MAY 22 1985

UCID- 20413 Vol. 1

**SMACS - A System of Computer Programs
for Probabilistic Seismic Analysis
of Structures and Subsystems**

Volume I

USER'S MANUAL

**Oleg R. Maslenikov*
James J. Johnson*
Lu Woon Tiong*
Michael J. Mraz*
Stan Bumpus
Michael A. Gerhard**

***Structural Mechanics Associates**

March, 1985

**Lawrence
Livermore
National
Laboratory**

This is an informal report intended primarily for internal or limited external distribution. The opinions and conclusions stated are those of the author and may or may not be those of the Laboratory.

This work was supported by the United States Nuclear Regulatory Commission under a Memorandum of Understanding with the United States Department of Energy.

DISTRIBUTION OF THIS REPORT IS UNLIMITED

TABLE OF CONTENTS

FOREWORD	v
PART I: OVERVIEW	
CHAPTER 1. INTRODUCTION.	1
CHAPTER 2. ANALYTICAL APPROACH OF SMACS.	3
CHAPTER 3. APPLICATION OF SMACS.	6
CHAPTER 4. REFERENCES.	10
PART II: PROGRAM USER'S MANUALS	
APPENDIX A - Program GLAY User's Manual	A-1
APPENDIX B - Program CLAN User's Manual	B-1
APPENDIX C - Program INSSIN User's Manual	C-1
APPENDIX D - Program SAPPAC User's Manual	D-1
APPENDIX E - Program LNGEN User's Manual	E-1
APPENDIX F - Program SMAX User's Manual	F-1
APPENDIX G - Program PRESTO User's Manual	G-1
APPENDIX H - Program CHANGO User's Manual	H-1

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

The following pre-processor codes are used to create necessary input files for SMAX.

- GLAY and CLAN for soil/foundation SSI characteristics.
- INSSIN for projection of dynamic structural properties to the foundation. Used in conjunction with program GEMINI.
- SAPPAC for projection of multiply-supported subsystem dynamic properties to support points on major structures. Used in conjunction with program GEMINI.
- LGEN for generating the experimental design.

The post-processor codes are:

- PRESTO for calculating and plotting probability distributions for individual response and basic events. Produces the response vectors used by program SEISM.
- CHANGO for plotting comparisons of basic events from different analyses.

This manual is presented in two volumes. Volume I, contained herein, consists of the SMACS user's manuals. Part I of this volume briefly discusses the SMACS system, the nature of the problems that it solves, and the way that SMACS is applied to these problems. Part II contains manuals describing the functions of the different programs, the flow of data through them, and the input and output files associated with them. In Volume II there is an example problem illustrating a SMACS analysis of a nuclear power plant.

FOREWORD

The SMACS (Seismic Methodology Analysis Chain with Statistics) system of computer programs, one of the major computational tools of the Seismic Safety Margins Research Program (SSMRP), links the seismic input with the calculation of soil-structure interaction, major structure response, and subsystem response. The seismic input is defined by ensembles of acceleration time histories in three orthogonal directions. Soil-structure interaction and detailed structural response are then determined simultaneously, using the substructure approach to SSI as implemented in the CLASSI family of computer programs.

Uncertainty is explicitly considered in each step of the seismic methodology chain (SMC). The variability in the seismic input is incorporated in the calculations by using ensembles of time histories. In the remaining links of the SMC, selected parameters are varied to represent all other sources of uncertainty. In the SSI link, the selected parameters are the shear modulus and material damping in the soil; in the major structure and subsystem links, they are modal frequencies and damping properties.

The modus operandi of SMACS is to perform repeated deterministic analyses, each analysis simulating an earthquake occurrence. By performing many such analyses and by varying the values of the input parameters, we are able to account for the uncertainty inherent in any deterministic analysis. Parameter values for each simulation are sampled from assumed probability distributions according to a Latin hypercube experimental design. The user may specify values of the coefficients of variation (COV) for the distributions of the input variables.

At the heart of the SMACS system is the computer program SMAX, which performs the repeated SSI response calculations for major structure and subsystem response. This report describes SMAX and the pre- and post-processor codes, used in conjunction with it, that comprise the SMACS system.

2. ANALYTICAL APPROACH OF SMACS

SMACS was developed to link the seismic input with the calculation of soil-structure interaction, major structure response, and subsystem response. The seismic input is defined by ensembles of acceleration time histories in three orthogonal directions (two horizontal and one vertical) on the ground surface in the free-field. Soil-structure interaction and detailed structural response are determined simultaneously, using the substructure approach to SSI, as implemented in the CLASSI family of computer programs (Ref. 3). The CLASSI methodology permits very detailed models of the structure to be considered in the final step of the substructure approach--analysis of the coupled soil-structure system. Detailed structural response in the form of time histories and peak values of accelerations, displacements, and forces can be computed. These results serve two main purposes in an analysis. First, a measure of response (peak value) can be compared with a fragility description of the structure, or a portion of it, to assess its ability to perform its design function during or after the earthquake. For buildings, this function includes providing support for systems contained within. For vessels, it includes providing a leak-tight pressure boundary. Second, because structures act as transmitters of vibratory motion to the support systems of emergency and power-generation equipment, structural response becomes an input to the calculations of subsystem response.

SMACS likewise calculates the time-history response of subsystems. Piping systems are analyzed assuming independent piping-support motions obtained from the detailed structural response. Three methods of performing multisupport time-history analyses were implemented and evaluated: the pseudostatic-mode method, the absolute displacement formulation, and the datun-acceleration/relative-displacement formulation. The pseudostatic-mode method was selected as the most efficient for our purposes.

As mentioned earlier, uncertainty is involved in every link of the SMC. This is accounted for in SMACS by performing repeated analyses using different definitions of the seismic input and varying different system parameters according to a latin hypercube experimental design. For example, the frequency and damping values of a structure or a substructure vary with

material properties, modelling technique, and method of analysis. The uncertainties in these parameters may be represented by probabilistic distributions of values for structural frequency and damping. SMACS uses the calculated frequency and damping as the median value of these distributions and accounts for uncertainty through multiplicative factors that are distributed lognormally. The same approach is used for variations in the shear modulus and material damping of soil. Then for each repeated analysis the above distributions are sampled according to the experimental design and the factors are applied to the median values. The results of the repeated analyses are distributions of response on the foundations, in the structures and on the subsystems.

The computational approach of the SSMP, which embodies the concepts of systems analysis, is implemented in the computer program SEISIM. The input requirements for SEISIM include system event trees and fault trees represented by Boolean expressions; fragility descriptions of structures, components, and systems; the seismic responses to the range of earthquake levels under consideration; and the probability of occurrence for each earthquake level.

Since the dynamic responses determined by SMACS constitute one of the inputs to SEISIM, a compatibility between the responses and the fragility descriptions of structures, components, and systems must be maintained. The points for which responses are computed must therefore be selected according to the requirements of the fault trees and the fragility descriptions. Likewise, response quantities must be chosen to correspond to those on which the fragility descriptions are based: The fragilities of the structures and floor-mounted components may be expressed in terms of structural accelerations, the fragilities of valves mounted on piping systems can be tied to their local accelerations, and the fragilities of the pipes themselves can be characterized by peak resultant moments over the duration of the earthquake.

The SMACS response calculations are intended to be as realistic as possible, incorporating uncertainties in each link of the seismic methodology chain--seismic input, SSI, major structure response, and subsystem response. The largest source of variability in the seismic input is accounted for in the

calculations by using ensembles of time histories. In the remaining links of the SMC, variations in selected parameters are intended to represent all sources of variability. In the SSI phase, the selected parameters are the shear modulus and material damping in the soil; in the major structure and subsystem phases, they are modal frequencies and damping properties.

3. APPLICATION OF SMACS

SMACS is the collective name for a system of computer programs which perform multiple SSI analyses, calculating detailed user-specified foundation response, in-structure response and, using multiple support excitation, subsystem (e.g. piping) response. At the core of this system is the computer program SMAX, which performs the SSI response analyses. SMAX uses input data from five sources:

- Seismic input in the form of acceleration time histories,
- Soil/foundation characteristics in the form of impedance matrices and wave scattering vectors,
- Dynamic characteristics of the major structures,
- Dynamic characteristics of specified subsystems,
- Property variation data in the form of multiplication factors on nominal soil, structural and subsystem properties.

For all data sets except seismic input, the input data required by SMAX is generated by one or more of five pre-processing computer programs. Figure 1 is a flow diagram showing the programs that comprise SMACS and their dependency relationships. The five pre-processing computer programs for SMAX are:

- GLAY and CLAN, which generate the nominal impedance matrices and wave scattering vectors for surface-founded structures. These programs were originally developed by Prof. J. E. Luco (UCSD) and Prof. H. L. Wong (USC).
- INSSIN, which projects the dynamic properties of structures, such as those calculated in a modal extraction analysis using programs like GEMINI, to the foundation in the form of modal participation factors and mass matrices.
- SAPPAC, which projects the dynamic and pseudostatic properties of multiply supported piping systems to the support locations. Input data to SAPPAC typically can be obtained from modal extraction analyses and pseudostatic analyses using programs such as GEMINI.

1. INTRODUCTION

A nuclear power plant is designed to ensure the survival of safety-related systems, buildings and equipment during a safe shutdown earthquake. Conventionally, in this design process, structures and substructures are modelled and analyzed deterministically for a given earthquake at a specific site. However, there are several uncertainties involved in the process; namely, uncertainties in modelling, in selecting soil and structural material properties, in choosing the characteristics of the earthquake accelerogram and in the analytical procedure itself. To quantify these uncertainties, a trend towards a probabilistic analysis of nuclear power plants followed by a risk analysis has evolved. A notable example of the development and implementation of this technique is the Seismic Safety Margins Research Program (SSMRP), conducted by Lawrence Livermore National Laboratory (LLNL) for the USNRC (Refs. 1 & 2). Total risk of failure as computed in the SSMRP is obtained by considering the entire spectrum of possible earthquakes, the entire range of uncertainties in shear modulus and material damping values of soil, and the entire range of variations in modal frequencies and damping values in the structures and subsystems.

The approach to risk analysis is embodied in the "seismic methodology chain" (SMC) comprising the following steps:

- (1) Determination of seismic input characteristics for a given site;
- (2) Calculation of soil-structure interaction effects;
- (3) Calculation of major structure (e.g. building) response;
- (4) Calculation of subsystem (e.g. piping) response; and
- (5) Calculation of probability of failure.

The seismic input consists of the earthquake hazard at the site as defined by the seismic hazard function; that is, the relationship between the probability of occurrence and the size of an earthquake as well as a description of the free-field motion. The soil-structure interaction step transforms the free-field ground motion into basemat or foundation response, usually assuming that the basemats are rigid. After the basemat response is obtained, detailed in-structure response is computed in a straightforward manner. The time histories of acceleration at the subsystem support points

are then used to calculate responses in the piping systems by a multiple support excitation method.

To account for uncertainties, the analyses are repeated a number of times, each time varying certain parameters such as frequency or stiffness and damping. The resulting sets of responses are then defined by probability distribution functions. Calculation of probability of failure consists of determining fragility functions, identifying accident scenarios, and, finally, combining these with the probabilistic representation of response to calculate probabilities of failure of structures and components and groups of structures and components and the probability of radioactive release.

The computer program system, SMACS (Seismic Methodology Analysis Chain with Statistics), developed at LLNL for the SSMRP, performs the first four steps of the SMC, using the seismic input and structural data to calculate soil-structure interaction, major structure response, and subsystem response. It also performs the statistical analysis of the resulting responses. The probabilistic representation of the responses calculated in SMACS is used by systems analysis computer program SEISM to calculate probabilities of failure.

- LGEN, which can be used to generate the multiplication factors to be applied to the nominal soil, structural and subsystem properties for each of the response calculations to account for random variations of these properties. These multiplication factors are calculated according to a Latin hypercube experimental design.

Program SMAX uses the data generated by these programs, along with ensembles of acceleration time histories to perform the repeated SSI response calculations. The results from SMAX are in the form of peak values and time histories of accelerations, displacements and forces. In particular, a file of response vectors, consisting of maximum responses of specified parameters, is generated for further use in the systems analysis program SEISIM.

Program SMAX produces response data for a number of earthquakes. To facilitate in evaluation and processing these results, there are two programs which can be used.

- PRESTO, which performs statistical operations on the raw data from the response vectors that SMAX produces to calculate best fit lognormal distributions for each response location. PRESTO generates response vectors that are used by program SEISIM for its system analysis calculations. If the individual responses are correlated to basic events defined for the SEISIM analysis, then condensed results can be produced. These results are the maxima of all responses associated with each basic event for each earthquake.
- CHANGO, which manipulates the data produced by PRESTO to produce other results which may be of interest to the user.

Before executing SMACS, one must first perform a modal analysis of the given structures with a fixed base. This modal extraction can be done using the computer program GEMINI (Ref. 4). The resulting modal frequencies are used as the median values in SMACS. For seismic input a desired number of synthetic earthquakes can be generated by any of a number of computer programs. SIMQ (Ref. 5), which generates acceleration time-histories to match a given target spectrum, is an example. The soil-structure interaction part of SMACS solves the problem through the use of substructuring. It uses the CLASSI family of programs to divide the soil-structure interaction problem into simpler ones; namely, determination of Green's functions; determination of foundation impedances; and analysis of the coupled soil-structure system. These three problems are solved, and then the results are superimposed to give

the response of structures. Once the time history of responses at subsystem support points in the structures is obtained, responses of subsystems or piping can be calculated. One important set of SMACS output is the set of response vectors. These response vectors can be used along with fragility curves of the structural components and subsystems to complete a risk analysis that will determine probability of failure and the risk of radioactive release. For the mathematical theory and methodology involved in various links of SMACS, the reader should refer to the SSMRP report on SMACS (Ref. 6).

The remainder of this manual consists of appendices which describe in greater detail how problems are solved using the SMACS system of computer programs. Appendices A through H in Part II are user's manuals for each of the programs of the system. Each is intended to be a standalone document and contains a brief description of the function of the program, the flow of data through the program, program execution procedures and compatibility with other programs as well as detailed descriptions of the input files to be prepared by the user.

Following the individual user's manuals, in Part III, is an example problem showing the complete flow of data through the SMACS system. In designing this problem we attempted to produce an example that would exercise as many parts of the system as possible yet still remain simple enough that most of it could be executed at reasonable cost to the user.

Fig. 1 Flow of Data through the SMACS System of Computer Programs.

4. REFERENCES

1. Smith, P. D., Chuang, T. Y., Bernreuter, D. L., Shieh, L. C., Bohn, M. P., Wells, J. E., Johnson, J. J., "Seismic Safety Margins Research Program - Phase I Final Report," Lawrence Livermore National Laboratory, UCRL-53021, also USNRC NUREG/CR-2015, 10 Volumes, September, 1981.
2. Bohn, M. P., Shieh, L. C., Wells, J. E., Cover, L. C., Bernreuter, D. L., Chen, J. C., Johnson, J. J., Bumpus, S. E., Mensing, R. W., O'Connell, W. J., Lappa, D. A., "Application of the SSMRP Methodology to the Seismic Risk at the Zion Nuclear Power Plant," Lawrence Livermore National Laboratory, UCRL-53483, also USNRC NUREG/CR-3428, January, 1984.
3. Wong, H. L., and Luco, J. E., "Soil Structure Interaction: A Linear Continuum Mechanics Approach (CLASSI)," Dept. of Civil Engineering, University of Southern California, CE79-03, 1980. (Draft)
4. R. C. Murray, "GEMINI - An Efficient Computer Program for Three Dimensional Linear Static and Seismic Structural Analysis," doctoral dissertation, University of California, Davis.
5. Gasparini, D. and Vanmarcke, E. H., "Evaluation of Seismic Safety of Buildings, Report No. 2, Simulated Earthquake Motions with Prescribed Response Spectra," Massachusetts Institute of Technology, prepared for the National Science Foundation, January, 1976.
6. Johnson, J. J., Goudreau, G. L., Bumpus, S. E., and Maslenikov, O. R., "SSMRP Phase I Final Report - SMACS - Seismic Methodology Analysis Chain with Statistics (Project VIII)," Lawrence Livermore National Laboratory, UCRL-53021, Vol. 9, also USNRC NUREG/CR-2015, Vol. 9, September, 1981.

PART I

OVERVIEW

PART II
PROGRAM USER'S MANUALS

PROGRAM GLAY, VERSION 3.0

USERS MANUAL

Program Description

Program GLAY was developed by Prof. J. E. Luco (UCSD) and Prof. H. L. Wong (USC) to calculate Green's functions, describing steady state dynamic force/displacement relationships between source and observer points located on the surface of a layered visco-elastic halfspace. It is part of the CLASSI family of programs developed by them. The calculations are performed at specified frequencies and at uniform intervals out to a specified maximum distance from the source. For each frequency, a table of Green's functions amplitudes in polar coordinates is produced. The methodology on which the calculations are based is discussed in the CLASSI Users Manual by Luco and Wong (Ref. 1) and in greater detail in a doctoral dissertation by R. J. Appell (Ref. 2). The results of the calculations are used by Program CLAN to calculate impedance matrices and incident wave scattering vectors for surface foundations of arbitrary shape.

The soil properties that GLAY requires to define the visco-elastic halfspace are the shear wave velocity, mass density, Poisson's ratio, material damping ratio and thickness of each layer. Typically, these are the equivalent-linear strain-compatible free-field soil properties calculated using a program like SHAKE (Ref. 3) for a free-field surface excitation of the same amplitude as that for which the subsequent SSI analyses will be performed. For each user-specified frequency, GLAY calculates a table of Green's function polar amplitudes at a calculated uniform radial interval out to the user-specified maximum radius from the source. All values are normalized by the shear modulus in the top soil layer and the radius. The Green's function matrix which relates the displacements at an observer point to a set of point loads at the source point, can be obtained in cartesian coordinates by interpolation between the amplitudes at the tabulated radii and then by their appropriate combination and multiplication by the normalization factor.

Program Data Flow

Program GLAY requires one input data file, prepared by the user, called GLAYIN (default file name) containing the definition of the halfspace and the frequencies at which Green's functions are to be computed. GLAY produces two text output files called GLAYLG, which summarizes the input data and provides execution timing data, and GRNFNC, which contains the Green's function tables. File GRNFNC contains input data for program CLAN. Figure 1 gives a graphic description of the data flow. A detailed description of the data is given later in this manual.

Access and Execution on the NMFEC

Program GLAY may be accessed on the NMFEC by using the following FILEM command:

RDS 7205 . SMACS GLAY

Program GLAY may be executed on the NMFEC with the following command:

GLAY (I=GLAYIN,O=GLAYLG,G=GRNFNC) / t v

The expression in brackets may be used if files with names other than the default names (shown) are to be used.

Inter-Program Compatibility

This version of GLAY is compatible with program CLAN, Version 2.0.

References

1. H. L. Wong and J. E. Luco, "Soil Structure Interaction: A Linear Continuum Mechanics Approach (CLASSI)," Dept. of Civil Engineering, Univ. of Southern California, CE79-03, 1980. (Draft)
2. R. J. Apsel, "Dynamic Green's Functions for Layered Media and Applications to Boundary-Value Problems" doctoral dissertation, Dept. of Applied Mechanics and Engineering Science, Univ. of California, San Diego, CA, 1979.
3. P. B. Schnabel and J. Lysmer, "SHAKE - A Computer Program for Earthquake Response Analysis of Horizontally Layered Sites," Dept. of Civil Engineering, Univ. of California, Berkeley, CA, EERC 72-12, 1972.

Fig. 1 Data Flow through Program GLAY.

DESCRIPTION OF INPUT FILE "GLAYIN"

Heading Line (10A8)

Columns	Variable	Description	Notes
1-80	IDENT	Identifying header information to be printed with the output	(1)

NOTES

- (1) The time, date and machine letter are printed along with this heading identifier information at the beginning of the GLAYLG output file. This information aids in quality assurance in a sequence of runs involving soil-structure interaction.

Control Information (3I5,5X,F10.0)

Columns	Variable	Description	Notes
1-5	NLY	Number of layers in the soil profile	(1)
6-10	NFRQ	Number of frequencies in the Green's function table	(2)
11-15	NITER	Number of iterations for convergence	(3)
21-30	RE	Radius of extreme foundation point	(4)

NOTES

- (1) NLY must be strictly greater than 1. If users are modelling an elastic half space, NLY can be input as 2 and two identical soil layers defined. NLY soil property sets are read in the next series of cards.
- (2) NFRQ is the number of frequencies to be read in. These frequencies will become the discretized points for which the Green's functions are calculated.
- (3) If a Green's function table did not converge to within a calculated tolerance in NITER iterations, a message is written to the GLAYLG file giving the average iteration error, the results are written to the GRNFNC file, and the calculations continue.
- (4) The value RE must be equal to or greater than the maximum distance between any two points on the foundation(s).

Soil Properties (5F10.0)

In this next section, NLY soil property lines must be provided. The first line describes the soil properties of the topmost (reference) layer, the second line describes the soil properties of the second layer, etc. The last (NLY-th) line describes the soil properties of the underlying semi-infinite halfspace. Values must be in consistent units.

Columns	Variable	Description	Notes
1-10	VS	Shear wave velocity	
11-20	RHO	Mass density	
21-30	POISSON	Poisson's ratio	
31-40	DAMP	Material damping ratio	(1)
41-50	THICK	Layer thickness	(2)

NOTES

- (1) DAMP must be expressed as a decimal fraction, not percent.
- (2) Because the last layer is the semi-infinite half space, THICK need not be specified and is assumed to be 10000.

Frequencies of Discretization (8F10.4) Provide NFRQ values

Columns	Variable	Description	Notes
1-10	FREQS(I)	Frequency at which to compute	(1)
11-20	I=1,NFRO	Green's function table	
71-80			

NOTES

- (1) Users must enter a total of NFRQ frequencies (Hz) in this series, 8 values to a line. The frequencies must be in increasing order. If the first frequency is zero, the program replaces it with the value of 0.01 Hz.

DESCRIPTION OF OUTPUT FILE "GRNFNC"

File GRNFNC contains the Green's function tables calculated by GLAY and is used by program CLAN to calculate subregion compliance matrices. Also contained in the file, for problem documentation and quality assurance purposes, is problem definition data including the properties of the soil layers and problem execution identification data (problem title and execution date and time).

Each Green's function table is preceded by the following control data:

NGRN = The number of radii at which the Green's function amplitudes are computed, including a program-defined point very near the source point.

DIV = The radial increment at which the Green's function amplitudes are computed, normalized by the factor ω/V_1 , where ω is the circular frequency (rad/sec) and V_1 is the shear wave velocity of the top layer

FREQ = The frequency (Hz) for which the calculation is performed

The Green's function table is constructed of a set of four complex values, GRR, GTR, GRZ, GZZ for each radial distance. These values represent, in polar coordinates, the displacement amplitudes and phase angles occurring at each radius, r , due to unit steady state point loads occurring at the point source at the frequency FREQ. At each radius, r , all four amplitudes are normalized by the factor $G_1 r$, where G_1 is the shear modulus in the top soil layer. The tabulated values are:

GRR = Radial displacement amplitude due to a unit horizontal force at the point source.

GTR = Tangential displacement amplitude due to a unit horizontal force at the point source.

GRZ = Radial displacement amplitude due to a unit vertical force at the point source.

GZZ = Vertical displacement amplitude due to a unit vertical force at the point source.

DESCRIPTION OF OUTPUT FILE "GLAYLG"

File GLAYLG is a text log file for information only. It contains the program compilation and job execution dates and times for documentation purposes, an echo-print of the data contained on the GLAYIN file, a summary of the soil properties as used by the program, any execution error messages, and timing information.

APPENDIX A

PROGRAM GLAY

USER'S MANUAL

November 1984

Oleg R. Maslenikov

NTS/Structural Mechanics Associates
Two Annabel Lane, Suite 101,
San Ramon, CA 94583

APPENDIX B

PROGRAM CLAN

USER'S MANUAL

November 1984

Oleg R. Maslenikov

NTS/Structural Mechanics Associates
Two Annabel Lane, Suite 101,
San Ramon, CA 94583

PROGRAM CLAN, VERSION 2.0
USERS MANUAL

Program Description

Program CLAN was developed by Prof. J. E. Luco (UCSD) and Prof. H. L. Wong (USC) to calculate dynamic soil impedance matrices and incident wave scattering vectors for multiple rigid surface foundations of arbitrary shape, including through-soil coupling. It is part of the CLASSI family of programs developed by them (Ref. 1). CLAN uses the Green's functions calculated by program GLAY, along with a user-defined discretized model of the rigid soil/foundation interface. Figure 1 shows an example of a model of the foundation geometry of a typical nuclear power plant including the discretization of the three foundations into a number of subregions. At a given frequency, the force/displacement relationships between each pair of subregions are calculated by integration of the Green's functions over the subregion areas. The impedance matrix is then calculated by applying the constraints of rigid body motion between all subregions on the same foundation. Wave scattering vectors are calculated by an extension of this procedure. A more detailed discussion of the methodology used in CLAN can be found in the CLASSI Users Manual by Luco and Wong (Ref. 1).

Program Data Flow

Program CLAN requires two input data files for its execution, CLANIN and GRNFNC, and produces two output files, CLANLG and IMPFNC.

CLANIN is a text input file prepared by the user. It contains certain control information as well as the definition and discretization of the geometry of the rigid soil/foundation interface. The input data contained in this file is described in detail in subsequent pages of this manual.

GRNFNC is a text input file that is generated by program GLAY and contains the Green's functions tables. For a detailed description of GRNFNC, refer to the GLAY Users Manual.

CLANLG is a text output file produced by CLAN. It provides a summary of the computer run which includes execution time and date, program version, summary of foundation geometry and execution time. It is for the users information only.

IMPFNC is a text data file produced by CLAN. It contains the impedance and scattering matrices for the foundation. IMPFNC is used as an input data file for program SMAX. A more detailed description of IMPFNC is found in subsequent pages of this manual.

The flow of data through program CLAN is summarized in Figure 2. Detailed descriptions of the data contained in files CLANIN and IMPFNC are found later in this manual. A detailed description of file GRNFNC is found in the GLAY manual.

Access and Execution on the NMFECC

Program CLAN may be accessed on the NMFECC with the following FILEM command:

```
RDS 7205 .SMACS CLAN
```

Program CLAN may be executed on the NMFECC with the following command:

```
CLAN (I=CLANIN,O=CLANLG,G=GRNFNC) / t v
```

The expression in brackets should be used if files with names other than the default names (shown) are to be used.

Inter Program Compatibility

This version of CLAN is compatible with

Program GLAY, Version 3.0

Program SMAX, Version 2.0

References

1. H. L. Wong and J. E. Luco, "Soil Structure Interaction: A Linear Continuum Mechanics Approach (CLASSI)," Dept. of Civil Engineering, University of Southern California, CE79-03, 1980. (Draft)

Fig. 1 Foundation model of a typical nuclear power plant showing the discretization used to compute impedance functions.

Fig. 2 Data Flow through Program CLAN.

DESCRIPTION OF INPUT DATA FILE "CLANIN"

Problem Identification Line (10A8)

Columns	Variable	Description	Notes
1-80	IDENT	Heading identifier information to be printed with the output	(1)

NOTES

- (1) The time, date and machine letter are printed along with this heading identifier information at the beginning of the CLANLG output file. This information serves as quality control in a sequence of runs involving soil-structure interaction

Plot Control Information (6I5,F10.0,5X,3A10)

Data on this line controls the generation of plot files for the foundation model geometry and for the impedance and scattering matrix components as functions of frequency. If no plots are desired, provide one blank line.

Columns	Variable	Description
1-5	IFPLT	Indicator for plotting foundation models -1 = Plot foundation only, then terminate 0 = Do not plot foundations 1 = Plot foundations then continue with calculations
6-10	IFTYP	Indicator for type of foundation plot 0 = Plot all foundations on a single plot 1 = Plot each foundation on a separate plot 2 = Make plots both ways
11-15	IFLAB	Indicator for foundation plot labelling 0 = Include subregion numbers 1 = Omit subregion numbers
16-20	IMPLT	Indicator for impedance plots -1 = Plot precalculated impedances 0 = Do not plot impedances 1 = Plot impedances calculated in this run
21-25	IMNRM	Indicator for normalized plots 0 = Plot normalized impedances 1 = Plot impedances in physical units.

Columns	Variable	Description
26-30	IMPRT	Indicator for printing impedances 0 = Do not print impedances in CLANLG 1 = Print impedances in CLANLG
31-40	FQCUT	Lower cut-off frequency for plotting impedances.
46-75	IDBOX	30-character box and ID to be included in plot file.

Problem Control Information (F10.0,3I5)

Columns	Variable	Description	Notes
1-10	CL	Characteristic length	(1)
11-15	NFDN	Number of foundations	
16-20	NFREQ	Length of frequency list	(2)
21-25	NWAVE	Number of incident wave cases	

NOTES

- (1) CL is used to make both the frequencies and impedances dimensionless. If CL is chosen to represent reasonably an overall dimension of the foundation (that is, the equivalent radius of the foundation or the length of the longest side of a rectangle), the impedance values will usually be near the range 1 to 10.
- (2) The number input for NFREQ is the length of the list of frequencies to be read in on the next set of input lines. If NFREQ is left blank or input as zero, all the frequencies from the GLAY run will be used and the user should not input any frequency list denoted LISTFQ on the next line series. If NFREQ is input as a positive value, it must not be larger than the number of frequencies used in the GLAY run.

Frequency List (10I5)

Columns	Variable	Description	Notes
1-50	LISTFQ	List of frequency numbers	(1)

NOTES

- (1) This line (series) is read only if NFREQ is input as a positive number on the previous line. Ten frequency numbers are read per line. Users must provide enough lines to make up the NFREQ items that the program expects to read here. Each number in the list should correspond to the sequential number of the frequency used in the GLAY run. These sequence numbers may be found in the GLAYLG and GRNFNC files. The list does not have to be in any particular order; however, CLAN will generate results in the order of increasing frequencies.

FOUNDATION INFORMATION

The following data sets, denoted FOUNDATION CONTROL, GEOMETRY REFERENCE AND SCALING, and SUBREGION GEOMETRY, must be provided NFDN times, once for each foundation. It should be noted that, for the definition of foundation geometry, the z-axis is always assumed to be vertical.

Foundation Control Information (315)

Columns	Variable	Description	Notes
1-5	NREG	The number of rectangular subregions used to model the foundation shape	(1)
6-10	ISYMX	Indicator of foundation symmetry about the local xz-plane	(2)
11-15	ISYMY	Indicator of foundation symmetry about the local yz-plane	(3)

NOTES

- (1) NREG is the number of foundation subregions actually included in the data file, regardless of symmetry indications.
- (2) If ISYMX is zero, no symmetry about the xz-plane is indicated. Otherwise, symmetry is invoked.
- (3) If ISYMY is zero, no symmetry about the yz-plane is indicated. Otherwise, symmetry is invoked.

Geometry Reference and Scaling Information (4F10.0)

Columns	Variable	Description	Notes
1-10	XREF	Global X-coordinate of the foundation reference point	(1)
11-20	YREF	Global Y-coordinate of the foundation reference point	(1)
21-30	QREF	The reference angle of rotation of the foundation	(2)
31-40	SCALE	Scale factor to apply to XREF, YREF and the subsequent subregion geometry	(3)

NOTES

- (1) The local reference point is located relative to the global reference axes, which are used in the structural definition. All subregion geometric data is given the local reference point. The impedance and scattering matrices are calculated so as to apply at the reference point. Each foundation reference point also defines the location of foundations relative to one another.
- (2) The reference angle is measured counter-clockwise between the foundation local x-axis and the global X-axis. Non-zero values for QREF have not been verified.
- (3) If SCALE is input as zero or left blank, a default value of 1.0 is used.

Subregion Geometry (4F10.0)

Columns	Variable	Description	Notes
1-10	XMIN	Minimum local x-coordinate of a subregion	(1,2)
11-20	XMAX	Maximum local x-coordinate of a subregion	
21-30	YMIN	Minimum local y-coordinate of a subregion	
31-40	YMAX	Maximum local y-coordinate of a subregion	

NOTES

- (1) All coordinates are relative to the foundation reference point. A total of NREG lines are input in this series. Each line describes one rectangular subregion.
- (2) The user should ensure that the maximum diagonal distance between any two points on any two subregions, including subregions on different foundations, is less (usually at least by ten percent) than the maximum distance which Green's functions are calculated by program GLAY.

Incident Wave Description (8F10.0)

The following data must be provided NWAIVE times, once for each incident wave case. Each planar incident earthquake wave which impinges on the foundation(s) is defined by a vertical angle and a horizontal angle, defining the direction of approach, and a definition of the components associated with that wave. Figure 3 provides an illustration of this. In program SMAX, each incident wave is paired with a free-field acceleration time history. For a typical earthquake problem, where three independent components of free-field earthquake motion are used, each component is usually associated with a single incident wave, so that three incident wave cases are defined for the CLAN problem. Typically, these are in the order of x-, y-, and z-components. It should be noted that more incident wave cases (NWAIVE) can be defined in the CLAN run, than are used in a single SMAX analysis (incident wave data can be generated for multiple SMAX analyses). However, the wave cases used for the same SMAX analysis must be defined consecutively. The incident wave definitions are applied to all frequencies.

Columns	Variable	Description	Notes
1-10	BOC	The ratio of the reference shear wave velocity in the soil profile (VS in GLAY) to the apparent wave velocity	(1)
11-20	TH	Horizontal angle that specifies the propagation direction of the incident wave	(2)
21-30 31-40	UL(Re) UL(Im)	The complex amplitude of the longitudinal (horizontal radial) wave component	(3)
41-50 51-60	UT(Re) UT(Im)	The complex amplitude of the transverse wave component	(3)
61-70 71-80	UV(Re) UV(Im)	The complex amplitude of the vertical wave component	(3)

NOTES

- (1) The apparent wave velocity is the velocity at which the incident wave appears to propagate on the ground surface; thus the ratio BOC is equal to the cosine of vertical angle θ in Figure 3. Note that for vertically propagating waves, the apparent wave velocity is infinite and the value of BOC is zero.
- (2) The angle is measured counter-clockwise from the global X-axis.
- (3) UL, UT, and UV are vibratory components propagating along the line from the earthquake source to the reference point. Figure 3 shows their directions of vibration. For vertically incident waves, UL, UT, and UV are each represented by the complex number (1.0, 0.0). For three independent components of vertically incident input motion UL, UT, and UV are individually set to (1.0, 0.0) while the other two values are set to (0.0, 0.0). In this case, three lines (NWAVE equals 3) must be provided.

Fig. 3 Convention for Definition of Incident Wave Components.

DESCRIPTION OF THE IMPEDANCE FUNCTION OUTPUT FILE "IMPFNC"

This section describes the structure of the impedance function file IMPFNC that CLAN produces. The file IMPFNC contains descriptive headers for accountability, control information to pass to SMAX, the original soil data from GLAY, frequency use information, and finally the normalized dimensionless impedance function tables. The descriptive headers, control information and the frequency use table are also output to the CLANLG output file. Each of these areas will be described briefly below.

Descriptive Headers

The first four lines of the IMPFNC file contain the CLAN execution date and time, the heading identifier line from CLAN, the GLAY execution date and time, and the heading line from GLAY respectively. The GLAY information is obtained from the GRNFNC file.

Control Information

The next five non-blank lines contain the input characteristic length, the input number of foundations, the number of frequencies used from the GLAY list, the input number of incident wave cases, and the number of soil layers from GLAY respectively.

Soil Data

The next data is the layered soil data echo printed from the GRFNC file.

Frequency Use Information

Following the soil data, the IMPFNC file has a list of the frequencies used in the calculation of the Green's function tables. Included with this list is a one word descriptor, either "yes" or "no", which indicates whether an impedance matrix and scattering vectors were calculated for that frequency

Impedance and Scattering Matrix Tables

Following the documentation data described above are tables of dimensionless impedance and wave scattering matrices calculated at each frequency specified by the user. Each table is preceded by the frequency, f , at which the calculation was made, and its corresponding dimensionless frequency, a_0 . The dimensionless frequency is related to f by the equation

$$f = \frac{V}{2\pi\ell} a_0$$

where V is the shear wave velocity of the reference soil layer (VS in program GLAY) and ℓ is the characteristic length, CL.

The dimensionless, complex impedance matrix, $[k]$, is NTDOF x NTDOF in size, where NTDOF is the total number of foundation degrees-of-freedom. For this version of CLAN, NTDOF equals six times NFDN, the number of foundations. Matrix $[k]$ is related to the impedance matrix in physical units, $[K]$, by the relationship

$$[K] = G\ell [D]^T [k] [D]$$

where G is the shear modulus in the reference soil layer and $[D]$ is the matrix

$$D = \begin{bmatrix} L & L & . & . & . & . \\ L & L & & & & \\ . & & & & & \\ . & & & & & \\ . & & & & & \\ . & & & & & \end{bmatrix} \quad L = \begin{bmatrix} 1 & 0 & . & . & . & 0 \\ 0 & 1 & & & & . \\ . & & 1 & & & . \\ . & & & \ell & & 0 \\ . & & & & \ell & 0 \\ 0 & . & . & . & 0 & \ell \end{bmatrix}$$

NFDN x NFDN 6 x 6

The dimensionless scattering matrix, $[s]$, is NDOF x NDOF in size. It is related to the scattering matrix, $[S]$, given in physical units, by the relationship

$$[S] = [D^*]^T [s]$$

where

$$D^* = \begin{bmatrix} L^* & L^* & . & . & . & . \\ L^* & L^* & & & & \\ . & & & & & \\ . & & & & & \\ . & & & & & \\ . & & & & & \end{bmatrix}, \quad L^* = \begin{bmatrix} 1 & 0 & . & . & . & 0 \\ 0 & 1 & & & & . \\ . & & 1 & & & . \\ . & & & 1/\ell & & . \\ . & & & & 1/\ell & 0 \\ 0 & . & . & . & 0 & 1/\ell \end{bmatrix}$$

NDOF x NDOF

6 x 6

PROGRAM INSSIN, VERSION 2.0

USERS MANUAL

Program Description

Program INSSIN was developed at LLNL to project the dynamic characteristics of a structural model to a specified reference point, usually on the foundation. It was originally developed to provide structural data for CLASSI program SSIN (Ref. 1), which performs soil-structure interaction (SSI) response calculations. Typically, the model is a detailed dynamic model of a large structure such as a reactor building or auxiliary building of a nuclear power plant. Although simpler models may be used, the structural model can be of such detail that its inclusion in a SSI analysis would be prohibitively expensive (if not impossible) by any method other than a substructure technique. Figure 1 shows a large half-model of a structure that was processed by INSSIN and used in a SSI response calculation.

INSSIN was designed to receive input data from an eigenvalue extraction analysis of the structural model, using finite element computer programs such as GEMINI (Ref. 2). INSSIN is currently coded to accept this data directly without any data manipulation. The dynamic data that INSSIN uses includes the fixed-base structural mode shapes and frequencies, mass matrix, nodal point geometry and degree-of-freedom/equation-number vector ("I.D. array"), and element stress/displacement matrices. Modal damping ratios are specified separately by the user. Equation numbering need not be in order of increasing node point numbers but may be the result of an optimization scheme. INSSIN uses this data to calculate the modal participation factors and the rigid-body mass matrix of the structure at the reference point for the six degrees-of-freedom defining motion on the rigid foundation. INSSIN also produces modal component vectors and rigid body transformation vectors for in-structure acceleration, displacement, force or stress components specified by the user.

The projected dynamic structural data produced by INSSIN can be used in SSI response calculations using programs such as SMAX or CLASSI program SSIN for which it was originally designed. These two programs have a limitation on

the coordinate system used in the analysis. This limitation is that the z-axis of the structural model must be vertical. Through appropriate manipulation of INSSIN output data, partial structural models (horizontal/vertical or symmetric/antisymmetric models) can be combined to produce full models having six foundation degrees-of-freedom for SSI analysis. When using INSSIN with SMAX or SSIN, the user should be careful to ensure that the reference point used in INSSIN is at the same location as the foundation reference point used in program CLAN to generate foundation impedance and scattering data.

Program Data Flow

Program INSSIN requires data from four input data files and produces two output files. Figure 2 shows the flow of data.

INSNIN is a text input file prepared by the user. It contains certain control information, the location of the reference point, modal damping ratios, and locations in the structure at which acceleration, displacement or force response is desired. A detailed description of this input data is found later in this manual.

GEMI is a text input file and is the user-prepared input file for the GEMINI eigenvalue extraction analysis. INSSIN reads structural model nodal point geometry from this file. Detailed descriptions of GEMI and TAPE10 may be found in the GEMINI users manual.

TAPE10 is a sequential binary file produced by GEMINI. It contains the results of the eigenvalue extraction. Detailed descriptions of GEMI and TAPE10 may be found in the GEMINI users manual.

TAPE1 is a sequential binary file produced by GEMINI. It contains the element stress/displacement matrices for the model and is used only when element force responses are desired.

INSNLG is a text print file produced by INSSIN. It summarizes problem execution and can contain a number of data sets depending on what the user specifies in the INSNIN file. It is for information only.

SSINST is a text data file produced by INSSIN. It contains modal frequencies, damping ratios and participation factors and the structure's rigid-body mass matrix for six degrees-of-freedom at the foundation reference point. Also included in SSINST are the modal component vectors and rigid-body transformation vectors for the user-specified response quantities, such as acceleration, displacement or force, for which results from the SSI response analysis are desired. SSINST is used as an input file to program SMAX.

Access and Execution on NMFECC

Program INSSIN can be accessed on the NMFECC with the following FILEM command:

RDS 7205 .SMACS INSSIN

Program INSSIN can be executed on the NMFECC with the following command:

INSSIN (I=INSNIN,S4=,GEMI,T1=TAPE1,T10=TAPE10,O=INSNLG,S=SSINST) / t v

For files with other than the default names (shown), append the appropriate expression in brackets.

Inter-program compatibility

This version of INSSIN is compatible with
Program GEMINI, Version
Program SMAX, Version 2.0

References

1. H. L. Wong and J. E. Luco, "Soil Structure Interaction: A Linear Continuum Mechanics Approach (CLASSI)," Dept. of Civil Engineering, University of Southern California, CE79-03, 1980. (Draft)
2. R. C. Murray, "GEMINI - An Efficient Computer Program for Three Dimensional Linear Static and Seismic Structural Analysis," doctoral dissertation, University of California, Davis.

Fig. 1 Finite element half-structure model of a typical Auxiliary/Turbine Building. The inset sketch shows the portion of the structure modeled.

Fig. 2 Data Flow through Program INSSIN.

DESCRIPTION OF INPUT FILE "INSSIN"

Record 1 (8A10): Model Identification

Columns	Variable	Description
1-80	IDENT	Identifying label that will be written to the SSINST data file. If this line is left blank, the identifier on the TAPE10 file will be used.

Record 2 (4X,A1): Vertical Axis Designation

This version of INSSIN accepts only the Z-axis as the vertical axis of the GEMINI model. PROVIDE ONE BLANK LINE.

Record 3 (11I3): Output File Options

List as many options as desired in any order.

Columns	Variable	Description
1-3	KSW(I)	1 = Write the node point coordinates to the INSLOG print file GEMINI
.		2 = Write the GEMINI ID-array to the print file. This array is used to find the equation numbers of given degrees-of-freedom as described by node point number and component.
.		3 = Write the inverse ID-array to the print file. This array performs the inverse function of obtaining nodes and components from the equation numbers.
.		4 = Write the mass matrix XM to the print file.
.		5 = Write the modal frequencies (rad/sec) and damping ratios to the print file.
31-33		6 = Write the modal participation factors (BETA matrix)
		7 = Write the rigid body mass matrix (BMASS) to the print file.
		8 = Write the stress-displacement matrix (SA) of the elements for which stress or force responses will be obtained.
		9 = Write the modal coordinates for all degrees-of-freedom connected to elements for which stress or force responses will be obtained.

- 10 = Write the SMODE and ALKP vectors for all responses.
 The SMODE vector consists of the modal coordinates of the degrees-of-freedom for which response is desired. Similarly, "pseudo-modal" coordinates are computed for specified element force components for which response is desired.
 The ALKP vector defines the rigid body motion at a given degree-of-freedom caused by unit motions at the foundation reference point. For motion relative to the foundation, the ALKP vector is zero.
- 11 = Write the complete ALFA matrix, which defines the rigid body motions of all degrees-of-freedom induced by unit motions of the rigid basemat.

Record 4 (715): Basemat Degrees-of-Freedom

This version of INSSIN accepts only models having 6 foundation degrees-of-freedom. PROVIDE ONE BLANK LINE.

Record 5 (3E10.3): Location of Foundation Reference Point

Columns	Variable	Description
1-10	XREF	X-coordinate relative to origin of model
11-20	YREF	Y-coordinate relative to origin of model
21-30	ZREF	Z-coordinate relative to origin of model

Record 6 (8F10.5): Modal Damping Ratios

Columns	Variable	Description
1-10	DAMP(J)	The damping ratio to be used for each mode in the subsequent SMACS run. If a single damping ratio is specified, that value will be used for all modes. A damping ratio of 0.0 can not be input; users must input some small value such as 0.001.
11-20	J=1,NF	
.		
.		
71-80		

Record 7 (3I5,1X,3A4,12X,I5,2F10.0): Kinematic Response Requests

Provide one line for each kinematic response component desired. Responses can be in any order. Multiple responses for the same component may be specified.

Columns	Variable	Description
1-5	NP	Node point number
6-10	ICOMP	Component number. The convention is the same as for the basemat components: 1 = X-translation 4 = XX-rotation 2 = Y-translation 5 = YY-rotation 3 = Z-translation 6 = ZZ-rotation
15	ITYPE	The type of response desired. This is a control variable that is passed to program SMAX via file SSINST. 0 = Absolute acceleration +1 = Absolute velocity -1 = Relative velocity +2 = Absolute displacement -2 = Relative displacement
17-28	LBL	Identifier up to 12 characters. INSSIN will embed the node number and component in the first 12 characters of the label output to the SSINST file.
45	IUSE	Flag indicating what measure of response is to be calculated (default=0). This is a control variable that is passed to program SMAX via the SSINST file. According to the value of IUSE, SMAX will calculate the appropriate measure of response and output the result to one of its output data files. 0 = Only the peak response will be output by SMAX to the RVEnn file 1 = The complete time history will be output by SMAX to the SPEnn file 2 = The peak response and spectral response will be output by SMAX to the RVEnn file. 3 = The spectral response only will be output by SMAX to the RVEnn file. 4 = The peak response and spectral responses at 4, 8, and 16 Hz will be output by SMAX to the RVEnn file.
46-55	FRQ	Frequency at which a spectral response is to be computed. The value of IUSE must be at least 2.
56-65	DMP	Damping ratio for which the above spectral response is to be computed.

Record 8 (2I5,6X,3A4,12X,15): Force or Stress Response Requests

Provide one line for each element force or stress component for which SMAX output is desired. Requests are grouped by element group; all element groups must be included. A group consists of a series of element force component request lines terminated by a blank line. Element numbers within an element group (truss, say) must be in ascending order. Element groups are processed in the same order as the original GEMINI element data. If no output is to be produced for an element group, then input one blank line for that group.

Columns	Variable	Description
1-5	NEL	Element number
6-10	II	Component number. The convention is the same as that used by GEMINI
17-28	LBL	Identifying label up to 12 characters. The element and force component numbers will be embedded by INSSIN in the first 12 characters of the label output to the SSINST file.
45	IUSE	Flag indicating what measure of response is to be calculated (default = 0). This is a control variable that is passed to program SMAX via the SSINST file. According to the value of IUSE, SMAX will calculate the appropriate measure of response and output the result to one of its output files. 0 = Only the peak response will be output by SMAX to the RVEnn file. 1 = The complete time history will be output by SMAX to the SPEnn file. 2 = The peak response will be output by SMAX to the RVEnn file.

APPENDIX C

PROGRAM INSSIN

USER'S MANUAL

November 1984

Oleg R. Maslenikov

NTS/Structural Mechanics Associates
Two Annabel Lane, Suite 101
San Ramon, CA 94583

PROGRAM SAPPAC, VERSION 2.0
USERS MANUAL

Program Description

Program SAPPAC is a linear dynamic response program developed by Dr. G. L. Goudreau (LLNL) for modal analysis of structural systems (Ref. 1), including multi-support excitations based on the method of "pseudostatic modes." SAPPAC uses the results of an eigenvalue extraction and a static support displacement analysis of the system by program GEMINI (Ref. 2) in its response calculations. When operating in conjunction with the SMACS system of programs, SAPPAC is used only to project the dynamic properties of a subsystem, such as a piping system, to its support locations; it acts as a preprocessor. Program SMAX uses this data, along with the subsystem support motions calculated in its major structure response phase, to obtain responses of interest in the subsystem. This manual describes how to obtain the subsystem data for use in SMAX. The user is referred to the more extensive SAPPAC manual (Ref. 1) for operation of all of SAPPAC's options. This version of SAPPAC differs from the version in Reference 1 in that the option to plot transfer functions has been removed because of required external routines not available outside LLNL.

Program Data Flow

Program SAPPAC requires data from up to six input data files and produces one output file. Figure 1 shows the flow of data.

SAPPACI is a text input file prepared by the user. It contains control information, and locations in the subsystem at which acceleration, displacement or force response is desired. A detailed description of this input data is found later in this manual.

TAPE10 is a sequential binary file produced by GEMINI. It contains results of the eigenvalue extraction. Detailed descriptions of TAPE10 may be found in the GEMINI users manual.

TAPE1 is a sequential binary file produced by GEMINI. It contains the element stress/displacement matrices for the subsystem model and is used only when element force responses are desired. Detailed descriptions of TAPE1 may be found in the GEMINI users manual.

TAPE8 is a sequential binary file produced by GEMINI. It contains support coupling stiffness matrices and element destination vectors. If only rigid body base translation (fixed base analysis) is to be prescribed, TAPE8 is not needed. Detailed descriptions of TAPE8 may be found in the GEMINI manual and the more extensive SAPPAC manual (Ref. 1).

TAPE9 is a sequential binary file produced by GEMINI. It contains the pseudostatic mode matrices required by the "pseudostatic mode" method of SAPPAC. If only fixed base excitation is desired ($NSUP=0$), this tape is not needed. A detailed description and the method of forming TAPE9 are found in the GEMINI user's manual.

TAPE2 is a sequential binary file produced by GEMINI. It contains element stiffness data and is used only when composite modal damping is desired. A detailed description and the method of forming TAPE2 are found in the GEMINI user's manual.

TAPE3 is a sequential binary file produced by SAPPAC. It contains control information, response request identification arrays, response request inertial and pseudostatic component recovery matrices, and the forcing function matrix. A detailed outline of variables and data in TAPE3 is given below.

SAPPACO is a text file produced by SAPPAC. It contains eigenvectors if requested and control data for user information.

SAPPAC reads TAPE10 modal data and response degree-of-freedom lists from SAPPAC1, and TAPE8 coupling stiffness data, TAPE9 pseudostatic modes and TAPE1 stress recovery data, if needed. Modal multi-support excitation matrix, modal participation factors, and response recovery matrices are computed. These compacted modal matrices are saved on TAPE3. If composite modal damping is desired, a separate GEMINI special data check must be run (with $MODEX=-1$ to save the TAPE2 element stiffness file, to be read by SAPPAC.

The GEMINI modal extraction analysis must generate a TAPE10 family of modal information, a TAPE1 stress recovery family, and a TAPE8 support stiffness family. The TAPE8 family will only be generated by GEMINI if potentially active support degrees-of-freedom are flagged. In the GEMINI nodal constraint specification, such potential degrees of support freedom must be numbered $1000i$, where $i = 1, 2, \dots, NSUP$ spans the set of $NSUP$ degrees-of-freedom. These need not be unique, since 10001 for all x direction, 10002 for all Y-direction, and 10003 for all Z-direction supports collapses many nodes to 3 groups of support freedom. These degrees-of-freedom are assumed constrained for the GEMINI modal extraction, and coupling stiffness matrices and flagged element destination vectors are saved on TAPE8. If only rigid

body base translation ("fixed base") is to be prescribed, TAPE8 is not needed, and this step may be skipped.

The "pseudostatic mode" method requires an additional GEMINI static run with no loads defined, and support flags set. The number of load cases must be set equal to the number of support groups defined. The resulting TAPE10 file must be switched to TAPE9 for the following SAPPAC run. Again, if only fixed base excitation is desired (NSUP=0), this may be omitted. The user should refer to the GEMINI user's manual (Ref. 2) for more detail.

Access and Execution on the NMFEC

Program SAPPAC can be accessed on the NMFEC with the following FILEM command:

RDS 7205 .SMACS SAPPAC

Program SAPPAC can be executed on the NMFEC with the following command:

SAPPAC (I=SAPPACI,O=SAPPAC,T1=TAPE1,T2=TAPE2,
T3=TAPE3,T8=TAPE8,T9=TAPE9,T10=TAPE10) / t v

For files with other than the default names (shown) append the appropriate expression in brackets.

Inter-Program Compatibility

This Version of SAPPAC is compatible with
Program GEMINI, Version
Program SMAX, Version 2.0

References

1. G. L. Goudreau, "Preliminary User's Manual for SAPPAC, A Multisupport Modal Time Integrator and Transfer Function Plotter for SAP4 Models," Lawrence Livermore National Laboratory, UCID-19054, July, 1982.
2. R. C. Murray, "GEMINI - An Efficient Computer Program for Three Dimensional Linear Static and Seismic Structural Analysis," doctoral dissertation, University of California, Davis.

Fig. 1 Flow of Data through Program SAPPAC.

DESCRIPTION OF INPUT FILE "SAPPACI"

Heading Line (7A10)

Columns	Variable	Description	Notes
1-70	IHED	Identifying header information to be printed with the output	

Control Information (7I15,5X,2I15,F10.0)

Columns	Variable	Description	Notes
1-5	NF	Number of modes to use to describe the eigensystem. Must not be greater than the number of modes on TAPE10.	
6-10	NSUP	Number of support degrees-of-freedom besides rigid body datum.	(1)
11-15	NB	Number of datum components (0-6). For SMACS, Use NB=0.	(1)
16-20	IB	Node number of datum. For SMACS, use IB=0.	(1)
21-25	NF1	First mode shape to be printed in output file SAPPACO.	(2)
26-30	NF2	Last mode shape to be printed in output file SAPPACO.	(2)
31-35	IPR	Projection option for formation of TAPE3. For SMACS use IPR=1 to project mode shapes onto normal coordinates and save matrices on TAPE3. If IPR is negative projected matrices are printed in output file SAPPACO.	
41-45	IZP	Transient integration option. For SMACS, use IZP=0.	
46-50	IDAMP	Modal damping ratio control line NDAMP = 0 following constant applies to all modes NDAMP > 0 read NF damping values from following lines. NDAMP < 0 activates composite modal damping. Read from following lines	
51-60	DAMP	Constant damping ratio used for all modes when NDAMP = 0.	

NOTES

- (1) Program SMAX expects data for subsystems that are multiply supported and processed using the pseudostatic mode method. Therefore NSUP must be greater than zero and NB and IB must equal zero.
- (2) Mode shapes are normalized to the maximum modal coordinate before printing to the output file.

Modal Damping Information (IF NDAMP >0) (8F10.0)

Columns	Variable	Description	Notes
1-80	DAMP(I)	Modal damping ratios for each mode	(1)

This line or lines are provided only if NDAMP is greater than zero on control data line.

Composite Modal Damping (if NDAMP <0) (8F10.0)

Columns	Variable	Description	Notes
1-80	DAMP(I)	One modal damping ratio must be supplied for each element group.	(1) (2)

This line or lines are provided only if NDAMP is less than zero on the control data line.

NOTES

- (1) GEMINI element stiffness file TAPE2 must be supplied. A special GEMINI data check run (with MODEX=-1) will generate the TAPE2 file. Composite modal damping ratios will be computed. This is the most I/O intensive operation, so expand buffers for TAPE2 and TAPE10. On the NMFECC this can be done on the execute line. Once computed, ratios can be edited out of SAPPACO for subsequent use in input.
- (2) This line or lines provided only if NDAMP is less than zero in control data line.

Acceleration Recovery List

Line 1 (IS)

Columns	Variable	Description	Notes
1-5	KKK	Not used. PROVIDE ONE BLANK LINE.	(1)

Line 2 (7IS) List of Responses

Repeat for each node for which accelerations are desired. Node numbers and component numbers may be in any order and may be repeated. Terminate list with one blank line.

Columns	Variable	Description	Notes
1-5	NP	Node Number	
6-35	IC	Degree-of-freedom list	(2)

NOTES

- (1) If IPR = 0, skip above input sequence. Otherwise, if no acceleration response requested, provide two blank lines.
- (2) If several response components are desired to be summed into vector resultant, all but last component number in the series should be set negative.
- (3) Variable NA in SAPPAC is the total number of acceleration components requested.

Displacement Recovery List

Line 1 (15)

Columns	Variable	Description	Notes
1-5	KKK	Not used. PROVIDE ONE BLANK LINE.	(1)

Line 2 (715) List of Responses

Repeat for each node for which displacements are desired. Node numbers and component numbers may be in any order and may be repeated. Terminate list with one blank line.

Columns	Variable	Description	Notes
1-5	NP	Node number	
6-35	IC	Degree-of-freedom list	(2) (3)

NOTES

- (1) If IPR = 0, skip above input sequence. Otherwise, if no displacement response requested, provide two blank lines.
- (2) If several response components are desired to be summed into vector resultant, all but last component number in the series should be set negative.
- (3) Variable NU in SAPPAC is the total number of displacement components

Force Recovery List

Provide the following data once for each element group in the GEMINI analyses. Groups must be in the order listed in the GEMINI input file GEMI.

Line 1 (15)

Columns	Variable	Description	Notes
1-5	KKK	Not used. PROVIDE ONE BLANK LINE.	(1) (3)

Line 2 (715) List of Responses

Repeat for each element for which forces are desired. Element numbers and component numbers may be in any order and may be repeated. Terminate list with one blank line.

Columns	Variable	Description	Notes
1-5	NEL	Element number	(3)
6-65	IC	Force component number list	(2) (3)

NOTES

- (1) If IPR = 0 skip above input sequence. Otherwise, if no force response requested, provide two blank lines for each element group.
- (2) If several response components are desired to be summed into a vector resultant, all but last component number in the series should be set negative.
- (3) Variable NS in SAPPAC is number of stress components requested.

Transient Response Input

This section skipped when used in conjunction with SMACS. Refer to more extensive SAPPAC Manual (Ref. 1.) for use and operation of this option.

DESCRIPTION OF OUTPUT FILE "TAPE3"

TAPE3 is a sequential binary file produced by SAPPAC that contains all essential subsystem information for response calculation based on the "pseudostatic mode" methodology. To be used by program SMAX, the name of this file must be changed to "T3Snn", where nn is the sequence number of the subsystem in the SMAX analysis. Once T3Snn files have been generated for all subsystems to be analyzed by SMAX, they must be collected into a library file named LIBT3. This is done using the NMFECC utility routine LIB.

Record 1 (16 Words)

Words	Entry	Type	Description
10	HED	10A8	Heading information identifying subsystem
1	NA	integer	Number of acceleration response requests
1	NU	integer	Number of displacement response requests
1	NS	integer	Number of stress recovery requests
1	NF	integer	Number of subsystem modes
1	NSUP	integer	Number of support degrees-of-freedom
1	NB	integer	Number of datum degrees-of-freedom

Record 2 (3*NA+3*NU+3*NS Words)

Words	Entry	Type	Description
3*NA	IRA(3,i), i=1,NA	integer	Node number, component number and equation number for each acceleration response request.
3*NU	IRU(3,i) i=1,NU	integer	Node number, component number and equation number for each displacement response request.
3*NS	IRS(3,i) i=1,NS	integer	Element number, component number and element type for each stress recovery response.

Record 3 (6*NF+NF+NF*NA+NF*NU+NF*NS+NA+NU+NS+6*NF words)

Words	Entry	Type	Description
NF	W(i),i=1,NF	real	Dynamic modal frequencies
NF	D(i),i=1,NF	real	Dynamic modal damping values, either constant, variable or composite
NF*NA	AIT(i,j) i=1,NF j=1,NA	real	Dynamic eigenvectors for acceleration response request inertial component recovery.
NF*NU	UIT(i,j) i=1,NF j=1,NU	real	Dynamic eigenvectors for displacement response request inertial component recovery.

Words	Entry	Type	Description
NF*NS	RS1T(i,j) i=1,NF j=1,NS	real	Stress recovery matrix for inertial component of requested stress response
NN	NNF(i) i=1,NN	integer	Acceleration response request node numbers, displacement response request node numbers and stress response request element numbers. Note: NN=NA+NU+NS
6*NF	ALF(i,j) i=1,6 j=1,NF	real	Modal participation factors

Record 4 (NSP*NF Words) Note: NSP = NSUP + NB

Words	Entry	Type	Description
NSP*NF	S21(i,j), i=1,NSP, j=1,NF	real	Displacement forcing function. Note: not completely calculated, divide by w^2 to become forcing function.

Record 5 (NSP*NS Words)

Words	Entry	Type	Description
NSP*NS	RS2T(i,j) i=1,NSP, j=1,NA	real	Stress recovery matrix for pseudostatic component of requested stress responses

Record 6 (NSP*NA Words)

Words	Entry	Type	Description
NSP*NA	AZT(i,j), i=1,NSUP, j=1,NA	real	Acceleration recovery matrix for pseudostatic component of requested acceleration responses, ie - pseudostatic modes

$$K_{11}^{-1} \quad K_{12}$$

Record 7 (NSP*NU Words)

Words	Entry	Type	Description
NSP*NU	UZT(i,j), i=1,NSUP, j=1,NU	real	Displacement recovery matrix for pseudostatic component of requested displacement responses, ie - pseudostatic modes

$$K_{11}^{-1} \quad K_{12}$$

APPENDIX D

PROGRAM SAPPAC

USER'S MANUAL

November 1984

Michael J. Mraz
Oleg R. Maslenikov
James J. Johnson

NTS/Structural Mechanics Associates
Two Annabel Lane, Suite 101,
San Ramon, CA 94583

PROGRAM LNGEN, VERSION 1.0

USERS MANUAL

Program Description

Program LNGEN generates a set of random numbers which are used in a subsequent SMAX analysis to represent variations in soil, structure and subsystem properties. The numbers are applied in SMAX as multipliers to soil properties (shear modulus and damping) structures and subsystems input parameters (frequencies and damping) to account for all sources of uncertainty.

The method used in calculating the multipliers is a Latin hypercube sampling technique which selects values from user-specified lognormal distributions of the soil, structure and subsystem properties. A coefficient of variation (COV) is input to describe the distribution of each parameter. For more detail on the methodology, refer to the SSMRP Phase I report on SMACS (Ref. 1).

Program Data Flow

Program LNGEN requires one input data file, LNGENI, and produces two output files, LNGENO and DATA. The data files are described in general below. Detailed descriptions can be found in subsequent pages of this manual or in other manuals

LNGENI is a text input file prepared by the user. It contains problem control information such as the number of structures, number of subsystems, number of earthquake problems and whether SSI or fixed base analysis is to be performed. The user also specifies a random seed for number generation and coefficients of variation for soil, structure and subsystem properties. A detailed description of this input file is given in subsequent pages.

LNGENO is a text print file produced by LNGEN. It contains a summary of the experimental design calculations performed by LNGEN.

DATA is a text data file produced by LNGEN. It contains multipliers to be applied to (1) soil modulus and material damping ratios in the case of an SSI analysis, (2) structure frequencies and damping ratios and (3) subsystems frequencies and damping ratios. This file is in SMAX format and is to be

appended to the SMAXI file. However, it is necessary for the user to enter control parameters for the input motion for each of the NPROBS earthquakes manually.

Access and Execution on the NMFECC

Program LNGEN can be accessed on the NMFECC with the following FILEM command:

```
RDS 7205 .SMACS LNGEN
```

Program LNGEN can be executed on the NMFECC with the following command:

```
LNGEN (I=LNGENI) / t v
```

For files with other than the default names (shown) append the appropriate expression in brackets.

Inter-program Compatibility

This version of LNGEN is compatible with
Program SMAX, Version 2.0

References

1. J. J. Johnson, G. L. Goudreau, S. E. Bumpus, O. R. Maslenikov, "Seismic Safety Margins Research Program, Phase I Final Report - SMACS - Seismic Methodology Analysis Chain with Statistics (Project VIII)," Lawrence Livermore National Laboratory, NUREG/CR-2015, Vol. 9.

DESCRIPTION OF DATA FILE "LNGENI"

Record 1 (415): Control Information

Columns	Variable	Description
1-5	NTSTR	Total number of structures
6-10	NSUB	Total number of subsystems
11-15	NPROBS	Number of earthquake problems
16-20	IFBXI	Indicator for fixed base analysis .eq.0, SSI analysis .ne.0, fixed base analysis

Record 2 (110): Random Seed

Columns	Variable	Description
1-10	ISEED	Integer seed for random number generation. If ISEED = 0, a random seed is obtained from the computer microsecond wall clock. Therefore ISEED must be greater than zero if this experimental design is to be repeatable.

Record 3 (8F10.5): Coefficients of Variation

Columns	Variable	Description
1-80	XCV(I)	Coefficients of variation as described below. Provide data in fields of 5 columns each.

If IFBXI.eq.0, $2*(NTSTR+NSUB)+2$ values must be provided

XCV(1) is the COV for soil shear modulus

XCV(2) is the COV for soil damping ratio

XCV(3) is the COV for the frequencies of structure 1

XCV(4) is the COV for the damping ratios of structure 1

XCV(5) is the COV for the frequencies of structure 2

XCV(6) is the COV for the damping ratios of structure 2

.

.

.

XCV($2*NTSTR+3$) is the COV for the frequency of subsystem 1

XCV($2*NTSTR+4$) is the COV for the damping ratio of subsystem 1

.

.

.

If IFBXI.ne.0, $2*(N1STR+NSUB)$ values must be provided

XCV(1) is the COV for the frequencies of structure 1

XCV(2) is the COV for the damping ratios of structure 1

XCV(3) is the COV for the frequencies of structure 2

XCV(4) is the COV for the damping ratios of structure 2

.

.

XCV($2*N1STR+1$) is the COV for the frequency of subsystem 1

XCV($2*N1STR+2$) is the COV for the damping ratio of subsystem 1

.

.

.

Description of Output File DATA

This is a text file to be edited by user and appended to the SMAXI input file. The information in this file is described in Data set #6 through Data set #9 in the SMAXI file description in the SMAX users manual. Record 2 of Data set #6 must be edited by the user.

APPENDIX E

PROGRAM LNGEN

USER'S MANUAL

November 1984

Lu Woon Tiong
Oleg R. Maslenikov

NTS/Structural Mechanics Associates
Two Annabel Lane, Suite 101,
San Ramon, CA 94583

PROGRAM SMAX, VERSION 2.0

USERS MANUAL

Program Description

Program SMAX was developed at LLNL to perform soil-structure interaction (SSI) major structure and subsystem response calculations for multiple analyses that include random variation of soil, structure and subsystem properties. SMAX is a marriage of the CLASSI program SSIN, developed by Prof. J. E. Lucio (UCSD) and Prof. H. L. Wong (USC) (Ref. 1), which calculates SSI response of major structures using Fourier analysis methods, and parts of program SAPPAC, developed by Dr. G. L. Goudreau (LLNL) (Ref. 2), which performs modal time history analyses for multiply-supported systems. SMAX uses the foundation impedance and incident wave scattering characteristics calculated by program CLAN, the dynamic properties of the major structures calculated by program INSSIN, and the dynamic properties of specified subsystems calculated by program SAPPAC in combination with ensembles of earthquake free-field acceleration time histories to define the problem. Properties of the soil, structures and subsystems can be varied by applying random multipliers to base (or "nominal") properties according to an experimental design calculated in program LNGEN. For each earthquake, SMAX first calculates foundation and in-structure response of the major structures, using the logic of program SSIN. Response time histories at specified locations in the structures then serve as the input motions to specified subsystems. The subsystem response is calculated using the ZIPP module taken from program SAPPAC. In addition to time histories of response, SMAX produces a response vector which contains measures of response in the structures and subsystems such as maximum accelerations, displacements and forces or spectral acceleration at specified frequencies. The response vector is used by program SEISIM in its systems analysis to calculate probabilities of component failure, system failure, core melt and radioactive release. The response vector can also be post-processed by program PRESTO to reduce the data to statistical terms and plot cumulative distribution functions of the results.

Program Data Flow

Program SMAX requires six input data files for its execution: SMAXI, IMPFNC, SSINST, SSINTH, SUBSYS and LIBT3. SMAX produces either three or four output files depending on the user's commands: SMKLOG, LIBSP, LIBT7 (if flagged), and LIBRV. These data files are described in general below. Detailed descriptions can be found either in subsequent pages of this manual or in other manuals. The flow of data is illustrated in Figure 1.

SMAXI is a text input data file prepared by the user. It contains certain problem control information such as the number of foundations, the number of structures on each foundation, the number of in-structure responses to calculate and the number of earthquake problems. It also contains the foundation mass matrices. The last part of the file contains the property variation multipliers. These are applied to the nominal soil shear modulus and damping ratio, and the nominal modal frequencies and damping ratios of the structures and subsystems. These factors can be obtained from program LNGEN which produces an output file of the appropriate format which can be appended to the first part of the SMAXI file. For a detailed description of the SMAXI file, refer to subsequent pages of this manual.

IMPFNC is a text input file produced by program CLAN. It contains the foundation impedance matrices and incident wave scattering vectors at the frequencies selected in the CLAN execution. In general, the number of frequencies for which this data is calculated is much less than the number of frequencies used to calculate SSI response in SMAX because the impedance and scattering functions are much smoother than the Fourier spectra of the input and response time histories. Intermediate points are therefore interpolated in SMAX. For a detailed description of the IMPFNC file, refer to the CLAN User's Manual.

SSINST is a text input file produced by program INSSIN. It contains the dynamic properties of the structures, including modal frequencies, damping ratios and participation factors and the rigid body mass matrix of the structure about the foundation reference point. Also included are the modal coordinate vector and the rigid body transformation vector for each of the locations at which in-structure response is to be calculated. For a detailed description of the SSINST file, refer to the INSSIN User's Manual. For multiple structures the SSINST file used by SMAX consists of a chain of the SSINST data from the individual INSSIN runs appended to each other. Their order must agree with the order in which the foundations are identified in the SMAXI file and in the CLAN run that generated the impedances.

SSINTH is an input file provided by the user. It can be either text or sequential binary. It contains the acceleration time histories that define the earthquake free-field motion of the ground surface. Typically, each earthquake would be defined by a triad of two orthogonal horizontal components and a vertical one. The format in which the data appears can be specified by the user in the SMAXI file.

SUBSYS is a text input file prepared by the user. It contains information identifying each subsystem and correlating it to its structural support locations. For a detailed description of the SUBSYS file, refer to subsequent pages of this manual.

LIBT3 is a LIB (library) file used by SMAX that contains for each subsystem one sequential binary file of dynamic properties. Each file is produced by a separate analysis using program SAPPAC. The files are named T3S01 through T3Snn for nn subsystems. For a detailed description of the T3Snn files, refer to the SAPPAC User's Manual. For a description of the LIB utility routine, refer to the NMFEC Summary of Utility Routines.

SMLOG is a text output file generated by SMAX. It contains a summary of the problem execution including the soil properties used in programs GLAY and CLAN to generate the foundation impedance and wave scattering functions, the dynamic properties of the structures, the maximum value of each specified in-structure response component and the response vector for each earthquake. This file is for the user's information only.

LIBRV is a LIB file produced by SMAX that contains for each earthquake a sequential binary file containing the response vector. The files are named RVE01 to RVEmm for mm earthquakes. The major structure response quantities contained in the response vector are preselected by the user when he executes program INSSIN. The subsystem response quantities are preselected by the user when he executes program SAPPAC. The same response components are obtained for each earthquake. For a detailed description of the RVEmm files, refer to subsequent pages of this manual. The LIBRV file is used for subsequent processing by programs PRESTO and SEISIM.

LIBSP is a LIB file, produced by SMAX, that contains for each earthquake one sequential binary file of selected in-structure response time histories from the SSI analysis. The files are named SPE01 to SPEmm for mm earthquakes. The time histories are selected by the user when he generates the structural data using program INSSIN. (Refer to the description of the INSNIN file in the INSSIN User's Manual). The same response components are obtained for each earthquake. For a detailed description of the SPEmm files, refer to subsequent pages of this manual.

LIBT7 is a LIB file produced by SMAX that contains one sequential binary file for each earthquake and each subsystem. The files are named T7E01S0 to T7EmmSnn for mm earthquakes and nn subsystems. Each file contains preselected response time histories for one subsystem and one earthquake. The response components are selected by the user when he executes program SAPPAC. For a detailed description of the T7EmmSnn files, refer to subsequent pages of this manual.

Access and Execution on the NMFECC

Program SMAX may be accessed on the NMFECC by using the following FILEM command.

RDS 7205 .SMACS SMAX

Program SMAX may be executed on the NMFECC with the following command:

SMAX (I=SMAXI,IMP=IMPFNC,ST=SSINST,TH=SSINTH,L=SMXLOG) / t v

For files with other than the default names (shown), append the appropriate expression in brackets.

Inter-program Compatibility

This version of SMAX is compatible with

Program CLAN, Version 2.0
Program INSSIN, Version 2.0
Program SAPPAC, Version 2.0
Program PRESTO, Version 1.0
Program LNGEN, Version 1.0

References

1. G. L. Goudreau, "Preliminary User's Manual for SAPPAC, A Multisupport Modal Time Integrator and Transfer Function Plotter for SAP4 Models," Lawrence Livermore National Laboratory, UCID-19054, July, 1982.
2. R. C. Murray "GEMINI - An Efficient Computer Program for Three Dimensional Linear Static and Seismic Structural Analysis," doctoral dissertation, University of California, Davis.

Fig. 1 Flow of Data through Program SMAX.

DESCRIPTION OF INPUT DATA FILE "SMAXI"

Data set #1: Global Control Parameters

Record 1 (10A8)

Columns	Variable	Description	Notes
1-64	IDENT	Up to 64 characters identifying the SMAX run	

Record 2 (4E10.0,2I10)

Columns	Variable	Description	Notes
1-10	G	Shear modulus in the reference soil layer.	(1)
11-20	VS	Shear wave velocity in the reference layer.	(1)
21-30	CL	Characteristic length of the foundation.	(1)
31-40	DRAT	Damping ratio in the reference layer.	(1)
41-50	LLCM	Length of LCM storage to be used. (Default = 105680)	

NOTES

- (1) If the foundation reference parameters are defaulted or set to zero, they will be set equal to the values in the IMPFNC file. If non-zero, they will be compared with those in the IMPFNC file and a warning will be written in the SMXLOG file if they are not equal. The values provided here, if not defaulted, will be used for the analyses.

Record 3 (3I5)

Columns	Variable	Description
1-5	NFDN	Number of foundations.
6-10	NTSTR	Total number of structures.
11-15	NPROBS	Number of earthquake problems to be solved (Default = 1). The same structure, foundation and subsystem configurations will be used for all problems. Repeat data sets #6 through #8 NPROBS times.

Data set #2: Structures Per Foundation

This data set is repeated NFDN times, once for each foundation.

Record 1 (15)

Columns	Variable	Description
1-5	NSTR	Number of structures supported on the foundation.

Data set #3: Foundation Mass Matrix

This data set, followed by data set #4 is repeated NFDN times, once for each foundation.

Records 1 to 6 (6E10.3)

Columns	Variable	Description
1-60	FMI	The mass matrix of the foundation about the reference point. It is read in 6 lines (row-by-row) with 6 values on each line. The reference point must agree with the one used for the CLAN and INSSIN runs.

Data set #4: Control Parameters for Structural Data

Data set #4 is read NSTR times, once for each structure supported on the foundation specified in data set #3.

Record 1 (215)

Columns	Variable	Description
1-5	NMODE	Number of modes representing the structure
6-10	NKEEP	Number of in-structure response components for which SSI response is desired. Foundation response is not included in this count.

Data set #5: Control Data for Ground Motions and Variations

Record 1 (515,5X,A1,A7,4A8)

Columns	Variable	Description
1-5	NCOM	Number of components of ground motion per earthquake. Maximum number of components for SSI analysis is 3. For a fixed-base analysis it is 6.
6-10	NCASE	Number of incident waves in the file IMPFNC. If NCASE equals 0, a fixed-base analysis will be performed with motions assumed to be in the order X, Y, Z, XX, YY, ZZ
11-15	NSTART	Number of the starting incident wave case to be read from IMPFNC. If NCASE>1 the scattering vectors are assumed to be ordered consecutively in IMPFNC. Note that NCOM+NSTART-1 must not be greater than NCASE.
16-20	NVIMP	Flag indicating if soil properties are to be varied 0 = No variation 1 = Vary by multipliers in data set #7
21-25	NVSTR	Flag indicating if structural properties are to be varied (No default) 0 = No variation 1 = Vary by multipliers in data set #8.
31-70	IFMT	Format in which input motions are to be read. The format specification must be left-justified in columns 31-80. If column 31 is blank, a default format of (10A8,/, (8F10.0)) will be read.

Data set #5A: Minimum and Maximum Frequency in Fourier Analysis

The input motions used by SMAX are in the time domain. The Fast Fourier Transform (FFT) is used to transform the input accelerograms into Fourier spectra and the problem is solved in the frequency domain. The computed SSI response motions are transformed to the time domain using the inverse FFT.

Record 1 (2F10.0)

Columns	Variable	Description
1-10	FMIN	Minimum frequency (Hz) to be used
11-20	FMAX	Maximum frequency (Hz) to be used

SMAX will use the maximum of FMIN and the lowest frequency in the IMPFNC file for the low frequency cutoff for the Fourier analysis. Likewise, it will use the minimum of FMAX and the highest frequency in the IMPFNC file for the high frequency cutoff. If the soil shear modulus is to be varied (data set #7) the IMPFNC frequency will first be adjusted before determining cutoff frequency.

The user must provide Data Sets #6 through #9 NPROBS times. Data sets #7 through #9 can be obtained, in the proper format, from program LGEN.

Data Set #6: Input Motion Control Parameters

Record 1

Separator in the input file to mark the beginning of a block of data pertaining to one of NPROBS problems. Skipped over by program.

Record 2 (2E10.0, 2I5)

Columns	Variable	Description
1-10	DT	Time step size for each accelerogram.
11-20	SCALE	Scale factor applied to each input accelerogram. A value of 0.0 defaults to 1.0.
21-25	NPOINT	Number of points in each input accelerogram.
26-30	NFFT	Number of fast Fourier transform points used. NFFT is twice the number of frequency increments that are used in the Fourier SSI response analysis. NFFT must be an integer power of two. If NFFT is less than NPOINT, the input motions will be truncated to NFFT points. If NFFT is greater than NPOINT, trailing zeroes will be added to the input motions.

Data set #7: Variation of Soil Properties

This data set will be read only if NCASE and NVIMP (data set #5) are both greater than zero. The soil shear modulus and damping ratio scale factors that are specified in this data set are used to modify the soil impedance matrix tables, which the CLAN program generates.

Record 1 (2F10.0)

Columns	variable	Description
1-10	GFAC	The scale factor for the variation in soil shear modulus.
11-20	DFAC	The scale factor for the variation in soil damping ratios.

Data set #8 Variation of Structural Properties

This data set will be read only if NVSTR (data set #5) is greater than zero. Records 1 and 2 will be read NSTR (data set #1) times, once for each structure in the SSI system. The order of the structural models is same as in data set #4.

Record 1 (8F10.0)

Columns	Variable	Description
1-10	VW(n)	Frequency variation factors to be applied sequentially to each of the individual modal frequencies of the structural model. Provide up to NMODE (data set #4) values. A value of zero will default to 1.0. If VW(n) is negative reading ceases and the absolute value of VW(n) is used for that and all remaining modes.
.		
.		
.		
71-80		

Record 2 (8F10.0)

Columns	Variable	Description
1-10	VD(n)	Damping variation factors to be applied sequentially to each of the individual modal damping ratios of the structural model. Provide up to NMODE (data set #4) values. A value of zero will default to 1.0. If VD(n) is negative reading ceases and the absolute value of VD(n) is used for that and all remaining modes.
.		
.		
.		
71-80		

Data set #1: Variation of Subsystem Properties

This data set will be read only if NSUB (specified in input file SUBSYS) is greater than zero. Records 1 and 2 will be read NSUB (specified in file SUBSYS) times, once for each subsystem.

Record 1: NAME

Column	Variable	Description
1-80	NAME	Up to 80 characters identifying subsystem. Used as separator in the input deck.

Record 2: FREQ, DMP, IND

This record is repeated for each mode of the subsystem or until DMP (see below) equals 1 is encountered.

Column	Variable	Description
1-10	FREQ	Subsystem frequency multipliers
11-20	DMP	Subsystem damping multipliers
21-25	IND	Indicator for generation of constant multipliers for remaining modes. 0 = multiplier for next mode to be read in 1 = multipliers for remaining modes to be set constant and equal to current mode

DESCRIPTION OF INPUT DATA FILE "SUBSYS"

Data set #1: Global Control Parameters

Record 1 (415)

Columns	Variable	Description
1-5	NSUB	Total number of subsystems
6-10	IBLC	Indicator for baseline correction of stress quantities .ge. 0, no baseline correction on stresses (default) .eq.-1, perform baseline correction on stresses and print correction to log .lt.-1, perform baseline correction on stresses, do not print correction.
11-15	ISUPL	Controls printing of subsystem support time history labels. .eq.0, do not print labels from SSINP (default) .eq.1, print labels of time histories associated with subsystem supports.
16-20	INTORV	Controls writing of subsystem responses to response vector file (RVENN-file) .eq.0, write total, pseudostatic and inertial responses (default) .eq.1, write total responses only .eq.2, write pseudostatic responses only .eq.3, write inertial responses only

Record 2 (3(2X,A8),9X,I1,2X,or8)

Columns	Variable	Description
3-10	LIBUSR	Name of user-supplied library of subsystem files. Default name is LIBT3S.
13-20	NLIBRV	Name for library file of response vectors to be created by program. Leave blank if response vector data are not to be libaried.
23-30	NLIBSP	Name for library file of in-structure response time- histories. Leave blank if time history data are not to be libaried.
40	INDT7	Controls writing of subsystem response time-histories to TAPE7. eq.0, do not write TAPE7 (default) ne.0, write TAPE7
43-50	NLIBT7	Name for library file of TAPE7. Leave blank if data are not to be libaried.

Data set #2: Subsystem Parameters

Provide Data Sets #2 and #3 NSUB times, once for each subsystem.

Record 1 (10A8)

Columns	Variable	Description
1-80	HED2	Subsystem identification

Record 2 (F10.0,I5)

Columns	Variable	Description
1-10	SCL	Scale factor for all support time-histories. Defaults to 1.0
11-15	KRO	Indicator for static correction to subsystem responses. 1e.0, no static correction gt.0, static response values to be read in from data set #3

Record 3 (16I5) (to be repeated until NSUP integers are input)

Columns	Variable	Description
1-80	NNBR	Response number in SSINP file to be used for subsystem excitation. A negative integer number will cause the particular record to be scaled by -1.0.

Data set #3: Static response values

This data set will be read only if KRO (data set #2) is greater than zero.

Record 1 (8F10.0) (to be repeated until NA values are input. Skip if NA = 0)

Columns	Variable	Description
1-80	RA(I,5) J=1,NA	Static acceleration values to be added to dynamic response.

Record 2 (8F10.0) (to be repeated until NU values are input. Skip if NU=0)

Columns	Variable	Description
1-80	RU(I,5) I=1,NV	Static displacement values to be added to dynamic response

Record 3 (8F10.0) (to be repeated until NS values are input. Skip if NS=0.)

Columns	Variable	Description
1-80	RS(I,5) I=1,NS	Static stress values to be added to dynamic response. Note: If IBLC (data set #1) is less than 0, the static values read in this section are overwritten by baseline correction stresses calculated within the program.

DESCRIPTION OF OUTPUT FILES "SPEmm"

Each of the sequential binary files, SPEmm, located in LIB (library) file LIBSP contains in-structure response data. The data is described below. Files SPEmm can be used for post-processing as desired.

Record 1 (10 Words)

Words	Entry	Type	Description
8	IDENT	8A8	User-specified title from SMAXI line 1.
1	IDATE	A8	Date of execution
1	ITIME	A8	Time of execution

Record 2 (8 Words)

Words	Entry	Type	Description
1	LFT	integer	Indicates type of file. LFT = 1 time history data
1	NOUT	integer	Number of responses per earthquake
1	NQKS	integer	Number of earthquakes = 1
1	NVSTR	integer	Structure variation flag (see SMAXI file) = 1
1	NVSOIL	integer	Soil variation flag (see SMAXI file) = 1
1	LENG	integer	Number of data points per response
1	DEL	real	Abscissa step size
1	ZERO	real	First Abscissa value in responses = 0

Each response in the file is described by record i or by the pair of records i,j depending on the value of IPEAK (below).

Record i (9 Words)

Words	Entry	Type	Description
1	ITH	integer	Response number
1	AMAX	real	Maximum value in the response
5	LABEL	(5A8)	Response label.
1	KEY	integer	Key to the type of response. -2 = relative displacement -1 = relative velocity 0 = absolute acceleration 1 = absolute velocity 2 = absolute displacement 3 = force or stress
1	IPEAK	integer	Flag to indicate if response follows 0 = response follows 0 = peak value only; no response follows

Record j (LENG Words)

Words	Entry	Type	Description
LENG	Y(L), L=1,LENG	real	Response data included if IPEAK is greater than 0.

DESCRIPTION OF OUTPUT FILES "T7EmmSnn"

The family of sequential binary files with root name T7EmmSnn is described below. Numeric characters "mm" embedded in the name identify the earthquake problem and "nn" identify the subsystem.

Record 1 (16 words)

Words	Entry	Type	Description
10	HED	(10A8)	Heading information identifying subsystem
1	NT	integer	Number of time points in response time-histories
1	DT	real	Step-size of response time-histories
1	NB	integer	Number of rigid body datum components (from file T3Snn)
1	NA	integer	Number of acceleration responses
1	NU	integer	Number of displacement responses
1	NS	integer	Number of stress responses

NT records follow. Each of the following NT records represent response values at a particular time point. No further information will be written, however, if both NA and NS equals zero.

Subsequent Records (NB+3*NU+3*NA+3*NS words)

Words	Entry	Type	Description
NB	(B(1),I=1,NB)	real	Rigid body datum
3*NU	(RU(i,3),RU(i,6),RU(i,7), i=1,NU)		Total, pseudostatic and inertial response for each displacement component
3*NA	(RA(i,3),RA(i,6),RA(i,7), i=1,NA)		Total, pseudostatic and inertial response for each acceleration component
3*NS	(RS(i,3),RS(i,6),RS(i,7), i=1,NS)		Total, pseudostatic and inertial response for each stress component

DESCRIPTION OF OUTPUT FILES "RVEmm"

This sequential binary data file contains only one record. To facilitate description, one could visualize the record to comprise several contiguous segments of information. The data structure is as follows.

Segment #1 (1 Word)

Words	Entry	Type	Description
1	IAT	int	Number of attributes provided in segments #2 and #3. For fixed base analysis, $IAT = NTSTR * 4$ where NTSTR is the total number of structures For SSI analysis, $IAT = NTSTR * 4 + 2$

Segment #2: (2 Words) Reference soil properties

This segment is not written for a fixed base analysis

Words	Entry	Type	Description
1	GIN	real	Reference soil shear modulus read from SMAXI
1	DRAT	real	Reference soil damping ratio read from SMAXI

Segment #3: (4 Words) Structure properties variation

This segment is repeated NTSTR times, once for each structure in the system

Words	Entry	Type	Description
1	W(n)	real	Frequency multiplier for structure n. The first mode multiplier is written. This assumes that the same multiplier was applied to all modes.
1	DAMP(n)	real	Damping multiplier for structure n. The multiplier for the first mode is written.
1	WIN(n)	real	Frequency of mode 1 for structure n.
1	DAMPIN(n)	real	Damping ratio of mode 1 for structure n.

Segment #4: (1 Word)

Words	Entry	Type	Description
	IRV	int	Number of responses in response vector

Segment #5: (IRV Words)

Words	Entry	Description
IRV	(RV(i),i=1,IRV)	Response values

Segment #6: (IRV Words)

Words	Entry	Description
IRV	(IDENTY(i), i=1,IRV)	Identifiers for responses.

Each IDENTY is an integer number coded from

$$\begin{aligned}\text{IDENTY} &= 10,000,000,000 * \text{ISTR} \\ &+ 100,000,000 * \text{ISUB} \\ &+ 10,000,000 * \text{NEI} \\ &+ 1,000 * \text{NODEL} \\ &+ 10 * \text{IDIR} \\ &+ \text{IRTPE}\end{aligned}$$

where ISTR = structure number
ISUB = subsystem number
NEI = response location
eq. 0, element
eq. 1, node
eq. 2, free-field motion
eq. 3, SSI foundation motion
NODEL = element, node or foundation number
IDIR = direction/component number for
kinematic/stress quantities
IRTPE = response type
eq. 0, peak value of response
eq. 1, 2 or 3, spectral value
eq. 4, subsystem pseudostatic response
eq. 5, subsystem inertial response

APPENDIX F

PROGRAM SMAX

USER'S MANUAL

November 1984

Oleg R. Maslenikov

Lu Woon Tiong

James J. Johnson

NTS/Structural Mechanics Associates

Two Annabel Lane, Suite 101,

San Ramon, CA 94583

PRESTO

This code post-processes the binary files in LIBRV as obtained as an output from SMACS. To run PRESTO we also need the file PRESIN and the file RVCOR. PRESIN is an input driver file which has such data as numbers of earthquakes, structures, substructures, etc. The RVCOR file tells how to combine the response vectors from SMACS to get the basic events. The RVCOR file will be needed even if we are doing only a raw data analysis. This combination is determined by the IPOS variable in the RVCOR file. The resulting output files are:

- PRSOUT - a printed listing of response vectors; and
- UPABO - the resulting response vectors or basic events plotted along with the fitted lognormal probability distribution.

Input to PRESTO

PRESTO is executed by typing:

PRESTO I=presin,O=prsout,SI=sizemi,RC=rvcor / t v

where the default names are given in small print. Any file or file substitution can be defaulted, in which case we do not have to type in that part of the substitution at execution time. Here we have

I = the input file,

O = the output file,

SI = the responses output for SEISEM

RC = the response correlation file.

The RVCOR and PRESIN files must be present at execution time or an error will result, even if the RVCOR file is not used. The input to PRESTO as contained in the file PRESIN is summarized below.

Card	Format	Content
1.	(10A8)	Title of up to 80 characters
2.	(3A10)	Box number and identifier for the plotted output files
3.	(4I5, 2X, A8, 3I5, F10.0, 4I5)	
		NBE = the number of basic events or raw samples to process
		NEQ = the number of earthquakes
		NSTR = the number of structures
		NSUB = the number of subsystems
		LIBNAM = the library (LIBRV) from SMACS that has the responses by earthquake. If this name is left blank, all the individual response files from SMACS are assumed to be out of the library before PRESTO is executed.
		IRAW = flag for doing raw data, which is the data as output directly from SMACS.
	.EQ. 0	No raw data calculation, compute basic event responses by combining raw data responses.
	.EQ. 1	Do raw data calculations with no combination to get basic events.
	.EQ. 2	Same as 1 except that the RVCOR file is used to give extra printout.

1 Initial startup line at LLNL.

IPLOT = plotting flag for raw data or basic events. Abscissa has units of the data or event being plotted.

.EQ. 0 no plots will be produced

.EQ. 1 abscissa and ordinate are linear with ordinate being cumulative probability in the range 0 to 1.0.

.EQ. 2 abscissa is logarithmic and ordinate is cumulative probability between 0 and 1.0.

.EQ. 3 abscissa is logarithmic and ordinate is normalized standard deviation values; i.e., a lognormal distribution will plot as a straight line.

IPRNT = flag to print response vectors and their IDENTITY identifiers

.EQ.-1 suppress printing

.EQ. 0 print all earthquakes

.GE. 1 print up to IPRNT earthquakes

XMULT = multiplier for all beta values (default is 1.0). See explanation below on beta values.

NBEVENT =sizing limit. This number must be greater than or equal to the number of basic events or raw responses that are being analyzed(NBE). The default and maximum size is set at 1200.

NRV = sizing limit for the number of raw data responses. This number must be greater than or equal to number of raw response events actually used. The default and maximum size is set at 1200.

NNUM = the number of upper points on the probability plots to have their corresponding earthquake number labeled.

NCHSIZ = the plot character size of the NNUM labeled numbers
.EQ. 0 small
.EQ.-1 even smaller

Card	Format	Content
4 thru 104	(2F10.0)	Beta factor array The beta factor array has dimensions of 101 by 8. the factors in the array are used only to obtain the basic events. These factors are read but not used when processing raw events only. Each factor that is used is referenced by the row (IROW) and column (JCOL) where it is located (see the writeup about the RVCOR file). A particular raw response from all earthquakes is multiplied by this factor in the PRESTO processing. These scaled raw responses are then used in finding maxima, which are the basic events.

In a separate file we must construct the RVCOR data. The RVCOR file is used to construct basic events. Each line in the RVCOR file does not necessarily correlate to an individual basic event. This data has the following content:

Format
(2A5,8I5)

IDBE = Ten character identifier for the failure.

MODEL = node, element, or foundation number.

NEI = node or element indicator

.EQ.0 an element force
.EQ.1 a nodal acceleration
.EQ.2 free-field acceleration
.EQ.3 foundation acceleration
.EQ.4 structure base force
.EQ.5 fixed base acceleration or input motion.

IROW = Ith row number where to find beta factor

ICOL = Jth column number where to find beta factor

IPOS = Ith position in the basic event vector that is being constructed from this and other responses.

APPENDIX G

PROGRAM PRESTO

USER'S MANUAL

March 1985

Stan Bumpus

Lawrence Livermore National Laboratory
7000 East Avenue
Livermore, CA 94550

APPENDIX H

PROGRAM CHANGO

USER'S MANUAL

March 1982

Michael A. Gerhard

Lawrence Livermore National Laboratory
7000 East Avenue
Livermore, CA 94550

CHANGO

the other half
OF
PRESTO

March 04, 1982

MICHAEL A. GERHARD

MECHANICAL ENGINEERING DEPARTMENT
NUCLEAR TEST ENGINEERING DIVISION

LAWRENCE LIVERMORE NATIONAL LABORATORY
UNIVERSITY OF CALIFORNIA
LIVERMORE, CALIFORNIA

CONTENTS

Introduction	1
Running CHANGO	2
General Information About CHANGO	3
Command Continuation	3
Command Stacking	3
Command Duplication	3
Free-Format Input	4
Keyword Equals	4
Command File	5
Interrupts	5
Handling Errors	6
Basic Commands	7
READ	7
RETURN	7
C	8
N	8
END	8
Graphics Commands	9
DEVICE	9
TAKE	10
Manipulation Commands	11
SETUP	11
RDSET	12
GAP	12
SORT	13
MAKE	13
Printing and Plotting Commands	14
WRITE	14
IDLIS	14
PLOT	14
Command Abbreviations	16

INTRODUCTION

CHANGO is an interactive code that may also be run in a batch mode. The interactive driver to CHANGO was derived from the interactive mesh generator SLIC. The command processor in CHANGO is a sophisticated new parser which prompts for all required data (unless it has already been provided). The error recovery system has been expanded to prompt the user for required corrections.

CHANGO was developed to post-process data from its companion code, PRESTO. CHANGO allows you to read in the PRESTO summary table and manipulate the data columns. The main manipulations that CHANGO can perform are:

1. Write the columns out in the print file.
2. Sort the complete set of columns.
3. Place a gap between the free-field, foundation and fixed-base accelerations and the rest of the data.
4. Plot the columns (with upto 4 columns overplotted) versus the response number.
5. Cross-plot the columns (with upto 4 columns plotted versus a single column).
6. Plot and print the decoded identifier associated with each response.
8. Create new columns as combinations of existing columns.

RUNNING CHANGO

CHANGO is executed by typing

```
CHANGO c=cmf l=lgf p=prf s=svf b=batch / t v
```

where the execution parameters are defined as

```
cmf = command file          (default is C$CMD)
lgf = log file              (default is CHGLOG)
prf = print file            (default is CPRNT)
svf = command save file     (default is CSAVE)
batch = flag for batch execution
```

These parameters cause several different actions to occur. The parameters "c=", "l=", "p=", and "s=" all serve to rename files that CHANGO uses. The parameter "c=" has the additional function in that by renaming the command file, CHANGO will immediately try to obtain commands from the named file. The parameter "b=" places CHANGO into *batch mode*. When CHANGO is in *batch mode*, no messages will be sent to the terminal. Also, a command file must be supplied on the execute line when going into *batch mode*. If the value of "b=" is not batch, CHANGO will not go into *batch mode*. These parameters may be entered in any order and may be omitted as well. When parameters are omitted, the defaults are assumed. For example, the execute line

```
CHANGO c=job1 b=batch / t v
```

would result in the following actions: CHANGO would enter into *batch mode* and would then immediately try to obtain commands from the file job1. The log of the execution would be found in file chglog; all printed data would be found in file cprnt; and CHANGO would not save any commands into file csave.

GENERAL INFORMATION ABOUT CHANGO

There are several features in CHANGO that make it easy to use: command continuation on another line; command stacking; command duplication without re-typing the complete command word; free-format input; use of keyword= for command options; and the ability to obtain and execute commands from a command file.

Command Continuation

The special character "&" allows continuation of a command. When this character is preceded and followed by a blank, CHANGO will prompt for more input if input is from the terminal. If input is from a command file, CHANGO will interpret the next record in the command file as a continuation of the current command. Since CHANGO accepts input in the form of a mock 80 column "card," the continuation character "&" must occur at or before column 80 because no data past 80 characters of input is read. A given command may have any number of continuations. An example of the use of the continuation character is shown below (following the CHANGO prompts which appear underlined).

```
start of command => command ? device mon= 1231 uxon box= &
continuation => box= ? "box r09 gerhard"
```

Command Stacking

Commands may be stacked for sequential execution. This procedure is handy when entering short command lines on the terminal (i.e. DEVICE and OPTIONS commands). Commands are separated by means of the special character "\". On older model terminals (Teletypes or ttys), the character "\" may occur as a shift of the L key, while on most 7-bit terminals it is a separate key. In the event of an error CHANGO will purge the stack and leave a clean slate. An example of how stacking might be used is shown below.

```
setup lowercas\plot 17 18 vs 5 xlab="column 5"\make 4 from 3 + 5
```

Command Duplication

Whenever the same command is to be executed several times in succession with different data, it is not convenient to type out the complete command word each time (apart from the first time). CHANGO allows the user to signal re-use of the most previous command word by preceding the data for the command with the special character ";". When the character ";" is used, it must occur as the first non-blank

character and must be followed by a blank. When the command is placed into the CSAVE file, the total command word is substituted for the duplication character ":". For example, the following command is issued.

```
command ? rdset prsout1 v3=5 v4= 9 clear.
```

If the next command is

```
command ? : prsout2 v1=3 v5=1 v3=8
```

This command will be interpreted as the RDSET command and the following image will be saved in the CSAVE file.

```
rdset prsout2 v1= 3 v5= 1 v3= 8
```

Free-Format Input

CHANGO has free-format input. Thus, all inputs may be delimited by either a comma "," or a blank ". These delimiters may appear as many times as one likes. While several blanks look ok, a string of commas seems silly. All numeric inputs may be input with or without the decimal point for both integer and real numbers. Some of the legal forms for numeric input are:

```
nn      .nnn   nn.   nn.nnn   +nnnn  -nn.nnnn
nnn.nnE+nn  nnn.nEnn  .nnnnE-nn  +.nnE-n
```

Other combinations can be produced and one should get the idea. The main restrictions are that numeric (and alphabetic) input may not have any imbedded blanks and that the values which "n" may take are limited to the digits 0 through 9. The character "E" indicates the use of the exponent form, in which digits following "E" signify powers of 10. These powers of 10 then multiply the number preceding the E).

Keyword Equals

In following with free-format input for CHANGO, options for commands may be entered in any order. This convenience is accomplished by the use of "keyword" and "keyword equals" designators (key and key=). Any option not named by either of the two designators is assigned its default value.

Command File

CHANGO allows the user to either enter commands interactively from the terminal or obtain commands from a disk file called the "command" file. CHANGO saves all successfully executed commands during each session in the file CSAVE, overwriting an existing CSAVE file. Generally the command file comes from a past CSAVE file. If the user wishes to save or reuse the resulting file CSAVE, he must switch its name. For example, after a CHANGO session, file CSAVE is created. At a future time, it is desired to run the same session with a few modifications; thus, the file CSAVE may be renamed JOBIN and modified with a text editor until it is in the desired form. When CHANGO is then initialized, the execute line could include c=jobin as one of the parameters. For further discussion of this feature see the READ command (page 7).

Interrupts

Execution of CHANGO may be interrupted at almost any time by typing (CTRL-E). CHANGO cannot be interrupted when it is performing an RDSET command (read a set of data). Upon interruption, CHANGO will abandon its current operation, end any graphics frames that are in progress, and prompt for the next command. If CHANGO is in batch mode, CHANGO will terminate rather than prompt for the next command. If CHANGO is obtaining commands from a command file, CHANGO will return to interactive input.

HANDLING ERRORS

CHANGO has an exceptional error recovery system. Two main types of errors can occur: errors that are recoverable within the command and errors that are non-recoverable. In either case, CHANGO prompts for terminal input for correction if appropriate and will not return to the command file, if in use, until the user so instructs.

The user can treat recoverable errors in either of two ways: by correcting the error, thus recovering, or by aborting the command. For recoverable errors CHANGO will print the offending input line and the appropriate error message. CHANGO will then prompt for data to continue the command. CHANGO will discretely identify the input field that initiated the error (i.e. "value of mon= must be positive"). All data after and including that field is lost. In this example the input line is cleared starting with the value for "mon=" and CHANGO prompts with mon= ?. Correction would start with a valid value for "mon=". To abort the command without recovering, do a carriage return (return key). This action results in the message "**ABORTED*" being printed on the terminal. If CHANGO is prompting for options ? after an error, it is possible to execute the command "as is" by supplying a blank input line (1 or 2 blanks followed by a carriage return) after the prompt. This action results in the execution of the command with all of the previously accepted options. Any time CHANGO prompts for options ?, a blank input line will cause CHANGO to process all of the command that has been accepted whereas a null line (carriage return with no blanks) will cause CHANGO to abort the command.

Non-recoverable errors result in CHANGO printing the offending input line, the appropriate error message and the word "**ABORTED*". CHANGO will abort all commands that cannot be corrected "mid-stream". Thus, the user should determine why such an error was encountered before attempting the command again. Typically, all errors having to do with file names result in an abort.

In the event of either type of error, if several commands have been stacked (see page 3), they are purged and CHANGO will print "**STACK CLEARED** THESE COMMANDS PURGED", followed by one line displaying the purged commands. Thus, those commands are lost and the user must re-enter them upon recovery. The offending command is printed with its appropriate error message after the stack cleared message is printed.

BASIC COMMANDS

There are several basic commands that allow the user to exercise control over the mode of execution, the source of command input, and termination. Two commands have been included for documenting the execution and for reporting the current stage of execution.

READ (filenam)

This command causes subsequent commands to be obtained from a command file, rather than from the terminal. If filenam is omitted, CHANGO will continue to use the command file that is currently open (if any). If the filenam that is included on the READ command is the same as the name of the currently opened command file, commands are obtained from filenam starting with the next available command (filenam is not rewound). If filenam is not currently open, the current command file is closed, filenam is opened and commands are obtained starting with the first record. NOTE: If an end-of-file has been encountered while reading filenam, another READ for that file will cause CHANGO to crash.

example: read job1

RETURN (comment to be printed upon return)

This command is applicable only to command files and is used to return control to interactive input from command file input. When RETURN is executed, the complete command line including the optional comment is printed on the terminal. NOTE: If CHANGO is in batch mode, CHANGO will terminate upon encountering a RETURN

example: return this is the end of the rdsets

C (printable comment)

This command allows the user to print comments from the command file to the terminal during the execution of the command file. The complete C line is printed. If C is executed interactively, the only effect is to save the comment in the CSAVE file. NOTE: If CHANGO is operating in *batch mode* no comments will be printed.

example: c now reading columns from prscut12

N (non-printable comment)

N is designed to be used in the command file for documentation and is not printed to the terminal during execution of the command file. N can be used to place spaces between comments in the command file by using a blank comment. All N commands are saved in the CSAVE file.

example: n
n input file for chango date 18-feb-82
n

END

All good interactive codes must have some way to terminate. END will obtain a log of the time used by CHANGO and then terminate CHANGO. The only other way that CHANGO can terminate is for CHANGO to encounter an error, a prompt, or an interrupt while in *batch mode*.

example: end

GRAPHICS COMMANDS

CHANGO is a graphics post-processor, thus it is necessary to exercise control over the various graphics devices, namely the TMDS and the generation of a UX80 plot file.

DEVICE < mon= box= tvon tvoff uxon uxoff rjet= >

This command is the basic graphics command. It attaches and/or releases the TMDS ("tvon" and "tvoff") and the UX80 file ("uxon" and "uxoff"); it sets or changes the TMDS monitor ("mon="); and it specifies the box number and id for the UX80 file ("box="). The default mode is "tvon" and "uxoff". This default means that upon execution of DEVICE, the only required data is the "mon=" option which will hook up the TMDS of your choice but will not save the frames in a UX80 plot file. The UX80 plot files may be output to an RJET automatically using the option "rjet=". The various options are discussed in greater depth below. NOTE: Frames are plotted only if the DEVICE command initializes either the TMDS or the UX80 file.

"mon=" -- The monitor to be attached. Monitors can be changed at will by executing DEVICE with a new monitor number. If DEVICE is executed and the status is "tvon" (default) but "mon=" is not specified, CHANGO will prompt for the monitor number.

"box=" -- The UX80 file box number and id. This information is required for attaching a UX80 file. The value associated with "box=" is a character string of up to 30 characters enclosed within quotes " ". The first 3 characters must be box. The first time a UX80 file is requested (either by "uxon" on the DEVICE command or by TAKE, page 10), a box and id is required. When "uxon" is requested, if "box=" is not specified on the DEVICE command, CHANGO will prompt for the box number and id. Once the box number and id have been input, it is not necessary to input them again. The initial box and id cannot be overridden.

"tvon" -- Activate the TMDS. "tvon" implies that a "tvoff" has already been done. The normal mode is "tvon". Thus one needs to input only the monitor number ("mon=") to hook up the TMDS.

"tvoff" -- Turn the TMDS off. This command is useful if all one wants is the UX80 plot files. Since DEVICE defaults to "tvon" and will thus prompt for a monitor number, "tvoff" is used to inhibit this default. The general use would be in *batch mode* operation where no interactive processing is done.

"uxon" -- Activate the UX80 file. This option will cause all

subsequent frames to be saved in the UX80 plot file until "uxoff" is specified.

"uxoff" -- Deactivate the UX80 plot file. This option turns off the automatic saving of all frames into the UX80 plot file. Thus, to save any subsequent frames, one must either reselect "uxon" or execute TAKE (see page 10).

"rjet=" -- The RJET number for plot output at termination. A value of zero or less will turn off the RJET request (default).

example: device mon = 1231 uxon box="box r09 test1"

TAKE < "box and id" >

This command will save the current frame into the UX80 plot file if the UX80 plot file has been previously initialized. If the current frame is still in-progress, it will be completed and then saved. If the UX80 plot file has never been requested by TAKE or by DEVICe (see page 9), then a box number and id must be input. The box and id may be included on the TAKE command by enclosing it within quotes " ", or if not included, CHANGO will prompt for it. In either case, the box number and id must be enclosed within quotes and must begin with the characters box. The TAKE command is useful when operating in an interactive mode, especially when doing overplotting, allowing one to save only those frames that are really desired. If "uxon" has been selected by the DEVICe command, TAKE need not be executed. NOTE: If the UX80 plot file has not been previously initialized, the current frame is not saved. It must be reconstructed to save it.

example: take "box r09 test2"

MANIPULATION COMMANDS

CHANGO has a variety of manipulation commands. The primitive commands set global parameters, define the maximum array size and read in sets of data from PRESTO summary tables. The remaining commands can create new columns as combinations of old ones, sort the columns according to structure and component, and place gaps between free-field data and structure data.

SETUP < ncol= pc1= pc2= pc3= pc4= full short uppercas lowercas
 minic smallc mediumc largec >

SETUP sets various plotting options and determines the maximum array size to be used.

"ncol=" -- Set the maximum array size to be used. This number is actually the maximum number of columns to allow. The column length is determined from the PRESTO file. The maximum "ncol=" allowed is 100. The default for "ncol=" is 20.

"pc1= ... pc4=" -- Set the character to be used for point plotting. Since one can plot upto four columns on the same plot, these options allow one to assign different characters to be plotted for each of the different columns. The default characters are "+", "#", "0" and "*". NOTE: Care should be taken to pick characters that fill the complete character block as all characters are positioned based on the center of the character block. Periods (.) and quotes (") are poor choices while pound signs (#), plus signs (+) and zeros (0) are good choices.

"full" -- Make all subsequent plots full size, square plots.

"short" -- Make all subsequent plots short, rectangular plots with the vertical dimension 70% of the horizontal dimension (used to output to the RJET without shrinking the plots).

"uppercas" -- Make all plotting characters uppercase (default).

"lowercas" -- Make all plotting characters lowercase.

"minic" -- Make all plot labels and text (except for plotting characters) miniature size.

"smallc" -- Make all plot labels and text small size (default).

"mediumc" -- Make all plot labels and text medium size.

"largec" -- Make all plot labels (not text) large size.

example: setup mediumc pc1= # ncol= 35

RDSET *filename* < v1= v2= v3= v4= v5= v6= clear. >

This command reads selected data vectors from the PRESTO file *filename* and stores them in the specified columns of the large data matrix. There are six vectors available from each PRESTO file. Thus, one can store zero to six of them in any of the available columns. Care should be taken, in that it only makes sense to read sets of data into the data matrix before gapping or sorting the matrix. To start a completely new set of data reads and manipulations, the first RDSET command should include the "clear." option. This option clears the work space and hopefully nothing weird should happen. NOTE: The seventh vector in the PRESTO summary table is the IDENTITY of the responses. This vector must match all previous IDENTITY vectors from other PRESTO summary tables, or CHANGO will not accept the RDSET command. There is one exception. If the "clear." option is used, it is a fresh start, and the IDENTITY vector encountered on this RDSET command becomes the vector other PRESTO summary tables must match.

"v1= ... v6=" -- Read selected vector and place it into designated data column. "v1=" through "v6=" designate the vector to read in the PRESTO summary table, and the number following it indicates which data column to store it into. For example, "v4= 14" means to read vector 4 from the PRESTO table and put it into data column 14. Thus, vector 4 from this PRESTO file would subsequently be referred to as column 14 for all plotting and manipulations.

"clear." -- Clear the workspace prior to reading in new vectors. This option is necessary when starting a new batch of vector reads after having already done GAP or SORT. If neither of these commands have been executed, "clear." is not needed.

example: rdset prsout1 v3=5 v4=11 v6=12 clear.

GAP { ngap }

The GAP command places gaps between certain types of data, for plot clarity. The default gap size is 5 spaces. A different gap size can be ordered by including a numeric value (ngap) on the command, where 1 ≤ ngap ≤ 50. The gaps will occur after free-field accelerations, foundation accelerations and fixed-base accelerations. If both free-field and foundation accelerations are present, two gaps will be constructed.

example: gap 15

SORT

The SORT command will sort the responses occurring after the free-field and foundation accelerations. The responses are already sorted by structure number. SORT will sort first by response component, and then by response type. Response components are sorted in ascending order; response types are sorted such that nodal responses are first, base forces are second and element forces are third.

example: sort

MAKE col1 from col2 (op col3) < scaleby >

The MAKE command constructs new columns as combinations of other columns. The parameters col1, col2 and col3 are *column numbers*. These numbers correspond to the *column numbers* utilized with the RDSET command (see page 12). The word *from* must separate the parameters col1 and col2. The most trivial construction is to copy one column into another (i.e., MAKE 5 from 12). New columns can be constructed as a combination of two columns, when the parameter pair "op col3" are included. Here op is one of the four basic mathematical operators "+", "-", "*", or "/". Thus col1 can be the sum, difference, product, or quotient of col2 and col3. The option "scaleby" can be used to scale col3 prior to the actual mathematical operation. "scaleby" is followed by a numeric scale factor. The default for "scaleby" is 1. NOTE: The terms col1, col2 and col3 refer to a complete column of data. Thus, the indicated MAKE operation is performed on each element of the appropriate columns.

example: make 17 from 5 + 3 scaleby 32.174

in identifying the characteristics of a particular response number. If the parameter pair "vs col2" is included in the command, the columns col1...col1c are plotted versus col2.

"t1=", "t2=" -- Set the plot title. There are two title lines (referred to as "t1=" and "t2=") that can be specified for each plot. The default for "t1=" is a blank line; the default for "t2=" is blank for response number plots and "CROSS PLOT" for cross-plots. Titles must be enclosed within quotes (" ") and are limited to 60 characters; excess characters are ignored.

"xlab=" -- Set the abscissa label. Default is "RESPONSE NUMBER" for response number plots and "ABSCISSA VALUE" for cross-plots. The abscissa value must be enclosed within quotes and is limited to 30 characters; excess characters are ignored.

"ylab=" -- Set the ordinate label. Default is "VALUE OF RESPONSE" for response number plots and "ORDINATE VALUE(S)" for cross-plots. The ordinate label must be enclosed within quotes and is limited to 30 characters; excess characters are ignored.

"xmin=", "xmax=" -- Set the minimum and maximum abscissa limits or cross-plots.

"ymin=", "ymax=" -- Set the minimum and maximum ordinate limits for all plots.

example: plot 4 8 ymin= -5 ymax= 7 t2="plot 1" ylab="accel"
(response number plot)

plot 18 26 31 vs 42 xlab="column 42" ylab= &
"columns 18 26 31" t1="test problem 1" xmax= 100
(cross-plot)

COMMAND ABBREVIATIONS

CHANGO will read only the first five characters of a command name. Thus longer command names can be shortened (i.e. RETURN and be input as RETUR). This manual indicates the required letters in the command names by placing them in all caps. The lower case letters are optional. Some feel it is advantageous to have even shorter commands. I have incorporated a set of unambiguous abbreviations. These abbreviations must be typed exactly. There is no option for typing something in between the abbreviation and the full command word. Following is an alphabetic list of the commands and their abbreviations. As noted previously, the required letters are in caps.

C		C
DEVIce		D
END		E
GAP		G
IDList		I
MAKE		M
N		N
PLOT		P
RDSET		RDS
READ		RD
RETURN		RT
SETUP		ST
SORT		S
TAKE		T
WRITE		W