Calhoun: The NPS Institutional Archive

DSpace Repository

Theses and Dissertations   1. Thesis and Dissertation Collection, all items

1989

Utilization of a PC for finite element modeling.

Collins, Robin A.

Monterey, California. Naval Postgraduate School

http://hdl.handle.net/10945/26329

UTILIZATION OF A PC FOR FINITE ELEMENT MODELING

A Special Research Problem

Presented to

The Faculty of the School of Civil Engineering

by

Robin A. Collins

August, 1989

GEORGIA INSTITUTE OF TECHNOLOGY
A UNIT OF THE UNIVERSITY SYSTEM OF GEORGIA
SCHOOL OF CIVIL ENGINEERING
ATLANTA, GEORGIA 30332

UTILIZATION OF A PC FOR
FINITE ELEMENT MODELING

A Special Research Problem

presented to
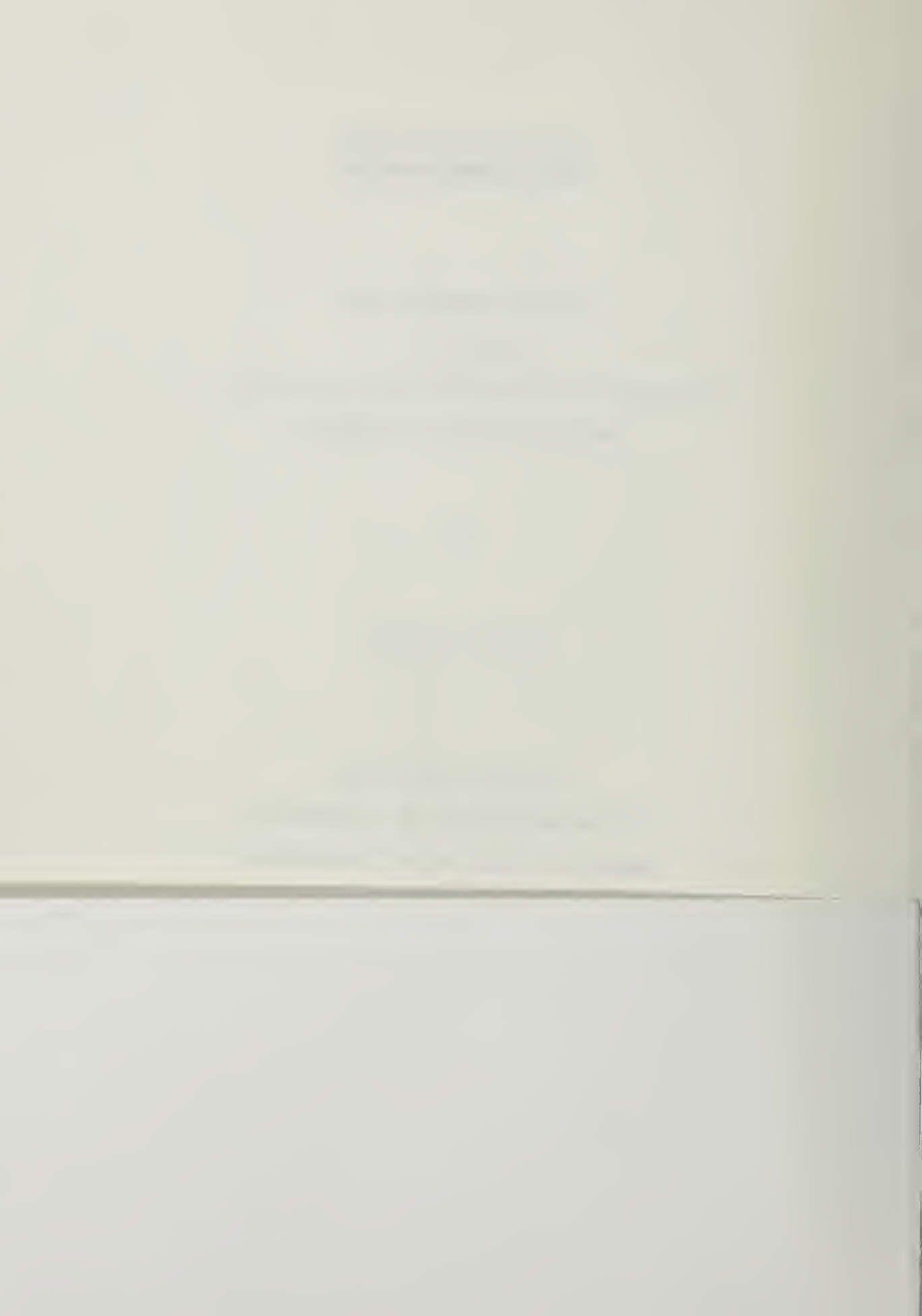
The Faculty of the School of Civil Engineering

Georgia Institute of Technology

by

Robin A. Collins

In Partial Fulfillment

of the Requirements for the Degree of

Master of Science in Civil Engineering

ACKNOWLEDGEMENTS

TABLE OF CONTENTS

CHAPTER ONE

INTRODUCTION

INTRODUCTION

In the past, Geotechnical Engineering has always been sort
of a "black science" as opposed to an "exact science",
where the ability to predict what will occur beneath the
earth's surface was often based primarily on experience.
Over the years this has slowly changed. With the advent
of the Finite Element method of analysis, Geotechnical
Engineering leaped forward towards becoming a more exact
science. The ability to predict is still based on
experience. However the predictions are much better
today.

Finite element analysis has a couple of draw backs though.
One draw back is that mesh generation is very tedious and
time consuming. The second draw back is that the overall
mathematical analysis is also very time consuming. With
the invention of the computer the mathematical analysis
draw back is pretty much avoided, although at a
substantial monetary cost. Performing a finite element
analysis on a mainframe computer takes a lot of time and
as a result cost a lot of money. As computers have become
more sophisticated and quicker, the running time has
decreased. However the cost per unit of time has
increased. Therefore, a dynamic finite element analysis
still costs a fair amount.

With the fast computers now available to perform finite
element analyses, they became much more common. However
they still have the first drawback of the tedious mesh
generation. That is until programs were developed that
perform mesh generation themselves.

Computers have continued to become more sophisticated.
With the inception of the 386 generation personal computer
(PC), many of them have a greater capability than the
mainframe computers of yesteryear. In addition to their
large capabilities these PCs are relatively inexpensive.

With the increasing capabilities of these PCs there there
is an increasing amount of work that can be accomplished
with them. As a result over the last forty years the
engineering field has gone from the very tedious hand
development of a finite element mesh and its' analysis, to
just recently being able to use a PC to develop and
analyze a finite element mesh.

This capability of using a PC to develop and analyze a
finite element mesh has opened a whole new area of
analyzing complex problems by the practitioner. Not only
has it opened new areas but it has substantially decreased
the cost. For the cost of performing three fairly complex
analyses on a mainframe computer, a consulting firm can
purchase a PC system capable of performing those same

analyses, a commercial finite element analysis program
with mesh generation and post processing capability, and
have personnel trained. As a result, the actual cost to
the consultant for performing a fourth analysis is the
personnel cost of developing the mesh, which would have
occurred any way, and the minimal cost of computer and
program maintenance.

There are many comprehensive finite element analysis
programs, for dynamic and static problems, available on
the market today. Programs such as STRUDL, NISA, ANSYS,
and NASTRAN are readily available, to just name a few, and
relatively easy to use. After obtaining an office
computer system and finite element program package, a new
user merely has to sit down and "use" the program to learn
how it operates. This "use" should consist of first
reading a few of the basic chapters in the user's manuals
and then trying to develop and analyze problems with
"known" solutions. Once the new user is able to obtain
analysis responses that match the "known" solutions they
are ready to go. As becoming intimately familiar with a
comprehensive computer program may take many months, it is
recommended that the new user "cross-check" the program
output, with another method, until they are very
comfortable with the program and its performance.

Until the start of this master's research problem the

author had very little computer experience and only a very
basic understanding of what a finite element analysis was.
This research problem consisted of providing a "new user"
the opportunity of learning, from scratch, how to generate
finite element meshes and analyze them using the NISA II
computer program. This involved loading the program onto
a PC, modeling problems that had solutions readily
available (ie verification problems obtained with the
user's manuals) and then analyzing problems and comparing
the response to theoretical responses or those provided by
a different finite element program (GTSTRUDL).

The following chapters will provide an individual that has
no finite element experience and/or NISA II program
experience with a more clarified user's manual,
highlighting the problem areas the author encountered, and
some comparisons between NISA answers and those provided
by theory or another finite element program (GTSTRUDL).

Appendix A provides a list of the equipment required to
operate the NISA II program and appendix B provides an
idea of the time required to perform analyses of various
sizes.

CHAPTER TWO

USER'S MANUAL

INTRODUCTION

There are basically three main methods of developing a
finite element model for analysis. Method one consists of
using a finite element modeling program like PATRAN. Once
the model has been created NISA II can preform the
analysis after the model has been interfaced with the
DISPLAY-PRE program via a neutral file system. Method two
consists of using a text or line editor, like EDLIN, and
writing an ASCII file following the format in Sections 5
to 8 (input & executive commands, model data, analysis
data, and modal dynamic data respectively) of the NISA II
User's Manual. The third method consists of using the
DISPLAY-PRE program portion of the NISA II package.

This user's manual will concentrate on how to develop
finite element models using the DISPLAY-PRE program, and
then how to perform a finite element analysis using the
NISA II program. Since writing an all inclusive users
manual is beyond the scope of this project, it is the
author's intent that this users manual clarify and
simplify the generation and analysis of finite element
models using the NISA II documentation. Therefore, this
chapter should be used concurrently with the actual NISA
II User Manuals.

The DISPLAY-PRE program is a user friendly program that,

in addition to providing two modes for creating models
(menu mode and command mode), automatically produces on
the screen for viewing, a geometric representation of the
model being created. The first mode of creating a model
is the menu mode which guides the user through the various
commands by way of a pyramid. This allows the user to
select a "main" command and then allows the selection of
an option from the "main" command's sub-menu. The second
mode is the command mode. This allows the user to bypass
the menu mode and input desired commands and sub-commands
directly (for someone who is familiar with the program the
command mode is the quicker method). The commands for
both modes are identical. To access the command mode once
the user is in the DISPLAY-PRE program merely type "c".

COMMAND FORMAT

The general format for the command input consists of:

Entry #	1	2	3	4	5
variable	name	function	input	output	data

Name: This is the "main" command.

Function: This is the desired option available from the
          "main" command.

Input: This is the list of identities that the function

uses to perform the command.

Output: This is the list of generated entities.

Data: This is the data used by the command to perform the
required function.

TYPICAL COMMANDS

The following commands are available in the DISPLAY-PRE
program and are typical of those used to create a finite
element model for use in Geotechnical Engineering.   For
the complete list of all available commands refer to the
DISPLAY II User's Manual.

Name   Description

GRD: This stands for grid which is defined as a
point in 3-D space.   The user can create, add,
delete, plot, search, etc. a grid using this
command.   Refer to pages 3.1-1 to 3.1-37 of the
DISPLAY II User's Manual.

LIN: This command is used to define a line.   Lines
can be created through two or more grids,
plotted, deleted, etc. using this command.
Refer to pages 3.2-1 to 3.2-64 of the DISPLAY-
PRE User's Manual.

PAT: This command stands for patch and is used to
       create or manipulate a patch. A patch is a
       surface defined by parametric cubic equations.
       Refer to pages 3.3-1 to 3.3-46 of the DISPLAY
       II User's Manual. Refer to the eigenvalue
       example to see a method of creating a patch and
       what one looks like.

HYP: This command stands for hyperpatch and is
       similar to the PAT command only it is used to
       create a 3-D solid entity. Refer to pages 3.4-
       1 to 3.4-28 of DISPLAY II User's Manual.

FEG: This command stands for finite element
       generation and is used to create the finite
       element mesh. Refer to pages 4.1-1 to 4.1-28
       of the DISPLAY II User's Manual.

FZM: This command stands for finite element
       generation with "topological zoom". With this
       command the user can create local high mesh
       densities without disturbing the rest of the
       mesh. Refer to pages 4.2-1 to 4.2-36 of the
       DISPLAY II User's Manual.

NOD: This command stands for node and can be used to

add, delete, search for, plot, etc. a node or
list of nodes.   Refer to pages 4.3-1 to 4.3-30
of the DISPLAY II User's Manual.

ELE: This command stands for element and is similar
to the node command.   Refer to pages 4.4-1 to
4.4-60 of the DISPLAY II User's Manual.

PRO: This command stands for property and is used to
specify the geometric properties of a property
index.   For ease of providing the specified
properties every element in the model is
assigned a property index number.   This allows
the user to input one command to assign the
geometric properties to a group of elements
instead of inputing one command for each
element.   Refer to pages 4.5-1 to 4.5-8 of the
DISPLAY II User's Manual.

MAT: This command stands for material and is used to
specify the actual material properties (example
Poisson's ratio) to a material index.   Like the
property index each element is assigned a
material index.   Therefore the user can assign
a material property to a group of elements with
one command.   Refer to pages 4.6-1 to 4.6-8 of
the DISPLAY II User's Manual.

The following example is provided to show how the property index and the material index is used to provide properties to various items. The item to be modeled consists of the following materials: a concrete beam 18 inches thick and E=4,074,280 psi (f'c=5000 psi), a concrete beam 18 inches thick and E=3,155,924 psi (f'c=3000 psi), a concrete slab (6 inches thick and E=4,074,280 psi (f'c=5000 psi), and a concrete slab 24 inches thick and E=3,155,924 psi (f'c=3000 psi). In this case the material and property index numbers could be assigned as follows:

Material	E (psi)	T (in)	MID #	PID #
beam	4,074,280	18	1	1
beam	3,155,924	18	2	1
slab	4,074,280	6	1	2
slab	3,155,924	24	2	3

FOR: This stands for force and allows the user to manipulate the forces in the model. Note: Forces are only applied to nodes. Refer to pages 4.7-1 to 4.7-14 of the DISPLAY II User's Manual.

PRE: This stands for pressure and allows the user to manipulate the pressures in the model. Note:

Pressures are only applied to elements. Refer
to pages 4.8-1 to 4.8-10 of the DISPLAY II
User's Manual.

DIS: This stands for displacement and allows the
       user to manipulate displacement constraints, ie
       boundary conditions, at specified nodes.
       Although the user's manual does not state it
       the author determined that the program will
       accept a maximum of ten variables between
       slashes "/" in entry 3 of this command. If
       greater than ten variables are required to
       define the displacement constraints, multiple
       DISP commands are required. Refer to the
       various verification examples and pages 4.9-1
       to 4.9-14 of the DISPLAY II User's Manual.

GENERAL EXECUTIVE COMMANDS

DEC: This command places the program in the mode to accept Executive commands. Executive commands tell the program what type of analysis to perform, what files to save for post-processing, etc. Refer to page 5.1-1 of the NISA II User's Manual. Appendix C is a photocopy of table 5.2.3 on page 5.2-7 of the NISA II User's Manual and provides an alphabetical listing of all executive commands available.

INS: This is an executive command that allows the insertion of data lines.

ANAL=: This tells the program what type of analysis to perform. For example "ANAL=STATIC" tells the program to perform a static analysis.

FILE=: This tells the program the prefix to be used when saving the desired data files. For example "FILE=S01" tells the program that the prefix S01 is to be used when saving any binary data file. The author determined that using greater than five alphanumeric characters in this command will cause problems with the

program trying to retrieve the internally
generated data files used for the dynamic
analysis and post processing.

Note this command is to be used in conjunction
with the following SAVE= command.

SAVE=:   This tells the program what data files are to
         be saved. For example "SAVE=26,27" tells the
         program to save the internally generated data
         files 26 and 27.   These files are named
         S0126.DAT and S0127.DAT respectively and are
         required if the user desires to perform any
         post-processing.   Section 3 of the NISA II
         User's Manual, especially pages 3.15-1 and
         3.15-2, describes a number of internally
         generated data files the program uses to
         perform its analysis.   All of these files are
         temporary files and will not be saved unless,
         specifically saved with this command.

QUI: This command either takes the program out of
         the INS mode or the DEC mode depending on which
         mode the program is in when the QUI command is
         input.

COMMAND RULES

The general rules governing command input are:

Commands can either be typed in capital letters or lower case letters.

Commas "," separate the variables (ie name, function, input, output, data). Two consecutive commas indicate a null (void) entry. <u>The program will automatically provide "DEFAULT" values to null entries, if applicable.</u>

Slashes "/" separate multiple entries in one variable (example: name, function, input #1/input #2/input #3, etc.). Like commas, two consecutive slashes indicates a null (void) entry. <u>Again the program will automatically provide "DEFAULT" values to null entries, if applicable.</u>

Units: Any unit (example feet, foot-pound, meter, etc.) can be used in this program. The only requirement is that the user maintains unit consistency.

For a more in depth description of the general rules refer to page 2.1-3 of the DISPLAY II User's Manual.

MODEL GENERATION: The following steps are required to
generate a finite element model:

Once the machine is on, get into the directory and
sub-directory that the appropriate files will be stored
in. For example on the machine the author used, all model
generation was accomplished in the sub-directory "TEST"
under the directory "EMRCNISA" on the C drive. Therefore,
to create a model the author typed in "cd emrcnisa\test"
once the C> was received. This changed the directory from
the C directory to the sub-directory "TEST" of the
directory "EMRCNISA".

Now type "NISA286" and hit the return. This will
automatically access the main menu of the NISA II program,
with the "DISPGDB" option highlighted. Hit return again
to access the DISPLAY-PRE program.

The program is now ready to accept the commands
required to generate the finite element model. As
discussed above this can be accomplished either in the
user friendly menu mode or the command mode.

To end the DISPLAY-PRE session type in the command
"END".

RUN ANALYSIS: To perform an analysis the following steps
are required:

Access the directory and sub-directory that the file

containing the model to be analyzed is in, then type
"NISA286" and hit return.

Now move the cursor, using the arrow keys, to the
desired analysis (Example: STATIC/EIGN/BUCK for either a
static, eigenvalue or buckling analysis) mode and hit
return. Note if a dynamic analysis is desired, an
eigenvalue analysis must be performed first so that the
natural frequencies of the model can be used in the actual
dynamic analysis (for more info refer to the dynamic
example).

No matter what type of analysis is to be performed
the program will ask for the input file name at this time.
For a static, eigenvalue, or buckling analysis type the
file name generated with the "WRI,NISA,(file name).NIS"
command and hit return. For a dynamic analysis type in
the file name that was created using a text editor.
Remember as stated above an eigenvalue analysis must be
accomplished before a dynamic analysis can be completed.

Now the program will ask for the output file name.
This is the file name the program will store the analysis
in. For ease of file maintenance, it is recommended to
use the same prefix as the input name and use the .out
suffix (ie "file name.OUT"). Note: if an output file name
is used that is currently employed in the originating
sub-directory, the file in that sub-directory will be
written over. After inputing the output file name hit the

return and wait for the analysis.

To end an analysis prior to its completion hit the control key "CTRL" and the "Break" key simultaneously. The program will then respond with "Strike any key when ready". At this time strike any key and the program will access the "main" menu. Now move the cursor, using the arrow keys, to "QUIT" and hit return. The program will now return back to the originating directory and sub-directory.

EXIT PROGRAM: To exit any portion of the DISPGDB (DISPLAY II) program, type end. The program will then ask if a data base file is to be written. Respond with either yes "Y" or no "N" and hit return. The program will now access the "main" menu. Now move the cursor, using the arrow keys, to "QUIT" and hit the return. The program will now return back to the originating sub-directory.

Refer to "RUN ANALYSIS" above for the method of exiting the program prior to the completion of an ongoing analysis.

VIEW A FILE: To view any file first access the directory and sub-directory that the file is in. Then use standard DOS commands such as "type" with the file name placed after the word "type" once a C>, D>, A>, etc. for the proper directory and sub-directory has been received. Example C>type s01.std; this will access the file s01.std

in the C drive and scroll it across the screen. To stop

the screen from scrolling hit the control key, "CTRL" and

the s key simultaneously. To start the scrolling again,

hit any key.

PRINT A FILE: To print a file add a ">prn" after the

above discussed type command. Example C>type s01.std>prn

The fill s01.std in the C drive will now be printed.

HELPFUL HINTS

Based on the author's experience, because the NISA II and
DISPLAY II programs are so user friendly, there really is
a limited number of helpful hints that can be provided to
the unfamiliar user. Most of the helpful hints are
methods of avoiding problems when developing either the
DISPLAY II file or the ASCII file, required for dynamic
analysis. Wherever a problem or a limitation, not listed
in the NISA II or DISPLAY II User's Manual, was discovered
a method of avoiding this problem is provided. These
methods are listed wherever they are appropriate, whether
it be in the general command, Executive command, or
verification problem section of chapter two.
Additionally, the following hints are provided:

1. Whether the user is experienced or a novice it is
recommended that the user sit down and develop the model,
to be analyzed, with paper and pencil prior to sitting at
the key board. The main reason is, it is much easier to
erase an unwanted or incorrect data line on paper than it
is to delete one once it has been inputed into the DISPLAY
II program. In fact, some command lines can not be
deleted. As a result if one of these lines is incorrectly
inputed, the user must start completely over inputing the
required model generation commands to correct the model.

2. Keep the hard disk free of unwanted clutter and
unnecessary files. Due to the program storing numerous

temporary files when in the process of performing an analysis a 10,000 degree of freedom model requires approximately 12 megabytes of disk space. A 10,000 DOF system may seem excessive for a Geotechnical Engineer and therefore why worry about space? The reason this "house keeping" needs to be accomplished is because a 1350 DOF system, which is easily within the range a Geotechnical Engineer would use, requires approximately 2 megabytes of disk space. If there is not enough disk space available two things will happen: 1. the program will not complete the analysis and 2. because the program does not complete its' analysis all of the temporary files created remain on the disk until deleted. The author at one point attempted a 1350 DOF model with 1.8 megabytes of space available and ended up with the program "crashing", 0 bytes of disk space available, and approximately 20 temporary files on the disk that had to be deleted. When the system "crashed" these temporary files are stored with the prefix the user inputed with the Executive command "File=" and the suffix ".tmp". For example if the Executive command "File=s01" was used and the system "crashed" one of the temporary files saved on the disk may be "s0128.tmp". Note: all the internally generated files are numbered 26., 27., 28., 32., etc.

EXAMPLE PROBLEMS

Because the NISA II Verification Problems Manual was written prior to the development of the DISPLAY-PRE program, those problems do not provide the DISPLAY-PRE commands required for model generation. As a result the following problems were selected from the Verification Problems Manual, as good representative examples of what a Geotechnical Engineer may encounter, for a step by step description of how to develop the model using DISPLAY-PRE and then how to perform the desired analysis.

In order to complete a dynamic analysis of a structure, facility, mass etc. two files are required to "design" the finite element model. The first file contains the actual geometric and material properties of the model and is developed using the DISPLAY II program. This file is then used to complete an eigenvalue analysis of the model. The eigenvalue analysis determines the natural frequencies of the system, which is then used in the dynamic analysis. The second file is written using a standard text or line editor, like EDLIN, and contains the loading characteristics of the model. Additionally, this file contains the commands required to tell the program what the desired output is. This file is generated following Section 8 "MODAL DYNAMIC DATA" of the NISA II User's Manual.

The following examples consists of the input file

(the actual commands inputed during the DISPLAY-PRE

session), the NISA file (the actual file that is created

by the DISPLAY-PRE pre-processing program and is used

during the finite element analysis), and a line by line

description of the input file. Additionally, the dynamic

problems contain the dynamic input file created using the

text or line editor and a line by line description of this

input file.

The line by line descriptions discussed above do not

discuss every line of the input file. However, they do

discuss every different line of input. If there is more

than one line of input performing the same function, for

example a model may contain five lines of input listing

the movement constraints the model will have, only one of

the input lines is discussed.

VERIFICATION PROBLEM 2.20

TITLE:

Beam on a tensionless elastic foundation

ELEMENT TYPE:

2-D beam element (NKTP = 13, NORDR = 1)
2-D spring element (NKTP = 18, NORDR = 1)
2-D gap element (NKTP = 42)

PROBLEM:

A 30 inch long beam (3 inches wide, 1 inch deep) rests on an elastic soil foundation (Fig. 2.20.1). The soil is active in compression only. A concentrated load of 8000 lbs. is applied at the midspan of the beam. The foundation modulus is 5000 lb/in^3.

The midspan displacement and the contact length of the beam are calculated.

PROPERTIES:

Material:
EX   =   30.0E6 psi   (Modulus of elasticity of beam material)
NUXY =   0.0   (Poisson's ratio)
Cross-section:
IZZ   =   0.25 in^4 (Second area moment about beam Z-axis)
A   =   3 in^2 (Cross-sectional area)

FINITE ELEMENT MODEL:

Due to symmetry, only one half of the structure is modeled using 2-D beam, 2-D spring and 2-D gap (or compression-only spring) elements (Fig. 2.20.1). The beam is divided into eight equal length beam elements with two nodes per element. The tensionless foundation is simulated by nine discrete equidistant springs. The two springs (near the midspan) are assumed to be active in tension and compression. These two springs support the beam initially. The remaining seven compression-only springs are modeled by gap elements with negative flexibility coefficients and zero gap width.

The spring constant (K) is taken as foundation modulus times beam-width times spacing between the springs (K = 5000*3*1.875 = 28125 lb/in). The flexibility coefficient (F) for a typical compression-only spring is the reciprocal of the spring constant (F = -1/K).
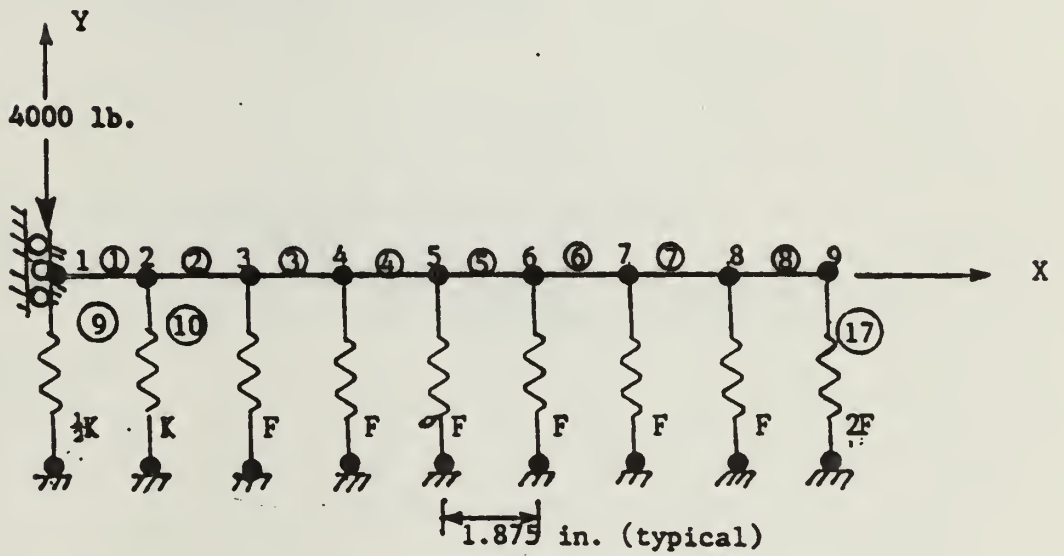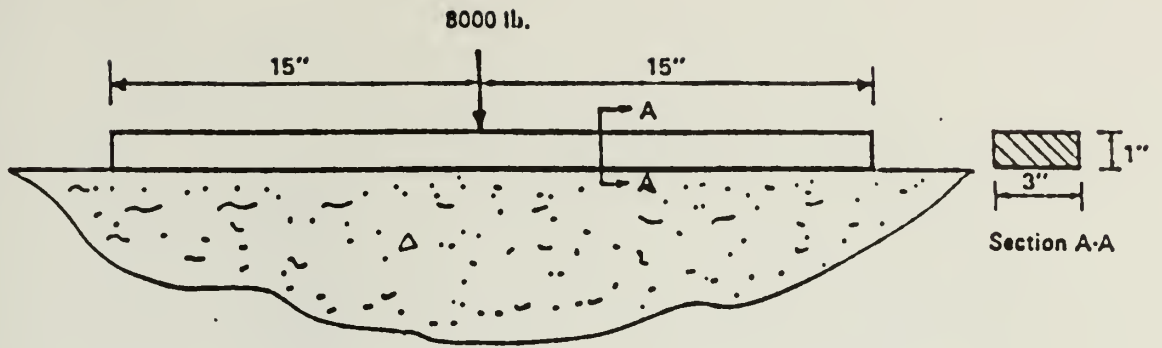
Figure 2.201 Geometry, loading and finite element model for
a beam on elastic foundation.

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,15
LIN,2GD,1,2,1
FEG,BAR,1,,,8/13,1/1
LIN,TRS,1,2,/-10
FEG,BAR,2,,,8/13
ELE,DEL,9T16
ELE,ADD,9,18/1//2,1/10
ELE,ADD,10,18/1//3,2/11
ELE,ADD,11,42/1//4,3/12
ELE,ADD,12,42/1//4,4/13
ELE,ADD,13,42/1//4,5/14
ELE,ADD,14,42/1//4,6/15
ELE,ADD,15,42/1//4,7/16
ELE,ADD,16,42/1//4,8/17
ELE,ADD,17,42/1//5,9/18
DISP,ADD,1,1,0//////0
DISP,ADD,2T9,1,0
DISP,ADD,10T18,1,0/0
FORCE,ADD,1,1,/-4000.
PROP,ADD,1,3/0.25/0/0
MAT,ADD,1,30.0E6/0.0
PROP,ADD,2,14062.5
PROP,ADD,3,28125
PROP,ADD,4,0./-0.35555E-4/0./1.0E-8/-90
PROP,ADD,5,0./-0.71111E-4/0./1.0E-8/-90
DEC
INS
ANAL=STATIC
FILE=S01
SAVE=26,27
*TITLE,ATTEMPT OF VERIFICATION PROBLEM S20.STD
*I3,1,0,1,0,0,0,0,0,0,0.0000E-01,0.0000E-01
*I4,1,POINT LOAD AT BEAM'S CENTER
QUI
QUI
WRI,NISA,S01.NIS
WRI,DBS,S01.BIN
END
N
```

*TITLE
ATTEMPT OF VERIFICATION PROBLEM S20.STD
*ELTYPE
  1, 13,   1
  2, 18,   1
  3, 42,   1
*RCTABLE
     1,   4
  3.0000, 0.2500, 0.0000, 0.0000,
     2,   1
14062.5
     3,   1
28125
     4,   5
0.0000,-.35555E-4,0.,1.E-8,-90
     5,   5
0.0000,-.71111E-4,0.,1.E-8,-90
*ELEMENT
     1,   1,   1,   1, 0
     1,   2,
     2,   1,   1,   1, 0
     2,   3,
     3,   1,   1,   1, 0
     3,   4,
     4,   1,   1,   1, 0
     4,   5,
     5,   1,   1,   1, 0
     5,   6,
     6,   1,   1,   1, 0
     6,   7,
     7,   1,   1,   1, 0
     7,   8,
     8,   1,   1,   1, 0
     8,   9,
     9,   0,   2,   2, 0
     1,   10,
   10,   0,   2,   3, 0
     2,   11,
   11,   0,   3,   4, 0
     3,   12,
   12,   0,   3,   4, 0
     4,   13,
   13,   0,   3,   4, 0
     5,   14,
   14,   0,   3,   4, 0
     6,   15,
   15,   0,   3,   4, 0
     7,   16,
   16,   0,   3,   4, 0
     8,   17,
   17,   0,   3,   5, 0
     9,   18,

```
      1 ,,,,        0.0000,       0.0000,       0.0000,
      2 ,,,,        1.8750,       0.0000,       0.0000,
      3 ,,,,        3.7500,       0.0000,       0.0000,
      4 ,,,,        5.6250,       0.0000,       0.0000,
      5 ,,,,        7.5000,       0.0000,       0.0000,
      6 ,,,,        9.3750,       0.0000,       0.0000,
      7 ,,,,       11.2500,       0.0000,       0.0000,
      8 ,,,,       13.1250,       0.0000,       0.0000,
      9 ,,,,       15.0000,       0.0000,       0.0000,
     10 ,,,,        0.0000,     -10.0000,       0.0000,
     11 ,,,,        1.8750,     -10.0000,       0.0000,
     12 ,,,,        3.7500,     -10.0000,       0.0000,
     13 ,,,,        5.6250,     -10.0000,       0.0000,
     14 ,,,,        7.5000,     -10.0000,       0.0000,
     15 ,,,,        9.3750,     -10.0000,       0.0000,
     16 ,,,,       11.2500,     -10.0000,       0.0000,
     17 ,,,,       13.1250,     -10.0000,       0.0000,
     18 ,,,,       15.0000,     -10.0000,       0.0000,
*MATERIAL
 EX    ,     1 , ,   0.30000E+08
 NUXY  ,     1 , ,   0.00000E+00
*LDCASE
0,1,0,0,0,0,0,0.0000E-01,0.0000E-01
*LCTITLE
POINT LOAD AT BEAM'S CENTER
*SPDISP
** SET ID =        1
      1,ROTZ,     0.000000
      1,UX   ,     0.000000
      2,UX   ,     0.000000
      3,UX   ,     0.000000
      4,UX   ,     0.000000
      5,UX   ,     0.000000
      6,UX   ,     0.000000
      7,UX   ,     0.000000
      8,UX   ,     0.000000
      9,UX   ,     0.000000
     10,UY   ,     0.000000
     10,UX   ,     0.000000
     11,UY   ,     0.000000
     11,UX   ,     0.000000
     12,UY   ,     0.000000
     12,UX   ,     0.000000
     13,UY   ,     0.000000
     13,UX   ,     0.000000
     14,UY   ,     0.000000
     14,UX   ,     0.000000
     15,UY   ,     0.000000
     15,UX   ,     0.000000
     16,UY   ,     0.000000
     16,UX   ,     0.000000
     17,UY   ,     0.000000
     17,UX   ,     0.000000
     18,UY   ,     0.000000
     18,UX   ,     0.000000
 *CFORCE
 ** SET ID =        1
      1,FY   ,-0.40000E+04
 *ENDDATA
```

2.23

COMMAND>GRD,ADD,1,0/0/0

This adds a GRID (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

COMMAND>GRD,TRS,1,2,15

This makes grid #2 by "translating" grid #1 15 units in the X direction (actually it translates grid #1 15/0/0 units where the format is X/Y/Z; since there are no "0"s typed in, they are automatically assigned). If it was desired that grid #2 be placed 15 units in the Y direction, the input would be either "GRD,TRS,1,2,0/15" OR "GRD,TRS,1,2,/15". Default values are "0".

COMMAND>LIN,2GD,1,2,1

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #, line #".

COMMAND>FEG,BAR,1,,,8/13,1/1

FEG stands for "Finite Element Generation". This entry creates a group of 8 bar/beam elements (of element type 13), in the global coordinate system, assigning a

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED"

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II Users Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system,

for example a force or displacement constraint
may be defined with directions which are not
aligned with the global coordinate system. For
more information on local coordinate systems
refer to pages 3.2-1 to 3.2-4 of the NISA II
User's Manual.

Entry 7A & 7B are the material and property ID
#'s respectfully. This set of #s, assigned to
one or more elements, is used later in the
input phase for assigning actual properties.
If left blank the program automatically assigns
a material ID # of 1 and a property ID # of 1.
These numbers do not have to be the same. A
group of elements can have the same material
properties (Poisson's ratio, Young's modulus,
etc) with some of the elements having one
geometric property and others having a
different geometric property. As a result this
group of elements would have one material ID #
and two property ID #s.

Entry 7C is the zoom factor, ie the length of
the last element to the length of the first
element. If all elements are the same length
then the zoom=1. If left blank the program
automatically assigns a value of 1. For more
information and a pictorial representation
refer to page B-3 of the DISPLAY II User's

Manual.

COMMAND>LIN,TRS,1,2,/-10

This creates a second line, #2, by "translating" line
#1 -10 units in the Y direction.

COMMAND>FEG,BAR,2,,,8/13

This creates a group of 8 bar/beam elements (of NKTP
type 13) along line #2, each element is defined with a
node on each end, and each node and elements is
automatically numbered by the program. This was done
solely to create the nodes required for defining the
spring elements. Note, the elements of this line are not
needed and will be deleted later.   Another method of
generating these nodes would be by generating each
individual node using the command "NOD,ADD,NODE #,X/Y/Z
(COORDINATES)/LOCAL COORDINATE SYSTEM #". By using the
FEG command the nine required nodes, for the required
springs, can be created at the same time. By using the
NOD command creating the nine nodes would require multiple
NOD commands.

COMMAND>ELE,DEL,9T16

This deletes elements 9 thru 16. Even though the

elements are deleted the nodes still remain.

COMMAND>ELE,ADD,9,18/1//2,1/10

This adds an element #9 with NKTP=18, order=1,
material ID # =0, property ID # =2 between nodes 1 and 10,
using the following format:
"ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this is
left blank the program will automatically assign
a number.

Entry 4A is the NKTP # (see entry 6B under the
FEG description above).

Entry 4B is the order #. Some elements only
have one order, NORDR=1, others have many
depending on the element type. For a pictorial
description of how this order works/varies refer
to section 4, the "element library" of the NISA
II User's Manual; specifically NKTP #s
1,2,3,4,5,& 32 provide good examples of some of
the element types with varying orders.

Entry 4C is the material ID #. Since these are
spring elements the material characteristics are
included with the property characteristics (see
the description of NKTP # 18 in the NISA II
User's Manual). Therefore, this element does
not need a material ID number. Accordingly it

2.28

is "0".

Entry 4D is the property ID #. (See entry 7B under the FEG description above.)

Entry 5A & 5B are the two nodes that defines the element.

Entry 6 is the vector number of a previously defined vector used in establishing a local coordinate system. This can be used for elements of NKTP = 11,12,& 38.

COMMAND>DISP,ADD,1,1,0/////0

This constrains node 1 against movement in the X direction and against rotation in the Z direction, and assigns this a constraint ID # of 1 with the following format: "DISP,ADD,3,4,5A-5F", where:

Entry 3 is the list of nodes with the assigned constraint. If it was desired that all nodes from 1 to 10 be constrained, with the same constraint, the input would be "1T10".

Entry 4 is the constraint ID #. The set ID under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that are constrained. This is accomplished by placing a "0" in the degree of freedom that must be constrained with the following format: "X/Y/Z/ROTX/ROTY/ROTZ". If the desired

constraints are in the Y and ROTX directions the
input would be either "/0//0" or "/0//0//".
Consecutive slashes indicate a null entry which
the program identifies and either ignores or
places a default value in the location. In this
case the program ignores the entry. For more
information on null entries refer to the general
command rules previously addressed in this
chapter or to pages 2.1-3 to 2.1-6 of the
DISPLAY II User's Manual.

COMMAND>FORCE,ADD,1,1,/-4000

This entry adds a force of -4000 pounds in the Y
direction, at node 1, with the following format:
"FOR,ADD,3,4,5", where:

Entry 3 is the node that the force is applied
to.

Entry 4 is the force set #. A value greater
than 1 is used for multiply loading conditions.
Example: suppose the response of a model to
different loading conditions is desired, like
different load amounts at the same node and/or
the same or different forces at different nodes.
To do this one force condition would be
identified with one force set # and the other
force condition would have a different force set

2.30

#. This allows a model to be analyzed with

different loading conditions at the same time

instead of performing one analysis with one

loading condition and then remodeling the model

with the different loading condition. In this

case suppose it is desired that the systems

response be determined for a force of -4000 lbs

and 7500 lbs, both in the Y direction. The

input would be: COMMAND>"FORCE,ADD,1,1,/-4000"

AND COMMAND>"FORCE,ADD,1,2,/7500".

When the analysis is performed, answers will be

received for both loading conditions.

Entry 5 is the force value in the desired

direction X/Y/Z/MX/MY/MZ. Example: if the

desired load condition is a force of 100 lbs in

the Z direction and a moment of 10 ft.lbs. in

the Y direction the input would be either

"//100//10" or "0/0/100/0/10/0". If a position

is left blank the program will automatically

place a "0" in it.

COMMAND>PROP,ADD,1,3/0.25/0/0

    This adds a cross sectional area of 3, a moment of

inertia of 0.25, a transverse shear coefficient of 0, and

a local y-coordinate for stress recovery of 0 to property

ID set #1. A local y-coordinate of "0" means that there

is no local coordinate system, rather the global coordinate system is being used. This command has the following format "PROP,ADD,3A/3B/3C...,4", where:

Entry 3 is the property ID set that the properties will be assigned to.

Entry 4 is the list of properties which are defined by the element type with each different property being separated by a /. (Refer to the specific element type in the element library section of the NISA II Users Manual for more info on what properties are listed here).

Entry 5 is the section orientation for the beam element.

Note: there seems to be a couple of problems with this command. First the input field is a maximum of 10 characters, and the program automatically places up to four "0" after the decimal point. Therefore, if the input is a five figure number (ex 28125) the program will list ********** in the (file name).NIS file. Second the program does not seem to accept any numbers in the second or fourth fields of entry number 3, automatically assigning its own numbers. Therefore, these lines must be manually changed in the (file name).NIS file using a text or line editor like EDLIN. The (filename).NIS is discussed in more detail later.

COMMAND>MAT,ADD,1,30.0E6/0.0

This adds the material properties of Youngs modulus
of elasticity = 30.0E6 psi and Poissons ratio = 0.0 to
material ID set #1 with the following format
"MAT,ADD,3,4A/4B/4C/..." where:

Entry 3 is the material ID set #.

Entry 4 is the list of material properties.
Like the property characteristics described
above, this list will vary for each element
type.

COMMAND>DEC

This places the program in the mode to accept
Executive Commands. For more info on executive commands
refer to the previous Executive command overview and pg
7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the
insertion of data lines.

COMMAND>ANAL=STATIC

This tells the program that the analysis to be
performed is a STATIC analysis.

2.33

COMMAND>FILE=S01

    This provides the file name prefix S01 for the
internally generated data files.

COMMAND>SAVE=26,27

    This tells the program to save files 26 and 27,
internally generated and named files. The name of these
files will be S0126.DAT and S0127.DAT respectively. <u>Note
if post processing is to be performed files 26.dat and
27.dat must be saved,</u> which is accomplished using this
command. Refer to the previous Executive command overview
and section 3 of the NISA II User's Manual.

COMMAND>*TITLE,ATTEMPT OF VERIFICATION PROBLEM S20.STD

    This tells the program that the title of this
analysis is "ATTEMPT OF VERIFICATION PROBLEM S20.STD",
which the program will then print at the top of the answer
output.

COMMAND>*I3,1,0,1,0,0,0,0,0,0.0000E-01,0.0000E-01

    The *I3 is the same as *LDCASE which is the load case
"control card" for the desired output. Refer to page 7.1-
2 of the NISA II Users Manual for an explanation of what
each number means. <u>Note: there is a problem with this</u>

2.34

command on the PC version of 88.7. No matter what entry
is placed after "*I3," the program automatically changes
it to "0,1,3,0,0,4,0" on the *LDCASE card in the "(file
name).NIS" file. The "(file name).NIS file is the file
that the NISA II program actually uses for the analysis.
Therefore, if different output than what is specified with
"0,1,3,0,0,4,0" is desired, the *LDCASE line in the (file
name).NIS file must be manually changed. This can be
accomplished using a text or line editor like EDLIN.

COMMAND>*I4,1,POINT LOAD AT BEAM'S CENTER

    The *I4 is the same as *LCTITLE which means load case
title. This is the title for the specific load case. In
this example the title "POINT LOAD AT BEAM'S CENTER" is
assigned to load case 1. If there were multiple load
cases, as discussed in "FORCE,ADD,..." above, multiple
load case title cards can be used, one for each load case.
In the description for entry 5 of "FORCE,ADD..." two
loadings were discussed, one force and one moment. In
that case the input could be:
        COMMAND>*I4,1,POINT LOAD
        COMMAND>*I4,2,MOMENT
or what ever the desired titles were. The program will
then place the specified title at the heading of the
answers for each load case.

COMMAND>QUI

    Quits the insert mode.

COMMAND>QUI

    Quits the DEC mode.

COMMAND>WRI,NISA,S01.NIS

    This takes the above data just input during this
session and writes a "NISA" file called "S01.NIS" that the
program uses for it's analysis.

COMMAND>WRI,DBS,S01.BIN

    This takes the above data and writes a binary data
base file called "S01.BIN".

COMMAND>END

    This ends the session.

                *

Data Base File: At this time the program will ask if a
data base file DISP.BIN is to be written. Since this has
already been done with the WRI,DBS,S01.BIN command above,
answer no "N".

The model is now ready to be analyzed. Refer to the

previous section "Program Usage" for guidance on how to

perform the analysis.

COMMAND INPUT DEMONSTRATION

In an attempt to assist the unfamiliar user with
understanding what the various DISPLAY-PRE commands
actually do, the demonstration below provides a pictorial
description of some of the various commands required in
the previous verification problem. This demonstration
does not contain all required commands only a few to
provide an idea of how the commands work.

GRD,ADD,1,0/0/0   This creates a point at coordinates
(0,0,0).

(0,0,0)
□
|

GRD,TRS,1,2,15   This creates grid #2 by translating
grid #1 15 units in the X direction.

(0,0,0)   (15,0,0)
□   □
1   2

LIN,2GD,1,2,1   This creates line #1 between grids #1
and #2.
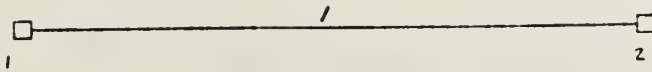
FEG,BAR,1,,,8/13,1/1   This creates 8 elements of NKTP
type #13 along line #1, and assigns MID=1 and PID=1.   Note
there is a node created at the end of each element.
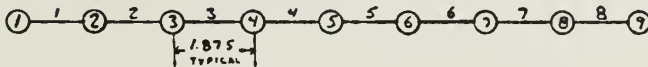
LIN,TRS,1,2,/-10   This creates line #2 by translating
line #1 -10 units in the Y direction.
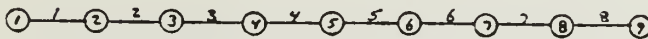
FEG,BAR,2,,,8/13   This creates 8 elements of NKTP type
13 along line #2 and allows the program to automatically
number them.   Again, as mentiond above, each element has a
node at each end.
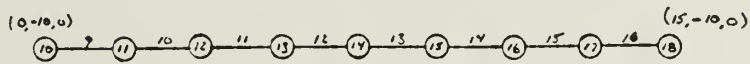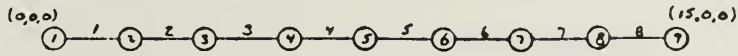
ELE,DEL,9T16   This deletes elements 9 through 16.
However, nodes 10 to 18 remain.
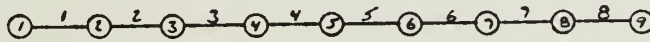
ELE,ADD,9,18/1//2,1/10   This adds element 9, of NKTP
type 18 and PID #2, between nodes 1 and 10.
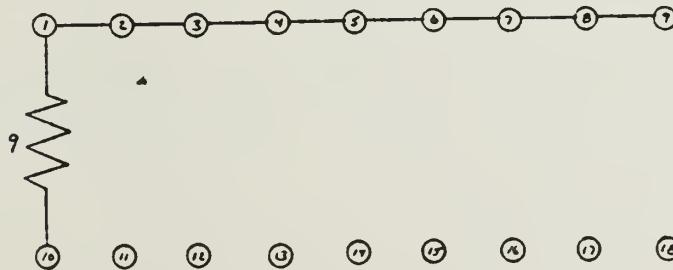
ELE.ADD.17.42/1//5.9/18   This adds element 17, of NKTP
type 42 and PID #5 between nodes 9 and 18.   Note the steps
to add elements 10 to 16 were not shown, but would
consists of basically the same command.
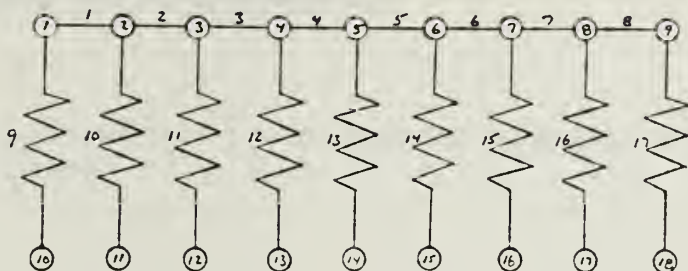
VERIFICATION PROBLEM 5.5

TITLE:

Natural frequencies of a simply supported square plate

ELEMENT TYPE:

3-D general shell element (NKTP = 20, NORDR = 2)

PROBLEM:

The first five natural frequencies of a simply supported square plate, with dimensions as shown in Fig. 5.5.1, are to be computed.

PROPERTIES:

Material:

EX   =   10.5E6 psi   (Modulus of elasticity)

NUXY =   0.3333   (Poisson's ratio)

DENS =   2.588E-4 lbs.-sec^2/in.^{4} (Mass density)

Geometry:

t = 1.0 in   (thickness)

FINITE ELEMENT MODEL:

The plate is modeled using sixteen 3-D general shell elements (NKTP = 20, NORDR = 2). The nodes on the periphery of the plate are constrained in the normal (UZ) direction. In-plane translations (UX and UY) for all the nodes are constrained to obtain only the bending modes.

SOLUTION PROCEDURE:

Conventional subspace iteration and lumped mass matrix formulation are used to extract the eigenvalues.

RESULTS & COMPARISON:

The first five natural frequencies of the simply supported plate obtained from NISA are compared to the theoretical (Ref. [1]) values in Table 5.5.1. The mode shape plots are shown in Figs. 5.5.2 through 5.5.6.

RÉFERENCE:

1.   S. Timoshenko, D. H. Young, and W. Weaver, Jr., *"Vibration Problems in Engineering"*, 4th edition, John Wiley & Sons, New York (1974).
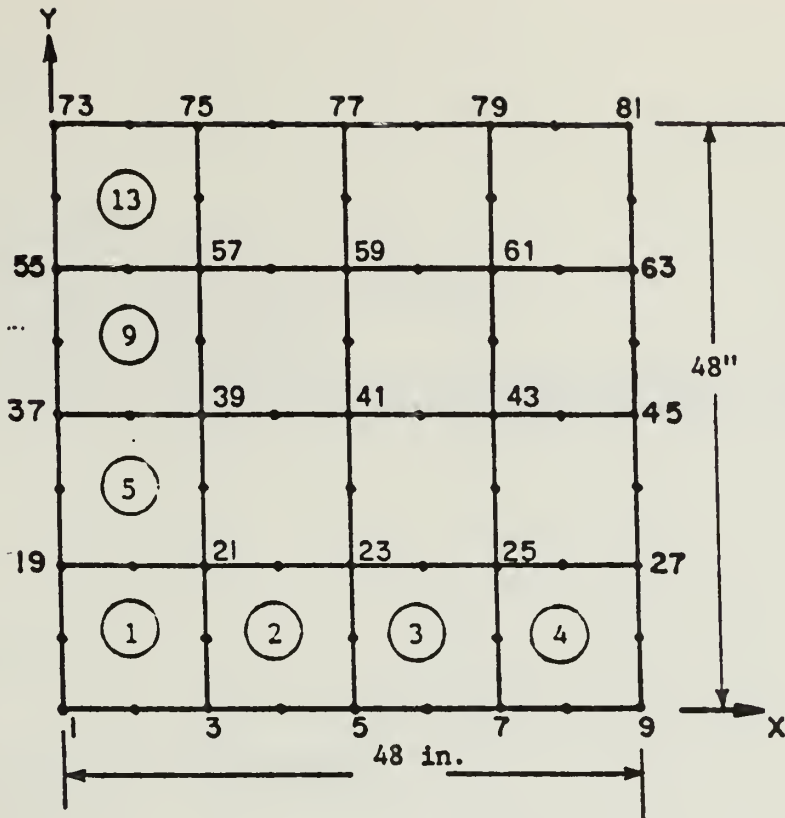
Figure 5.5.1 Finite Element Mesh for a Simply Supported Square Plate
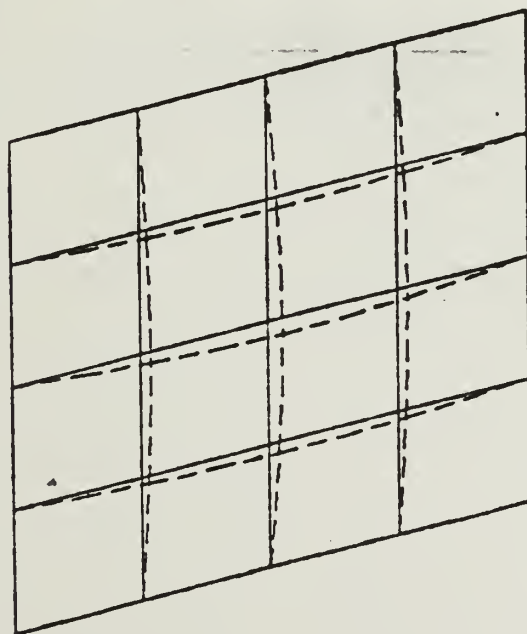
Figure 5.5.2 Mode No. 1: Frequency = 83.979 Hz

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,48
LIN,2GD,1,2,1
LIN,TRS,1,2,0/48
PAT,2LN,1,2,1
FEG,QUA,1,,,2/4/4/20
PROP,ADD,1,1/1/1/1/1/1/1/1
MAT,ADD,1,10.5E6/0.3333//////////2.5879E-4
DISP,ADD,1/9/57/65,1,0/0/0/0/0
DISP,ADD,2T8/58T64,1,0/0/0//0
DISP,ADD,10/14T15/23T24/28T29/37T38/42T43/51T52/56,1,0/0/0/0
DISP,ADD,11T13/16T22/25T27/30T36/39T41/44T50/53T55,1,0/0
DEC
INS
ANAL=EIGEN
FILE=E01
SAVE=26,27
RESE=ON
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*LDCASE,1
*EIGCNTL,5,0,20,0,0.0,0.0,1.0E-5
*MODEO,1,1,1,1
*EIGO,1,1,0,3,1,0,1
QUI
QUI
WRI,NISA,E01.NIS
WRI,DBS,E01.BIN
END
N
```

```
** EXECUTIVE
ANAL=EIGEN
FILE=E01
SAVE 26,27
RESE ON
AUTO-ON
EIGE=SUBS,CONV
MASS=LUMPED
*ELTYPE
   1,   20,       2
*RCTABLE
        1,        8
   1.0000,    1.0000,    1.0000.    1.0000,    1.0000,    1.0000,    1.0000,    1.00
*ELEMENT
        1,        1,        1,        1, 0
        1,        2,        3,       11,       17,       16,       15,       10,
        2.        1,        1,        1, 0
        3,        4,        5,       12,       19,       18,       17,       11,
        3,        1,        1.        1, 0
        5,        6,        7,       13,       21.       20.       19.       12,
        4,        1,        1,        1, 0
        7,        8,        9,       14,       23,       22,       21,       13,
        5,        1,        1,        1, 0
       15,       16,       17,       25,       31,       30,       29,       24,
        6,        1,        1,        1, 0
       17,       18,       19,       26,       33,       32,       31,       25,
        7,        1,        1,        1, 0
       19,       20,       21,       27,       35,       34,       33,       26,
        8,        1,        1,        1, 0
       21,       22,       23,       28,       37,       36,       35,       27,
        9,        1,        1,        1, 0
       29,       30,       31,       39,       45,       44,       43,       38,
       10,        1,        1,        1, 0
       31,       32,       33,       40,       47,       46,       45,       39,
       11,        1,        1,        1, 0
       33,       34,       35,       41,       49,       48,       47,       40,
       12,        1,        1,        1, 0
       35,       36,       37,       42,       51,       50,       49,       41,
       13,        1,        1,        1, 0
       43,       44,       45,       53,       59,       58,       57,       52,
       14,        1,        1,        1, 0
       45,       46,       47,       54,       61,       60,       59,       53,
       15,        1,        1,        1, 0
       47,       48,       49,       55,       63,       62,       61,       54,
       16,        1,        1,        1, 0
       49,       50,       51,       56,       65,       64,       63,       55,
*NODES
        1 ,,,,       0.0000,    0.0000,    0.0000,
        2 ,,,,       6.0000,    0.0000,    0.0000,
        3 ,,,,      12.0000,    0.0000,    0.0000,
        4 ,,,,      18.0000,    0.0000,    0.0000,
        5 ,,,,      24.0000,    0.0000,    0.0000,
        6 ,,,,      30.0000,    0.0000,    0.0000,
        7 ,,,,      36.0000,    0.0000,    0.0000,
        8 ,,,,      42.0000,    0.0000,    0.0000,
        9 ,,,,      48.0000,    0.0000,    0.0000,
```

```
10 ,,,,      0.0000,    6.0000,    0.0000,
11 ,,,,     12.0000,    6.0000,    0.0000,
12 ,,,,     24.0000,    6.0000,    0.0000,
13 ,,,,     36.0000,    6.0000,    0.0000,
14 ,,,,     48.0000,    6.0000,    0.0000,
15 ,,,,      0.0000,   12.0000,    0.0000,
16 ,,,,      6.0000,   12.0000,    0.0000,
17 ,,,,     12.0000,   12.0000,    0.0000,
18 ,,,,     18.0000,   12.0000,    0.0000,
19 ,,,,     24.0000,   12.0000,    0.0000,
20 ,,,,     30.0000,   12.0000,    0.0000,
21 ,,,,     36.0000,   12.0000,    0.0000,
22 ,,,,     42.0000,   12.0000,    0.0000,
23 ,,,,     48.0000,   12.0000,    0.0000,
24 ,,,,      0.0000.   18.0000,    0.0000,
25 ,,,,     12.0000,   18.0000,    0.0000,
26 ,,,,     24.0000,   18.0000,    0.0000,
27 ,,,,     36.0000,   18.0000,    0.0000,
28 ,,,,     48.0000.   18.0000,    0.0000,
29 ,,,,      0.0000,   24.0000,    0.0000,
30 ,,,,      6.0000,   24.0000,    0.0000,
31 ,,,,     12.0000,   24.0000,    0.0000,
32 ,,,,     18.0000,   24.0000,    0.0000,
33 ,,,,     24.0000,   24.0000,    0.0000,
34 ,,,,     30.0000,   24.0000,    0.0000,
35 ,,,,     36.0000,   24.0000,    0.0000,
36 ,,,,,    42.0000,   24.0000,    0.0000,
37 ,,,,     48.0000,   24.0000,    0.0000,-
38 ,,,,      0.0000,   30.0000,    0.0000,
39 ,,,,     12.0000,   30.0000,    0.0000,
40 ,,,,     24.0000,   30.0000,    0.0000,
41 ,,,,     36.0000,   30.0000,    0.0000,
42 ,,,,     48.0000,   30.0000,    0.0000,
43 ,,,,      0.0000,   36.0000,    0.0000,
44 ,,,,      6.0000,   36.0000,    0.0000,
45 ,,,,     12.0000,   36.0000,    0.0000,
46 ,,,,     18.0000,   36.0000,    0.0000,
47 ,,,,     24.0000,   36.0000,    0.0000,
48 ,,,,     30.0000,   36.0000,    0.0000,
49 ,,,,     36.0000,   36.0000,    0.0000,
50 ,,,,     42.0000,   36.0000,    0.0000,
51 ,,,,     48.0000,   36.0000,    0.0000,
52 ,,,,      0.0000,   42.0000,    0.0000,
53 ,,,,     12.0000,   42.0000,    0.0000,
54 ,,,,     24.0000,   42.0000,    0.0000,
55 ,,,,     36.0000, ▲ 42.0000,    0.0000,
56 ,,,,     48.0000,   42.0000,    0.0000,
57 ,,,,      0.0000,   48.0000,    0.0000,
58 ,,,,      6.0000,   48.0000,    0.0000,
59 ,,,,     12.0000,   48.0000,    0.0000,
60 ,,,,     18.0000,   48.0000,    0.0000,
61 ,,,,     24.0000,   48.0000,    0.0000,
62 ,,,,     30.0000,   48.0000,    0.0000,
63 ,,,,     36.0000,   48.0000,    0.0000,
64 ,,,,     42.0000,   48.0000,    0.0000,
65 ,,,,     48.0000,   48.0000,    0.0000,
```

```
*MATERIAL
 EX     ,       1 . ,   0.10500E+08
 NUXY  ,       1 . .   0.33300E+00
 DENS  ,       1 , ,   0.25879E-03
*I2
5,0,20,0,0.0,0.0,1.0E-5
*SPDISP
** SET ID =         1
        1,ROTY,     0.000000
        1,ROTX,     0.000000
        1,UZ  ,     0.000000
        1,UY  ,     0.000000
        1,UX  ,     0.000000
        2,ROTY,     0.000000
        2,UZ  ,     0.000000
        2,UY  ,     0.000000
        2,UX  .     0.000000
        3,ROTY,     0.000000
        3,UZ  ,     0.000000
        3,UY  ,     0.000000
        3,UX  ,     0.000000
        4,ROTY.     0.000000
        4,UZ  ,     0.000000
        4,UY  ,     0.000000
        4,UX  ,     0.000000
        5,ROTY,     0.000000
        5,UZ  ,     0.000000
        5,UY  ,     0.000000
        5,UX  ,     0.000000
        6,ROTY,     0.000000
        6,UZ  ,     0.000000
        6,UY  ,     0.000000
        6,UX  ,     0.000000
        7,ROTY,     0.000000
        7,UZ  ,     0.000000
        7,UY  ,     0.000000
        7,UX  ,     0.000000
        8,ROTY,     0.000000
        8,UZ  ,     0.000000
        8,UY  ,     0.000000
        8,UX  ,     0.000000
        9,ROTY,     0.000000
        9,ROTX,     0.000000
        9,UZ  ,     0.000000
        9,UY  ,     0.000000
        9,UX  ,     0.000000
       10,ROTX,     0.000000
       10,UZ  ,     0.000000
       10,UY  ,     0.000000
       10,UX  ,     0.000000
       11,UY  ,     0.000000
       11,UX  ,     0.000000
       12,UY  ,     0.000000
       12,UX  .     0.000000
       13,UY  ,     0.000000
       13,UX  ,     0.000000
       14,ROTX,     0.000000
       14,UZ  ,     0.000000
       14,UY  ,     0.000000
       14,UX  ,     0.000000
```

2.47

```
15,ROTX,      0.000000
15,UZ   ,     0.000000
15,UY   ,     0.000000
15,UX   ,     0.000000
16,UY   ,     0.000000
16,UX   ,     0.000000
17,UY   ,     0.000000
17,UX   ,     0.000000
18,UY   .     0.000000
18,UX   ,     0.000000
19,UY   ,     0.000000
19,UX   ,     0.000000
20,UY   ,     0.000000
20,UX   ,     0.000000
21,UY   ,     0.000000
21,UX   ,     0.000000
22,UY   ,     0.000000
22,UX   ,     0.000000
23,ROTX,      0.000000
23,UZ   ,     0.000000
23,UY   ,     0.000000
23,UX   ,     0.000000
24,ROTX,      0.000000
24,UZ   ,     0.000000
24,UY   ,     0.000000
24,UX   ,     0.000000
25,UY   ,     0.000000
25,UX   ,     0.000000
26,UY   ,     0.000000
26,UX   ,     0.000000
27,UY   ,     0.000000
27,UX   ,     0.000000
28,ROTX,      0.000000
28,UZ   ,     0.000000
28,UY   ,     0.000000
28,UX   ,     0.000000
29,ROTX,      0.000000
29,UZ   ,     0.000000
29,UY   ,     0.000000
29,UX   ,     0.000000
30,UY   ,     0.000000
30,UX   ,     0.000000
31,UY   ,     0.000000
31,UX   , .   0.000000
32,UY   ,     0.000000
32,UX   ,     0.000000
33,UY   ,     0.000000
33,UX   ,     0.000000
34,UY   ,     0.000000
34,UX   ,     0.000000
35,UY   ,     0.000000
```

```
35,UX    ,     0.000000
36,UY    ,     0.000000
36,UX    ,     0.000000
37,ROTX ,      0.000000
37,UZ    ,     0.000000
37,UY    ,     0.000000
37,UX    ,     0.000000
38,ROTX ,      0.000000
38,UZ    ,     0.000000
38,UY    ,     0.000000
38,UX    ,     0.000000
39,UY    ,     0.000000
39,UX    ,     0.000000
40,UY    ,     0.000000
40,UX    ,     0.000000
41,UY    ,     0.000000
41,UX    ,     0.000000
42,ROTX ,      0.000000
42,UZ    ,     0.000000
42,UY    ,     0.000000
42,UX    ,     0.000000
43,ROTX ,      0.000000
43,UZ    ,     0.000000
43,UY    ,     0.000000
43,UX    ,     0.000000
44,UY    ,     0.000000
44,UX    ,     0.000000
45,UY    ,     0.000000
45,UX    ,     0.000000
46,UY    ,     0.000000
46,UX    ,     0.000000
47,UY    ,     0.000000
47,UX    ,     0.000000
48,UY    ,     0.000000
48,UX    ,     0.000000
49,UY    ,     0.000000
49,UX    ,     0.000000
50,UY    ,     0.000000
50,UX    ,     0.000000
51,ROTX ,      0.000000
51,UZ    ,     0.000000
51,UY    ,     0.000000
51,UX    ,     0.000000
52,ROTX ,      0.000000
52,UZ    ,     0.000000
52,UY    ,     0.000000
52,UX    ,     0.000000
53,UY    ,     0.000000
53,UX    ,     0.000000
54,UY    ,     0.000000
54,UX    ,     0.000000
```

```
            55,UY   ,      0.000000
            55,UX   ,      0.000000
            56,ROTX,       0.000000
            56,UZ   ,      0.000000
            56,UY   ,      0.000000
            56,UX   ,      0.000000
            57,ROTY,       0.000000
            57,ROTX,       0.000000
            57,UZ   ,      0.000000
            57,UY   ,      0.000000
            57,UX   ,      0.000000
            58,ROTY,       0.000000
            58,UZ   ,      0.000000
            58,UY   .      0.000000
            58,UX   ,      0.000000
            59,ROTY,       0.000000
            59,UZ   ,      0.000000
            59,UY   ,      0.000000
            59,UX   ,      0.000000
            60,ROTY,       0.000000
            60,UZ   ,      0.000000
            60,UY   ,      0.000000
            60,UX   ,      0.000000
            61,ROTY,       0.000000
            61,UZ   ,      0.000000
            61,UY   ,      0.000000
            61,UX   ,      0.000000
            62,ROTY,       0.000000
            62,UZ   ,      0.000000
            62,UY   ,      0.000000
            62,UX   ,      0.000000
            63,ROTY,       0.000000
            63,UZ   ,      0.000000
            63,UY   ,      0.000000
            63,UX   ,      0.000000
            64,ROTY,       0.000000
            64,UZ   ,      0.000000
            64,UY   ,      0.000000
            64,UX   ,      0.000000
            65,ROTY,       0.000000
            65,ROTX,       0.000000
            65,UZ   ,      0.000000
            65,UY   ,      0.000000
            65,UX   ,      0.000000
*MODEO
1,1,1,1
*EIGO
1,0,3,1,0,1
*ENDDATA
```

2.50

COMMAND>GRD,ADD,1,0/0/0

This adds a grid (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

COMMAND>GRD,TRS,1,2,48

This makes grid #2 by "translating" grid #1 48 units in the X direction. Actually it translates grid #1 48/0/0 units where the format is X/Y/Z: since there are no "0"s typed in, they are automatically assigned. If it was desired that #2 be placed 48 units in the Y direction, the input would be either "GRD,TRS,1,2,0/48" or "GRD,TRS,1,2,/48". Default values are "0".

COMMAND>LIN,2GD,1,2,1

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #".

COMMAND>LIN,TRS,1,2,0/48

This creates line #2 by translating line #1 48 units in the Y direction. This command could be either be input as is or as "LIN,TRS,1,2,/48". The format is

"LIN,TRS,3,4,5A/5B/5C" where:

>   Entry 3 in the line to be translated.

>   Entry 4 is the line # to be created.

>   Entry 5 is the direction (X/Y/Z) entry 3 is to
>   be translated in. Default values in this entry
>   are "0"s.

COMMAND>PAT,2LN,1,2,1

PAT stands for patch, which is defined as a plate of
defined size in 2 D space or a surface of defined size in
3 D space.. This entry creates patch #1 between lines #1
and #2 with the following format: "PAT,2LN,LINE ID #1,
LINE ID #2, PATCH #". Refer to the example at the end of
verification problem break down for an example of how a
is created and what it looks like.

COMMAND>FEG,QUA,1,,,2/4/4/20

FEG stands for "Finite Element Generation". This
entry creates a group of 16 quadrilateral shell elements,
4 elements by 4 elements, in the plate created by the
above patch. This is accomplished with the following
format: "FEG,QUA,3,4,5,6A/6B/6C/6D,7A/7B/7C/7D where:

>   Entry 3 is the patch ID number that is to be
>   "FEGED".

>   Entry 4 is element ID list. This is the list of

the element numbers to be created. If left
blank the program will automatically number the
elements being created.

Entry 5 is the list of node numbers to be
created. Like the element entry above if this
entry is left blank the program will
automatically number the nodes.

Entry 6A is the order of the elements to be
created. In this case the elements are second
order shell elements, of NKTP type 20, and
contain 8 nodes per element. Refer to the
Element Section of the DISPLAY II User's Manual.

Entry 6B and 6C are the number of elements to be
generated along the parametric directions C1 and
C2. Parametric directions are internally
assigned by the program to two edges of a patch
and three edges of a hyperpatch and are used by
the program for defining mesh density of the
patch. For example suppose it is desired to
"break" a line 1/3 of the way from the end.
Where is the end and where is the start? The
program uses parametric directions to show where
the start (where the directions intersect) and
end is. Refer to pages 2.2-7 to 2.2-10 of the
DISPLAY II User's Manual.

Entry 6D is the NKTP #, which stands for the
elements type identification number. Each

element type has a different NKTP #. For an
explanation of exactly what a specific NKTP #
is, refer to the element library section of the
NISA II User's Manual, pages 4.0-1 to 4.120-5.
Entry 7A and 7B are the material and property ID
#'s respectfully. This set of #s, assigned to
one or more elements, is used later in the input
phase for assigning actual properties. If left
blank the program automatically assigns a
material ID # of 1 and a property ID # of 1. In
this case all the elements have the same
material characteristics and property
characteristics and therefore all have the same
MID # and the same PID #.
Entry 7C and 7D are the zoom values to be
assigned along the parametric directions
discussed in entry 6B and 6C above. If left
blank the program assigns zoom = 1 to both.

COMMAND>PROP,ADD,1,1/1/1/1/1/1/1/1

This adds a thickness of 1 unit to all eight
nodes of the elements with PID = 1 with the following
format "PROP,ADD,3,4A/4B/4C..." where:
Entry 3 is the property index # (PID).
Entry 4 is the list of properties which are
defined by the element type with each different

2.54

property separated by a /. Refer to the
specific element type in the element library
section of the NISA II User's Manual for more
info on what properties are listed here.

COMMAND>MAT,ADD,1,10.5E6/0.3333////////2.5879E-4

    This adds the material properties of modulus of
elasticity (EX) = 10.5E6 psi, Poisson's ratio (NUXY) =
0.333 and mass density = 2.5879E-4 lbs.-sec squared per
inches to the fourth power to all the elements with
material ID #1. This is accomplished with the following
format: MAT,ADD,3,4A/4B/4C..." where:

        Entry 3 is the material ID #.

        Entry 4 is the list of material properties.
        Like the property characteristics described
        above, this list will vary for each element
        type with each separate property being separated
        by a slash "/".

COMMAND>DISP,ADD,1/9/57/65,1,0/0/0/0/0

    This constrains nodes 1,9,57, and 65 in the X, Y, Z,
Rotation in X and Rotation in Y, and assigns a constraint
ID # of 1 with the following format:
"DISP,ADD,3A/3B/3C...,4,5A-5F", where:

        Entry 3 is the list of nodes to be constrained.

Entry 4 is the constraint ID #. The set ID
under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that
are constrained. This is accomplished by
placing a "0" in the degree of freedom that must
be constrained with the following format:
"X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or
constrained directions not listed, indicate a
null entry which the program either ignores or
places a default value in that place. In this
case the program ignores the null entries. For
more information on null entries refer to the
general command rules previously addressed in
this chapter of to pages 2.1-3 to 2.1-6 of the
DISPLAY II User's Manual.

COMMAND>DEC

    This places the program in the mode to accept
Executive Commands. For more info on executive commands
refer to the previous Executive command overview and page
7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

    This places the program in the mode to allow the
insertion of data lines.

COMMAND>ANAL=EIGEN

This tells the program that the analysis to be
performed is an EIGENVALUE analysis.

COMMAND>FILE=E01

This provides the file name prefix E01 for the
internally generated binary data files.

COMMAND>SAVE=26,27

This tells the program to save the internally
generated binary data files 26 and 27. The names of these
files will be E0126.dat and E0127.dat respectively. Note
this command must be used to save the data files 26 and
27, which are required for post-processing or dynamic
analysis.

COMMAND>RESE=ON

This command tells the program to look at the
numbering of the elements and resequence them (ie renumber
them internally) to make the finite element analysis more
efficient. The input of this command is not required as
the program will automatically perform this function.
When creating a finite element model there is definite
"pecking order" of how elements and nodes should be

numbered. The analysis of a model that does not follow

this order will take longer than one that does. This

command has the program check the model to see if it

conforms to the "pecking order". If the model does not

then the program will internally change the element

numbering so that it does. The change will not effect the

user as it is transparent to the user and the element IDs

that the user uses are not changed. For more info refer

page 5.3-5 of the NISA II User's Manual

COMMAND>AUTO=ON

This command activates the automatic spurious normal

rotation constraint for shells. This input is not

specifically required as the default mode is for this to

be on. Nodes in basic shell theory have 5 degrees of

freedom. When transferring a model from the non-global

system to a global system rotation in the theta Z

direction may occur. Additionally rotation perpendicular

to the theta Z direction may occur. If this happens then

the shell (ie model) will not be defined and therefore

cannot be solved. ▴Having the AUTO CONSTRAINT on will

constrain this perpendicular rotation, thereby allowing

the model to be defined.

COMMAND>EIGE=SUBS,CONV

This command tells the program that the eigenvalue
extraction method is to be conventional subspace.   This
command produces some effect on the speed of the analysis.
If the eigenvalues are far apart then having the method be
accelerated subspace will cause the program to
aggressively shift the frequencies in order to speed up
the convergence onto the natural frequencies.   Refer to
pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II User's
Manual.

COMMAND>MASS=LUMPED

This command tells the program to activate the lumped
matrix formulation, ie it tells the program to solve the
stiffness matrix diagonally instead of consistent with the
way the matrix was developed.   Refer to page 3.5-5 of the
NISA II User's Manual.

COMMAND>*LDCASE,1

This is the load case "control card" for the desired
output.   Refer to page 7.1-2 of the NISA II User's Manual
for an explanation of what the number(s) following the
card heading mean.   Note when performing an Eigenvalue
analysis this card is not applicable.   However, using this
card was the only method the author found of "saving" the

displacement constraints that were input previously. A
more difficult method of inputing the displacement

constraints would be to input them directly into the NIS

file using a text or line editor, like EDLIIN.   Therefore

the user should either use this command to save the

displacement constraints and then delete the two specific

load case lines in the NIS file by using a text or line

editor, like EDLIN, or input the displacement constraints

directly into the NIS file. (The NIS file will be

explained below.)   The load case lines that need to be

deleted consists of one line containing *LDCASE and a

second line containing the string of numbers input

immediately after the word *LDCASE above.

COMMAND>*EIGCNTL,5,0,20,0,0.0,0.0,1.0E-5

     This command tells the program what the control

parameters are for the eigenvalue extraction.   Refer to

pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an

explanation of what each number means.

COMMAND>*MODEO,1,1,1,1

     This command tells the program what modes the

analysis is required on.   The format is "MODEO,2,3,4,5"

where:

          Entry 2 is the first mode number to be

analyzed.

Entry 3 is the last mode number to be analyzed.

Entry 4 is the increment between the first and
last modes that are to be analyzed.

Entry 5 is the output ID # that refers to the ID
listed in the output command *EIGO.

For more info refer to page 7.1-15 of the NISA II User's
Manual.

COMMAND>*EIGO,1,1,0,3,1,0,1

This command tells the program what output options
to provide.   For an explanation of what each number means
refer to pages 7.5-2 to 7.5-4 of the NISA II User's
Manual.

Note: there seems to be a problem with the MODEO and EIGO
commands being accepted by the DISPLAY-PRE program and
then saved in the NIS. file.   As a result the above
commands must be inputed directly into the ".NIS" file
using either a text or line editor.   This is accomplished
with the following format:

line x: *MODEO

line x+1: modeo data

line x+2: *EIGO

line x+3: eigo data

where line x+4 is the ENDDATA line

The EIGO line must be the first output line listed. If
any other output is requested (Example REGIONS, the
regions for averaged nodal stresses printout; STRSFIL, the
nodal stress filtering data; etc.; it must be listed after
the EIGO data line. Refer to Section 7.5 OUTPUT CONTROL
DATA of the NISA II User's Manual.

COMMAND>QUI

     Quits the insert mode.

COMMAND>QUI

     Quits the DEC mode.

COMMAND>WRI,NISA,E01.NIS

     This takes the above data just input during this
session and writes a "NISA" file called "E01.NIS" that the
program uses for it's analysis.

COMMAND>WRI,DBS,E01.BIN

     This takes the above data and writes a binary data
base file called "E01.BIN"

```
COMMAND>END
```

This ends the session.

Data Base File: At this time the program will ask if a
data base file DISP.BIN is to be written. Since this has
already been done with the WRI,DBS,E01.BIN command above,
answer no "N".

The model is now ready to be analyzed. Refer to the
previous section "Program Usage" for guidance on how to
perform the analysis.

COMMAND INPUT DEMONSTRATION

In an attempt to assist the unfamiliar user with
understanding what the various DISPLAY-PRE commands
actually do, specifically the creation of a patch, the
demonstration below provides a pictorial description of
some of the various commands required in the previous
verification problem. This demonstration does not contain
all required commands only a few to provide an idea of how
the commands work.

GRD,ADD,1,0/0/0   This creates a point at coordinates
(0,0,0).

            (o,o,o)
              □
              I

GRD,TRS,1,2,48   This creates point (grid) #2 by
translating grid #1 48 units in the X direction.

            (o,o,o)   (vs,o,o)
              □   □
              I   2

LIN,2GD,1,2,1   This creates line #1 between grids #1
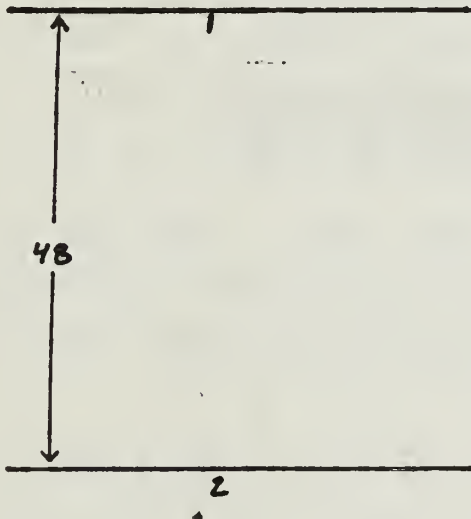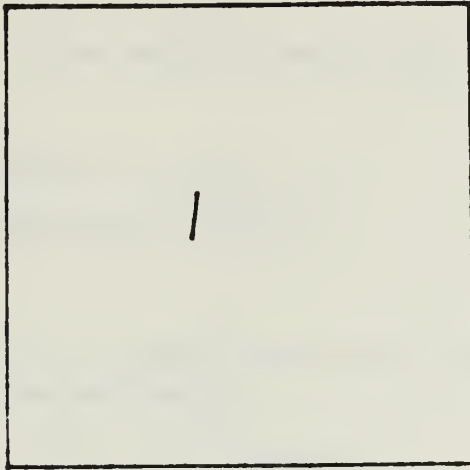and #2.

I

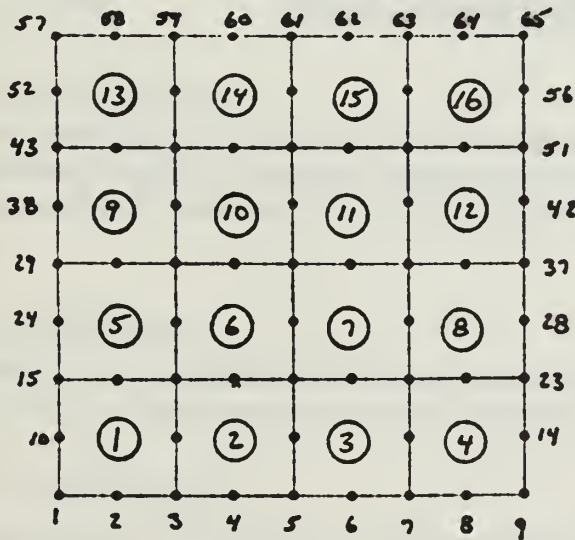LIN,TRS,1,2,0/48   This creates line #2 by
translating line #1 48 units in the Y direction.

48

2

PAT,2LN,1,2,1   This creates patch #1 between lines #1
and #2.

FEG,QUA,1,,,2/4/4/20   This creates a group of 16
quadrilateral shell elements of NORDR 2, 4 elements by 4
elements, in the patch created above. Due to space
limitation not all the nodes are numbered.

VERIFICATION PROBLEM 6.3

TITLE:
Transient response of a beam subjected to distributed load

ELEMENT TYPE:
2-D beam element (NKTP = 13, NORDR = 1)

2-D translational spring element (NKTP = 18)

PROBLEM:
A beam supported by two springs of stiffness $K/2 = \pi^4 EI/2L^3$ at its ends, as shown in Fig. 6.3.1, is subjected to a distributed load given by:

$$p(x) = 1-2 \, |x|/L \qquad \text{for } -L/2 \leq x \leq L/2$$

The variation of the load with time is shown in Fig. 6.3.2. Lateral motion of the beam is constrained. The natural frequencies and time history response of the mid-point of the beam are to be computed.

PROPERTIES:
Material:

EX	=	3.0×10^7 psi	(Modulus of elasticity)
NUXY	=	0.3	(Poisson's ratio)
DENS	=	0.000728 lbs.-sec^2/in^4	(Mass density)

Cross-Section:

A	=	4.0 in^2	(Cross-sectional area)
IZZ	=	1.3333 in^4	(Moment of inertia about beam Z-axis)

FINITE ELEMENT MODEL:
The finte element model is shown in Fig. 6.3.3. Since the applied load and the system are symmetric, only one half of the beam is modeled. Symmetry boundary conditions (UX = 0.0, ROTZ = 0.0) are applied at node 1. Lateral motion is constrained (UX = 0.0) at nodes 2 and 3, and node 4 is fixed (UX = 0.0, UY = 0.0).

SOLUTION PROCEDURE:
Consistent mass matrix formulation and subspace iteration technique is used in evaluating the natural frequencies. First two symmetric modes are used for the transient dynamic analysis.

RESULTS & COMPARISON:
The natural frequencies of the first two symmetric modes are compared to the theoretical values in Table 6.3.1. The peak transient response of the mid-point of the beam is compared to the theoretical value in Table 6.3.2. The displacement history of the mid-point of the beam is shown in Fig. 6.3.4.
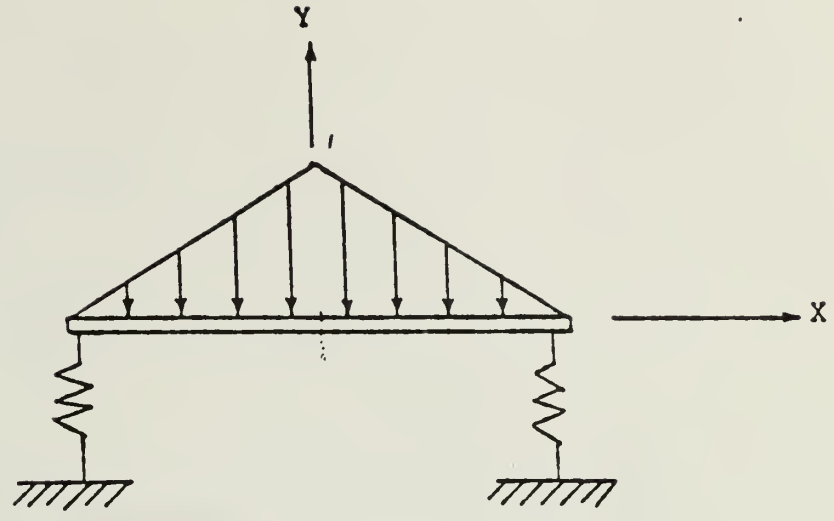
Figure 6.3.1   Beam with Distributed Load
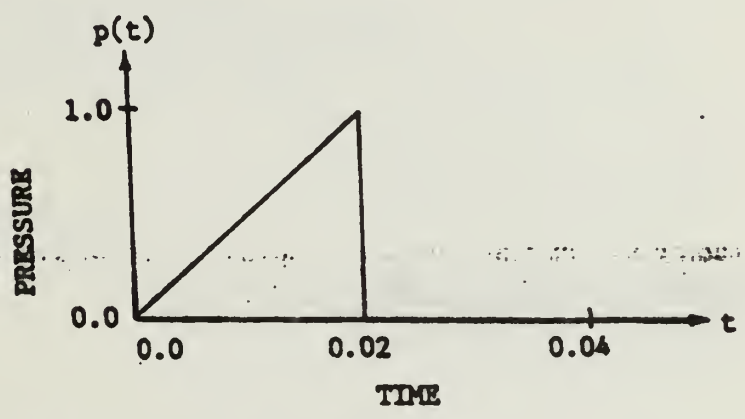
Figure 6.3.2   Load-Time History
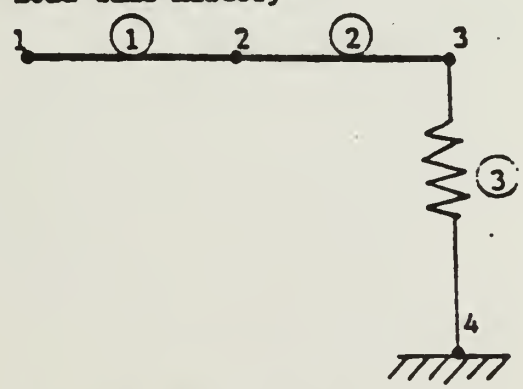
Figure 6.3.3   Finite Element Model

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,40
LIN,2GD,1,2,1
FEG,BAR,1,,,2/13,1/1
NOD,ADD,4,40/-4
ELE,ADD,3,18/1//2,3/4
DISP,ADD,1,1,0//////0
DISP,ADD,2T3,1,0
DISP,ADD,4,1,0/0
PROP,ADD,1,4./1.3333
MAT,ADD,1,3.0E7/0.3/////////7.28E-4
PROP,ADD,2,3.805E3
DEC
INS
ANAL=EIGENVALUE
AUTOCONSTRAINT=OFF
RESEQUENCING OF ELEMENTS=ON
FILE=ETD01
SAVE=26,27
EIGEN EXTRACTION=SUBSPACE
MASS MATRIX FORMULATION=CONSISTENT
*TITLE,ATTEMPT OF VERIFICATION PROBLEM ET03.STD
*I2,5,-1,20,0,0.0,0.0,1.0E-5
*I3,1
QUI
QUI
WRI,NISA,ETD01.NIS
WRI,DBS,ETD01.BIN
END
N
```

```
** EXECUTIVE
ANAL=EIGENVALUE
AUTOCONSTRAINT=OFF
RESEQUENCING OF ELEMENTS=ON
FILE=ETD01
SAVE=26,27
EIGEN EXTRACTION=SUBSPACE
MASS MATRIX FORMULATION=CONSISTENT
*TITLE
ATTEMPT OF VERIFICATION PROBLEM ET03.STD
*ELTYPE
   1,   13,        1
   2,   18,        1
*RCTABLE
        1,        2
   4.0000,    1.3333,
        2,        1
3805.0000,
*ELEMENT
        1,        1,        1,        1, 0
        1,        2,
        2,        1,        1,        1, 0
        2,        3,
        3,        0,        2,        2, 0
        3,        4,
*NODES
        1 ,,,,       0.0000,       0.0000,       0.0000,
        2 ,,,,      20.0000,       0.0000,       0.0000,
        3 ,,,,      40.0000,       0.0000,       0.0000,
        4 ,,,,      40.0000,      -4.0000,       0.0000,
*MATERIAL
  EX    ,        1 , ,   0.30000E+08
  NUXY ,        1 , ,   0.30000E+00
  DENS ,        1 , ,   0.72800E-03
*I2
5,-1,20,0,0.0,0.0,1.0E-5
*SPDISP
** SET ID =          1
        1,ROTZ,      0.000000
        1,UX   ,     0.000000
        2,UX   ,     0.000000
        3,UX   ,     0.000000
        4,UY   ,     0.000000
        4,UX   ,     0.000000
*ENDDATA
```

COMMAND>GRD,ADD,1,0/0/0

This adds a grid (a point in 3-D space), calls it grid #1, and puts it at coordinates (0,0,0).

COMMAND>GRD,TRS,1,2,40

This makes grid #2 by "translating" grid #1 40 units in the X direction. Actually it translates grid #1 40/0/0 units where the format is X/Y/Z: since there are no "0"s typed in, they are automatically assigned. If it was desired that #2 be placed 40 units in the Y direction, the input would be either "GRD,TRS,1,2,0/40" or "GRD,TRS,1,2,/40". Default values are "0".

COMMAND>LIN,2GD,1,2,1

This creates a line starting at grid #1 and ending at grid #2, calling it line #1. The format is "LIN,2GD,starting grid #,ending grid #".

COMMAND>FEG,BAR,1,,,2/13,1/1

FEG stands for "Finite Element Generation". This entry creates a group of 2 bar/beam elements (of element type 13), in the global coordinate system, assigning a

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED".

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II User's Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system, for example a

force or displacement constraint may be defined
with directions which are not aligned with the
global coordinate system. For more information
on local coordinate systems refer to pages 3.2-1
to 3.2-4 of the NISA II User's Manual.

Entry 7A & 7B are the material and property ID
#'s respectfully. This set of #s, assigned to
one or more elements, is used later in the input
phase for assigning actual properties. If left
blank the program automatically assigns a
material ID # of 1 and a property ID # of 1.
These numbers do not have to be the same. A
group of elements can have the same material
properties (Poisson's ratio, Young's modulus,
etc.) with some of the elements having one
geometric property and others having a different
geometric property. As a result this group of
elements would have one material ID # and two
property ID #s.

Entry 7C is the zoom factor, ie the length of
the last element to length of the first element.
If all elements are the same length then the
zoom = 1. If left blank the program
automatically assigns a value of 1. For more
information and a pictorial representation refer
to page B-3 of the DISPLAY II User's Manual.

COMMAND>NOD,ADD,4,40/-4

This command adds a node at coordinates (40,-4,0) and
calls it node # 4.   This is accomplished with the
following format: "NOD,ADD,3,4A/4B/4C/4D" where:

Entry 3 is the node # to be generated.

Entry 4A, 4B, & 4C are the (X/Y/Z) coordinates
of the node.

Entry 4C is the local coordinate system in which
the coordinates are to be specified. If this is
left blank then the node is in the global
system.

COMMAND>ELE,ADD,3,18/1//2,3/4

This adds an element #3 with NKTP = 18, order = 1,
property ID # = 2 between nodes 3 and 4, using the
following format: "ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this
is left blank the program will automatically
assign a number.

Entry 4A is the NKTP # (see entry 6B under the
FEG description above).

Entry 4B is the order #. Some elements only
have one order, NORDR=1, others have many
depending on the element type. For a pictorial
description of how this order works/varies refer
to section 4, the "element library" of the NISA

2.74

II User's Manual; specifically NKTP #s

1,2,3,4,5,& 32 provide good examples of some of

the element types with varying orders.

Entry 4C is the material ID #. This # is used

later in the input phase to assign actual

material properties to this element.

Entry 4D is the property ID #. (See entry 7B

under the FEG description above.)

Entry 5A & 5B are the two nodes that define the

element. Most elements require two nodes, one

at each end. However since this particular

element is a mass concentration it only requires

one node.

Entry 6 is the vector number of a previously

defined vector used in establishing a local

coordinate system. This can be used for

elements of NKTP = 11, 12, & 38.

COMMAND>DISP,ADD,1,1,0/////0

This constrains node #1 in the X and ROTZ directions

and assigns a constraint ID # of 1 with the following

format: "DISP,ADD,3A/3B/3C...,4,5A-5F", where:

Entry 3 is the list of nodes to be constrained.

Entry 4 is the constraint ID #. The set ID

under which the constraints are specified.

Entry 5A to 5F are the degree(s) of freedom that

are constrained. This is accomplished by
placing a "0" in the degree of freedom that must
be constrained with the following format:
"X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or
constrained directions not listed, indicate a
null entry which the program either ignores or
places a default value in that place. In this
case the program ignores the null entries. For
more information on null entries refer to the
general command rules previously addressed in
this chapter of to pages 2.1-3 to 2.1-6 of the
DISPLAY II User's Manual.

COMMAND>PROP,ADD,1,4./1.3333

    This adds a cross sectional area of 4.0 inches
squared and a moment of inertia on 1.3333 inches to the
forth to all the elements with property ID # 1. This is
accomplished with the following format
"PROP,ADD,3,4A/4B/4C..." where:

        Entry 3 is the property index # (PID).
        Entry 4 is the list of properties which are
        defined by the element type with each different
        property separated by a /. Refer to the
        specific element type in the element library
        section of the NISA II User's Manual for more
        info on what properties are listed here.

COMMAND>MAT,ADD,1,3.0E7/0.3////////7.28E-4

This adds the material properties of Modulus of
elasticity (EX) = 3.0E7 psi, Poisson's ratio (NUXY) = 0.3,
and mass density (DENS) = 7.28E-4 lbs.-sec squared per
inches to the forth. This is accomplished with the
following format:" MAT,ADD,3,4A/4B/4C/..." where:

Entry 3 is the material ID st #.

Entry 4 is the list of material properties.
Like the property characteristics described
above, this list will vary for each element
type.

COMMAND>DEC

This places the program in the mode to accept
Executive Commands. For more info on executive commands
refer to the previous Executive command overview and page
7.10-1 of the DISPLAY II User's Manual.

COMMAND>INS

This places the program in the mode to allow the
insertion of data lines.

COMMAND>ANAL=EIGENVALUE

This tells the program that the analysis to be
performed is an EIGENVALUE analysis.

COMMAND>AUTOCONSTRAINT=OFF

This command deactivates the automatic spurious
normal rotation constraint. Since these elements are
bar/beam elements this command, which basically applies to
shell elements, is not really applicable. However, the
following explanation is provided for what this command
does if it is activated. Nodes in basic shell theory have
5 degrees of freedom. When transferring a model from the
non-global system to a global system rotation in the theta
Z direction may occur. Additionally rotation
perpendicular to the theta Z direction may occur. If this
happens then the shell (ie model) will not be defined and
therefore cannot be solved. Having the AUTO CONSTRAINT on
will constrain this perpendicular rotation, thereby
allowing the model to be defined.

COMMAND>RESEQUENCING OF ELEMENTS=ON

This command tells the program to look at the
numbering of the elements and to resequence them (ie
renumber them internally) to make the finite element
analysis more efficient. This command is not really

required as the he program is set up to automatically
perform this function. When creating a large finite
element model this process (ie RESE=ON) is usually
necessary because there is a definite "pecking order" of
how elements and nodes should be numbered. The analysis
of a model that does not follow this order will take
longer than one that does. This command has the program
check the model to see if it conforms to the "pecking
order". If the model does not then the program will
internally change the element numbering so that it does.
The change will not effect the user as it is transparent
to the user and the element IDs that the user uses are not
changed. For more info refer page 5.3-5 of the NISA II
User's Manual.

COMMAND>FILE=ETD01

     This provides the file name prefix ETD01 for the
internally generated binary data files.

COMMAND>SAVE=26,27

     This tells the program to save the internally
generated binary data files 26 and 27. The names of these
files will be ETD0126.dat and ETD0127.dat respectively.
Note this command must be used to save the data files 26
and 27, which are required for post-processing or dynamic

analysis.

COMMAND>EIGEN EXTRACTION=SUBSPACE

This command tells the program that the eigenvalue
extraction method is to be accelerated subspace. The
accelerated selection of the subspace type is the default
setting and therefore the accelerated did not have to be
specified. This command produces some effect on the speed
of the analysis. If the eigenvalues are far apart then
having the method be accelerated subspace will cause the
program to aggressively shift the frequencies in order to
speed up the convergence onto the natural frequencies.
Refer to pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II
User's Manual.

COMMAND>MASS MATRIX FORMULATION=CONSISTENT

This command tells the program to activate the
consistent matrix formulation, ie it tells the program to
solve the stiffness matrix consistent with the way the
matrix was developed. Using a consistent mass matrix
formulation usually gives more accurate results than the
other method available, which is lumped. Refer to page
3.5-5 of the NISA II User's Manual.

COMMAND>*TITLE, ATTEMPT OF VERIFICATION PROBLEM ET03.STD

This command tells the program that the title of this
analysis is "ATTEMPT OF VERIFICATION PROBLEM ET03.STD",
which will be printed at the top of the answer output.

COMMAND>*I2,5,-1,20,0,0.0,0.0,1.0E-5

This command tells the program what the control
parameters are for the eigenvalue extraction. Refer to
pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an
explanation of what each number means.

COMMAND>*I3,1

*I3 is the same as *LDCASE which is the load case
"control card" for the desired output. Refer to page 7.1-
2 of the NISA II User's Manual for an explanation of what
the number(s) following the card heading mean. <u>Note when</u>
<u>performing an Eigenvalue analysis this card is not</u>
<u>applicable.</u> <u>However, using this card was the only method</u>
<u>the author found of "saving" the displacement constraints</u>
<u>that were input previously.</u> A more difficult method of
inputing the displacement constraints would be to input
them directly into the NIS file using a text or line
editor, like EDLIIN. Therefore the user should either use
this command to save the displacement constraints and then
delete the two specific load case lines in the NIS file by

2.81

using a text or line editor, like EDLIN, or input the displacement constraints directly into the NIS file. (The NIS file will be explained below.) The load case lines that need to be deleted consists of one line containing *LDCASE and a second line containing the string of numbers input immediately after the word *LDCASE above.

COMMAND>QUI

Quits the insert mode.

COMMAND>QUI

Quits the DEC mode.

COMMAND>WRI,NISA,ETD01.NIS

This takes the above data just input during this session and writes a "NISA" file called "ETD01.NIS" that the program uses for it's analysis.

COMMAND>WRI,DBS,ETD01.BIN

This takes the above data and writes a binary data base file called "ETD01.BIN"

COMMAND>END

This ends the session.

Data Base File: At this time the program will ask if a
data base file DISP.BIN is to be written. Since this has
already been done with the WRI,DBS,ETD01.BIN command
above, answer no "N".

The model is now ready to have an eigenvalue analysis
performed on it. Once this analysis has been completed
a dynamic analysis can be preformed. Remember that files
26.dat and 27.dat must be saved during the eigenvalue
analysis in order to be used to complete the dynamic
analysis. Refer to the previous section "Program Usage"
for guidance on how to perform the analysis.

```
ANAL=TRANSIENT
DAMP=VISCOUS
FILE=ETD01
STARTING TIME=0.0
DELTA TIME=0.0001
ENDTIME=0.08
*TITLE
ATTEMPT OF VERIFICATION PROBLEM TD03.STD
*MODESECTION
1,2,1,1
*DAMPING
1,0.0
*TIMEFUNCTION
LINEARLY INCREASING LOAD
1001,3,0,1.0
0.0,0.0,0.02,1.0,0.0201,0.0
*ARRIVALTIME
101,0.0
*DPRESSURE
1,1001,101,1,1
1.0,0.5
2,1001,101,1,1
0.5,0.0
*RSET
1001,1,UY
*HISTORY
0.0,0.08,1,1,1001
*ENDDATA
```

ANAL=TRANSIENT

This tells the program that the analysis to be performed is a transient dynamic analysis.

DAMP=VISCOUS

This tells the program that the model has viscous damping in it.

FILE=ETD01

This tells program that the prefix ETD01 is to be used for the internally generated binary data file. This prefix must be the same as that used for the eigenvalue portion of the problem. The reason for this is because the dynamic analysis must use the 26.dat and 27.dat files generated by the eigenvalue analysis, for the its' analysis. This command provides the prefix name to look for the required data.

STARTING TIME=0.0

This tells the program that the starting time of the forcing function is 0.0.

DELTA TIME=0.0001

This tells the program to analyze the response of the system from the starting time to the ending time in 0.0001 time increments. This value must be less than the difference between the starting time and the ending time or the program will not provide a response.

ENDTIME=0.08

This tells the program that the ending time of the forcing function is 0.08.

From this point on all commands will consists of more than one line of data. As a result a "*" indicates the start of a new set of data.

*TITLE

This is a two line data group. This line tells the program that the following line of input will be the analysis title.

ATTEMPT OF VERIFICATION PROBLEM TD03.STD

This is the title provided to the analysis which the program will then print at the top of the answer output.

*MODESECTION

This is a multi line data group. This line tells the
program the following line(s) of input will contain the
data specifying what modes are to be analyzed. This
command is always required, refer to page 8.4-2 of the
NISA II User's Manual.

1,2,1,1

This tells the program that modes 1 and 2, with an
increment of 1, are to be analyzed with damping ID #1's
damping value. This is accomplished with the following
format: Entry 1,2,3,4 where:

Entry 1 is the first mode to be analyzed.

Entry 2 is the last mode to be analyzed.

Entry 3 is the increment between modes to be

analyzed. Suppose the modes to be analyzed were

modes 1, 5 and 10 then entry 1 would be 1, entry

2 would be 10 and entry 3 would be 2.

Entry 4 is the damping ID # that corresponds to

the damping parameters set previously. The

damping ID # allows a system to be analyzed for

various damping values without remodeling the

whole system each time. Suppose it was desired

to analyze this problem with two different

damping values, say 30% and 70% of critical. To

do this, two damping data groups would be
required. Part of the damping data input is an
arbitrary ID #. Let's let the 30% damping have
an ID # of 1030 and the 70% have an ID # of 170
(note these numbers are totally arbitrary). Now
to find the response of this system with those
two values input:

*MODESECTION

1,2,1,1030

1,2,1,170

*TIMEFUNCTION

This is a multi line data group. This line tells the
program that the following lines of input will provide the
data used to specify the time function. This command is
always required for transient dynamic analysis. Refer to
page 8.5-3 of the NISA II User's Manual.

LINEARLY INCREASING LOAD

This is the title used to describe the time function.

1001,3,0,1.0

This tells the program that three points will be used
to define the forcing function in terms of abscissa and
ordinates, has a scale of 1.0, and a time function ID # of
1001. This is accomplished with the following format:
Entry 1,2,3,$,4,5,6 where:

> Entry 1 is the time function ID #. This is an
> arbitrary number that matches the number used in
> the *GROUND, *DPRESSURE and/or *DCFORCE data
> lines discussed later.
>
> Entry 2 is the number of points used to define
> the time function.
>
> Entry 3 provides the definition method. 0= the
> time function is defined in terms of abscissa
> and ordinates. 1= the time function is defined
> at equal intervals.
>
> $ tells the program to skip all entries between
> the last one inputed and entry 4.
>
> Entry 4 is the scale factor used for the time
> function. Basically there are two numbers that
> can be input here. The correct number depends
> on the definition of the time function. The
> easiest way to explain this is with an example.
> Therefore, let the time function = 200 sine 400
> theta. Now if the amplitude was based on 200
> sine 400 theta (ie theta = 0.5236 radians and

2.89

the force = 173.2) then the scale factor is 1.
If the amplitude was based on sine 400 theta (ie
theta = 0.5236 radians and the force = 0.866)
then the scale factor = 200. However any number
can be used as the scale factor as long as the
user maintains consistency through out the
modal.

Entry 5 is the starting time for the load
function.

Entry 6 is the time increment used for the time
function. If entry 3 is "0" this entry is not
used.

0.0,0.0,0.02,1.0,0.0201,0.0

    This defines the time function by using a maximum of
three pairs of numbers per line, with the total number of
pairs equal to the number listed in entry 2 above. The
pairs of numbers consists of the first number referring to
the time at point x and the second number of the pair is
the function value at point x.

*ARRIVALTIME

    This is two line data group that tells the program
that the following line of input will specify arrival
times used in ground motion, concentrated forces and/or

pressure data.

101,0.0

This tells the program that arrival time ID # 101 has
an arrival time of 0.0. This is accomplished with entry 1
being an arbitrary number (referenced in the *GROUND,
*DPRESSURE and/or *DCFORCE data lines) assigned as the ID
and entry 2 being the arrival time.

*DPRESSURE

This is a multi line data group. This line tells the
program that the following lines of input will contain
data specifying the pressure loading of elements.

1,1001,101,1,1

This tells the program that there is one additional
line of input required to complete the pressure loading
data, that the pressure is acting on face 1 of element 1,
and the time function ID # is 1001 and the arrival time ID
is 101. This is accomplished with the following format:
Entry 1,2,3,4,5,6,7,$,8 where:

        Entry 1 is the element that the pressure is
        acting on.
        Entry 2 is the time function ID #.

Entry 3 is the arrival time ID #. Note: The
parameters that entries 2 and 3 refer to depend
on the type of dynamic analysis being performed.
For example if a frequency response analysis is
to be performed entry 2 refers to the amplitude
spectrum ID # and entry 3 refers to the phase
spectrum ID #. Refer to pages 8.6-2 and 8.6-8
of the NISA II User's Manual.

Entry 4 is the face number that the pressure
acts on.

Entry 5 is the number of additional lines
required to complete the pressure loading data.

Entry 6 is the last element number that the
pressure is acting on.

Entry 7 is the increment between entry 1 and
entry 6 elements the pressure is acting on.

$ tells the program to skip all entries between
the last one inputed and entry 8.

Entry 8 is the uniform pressure load intensity.
This entry is used only if entry 5 = 0.

1.0,0.5

 This tells the program that the pressure on the
element referenced above (element #1) varies from a
pressure of 1.0 on the left node to a pressure of 0.5 on
the right node of the element. Refer to page 8.6-9 of the

*RSET

This is a multi line data group. This line tells the
program that the following line(s) of input will define
the nodal/element response quantities that are to be
provided by the program. This command is required. Refer
to page 8.7-2 of the NISA II User's Manual.

1001,1,UY

This tells the program that the response at node 1,
in the Y direction, is to be provided with a response ID #
of 1001. This is accomplished with the following format:
Entry 1,2,3,4,5,$,6 where:

Entry 1 is an arbitrarily assigned response ID
#.

Entry 2 is the node or element # to be analyzed.

Entry 3 is the response to be specifically
provided.

Entry 4 is the last node or element # to be
analyzed. If the response at only one node or
element is required then this entry must be the
same as entry 2. If not the program will state
that there is an error in this data line and not
provide a response.

Entry 5 is the node or element increment. If
the response at only one node or element is
required then with entry 4 equaling entry 2 this
entry must be one "1". If not the program will
state that there is an error in this data line
and not provide a response.

$ This tells the program to skip all entries
between the last one inputed and entry 6.

Entry 6 is the layer number for composite
elements.

Note: Additional input lines can be used to specify
responses for different nodes/elements and/or different
responses. For example suppose the responses at node 1 in
both the X and Y directions was desired. Then there would
be two lines of input, one with entry 3 containing UX and
one with entry 3 containing UY.

*HISTORY

This is a two line data group. This line tells the
program that the following line of input will request the
response history computation for printout and/or post
processing. The NISA II User's Manual states that this
command group is optional. However the author determined,
by running various analyses, that this command is not
optional. If this command and/or the *RSET command is not
used then the program will have a "fatal" error and will

provide the following response "FATAL ERROR", "NO RESPONSE
OUTPUT HAS BEEN REQUESTED".

0.0,0.08,1,1,1001

This tells the program that the time history, maximum
values and time of occurrence of maximum values are to be
printed starting at time 0.0, ending at time is 0.08, with
increments of 1, and that the response ID # is 1001. This
is accomplished with the following format:
Entry 1,2,3,4,$,5-14 where:

Entry 1 is the starting time. This entry must
equal the value placed in the "STARTING TIME"
Executive command.

Entry 2 is the ending time. This entry must
equal the value placed in the "ENDTIME"
Executive command.

Entry 3 is the history calculation increment.
This increment must be smaller than the
difference in time between the start time and
the end time. If it is not the program will
not provide a response.

Entry 4 is the printout option. 0 - response
history calculated but not printed. 1 - time
history, maximum values and time of occurrence
of maximum values are printed. 2 - only maximum
values and times of occurrence are printed.

2.95

$\underline{\$}$ tells the program to skip all entries
between the last one inputed and entry 5.
$\underline{\text{Entry 5 to 14}}$ are the response set ID #s that
the time history response is required. These
numbers should be referenced in the *RSET input.

*ENDDATA

This tells the program that this is the last line of
data.

Once this file is completed and the eigenvalue analysis
has been performed the model is ready to have the dynamic
analysis performed. To perform the dynamic analysis
access the directory and sub-directory that the above file
is saved in, type "NISA286" and hit the return. Now move
the cursor, using the arrow keys, to the "DYNAMIC"
selection and hit the return. At this time the program
will ask for the input file name. Type in the name of the
file above and hit the return. Now the program will ask
for the output file name. Type in the desired output name
and hit return. Now wait for the analysis.

VERIFICATION PROBLEM 8.2

TITLE:

Frequency response analysis of a 2-DOF system

ELEMENT TYPE:

2-D mass element (NKTP = 25)
2-D translational spring (NKTP = 18)

PROBLEM:

A 2DOF system (Fig. 8.2.1) is subjected to a harmonic force acting at node 2. Damping is neglected.

The response of the system is to be computed for an exciting frequency range of 0.0 to 7.5 HZ at intervals of 0.25 HZ.

PROPERTIES:

Material:
Translational mass   = 200 lb. sec.^{2}/in.
Translational spring constant   = 0.5 lb./in.

FINITE ELEMENT MODEL:

The finite element model consists of 4 active nodes and 5 elements (2 mass elements and 3 spring elements). Zero displacements are prescribed at the end points.

SOLUTION PROCEDURE:

An eigenvalue analysis using lumped mass formulation and conventional subspace iteration technique is carried out first, from which 2 eigenvalues and eigenvectors are obtained. The frequency response run is performed and displacement response (amplitude and phase lag) is computed.

RESULTS & COMPARISON:

The natural frequencies computed by NISA are compared with the theoretical results in Table 8.2.1. The displacement response results are presented in Table 8.2.2 along with theoretical results. The amplitude and phase spectra for the displacement at node 2 are shown in Figs. 8.2.2 and 8.2.3.

REFERENCE:

1. W.T. Thomson, *"Theory of Vibration with Applications,"* Prentice Hall Inc., Englewood Cliffs, N.J., 2nd. edition (1981).
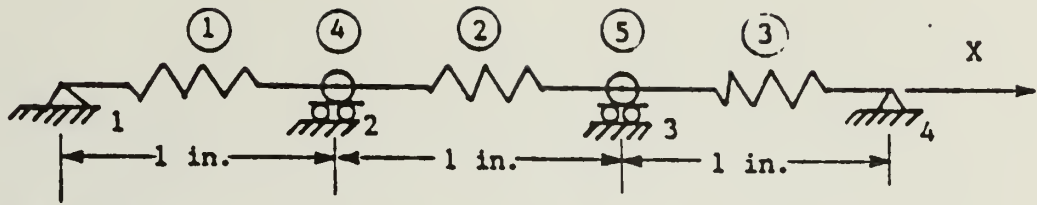
Figure 8.2.1   Two Degrees of Freedom System
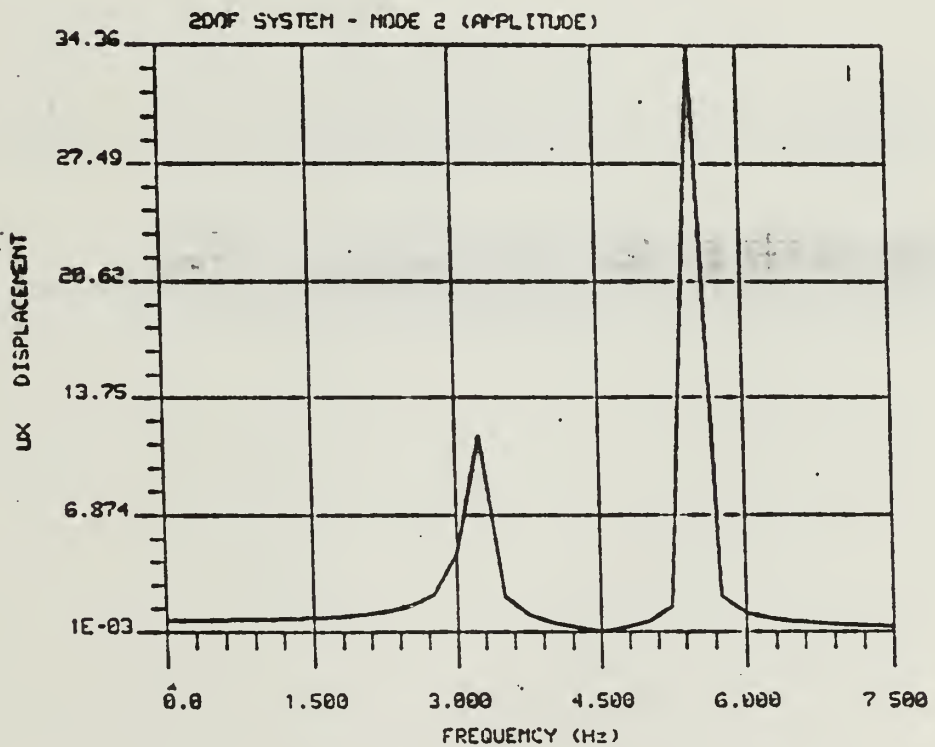
Figure 8.2.2   Response amplitude Spectrum for x-Displacement
             at node 2

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,3
LIN,2GD,1,2,1
FEG,BAR,1,,,3/18,1/1
ELE,ADD,4,25/1/2/2,2
ELE,ADD,5,25/1/2/2,3
DISP,ADD,1/4,1,0/0
DISP,ADD,2/3,1,/0
PROP,ADD,1,0.5
PROP,ADD,2,200.
DEC
INS
ANAL=EIGEN
FILE=EF01
SAVE=26,27
RESE=OFF
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*LDCASE,1
*EIGCNTL,2,0,20,0,0.0,0.0,1.0E-05
QUI
QUI
WRI,NISA,EF01.NIS
WRI,DBS,EF01.BIN
END
N
```

```
**  EXECUTIVE
ANAL=EIGEN
FILE=EF01
SAVE=26,27
RESE=OFF
AUTO=ON
EIGE=SUBS,CONV
MASS=LUMPED
*ELTYPE
   1,   18,      1
   2,   25,      1
*RCTABLE
      1,       1
200.0000
      2,       1
0.5000
*ELEMENT
      1,      1,      1,      1, 0
      1,      2,
      2,      1,      1,      1, 0
      2,      3,
      3,      1,      1,      1, 0
      3,      4,
      4,      2,      2,      2, 0
      2,
      5,      2,      2,      2, 0
      3,
*NODES
      1 ,,,,      0.0000,      0.0000,      0.0000,
      2 ,,,,      1.0000,      0.0000,      0.0000,
      3 ,,,,      2.0000,      0.0000,      0.0000,
      4 ,,,,      3.0000,      0.0000,      0.0000,
*I2
2,0,20,0,0.0,0.0,1.0E-05
*SPDISP
**  SET ID =         1
      1,UY   ,    0.000000
      1,UX   ,    0.000000
      2,UY   ,    0.000000
      3,UY   ,    0.000000
      4,UY   ,    0.000000
      4,UX   ,    0.000000
*ENDDATA
```

COMMAND>GRD,ADD,1,0/0/0

This adds a grid (a point in 3-D space), calls it
grid #1, and puts it at coordinates (0,0,0).

COMMAND>GRD,TRS,1,2,3

This makes grid #2 by "translating" grid #1 3 units
in the X direction. Actually it translates grid #1 3/0/0
units where the format is X/Y/Z: since there are no "0"s
typed in, they are automatically assigned. If it was
desired that #2 be placed 3 units in the Y direction, the
input would be either "GRD,TRS,1,2,0/3" or
"GRD,TRS,1,2,/3". Default values are "0".

COMMAND>LIN,2GD,1,2,1

This creates a line starting at grid #1 and ending at
grid #2, calling it line #1. The format is
"LIN,2GD,starting grid #,ending grid #".

COMMAND>FEG,BAR,1,,,,3/18,1/1

FEG stands for "Finite Element Generation". This
entry creates a group of 3 bar/beam elements (of element
type 18), in the global coordinate system, assigning a

2.101

material ID # of 1 and a property ID # of 1, and a zoom of 1. Each element is defined with a node on each end and both nodes and elements are automatically numbered by the program. This is accomplished using the following format: "FEG,BAR,3,4,5,6A/6B/6C,7A/7B/7C" where:

Entry 3 is the number of lines to be "FEGED".

Entry 4 is the list of generated elements. If left blank the program will automatically number the elements.

Entry 5 is the list of generated nodes. If left blank the program will automatically number the nodes.

Entry 6A is the number of elements to be generated for each line.

Entry 6B is the NKTP #, which stands for the element type identification number. Each element type has a different NKTP #. For an explanation of exactly what a specific NKTP # is, refer to the element library section of the NISA II User's Manual, pages 4.0-1 to 4.120-5.

Entry 6C is the node # that has a local coordinate system the user may wish to use. This entry is used only if the beam is not to be defined in the global coordinate system (ie the Cartesian, Cylindrical, or Spherical systems). At times it is more convenient to refer to items with a local coordinate system, for example a

force or displacement constraint may be defined
with directions which are not aligned with the
global coordinate system. For more information
on local coordinate systems refer to pages 3.2-1
to 3.2-4 of the NISA II User's Manual.

Entry 7A & 7B are the material and property ID
#'s respectfully. This set of #s, assigned to
one or more elements, is used later in the input
phase for assigning actual properties. If left
blank the program automatically assigns a
material ID # of 1 and a property ID # of 1.
These numbers do not have to be the same. A
group of elements can have the same material
properties (Poisson's ratio, Young's modulus,
etc.) with some of the elements having one
geometric property and others having a different
geometric property. As a result this group of
elements would have one material ID # and two
property ID #s.

Entry 7C is the zoom factor, ie the length of
the last element to length of the first element.
If all elements are the same length then the
zoom = 1. ˆ If left blank the program
automatically assigns a value of 1. For more
information and a pictorial representation refer
to page B-3 of the DISPLAY II User's Manual.

This adds an element #4 with NKTP = 25, order = 1, material ID # = 2, property ID # = 2 at node 2, using the following format: "ELE,ADD,3,4A/4B/4C/4D,5A/5B,6" where:

Entry 3 is the element # to be added. If this is left blank the program will automatically assign a number.

Entry 4A is the NKTP # (see entry 6B under the FEG description above).

Entry 4B is the order #. Some elements only have one order, NORDR=1, others have many depending on the element type. For a pictorial description of how this order works/varies refer to section 4, the "element library" of the NISA II User's Manual; specifically NKTP #s 1,2,3,4,5,& 32 provide good examples of some of the element types with varying orders.

Entry 4C is the material ID #. This # is used later in the input phase to assign actual material properties to this element.

Entry 4D is the property ID #. (See entry 7B under the FEG description above.)

Entry 5A & 5B are the two nodes that define the element. Most elements require two nodes, one at each end. However since this particular element is a mass concentration it only requires one node.

Entry 6 is the vector number of a previously
defined vector used in establishing a local
coordinate system. This can be used for
elements of NKTP = 11, 12, & 38.

COMMAND>DISP,ADD,1/4,1,0/0

This constrains nodes 1 and 4 in the X and Y
direction and assigns a constraint ID # of 1 with the
following format: "DISP,ADD,3A/3B/3C...,4,5A-5F", where:

Entry 3 is the list of nodes to be constrained.
Entry 4 is the constraint ID #. The set ID
under which the constraints are specified.
Entry 5A to 5F are the degree(s) of freedom that
are constrained. This is accomplished by
placing a "0" in the degree of freedom that must
be constrained with the following format:
"X/Y/Z/ROTX/ROTY/ROTZ". Consecutive slashes or
constrained directions not listed, indicate a
null entry which the program either ignores or
places a default value in that place. In this
case the program ignores the null entries. For
more information on null entries refer to the
general command rules previously addressed in
this chapter of to pages 2.1-3 to 2.1-6 of the
DISPLAY II User's Manual.

2.105

```
COMMAND>PROP,ADD,1,0.5
```

This adds a translational spring constant of 0.5 lb./in. to all the elements with property ID # 1. This is accomplished with the following format "PROP,ADD,3,4A/4B/4C..." where:

Entry 3 is the property index # (PID).

Entry 4 is the list of properties which are defined by the element type with each different property separated by a /. Refer to the specific element type in the element library section of the NISA II User's Manual for more info on what properties are listed here.

```
COMMAND>DEC
```

This places the program in the mode to accept Executive Commands. For more info on executive commands refer to the previous Executive command overview and page 7.10-1 of the DISPLAY II User's Manual.

```
COMMAND>INS
```

This places the program in the mode to allow the insertion of data lines.

COMMAND>ANAL=EIGEN

This tells the program that the analysis to be performed is an EIGENVALUE analysis.

COMMAND>FILE=EF01

This provides the file name prefix EF01 for the internally generated binary data files.

COMMAND>SAVE=26,27

This tells the program to save the internally generated binary data files 26 and 27. The names of these files will be EF0126.dat and EF0127.dat respectively. Note this command must be used to save the data files 26 and 27, which are required for post-processing or dynamic analysis.

COMMAND>RESE=OFF

This command tells the program not to look at the numbering of the elements nor to resequence them (ie renumber them internally) to make the finite element analysis more efficient. The program is set up to automatically perform this function. Since this model is so small the process is not really necessary. However

2.107

when creating a large finite element model this process
(ie RESE=ON) is usually necessary because there is
definite "pecking order" of how elements and nodes should
be numbered. The analysis of a model that does not follow
this order will take longer than one that does. This
command has the program check the model to see if it
conforms to the "pecking order". If the model does not
then the program will internally change the element
numbering so that it does. The change will not effect the
user as it is transparent to the user and the element IDs
that the user uses are not changed. For more info refer
page 5.3-5 of the NISA II User's Manual

COMMAND>AUTO=ON

     This command activates the automatic spurious normal
rotation constraint for shells. This input is not
specifically required as the default mode is for this to
be on. Nodes in basic shell theory have 5 degrees of
freedom. When transferring a model from the non-global
system to a global system rotation in the theta Z
direction may occur. Additionally rotation perpendicular
to the theta Z direction may occur. If this happens then
the shell (ie model) will not be defined and therefore
cannot be solved. Having the AUTO CONSTRAINT on will
constrain this perpendicular rotation, thereby allowing
the model to be defined.

COMMAND>EIGE=SUBS,CONV

This command tells the program that the eigenvalue
extraction method is to be conventional subspace.   This
command produces some effect on the speed of the analysis.
If the eigenvalues are far apart then having the method be
accelerated subspace will cause the program to
aggressively shift the frequencies in order to speed up
the convergence onto the natural frequencies.   Refer to
pages 3.5-1 to 3.5-4 and 5.3-6 in the NISA II User's
Manual.

COMMAND>MASS=LUMPED

This command tells the program to activate the lumped
matrix formulation, ie it tells the program to solve the
stiffness matrix diagonally instead of consistent with the
way the matrix was developed.   Refer to page 3.5-5 of the
NISA II User's Manual.

COMMAND>*LDCASE,1

This is the load case "control card" for the desired
output.   Refer to page 7.1-2 of the NISA II User's Manual
for an explanation of what the number(s) following the
card heading mean.   Note when performing an Eigenvalue
analysis this card is not applicable.   However, using this

card was the only method the author found of "saving" the
displacement constraints that were input previously. A
more difficult method of inputing the displacement
constraints would be to input them directly into the NIS
file using a text or line editor, like EDLIIN. Therefore
the user should either use this command to save the
displacement constraints and then delete the two specific
load case lines in the NIS file by using a text or line
editor, like EDLIN, or input the displacement constraints
directly into the NIS file. (The NIS file will be
explained below.) The load case lines that need to be
deleted consists of one line containing *LDCASE and a
second line containing the string of numbers input
immediately after the word *LDCASE above.

COMMAND>*EIGCNTL,2,0,20,0,0.0,0.0,1.0E-5

    This command tells the program what the control
parameters are for the eigenvalue extraction. Refer to
pages 7.1-5 to 7.1-6a of the NISA II User's Manual for an
explanation of what each number means.

COMMAND>QUI

    Quits the insert mode.

COMMAND>QUI

    Quits the DEC mode.

COMMAND>WRI,NISA,EF01.NIS

    This takes the above data just input during this
session and writes a "NISA" file called "EF01.NIS" that
the program uses for it's analysis.

COMMAND>WRI,DBS,EF01.BIN

    This takes the above data and writes a binary data
base file called "EF01.BIN"

COMMAND>END

    This ends the session.

Data Base File: At this time the program will ask if a
data base file DISP.BIN is to be written. Since this has
already been done with the WRI,DBS,EF01.BIN command above,
answer no "N".

The model is now ready to have an eigenvalue analysis
performed on it. Once this analysis has been completed
a dynamic analysis can be preformed. <u>Remember that files</u>

<u>26.dat and 27.dat must be saved during the eigenvalue</u>

<u>analysis in order to be used to complete the dynamic</u>

<u>analysis.</u> Refer to the previous section "Program Usage"

for guidance on how to perform the analysis.

```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=EF01
SAVE=26,27
FLOW=0.0
FUPP=10.0
GENF=OFF
INPH=LINEAR
INPO=LINEAR
MRES=OFF
*TITLE
ATTEMPT OF VERIFICATION PROBLEM 8.2
*MODE
1,2,1
*ADDF
0.0,7.5,30
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
PHASE SPECTRUM
1002,0,3,1.0,0.0
*DCFO
2,FX,1001,1002,$,200.0
*RSET
1001,2,UX,3,1
*SPOUT
1001,0,1,1001
*ENDDATA
```

ANAL=FREQUENCY

This tells the program that the analysis to be performed is a Frequency Response analysis.

DAMP=VISCOUS

This tells the program that the model has viscous damping in it.

FILE=EFO1

This tells program that the prefix EFO1 is to be used for the internally generated binary data files. This prefix must be the same as that used for the eigenvalue portion of the problem. The reason for this is because the dynamic analysis must use the 26.dat and 27.dat files generated by the eigenvalue analysis, for the its' analysis. This command provides the prefix name to look for the required data.

SAVE=26,27

This command tells the program to save the internally generated data files, 26.dat and 27.dat. Note this command is not required, as the dynamic program

automatically saves these two files.

FLOW=0.0

This tells the program that the lower limit of the
exciting frequencies is 0.0 cycles per time.

FUPP=10.0

This tells the program that the upper limit of the
exciting frequencies is 10.0 cycles per time.

Note: In a Frequency Response analysis the program
determines the response of the system at various exciting
frequencies ranging from the lower limit (FLOW value) to
the upper limit (FUPP value).

GENF=OFF

This command tells the program to suppress the
generation of frequency points between modes. A system
with more than one degree of freedom has more than one
natural frequency. 'The first mode (or fundamental mode)
refers to the mode associated with the lowest natural
frequency. During a frequency response analysis the
program determines the response of the system to
frequencies varying between those set with the upper and

lower limit commands. With this command set to off
(GENF=OFF) only the response to the natural frequencies
occurring between the upper and lower limits will be
analyzed. However if this command is on (GENF=ON) then
the program will also determine the system's response to
various other frequencies equally spaced between the
natural frequencies within the upper and lower limits.

INPH=LINEAR

     This command tells the program that the effect of any
specified phase angle is computed along a linear ordinate
and abscissa. The other choices would be to have the
ordinate and/or the abscissa be logarithmic.

INPO=LINEAR

     This command tells the program that the effect of any
specified power spectral density, amplitude spectrum or
shock spectrum is computed along a linear ordinate and
abscissa. The other choices would be to have the
ordinate and/or the abscissa be logarithmic.

MRES=OFF

     This tells the program not to (ie to suppress modal
response) printout the displacement, velocity and

acceleration responses for all modes. As a result only
those modes specified will have responses printed.

From this point on all commands will consists of more than
one line of data. As a result a "*" indicates the start
of a new set of data.

*TITLE

     This is a two line data group. This line tells the
program that the following line of input will be the
analysis title.

ATTEMPT OF VERIFICATION PROBLEM 8.2

     This is the title provided to the analysis which the
program will then print at the top of the answer output.

*MODE

     This is a multi line data group. This line tells the
program that the following line(s) of input will contain
the data specifying what modes are to be analyzed. This
command is always required, refer to page 8.4-2 of the
NISA II User's Manual.

This tells the program that modes 1 and 2, with an
increment of 1 between them, are to be analyzed with zero
"0" damping.   This is accomplished with the following
format: Entry 1,2,3,4 where:

Entry 1 is the first mode to be analyzed.

Entry 2 is the last mode to be analyzed.

Entry 3 is the increment between modes to be

analyzed.   Suppose the modes to be analyzed were

modes 1, 5 and 10 then entry 1 would be 1, entry

2 would be 10 and entry 3 would be 2.

Entry 4 is the damping identification # that

corresponds to the damping parameters set

previously.   The damping ID # allows a system to

be analyzed for various damping values without

remodeling the whole system each time.   In this

example there is zero "0" damping.   Therefore,

there is no damping data input.   However,

suppose it was desired to analyze this problem

with two different damping values, say 30% and

70% of critical.   To do this two damping data

groups would be required.   Part of the damping

data input is an arbitrary ID #.   Let's let the

30% damping have an ID # of 1030 and the 70%

have an ID # of 170 (note these numbers are

totally arbitrary).   Now to find the response of

this system with those two values input:

2.118

```
                    *MODE

                1,2,1,1030

                1,2,1,170
```

*ADDF

    This is a two line data group. This line tells the
program that the following line of input will contain the
data specifying the exciting frequencies that are to be
added to the system.

```
0.0,7.5,30
```

    This tells the program that the system is to be
analyzed for 30 exciting frequencies equally spaced
between 0.0 cycles per time and 7.5 cycles per time. This
is accomplished with the following format: Entry 1,2,3
where:

            Entry 1 is the value of the starting frequency.

            Entry 2 is the value of the ending frequency.

            Entry 3 is the number of equal increments

            between entry 1 and 2 that are to be analyzed.

*SPEC

    This is a multi line data group. This line tells the
program that the following lines of input will provide the

spectrum definition. This is accomplished with multiple
two line series where the first line provides the spectrum
description and the second line provides the data.

AMPLITUDE SPECTRUM

This tells the program that the following line of
data defines the amplitude spectrum.

1001,0,3,1.0,1.0

This tells the program that the amplitude spectrum
has a constant value of 1.0, a scale factor of 1.0 and an
ID # of 1001. This is accomplished with the following
format: Entry 1,2,3,4,5 where:

Entry 1 is an arbitrarily assigned #.

Entry 2 is the number of points used to define
the spectrum. If the spectrum is constant this
entry is ignored. If the spectrum is not
constant this entry must be greater than or
equal to two.

Entry 3 is a number between 0 and 3 which is
used to define the input spectrum. "0" -
spectrum is a function of frequency
(cycles/time); "1" - spectrum is a function of
frequency (radians/time); "2" - spectrum is a
function of period; "3" - spectrum is constant.

Entry 4 is the scale factor for the spectrum
data.

Entry 5 is the constant spectrum value.

Note if the spectrum is not constant then the next line(s)
of input would define the spectrum. This would be
accomplished with each line containing a maximum of three
pairs of numbers. The first number of the pair is the
frequency/period value. The second number is the spectrum
value that corresponds with the frequency/period value.
The total number of pairs must equal the value placed in
entry 2 above.

PHASE SPECTRUM

This tells the program that the following line of
data defines the phase spectrum.

1002,0,3,1.0,0.0

This tells the program that the phase spectrum has a
constant value of 0.0, a scale factor of 1.0 and has an ID
# of 1002. Refer to the above description for format and
entry discussions.

For every line of Amplitude Spectrum data there has to be
a corresponding line of Phase Spectrum data. The first
entry of the data lines must be different. For example

2.121

suppose there are two dynamic forces being applied to a
model. One of the forces is out of phase with the other
force by 90 degrees. This would be modeled as follows:

        Amplitude Spectrum

        1001,0,3,1,1

        Amplitude Spectrum

        1002,0,3,1,1

        Phase Spectrum

        1003,0,3,1,0

        Phase Spectrum

        1004,0,3,1,1.571

The description "Amplitude Spectrum" or "Phase Spectrum"
must proceed every line of data. Also the Phase Spectrum
data line 1003 corresponds to Amplitude Spectrum data line
1001 and 1004 corresponds with 1002. Refer to the above
description for an explanation of what each line of data
means.

*DCFO

    This is a multi line data group. This line tells the
program that the following line(s) of input will contain
the data used to specify a concentrated nodal force.

2,FX,1001,1002,$,200.0

    This tells the program there is a concentrated force
of 200.0 units in the X direction at node 2 for both the

amplitude and phase spectrums.   This is accomplished with
the following format: Entry 1,2,3,4,5,6,$,7 where:

       <u>Entry 1</u> is the node number the force or moment
is applied to.

       <u>Entry 2</u> is the force or moment direction.

       <u>Entries 3 & 4</u> are the ID #s that corresponds to
the amplitude and phase spectrums.

       <u>Entry 5</u> is the last node that the force or
moment is applied to.

       <u>Entry 6</u> is the increment between the first and
last nodes this force or moment will be applied
to.

       <u>$</u>   This tells the program to skip all entries
between the last one inputed and entry 7.

       <u>Entry 7</u> is the value of the force multiplier.
Basically there are two numbers that can be
input here. The correct number depends on the
spectrum definition. The easiest way to explain
this is with an example. Therefore, let the
forcing function = 200 sine 400 theta. Now if
the amplitude definition was based on 200 sine
400 theta (ie theta = 0.5236 radians and the
force = 173.2) then the force multiplier is 1.
However if the amplitude was based on sine 400
theta (ie theta = 0.5236 radians and the force =
0.866) then the force multiplier = 200.
However, any number can be used as the force

multiplier, as long as the user maintains
consistency through out the modal.

*RSET

This is a multi line data group. This line tells the
program that the following line(s) of input will define
the nodal/element response quantities that are to be
provided by the program. This command is required and
there must be a minimum of one line of *RSET data for each
line of AMPLITUDE SPECTRUM. Refer to page 8.7-2 of the
NISA II User's Manual.

1001,2,UX,3,1

This tells the program that the response at nodes 2
and 3, in the X direction, is to be provided with a
response ID # of 1001. This is accomplished with the
following format: Entry 1,2,3,4,5,$,6 where:

      Entry 1 is an arbitrarily assigned response ID
#.

      Entry 2 is the node or element # to be analyzed.

      Entry 3 iş the response to be specifically
provided.

      Entry 4 is the last node or element # to be
analyzed.

      Entry 5 is the node or element increment

2.124

(between the entry 2 node/element and entry 4 node/element).

$\underline{\$}$ This tells the program to skip all entries between the last one inputed and entry 6.

Entry 6 is the layer number for composite elements.

Note: Additional input lines can be used to specify responses for different nodes/elements and/or different responses. For example suppose the responses at node 2 in both the X and Y directions was desired. Then there would be two lines of input, one with entry 3 containing UX and one with entry 3 containing UY.

*SPOUT

This is a multi line data group. This line tells the program that following line(s) of data will provide the required output responses.

1001,0,1,1001

This tells the program that the output spectr, response maxima and, the frequencies at which they occur are to be printed, with a zero "0" phase angle shift, for response ID set 1001. This output will have an identification number 1001, which is referenced during post processing. This is accomplished with the following

format: Entry 1,2,3,$,4-13 where:

Entry 1 is the output identification # to be
referenced during post processing.

Entry 2 is the phase angle shift.

Entry 3 is the printout option: "0" = output
spectra is calculated but only the response
maxima and the frequencies at which they occur
are printed; "1" = output spectra, response
maxima and the frequencies at which they occur
are printed.   "2" = the response is printed at
the specified phase angle shift.

$ tells the program to skip all entries between
the last one inputed and entry 4.

Entry 4 to 13 are the response set
identification number(s) (up to ten) for which
the output is required. This (these) number(s)
correspond to the response output listed in the
RSET command.

*ENDDATA

    This tells the program that this is the last line of
data.   .

Once this file is completed and the eigenvalue analysis
has been performed the model is ready to have the dynamic
analysis performed.   To perform the dynamic analysis

access the directory and sub-directory that the above file
is saved in, type "NISA286" and hit the return. Now move
the cursor, using the arrow keys, to the "DYNAMIC"
selection and hit the return. At this time the program
will ask for the input file name. Type in the name of the
file above and hit the return. Now the program will ask
for the output file name. Type in the desired output name
and hit return. Now wait for the analysis.

CHAPTER THREE

PROGRAM UTILIZATION AND COMPARISON

INTRODUCTION

Chapter Two provided a user's manual for the NISA II
program as well as a detailed step by step explanation of
how to generate a finite element model and analysis it for
Static, Eigenvalue, Transient Dynamic, and Frequency
Response conditions. This "user's manual" will now be
utilized to model and analyze two different problems, a
rigid frame on a slab on springs and a circular foundation
on a halfspace.

FRAME WITH SLAB ON SPRINGS

The following example consists of a Rigid frame on a
concrete slab on springs.   This example was generated
using DISPLAY II and then analyzed for both a static
response and a frequency response using the NISA II
program.   This analysis is presented in the following
order: Model sketch, DISPLAY II input, input explanation,
frequency response input, input explanation, and response
output.   A GTSTRUDL response is currently being analyzed
and will be placed into the Appendix D of this report, for
a comparison with NISA II, when it is completed.

This example consists of a rigid frame on a concrete slab, on springs. Machinery located in the center of the rigid frame causes a sinusoidal dynamic force of 60,000 # in the X direction and 30,000 # in the Z direction. These two forces are evenly distributed between two points, such that Fx=30,000 # and Fz=15,000 # at each point, are occurring at a frequency of 400 rpm, and Fz is 90 degrees out of phase with Fx. The weight of the machinery is distributed at the top of the rigid frame as follows:
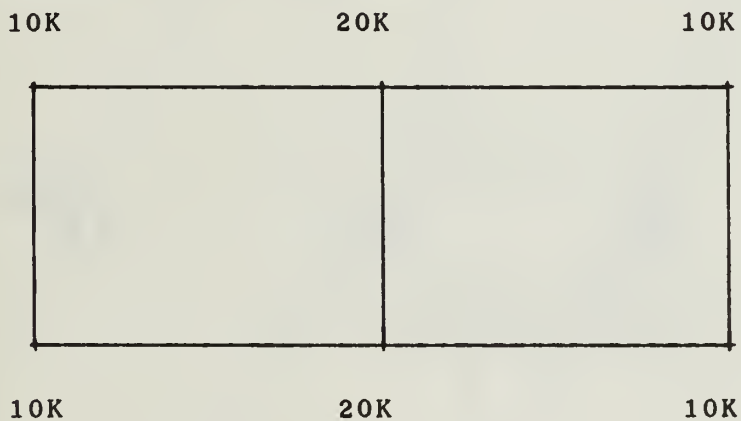
Material properties are as follows:

columns (c1)= W18x119   beams (b1)= W36x230

concrete slab 3ft thick   (b2)= W30x173

    poissons ratio = 0.17

    E = 3.6E6 psi

    gamma = 150 pcf

Springs (kips per inch)

K1 = 586   K2 = 1170   K3 = 2340   K4 = 2500   K5 = 5000

3.4
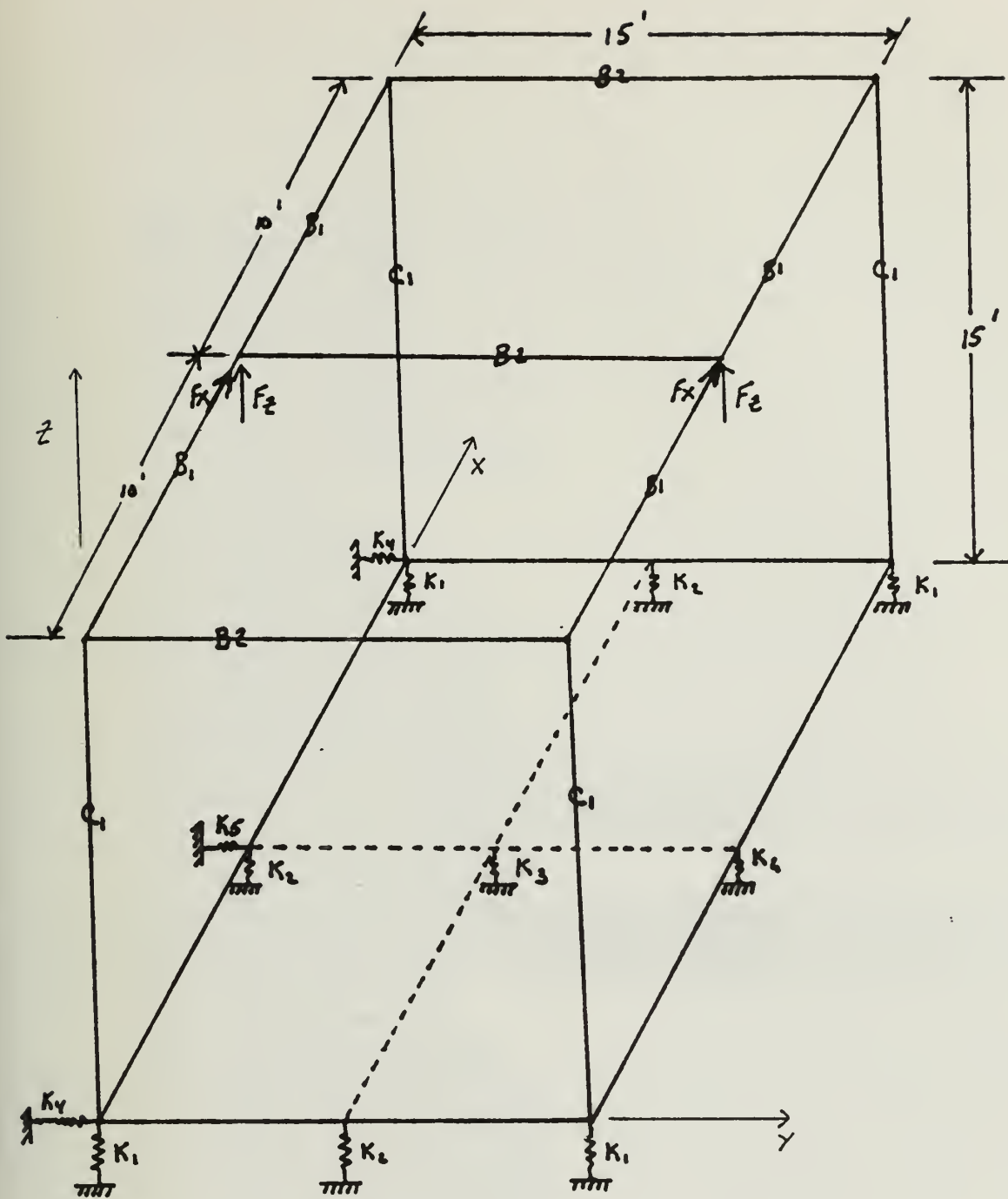
```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,240
LIN,2GD,1,2,1
LIN,TRS,1,2,/180
PAT,2LN,1,2,1          .
VEW,ABS,-80/0/10
PLO
PAT,TRS,1,2,//-20
FEG,QUA,1,,,1/2/2/20,1/1
FEG,QUA,2,,,1/2/2/20,1/1
ELE,DEL,5T8
NOD,ADD,19,/-20
NOD,ADD,20,120/-20
NOD,ADD,21,240/-20
ELE,ADD,5,17/1//2,1/10
ELE,ADD,6,17/1//3,2/11
ELE,ADD,7,17/1//2,3/12
ELE,ADD,8,17/1//3,4/13
ELE,ADD,9,17/1//4,5/14
ELE,ADD,10,17/1//3,6/15
ELE,ADD,11,17/1//2,7/16
ELE,ADD,12,17/1//3,8/17
ELE,ADD,13,17/1//2,9/18
ELE,ADD,14,17/1//5,1/19
ELE,ADD,15,17/1//6,2/20
ELE,ADD,16,17/1//5,3/21
NOD,ADD,22,//180
NOD,ADD,23,120//180
NOD,ADD,24,240//180
NOD,ADD,25,/180/180
NOD,ADD,26,120/180/180
NOD,ADD,27,240/180/180
ELE,ADD,17,12/1/2/7,1/22
ELE,ADD,18,12/1/2/7,3/24
ELE,ADD,19,12/1/2/7,7/25
ELE,ADD,20,12/1/2/7,9/27
ELE,ADD,21,12/1/2/8,22/23
ELE,ADD,22,12/1/2/8,23/24
ELE,ADD,23,12/1/2/8,25/26
ELE,ADD,24,12/1/2/8,26/27
ELE,ADD,25,12/1/2/8,22/25
ELE,ADD,26,12/1/2/8,23/26
ELE,ADD,27,12/1/2/8,24/27
ELE,ADD,28,25/1//10,22
ELE,ADD,29,25/1//11,23
ELE,ADD,30,25/1//10,24
ELE,ADD,31,25/1//10,25
ELE,ADD,32,25/1//11,26
ELE,ADD,33,25/1//10,27
PRO,ADD,1,36/36/36/36
```

```
MAT,ADD,1,3.6E6/0.1///////////2.246E-4
PRO,ADD,2,5.86E5
PRO,ADD,3,1.17E6
PRO,ADD,4,2.34E6
PRO,ADD,5,2.5E6
PRO,ADD,6,5.0E6
PRO,ADD,7,35.1/253/2190
PRO,ADD,8,67.6/940/15000
PRO,ADD,9,50.8/598/8200
MAT,ADD,2,2.9E7/0.3//////////7.332E-4
PRO,ADD,10,25.88
PRO,ADD,11,51.76
DISP,ADD,1T3,1,0///0/0/0
DISP,ADD,4T9,1,0/0//0/0/0
DISP,ADD,10T21,1,0/0/0/0/0/0
DISP,ADD,22T27,1.///0/0/0
FOR,ADD,1,1,//-386.4
DEC
INS
ANAL=STATIC
FILE=GTSTAT
SAVE=26,27
*TITLE,ANALYSIS OF CE6193 GTSTRUDL EXAMPLE
*LDCASE,1
*EIGCNTL,1,0,20,0,0.0,0.0,1.0E-5
*MODEO,1,20,1,1
*EIGO,1,1,0,3,1,0,1
QUI
QUI
WRI,NISA,GTSTAT.NIS
WRI,NISA,GTEIGE.NIS
WRI,DBS,GTSTAT.BIN
WRI,DBS,GTEIGE.BIN
END
N
```

The two previous pages contain the commands required to generate this model using the DISPLAY II program. With the exception of the following clarifications/explanations all commands are explained in Chapter Two.

COMMAND>VEW,ABS,-80/0/10

This command tells the program to change the angle the user is viewing the model with. This actual command will rotate the model clockwise 80 degrees about the X axis and 10 degrees counterclockwise about the Z axis (counterclockwise angles are positive). This "new" viewing angle will now be used for all future model "plots".

COMMAND>PLO

This command tells the program to erase the current screen picture and to replot the model. The previous command changed the viewing angle and this new "plot" will be with the new viewing angle.

COMMAND>FOR,ADD,1,1,//-386.4

The explanations for this command provided in Chapter Two indicate that this command tells the program that there is a force on 386.4 units in the -Z direction on node #1. However upon examining the figure on page 3.4 it

is noted that node #1 has no such force. Therefore why is this command added? It was discovered while developing this model that if a static analysis is to be performed considering just the mass of the model then the command "BODYFORCE" must be used. However the DISPLAY II program does not accept this command. As a result the author used the command "FOR,ADD,..." for two purposes:

> 1. As a reminder to add the "BODYFORCE" command to the ".NIS" file created by the DISPLAY II program.
>
> 2. As an easy method of determining the correct location of inserting the "BODYFORCE" command to the ".NIS" file. This is accomplished by changing the "FORCE" data lines in the ".NIS" file to "BODYFORCE" data lines using a line or text editor. Refer to page 7.3-11 of the NISA II User's Manual for the correct "BODYFORCE" format.

COMMAND>*MODEO,1,20,1,1

This command and the "*EIGO" command were both discussed in Chapter Two. However this is a reminder that the DISPLAY II program does not accept these commands and that they must be inserted into the ".NIS" file using a line or text editor.

```
COMMAND>WRI,...
```

All the examples discussed in Chapter Two had one "WRI,NISA,..." and one "WRI,DBS,..." command. This model has two of each. This may not be required; however, since the .NIS file for a static analysis contains a few different commands than the .NIS file for an eigenvalue analysis, the author felt that the easiest way to perform both analyses was to let the DISPLAY II program create two .NIS files, one for the static analysis and one for the eigenvalue analysis. Once the two files are created, each may then be easily tailored using a text or line editor, for the specific analysis.

```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=GTEIGE
GENF=OFF
*TITLE
FREQUENCY RESPONSE OF CE6193 GTSTRUDL EXAMPLE
*MODE
1,1,1,1001
*DAMP
1001,.1
*ADDF
41.888
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
AMPLITUDE SPECTRUM
1002,0,3,1,1
PHASE SPECTRUM
1003,0,3,1.0,0
PHASE SPECTRUM
1004,0,3,1,1.571
*DCFO
23,FX,1001,1003,$,30000.0
26,FX,1001,1003,$,30000.0
23,FZ,1002,1004,$,15000.0
26,FZ,1002,1004,$,15000.0
*RSET
1001,1,UZ,9,1
1001,22,UZ,27,1
1002,1,UZ,9,1
1002,22,UZ,27,1
*SPOUT
1001,0,0,1001
1002,0,0,1002
*ENDDATA
```

FREQUENCY RESPONSE INPUT EXPLANATION

All the commands listed in the frequency response input
have been explained in Chapter Two. Accordingly a line by
line description is not provided. However the following
general description is provided.

This model consists of a rigid frame, on a concrete
slab, on springs. There are two forces being applied to
nodes 23 and 26, one in the X direction and one in the Z
direction. The Z direction force is 90 degrees out of
phase with the X direction force. An analysis of the
displacement is the Z direction for nodes 1 to 9 and 22 to
27 is required.

CIRCULAR FOUNDATION ON A HALFSPACE

The following example consists of a circular foundation on a halfspace. A halfspace has no boundaries along three edges, typically along the X directions and one of either the Y or Z directions. Since an infinite length cannot really be modeled, a halfspace is modeled by making the halfspace boundaries many times larger than the diameter of the foundation on the halfspace. In this model the foundation has a diameter of 18 feet and the boundaries are located 234 feet from the foundation's center. Additionally the use of symmetry is used in this model. As a result only one half of the model is analyzed. This example was generated using DISPLAY II and then analyzed for a frequency response using the NISA II program. The NISA II analysis was then compared to theoretical analyses of the same model. This analysis is presented in the following order: Model sketch, DISPLAY II input, input explanation, frequency response input, input explanation.

Appendix E contains the analyses comparisons, hand calculations, graphs, and computer printouts used for the theoretical analyses.
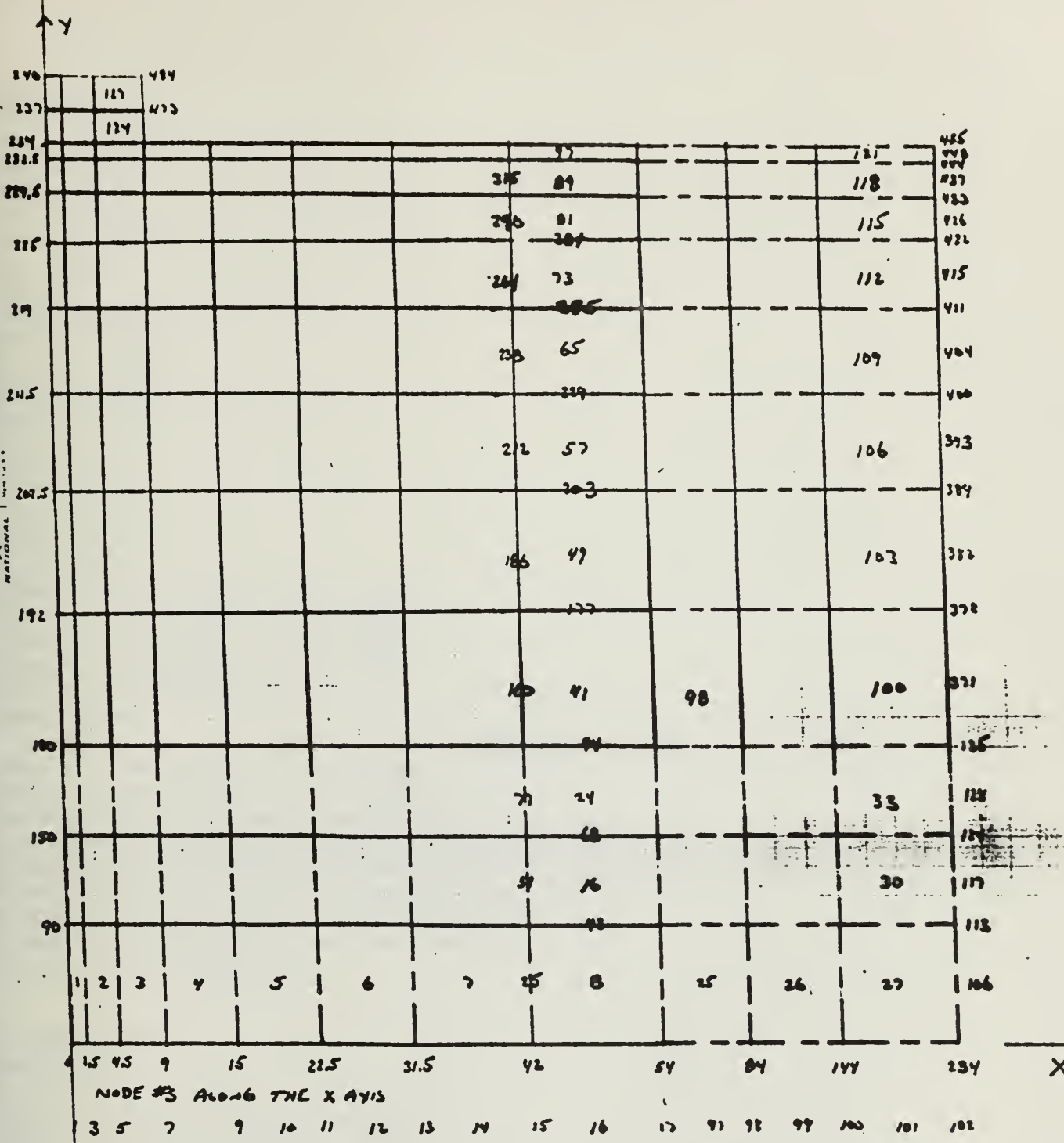
THIS IS A 127 ELEMENT MESH; 9 FT RADIUS BY 6 FT THICK FOUNDATION ON
THE SURFACE. THE SOIL IS REPRESENTED BY 121 ELEMENTS (234' x 234';
11 ELEMENTS X 11 ELEMENTS). ELEMENTS CONTAIN 8 NODES EACH.
THE NUMBERS ALONG THE AXES REPRESENT THE DISTANCE FROM THE ORIGIN
THE SOLID ELEMENTS DRAWN APPROXIMATELY TO SCALE (1" = 125'). THE DASHED
ELEMENTS ARE NOT TO SCALE. ALL NODES ALONG THE OUTSIDE VERTICAL EDGES
ARE PROHIBITED FROM MOVEMENT IN THE X DIRECTION. ALL NODES ALONG THE
BOTTOM HORIZONTAL EDGE ARE PROHIBITED FROM MOVEMENT IN BOTH THE X + Y
DIRECTIONS.

3.13

```
GRD,ADD,1,0/0/0
GRD,TRS,1,2,54
GRD,TRS,1,3,234
LIN,2GD,1,2,1
LIN,2GD,2,3,2
LIN,TRS,1,3,/180
LIN,2GD,1,5,/234
LIN,DEL 2GD
LIN,DEL,4
LIN,TRS,1,5,/234
LIN,TRS,2,4,/180
LIN,TRS,2,6,/234
GRD,ADD,10,9/234
LIN,2GD,6,10,7
LIN,TRS,7,8,/6
PAT,2LN,1,3,1
PAT,2LN,2,4,2
PAT,2LN,3,5,3
PAT,2LN,4,6,4
PAT,2LN,7,8,5
FEG,QUA,1,,,,2/8/3/3,1/1/8/0.3333
FEG,QUA,2,,,,2/3/3/3,1/1/3/0.3333
FEG,QUA,3,,,,2/8/8/3,1/1/8/0.125
FEG,QUA,4,,,,2/3/8/3,1/1/3/0.125
FEG,QUA,5,,,,2/3/2/3,2/2/3/1
NOD,MER,1T484
N
MAT,ADD,1,6.0E5/0.45/////////3.88
MAT,ADD,2,5.76E8/0.45/////////4.66
DISP,ADD,478/474/467/463/344/335/318/309,1,0
DISP,ADD,292/283/266/257/240/231/214/205,1,0
DISP,ADD,188/179/162/153/79/70/53/44,1,0
DISP,ADD,27/18/106/113/117/124/128/135,1,0
DISP,ADD,371/378/382/389/393/400/404/411,1,0
DISP,ADD,415/422/426/433/437/444/448/455,1,0
DISP,ADD,1T17/97T102,1,0
DEC
INS
ANAL=EIGENVALUE
FILE=SOLID
SAVE=26,27
*TITLE,CIRCULAR FDN ON A HALFSPACE
*EIGCN,40,0,40,0,0.0,0.0,1.0E-5
*LDCASE,1
QUI
QUI
WRI,NISA,SOLID.NIS
WRI,DBS,SOLID.BIN
END
N
```

The previous page contains the commands required to
generate this model using the DISPLAY II program. With
the exception of the following clarifications/explanations
all commands are explained in Chapter Two.

COMMAND>NOD,MER,1T484

     This command tells the program to review all nodes
between 1 and 484 (all the nodes of the model) and to
combine any two or more nodes together, into one, that are
located within the "default" distance of each other. This
is accomplished with the following format: "NOD,MER,3,4"
where:

               Entry 3 is the list of nodes to be merged.
               Entry 4 is the "tolerance", ie. the specified
               distance between nodes.

If a model was generated with more than one patch this
command must be used to "tie" the patches together. If
this command is not used on a model created by more than
one patch, it cannot be analyzed. Refer to pages 4.3-12 to
4.3-14 of the DISPLAY II User's Manual for more
information.

COMMAND>N

If no tolerance is specified in the above command the
program will ask if the user would like to modify the

default tolerance (Y/N)? The author chose to use the
default tolerance for this merge and therefore responded
with "N".

The element type used in this model and analysis was NKTP
3, an axisymmetric solid element. This was the first
analysis the author performed using this type of element
as well as the first model generated using multiple
patches. While performing this model generation and
analysis the following restrictions, not found in any of
the NISA II User's Manuals, were discovered:

   The NISA II User's Manual states that "The element
must lie in the right half of the global XY plane where
the global X-axis is the radial direction, and the global
Y-axis is the axis of revolution or axis of symmetry."
The author determined that not only does this element have
to remain in the right half of the global XY plane but it
must also remain in the first quadrant of the global
coordinate system, ie all nodal coordinates must be
positive.

   All the element meshes must be generated starting
with the patch closest to the origin of the coordinate
system, move left along the X-axis, then move up to the
next layer of patches and move left along the X-axis from
the Y-axis again. The following figure contains 5
patches, blocks 1 to 5. To correctly generate a finite
element mesh from these patches, start with patch 1 and

finish with patch 5, completing the generation in
numerical order.

Y |
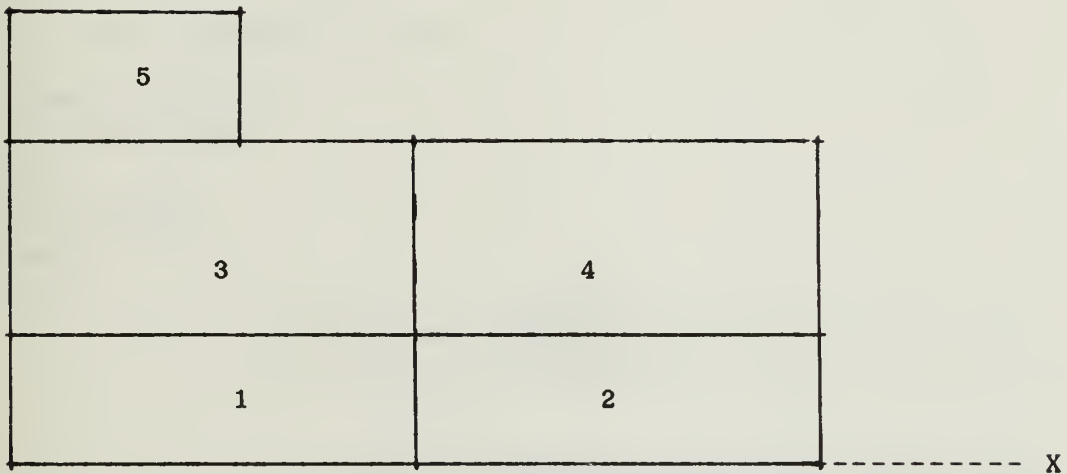
```
ANAL=FREQUENCY
DAMP=VISCOUS
FILE=SOLID
GENF=OFF
*TITLE
SINUSOIDAL LOADING ON A CIRCULAR FDN ON A HALFSPACE
*MODE
1,39,1,1001
*DAMP
1001,0
*ADDF
4.0,8.0,1
*SPEC
AMPLITUDE SPECTRUM
1001,0,3,1.0,1.0
PHASE SPECTRUM
1002,0,3,1.0,0.0
*DCFO
478,FY,1001,1002,478,1,50000
*RSET
1001,344,UY,360,1
1002,344,UX,360,1
*SPOUT
1001,0,0,1001
1002,0,0,1002
*ENDDATA
```

FREQUENCY RESPONSE INPUT OF A CIRCULAR FOUNDATION ON A HALFSPACE

All the commands listed in the frequency response input, shown on the previous page, have been explained in Chapter Two. Accordingly a line by line description is not provided. However the following general description is provided.

This model consists of an 18 foot diameter circular foundation on a halfspace. The halfspace is modeled by setting a displacement constraint in the X and Y direction 234 feet below the base of the foundation and setting a displacement constraint in the X direction along the center of the foundation and 234 feet to the right of the foundation's center. There is one force being applied to the center of the foundation, node 478, in the Y direction. An analysis of the displacements in the X and Y directions for nodes 244 to 360 is required. These nodes are located on the surface (top) of the halfspace starting at the center of the foundation and ending 54 feet away.

CHAPTER FOUR

CONCLUSIONS

The following conclusions can be made based on the results
of this research problem:

1.   There are many different Finite Element Analysis
programs available on the market today.   One of the
programs available for use on a personal computer is
called NISA II and is produced by Engineering Mechanics
Research Corporation located in Troy Michigan.   NISA II is
a very powerful program that is relatively user friendly,
once the user becomes familiar with it, provides rapid
analysis, and is fairly inexpensive to use on a personal
computer.   Additionally it provides answers in good
agreement with check cases.

2.   Even though a program is considered user friendly, as
in the case of the NISA II program, it still requires an
extensive amount of "user time" for someone who is
unfamiliar with the program to become proficient in its
utilization.   A large amount of this user time is
typically spent trying to solve/understand roadblocks
encountered that are not clearly explained in the program
provided user manuals.   Once an individual becomes
proficient with a program, "reasonably" correct answers
should be provided.

3.   The NISA II User's Manuals provide a broad
explanation on how to use the program.   This is fine for

someone who is familiar with the basics of a finite
element modeling program. This report, on the other hand,
is designed to provide a much more basic user's manual on
how to use the NISA II computer program. As a result this
report gives complete step by step examples on how to
develop and analyze various types of static and dynamic
problems, as well as invaluable hints on where to avoid
problems not discussed in the NISA II User's Manuals.

4.   The comparison between NISA and other published
values, of an axisymmetric solid on a halfspace under a
sinusoidal loading, show the following:
     A.   All values, even those of the published
     solutions, tend to have rather large differences.
     B.   NISA II values tend to compare reasonable well,
     not varying any greater than any of the other
     solutions.
     C.   The NISA II values were for a "model" with rigid
     boundaries. These boundaries were located so far
     away from the loaded solid, that it was presumed they
     would have no effect. However as no other finite
     element solution was obtained, for a comparison, this
     can not be completely determined.

5.   When comparing the NISA II solutions to those
calculated using Lysmer it was noted that Lysmer provided
displacements in excess of six times the amount provided

by NISA II. The following are possible reasons:

A.   The NISA II model had rigid boundaries, which may have allowed energy to bounce off, where as the Lysmer finite element model had energy adsorbing boundaries.

B.   The Lysmer curve of frequency verses displacement had only one peak. The NISA II curve of frequency verses displacement gave multiple peaks. The reason for the multiple peaks could be due to the rigid boundaries and therefore, this needs to be looked into more.

C.   As a result of the above it is highly recommended that an exact comparison between NISA and another Finite Element program be accomplished. Based on discussions between the author and the author's advisor it is understood that this comparison will be done in the near future.

CHAPTER FIVE

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Gazetas, G., "Summary of Analysis of Machine Foundation Vibrations: State-of-the-Art". Soil Dynamics and Earthquake Engineering, Vol. 2, No. 1, 1983.

2. Luco, J.E., and Westmann, R.A., "Dynamic Response of Circular Footings", UCLA-ENG-7113, April 1971.

3. Lysmer, Report by Lysmer, University of California, Berkley, from CE-6193 Dynamics of Massive Media class notes.

4. NAVFAC, "Soil Dynamics, Deep Stabilization, and Special Geotechnical Construction", NAVFAC DM-7.3

5. Engineering Mechanics Research Corporation, NISA II User's Manual, EMRC Troy Michigan 48099.

6. Engineering Mechanics Research Corporation, DISPLAY II User's Manual, EMRC Troy Michigan 48099

7. Engineering Mechanics Research Corporation, Verification Problems Manual, EMRC Troy Michigan 48099

APPENDIX A
REQUIRED EQUIPMENT

REQUIRED EQUIPMENT

In order to use the NISA II program, version 88.7, on a PC
the following minimum equipment is required:

1. IBM PC-XT or IBM PC-AT compatible Personal
Computer. The author used a HP 386.

2. 640K bytes of central memory.

3. IBM compatible Color Graphics Adaptor (CGA) and
monitor.

4. A 20 MB hard disk with a 360 KB floppy drive.

5. A Math Coprocessor (Intel 8087 of 80287 chip).

6. A 132 character line printer.

7. MS-DOS Version 2.1 of above operating system.

8. A Parallel port for the protection device
supplied by EMRC.

Additionally the following equipment is Optional:

1. IBM compatible Enhanced Graphics Adaptor (EGA)
and Enhanced Display monitor.

2. IBM compatible Professional Graphics Controller
(PGA) and Professional Display monitor.

3. Ink jet printers (ie Tektronix 4696) and screen
dump software for EGA and PGA.

The author encountered a problem during the initial set up

and operation of the program. This problem was due to
the MS-DOS version (DOS 3.3) and all the associated
software requiring too much space. As a result the
hardware did not have 600K bytes of RAM available for the
NISA II and DISPLAY II programs. To over come this
problem the author deleted everything except the main DOS
3.3 from the operating systems start up path. This
resulted in having about 596K of RAM available for the
NISA II and DISPLAY II programs to operate, which was
enough.

APPENDIX B

RUNNING TIME

.

The author used a HP VECTRA QS/16 to perform all analysis
on. The actual time required to perform an analysis
depends more on the type of problem being analyzed than
anything else. However in an attempt to provide some type
of guide to running time the following graphs are
provided. As shown on these graphs, whether it is the log
total time or just cpu time, the number of degrees of
freedom is the controlling factor on length of time
required to perform an analysis.

In addition to the above "guide", the following is
provided. Two eigenvalue analyses were performed on a
model with: 860 DOF, 430 nodes, 127 elements, and a wave
front of 84. One analysis used "accelerated subspace"
eigenvalue extraction and the other analysis used
"inverse" eigenvalue extraction. The accelerated
subspace method took approximately 3993 seconds (about 67
minutes) to provide the first 40 eigenvalues. The inverse
method took approximately 7708 seconds (about 1 1/2 hours)
to extract 30 eigenvalues starting with a frequency of 2
cps.

LOG RUNNING TIME

B.2
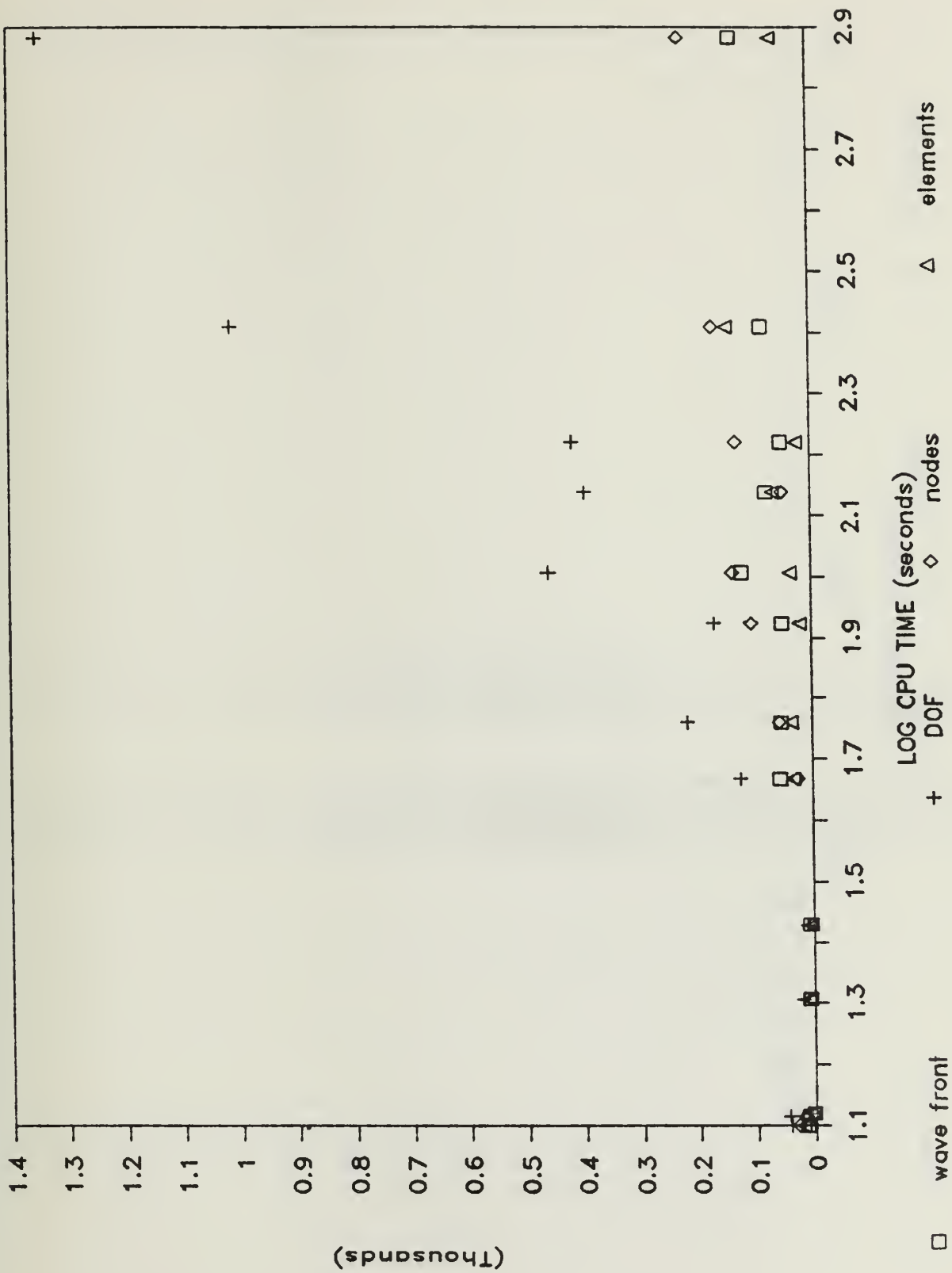
RUNNING TIME

LOG RUNNING TIME

(Thousands)

LOG TOTAL TIME (seconds)

□   wave front   +   DOF   ◇   nodes   △   elements

B.4

RUNNING TIME

B.5
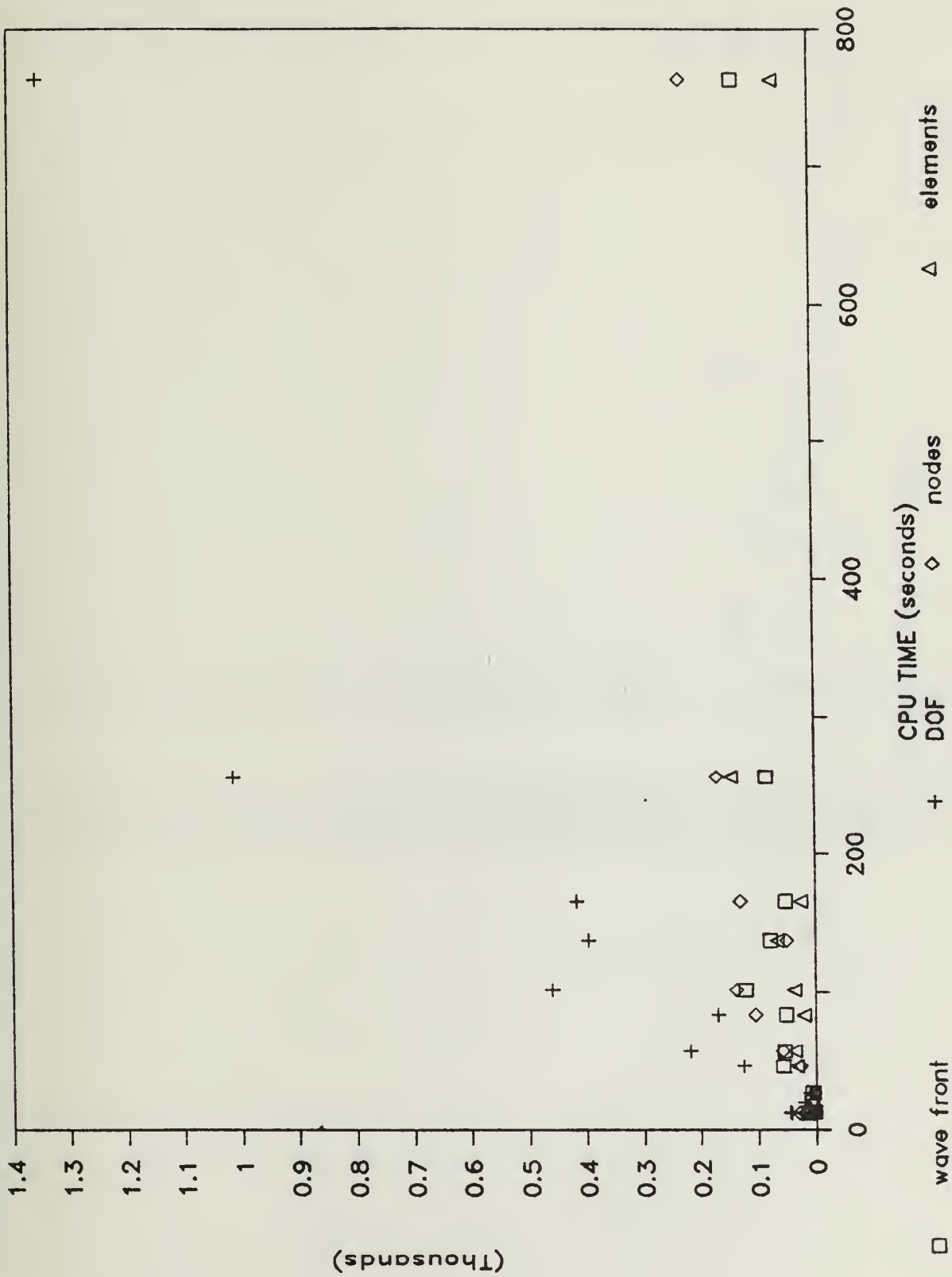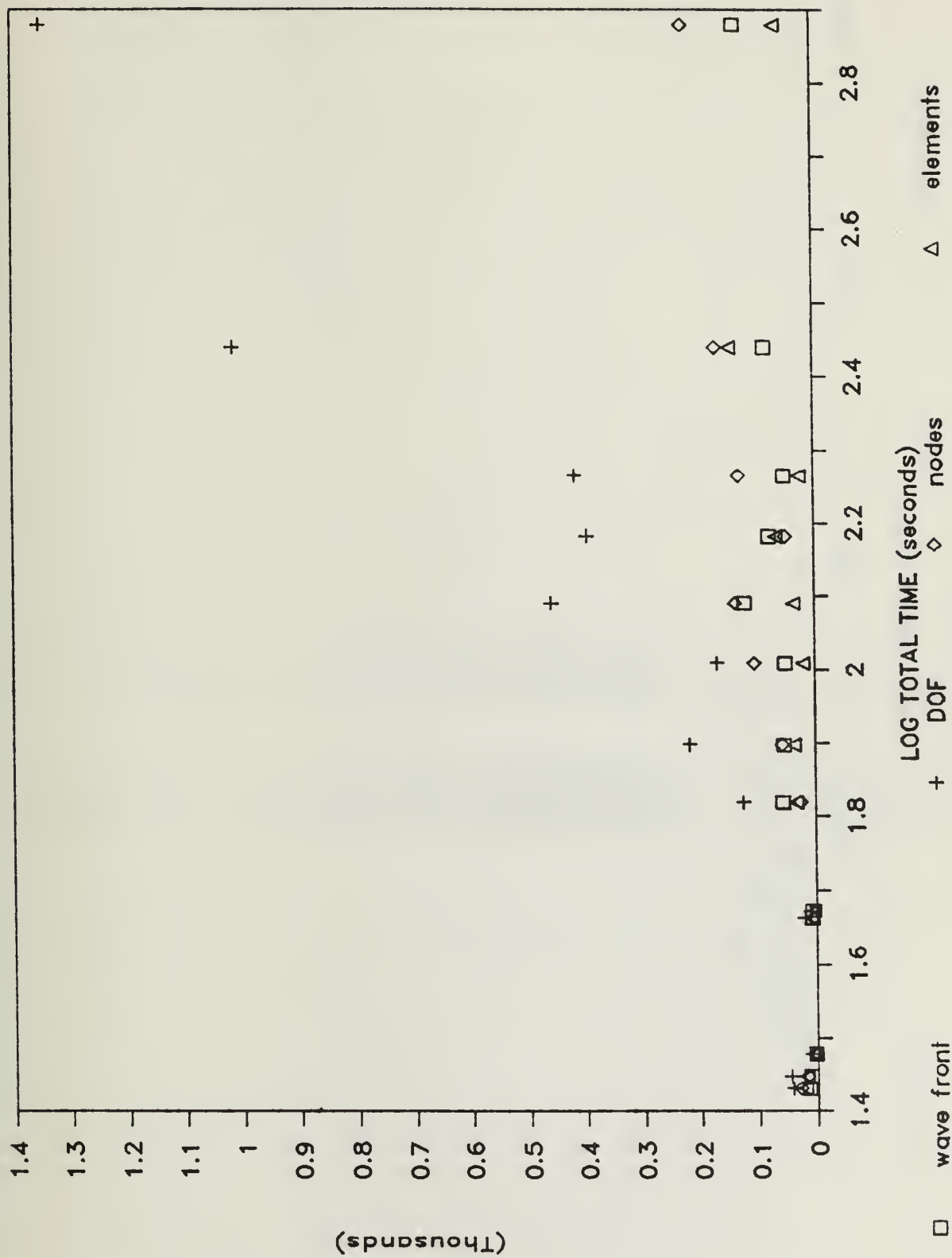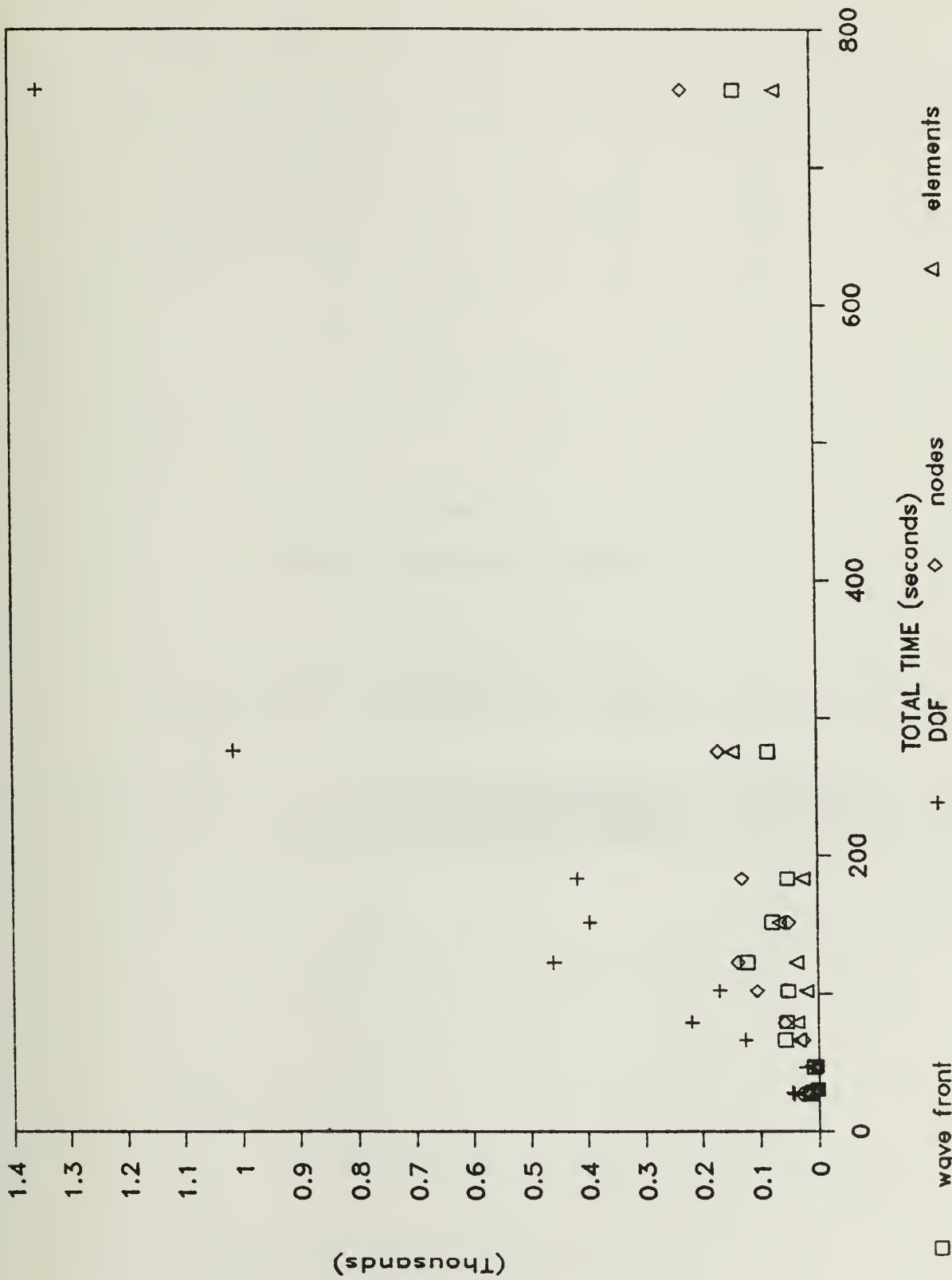
APPENDIX C

NISA II EXECUTIVE COMMANDS

Table 5.2.3 Alphabetical list of *all* executive commands for *all* analyses

Executive Command Name[1]	Description	Applicable Analysis Types[2]
ACUTOFF	Range of model coupling	RA
ANALYSIS	Specify analysis type	ALL1, ALL2
AUTO CONSTRAINT	Spurious normal rotation constraint	ST, EV, BU
BLANK COMMON	Blank common storage unit	ALL1, ALL2
DAMPING	Type of damping	ALL2
DELT TIME	Time increment	TR
DIRECTION	Directional superposition procedure	SHO
EIGEN EXTRACTION	Eigenvalue extraction method	EV, BU
ELEMENT ECHO	Echo element input	ALL1
EXECUTION	Select execution/checking run	ALL1, ALL2
ENDTIME	Ending time	TR
FILENAME	Specify NISA file prefix	ALL1, ALL2
FLOWER	Lower limit of exciting frequency	RA, FR
FRQRDF	Reduction factor for exciting frequency point generation	RA, FR
FSMALL	Frequency cut-off to identify rigid body modes	ALL2
FUPPER	Upper limit of exciting frequency	RA, FA
GENFREQUENCY	Control generation of exciting frequency points	RA, FA
GFACTOR	Acceleration due to gravity	ALL2
INITIAL TEMPERATURE	Define initial temperature	TH
INPOLATION	Type of interpolation	RA, FR, SHO
INPHASE	Type of interpolation for phase spectra	FR
INTEGRATION	Type of integration procedure	RA
MASS FORMULATION	Mass formulation for eigenvalue analysis	EV

(1) Minimum abbreviations are in bold face

(2) ST: Linear static   EV: Eigenvalue   BU: Buckling
NL: Nonlinear static   SH: Steady state heat transfer   TH: Transient heat transfer
TR: Transient dynamics   FR: Frequency response   RA: Random vibration
SHO: Shock spectrum
ALL1: All static and heat transfer analyses (ST, EV, BU, NL, SH, TH)
ALL2: All modal dynamic analyses (TR, FR, RA, SHO)

Table 5.2.3 Alphabetical list of *all* executive commands for *all* analyses (continued)

Executive Command Name[1]	Description	Applicable Analysis Types[2]
MAXCPU TIME	Limit of CPU seconds	NL
MRESPONSE	Control printout of modal responses	FR
NLTYPE	Specify nonlinear static analysis type	NL
NODE ECHO	Echo node input	ALL1
ORTHOTROPIC	Orthotropic direction definition	SH, TH
RESEQUENCE	Element resequencing	ALL1
RESTART	Restart options	SH, TH
SAVE FILE	Save specific NISA files	ALL1, ALL2
SIGMA	Stefan-Boltzmann coefficient	SH, TH
SORT STRESS	Sort highest stresses	ST
STARTING TIME	Starting time	TR
STEP	Total number of steps for heat transfer analysis	TH
STRESS STIFFENING	Stress stiffening option	ST
WARNING	Set warning flag	ALL1, ALL2

(1) Minimum abbreviations are in bold face

(2) ST: Linear static   EV: Eigenvalue   BU: Buckling
    NL: Nonlinear static   SH: Steady state heat transfer   TH: Transient heat transfer
    TR: Transient dynamics   FR: Frequency response   RA: Random vibration
    SHO: Shock spectrum
    ALL1: All static and heat transfer analyses (ST, EV, BU, NL, SH, TH)
    ALL2: All modal dynamic analyses (TR, FR, RA, SHO)

APPENDIX D

NISA II VERSES GTSTRUDL

DISPLACEMENT COMPARISONS

NODE	Vertical Displacement (inches)				Horizontal Displacement (inches)			
	NISA		6T STRUDL		NISA		6T STRUDL	
	Static	Dynamic	Static	Dynamic	Static	Dynamic	Static	Dynamic
1	1.717×10^{-2}	2.611×10^{-7}			0	0		
2	1.684×10^{-2}	2.657×10^{-7}			0	0		
3	1.717×10^{-2}	1.521×10^{-7}			0	0		
4	1.694×10^{-2}	1.204×10^{-7}			0	0		
5	1.696×10^{-2}	1.807×10^{-7}			0	0		
6	1.694×10^{-2}	1.647×10^{-7}			0	0		
7	1.717×10^{-2}	2.444×10^{-7}			0	0		
8	1.689×10^{-2}	1.388×10^{-7}			0	0		
9	1.717×10^{-2}	1.215×10^{-7}			0	0		
22	1.807×10^{-2}	1.543×10^{-7}			0	3.371×10^{-3}		
23	2.756×10^{-2}	5.64×10^{-8}			0	3.372×10^{-3}		
24	1.807×10^{-2}	2.635×10^{-8}			0	3.713×10^{-3}		
25	1.807×10^{-2}	9.616×10^{-8}			0	3.371×10^{-3}		
26	2.756×10^{-2}	1.86×10^{-7}			0	3.372×10^{-3}		
27	1.802×10^{-2}	1.032×10^{-7}			0	3.371×10^{-3}		

APPENDIX E

THEORETICAL DISPLACEMENT CALCULATIONS
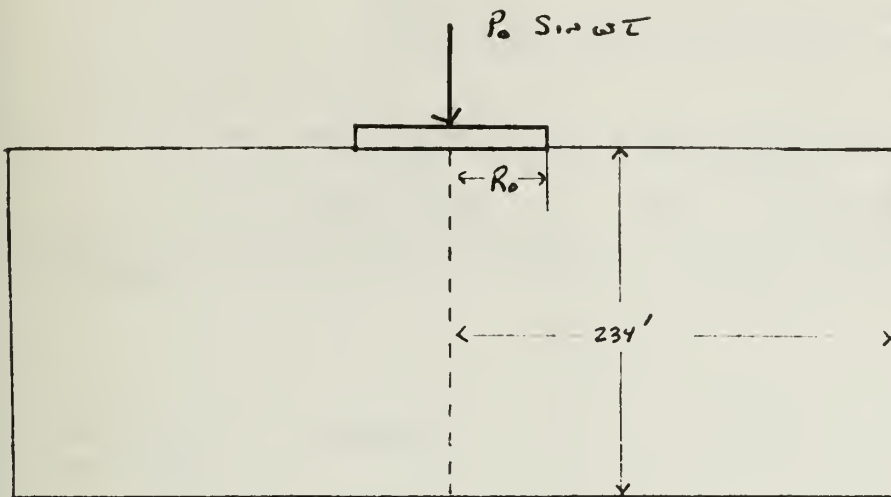
6 FT THICK CONCRETE CIRCULAR FOUNDATION ON SOIL

FDN MASS = 5145.4 $\#SEC^2/FT$ OR $0 \#SR^2/FT$

SOIL PROPERTIES

$\rho = 3.88 \#SEC^2/FT^4$

$G = 206,000$ psf

$\nu = 0.45$

FORCING FUNCTION $P_0 \sin \omega t$

$P_0 = 50,000 \#$

$\omega = 12.57$ RADIANS/SEC   OR   18.85 RADIANS/SEC

CIRCULAR FOUNDATION ON A HALFSPACE
DISPLACEMENT CALCULATION SUMMARY

	f = 2 CPS FDN HAS MASS	f = 3 CPS FDN IS MASSLESS	f = 3 CPS FDN HAS MASS
NISA	0.00137 Ft	0.00231 Ft	0.00211 Ft
GAZETAS By Computer	0.00186 Ft	0.00174 Ft	0.00184 Ft
NAUFAC Hand Cal.	0.00344 Ft	0.00355 Ft (mass = 1×10^{-5} #sec²/ft)	0.00332 Ft
LUCO Hand Cal		0.00344 Ft	
LYSMER Hand Cal	0.00916 Ft		0.00247 Ft

FOR 2CPS NISA II ANALYZED THE PROBLEM USING 33 EIGENVALUES
STARTING WITH THE FUNDAMENTAL VALUE OF 0.635 CPS AND
ENDING WITH 3.57 CPS. DUE TO THE LARGE DIFFERENCE BETWEEN
LYSMER AND NISA, AND THE FACT THAT 2CPS IS ON THE PEAK
OF LYSMER'S CURVE; A NISA ANALYSIS WAS PERFORMED ON
THE MODEL WITH f = 1.0 TO 3.0 IN INCREMENTS OF 0.1 CPS.
THE RESULTS ARE SHOWN ON THE FOLLOWING GRAPH

FOR 3CPS NISA II ANALYZED THE PROBLEM USING 40 EIGENVALUES
RANGING FROM ABOUT 2.1 AND ENDING WITH ABOUT 4.2

FORCING FUNCTION FREQUENCY
VS
DISPLACEMENT

AMPLITUDE (INCHES)

0.0094

0.009

0.008

0.007

0.006

0.005

0.004

0.003

0.002

0.001

1.0 1.2 1.4 1.6 1.8 2.0 2.2 2.4 2.6 2.8 3.0 CPS

E.3
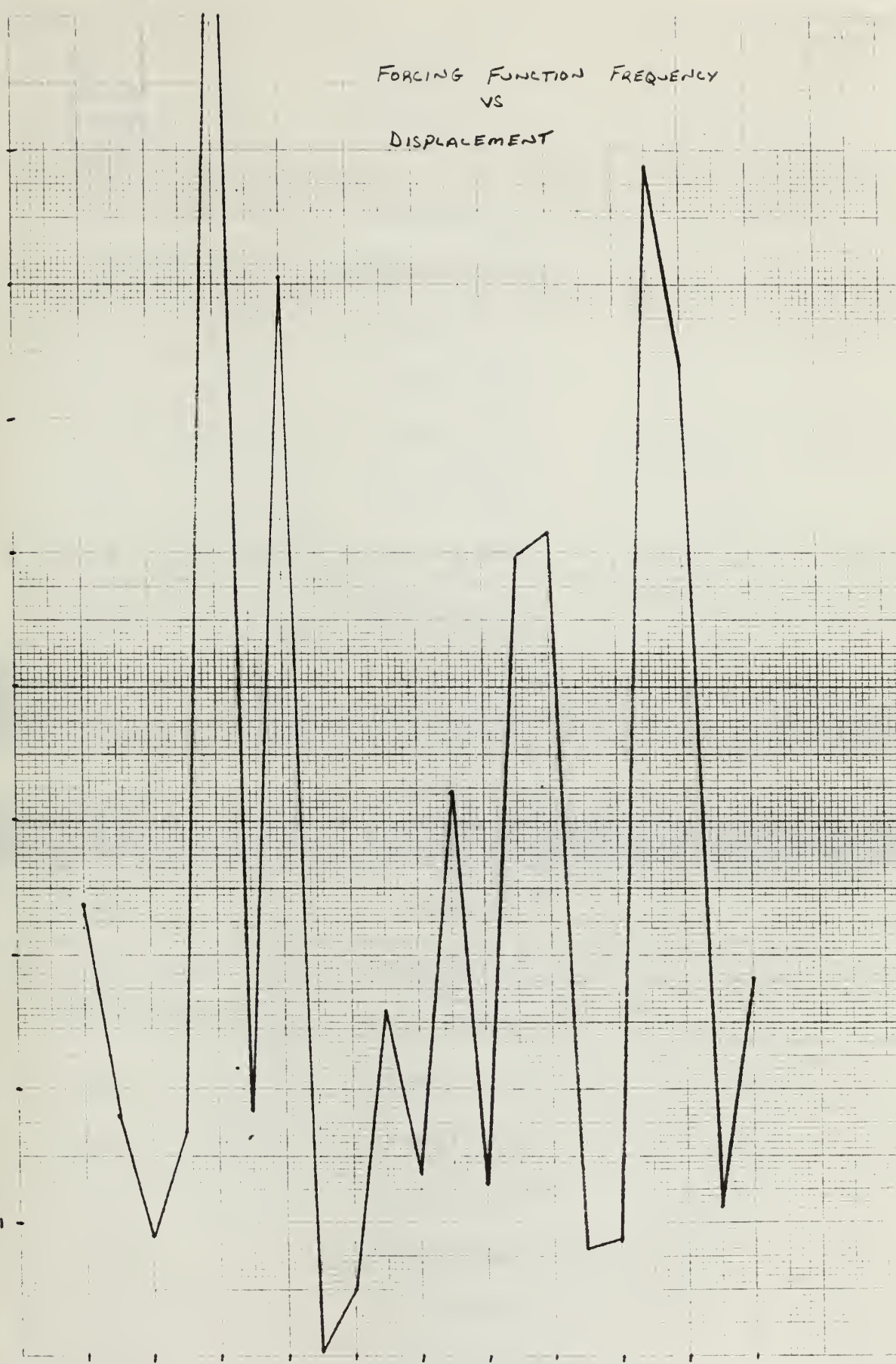
FORCING FUNCTION FREQUENCY (cyc/sec)	FOUNDATION CENTER DISPLACEMENT (inches)
1.0	3.338813×10^{-3}
1.1	1.801127×10^{-3}
1.2	9.005687×10^{-4}
1.3	1.672994×10^{-3}
1.4	3.742432×10^{-2}
1.5	1.840101×10^{-3}
1.6	8.047797×10^{-3}
1.7	3.945689×10^{-5}
1.8	5.197638×10^{-4}
1.9	2.562883×10^{-3}
2.0	1.373975×10^{-3}
2.1	4.226132×10^{-3}
2.2	1.285588×10^{-3}
2.3	5.962294×10^{-3}
2.4	6.135445×10^{-3}
2.5	8.096274×10^{-4}
2.6	8.681937×10^{-4}
2.7	8.828307×10^{-3}
2.8	7.418144×10^{-3}
2.9	1.142477×10^{-3}
3.0	2.810870×10^{-3}

E.4

EIGENVALUES USED FOR THE 2CPS NISA ANALYSIS

NUMBER	RAD/SEC	CYC/SEC	RATIO
******	*******	*******	*****
1	3.990103E+00	6.350446E-01	0.000000E-01
2	5.141340E+00	8.182696E-01	0.000000E-01
3	6.500898E+00	1.034650E+00	0.000000E-01
4	6.730453E+00	1.071185E+00	0.000000E-01
5	8.702491E+00	1.385045E+00	0.000000E-01
6	8.804094E+00	1.401215E+00	0.000000E-01
7	1.017533E+01	1.619454E+00	0.000000E-01
8	1.128642E+01	1.796290E+00	0.000000E-01
9	1.137774E+01	1.810823E+00	0.000000E-01
10	1.183149E+01	1.883040E+00	0.000000E-01
11	1.348654E+01	2.146450E+00	0.000000E-01
12	1.372117E+01	2.183792E+00	0.000000E-01
13	1.384859E+01	2.204072E+00	0.000000E-01
14	1.448125E+01	2.304763E+00	0.000000E-01
15	1.522606E+01	2.423303E+00	0.000000E-01
16	1.545846E+01	2.460290E+00	0.000000E-01
17	1.590396E+01	2.531194E+00	0.000000E-01
18	1.617381E+01	2.574141E+00	0.000000E-01
19	1.697778E+01	2.702097E+00	0.000000E-01
20	1.744101E+01	2.775824E+00	0.000000E-01
21	1.847239E+01	2.939973E+00	0.000000E-01
22	1.874823E+01	2.983873E+00	0.000000E-01
23	1.899211E+01	3.022689E+00	0.000000E-01
24	1.971806E+01	3.138227E+00	0.000000E-01
25	1.977497E+01	3.147284E+00	0.000000E-01
26	2.018043E+01	3.211815E+00	0.000000E-01
27	2.038338E+01	3.244116E+00	0.000000E-01
28	2.100639E+01	3.343270E+00	0.000000E-01
29	2.143819E+01	3.411994E+00	0.000000E-01
30	2.167573E+01	3.449800E+00	0.000000E-01
31	2.212480E+01	3.521272E+00	0.000000E-01
32	2.225399E+01	3.541833E+00	0.000000E-01
33	2.241003E+01	3.566667E+00	0.000000E-01

$$f = 2\,cps$$

NAVFAC / RICHART METHOD w/ GAZETAS K   FOR HAS MASS

$$K_v = \frac{4GR}{1-\nu}\left(1 + 1.28\,\frac{R}{H}\right)$$

$$= \frac{(4)(206000\,psf)(9\,ft)}{(1-0.45)}\left[1 + 1.28\left(\frac{9}{234}\right)\right]$$

$$= 14,147,446.2 \ \#/ft$$

$G = 206000\ psf$
$R = 9'$
$H = 234'$
$\nu = 0.45$

$$f_n = \frac{1}{2\pi}\sqrt{\frac{K_v}{m}}$$

$$= \frac{1}{2\pi}\sqrt{\frac{14147446.2\ \#/ft}{5145.4\ \#\,sec^2/ft}}$$

$$= 8.35 \ cycles/sec$$

W FDN $= 165682\ \#$ $\therefore\ m = 5145.4\ \dfrac{\#\,sec^2}{ft}$

MASS RATIO

$$B_s = \left(\frac{1-\nu}{4}\right)\left(\frac{m}{\rho R^3}\right)$$

$$= \left(\frac{1-0.45}{4}\right)\left(\frac{5145.4\ \#\,sec^2/ft}{(3.88\ \#\,sec^2/ft^4)(9\,ft)^3}\right)$$

$$= 0.25$$

$m = 5145.4\ \#\,sec^2/ft$
$\rho = 3.88\ \#\,sec^2/ft^4$

SHEAR WAVE VELOCITY

$$V_s = \sqrt{\frac{G}{\rho}} = \sqrt{\frac{206000\ \#/ft^2}{3.88\ \#\,sec^2/ft^4}} = 230.4 \ ft/sec$$

$$f_s = \frac{V_s}{4H} = \frac{230.4\ ft/sec}{4(234\,ft)} = 0.246 \ cycles/sec$$

$$f = 2 \ cycles/sec \ > \ f_s$$

ASSUME V_s ROCK $\neq \infty$ $+ \ \dfrac{V_s}{V_R} \simeq 0.5$

E.6

$$f = 2 \, cps \quad \therefore \quad \frac{f}{f_s} = 8.12 \quad \therefore \quad \text{RIGID LAYER HAS LITTLE TO NO}$$

EFFECT + SYSTEM DAMPING

$\approx$ HALF SPACE DAMPING

$$\frac{C_v}{C_{HS}} = 1$$

$\therefore$ DAMPING RATIO $\quad D_z = \dfrac{0.425}{\sqrt{B_z}}$

$$= \frac{0.425}{\sqrt{0.25}} = 0.85$$

MAGNIFICATION FACTOR

$$M = \frac{1}{\left[\left(1-\left(\frac{\Omega}{\omega_n}\right)^2\right)^2 + \left(2(D_z)\left(\frac{f}{f_n}\right)\right)^2\right]^{1/2}}$$

$$= \frac{1}{\sqrt{\left(1-\left(\frac{2}{8.35}\right)^2\right)^2 + \left(\left(2(0.85)\left(\frac{2}{8.35}\right)\right)\right)^2}}$$

$$= 0.974$$

$$A_s = \frac{b}{K_v} = \frac{50,000 \, \#}{14147446.2 \, \#/Ft} = 0.00353 \, Ft$$

$$\delta_{DYN} = A_s \cdot M = (0.00353 \, Ft)(0.974) = 0.00344 \, Ft$$

E.7

```
*                                                    *
*            DYNAMIC ANALYSIS                         *
*         using ' GAZETAS ' METHOD                    *
*      Earthquake engineering vol.2 no1,1983          *
*                                                    *
*                                                    *
*              IBM version 1.0                        *
*                                                    *
*                                                    *
 ****************************************************
```

$f = 2$ cps   FDN HAS MASS

```
 *************************************************
 *           I n p u t  -  d a t a              *
 *************************************************
```

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4)= .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c]   :

HORIZONTAL DYNAMIC STIFFNESS [KH] = 1

VERTICAL DYNAMIC STIFFNESS [KV] = .97

ROCKING DYNAMIC STIFFNESS [KR] = .94

TORSIONAL DYNAMIC STIFFNESS [KT] = .96

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .84

HORIZONTAL DAMPING COEFFICIENT [CH] = .59

ROCKING DAMPING COEFFICIENT [CR] = .11

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

ROCKING MASS OF INERTIA - CG /AXIS X (KIP-SEC^2-FT) = 119.631

TORSION MASS OF INERTIA - CG /AXIS Z (KIP-SEC^2-FT) = 208.388

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 5.145

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 120

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3


```
**************************************************
*          O U T P U T  -  D A T A           *
**************************************************
```

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.78

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.862894E-03 ←

VERTICAL PHASE ANGLE (RAD)= -.4137561

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -4.803612E-02

HORIZONTAL DISPLACEMENT (FT)= 0

HORIZONTAL PHASE ANGLE (RAD)= 1.20483

ROTATIONAL DISPLACEMENT (FT)= 0

ROTATIONAL PHASE ANGLE (RAD)= 1.20483

***** End of Dynamic Analysis *****

```
R(1)  =   9
R(3) =   9
R(5) =   9
R(7) =   9
R(9) =   9
AI(1) =   .2895918
AI(3) =   .4123002
AI(5) =   5.399169E-02
AI(7) =   0
AI(9) =   4.417502E-02
KI(1) =   9753.052
KI(2) =   2824.404
KI(3) =   25387.6
KI(4) =   10791.04
KI(5) =   688816.8
KI(6) =   39564.24
KI(7) =   0
KI(8) =   0
KI(9) =   768891
KI(10) =   35381.02
HR= 8940.587   HI= 2824.404   CR=-29259.16   CI=-8473.211   RR= 757702.9   RI=
   64983.87
NR= 5.806465E+09                NI= 2.225215E+09                V1= 24575.13  V2=
   10791.04      V3= 26839.96   V4= .4137561   V5= 1.862894E-03
PV=-.4137561   O1= 735983.7   O2= 35381.02   O3= 736833.6   O4= 4.803612E-02
      O5= 0      PO=-4.803612E-02
RA= 760484.4   RB= 8.555496E-02              CA= 30461.35   CB= 3.423471  HA=
   9376.106      HB= .3059867
H1= 0          H2= 8.555496E-02              H3= 0          H4= 3.423471  H5= 0
   H6= 0
H7= 0          H8= 0          H9= 0          PH= 1.20483
I1= 0          I2= 0          I3= 0          I4= 1.570796   I5= 6.218248E+09
   I6= .3659669
R1= 0          R2= .3059867   R3= 0          R4= 3.423471   R5= 0          R6= 0
R7= 0          R8= 0          R9= 0
S1= 0          S2= 0          S3= 0          S4= 1.570796
PR= 1.20483
End of internal storage data
```

LYSMER   FOUNDATION HAS MASS

$f = 2 cps$

$a_0 = \omega \dfrac{r_0}{V_s}$ $r_0 = 9'$ $\omega = (2cps)(2\pi\ rad/s) = 12.57\ rad/sec$

$\qquad = (12.57)\left(\dfrac{9\ ft}{230\ ft/sec}\right)$ $V_s = \sqrt{\dfrac{G}{\rho}} = \sqrt{\dfrac{200000\ psf}{288\ \frac{slug}{ft^4}}} = 230\ ft/sec$

$\qquad = 0.49$

$\therefore$ FROM CHART $M = 2.47$

$\delta_{max} = \dfrac{(1-\mu)\ P_0}{4 G\ r_0}\ M = \dfrac{(1-0.45)(50,000\ \#)}{4(200000\ psf)(9\ ft)}\ 2.47$

$\qquad\qquad\qquad = 0.00916\ ft$

$f = 3\ cps$

$\therefore a_0 = (3cps)(2\pi\ rad/s)\left(\dfrac{9\ ft}{230\ ft/sec}\right) = 0.74$

$\therefore$ FROM CHART $M = 0.74$

$\delta_{max} = \dfrac{(1-\mu)\ P_0}{4 G\ r_0}\ M = \dfrac{(1-0.45)(50,000\ \#)}{4(200000\ psf)(9\ ft)}(0.74)$

$\qquad\qquad\qquad = 0.00274\ ft$

E.11
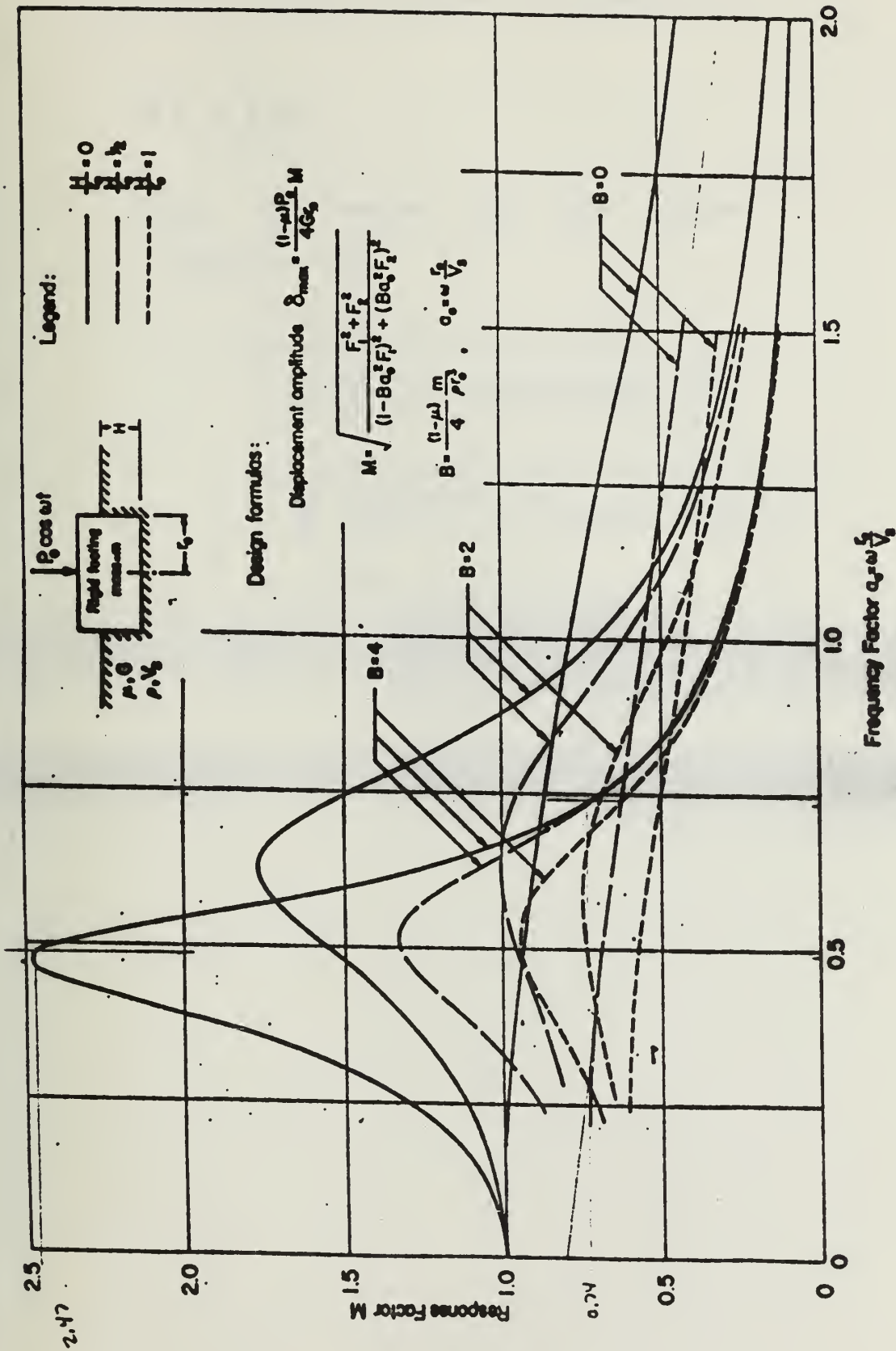
FIG.17 – STEADY STATE SPECTRA FOR EMBEDDED FOOTINGS

P₀ cosωt

NANFAC / RICHART METHOD W/ GAZETAS K

$f = 3$ CPS

SAME AS NANFAC FOR 2 CPS EXCEPT
DIFFERENT m

$$\therefore \quad m = \frac{1}{\left\{\left(\left(1 - \left(\frac{f}{f_n}\right)^2\right)^2 + \left((2)(D_+) \left(\frac{f}{f_n}\right)\right)^2\right]^{1/2}\right.}$$

$$= \frac{1}{\left\{\left(1 - \left(\frac{3}{8.35}\right)^2\right)^2 + \left((2)(0.85) \left(\frac{3}{8.35}\right)\right)^2\right]^{1/2}\right.}$$

$$= 0.94$$

$$\delta = A_s\, m = \frac{P_o}{K_i}\, m = \left(\frac{50\,000}{\text{141730462}}\right)(0.94) = 0.00332 \;\text{ft}$$

```
***************************************************
*                                                 *
*              DYNAMIC ANALYSIS                    *
*           using ' GAZETAS ' METHOD              *
*       Earthquake engineering vol.2 no1.1983     *
*                                                 *
*                                                 *
*              IBM version 1.0                     *
*                                                 *
*                                                 *
***************************************************
```

$f = 3 cps$ FDN HAS MASS

```
**************************************************
*            I n p u t  -  d a t a            *
**************************************************
```

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4)= .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c]   :

HORIZONTAL DYNAMIC STIFFNESS [KH] = .99

VERTICAL DYNAMIC STIFFNESS [KV] = .9

ROCKING DYNAMIC STIFFNESS [KR] = .9

TORSIONAL DYNAMIC STIFFNESS [KT] = .92

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .85

HORIZONTAL DAMPING COEFFICIENT [CH] = .6

ROCKING DAMPING COEFFICIENT .[CR] = .1

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

E.14

TORSION MASS OF INERTIA - CG /AXIS Z (KIP-SEC 2-FT) = 208.388

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 5.145

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 180

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3

****
*   O U T P U T - D A T A   *
****

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.78

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.837587E-03

VERTICAL PHASE ANGLE   (RAD)= -.6459611

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -7.989973E-02

HORIZONTAL DISPLACEMENT (FT)= 0

E.15

ROTATIONAL PHASE ANGLE (RAD)= .9749001

***** End of Dynamic Analysis *****

VS= 230.4187
R(1) = 9
R(3) = 9
R(5) = 9
R(7) = 9
R(9) = 9
AI(1) = .4417502
AI(3) = .6258128
AI(5) = 7.362503E-02
AI(7) = 0
AI(9) = 6.626253E-02
KI(1) = 9655.521
KI(2) = 4308.412
KI(3) = 23555.5
KI(4) = 16379.26
KI(5) = 659505.4
KI(6) = 53951.23
KI(7) = 0
KI(8) = 0
KI(9) = 736853.8
KI(10) = 53071.52
HR= 7827.477 HI= 4308.412 CR=-28966.57 CI=-12925.24 RR= 703899.6 RI=
  92726.94
NR= 4.436252E+09   NI= 3.009706E+09   V1= 21727.45 V2=
  16379.26   V3= 27209.6   V4= .6459611 V5= 1.837587E-03
PV=-.6459611 O1= 662812.5 O2= 53071.52 O3= 664933.8 O4= 7.969973E-02
    O5= 0   PO=-7.989973E-02
RA= 709980.9 RB= .130979   CA= 31719.45 CB= 3.56129   HA= 8934.864 HB=
  .5031669
H1= 0   H2= .130979   H3= 0   H4= 3.56129   H5= 0   H6= 0
H7= 0   H8= 0   H9= 0   PH= .9749001
I1= 0   I2= 0   I3= 0   I4= 1.570796 I5= 5.362501E+09
    I6= .5958963
R1= 0   R2= .5031669 R3= 0   R4= 3.56129   R5= 0   R6= 0
R7= 0   R8= 0   R9= 0
S1= 0   S2= 0   S3= 0   S4= 1.570796
PR= .9749001
End of internal storage data

NAUFAL / RICHART METHOD W/ GAZETAS K $f = 3 \, CPS$

ASSUME FDN HAS O MASS ∴ LET FDN $M = 1 \times 10^{-5}$

FROM PREVIOUS CALC. $K_v = 14147446.2 \, \#/FT$

$$f = \frac{1}{2\pi}\sqrt{\frac{K}{m}} = \frac{1}{2\pi}\sqrt{\frac{14147446.2}{1 \times 10^{-5}}} = 189304 \, CPS$$

MASS RATIO

$$B_z = \left(\frac{1-\nu}{4}\right)\left(\frac{m}{\rho R_0^3}\right) = \left(\frac{1-0.45}{4}\right)\left(\frac{1 \times 10^{-5}}{(3.90)(9)^3}\right) = 4.86 \times 10^{-10}$$

SHEAR WAVE = 230.4 FT/SEC   (FROM BEFORE)

$f_s = 0.246 \, c/s$ (FROM BEFORE)

$f = 3 \, cps > f_s$ ∴ ASSUME $V_{ROCK} \neq \infty$ & $\frac{V_s}{V_R} < 0.15$

$\frac{f}{f_s} = 12.2$ ∴ RIGID LAYER HAS NO EFFECT

$\frac{C_v}{C_{v∞}} = 1$

∴ DAMPING RATIO $= \frac{0.425}{\sqrt{B_z}} = \frac{0.425}{\sqrt{4.86 \times 10^{-10}}} = 19276.0$

$$m = \frac{1}{\left[\left(1-\left(\frac{f}{f_n}\right)^2\right)^2 + \left(2(D_z)\left(\frac{f}{f_n}\right)\right)^2\right]} = \frac{1}{\left[\left(1-\left(\frac{3}{189304}\right)^2\right)^2 + \left((2)(19276)\left(\frac{3}{189304}\right)\right)^2\right]^{1/2}}$$

$= 0.995 = 1$

$$\delta = A_s m = \frac{P_0}{K} m = \left(\frac{50,000}{14147446.2}\right)(1) = 0.00353 \, FT$$

LUCO

MASSLESS FOUNDATION

$f = 3$ CPS $\qquad$ $V = 50,000$ lbs $\qquad$ $\mu = G = 206000$ psf

$a_0 = \omega a (\rho/\mu)^{1/2}$ $\qquad$ $a = 9$ ft $\qquad$ $\rho = 3.88 \frac{\# SL^2}{ft^4}$

$\Delta V = C_W (a_0) V$ $\qquad$ $\gamma = \sigma = 0.45$

$(3 \text{ c/s})(2\pi \text{ Rad/c}) = 18.85$ RADIANS/sec

$a_0 = (18.85 \text{ Rad/sec})(9 \text{ ft}) \left(\dfrac{3.88 \text{ \# sec}^2/ft^4}{206000 \text{ \#/ft}^2} \right)^{1/2} = 0.74$

∴ FROM LUCO CHART

$\dfrac{4}{1-\sigma} \mu a\, C_W (a_0)_{IMAG.} = 0.76 = X_i$

$\dfrac{4}{1-\sigma} \mu a\, C_W (a_0)_{REAL} = 0.53 = X_R$

$$X = \frac{4}{1-\sigma} \mu a\, C_W\, a_0$$

∴ $C_W = \dfrac{X}{\left(\frac{4}{1-\sigma}\right)(\mu)(a)(a_0)}$

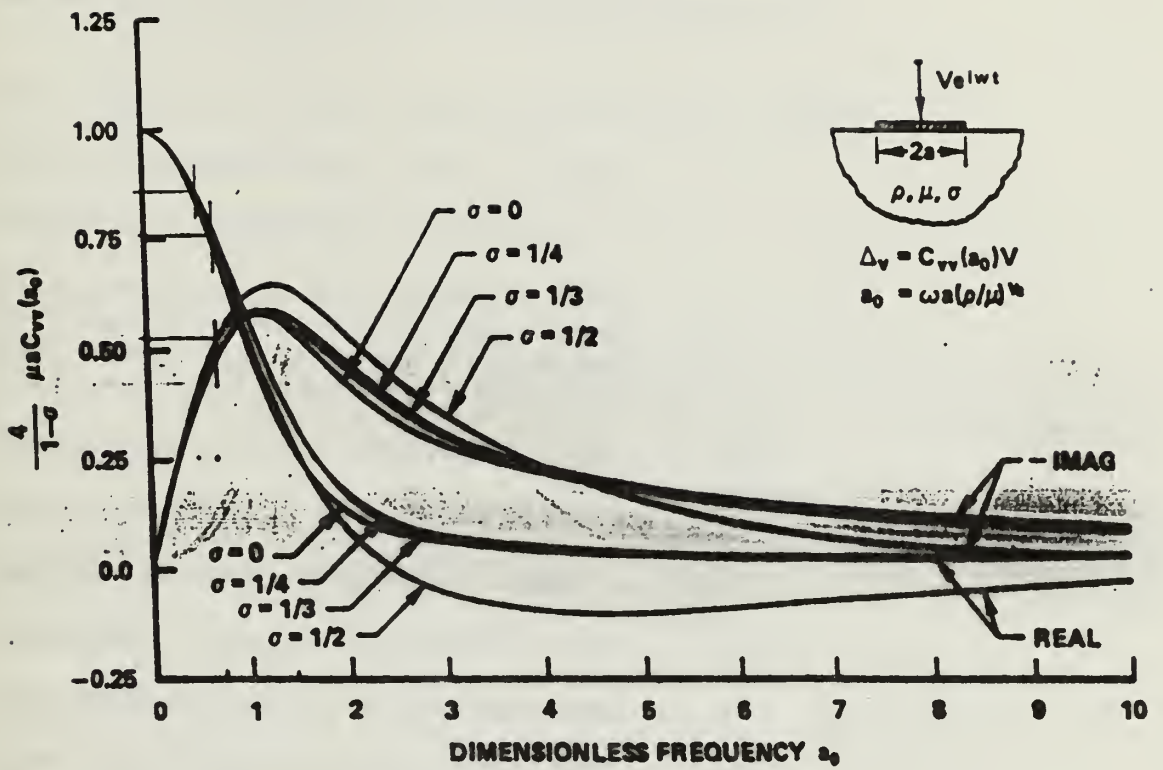∴ $C_W\,_{IMAG} = \dfrac{0.76}{\left(\frac{4}{1-0.45}\right)(206000 \text{ psf})(9 \text{ ft})(0.74)} = 7.617 \times 10^{-8} \text{ ft/\#}$

∴ $C_W\,_{REAL} = \dfrac{0.53}{\left(\frac{4}{1-0.45}\right)(206000 \text{ psf})(9 \text{ ft})(0.74)} = 5.312 \times 10^{-8} \text{ ft/\#}$

E.18

LUCO   MASSLESS FDN   CONT.

$$\Delta V_i = C_{VVi} \; a_o \; V$$

$$= (2.617 \times 10^{-8} \; Ft/\#)(0.74)(50000 \; \#)$$

$$= 0.00282 \; FT$$

$$\Delta V_R = C_{VVR} \; a_o \; V$$

$$= (5.312 \times 10^{-8} \; Ft/\#)(0.74)(50000 \; \#)$$

$$= 0.00197 \; FT$$

$$\Delta V_{TOTAL} = \sqrt{(V_i)^2 + (V_R)^2}$$

$$= 0.00344 \; FT$$

```
**********************************************
*                                            *
*        DYNAMIC ANALYSIS                     *
*      using ' GAZETAS ' METHOD               *
*  Earthquake engineering vol.2 no1.1983      *
*                                             *
*            IBM version 1.0                  *
*                                             *
**********************************************
```

$f = 3 cps$ FDN HAS NO MASS

```
************************************************
*........ I n p u t - d a t a            *
************************************************
```

MASS DENSITY OF SOIL (KIPS-SEC^2/FT^4) = .00388

DEPTH TO BEDROCK [H] (FT) = 234

FOUNDATION EMBEDMENT [D] (FT) = 0

SHEAR MODULUS OF SOIL (KSF) = 206

Data for little [k] and little [c]   :

HORIZONTAL DYNAMIC STIFFNESS [KH] = .99

VERTICAL DYNAMIC STIFFNESS [KV] = .9

ROCKING DYNAMIC STIFFNESS [KR] = .9

TORSIONAL DYNAMIC STIFFNESS [KT] = .92

HORIZONTAL-ROCKING STIFFNESS [KHR] = 1

VERTICAL DAMPING COEFFICIENT [CV] = .85

HORIZONTAL DAMPING COEFFICIENT [CH] = .6

ROCKING DAMPING COEFFICIENT [CR] = .1

HORIZONTAL-ROCKING DAMPING COEFFICIENT [CHR] = 0

TORSIONAL DAMPING COEFFICIENT [CT] = 9.000001E-02

POISSON RATIO = .45

MASS OF FOUNDATION + MACHINERY (KIP-SEC^2/FT) = 0

TORSIONAL EXCITING MOMENT (FT-KIP) = 0

HORIZONTAL EXCITING FORCE (KIPS) = 0

ROCKING EXCITING MOMENT (FT-KIP) = 0

VERTICAL EXCITING FORCE (KIPS) = 50

EQUIVALENT RADIUS OF FOUNDATION (FT) = 9

OPERATING FREQUENCY OF SYSTEM (RPM) = 180

VERTICAL PHASE ANGLE IN DEGREES = 0

HORIZONTAL PHASE ANGLE IN DEGREES = 0

ROCKING PHASE ANGLE IN DEGREES = 0

TORSIONAL PHASE ANGLE IN DEGREES = 0

HYSTERETIC MATERIAL DAMPING (%)= 0

DISTANCE FROM CENTER OF GRAVITY TO BASE OF FOUNDATION (FT)= 3

```
********************************************
*       O U T P U T  -  D A T A        *
********************************************
```

HORIZONTAL STATIC STIFFNESS (KIP/FT)= 9753.052

VERTICAL STATIC STIFFNESS (KIP/FT)= 26172.76

HORIZONTAL-ROCKING STATIC STIFFNESS (KIP/FT)= 0

TORSIONAL STATIC STIFFNESS (KIP/FT)= 800928.1

ROCKING STATIC STIFFNESS (KIP/FT)= 732783.8

VERTICAL DISPLACEMENT (FT)= 1.742741E-03

VERTICAL PHASE ANGLE (RAD)= -.6075967

TORSIONAL DISPLACEMENT (FT)= 0

TORSIONAL PHASE ANGLE (RAD)= -7.190033E-02

HORIZONTAL DISPLACEMENT (FT)= 0

ROTATIONAL PHASE ANGLE   (RAD)=   1.069473

***** End of Dynamic Analysis *****

VS= 230.4187
R(1) = 9
R(3) = 9
R(5) = 9
R(7) = 9
R(9) = 9
AI(1) = .4417502
AI(3) = .6258128
AI(5) = 7.362503E-02
AI(7) = 0
AI(9) = 6.626253E-02
KI(1) = 9655.521
KI(2) = 4308.412
KI(3) = 23555.5
KI(4) = 16379.26
KI(5) = 659505.4
KI(6) = 53951.23
KI(7) = 0
KI(8) = 0
KI(9) = 736853.8
KI(10) = 53071.52
HR= 9655.521   HI= 4308.412   CR=-28966.57   CI=-12925.24   RR= 746405.1 RI=
   92726.94
NR= 6.135424E+09   NI= 3.362348E+09   V1= 23555.5   V2=
   16379.26   V3= 28690.44   V4= .6075967   V5= 1.742741E-03
PV=-.6075967   O1= 736853.8   O2= 53071.52   O3= 738762.6   O4= 7.190033E-02
      O5= 0   PO=-7.190033E-02
RA= 752142.8   RB= .1235981   CA= 31719.45   CB= 3.56129   HA= 10573.15   HB=
   .4196996
H1= 0   H2= .1235981   H3= 0   H4= 3.56129   H5= 0   H6= 0
H7= 0   H8= 0   H9= 0   PH= 1.069473
I1= 0   I2= 0   I3= 0   I4= 1.570796   I5= 6.996343E+09
      I6= .5013234
R1= 0   R2= .4196996   R3= 0   R4= 3.56129   R5= 0   R6= 0
R7= 0   R8= 0   R9= 0
S1= 0   S2= 0   S3= 0   S4= 1.570796
PR= 1.069473
End of internal storage data