBER	=	***IMAX***
PEAK NEGATIVE ACCEL.	=	***XDDMIN***
AT RECORD NUMBER	=	***IMIN***
PEAK ACCELERATION	=	***PEAK***

The program queries if this data is correct. The name of the output file to store the Fourier components is next requested.

OUTPUT FOURIER COMPONENT FILE NAME?

Enter the output Fourier component file name using a maximum of 20 characters. The program checks the name of the output file.

OUTPUT FOURIER FILE NAME = *AOUT*
IS THIS NAME CORRECT (Y/N)?

If ***N*** is selected, the output file name is again requested. If ***Y*** is selected, a title card for the output Fourier file is requested.

TITLE CARD FOR OUTPUT FILE ?

Enter the title card for the output Fourier file using a maximum of 80 characters. The title card for the Fourier component file is printed on the screen.

NEW OUTPUT HEADER CARD = *ANAME*
IS THIS TITLE CARD OK (Y/N)?

If ***Y*** is selected, the program then continues on to calculate the Fourier transform of the time history record. If ***N*** is selected, the title for the output file is requested again.

CRSSOIL.ANALYSIS 4:

After selecting CONVOLUTION ANALYSIS, the following menu appears:

```
**********************************************************
*                                                        *
*                      CARES SOIL                        *
*                                                        *
*          READ FOURIER INPUT MOTION AND SOIL DATA,      *
*          COMPUTE FINAL SOIL STRAIN AND PROPERTIES      *
*                    (NONLINEAR ANALYSIS)                *
*                          THEN                          *
*          COMPUTE FOURIER COMPONENTS OF OUTPUT          *
*                                                        *
**********************************************************
```

The program can be used with an existing input file or allows the user to form an input file through prompts. The program asks:

IS INPUT FILE ALREADY FORMED (Y/N) ?

The user enters ***Y*** for yes or ***N*** for no.

INPUT FILE EXISTS:

If **YES**, an input file exists and the program requests the input file name:

NAME OF INPUT FILE?

Input the name of the existing data input file, using less than 20 characters. Proceed to section on soil degradation input information.

INPUT FILE DOES NOT EXIST:

If an input file does not exist, the input data file is formed by the user responding to screen prompts. A data file of these responses is then formed for future use.

Two options are allowed for the analysis, soil amplification for a pulse of unit amplitude at each frequency component over the frequency range or for the complete convolution solution for a specified input time history represented by a Fourier input file. The program prompts:

OPTION 1 = SOIL AMPLIFICATION DUE TO A SPECIFIED TIME HISTORY
OPTION 2 = SOIL AMPLIFICATION DUE TO A UNIT PULSE

Type in the **OPTION #.** The program then checks if this option is correct.

For **OPTION 1: ANALYSIS WILL BE PERFORMED FOR A SPECIFIED TIME HISTORY**
For **OPTION 2: ANALYSIS WILL BE PERFORMED ON A UNIT PULSE**

IS OPTION = "OPTION #" CORRECT?

If the response is *Y* program proceeds; if *N* the query is repeated. The program then prompts for the name of the input file to be formed.

NAME OF INPUT FILE TO BE FORMED?

Input the name of the input file, ***ARUN***, using less than 20 characters.

PROBLEM TITLE CARD?

Enter a title card using a maximum of 80 characters. This title card will be used as an identifier for the output data files.

NUMBER OF SOIL COLUMN LAYERS?

Input number of soil layers, ***MLAYER***. The maximum number of soil layers allowed is 30. If a rock outcrop is to be included in the soil column, the maximum number of soil layers is equal to **(MLAYER-1)** or 29. The program calculates the effective shear stress and effective vertical stress ratios for liquefaction studies. Therefore, the depth to the ground water table must be specified:

IS GROUND WATER TO BE CONSIDERED IN THE PROBLEM (Y/N) ?

If a ground water table exists in the problem, enter *Y* to include the effect of ground water in the effective stress computation. If ground water does not exist, enter *N* . If *Y* is input, the program requests:

DEPTH TO GROUND WATER TABLE (ft)

Enter the depth to ground water table , ***ZGWT***. A review of the input information is then displayed on the screen.

NUMBER OF SOIL LAYERS = ***MLAYER***

If ground water exists the screen will display:

DEPTH TO GROUND WATER = *ZGWT*

If no ground water exists, the screen will display:

GROUND WATER IS NOT CONSIDERED IN THIS PROBLEM

then **IS INPUT CORRECT (Y/N)**

Enter ***Y*** if the information is correct or ***N*** if there is an error. If ***N*** is input, questions related to the number of layers and the ground water table are asked again. The layer at which the ground motion is to be input is then requested.

LAYER INTERFACE OF INPUT MOTION?
(SURFACE= 0, OUTCROP= MLAYER +1)

Enter the interface at which the input motion is to be located, ***NSPEC***. If the input motion is located at the ground surface, enter ***0*** . If the motion is located at the base of one of the soil layers, input the ***soil layer #***. If the input motion is located at the rock outcrop, enter the ***maximum soil layer # +1***. The information is then checked.

NUMBER OF SOIL LAYERS **MLAYER**
INPUT LOCATION **NSPEC**
IS INPUT CORRECT (Y/N)?

If input is correct, enter ***Y*** and the program continues. If the answer is ***N***, the prompts above will be repeated.

For the case of a specified time history applied to the soil column, or ***IOPT*** = 1, the Fourier Component Input file name is then requetsed. When the problem is for a unit input, or ***IOPT*** = 2, this question will be omitted.

INPUT FOURIER COMPONENT FILE NAME ?

Enter the Fourier component file name using a maximum of 20 characters. The input is then checked. For a specified input time history:

FOURIER COMPONENT FILE : *AFILE*
IS INPUT AT INTERFACE *NSPEC*
IS THIS CORRECT (Y/N) ?

or for a unit pulse

UNIT PULSE
IS INPUT AT INTERFACE *NSPEC*
IS THIS CORRECT (Y/N)

Enter whether the information is correct. If the information is incorrect, enter *N* and the cycle of questions will be repeated. For each of the **MLAYER** soil layers, properties of the soil layers are then specified.

SOIL COLUMN CONFIGURATION
LAYER "I"

LAYER THICKNESS (ft)?	Enter layer thickness, t , for layer "I"
UNIT WEIGHT (#/ft3)?	Enter total unit weight, γ, for layer "I"
SHEAR MODULUS (#/ft2)?	Enter shear modulus, G , for layer "I"

After the information about a layer is input, the layer information is echo printed.

CHECK LAYER DATA : "I"

LAYER THICKNESS (ft)	t
UNIT WEIGHT (#/ft3)	γ
SHEAR MODULUS (#/ft2)	G
IS INPUT CORRECT (Y/N)?	

If the layer description is correct, enter *Y* . If information is incorrect, enter *N* and information can be reentered. A summary list of the layer properties is then printed and correctness requested. If the motion input is not specified at the rock outcrop, the program will prompt if the rock outcrop is to be considered in the analysis.

IS OUTPUT MOTION REQUIRED AT ROCK OUTCROP (Y/N)

If input motion is specified at the rock outcrop or if *Y* is selected, the questions pertaining to the rock outcrop properties are asked.

ROCK UNIT WEIGHT (#/ft3)?	Enter rock unit weight (#/ft3), γ_{rock}
ROCK SHEAR MODULUS (#/ft2)?	Enter rock shear modulus (#/ft2), G
ROCK DAMPING (%)	Enter rock hysteretic damping (%), D

The outcrop properties are then checked.

CHECK ROCK DATA

ROCK UNIT WEIGHT (#/ft3)	γ rock
ROCK SHEAR MODULUS (#/ft2)	G

ROCK DAMPING (%) **D**

IS INPUT CORRECT (Y/N)?

Enter ***Y*** if input is correct or ***N*** if corrections are required in this data block

Output information is next considered.

OUTPUT INFORMATION

When the option for amplification of a specified input time history is chosen (***IOPT*** =1), effective strain will be calculated to determine degraded soil properties, namely, the shear modulus and hysteretic damping, of each layer of the soil column. The effective strains are required for the soil degradation models. Two (2) options are available for the calculation of effective strains. Strain can be calculated in the time domain by applying the inverse FFT to the computed Fourier strain components. Alternatively, an approximate effective strain can be calculated from the Fourier components of the strains by using Parseval's equality. The calculation in the time domain is more accurate. However the process is slower than the calculation in the frequency domain.

IS STRAIN CALCULATED IN THE TIME (T) OR FREQUENCY (F) DOMAIN

Enter ***T*** if the calculation is to be calculated in the time domain or ***F*** if the calculation is to be done in the frequency domain. The user can also use only a portion of the frequencies in the strain calculation, by allowing the user to specify a cut-off frequency in the strain computation.

DO YOU WANT TO USE A CUT-OFF FREQUENCY FOR THE STRAIN CALC (Y/N) ?

If the user desires the calculation to include all the Fourier components in the strain calculation, enter ***N***. If an upper frequency limit is desired for the strain calculation, enter ***Y*** and an additional question will then be asked for the cut-off frequency:

STRAIN CUT-OFF FREQUENCY (hz)..?

Enter the value of the cutof frequency, ***FCUTS*** . All frequencies above this value will be neglected in the strain computations. Otherwise, the cut-off frequency will be set to 1000 hz as a default. The program then requests the acceptable error in the strain computation.

MAXIMUM ALLOWABLE ERROR IN STRAIN CALCULATION (%) ?

The strain calculation is an iterative process which involves an assumed value in each soil layer at the beginning of the iteration and a calculated strain obtained at the end of the iteration. The

acceptable error between the assumed and the calculated value indicates the end to the iterative process. The lower the allowable strain error, the more iterations that will be required to complete the convolution calculation. A typical value used for the maximum allowable error is 5%.

A file is created that includes an echo of the input information and the final soil properties. The user is queried for the name of this final soil properties file:

FINAL SOIL PROPERTIES FILE NAME ?

The user inputs the final soil properties file name using a maximum of 20 characters. The input information related to the final properties for the soil profile are displayed on the screen. For a strain calculation in the time domain, the screen displays

STRAIN CALCULATION IN TIME DOMAIN
MAXIMUM ALLOWABLE ERROR = ***ERRMAX***

while for a calculation in the frequency domain, it displays

STRAIN CALCULATION IN FREQUENCY DOMAIN
MAXIMUM ALLOWABLE ERROR = ***ERRMAX***

For either cases, the screen then continues to display

STRAIN CUT-OFF FREQUENCY = ***FCUTS*** **cps**
FINAL SOIL PROPERTIES FILE NAME: ***FSOIL***
IS DATA CORRECT (Y/N)

If all the information related to final soil properties is correct, enter ***Y***. If there are any errors, enter ***N*** and the queries for strain and final soil properties will be repeated.

Motion output can be specified at multiple locations in the soil column, up to a maximum of six locations. The motion at each depth is obtained in terms of a Fourier component file which can then be converted to a time history as described previously. The program queries:

NUMBER OF OUTPUT DEPTHS FOR FOURIER COMPONENTS
(FOR STRAIN-ONLY CALCULATION INPUT 0).....?

If no output Fourier files are required, enter ***0*** and output will be restricted to the iterated stress and strain solution in each soil layer. If output Fourier files are desired, enter the number of depths, ***NDEPTHS*** (up to a maximum of 6 output depths). If **NDEPTHS** is set equal to 0, the program types:

ANALYSIS IS FOR STRAIN ONLY
IS INPUT CORRECT (Y/N)?

If the answer is *Y*, the program continues and the input data file is completed. If the response is *N*, the program queries for number of output depths requested. If **NDEPTHS** is not set equal to 0, information about depths and file names for storage of the output Fourier records is then requested. If rock data is included and the input motion is not specified at the rock outcrop, the program assumes that output files are to include the outcrop motion.

OUTCROP MOTION FILE NAME?

Input the name of the outcrop motion file, using a maximum of 20 characters. For the remaining depths, the depth (in ft from the ground surface) and the name of the output file for that depth is requested.

DEPTH FOR OUTPUT "I" (ft)?

Enter depth for output *I* in feet.

OUTPUT FILE NAME FOR DEPTH "I"?

Enter output file name using a maximum of 20 characters.

After the cycle of depths is completed, the information is echo printed on the screen for the user to review.

CHECK OUTPUT INFORMATION

NUMBER OF OUTPUT DEPTHS	**NDEPTHS**

If output is given at the outcrop:

OUTCROP MOTION FILE NAME	**OUTPUT FILE NAME**

And for the remaining depths:

OUTPUT FILE NUMBER	**I**
OUTPUT DEPTH (ft)	**DEPTH**
OUTPUT FILE NAME	**OUTPUT FILE NAME**

IS INPUT CORRECT (Y/N)?

If *Y* , the program continues. If *N* , the program requests that the output information be resubmitted.

SOIL DEGRADATION INPUT

For either the case of an existing soil profile input file or the case of a new soil profile, information on the type of soil degradation to be used for each soil layer is required. To allow for the option of using an existing soil degradation file or to create a new soil degradation file, the following option is included:

DOES A SOIL DEGRADATION FILE EXIST (Y/N)?

If *Y* , the program prompts to enter the soil degradation file name.

SOIL DEGRADATION FILE NAME?

Enter the name of the soil degradation file using less than 20 characters.The input name for the soil degradation file is then checked:

SOIL DEGRADATION FILE: ***DEGRAD***
IS THIS CORRECT (Y/N)

In the case of an error in the input soil degradation name, enter *N* and the question is repeated. If the file name is correct, enter*Y* . The program then continues and calculates the soil amplification for each of the frequencies specified in the input Fourier component file.

SOIL DEGRADATION FILE PREPARATION:

If a soil degradation file does not exist, the program will then generate a degradation data file through prompts asking for data through keyboard entry, responding to the following questions.

SOIL DEGRADATION FILE NAME TO BE FORMED?

Enter the name of the file using up to 20 characters The program then request the header card to be used for the file:

TITLE FOR SOIL DEGRADATION FILE?

Enter the title card for the soil degradation file using less than 80 characters. The information supplied by the user is echo printed.

SOIL DEGRADATION FILE: ***DEGRAD***
TITLE: ***ATITL1***
IS THIS CORRECT (Y/N)

If any of the information is incorrect, enter *N* and the questions will be repeated.

Five soil degradation models are available for each soil layer in the soil column from which to choose, namely: (1) the SEED-IDRISS 1970 model, (2) the IDRISS 1990 model, (3) the GEI

model, (4) the CONSTANT model and (5) OTHER. The constant model refers to a model with no soil degradation. Option 5 allows the user to submit his/her own soil degradation model.

The following menu appears on the screen

INPUT # OF SOIL DEGRADATION MODEL TO BE USED
(1) SEED-IDRISS 1970
(2) IDRISS 1990
(3) GEI
(4) CONSTANT
(5) OTHER

Enter the soil model desired, ***ITYPE*** . The program echoes the soil degradation type selected

SOIL DEGRADATION MODEL USED = ITYPE
IS SOIL DEGRADATION MODEL # CORRECT (Y/N)?

If ***Y*** is typed, the program continues. If ***N*** is typed, the program again requests the soil degradation type.

Soil degradation models "1" and "2" are based on the type of soil, sand or clay, in the soil layer. For each soil layer, the soil type is requested.

SOIL TYPE OF LAYER "I" SAND (S) OR CLAY (C)?

Input soil type ***S*** for sand or ***C*** for clay. Degradation type 3 (the GEI model) is considered appropriate for sands or silty sands. All degradation data is already included in the code for **ITYPE =1, 2 or 3**. Soil degradation for **ITYPE = 4** is equivalent to a model in which there is no shear modulus degradation and constant damping. The ratio G/G_{max} is set to unity. However, for each soil layer, the constant value of hysteretic damping ratio (%) is required.

DAMPING VALUE (%) FOR LAYER "I"?

Input damping, ***D***, for layer "I".The program allows for the user to check this number by echo printing the damping value.

CONSTANT DAMPING FOR LAYER "I" = *D* %
IS DAMPING CORRECT (Y/N)?

If damping is correct, enter ***Y***. If ***N*** , the value for damping will again be requested. The soil degradation for **ITYPE = 5** allows for a user specific model to be entered for each soil layer at a specified number of strain values through screen prompts.

NUMBER OF STRAIN VALUE POINTS (max 15) ?

Enter the number of strain value points, ***NPTS*** , at which data for each layer will be specified (with a maximum of 15 points). For each point, the strain value is requested.

STRAIN VALUE (%) FOR POINT "IPT"?

Enter the shear strain value (in %) for strain point IPT. If there are any errors in the input, the program allows for reentry after the last strain point.

ARE THESE STRAIN VALUES CORRECT (Y/N)

If the answer is ***Y*** , the program continues; if not, the program requests reentry of the strain data. For each soil layer, the G/G_{max} ratio and the hysteretic damping ratio is required at each strain point.

G/GMAX FOR LAYER "I" & POINT "IPT"

Enter the G/G_{max} ratio. After all points are entered for layer "I", the G/G_{max} values are echo printed on the screen for the user to review.

G/GMAX FOR LAYER "I" AND POINT "IPT" = *G/Gmax*
ARE G/Gmax RATIOS CORRECT (Y/N)?

Enter ***Y*** if values are correct. Enter ***N*** if there any errors. Next, the hysteretic damping ratio is input at each strain point.

HYSTERETIC DAMPING (%) FOR LAYER "I" & STRAIN POINT "IPT"

Enter Damping, ***D***. After all points are entered for layer "I", the damping values are echo printed on the screen for the user to review.

DAMPING FOR LAYER "I" AND POINT "IPT" = *D* (%)
ARE DAMPING VALUES CORRECT (Y/N)?

Enter ***Y*** if values are correct. Enter ***N*** if there any errors. If ***N*** is entered, the program returns to the point of reentering the damping values. For both cases of an existing and a newly created soil column input file, the entire input is displayed on the screen.

CRSSOIL.ANALYSIS 5:

Analysis #5 calculates the inverse Fourier Transform of a given Fourier component file to generate a corresponding time history, using the FFT. When Analysis #5 is selected from the CRSSOIL menu, the following appears on the screen:

PROGRAM TO COMPUTE TIME HISTORY

INPUT FOURIER COMPONENT FILE NAME?

Input the name of the Fourier component file, ***AFILE*** , using a maximum of 20 characters. The Fourier component file should be in the following format:

4 DESCRIPTOR CARDS	
MAXIMUM TIME (SECS)	T35, E15.5
DELTA TIME (SECS)	T35, E15.5
NO. OF TIME RECORDS	T35, I10
NO. OF FOURIER COMPONENTS	T35, I10
DELTA FREQUENCY (CPS)	T35, E15.5
MAXIMUM FREQUENCY (CPS)	T35, E15.5
MAX. ACCEPTABLE FREQ. (CPS)	T35, E15.5
PEAK ACCELERATION (IPS2)	T35, E15.5

7 LINES which indicate the descriptors:
COSINE AND SINE COEFFICIENTS (29x, 2E12.0) FOR EACH FOURIER COMPONENT

The user inputs the name of the input Fourier component file, ***AFILE*** , using a maximum of 20 characters. The program then asks

FOURIER COMPONENT FILE NAME = *AFILE*
IS THIS FILE NAME CORRECT (Y/N)?

If ***Y*** is selected, the program continues on. If ***N*** is selected, the input file name is again requested. The program then requests the name of the output acceleration time history file to which the time history is written.

ENTER OUTPUT TIME HISTORY FILE NAME?

Enter the output time history file name, ***ATHIST*** , using a maximum of 20 characters. The program checks the name of the output file. If it is correct, the inverse Fourier Transform is executed. If not, the file name is again requested.

CRSSOIL.ANALYSIS 6:

Analysis #6 calculates the response spectrum of a given time history record. When Analysis #6 is selected, the following appears on the screen:

CODE TO TAKE ACCELEROGRAM PULSE FROM AN INPUT FILE
AND
COMPUTE SPECTRA AT DAMPING (%) AND STORE ON FILE SPECT
ACCELEROGRAM PULSE d = IN/SEC2**

The program asks for the acceleration time history record.

INPUT TIME HISTORY FILE NAME?

Input the file name of the time history, ***AINPT*** ,using a maximum of 20 characters. The time history should have the following format:

4 DESCRIPTOR CARDS
NO. OF TIME RECORDS, DELTA TIME (SECS) (I5, F10.4)
TIME HISTORY (IN/SEC**2) stored 7 to a card (7F10.0)

The user then inputs the name of the response spectrum file, ***SPCT*** , using a maximum of 20 characters.

OUTPUT RESPONSE SPECTRUM FILE NAME ?

The program then checks if these file names, ***AINPT*** and ***SPCT*** , are correct.

The program next requests the equipment damping for which the spectrum is to be computed. Enter the value for damping in percent.

DAMPING (%)?

The program then echo prints the value of the spectral damping and asks if this is correct.

DAMPING = *BETA* %
IS THIS CORRECT (Y/N) ?

If correct, enter ***Y*** and the response spectrum is computed. If incorrect, enter ***N*** and the damping is again requested.

CRSSOIL.ANALYSIS 7:

Analysis #7 calculates the response spectrum of a given time history record. When Analysis #7 is selected, the following appears on the screen:

PROGRAM TO COMPUTE POWER SPECTRAL DENSITY FUNCTION FROM A FOURIER COMPONENT FILE

The program asks for the Fourier component file name which is assumed to be in the CARES format.

INPUT FOURIER COMPONENT FILE NAME

Enter the Fourier component file name using a maximum of 20 characters. The file name is assumed to be CARES format. The file name is checked:

FOURIER COMPONENT FILE NAME = AINPT
IS THIS FILE NAME CORRECT (Y/N) ?

If there is an error in the file name ***AINPT***, enter ***N*** and supply the correct file name. If the file name is correct, enter ***Y*** and the name of the power spectral density function file name will be asked for.

ENTER OUTPUT PSD FILE NAME ?

Input the desired name for the output file name using a maximum of 20 characters. After entry of this name, the name is checked:

OUTPUT PSD FILE NAME = *APSD*
IS THIS FILE NAME CORRECT (Y/N) ?

If there is an error, enter ***N*** . Otherwise enter ***Y*** and the program will proceed to calculate the PSD function of the Fourier component file.

3.2 CRSSTRUC:

CRSSTRUC is the CARES module that performs the structural response calculation, including the effects of soil-structure interaction. Included in the code are the options to (1) calculate the Fourier transform of a given time history, (2) generate the data input file required for input to the SSI analysis, and (3) perform the soil-structure response analysis. The following menu appears when the module is initiated:

```
**************************************************
**************************************************
**********     CARES (STRUCTURE)          **********
**************************************************
**************************************************
*   ANALYSIS #                                   *
*                                                *
*                                                *
* 1.  GENERATES THE FOURIER COEFFICIENTS FOR A   *
*     GIVEN ACCELEROGRAM                         *
*                                                *
* 2.  FORMS THE DATA FILE REQUIRED AS INPUT TO   *
*     STRUCTURAL RESPONSE SSI ANALYSIS           *
*                                                *
* 3.  PERFORMS THE SSI ANALYSIS                  *
*                                                *
* 4.  EXITS FROM CARES                           *
**************************************************
```

INPUT ITEM FOR THE ANALYSIS..(1,2,3, or 4)

The user inputs the analysis number, ***ITYPE***, of the category of interest. The program checks if this is the correct analysis number.

ANALYSIS # = *ITYPE*

IS THIS CORRECT (Y/N)

If ***ITYPE*** is correct, the program will initiate the routine selected. If ***ITYPE*** is incorrect, the menu will again appear and the value of ***ITYPE*** will again be requested. After the completion of an analysis, the menu will again appear and another analysis number can be selected.

CRSSTRUC.ANALYSIS 1:

Analysis #1 calculates the Fourier transform of a given discrete time history record using the FFT, assuming that the time records are uniformly spaced at a given time increment. This module is identical to that in CARES.SOIL.ANALYSIS #3 and has been included in this module for convenience only. When Analysis #1 is selected from the CARES.STRUCTURE menu, the following screen comment appears:

PROGRAM TO COMPUTE FOURIER
COMPONENTS OF ACCELEROMETER RECORD

The program requests the title card that is to be used as the first line of the output Fourier component file.

TITLE NAME FOR OUTPUT FILE?

Input the title using a maximum of 80 characters. The input time history file is assumed to have the following format:

4 DESCRIPTION CARDS
# of RECORDS, DELTA TIME (I5,F10.0)
ACCELERATION RECORD, 7 to a card, formatted as 7F10.0

The units of the output Fourier components will be the same as the units of the input time record.

The program next requests the user for the input time history file name.

TIME HISTORY FILE NAME?

Input the name of the input time history file, ***AFILE*** , using a maximum of 20 characters. The program checks if this input is correct.

INPUT DATA FILE NAME = *AFILE*

IS THIS FILE NAME CORRECT (Y/N)?

If *Y* is selected, the program continues to the next question. If *N* is selected, the input file name is requested again. A scaling factor is included in the Fourier transform analysis to allow for scaling or redimensioning of the time history values.

ENTER MULTIPLICATION FACTOR FOR TIME HISTORY VALUES?

Enter a scaling factor, ***SCALE***. If no change is desired in the time history record, enter a scale factor equal to 1.0. The program checks with the user if the scale factor input by the user is correct

MULTIPLICATION FACTOR= ***SCALE***

IS THIS MULTIPLICATION FACTOR CORRECT (Y/N)?

If ***SCALE*** is correct, enter ***Y*** and the program will continue. If ***N*** is selected, the program again asks for the multiplication factor.

The time history record is then read and the following summary of the time history record is printed on the screen

NO. OF RECORDS	=	***NRCDS***
TIME INCREMENT (SEC)	=	***DT***
PEAK POSITIVE ACCEL.	=	***XDDMAX***
AT RECORD NUMBER	=	***IMAX***
PEAK NEGATIVE ACCEL.	=	***XDDMIN***
AT RECORD NUMBER	=	***IMIN***

The name of the output file to store the Fourier components is next requested.

OUTPUT FOURIER COMPONENT FILE NAME?

Enter the output Fourier component file name using a maximum of 20 characters. The program checks the name of the output file.

OUTPUT DATA FILE NAME = ***AOUT***

IS THIS FILE NAME CORRECT (Y/N)?

If ***Y*** is selected, the program continues. If ***N*** is selected, the output file name is asked for again. The program then continues to execute the Fourier transform of the time history.

CRSSTRUC.ANALYSIS 2:

This option prepares a structural data file. Input from the screen is in free format. However, formats are given for all input parameters to assist in editing the file once it has been created. After selecting Analysis 2, the following screen message appears:

```
**************************************************
*                                                *
*                                                *
*     C A R E S      S Y S T E M                 *
*                                                *
*       (MODULE: SEISMIC/STRUCTURES)             *
*                                                *
*                                                *
*   STRUCTURES ARE MODELED USING TWO OR THREE    *
*   DIMENSIONAL BEAM AND SPRING ELEMENTS WITH    *
*   INPUT MOTION APPLIED TO THE MODEL.           *
*   THE NODAL SEQUENCE SHOULD BE SELECTED TO     *
*   GIVE SMALL BANDWIDTH                         *
**************************************************
```

PRESS RETURN KEY WHEN READY TO CONTINUE

As indicated , this option is used to prepare the data file required as input to the SSI calculation performed in Analysis 3. Proceed by pressing ***RETURN.***

ENTER NAME FOR THE FILE STORING INPUT DATA

Type in a name for the file to store the SSI model data ***FILENM*** (A20)

SSI ANALYSIS DATA FILE = FILENM
IS THIS NAME CORRECT..... (Y/N) ?

N ---- go back to previous command
Y-----proceed

If yes ,

ENTER TITLE FOR THE PROBLEM:

Type in a title for the analysis : ***TITLE*** (/A80)

Control information is next requested:

ENTER CONTROL CARD:
NOD NSTIF1 NSTIF2 NSTIF3 IANAL
NOD = NUMBER OF NODES (<500)
NSTIF1 = NUMBER OF BEAM ELEMENTS
NSTIF2 = NUMBER OF SPRING ELEMENTS
NSTIF3 = NUMBER OF SHEAR WALL ELEMENTS
IANAL = 0 FREQUENCY DOMAIN SHAKER
= 1 FREQUENCY DOMAIN TIME HISTORY

Type in parameters in one line ***NOD, NSTIF1, NSTIF2 , NSTIF3, IANAL*** (5I5).

These parameters are then displayed for check purposes:

CONTROL PARAMETERS

NUMBER OF NODES = NOD
NUMBER OF BEAM ELEMENTS = NSTIF1
NUMBER OF SPRING ELEMENTS = NSTIF2
NUMBER OF SHEAR WALL ELEMENTS = NSTIF3
ANALYSIS TYPE = IANAL
IS THE CONTROL CARD OK..... (Y/N)?
N------ go back to previous command
Y------ proceed

If yes, the nodal and lumped mass data will be input starting with node 1:

ENTER NODAL COORDINATES AND LUMPED WEIGHTS
INPUT NUMBER = 1
NODE NO. X-COORD. Y-COORD. Z-COORD. WEIGHT
N (FT) (FT) (FT) (KIP)

Type in ***coordinate data*** accordingly (I5, 3E12.4, E15.4). The same command will be repeated for each node until all nodes are processed

ENTER NO. OF NODAL ROTARY AND TORSIONAL INERTIAS:

Type in the ***number of rotary and torsional inertias*** (I5). If none, enter ***0***

NO. OF NODAL ROTARY AND TORSIONAL INERTIAS
IS THIS NUMBER OK..... (Y/N) ?

N go back to previous command
Y ***continue***

If the number of rotary inertias is greater than zero , input the following for each:

INPUT NUMBER = I
NODE NO. DIR.(4-X,5-Y,6-Z) INERTIA(KIP-FT*SEC2)**

Type in ***node number, direction,and the inertia value*** (2I5, E15.4)

The same procedure is repeated for all rotary degrees of freedom. If the number of rotary inertias is zero, the above input prompt is skipped .

ENTER NO. OF COUPLED DEGREES OF FREEDOM:

Type in the ***number of coupled degrees of freedom*** (I5)

NO. OF COUPLED DEGREES OF FREEDOM =
IS THIS NUMBER OK..... (Y/N) ?

N ---- go back to previous command
Y-----proceed

If yes,

INPUT NUMBER =
SLAVE NO. MASTER NO.
DIR.(1=X,2=Y,3=Z,4=RX,5=RY,6=RZ)

Type in ***slave, master, DOF*** (3I5)

The same procedure is repeated until all coupled data is input.

Restraints to structural degrees of freedom are specified next. The following prompt is given:

ADDITIONAL DOF RESTRAINTS CAN BE GIVEN IN 3 WAYS:
NREST1 = # INDIVIDUAL RESTRAINTS - SPECIFY: NODE, DOF
NREST2 = # NODES WITH ALL DOF RESTRAINED - SPECIFY NODES
NREST3 = # DOF RESTRAINED AT ALL NODES - SPECIFY DOF

Type in ***NREST1, NREST2, NREST3*** (3I5)

If ***NREST1*** is not equal to zero, repeat the following ***NREST1*** times:

INDIVIDUAL RESTRAINT (#) NODE DOF ?

Type in ***node, DOF*** (2I5)

If ***NREST2*** is not equal to zero, read node numbers with all ***DOF*** restrained:

NODES WITH ALL DOF RESTRAINED NODE (i) ,i=1,NREST2# ?

Type in ***node 1, node 2,....*** (14I5)

If ***NREST3*** is not equal to zero, read ***DOF*** restrained at all nodes:

DOF RESTRAINED AT ALL NODES: REST (#) DOF # ?

Type in ***DOF 1, DOF 2,....***(6I5)

The number of rigid links are specified next beginning with the following prompt:

ENTER NO. OF RIGID LINKS CONNECTING SLAVE TO MASTER NODES:
LIMITATIONS: SSI NODE CANNOT BE SLAVE;
SLAVE NODE CANNOT BE MASTER TO ANOTHER SLAVE

Type in ***number of rigid links*** (I5)

NO. OF RIGID LINKS =
IS THIS NUMBER OK..... (Y/N) ?
N ---- go back to previous command
Y-----proceed

If the number of rigid links is not zero, repeat the following for each rigid link

INPUT NUMBER = 1
MASTER NODE AT END OF LINK, SLAVE NODE AT END OF LINK

Type in ***master node number and slave node number*** (2I5).

The nodes at which output data is saved is specified. Note that post proceesing of structural response data (e.g., spectra, time histories) can only be done at these nodes:

HOW MANY OUTPUT NODES DO YOU WANT FOR POST-PROCESSING ?
FILE WILL BE SAVED CONTAINING FOURIER COMPONENTS OF RESPONSE AT EACH DOF FOR EACH NODE

Type in the ***number of nodes to be saved for post-processing*** (I5)

NO. OF NODES FOR POST-PROCESSING =
IS THIS NUMBER OK..... (Y/N) ?

N ---- go back to previous command
Y ---- proceed

If yes ,**SPECIFY NODE NUMBERS FOR POST-PROCESS:**

Type in ***all desired node numbers*** (14I5) for post-processing

Element data input will start next :

If **NSTIF1** is greater than 0, the beam data is specified:

BEAM ELEMENT DATA

ENTER NUMBER OF MATERIAL GROUPS (<200):

Type in ***number of material groups*** (I4)

FOR MATERIAL GROUP NUMBER = I
ENTER: E(YOUNG'S MOD) G(SHEAR MOD) DAMPING RATIO
(KSF) (KSF) (%)

Type in ***Young's modulus, shear modulus and damping ratio*** (3E20.4)

The same data is input for each of the material groups.

ENTER NUMBER OF SECTION PROPERTY GROUPS (<200):
LOCAL AXES: X-AXIS OF MEMBER; Y- IS IK NODES PLANE

Type in the ***number of section property groups*** (I4)

The following is repeated for each of the section property groups:

FOR SECTION PROPERTY GROUP NUMBER = I
ENTER : AREA SH AREA-Y SH AREA-Z IY IZ J
----------SQ FEET---------- --FT4--**

Type in the ***cross sectional area, shear area for bending about local y axis, shear area for bending about local z axis, moment of inertia about local y axis, moment of inertia about local z axis, torsional inertia*** (6E15.3)

Repeat the following for each of the NSTIF1 beam elements:

FOR BEAM NUMBER = I
ENTER: START, END, K-NODES, MAT# , SECT#

Type in the ***required data*** (5I5).

If **NSTIF2** is greater than 0, the spring element data is then input:

SPRING ELEMENT DATA

FOR SPRING ELEMENT NUMBER = I
ENTER NODE NOS: START, END, K-NODE (ON LOC. X-Y PL)
LATERAL STIFFNESSES: KX, KY, KZ (AXIAL DIRECTION)
ROTARY STIFFNESSES: MX, MY, MZ (VECTOR RULE)
COUPLED STIFFNESS:MX-KX, MY-KX, MZ-KX, MY-KY,
MZ-KY, MZ-KZ

Type in ***node data, lateral stiffnesses, rotary stiffnesses, coupled stiffnesses*** (3I5, 6E11.4, 6E12.4)

If **NSTIF3** is greater than 0, the shear wall data is specified:

SHEAR WALL DATA

ENTER NUMBER OF MATERIAL GROUPS (<200):

Type in ***number of material groups*** (I4)

FOR MATERIAL GROUP NUMBER = I
ENTER: E(YOUNG'S MOD) POISSON' RATIO DAMPING RATIO
(KSF) (%)

Type in ***Young's modulus, poisson's ratio and damping ratio*** (3E20.4)

The same data is input for each of the material groups.

The following is specified for each of the NSTIF3 shear walls (note that the nodes I J K L must be in order around the element):

SHEAR WALL (#) INPUT: MAT #; THICKNESS; NODES I J K L

Type in the ***required data*** (I5, F10.4, 4I5)

The SSI data is defined next:

SELECT SSI MODEL..ITYPE:
IF CIRCULAR FOUNDATION (1, 2 , OR 3)
1.FREQUENCY INDEPENDENT PARAMETER MODEL
2.BEREDUGO NOVAK MODEL
3.KAUSEL MODEL

IF RECTANGULAR FOUNDATION (5, 6, OR 7) 1
5.FOR ENTIRE RECT FOUNDATION
6.FOR UNIT WIDTH IN X-DIRECTION
7.FOR UNIT WIDTH IN Y-DIRECTION
AND
NODE AT WHICH SSI IS APPLIED (INODE)

ITYPE, INODE = ?

Type ***ITYPE, INODE*** (2I5)

If the foundation is circular:

ENTER SOIL PROPERTIES AND GEOMETRY:

SOIL BENEATH FOUNDATION
G = SHEAR MODULUS (KSF)
MU = POISSON RATIO
DEN = SOIL DENSITY (K/FT3)**

SOIL TO SIDE OF FOUNDATION
GS = SHEAR MODULUS (KSF)
MUS = POISSON RATIO
DENS = SOIL DENSITY (K/FT3)**
H = DEPTH OF BURIAL (FT)
R = RADIUS OF FOUNDATION (FT)
DBETA = SOIL DAMPING (%)

Type in ***G, MU, DEN, GS, MUS,*** **DENS, H, R, DBETA**
(3E15.4/, 3E15.4/, 3F10.3)

If the foundation is rectangular:

ENTER SOIL PROPERTIES AND GEOMETRY:

SOIL BENEATH FOUNDATION
G = SHEAR MODULUS (KSF)
MU = POISSON RATIO
DEN = SOIL DENSITY (K/FT3)**

SOIL TO SIDE OF FOUNDATION
GS = SHEAR MODULUS (KSF)
MUS = POISSON RATIO
DENS = SOIL DENSITY (K/FT3)**
DBETA= SOIL DAMPING (%)
H = DEPTH OF BURIAL (FT)
LX = LENGTH OF FOUNDATION IN X DIRECTION (FT)
LY = LENGTH OF FOUNDATION IN Y DIRECTION (FT)

Type in *G, MU, DEN, GS, MUS, DENS, DBETA, H LX LY*
(3E15.4/, 3E15.4/ 4F10.3)

The structural damping data is input next.

INPUT STRUCTURAL DAMPING TYPE:
1. READ IN MASS AND STIFFNESS MATRIX MULT
2. CONSTANT RATIO BASED ON FIRST 2 MODES
3. COMPOSITE

Type in *IDMPT* (I5)(damping type)

If IDMPT = 1:

MASS MATRIX MULTIPLIER = alpha
STIFFNESS MATRIX MULTIPLIER = beta

Type in *alpha, beta* (7E15.4)

If IDMPT = 2:

DAMPING RATIO = ?

Type in *damping ratio* **(F10.0)**

If IDMPT = 3, no additional data is required

The structural model and the soil data have now been completed. Next, the specification for the loading will be input. The type of load input required is specified with IANAL on the control card.

For the shaker analysis (*IANAL* = 0):

SET UP DATA FOR SHAKER ANALYSIS:

ENTER: NFREQ,INLD,IDRLD, PFLD IN ONE LINE:

NFREQ	**= TOTAL NUMBER OF FREQUENCIES**
	>O NFREQ SUPPLIED
	<0 SPECIFY FSTRT AND DFREQ
	(FSTRT = START FREQUENCY,
	DFREQ = FREQUENCY INCREMENT)
INLD	**= NODE TO WHICH LOAD IS APPLIED**
IDRLD	**= LOAD DIRECTION**
	(1=X,2=Y,3=Z,4=RX,5=RY,6=RZ)
PFLD	**= LOAD MAGNITUDE(FORCE: KIPS;**
	MOMENT: KIPS-FT)

Type in the ***requested parameters*** (3I5, F12.4)

There are two ways to specify frequency points:

If NFRQ > 0 **INPUT NFRQ FREQUENCIES (CPS):**

Type in ***NFREQ FREQUENCIES*** (8F10.3)

If NFRQ < 0 **INPUT FSTRT, DFREQ**

Type in ***FSTRT, DFREQ*** (2F12.4)

For the time history analysis (**IANAL = 1**):

FREE FIELD PULSE SPECIFICATION
INPUT PULSE DIRECTIONS: IPLS(I),I=1,3 IN ONE LINE:

Type in the ***directions*** (3I5) at which input pulses are applied. For directions without inputs, enter a 0 (do not leave blank). For each direction which has an input, the name of the file containing the pulse will be requested

FOR PULSE NO. = I ENTER THE FILE NAME:
ENTER SCALE FACTOR :

Type in ***name of the file*** (A20) containing the pulse (generated with Analysis 1) and ***scale factor*** (F10.3). If you do not wish to scale the pulse, input scale factor as one

This procedure will be repeated for directions which have inputs -

The input data for model is complete.

DO YOU WANT TO MAKE ANOTHER MODEL..... (Y/N) ?

Y ----- start the same procedure again from the beginning

N ----- leave this option and return to the CARES main menu page

CRSSTRUC.ANALYSIS 3:

After selecting Analysis 3, the following message appears:

```
**************************************************
*                                                *
*                                                *
*              C A R E S    S Y S T E M          *
*                                                *
*           (MODULE: SEISMIC/STRUCTURES)         *
*                                                *
*                                                *
*  PERFORMS EVALUATION OF DYNAMIC RESPONSES      *
*  OF STRUCTURES SUBJECTED TO SEISMIC MOTION     *
*  USING LUMPED MODEL AND FREQUENCY DEPENDENT    *
*  SOIL INTERACTION COEFFICIENTS                 *
*                                                *
**************************************************
```

The general information, during the SSI calculation is stored on file named **SSI.RUN**

GENERAL OUTPUT IS STORED IN FILE<SSI.RUN>

The program asks for the input file that is constructed by using Analysis #2.

ENTER INPUT-DATA FILE NAME

The user inputs the input file name, ***AFILE*** using a maximum of 20 characters. The program checks if this file name is correct.

INPUT-DATA FILE NAME = *AFILE*

IS THIS NAME CORRECT (Y/N)?

If ***Y*** is selected, the program continues to the next question. If ***N*** is selected, the input file name is asked for again.

Enter the name of the output file name to which final data will be written:

POST-PROCESS FILE NAME ?

The user inputs the input file name, ***AOUTF*** using a maximum of 20 characters. The program checks if this file name is correct.

POST-PROCESS FILE NAME = *AOUTF*

IS THIS NAME CORRECT (Y/N) ?

If ***Y*** is selected, the program continues to the next question. If ***N*** is selected, the post-process file name is asked for again.

For calculations of large structures, calculation using all the input Fourier components is time consuming. An option is provided to allow the user to enter a cut-off frequency. The maximum frequency of the input Fourier components is printed on the screen:

MAXIMUM FREQUENCY FOR ANALYSIS = cps

Enter the desired cut-off frequency, ***FMAX***.

CUT-OFF FREQUENCY FOR ANALYSIS (CPS) ?

The code checks if this value is correct

CUTOFF FREQUENCY = *FMAX* cps

IS THIS CORRECT (Y/N)?

If ***Y*** is selected, the program calculates the structure response. If ***N*** is selected, the cut-off frequency is asked for again. Solution consists of Fourier components at all output nodes

3.3 CRSPOST:

CRSPOST is the module that is used for postprocessing of input or output files from either CRSSOIL or CRSSTRUC. Included in the module are the options to: 1) generate an acceleration time history from a given Fourier component file, 2) calculate the response spectrum from a given time history, 3) calculate the response spectrum for an SSI output data file, and 4) generate PSD related files. When the program begins, the following menu appears:

```
**************************************************
**************************************************
*****          CARESPOST (ANALYSIS)          ******
**************************************************
**************************************************
*   ANALYSIS #                                   *
*                                                *
* 1.  COMPUTES TIME HISTORY FROM AN INPUT        *
*     FOURIER COMPONENT FILE                     *
*                                                *
* 2.  GENERATES RESPONSE SPECTRA FOR GIVEN       *
*     ACCELERATION TIME HISTORY                  *
*                                                *
* 3.  GENERATES RESPONSE SPECTRA FOR GIVEN       *
*     NODE FROM SSI OUTPUT DATA FILE             *
*                                                *
* 4.  PSD-RELATED ACCELERATION TIME HISTORY      *
*     SPECTRAL ANALYSES                          *
*                                                *
* 5.  EXIT FROM CRSPOST                          *
**************************************************
```

INPUT ITEM FOR THE ANALYSIS?

The user inputs the analysis number, ***ITYPE***, of the category of interest. The program checks if this is the correct item number.

ANALYSIS # = *ITYPE*

IS THIS CORRECT (Y/N)?

If ***ITYPE*** is correct, the main program will call the routine of the item selected. If ***ITYPE*** is incorrect, the menu will again appear and ***ITYPE*** can be retyped. After the completion of an analysis, the menu will again reappear and another analysis number can be selected.

CRSPOST.ANALYSIS 1:

Analysis #1 calculates the inverse Fourier transform of a given Fourier component file to generate a corresponding time history, using the Fast Fourier Transform. When Analysis #1 is selected from the CRSPOST menu, the following appears on the screen:

PROGRAM TO COMPUTE TIME HISTORY

The program asks for the name of the Fourier component file.

ENTER FOURIER COMPONENT FILE NAME = ?

Input the Fourier component file, ***AFILE***, using a maximum of 20 characters. The Fourier component file should have the following format:

4 DESCRIPTION CARDS	
MAXIMUM TIME (SECS)	T35, E15.5
DELTA TIME (SECS)	T35, E15.5
# OF RECORDS	T35, I10
# OF FOURIER COMPONENTS	T35, I10
DELTA FREQUENCY (CPS)	T35, E15.5
MAXIMUM FREQUENCY (CPS)	T35, E15.5
MAX. ACCEPTABLE FREQ. (CPS)	T35, E15.5
PEAK ACCELERATION (IPS2)	T35, E15.5

7 LINES THAT ARE SKIPPED

FOR THE NUMBER OF FOURIER COEFFICIENTS, NCOMPS:
A_n COSINE COEFFICIENT & B_n SINE COEFFICIENT (T30,2E12.0)

The user inputs the name of the input time history, ***AFILE***, using a maximum of 20. The program checks if this input is correct.

FOURIER COMPONENT FILE NAME = *AFILE*
IS THIS FILE NAME CORRECT (Y/N)?

If ***Y*** is selected, the program continues to the next question. If ***N*** is selected, the input file name is asked for again. Enter the name of the output file to which the time history is to be written.

ENTER OUTPUT TIME HISTORY FILE NAME

Enter the output time history file name, ***ATHIS***T, using a maximum of 20 characters. The program checks the name of the output file.

TIME HISTORY FILE NAME = *ATHIST*

IS THIS FILE NAME CORRECT (Y/N)?

If ***Y*** is selected, the program executes the inverse Fourier transform. If ***N*** is selected, the time history file name is asked for again.

CRSPOST.ANALYSIS 2:

Analysis #2 calculates the response spectrum of a given input time history record. When Analysis #2 is selected from the CRSPOST menu, the following appears on the screen:

CODE TO TAKE ACCELEROGRAM PULSE FROM AN INPUT FILE
AND
COMPUTE SPECTRA AT DAMPING (%) AND STORE ON FILE SPECT
ACCELEROGRAM PULSE d = IN/SEC2**

The program asks for the acceleration time history record.

INPUT TIME HISTORY FILE NAME ?

Input the file name with the time history, ***AINPT***, using a maximum of 20 characters. The time history should have the following format:

DESCRIPTION CARD 1
DESCRIPTION CARD 2
DESCRIPTION CARD 3
DESCRIPTION CARD 4

NUMBER OF RECORDS, DELTA TIME (SECS) I5,F10.4
TIME HISTORY (IN/SEC2) stored 7 to a card in the format 7F10.0

The user inputs the name of the output response spectrum file name, ***SPCT***, using a maximum of 20 characters.

OUTPUT RESPONSE SPECTRUM FILE NAME?

The program checks if the file names are correct.

INPUT FILE NAME = *AINPT*
OUTPUT FILE NAME = *SPCT*
IS THIS CORRECT (Y/N)?

If ***Y*** is selected, the program continues to the next question. If ***N*** is selected, this input is asked for again. Enter the equipment damping, ***BETA***, in percent at which the response spectrum is to be calculated.

DAMPING(%)?

The program echoes the value for damping for the user to check

DAMPING = *BETA* (%)
IS THIS CORRECT (Y/N)?

If ***Y*** is selected, the program calculates the response spectrum for damping ***BETA***. If ***N*** is selected, damping is asked for again. The spectrum for the input record is then calculated.

CRSPOST.ANALYSIS 3:

Analysis #3 calculates the response spectrum at structural node points from fourier spectral output previously generated. After selecting Analysis #3, the following message appears:

```
**********************************************************
* PROGRAM TO TAKE FOURIER COMPONENTS (ft/sec2)           *
* FROM SIM CODE OUTPUT AND :                             *
*  1 -  DETERMINE THE TIME HISTORY (in/sec2) AT A        *
*       SPECIFIED GAGE LOCATION AND DEGREE OF FREEDOM    *
*  2 -  COMPUTE SPECTRA AT UP TO 5 DAMPING RATIOS        *
**********************************************************
```

GENERAL INFORMATION IS STORED ON FILE : SPECTRUM.RUN.

Enter the input file name, ***AFILE***, from the SSI analysis using a maximum of 20 characters.

INPUT FILE NAME FROM SSI ANALYSIS

The program then checks if this input file is correct

FILE NAME FROM SSI ANALYSIS = ***AFILE***
IS THIS NAME CORRECT (Y/N)

Enter Y if the file name is correct. Enter ***N*** if file is incorrect and needs to be reentered. The program reads ***AFILE***, and prints on the screen

FOLLOWING DATA FROM SSI ANALYSIS OUTPUT
NUMBER OF NODES
NUMBER OF DEG OF FREEDOM
NUMBER OF FREQUENCIES
NO OF DISPL SOL ON FILE
NODE DOF

The nodes and DOF for each of the nodes that are available from ***AFILE*** are printed on the screen. Enter the number of locations, ***NSPECT***, that response spectra are to be calculated.

INPUT NO. OF LOCATIONS AT WHICH YOU WISH TO GENERATE RESPONSE SPECTRA

The program checks if this value is correct

TOTAL NO. OF LOCATIONS FOR SPECTRA = ***NSPECT***

IS THIS NUMBER CORRECT (Y/N)

If this value is correct, type ***Y*** and the program will continue. Typing ***N*** will cause the question to be asked again. For each of the ***NSPECT*** locations, the node number, ***INOD***, of the location and the direction, ***IDF***, are requested.

LOCATION NO. *I*
INPUT NODE NUMBER &DIRECTION

For location ***I***, enter the node number and the degree of freedom number. Data for the input node and the direction are input on the same line. After each input, the data is checked

CHECK LOCATION # *I*:
NODE NUMBER = *INOD*
DEGREE OF FREEDOM = *IDF*
IS THIS CORRECT (Y/N)

If the value is correct, type ***Y***. The response spectrum for each of the locations, ***INOD***, and degrees of freedom, ***IDOF***, is then calculated.

START SOLUTION FOR NODE *INOD* DIRECTION *IDOF*

The response spectra can be calculated for multiple damping ratios. Enter the number of damping ratios that are to be used. The maximum number of damping ratios that can be used is 5.

INPUT NO. OF DAMPING RATIOS TO BE USED (MAX. 5)

For each damping ratio, the user is asked for the damping value, %, and the output file of the response spectrum.

INPUT THE *I* th DAMPING RATIO (%)

Enter the damping ratio, ***SPDAMP(I)***, in percent.

SPECTRA OUTPUT FILE NAME FOR THE DAMPING

Enter the output file name, ***SPECT(I)***, for the response spectrum using a maximum of 20 characters. The damping ratio and the spectra output file are entered for the number of damping cases. The program then calculates the response spectrum for each of the damping ratios for the particular location and degree of freedom. For each damping ratio, ***I***, the name of the output file is printed on the screen and the acceleration time history file name asked for.

RESULTING SPECTRUM FOR DAMPING = *SPDAMP(I)* IS STORED ON THE FILE : *SPECT(I)*

ASSIGN A FILE NAME TO STORING ACCELERATION TIME HISTORY FOR LOCATION *INOD* DIRECTION *IDF*

The program checks if this file name is correct

ACCEL. HISTORY FILE NAME FOR LOCATION *INOD* DIRECTION IDF
= *ACCEL*
IS THIS NAME CORRECT (Y/N)

If the file name is correct, enter ***Y*** and the response spectrum at this location will be calculated. If ***N*** is entered, the question will be asked again.

CRSPOST.ANALYSIS 4:

After selecting Analysis #4, the following message appears:

C A R E S S Y S T E M
(MODULE: SEISMIC)
OPTION 7: PSD-RELATED TIME HISTORY AND RESPONSE SPECTRA ANALYSIS.
PERFORMS THE FOLLOWING THREE TASKS

(1) GENERATES ACCEL TIME HISTORIES ACCORDING TO A USER-SUPPLIED TARGET PSD FUNCTION

(2) PERFORMS OPTIONAL CLIPPING OF THE GENERATED ACCEL TIME HISTORY AND THEN CALCULATES THE PSD FUNCTION OF THE CLIPPED OR NON-CLIPPED ACCEL TIME HISTORY

(3) MULTIPLIES THE CLIPPED OR NON-CLIPPED ACCEL TIME HISTORY BY AN ENVELOPE FUNCTION AND THEN CALCULATE THE ACCEL, VELO, DISPL, AND PSEUDO-VELO RESPONSE SPECTRA

PRESS RETURN KEY WHEN READY TO CONTINUE
Press ***RETURN KEY***

The machine then indicates
PAUSE: To resume execution, type: go
Any other input will terminate the program.
Execution resumed after PAUSE.
The user should respond ***go*** to continue.

The acceleration time history is generated according to a user-supplied, one-sided target power spectral density function which is of "polygonal" shape. The parameters defining the shape of the target spectrum are defined in Ref. 3 and are shown in Figure 3.1 following. The shape is defined by the maximum psd value (in in^2/sec^3) S_1 and linear segments (on a log-log plot as indicated in Figure 3.1). Each of the interface frequency points are input as well as the slopes of the intervening line segments. The specific values of these parameters for the Reg. Guide 1.60 PSD criteria PSD are presented in Appendix E. The program describes this input stream as:

IN ORDER TO DESCRIBE USER-SUPPLIED TARGET PSD ENTER THE FOLLOWING PARAMETERS:
SI , WI , W2, W3, EO, El , E2, E3

SI--MUST BE POSITIVE AND MEASURED IN IN2/SEC**3**
0 < WI < W2 < W3 < W_{UPPER} MUST BE SATISFIED
AND W1, W2, W3 ARE MEASURED IN RAD/SEC

Type in ***S1, W1, W2, W3, EO, El, E2, E3***. Data typed in will then be displayed.
ARE THESE DATA CORRECT..... (Y/N) ?

N.... reentry of data is required ***Y***.... continue

Parameters of the acceleration pulse are next defined

ENTER THE FOLLOWING PARAMETERS TO DEFINE ACCEL:
NPT, NPTPWR, DT
NPT -- NO. OF PTS IN THE ACCEL AND MUST BE A POWER OF 2, UP TO MAX NO. OF 4096
NPTPWR -- POWER TO WHICH 2 IS RAISED TO OBTAIN NPT
DT -- TIME STEP (SEC)

*Type in **NPT , NPTPWR , DT***
The program will then display the parameters defining the acceleration pulse.

TOTAL ACCEL DIGITIZED PTS =
POWER TO WHICH 2 IS RAISED =
TIME STEP =
ARE THESE DATA CORRECT (Y/N) ?

N .... Reentry **of** these parameters is required
Y.....Proceed

ENTER THE FOLLOWING PARAMETERS:

WUPPER, F2UPPER, NENVEL, ICLIP
WUPPER-- UPPER CUT-OFF FREO(RAD/SEC) BEYOND WHICH
TARGET PSD IS CONSIDERED TO BE EQUAL TO ZERO
F2UPPER - CLIPPING VALUE (IN/SEC--2) FOR ACCEL HISTORY
NENVEL - ACCEL POINT AT WHICH RISE OF ENVELOPE FUNC ENDS
CONSTANT PART OF ENVELOPE FUNC BEGINS
MUST BE OBEYED
ICLIP --- IF ICLIP=0 DO NOT PERFORM CLIPPING

IF ICLIP=1 PERFORM CLIPPING

These parameters are then displayed:

WUPPER F2UPPER
NENVEL ICLIP
ARE THESE DATA CORRECT.... (Y/N) ?

N..... Reentry of these data is required
Y..... Proceed

ENTER A POSITIVE NUMBER FOR RANDOM NUMBER GENERATOR:
Type in ***MGEN***

Now user is asked to specify the desired response spectrum:

ENTER THE FOLLOWING PARAMETERS FOR CALCULATION OF RESPONSE SPECTRA:
DAMP, ISPECTR
DAMP -- PERCENTAGE OF DAMPING FOR THE SPECTRA
ISPECTR = 1 , ACCEL RESPONSE SPECTRUM
ISPECTR = 2, VELOCITY RESPONSE SPECTRUM
ISPECTR = 3, DISPLACEMENT RESPONSE SPECTRUM
ISPECTR = 4, PSEUDO-VELOCITY RESPONSE SPECTRUM

Type in ***DAMP, ISPECTR***

DAMP= ISPECTR =
ARE THESE DATA CORRECT..... (Y/N) ?

N..... Reentry of these data is required
Y..... Proceed

SOLUTION PROCESS BEGINS:
DO YOU WANT TO SAVE THE TARGET PSD..... (Y/N) ?

If N .... Skip next question
If Y ....Proceed

NAME OF THE FILE STORING THE TARGET PSD:
Type in ***FILENAME***

WISH TO SAVE ACCEL TIME HISTORY..... (Y/N) ?

If ***N***.... Skip next three questions
if ***Y***

ENTER NAME OF FILE STORING ACCEL HISTORY:
Type in ***FILENAME***

TYPE TITLE 1 FOR THIS RECORD (< 78 CHARACTERS)
Type in ***TITLE1***

TYPE TITLE 2 FOR THIS RECORD (< 78 CHARACTERS)
Type in ***TITLE2***

ENTER FILE NAME FOR SAVING GENERATED PSD:
Type in ***FILENAME***

FILENAME FOR RESPONSE SPECTRUM FILE:
Type in ***FILENAME***

These files will be saved and the program returns to the CARES main menu.

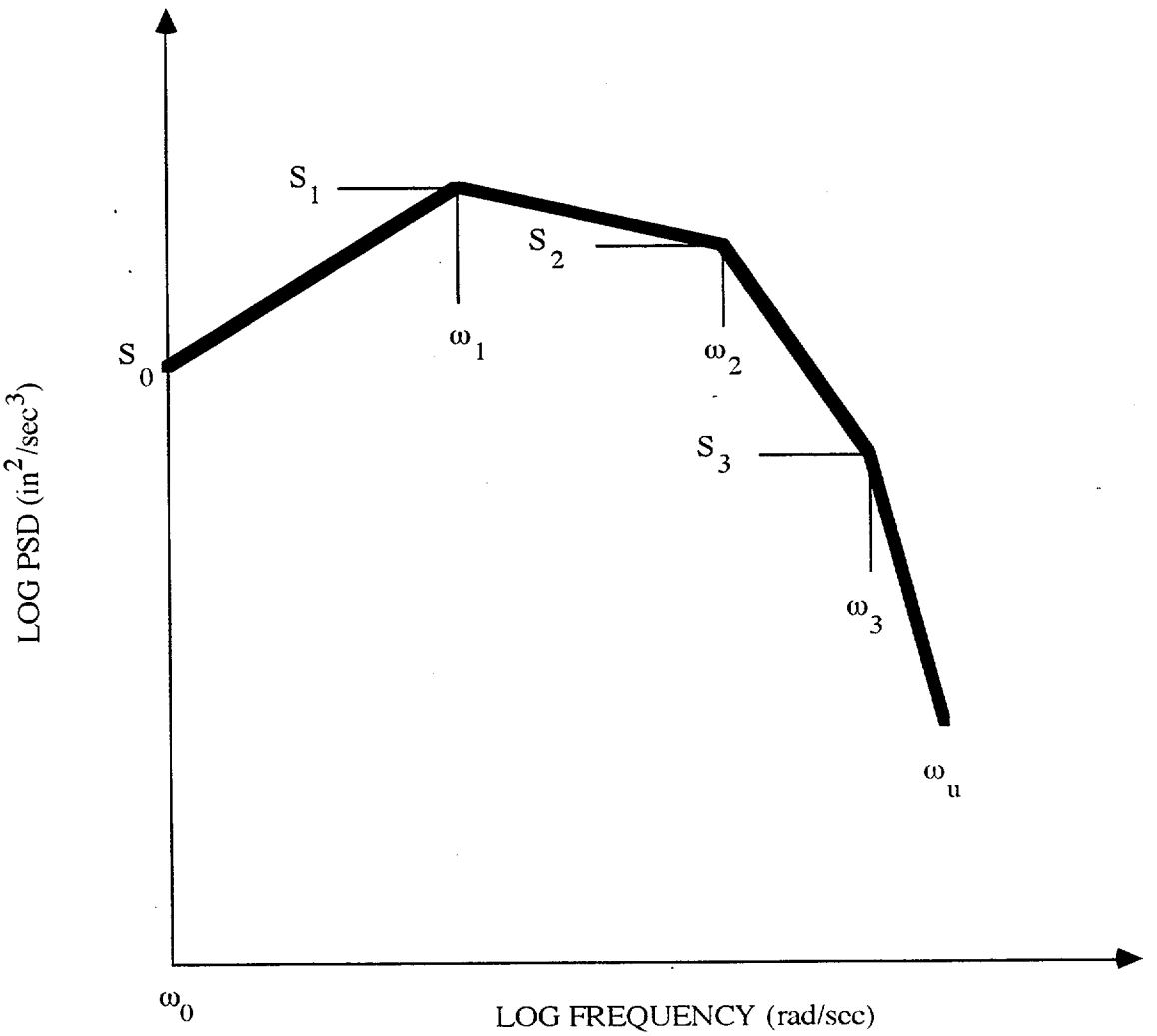

FIGURE 3.1 POLYGONAL TARGET PSD FUNCTION

3.4 CRSPLOT:

CRSPLOT contains the plotting codes used to create plots of output data. When PLOT is executed, the following menu appears.

```
**************************************************
*                                                *
*              CRSPLOT    SYSTEM                 *
*                                                *
*              (MODULE: SEISMIC)                 *
*                                                *
* (GENERAL PLOTTING PROGRAM FOR CARES SYSTEM)    *
*   PLOTTING OPTIONS:                            *
*                                                *
*1.  CONVERTS  INPUT ACCELERATION TIME HISTORY   *
*    DATA FILE TO CARES FORMAT                   *
*                                                *
*2.  PLOT TIME HISTORIES                         *
*                                                *
*3.  PLOT RESPONSE SPECTRA                       *
*                                                *
*4.  PLOT AMPLIFICATION /PSD FUNCTIONS           *
*                                                *
*5.  PLOT GENERAL X-Y GRAPHS                     *
*                                                *
*6.  EXIT FROM CARES                             *
**************************************************
```

and the following question asked:

SELECT AN OPT. YOU WISH TO EXECUTE (1 thru 6)

1. CONVERT ACCELERATION FILE
2. PLOT ACCELERATION TIME HISTORIES.
3. PLOT RESPONSE SPECTRA
4. PLOT AMPLIFICATIONS / PSD FUNCTIONS.
5. PLOT GENERAL X-Y GRAPHS
6. EXIT FROM THE PROGRAM

The user inputs the option number, ***NOPT***, for the particular plot desired. The program checks if this is the correct option number.

RUN OPTION = ***NOPT***
IS THIS OPTION CORRECT (Y/N)

If ***NOPT*** is correct, the plot subroutine will transfer to the routine selected. If ***ITYPE*** is incorrect, the menu will again appear and the ***NOPT*** can be retyped. After the completion of an option, the menu will reappear and another option number can be selected.

CRSPLOT.ANALYSIS 1:

The analysis performed is the same as that described under CRSSOIL.ANALYSIS #1. After selecting ANALYSIS #1, the title of the analysis appears on the screen:

TIME HISTORY FILE CONVERSION:
CONVERTS A GIVEN TIME HISTORY TO CARES FORMAT

The program is used to convert an existing time history file to the format used in CARES and to reduce the total number of time history records. The program asks for the name of the input time history:

INPUT FILE NAME?

Input the name of the existing time history file using a maximum of 20 characters. Next, the number of header cards is requested.

NUMBER OF HEADER CARDS IN INPUT FILE?

Input the number of lines in the input file that precede the time history records. This determines the number of lines that are required to be skipped before reading the first time history record. Parameters which define the time history are then requested.

TIME INCREMENT FOR INPUT TIME HISTORY (secs)?

Input the time increment of the input time history file.

NUMBER OF RECORDS IN INPUT FILE (LE. 20,000)?

Input the total number of records in the input time history file. The program allows for any format to be used for the time history records.

ENTER THE FORMAT OF THE INPUT TIME HISTORY RECORDS

Enter the format that is used for the time history records, eg. 6F10.3. The program checks if this format is correct.

THE FORMAT OF THE TIME HISTORY RECORDS = *FORMAT*
IS THIS CORRECT (Y/N)

If the format is correct, enter ***Y***. If the format is wrong, enter ***N*** and the format will be asked for again.

Next, enter the name for the output file, using less than 20 characters.

OUTPUT FILE NAME (UP TO 20 LTRS)?

Input the name of the new time history file using a maximum of 20 characters.

HEADER CARD FOR OUTPUT FILE (LE. 80 LTRS)?

Input the title card for the new time history file. A maximum of 80 characters can be used.

The program also allows for the option of reducing large time histories by skipping time history records.

ENTER RECORD INCREMENT TO BE USED?
eg. ENTER 1 TO USE EVERY TIME HISTORY RECORD
eg. ENTER 2 TO USE EVERY OTHER TIME HISTORY RECORD

Enter "1" if every time history record is to be used. Enter "2" if every other record is to be used. The final number of time history records, when using CARES, must be less than 8200.

The program permits the user to use a segment of the input time history . The program asks:

IS ONLY A SEGMENT OF THE INPUT TIME HISTORY WANTED (Y/N)

If only a portion of the time history is required, enter ***Y***. If the entire time history is required to be used, enter ***N***. If ***Y*** is entered, the code further asks for the start and finish times to be used from the input time history:

ENTER STARTING TIME (sec

Enter the time in which to begin the time history at.

ENTER STOPPING TIME (sec)

Enter the stopping time. The new time history will begin at the starting time established by the user and end at the stopping time. Therefore, allowing for a window of the input time history to be used. In order to scale the time history, a scale factor is also provided

ENTER SCALE FACTOR?

Enter the scale factor that will be applied to each time history record.

The program will then read the old data file, scale it by the scale factor and prepare the new output file in CARES format.

CRSPLOT.ANALYSIS 2:

For ANALYSIS #2, the screen displays:

```
**********************************************************
*                                                        *
*                  CRSPLOT    SYSTEM                     *
*                                                        *
*                  (MODULE: SEISMIC)                     *
*                  TIME HISTORY PLOT                     *
*                                                        *
*                  PLOT TIME HISTORIES FOR               *
*                  CARES SYSTEM                          *
*                                                        *
**********************************************************
```

ENTER TIME HISTORY TYPE #:

ACCELERATION	**1**
VELOCITY	**2**
DISPLACEMENT	**3**

Three types of time history plots are available to be plotted. The type number designates what names are used for the titles of the plot. Enter the number of the type of plot desired. The input is checked:

For an acceleration time history:

INPUT TIME HISTORY IS FOR ACCELERATION (in/sec2)

For a velocity time history:

INPUT TIME HISTORY IS FOR VELOCITY (in/sec)

For a displacement time history:

INPUT TIME HISTORY IS FOR DISPLACEMENT (in)

IS THIS CORRECT (Y/N)

If the type of time history is correct, enter ***Y*** and the program will continue. If the type is incorrect, enter ***N*** and the type of time history will be asked for again. The plot routine for time histories allows for more than one time history. Therefore, the user is asked for the number of time histories to be plotted:

 # OF TIME HISTORY CURVES TO BE PLOTTED (3 MAX)

Type in the number of curves that need to plotted using a maximum of three (3) curves and a maximum of 20 characters for the name of each curve. For each curve, the file name is asked for:

ENTER THE TIME HISTORY FILE NAME FOR CURVE "I"

and checked

FILE NAME FOR CURVE I = ***AFILE***
IS THIS CORRECT (Y/N) : ?

After each of the file names have been submitted, the program calculates the maximum and minimum values of the time and time history values of the input files. The values are printed on the screen and the user is asked to enter the maximum and minimum values for the x and y axis. For the x axis:

MAXIMUM VALUE OF TIME = ***XMAX***
MINIMUM VALUE OF TIME = ***XMIN***

ENTER DESIRED MAX. TIME (sec)

Enter the maximum value for the time axis.

ENTER DESIRED MIN. TIME (sec)

Enter the minimum value for the time axis. Both the maximum and minimum values of the time axis are checked:

TIME AXIS FOR PLOT WILL HAVE:
A MINIMUM VALUE OF ***XMIN*** **sec**
A MAXIMUM VALUE OF ***XMAX*** **sec**
IS THIS CORRECT (Y/N)

If these values for the time axis are correct, enter ***Y***. If these values are incorrect, enter ***N***. The time axis is segmented by major and minor tick marks. Within each major tick interval, there are minor tick intervals. The user is asked to decide on the time length of these intervals.

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE X AXIS (sec)

Enter the length of the major tick interval for the time axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE X AXIS (sec)

Enter the length of the minor tick interval. An echo of the input is printed on the screen.

MAJOR TICK MARKS ARE AT *XDEL* sec INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL

IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***.

Next, the program calculates the maximum and minimum values of the y axis. The values are printed on the screen and the user is asked to enter the maximum and minimum values for the y axis.

For an acceleration time history:

MAXIMUM VALUE OF ACCELERATION = *AMAX* g
MINIMUM VALUE OF ACCELERATION = *AMIN* g
ENTER DESIRED MAX. ACCELERATION (g)

Enter the maximum acceleration value for the y axis in g's . The code will then ask for the minimum acceleration value for the y axis.

ENTER DESIRED MIN. ACCELERATION (g)

Enter the minimum acceleration value for the y axis in g's . The maximum and minimum values for the y axis will be given for the user to review.

Y AXIS FOR PLOT WILL HAVE:
A MINIMUM ACCELERATION OF *YMIN* g
A MAXIMUM ACCELERATION OF *YMAX* g
IS THIS CORRECT (Y/N)

If the data are correct, enter ***Y***. If any of the data is incorrect, enter ***N*** and the questions related to the y axis will be repeated.

For a velocity time history:

MAXIMUM VALUE OF VELOCITY = *VMAX* in/sec
MINIMUM VALUE OF VELOCITY = *VMIN* in/sec
ENTER DESIRED MAX. VELOCITY (in/sec)

Enter the maximum velocity value for the y axis in in/sec . The code will then ask for the minimum velocity value for the y axis.

ENTER DESIRED MIN. VELOCITY (in/sec)

Enter the minimum velocity value for the y axis in in/sec . The maximum and minimum values for the y axis will be given for the user to review.

Y AXIS FOR PLOT WILL HAVE:
A MINIMUM VELOCITY OF *YMIN* in/sec
A MAXIMUM VELOCITY OF *YMAX* in/sec
IS THIS CORRECT (Y/N)

If the data are correct, enter ***Y***. If any of the data is incorrect, enter ***N*** and the questions related to the y axis will be repeated.

For a displacement time history:

MAXIMUM VALUE OF DISPLACEMENT = *DMAX* in
MINIMUM VALUE OF DISPLACEMENT = *DMIN* in
ENTER DESIRED MAX. DISPLACEMENT (in)

Enter the maximum displacement value for the y axis in inches. The code will then ask for the minimum displacement value for the y axis.

ENTER DESIRED MIN. DISPLACEMENT (in)

Enter the minimum velocity value for the y axis in inches . The maximum and minimum values for the y axis will be given for the user to review.

Y AXIS FOR PLOT WILL HAVE:
A MINIMUM DISPLACEMENT OF *YMIN* in
A MAXIMUM DISPLACEMENT OF *YMAX* in
IS THIS CORRECT (Y/N)

If the data are correct, enter ***Y***. If any of the data is incorrect, enter ***N*** and the questions related to the y axis will be repeated.

The y axis is segmented by major and minor tick marks. Within each major tick interval, there will be minor tick intervals. The user is asked to decide on the length of these intervals. The units of the length will be dependent on the type of time history plot.

For an acceleration time history:

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS (g)

Enter the length of the major tick interval for the acceleration axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS (g)
Enter the length of the minor tick interval.

An echo of the input is printed on the screen.
MAJOR TICK MARKS ARE AT *YDEL* g INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***.

For a velocity time history:
INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS (in/sec)
Enter the length of the major tick interval for the velocity axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS (in/sec)
Enter the length of the minor tick interval. An echo of the input is printed on the screen.

MAJOR TICK MARKS ARE AT *YDEL* in/sec INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***.

For a displacement time history:
INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS (in)
Enter the length of the major tick interval for the displacement axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS (in)
Enter the length of the minor tick interval. An echo of the input is printed on the screen.

MAJOR TICK MARKS ARE AT *YDEL* in INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***.

A title is printed on the plot and specified by the user.

ENTER THE PLOT TITLE (MAX 60 CHARACTERS)

Enter the title for the plot using a maximum of 60 characters. The title name is echoed on the screen:

TITLE FOR PLOT WILL BE:
TIME HISTORY PLOT: ***ATITLE***
IS THIS CORRECT (Y/N)

If the title is correct, enter ***Y*** and the data will be plotted on the screen. If there is an error in the title name, enter ***N*** and the name can be reentered. After the data is plotted on the screen, the user has the option to save the plot to a file which then can printed or reviewed.

SAVE PLOT TO A FILE (Y/N)

If the user does not need to save the plot, enter ***N*** and the CARES PLOT menu will appear on the screen. If the user wants to save the plot, enter ***Y*** and the name of the file to save the plot will be asked for:

FILE NAME TO SAVE PLOT

Enter the name of the plot file by using a maximum of 20 characters.

CRSPLOT.ANALYSIS 3:

For Analysis #3, the program prints:

```
**************************************************
*                                                *
*              CRSPLOT SYSTEM                    *
*                                                *
*              (SEISMIC MODULE)                  *
*                                                *
*              RESPONSE SPECTRA PLOT             *
*                                                *
*     PLOT RESPONSE SPECTRA FOR CARES SYSTEM     *
*                                                *
**************************************************
```

Analysis # 3 has the capability of plotting a maximum of 6 response spectra on a single plot. The program asks for the number of response spectra that are to be plotted.

 # OF RESPONSE SPECTRA TO BE PLOTTED... (MAX 6)

Enter the number of spectra that are to be used. The file name is asked for each of the spectra:

FOR CURVE *#I*
INPUT FILE NAME FOR SPECTRUM PLOT : ?

Enter the response spectrum data file name using a maximum of 22 characters. After the file name is input, it is reprinted on the screen for the user to check:

RESPONSE SPECTRUM FILE NAME TO BE PLOTTED = *AFILE*
IS THIS CORRECT (Y/N)

If the file name is correct, enter ***Y***, otherwise enter ***N***.

The x axis is in log scale with a maximum of 100 cps while the y axis is in arithmetic scale with a maximum value set by the user:

PEAK RESPONSE SPECTRAL ACCELERATION VALUE OF PLOT = *SPMAX* g
INPUT THE MAXIMUM VALUE OF THE Y AXIS THAT YOU WANT (g)

Enter the maximum value of the spectral acceleration to use for the y axis. The y axis is subdivided by major and minor tick marks.

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS (g)

Enter the value for the major tick marks. The major tick marks establish major tick intervals. These intervals are further subdivided into minor tick intervals.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS (g)

Enter the value for the minor tick marks. The information on the tick intervals is repeated on the screen for the user to review:

MAJOR TICK MARKS ARE AT *YTIC* g INTERVALS AND
THERE ARE *ITIC* MINOR INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for the tick marks are incorrect, enter ***N*** and the questions related to the tick intervals will be repeated, otherwise enter ***Y*** and the title used for plot will be asked for:

ENTER THE PLOT TITLE (MAX 60 CHARACTERS)

The title will be printed on the plot and should have a maximum of 60 characters. The title is echoed on the screen for the user to check:

TITLE FOR PLOT WILL BE:
RESPONSE SPECTRA: *ATITLE*
IS THIS CORRECT (Y/N)

Enter ***Y*** if this is correct. If there is an error in the title, enter ***N*** and the question for the plot title will be repeated.

On the plot, a legend is supplied which identifies the various curves. For each of the curves, the program asks for the name to use to identify the curve:

For the 1st curve:

ENTER LABEL FOR 1st DATA SET USING LESS THAN 10 CHARACTERS

and for the subsequent files:

ENTER LABEL FOR DATA SET # "I"
USING LESS THAN 10 CHARACTERS

After all the curves are input, the names of the curves are echoed on the screen for the user to check:

LABEL FOR CURVE "I" : ANAME

.
.
.

LABEL FOR CURVE "N" : ANAME

IS THIS CORRECT (Y/N)

If the names are correct to use for the legend, enter ***Y*** and the data will be plotted on the screen. If there is an error in any of the names, enter ***N*** and the names will be asked for again.

After the data is plotted on the screen, the user has the option to save the plot to a file which then can be printed or reviewed.

SAVE PLOT TO A FILE (Y/N)

If the user does not need to save the plot, enter ***N*** and the CARES PLOT menu will appear on the screen. If the user wants to save the plot, enter ***Y*** and the name of the file to save the plot will be asked for.

FILE NAME TO SAVE PLOT

Enter the name of the file to save the plot by using a maximum of 20 characters.

CRSPLOT.ANALYSIS 4:

For Analysis #4, the program prints:

```
**********************************************************************
*                                                                    *
*                        CARESPLOT SYSTEM                            *
*                                                                    *
*                        (MODULE SEISMIC)                            *
*                  AMPLIFICATION SPECTRA /PSD PLOTS                  *
*                PLOT AMPLIFICATION SPECTRA AND PSD FOR              *
*                  CARES SYSTEM                                      *
**********************************************************************
```

SELECT INPUT OPTION.....(1,2 OR 3) ?

1. AMPLIFICATION PLOT FROM CONVOLUTION ANALYSIS.
2. AMPLIFICATION PLOT FROM SSI ANALYSIS.
3. PSD FUNCTION PLOT

Type ***1, 2 OR 3***

(A) AMPLIFICATION PLOT FROM CONVOLUTION:

For an amplification plot from a convolution analysis, the program queries:

INPUT FILE NAME FOR AMPLIFICATION PLOT: ?
Type in the name of the file containing the amplification data: ***FILENM***

FILE NAME FOR AMPLIFICATION TO BE PLOTTED = *FILENM*
IS THIS CORRECT (Y/N)
If the name is incorrect, enter *N* and the file name will be asked for again.

(B) AMPLIFICATION PLOT FROM SSI ANALYSIS:

For an amplification plot from SSI analysis, the program queries:

INPUT FILE NAME FOR AMPLIFICATION PLOT: ?
Type in name of the file containing the amplification data: ***FILENM***

FILE NAME FOR AMPLIFICATION TO BE PLOTTED = *FILENM*
IS THIS NAME CORRECT (Y/N)

If the name is incorrect, enter ***N*** and the file name will be requested again. Information on the SSI input data is then displayed

FOLLOWING DATA FROM SSI ANALYSIS OUTPUT
NUMBER OF NODES = NOD
NUMBER OF DEG OF FREEDOM = NDOF
NUMBER OF FREQUENCIES = NCOMP
NODE DOF:
INODE INODE DEGREE OF FREEDOM

.

.

NNODE NNODE DEGREE OF FREEDOM

Because of the complexity of the amplification curve, only one curve is plotted on a given graph. The user must specify the selected gage and degree of freedom:

TOTAL NO. OF LOCATIONS FOR PLOT = 1
LOCATION NO. 1
INPUT : NODE NUMBER & DIRECTION:

Enter the node number and the degree of freedom for the amplification curve that is to be plotted. After this information is given, the information is printed on the screen:

CHECK LOCATION # 1
NODE # = INODE
DEGREE OF FREEDOM = DOF
IS THIS CORRECT.....(Y/N) ?

If this is the correct gage location and degree of freedom, enter ***Y***, otherwise enter ***N***.

(C) PSD PLOT:

For a PSD function plot, the program requests:

INPUT FILE NAME FOR PSD PLOT : ?

Enter the name of the PSD function file using a maximum of 22 characters. After the file name is entered, it is displayed on the screen for the user to review.

FILE NAME FOR PSD FUNCTION TO BE PLOTTED = *AFILE*
IS THIS CORRECT (Y/N)

If the file name is correct, input ***Y***. If there is an error in the file name, input ***N*** and the filename will be requested again.

For amplification and PSD plots, the horizontal frequency axis is plotted to a $\log_{10}$ scale and has a maximum value of 100 cps. The program notes:

X-AXIS IS IN LOG SCALE WITH MAX VALUE = 100 cps

For the amplification plots, the program indicates:

Y-AXIS IS IN ARITHMETIC SCALE
MAX FOURIER MAGNITUDE = *YMAX* in/sec2
ENTER THE MAXIMUM VALUE FOR THE Y AXIS (in/sec2)

Enter the maximum value to use for the y axis. The y axis is subdivided by major and minor tick marks.

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS (in/sec2)

Enter the value for the major tick marks. The major tick marks establish major tick intervals. These intervals are further subdivided into minor tick intervals.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS (in/sec2)

Enter the value for the minor tick marks. The information on the tick intervals is repeated on the screen for the user to review:

MAJOR TICK MARKS ARE AT *YTIC* in/sec2 INTERVALS AND
THERE ARE *ITIC* MINOR INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for the tick marks are incorrect, enter ***N*** and the questions related to the tick intervals will be repeated, otherwise enter ***Y***.

The y-axis of the PSD plot (in in^2/sec^3 units) is plotted to a $\log_{10}$ scale. The maximum value for the y axis is supplied by the user. The program indicates:

X-AXIS IS IN LOG SCALE WITH MAX VALUE = 100 CPS
Y-AXIS IS IN LOG SCALE
MAX PSD FUNCTION = *PSDMAX* in2/sec3

ENTER VALUE OF Ymax (10 Ymax)**

Enter the $\log_{10}$ of the maximum value that is to be used for the y axis (eg, for a maximum value of 100, enter 2) . The program then responds:

MAXIMUM Y AXIS VALUE = *PSDMAX* in2/sec3
MINIMUM Y AXIS VALUE = *PSDMIN* in2/sec3
IS THIS CORRECT (Y/N) ?

If these scale limits are correct, respond ***Y*** . Otherwise respond ***N*** and the scale limits will be requested again.

For both the amplification plots and the PSD plot, a title is printed which is supplied by the user:

ENTER THE PLOT TITLE (MAX 60 CHARACTERS)

The title should have a maximum of 60 characters. The title is echoed on the screen for the user to check. For amplification plots, the program indicates:

TITLE FOR PLOT WILL BE:
FOURIER MAGNITUDE : ATITLE
IS THIS CORRECT (Y/N)

For PSD plots, the program indicates:

TITLE FOR PLOT WILL BE:
PSD FUNCTION : ATITLE
IS THIS CORRECT (Y/N)

Enter ***Y*** if this is correct. If there is an error in the title, enter ***N*** and the question for the plot title will be repeated.

On the plot, a legend is supplied which identifies the curve. The program asks for the name to use to identify the curve:

ENTER LABEL FOR 1st DATA SET USING LESS THAN 10 CHARACTERS

After the curve name is input, the name of the curve is echoed on the screen for the user to check:

LABEL FOR CURVE 1 : *ANAME*

If the name is correct to use for the legend, enter ***Y*** and the data will be plotted on the screen. If there is an error in the name, enter ***N*** and the name will be asked for again. After the data is plotted on the screen, the user has the option to save the plot to a file which then can printed or reviewed.

SAVE PLOT TO A FILE (Y/N)

If the user does not need to save the plot, enter ***N*** and the CARES PLOT menu will appear on the screen. If the user wants to save the plot, enter ***Y*** and the name of the file to save the plot will be asked for.

FILE NAME TO SAVE PLOT

Enter the file name to save the plot by using a maximum of 20 characters.

CRSPLOT.ANALYSIS 5:

For Analysis #5, the program prints:

```
**********************************************************
*                                                        *
*                    CRSPLOT SYSTEM                      *
*                                                        *
*                    (SEISMIC MODULE)                    *
*                                                        *
*                    GENERAL X-Y PLOT                    *
*                                                        *
*          PLOT GENERAL X-Y CURVES FOR CARES SYSTEM      *
*                                                        *
**********************************************************
```

THIS IS X-Y PLOT PROGRAM
MAX NO. OF CURVES ALLOWED IS 6
INPUT NO. OF CURVES DESIRED..... ?

Type in the number of curves ***NCURVE***

INPUT OPTION..... (1/2) ?
1. TYPE INPUT INTERACTIVELY
2. READ INPUT FROM A FILE

If *1* : Type input interactively

PREPARE TO READ NCURVE CURVES INTERACTIVELY
FOR CURVE NO. = 1 , ENTER NO. OF PAIRS OF DATA

Type in the ***number of pairs of data***

FOR DATA NO. 1 :
ENTER: X ? Y ?

Type in values for *X* and *Y*

The same procedure will be repeated for all pairs of data. Then, the other curves will be processed.

If *2* : Read input from a file:

INPUT NAME OF DATA FILE

Type in name of the file containing the X-Y data: ***FILENM***

DATA FILE NAME ***FILENM***

Required data file format:	Card no.	Name	Formal
	1	nrcds	(I5)
	2	X1, Y1	(free)

After total number of points for this curve are read in, the same procedure BEGINNING with card 1 will be repeated for another curve until all curves are read in. Information on the plotting data is then displayed

MAX. VALUE OF X-AXIS **= *XMAX***
MIN. VALUE OF X-AXIS **= *XMIN***
WHAT VALUE OF XMAX DO YOU WANT..... ?

Enter the maximum value for the x-axis.

WHAT VALUE OF XMIN DO YOU WANT..... ?

Enter the minimum value for the x-axis.

The x axis is subdivided by major and minor tick marks. Within each major tick interval, there will be minor tick intervals. The user is asked to decide on the length of these intervals.

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE X AXIS

Enter the length of the major tick interval for the x axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE X AXIS

Enter the length of the minor tick interval. An echo of the input is printed on the screen.

MAJOR TICK MARKS ARE AT *XDEL* INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL

IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***. Next, the program calculates the maximum and minimum values of the y axis. The values are printed on the screen and the user is asked to enter the maximum and minimum values for the y axis.

MAX. VALUE OF Y-AXIS **= *YMAX***

MIN. VALUE OF Y-AXIS = *YMIN*

WHAT VALUE OF YMAX DO YOU WANT..... ?

Enter the maximum value for the y-axis.

WHAT VALUE OF YMIN DO YOU WANT..... ?

Enter the minimum value for the y-axis. In a similar fashion as the x-axis, the y-axis is subdivided by major and minor tick marks:

INPUT INCREMENT FOR MAJOR TICK MARKS ON THE Y AXIS

Enter the length of the major tick interval for the y-axis.

INPUT INCREMENT FOR MINOR TICK MARKS ON THE Y AXIS

Enter the length of the minor tick interval.

An echo of the input is printed on the screen.

MAJOR TICK MARKS ARE AT *YDEL* INTERVALS AND
THERE ARE *ITIC* MINOR TICK INTERVALS PER MAJOR TICK INTERVAL
IS THIS CORRECT (Y/N)

If the values for major and minor tick marks are acceptable, enter ***Y***, otherwise enter ***N***.

A title is printed on the plot and needs to be specified by the user.

ENTER THE PLOT TITLE (MAX 60 CHARACTERS)

Enter the title for the plot using a maximum of 60 characters. The title name is echoed on the screen:

TITLE FOR PLOT WILL BE:
GENERAL X-Y PLOT: *ATITLE*
IS THIS CORRECT (Y/N)

If the title is correct, enter ***Y*** and the titles for the x and y axis will be asked for. If there is an error in the title name, enter ***N*** and the name can be reentered.

ENTER LABEL FOR X-AXIS :

Enter the name to use for the x axis using a maximum of 20 characters. The label for the y axis is next asked for:

ENTER LABEL FOR Y-AXIS:

Enter the name for the label of the y axis using a maximum of 20 characters. On the plot, a legend is supplied which identifies the various curves. For each of the curves, the program asks for the name to use to identify the curve:

For the 1st curve:

ENTER LABEL FOR 1st DATA SET USING LESS THAN 10 CHARACTERS

and for the subsequent files:

ENTER LABEL FOR DATA SET # "I"
USING LESS THAN 10 CHARACTERS

After all the curves are input, the names of the curves are echoed on the screen for the user to check:

LABEL FOR CURVE "I" : ANAME

.

.

LABEL FOR CURVE "N" : ANAME

IS THIS CORRECT (Y/N)

If the names are correct to use for the legend, enter ***Y*** and the data will be plotted on the screen. If there is an error in any of the names, enter ***N*** and the names will be asked for again.

After the data is plotted on the screen, the user has the option to save the plot to a file which then can printed or reviewed.

SAVE PLOT TO A FILE (Y/N)

If the user does not need to save the plot, enter ***N*** and the CARES PLOT menu will appear on the screen. If the user wants to save the plot, enter ***Y*** and the name of the file to save the plot will be asked for.

FILE NAME TO SAVE PLOT

Enter the name of the file to save the plot to using a maximum of 20 characters.

4.0 SAMPLE PROBLEMS

The sample problems developed to demonstrate the use of the CARES Version 1.2 and to validate the code are discussed in this Section of the report. The sample problems are of two types. The first is to demonstrate the use of the code for a typical problem. This is done by following a problem from the specification of the input free field motion through the generation of floor response spectra in the facility. This sample problem is discussed in Section 4.1 and contains detailed descriptions of the code operation. The second type of sample problem focuses on code validation. This validation is done assuming that the CARES Version 1.0 is correct so that the sample problems are designed to test the modifications that have been made to the code in the development of Versions 1.1 and 1.2. Modifications have been made to the structures portion of the code (e.g., seismic response of structure) and to the soils portion of the code (e.g., convolution). The sample problems for the structures and soils validations are discussed in Sections 4.2 and 4.3 respectively.

4.1 Sample Problem Demonstrating Use of CARES Version 1.2

A sample problem is executed from start to finish to illustrate the use of CARES Version 1.2. The application used for this demonstration is the quarter scale model containment structure located in Hualien, Taiwan. NRC is participating in an experimental program in Hualien. A quarter scale model containment structure has been constructed in a seismically active region in Taiwan. Predictions are being made of the expected seismic response of the structure and these predictions will be tested against the measured response.

Data describing the sample problem is shown in Fig. 4.1. The soil beneath the foundation has a shear wave velocity of about 1050 fps down to a depth of 131' and larger shear wave velocities below 131'. The upper 16.4' has a shear wave velocity of about 525 fps. The depth to bedrock is approximately 600'. The input free field motion is specified at a rock outcrop.

The screen dumps from running this problem with CARES are shown in Appendix A. The bold face type is added comments while all other printing is copied from the screen during execution of the problem.The soils module is first run and the following options are executed:

Analysis 1 - (pages A.1 - A.2) converts time history to CARES format
Analysis 3 - (pages A.2 - A.4) computes Fourier components of the input record
Analysis 4 - (pages A.4 - A.14) performs the convolution analysis
Analysis 5 - (pages A.15 - A.16) computes the surface time history from the Fourier components of the surface motion
Analysis 6 - (pages A.16 - A.18) computes a response spectra for the free field surface

motion.

The structures module is then executed and the following options executed:

Analysis 2 - (pages A.19 - A.26) generates the structural input file
Analysis 3 - (pages A.27 - A.30) performs the SSI analysis

The post processing module is next executed and the following options used:

Analysis 4 - (pages A.31 - A.36) generates floor response spectra

The plotting module is then executed with the following options used:

Analysis 2 - (pages A.37 - A.45) plots accelerograms
Analysis 3 - (pages A.45 - A.51) plots response spectra

An example of the generation of a accelerogram to fit a prescribed response spectra is shown on pages A.52 through A.68. Next, the Hualien sample problem is run with three simultaneous motions. The convolution of the two additional motions, the orthogonal horizontal motion as well as the vertical motion, is similar to the procedure that has already been mentioned. After the amplification of the rock motions, the surface motions are simultaneously applied to the quarter scale model containment structure. The screen dump for the input to Analysis 2 is included on pages A.69-A.76 to illustrate any differences with the previous problem.

4.2 Structures Module Validation Problems

Three basic changes were made to CRSSTRUC (the SSI structural response calculation) of CARES Version 1.0. First, the storage scheme used for the stiffness and dynamic matrices were modified to take advantage of symmetry and the sparsity of the matrices. This was done so that larger problems could be solved with CARES. Sample problem S.1 was done to check that the new storage scheme gives the same results as obtained with Version 1.0. Second, the option of including rigid links between nodes was added. This is validated with sample problem S.2. Third, the ability of different damping ratios for different elements in the structure was incorporated into Version 1.1 by the use of composite damping. This was validated with sample problem S.3. A fourth validation problem (S.4) was solved to validate the code against analytic solutions. All of the structure sample problems were solved with the final Version 1.2 after all of the modifications had been made. The problems and results are discussed below and some of the detailed output from the problem solutions S.1 through S.4 is contained in Appendix C. All of the **bold** typing in Appendix C is comments while the remaining material is copies of computer output.

Two additional changes were made during the development of Version 1.2. First, a shear wall element has been added to the catalog of structural elements in CARES. This is validated in sample problem S.5. Second, SSI coefficients for rectangular foundations have been added. This addition involved the programming of equations given in Reference 17. As such there are no validation problems.

Sample Problem S.1

This problem was run to demonstrate that Version 1.2 obtains the same results as Version 1.0. The problem was set up so that only the modifications associated with the storage allocation portions of the SSI response option were employed (i.e., there were no rigid links and composite damping was not used). A configuration of the problem is shown on Fig. 4.2. A single beam is attached to the free field with a Type 1 SSI model and the top of the beam is subjected to a shaker load (IANAL = 0). This problem was run using both Versions of the CARES code. The input files, hardcopy output of the SSI computation, and the resulting Fourier components of the structural deformations are shown in Appendix C. The files containing the deformation Fourier components are identical for both Versions of the code. Since all of the modifications involving storage reallocations made in Version 1.2 were invoked during this solution, it is concluded that Version 1.2 gives the same results as Version 1.0.

Sample Problem S.2

This problem was run to demonstrate that the rigid link capability installed in Version 1.2 operates properly. A three beam model was used for this purpose as shown in Fig. 4.3. The beam from node 2 to node 4 was intended to be rigid and modeled in Prob S.2A with a rigid link and in Prob S.2B with a very stiff beam (4 times stiffer than the other beams). A shaker loading in the horizontal direction was applied at node 2 and a Type 1 SSI model was attached to node 1. The computed responses to both problems should be the same.

The input files used for each of the models is shown in Appendix C. The output from the Option 3 run for the rigid link case is also included in the Appendix. The output for the stiff beam case is very similar and is therefore not included. The actual deformations results are included in the files **"s2av1.2.ou"** and **"s2bv1.2.ou"** which are both included in the Appendix. These files contain the Fourier components of the deformations at nodes 1, 2, 4, and 5. An inspection of these files indicate that the results are identical. For example the two 5 cps components of the vertical (3) direction deformation are (-0.716E-07 and -0.635E-06) for the rigid link case and (-0.707E-07 and -0.639E-06) for the stiff beam case. The responses are even closer for the horizontal deformation components. It is therefore concluded that the rigid link addition is correct.

Sample Problem S.3

Sample problem S.3 is run to validate the composite damping addition to Analysis 3 of the structures module. A uniform beam is used having the following properties:

Length	= 100 ft
Moment of Inertia	= 20000 ft^4
Young's Modulus	= 10000 k/ft^2
Mass Per Unit Length	= 10 k sec^2/ft^2

Shear areas and section properties are selected so that shear deformations are negligible (required so that analytic solutions can be found to compare with the CARES results).

Damping in the first 50 feet of length of the beam is specified to be 7 % and damping is specified to be 2 % in the remainder of the beam.The composite damping model in CARES has the objective of computing a composite damping value in each of the first two modes of the structure representing this variable damping. The composite damping value is determined by computing a weighted average of damping in the various parts of the structure. The weighting function is the strain energy stored in the beam for deformation shape equal to the mode shape.

Since composite damping applies only to the structure, the analytic solution is first developed and then compared with a solution obtained from CARES. The first two frequencies of the cantilever beam are (Ref. 13):

$$f_1 = [1.875]^2 [E I / m L^4]^{1/2} / 2\pi = 0.79 \text{ cps}$$

$$f_2 = [4.712]^2 [E I / m L^4]^{1/2} / 2\pi = 5.00 \text{ cps}$$

The mode shapes (X) are given as:

$$X_n = B_n (\sinh a_n x - \sin a_n x) + \cosh a_n x - \cos a_n x$$

where

n = mode number

$a_1 L = 1.875$

$a_2 L = 4.712$

$B_n = -(\cos a_n L + \cosh a_n L) / (\sin a_n L + \sinh a_n L)$

$B_1 = -0.734$

$B_2 = -1.018$

The strain energy (S_e) stored in a beam element is given by:

$$S_e = \int_0^L \frac{E I (X_n'')^2}{2} dx$$

where the primes denote differentiation with respect to the coordinate measuring length along the beam. The value of composite damping (p_{cn}) is then found from:

$$p_{cn} = \frac{1}{S_e}\left[p_a \int_{l_a} \frac{E I (X_n'')^2}{2} dx + p_b \int_{l_b} \frac{E I (X_n'')^2}{2} dx \right]$$

where the integrals in the numerator are evaluated over the beam length for which the specified damping is applicable. The resulting composite damping values for the sample problem can then be computed to be:

$$p_{c1} = 0.0675$$
$$p_{c2} = 0.0482$$

The same problem is now run with CARES and the beam is divided into 10 beam segments. The input to the problem and resulting output contained in file ("SSI.RUN") are shown in Appendix C. The resulting frequencies are found in CARES to be:

$$\omega_1 = 4.950 \; ; \; f_1 = 4.950 / 2\pi = 0.79 \text{ cps}$$
$$\omega_2 = 30.674 \; ; \; f_2 = 30.674 / 2\pi = 4.88 \text{ cps}$$

These frequencies are in excellent agreement with the analytic solutions of (0.79 and 5.00 cps). Closer agreement could be obtained for the second mode by dividing the beam into finer elements. The composite damping values computed in CARES are:

$$p_{c1} = 0.0674$$
$$p_{c2} = 0.0474$$

These are also in excellent with the analytic solutions of (0.0675 and .0482).

Sample Problem S.4

Analytic solutions (Ref. 14) are available for single degree of freedom systems subjected to sinusoidal forcing functions. The ratio of the dynamic peak displacement of the single degree of freedom system to the static displacement (when the system is subjected to a static load equal to the peak value of the sinusoidal forcing function) is:

$$X_{dyn} / X_{stat} = 1 / [(1 - \lambda^2)^2 + (2 \zeta \lambda)^2]^{1/2}$$

where

$\lambda = \omega / \omega_n$

ω = frequency of forcing function

ω_n = frequency of single degree of freedom system

λ = damping ratio of single degree of freedom system

This solution is used to validate the structural module in CARES. A two mass model subjected to vertical excitations is used as shown in Fig. 4.4. Two solutions are obtained: the first with the beam properties selected so that the beam is stiff and the flexibility is contained in the vertical SSI model. Young's modulus for the beam is specified as 18,000,000 ksf the area is 10 square feet, and the length is 100 feet. This results in a beam frequency of 213 cps and therefore the beam is rigid. The soil shear modulus (G) is specified as 1000 ksf, and the foundation radius (R) is specified as 2.65 feet. The vertical SSI spring (K_V) and damper (C_V) are:

$$K_V = 4 G R / (1-\mu) = 15{,}820 \text{ k/ft}$$

$$C_V = 3.4 [G \rho]^{1/2} R^2 / (1-\mu) = 62.8 \text{ k sec / ft}$$

The vertical SSI frequency (f_V) and damping ratio (ζ) are then:

$$f_V = [K_V / M]^{1/2} / 2\pi = 2 \text{ cps}$$

$$\zeta_V = C_V / 2 [K_V M]^{1/2} = 0.025$$

The input file for this problem is shown in Appendix C together with the **"SSI.RUN"** file and the Fourier components of the displacement (see file **"s4av1.2.ou"**). The peak amplification of the displacement at node 2 can be computed as the square root of the sum of the squares for the

two Fourier components listed in the file. This is done and the results compared with the analytic solution (determined as described above) on Table 4.1. As may be seen there is excellent agreement between the two results.

The second solution is obtained with the model parameters set so that the SSI spring is effectively rigid and the beam is flexible. The radius of the foundation is set to 297.6 feet so that the SSI frequency is about 80 cps and is therefore rigid. The beam Young's modulus is specified as 79,000 ksf so that its stiffness (K_b) is:

$$K_b = A\,E / L = 7{,}900 \text{ k/ft}$$

The beam frequency is:

$$f_b = [K_b / M]^{1/2} / 2\,\pi = 2 \text{ cps}$$

The structural damping is specified as type 1 with the mass multiplier ($\alpha = 1.26$) and the stiffness multiplier ($\beta = 0$). The ratio of critical damping (ζ_v) is therefore:

$$\zeta_v = \alpha M / 2\,[K_b\,M]^{1/2} = 0.05$$

The input file for this problem is shown in Appendix C together with the **"SSI.RUN"** file and the Fourier components of the displacement (see file **"s4bv1.2.ou"**). The peak amplification of the displacement at node 2 can be computed as the square root of the sum of the squares for the two Fourier components listed in the file. This is done and the results compared with the analytic solution (determined as described above) on Table 4.2. As may be seen there is excellent agreement between the two results.

Table 4.1

AMPLIFICATION WITH RIGID STRUCTURE (INPUT FROM FILE s4av1.2.ou)
SYSTEM FREQUENCY = 2.0 CPS SYSTEM DAMPING = 0.025
AMPL AT NODE 2 IN DIRECTION 3

FREQUENCY	CARES AMPL	THEO AMPL
.00	0.1000E+01	0.1000E+01
.25	0.1016E+01	0.1016E+01
.50	0.1067E+01	0.1067E+01
.75	0.1163E+01	0.1163E+01
1.00	0.1331E+01	0.1333E+01
1.25	0.1632E+01	0.1639E+01
1.50	0.2271E+01	0.2277E+01
1.75	0.4151E+01	0.4194E+01
2.00	0.1997E+02	0.2000E+02
2.25	0.3738E+01	0.3683E+01
2.50	0.1783E+01	0.1767E+01
2.75	0.1127E+01	0.1119E+01
3.00	0.8021E+00	0.7986E+00
3.25	0.6115E+00	0.6088E+00
3.50	0.4862E+00	0.4844E+00
3.75	0.3990E+00	0.3972E+00
4.00	0.3340E+00	0.3331E+00
4.25	0.2849E+00	0.2843E+00
4.50	0.2469E+00	0.2461E+00
4.75	0.2168E+00	0.2154E+00

Table 4.2

AMPLIFICATION WITH RIGID SSI (INPUT FROM FILE s4bv1.2.ou)
SYSTEM FREQUENCY = 2.0 CPS SYSTEM DAMPING = 0.05
AMPL AT NODE 2 IN DIRECTION 3

FREQUENCY	CARES AMPL	THEO AMPL
.00	0.1000E+01	0.1000E+01
.25	0.1016E+01	0.1016E+01
.50	0.1063E+01	0.1066E+01
.75	0.1159E+01	0.1163E+01
1.00	0.1326E+01	0.1330E+01
1.25	0.1631E+01	0.1632E+01
1.50	0.2246E+01	0.2253E+01
1.75	0.3972E+01	0.3997E+01
2.00	0.9922E+01	0.1000E+02
2.25	0.3468E+01	0.3467E+01
2.50	0.1735E+01	0.1735E+01
2.75	0.1108E+01	0.1110E+01
3.00	0.7931E+00	0.7943E+00
3.25	0.6061E+00	0.6066E+00
3.50	0.4828E+00	0.4831E+00
3.75	0.3956E+00	0.3964E+00
4.00	0.3322E+00	0.3326E+00
4.25	0.2832E+00	0.2839E+00
4.50	0.2453E+00	0.2458E+00
4.75	0.2152E+00	0.2152E+00

Sample Problem S.5

This problem is used to validate the shear wall model in CARES with the details of the problem shown on Figure 4.5. A 150' high by 50' wide wall is modeled with 3 shear wall elements. The wall is 1' thick, has a shear modulus of 200,000 ksf, and damping equal to 2 %. A harmonic load is applied to the top of the wall and a mass (m) of 26.77 k sec^2 /ft is concentrated at the top of the wall. The SSI parameters are set so that the wall behaves as if it is on a rigid foundation.

The stiffness (k) of the wall is:

$$k = \text{Area} * \text{Shear Modulus} / \text{Height} = 50 * 1 * 200000 / 150 = 66{,}667 \text{ k/ft}$$

The frequency of the wall (f) is:

$$f = [k/m]^2 / 2\pi = [\, 66667 / 26.77]^{1/2} / 2\pi = 7.94 \text{ cps}$$

The solution for the displacement at the top of the wall is given in the discussion of sample problem S.4 above. The damping ratio for the structure is 2 %. This analytic solution is shown on Figure 4.6 together with the CARES solution. As may be seen the agreement is excellent.

4.3 Soils Module Validation Problems

Three basic changes from version Version 1.0 were made to the free-field analysis section of CARES.SOIL. First, the method used to calculate the Fourier components of a time history was changed to include a FFT method. The FFT method is also used in determining the time history from given Fourier components. Sample problem S.6 validates this change by comparing the results with the Fourier transform method used in CARES Version 1.0. The other basic changes were made to the convolution subroutine included in CARES.SOIL. The first modification to the convolution subroutine is the option to define a rock outcrop the motion. The second modification was made in the details of the convolution analysis. Two sample problems are included to validate this modification. Sample problem S.7 compares the amplification of a unit pulse input at a rock outcrop and convolved upward through a single soil layer. These results are compared with the available analytic solution. Sample problem S.8 indicates the results of convolving a given motion upward through a specific soil column to the surface. A comparison of the CARES Version is made with the comparable results from SHAKE.

Sample Problem S.6

This problem was run to validate the new FFT method used in CARES. The time history used for this problem is the same as used in Section 4.1. The time history of this motion is shown in Figure 4.7. Fourier components of the motion are determined using both CARES Version 1.0 and CARES Version 1.2. The magnitude and phase angle of the Fourier components from both methods are shown in Figures 4.8 and 4.9.

Sample Problem S.7

Sample problem S.7 compares the results of the convolution module of CARES with the analytic solution. A unit pulse is specified at the rock outcrop and convolved through a single soil layer to the surface. The properties of the soil column are:

Soil Layer:

Thickness	=	100'
Shear Velocity	=	750 ft/sec
Unit Weight	=	125 #/ft3
Damping	=	5%

Rock Halfspace:

Shear Velocity	=	4500 ft/sec
Unit Weight	=	140 #/ft3
Damping	=	2%

Amplification of the unit pulse using both methods is shown in Figure 4.10 The results from CARES are identical with the analytic solution.

Sample Problem S.8

Sample problem S.8 compares the results of CARES with those from SHAKE. The sample problem provided in the SHAKE manual, but using a constant damping for the rock half-space, is run using CARES. The description of the soil column is given in Figure 4.11. The input motion is placed at the rock outcrop and convolved upward through eight soil layers. The Seed-Idriss 1970 soil degradation model is used in the analysis. CARES allows for the strain to be calculated in the time domain or the frequency domain. Both methods were used for this comparison problem. The response spectra at the surface and the rock outcrop are presented in Figure 4.11. Final effective strains for the soil layers are given in figure 4.13. The response spectra and layer strains from CARES and SHAKE are effectively the same.

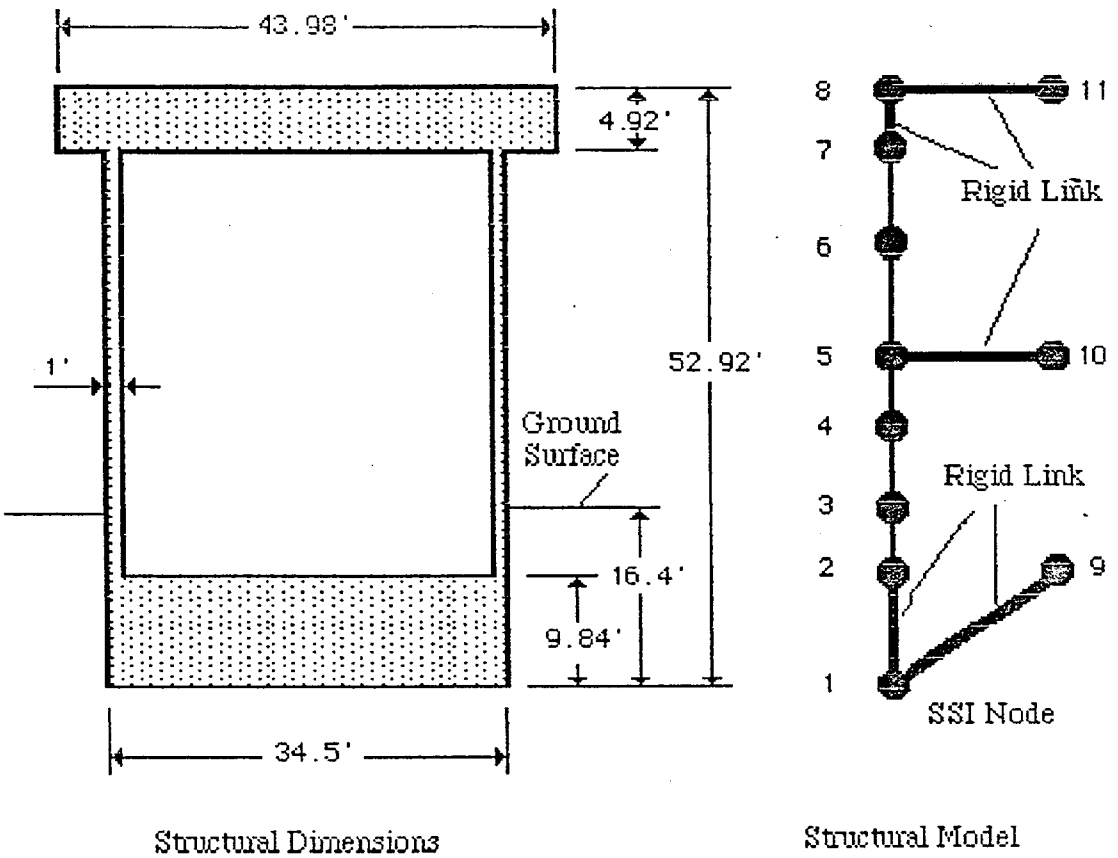

Figure 4.1 Sample Problem of Hualien Test Structure

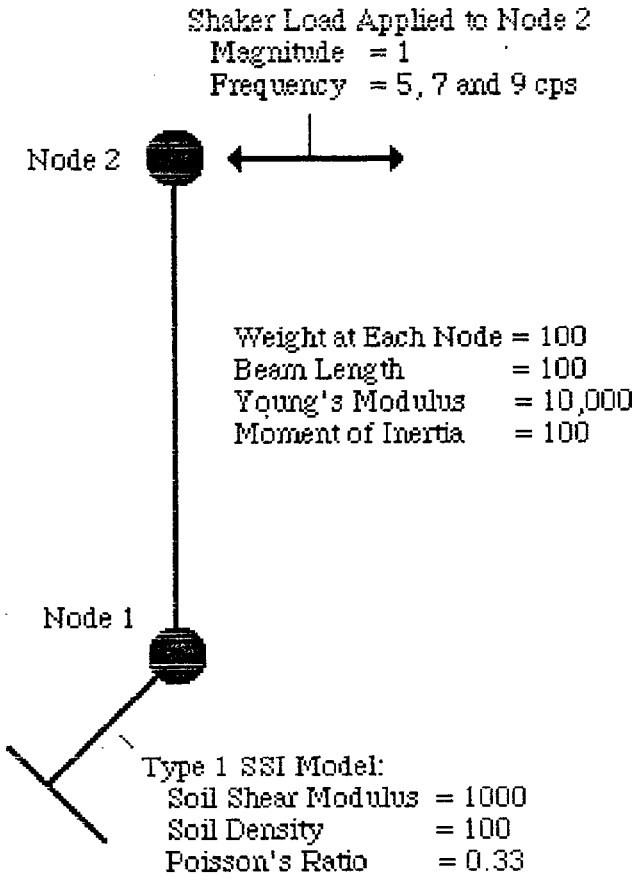

Figure 4.2 Sample Problem S.1

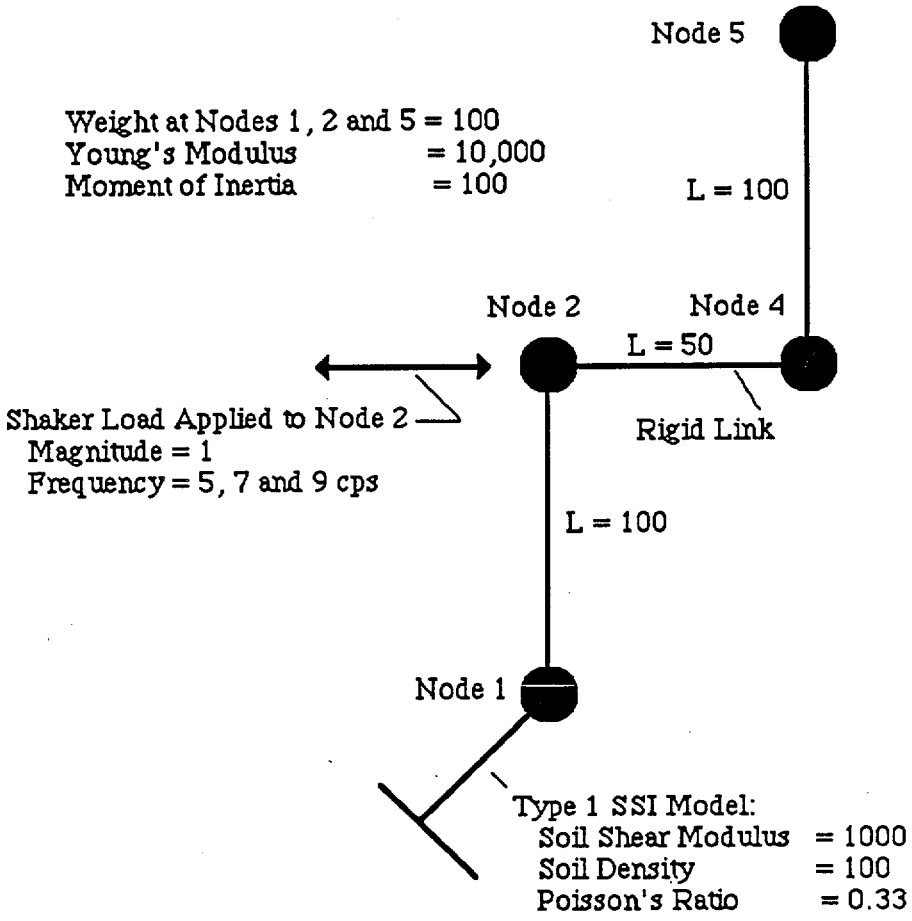

Figure 4.3 Sample Problem S.2

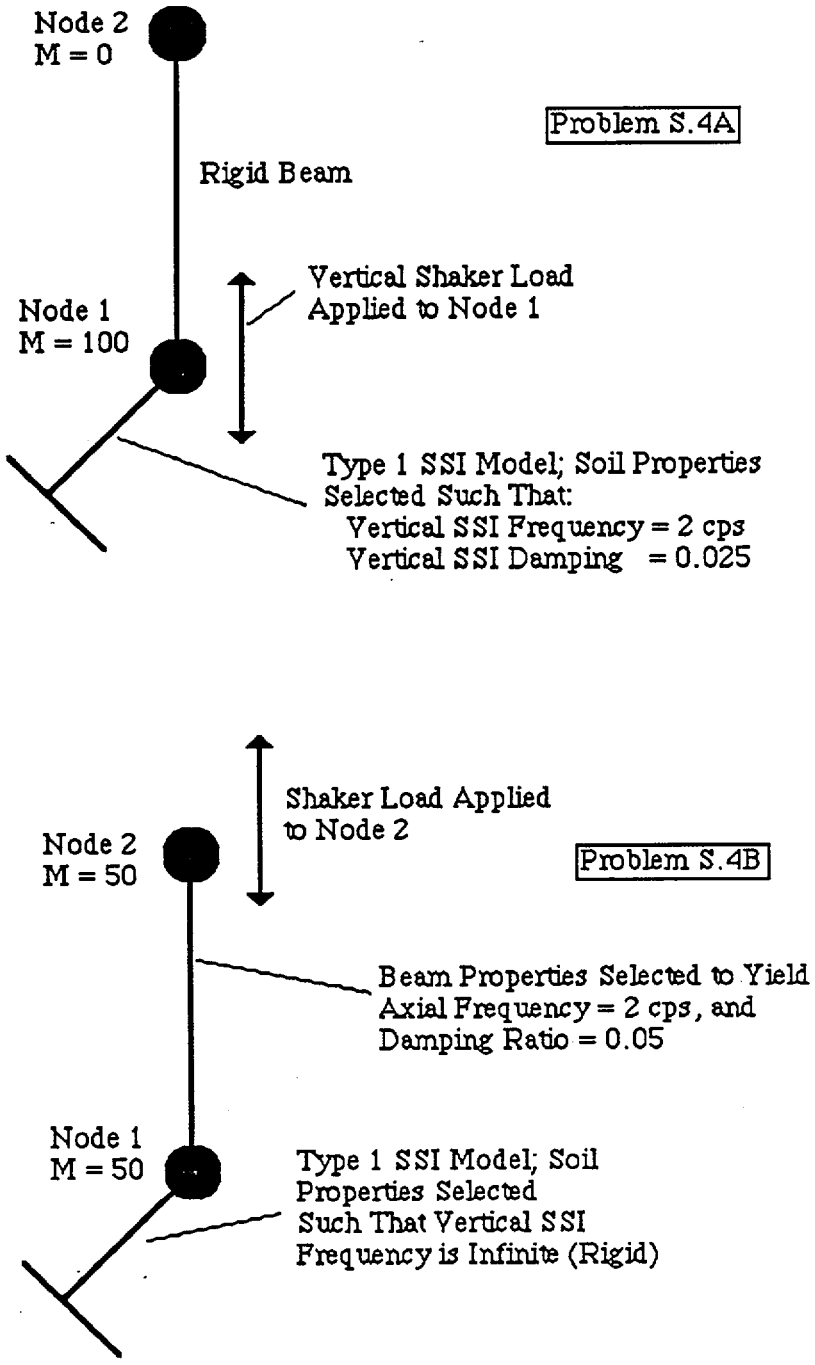

Figure 4.4 Sample Problem S.4

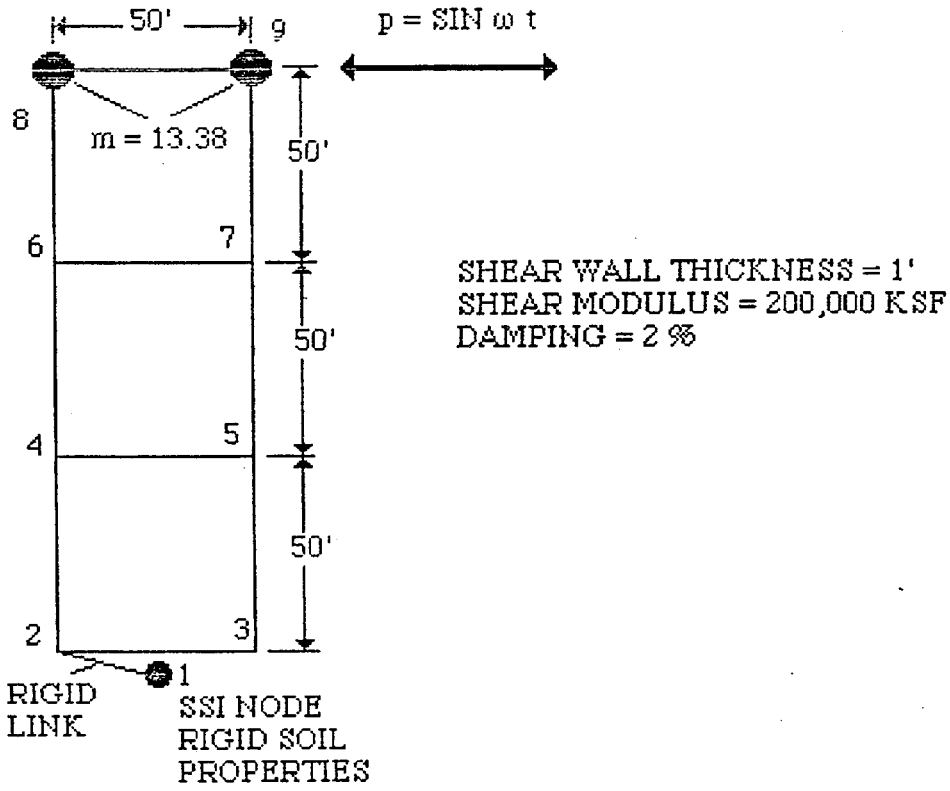

Figure 4.5 Sample Shear Wall Problem

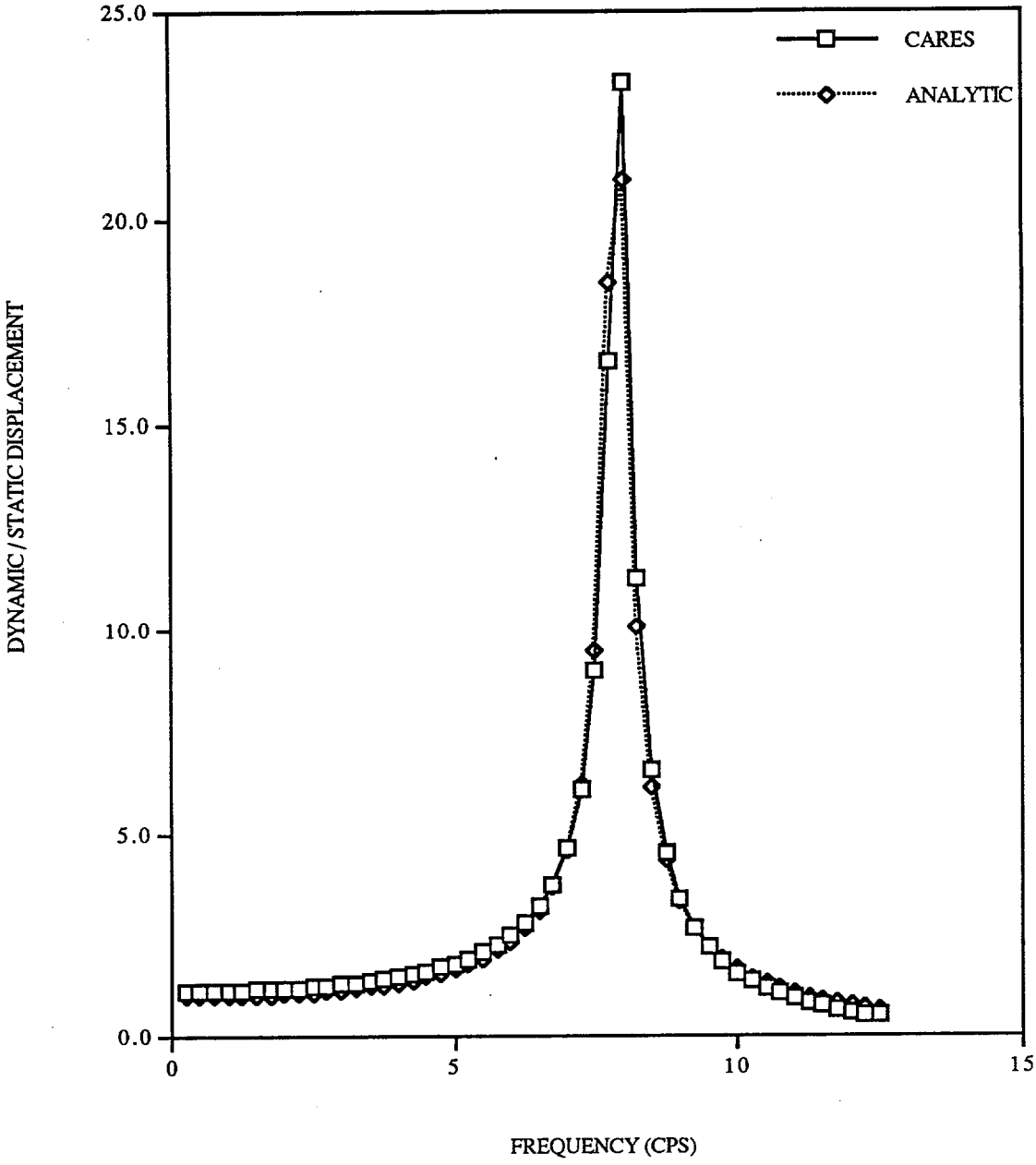

Figure 4.6 Results From Sample Problem S.5

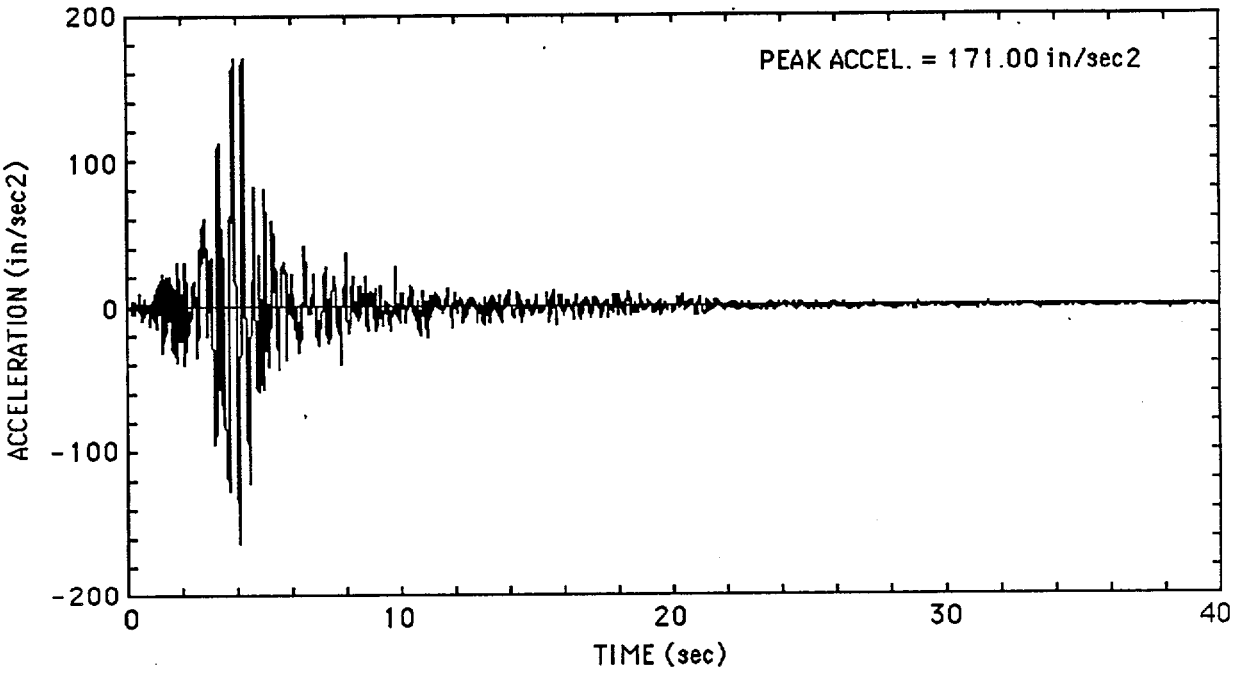

Figure 4.7 Input Time History for Sample Problem S.6

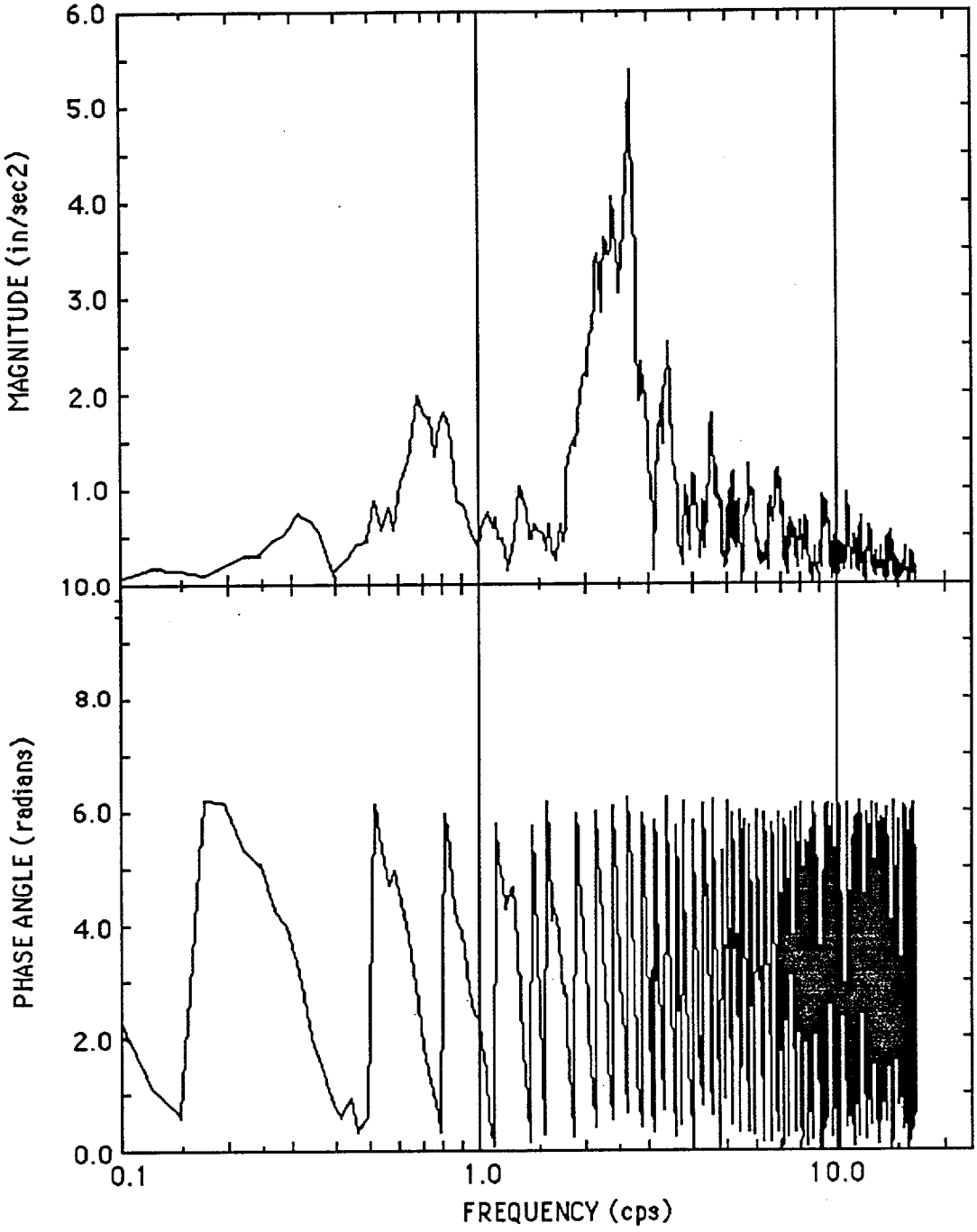

Figure 4.8 Fourier Components of Outcrop Time History
Using CARES Version 1.0

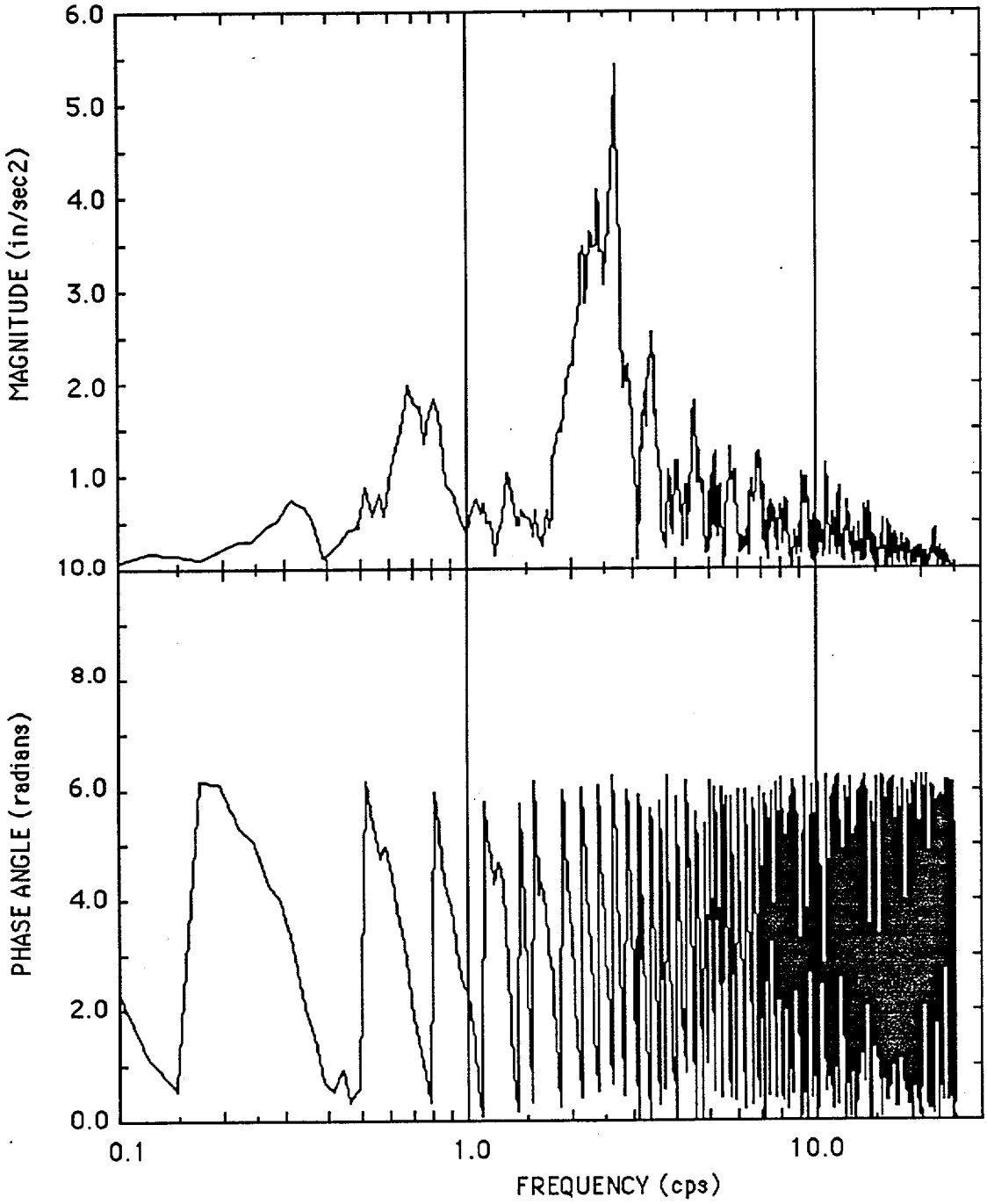

Figure 4.9 Fourier Components of Outcrop Time History
Using CARES Version 1.2

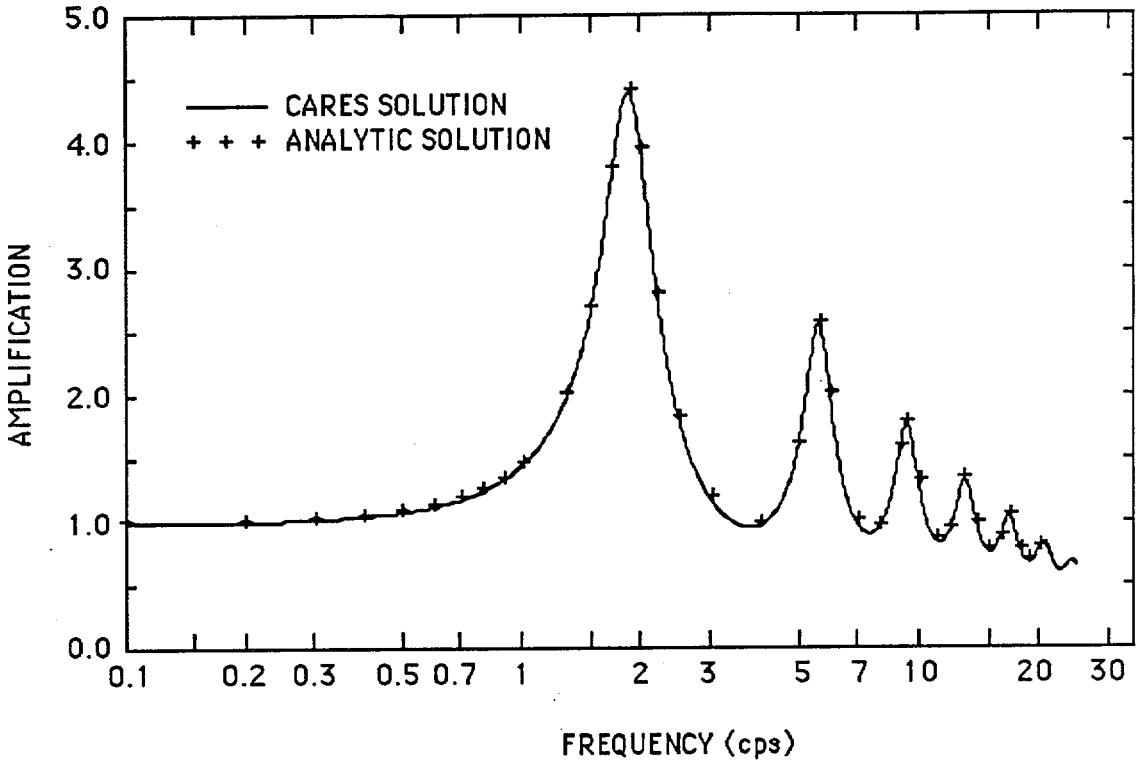

Figure 4.10 Amplification of Outcrop Motion for Single Layer

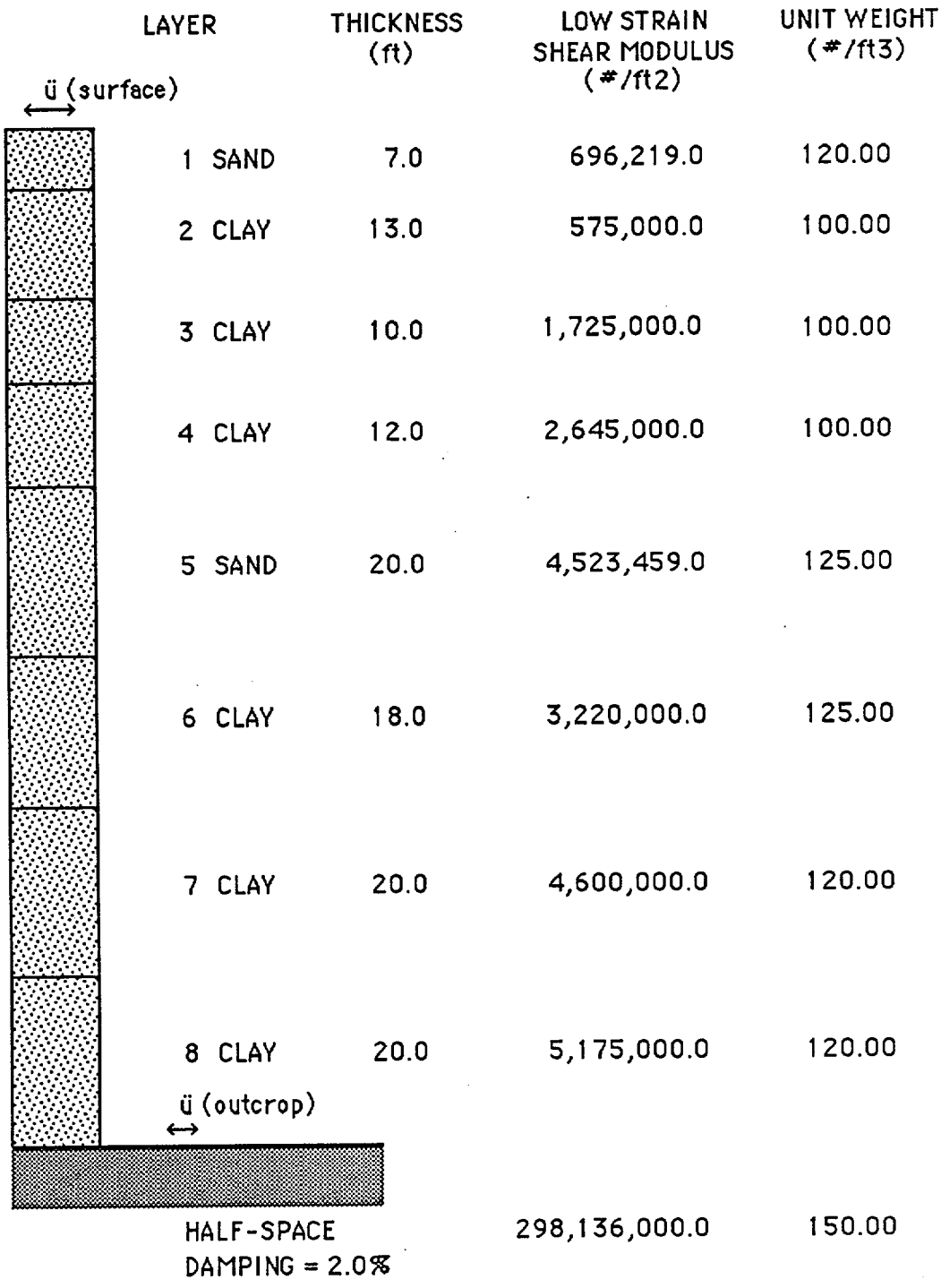

Figure 4.11 Soil Column Description for Sample Problem S.8

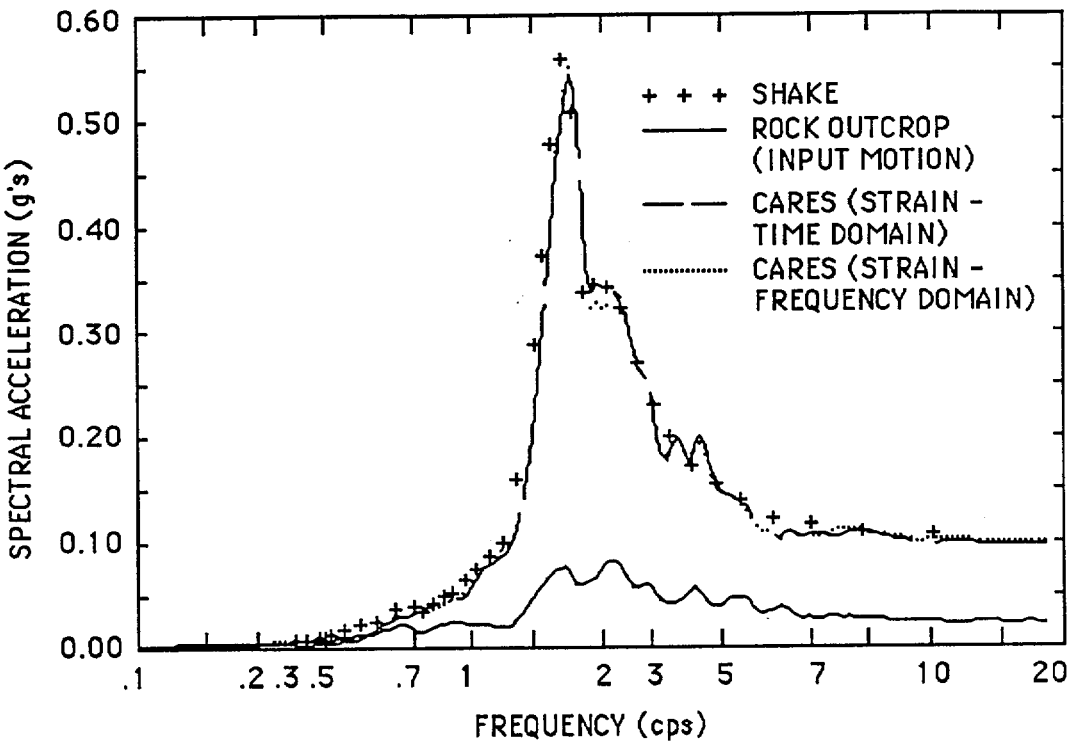

Figure 4.12 Comparison of Surface Spectra from SHAKE and CARES

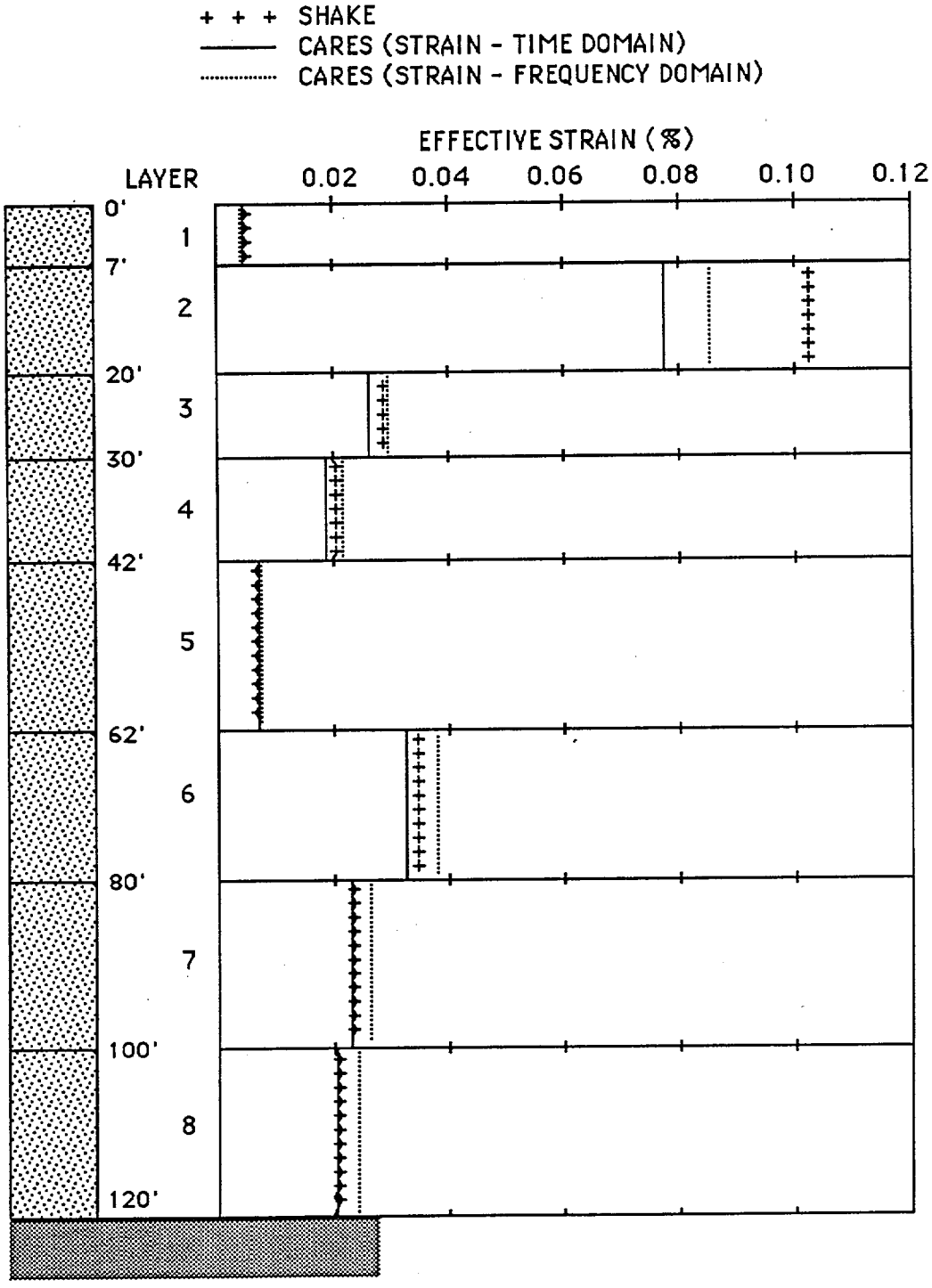

Figure 4.13 Comparison of Effective Strains from SHAKE and CARES

5.0 AP 600 MODEL

A CARES model of the AP 600 nuclear island is discussed in this section of the report. This model is based on a SASSI model developed for the island by Westinghouse. The SASSI model dated July 7, 1995 is the basis for the CARES model. The CARES model is developed from a listing of the SASSI "HOUSE" input data and sketches of the island provided by Westinghouse. The "HOUSE" portion of SASSI develops the data describing the structure and its connection to the soil. No attempt has been made to verify the SASSI model from actual design details of the AP 600.

5.1 Description of SASSI Model

The SASSI "HOUSE" model contains descriptions of six components of the nuclear island:

1. The soil displaced by the buried portion of the island is described with solid elements.

2. The structural foundation is divided into two parts. The top surface of the 6' thick basemat is at elevation 66.5'. The basemat is modeled with solid elements. The mass of the basemat is computed within SASSI based on the element geometry. The walls of the foundation rise from the basemat to elevation 100' and are modeled with plate elements. The mass of the walls is not computed within the code but is input together with the masses of the other structural elements. The entire foundation (including the displaced soil elements) is modeled with node numbers ranging from 1 to 789.

3. The Coupled Shield and Auxiliary Building (CSAB) rises from the top of the basemat to elevation 307.25' and is modeled as a stick. The CSAB is connected to the foundation at elevations 66.5', 82.5' and 100'. The diaphragms of the CSAB are considered to be rigid (all nodes on a given diaphragm are connected with very stiff 3-D beam elements). The masses are lumped at the diaphragm elevations at nodes called "Mass Centers." The diaphragms are connected with two different types of elements. First, 3-D beam elements are used to model the flexural stiffness (resistance to relative horizontal displacements and rotations). Nodes called "Shear Centers" on adjacent diaphragms are used to locate the 3-D beams. Second, spring elements are used to model the vertical stiffness (resistance to relative vertical displacements) between diaphragms. These springs only have stiffness resisting displacements in the vertical direction. Nodes called "Centroids" are used to locate the spring elements. The CSAB is modeled with nodes numbers in the range 3001 to 3078.

4. The Steel Containment Vessel (SCV) is modeled with a single stick and rises from elevation 100' to elevation 256.3'. The SCV is connected to the foundation at elevation 100'. Each node in the stick has an associated mass and is connected to adjacent nodes with 3-D beam elements. The polar crane is modeled with a mass connected to the stick with a spring providing stiffness for the three translational degrees of freedom. The SCV is modeled with node numbers in the range 3101 to 3199.

5. The Containment Internal Structure (CIS) is modeled in the same manner as the CSAB and rises from elevation 82.5' to elevation 158'. It is connected to the foundation at elevations 82.5' and 100'. The CIS is modeled with node numbers in the range 3200 to 3299.

6. The Reactor Coolant Loop (RCL) is modeled with 3-D beam elements. The RCL is modeled with node numbers in the range 4000 to 4701.

5.2 Development of CARES Model from SASSI Model

The AP 600 CARES model is described in this section. The major required modification of the SASSI model is in the representation of soil - structure interaction effects. The CARES code is based on the representation of soil - structure interaction effects with simple spring-dashpot models connecting a "SSI" node to the free field. This SSI node (node number 19) is placed as shown on Figure 5.1 with respect to the horizontal plan of the actual basemat. This location is at the horizontal plan centroid of the mat. The node 19 is located at elevation 60.5' (the base of the basemat). The horizontal X-Y coordinate system used to describe the model is also shown on this figure. The basemat masses (both translational and rotational) are lumped at node 19. The SSI coefficients are computed within the CARES code based on a specified embedded rectangular foundation. The dimensions of the foundation are selected so as to maintain the plan area of the foundation. The 254' width of the foundation in the "X" direction is maintained, so that the width in the "Y" direction becomes 127'. This equivalent rectangular plan of the foundation is also shown on Figure 5.1. The depth of embedment is maintained at 39.5'. Of course various soil properties may be specified as desired.

The foundation (basemat and walls) of the island are assumed to be rigid and are not included in the CARES model. This is an essential assumption of the CARES program since it relies on lumped SSI parameters which are only available for rigid foundations.

The CSAB, SCV, and CIS are modeled in CARES exactly as they are modeled in SASSI with one exception. The SASSI model uses very stiff beams to represent rigid connections (e.g.,

between nodes on common diaphragms). These nodes are connected with rigid links in the CARES model. The models are shown on Figures 5.2,5.3, and 5.4 respectively. The node numbers are shown and are equal to the node numbers in SASSI minus 3000. The nodes are located on the figures at the proper elevations but their locations in the horizontal plane are not related to their actual X-Y coordinates. The figures become too cluttered when this is done. The node numbers which are mass centers are shown as solid circle (•), the node numbers which are shear centers are shown as crosses (x), and the nodes which are centroids are shown as open circles (o).

The beams between diaphragms are shown as lines with element numbers given with **bold** numbers placed alongside the beam. The axial spring connection between diaphragms are shown with wavy lines with the spring number shown with a ***bold italic*** number. The rigid connections between nodes on a common diaphragm and between the SSI node (19) and the nodes on the stick models are shown as lines with double thickness lines.

The RCL is not included in the CARES model. The mass of the RCL is only about 1.7 % of the total mass of the modeled portions of the island so that it does not affect the overall response on the nuclear island. Its inclusion would greatly increase the size of the problem in that it both adds many degrees of freedom to the model and significantly increases the bandwidth of the stiffness matrix. Both of these factors result in a problem with significantly longer running times and thereby would make it more time consuming to run parametric studies. The inclusion of the RCL would add about 100 nodes to the model.

5.3 Sample Results

The AP 600 is run with CARES for the case with a uniform soil having a shear wave velocity of 3400 fps. The system response is computed for input motion in the X, Y, and Z (vertical) directions using the deconvolved free field motion (at the basemat) as input. The Fourier components defining the input motion are H140.FC, H240.FC, and V40.FC in the three directions respectively. The deconvolved motions in each case were determined from the surface motions used in SASSI. Response spectra (5 %) in the X direction are computed at nodes 16, 110, and 204, in the Y direction at nodes 16 and 110, and in the Z direction at nodes 110 and 204. Nodes 16 and 110 are located at the top of the CSAB, and 220' elevation of the SCV, and Node 204 is located at the 135' elevation of the CIS. A listing of the SSI.RUN file is given in Appendix D defining all parameters of the model for the run with the input in the X direction. The only changes in the SSI.RUN files for the other two runs (in the Y and Z directions) are at the very end on the file where the direction and file name for the input motion is given.

The H140.FC input motion is used in the X direction with the response spectra in the X direction shown on Figures 5.5 through 5.7 for the three nodes. The H240.FC input motion is used in the Y direction with the response spectra in the Y direction shown on Figures 5.8 through 5.9 for the two nodes. Spectra obtained from the SASSI code are shown on the same curves for the purposes of comparison. As may be seen the SASSI spectra for node 16, in both directions, are about 20 % higher than the CARES spectra for frequencies larger than 5 cps. A possible cause of this difference is the modeling of the "rigid" floor diaphragms in the CSAB. SASSI models these with stiff beam elements while the CARES model uses rigid links. The flexibility in the SASSI beams could account for this difference. The SASSI spectra are slightly higher (about 15 %) than the CARES spectra at node 110 (the SCV) at the primary frequency of 10 cps. Since this occurs primarily at the primary frequency rather than over some broad frequency range it is likely that this is the result of differences in the damping models used in the two codes. The results indicate that the CARES effective damping is more than the SASSI effective damping. The SASSI and CARES results are quite similar for node 204 (CIS) although the CARES spectra are somewhat higher at the primary frequency of the building (about 15 cps).

The V40.FC is used in the vertical direction with the response spectra in the Z direction at node 204 shown on Figure 5.10. The SASSI spectra is shown on the same figure and can be seen to be in good agreement with the CARES spectra.

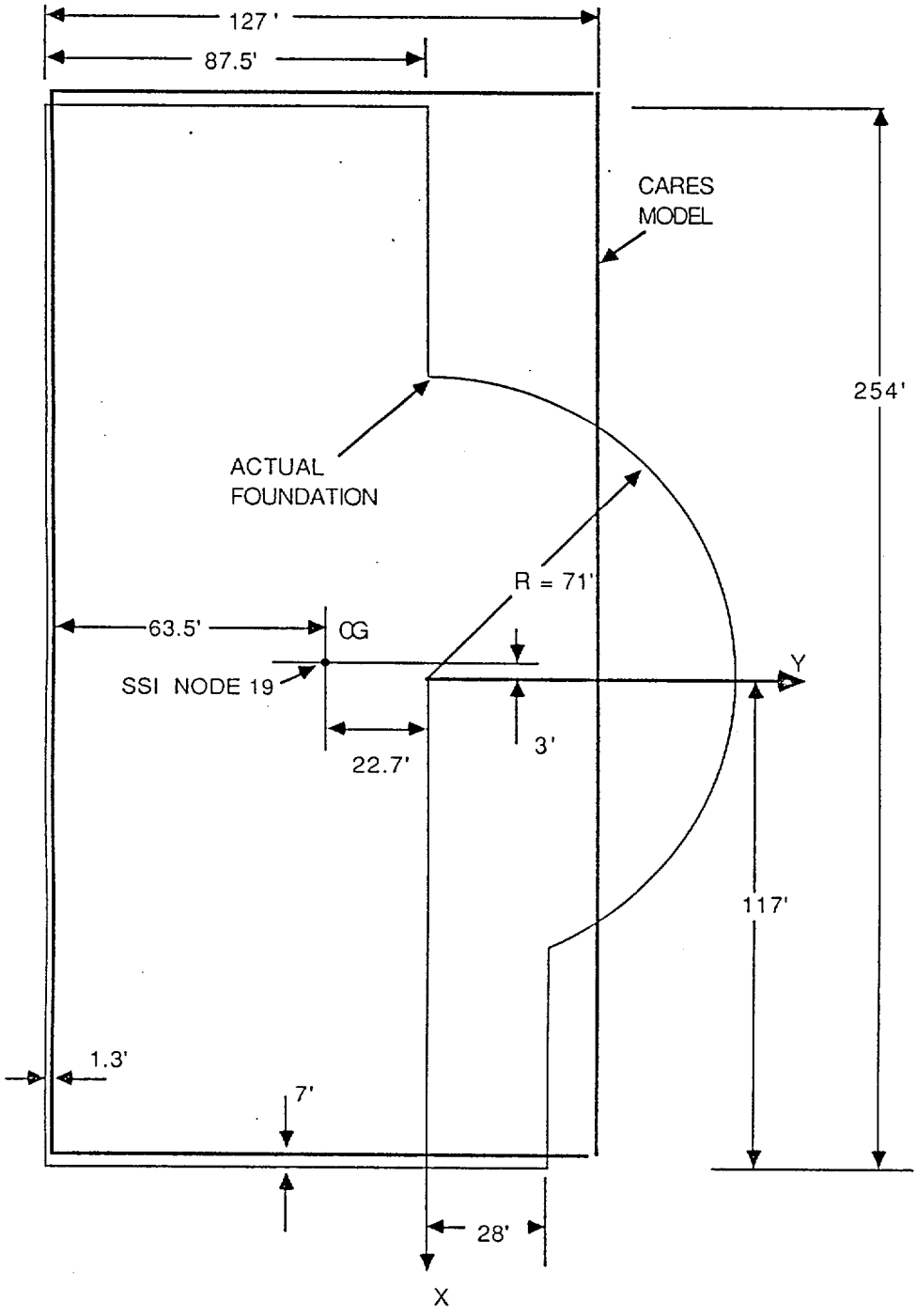

Figure 5.1 AP 600 Foundation Plan

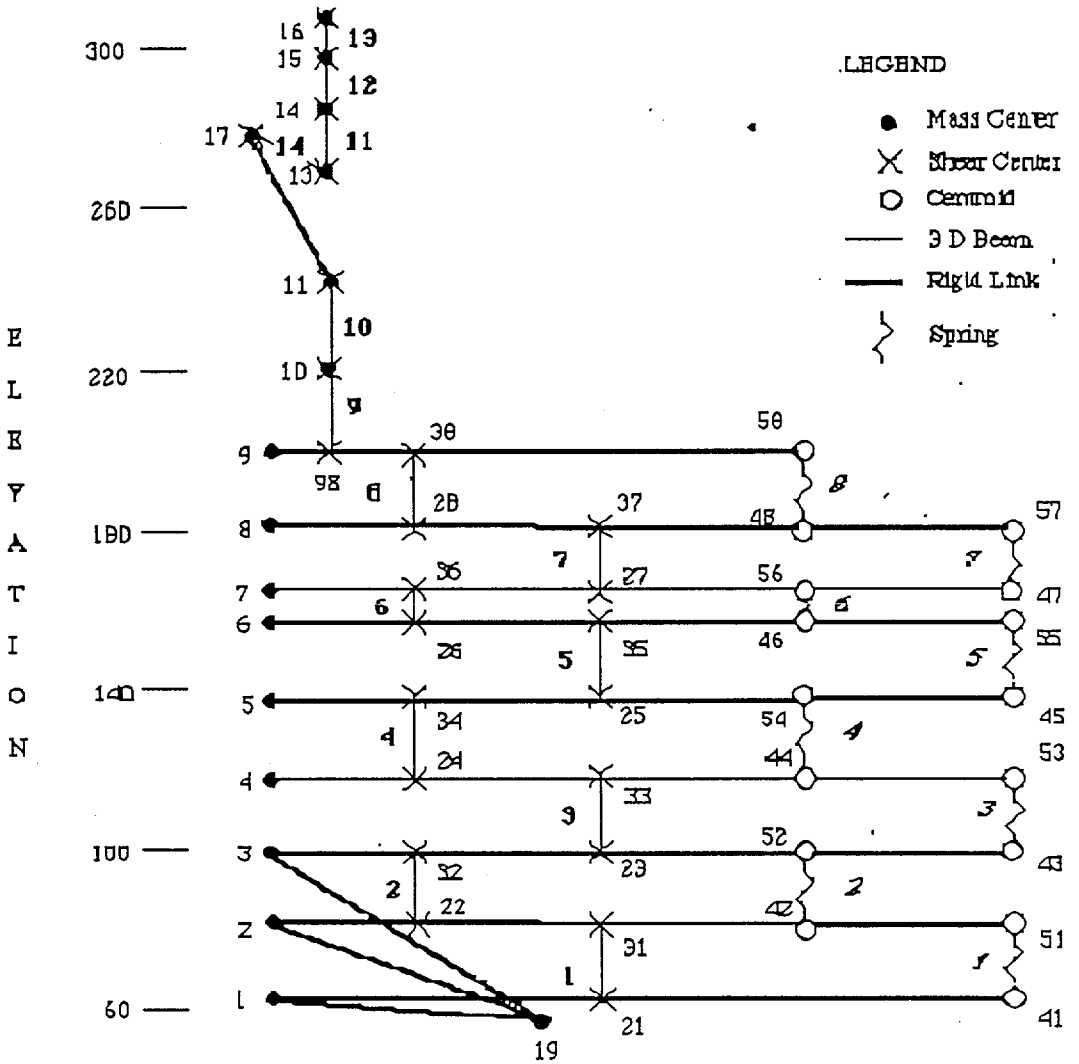

Figure 5.2 Stick Model of Coupled Shield and Auxiliary Building

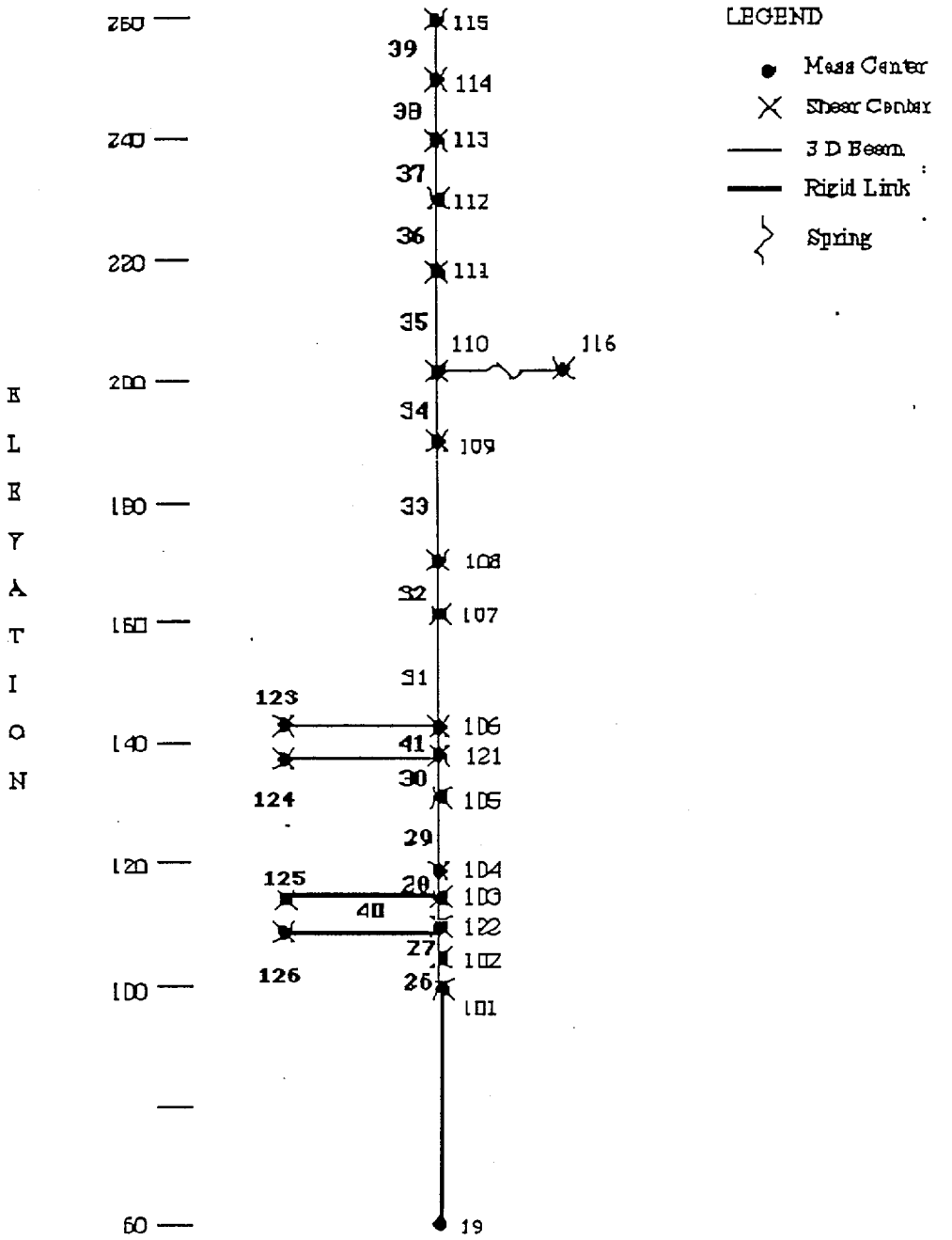

Figure 5.3 Stick Model of Steel Containment Vessel

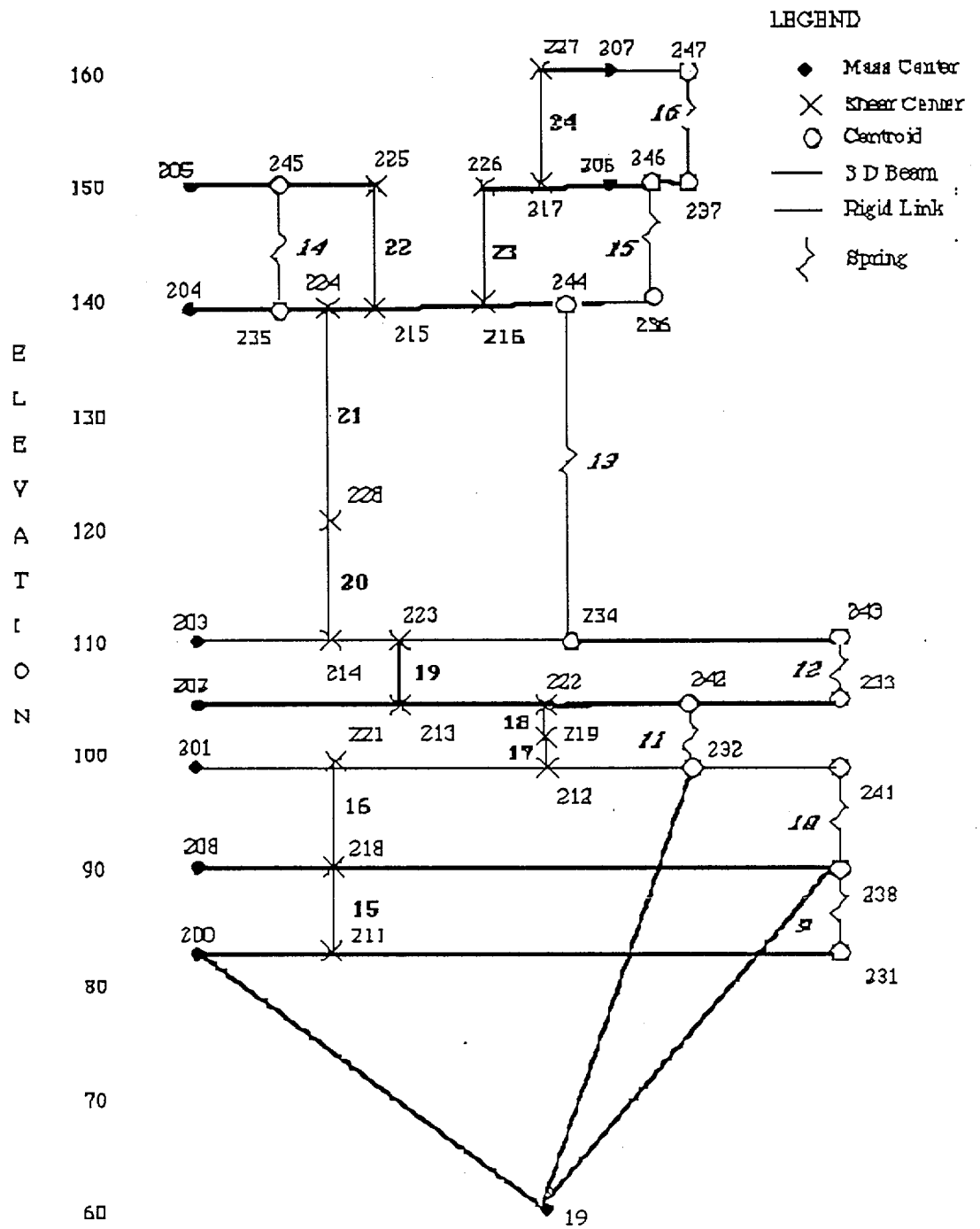

Figure 5.4 Stick Model of Containment Internal Structure

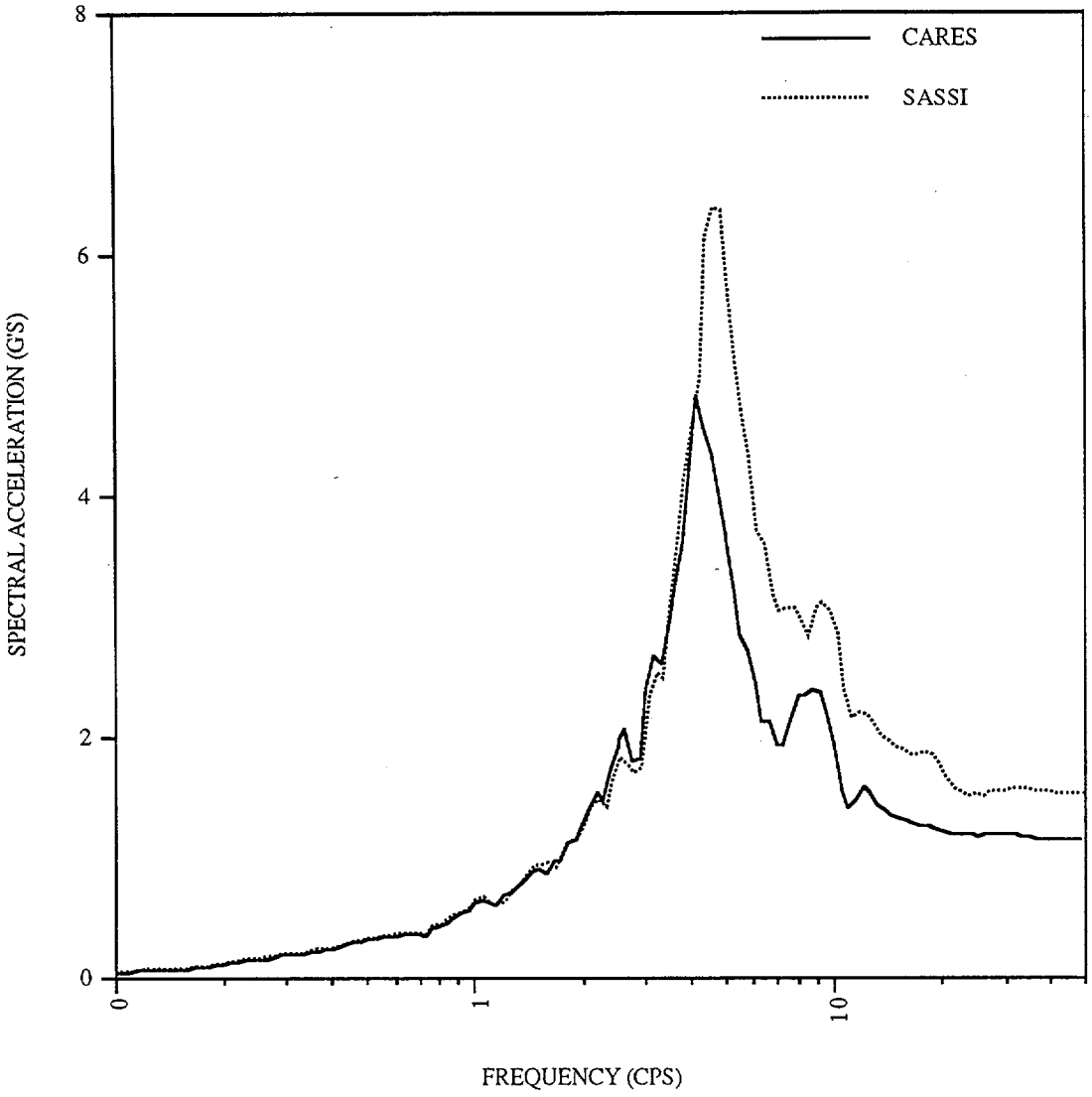

Figure 5.5 Response Spectra (5 %) at Node 16 of CSAB in Horizontal X Direction Due to Horizontal Input in X Direction (H1)

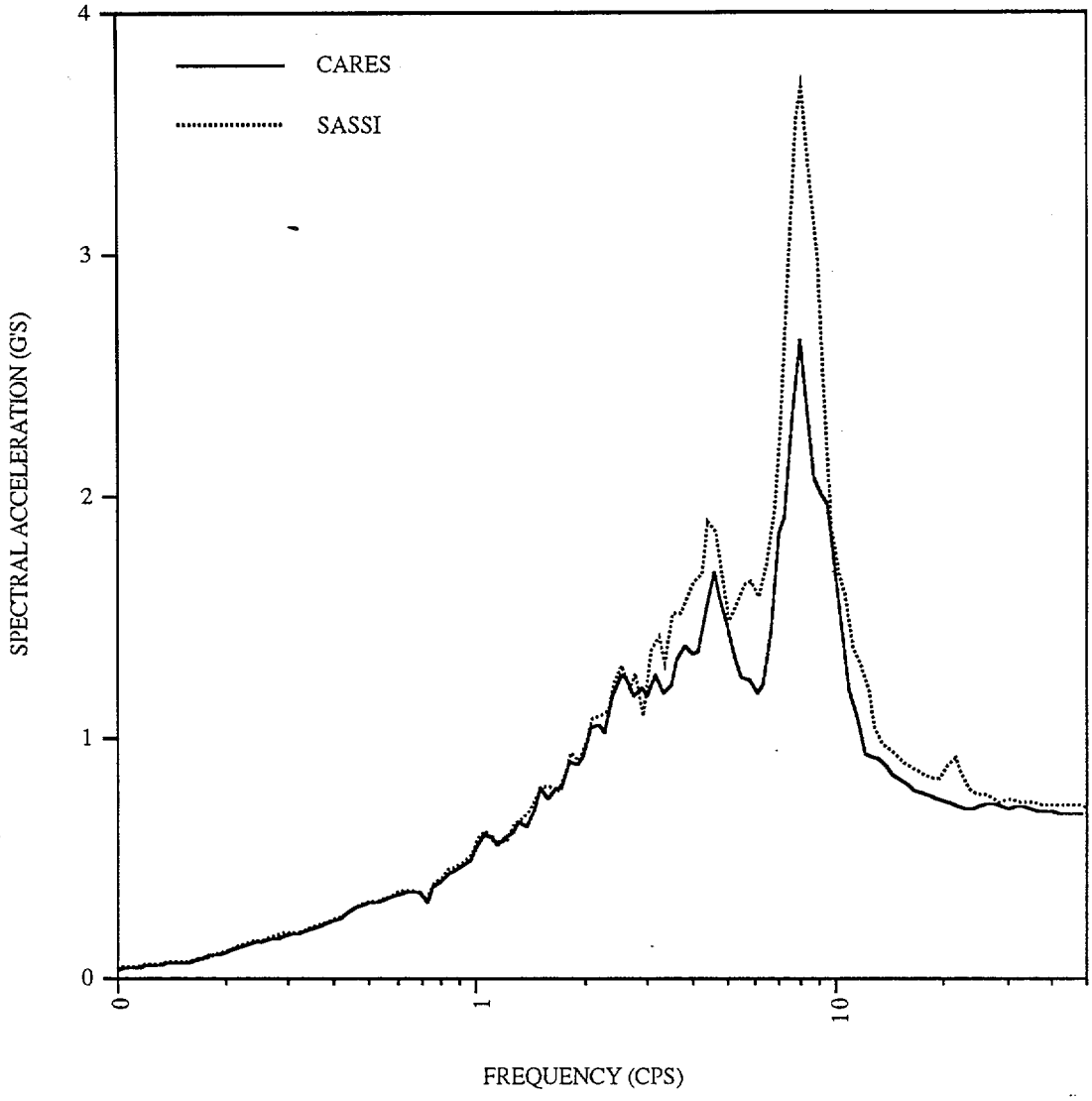

Figure 5.6 Response Spectra (5 %) at Node 110 of SCV in Horizontal X Direction Due to Horizontal Input in X Direction (H1)

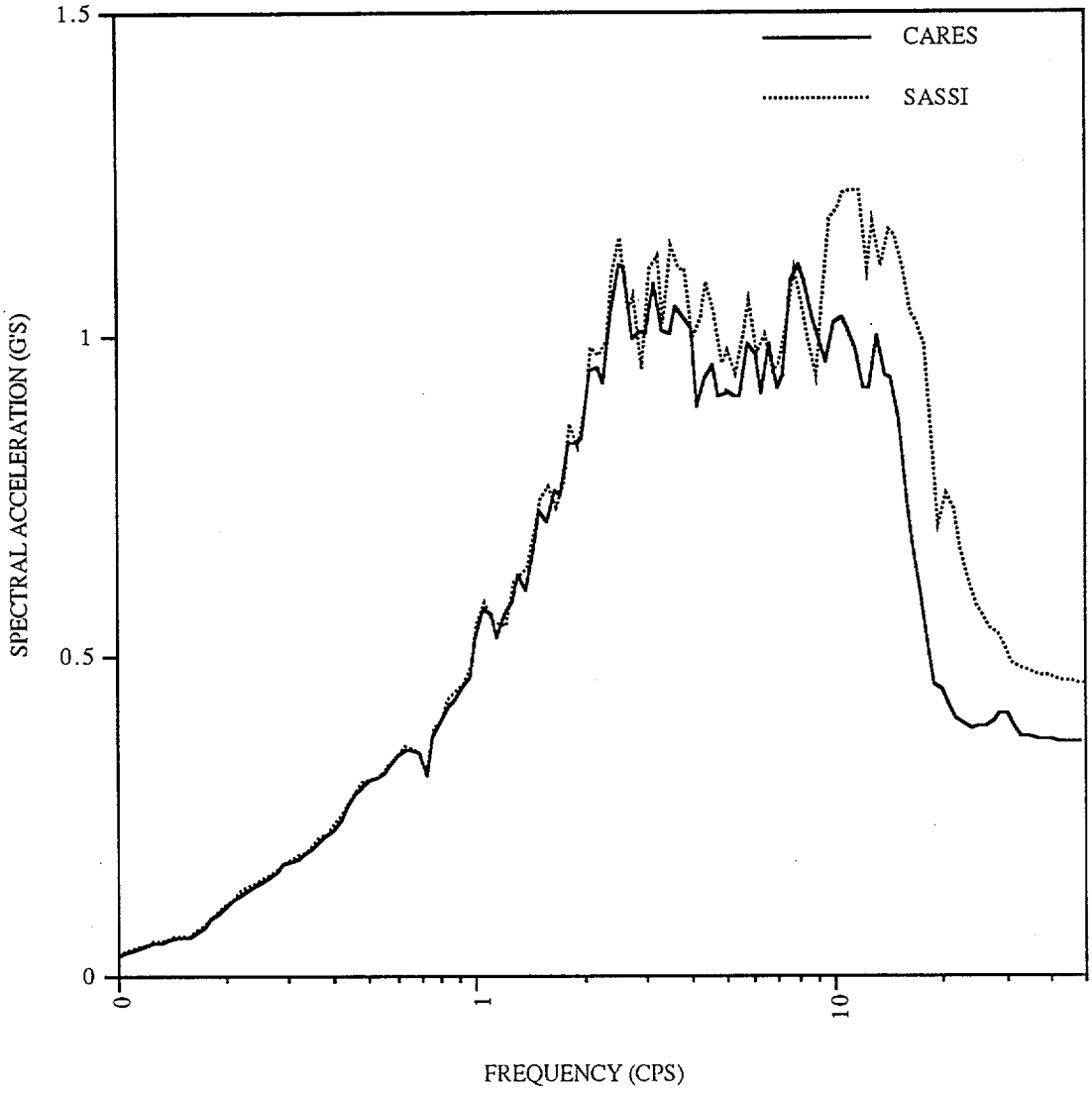

Figure 5.7 Response Spectra (5 %) at Node 204 of CIS in Horizontal X Direction Due to Horizontal Input in X Direction (H1)

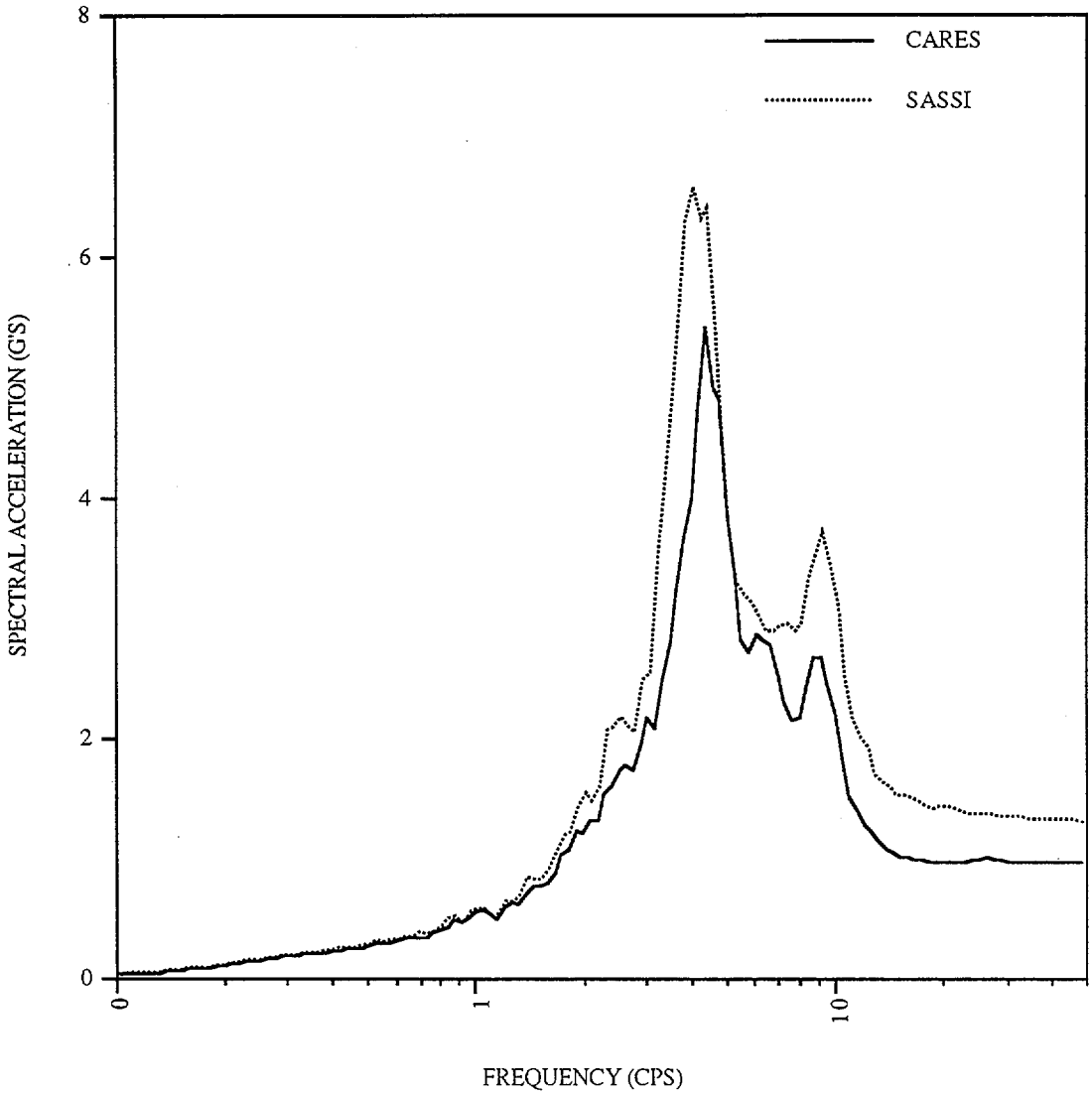

Figure 5.8 Response Spectra (5 %) at Node 16 of CSAB in Horizontal Y Direction Due to Horizontal Input in Y Direction (H2)

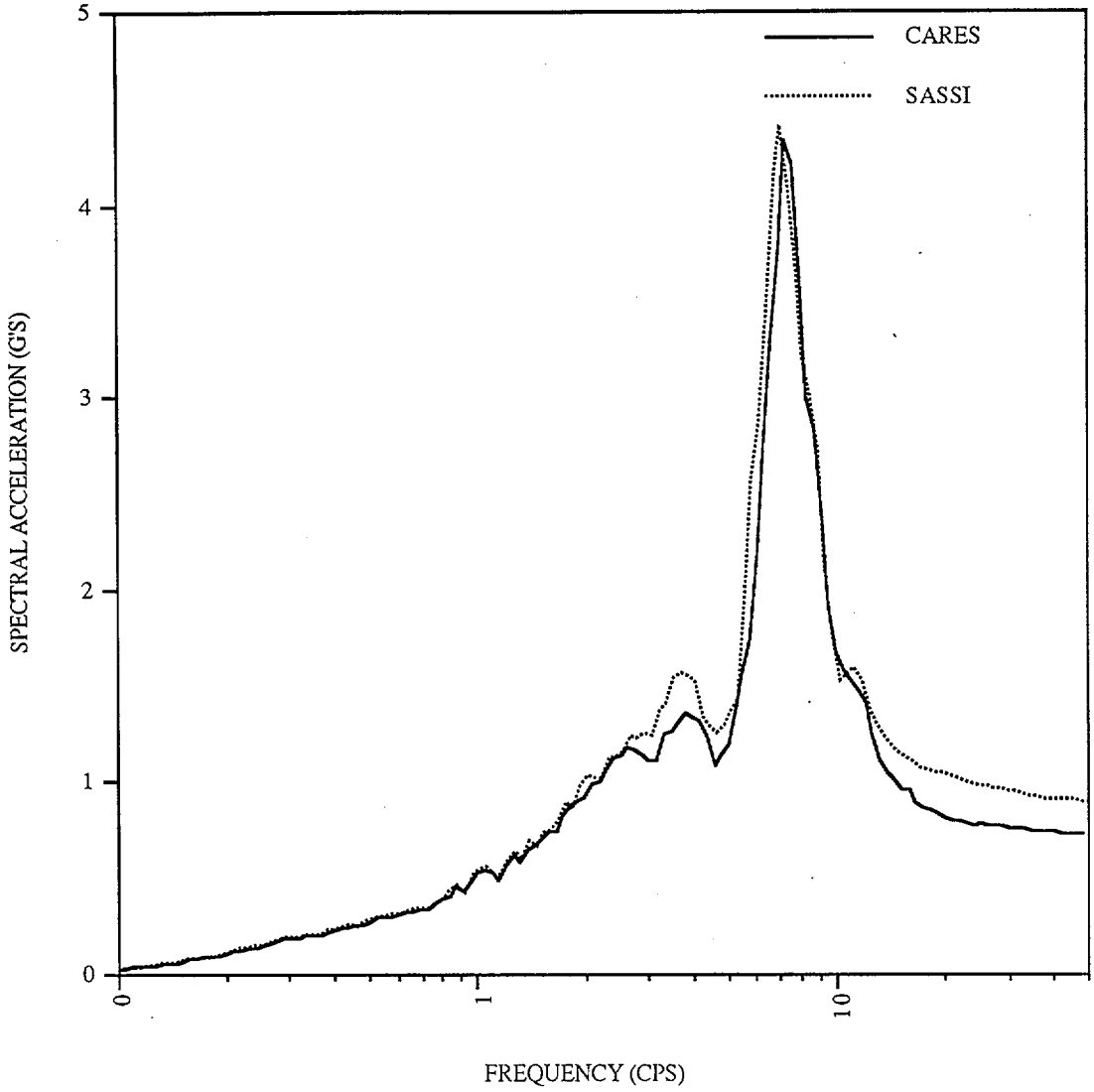

Figure 5.9 Response Spectra (5 %) at Node 110 of SCV in Horizontal Y Direction Due to Horizontal Input in Y Direction (H2)

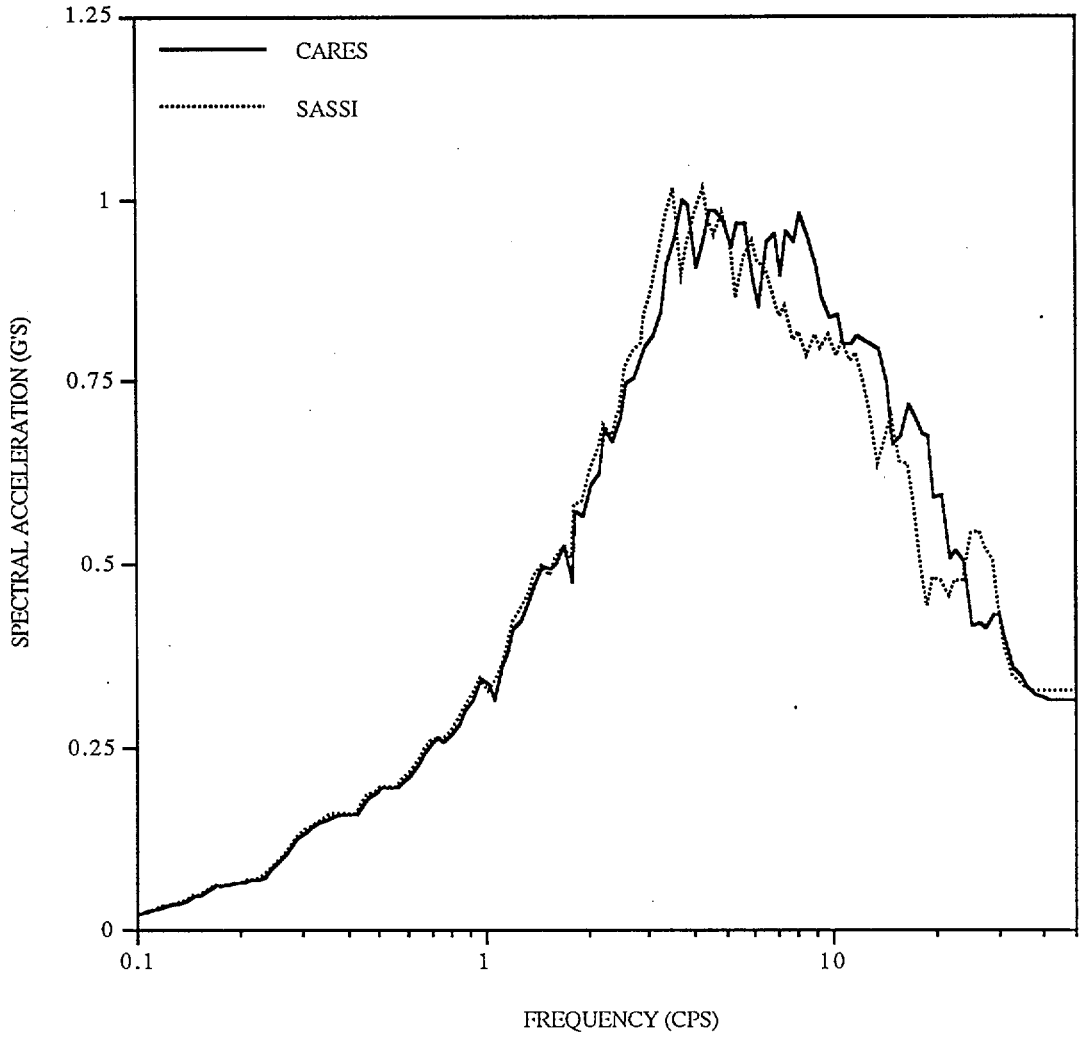

Figure 5.10 Response Spectra (5 %) at Node 204 (CIS) in Vertical Dirrection Due to Vertical Input

6.0 SUMMARY

This report presents a summary of the improvements and additions made to the CARES Code Version 1.0 during the development of Version 1.1 and 1.2 to allow the user greater flexibility in analyzing seismic problems of current interest. This updated Code as described herein has been made operational on a Unix SPARC station. In the revisions contained in this version, the following primary additions have been made to extend the capabilities of the analysis.

a. In the structural module, several aspects have been modified, with the primary change being associated with the major expansion of the size of the structural stick model that can be treated. The original version of CARES was designed for analysis of relatively small lumped mass stick models having relatively few degrees of freedom. The structural models currently being evaluated in the various advanced reactor studies are significantly larger than previously envisioned in the original development of the CARES Code. As part of this upgrade in the structural module, a shear wall element model has been added to the element library. The capability for inclusion of rigid links within the structural model and an improved algorithm for handling composite damping have also been added. In addition, SSI dynamic impedance coefficients for rectangular foundations are now included in the code. An approximate scheme has been included to account for embedment effects in the SSI analysis.

b. An improved Fast Fourier Transform (FFT) method was incorporated into the code to reduce run times required to convert between the time and frequency domains. This improvement is important when analyzing the larger stick models with many more degrees of freedom than previously treated.

c. In the free-field module, several changes have also been incorporated which extend the capability of the Code. The first change allows for the inclusion of a rock outcrop model within the soil column formulation. In addition, this module has been updated to allow users the option to incorporate a variety of soil degradation models within the analysis. Such modern developments have been found to be extremely important in site response evaluations at deep soil sites.

A number of other improvements have been incorporated along with these modifications to either extend the analysis capability or allow the user greater flexibility in analysis.

As a result of these extensions to the capability of CARES, and the much larger problem which can now be treated, a number of suggestions and cautions are presented below which should be considered prior to running on the SPARC station.

1. For very large models, running times can become very extensive, since solutions must be obtained at each frequency contained in the ground motion expansion. To reduce the number of frequencies at which solutions are required, a frequency interpolation routine can be added, similar to that used in the SASSI Code, to speed solution time.

2. The run time for a particular structural model is proportional to the bandwidth of the structural matrices developed, which, in turn, are directly related to the nodal numbering scheme used to develop the stick model. The bandwidth is controlled by the largest difference in number of the nodes at the ends of a particular element. To reduce run times, the numbering scheme used for the model should be reviewed carefully to try to minimize these differences. As an alternative, a bandwidth minimizer could be added to perform this function.

3. The simplified SSI models incorporated into CARES consider frequency dependent impedance functions generated between the structural foundation and the free-field halfspace, along with sidewall interaction components for buried facilities. The effects of kinematic interaction (wave passage effects), however, are not included in the model, although such simplified additions can be incorporated in a relatively straight forward manner to account for this effect.

4. Finally, although options are available for treating sidewall interaction effects for buried facilities, the vertical spatial variation of the free-field motions has not been incorporated into the model. Again, the Code can be easily extended to incorporate this effect, although for most problems of interest is probably not significant.

Version 1.2 of CARES was used to analyze the seismic response of the AP600 nuclear island. The model of the nuclear island was developed from a SASSI output listing of the model provided by the Westinghouse Corporation. Floor response spectra obtained from CARES were shown to compare reasonably well with spectra generated using the SASSI Code.

7.0 REFERENCES

1. C. J. Costantino, C. A. Miller, "Soil-Structure Interaction Methods: SLAVE Code", NUREG/CR-1717, Brookhaven National Laboratory, vol. II, September, 1979

2. C. A. Miller, C. J. Costantino, "Soil-Structure Interaction Methods: SIM Code", NUREG/CR-1717, Brookhaven National Laboratory, vol. III, September, 1979

3. J. Xu, A. J. Philippacopoulos, C.A. Miller, C. J. Costantino, "CARES (Computer Analysis for Rapid Evaluation of Structures)", Version 1.0, NUREG/CR-5588, by Brookhaven National Laboratory for the U.S. Nuclear Regulatory Commission, vols. 1 through 3, July 1990

4. P. B. Schnabel, J. Lysmer, H. B. Seed, "SHAKE: A Computer Program for Earthquake Response Analysis of Horizontally Layered Sites", Report No. EERC 72-12, University of California, Berkeley, CA, December, 1972

5. H. B. Seed, I. M. Idriss, "Soil Moduli and Damping Factors for Dynamic Response Analyses", Report No. EERC-70-10, University of California, Berkeley, CA, December, 1970

6. K. Coppersmith , "Ground Motion Following Selection of SRS Design Basis Earthquake and Associated Deterministic Approach", Final Report, Project No. 1724, Geomatrix Consultants, San Francisco, CA, January, 1991

7. I. M. Idriss, "Response of Soft Soil Sites During Earthquakes", Proceedings of the H. B. Seed Memorial Symposium, Berkeley, CA, May, 1990

8. C. J. Costantino, E. Heymsfield, Y. T. Gu, "Site Specific Estimates of Surface Ground Motions for the K-Reactor Site, Savannah River Site, Topical Report, Dept. of Civil Engineering, City College of New York, Report No. CEERC-91-003, for the Office of Reactor Restart (DP-12) U. S. Dept. of Energy, March, 1991

9. J. W. Cooley, J. W. Tukey, "An Algorithm for Machine Computation of Complex Fourier Series", Mathematics of Computation, vol. 19, no. 90, pages 297-301, April, 1965

10. A. White, P. Gray, "Fast Fourier Transform: A Pragmatic Approach", Monograph EE-M-2, Department of Electrical Engineering, North Carolina Agricultural and Technical State University, Greensboro, North Carolina, April, 1973

11. M. W. Salmon, S. A. Short, R. P. Kennedy, "Strong Motion Duration and Earthquake Magnitude Relationships", June, 1992

12. P. C. Jennings, G. W. Housner, N. C. Tsai, " Simulated Earthquake Motions", EERL, California Institute of Technology, April, 1968

13. R. W. Clough, J. Penzien, "Dynamics of Structures", McGraw-Hill Publishers, Second Edition, 1993

14. W. T. Thomson, "Theory of Vibration with Applications", Prentice-Hall Publishers, Fourth Edition, 1993

15. K. J. Bathe, E. L. Wilson, "Numerical Methods in Finite Element Analysis", Prentice-Hall Publishers, 1976

16. D. A. Gasparini, A. D. Choudhury, L. W. Curry, "Damping for Frames with Constrained Viscoelastic Layers", Journal, Structures Division, ASCE, vol. 106, no. ST1, Jan. 1980

17. A. Pais, E. Kausel, "Stochastic Response of Foundations," Massachussets Institute of Technology Research Report R85-6, February 1985