NUREG/CR-1717
BNL-NUREG-51263
VOL. IV
AN, RD

SOIL-STRUCTURE INTERACTION METHODS
SLAM CODE

C.J. Costantino
C.A. Miller

THE CITY UNIVERSITY OF NEW YORK
Under Contract to the
STRUCTURAL ANALYSIS GROUP
of
BROOKHAVEN NATIONAL LABORATORY
UPTON, NEW YORK 11973

September 1979

Prepared for the
U.S. NUCLEAR REGULATORY COMMISSION
OFFICE OF NUCLEAR REGULATORY RESEARCH
CONTRACT NO. DE-AC02-76CH00016
NRC FIN NO. A-3216-9

8101260144

NOTICE

Available from
GPO Sales Program
Division of Technical Information and Document Control
U.S. Nuclear Regulatory Commission
Washington, D.C. 20555
and
National Technical Information Service
Springfield, Virginia 22161

CONTENTS

Contents--Cont'd

1.0 INTRODUCTION

This report presents a detailed description of SLAM Code, a large finite element computer program to treat the two-dimensional (axisymmetric or planar) wave propagation problem through arbitrary nonlinear materials and the interaction of these motions with a flexible structure embedded within or on the soil. The acronym SLAM stands for Stresses in Layered Arbitrary Media; the Code, originally developed for the Air Force, has been in a constant state of development and improvement. Prior to presenting the details of the code operation and usage, a brief summary of the method of analysis and capabilities of the Code will be presented.

1.1 General Capabilities of the Code

The basic configuration of interest is shown in Figure 1 and consists of a general flexible structure embedded within a soil/rock foundation composed of an arbitrary number of material layers or zones, each layer possessing its own, generally nonlinear, constitutive law. To this system, loadings can be applied in any or all of the following ways:

1. Specified pressure histories can be applied to either boundary or interior surfaces in an arbitrary fashion, including the effects of moving loads.
2. Specified input motions (displacement or velocity pulses) can be applied to either boundary or interior surfaces or nodes, and
3. Specified force histories can be applied to the structure in terms of its generalized forces.

The general characteristics of the Code can be summarized as follows. More details on the specific methods of analysis will be presented later. The wave propagation from the input locations into the free-field is treated by finite element methods of analysis, including the effects of nonlinear properties of the soil. The finite element approach has been taken in this development to allow the user a general flexibility in treating problems of rather complex geometry (material layering, structural inclusions, etc.).

To treat the interaction of the free-field soil/rock with the embedded structure, two methods of analyses can be used, both available in SLAM Code. First, the finite element mesh can be continued throughout the structure as well as through the free-field. Thus, no special considerations need be made in the developed Code (save for possible separation and sliding effects which will be discussed later). Such an approach would be desirable when either nonlinear dynamic response of the structure or wave propagation through the structural wall is of concern.

When such is not the case, the use of finite elements through the structure can lead to serious drawbacks. Many more elements would be required to treat the stucture in this manner, increasing computer running time. Of more importance, however, is the following consideration. The stiffest material encountered in a typical problem is usually the structural material. In addition, the smallest sized elements in the problem occur through the relatively thin wall of the structure. This combination leads to extremely high frequencies in the mesh which, in turn, lead to extremely small time steps in the required numerical integration procedure. This occurs because the mesh in the structural wall is able to transmit the high frequency through-the-thickness waves which will develop.

Whenever this refinement in the solution is unwarranted, a second method of structural analysis is available in the Code. In this case, the structure is represented by its rigid body modes together with its lower free-free elastic modes of vibration. In this case, the critical frequencies of the system are those associated with the free-field as well as those lower frequencies of interest of the structure, and computer running times again become similar to those reeded to solve the free-field problem alone.

By either method of structural representation (finite elements or flexible modes), potential separation and sliding between the strucutre and the soil is accounted for by making use of a special (zero thickness line) element which is placed between the structural and soil nodes in the mesh. This element can be used with a simple Coulomb shear material model to limit both tensions and shears transferred between the soil and the structure.

An additional characteristic of the SLAM Code which has been incorporated concerns the use of non-reflecting or "quiet" boundaries. These boundaries are used to limit the size of the required mesh while at the same time minimizing the effects of reflections caused by artificial boundaries of the mesh. Such characteristics are desirable when long duration responses of the structure are desired.

To summarize then, the following general characteristics of the SLAM Code are available to investigate various two-dimensional (axisymmetric or plane) wave problems.

1. The free-field is represented by finite element methods including nonlinear material property effects.
2. The structure can be arbitrarily embedded within the free-field and can be represented by either finite elements or by its rigid body and elastic free-free mode shapes.
3. Potential separation and sliding between the structure and the soil can be treated by means of a special zero thickness element.
4. Non-reflecting boundaries can be used to minimize the size of required meshes for a given problem.

1.2 Free-Field Analysis

To treat the free-field wave propagation problem, the soil/rock material is divided into small elements, these elements being connected to each other at their vertices. The types of elements used in SLAM Code are rectangular elements, triangular elements and a zero thickness element which is used to simulate crack and soil-structure interface conditions. This latter element will be described more fully in a following section. The data that is developed is the motion history (displacements, velocities, accelerations, stresses, etc.) at these node points or vertices as a function of time. This method of mesh formulation is a physical one, as opposed to the more abstract approach of finite difference methods.

In the analysis, any material constitutive laws can be used, provided of course, that it can be suitably described for inclusion in the program. The specific soil/rock models used in the current code will be described in a later paragraph. The computational procedure starts from some time at which the complete solution is known; that is, displacement, velocities, and accelerations of all the nodes are specified, as well as the entire stress and strain history up to and including this time. Typically, this time is the zero or initial time, although it need not be. The problem then is to determine these same variables at the following instant of time, suitably taking into account the nonlinearities introduced by the material properties.

A typical interior node point of a two-dimensional mesh is shown in Figure 2, this node being connected to its surrounding nodes through the interconnecting elements. The equations of motion for this node can be written symbolically as:

$$\begin{aligned} M_N \ddot{U}_N &= F^A_{UN} - F^R_{UN} \\ M_N \ddot{W}_N &= F^A_{WN} - F^R_{WN} \quad , \end{aligned} \tag{1}$$

where M_N is the total nodal mass composed of the mass contributions from each adjacent element, (F^A_{UN}, F^A_{UN}) are the horizontal and vertical forces applied to the nodes (if any) and (F^R_{UN}, F^R_{WN}) are the node-resisting forces developed by the distortions of the surrounding elements, the summation being taken over all of the surrounding elements. Clearly, a displacement field causing only rigid body motions of the elements will develop no resisting forces at the nodes. The details for computing the node resisting forces from the element distrotion are presented in references 1, 3, 5, and 6.

Combining the equations for all the nodes, a set of second order equations are developed for the entire mesh which can be written symbolically as:

$$M\ddot{x} + Kx = F^A + F^N \quad , \tag{2}$$

where M is a diagonal mass matrix, x is a displacement vector consisting of the horizontal and vertical displacements of the nodes, K is the usual banded system elastic stiffness matrix and F^A is the vector of applied nodal forces. F^N is a vector of corrective forces which account for the nonlinearities in the material stress-strain properties or deviations from the elastic case. These nonlinear correction forces are computed at each time step for each element surrounding a given node.

The numerical integration scheme used to treat the wave propagation phenomena follows directly from Eq. (2). At a given time, t, when the nodal displacements and velocities are known (together with the previous histories), the nodal accelerations are computed by determining the applied loads (determined from any applied surface pressures), and the nonlinear correction forces (by knowing the current displacements and the previous element displacement and stress histories). Knowing the accelerations at this time, the displacements of the nodes can be determined at the following time, $t + \Delta t$, by a suitable numerical integration scheme. Knowing the new displacements, the cycle can be started again by determining the accelerations from Eq (2) at time $t + \Delta t$, etc. In this manner, the solution is marched out in time to obtain the complete history of the motion of all the nodes. Examples of various solutions obtained in this manner are presented in reference 6.

1.3 Material Constitutive Laws

The computer program developed to treat this problem contains a catalogue of material stress-strain laws which can be added to with little difficulty without changing the operation of the Code. Each material occurring in a particular problem can then be allowed to have any of the material properties available in the catalogue. The current catalogue allows the specification of the following stress-strain relations:

1. Elastic material, either isotropic or anisotropic.
2. Linear compressible fluid.
3. Elastic plastic material satisfying the Mises yield criterion with arbitrary strain hardening effects.
4. Elastic plastic material satisfying the Coulomb-Mohr yield criterion.
5. A nonlinear material law which contains a stiffening effect under hydrostatic pressure as well as a plastic dissipation under deviatoric strains to account for compaction effects in soils.
6. A special crack model which allows shear and tension transfer in a specified direction to be limited by a simple Coulomb shear law.

The last four of this list are the only nonlinear laws currently available in the Code, and have been included in an attempt to at least crudely approximate some known responses of soil/rock materials. Quite apparently, none of these models are completely adequate but until further advances in the state-of-the-art occur, only such approximations are available for applications to earth media.

1.4 Soil-Structure Interaction

The treatment of the interaction between the structure and soil begins with the assumption of continuity, that is, the nodes at the soil-structure interface are assumed to be attached to the structure and move with it. Separation and sliding between the soil and the structure is accounted for separately and will be discussed in a following paragraph. Let the displacements of the nodes attached to the structure be defined by the vector x_f a subset of the free-field displacement vector, x, defined in Eq. (2) . If there are p such nodes and if two-dimensional motion is considered, the components of x_f are then

$$\{x_f\} = \{u_1, w_1, u_2, w_2 \ldots u_p, w_p\} \tag{3}$$

where (u, w) are the horizontal and vertical displacements of the nodes.

The equations of motion of the structure are defined by its modal equations

$$M_s\ddot{y}_s + K_s y_s = Q_s \tag{4}$$

where y_s is the mode vector of the structural degrees of freedom and consists of the rigid body coordinates. M_s is a diagonal mass matrix consisting of the modal masses, K_s is a diagonal stiffness matrix, and Q_s is the vector of applied structural modal loadings. The displacements of the structure at the locations of the attached nodes are obtained by superimposing the modal vectors, or

$$x_f = Fy_s \tag{5}$$

where F is a matrix composed of the structural mode shapes.

From Eq. (2), the equations of motion of the nodes attached to the structure are

$$M_f\ddot{x}_f + F_f^R = -P \tag{6}$$

where P is the vector of interaction forces developed between the nodes and the structure. With these interaction forces, the corresponding modal loads applied to the structure are then

$$Q_s = F^T P \tag{7}$$

where the superscript indicates the transpose of the matrix. Substituting Eqs. (6) and (7) into Eq. (4), the equations of motion for the structure become

$$\overline{M}_s \ddot{y}_s + K_s v_s = -F^T F_f^R \tag{8}$$

where $\overline{M}_s$ is a nondiagonal mass matrix including the inertial coupling between the structure and the free-field, and is defined by

$$\overline{M}_s = M_s + F^T M_f F \ . \tag{9}$$

From this point on, the solution to the interaction problem procedes in a similar manner to the free-field problem. At a particular instant of time, the displacements of all the free-field nodes and structural mode displacements are known. The accelerations of all the nodes (except the attached nodes) are computed as before from Eq. (2). The resisting force vector, F_f^R, of Eq. (6) is determined during this computation. The modal accelerations of the structure are then computed from Eq. (8). The displacements of the free-field nodes and the modal displacements are then determined at the following instant of time by the integration algorithm. The displacements of the attached nodes are then computed from Eq. (5) and the solution then marched out in time as before.

1.5 Separation and Sliding Between Soil and Structure

In treating this separation and sliding problem, it is desirable to use a technique which does not deviate from the method of analysis outlined above. To accomplish this objective, a new finite element model was developed. For two-dimensional problems (axisymmetric or planar motion), a rectangular element is used which has a finite dimension in one direction and a zero dimension in the normal direction (see Figure 3). The properties of this element are determined by applying the limit process to the properties of the finite size rectangular element normally used. Of the four nodes comprising this element, two at one end have the same coordinates and two at the other end have the same coordinates. This element is situated between the soil and the structure so that the side of finite length lies on the interface. Two nodes are then located on and attached to the structure while two nodes are attached to the free-field nodes. Shear and tension transfer across this element can be governed by any of the nonlinear material properties available in the catalogue. The details of this element formulation are presented in Appendix C.

1.6 Quiet or Non-Reflecting Boundary

In order to obtain motion histories of adequate duration, mesh boundaries must be placed far enough from the area of interest so that no unwanted reflections from the mesh boundaries will arrive within the time period of interest. This requirement can lead to large meshes which, in turn, lead to long computer running times. In an attempt to overcome this situation, the "quiet" or non-reflecting boundary has been introduced to eliminate or at least reduce any unwanted reflections. These boundaries are placed away from the zones of interest but not as far as ordinary boundaries of the mesh would need to be located. Of course, the use of quiet boundaries implies that no effects of changes in materials past the boundary can be accounted for; that is, the use of the quiet boundary implies that the materials outside of the mesh are the same as those inside the mesh.

Various methods are available with which to achieve the quiet boundary. If only plane one-dimensional elastic wave problems are of interest (clearly not of interest herein), a simple and, what is more important, stable scheme can be devised. Since it is known that for such problems the stresses at any location are proportional to the particle velocities, or

$$\sigma = \rho c v \tag{10}$$

where ρ is the material mass and c is the material wave speed, a simple dashpot can be placed at the end of the rod with a damping constant equal to ρc.

If the rod material has nonlinear material properties, this can be used but with a time-varying damping constant depending upon the current wave speed at the end of the rod (a more difficult task from a programming point of view). For two-dimensional problems, this approach cannot be used since, even for elastic materials, there are two wave speeds and the simple relation of Eq. (10) no longer applies. All other methods of analysis are based upon predicting the velocities of the nodes on the quiet boundary from the velocities of the nodes immediately adjacent to the boundary at previous time steps.

Assuming continuity of velocities between nodes, the velocity of the quiet boundary node can be determined by the Taylor expansion:

$$v_j^i = v_{j-1}^{i-1} + a(\partial v/\partial x)_{j-1}^{i-1} + \Delta t(\partial v/\partial t)_{j-1}^{i-1} + \ldots \tag{11}$$

where the subscript j refers to the boundary node, the subscript j-1 refers to the node adjacent to the boundary node, the superscript i refers to the current time and the superscript i-1 refers to the previous time steps. In Eq. (11), the parameter, a, is the distance between the nodes j-1 and j, and the coordinate, x, refers to the direction from j-1 to j.

The time derivative in Eq. (11) is merely the acceleration of node j-1 at the previous time step. To approximate the space derivation, various forms can be used such as:

$$
(\partial v/\partial x)_{j-1}^{i-1} = \begin{cases} (v_j^{i-1} - v_{j-1}^{i-1})/a & \text{(forward differencing)} \\ (v_{j-1}^{i-1} - v_{j-2}^{i-1})/a & \text{(backward differencing)} \\ (v_j^{i-1} - v_{j-2}^{i-1})/2a & \text{(central differencing)} \end{cases} \tag{12}
$$

For the problems run to date with known solutions (one-dimensional rod problems), the central differencing scheme yielded the best results.

It should be pointed out that other techniques can and have been used. For example, if steady motion is assumed, the time derivative can be related to the space derivative by

$$
\partial v/\partial t = -c(\partial v/\partial x) \quad . \tag{13}
$$

Such an approach has been used in reference 7 for similar problems. However in that report, the dilatational wave speed was used for horizontal motion while the shear wave speed was used for the vertical motion. It is felt that such an approach is not necessary and, although adequate for one-dimensional elastic problems, is questionable for the general problem of interest.

All of these methods, however, are fundamentally unstable in their approach. That is, if the predicted velocity at the boundary is in error at a given time, this error will propagate back into the mesh at the following time steps. In turn, the errors so propagated will influence the predicted velocities of the boundary nodes at later times so that after significant times, it can be anticipated that major errors in the computation may develop. The actual estimate of the errors involved must await further experimentation with larger problems run for longer times.

2.0 General Formation of SLAM Code

The remainder of this report emphasizes the usage and operation of SLAM Code, while at the same time presenting some more detailed information on the analysis not contained in the references.

The program is divided into three primary segments which can be labeled as LINK1, LINK2, and LINK3. The function of these segments or links are:

- LINK1 - generates data and tables required for the numerical integration phase,
- LINK2 - computes the motion history of the finite element mesh and the embedded structure by a numerical integration technique, and outputs the history data generated, and
- LINK3 - computes shock spectra at desired locations.

Several pre- and post-processor programs have been previously developed to assist the user in preparing input data and analyzing output results. A program has been written to generate the mesh data (node and element data) required for a limited though important class of problems. This "mesh generator" program eliminates the need to manually key-punch this extensive data. A mesh plotting program is also available to plot the mesh, generated by hand or by the mesh generator program, to assist the user checking the accuracy and adequacy of the developed data. This program is discussed in Appendix D.

The subroutines comprising SLAM Code are shown in Table 1. The control of the program is maintained by the MAIN program which initiates calls to the proper segments. Each segment, in turn, initiates calls to its various sub-programs to generate the required data. The program makes extensive use of tape storage which helps satisfy two requirements, namely, to provide as large a flexibility as possible in the problem, and to maximize core storage available (increasing the problem size that may be treated). The program was initially written for a machine with 32K-word high speed memory and 12 physical tapes. The program, as currently developed, makes use of 8 tapes (or files) as well as the usual input and output files, together with about 45K word memory.

The need for auxiliary tape storage has also been caused by an additional capability introduced into the code. Much data generated (motion history, required tables, etc.) are permanently saved on tape for either a restart of the problem solution or for later analysis of the data. This additional capability increases the versatility of the code from a user's viewpoint. However, this increased capability is achieved (as always) at a price, this price being increased running time. The continued use of auxiliary (tape or disk) storage increases running time significantly since these peripheral processes interrupt the central processing or operation of the code. Significant time savings can easily be achieved by eliminating the tape handling aspects of the program. At the same time, however, the versatility of the code is significantly decreased.

In the following sections of this report, the details of the code operation are presented, together with the analysis used where required. The data setup and format are presented in a final segment of this report.

3.0 Description of LINK1

The first section of the program consists of a series of nine segments, L1A to L1J, which are used for table generation. All the data needed to describe the mesh and its properties is read (off cards) and used to generate three tables (stiffness tables, stress tables and nonlinear element tables) which are required for operation in the second link. In the following paragraphs, these sections of the Code are discussed.

The first four segments of the program (L1A to L1D) are used to set up and process the data required in the following parts of the program. The primary function performed in this section is to take the mesh data (node numbering scheme) read from input data cards and reorder (or renumber) the nodes into a form more suitable for the computations to follow. Basically, the objective is to minimize the bandwidth of the stiffness matrix which is to be formed in the following sections of the Code.

The stiffness matrix is to be maintained on tape for the integration or solution phase of the problem (in LK2B) since, in general, there is not enough core storage available to maintain the matrix in high speed memory continuously. Thus the stiffness data at each step of the computations are read into core in a series of "clusters" into a buffer storage area. Each cluster is composed of the data of N consecutive rows of the stiffness matrix. The value of N has been arbitrarily set to 100 in this version of the Code which allows sufficient storage to be treated in a 45K word machine. Obviously, the larger this cluster size, the more efficient the integration (less tape reads per time step). For most problems of interest, this cluster size cannot be too significantly reduced.

In forming the system stiffness matrix (in L1E), each element is treated one at a time, an element stiffness matrix computed and then distributed in usual fashion to the system matrix. Therefore (since data for only 100 nodes are maintained in core at one time), each element must have node numbers which differ from each other by less than 101. If this is not satisfied, the element matrix cannot be distributed to the system stiffness matrix. In addition, the smaller the maximum difference between element nodes (bandwidth), the more efficient the stiffness matrix formation (less tape usage).

Since the element and node data are entered in an arbitrary fashion by the user, the node numbers must be reordered to ensure satisfaction of the above criteria. In addition, the stiffness matrix storage requirements can be further reduced if the bandwidth is suitably small. However, this option has not been used in this version of the Code. This additional saving of core storage is achieved by taking advantage of the symmetry property of the stiffness matrix. The details will be described in Section IV of the report when describing subroutine ACCEL.

Segment L1A

This subroutine is used to read all the data associated with the mesh from data cards. This data includes:

a. Node point data

b. Element data

c. Output element data

d. Material property data

e. Loaded node point data

The details of this data are deferred to a later section. For each material (or zone) in the problem, the material elastic stress-strain matrix is determined from subroutine ELAST and stored for use later in the program. The element node data is ordered into the proper sequence in subroutine ORDER and all the element data placed on Tape 1 for future use. In addition, those elements composed of materials with nonlinear material properties are stored (together with requisite information) on Tape 14.

ORDER

Subroutine ORDER is used to reorder the element node data so that the Nodes I, J, K and L are placed in clockwise order. In addition, the variable KASE is determined to add to the element data list placed onto logical Tape 1 in subroutine L1A. The variable indicates the following:

KASE = 1 general triangular element

= 2 triangle with one node on axis of symmetry (for axisymmetric problem only)

= 3 triangle with two nodes on axis

= 4 general rectangular element

= 5 rectangle with one node on axis

= 6 rectangle with two nodes on axis.

Segment L1B

The purpose of this routine is to form the "adjacency" table which is made up of the following arrays:

(a) NADJNP (I) = the number of nodes connected or adjacent to the Node I

(b) NADJEL (I) = the number of elements surrounding the Node I, and

(c) NPADJ(I,J) = the numbers of the nodes adjacent to the Node I (J=1, NADJNP(I)).

Each element data list is read off Tape 1, one element at a time. A call is made to subroutine ADJNP to distribute the element node numbers to the proper location in the Table NPADJ. At the end of this operation, each row of the matrix NPADJ is scanned to determine the array NADJNP (in subroutine VADJNP).

Segment L1C

The maximum difference (termed the bandwidth) between adjacent nodes is then computed based on the original numbering scheme. This value is then printed out (value of MAXBD). A call is then made to subroutine PATH to determine an initial renumbering scheme. This subroutine is used to generate trial vectors NPTN(I) and NPTP(I) based upon the input path or start data stored in vector NSTART(I). These arrays are:

NPTP(I) = new node numbers in the original order,

(I = 1, NUMNP)

NPTN(I) = original node number in the new or revised order,

(I = 1, NUMNP)

NSTART(I) = start nodes,

(I = 1, NUMST)

Another array (ICP(I), I = 1, NUMGP) is also formed in subroutine PATH for additional information only. The details of the computation will be deferred until subroutine PATH is described.

Again, the bandwidth (MAXBD) is computed for this new or trial numbering scheme. This trial data is then entered into subroutine MINI which continues the minimization process to obtain (hopefully) a minimum bandwidth (or optimum numbering scheme). A haphazard set of start nodes should not be used as this may not lead to an increase in the bandwidth. The algorithm used is deferred to the description of subroutine MINI. With this new numbering scheme, the adjacency table is altered to reflect the new node numbering scheme. The node numbers of the loaded node points are then revised.

A call is then made to subroutine SIZE in which clustering information is determined. This, basically, is to determine how the stiffness (as well as nonlinear element) data are to be "clustered" on their respective files.

PATH

This routine obtains the first trial for a revised node numbering scheme. The routine, together with subroutine MINI, forms the algorithm to minimize the bandwidth and represents a compromise among various optimizing algorithms available. Ideally, the user would like to have an algorithm which is independent of any input, save for the input mesh data (or original numbering scheme). Various minimizing schemes are available and have been tested to obtain reordered numbering schemes. In general, these use more machine time than the algorithm used herein. For most of the problems of interest, however, a reasonable starting point for a renumbered system can be chosen by the user. For other reasons (primarily data input information), another "nonoptimum" scheme is more convenient for the original numbering system.

In any case, the routine uses as input a series of start nodes (NSTART(I), I = 1, NUMST) which are read in from data cards. These start nodes are usually a series of nodes lying on a surface at one side of the mesh. These nodes form the nodes in the first "partition" of the node data and are numbered from 1 to NUMST in the reordered system. All the nodes connected to these nodes are placed in the second "partition" and are numbered consecutively in the reordered system. Continuing, the nodes connected to this second "partition" are placed into a third "partition" and numbered consecutively, etc. Thus, the PATH routine forms the shortest path between any two nodes in the system.

Alternatively, the routine uses as a first trial a renumbering scheme wherein the nodes adjacent to each other are numbered close to each other. In this process, the two vectors NPTN(I) and NPTP(I) are formed which keep track of the relation between the new and original numbering schemes. The vector NPTN(I) indicates the original node numbers placed in their reordered system. Conversely, the vector NPTP(I) contains the new node numbers placed in their original order.

MINI

The trial numbering scheme is refined in this subroutine by forming a vector S(I). This vector is obtained by the relation

$$S(I) = \frac{1}{(1+K)} \{I + \sum_{J=1}^{K} N_J\} \tag{14}$$

where I is the node number, N_J is the number of the adjacent node and the sum is taken over the K adjacent node points. The number S(I) therefore represents an average node number. If there are significant differences between adjacent nodes, the value of S(I) will usually be different than the Node Number I. Ideally, if the nodes are numbered properly with all adjacent nodes numbered close to each other, the value of S(I) will be close to I for all the nodes.

The nodes are reordered again by sorting the vector S(I) into ascending order (performed in subroutine SORT1). The new bandwidth is then computed.

If this is less than the previous, the operation is performed again. If the bandwidth increases, the previous numbering scheme is used as the final revised numbering system.

The algorithm used can be shown by example to be highly sensitive to the original start nodes used as input, and does not lead, in general, to an optimum set (or minimum bandwidth). However, for most problems of interest, it has led to an adequate scheme, using relatively little machine time. Other algorithms are being tested with the view of obtaining systems which are more user independent while at the same time being simple.

SORT1

See description of SORT2.

SIZE

This routine determines the node cluster information stored in the following arrays:

NPLOW(I) = first node number of the cluster

NPHIGH(I) = last node number of the cluster

NPOUT(I) = one less than the smallest node number attached to any node within the cluster

NUMCP(I) = largest node number attached to any node in the cluster

The clustering used herein is in multiples of 100. Thus, NPLOW(I) = 1, 101, 201, etc., while NPHIGH(I) = 100, 200, 300, etc. The total number of clusters is determined and stored in NUMCLS.

Segment L1D

The final operation on the input data is performed in this section of the code. The original element data records are taken from Tape 1, the node numbers are changed to the revised scheme, and a number (KEY) added to indicate the smallest node number of the element.

This revised data is placed onto logical Tape 3. A call is then made to subroutine GSORT which reorders or sorts the element data according to the lowest node number of the element (KEY) and places this revised data onto logical Tape 1. Thus, when forming the stiffness data, all nodes numbered less than the current value of KEY associated with the particular element being treated will be unaffected by this element and all of the following elements.

Similarly, the node data which were maintained on Tape 14 are posted in GSORT according to the new numbering scheme and placed in ascending order onto Tape 4. The nonlinear element data are then taken off Tape 14, cluster number determined (JJ), and reordered by ascending cluster number.

GSORT

This routine is a control routine to sort a series of data maintained on an input file (INTAPE). The data are stored in a buffer array (IARRAY) which can store up to MXRCDS data records, each record containing NWRDS words. The data is to be reordered in increasing order of the word specified by NKEY which indicates which word of the data record is the key for sorting. The resorted data are placed on an output file (IOUTAP). Two intermediate files (INT1 and INT2) may be used as intermediate storage in the sorting operations.

If all the data records to be sorted can be stored in the buffer area, a core sorting routine (SORT2) is used. If more records are available than can fit in core, a tape sorting routine (TSORT) is used. For this case, the two intermediate files will be used.

SORT2 and TSORT

These two routines are used for sorting data records of various lengths according to ascending order of one word of each record specified by NKEY. The routines are based upon a simple "bubble sort" method in which two adjacent records are compared (according to the NKEY word of the record) and interchanged, if desired. In SORT2, all the data to be sorted are maintained in machine core and the sorti done continuously. In TSORT, the data are taken from tapes in clusters, merged together and written onto output files. The same "bubble sort" is used for each cluster with an additional sort superimposed between each tape cluster.

Segment L1E

This segment of the program generates the elastic stiffness matrix of each element and assembles the system stiffness matrix. The specific formulas used in the development are contained in reference 3 for both the triangular and rectangular elements and need not be repeated herein. The development of the stiffness matrix for the zero thickness or cracked element is presented in Appendix C. The node point data are maintained (in the revised order) on logical Tape 4, the adjacency data on logical Tape 8 and the element data on logical Tape 1. The buffer stiffness matrix is first zeroed out and the node and adjacency data for up to 100 nodes taken into core.

The element data are taken off the tape one at a time. For each element, the data read from logical Tape 1 consists of

KEY = lowest node number

NUME = element number

IZONE = material zone number

KASE = element type (as described previously)

NTI, etc. = new node numbers

NCRACK = 0 - regular element, = 1 - crack element

The element stiffness matrix ([CK]) is then computed in STIFF. (A call from STIFF to INTER is made to compute the required element integrals.)

The element stiffness matrix is then adjusted in ADJUSK if any of the element nodes are roller supported (ITYPE = 1). The matrix is adjusted to the directions parallel and perpendicular to the roller support. The element matrix is then distributed to the system matrix in DISTK. The mass matrix is computed and distributed in subroutine MASS. This element data are then placed on logical tape 12 temporarily for later use in determining the stress matrix.

This procedure is continued until the first element is reached whose largest node number exceeds the node numbers in core. At this time, the completed stiffness matrix (I = 1, KEY-1) is printed, if desired, in subroutine PRNK and the completed stiffness data written off onto logical Tape 10. The remaining data are moved up in core, more node and adjacency data taken into core, and the computations continued until all the elements have been treated. In addition, the highest main diagonal "frequency" is computed as the stiffness data are completed. This data are required for choosing integration time steps and are discussed further in a later section.

Segment L1F

This routine is used to determine the elastic stress displacement matrix required for stress computation for those output elements which are composed of linear elastic material. Basically, the stresses are computed at the centroid of each element by the relation (see Ref. 3)

$$\{\sigma\} = [S]\{x\} \qquad (15)$$

where $\{x\}$ is th[illegible]ment displacement vector. For those output elements which are composed of nonlinear material, the element stresses are required in the integration computation. Thus, this step is omitted for these elements. The development of the [S] matrix for the crack element is also presented in Appendix C.

Segment L1G

Subroutine L1G is used to form the pressure coefficients CPRESS. These coefficients are used in LINK2 to convert applied pressure data to equivalent nodal forces. The coefficients are computed in subroutine COEF and are based upon the relations presented in Ref. 3.

Segment L1I

This subroutine is used to cluster the stiffness data developed in L1E into nodal clusters of 100 each and store these on logical Tape 10 for use in the integration step. In addition, the nonlinear elements are clustered, with all those elements affecting any node in a cluster filed together.

Segment L1J

This routine is used to generate the initial nonlinear element tape required in LINK2 to compute the nonlinear nodal forces. This data is generated for all the elements composed of potentially nonlinear materials. The specific computations are presented in Ref. 1 and also in the description of subroutine PLASTF.

4.0 Description of LINK2

The following paragraphs present a description of the operation of the integration (or solution) section of the SLAM Code. The motion history of all the nodes as well as that of the embedded structure (if any), are stored on tape for possible use in LINK3 or with any plotting programs, if desired. Output is also printed out at selected time intervals during the integration.

The equations of motion for both the nodes and the embedded structure are presented in Section I in Equations (1) and (8), respectively. To correspond to the notation used in the computer program, the resisting forces at Node i can be written as

$$\begin{Bmatrix} F^{R}_{ui} \\ \\ F^{R}_{wi} \end{Bmatrix} = \sum_{j=1}^{s} \begin{bmatrix} K^{uu}_{ij} & K^{uw}_{ij} \\ \\ K^{wu}_{ij} & K^{ww}_{ij} \end{bmatrix} \begin{Bmatrix} u_j \\ \\ w_j \end{Bmatrix} - \begin{Bmatrix} F^{nu}_{i} \\ \\ F^{nw}_{i} \end{Bmatrix} \qquad (16)$$

where (F^{nu}_i, F^{nw}_i) are the horizontal and vertical force components of the correction forces to account for the difference in material behavior from the linear elastic case. The K matrix of Equation (16) represents the usual stiffness matrix for linear elastic systems. The K terms multiplied by the corresponding node displacements (u_j, w_j) represent the elastic resisting forces developed by the distortion of the elements surrounding a particular node i. The summation in Equation (16) is taken over all of the nodes adjacent to the particular node i, plus the node i itself.

The numerical integration procedure is a simple one and is based on the wellknown Newmark Beta method (Ref. 2) with the parameter β chosen to be zero.

Thus the procedure is basically a predictor method of numerical integration. To simplify the description further, the equations of motion can be written simply as

$$\begin{aligned}
&\text{(a)}\quad Mx = F^A - F^R \\
&\text{(b)}\quad \bar{M}_s x_s + K_s x_s = -F^T F^R + Q_s^A
\end{aligned} \qquad (17)$$

where Q_s^A are the applied (if any) structural modal loads.

At some time t, let it be assumed that the complete solution is known, that is, x(t) and x(t) are specified together with all of the previous history required. Knowing this data, the applied and resisting forces developed at this time, t, can be computed (the details of which will be discussed in a later description). From Equation (17), the node and structural mode accelerations, x(t) and $x_s(t)$, can then be calculated. The objective is now to determine the displacements and velocities at the following instant of time, or $x(t + \Delta t)$ and $x(t + \Delta t)$, as well as the corresponding structural terms. This is done by the extrapolation formulae

$$\begin{aligned}
x(t + \Delta t) &= x(t) + x(t)\Delta t + x(t)\frac{(\Delta t)^2}{2} \\
x(t + \Delta t) &= x(t) + (x(t) + x(t + \Delta t))\frac{(\Delta t)}{2}
\end{aligned} \qquad (18)$$

Note that in determining the velocities at $(t + \Delta t)$, the accelerations at this time, $x(t + \Delta t)$, are required. For forces [F^A, F^F of Equation (17)] that are dependent only on time and displacements (no velocity dependence), the procedure is to compute the predicted displacements, $x(t + \Delta t)$ and $x_s(t + \Delta t)$, and to use these to obtain the predicted accelerations $x(t + \Delta t)$, which can then be used to compute the predicted velocities, $x(t + \Delta t)$. A simplified flow diagram for this procedure is shown in Figure 4. Prior to discussing the details of the integration phase of the solution, a description of the tape usage is presented.

Tape Usage:

There are four primary files used in the integration step of the solution, these tapes being labeled as follows:

(a) The Restart Tape (Logical 8) which contains all the data necessary for restarting the problem.

(b) The Stiffness Tape (Logical 10 and/or 1) which contains the K matrix of Equation (1).

(c) The Nonlinear Element Tapes (Logicals 12 and 3) which contain all data required to compute the nonlinear forces, and

(d) The History Tape (Logical 14) which contains the motion history records for a selected set of element. node and structure data.

The restart tape contains all the problem parameters associated with both the free-field and the structure together with the motion (displacement, velocity and acceleration) for all nodes and structural modes at a particular time. This tape is written at a specified (by input data) number of time intervals during the integration. To restart the problem at a later date, these data are read into the code and the integration continued.

The history data tape contains a restricted set of data for all time steps through the integration, thus providing a "continuous" record of the motion history. This tape can then be used later as input to a plotting program to automatically plot the results generated.

As mentioned previously, the element tapes contain all the data required to compute the nonlinear correction forces for those elements composed of (potentially) nonlinear materials. During the acceleration computation initiated by the call to ACCEL, the nonlinear data from the previous time step is read off one logical tape, nonlinear forces computed, the new stress-strain state for each nonlinear element updated, and written off onto a second logical tape. For the following time step, this second tape is used as input, the data again updated and written off back onto the first tape. This procedure is continued through the integration process alternately reading and writing the element tapes to continually update the element data. The two tapes used to store the element data are Logicals 3 and 12.

Integration Step Size

In the generation of the elastic stiffness tables performed in LINK1, a pseudo-period is determined by the following method. The equations of motion for each node (Equation (1)) can be written as

$$[\,M\,]\{\ddot{x}\} + [K]\{x\} = \{F^A\} + \{F^N\} \qquad (19)$$

if typical matrix notation is used. A dynamic matrix can be found from

$$\{\ddot{x}\} + [\,M\,]^{-1}[K]\{x\} = [\,M\,]^{-1}(\{F^A\} + \{F^R\}) \qquad (20)$$

where $[\,M\,]^{-1}[K]$ is the dynamic matrix, and $[\,M\,]$ is a diagonal mass matrix (lumped mass configuration). If the dynamic matrix is diagonalized, the diagonal terms will, of course, correspond to the elastic frequencies of the system. A reasonable approximation to the highest frequency of the system can be chosen as the largest value of the main diagonal of the dynamic matrix, avoiding the necessity of diagonalizing the system. A similar operation is performed on the coupled structural equations [Equation (17b)] to obtain the highest frequency for the soil-structure system.

This highest frequency approximation is used to determine the shortest period in the system or

$$T_{min} = \frac{2\pi}{\omega_{max}} \tag{21}$$

where ω_{max} is the largest frequency. The integration interval must be some fraction of this period; this fraction being determined by the stability of the integration routine used. For the system used in SLAM Code, it has been found that the time increment should be less than 1/18 of the smallest pseudo-period. The period is printed out in LINK1, after the stiffness and mass tables are completed.

The data read in LK2B (variables ET, KDT and KINT) are used to determine the time increment used. If KDT is set to zero, the time increment used (DT) is taken as the variable read in ET. This is then checked against the minimum time given by

$$\overline{DT} = T_{min}/KINT \tag{22}$$

If $\overline{DT}$ is less than DT as read, the program is halted; otherwise, the integration procedure begins with the time increment as read. If KDT is set to one, the time increment is chosen as the value $\overline{DT}$ obtained from Equation (22).

Relation Between Mesh and Integration Step Size

As mentioned above, the time step required for stable integration is determined by the highest frequency in the system (or highest wave speed). In most problems of interest, the computed response is desired so as to provide reasonable results over a particular frequency range of interest. The low frequency range, in fact, defines the overall size of the mesh since a suitably long record length is required to adequately define the low frequency content. For example, if data are desired around the 1 cps frequency range, the record length should be at least several seconds long. Boundaries (including transmitting or nonreflecting boundaries) must then be placed suitably far from the zone of interest so as not to obtain spurious signals during the time of interest from the mesh boundaries.

At the high frequency end of the spectrum, the required mesh or element size can be estimated from the approximate relation

$$f = \frac{c}{4a}$$

where c is the dilatational wave speed and a is the dimension of the smallest element in the mesh. If the high frequency response is specified, the size of the elements required can be determined from the above relation.

ACCEL

This subroutine computes the accelerations at each node in the free-field and for each structural mode at a given instant of time. From Equation (1), the node accelerations are composed of

- applied nodal forces
- nonlinear correction forces
- elastic stiffness forces.

The stiffness forces are determined by multiplying the stiffness matrix coefficients by the appropriate node displacement; the stiffness matrix being obtained from tape in blocks or "clusters" of 100.

The data read off the tape is stored in a buffer area in core (as specified by the COMMON/B/ storage allocation). As will be noted in a later description of subroutine PLASTF, this same storage area is used when computing the nonlinear correction forces.

The parameters associated with the stiffness tables have the following significance:

NADJNP(I) = number of nodes adjacent to or connected to a particular node (less than or equal to 8)

ITYPE(I) = the type of node (0 is free in both directions, 1 means free in one direction, 2 means fixed in both directions. See description of data input).

THETA(I) = angle of roller support, if ITYPE = 1. (See description of data input).

MASS(I) = node mass

Referring to Equation (16), the stiffness force terms can be written for Node i as:

$$\begin{aligned} &\text{u-equation:} \quad K^{uu}_{ii} u_i + K^{uw}_{ii} w_i + \sum_{j=1}^{N} (K^{uu}_{ij} u_j + K^{uw}_{ij} w_j) \\ &\text{w-equation:} \quad K^{wu}_{ii} u_i + K^{ww}_{ii} w_i + \sum_{j=1}^{N} (K^{wu}_{ij} u_j + K^{ww}_{ij} w_j) \end{aligned} \qquad (23)$$

Thus, the stiffness variables in the program can be related to Equation (23) by the following table:

Program Variable	Stiffness Variables
SNPUU(I)	K_{ii}^{uu}
SNPUW(I)	K_{ii}^{uw}, K_{ii}^{wu}
SNPWW(I)	K_{ii}^{ww}
NADJNP(I)	N
SADUU(I,J)	K_{ij}^{uu}
SADUW(I,J)	K_{ij}^{uw}
SADWU(I,J)	K_{ij}^{wu}
SADWW(I,J)	K_{ij}^{ww}

The adjacent stiffness values (SADUU to SADWW) are stored in compact form. To determine the node number associated with these values (the j of Equation (11)), the Table NPADJ(I,J) is used. That is, to determine the force contribution from the first term of, say, SADUU (SADUU(I,1)), this term must be multiplied by the u-displacement of Node NPADJ(I,1). The term MXADJP refers to the maximum number of nodes that are allowed to be adjacent to any node which is previously set to 8.

In the node force computation, the first step is to determine the pressures applied to the nodes that are loaded (the node numbers specified by the Vector NPLOAD(I,J)). Up to 100 nodes can be specified as potentially loaded nodes along a single surface with two such surfaces allowed. The R and Z coordinates of these nodes are contained in the arrays RAD and ZAD which are maintained in COMMON. With minor corrections, an arbitrary number of loaded surfaces may be allowed. This procedure will be described in the discussion of subroutine PRESS. The pressures applied to these nodes are determined by the call to subroutine PRESS. Both the horizontal (PRESSU) and the vertical (PRESSW) pressures are returned to ACCEL. The PRESS routine is independent of the rest of the program and any routine required for a particular program can be substituted.

The pressures are converted to node point forces by using the coefficients CPRESS which are determined in L1G and transferred through COMMON. For example, if the horizontal applied force to Node j is to be computed, the following relation is used:

$$F_j^{Au} = p_{j-1}^u C_j^1 + p_j^u C_j^2 + p_{j+1}^u C_j^3 \qquad (24)$$

where p_j^u is the horizontal pressure at Node j, and the C_j^i's are coefficients converting pressures at Nodes j-1, j and j+1 to the horizontal force at Node j. The vertical force computation is similar, except that the vertical pressures are substituted for the horizontal pressures. The development of the coefficients C_j^i (i=1, 2, 3) is presented in Ref. 3.

After the pressures are computed at all the loaded node points, the nonlinear forces are determined by the call to PLASTF. If the number of nonlinear elements (NUMPEL) are zero, this step is, of course, omitted. In PLASTF, the nonlinear forces are stored in the proper acceleration locations. The node stiffness data is then read and stored into the buffer storage region (this region was already used in PLASTF to compute the nonlinear correction forces). For this cluster of data, the applied forces developed at all loaded nodes within this cluster are computed. If the node type (ITYPE) is 1 (roller support), the applied force is converted to force components parallel and perpendicular to the roller. The applied node forces are then added to the previously determined nonlinear force components.

The remaining force computation is the elastic stiffness forces which are computed for the cluster in core. This computation is accomplished by multiplying the stiffness data by the appropriate node displacement. Again, this is added to the acceleration data already generated. The final accelerations for all the nodes within the cluster are determined by dividing the accumulated force data by the appropriate node mass. The procedure is repeated for all the node clusters until the accelerations of all the nodes are computed.

Prior to reading into core the data for the next cluster, the resisting forces at the nodes attached to the structure (the KX terms minus the nonlinear correction terms) are stored in a separate array for later use in computing the structural accelerations. At the completion of the free-field calculations, the structural accelerations are then determined.

PLASTF

This subroutine is used to compute the nonlinear correction forces developed by any material which exhibits nonlinear or nonelastic behavior. These correction forces are computed on an element-by-element basis with the nonlinearities for each element determined at each time step. If the mesh is composed of materials (zones), all of which possess potential nonlinearities, data for all the elements must be stored on the element tapes. If, however, only some materials in the problem have nonlinear properties, only those elements which are within these zones of materials must be maintained on the element tape.

For each element of interest in this computation, a block of data (61 words) is used for transmission and updating of data. The buffer area (COMMON/B/) contains space to accept 78 elements at one time into core. Thus, the element tapes are blocked in clusters of 78 elements. In each pass (integration step) through PLASTF, the required data is read in from the input tape, a cluster at a time, updated, and written onto the output element tape. The next time step uses this output tape as input, updates and generates a new output tape which was the previous input tape, etc.

The computation of the nonlinear correction forces is based on the following analysis (Ref. 1). The stress-strain relation for the nonlinear materials considered is

$$\{\sigma\} = [C](\{\epsilon^T\} - \{\epsilon^N\}) \tag{25}$$

where $\{\epsilon^T\}$ is the total strain vector $\{\epsilon_r, \epsilon_\theta, \epsilon_z, \gamma\}$, $\{\epsilon^N\}$ is the nonlinear strain vector, [C] is the elastic stress-strain relation, and $\{\sigma\}$ is the stress vector $\sigma_r, \sigma_\theta, \sigma_z, \tau\}$. The total strains are related to the node displacements by

$$\{\epsilon^T\} = [B]\{x\} \tag{26}$$

where $\{x\}$ is the node displacement vector $\{u_i, w_i, u_j, w_j, u_k, w_k, u_l, w_l\}$, u and w are the node displacements as before, and B is a matrix defined by the element geometry. If the element is a triangle, the node displacement vector is a (1x6) array with the last two elements (Node 1) missing. Thus, the B matrix is either a (4x6) or (4x8) array depending upon whether the element is triangular or rectangular.

The element nonlinear correction force can be determined by the relation (Ref. 1)

$$\{R^N\} = [P]\{\varepsilon^N\} \tag{27}$$

where $\{R^N\}$ is the vector $\{R_i^{nu}, R_i^{nw}, \ldots\}$ and is either a (1x6) or (1x8) array depending on the element shape. Once the element nonlinear correction force is determined, it is merely distributed to the nodes of the element. The total nonlinear correction force $\{F^N\}$ of Equation (2) is determined as the sum of all the element nonlinear forces surrounding the nodes.

The procedure followed in PLASTF is then fairly straight forward. At any time, t, the current node displacements as well as the past history of the elements are known (or at least all the data that are required is maintained). The object is to determine the nonlinear strain vector $\{\varepsilon^N\}$ for each element, which depends upon the particular constitutive law of the element material. The element cluster data from the previous time increment is read into the buffer region in core.

The data on this tape is the following:

NUMCEL = number of elements already investigated plus the number of elements being read in this cluster

NELBUF = number of elements being read in this cluster

NOOFEL = original element number read in from cards

IZONE = counter describing which material zone the element lies in

NP = node numbers of the element nodes

B = B matrix of Equation (26)

EPSTI1 = total strain vector at centroid of element from previous time increment

EPSPI1 = nonlinear strain vector from previous time step

SIGI1 = previous stress vector

DUM = general data applicable to each material which must be used for that material analysis.

The next step is to determine the {x} vector of Equation (26) from the known node displacements (u, w). If a node is on a roller support (ITYPE=1), the value of u is taken in the direction of the roller while w is maintained as zero. The total strains at the current time, EPSTI, are computed from Equation (26). With this data, entry is made into the material subroutines (MISES, COULMR, COMPCT or NCOUL) depending on the value of the parameter IPLAST. The current values of the various parameters are returned from the subroutine and the necessary data updated.

The element nonlinear forces are then computed (FPLAST) from Equation (27). These forces are then stored in the proper node acceleration locations (UDDN, WDDN). The new data are then written off onto the element update tape.

MISES

This routine is used to determine the current stress and strain state for a material satisfying the Mises yield condition (Prandtl-Reuss flow equations) with arbitrary strain-hardening properties. The yield surface is defined by the magnitude of effective stress

$$S = \frac{1}{\sqrt{2}}\{(\sigma_r - \sigma_\theta)^2 + (\sigma_\theta - \sigma_z)^2 + (\sigma_z - \sigma_r)^2 + 6\tau^2\}^{1/2} \qquad (28)$$

The strain hardening is defined by linear segments as shown in Fig. 5. Up to 10 segments are provided for and these segments are defined by the parameters SSTAR(J) indicating the effective stress at the beginning of the segment and HSTAR(J) indicating the slope of the segment.

If the previous stress-state lay on the yield surface, the plastic strain-rate vector is defined from the usual normality condition. If the plastic flow occurring over the increment is assumed to follow the initial strain-rate vector, the plastic strain increments can be written as

$$\{\Delta\varepsilon^P\} = \Delta e^P\{\dot{\varepsilon}^P\} = \frac{\Delta e^P[G]\{\sigma\}_{i-1}}{2s_{i-1}} \tag{29}$$

where $\{\Delta\varepsilon^P\}$ is the plastic strain increment vector, $\{\dot{\varepsilon}^P\}$ a vector proportional to the plastic strain-rate vector, s_{i-1} is the effective stress at the beginning of the increment, $\{\sigma\}_{i-1}$ the corresponding stress-state, Δe^P the increment in effective plastic strain and G a matrix defined by

$$[G] = \begin{bmatrix} 2 & -1 & -1 & 0 \\ -1 & 2 & -1 & 0 \\ -1 & -1 & 2 & 0 \\ 0 & 0 & 0 & 6 \end{bmatrix} \tag{30}$$

The total effective plastic strain is then given by

$$e^P = \Sigma\Delta e^P \tag{31}$$

where the sum is obtained by adding the effective strain increments developed during each time increment.

The procedure to determine the current stress state from the previous known state is as follows: A fictitious stress state is first computed by assuming that no new plastic strains will occur over the time increment. If this were the case, the new stress state would be

$$\{\bar{\sigma}\}_i = [C](\{\varepsilon^T\}_i - \{\varepsilon^P\}_{i-1}) \tag{32}$$

where the subscript i refers to the current time step. The effective stress associated with this state, $\bar{s}_i$, is obtained from Equation (28). If $\bar{s}_i \leq s_{i-1}$, then no yielding took place and the barred state is the actual state. In this case, $\{\varepsilon^N\}_i = \{\varepsilon^P\}_i = \{\varepsilon^P\}_{i-1}$. (Note that for this material, the nonlinear strains correspond to the plastic strains and the superscripts N and P are interchangeable.) If $\bar{s}_i > s_{i-1}$, then the fictitious stress state is inadmissable and a current state must be found, together with the amount of increased yielding. The new stress state is given by

$$\{\sigma_i\} = [C](\{\varepsilon^T\}_i - \{\varepsilon^P\}_{i-1} - \Delta e^P\{\dot{\varepsilon}^P\}_{i-1}) \tag{33}$$

where it is assumed that yielding follows the initial strain rate vector. From Equations (32) and (33), the correct stress state is

$$\{\sigma\}_i = \{\bar{\sigma}\}_i - \Delta e^P[C]\{\dot{\varepsilon}^P\}_{i-1} \tag{34}$$

This new stress state must also lie on the yield surface, that is, the new effective stress must satisfy

$$s_i = s_{i-1} + H\Delta e^P \tag{35}$$

Substituting Equation (34) into Equation (28) and equating the result to Equation (35), the following relationship can be obtained to determine the plastic strain increment,

$$\alpha\delta^2 - \beta\delta + \gamma = 0 \qquad (36)$$

For the plane strain or axisymmetric problem, the parameters of Equation (36) are given by

$$\begin{aligned}
\delta &= (E/s_{i-1})\Delta e^P \\
\alpha &= A - (H/E)^2 \\
\beta &= B + 2(H/E) \\
\gamma &= (\bar{s}_i/s_{i-1})^2 - 1 \\
A &= [3/2(1+v)]^2 \qquad (37) \\
B &= [3/(1+v)](1/s_{i-1})(\bar{\sigma}_r\varepsilon_r + \bar{\sigma}_\theta\varepsilon_\theta + \bar{\sigma}_z\varepsilon_z + \bar{\tau}_\gamma)
\end{aligned}$$

where H is the slope of the effective stress-plastic strain curve (Fig. 5) and E is Young's Modulus. For the plane stress problem

$$\begin{aligned}
A &= 9/4(1+v)^2 - [(2-v)(1-2v)/4(1-v^2)^2][(\sigma_r + \sigma_z)/s]^2 \\
B &= [1/(1-v^2)]\{[(5-4v)/2][(\bar{\sigma}_r\sigma_r + \bar{\sigma}_z\sigma_z)/s^2] + \\
&\quad [(5v-4)/2][(\bar{\sigma}_r\sigma_z + \bar{\sigma}_z\sigma_r)/s^2] + [9(1-v)(\bar{\tau}\tau/s^2)] \qquad (38)
\end{aligned}$$

where the barred stresses refer to those computed from Equation (32) and the unbarred stresses to the actual stresses at the beginning of the increment.

In the subroutine, this procedure is followed directly with some minor additions. The variables SSTAR, ESTAR, HSTAR refer to the effective stress and plastic strain at the beginning of the segment of

Fig. 5 and HSTAR is the forward slope of the segment. The variable NOYILD indicates the number of segments in the yield curve of Fig. 5. Other variables of interest are

SYI1 = yield stress

SMAXI1 = maximum effective stress previously obtained, $\leq$ SYI1

SI1 = the effective stress at the previous time step, $\leq$ SYI1

EEFFI1 = effective plastic strain at the previous time

EPSDI1 = plastic strain rate vector at the previous time step.

The first step in the procedure is to compute the barred stresses of Equation (32). If yielding is to occur (SBAR > SYI1), the remaining procedure is started. If the previous stress state was below the yield surface, the stresses and strains are adjusted to obtain a new previous state lying on the yield surface. The value of δ of Equation (36) is then computed, which, by definition, must be positive. If not, an error flag is detected, data printed out and the probram halted (Cards 1740 to 1950). This usually occurs when the total strain increment $(\{\varepsilon^T{}_i\} - \{\varepsilon^T\}_{i-1})$ is relatively large, so that a solution based on the initial strain-rate vector is obtainable. It can easily be shown that if this increment is small, a solution always exists.

As a further check on the solution obtained, an iteration cycle is then begun to determine an improved value of $\Delta\varepsilon^P$ (variable DEEFF). If the initial solution is good, obviously only a few cycles need be performed. The iteration cycle is set to continue until the error is computed stress is within 1 percent of the yield surface. This iteration cycle is also needed if, during the increment, the slope of the plastic yield curve changes so that adifferent slope H must be used.

COULMR

This subroutine is used for materials which obey the Coulomb-Mohr yield condition with its associated flow rule (normality condition). The yield surface is defined by

$$f = \alpha I_1 + \sqrt{I_2'} = k \tag{39}$$

where

$$I_1 = \sigma_r + \sigma_o + \sigma_z$$

$$I_2' = \frac{1}{6}[(\sigma_r - \sigma_o)^2 + (\sigma_o - \sigma_z)^2 + (\sigma_z - \sigma_r)^2] + \tau^2 \tag{40}$$

and α and k are material properties related to the usual soil properties, the angle of internal friction, ϕ, and cohesion, c. If these properties are determined from a triaxial test series, the parameters of the yield surface may be found by

$$\alpha = (2c/\sqrt{3})[\sin\phi/(3 - \sin\phi)]$$

$$k = (6c/\sqrt{3})[\cos\phi/(3 - \sin\phi)]. \tag{41}$$

As for the Mises material, the plastic strain rate vectors are obtained from the normality condition, or

$$\{\dot{\varepsilon}^P\} = \alpha\{I\} + (1/6\sqrt{I_2'})[G]\{\sigma\} \tag{42}$$

where {I} is a unit vector {1,1,1,0} and G is the matrix defined by Equation (30).

In principal stress space, the yield surface is a cone (Fig. 6) whose axis makes equal angles with the stress axes. If we call this axis the Λ line, the angle between the Λ line and the normal to the yield surface can be found from

$$\cos\theta = [\alpha^2/(\alpha^2 + 1/6)]^{1/2} \qquad (43)$$

As before, we proceed by first assuming that no plastic strain occurs over the increment; a fictitious stress state can be computed from

$$\{\bar{\sigma}\}_i = [C](\{\varepsilon^T\}_i - \{\varepsilon^P\}_{i-1}) \qquad (44)$$

If this new stress state lies below or on the yield surface, it is acceptable and is the true state. If it lies above the yield surface, it is inadmissible and the correct stress vector must be determined. This new stress state lies on the yield surface also, and once again, it is assumed that the plastic strain increment is proportional to the initial strain rate vector. As in Equation (34), the new stress vector is given by

$$\{\sigma\}_i = \{\bar{\sigma}\}_i - \lambda[C]\{\dot{\varepsilon}^P\}_{i-1} \qquad (45)$$

where λ is a factor of proportionality to be found. To conform to the notation of the program, a vector $\{A\}$ is defined by

$$\{A\} = \begin{Bmatrix} A_x \\ A_y \\ A_z \\ A_w \end{Bmatrix} = [C]\{\dot{\varepsilon}^P\}_{i-1} \qquad (46)$$

so that

$$\{\sigma\}_i = \{\bar{\sigma}\}_i - \lambda\{A\} \tag{47}$$

The stress invariants of the new stress state are

$$\begin{aligned} I_1 &= \bar{I}_1 - \lambda B_1 \\ I_2' &= \bar{I}_2' - 2\lambda B_2 + \lambda^2 B_3 \end{aligned} \tag{48}$$

where

$$B_1 = A_x + A_y + A_z$$

$$\begin{aligned} B_2 = \tfrac{1}{6}\{&(\bar{\sigma}_r - \bar{\sigma}_\theta)(A_x - A_y) + (\bar{\sigma}_\theta - \bar{\sigma}_z)(A_y - A_z) + \\ &(\bar{\sigma}_z - \bar{\sigma}_r)(A_z - A_x)\} + \bar{\tau}A_w \end{aligned}$$

$$B_3 = \tfrac{1}{6}\{(A_x - A_y)^2 + (A_y - A_z)^2 + (A_z - A_x)^2\} + A_w^2$$

Substituting Equations (48) into the yield condition, the parameter λ can be found from

$$D_1\lambda^2 + D\lambda + D_3 = 0 \tag{49}$$

where

$$D_1 = (B_3 - \alpha^2 B_1^2)$$

$$D_2 = (2\alpha^2\bar{I}_1 B_1 - 2B_2 - 2\alpha k B_1)$$

$$D_3 = (\bar{I}_2' - k^2 + 2\alpha k\bar{I}_1 - \alpha^2\bar{I}_1^2)$$

As for the Mises material, if the strain increment is too large, a solution for λ (which must be positive) cannot always be found.

The previous analysis, of course, is based on the assumption that the initial stress state is not at the apex of the cone. If it is, the strain-rate vector is undefined since the stress invariant I_2' is zero. The procedure used in the subroutine is as follows: It is first assumed that the stress state, $\{\sigma\}_i$, remains in the corner. Since the stress is then the same as the previous stress, $\{\sigma\}_{i-1}$, the elastic strain is the same as previously. Therefore, the entire strain increment had to be plastic, that is,

$$\begin{aligned}\{\Delta\varepsilon^P\} &= \{\varepsilon^T\}_i - \{\varepsilon^T\}_{i-1} \\ &= \{\varepsilon^T\}_i - \{\varepsilon^E\}_{i-1} - \{\varepsilon^P\}_{i-1} \\ &= \{\varepsilon^T\}_i - \{\varepsilon^P\}_{i-1} - [k(1-2\nu)/3\alpha E]\{I\}\end{aligned} \tag{50}$$

If the angle between this strain increment vector and the Λ line is less than θ (defined by Equation (43)), this assumption is correct.

If the angle is greater than θ, the stress vector $\{\bar{\sigma}\}_i$ is used to define the current stresses. The deviatoric component of $\{\bar{\sigma}\}_i$ is reduced until the stress point lies on the yield surface. This new stress point is taken as the current stress point.

In the subroutine, the variable KORNER is used to determine if the initial stress state lies at the cone apex or not. If KORNER=0, the stress state at the beginning of the increment is not at the apex; while if KORNER=1, the initial state lies at the apex. The routine follows the analysis outlined above. The only variation is an iteration routine which refines the solution for λ after it is computed from the above procedure.

COMPCT

This subroutine is used to determine the nonlinear strains developed in a material which exhibits irreversible compacting properties under hydrostatic stress. The analysis used is based upon the constitutive relations developed in Ref. 4. Basically, the approach is more empirical than the previous two constitutive relations but at the same time attempts to reproduce known properties of actual soils.

The yield surface description of this material has the following form

$$f = \sqrt{I_2'} - k_e(I_1) = 0 \tag{51}$$

where

$$k_e(I_1) = \begin{cases} k - \alpha I_1[1 + (I_1/2c)], & \text{for } (I_1 + c) \geq 0 \\ k + (\alpha c/2) & , \text{for } (I_1 + c) < 0 \end{cases} \tag{52}$$

where (α, k) are the Coulomb-Mohr parameters defined in Equation (41) and c is another material parameter which must be specified as input. By comparison with the previous analyses, it can be noted that at low values of hydrostatic pressures, the yield condition is similar to a Coulomb-Mohr description, while at high pressures, it approaches the Mises yield criterion (Fig. 7). The parameter c controls the shape of the yield surface.

In the subroutine COMPCT, two options are allowed the user which, in effect, allow him to simulate four (4) different material constitutive laws. It remains to be seen, however, whether these laws are in fact general enough to simulate real soil properties. In any case, it is up to the analyst to decide which options to use to best suit his purpose. The first option concerns the choice of relation between the hydrostatic stress and the hydrostatic strain. Both options allow for the inclusion of hysteretic effects in the hydrostatic component, which is not available in either the Mises or the Coulomb-Mohr representations.

The second option allows for a choice of flow rule to use with the yield surface. If an associated flow rule is chosen, the usual normality criterion is enforced stating that the plastic strain-rate vector is normal to the yield surface if the material is being loaded. An alternate option can be used, this allowing a nonassociated flow rule to define the strain-rate vector. This nonassociated flow rule uses the Mises flow criterion throughout the yield surface. By noting Fig. 7, this implies that the Mises flow rule applies even for values of the first invariant of stress I_1 greater than the parameter c. This option plays a significant role in specifying the material behavior and has as much validity as the associated flow rule for real soil materials.

(a) Hydrostatic Properties

As mentioned, two different hydrostatic stress-strain relations are incorporated in the subroutine which we will denote as Case A and Case B. The first relation (Case A) is written as a relation between the bulk modulus and the first stress invariant (I) or

$$K_L = \begin{cases} K_o - K_1(I_1 + \gamma c) + K_2(I_1 + \gamma c)^2, \text{ for } |I_1| > \gamma c \\ (K_o - \gamma c K_1/4) + [(K_1/\gamma c)(I_1 + \gamma c/2)^2], \text{ for } |I_1| \leq \gamma c \end{cases} \tag{53}$$

where K_L is the bulk modulus for the loading case (that is $\dot{I}_1 > 0$), and K_o, K_1, K_2 and γ are material parameters. For an unloading situation ($\dot{I}_1 < 0$), the bulk modulus is defined as

$$K_u = \begin{cases} K_o - K_1 I_1 + K_2(I_1)^2, \text{ for } (I_1 + I_1^L) < 0 \\ K_4(1 - \alpha I_1/k)^{1/2}, \text{ for } (I_1 + I_1^L) > 0 \end{cases} \tag{54}$$

where

$$K_4 = [K_o + K_1 I_1^L + K_2 (I_1^L)^2]/(1 + \alpha I_1^L/k)^{1/2}$$

and I_1^L is another material property. The relationship between K and I is shown in Fig. 8 and was developed to represent data for a particular soil type (Ref 4). The value of I_1^L is chosen to represent fluid or linear behavior at high hydrostatic stresses. Below this value, the material is much stiffer on unloading than on loading.

The second representation of the hydrostatic properties is based upon a trilinear model as shown in Fig. 9. The unloading modulus (K_u) must be greater than the loading moduli (K_o, K_1, K_2). The parameters I_1, I_1', K_o, K_1, K_2 and K_u must be supplied as input data to the code.

In both cases, no information on the tension side is required since no hydrostatic tensile stresses are allowed in the subroutine.

(b) Shear Properties

As mentioned previously, two different flow rules are specified in the subroutine. The nonassociated flow rule, being simpler, is discussed first. The first step in the analysis is to divide the stress and strain vectors into their hydrostatic and deviator components, or

$$\begin{aligned} \{\sigma\} &= \{s\} + H\{I\} \\ \{\varepsilon\} &= \{e\} + E\{I\} \end{aligned} \qquad (55)$$

where {I} is the unit vector {1,1,1,0}, as before. The stress-strain relation is simply:

$$\begin{aligned} \dot{E} &= (1/3K)\dot{H} \\ \{\dot{e}\} &= (1/2G)\{\dot{s}\} + \lambda\{s\} \end{aligned} \qquad (56)$$

where K is the bulk modulus, G is the shear modulus and λ is a factor of proportionality (greater than or equal to zero) to account for plastic strain increments.

The invariant relationships are

$$\begin{aligned} I_1 &= 3H \\ \dot{I}_1 &= 3\dot{H} \end{aligned} \tag{57}$$

and

$$\begin{aligned} I_2' &= -(1/2)\{s\}'[F]\{s\} \\ \dot{I}_2' &= -\{s\}'[F]\{\dot{s}\} \end{aligned} \tag{58}$$

where F is the matrix

$$[F] = \begin{bmatrix} 0 & 1 & 1 & 0 \\ 1 & 0 & 1 & 0 \\ 1 & 1 & 0 & 0 \\ 0 & 0 & 0 & -2 \end{bmatrix} \tag{59}$$

The yield condition can be written as

$$I_2' = k_e^2$$

or

$$\dot{I}_2' = \beta \dot{I}_1 \tag{60}$$

where k_e is a function of I_1 only and is defined in Equations (51) and (52). The term β is

$$\beta = 2k_e k_e' \tag{61}$$

where

$$k_e' = dk_e/dI_1$$

Substituting Equation (56) into (60) and using the relations of Equation (58), the parameter λ may be found from

$$\lambda = (1/4GI_2')\left\{-\beta\dot{I}_1 - \{s\}'[F]\{\dot{s}\}\right\} \tag{62}$$

where the vector $\{\dot{s}\}$ is defined as

$$\{\dot{s}\} = 2G\{\dot{e}\} \tag{63}$$

From Equation (56), $\dot{I}_1$ can be found from

$$\dot{I}_1 = 9K\dot{E}. \tag{64}$$

In the subroutine, Relation (62) is used to obtain the value of λ by using the values of the variables at the beginning of the increment together with the known increments in strain.

The development for the associated flow rule is similar to the above except for the influence of plastic flow on the hydrostatic stresses. The stress-strain rate relations are

$$\begin{aligned} \dot{E} &= (1/3K)\dot{H} - \lambda k_e' \\ \{\dot{e}\} &= (1/2G)\{\dot{s}\} + (\lambda/2\sqrt{I_2'})\{s\}. \end{aligned} \tag{65}$$

Substituting these into Equation (60), the parameter λ can be found as

$$\lambda = \frac{(-9K\beta\dot{E} - \{s\}'[F]\{\dot{s}\})}{(2G\sqrt{I_2'} + 9K\beta k_e')}. \tag{66}$$

NCOUL

This routine is used for materials whose shear strength transfer is defined by a simple two-dimensional Mohr envelope, rather than the complex relations for the general Coulomb-Mohr material defined previously. A Mohr envelope is defined again by the cohesion and friction angle. The element (the zero thickness crack model) has a given orientation defined by the angle of the element to the horizontal. All stresses and strains are rotated to this orientation.

A fictitious stress state is first computed assuming that no shear slip occurs during the increment and that both normal and shear stresses are computed. If the shear stress exceeds the allowable, defined by the relation

$$\tau_{all.} = c - \sigma_n \tan\phi, \tag{67}$$

the shear stress is then reduced to the allowable. This new stress state is rotated back to the (R,Z) directions and nonlinear "strains" computed from

$$\{\varepsilon^N\} = \{\varepsilon^T\} - [c]^{-1}\{\sigma\} \tag{68}$$

These plastic strains although not related to any "flow rule" for the material will yield the correct stress state in the usual calculations performed in the remainder of the code.

5.0 Special Routines for LINK2

In the integration phase of the program, several subroutines are primarily problem dependent; these routines specifying essentially the loading and boundary inputs to the problem. These routines, therefore, can be added and changed at will to suit the particular problem of interest, and the only concern need be the transfer of the proper data between these routines and the calling programs. The specific routines of concern are:

PRESS - compute any applied surface pressures

QUIET - compute quiet boundary motions

BOUND - input specific boundary motions

QGENER - compute applied generalized loadings to the embedded structure.

Subroutine PRESS

Subroutine PRESS must be specified for each problem to be solved. Obviously, some subroutines may be written to apply to a wide class of problems, but this decision is left to the user. Basically, the data input to PRESS transferred through the list is the following:

MXLINE = maximum number of loaded surfaces (limited to 2)

MXLOAD = maximum number of loaded node points on a given surface (100)

LOADNP = the actual number of loaded node points on a surface

LINES = number of surfaces (1 or 2)

RAD = radial coordinates of the loaded nodes (inches)

ZAD = vertical coordinates of the loaded nodes (inches)

T = time (seconds)

TX = constant to convert T to real time (seconds)

The output from the subroutine is PRESSU and PRESSW, the horizontal and vertical pressure components applied to the nodes.

As mentioned previously, the current formulation accepts pressure applied along two loaded surfaces of connected nodes. To include several more such loaded surfaces in the problem, only minor changes must be made. In this description, it will be assumed that M-loaded surfaces will be allowed and that N ($\leq$ M) surfaces will be used in a particular problem.

(a) The COMMON region variables should be changed to LOADNP(M), NPLOAD(M,100), RAD(M,100), ZAD(M,100), CPRESS(M,100,3).

(b) The COMMON/A/ region variables should be changed to PRESSU(M,100), PRESSW(M,100). The changes in (a) and (b) will allow up to 100 nodes on each of the M-loaded surfaces. These can be decreased by changing the dimension of these variables as desired.

(c) In the MAIN routine, set MXLINE=M.

(d) In LIG, make sure the proper data for N-loaded surfaces are read from cards (LINES=N).

The specific pressure routine included in this version of SLAM Code is designed to generate the overpressure history for ground ranges in the 10 to 50 psi region. The pressure pulse is assumed to be a steady one (constant shape) which travels from left to right at a constant shock velocity (U). The computer time (t) is assumed to start when a ground disturbance reaches the left hand boundary of the mesh. The airblast pulse is then assumed to be delayed by an amount (t_D), such that at a specific time, t, in the computations, the air pressure shock front has reached a range

$$R = U(t - t_D). \tag{69}$$

Conversely, at a given range, R, the arrival time, t_A, of the shock front is

$$t_A = t_D + R/U. \tag{70}$$

The parameters of the pressure pulse form are taken from Ref. 9. The input to the problem is specified as the peak overpressure (P_o), the weapon yield (W) and the delay time, t_D. At a given ground range (R), the pressure profile is given by

$$P(t) = \begin{cases} 0 & , \text{for } t < t_A \\ P_o \epsilon^{-\alpha\tau}(1 - \tau) & , \text{for } t_A \leq t \leq (t_A + t_p) \\ 0 & , \text{for } (t_A + t_p) < t \end{cases} \tag{71}$$

where

$\tau = (t - t_A)/t_p$

t_A = arrival time (Equation (70))

t_p = positive phase duration.

The parameters of Equation (71) are defined by

$$U = 1100[1 + (6/7)(P_o/14.7)]^{1/2}$$

$$\alpha = (\log P_o)/1.53 \tag{72}$$

$$t_p = (W)^{1/3}[(6.25 - \log P_o)/18]$$

where the units are P_o in psi, U in fps, and W in KT.

Subroutine QUIET

As discussed in Section I of this report, the quiet boundary condition makes use of the central difference scheme. The data transferred to the subroutine and contained in the list are:

T	= current time in the machine
DT	= integration time step
UD,WD	= velocity vectors of all the node points
UDD,WDD	= corresponding acceleration vectors
NUMNP	= number of node points in problem
MAXNP	= maximum number of node points
NPTN	= vector converting original numbering scheme to new numbering scheme.

The QUIET subroutine is entered at each time step in the integration after all the node accelerations have been computed in ACCEL. The objective is to modify the accelerations of the quiet boundary nodes such that these nodes behave as though other node points exist outside the mesh. Any algorithm desired can be used in place of the one included in this version of SLAM Code.

Subroutine BOUND

This subroutine is used to input any motion time history (either displacements or velocities) to a particular set of node points or a boundary surface. The variables transferred through the subroutine list are:

T	=	machine time
MAXNP	=	maximum number of node points
NUMNP	=	number of node points
UN,WN	=	either node displacement or velocity vectors
NPTN	=	vector converting original numbering scheme to new numbering scheme
ISWTCH	=	counter to indicate if either displacements or velocities are specified.

As may be noted from Subroutine LK2B, calls to Subroutine BOUND are initiated both before and after the accelerations are computed in ACCEL. If displacements are to be specified (ISWTCH=1), these should be specified before the node accelerations are computed. If velocities are to be specified (ISWTCH=2), these should be specified after the node accelerations are computed.

The specific routine included herein is one in which displacements are to be specified from data cards. This routine allows specification of the displacement history along a vertical line of nodes in the mesh. Horizontal and vertical displacement records are entered at several depths (up to 4) and displacements computed at the nodes between these depths by linear interpolation. Nodes above the first depth receive the same motion as the first record while nodes below the last depth receive the same motion as the last record.

Subroutine OGENER

This subroutine is used to generate the modal forces applied to the embedded structure. Again, any subroutine can be used to satisfy a given problem. The particular one included herein takes specific data read from cards at given times. These input times are converted to machine time by the parameter TSTART or

$$t = t_{input} + t_{start} \quad . \tag{73}$$

For the data specified, space has been allowed for up to twenty generalized forces (associated with 3 rigid body + the free-field modes) for twenty time periods. For times between these input times, linear interpolation is used, while for times greater than that associated with the last force record, the generalized forces are set to zero. More space can be easily supplied if more digitized force data are to be used as input.

The data transferred to the calling program (ACCEL) through the subroutine list are:

MXSEQS = maximum number of modal equations of motion for structure

NUMEQS = number of structural modal equations

T = machine time

QSA = generalized force vector associated with this machine time

NUMSTR = embedded structural type.

6.0 Description of LINK3

As discussed previously, the output for a selected set of node points and elements as well as the structural modal coordinates are stored on Logical 14 for potential use in LINK3 or for plotting purposes. If the parameter KSPEC in the MAIN program is set equal to 1, LINK3 is performed. In LINK3, the shock spectra for a selected set of free-field node points or structural boundary node points can be computed. The nodes specified for shock spectra calculations must be from among the nodes whose history has been stored on Logical 14 (originally specified as the output nodes in LINK2). Not all of the output nodes need have spectra computed for its generated motion.

Either or both of the horizontal and vertical shock spectra can be computed for the free-field nodes as well as the structural nodes. For each node point in the free-field, the acceleration history is taken from Logical 14, stored in core, and a call to subroutine SPECTA initiated to compute the shock spectra. For the structural boundary node points (if any), the motion history is computed from modal component histories. The shock spectra is computed as before.

Subroutine SPECTA

The subroutine computes the shock spectra for a given acceleration-time history supplied as input through the subroutine list. The frequency band for spectra computation has been set in the DATA statement from 1 cps to 1,000 cps. The shock spectra is computed from the solution of the simple linear oscillator with a specified damping subjected to the acceleration input.

7.0 Output From Code

As discussed previously, two types of output are generated, tape storage and printed storage. In subroutine FORM, three output parameters are read from cards, namely, IOTAPE, IOPAPE and IOSAVE.

IOTAPE indicates the number of integration steps between tape writes for the motion history onto Logical Tape 14. A selected amount of data generated at each time step, concerned with output elements, output nodes and structural motion is stored permanently onto this tape for later use (plot and/or analysis).

IOPAPE indicates the number of integration steps between paper printout of the same data. Usually, printout is not desired as often as the data is written onto the save tape.

The parameter IOSAVE is used to indicate when the entire problem parameters and motions are stored onto the restart tape (Logical Tape 8). This is usually done at the beginning of the problem and at the end of the run. However, if physical tapes are used in the run, breakdowns usually occur due to excessive tape wear during the run. In this case, the save tape should be written more often so that restart can begin at a later time in the run.

The output data that is stored on the tape is data associated with elements, nodes and the embedded structure. The elements for which output is desired are specified from cards in LIA by the parameters [NELOUT(I), I = 1, MELOUT]. The data associated with this are the following:

NELOUT	=	element number
STRESS (4)	=	stresses at the centroid of the element (σ_r, σ_θ, σ_z, τ)
STRMAX	=	maximum principal stress
STRMIN	=	minimum principal stress
ANGLE	=	angle of principal stresses from horizontal

The nodes for which output is desired are specified from cards in FORM by the parameters [NPOUT(I), I=1, NUMOUT]. The data associated with this are the following:

MOOLD	=	original node number
MONEW	=	new node number
UDISPL, WDISPL	=	horizontal and vertical displacements (inches)
UVEL, WVEL	=	horizontal and vertical velocities (ips)
UACCEL, WACCEL	=	horizontal and vertical accelerations (g's)
PU,PW	=	horizontal and vertical pressures (psi)

The structural data stored (and printed) are the following:

XS(I)	=	modal displacements
XSD(I)	=	modal velocities
XSDD(I)	=	modal accelerations
QSA(I)	=	modal loads applied to the structure
FRSTRC(J)	=	resisting forces developed at the attached nodes to the structure.

Each time the restart tape or the history tape is written, a message is printed out so that the user can monitor the latest state of the data storage.

NELOUT	=	element number
STRESS (4)	=	stresses at the centroid of the element (σ_r, σ_θ, σ_z, τ)
STRMAX	=	maximum principal stress
STRMIN	=	minimum principal stress
ANGLE	=	angle of principal stresses from horizontal

The nodes for which output is desired are specified from cards in FORM by the parameters [NPOUT(I), I=1, NUMOUT]. The data associated with this are the following:

MOOLD	=	original node number
MONEW	=	new node number
UDISPL, WDISPL	=	horizontal and vertical displacements (inches)
UVEL, WVEL	=	horizontal and vertical velocities (ips)
UACCEL, WACCEL	=	horizontal and vertical accelerations (g's)
PU,PW	=	horizontal and vertical pressures (psi)

The structural data stored (and printed) are the following:

XS(I)	=	modal displacements
XSD(I)	=	modal velocities
XSDD(I)	=	modal accelerations
QSA(I)	=	modal loads applied to the structure
FRSTRC(J)	=	resisting forces developed at the attached nodes to the structure.

Each time the restart tape or the history tape is written, a message is printed out so that the user can monitor the latest state of the data storage.

8.0 Summary

This report has attempted to describe in detail the analysis and program operation of SLAM Code. The program has been written to provide maximum flexibility to the user, making heavy usage of tape files for auxiliary storage. This additional flexibility has been obtained at a price of slower operation (additional I/O procedures). The Code can be easily modified, of course, to speed up the operation by eliminating much of the tape handling procedures. The user must decide, therefore, which is more important to solve his problems.

REFERENCES

1. C.J. Costantino, "Stress Waves in Layered Arbitrary Media", Report No. SAMSO TR 68-181, Space and Missile Systems Organization, Norton AFB, California, July 1968.

2. S.P. Chan et al, "Transient Analysis of Forced Vibrations of Complex Structural-Mechanical Systems", Journal Royal Aeronautical Society, Vol. 66, pp. 457-460, July 1962.

3. C.J. Costantino, "Automated Design of Advanced Hardened Facilities", Report No. BSD-TR-66-256, Space and Missile Systems Organizations, Norton AFB, California, September 1966.

4. I. Nelson, M.L. Baron, "Investigation of Ground Shock Effects in Nonlinear Hysteretic Media", Paul Weidlinger, Consulting Engineer for Waterways Experiment Station, Contract Report S-68-1, March 1968.

5. C.J. Costantino, "Finite Element Approach to Stress Wave Problems", Journal, Engineering Mechanics Division, ASCE, Vol. 93, p. 153, April 1967.

6. C.J. Costantino, "Two-Dimensional Wave Propagation Through Nonlinear Media", Journal of Computational Physics, Vol. 4, p 147, August 1969.

7. J. Isenberg, "Interaction Between Soil and Nuclear Reactor Foundations During Earthquakes", Agbabian-Jacobsen Associates, Report No. ORO-3822-5, June 1970.

8. C.A. Miller, C.J. Costantino, "Structure-Foundation Interaction of a Nuclear Power Plant with a Seismic Disturbance", Nuclear Engineering and Design, Vol. 14, December 1970.

9. S. Glasstone, "The Effects of Nuclear Weapons", U.S. Department of Defense, April 1962.

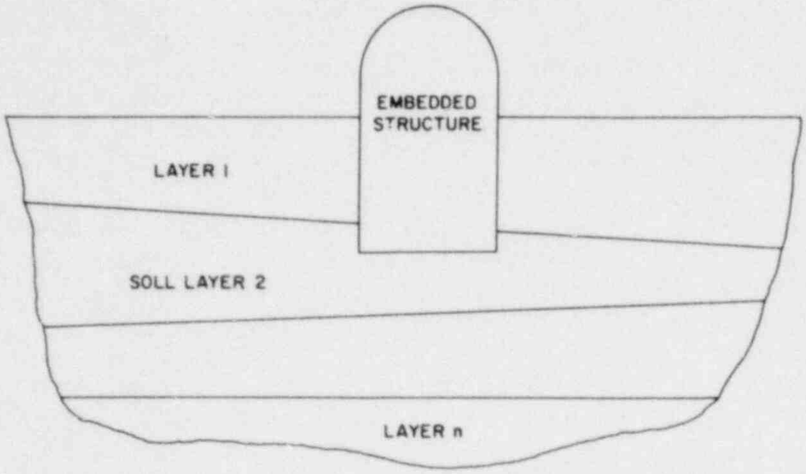


Figure 1. General Configuration

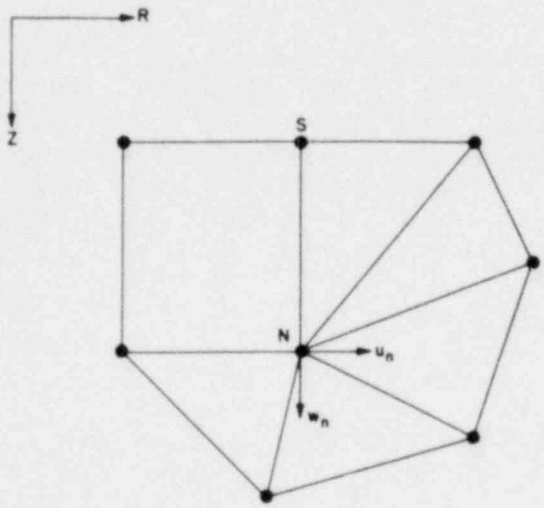


Figure 2. Typical Int rior Node, N, and Surrounding Nodes,S

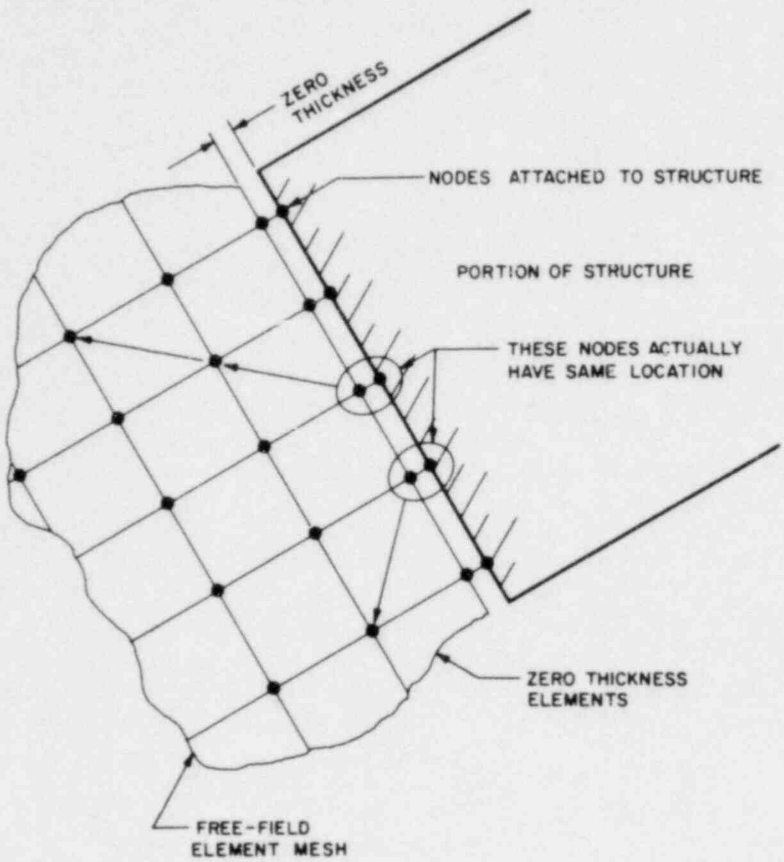


Figure 3. Example of Crack Elements

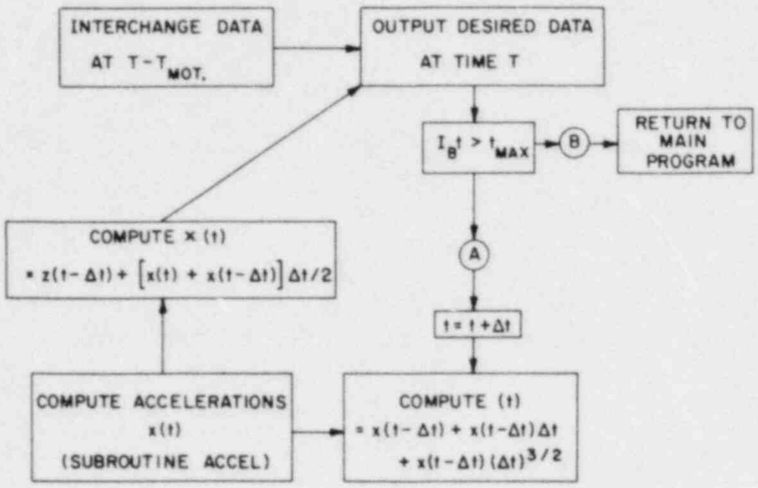


Figure 4. Basic Flow Diagram of LK2B Subroutine

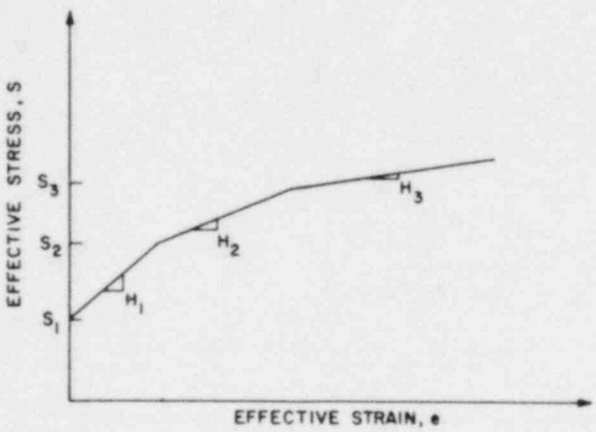


Figure 5. Mises Hardening Relationship

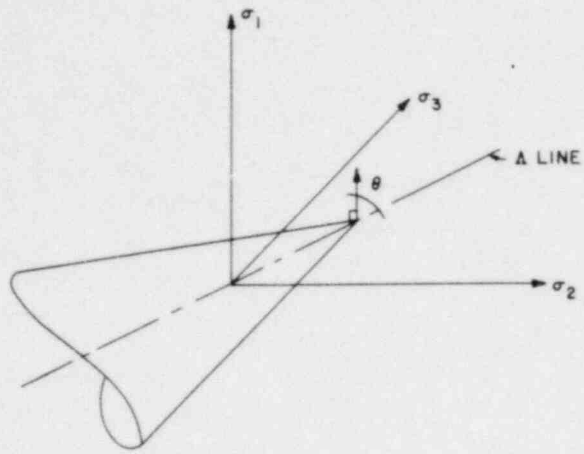


Figure 6. Coulomb-Mohr Yield Criteria

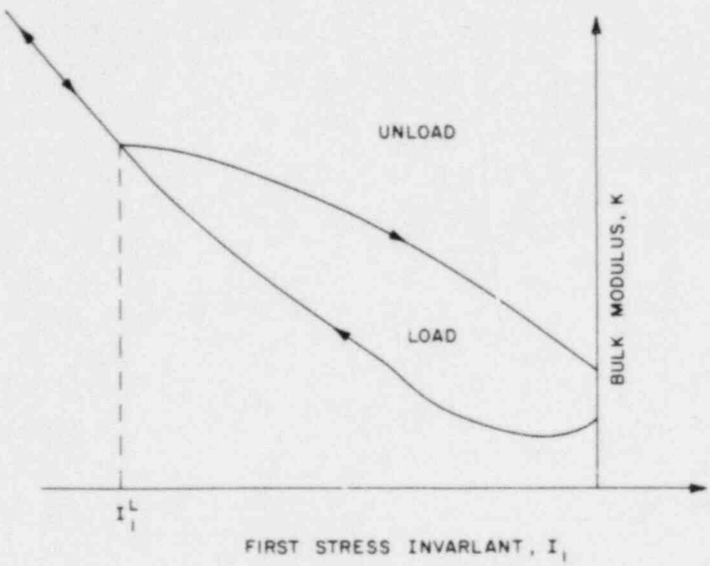


Figure 7. Yield Criteria for Compct Material

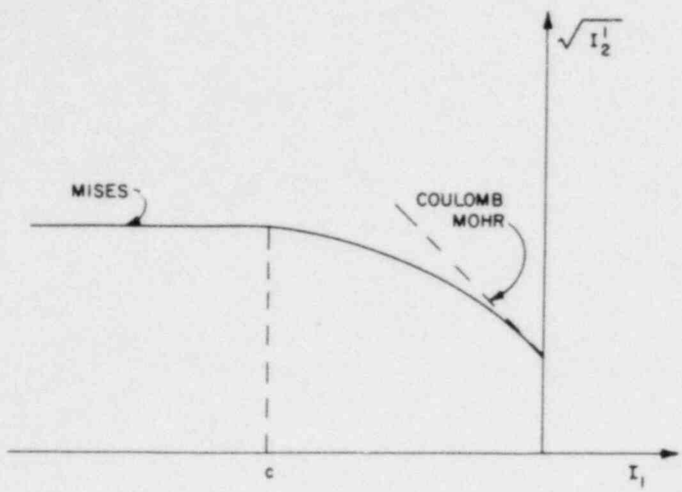


Figure 8. Bulk Modulus Relation - Case: A

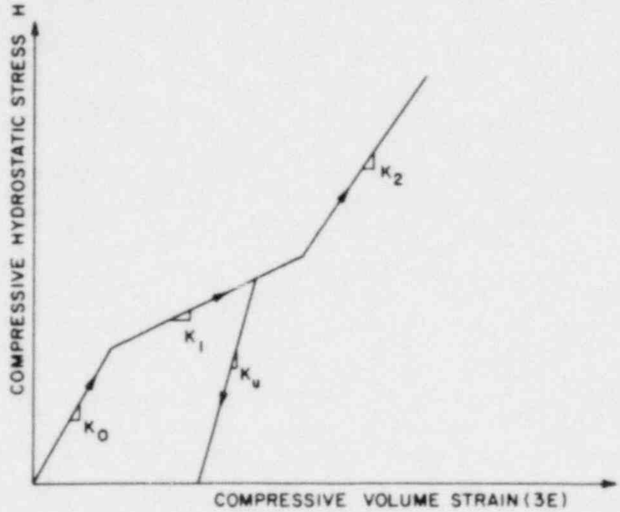


Figure 9. Bulk Modulus Relation - Case: B

APPENDIX A

CONTROL CARDS

APPENDIX A

CONTROL CARDS

The following sets of control cards are used for a SLAM run:

1. Job Card: t, STMFZ, Ta, Pb.

   t = title
   a = running time in octal seconds
   b = job priority 1 to 3 with 1 lowest

2. Account Card: Account (a,b)

   a = ID
   b = Account Number

3. Tape Request for Restart (only needed if tapes are to be saved):

   REQUEST, TAPE 8, *PF.
   REQUEST, TAPE 14, *PF.
   REQUEST, TAPE 20, *PF.

4. SLAM Program: FILE, TAPE, RT=W, BT=J, MBL=5120
   STAGE, TAPE, NT, PE, E, VJN=K2690

   OR

   ATTACH (OLDPL, SLAM1, ID = ZZGCAM)

5. Calcomp Tapes

   ATTACH (LIB1, FR80LIB)
   ATTACH, LIB2, CALCOMPLIB.
   LIBRARY (LIB1, LIB2)

6. ID cards for Microfiche output (only if microfiche requested)

   ID CARD (TAPE 99, a, T129, $ b$, PLFI)
   ID CARD (MIKE, $ B$, a, T129, PRFI)

   a = Name
   b = Title

7. UPDATE (F)

8. FTN (I= COMPILE, L=0)

9. LDSET (PRESET = ZERO)

10. LGO (PL = a)

    a = number of lines of output

11. Save Tapes for Restart (only if required):

```
EXIT (U)
CATALOG (TAPE 8, SAVE 8, ID = a, RP = 10)
CATALOG (TAPE 14, SAVE 14, ID = a, RP = 10)
CATALOG (TAPE 20, SAVE 20, ID = a, RP = 10)
```

Note: If the problem is a Restart, then the following cards are required instead of the 3 cards above.

```
ATTACH (TAPE 8, SAVE 8, ID = a)
ATTACH (TAPE 14, SAVE 14, ID = a)
ATTACH (TAPE 20, SAVE 20, ID = a)
```

12. Copy output onto microfiche (only if microfiche requested)

```
REWIND (OUTPUT)
COPY (PUTPUT, MIKE)
```

13. EOR

14. EOR

15. SLAM Data

16. EOF

APPENDIX B

Data Deck Input

APPENDIX B

Data Deck Input

1. Deck Setup

The Data Deck is read from cards in specific subroutines. Generally, five card columns are used for integer numbers and ten card columns are used for floating point numbers. The restart condition is determined from the first card by the Parameter KRUN. If KRUN = 0, an initial run is performed and all data cards are required. If KRUN = 2, only the second and third links (integration and spectra calculations) are performed. For this condition, however, previously calculated data from an initial run must be available from tape, namely, the Restart Tape (Logical Tape 8), and the Stiffness Save Tape (Logical Tape 20) which are required as input to the program. The calculations are then continued from the last time on the restart tape (last time from previous run) to a new final time TMAX. The generated history tape (Logical Tape 14) which contains the motion data for the output nodes, the stress data for the output elements and the structural mode histories, can either be placed on a new save tape or the new generated history data can be added to the previous history tape generated from the initial problem run. A data generator is available to generate much of the following data. This is described in Appendix D.

For an initial run (KRUN = 0), the data deck input format is composed of the following clusters of data:

Data Tape	Subroutine
Restart Condition	MAIN Program
Mesh Data	L1A
Pressure Surface Data	L1G

Data Type	Subroutine
Run Time Data	LNK2
Output Node Data	FORM
Pressure Data	PRESS
Output Tape Data	OUTPUT
Boundary Input Motion Data	BOUND
Quiet Boundary Data	QUIET
Shock Spectra Data	LNK3

For a restart condition (KRUN = 2), the data deck format is composed of the following clusters of data:

Data Type	Subroutine
Restart Condition	MAIN Program
Run Time Data	LNK2
Output Node Data	FORM
Boundary Input Motion Data	BOUND
Pressure Data	PRESS
Quiet Boundary Data	QUIET
Output Tape Data	OUTPUT
Shock Spectra Data	LNK3

2. Restart Condition Data

This data is read in the MAIN Program. It consists of the parameters (the card format for this and all other cards is shown in parentheses): KRUN, KSPEC, ANAME (2I5,11A6) If KRUN = 0, initial run; if KRUN = 2, restart condition and Logicals 8, 20 (and possibly 14) are required with card data. If KSPEC = 0, the shock spectra calculations of LK3 are omitted, While if KSPEC = 1, shock spectra are computed. If KPLOT = 0, no CALCOMP plots are generated; if KPLOT = 1, CALCOMP plots augmented. ANAME is a program title name.

3. Mesh Data

The Mesh Data is read in Subroutine L1A. It consists of node data, element data, material property data and output element data.

Card Group	Variables	Format
1.0	ANAME (Mesh Title)	(12A6)
	All these variable designations allow the use of alphanumeric characters to aid in data output interpretation.	
1.1	NUMNP, NUMEL, ISTRES, IPRINT	(4I5)
	NUMNP - Number of Node Points ($\leq$ 1000)	
	NUMEL - Number of Elements	
	ISTRES - Stress Condition, = 0 axisymmetric problem = 1 plane strain problem = 2 plane stress problem	
	IPRINT - Intermediate printout of tables in L1A; if = 0, no printout	
2.0	ANAME (Node Point Data)	(12A6)
2.1	N, R(N), Z(N), ITYPE(N), THETA(N)	(I5, 2E10.0, I10, E10.0)
	(Card 2.1 is repeated NUMNP times)	
	N - Node Point Number	
	R - Radial coordinate (ft), positive to right	
	Z - Depth coordinate (ft), positive downward	

Card Group	Variables	Format
	ITYPE - Restraint Condition of Node, = 0 unrestrained = 1 roller support = 2 fixed node	
	THETA - If ITYPE=1, THETA is the angle (in degrees) of the roller support measured from the horizontal, positive clockwise	
3.0	ANAME (Zone or Material Property Data)	(12A6)
3.1	NZONES	(I5)
	NZONES = Number of Materials (or Zones) in Problem ($\leq$ 5) (Remainder of Card Group 3 is concerned with data for each zone and is therefore repeated NZONES times)	
3.2	IZ, ANAME	(I5, 12A6)
	IZ = Zone Number of Material for which Material Property Data is to follow	
	ANAME = Material Zone Name	
3.3	IELAST, IPLAST, WGT, E1, ..., E5	(2I5, 6E10.0)
	IELAST - Specifies type of elastic stress-strain relation of material. If IELAST = 1 isotropic = 2 anisotropic = 3 linear compressible fluid	
	IPLAST - Specifies type of nonlinearity in material stress-strain law, = 0 linear material = 1 von Mises material = 2 Coulomb-Mohr material = 3 compacting material = 4 crack material. If IPLAST$\neq$0, IELAST must equal 1 since only isotropic plastic behavior has been included.	

Card Group	Variables	Format
	WGT - Unit Weight of Material (pcf)	

E1, E2, E3, E4, E5 -

(a) If IELAST = 1,

E1 is Young's Modulus (psi)
E2 is Poisson's Ratio, and
E3 to E5 are omitted.

(b) If IELAST=2, these are coefficients of anisotropic material described in Ref. 1, Vol. 1, p. [illegible]6.

(E1=a, E2=$\bar{a}$, E3=b, E4=μ, E5=$\bar{\mu}$).

(c) If IELAST=3, E1 is the fluid bulk modulus and E2 to E5 are omitted.

(If IPLAST=0, material is elastic and remainder of zone data is omitted).

Card Group	Variables	Format
3.4.1	NOYILD	(I5)
3.4.2	(SSTAR(J), J=1, NOYILD)	(7E10.0)
3.4.3	(HSTAR(J), J=1, NOYILD)	(7E10.0)

NOYILD - Number of linear segments in the material stress-strain compression data (< 10) not counting the initial linear segment.

SSTAR - Stress at the beginning of the linear segment (psi)

HSTAR - Slope of the linear segment (psi)

(If IPLAST≠1, Card Group 3.4 is omitted).

Card Group	Variables	Format
3.5.1	COHESN, FRCTAN	(2E10.0)

COHESN - Soil Cohesion (psi) associated with Mohr failure envelope.

Card Group	Variables	Format
	FRCTAN - Friction Angle in Degrees	
	(Note: These parameters correspond to the usual cohesion and friction angle obtained from triaxial test results on soil/rock samples).	
	(If IPLAST≠2, Card Group 3.5.1 is omitted).	
3.6.1	LINEAR, IASSOC, AGO, ALPHA, CAPPA, CONSTC	(2I5, 4E10.0)
	LINEAR - Compacting Media Counter =0 variable bulk modulus material =1 trilinear bulk modulus material	
	IASSOC - Flow Rule Counter =0 nonassociated flow rule =1 associated flow rule	
	AGO - Shear Modulus (psi)	
	ALPHA - Yield Coefficient α	
	CAPPA - Yield Coefficient k	
	CONSTC - Yield Coefficient c	
3.6.2	AKO, AK1, AK2, GAMMA, AJ1LIQ	(6E10.0)
	AKO - Bulk Modulus Coefficient K_o for Variable Bulk Modulus Media	
	AK1 - Bulk Modulus Coefficient K_1 for Variable Bulk Modulus Media	
	AK2 - Bulk Modulus Coefficient K_2 for Variable Bulk Modulus Media	
	GAMMA - Bulk Modulus Coefficient γ for Variable Bulk Modulus Media	
	AJ1LIQ - Bulk Modulus Coefficient J_1^L for Variable Bulk Modulus Media	
	(Note: Card 3.6.2 omitted if LINEAR=1).	

Card Group	Variables	Format
3.6.3*	AKO, AK1, AK2, AKU, AJ10, AJ11	(6E10.0)
	AKO - Initial Bulk Modulus K_o for Trilinear Media	
	AK1 - Second Bulk Modulus K_1 for Trilinear Media	
	AK2 - Compacting Bulk Modulus K_2 for Trilinear Media	
	AKU - Unloading Bulk Modulus K_u for Trilinear Media	
	AJ10 - Hydrostatic Pressure at Which Second Bulk Modulus Applies.	
	AJ11 - Hydrostatic Pressure at Which Compacting Bulk Modulus Applies.	
	(Note: Card 3.6.3 omitted if LINEAR=0. Card Group 3.6 omitted if IPLAST≠3).	
3.7.1	COHESN, FRCTAN	(2E10.0)
	COHESN - Cohesion (psi) for Cracked Element for Simple Coulomb-Mohr Slip Model	
	FRCTAN - Friction Angle (degrees)	
	(Card 3.7.1 omitted if IPLAST≠4).	
4.0	ANAME (Output Element Data)	(12A6)
4.1	MELOUT	(I5)
	MELOUT - Number of Elements for Which Stress Output is desired (≤100)	
4.2	(NELOUT(J), J=1, MELOUT)	(14I5)
	NELOUT - Element Numbers of Output Elements	
	(Card 4.2 omitted if MELOUT=0).	
5.0	ANAME (Element Data)	(12A6)
5.1	NUME, IZONE, NPI, NPJ, NPK, NPL, NCRACK	(7I5)
	(Card 5.1 repeated NUMEL times)	

* The data for the compacting model is entered with hydrostatic compressive stresses taken as positive numbers. See Figures 8 and 9.

POOR ORIGINAL

Card Group	Variables	Format
	NUME - Element Number	
	IZONE - Zone of Material in Which Element Occurs	
	NPI, NPJ, NPK, NPL - Node Point Numbers of the Nodes of the Element. If triangular element, NPL=0.	
	NCRACK - If 0 regular element if 1 crack model. Node numbers must be in clockwise order for crack model.	
6.0	ANAME (Starting Node Data)	(12A6)
6.1	NUMST	(I5)
	NUMST - Number of Starting Nodes for Renumbering Algorithm	
6.2	[NSTART(I), I=1, NUMST]	(14I5)
	NSTART - Node Numbers for Start Nodes	

4. Pressure Surface Data

These data are read in Subroutine LIG. They describe the node points to which pressures will be applied (if any).

Card Group	Variables	Format
1.0	ANAME (Loaded Node Point Data)	(12A6)
1.1	LINES	(I5)
1.2	LOADNP	(I5)
1.3	(NPLOAD(I), I=1, LOADNP)	(14I5)
	LINES - Number of Loaded Surfaces (up to two allowed)	
	LOADNP - Number of Nodes on the Loaded Surface ($\leq$ 100)	

Card Group	Variables	Format
	NPLOAD - Node Point Numbers of the Loaded Surface. Entered in consecutive order starting with the first loaded node point and moving in the direction on the surface such that the outer normal is on the left.	
	(Cards 1.2 and 1.3 repeated LINES times) (Cards 1.2 and 1.3 omitted if LINES=0)	

5. Initial Stress Data

The Initial Stress Data is made in LK1J. Initial stresses are read for NSTRSS elements in the following forms.

Card Group	Variables	Format
1.0	ANAME (Initial Stress)	(12A)
2.0	NSTRSS	(I5)
3.0	These cards are repeated NSTRSS times	
	NEL, (SIGINLJ), J=1,4)	(I5,6E10.0)
	(EPSTINLJ), J=1,4)	(5X,6E10.0)
	(EPSPINLJ), J=1,4), EFFIN	(5X,6E10.0)

NEL = Element number

SIGIN = INITIM STRESS VECTOR

EPSTIN = Initial total strain vector

EPSPIN = Initial plastic strain vector

EFFIN = Initial effective plastic strain

6. Structural Data

The Structural Data is read in LK2A. Special structural types have been included and are specified by the counter NUMSTR. If NUMSTR = 0, no embedded structure occurs in the problem and only a free-field wave problem is investigated. If NUMSTR = 1, the embedded structure is considered to be a rigid body with three degrees of freedom (two translation and a rotation). If NUMSTR = 2, the structure is a general flexible circular tunnel lining including shell bending according to the inextensional bending theory. If NUMSTR = 3, the same circular tunnel model is treated but only horizontal, e.g., motion is included. If NUMSTR = 4, a general structural model is treated, with general modal data used as input.

Card Group	Variables	Format
1.0	ANAME (Structural Data)	(12A6)
1.1	NUMSTR	(I5)
	(If NUMSTR=0, remainder of structural data is omitted.)	
1.2	NMSTNP	(I5)
	NMSTNP - Number of Nodes Attached to Structure (≤ 50).	
1.3	(NPSTRC(I), I=1, NMSTNP)	(14I5)
1.4	RCG, ZCG	(2E10.0)
	RCG, ZCG - Coordinates of c.g. of Structure (ft)	
1.5	ANAME (Structural Type Name)	(12A6)
2.0	WEIGHT, ROTARY	(2E10.0)
	WEIGHT - Weight of Rigid Structure (lbs)	
	ROTARY - Rotary Weight (lb-ft^2)	
	(Card 2.0 omitted if NUMSTR$\neq$1)	
3.0	RADIUS, THICK, PCF, EMOD, XNU NBMODE	(5E10.0,I5)
	RADIUS - Radius of Cylinder (ft)	
	THICK - Thickness of Liner (in)	
	PCF - Unit Weight of Liner Material (pcf)	
	EMOD - Elastic Modulus of Liner Material	
	XNU - Poisson's Ratio	

Card Group	Variables	Format
	NBMODE - Number of Bending Modes (≤ 8)	
	(Card 3.0 omitted if NUMSTR not equal to 2 or 3)	
4.0	NSMODE, NSDOF	(2I5)
	NSMODE - Number of free-free modes of structure (≤ 20 including rigid body modes).	
	NSDOF - Number of structural degrees of freedom for each mode vector for structure (≤ 150).	
4.1	NHOR(I), NVER(I)	(2I5)
	NHOR - Location in Mode Vector of Horizontal Degree of Freedom Associated with Attached Node I	
	NVER - Location in Mode Vector of Vertical Degree of Freedom	
	(Card 4.1 is repeated NMSTNP times. Input order must be the same as input order of NPSTR of Card 1.3.)	
4.2	FREQ, GMASS	(2E12.5)
	FREQ - Frequency of Free-free Mode (cps). Zero for rigid body modes.	
	GMASS - Generalized Mass Associated with Mode	
4.3	(AMODE(I), I=1, NSDOF)	(7E11.4)
	AMODE - Mode Vector for Particular Mode	
	(Cards 4.2 and 4.3 repeated NSMODE times)	

7. Run Time Data

1.0	TMAX, DT, KDT, KINT, KTAPE	(2E10.0, EI5)
	TMAX - Maximum or Final Time (sec) of integration	
	DT - Time Increment (sec) of integration	
	KDT = 0, use DT as read = 1, choose DT from period	

Card Group	Variables	Format
	KINT - Interval for choosing DT. (usually 20)	
	KTAPE = 0, uses two tapes for stiffness matrix = 1, use one tape	
8. Output Node Data		
1.0	ANAME (output Node Data)	(12A6)
1.1	NUMOUT, IOTAPE, IOPAPE	(3I5)
	NUMOUT - Number of Node Points for which output history is desired (< 100)	
	IOTAPE - Number of Intervals between Output Writes onto Logical 14, the Output History Tape (Note that the time increment at which records are written on 14 determines the maximum frequency attainable in response spectra)	
	IOPAPE - Number of Intervals between Output Printing	
1.2	[NOOLD(I), I=1, NUMOUT]	(14I5)
	NOOLD - Output Node Numbers (Note that spectra may only be determined for these nodes)	
	(Card 1.2 omitted if NUMOUT = 0.)	
1.3	ANAME (Save Tape Data Title)	(12A6)
1.4	IOSAVE	(I5)
	IOSAVE - Number of Intervals between Save Tape (Logical 8) Writes.	
9. Quiet Node Data		
1.0	ANAME (Quiet Boundary Data)	(18A4)
1.1	NRIGHT, NBOT, NLEFT, ICORNER	(14F5)
	NRIGHT - Number of Quiet Nodes in Right Boundary	
	NBOT - Number of Quiet Nodes in Bottom Boundary	
	NLEFT - Number of Quiet Nodes in Left Boundary	

Card Group	Variables	Format
	ICORNER - Left Corner Condition = 0, Regular Bottom Boundary = 1, Quiet Vertically only	
1.2	NNODE(I), I=1, NRIGHT	(14I5)
	NNODE(I) - Node Number in Bottom	
1.3	NNODE(I), I=1, NLEFT	(14I5)
	NNODE(I) - Node Numbers in Left Boundary	
2.0	NUMB, IZONE, NPI, NPJ, NPK, NPL, NL, DR, DZ, repeat NUMEL times	(7I5,2E10.0)
	NUMEL - Number of Rectangular Elements on Boundary	
	NUMB - Element Number	
	IZONE - Material Zone	
	NPI, NPI, NPK, NDL - Corner Nodes of Element	
	NL =0, Regular Element =1, Cracked Element	
	DR - Width of Element (ft)	
	DZ - Depth of Element (ft)	
10. Boundary Data		
1.0	ANAME (Boundary Data)	(12A6)
1.1	NNODES, IOISPL, ITAPE, ICOMP, IDIM	(14I5)
	NNODES - Number of Boundary Nodes	
	IOISPL =0, both Horizontal and Vertical Record =1, Horizontal Record only =2, Vertical Record only	
	ITAPE =0, Input off Cards =1, Input off Tape 18	

Card Group	Variables	Format
	ICOMP =1, Displacements =2, Accelerations	
	IDIM =1, Units (in) =2, Units (ft) =3, Units	
1.2	(NODES(I), I=1, NNODES)	(14I5)
	NODES(I) - Node Numbers	
	(Orbit Set 2.0, 2.1, and 2.2 if ITAPE = 1	
2.0	NRCDS, TSTART, DT	(I5,2E10.0)
	NRCDS - Number of records	
	TSTART - Starting Time of Problem with Respect to Record (sec)	
	DT - Time Increment of Record (sec)	
2.1	(UDISPL(I), I=1, NRCDS)	(8E9.0)
	Omit if IOISPL=2 = Horizontal Record	
2.2	(WDISPLLJ), I=1, NRCDS)	(8E9.0)
	omit if IDIJPL=1	
	WDISPL(I) = Vertical Record	

11. Pressure Data

The Pressure Data are read in Subroutine PRESS which computes the pressure applied to the loaded surfaces specified in Subroutine LIG. If the number of LINES is zero, these data are omitted. Any routine can be used as desired. The specific routine used is described in Section V of this report.

Card Group	Variables	Format
1.0	ANAME	(12A6)
1.1	YIELD, PO, TPHASE	(3E10.0)
	YIELD - Weapon Yield in KT	
	PO - Peak Overpressure in psi	
	TPHASE - Arrival time (sec) of pulse at left boundary of mesh	
	(Cards 1.0 and 1.1 are omitted if LINES=0).	

12. Generalized Force Data

The routine is used to specify the applied generalized forces to the embedded structure (if any). As discussed in Section V, the specific routine is included herein as an example of its construction.

Card Group	Variables	Format
1.0	ANAME	(12A6)
1.1	NRCDS, TSTART	(I5, E10.0)
2.0	TIME(I)	(F6.3)
2.1	[QFORCE(I,J), J=1, NUMEQS]	(5E12.5)
	NRCDS - Number of Time Records for Generalized Force Data ($\leq$ 20)	
	TSTART - Starting Time (sec) which when added to TIME (I) converts the record time to machine or problem time	
	TIME - Time at which Generalized Forces are specified	
	QFORCE - Generalized Force Vector	
	(Cards 2.0 and 2.1 are repeated NRCDS times) (Cards 2.0 and 2.1 are omitted if NRCDS=0).	

13. Output Tape Data

These data are read in subroutine OUTPUT and concern the record formation on the Output History Tape (Logical 14).

Card Group	Variables	Format
1.0	ANAME (Output Tape Data)	(12A6)
1.1	JRCDS	(I5)
	JRCDS - Number of Time Histories Already on the Output History Tape from a Previous Run. New Records will be placed after the last record.	

14. Shock Spectra Data

These data are read in Subroutine LK3. All of these data are omitted if KSPEC (MAIN Program) is set to zero.

Card Group	Variables	Format
1.0	ANAME (Shock Spectra Data)	(12A6)
1.1	NNODES, MNODES, LDOFS, NBETA	(4I5)
	NNODES - Number of Free-field Nodes ($\leq$ 100) for which Spectra is desired.	
	MNODES - Number of Interaction Nodes ($\leq$ 50) for which Spectra is desired.	
	LDOFS - Number of Structural DOFs for Spectra ($\leq$ 50)	
	NBETA - Number of Damping Values for Spectra ($\leq$ 10)	
	ISPEC - =0; Both Horizontal and Vertical Spectra Computed =1; Only Horizontal Spectra Computed =2; Only Vertical Spectra Computed	
	IPLTOL - =0; No On-Line Spectra Plots =1; Spectra to be Plotted on Line	
	IPLTCC - = 1, Calcomp Spectra Plots	

2.0	[NODES(I), I=1, NNODES]
	NODES - Free-field Node Numbers
	(Card 2.0 omitted if NNODES=0).
3.0	[NSTRC(I), I=1, MNODES]
	NSTRC - Interaction Node Numbers
	(Card 3.0 omitted if MNODES=0).
4.0	[NDOF(I), I=1, LDOFS]
	NDOF - Structural Node Numbers
5.0	[XBETA(I), I=1, NBETA]
	XBETA - Damping Values, %.

14. Integration Interval

As mentioned in paragraph 6.0, the integration interval should be chosen such that DT is about 1/20 of the shortest period of the system, which is printed in LINK1. The shortest period is, of course, a function of the elastic modulus of the material as well as the sizes of the elements connecting the nodes. The smaller the element size (distance between nodes), the higher the stiffness, and the higher the elastic modulus, the higher the stiffness. The shortest period then corresponds to those nodes which have the highest stiffness.

For yielding materials (Mises or Coulomb-Mohr), the elastic stiffness is the controlling factor on the integration interval. For the compacting media, however, the unloading modulus may be significantly higher than the initial modulus. In this case, then, the time interval must be suitably decreased.

APPENDIX C

CRACK MODEL

APPENDIX C

Crack Model

The model used to treat the crack discontinuity problem is developed from that of the general rectangular element. A typical rectangular element is shown in Fig. C-1 which has side dimensions a and b. The analysis for this typical element begins with the definition of the stress-strain relation or:

$$\{\sigma\} = [C](\{\varepsilon^T\} - \{\varepsilon^N\}) \qquad (1)$$

where $\{\sigma\}$ is the element stress vector, $\{\varepsilon^T\}$ is the total strain vector and $\{\varepsilon^N\}$ is the nonlinear or correction strain vector. The matrix [C] is the usual elastic stress-strain relation defined in References 1 and 6.

The total strains in the element are related to the node point displacements by the relation

$$\{\varepsilon^T\} = [B]\{x\} \qquad (2)$$

Included in the matrix [B] is the assumption for the displacement variation over the element. Again, the details of this formulation are presented in References 1 and 6.

Applying a virtual displacement to the element node points, the virtual work can be determined from:

$$\delta W_i = \int_V \{\delta\varepsilon\}^T\{\sigma\}dV \qquad (3)$$

where the superscript T indicates the transpose of the vector. Substituting Equations (1) and (2) into (3), the virtual internal work is:

$$\delta W_i = \{\delta x\}^T[k]\{x\} - \{\delta x\}^T\{F^N\} \tag{4}$$

where the matrix [k] is defined by:

$$[k] = \int_V [B]^T[C][B]dx \tag{5}$$

and the correction vector $\{F^N\}$ by:

$$\{F^N\} = \int_V [B]^T[C]\{\varepsilon^N\}dV \tag{6}$$

Both integrals in Equations (5) and (6) are taken over the volume of the element.

The external work done by a set of equivalent node point forces during this virtual displacement is simply:

$$\delta W_e = \{\delta x\}^T\{F\} \tag{7}$$

Equating the internal to the external virtual work, the equilibrium equations for the element are then:

$$\{F\} = [k]\{x\} - \{F^N\} \tag{8}$$

The procedure for the zero thickness element begins by first defining an equivalent element strain vector defined by:

$$\begin{aligned} \{\bar{\varepsilon}^T\} &= b\{\varepsilon^T\} \\ \{\bar{\varepsilon}^N\} &= b\{\varepsilon^N\} \end{aligned} \tag{9}$$

where b is the thickness of the rectangular element. The strain-displacement relation becomes (from Equation 2)

$$\{\bar{\varepsilon}^T\} = [\bar{B}]\{x\} \qquad (10)$$

where:

$$[\bar{B}] = b[B] \qquad (11)$$

Similarly, the stress-strain relation becomes [from Equation (1)]

$$\{\sigma\} = [\bar{C}](\{\bar{\varepsilon}^T\} - \{\bar{\varepsilon}^N\}) \qquad (12)$$

where:

$$[\bar{C}] = (1/b)[C] \qquad (13)$$

is the equivalent stress-strain matrix. It may be noted that the stress-strain relation [Equation (13)] is now a function of the element thickness.

If this modified stress-strain relation is applied to the stiffness and correction force terms of Equations (5) and (6) and the limit of these terms taken as the element thickness, b, goes to zero (except that the stress-strain matrix of [Equation (13)] remains finite), both the stiffness matrix and correction force vector for this zero thickness element can be obtained. The coefficients of the stress-strain matrix, however, must still be chosen, although these terms are apparently arbitrary.

If the material of the "crack" is considered to be the same as that in the zones about the crack, the only item that must be determined is the magnitude of the coefficients since their magnitude relative to each other is determined. In a particular problem, this, in general, is unimportant since it only affects the relative displacements of the nodes on both sides of the crack.

If compressive stresses are tranferred through the element, actual computations indicate that Nodes i and l, and Nodes j and k, move normal to each other, (nodes may overlap), although the actual differences are small and unimportant. The actual size of these differences are controlled by the magnitude of the coefficients of the [C] matrix. For example, in a typical problem, if the coefficients of the [C] matrix are taken to be the same as those of the original [C] matrix for the surrounding materials, the differences during compression of the element occur in the third significant figure, clearly an insignificant amount.

During tension and/or shear of the element, the "material" of the crack can be made to follow any of the nonlinear laws of the material catalogue which limit the amount of tension and/or shear that may be transferred. In addition, another simple model that limits these values (Subroutine NCOUL) assumes that a simple functional model controls the stress transfer.

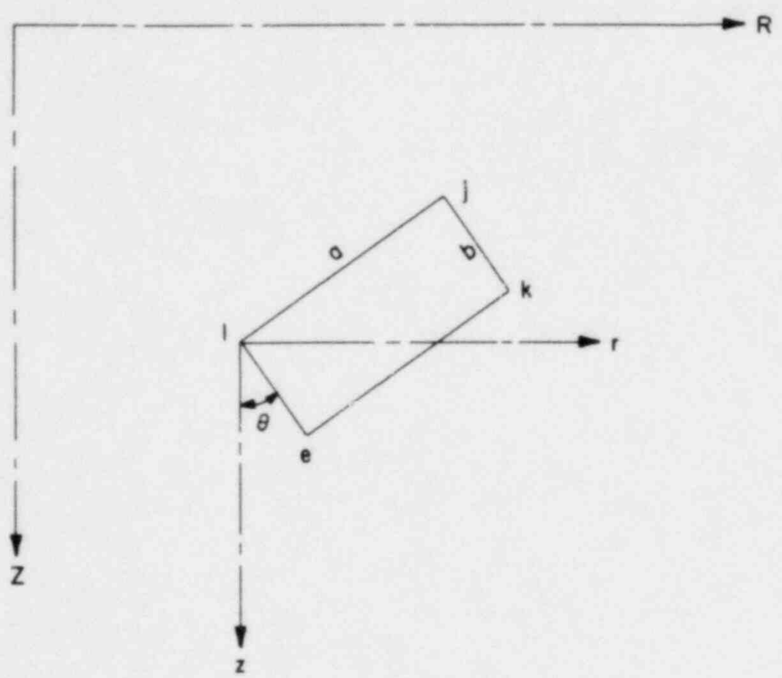


Figure C-1. Typical Rectangular Element

APPENDIX D

SLAM DATA GENERATOR

APPENDIX D

SLAM DATA GENERATOR

1.0 Introduction

The major portion of the input required for SLAM Code concerns the mesh (node and element data) description. These data follow very similar patterns however for most seismic analysis problems. A preprocessor program was written to generate this data.

2.0 Description of Problem

The SLAM data generator is written to generate the data for the configuration shown in Figure D-1. Only one-half of the problem needs to be considered since a vertical axis through the facility is an axis of asymmetry.

The user specifies the overall geometry of the problem with the parameters: H, W, H_s, W_s, the number of soil layers, and the depth to the top of each layer. The details of the mesh are specified in terms of the total number of elements wanted horizontally and the number of element vertically within each of the soil layers.

A mesh is then generated satisfying these requirements. The left boundary is taken as horizontal rollers to satisfy the asymmetric boundary conditions; quiet boundaries are placed along the right boundary; and the bottom boundary is restrained vertically and set to receive the horizontal accelerogram. The node and element numbers start at the upper left and are numbered down one column, to the top of the next column, then down that column, etc. The following card groups required for SLAM Code are generated:

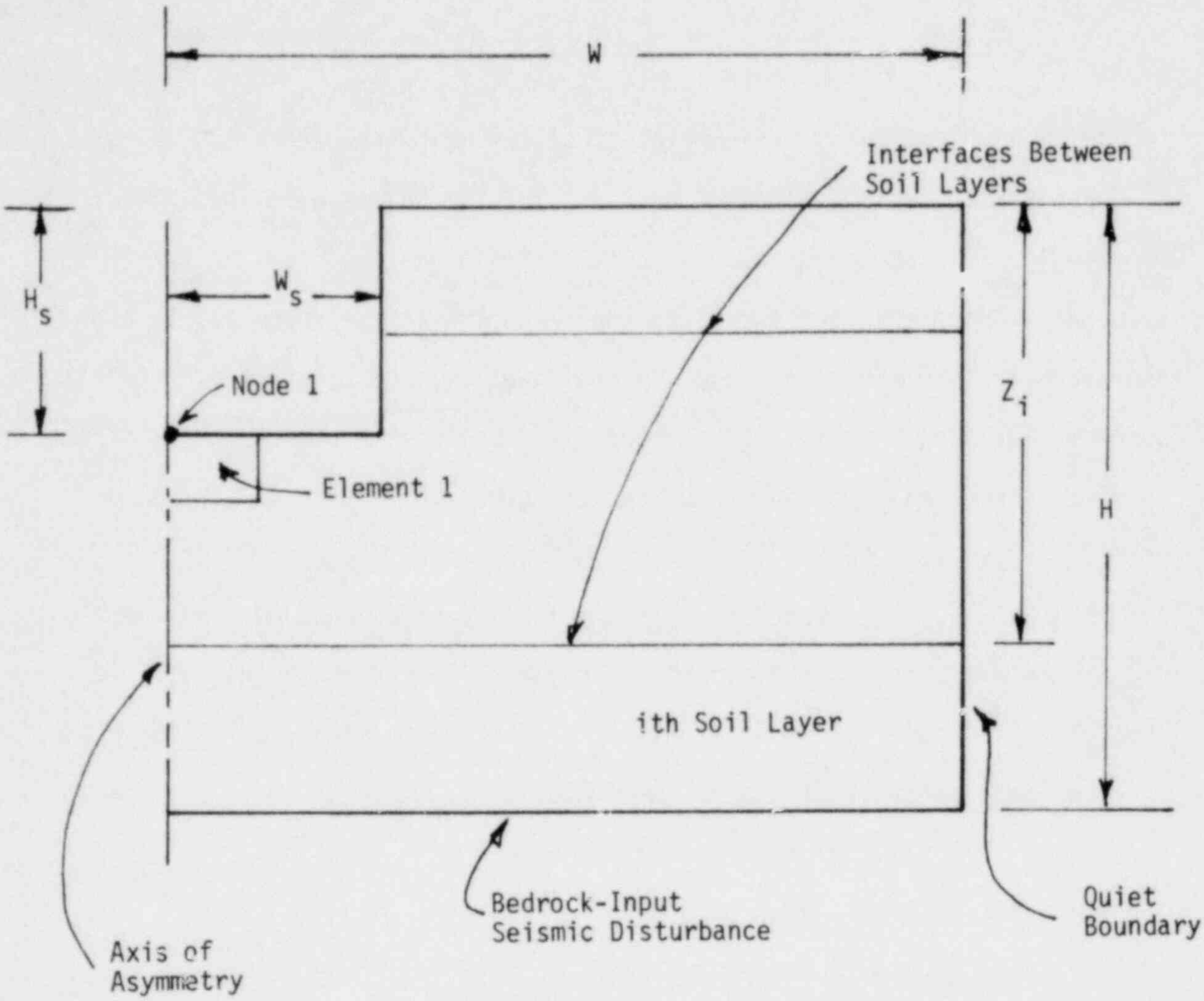


Figure D-1. Configuration for Data Generation

- Control Card
- Mesh Data Groups 1.0, 2.0, 5.0, 6.0
- Pressure Surface Data
- Initial Stress Data
- Quiet Node Data
- Output Tape Data

In addition to the cards Calcomp plots of the mesh are generated.

The structure is defined using the same material as used in SIM Code (Volume 3). The user must also specify: the foundation nodes on the structure (these are used to attach the structure to the free field); and the datum for the elevations used in the structural description (so that its correct elevation relative to the free field may be calculated). All of the required structural data for SLAM Code are generated.

3.0 Interface with SLAM

The SLAM data generated are printed and may be punched on cards or written on an output tape (Tape 14). The data is written in the required order for SLAM Code with special flags inserted where additional SLAM input is required. These flags are of the form:

******Name of Data Set to be Added******

The following data sets must be added:

- Mesh Data Groups 3.0 and 4.0 (Material Property Data, Output Element Data)
- Run Time Data
- Output Node Data
- Boundary Data
- Shock Spectra Data

Note that it is possible to run the data generator a second time (with KRUN = 2) and add these SLAM data cards to the end of the first run data set. In this case a complete set of SLAM input data is generated.

4.0 Use of Generator

4.1 Required Control Cards

The following control cards are required:

1) Job Card: t, STMFZ, Ta, Pb

t = title

a = running time in octal seconds

b = job priority

2) Account Card: ACCOUNT(a,b)

a = ID

b = Account Number

3) Call in Program: ATTACH(OLDPL, GENI, ID=ZZG CAM)

4) Calcomp Tapes: ATTACH(LIBI, FR80LIB)

ATTACH LIB2, CALCOMPLIB.

LIBRARY (LIB1, LIB2)

ID CARD(TAPE99, a, T129, b, PLFI)

a = Name

b = Title for Plots

5) Request PF for SLAM Data

(Only if Data is on Tape) REQUEST, TAPE14, *PF.

6) UPDATE(F)

7) FTN(I=COMPILE, L=0)

8) LGO

9) Save SLAM Data Tape

(Only if Data is on Tape) CATALOG (Tape14, 9, ID=b)

a = Tape Name

b = ID

10) EOR

11) EOR

12) Data

13) EOR

4.2 Data Requirements

# of Cards	Format	Variables	Comments
1	2I5	KRUN	= 0 Do not write SLAM Data on Punch on Tape 14
			= 1 Write SLAM Data on Punch on Tape 14. Omit sets 2, 3, 9, 10, 12
			= 2 Write SLAM Data on Punch or Tape 14. Data for sets 2, 3, 9, 10, 12, 14 required
		ITAPE	= 1 Punch SLAM Data if KRUN = 1 or 2 = 2 Write SLAM Data on Tape 14 if KRUN = 1 or 2
1	8A10	ANAME	Problem title
1	14I5	NOD	# of Nodes in Structure
		MD	# of Modes
		NBM	# of Shear Beam
		NEL	# of Element
		NROT	# of Rotary inertia
NOD	I5,2F10.0	J	Node number
		Y(J)	Elevation of node (J) in feet
		W	Weight attached to node J in Kips
NROT	I5,F10.0	J	Node Number
		WR	Rotary inertia attached to node J
NBM	I5,5E10.0	K	Beam Number
		KR	= 0 no moment release
			= 1 release at KS
			= 2 release at KE

# of Cards	Format	Variables	Comments
		KS	Start node
		KE	End node
		A	Cross sectional area
		As	Shear area
		XI	Moment of inertia
		E	Young's Modulus
		G	Shear Modulus
NEL-NBM	3I5,3E10.0	K	Spring number
		KS	Start node
		KE	End node
		SL	Lateral stiffness
		SR	Rotational stiffness
		SLR	Coupled stiffness
1	I5,2F10.0	IFOUND	# of free field nodes in structure
		Depth	Depth of burial (feet)
		Datum	Datum for structural elevation in free field coordinate system
IFOUND	14F5	INODE(I)	Structural node numbers attached to free field
1	20AX	ANAME	Mesh Name
1	2F5,4F10.0	NL	# of Layers
		NH	# of elements horizontal
		H	Depth of Mesh (feet)
		W	Width of Mesh (feet)

# of Cards	Format	Variables	Comments
		HS	Depth of structure (feet)
		WS	Width of structure (feet)
NL	2I5,F10.0	NV(I)	# of element vertically in layer (I)
		IZONE(I)	Material numbers for layer (I)
		ZL(I)	Depth of layer(I)

Omit the following if KRUN N.E. 2

Material Properties Data for SLAM

Output Element Data

Time Data

Output Node Data

Boundary Data

Shock Spectra Data

APPENDIX E

SAMPLE PROBLEM

Sample output for the problem discussed in Volume 1 follows.

```
   SAMPLE PROBLEM 1
COMPLETE PROGRAM TO BE RUN
RESPONSE SPECTRA TO BE OUTPUT
CALCOMP PLOTS TO BE GENERATED
```

```
      SAMPLE PROBLEM 1

NO. OF NODE POINTS=  215
NO. OF ELEMENTS   =  184
IPRINT            =   -[illegible]
ISTRES            =    1   PLANE STRAIN PROBLEM
```

NODE POINT DATA

NODE PT NO.	TYPE	THETA (DEGREES)	RADIUS (FT)	DEPTH (FT)
1	1	0.	0.	8.0000000E+01
2	1	0.	0.	1.0000000E+02
3	1	0.	0.	1.2000000E+02
4	1	0.	0.	1.4000000E+02
5	1	0.	0.	1.6000000E+02
6	1	0.	0.	1.8000000E+02
7	1	0.	0.	2.0000000E+02
8	1	0.	0.	2.2000000E+02
9	1	0.	0.	2.4000000E+02
10	1	0.	0.	2.6000000E+02
11	1	0.	0.	2.8000000E+02
12	1	0.	0.	3.0000000E+02
13	1	0.	0.	3.2000000E+02
14	1	0.	0.	3.4000000E+02
15	1	0.	0.	3.6000000E+02
16	1	0.	0.	3.8000000E+02
17	1	0.	0.	4.0000000E+02
18	0	0.	4.2500000E+01	8.0000000E+01
19	0	0.	4.2500000E+01	1.0000000E+02
20	0	0.	4.2500000E+01	1.2000000E+02
21	0	0.	4.2500000E+01	1.4000000E+02
22	0	0.	4.2500000E+01	1.6000000E+02
23	0	0.	4.2500000E+01	1.8000000E+02
24	0	0.	4.2500000E+01	2.0000000E+02
25	0	0.	4.2500000E+01	2.2000000E+02
26	0	0.	4.2500000E+01	2.4000000E+02
27	0	0.	4.2500000E+01	2.6000000E+02
28	0	0.	4.2500000E+01	2.8000000E+02
29	0	0.	4.2500000E+01	3.0000000E+02
30	0	0.	4.2500000E+01	3.2000000E+02
31	0	0.	4.2500000E+01	3.4000000E+02
32	0	0.	4.2500000E+01	3.6000000E+02
33	0	0.	4.2500000E+01	3.8000000E+02
34	1	0.	4.2500000E+01	4.0000000E+02
35	0	0.	8.5000000E+01	8.0000000E+01
36	0	0.	8.5000000E+01	1.0000000E+02
37	0	0.	8.5000000E+01	1.2000000E+02
38	0	0.	8.5000000E+01	1.4000000E+02
39	0	0.	8.5000000E+01	1.6000000E+02
40	0	0.	8.5000000E+01	1.8000000E+02
41	0	0.	8.5000000E+01	2.0000000E+02
42	0	0.	8.5000000E+01	2.2000000E+02
43	0	0.	8.5000000E+01	2.4000000E+02
44	0	0.	8.5000000E+01	2.6000000E+02
45	0	0.	8.5000000E+01	2.8000000E+02
46	0	0.	8.5000000E+01	3.0000000E+02
47	0	0.	8.5000000E+01	3.2000000E+02
48	0	0.	8.5000000E+01	3.4000000E+02
49	0	0.	8.5000000E+01	3.6000000E+02
50	0	0.	8.5000000E+01	3.8000000E+02
51	1	0.	8.5000000E+01	4.0000000E+02
52	0	0.	1.2750000E+02	8.0000000E+01
53	0	0.	1.2750000E+02	1.0000000E+02
54	0	0.	1.2750000E+02	1.2000000E+02
55	0	0.	1.2750000E+02	1.4000000E+02
56	0	0.	1.2750000E+02	1.6000000E+02
57	0	0.	1.2750000E+02	1.8000000E+02

58	0	0.	1.2750000E+02	2.0000000E+02
59	0	0.	1.2750000E+02	2.2000000E+02
60	0	0.	1.2750000E+02	2.4000000E+02
61	0	0.	1.2750000E+02	2.6000000E+02
62	0	0.	1.2750000E+02	2.8000000E+02
63	0	0.	1.2750000E+02	3.0000000E+02
64	0	0.	1.2750000E+02	3.2000000E+02
65	0	0.	1.2750000E+02	3.4000000E+02
66	0	0.	1.2750000E+02	3.6000000E+02
67	0	0.	1.2750000E+02	3.8000000E+02
68	1	0.	1.2750000E+02	4.0000000E+02
69	0	0.	1.7000000E+02	0.
70	0	0.	1.7000000E+02	2.0000000E+01
71	0	0.	1.7000000E+02	4.0000000E+01
72	0	0.	1.7000000E+02	6.0000000E+01
73	0	0.	1.7000000E+02	8.0000000E+01
74	0	0.	1.7000000E+02	1.0000000E+02
75	0	0.	1.7000000E+02	1.2000000E+02
76	0	0.	1.7000000E+02	1.4000000E+02
77	0	0.	1.7000000E+02	1.6000000E+02
78	0	0.	1.7000000E+02	1.8000000E+02
79	0	0.	1.7000000E+02	2.0000000E+02
80	0	0.	1.7000000E+02	2.2000000E+02
81	0	0.	1.7000000E+02	2.4000000E+02
82	0	0.	1.7000000E+02	2.6000000E+02
83	0	0.	1.7000000E+02	2.8000000E+02
84	0	0.	1.7000000E+02	3.0000000E+02
85	0	0.	1.7000000E+02	3.2000000E+02
86	0	0.	1.7000000E+02	3.4000000E+02
87	0	0.	1.7000000E+02	3.6000000E+02
88	0	0.	1.7000000E+02	3.8000000E+02
89	1	0.	1.7000000E+02	4.0000000E+02
90	0	0.	2.0833300E+02	0.
91	0	0.	2.0833300E+02	2.0000000E+01
92	0	0.	2.0833300E+02	4.0000000E+01
93	0	0.	2.0833300E+02	6.0000000E+01
94	0	0.	2.0833300E+02	8.0000000E+01
95	0	0.	2.0833300E+02	1.0000000E+02
96	0	0.	2.0833300E+02	1.2000000E+02
97	0	0.	2.0833300E+02	1.4000000E+02
98	0	0.	2.0833300E+02	1.6000000E+02
99	0	0.	2.0833300E+02	1.8000000E+02
100	0	0.	2.0833300E+02	2.0000000E+02
101	0	0.	2.0833300E+02	2.2000000E+02
102	0	0.	2.0833300E+02	2.4000000E+02
103	0	0.	2.0833300E+02	2.6000000E+02
104	0	0.	2.0833300E+02	2.8000000E+02
105	0	0.	2.0833300E+02	3.0000000E+02
106	0	0.	2.0833300E+02	3.2000000E+02
107	0	0.	2.0833300E+02	3.4000000E+02
108	0	0.	2.0833300E+02	3.6000000E+02
109	0	0.	2.0833300E+02	3.8000000E+02
110	1	0.	2.0833300E+02	4.0000000E+02
111	0	0.	2.4666700E+02	0.
112	0	0.	2.4666700E+02	2.0000000E+01
113	0	0.	2.4666700E+02	4.0000000E+01
114	0	0.	2.4666700E+02	6.0000000E+01
115	0	0.	2.4666700E+02	8.0000000E+01
116	0	0.	2.4666700E+02	1.0000000E+02
117	0	0.	2.4666700E+02	1.2000000E+02
118	0	0.	2.4666700E+02	1.4000000E+02
119	0	0.	2.4666700E+02	1.6000000E+02
120	0	0.	2.4666700E+02	1.8000000E+02
121	0	0.	2.4666700E+02	2.0000000E+02
122	0	0.	2.4666700E+02	2.2000000E+02
123	0	0.	2.4666700E+02	2.4000000E+02

124	0	0.	2.4666700E+02	2.6000000E+02
125	0	0.	2.4666700E+02	2.8000000E+02
126	0	0.	2.4666700E+02	3.0000000E+02
127	0	0.	2.4666700E+02	3.2000000E+02
128	0	0.	2.4666700E+02	3.4000000E+02
129	0	0.	2.4666700E+02	3.6000000E+02
130	0	0.	2.4666700E+02	3.8000000E+02
131	1	0.	2.4666700E+02	4.0000000E+02
132	0	0.	2.8500000E+02	0.
133	0	0.	2.8500000E+02	2.0000000E+01
134	0	0.	2.8500000E+02	4.0000000E+01
135	0	0.	2.8500000E+02	6.0000000E+01
136	0	0.	2.8500000E+02	8.0000000E+01
137	0	0.	2.8500000E+02	1.0000000E+02
138	0	0.	2.8500000E+02	1.2000000E+02
139	0	0.	2.8500000E+02	1.4000000E+02
140	0	0.	2.8500000E+02	1.6000000E+02
141	0	0.	2.8500000E+02	1.8000000E+02
142	0	0.	2.8500000E+02	2.0000000E+02
143	0	0.	2.8500000E+02	2.2000000E+02
144	0	0.	2.8500000E+02	2.4000000E+02
145	0	0.	2.8500000E+02	2.6000000E+02
146	0	0.	2.8500000E+02	2.8000000E+02
147	0	0.	2.8500000E+02	3.0000000E+02
148	0	0.	2.8500000E+02	3.2000000E+02
149	0	0.	2.8500000E+02	3.4000000E+02
150	0	0.	2.8500000E+02	3.6000000E+02
151	0	0.	2.8500000E+02	3.8000000E+02
152	1	0.	2.8500000E+02	[illegible].0000000E+02
153	0	0.	3.2333300E+02	0.
154	0	0.	3.2333300E+02	2.0000000E+01
155	0	0.	3.2333300E+02	4.0000000E+01
156	0	0.	3.2333300E+02	6.0000000E+01
157	0	0.	3.2333300E+02	8.0000000E+01
158	0	0.	3.2333300E+02	1.0000000E+02
159	0	0.	3.2333300E+02	1.2000000E+02
160	0	0.	3.2333300E+02	1.4000000E+02
161	0	0.	3.2333300E+02	1.6000000E+02
162	0	0.	3.2333300E+02	1.8000000E+02
163	0	0.	3.2333300E+02	2.0000000E+02
164	0	0.	3.2333300E+02	2.2000000E+02
165	0	0.	3.2333300E+02	2.4000000E+02
166	0	0.	3.2333300E+02	2.6000000E+02
167	0	0.	3.2333300E+02	2.8000000E+02
168	0	0.	3.2333300E+02	3.0000000E+02
169	0	0.	3.2333300E+02	3.2000000E+02
170	0	0.	3.2333300E+02	3.4000000E+02
171	0	0.	3.2333300E+02	3.6000000E+02
172	0	0.	3.2333300E+02	3.8000000E+02
173	1	0.	3.2333300E+02	4.0000000E+02
174	0	0.	3.6166700E+02	0.
175	0	0.	3.6166700E+02	2.0000000E+01
176	0	0.	3.6166700E+02	[illegible]0000000E+01
177	0	0.	3.6166700E+02	[illegible]0000000E+01
178	0	0.	3.6166700E+02	6.0000000E+01
179	0	0.	3.6166700E+02	1.0000000E+02
180	0	0.	3.6166700E+02	1.2000000E+02
181	0	0.	3.6166700E+02	1.4000000E+02
182	0	0.	3.6166700E+02	1.6000000E+02
183	0	0.	3.6166700E+02	1.8000000E+02
184	0	0.	3.6166700E+02	2.0000000E+02
185	0	0.	3.6166700E+02	2.2000000E+02
186	0	0.	3.6166700E+02	2.4000000E+02
187	0	0.	3.6166700E+02	2.6000000E+02
188	0	0.	3.6166700E+02	2.8000000E+02
189	0	0.	3.6166700E+02	3.0000000E+02

POOR ORIGINAL

190	0	0.	3.6166700E+02	3.2000000E+02
191	0	0.	3.6166700E+02	3.4000000E+02
192	0	0.	3.6166700E+02	3.6000000E+02
193	0	0.	3.6166700E+02	3.8000000E+02
194	1	0.	3.6166700E+02	4.0000000E+02
195	0	0.	4.0000000E+02	0.
196	0	0.	4.0000000E+02	2.0000000E+01
197	0	0.	4.0000000E+02	4.0000000E+01
198	0	0.	4.0000000E+02	6.0000000E+01
199	0	0.	4.0000000E+02	8.0000000E+01
200	0	0.	4.0000000E+02	1.0000000E+02
201	0	0.	4.0000000E+02	1.2000000E+02
202	0	0.	4.0000000E+02	1.4000000E+02
203	0	0.	4.0000000E+02	1.6000000E+02
204	0	0.	4.0000000E+02	1.8000000E+02
205	0	0.	4.0000000E+02	2.0000000E+02
206	0	0.	4.0000000E+02	2.2000000E+02
207	0	0.	4.0000000E+02	2.4000000E+02
208	0	0.	4.0000000E+02	2.6000000E+02
209	0	0.	4.0000000E+02	2.8000000E+02
210	0	0.	4.0000000E+02	3.0000000E+02
211	0	0.	4.0000000E+02	3.2000000E+02
212	0	0.	4.0000000E+02	3.4000000E+02
213	0	0.	4.0000000E+02	3.6000000E+02
214	0	0.	4.0000000E+02	3.8000000E+02
215	1	0.	4.0000000E+02	4.0000000E+02

ELEMENT DATA

ELEMENT NO.	ZONE NO.	NPI	NPJ	NPK	NPL	NCRACK
1	1	1	18	19	2	-0
2	1	2	19	20	3	-0
3	1	3	20	21	4	-0
4	1	4	21	22	5	-0
5	1	5	22	23	6	-0
6	1	6	23	24	7	-0
7	1	7	24	25	8	-0
8	1	8	25	26	9	-0
9	1	9	26	27	10	-0
10	1	10	27	28	11	-0
11	1	11	28	29	12	-0
12	1	12	29	30	13	-0
13	1	13	30	31	14	-0
14	1	14	31	32	15	-0
15	1	15	32	33	16	-0
16	1	16	33	34	17	-0
17	1	18	35	36	19	-0
18	[illegible]	19	36	37	20	-0
19	1	[illegible]	37	38	21	-0
20	1	[illegible]	38	39	22	-0
21	1	[illegible]	39	40	23	-0
22	1	23	40	41	24	-0
23	1	24	41	42	25	-0
24	1	25	42	43	26	-0
25	1	26	43	44	27	-0
26	1	27	44	45	28	-0
27	1	28	45	46	29	-0
28	1	29	46	47	30	-0
29	1	30	47	48	31	-0
30	1	31	48	49	32	-0
31	1	32	49	50	33	-0
32	1	33	50	51	34	-0
33	1	35	52	53	36	-0
34	1	36	53	54	37	-0
35	1	37	54	55	38	-0
36	1	38	55	56	39	-0
37	1	39	56	57	40	-0
38	1	40	57	58	41	-0
39	1	41	58	59	42	-0
40	1	42	59	60	43	-0
41	1	43	60	61	44	-0
42	1	44	61	62	45	-0
43	1	45	62	63	46	-0
44	1	46	63	64	47	-0
45	1	47	64	65	48	-0
46	1	48	65	66	49	-0
47	1	49	66	67	50	-0
48	1	50	67	68	51	-0
49	1	52	73	74	53	-0
50	1	53	74	75	54	-0
51	1	54	75	76	55	-0
52	1	55	76	77	56	-0
53	1	56	77	78	57	-0
54	1	57	78	79	58	-0
55	1	58	79	80	59	-0
56	1	59	80	81	60	-0
57	1	60	81	82	61	-0

124	1	130	151	152	131	-0
125	1	132	153	154	133	-0
126	1	133	154	155	134	-0
127	1	134	155	156	135	-0
128	1	135	156	157	136	-0
129	1	136	157	158	137	-0
130	1	137	158	159	138	-0
131	1	138	159	160	139	-0
132	1	139	160	161	140	-0
133	1	140	161	162	141	-0
134	1	141	162	163	142	-0
135	1	142	163	164	143	-0
136	1	143	164	165	144	-0
137	1	144	165	166	145	-0
138	1	145	166	167	146	-0
139	1	146	167	168	147	-0
140	1	147	168	169	148	-0
141	1	148	169	170	149	-0
142	1	149	170	171	150	-0
143	1	150	171	172	151	-0
144	1	151	172	173	152	-0
145	1	153	174	175	154	-0
146	1	154	175	176	155	-0
147	1	155	176	177	156	-0
148	1	156	177	178	157	-0
149	1	157	178	179	158	-0
150	1	158	179	180	159	-0
151	1	159	180	181	160	-0
152	1	160	181	182	161	-0
153	1	161	182	183	162	-0
154	1	162	183	184	163	-0
155	1	163	184	185	164	-0
156	1	164	185	186	165	-0
157	1	165	186	187	166	-0
158	1	166	187	188	167	-0
159	1	167	188	189	168	-0
160	1	168	189	190	169	-0
161	1	169	190	191	170	-0
162	1	170	191	192	171	-0
163	1	171	192	193	172	-0
164	1	172	193	194	173	-0
165	1	174	195	196	175	-0
166	1	175	196	197	176	-0
167	1	176	197	198	177	-0
168	1	177	198	199	178	-0
169	1	178	199	200	179	-0
170	1	179	200	201	180	-0
171	1	180	201	202	181	-0
172	1	181	202	203	182	-0
173	1	182	203	204	183	-0
174	1	183	204	205	184	-0
175	1	184	205	206	185	-0
176	1	185	206	207	186	-0
177	1	186	207	208	187	-0
178	1	187	208	209	188	-0
179	1	188	209	210	189	-0
180	1	189	210	211	190	-0
181	1	190	211	212	191	-0
182	1	191	212	213	192	-0
183	1	192	213	214	193	-0
184	1	193	214	215	194	-0

NO. OF ELASTIC OUTPUT ELEMENTS= 9
NO. OF PLASTIC OUTPUT ELEMENTS= 0

ZONE PROPERTY DATA

NO. OF ZONES= 1

ZONE NUMBER= 1 ELASTIC

IELAST	=	1
IPLAST	=	0
UNIT WEIGHT	=	1.30000E+02 PCF
ELASTIC MODULUS	=	3.47220E+05 PSI
POISSON,S RATIO	=	3.50000E-01

NO. OF OUTPUT ELEMENTS IS 9

OUTPUT ELEMENT NOS. ARE

1 17 33 49 65 66 67 68 69

58	1	61	82	83	62	-0
59	1	62	83	84	63	-0
60	1	63	84	85	64	-0
61	1	64	85	86	65	-0
62	1	65	86	87	66	-0
63	1	66	87	88	67	-0
64	1	67	88	89	68	-0
65	1	69	90	91	70	-0
66	1	70	91	92	71	-0
67	1	71	92	93	72	-0
68	1	72	93	94	73	-0
69	1	73	94	95	74	-0
70	1	74	95	96	75	-0
71	1	75	96	97	76	-0
72	1	76	97	98	77	-0
73	1	77	98	99	78	-0
74	1	78	99	100	79	-0
75	1	79	100	101	80	-0
76	1	80	101	102	81	-0
77	1	81	102	103	82	-0
78	1	82	103	104	83	-0
79	1	83	104	105	84	-0
80	1	84	105	106	85	-0
81	1	85	106	107	86	-0
82	1	86	107	108	87	-0
83	1	87	108	109	88	-0
84	1	88	109	110	89	-0
85	1	90	111	112	91	-0
86	1	91	112	113	92	-0
87	1	92	113	114	93	-0
88	1	93	114	115	94	-0
89	1	94	115	116	95	-0
90	1	95	116	117	96	-0
91	1	96	117	118	97	-0
92	1	97	118	119	98	-0
93	1	98	119	120	99	-0
94	1	99	120	121	100	-0
95	1	100	121	122	101	-0
96	1	101	122	123	102	-0
97	1	102	123	124	103	-0
98	1	103	124	125	104	-0
99	1	104	125	126	105	-0
100	1	105	126	127	106	-0
101	1	106	127	128	107	-0
102	1	107	128	129	108	-0
103	1	108	129	130	109	-0
104	1	109	130	131	110	-0
105	1	111	132	133	112	-0
106	1	112	133	134	113	-0
107	1	113	134	135	114	-0
108	1	114	135	136	115	-0
109	1	115	136	137	116	-0
110	1	116	137	138	117	-0
111	1	117	138	139	118	-0
112	1	118	139	140	119	-0
113	1	119	140	141	120	-0
114	1	120	141	142	121	-0
115	1	121	142	143	122	-0
116	1	122	143	144	123	-0
117	1	123	144	145	124	-0
118	1	124	145	146	125	-0
119	1	125	146	147	126	-0
120	1	126	147	148	127	-0
121	1	127	148	149	128	-0
122	1	128	149	150	129	-0
123	1	129	150	151	130	-0

STARTING NODE DATA

NO. OF START NODES= 3

STARTING NODE NUMBERS

1

2

18

POOR ORIGINAL

NO. OF GRAND PARTITIONS= 16

PARTITION	NEW NODE NO.
1	3
2	6
3	15
4	24
5	37
6	54
7	75
8	100
9	123
10	148
11	160
12	171
13	182
14	193
15	204
16	215

ORIGINAL	BANDWIDTH=	22
NEW	BANDWIDTH=	27
NEW	BANDWIDTH=	36

CLUSTERING

NO. OF CLUSTERS= 3

NPLOW	NPHIGH	NPOUT	NUMCP	NELCLS	NMPCLS
1	100	0	123	0	0
101	200	75	212	0	0
201	215	138	215	0	0

NO. OF NONLINEAR ELEMENTS ON TAPE= 0

SMALLEST MAIN DIAGONAL FREQUENCY (SEC/CYCLE)

```
NODE POINT=   69      PERIODU=     3.44166E-02
NODE POINT=  215      PERIODW=     2.36778E-02
NODE TYPE =    1
```

LOADED NODE POINT DATA

NO. OF LOADED SURFACES= 0

INITIAL STRESS DATA

NO. OF ELEMENTS=   0

STRL ,RAL DATA

STRUCTURAL TYPE= 4

NO. OF NODES ON STRUCTURE= 9

NODE	RADIUS (FT)	DEPTH (FT)
1	0.	8.00000E+01
18	4.25000E+01	8.00000E+01
35	8.50000E+01	8.00000E+01
52	1.27500E+02	8.00000E+01
69	1.70000E+02	0.
70	1.70000E+02	2.00000E+01
71	1.70000E+02	4.00000E+01
72	1.70000E+02	6.00000E+01
73	1.70000E+02	8.00000E+01

RADIAL COORDINATE OF CG (FT)= 0.
VERTICAL COORDINATE OF CG (FT)= 2.92360E+01

POOR ORIGINAL

SPEC STRUCTURE DATA

SPECIAL STRUCTURES DATA

NO. OF STRUCTURAL MODES = 7
NO. OF STRUCTURAL DOF = 59

INPUT SEQ.	NODE NO.	HOR DOF	VER DOF
1	1	42	43
2	18	44	45
3	35	46	47
4	52	48	49
5	69	50	51
6	70	52	53
7	71	54	55
8	72	56	57
9	73	58	59

MODE SHAPE DATA

MODE= 1 FREQ CPS= 1.60620E+00 GEN MASS= 5.04300E-01

MODE VECTOR

2.05600E-02	1.8600JE-02	1.24300E-02	5.98200E-03	1.76500E-03	-2.90800E-03	-6.15400E-03
-1.03700E-02	-1.47500E-02	-1.98300E-02	4.92800E-03	-3.46900E-04	-5.62400E-03	-1.09000E-02
-1.35300E-02	-2.95400E-03	-8.21700E-03	1.25600E-02	1.00000E+00	9.12600E-03	6.87500E-04
-1.01200E-02	-5.11600E-04	-5.07000E-03	-9.58000E-03	-1.41500E-02	-1.87200E-02	-2.37900E-02
-2.78000E-02	-3.23700E-02	-3.68700E-02	-4.14100E-02	-4.59500E-02	-5.04200E-02	-5.48900E-02
-5.93300E-02	-6.37600E-02	-6.81700E-02	-7.25700E-02	-7.69400E-02	-8.12800E-02	2.05600E-02
0.	2.05600E-02	2.22600E-02	2.05600E-02	4.45200E-02	2.05600E-02	6.67800E-02
-1.98300E-02	8.92800E-02	-1.03700E-02	1.11600E-01	1.76500E-03	1.33900E-01	1.24300E-02
1.56100E-01	2.05600E-02	1.78100E-01				

MODE= 2 FREQ CPS= 6.50500E+00 GEN MASS= 1.10250E+00

MODE VECTOR

-9.95800E-04	7.87000E-04	-5.31100E-04	-2.12400E-03	-3.19600E-03	-3.01300E-03	-2.81800E-03
-2.48300E-03	-2.04800E-03	-1.63700E-03	-1.25500E-03	-1.40500E-03	-1.50400E-03	-1.63400E-03
-1.91200E-03	-4.38300E-03	-5.67400E-03	1.00000E+00	-1.05500E-01	3.03200E-03	1.16400E-03
-1.22100E-03	-1.19900E-02	-2.99200E-02	-4.80000E-02	-6.65200E-02	-8.51500E-02	-1.05700E-01
-1.21600E-01	-1.39300E-01	-1.56400E-01	-1.73100E-01	-1.89100E-01	-2.04200E-01	-2.18500E-01
-2.34900E-01	-2.50400E-01	-2.65000E-01	-2.78300E-01	-2.90200E-01	-3.00100E-01	-9.95800E-04
0.	-9.95800E-04	4.33300E-03	-9.95800E-04	8.66700E-03	-9.95800E-04	1.30000E-02
-1.63700E-03	1.73200E-02	-2.48300E-03	2.26100E-02	-3.19600E-03	3.19500E-02	-5.31100E-04
3.44900E-02	-9.95800E-04	3.46700E-02				

MODE= 3 FREQ CPS= 7.87210E+00 GEN MASS= 3.04060E+00

MODE VECTOR

1.28700E-01	1.18400E-01	8.29200E-02	4.52700E-02	1.97400E-02	-2.01700E-02	-4.67700E-02
-8.18200E-02	-1.15800E-01	-1.54000E-01	3.10900E-02	-9.55300E-03	-5.05500E-02	-9.06100E-02

-1.0[illegible]E-01	-6.79200E-03	-3.62200E-02	5.19200E-01	[illegible]3100E-03	1.66300E-02	-3.16000E-02
-9.04800E-02	5.05800E-02	1.14000E-01	1.78400E-01	[illegible].44600E-01	3.11200E-01	3.84300E-01
4.40300E-01	5.01600E-01	5.59800E-01	6.15800E-01	6.68300E-01	7.16100E-01	7.59400E-01
8.14300E-01	8.64100E-01	9.08800E-01	9.47200E-01	9.76900E-01	1.00000E+00	1.28700E-01
0.	1.28700E-01	1.24000E-01	1.28700E-01	2.48100E-01	1.28700E-01	3.72100E-01
-1.54000E-01	5.24000E-01	-8.18200E-02	6.48700E-01	1.97400E-02	7.41900E-01	8.29200E-02
8.70000E-01	1.28700E-01	9.92200E-01				

MODE= 4 FFEQ CPS= 1.27950E+01 GEN MASS= 4.43900E-01

MODE VECTOR

-6.21600E-03	-6.46700E-03	-7.11700E-03	-7.81800E-03	-7.92200E-03	-6.31700E-03	-5.03100E-03
-3.18200E-03	-1.10800E-03	-2.68400E-03	-5.32600E-03	-3.08500E-03	-9.77500E-04	6.52300E-04
5.71800E-04	-8.98100E-03	-9.95000E-03	5.47400E-04	-2.50100E-03	1.00000E+00	6.26400E-01
1.01700E-01	-7.88900E-03	-7.63400E-03	-7.19100E-03	-6.56100E-03	-5.76700E-03	-4.69500E-03
-3.72000E-03	-2.50400E-03	-1.23600E-03	8.54000E-05	1.41400E-03	2.70300E-03	3.94000E-03
5.58100E-03	7.12800E-03	8.56800E-03	9.84700E-03	1.09300E-02	1.16800E-02	-6.21600E-03
0.	-6.21600E-03	2.90500E-03	-6.21600E-03	5.81100E-03	-6.21600E-03	8.71600E-03
-2.68400E-03	1.93700E-02	-3.18200E-03	2.11500E-02	-7.92200E-03	1.83500E-02	-7.11700E-03
2.04800E-02	-6.21600E-03	2.32400E-02				

MODE= 5 FREQ CPS= 1.89620E+01 GEN MASS= 1.57530E+00

MODE VECTOR

2.52000E-02	2.37600E-02	1.86200E-02	1.05900E-02	3.90500E-03	4.25800E-03	4.12400E-03
3.55500E-03	2.14500E-03	-2.64100E-04	1.55900E-02	1.26600E-02	9.01200E-03	4.88900E-03
3.02800E-03	1.07000E-03	-2.14000E-03	6.81000E-03	9.17700E-03	-2.00800E-02	-1.08900E-02
1.83100E-03	-8.67000E-02	-2.63600E-01	-4.26200E-01	-5.71500E-01	-6.91100E-01	-7.81700E-01
-8.13300E-01	-8.05000E-01	-7.61200E-01	-6.82600E-01	-5.73100E-01	-4.39100E-01	-2.84900E-01
-6.66300E-02	1.62000E-01	3.95700E-01	6.23100E-01	8.33800E-01	1.00000E+00	2.52000E-02
0.	2.52000E-02	1.40600E-02	2.52000E-02	2.81200E-02	2.52000E-02	4.21800E-02
-2.64100E-04	5.02500E-02	3.55500E-03	6.30700E-02	3.90500E-03	7.94100E-02	1.86200E-02
9.73600E-02	2.52000E-02	1.12500E-01				

MODE= 6 FREQ CPS= 0. GEN MASS= 1.22800E+02

MODE VECTOR

1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00
0.	1.00000E+00	0.	1.00000E+00	0.	1.00000E+00	0.
1.00000E+00	0.	1.00000E+00	0.	1.00000E+00	0.	1.00000E+00
0.	1.00000E+00	0.				

MODE= 7 FREQ CPS= 0. GEN MASS= 1.02800E+05

MODE VECTOR

-4.77600E+01	-4.37600E+01	-3.20100E+01	-1.97600E+01	-1.17600E+01	-2.76400E+00	3.23600E+00
1.12400E+01	1.95700E+01	2.92400E+01	-1.77600E+01	-7.76400E+00	2.23600E+00	1.22400E+01
1.72400E+01	-2.76400E+00	7.23600E+00	6.99900E+01	1.02100E+02	-2.57600E+01	-9.76400E+00
1.07400E+01	-7.66400E+00	5.35700E-01	8.63600E+00	1.68400E+01	2.50400E+01	3.41400E+01

4.134[illegible]E+01	4.95400E+01	5.76400E+01	6.58400E+01	[illegible]40400E+01	8.21400E+01	9.02400E+01
9.82400E+01	1.06200E+02	1.14200E+02	1.22200E+02	1.30200E+02	1.38200E+02	-4.77600E+01
0.	-4.77600E+01	4.25000E+01	-4.77600E+01	8.50000E+01	-4.77600[illegible]	1.27500E+02
2.92400E+01	1.70000E+02	1.12400E+01	2.12500E+02	-1.17600E+01	2.55000E+02	-3.20100E+01
2.97500E+02	-4.77600E+01	3.40000E+02				

PERIOD= 36776E-02
PERSTR= .32912E-02

MAX. TIME DURATION	=	7.00000E+00	SEC
TIME INCREMENT	=	1.57852E-03	SEC
KDT	=	1	
KINT	=	15	
KTAPE	=	1	

OUTPUT NODE DATA

NUMOUT= 14
IOTAPE= 4
IOPAPE= 200

OUTPUT NODES

1 4 7 10 13 17 18 35 52 69 70 71 72 73

SAVE TAPE INTERVAL

SAVE TAPE DATA

IOSAVE= 500

NRCDS ON LOGICAL 14= 3692

QUIET UNDARY DATA

```
NO. OF NODES ON RIGHT  BOUNDARY=   20
NO. OF NODES ON BOTTOM BOUNDARY=   -0
NO. OF NODES ON LEFT   BOUNDARY=   -0
LEFT CORNER CONDITION          =   -0
TOTAL NO. OF BOUNDARY NODES    =   20
```

RIGHT BOUNDARY NODE NUMBERS
195 196 197 198 199 200 201 202 203 204 205 206 207 208 209 210 211 212 213 214

NO. OF BOUNDARY RECTANGULAR ELEMENTS= 19

NUMB	NUME	IZONE	NODES				DR(FT)	DZ(FT)
1	165	1	174	195	196	175	3.83300E+01	2.00000E+01
2	166	1	175	196	197	176	3.83300E+01	2.00000E+01
3	167	1	176	197	198	177	3.83300E+01	2.00000E+01
4	168	1	177	198	199	178	3.83300E+01	2.00000E+01
5	169	1	178	199	200	179	3.83300E+01	2.00000E+01
6	170	1	179	200	201	180	3.83300E+01	2.00000E+01
7	171	1	180	201	202	181	3.83300E+01	2.00000E+01
8	172	1	181	202	203	182	3.83300E+01	2.00000E+01
9	173	1	182	203	204	183	3.83300E+01	2.00000E+01
10	174	1	183	204	205	184	3.83300E+01	2.00000E+01
11	175	1	184	205	206	185	3.83300E+01	2.00000E+01
12	176	1	185	206	207	186	3.83300E+01	2.00000E+01
13	177	1	186	207	208	187	3.83300E+01	2.00000E+01
14	178	1	187	208	209	188	3.83300E+01	2.00000E+01
15	179	1	188	209	210	189	3.83300E+01	2.00000E+01
16	180	1	189	210	211	190	3.83300E+01	2.00000E+01
17	181	1	190	211	212	191	3.83300E+01	2.00000E+01
18	182	1	191	212	213	192	3.83300E+01	2.00000E+01
19	183	1	192	213	214	193	3.83300E+01	2.00000E+01

MATERIAL ZONE DATA

ZONE	E(PSI)	XNU	E EQUIV	XNU EQUIV	CP(IN/SEC)	CS(IN/SEC)	RHO(LB-SEC2/IN4)
1	3.47220E+05	3.50000E-01	3.47220E+05	3.50000E-01	5.34996E+04	2.57004E+04	1.94698E-04

IELEM = 0 1

IELEM = 1 2

BOUNDARY ELEMENT CONNECTIVITY

BOUNDARY NODE NUMBERS	ELEMENT	ELEMENT	RHO*CP (LB-SEC/IN3)	RHO*CS (LB-SEC/IN3)	CONST
RIGHT	ABOVE	BELOW			
195	0	1	1.04163E+01	5.00382E+00	1.20000E+02
196	1	2	1.04163E+01	5.00382E+00	2.40000E+02
197	2	3	1.04163E+01	5.00382E+00	2.40000E+02
198	3	4	1.04163E+01	5.00382E+00	2.40000E+02
199	4	5	1.04163E+01	5.00382E+00	2.40000E+02
200	5	6	1.04163E+01	5.00382E+00	2.40000E+02
201	6	7	1.04163E+01	5.00382E+00	2.40000E+02
202	7	8	1.04163E+01	5.00382E+00	2.40000E+02
203	8	9	1.04163E+01	5.00382E+00	2.40000E+02
204	9	10	1.04163E+01	5.00382E+00	2.40000E+02
205	10	11	1.04163E+01	5.00382E+00	2.40000E+02
206	11	12	1.04163E+01	5.00382E+00	2.40000E+02
207	12	13	1.04163E+01	5.00382E+00	2.40000E+02
208	13	14	1.04163E+01	5.00382E+00	2.40000E+02
209	14	15	1.04163E+01	5.00382E+00	2.40000E+[illegible]
210	15	16	1.04163E+01	5.00382E+00	2.40000E+02
211	16	17	1.04163E+01	5.00382E+00	2.40000E+02
212	17	18	1.04163E+01	5.00382E+00	2.40000E+02
213	18	19	1.04163E+01	5.00382E+00	2.40000E+02
214	19	0	1.04163E+01	5.00382E+00	1.20000E+02

T= TREAL= 0.

ELEMENT	SIGMAR	SIGMAT	SIGMAZ	TAU	SIGMX	SIGMN	ANGLE
1	0.	0.	0.	0.	0	0.	0.
17	0.	0.	0.	0.	[illegible]	0.	0.
33	0.	0.	0.	0.	0	0.	0.
49	0.	0.	0.	0.	0.	0.	0.
68	0.	0.	0.	0.	0.	0.	0.
69	0.	0.	0.	0.	0.	0.	0.
67	0.	0.	0.	0.	0.	0.	0.
66	0.	0.	0.	0.	0.	0.	0.
65	0.	0.	0.	0.	0.	0.	0.

NODE	DISP. U (IN)	VEL.UD (IPS)	ACCEL.UDD (G)	PRESSU (PSI)	DISP. W (IN)	VEL.WD (IPS)	ACCEL.WDD (G)	PRESSW (PSI)
1	0.	0.	0.	0.	0.	0.	0.	0.
4	0.	0.	0.	0.	0.	0.	0.	0.
7	0.	0.	0.	0.	0.	0.	0.	0.
10	0.	0.	0.	0.	0.	0.	0.	0.
13	0.	0.	0.	0.	0.	0.	0.	0.
17	0.	0.	0.	0.	0.	0.	0.	0.
18	0.	0.	0.	0.	0.	0.	0.	0.
35	0.	0.	0.	0.	0.	0.	0.	0.
52	0.	0.	0.	0.	0.	0.	0.	0.
69	0.	0.	0.	0.	0.	0.	0.	0.
70	0.	0.	0.	0.	0.	0.	0.	0.
71	0.	0.	0.	0.	0.	0.	0.	0.
72	0.	0.	0.	0.	0.	0.	0.	0.
73	0.	0.	0.	0.	0.	0.	0.	0.

STRUCTURAL RESPONSE

MODE	DISP. X	VEL. XSD	ACCEL. XSDD	APPL. LOAD Q
1	0.	0.	0.	0.
2	0.	0.	0.	0.
3	0.	0.	0.	0.
4	0.	0.	0.	0.
5	0.	0.	0.	0.
6	0.	0.	0.	0.
7	0.	0.	0.	0.

HAVE WRITTEN SAVE TAPE 8, T= 0.

HAVE WRITTEN OUTPUT TAPE, T= 7.89261E-03 SEC IRCDS= 2

HAVE WRITTEN OUTPUT TAPE, T= 1.57852E-02 SEC IRCDS= 3

HAVE WRITTEN OUTPUT TAPE, T= 2.36778E-02 SEC IRCDS= 4

T= ..34566E-01 TREAL= 6.34566E-01

ELEMENT	SIGMAR	SIGMAT	SIGMAZ	TAU	SIGMX	SIGMN	ANGLE
1	-6.5076E-01	-3.5041E-01	-3.5041E-01	2.2282E+01	2.1782E+01	-2.2783E+01	6.7597E+01
17	-2.0400E+00	-1.0984E+00	-1.0984E+00	2.0962E+01	1.9398E+01	-2.2537E+01	6.7822E+01
33	-2.0919E+00	-1.1264E+00	-1.1264E+00	1.8936E+01	1.7333E+01	-2.0551E+01	6.7865E+01
49	-8.2517E+00	-4.4432E+00	-4.4432E+00	1.3864E+01	7.6465E+00	-2.0341E+01	6.9455E+01
68	-5.0806E+01	-2.7357E+01	-2.7357E+01	2.7187E+01	-9.4748E+00	-6.8689E+01	7.3332E+01
69	-4.5981E+01	-2.4759E+01	-2.4759E+01	1.4975E+01	-1.7017E+01	-5.3724E+01	7.6330E+01
67	-3.4129E+01	-1.8377E+01	-1.8377E+01	3.5168E+01	9.7859E+00	-6.2292E+01	7.0656E+01
66	-8.1570E+00	-4.3922E+00	-4.3922E+00	3.9471E+01	3.3241E+01	-4.5790E+01	6.8183E+01
65	1.3935E+01	7.5033E+00	7.5033E+00	3.3343E+01	4.4217E+01	-2.2779E+01	0.

NODE	DISP. U (IN)	VEL.UD (IPS)	ACCEL.UDD (G)	PRESSJ (PSI)	DISP. W (IN)	VEL.WD (IPS)	ACCEL.WDD (G)	PRESSW (PSI)
1	7.7449E-01	4.3573E+00	2.1777E+00	0.	0.	0.	0.	0.
4	9.1303E-01	4.8837E+00	5.2667E-02	0.	0.	0.	0.	0.
7	1.0878E+00	5.4264E+00	-7.8210E-04	0.	0.	0.	0.	0.
10	1.2858E+00	5.6443E+00	2.9427E-0	0.	0.	0.	0.	0.
13	1.5072E+00	5.9471E+00	6.8824E-02	0.	0.	0.	0.	0.
17	1.8432E+00	-4.7069E+02	-5.1967E+00	0.	0.	0.	0.	0.
18	7.7449E-01	4.3573E+00	2.0993E+00	0.	-9.6612E-03	-5.5661E-02	-1.7773E-01	0.
35	7.7449E-01	4.3573E+00	1.7991E+00	0.	-1.9311E-02	-1.1195E-01	-5.7094E-01	0.
52	7.7449E-01	4.3573E+00	1.0630E+00	0.	-2.8972E-02	-1.6782E-01	-1.5802E+00	0.
69	4.9023E-01	3.7879E+00	5.3863E-01	0.	-3.4770E-02	-2.8397E-01	-9.5051E-01	0.
70	5.5737E-01	3.9075E+00	5.5443E-01	0.	-4.4174E-02	-3.4004E-01	-4.0567E-01	0.
71	6.4435E-01	4.0409E+00	1.4002E+00	0.	-5.7117E-02	-3.2377E-01	3.1799E-01	0.
72	7.1811E-01	4.2160E+00	-2.4778E+00	0.	-6.6494E-02	-3.9036E-01	1.2023E-01	0.
73	7.7449E-01	4.3573E+00	-1.8571E+00	0.	-7.7214E-02	-4.4695E-01	7.8585E-01	0.

STRUCTURAL RESPONSE

MODE	DISP. X	VEL. XSD	ACCEL. XSDD	APPL. LOAD Q
1	2.5528E+00	2.0411E+01	1.6451E+02	0.
2	4.6788E-02	-1.2052E-01	-4.9735E+01	0.
3	1.1779E-01	-2.3124E+00	3.2645E+01	0.
4	2.4721E-03	-1.6104E-01	3.6396E+00	0.
5	1.0424E-02	-9.4792E-02	2.8749E+00	0.
6	6.1511E-01	3.9880E+00	1.0361E+01	0.
7	-1.9164E-03	-5.2034E-03	-1.6646E-01	0.

HAVE WRITTEN OUTPUT TAPE, T= 6.39301E-01 SEC IRCDS= 82

HAVE WRITTEN OUTPUT TAPE, T= 6.47.94E-01 SEC IRCDS= 83

HAVE WRITTEN OUTPUT TAPE, T= 6.55086L-01 : SEC IRCDS= 84

HAVE WRITTEN OUTPUT TAPE, T= 6.62979E 01 SEC IRCDS= 85

ERROR IN NODE DATA

NPOLD= 16
MOOLD
1 4 10 13 17 18 35 52 69 70 71 72 73

SHOCK SPECTRA FOR NODE NUMBER 1

HORIZONTAL SPECTRA

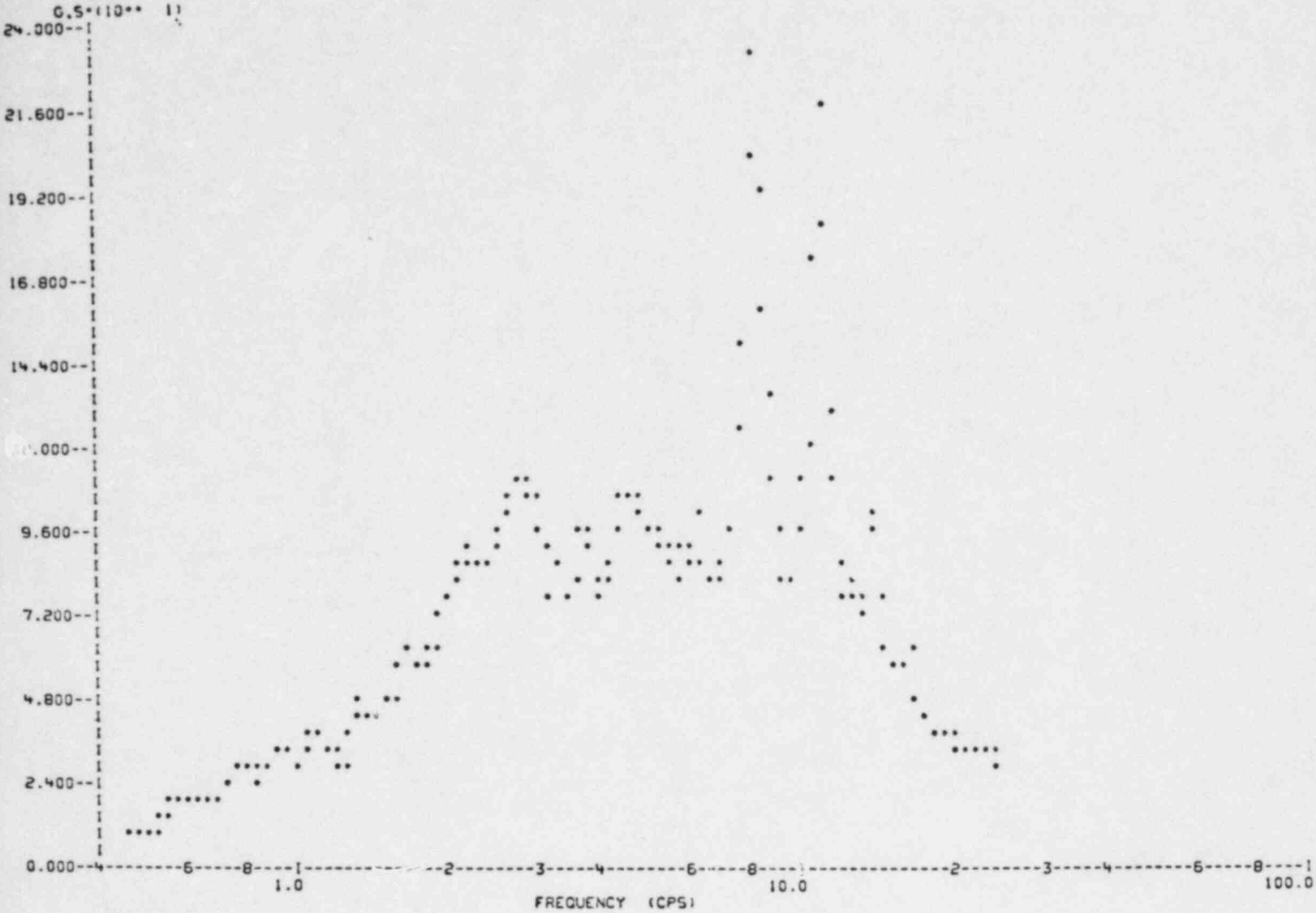
```
NO. OF INPUT RECORDS     =      888
FINAL DISPLACEMENT (IN.)=     .37161E+02
FINAL VELOCITY (IN/SEC) =     .81281E+01
PEAK ACCELERATION (G,S) =     .29773E+00

NO. OF RECORDS ADDED     =      90
FINAL NO. OF RECORDS     =      978
FINAL VELOCITY (IN/SEC) =    0.
FINAL DISPLACEMENT (IN.)=     .39972E+02
TMAX. (DURATION, SECS)  =     .77111E+01
```

DAMPING RATIO= 2.00000E+00 PERCENT

FREQUENCY (CPS)	ACCELERATION (G,S)	VELOCITY (IPS)	DISPLACEMENT (IN)	MAX. TIME (SECS)
.50000	.09369	11.52377	3.66813	6.67715
.51171	.08933	10.73620	3.33926	6.67715
.52369	.08997	10.56541	3.21097	6.68504
.53595	.09701	11.13102	3.30548	6.69293
.54849	.11004	12.33803	3.58010	6.69293
.56133	.12696	13.90939	3.94373	6.60504
.57448	.14470	15.48999	4.29141	6.66925
.58792	.16962	17.74241	4.80298	7.36380
.60169	.19337	19.76446	5.22797	7.32434
.61578	.21262	21.23479	5.48840	7.27698
.63019	.22495	21.95227	5.54405	7.22963
.64495	.22877	21.81427	5.38317	7.18227
.66004	.22376	20.84822	5.02708	7.14261
.67550	.21689	19.74611	4.65241	7.71108
.69131	.22109	19.66770	4.52793	7.71100
.70750	.21860	19.00159	4.27451	7.71108
.72406	.22133	18.79873	4.13214	7.71108
.74101	.23632	19.61252	4.21240	7.71108
.75836	.25929	21.02670	4.41283	7.71108
.77611	.28077	22.24798	4.56232	7.66372
.79428	.29452	22.80300	4.56917	7.60050
.81288	.29398	22.24074	4.35456	7.53744
.83191	.29980	22.16202	4.23988	6.16413
.85138	.31175	22.51807	4.20945	6.09309
.87132	.30922	21.82460	3.98649	6.06152
.89172	.28588	19.71553	3.51806	5.99046
.91259	.30307	20.42341	3.56182	4.71109
.93396	.32261	21.24239	3.61990	4.66453
.95582	.33762	21.72264	3.61706	4.61718
.97820	.34648	21.78232	3.54403	4.58561
1.00110	.34372	21.11470	3.35602	4.54614
1.02454	.34175	20.51319	3.18659	4.90024
1.04652	.33191	19.46721	2.95492	4.94867
1.07307	.33564	19.23548	2.85295	6.18780
1.09819	.38040	21.30169	3.08714	6.12466
1.12390	.41589	22.75665	3.22255	6.07731
1.15022	.41760	22.32757	3.08946	6.03784
1.17714	.39539	20.65650	2.79285	5.99049
1.20470	.37875	19.33432	2.55429	7.54533
1.23291	.37239	18.57469	2.39779	7.40008
1.26177	.33628	16.39015	2.06739	7.45062
1.29131	.30184	14.37489	1.77172	7.43484
1.32154	.31948	14.86678	1.79043	7.43404
1.35248	.39081	17.76997	2.09111	7.41905
1.38414	.46292	20.56731	2.36492	7.37959
1.41655	.48961	21.21241	2.38330	7.33223
1.44971	.44638	18.93548	2.07681	7.28488
1.48365	.43938	18.21252	1.95370	2.34410
1.51839	.46025	18.64090	1.95391	2.33621
1.55393	.47855	18.93883	1.93973	2.32832
1.59031	.49406	19.10527	1.91201	2.32043
1.62754	.50662	19.14283	1.87195	2.31253
1.66565	.51612	19.05556	1.82079	2.30464
1.70464	.60376	21.78137	2.03363	5.44590
1.74455	.66467	23.43043	2.13755	5.41433
1.76539	.64178	22.10613	1.97060	5.36697

1.82719	[illegible]	[illegible]	[illegible]	[illegible]
1.86997	[illegible]	[illegible]	[illegible]	2.50196
1.91375	[illegible]	20.02105	[illegible]	2.49406
[illegible]	[illegible]	19.99498	[illegible]	2.48617
2.00440	[illegible]	19.96533	1.58530	2.48617
2.05133	.74388	22.30091	1.73024	5.33540
2.09935	.79554	23.30424	1.76673	5.30383
2.14850	.79709	22.81531	1.69009	2.90448
2.19880	.84352	[illegible]	1.70765	2.88869
2.25028	.88245	[illegible]	1.70566	2.88080
2.30296	.90576	24.18709	1.67154	2.86502
2.35688	.91332	23.83099	1.60926	2.84923
2.41205	.90784	23.14616	1.52726	2.83345
2.46852	.89275	22.24071	1.43394	3.42539
2.52632	.89549	21.79874	1.37330	3.40171
2.58546	.90461	21.51705	1.32454	3.38593
2.64559	.93387	21.70488	1.30554	3.36225
2.70794	.97927	22.23931	1.30708	3.34647
2.77133	1.01834	22.59764	1.29776	3.33068
2.83621	1.06632	23.12087	1.29743	2.77031
2.90261	1.11135	23.54620	1.29108	2.75452
2.97057	1.12600	23.31069	1.24892	2.73873
3.04011	1.12171	22.69072	1.18790	2.73084
3.11129	1.09880	21.71884	1.11101	2.71506
3.18412	1.06349	20.54009	1.02667	2.70716
3.25867	1.00598	18.98477	.92722	2.69927
3.33496	.91817	16.93131	.80802	2.69138
3.41304	.79914	14.39927	.67146	2.68349
3.49294	.86677	15.26050	.69534	5.20123
3.57471	.87261	15.01192	.66837	5.17755
3.65840	.78386	13.17669	.57324	5.01181
3.74405	.85572	14.05559	.59749	2.69138
3.83171	.97727	15.68483	.65149	2.68349
3.92141	1.00655	15.78514	.64066	2.66770
4.01322	.92102	14.11339	.55970	2.65981
4.10717	.80758	12.09209	.46857	3.12547
4.20333	.83069	12.15353	.45018	4.40408
4.30173	.83537	11.94245	.44165	4.38029
4.40244	.87142	12.17284	.44007	4.13573
4.50551	.96400	13.15804	.46400	3.21229
4.61099	1.07127	14.28769	.49316	3.19651
4.71894	1.08680	14.16325	.47768	2.96762
4.82942	1.08692	13.84079	.45613	2.95973
4.94248	1.06523	13.25427	.42681	2.95184
5.05819	1.01398	12.32796	.38790	3.03865
5.17661	.97560	11.59006	.35634	5.16966
5.29780	.96258	11.17371	.33568	2.81766
5.42183	.99025	11.23198	.32971	2.80977
5.54876	.91838	10.17852	.29195	2.80188
5.67867	.90243	9.77290	.27390	5.73793
5.81161	.93140	9.65596	.26901	5.73003
5.94767	.91804	9.49227	.25401	6.13255
6.08691	.82557	8.34096	.21809	2.50196
6.22942	.89140	8.79999	.22483	6.18780
6.37526	.95765	9.23777	.23062	5.02759
6.52451	1.04244	9.82565	.23968	5.01970
6.67725	.87896	8.09517	.19295	5.01181
6.83358	.81647	7.34763	.17113	2.12311
6.99357	.85282	7.49926	.17066	5.84842
7.15729	.82537	7.09182	.15770	5.84053
7.32486	.90928	7.63407	.16587	2.11522
7.49634	.96847	7.94502	.16858	2.49406
7.67184	.98273	7.87753	.16342	5.00391
7.85145	1.27893	10.01738	.20306	5.43011
8.03526	1.53340	11.73582	.23245	5.83264
8.22338	2.05423	[illegible].36226	.29732	5.82474

HORIZ... RESPONSE SPECTRA
NODE POINT= 1 EQUIPMENT DAMP. = 2.00000E-02
RESPONSE SPECTRA IN G.S



8.41590	[illegible]	[illegible]	.32034	[illegible]
8.61293	1.96136	[illegible]	.25878	6.51140
8.81457	1.61590	11.27379	.20356	5.25884
9.02093	1.36660	9.31638	.16437	5.61164
9.23212	1.11788	7.44644	.12837	5.26437
9.44826	.99124	6.45187	.10868	5.25648
9.66946	.85705	5.45080	.08972	3.68585
9.89583	.84326	5.24044	.08428	4.90920
10.1275:	.84926	5.15696	.08104	4.90131
10.36461	.99768	5.91967	.09090	4.90131
10.60726	1.10937	6.43179	.09650	5.25648
10.85559	1.22004	6.91159	.10133	5.24858
11.10973	1.76290	9.75846	.13980	5.24858
11.36983	2.17686	11.77427	.16482	5.58797
11.63601	1.86684	9.86644	.13495	5.79317
11.90843	1.34146	6.92758	.09259	5.13020
12.18722	1.11746	5.63879	.07364	5.13020
12.47254	.89105	4.39346	.05606	5.73793
12.76454	.80627	3.88448	.04843	5.73793
13.06338	.86343	4.06472	.04952	5.69057
13.36921	.78445	3.60843	.04296	5.69057
13.68220	.74923	3.36759	.03917	4.99602
14.00252	.80105	3.51810	.03999	5.68268
14.33034	1.05566	4.53029	.05031	5.68268
14.66583	.98437	4.12771	.04479	5.67478
15.00918	.79694	3.26534	.03463	5.48536
15.36056	.65880	2.63758	.02733	5.48536
15.72018	.59975	2.34622	.02375	5.48536
16.08821	.57611	2.20219	.02179	5.47747
16.46486	.58041	2.16788	.02096	5.47747
16.85032	.60562	2.21029	.02088	4.83817
17.24481	.64624	2.30457	.02127	4.83028
17.64854	.49997	1.74219	.01571	4.83028
18.06171	.45895	1.56266	.01377	4.83028
18.48456	.43173	1.43636	.01237	4.83028
18.91731	.40753	1.32482	.01115	2.47828
19.36019	.40474	1.28564	.01057	5.13009
19.81344	.40089	1.24428	.00999	5.13809
20.27730	.38418	1.16514	.00915	5.48536
20.75202	.37429	1.10920	.00851	5.48536
21.23786	.36589	1.05950	.00794	5.48536
21.73507	.35913	1.01613	.00744	5.48536
22.24391	.35278	.97532	.00698	5.48536
22.76468	.34777	.93948	.00657	5.48536
23.29763	.34344	.90656	.00619	5.48536
23.84306	.33976	.87633	.00585	5.48536
24.40126	.33665	.84844	.00553	5.48536
24.97253	.33369	.82174	.00524	5.48536

POOR ORIGINAL

```
CALCOMP SPECTRA P[illegible] READY
TITLE    *HORIZONTAL [illegible]SPONSE SPECTRA
NODE NO.*     1
DAMPING *     2.00000E+00  PERCENT
          PEAK G.S *     2.31811E+00
          MAX SCALE*     2.70000E+01  G.S
          FIRSTY    *    0.            G.S
          DELTAA    *    4.90909E-01  G.S/INCH
          FIRSTX    *    0.
          DELTAX    *    1.00000E+00
          IPOWER    *   -1

SPECTRA FOR DOF     1
NO. OF INPUT RECORDS       *        888
FINAL DISPLACEMENT (IN.)*       .14864E+01
FINAL VELOCITY (IN/SEC) *       .16256E+01
PEAK ACCELERATION (G.S) *       .29773E+00

NO. OF RECORDS ADDED       *         90
FINAL NO. OF RECORDS       *        978
FINAL VELOCITY (IN/SEC) *     0.
FINAL DISPLACEMENT (IN.)*       .15989E+01
TMAX. (DURATION, SECS)     *    .15422E+01
```

U.S. GOVERNMENT PRINTING OFFICE: 614-090-#33