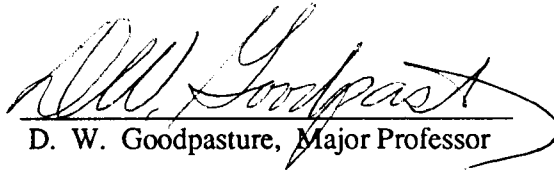
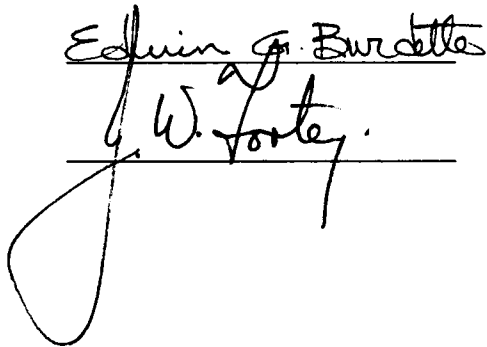


To the Graduate Council:

I am submitting herewith a thesis written by R. Craig Henderson entitled "Analysis and Plotting Capabilities of GTSTRUDL using the CDC Cyber System." I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Civil Engineering.


D. W. Goodpasture, Major Professor

We have read this thesis and
recommend its acceptance:


Edwin G. Burdette
J. W. Fortey

Accepted for the Council:


Vice Provost
and Dean of The Graduate School

STATEMENT OF PERMISSION TO USE

In presenting this thesis in partial fulfillment of the requirements of a Master's degree at the University of Tennessee, Knoxville, I agree that the Library shall make it available to borrowers under rules of the Library. Brief quotations from this thesis are allowable without special permission, provided that accurate acknowledgment of the source is made.

Permission for extensive quotation from or reproduction of this may be granted by my major professor, or in his absence, by the Head of Library Services when, in the opinion of either, the proposed use of the material is for scholarly purposes. Any copying or use of the material in this thesis for financial gain shall not be allowed without my written permission.

Signature R. Craig Henderson

Date 11/23/87

**ANALYSIS AND PLOTTING CAPABILITIES OF GTSTRUDL
USING THE CDC CYBER STSTEM**

A Thesis
Presented for the
Master of Science
Degree
The University of Tennessee, Knoxville

R. Craig Henderson
December 1987

ABSTRACT

The purpose of this manual is to familiarize the GTSTRUDL user with the applicable CIEM and GTSTRUDL procedural formats that are necessary for efficient structural analysis. The user is first introduced to the NOS system upon which GTSTRUDL programs are formulated and executed. Subsequently a breviary of the analysis process from login to completion is discussed and explained. Finally explicit GTSTRUDL instructions on matrix, finite element, and dynamic analyses in addition to structural steel design are included with detailed accompanying examples.

Typically GTSTRUDL, regardless of the specificity of the analysis technique, arranges input data and output requests in consistent sequences from program to program. With this in mind, problem examples in this text, for the purpose of clarity and brevity in descriptions and explanations will be organized in the following format:

STRUCTURAL GEOMETRY
STRUCTURAL COMPONENT CHARACTERISTICS
LOADINGS
ANALYSES
TEXT OUTPUT
GRAPHICAL OUTPUT

Finally, it is the intent of this manual to be supplementary to and in conjunction with GTSTRUDL Vol. 1,2,3, as well as the CCIEM NOS 2 USER'S GUIDE Vol 1,2.

TABLE OF CONTENTS

<u>CHAPTER</u>	<u>PAGE</u>
I. USING NOS	2
Process Files	5
II. CIEM PROCEDURAL FORMAT	10
Login	11
Setting the Screen	11
GTSTRUDL Submission File	12
Creation of TRUSS 1	14
Submit to the Screen	15
Examine TRUSS1 on the Scope	16
Submit to the Printer	17
Submit to the Plotter	18
III. GTSTRUDL PLANAR ANALYSIS	25
Plane Truss	25
Plane Frame	40
Plane Grid	44
IV. FINITE ELEMENT ANALYSIS	49
Mesh Generation	49
Complete Analysis	60
V. SPACE FRAME ANALYSIS	76
Space Frame Example	76
LIST OF REFERENCES	107
VITA	109

LIST OF FIGURES

FIGURE	PAGE
2-1. Calcomp 1043	20
2-2. Media Installation	21
2-3. Control Panel	23
3-1. Truss Geometry	29
3-2. Truss Loading and Dimensions	30
3-3. Coordinate Specifications	33
3-4. Deflected Shape: Joint Loading	41
3-5. Deflected Shape: Member Elongation	42
3-6. Deflected Shape: Support Settlement	43
3-7. Grid Dimensions and Loads	45
4-1. Rectangular Plate Geometry	50
4-2. Skewed Plate with Uniform Mesh	54
4-3. Skewed Plate	58
4-4. Uniform Cylindrical Mesh	62
4-5. Mesh Combination	65
4-6. Window on Fine Mesh	66
5-1. Unbraced Space Frame	77
5-2. Braced Space Frame	80
5-3. Wind Loading [center section]	82
5-4. Deflected Shape due to Wind	85
5-5. Mode #1 [Unbraced]	89
5-6. Mode #2 [Unbraced]	90
5-7. Mode #3 [Unbraced]	91
5-8. Mode #4 [Unbraced]	92
5-9. Mode #5 [Unbraced]	93
5-10. Mode #1 [Braced]	95
5-11. Mode #3 [Braced]	96
5-12. Mode #5 [Braced]	97

LIST OF TABLES

TABLE		PAGE
1-1.	NOS Command Index	3
3-1.	Minimum Member Property Requirements	26
3-2.	Global Degrees of Freedom	27
3-3.	Materials Constants	35
4-1.	Plate Elements	71
5-1.	GTSTRUDL Selection Procedures	103

CHAPTER I

USING NOS

This section consists of commands necessary for the creation, manipulation, and execution of GTSTRUDL data on the NOS 2 operating system for the CDC CYBER 180/830 supermini computer. The acronym, NOS, refers to the Network Operating System, maintained and utilized by the Control Data Corporation for data management on their 180 series computers. Available also is the NOS/VE (Network Operating system/Virtual Environment) which allows for the development of future applications as well as multiple times faster array manipulation and processing. Unfortunately the 1987 version of GTSTRUDL is not formatted for use on NOS/VE hence for the purpose of acquiring this more powerful system, dual state processing will be implemented in the near future.

Table 1.1 includes some essential NOS commands along with their respective explanations and formats. The format sequences listed are for the purpose of showing the user how command words are implemented in the context of actual NOS statements. For this reason, files or instructions following the first command may change as the user deems necessary. For example the change command may be used for other status changes - not just from public to private as listed. Additionally the word file refers to the user's indirect access files.

It should be noted that after a file is created and saved on the CYBER, explained in Chapter 2, it is stored as an indirect access file. For the user to make changes or manipulate the file in any way it must be made local by use of the GET command. For a complete understanding of the NOS system and more detailed explanations of these as well as other useful commands not listed in this section it is necessary to consult the NOS 2 USER'S GUIDE.

TABLE 1.1
NOS Command Index

<i>COMMAND</i>	<i>EXPLANATION</i>	<i>FORMAT</i>
<i>APPEND,</i>	APPENDS INDIRECT ACCESS FILES WITH LOCAL FILES	/APPEND,FILE,lfn
<i>BATCH</i>	INITIATES BATCH MODE	/BATCH
<i>BEGIN,</i>	BEGINS EXECUTION OF JOB	/BEGIN,,FILE
<i>BYE</i>	LOGS OFF SYSTEM	
<i>CATLIST</i>	LISTS INDIRECT FILES	
<i>CHANGE,</i>	CHANGES STATUS OF FILES	/CHANGE,FILE/CT=PU /CHANGE,FILE/CT=PR
<i>COPYSBF,</i>	COPIES FOR ROUTING SEE 'ROUTE'	/COPYSBF,FILE,A
<i>COPYEI,</i>	COPIES FROM FILE1 TO FILE2. (RENAMES)	/COPYEI,FILE1,FILE2
<i>CLEAR</i>	CLEARs ALL LOCAL FILES	
<i>DAYFILE</i>	LISTS THE USER DAYFILE	/DAYFILE
<i>DROP,</i>	DELETES JOB SEQUENCE FROM ACTIVE QUEUE.	/DROP,ABCD.
<i>ENQUIRE,F</i>	LISTS LOCAL FILES	
<i>ENQUIRE,JSN</i>	DISPLAYS JOB STATUS	
<i>FSE,</i>	CREATES OR FINDS A SPECIFIED FILE	/FSE,FILE /FSE,FILE,G (GETS)
<i>GET,</i>	BRINGS FILE INTO LOCAL AREA FROM INDIR- ECT OR OTHER USERS IF PERMITTED.	/GET,FILE

TABLE 1.1 (CONTINUED)

<i>COMMAND</i>	<i>EXPLANATION</i>	<i>FORMAT</i>
<i>HELPME</i>	PROVIDES ONLINE INSTRUCTION	<i>/HELPME</i>
<i>LIST,</i>	LISTS W/OUT HEADING THE CONTENTS OF A FILE	<i>/LIST,FN=FILE</i>
<i>LNH,</i>	LIST WITH NO HEADING	<i>/LNH,FN=file</i>
<i>NULL</i>	PERMITS ABBREVIATIONS	<i>/NULL</i>
<i>PASSWORD,</i>	CHANGES PASSWORD	<i>/PASSWORD,OLD,NEW</i>
<i>PERMIT,</i>	PERMITS OTHERS TO COPY A FILE FROM YOUR AREA	<i>/PERMIT,FILE,USER</i>
<i>PRIMARY,</i>	CONVERTS LOCAL FILES TO	<i>/PRIMARY,file</i>
<i>PURGE,</i>	CLEARs INDIRECT FILES	<i>/PURGE,FILE</i>
<i>R,</i>	RETYPEs LAST FSE COMMAND	
<i>RENAME,</i>	CHANGES THE NAME OF A FILE	<i>/RENAME,NEW=OLD</i>
<i>R,X</i>	RETYPEs LAST FSE COMMAND BEGINNING WITH 'X'	
<i>REPLACE,</i>	REPLACES FILE 1 W/ FILE 2	<i>/REPLACE,NPFILE</i>
<i>RETURN,</i>	CLEARs SPECIFIED LOCAL	<i>/RETURN,NPFILE</i>
<i>REWIND,</i>	PLACE FILE POINTER AT THE FILE BEGINNING	<i>/REWIND,*</i>
<i>ROUTE,</i>	SENDS OUTPUT TO A SPECIFIED DEVICE	<i>ROUTE,A,DC=PR</i>
<i>SAVE,</i>	SAVES COPIED FILES	<i>SAVE,FILE</i>

TABLE 1.1 (CONTINUED)

COMMAND	EXPLANATION	FORMAT
<i>SUBMIT,</i>	SUBMIT JOB FOR EXECUTION	/SUBMIT,FILE
<i>WRITE,</i>	WRITES x FILE ONTO LFN	/WRITE,lfn,x

and routing to the printer. It should be noted that commands five through nine would normally be executed individually which after several sequences becomes avoidably time consuming.

This file is initiated with:

```
/BEGIN,,BOB
```

at which point the computer will prompt the user for the filename to be printed. An alternative to this is to type:

```
/BEGIN,,BOB,FILENAME
```

which carries out the .PROC commands directly.

.PROC #2: SCREEN Submission

```
.PROC,SCRRUN*I,  
FILE[INPUT FILE NAME].  
BEGIN,DATAP,SCRRUN,FILE.  
PROC,DATAP,FILE.  
GET,STRUDL/UN=APPLLIB.  
GET,FILE.  
BEGIN,,STRUDL,I=FILE.  
/EOF
```

Explanation: This PROC. is initiated with the command:

```
/BEGIN,,SCRRUN
```

It is and extremely useful .PROC in that it will become necessary to examine GTSTRUDL output for errors numerous times during the process of analysis

and debugging. Additionally this .PROC may be used in conjunction with others for the purpose of replacing long sequences of routinely used commands.

.PROC #3: PRINTER Submission for GTSTRUDL

```
.PROC,PRRUN*I,  
FILE[INPUT FILE NAME]  
BEGIN,DATAP,PRRUN,FILE.  
.PROC,DATAP,FILE.  
SUBMIT,TWO,TO.  
NOTE,$DO ENQ,JSN= AND QGET,JSN TO CHECK JOB  
.DATA,TWO.  
/JOB  
RUN1,T1500.  
USER.  
GET,STRUDL/UN=APLLIB.  
GET,FILE.  
BEGIN,STRUDL,STRUDL,I=FILE.  
DAYFILE.  
ROUTE,OUTPUT,DC=PR.  
/EOF
```

Explanation: This file's purpose is for the submission of GTSTRUDL data files for print out. The first three lines designate this file as an interactive .PROC job containing input in the form of a file name. The 'SUBMIT' command sends output to the screen by default whereas the last nine lines link the data file with GTSTRUDL, execute, and route to the printer as later defined in Chapter 2 (SUBMISSION FILE).

.Proc # 3: Plotter submission

```
.PROC,PUTPLOT.  
GET,STPLOT.  
GET,UNIPROC/UN=APLLIB.  
BEGIN,UNIPRE,UNIPROC,C=STPLOT.  
REPLACE,NPFILE.  
REPLACE,PUNCH.  
REWIND,*.  
COPYEI,NPFILE,A.  
REWIND,*.  
ROUTE,A,DC=PL,UN=CALCOMP.
```

Explanation: This .PROC file routes the available plot data to the Calcomp plotter subsequent to screen submission. It is important to realize that commands two through four link the user's GTSTRUDL data file with plotting codes from the APPLICATION LIBRARY, thereby creating a neutral plot file(NPFILE). Thus it is necessary to submit to the screen (/BEGIN,,SCRRUN) immediately before the initialization of this .PROC file in order for the above linkage to transpire. Hence the commands:

```
/BEGIN,,SCRRUN  
/BEGIN,,PUTPLOT
```

may be executed concurrently for the purpose of complete screen and plotter submission which ordinarily would require twelve sequential commands repeated numerous times for various necessary plots.

.PROC # 4: File Copy Procedure

```
.PROC,YPOC*I,  
FILE[INPUT FILE NAME].  
INP[INPUT FILE2 NAME].  
GET,FILE.  
REWIND,*.  
COPYEI,FILE,INP.  
REWIND,*.  
SAVE,INP.
```

Explanation: Frequently it is necessary to create and submit GTSTRUDL data files whose content is essentially very similar. Rather than retype a long file it is advisable to merely copy the old file to a new file name upon which the user may edit as desired. The commands necessary for this procedure are listed above within the .PROC called 'YPOC'. After the user acquires this .PROC, the seven commands required for this process may be replaced with the following:

```
/BEGIN,,YPOC,Oldfile,Newfile
```

CHAPTER II

CIEM PROCEDURAL FORMAT

As will become apparent there are several requisite steps for the effective examination and manipulation of GTSTRUDL output. This section is essentially a 'new' user's guide to the CIEM/GTSTRUDL format with an emphasis on the most expedient use of the four available hardware devices including: screen, scope, printer, plotter. For this purpose this section has been segmented into the following procedural arrangement:

- 1) Login
- 2) Set the screen
- 3) Copy or create SUBMISSION FILE
- * 4) Create file TRUSS1
- ** 5) Submit STRUDL1 and TRUSS1 to the screen
- 6) Examine STRUDL1 and TRUSS1 on the scope
- 7) Submit STRUDL1 and TRUSS1 to the printer
- 8) Submit STRUDL1 and TRUSS1 to the plotter

Each of these steps is outlined on the subsequent pages. Although this procedural order is preferred, steps five through eight may be carried out in any order or omitted completely.

* STRUDL1 is intended to be created or copied only once, whereas the other steps are suggested - for complete evaluation - each time a new file (i.e., TRUSS1, 2, 3, etc.) is created.

** All names listed in this manual are for example purposes only. The user is encouraged to be creative and descriptive in the naming of files.

LOGIN

About ten seconds after the VIKING 721 (located along the wall in the third CIEM room) is turned on, a cursor will appear in the upper left corner of the screen. At this time press the carriage return until the computer prompts you for 'USER NAME'. Enter the user name given to you (probably the first seven letters of your last name) and press carriage return. The computer will then prompt you for 'PASSWORD' (seven easily remembered letters or digits) which you will enter and verify. Press carriage return and proceed.

SETTING THE SCREEN

In the CIEM room there are three types of processing equipment, CYBER, PRIME and IBM. However, throughout the duration of GTSTRUDL use, the CYBER system will be utilized for processing and execution. As such, the CYBER operates on a system of commands called NOS which in turn utilize two types of editing techniques, LINE MODE and FULL SCREEN EDITOR (FSE). Either of these techniques is available to the user for the purpose of file editing, however, FSE is a more powerful and expedient method for the creation and manipulation of GTSTRUDL-related input and thus will be focused on in this manual. The command necessary for the initialization of FSE is as follows:

TYPE: /SCREEN,721

If these commands are omitted the computer will reply with '??', indicating line mode has been initiated. If this was unintended, type 'Q', press carriage return, and retype the desired screen commands. The 'T' at the end of the screen commands initiates touch capability (i.e., fingertip movement of the cursor) and, though useful, may be omitted.

GTSTRUDL SUBMISSION FILE

For the execution of GTSTRUDL and examination of the subsequent results, it is necessary to create or copy what will be referred to herein as a submission file. This file, which for explanatory purposes, will be called 'STRUDL1' in the context of this manual, must be submitted simultaneously with the user's GTSTRUDL data file (STEP #4: containing geometry, member properties etc.). The process for creating STRUDL1 is as follows:

TYPE: /FSE,STRUDL1

After typing this command, hit carriage return. A clear workspace will appear upon which the following commands should be typed:

```
/JOB
RUN1,T500
GET,STRUDL/UN=APLLIB
GET,TRUSS1
BEGIN,STRUDL,STRUDL,I=TRUSS1
DAYFILE
ROUTE,OUTPUT,DC=PR
/EOF
```

After this file has been typed in, it is necessary to save this file. To do this, press the 'HOME' key (#5 on the right most keypad of the 721 and F9 at the top of the 4125). This brings the cursor to the top of the screen. Type 'QR' to save this file. The computer will respond with '/' indicating it is ready for the next NOS command.

Instead of creating this file, it is easier to copy from the instructor or APPLLIB. The process for copying from another user's area, after he/she has permitted you to do so, is as follows:

```
TYPE:      /GET,FILENAME/UN=username  
          /COPYEI,FILENAME,STRUDL1  
          /SAVE,STRUDL1
```

After you have completed this process:

```
TYPE:      /CATLIST
```

This produces a list of indirect files on the screen which should include STRUDL1.

The first and last lines of the submission file, as noted by the '/', are NOS commands whose purpose is to inform the computer of the beginning and ending (EOF=end of file) points of this job. The second line sets GTSTRUDL's run-time limit and may be increased as necessary(i.e., RUN1,T1500). Lines three through six are used strictly for the merging and simultaneous execution of the created GTSTRUDL data file (TRUSS1) and the GTSTRUDL code which was obtained from the Application Library (APPLLIB). The sixth line appends the GTSTRUDL output with a listing of all data pertinent to the previous execution. This appended information is referred to as a 'DAYFILE' and may be helpful to the user in the debugging of GTSTRUDL. It should also be noted that lines three through six may be executed in NOS, independently of the other commands in the submission file. This procedure will be discussed later in section 5 :SUBMITTING TO THE SCREEN. Finally, line seven prints all input from the data file (TRUSS1) as well as the analysis and output from GTSTRUDL code on the line printer adjacent to the 721's. It is important to note that lines four and five of STRUDL1 must be changed before every new data file is executed. In other words, TRUSS1 must be changed to TRUSS2 etc.

CREATION OF TRUSS1

For most structural applications, GTSTRUDL has adopted a very flexible format for the input of data into files for processing. Indeed, the pattern of input is so flexible that incorrect data, incorrect requests or both may be entered and processed with no indication of errors in input or output. For this reason as well as for continuity, it is suggested by this author that for repetitive or similar problem descriptions a strict pattern of data arrangement be adhered to.

An example format follows:

TYPE: /FSE,TRUSS1

STRUDL `PROBLEM TITLE` `DESCRIPTION`

TYPE PLANE TRUSS \$ OR FRAME etc.

UNITS KIP FEET

JOINT COORDINATES

\$

\$

\$

\$

MEMBER INCIDENCES

\$

\$

\$

\$

JOINT RELEASES

\$

\$

MEMBER RELEASES \$FOR HINGES

\$

\$

UNITS INCH

CONSTANTS

\$

MEMBER PROPERTIES PRISMATIC

\$

\$

LOADING 1 `ALL LOADS`

JOINT LOADS

\$

```

$
MEMBER LOADS
$
$
STIFFNESS ANALYSIS
SAVE `STEP`
PLOT DEVICE $.....
$
$ ENTER ALL NECESSARY PLOT COMMANDS
$
FINISH

```

As stated earlier, this arrangement of input is very flexible and may be altered as experience deems necessary. However, certain segments of the above are indeed order important and rearrangement of these segments may cause GTSTRUDL errors, incorrect output, and/or inability to plot or print desired output. Additionally, it is a useful and convenient procedure to create the above file, and each time a new truss file is needed, merely copy this to another file name and edit. This expedites subsequent analyses by eliminating repetitive typing.

SUBMIT TO THE SCREEN

This procedure, as alluded to earlier, may be carried out by using three commands within the submission file interactively with the FULL SCREEN EDITOR as follows:

```

TYPE:      /GET,STRUDL/UN=APPLLIB
           /GET,TRUSS1
           /BEGIN,,STRUDL,I=TRUSS1

```

These commands procure GTSTRUDL from the APPLICATIONS LIBRARY, link it with the previously created TRUSS1, and submit the output to the screen by default since no other device is specified.

EXAMINE TRUSS1 ON THE SCOPE

Before attempting to obtain the printout of results it is advisable to examine the structure for errors in geometry and topology. Although the line printer is capable of printing the structure's geometry, it is of poor quality and virtually useless for close inspection. Therefore, it is advisable to view the structure on the scope. This is possible on the three Tectronix 4125 computers. The procedure for scope viewing is as follows:

1) LOGIN ON 4125 (SEE STEP 1)

2) SUBMIT TO SCREEN (SEE STEP 5)

After the computer has listed data and output on the screen it will, as usual, prompt the user with '/'. At this point the procedure is as follows:

3) TYPE: /BEGIN,STRUDL,STRUDL

The computer will respond with '??'.

4) TYPE: ?? RESTORE 'STEP'

'Step' is stored in TRUSS1 (see step 4). After restoration, the computer will respond with '?'.

5) TYPE: ? PLOT DEVICE SCOPE 4114

6) TYPE: ? PLOT PROJECTION

At this juncture, there are several commands available for viewing the structure. These commands, and an explanation of their function follow:

a) ? ROTATE x 45

ROTATES THE STRUCTURE 45 DEGREES;
WHERE X MAY BE REPLACED BY Y OR Z AND
45 MAY BE REPLACED WITH +/- 0 TO 360

b) ? PLOT PROJ DEFORMED STRUCTURE LOADING1

c) ? PLOT FORMAT BOUNDARY SHRIN
? AGAIN BD

PLOTS FINITE ELEMENTS MESH

d) ? LABEL ALL

E) ? WINDOW

THIS COMMAND IS USED FOR ENLARGE AND
VIEWING INTRICATE SECTIONS OF A
STRUCTURE. AFTER THE WINDOW
COMMAND, CROSSHAIRS APPEAR ON THE
SCREEN. MOVE THESE TO THE LOWER LEFT
CORNER OF THE DESIRED VIEWING AREA
AND PRESS 'P'. ANOTHER SET OF CROSSHAIRS
WILL APPEAR. MOVE THESE TO THE UPPER
RIGHT CORNER OF THE DESIRED VIEWING AREA
AND PRESS 'P'. THIS PROCEDURE WILL
PLOT THE DESIRED SECTION IN THE
WINDOW.

SUBMIT TO THE PRINTER

The components of this procedure have been described earlier in
sections three and four. The command carried out in NOS is as follows:

TYPE: /SUBMIT,STRUDL1,TRUSS1

As mentioned previously, the user must be sure to change the name of the data file within STRUDL1 (on lines four and five) each time a new structure is to be analyzed. The output for this submission will be sent to the printer adjacent to the 721 terminals.

SUBMIT TO THE PLOTTER

The procedure for hardcopy plotting of a structure's geometry require's three pieces of computer hardware:

- 1) Viking 721 - ROOM #3 (REFERRED TO HEREIN AS T3)
- 2) Viking 721 - ROOM #2(REFERRED TO HEREIN AS T2)
- 3) CALCOMP PLOTTER.

In general the user will run and route TRUSS1 from T3, it will be received and routed to the plotter by T2. This intermediate step (i.e., the use of T2) is necessary because plotting is carried out by a direct linkage between the control terminal and the Calcomp plotter. The GTSTRUDL neutral file is initiated using the command PLOT DEVICE PLOTTER NEUTRAL at the end of TRUSS1. This neutral file is converted into a plot a FORTRAN 77 program called STPLOT which is compiled and linked to the installation Plotter library. STPLOT may be transferred from page 2.7.1-19 of GTSTRUDL VOL. #3, however, a much more efficient method would be to copy this from the instructor or APPLLIB. This and subsequent commands follow:

i) TYPE: /GET,STPLOT/UN=username
 /COPYE1,STPLOT,STPLOT
 /SAVE,STPLOT
 /CATLIST

ii) TYPE: /CLEAR

This command deletes all local files including any previous NEUTRAL PLOT FILES(NPFILES). This is necessary after the creation of the first NPFILE because each subsequent creation will add to the NPFILE - not replace it.

iii) Submit to the screen as per STEP #5

iv) TYPE: /GET,STPLOT
 /GET,UNIPROC/UN=APPLLIB
 /BEGIN,UNIPRE,UNIPROC,C=STPLOT

T3 should respond with '/REVERT. UNIPOST PROCEDURE COMPLETE'. This sequence of commands produces the NPFILE and places it in the user's local area. To verify this

TYPE: /ENQUIRE,F

which produces a listing of all local files, one of which should be NPFILE.

v) At this point it is necessary to prepare T2 to receive information from T3. The preparation process for T2, usually carried out by a CIEM employee, is as follows:

- a) TURN ON THE CALCOMP PLOTTER. (See figure 2.1 and 2.2)
 [Calcomp User's Guide].
- b) RAISE COVER AND PLACE PAPER UNDER PINCH
 ROLLERS SUCH THAT BOTTOM EDGE IS ALIGNED
 WITH THE DOTTED LINE ON THE DRUM.

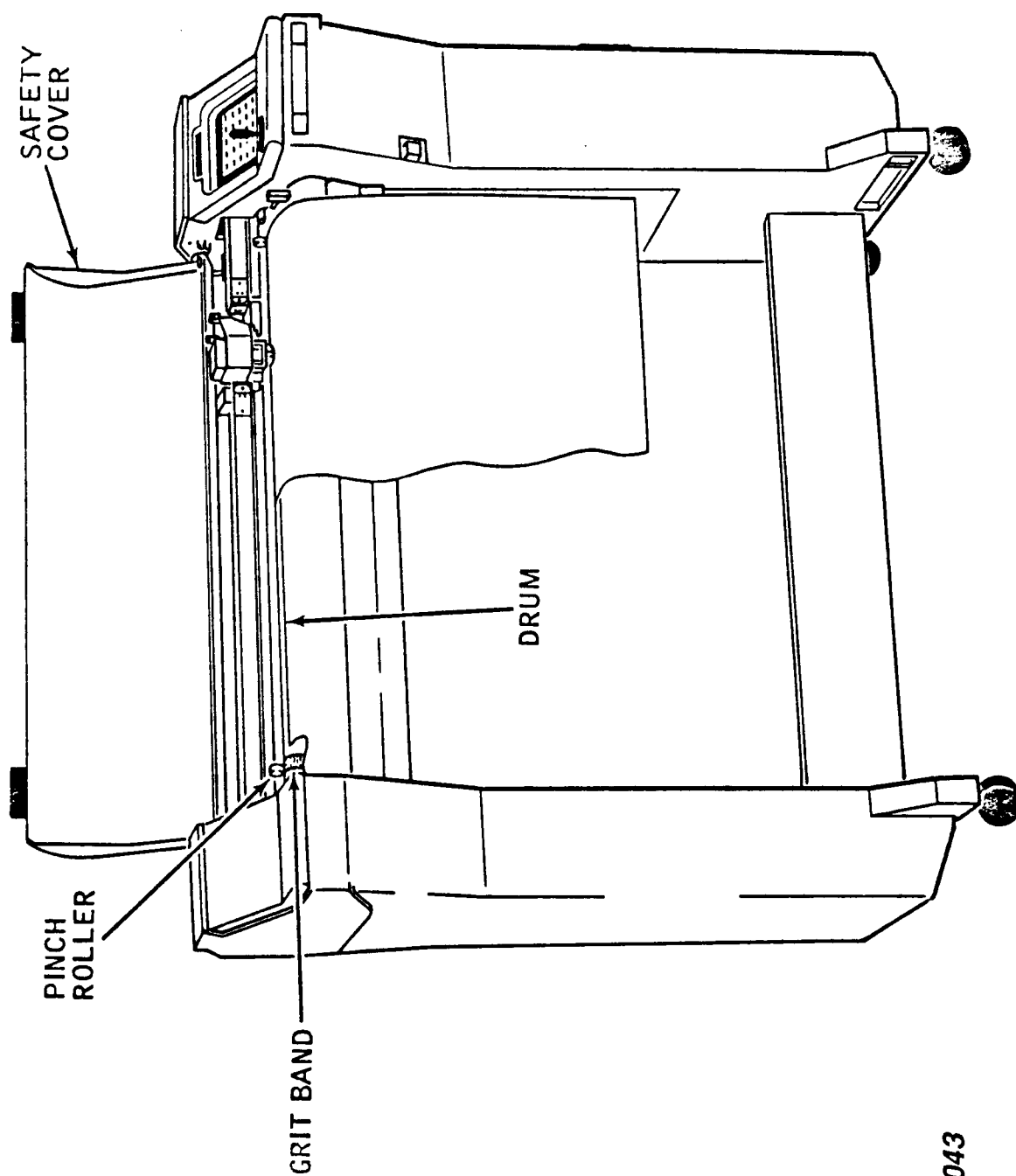


FIGURE 2.1: Calcomp 1043

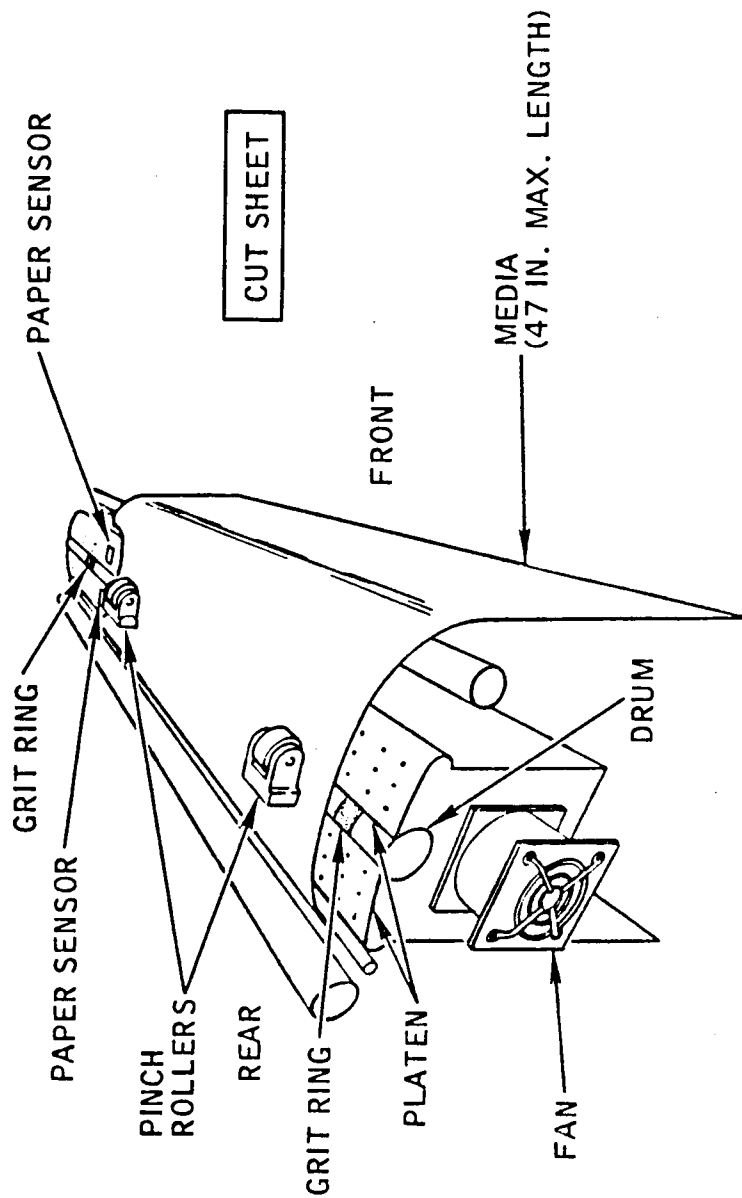


FIGURE 2-2. Media Installation

c) LOWER COVER AND PRESS THE LOAD PAPER
BUTTON.

d) TURN ON T2. WHEN T2 REQUESTS USER NAME AND
PASSWORD

TYPE: CALCOMP
 CALCOMP

vi) After the plotter and T2 have been readied to receive
and assimilate plot data, it is necessary to route this information from T3.
To do this:

TYPE: /REPLACE,NPFILE
 /REWIND,*
 /COPYEI,NPFILE,A
 /REWIND,A
 /ROUTE,A,DC=PL,UN=CALCOMP

When completed T3 should assign a job name to the user's plot job (i.e. ABCD
ROUTED).

vii) At this point the message display on the Calcomp control panel
should say 'PRESS AUTO TO PLOT'. (See Figure 2.3) [Calcomp User's Guide] It is
necessary to determine whether T2 has received its instruction from T3.
To do so

TYPE: /ENQUIRE,JSN on T2

This informs the user of all the active jobs on T2 and their status. The above
routed job should be among these listed (i.e., ABCD PLOT QUEUE. To start the
actual plotting process:

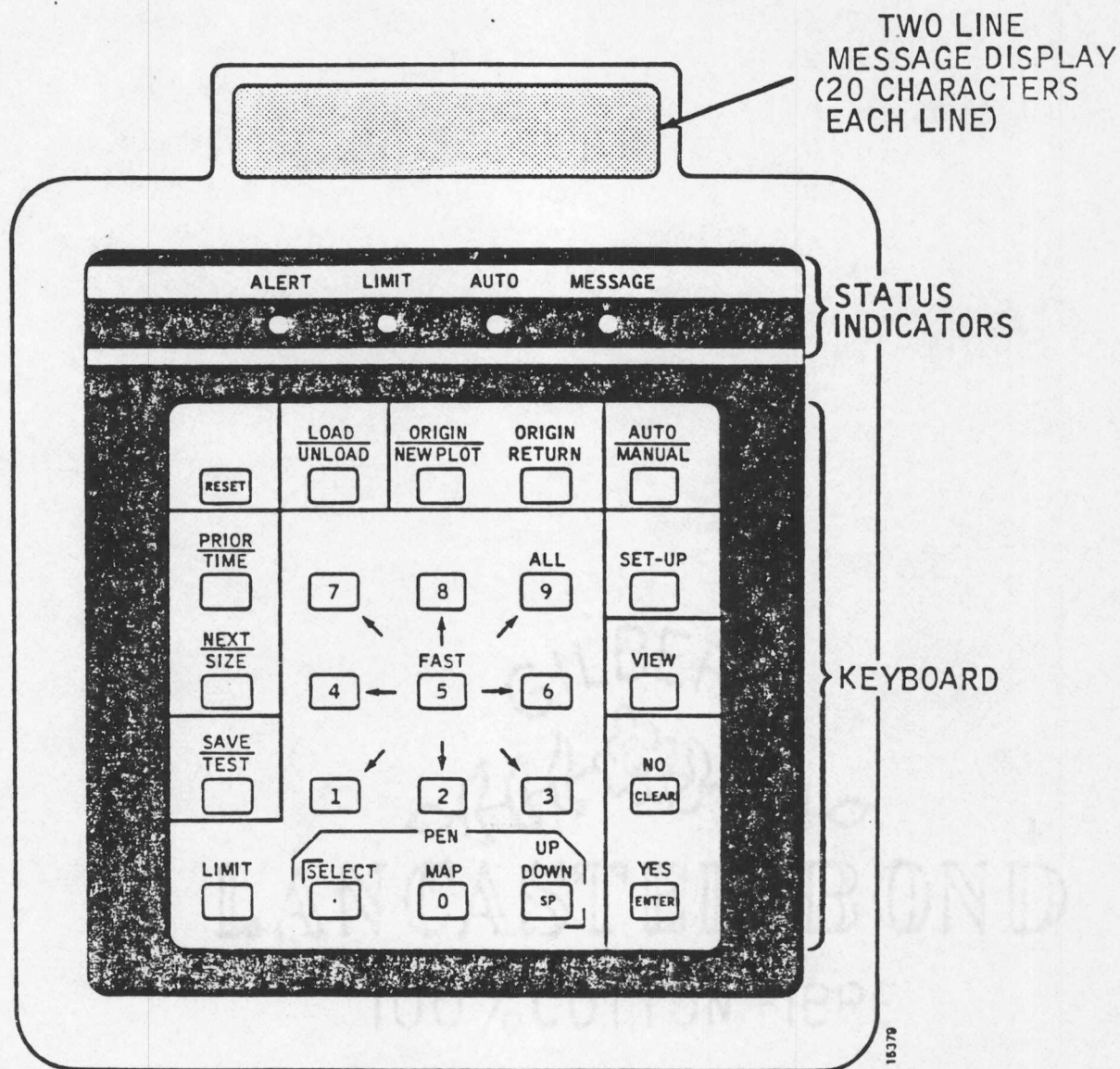


FIGURE 2-3. Control Panel

TYPE: /BEGIN,,PLOTS

This FSE command executes 'PLOTS' which is a process file containing all necessary information for the transfer of plot data to CALCOMP. Immediately after typing the above and pressing carriage return, press AUTO on the plotter. At this time plotting should begin.

CHAPTER III

GTSTRUDL PLANAR ANALYSIS

The term 'planar analysis' in GTSTRUDL refers to any structure whose member geometry and topology is located within any one of the three global planes XY, XZ, YZ. (For the purpose of this discussion the terms geometry and topology are associated with the structure's physical shape and joint connectivity respectively.) However, GTSTRUDL requires that this definition be categorized further using the TYPE command which distinguishes between various member characteristics by establishing the number of kinematic degrees of freedom. It is important to note that the TYPE command must be given prior to topology specification and classifies individual members only, as opposed to entire structures. Finally this section includes discussions of the three types of planar structures PLANE TRUSS, PLANE FRAME, and PLANE GRID as well as correlating examples of each.

PLANE TRUSS

This particular structure type is defined by GTSTRUDL as one in which the joining of members is established via pinned connections thus limiting the kinematic degrees of freedom to forces in the plane of the structure. In other words the PLANE TRUSS responds only to forces applied in the plane of the structure, disregarding without exception externally applied moments. Additionally due to the theoretically infinite rotational capacity of the pinned connections the structure deforms to equilibrium such that only axial member forces are produced without internal moments and shears. For a list of minimum member property requirements and relevant global degrees of freedom see Tables 3.1, 3.2 [GTSTRUDL VOL 1].

TABLE 3-1. Minimum Member Property Requirements

Member TYPE	Member Parallel to Global Plane	Direction of Member Local x-axis	BETA ANGLE (Deg.)	Minimum Required Properties	
				Area	Inertia
PLANE TRUSS	any global plane	any	N/A	AX	-
PLANE FRAME	XY plane	any	0	AX	IZ
			+90 -90	AX	IY
	XY plane	any	0	AX	IY
			+90 -90	AX	IZ
	YZ plane	parallel to Global Y	0	AX	IY
			+90 -90	AX	IZ
		not parallel to Global Y	0	AX	IZ
			+90 -90	AX	IY
			0	-	IX IY
			+90 -90	-	IX IZ
PLANE GRID	XY plane	any	0	-	IX IY
			+90 -90	-	IX IZ
	XZ plane	any	0	-	IX IZ
			+90 -90	-	IX IY
	YZ plane	parallel to Global Y	0	-	IX IZ
			+90 -90	-	IX IY
		not parallel to Global Y	0	-	IX IY
			+90 -90	-	IX IZ
			0	-	IX IY
			+90 -90	-	IX IZ
SPACE TRUSS	N/A	N/A	N/A	AX	-
SPACE FRAME	N/A	N/A	any	AX	IX IY IZ

TABLE 3-2. *Global Degrees of Freedom*

Incident Element Types	Element Parallel to Global Plane or	Relevant Joint Global or Plannar** Degrees-of-Freedom (which may be Released)
PLANE TRUSS	XY plane XZ plane YZ plane	FORCE X, Y FORCE X, Z FORCE Y, Z
PLANE FRAME	XY plane XZ plane YZ plane	FORCE X, Y MOMENT Z FORCE X, Z MOMENT Y FORCE Y, Z MOMENT X
PLANE GRID	XZ plane XZ plane YZ plane	FORCE Y MOMENT X, Z FORCE Y MOMENT X, Z FORCE X MOMENT Y, Z
SPACE TRUSS	any plane	FORCE X, Y, Z
SPACE FRAME	any plane	FORCE X, Y, Z MOMENT X, Y, Z
PLANE STRESS	XY plane	FORCE X, Y or X_p, Y_p^{**}
PLANE STRAIN	XZ plane	FORCE Y, Z or X_p, Y_p^{**}
	YZ plane	FORCE Y, Z or X_p, Y_p^{**}
	any other plane*	FORCE X, Y, Z or X_p, Y_p^{**}

EXAMPLE # 3 a & b: Plane Truss [Frame]

STRUDL ` SPHERICALLY GENERATED PLANE TRUSS'

UNITS KIP FEET

TYPE PLANE TRUSS [FRAME]

\$

\$ **STRUCTURAL GEOMETRY**

\$

\$ SEE FIGURE 3.1 AND 3.2

\$

\$ INPUT THE BOTTOM JOINTS IN SPHERICAL COORD.

\$

JOINT COORD SPHER

1 15 -90 90

7 15 90 90

8 7.5 90 90

9 0 0 90

10 7.5 -90 90

\$

\$ GENERATE THE SPHERICAL JOINTS AND MEMBERS

\$

GENERATE BETWEEN SPHER JOIN 1 7

XDIR 6 PARTS EQUAL

GENERATE 6 MEMB ID 1 1 FROM 1 1 TO 2 1

\$

\$ GENERATE THE BOTTOM MEMBERS

\$

GENE 3 MEMB ID 7 1 FROM 7 1 TO 8 1

\$\$INPUT THE DIAGONALS

\$

MEMB INCID

10 10 1

11 2 10

12 3 10

13 4 10

14 4 9

15 4 8

16 5 8

17 6 8

\$

\$ **STRUCTURAL COMPONENT CHARACTERISTICS**

\$

CONSTANTS

MATER STEEL

UNITS INCH

MEMB PROP PRISMATIC

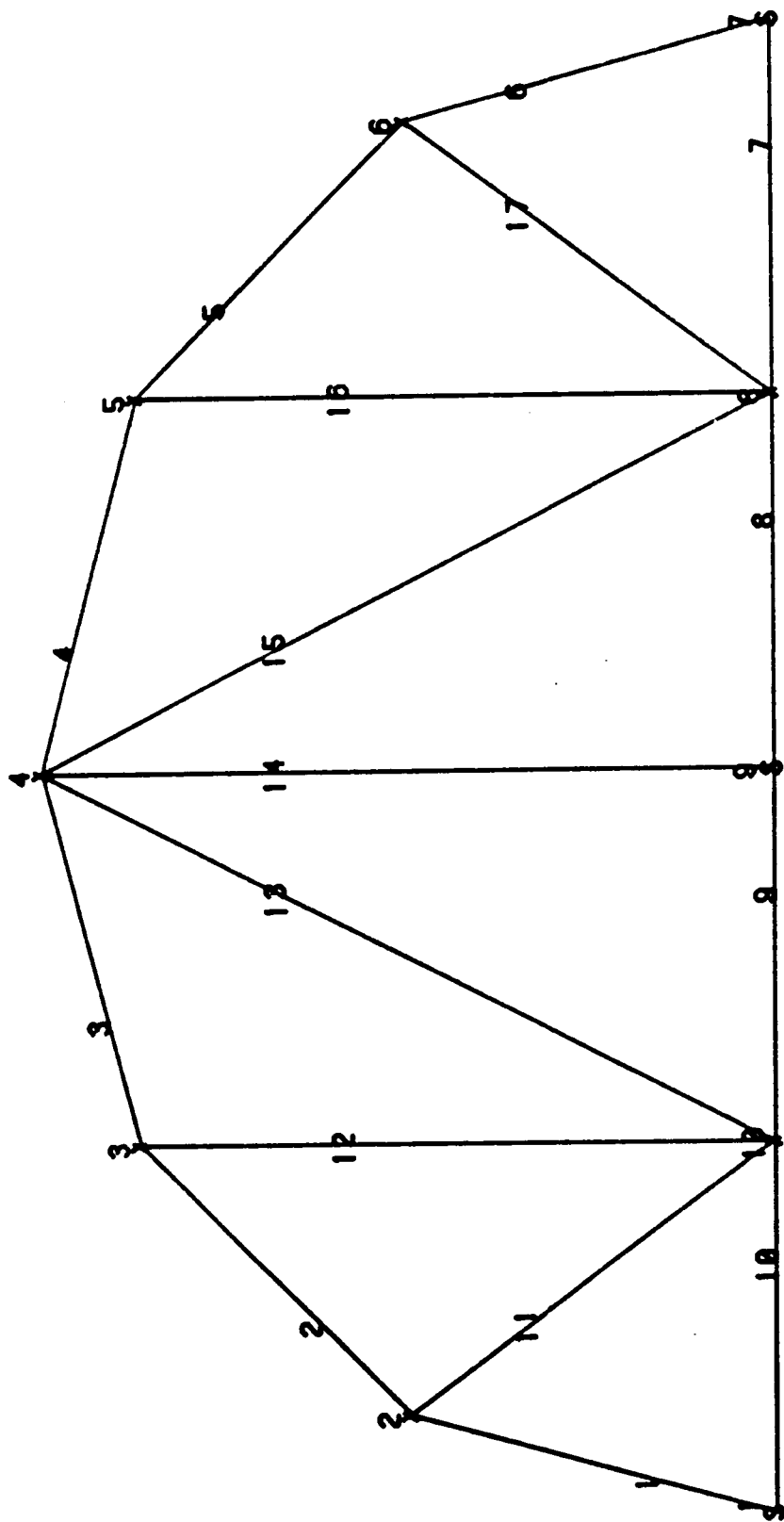


FIGURE 3-1. Truss Geometry

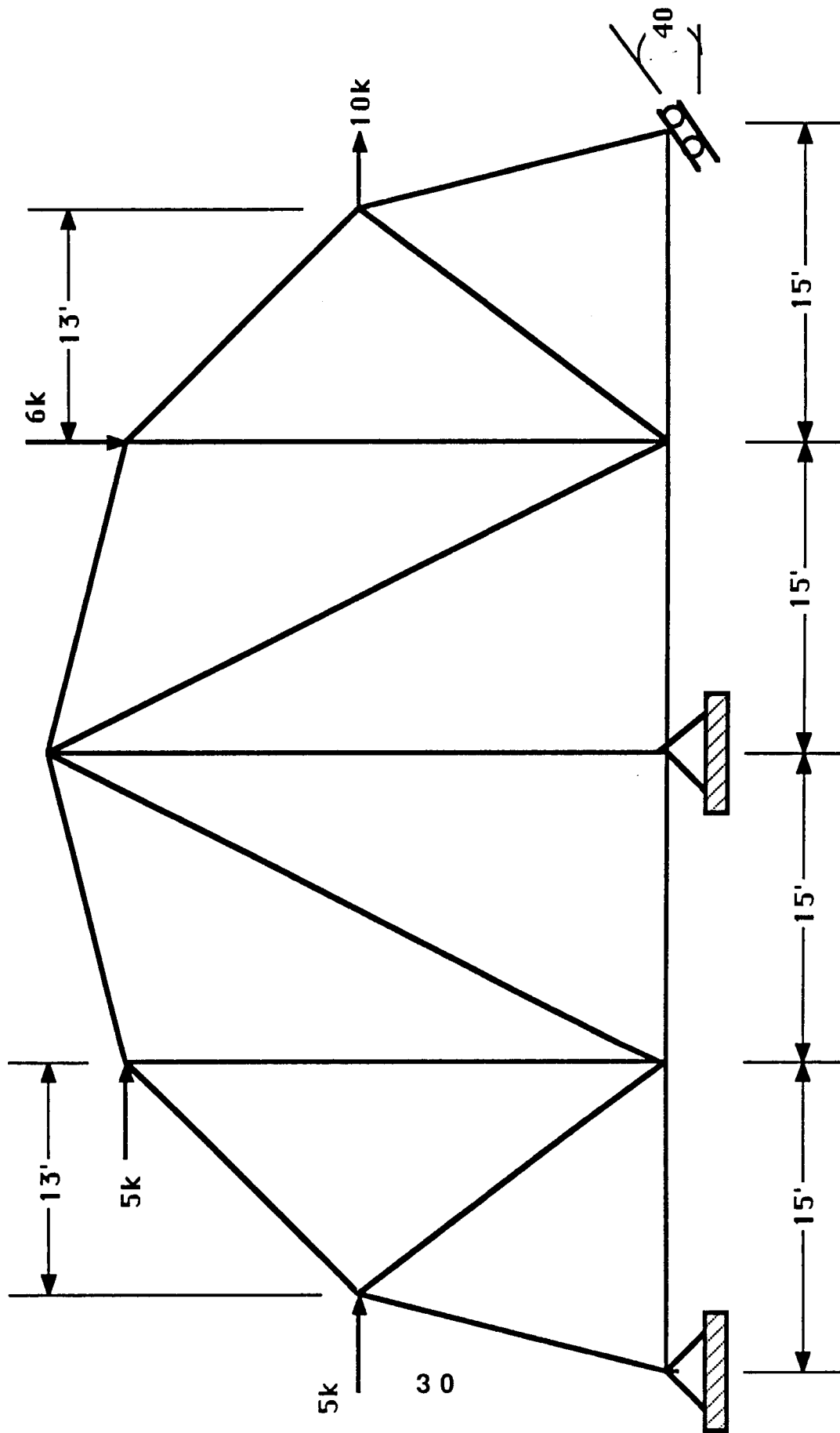


FIGURE 3-2. Truss Loading and Dimensions

```

1 TO 17 AX 4 [IZ 100]
UNIT FEET
STATUS S 1 7 9
JOINT RELEASES
[1 MOM Z]
7 FORCE X [MOM Z] TH1 40
[9 MOM Z]
$
$          LOADING
$
$      INPUT THE FOUR LOADINGS:      1) JOINT FORCE
$                                     2) MEMBER ELONGATION
$                                     3) SUPP. SETTLEMENT
$                                     4) COMBINATION
$
LOAD 1 'JOINT FORCE'
JOINT LOAD
2 3 FOR X 5
5 FOR Y -6
6 FOR X 10
$
LOAD 2 'MEMBER ELONGATION'
MEMB DIST
16 CONC FR L 1.0 DISP 0.25
$
UNIT INCH
$
LOAD 3 'SUPPORT SETTLEMENT'
JOINT DISP
1 DISP Y -0.5
$
$      COMBINE SUPPORT SETTLEMENT WITH A MAG-
$      NIFIED MEMBER ELONGATION (FACTOR = 1.5)
$
LOAD COMB 4 'SETTLE AND ELONGATE' COMB 2 1 3 1.5
$
LOAD LIST ALL
$
$          ANALYSIS
$
$      STIFFN ANA
$
$          TEXT OUTPUT
$
$      LIST REACTIONS FORCES DISP ALL
$      SAVE 'TRUSS'
$

```

```

$          GRAPHICAL  OUTPUT
$
PLOT DEVICE PLOTTER NEUTRAL LENGTH 8 WIDTH 8
PLOT FORMAT OMIT JOINT NAMES
PLOT FORMAT SIZE CHAR 2
PLOT PLANE DEFORM STRUC LOAD ALL
OVERLAY
FINISH

```

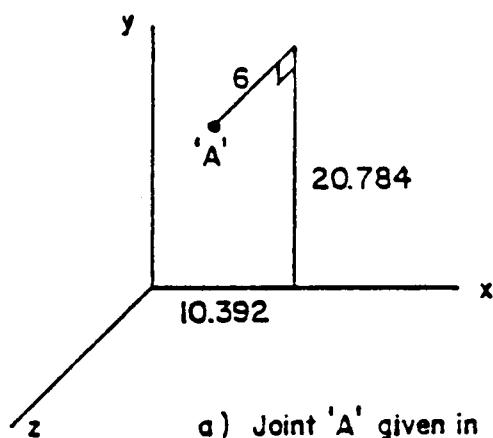
NOTE: TEXT IN BRACKETS [] ARE THE INCLUSIONS NECESSARY
FOR FRAME ANALYSIS.

STRUCTURAL GEOMETRY Typically GTSTRUDL geometry data may be entered by one or, more frequently, a combination of the following ways:

- 1) Input of joint coordinates and member incidences
- 2) Automatic GTSTRUDL generation.

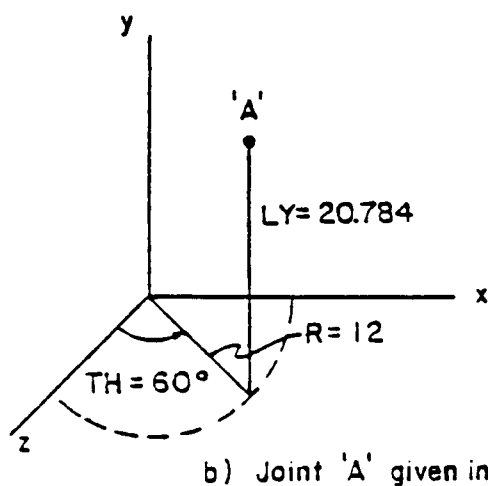
For this purpose GTSTRUDL employs three coordinate systems- Cartesian, Spherical, and Cylindrical- for the complete description of joints, members, and elements in planar or three dimensional settings (See figure 3.3 coordinate specifications) [GTSTRUDL VOL. 1]. Throughout the scope of this text the three systems are utilized and described for both the coordinate input and generation techniques of geometry formulation.

In the above TRUSS example spherical joint data in conjunction with GTSTRUDL generation was used for geometry and topology input. (see FIGURE 3.1) First the joint coordinates for the bottom of the truss were entered spherically by describing each point with a radius, angle from the Y-AXIS, and angle from the Z-AXIS. Notice that all angles from the Z-AXIS are 90 degrees, hence the truss is located in the XY plane. As noted above all planar structures must be located in one of the three global planes. Next, the Generate Between command was used to spherically generate the five joints at the top of the truss and subsequently the six members that form their connectivity. Notice that generation originates at member one and is incremented by one for subsequent members from joints 1 increment 1 to 2 increment 1 etc. Almost invariably when using any type of generation



JOINT COORDINATES CARTESIAN
'A' 10.392 20.784 6

$x = 10.392$
 $y = 20.784$
 $z = 6.0$

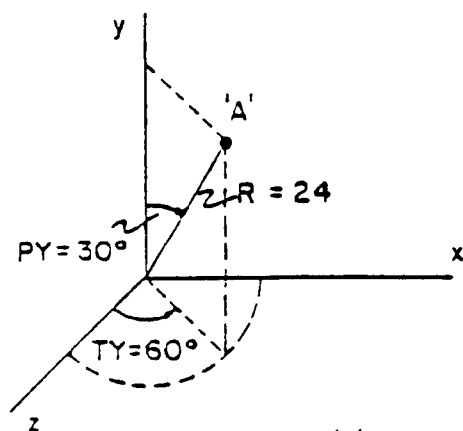


UNITS DEGREES

JOINT COORDINATES CYLINDRICAL

'A' 12 60 20.784

$R = 12.0$
 $TH = 60^\circ$
 $LY = 20.784$



UNITS DEGREES

JOINT COORDINATES SPHERICAL

'A' 24 30 60

$R = 24.0$
 $PY = 30^\circ$
 $TY = 60^\circ$

FIGURE 3-3. *Coordinate Specifications*

technique a member, joint or element within the generation command will be followed by an increment necessary to define the following member, joint or element.

Finally, the generate command was used to generate three of the four members on the bottom of the truss. The member incidences command was used to input the truss diagonals as well as the left-most, bottom member due to the erratic nature of their connectivity, topology, and numbering. This process attests to the fact that, in general, generation is the preferable, expedient choice for topology input on those structures and parts of structures whose geometry is uniform.

STRUCTURAL COMPONENT CHARACTERISTICS: After the structural geometry has been input in its entirety, it is then necessary to establish the desired structural behavior by enumerating certain limiting characteristics. These fall under the categories constants, member properties, status, and releases. It is the intent of the constants command to assign material properties to the structural members and elements. This may be accomplished via individually listing numerical values for specified constants (modulus of elasticity, density etc.) or by naming a particular material such as aluminum, concrete, steel (see Table 3.3) [GTSTRUDL VOL 1]. Likewise, the member properties command describes the size and shape of the member by specifying the cross sectional area (and moment of inertia for all member types- other than plane truss). For a complete listing of minimum member property requirements see Table 3.2. The Status command allows for the establishment of supports after the specification of structural geometry, usually by generation, has taken place. Normally, when the generation technique has not been utilized, the input of structural geometry and the prescribing of joints as free or support is performed concurrently, however, this is not a requirement. Note that when a joint is established as a support it is completely fixed against rotation and translation in frame type structures and against translation only in truss type structures.

TABLE 3.3: *Materials Constants*

Material	E (psi)	G (psi)	Poisson's Ratio	Density (pci)	CTE in/in/°f
ALUMINUM	10×10^6	3.75×10^6	0.33	0.098	1.3×10^{-5}
CONCRETE	3.6×10^6	1.44×10^6	0.17	0.0868	5.5×10^{-6}
STEEL	29×10^6	11×10^6	0.30	0.2833	6.5×10^{-6}

Finally the last component characteristic is the Release command of which there are two types: joint and member. The Joint Releases command modifies the status command by eliminating the unnecessary restrictions on rotation and/or translation of a joint. Similarly the member releases command eliminates the force or moment restrictions on the start and/or end of a member as deemed necessary by the analyst. The release of the Z-direction moment for the production of hinged members is a frequently used example of member release.

In the above Truss example, three joints-one, seven and nine- have been designated by the status command as supports, and thus immediately after GTSTRUDL's assimilation of their status they are deemed unable to translate. Hence it is necessary, if rotation or translation is desired, to release their fixity in the direction in which movement is required. This release has been accomplished by way of the joint releases command in which joints one and nine have been denoted 'pinned connections' by virtue of their inability to resist moment in the Z-direction. Likewise, joint seven has been deemed 'roller connected' at an angle of 40 degrees by the release of the force x and moment z at an angle, theta 1, equal to 40. The Theta angle refers to the inclination of the roller connection within any one of the three globally stipulated planes as follows:

- A) TH1: XY PLANE about the Z-AXIS
- B) TH2: XZ PLANE about the Y-AXIS
- C) TH3: YZ PLANE about the X-AXIS.

LOADING: GTSTRUDL has attempted to provide the engineer with a diverse range of internally and externally applied loads and loading combinations. In this section linear static loading (as opposed to linear dynamic loading-discussed in Section 5: Space Frame Analysis) will be discussed and various essential details inherent in the loading process, and in reference to the above planar truss analysis, will be explained.

Two fundamental loading conditions, independent and dependent, are available to the user for the delineation of loading types, locations and magnitudes. The first of these, independent, includes the following seven statically applied loading types:

- 1) Joint loads
- 2) Joint displacements
- 3) Member loads
- 4) member temperature loads
- 5) Member distortions load
- 6) Joint temperature load
- 7) Element loads.

The second of these loading conditions, dependent, allows the combination of any above dependent loading types in addition to an associated magnification factor. In the above Truss example, both loading conditions, dependent and independent, as well as three of the seven loading types have been included for the purpose of linear static analysis. Loading one, joint loads, probably the most frequently used but least realistic of the seven loading types, exemplifies the application of negative and positive planar forces on joints two, three, five and six.

Loading two, the elongation of member sixteen, is an extremely realistic loading type in that, often, fabrication errors produce members with dimensions longer or shorter than originally designed for thus creating deflections and ultimately loads in the remainder of the structure. The format requirements for the input of these elongations include, first, the member number followed by the specification of the type of elongation, uniform or concentrated. Concentrated elongations are applied instantaneously whereas uniform elongations occur with constant rate of change between some stipulated beginning and ending. Next it is necessary to establish the point at which elongation will commence either by some fractional multiplier of the actual length or by directly inputting the length

from the member origin. Finally a rotational or displacement distortion must be input, indicating whether twist, bending, translation, or shear is required. In the truss example, member 16 has a concentrated translational member distortion of 0.25 inches along its local X-AXIS (by default) occurring instantaneously at the end of the member.

Load three consists of a vertical displacement of 0.5 inches at support number one. Other options and/or modifications of this command include horizontal displacement in the X or Z (nonplanar) directions as well as rotations about the X, Y, or Z axes for frame structures. Dependent loading four consists of the combination of loadings two and three with magnification factors of one and 1.5 respectively. The loading combination command is extremely versatile in its applications in that, frequently, various loadings and their accompanying analysis results must be reviewed as simultaneous functions for the purpose of determining the worst case scenario. Additionally GTSTRUDL allows for the augmenting of load cases using stipulated magnification factors which may be used realistically to simulate safety-factored loading.

Finally the LOAD LIST command is a necessary part of any GTSTRUDL program that requires the activation or inactivation of certain previously specified loadings. This command is most frequently used in conjunction with loading combinations when it is necessary to create and utilize various loading sequences but ignore their individual components. For instance in the above TRUSS example, the user may wish to analyze both dependent loading four, which is a combination of independent loadings two and three, and independent loading one but not independent loadings two and three individually. For this purpose the command LOAD LIST 1 4 would be used.

ANALYSIS: Within the example TRUSS a linear static stiffness analysis was requested which includes the following operations, checks, and processes:

- 1) Original band width analysis
- 2) Renumbered band width analysis

- 3) Consistency check for members
- 4) Bandwidth reduction
- 5) Generate the individual stiffness matrices
- 6) Process the loadings
- 7) Assemble the stiffness matrix
- 8) Solve
- 9) Statics check
- 10) Generate combined results

TEXT OUTPUT: Although further commands become necessary when member selection is desired for design purposes (discussed in later sections), the complete examination of GTSTRUDL text output for stiffness analysis results may be solely accomplished via the LIST and PRINT commands. The PRINT command is very useful and practical as a debugging device for GTSTRUDL programs, in that upon its request GTSTRUDL produces a listing of specific structural data that was input earlier by the user. Hence the user is able to ascertain GTSTRUDL's translation and understanding of the structural characteristics which he/she is attempting to convey. For instance within the TRUSS example above, though it was the intent of the user to rotate the far left support 40 degrees in the XY plane and release the force in the X-direction other variations of these parameters may have actually been input thus the observance of how GTSTRUDL perceived the previous parameter input can be very important.

In addition to this it will normally be necessary to request of GTSTRUDL certain numerical analyses for the purpose of determining the magnitudes of shears, axial forces, displacements etc. The LIST command was designed to accomplish this by enumerating component displacements, loads, reactions, and forces.

GRAPHICAL OUTPUT: As explained previously the user has three options for the selection of a graphical output medium - the scope,

printer, and plotter. The scope device was discussed in considerable detail in CHAPTER 2: CCIEM PROCEDURAL FORMAT and thus within the following bounds of this text the crux of graphical output explanations will focus on the Calcomp plotter device as a means of graphical display

In the above TRUSS example it is essential to first define that device upon which graphical output will be plotted. When using the Calcomp, a FORTRAN neutral plot file must be created and subsequently routed for plotting, hence it is necessary to append the plot device statement with the neutral plot description for this purpose. Next it may be necessary to stipulate the plot area parameters by way of the length and width modifiers which allow the user to communicate to the plotter the longer and shorter dimensions of the plot area respectively.

Typically after the specification of a device it is then time for the organization and orientation of structural geometry within the bounds of the plot area by the use of several plot format commands. Only one of these was required for the simple truss structure in the above example; this being the format of the character size. By default GTSTRUDL formats the character size as the smallest value, four, of the four available sizes. It may be necessary to use this small lettering when plotting large plots with numerous labelings, however, for this example size number two was sufficiently legible.

Finally GTSTRUDL has defined provisions for the plotting of the deformed structure due to certain stipulated loadings for the purpose of visually ascertaining the points of maximum deflection. This command used in conjunction with the overlay command makes it possible to view loading deformations relative to the original structure shape. (see FIGURES 3.4, 3.5, 3.6)

PLANE FRAME

The structural type, plane frame, is one in which the linkage of members is accomplished through moment resisting connections thus producing three kinematic degrees of freedom (see table 3-2) which include

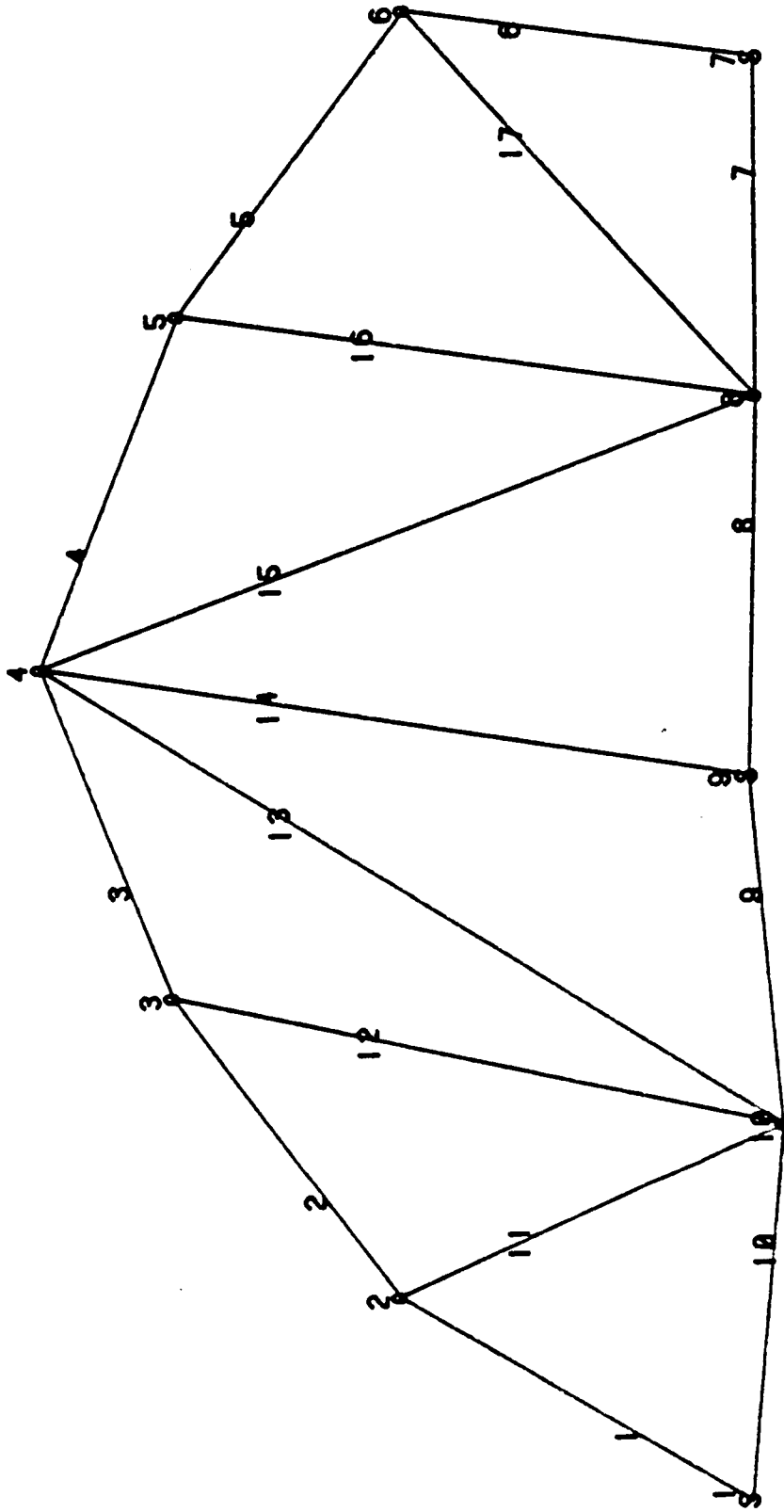


FIGURE 3-4. Deflected Shape - Joint Loading

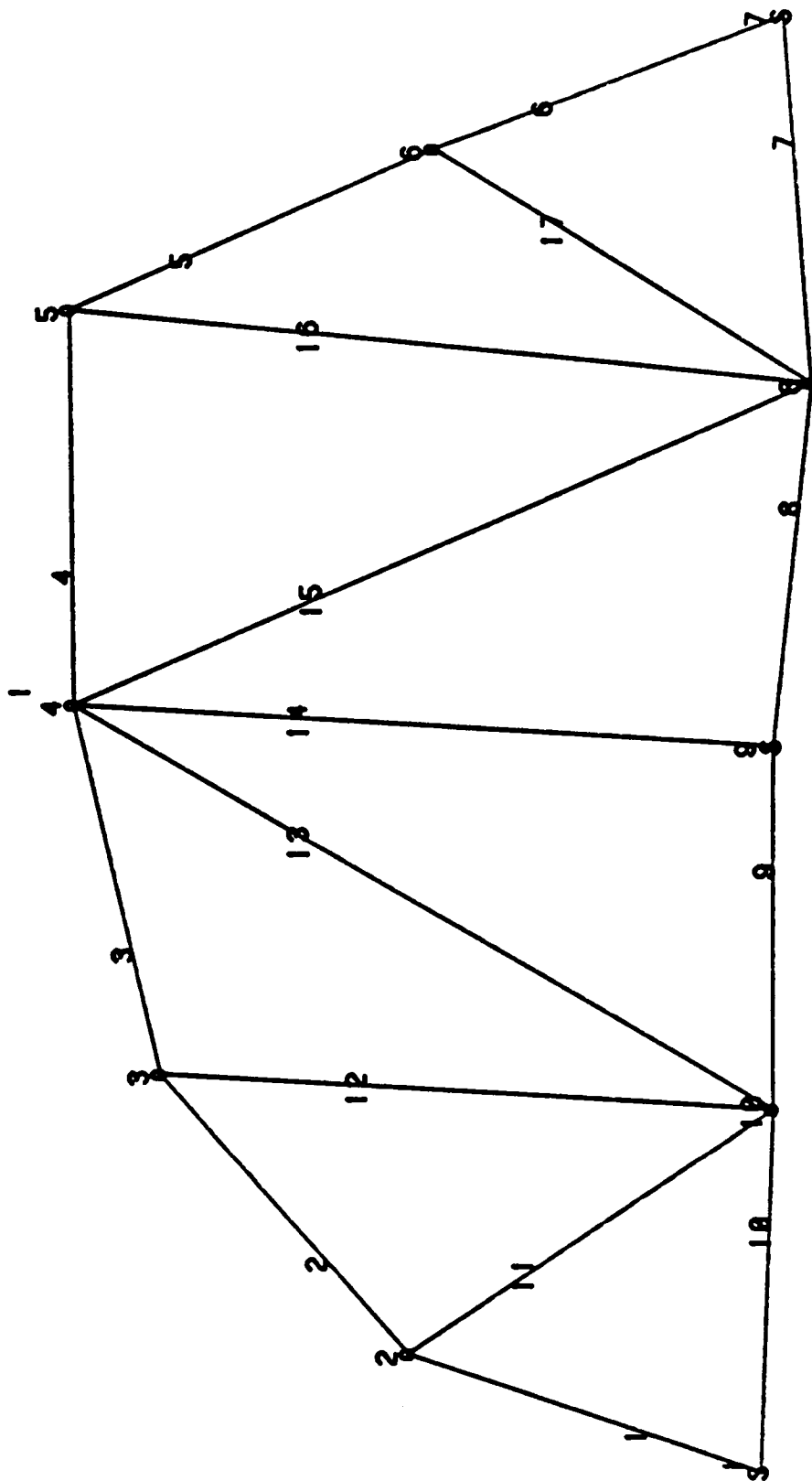


FIGURE 3-5. Deflected Shape - Member Elongation

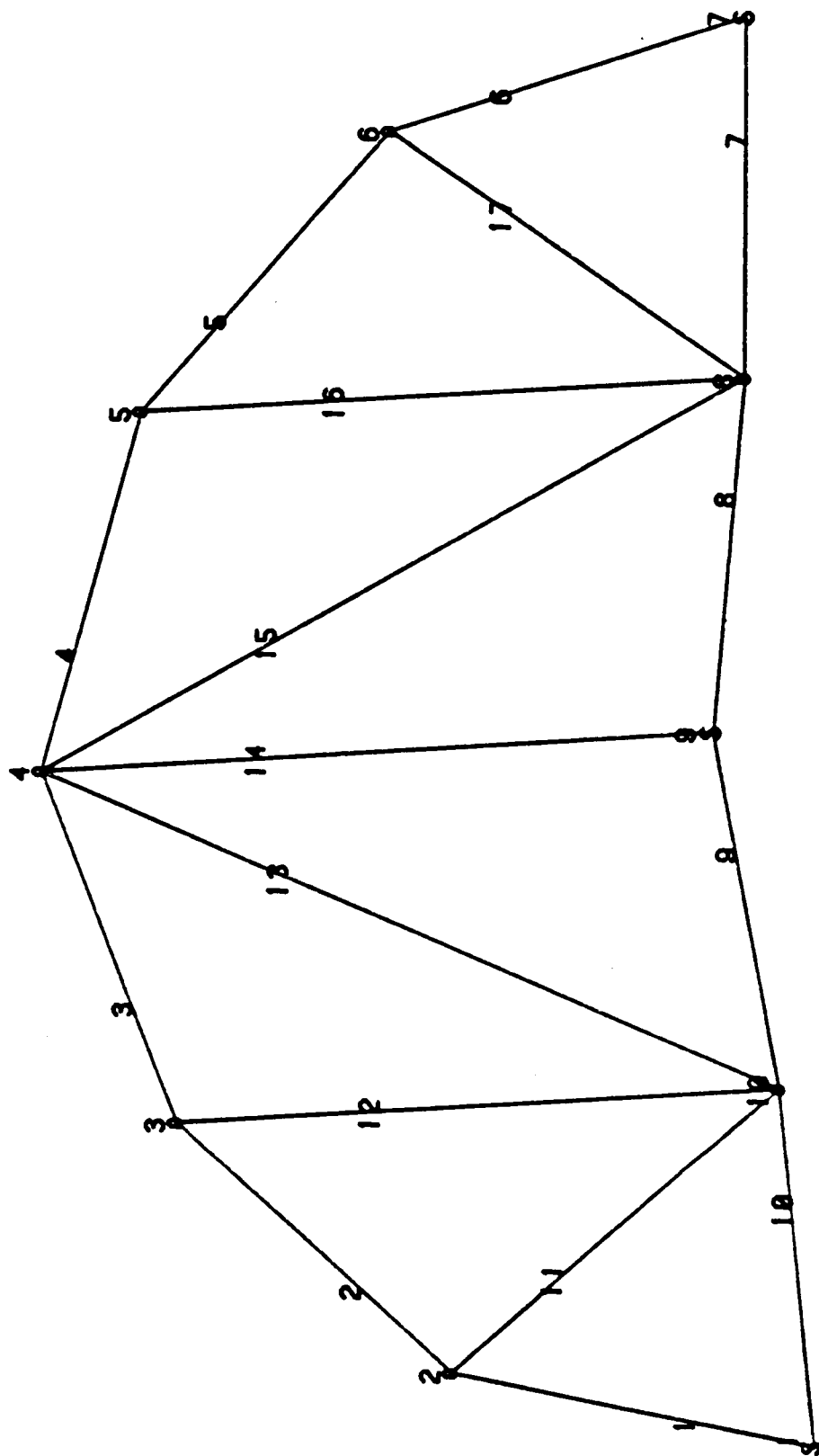


FIGURE 3-6. Deflected Shape - Support Settlement

forces in the plane of the member moments acting about the axis perpendicular to the plane of the member (i.e., also in the plane of the member). Due to this added degree of freedom, frame members, unlike their truss counterparts, may experience shears and moments in their respective planes, however, other than dissimilarities in the available loadings and their resultant forces the deviation relative to GTSTRUDL input between the two structural types is minute. For the purpose of accentuating these small differences, the above TRUSS analysis has been altered and adapted for planar frame analysis as noted previously by the bracketed additions. These additions generally include factors and properties related to rotations and moments about the perpendicular Z-axis. For instance member properties must include moments of inertia for all members due to the structures newly acquired resistance and capacity to handle inertial forces resulting from the member shape. Additionally pin and roller connections require the release of their Z-direction rotation which is assumed by default for truss analysis.

PLANE GRID

The planar grid type structure, similar to the previously mentioned plane frame, has the capacity to resist moment due to the fixity of its joints, however, unlike the frame, moment as well as torsional effects occur about the two axes which comprise the plane of the member. Additionally loadings occur perpendicular to the plane of the member as do the shear forces which result. (see Table 3.2 for the relevant kinematic degrees of freedom).

Example 3c: Plane Grid

STRUDL 'GRID ANALYSIS'
TYPE PLANE GRID

\$

\$ STRUCTURAL GEOMETRY

\$

\$ Sec Figure 3.7

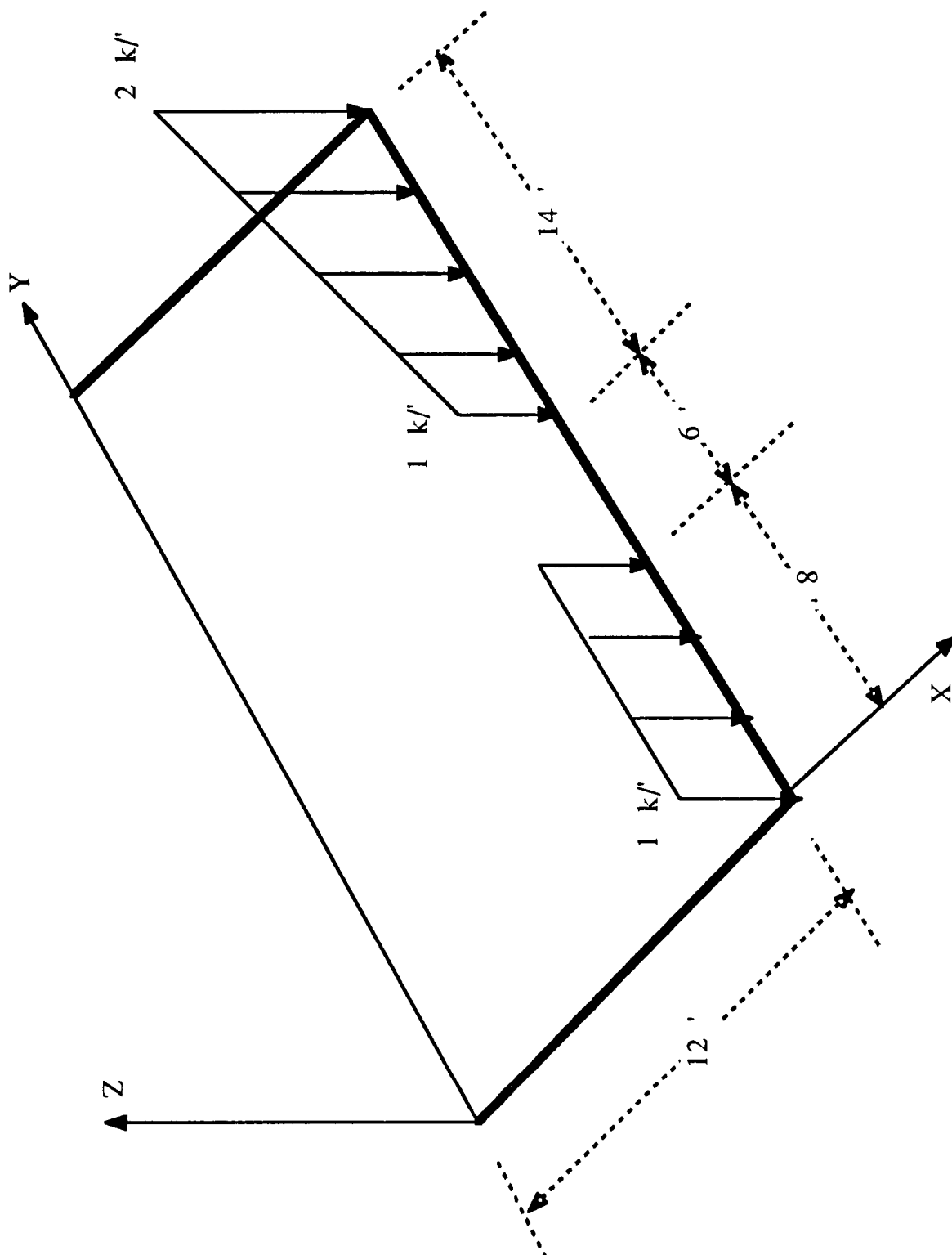


FIGURE 3-7. Grid dimensions and loads

\$
 UNITS KIP FEET
 JOINT COORD
 1 0 0 S
 2 12 0
 3 12 24
 4 0 24 S
 \$
 MEMBER INCID
 1 1 2
 2 2 3
 3 3 4
 \$
 \$ STRUCTURAL COMPONENT CHARACTERISTICS
 \$
 UNITS INCH
 CONSTANTS
 E 30000.ALL
 MEMBER PROPERTIES 1 TO 3 IX 100. IY 100.
 \$
 \$ LOADING
 \$
 LOADING 1 'MEMBER LOADS'
 MEMB LOAD
 2 FOR Z UNI W -1. LA 0 LB 8
 2 FOR Z UNI LIN WA -1. WB -2. LA 14 LB 24
 \$
 \$ ANALYSIS
 \$
 STIFFN ANA
 SCAN OFF
 \$
 \$ TEXT OUTPUT
 \$
 LIST FORCES DISP REACTIONS ALL
 \$
 \$ GRAPHICAL OUTPUT
 \$
 SAVE 'STEP'
 CHANGES
 CONSTANTS
 BETA 0 1 2
 BETA 60 3
 MEM PROP TABLE 'SSTEELW78'
 1 TO 3 TABLE 'W10X22'
 ADDITIONS
 STIFFN ANA

LIST FORCES DISP REACTIONS ALL
SAVE 'GRID'
FINISH

STRUCTURAL GEOMETRY: As mentioned previously, the preferable choice for the detailing of structural geometry in the vast majority of those structures with uniform topology is generation, however, the relative simplicity of the above grid example makes both methods equally manageable. Therefore, for the purpose of demonstrating the classification of supports without the status command the manual input method has been chosen. As is obvious this method entails the the explicit entry of joint coordinates and their subsequent connectivity via member incidences. Also when working within the XY plane the user need not specify the zero coordinates for the Z-axis, however, if other coordinate planes are chosen, all values, zero or nonzero, must be enumerated. Finally, the support command causes fixity against rotation and translation as defined in Table 3-2 and may be released in the direction of fixity if so desired.

STRUCTURAL COMPONENT CHARACTERISTICS: Occasionally it may be necessary to appropriate explicit values for the structural constants rather than using one of the three stored GTSTRUDL material names. The values GTSTRUDL uses for their material constants are listed in Table 3-3 and may vary significantly from the values the user deems necessary for his/her particular application. For instance a value of 30000 ksi was used in the grid example as compared to GTSTRUDL's 29000 ksi producing a substantial discrepancy in the stiffness of each member and ultimately changes in member stresses and deflections.

Also it fundamentally important to realize that because rotations occur about the axes within the plane of the member, moments of inertia must be included for those directions as well, as listed in Table 3-2. In other

words, regardless of the structure type, moments of inertia must be included in the direction in which rotation will occur.

LOADING: As has been determined, loading must occur in the direction perpendicular to structural plane of orientation. In the above example, grid member two has been loaded across two sections of its surface. Though there are two loadings they are each classified under one load heading. Thus for example, though it may be useful depending on the circumstances, it is unnecessary to establish five individual loadings for five different kinds of loads (i.e., joint loads, member loads, elongations may all be included under independent loading 1).

In this example, the first section of loading on member two is uniformly distributed with a Z magnitude of negative one, beginning at the member origin ($LA = 0$) and terminating at eight feet from the origin ($LB = 8$). The second section increases linearly from Z magnitude negative one to negative two beginning at fourteen feet ($LA = 14$) from the member origin and terminating at the end of the member, 24 feet ($LB = 24$) from the member origin. Note also that these load distances could have been input fractionally by specifying a fractional value times the actual length.

The sections ANALYSIS and OUTPUT contain a linear static stiffness analysis and a listing of forces, displacements and reactions similar to those catalogued in the above frame and truss analysis.

DATA BASE MANAGEMENT: It was then desired to observe the behavior of the structure with a modification of the member properties from prismatic to steel wide flanges in addition to a beta rotation of sixty degrees. The delineation of a beta angle describes the orientation (rotation) of non-symmetrical cross sections from an initially assumed reference position of a members local Y and Z axes. The beta angle is measured about the member's local X-axis relative to a right-handed rotation (+/_ 0 to 360 degrees)

CHAPTER IV

FINITE ELEMENT ANALYSIS

MESH GENERATION

This section contains several examples and explanations on the generation of finite element mesh on symmetric structures. The purpose of mesh generation is to expedite problem solutions by supplementing or replacing user calculated joint coordinates and elements on structures whose geometry is predominantly if not completely conducive to a faster GTSTRUDL generation. The structures which are normally applicable to such generation are ones which require large, uniformly sized elements.

EXAMPLE # 4a: RECTANGULAR PLATE

```
-
STRUDL 'MESH GENERATION' 'RECTANGULAR PLATE'
TYPE PLATE BENDING
UNITS KIP INCHES
$
$      SEE FIGURE 4.1
$
$      GENERATE NODES AND ELEMENTS HORIZONTALLY
$      NUMBER  NODES AND ELEMENTS VERTICALLY
$
1)  GENERATE 5 JOINTS ID 1 11 X 0.0 0.0 Y 0.0 12.0
2)  MODIFY 10 ID 1 X 0.0 Y 12.0
3)  GENE 4 ELEM ID 1 10 FROM 1 11 TO 2 11 TO 13 11 TO 12 11
4)  MODIFY 9 ID 1 TO 1 TO 1 TO 1
$
ELEMENT PROPERTIES
1 TO 40 TYPE 'BPHQ' THICKNESS 1.0
PLOTTER DEVICE PLOTTER NEUTRAL
PLOT FORMAT SIZE HORI 9 VERT 4
PLOT FORMAT ORIENTATION NONSTANDARD
PLOT FORMAT SIZE CHAR 2
PLOT PLANE
```

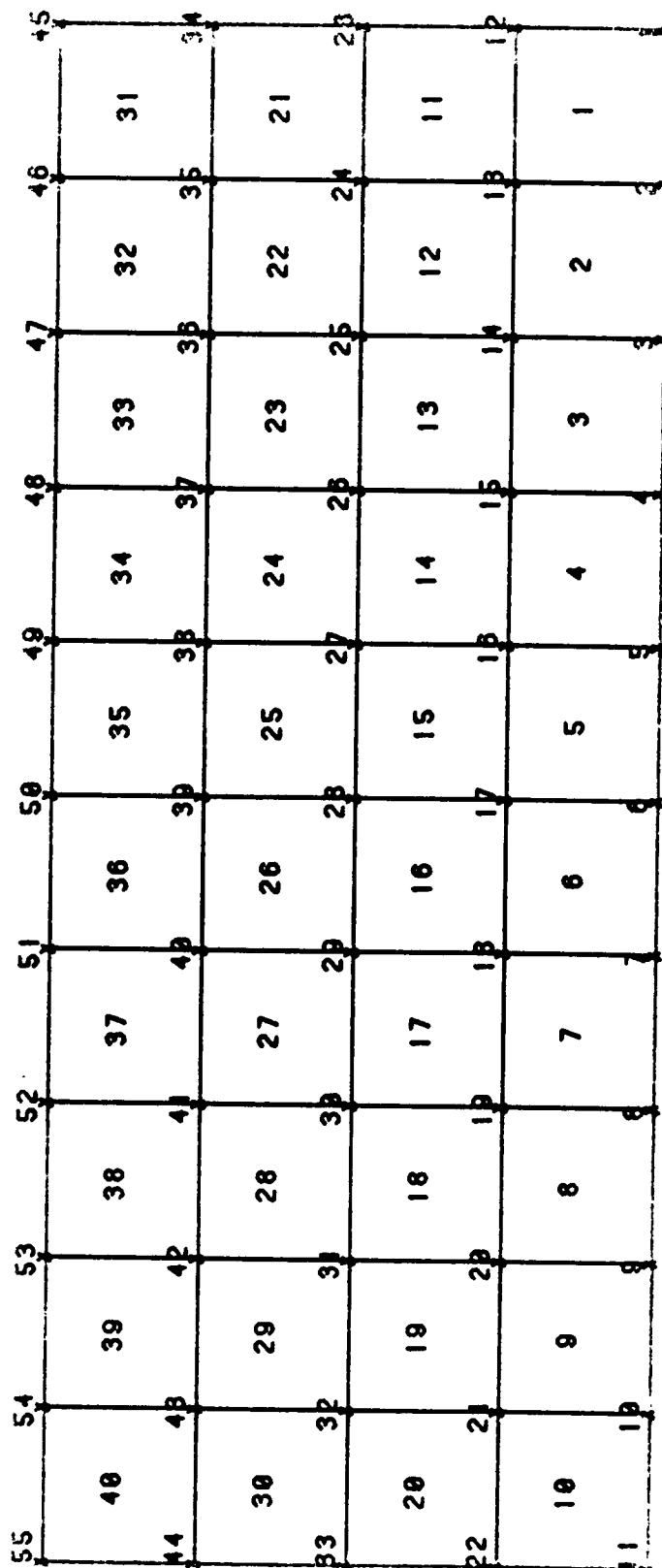


FIGURE 4-1. Rectangular Plate Geometry

SAVE 'STEP'
FINISH

EXPLANATION:

- 1) GENERATE 5: generates 5 joints
- ID 1 : starts generation at joint 1
- 11 : horizontally, joints are numbered in intervals of 11
 (i.e., 1,12,23,34,45 etc.)
- X...Y... : the location of the first joint is at the origin (0,0).
 each successive joint will be spaced 12 inches in
 the X direction and zero in the Y.
- 2) MODIFY 10 : modifies the first line of joints just generated 10
 times (i.e., 11 rows of 5 joints each)
- ID 1 : identifies starting joint (i.e. 1)
- X..Y... : each successive row of joints is located 0 inches
 (X-direction) and 12 inches (Y-direction) from the
 last.
- 3) GENER 4 : generate 4 elements
- ID 1 : starting at element 1
- 10 : horizontally, elements are numbered in intervals of
 10 (i.e., 1,11,21,31 etc.)

FROM.. : element 1 is numbered 1, 2, 13, 12 (numbered clockwise or counter clockwise) and the joints of each successive element, horizontally will increase by 11 (i.e., the second element, 11, consists of joints (12, 13, 24, 23))

4 MODIFY 9 : modifies the last line of elements just generated, 9 times.

ID 1 : identifies the starting joint (i.e., 1)

FROM.. : the interval is 1 between the joints defining the first row of elements generated and each successive row.

Immediately following the generation of the rectangular plate it is essential that the user define the type and thickness of all elements which will be involved in any subsequent operations. Should this step be omitted GTSTRUDL, after reaching the first command involving the previously generated elements, will display the warning 'ELEMENT PROPERTIES NOT DEFINED FOR ELEMENTS...'. As always for Calcomp plotting, the 'plotter neutral' directive must initiate the sequence of plot commands in order for the neutral plot file to be readied for the instructions that follow. These instructions, typically, entail the placement and orientation of the plot within the bounds of the plot area by way of several format statements. The first of these limits the plot area and vertical dimensions of nine and four respectively in currently activate units. It should be noted that the horizontal dimension would normally be plotted on the shorter media dimension however for the purposes of clarification the nonstandard command was issued, rotating the plot ninety degrees. Finally a character size of two, out of a possible four, was requested, instructing GTSTRUDL to print using the second largest lettering size - one being the largest.

EXAMPLE #4b: SKEWED PLATE/UNIFORM MESH

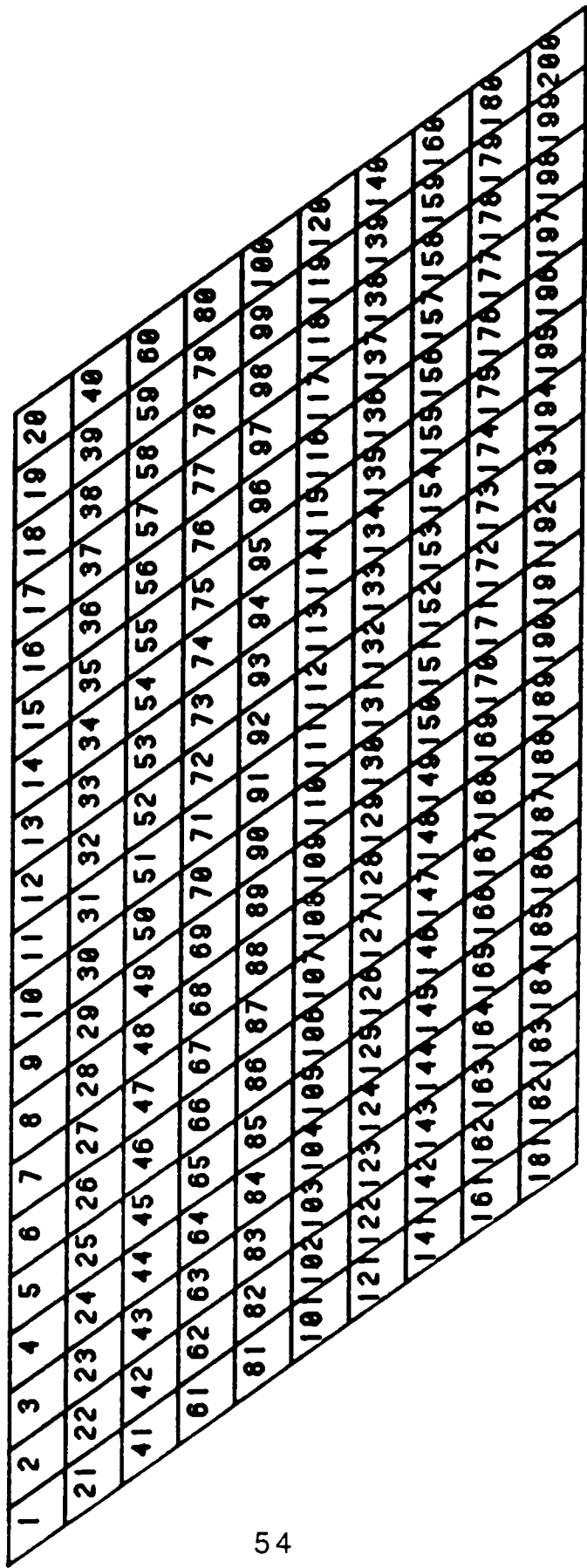
```
STRUDL 'SKEWED PLATE'
TYPE PLATE BENDING
UNITS KIP INCHES
$
$      SEE FIGURE 4.2 A and B
$
$      GENERATE JOINTS AND ELEMENTS VERTICALLY
$      NUMBER JOINTS AND ELEMENTS HORIZONTALLY
1) GENERATE 11 JOINTS ID 1 21 X 0.0 8.45 Y 120.0 -12.0
2) MODIFY 20 ID 1 X 12.0 Y 0.0
3) GEN 10 ELE ID 1 20 FROM 1 21 TO 2 21 TO 23 21 TO 22 21
4) MODIFY 19 ID 1 FROM 1 TO 1 TO 1 TO 1
$
ELEMENT PROPERTIES
1 TO 200 TYPE 'BPHQ' THICKNESS 1.0
PLOT DEVICE PLOTTER NEUTRAL
PLOT FORMAT ORIENTA NONSTAND
PLOT FORMAT OMIT ELEMENT NAMES
PLOT FORMAT SIZE CHAR 2 HOR 9 VERT 4
PLOT PLANE
PLOT FORMAT OMIT JOINT NAMES
PLOT PLANE
SAVE 'STEP'
FINISH
```

EXPLANATION:

1) GENERA 11 : generates 11 joints

 ID 1 : starts at joint 1

 21 : vertically joints are numbered in intervals of
 21.(i.e., 1,22,43,64 etc.)



a. Elements Listed

FIGURE 4-2. Skewed Plate with Uniform Mesh

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21
22	23	24	25	26	27	28	29	30	31	32	33	34	35	36	37	38	39	40	41	42
43	44	45	46	47	48	49	50	51	52	53	54	55	56	57	58	59	60	61	62	63
64	65	66	67	68	69	70	71	72	73	74	75	76	77	78	79	80	81	82	83	84
85	86	87	88	89	90	91	92	93	94	95	96	97	98	99	100	101	102	103	104	105
106	107	108	109	110	111	112	113	114	115	116	117	118	119	120	121	122	123	124	125	126
127	128	129	130	131	132	133	134	135	136	137	138	139	140	141	142	143	144	145	146	147
148	149	150	151	152	153	154	155	156	157	158	159	160	161	162	163	164	165	166	167	168
169	170	171	172	173	174	175	176	177	178	179	180	181	182	183	184	185	186	187	188	189
190	191	192	193	194	195	196	197	198	199	200	201	202	203	204	205	206	207	208	209	210
211	212	213	214	215	216	217	218	219	220	221	222	223	224	225	226	227	228	229	230	231

b. Joint listed

Figure 4-2. (Continued)

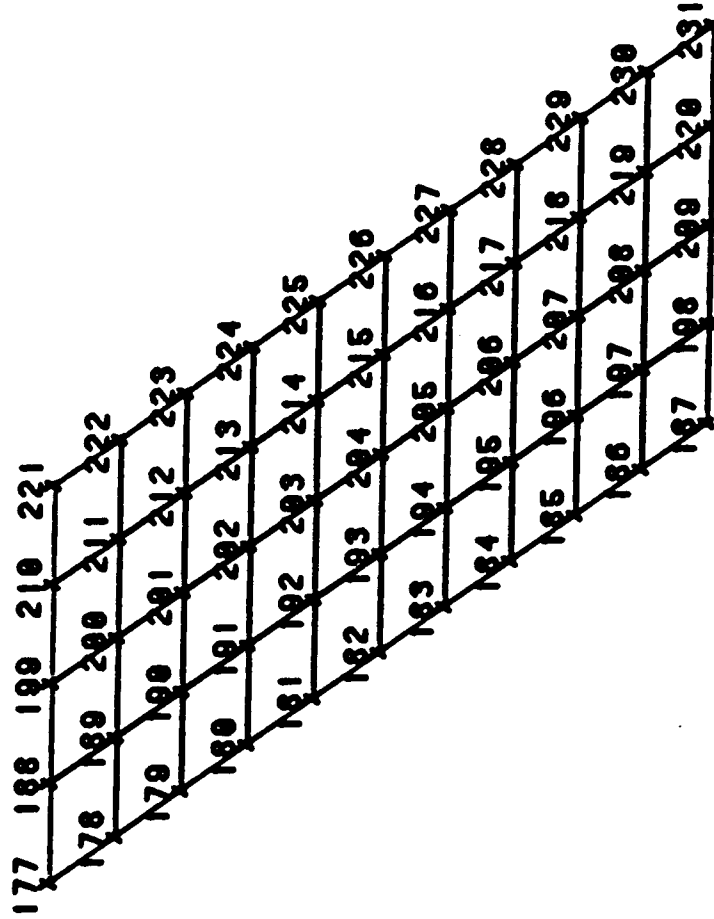
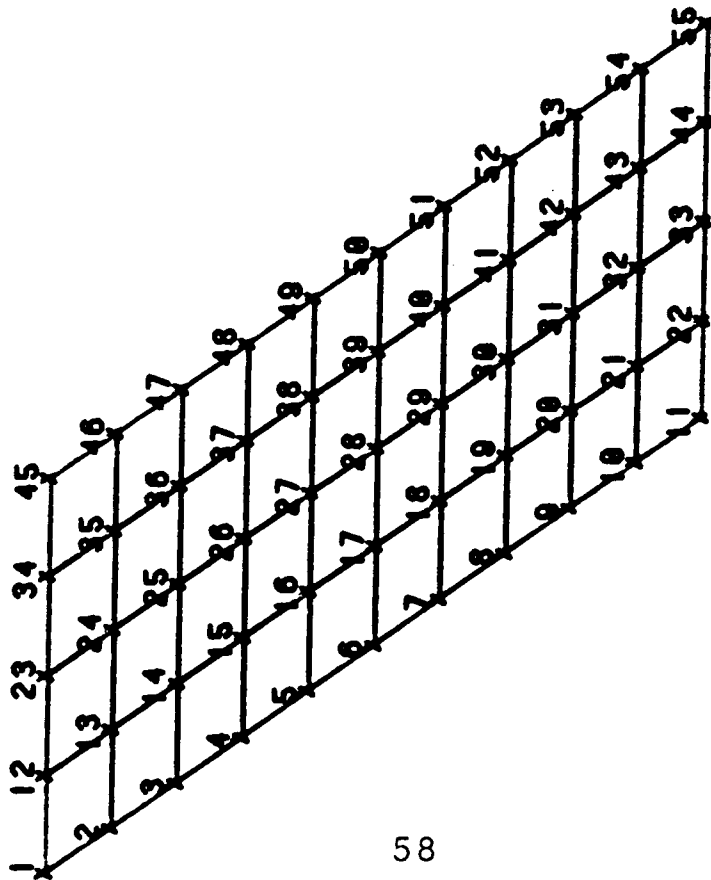
- X...Y... : the location of the first joint is at (0,120). Each successive joint will be spaced 8.45 inches in the X-direction and - 12.0 inches in the Y-direction.
- 2) MODIFY 20 : modifies the first line of joints just generated 20 times. (i.e., 21 columns of 11 joints each)
- ID 1 : identifies starting joint as 1
- X 12 Y 0 : each successive column of joints is located 12 inches in the X-direction and 0 inches in the Y-direction from the last.
- 3) GENERA 10 : generates 10 elements
- ID 1 : starting at element 1
- 20 : vertically elements are numbered in intervals of i.e., 1,21,41,61,etc.)
- FROM... : element 1 is numbered 1,2,23,22 and the joints of each successive element,vertically,will increase by 21, (i.e., the second element, 21, consists of joints 22, 23, 42, and 43.
- 4) MODIFY 9 : modifies the first column of elements just generated 9 times.
- ID : identifies starting element 1

FROM : the interval is 1 between the joints defining
the first column of elements generated and
each successive element.

This series of plot commands is very similar to those included in the above rectangular plate example, however, slight variations do exist. The first of these is the inclusion of the horizontal and vertical statements along with the character size. Each of these commands fall under the category of size and may be listed together or separately at the user's discretion. Additionally it was necessary to plot the same skewed plate two times for the purpose of showing both joints and elements in an uncluttered fashion. To do this it was only necessary to format the required names again rather than the entire sequence.

EXAMPLE #4c: SKEWED PLATE/FINE MIDDLE MESH

```
STRUDL 'SKEWED PLATE WITH FINE MIDDLE MESH'  
TYPE PLATE BENDING  
UNITS KIPS INCHES  
$  
$ SEE FIGURE 4.3 A and B  
$ LINES AND ELEMENTS GENERATED X numbered Y  
$  
GENERATE 6 JOINTS ID 1 11 X 0.0 12.0 Y 120.0 0.0  
MODIFY 10 ID 1 X 8.45 Y -12.0  
GEN 5 ELE ID 1 10 FROM 1 11 TO 2 11 TO 13 11 TO 12 11  
MODIFY 9 ID 1 FROM 1 TO 1 TO 1 TO 1  
GENERATE 10 JOINTS ID 67 11 X 66.0 6.0 Y 120.0 0.0  
MODIFY 10 ID 1 X 8.45 Y -12.0  
GEN 10 ELE ID 51 10 FROM 56 11 TO 57 11 TO 68 11 TO 67 11  
MODIFY 9 ID 1 FROM 1 TO 1 TO 1 TO 1  
GEN 5 JOINTS ID 177 11 X 132.0 12.0 Y 120.0 0.0  
MODIFY 10 ID 1 X 8.45 Y -12.0  
GEN 5 ELE ID 151 10 FRO 166 11 TO 167 11 TO 178 11 TO 177 11  
MODIFY 9 ID 1 FROM 1 TO 1 TO 1 TO 1  
ELEMENT PROPERTIES  
1 TO 200 TYPE 'BPHQ' THICKNESS 1.0  
PLOT DEVICE PLOTTER NEUTRAL
```



a. Deletion of middle mesh

FIGURE 4-3. Skewed Plate

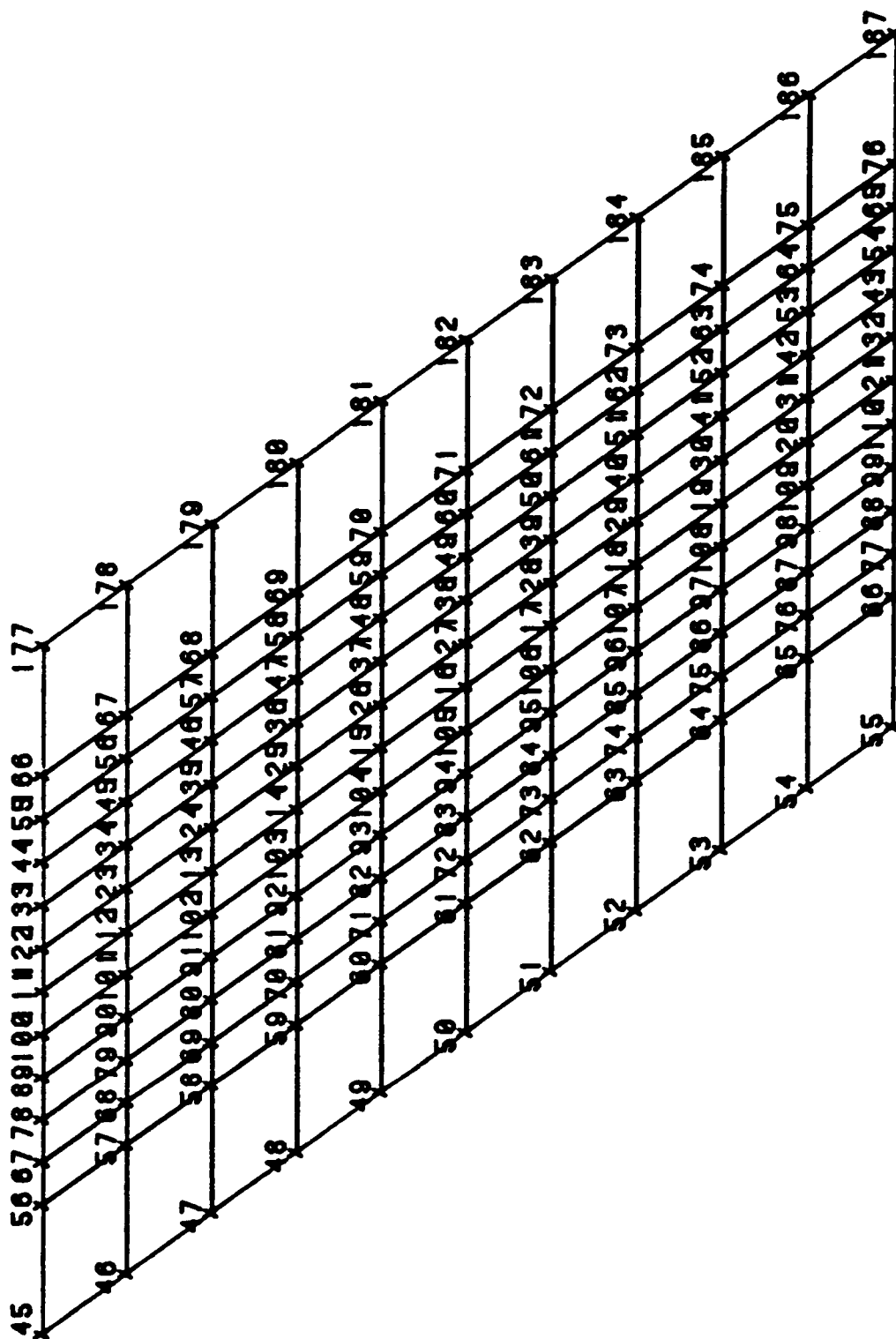


Figure 4-3. (continued)

b. Window on Fine Middle Mesh

```
PLOT FORMAT ORIE NONSTAN
  PLOT FORMAT OMIT ELEM NAMES
  PLOT FORMAT SIZE CHAR 2 HORI 9 VERT 4
  PLOT PLANE
  PLOT FORMAT OMIT JOINT NAMES
  PLOT PLANE
  SAVE 'STEP'
  FINISH
```

EXPLANATION:

This type of generation technique is similar in format to the first two examples except that additional sets of generate/modify commands are needed for the various stages of mesh. As is obvious, two sets per mesh, one for the generation and modification of joints and another for elements). Also a point of interest is that the identification (ID) is not necessarily 1. It is from whatever joint or node the beginning of the generation process starts.

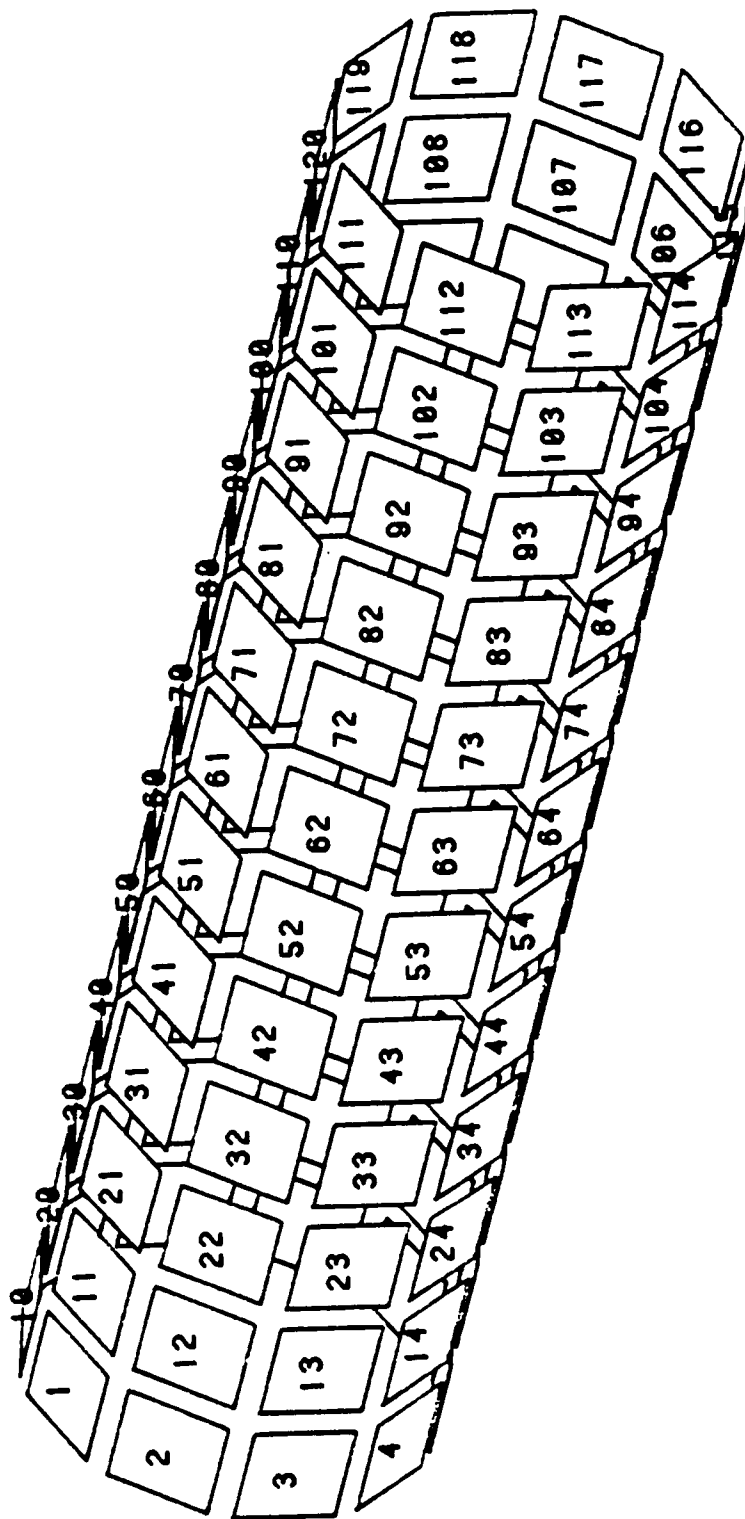
In summary two GENERATE` commands are necessary for each section of mesh. The ID refers to the beginning joint or element in a section from which generation originates. It has been found by this author that GTSTRUDL requires the generation of joints and elements in one direction and the modification and numbering in the opposite direction.

COMPLETE ANALYSIS

This section, unlike the previous mesh examples, contains a complete plate bending finite element analysis along with the accompanying explanation. It should be understood that the examples within this section are for the purpose of outlining a GTSTRUDL format for finite element analysis and in no way attempt to cover or explain the finite element process generally. The understanding of this process is at the very least a corequisite with this manual.

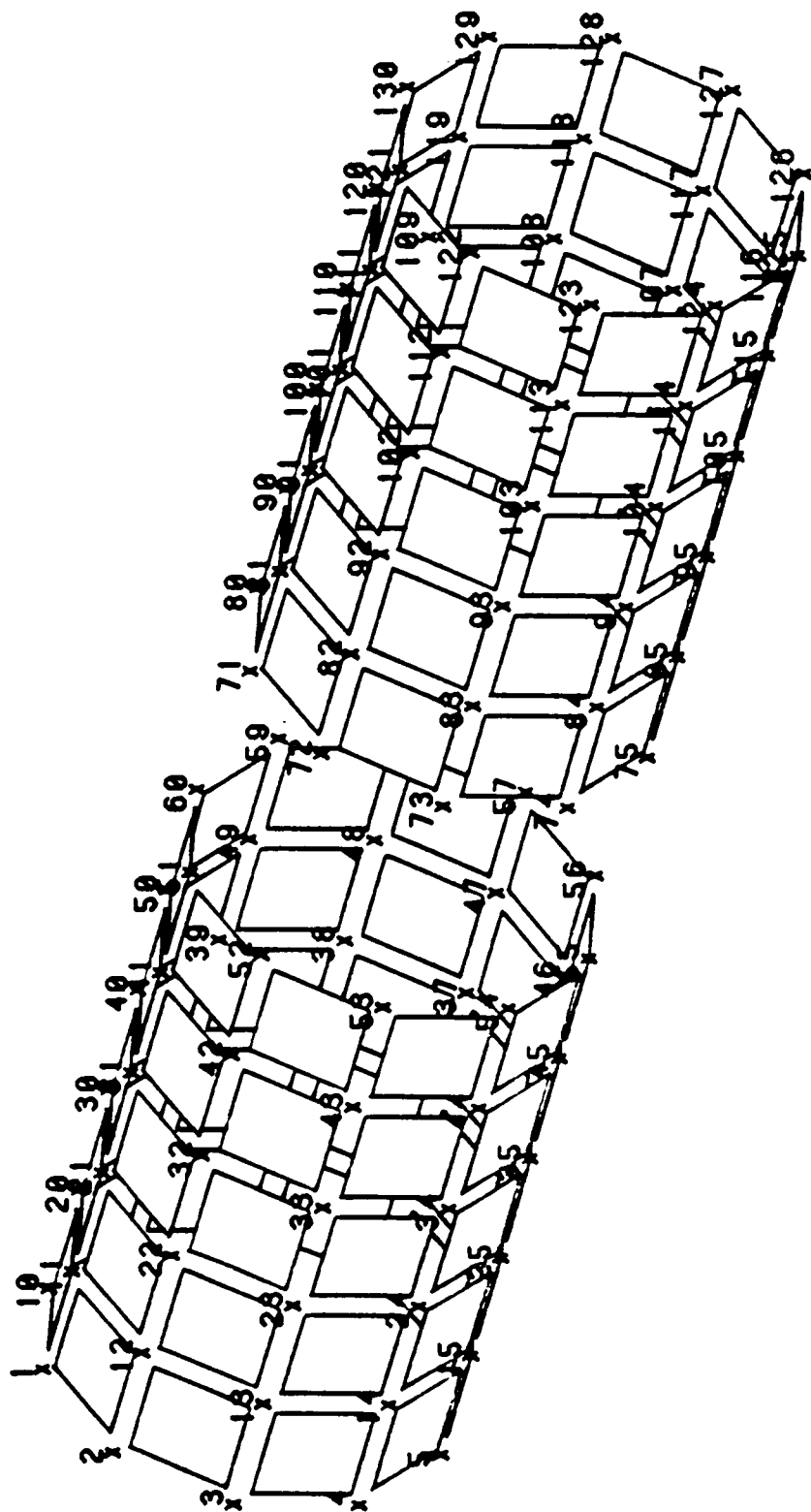
EXAMPLE #4d: 3-D PLATE BENDING

```
STRUDL '3-D CYLINDER FINITE ELEMENT MODEL'
$
$   THIS GTSTRUDL MODEL ANALYZES A 36 INCH STEEL PIPE
$   TO LOCATE THE POINT AT WHICH PLASTIC LOADING OCCURS
$
$   STRUCTURAL   GEOMETRY
$
$   SEE FIGURE 4.4 A and B
$
UNITS INCH DEGREES KIPS
GENER 10 JOINTS CYLIN ID 1 1 R 5 TH 0 36 LX 0
REPEAT 12 TIMES ID 10 LX 3
GENER 9 ELEMENTS ID 1 1 F 1 1 T 2 T 12 T 11
REPEAT 11 TIMES ID 10 F 10
GENER 12 ELEM ID 10 10 F 10 10 T 1 T 11 T 20
$
$   STRUCTURAL   COMPONENT   CHARACTERISTICS
$
STATUS S 1 TO 10 121 TO 130
JOINT RELEASES
1 TO 10 121 TO 130 MOM X MOM Y
CONSTANTS E 29000. ALL
POISSON 0.3 ALL
ELEMENT PROPERTIES
1 TO 120 TYPE 'SBHQ' THICKNESS 0.5
$
$   LOADING
$
LOADING 1 'PRESSURE'
ELEMENTS LOADS
1 TO 120 SURFACE FORCES LOCAL PZ -.3
LOADING 2
ELEMENT LOADS
1 TO 120 SURFACE FORCES LOCAL PZ -7.
$
$
$   ANALYSIS
$
STIFFN AN
$
$   TEXT OUTPUT
$
```



a. Elements Labeled

FIGURE 4-4. Uniform Cylindrical Mesh



b. Joints Labeled

FIGURE 4-4. (continued)

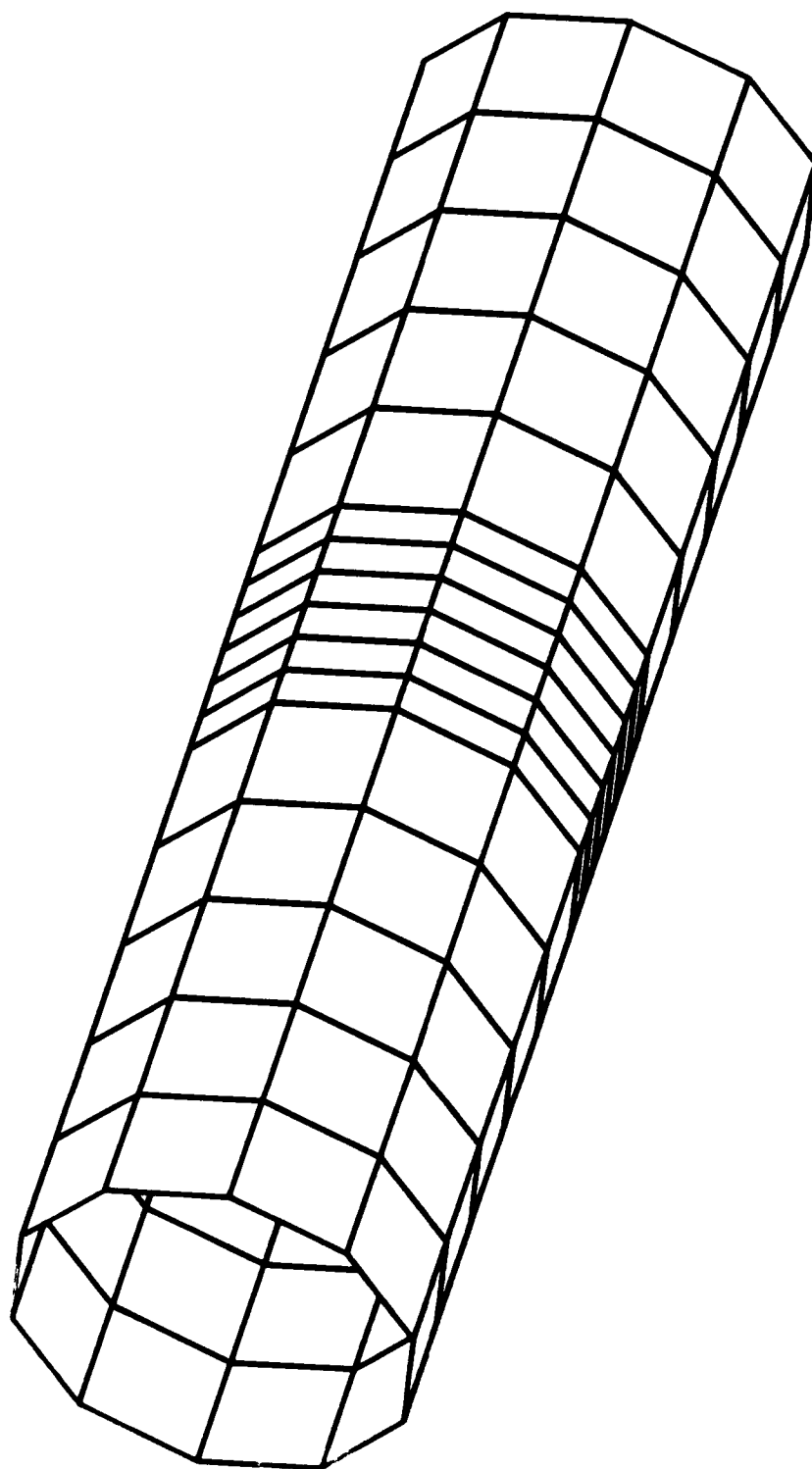
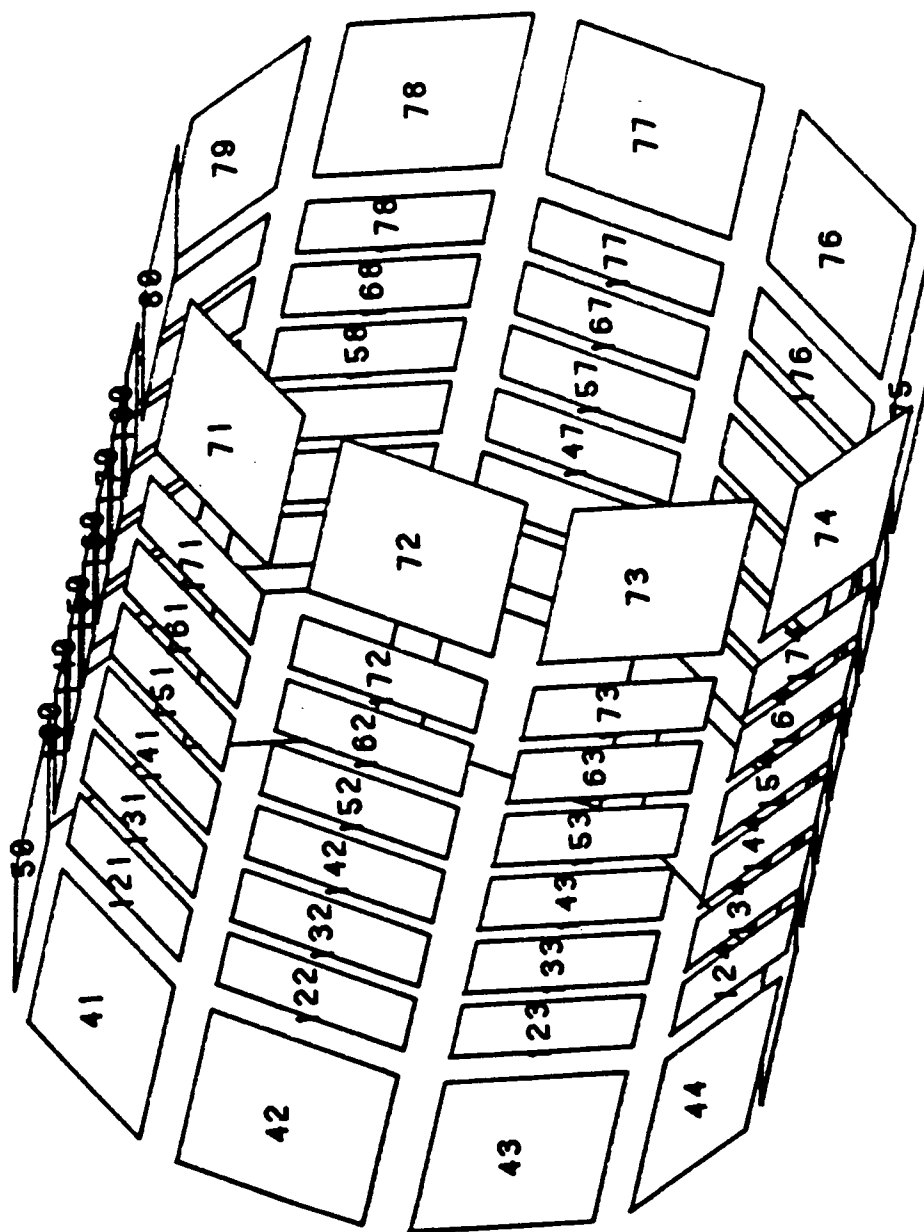


FIGURE 4-5. Mesh Combination



a. Elements

FIGURE 4-6. Window on Fine Mesh


```

$
LOAD 3 'JOINT LOADS ON THE END'
JOINT LOAD
121 TO 130 FORC X -10
$
$    ANALYSIS
$
STIFFN ANA
$
$    TEXT  OUTPUT
$
CALC AVE PRINC STRESS AT THE TOP FOR ALL
$
$    GRAPHICAL  OUTPUT FOR PIPE
$
$    FIGURES 4.4 A and B
$
RESTORE 'PIPE'
PLOT DEVICE PLOTTER NEUTRAL
PLOT FORMAT SCALE LENGTH 4
PLOT FORMAT SIZE CHAR 2
PLOT FORMAT BOUND SHRINK
PLOT FORMAT OMIT ELEMENT NAMES
PLOT FORMAT HIDDEN LINES REMOVED
PLOT PROJ BOUN HIDDEN 0 -40 20
PLOT FORMAT OMIT JOINT NAMES
PLOT PROJ BOUN HIDDEN 0 -40 20
$
$    GRAPHICAL  OUTPUT FOR PIPE 2
$
$    FIGURES 4.5 A and B
$
RESTORE 'PIPE2'
PLOT DEVICE PLOTTER NEUTRAL
PLOT FORMAT SIZE CHAR 2
PLOT FORMAT SCALE LENGTH 4
PLOT FORMAT BOUND SHRINK
PLOT FORMAT OMIT ELEM NAMES
PLOT PROJ BOUN HIDDEN 0 -40 20
PLOT FORMAT OMIT JOINT NAMES
PLOT PROJ BOUN HIDDEN 0 -40 20
$

$    WINDOW IN ON PARTIAL STRUCTURE
$
$    FIGURE 4.6 A
$

```

```

PLOT DEVICE PLOTTER NEUTRAL LENGTH 9 WIDTH 8
PLOT FORMAT SIZE CHAR 2 VERT 5
PLOT FORMAT BOUND SHRIN
PLOT FORMAT TITLE ' WINDOW IN PARTIAL STRUCTURE'
PLOT FORMAT LIMITS JOINT XMIN 41 XMAX 81
PLOT FORMAT OMIT JOINT NAMES
PLOT FORMAT HIDDEN LINES REMOVED
PLOT PROJ BOUND HIDDEN 0 -40 20
$
$      REMOVE ELEMENT SECTION AND REPLOT
$
$      FIGURE 4.6 B
$
DELETE ELEMENTS 42 43 72 73 122 TO 172 BY 10
DELETE ELEM 123 TO 173 BY 10
PLOT FORMAT OMIT ELEM NAMES
PLOT PROJ BOUND HIDDEN 0 -40 20
FINISH

```

STRUCTURAL GEOMETRY: The criteria for the decision as to whether manual input or generation should be used for geometric data involves 1) uniformity of the numbering and 2) uniformity of the geometry. Typically within finite element analyses, uniformity of numbering is a primary consideration due to the necessity of band width reduction. However, GTSTRUDL provides both an automatic renumbering as well an optional band width reduction command, hence uniformity of numbering is rarely a viable consideration. Geometric uniformity, on the other hand, for at least section of the structure is virtually a requirement.

Referring to Figure 4.4, cylindrical generation begins with the formation of the first ten joints beginning at joint one and incrementing the following joints by one. Each joint will be oriented about the X-axis ($LX=0$) at a radius of five inches, beginning at zero degrees (Y-axis) and continuing at angular increments of 36 degrees, thus rotating joint ten 324 degrees from the origin at joint one. The following repeat command iterates this process twelve times, incrementing each joint three inches in the positive X-direction starting at joint ten.

The creation of the first nine elements is done similarly, beginning at element one, consisting of joints 1, 2, 12, 11, and propagating both joints and elements at intervals of one. The process is then repeated 11 times

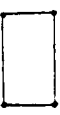
starting at joint ten and incrementing joints 1, 2, 12, 11 by ten. Note that within the repeat command the ID number as well as the increment refers to a joint rather than an element.

Finally the last twelve elements are generated linearly along the X-axis starting at element 10, consisting of joints 10, 1, 11, 20, and incrementing both elements and joints a value of ten. It is fundamentally important to realize that the last generation statement was executed as is, due to the inconsistency and nonuniformity of the element numbering for elements 10, 20, 30 etc. Also, to reiterate, within the generate command, ID refers to the first joint or element that is being generated, however, within the repeat or modify commands ID refers to the beginning joint.

STRUCTURAL COMPONENT CHARACTERISTICS: By judiciously selecting support joints, it is the user's intent to simulate actual structural conditions. In the above example, a 36 inch pipe section will be connected to other similar sections such that translation of the end joints due to the pressure loading will be inhibited but rotation at these joints will not. For this purpose joints 1 through 10 and 121 through 130 are classified as supports and the available moment fixity is released.

The constants, modulus of elasticity and Poisson's ratio, were chosen uniformly throughout the structure as defined in the section 'PLANAR ANALYSIS'. It was then necessary to choose the category of element properties within the PLATE element section, listed in table 4-1 that best fit this particular application. A detailed discussion of available element types and their respective characteristics is outlined in the GTSTRUDL finite element library and is beyond the scope of this text. However suffice it to say that under the PLATE type element there are eight element names of which there are three triangular and five quadrilateral. Six of the eight element names accommodate five kinematic degrees of freedom while the other two accommodate six. The 'SBHQ' type element, used in the above example, is among the five degree of freedom quadrilaterals, which allows

TABLE 4-1. Plate Elements

Element					Input				Output							
Element Name	Shape	No. of Nodes	D.O.F.	Rigidity Matrix	Joint Loads				Element Loads		Stress/Moment	Shear Resultant	Strain/Curvature	Principal Stress/Moments	Principal Strain/Curvature	Displacements
					Concentrated	Temperature Change	Temperature Gradient	Edge	Surface	Body						
TYPE — PLATE																
SBCT		3	u_1, u_2, u_3, u_4, u_5	x	x	x	x		x	x	*	C	ε ₁			N
SBCR		4	$u_1, u_2, u_3, u_4, u_5, u_6$	x	x	x	x		x	x	*	N				N
SBCRCSH		4	u_1, u_2, u_3, u_4, u_5	x	x	x	x		x	x	*	N				N
SBHQ		4	u_1, u_2, u_3, u_4, u_5	x					x	x	N	N	N			N
SBHQCSH		4	u_1, u_2, u_3, u_4, u_5	x	x				x	x	N	N	N			N
SBHT		3	u_1, u_2, u_3, u_4, u_5	x	x				x	x	N	N	N			N
SBHT 6		3	$u_1, u_2, u_3, u_4, u_5, u_6$	x	x				x	x	N	N	N			N
SBHQ 6		4	$u_1, u_2, u_3, u_4, u_5, u_6$	x	x				x	x	N	N	N			N

for joint concentrated and element surface loads in addition to formatting moment, shear, strain, and displacement output at the nodes.

LOADING: It was the intent of this analysis to determine at what approximate fluid pressure, acting normal to the surface of the elements, would cause inelasticity in the element material. In order to achieve such a simulation GTSTRUDL offers the surface forces command which applies a pressure type loading to the prescribed element face in the currently active units of force per length squared. Two loadings of 0.3 ksi and 7 ksi in the negative Z-direction were analyzed and found to produce zero plasticity and over ninety percent plasticity on the minimum (S1) and maximum (S2) faces respectively, with an assumed, approximate yield stress of 40 ksi. For a more accurate approximation of the first yield loading the engineer may choose to interpolate or continue with another GTSTRUDL analysis.

It should be noted that loading on most structures if left unstipulated is projected in the global, as opposed to local, direction. However for finite element loadings the local direction is the default, thus the reason for omitting the global or local specification to achieve the desired loading in the above example.

TEXT OUTPUT: The LIST and PRINT commands were defined in the Planar Analysis section and are similarly formatted for finite elements, however the tabulation of forces, reactions, etc., have been replaced with stresses and strains within the LIST command. A much more useful command, with respect to the processing of results by the user, is the calculate average command. Occasionally finite elements within a mesh may be incident upon four or more nodes, all producing variations in stress output due to particularly high stress gradients in that region in addition to the fact that continuity of stress is not enforced by GTSTRUDL. To obtain a more useful representation of nodal stress at areas of high stress concentrations due to material or geometric discontinuities, GTSTRUDL

allows for the calculation of weighted stress averages. This procedure involves the summing of all elemental stresses at a node and dividing by the number of elements used in the average. The output provided when the CALCULATE or LIST commands are used includes S1 (minimum normal stress), S2 (maximum normal stress), TMAX (maximum shear stress) and THETA (angle between horizontal and rotated normal stress).

DATA BASE MANAGEMENT: Often it is essential for the engineer to formulate and subsequently vary a sequence of GTSTRUDL commands for the purpose of analysis, reanalysis and comparison. Also it may be necessary to iterate this procedure several times in order to achieve the desired range of accuracy regarding the analysis results. To better equip the user to effectuate these iterations, GTSTRUDL allows for data base management via the changes, deletions, and additions commands. The above pipe analysis may be used as an introductory demonstration of how the management system operates.

The main consideration in the above data base management example was the manipulation and rearrangement of problem constraints from a pressure loaded pipe with supports at the ends to an end loaded pipe with supports at the center section. Due to the new support location and orientation, high stress concentrations will be formed within the center elements, thus necessitating finer mesh in this region. The following paragraphs explain how this process was carried out.

STRUCTURAL GEOMETRY: The support conditions exist as a four inch connection between the bottom center of the pipe and the ground. With this in mind it was decided that six inches of fine mesh would be sufficient to encompass the high stress gradients resulting from these supports. Thus it was necessary to delete two rows of three inch elements from 51 to 70 and replace these with six rows of one inch elements.

The generation of this finer mesh was performed as before except

that the initial joint ID was 131 and the increment of each repetition (LX) was one inch. Additionally elements 130 and 180 had to be input manually due to their nonuniform numbering sequence.

STRUCTURAL COMPONENT CHARACTERISTICS: All element properties remained unchanged as 'SBHQ'. However support status must be redesignated for all joints using the status command as indicated. Notice that status, both free and support, may be imposed by increments.

LOADING: It was then necessary to load the fine mesh structure in the negative X-direction using the joint load command. A value of 10 kips at each joint (100 kips total) was selected as an initial value for the determination of first yield and proved in retrospect to produce zero percent plasticity; thus a second value, as before, should be attempted.

GRAPHICAL OUTPUT: It was then necessary to create a Neutral Plot File containing the following plot images:

- 1) PIPE: JOINTS OMITTED
- 2) PIPE: ELEMENTS OMITTED
- 3) PIPE2: JOINTS OMITTED
- 4) PIPE2: ELEMENTS OMITTED
- 5) PIPE2: WINDOW WITH JOINTS OMITTED
- 6) PIPE2: CUT WINDOW WITH ELEMENTS OMITTED

Due to the placement of graphical output at the end of the GTSTRUDL program it is necessary to restore the original pipe with uniform mesh. The boundary shrink command is used for the purpose of examining each element individually as described in CHAPTER 2. Also for clarity and understanding of the joint and element placement, a rotation of 0, -40, and 20 were implemented in the Z, Y and X directions respectively, along with

the removal of hidden lines. Additionally it was essential that the plot image fit within the constraints of 8.5 x 11 inch paper. To accomplish this the scale command was executed with a length stipulation of four. This causes graphical output to be four units per inch; the currently active units being inches. It should be noted that each time the plot projection command is issued another plot image is stored in the neutral plot file, thus two files have been stored so far for pipe.

Next a plot of the three dimensional fine mesh is desired hence the restoration of PIPE2 is required. Similar to the above procedure, a plot of joints and elements using the same geometric rotation was carried out. Finally a window on the fine mesh section is plotted using the limits command. The specification of this command requires the stipulation of joints, elements, and/or coordinates as limiting factors for the window. In this case joints 41 and 81 were chosen as the minimum and maximum values for the plot window respectively. Using the same window and rotation, a section of elements were removed for the viewing of the structure's interior and posterior.

CHAPTER V

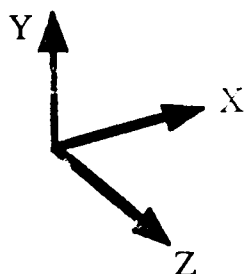
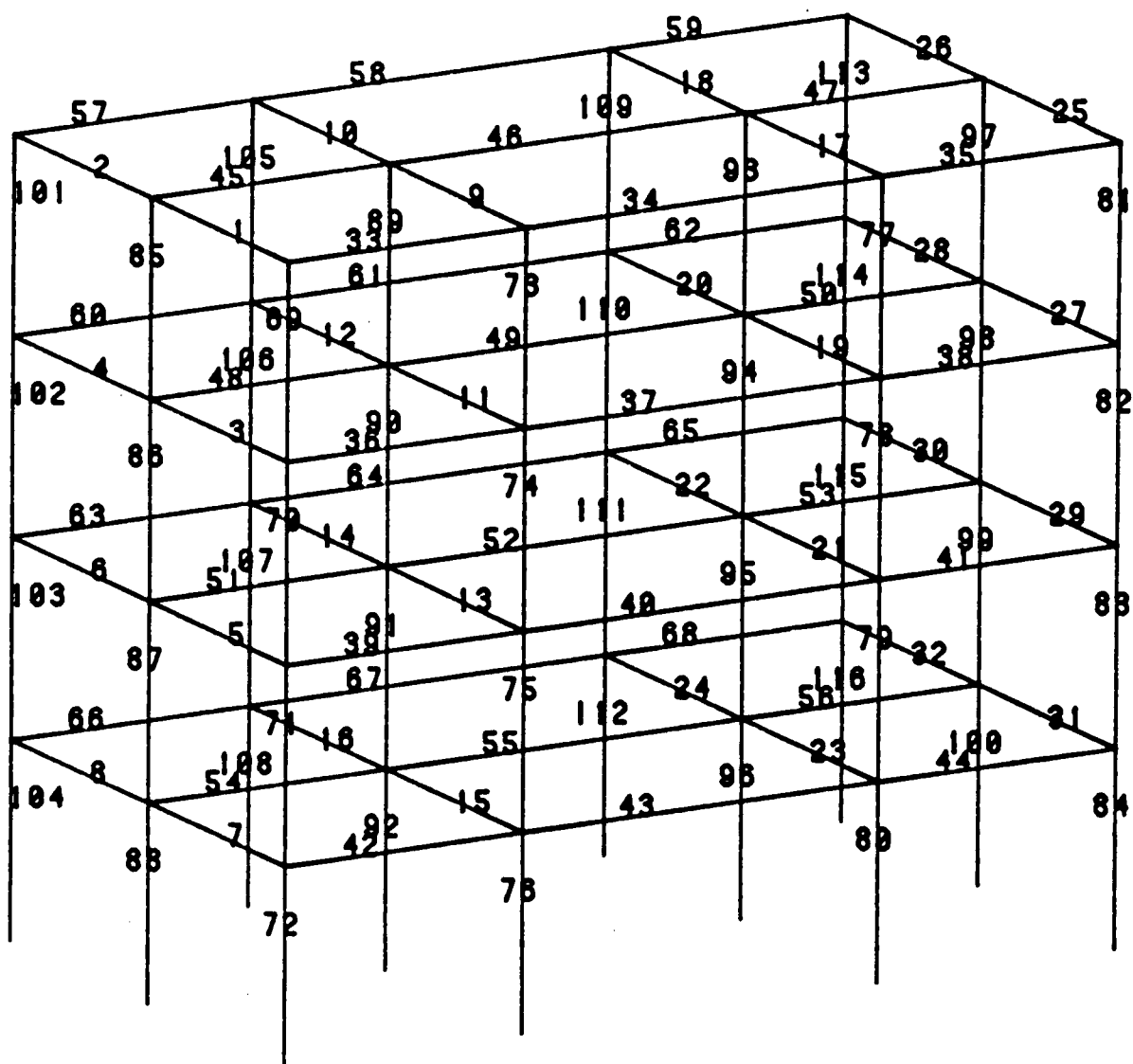
SPACE FRAME ANALYSIS

The structures considered within this text have become progressively more complex, from the PLANE TRUSS with two kinematic degrees of freedom at each joint to the present space frame with six. The degrees of freedom that must be considered, as listed in table 3.2 , are axial, torsion, two shears and two moments.

The structure that will be analyzed in the following example is a four-story space frame with three bents and two bays. A calculated wind load from the Standard Building Code will be applied from which GTSTRUDL will perform a stiffness analysis and choose the appropriate beams and columns from the 1978 AISC code. The structure will then be braced using the DATA BASE MANAGEMENT functions and reanalyzed for beam and column size. Finally both structures will be subjected to a dynamic analysis and the resulting mode shapes compared.

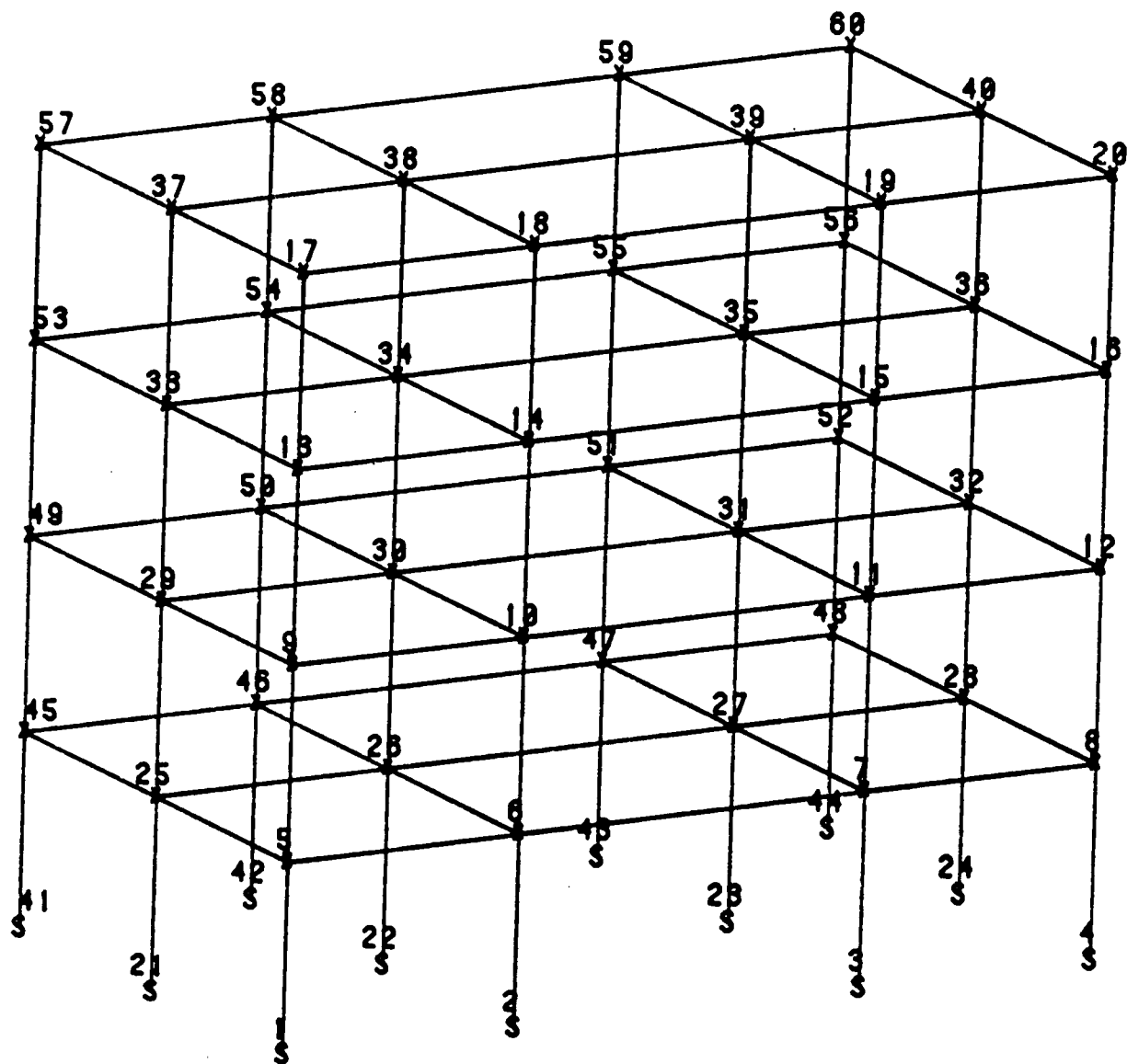
SPACE FRAME EXAMPLE

```
STRUDL '4 STORY SPACE FRAME ' 'DESIGN FOR WIND AND EARTH'
TYPE SPACE FRAME
$
$   STRUCTURAL   GEOMETRY
$
$   SEE FIGURES 5-1 A and B
$
UNITS KIP FEET
GENERA 3 JOIN ID 1 20 Z 0 -20
REPEA 4 ID 4 Y 15
REPEA 1 ID 1 X 20
GENER 3 JOIN ID 3 20 Z 0 -20
REPEA 4 ID 4 Y 15
REPEA 1 ID 1 X 20
$
```



a. Members

FIGURE 5-1. Unbraced Space Frame



b. Joints

FIGURE 5-1. (Continued)

```

$
$      GENERATE THE HORIZONTAL ( Z-DIR ) MEMBERS ( BEAMS )
$
$      4 ROWS OF 8 = 32 X - DIR BEAMS
$
GENER 2 MEMB ID 1 1 FROM 17 20 TO 37
REPEA 3 ID 2 F -4
REPEA 1 ID 8 F 1
$
GENER 2 MEMB ID 17 1 FROM 19 20 TO 39
REPEA 3 ID 2 F -4
REPEA 1 ID 8 F 1
$
$      GENERATE HORIZONTAL ( X-DIR ) BEAMS
$      3 ROWS OF 12 = 36 X-DIR (BEAMS)
$
GENER 1 MEMB ID 33 FROM 17 1 TO 18
REPEA 3 ID 3 F -4
REPEA 2 ID 12 F 20
$
GENER 1 MEMB ID 34 FROM 18 1 TO 19
REPEA 3 ID 3 F -4
REPEA 2 ID 12 F 20
$
GENER 1 MEMB ID 35 FROM 19 TO 20
REPE 3 ID 3 F-4
REPEA 2 ID 12 F 20
$
$      GENERATE THE COLUMNS ( Y-DIR )
$      3 ROWS OF 16 =48 COLUMNS
$
GENER 4 MEMB ID 69 FROM 17 -4 TO 13
REPEA 2 ID 16 F 20
REPEA 1 ID 4 F 1
$
GENERA 4 MEMB ID 77 FROM 19 -4 TO 15
REPEA 2 ID 16 F 20
REPEA 1 ID 4 F 1
$
$
$      BRACING FOR WIND
$
$      SEE FIGURE 5-2
$
MEMB INCID
$
$      FRONT LEFT
$

```

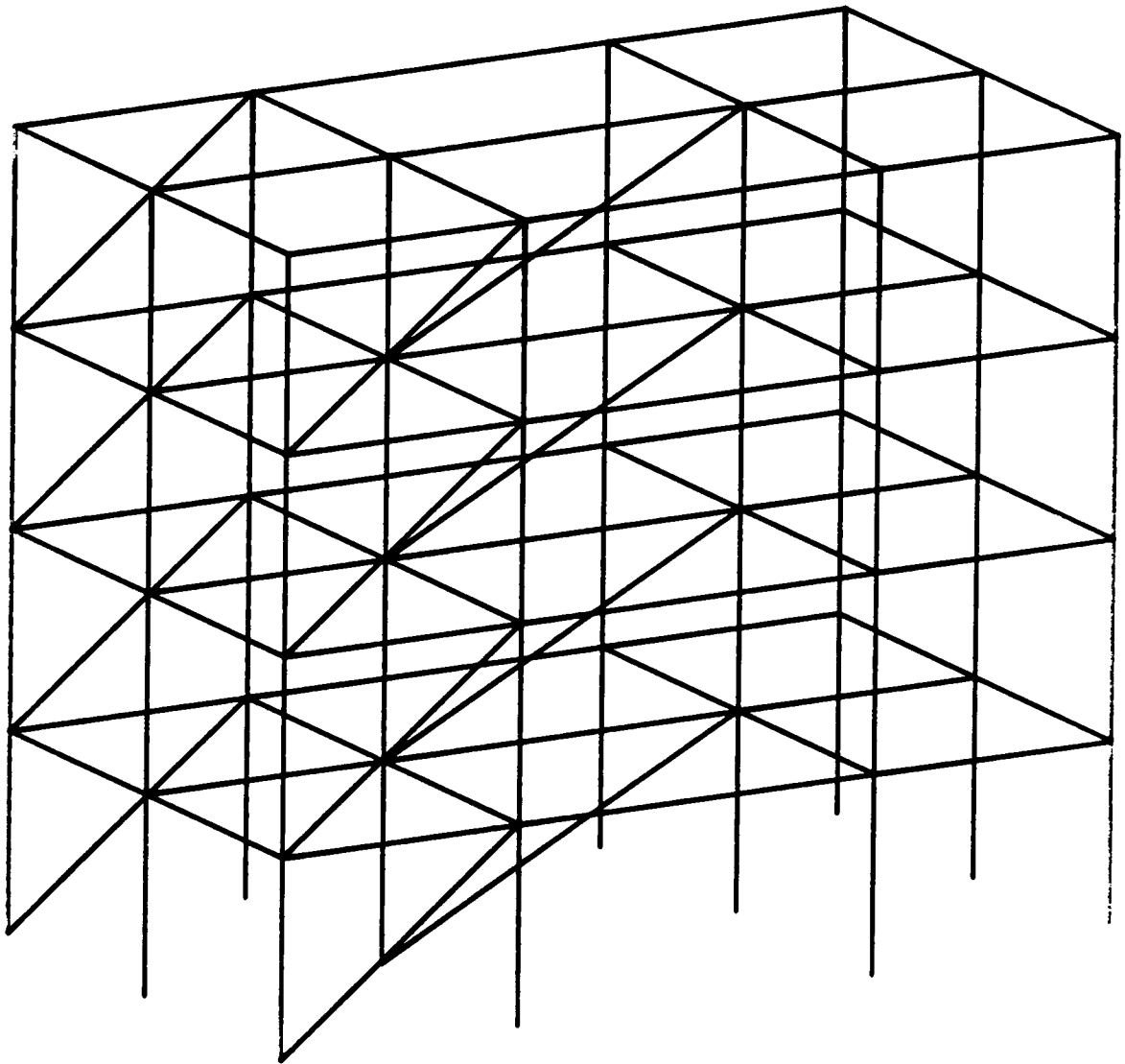


FIGURE 5-2. *Braced Space Frame*

117 1 2
 118 5 10
 119 9 14
 120 13 18
 \$
 \$ CENTER
 \$
 121 22 27
 122 26 31
 123 30 35
 124 34 39
 \$
 \$ BACK LEFT
 \$
 125 41 46
 126 55 50
 127 49 54
 128 53 58
 \$
 \$
 \$ STRUCTURAL COMPONENT CHARACTERISTICS
 \$
 STATUS SUPPORT 1 TO 4 21 TO 24 41 TO 44
 \$
 MEMB RELEASES
 1 TO 68 START MOM Z Y Z END MOM X Y Z
 117 TO 128 START MOM X Y Z END MOM X Y Z
 UNITS KIP INCHES
 CONSTANTS
 MATERIAL STEEL
 MEMB PROP TABLE `STEELW78'
 1 TO 116 TABLE `W16X50'
 117 TO 128 TABLE `W4X13'
 UNIT FEET
 \$
 \$ LOADING
 \$
 \$ SEE FIGURE 5-3
 \$
 LOAD 1 `COLUMN WIND LOAD'
 MEMB LOADS
 71 72 103 104 FORCE X GLOB UNI W 0.116 LA 0 LB 15
 70 102 FORC X GLOB UNI W 0.16 LA 0 LB 15
 69 101 FORC X GLOB UNI W 0.16 LA 0 LB 5
 69 101 FORC X GLOB UNI W 0.192 LA 5 LB 15
 \$
 \$ CENTER SECTION WINDWARD
 \$

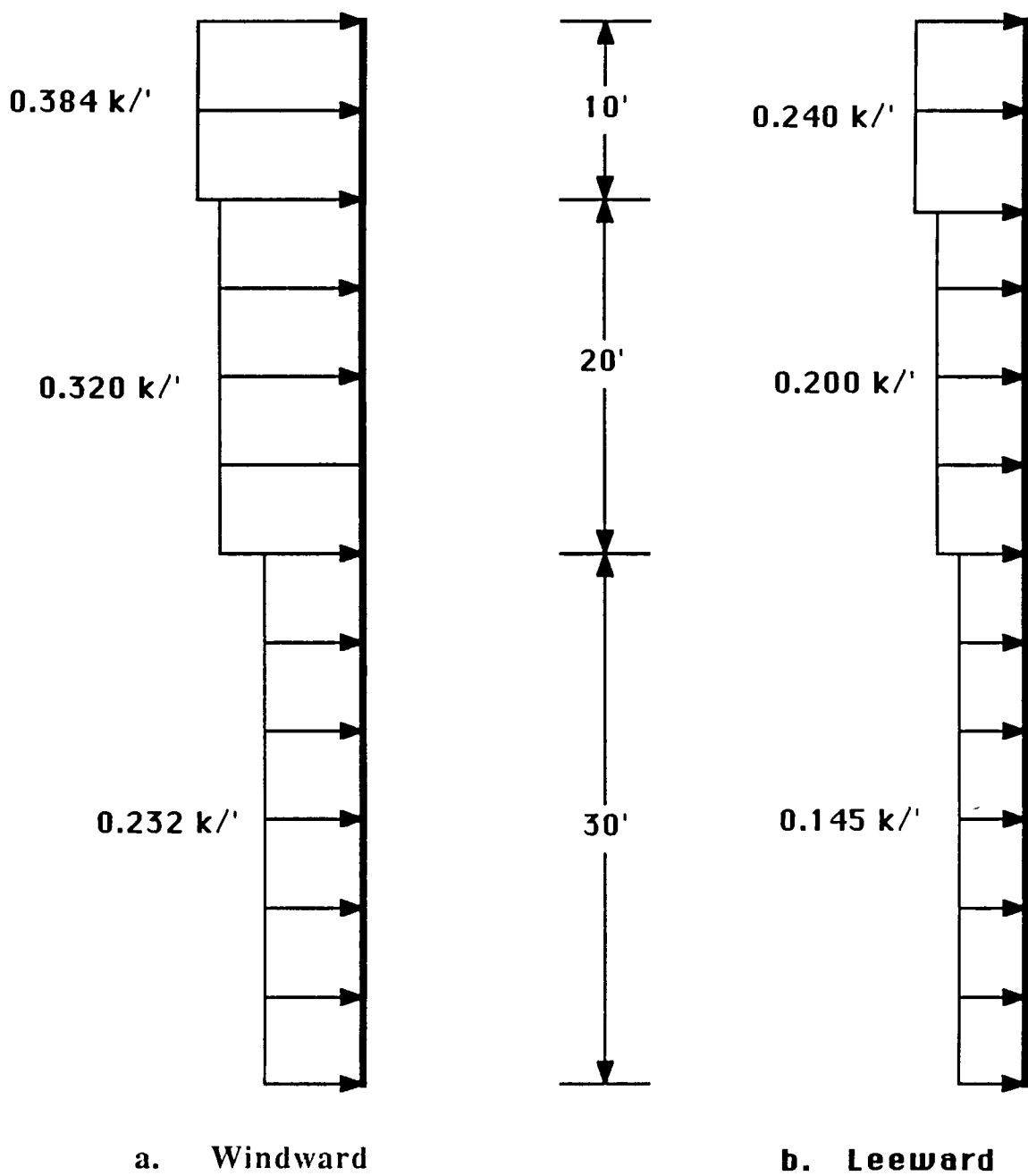
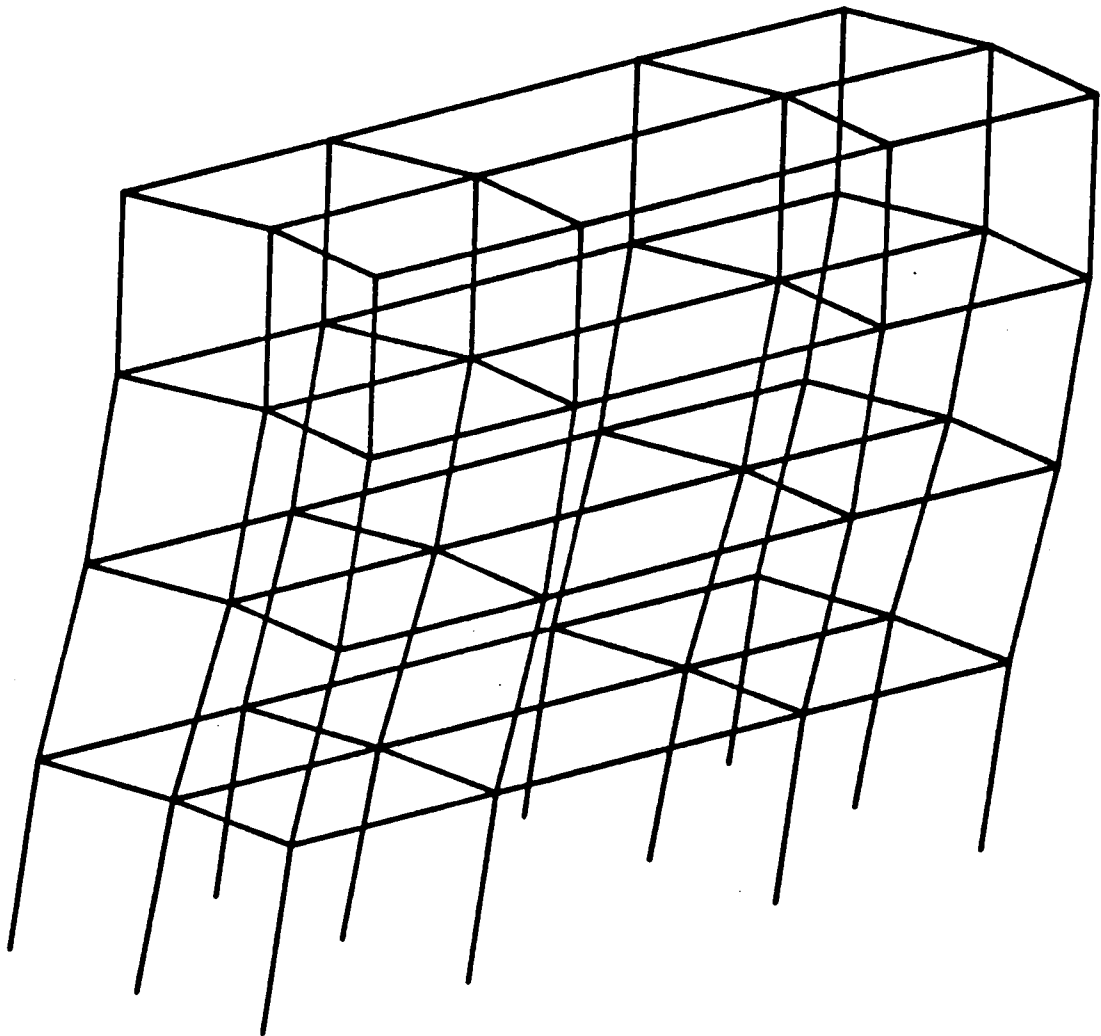


FIGURE 5-3. Wind Loading [center section]

```

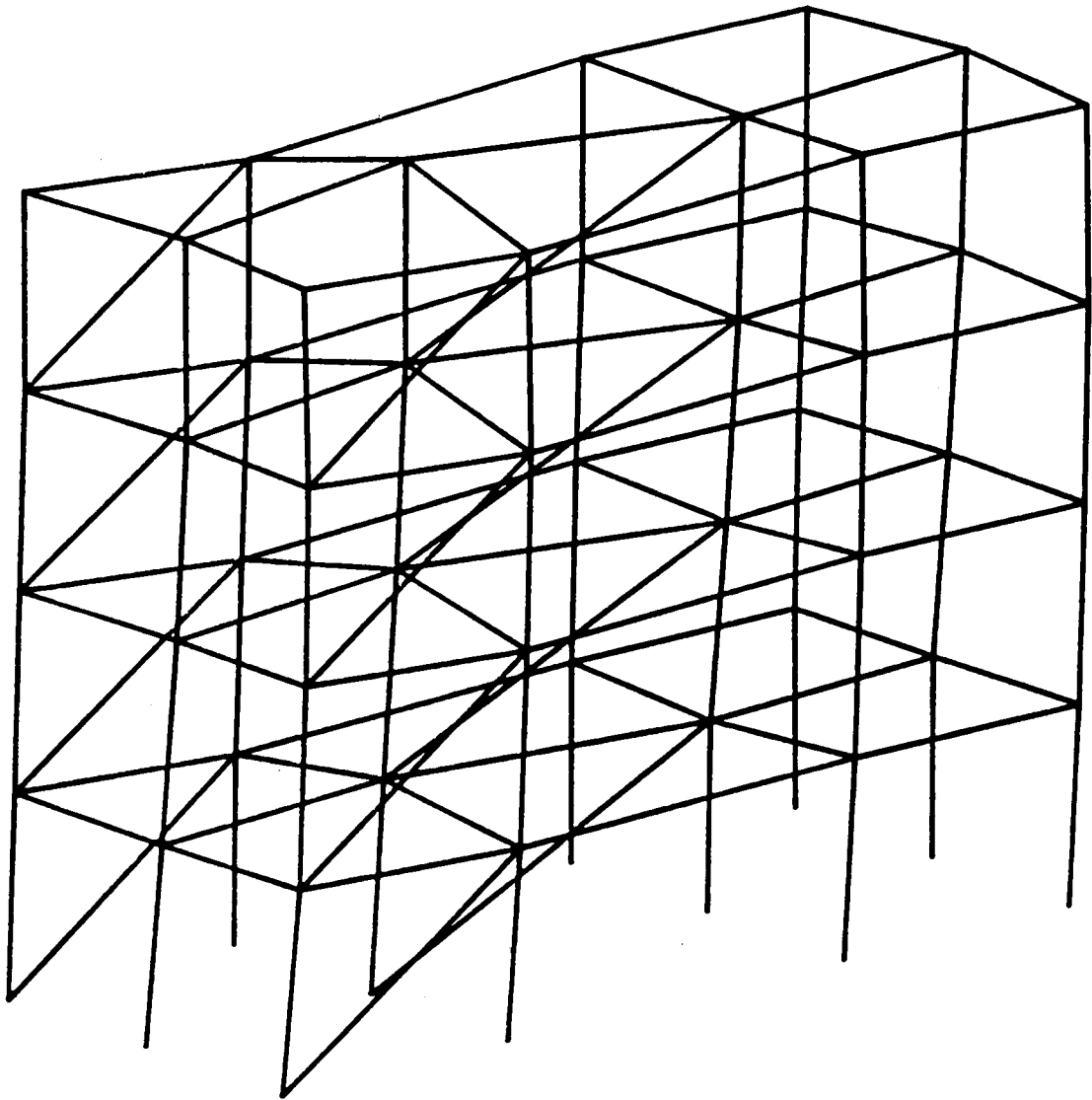
87 88 FORC X GLOB UNI W 0.232 LA 0 LB 15
86 FORC X GLOB UNI W 0.32 LA 0 LB 15
85 FORC X GLOB UNI W 0.384 LA 0 LB 5
85 FORC X GLOB UNI W 0.384 LA 5 LB 15
$
$      END SECTIONS (LEEWARD)
$
83 84 115 116 FORC GLOB UNI W 0.73 LA 0 LB 15
114 82 FORC X GLOB UNI W 0.1 LA 0 LB 15
81 113 FORC X GLOB UNI W 0.1 LA 0 LB 5
81 113 FORC X GLOB UNI W 0.12 LA 5 LB 15
$
$      CENTER SECTION (LEEWARD)
$
99 100 FORC GLOB UNI W 0.145 LA 0 LB 15
98 FORC X GLOB UNI W 0.2 LA 0 LB 15
97 FORC X GLOB UNI W 0.2 ;A 0 LB 5
97 FORC X GLOB UNI W 0.24 LA 5 LB 15
$
$      ROOF FRONT AND BACK
$
33 57 FOR Y GLOB UNI W 0.24 LA 0 LB 20
34 58 FORC Y GLOB UNI W 0.24 LA 0 LB 3.3
34 58 FORC Y GLOB UNI W 0.18 LA 3.3 LB 30
35 59 FORC Y GLOB UNI W 0.18 LA 0 LB 20
$
$      ROOF MIDDLE SECTION
$
45 FORC Y GLOB UNI W 0.480 LA 0 LB 20
46 FORC Y GLOB UNI W 0.480 LA 0 LB 3.3
46 FORC Y GLOB UNI W 0.36 LA 3.3 LB 30
47 FORC Y GLOB UNI W 0.36 LA 0 LB 20
$
$      DEAD LOAD MUST BE CHANGED AFTER THE 'SELECT'
$      AND 'TAKE' COMMAND
$
DEAD LOAD 2 'BEAM AND COLUMN WEIGHT' DIR -Y
$
SAVE 'SPACE1'
$
$      ANALYSIS
$
LOAD LIST ALL
STIFFNESS ANALYSIS REDUCE BAND
SELECT MEMB 1 TO 128 WITH 'GENERAL ' USE CODE '78AISC'
$
$      CHOOSE THE BEAM SECTIONS
$

```

a. *Unbraced*

FIGURE 5-4. *Deflected Shape due to Wind*



b. *Braced*

FIGURE 5-4. (Continued)

\$
RESTORE 'SPACE1'
\$

\$ STRUCTURAL GEOMETRY AND CHARACTERISTICS

\$
DELETIONS
MEMB 117 TO 128
MEMBER RELEASES
ADDITIONS

\$
\$ ANALYSIS

\$
STIFFN ANA
SELECT MEMB 1 TO 128 WITH 'GENERAL ' USE CODE `78AISC'

\$
\$ CHOOSE THE BEAM SECTIONS

\$
TAKE MEMB 1 2 9 10 17 18 25 26 33 TO 3 45 TO 47 57 TO 59-
LARG 'SZ' OF MEMB 1 2 9 10 17 18 25 26 33 TO 35-
45 TO 47 57 TO 59

\$
TAKE MEMB 3 4 11 12 19 20 27 28 36 TO 38 48 TO 50 60 TO 62-
LARG 'SZ' OF MEMB 3 4 11 12 19 20 27 28 36 TO 38-
48 TO 50 60 TO 62

\$
TAKE MEMB 5 6 13 14 21 22 29 30 39 TO 41 51 TO 52 63 TO 56-
LARG 'SZ' MEMB 5 6 13 14 21 22 29 30 39 TO 41 51-
TO 53 63 TO 65

\$
TAKE MEMB 7 8 15 16 23 24 31 32 42 TO 44 54 TO 56 66 TO 68-
LARG 'SZ' OF MEMB 7 8 15 16 23 24 31 32 42 TO 44-
54 TO 56 66 TO 68

\$
\$ CHOOSE THE COLUMN SECTIONS

\$
TAKE MEMB 69 TO 72 LARGEST 'AX' OF MEMB 69 TO 72
TAKE MEMB 73 TO 76 LARGEST 'AX' OF MEMB 73 TO 76
TAKE MEMB 77 TO 80 LARGEST 'AX' OF MEMB 77 TO 80
TAKE MEMB 81 TO 84 LARGEST 'AX' OF MEMB 81 TO 84
TAKE MEMB 85 TO 88 LARGEST 'AX' OF MEMB 85 TO 88
TAKE MEMB 89 TO 92 LARGEST 'AX' OF MEMB 89 TO 92
TAKE MEMB 93 TO 96 LARGEST 'AX' OF MEMB 93 TO 96
TAKE MEMB 97 TO 100 LARGEST 'AX' OF MEMB 97 TO 100
TAKE MEMB 101 TO 104 LARGEST 'AX' OF MEMB 101 TO 104
TAKE MEMB 105 TO 108 LARGEST 'AX' OF MEMB 105 TO 108
TAKE MEMB 109 TO 112 LARGEST 'AX' OF MEMB 109 TO 112

TAKE MEMB 113 TO 116 LARGEST 'AX' OF MEMB 113 TO 116
 \$
 CHECK MEMB 1 TO 68 AS BEAM USE CODE '78AISC'
 CHECK MEMB 69 TO 116 AS COLUMN USE CODE '78AISC'
 \$
 PRINT MEMB PROP
 STEEL TAKEOFF ITEMIZE
 STIFFN ANALYS
 \$
 \$ TEXT OUTPUT
 \$
 LIST REACT FORCE DISP ALL \$ SEE FIGURE 5-4b
 SAVE 'SPACE3'
 \$
 \$ APPLY THE DYNAMIC LOADING TO THE BRACED STRUCTURE.
 \$
 RESTORE 'SPACE2'
 DELETIONS
 LOAD 1 2
 ADDITIONS
 DYNAMIC DEGREES OF FREEDOM WITH STATIC CONDENSATION
 GENERATE FROM TRANS X
 INERTIA OF JOINTS LUMPED
 DYNAMIC ANALYSIS MODAL REDUCE BAND
 LIST DYNAMIC EIGENVECTORS 1
 SAVE 'SPACE4'
 \$
 \$ APPLY THE DYNAMIC LOADING TO THE UNBRACED STRUCTURE.
 \$
 \$ SEE FIGURES 5-5 TO 5-9
 \$
 RESTORE 'SPACE3'
 DELETIONS
 LOAD 1 2
 ADDITIONS
 DYNAMIC DEGREES OF FREEDOM WITH STATIC CONDENSATION
 GENERATE FROM TRANS X
 INERTIA OF JOINTS LUMPED
 DYNAMIC ANALYSIS MODAL REDUCE BAND
 LIST DYNAMIC EIGENVECTORS EIGENVALUES 1
 \$
 \$ GRAPHICAL OUTPUT
 \$
 \$ MODE SHAPES UNBRACED: SEE FIGURE 5-5 TO 5-9
 \$
 PLOT DEVICE PLOTTER NEUTRAL LENGTH 7 WIDTH 5
 PLOT FORMAT OMIT ALL NAMES
 PLOT FORMAT TITLE 'MODE SHAPE NUMBER 1 FOR BRACED STRUCT.'

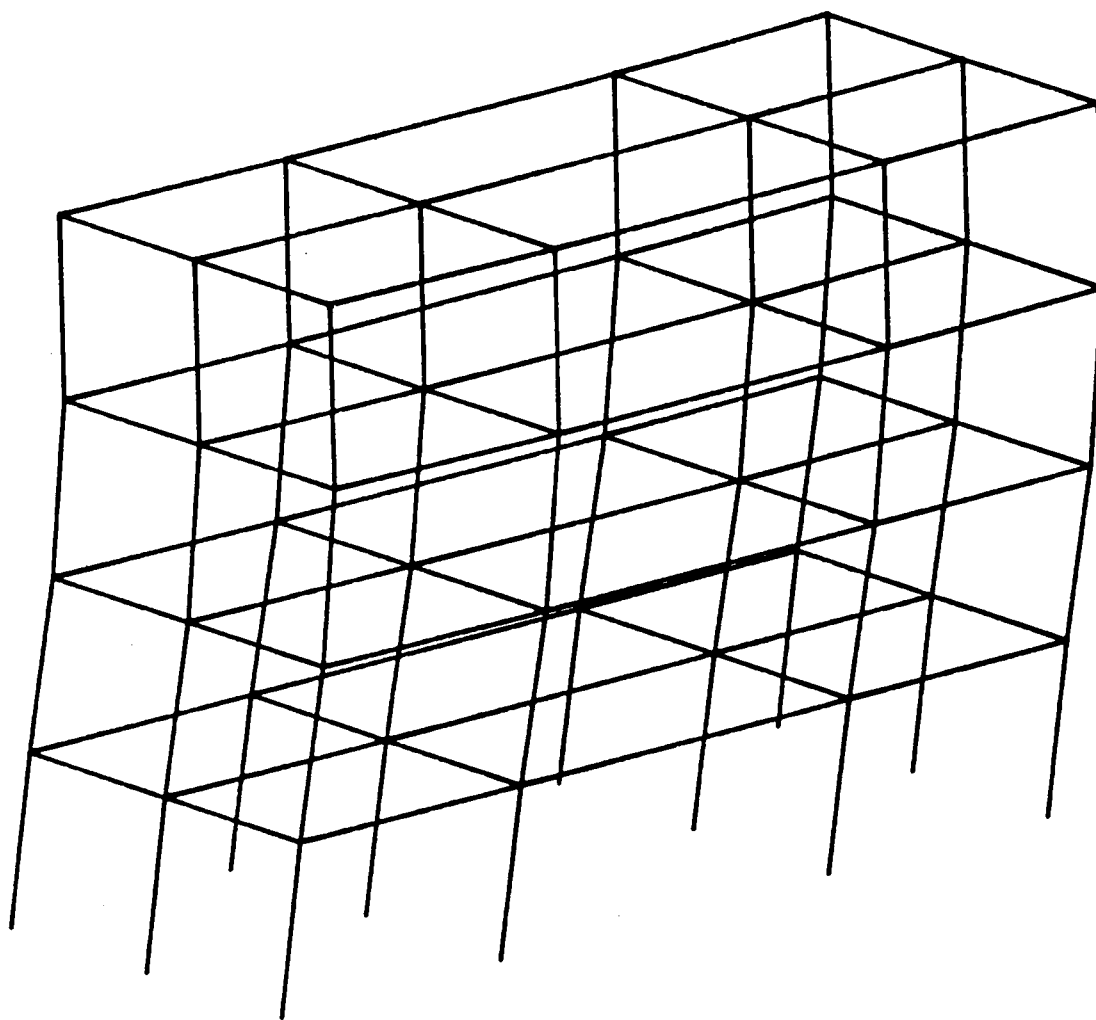


FIGURE 5-5. *Mode # 1 [unbraced]*

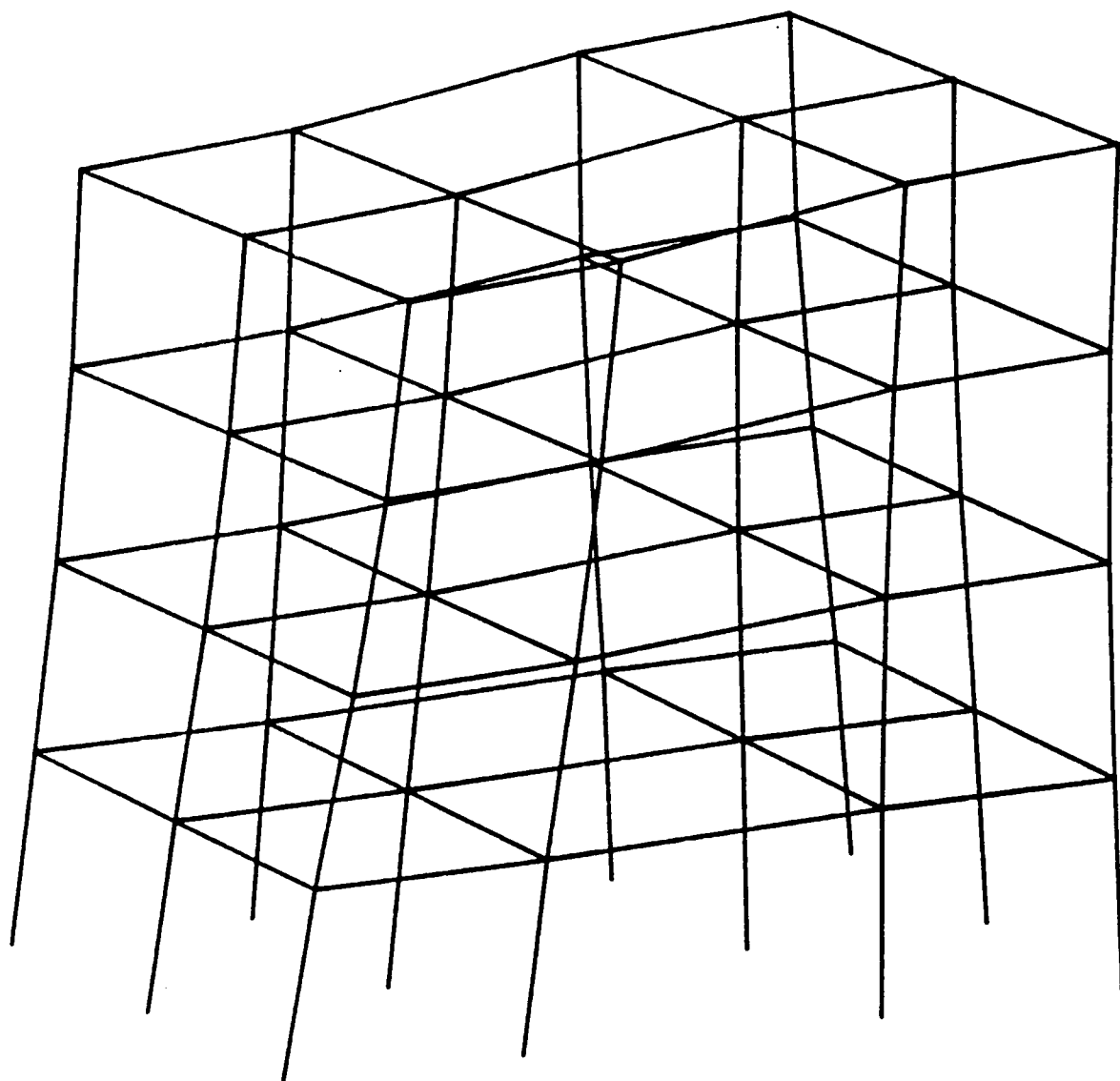


FIGURE 5-6. Mode # 2 [unbraced]

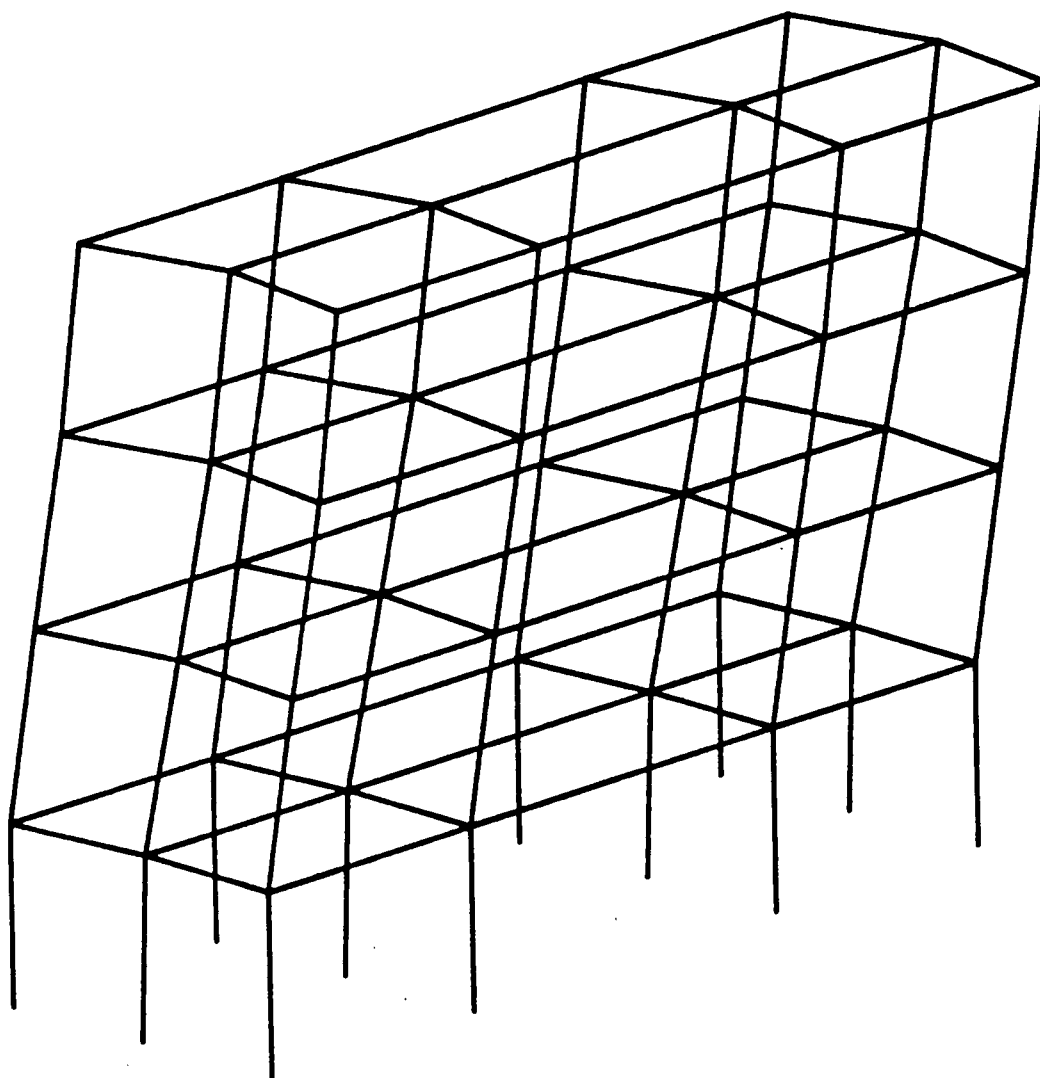


FIGURE 5-7. Mode # 3 [unbraced]

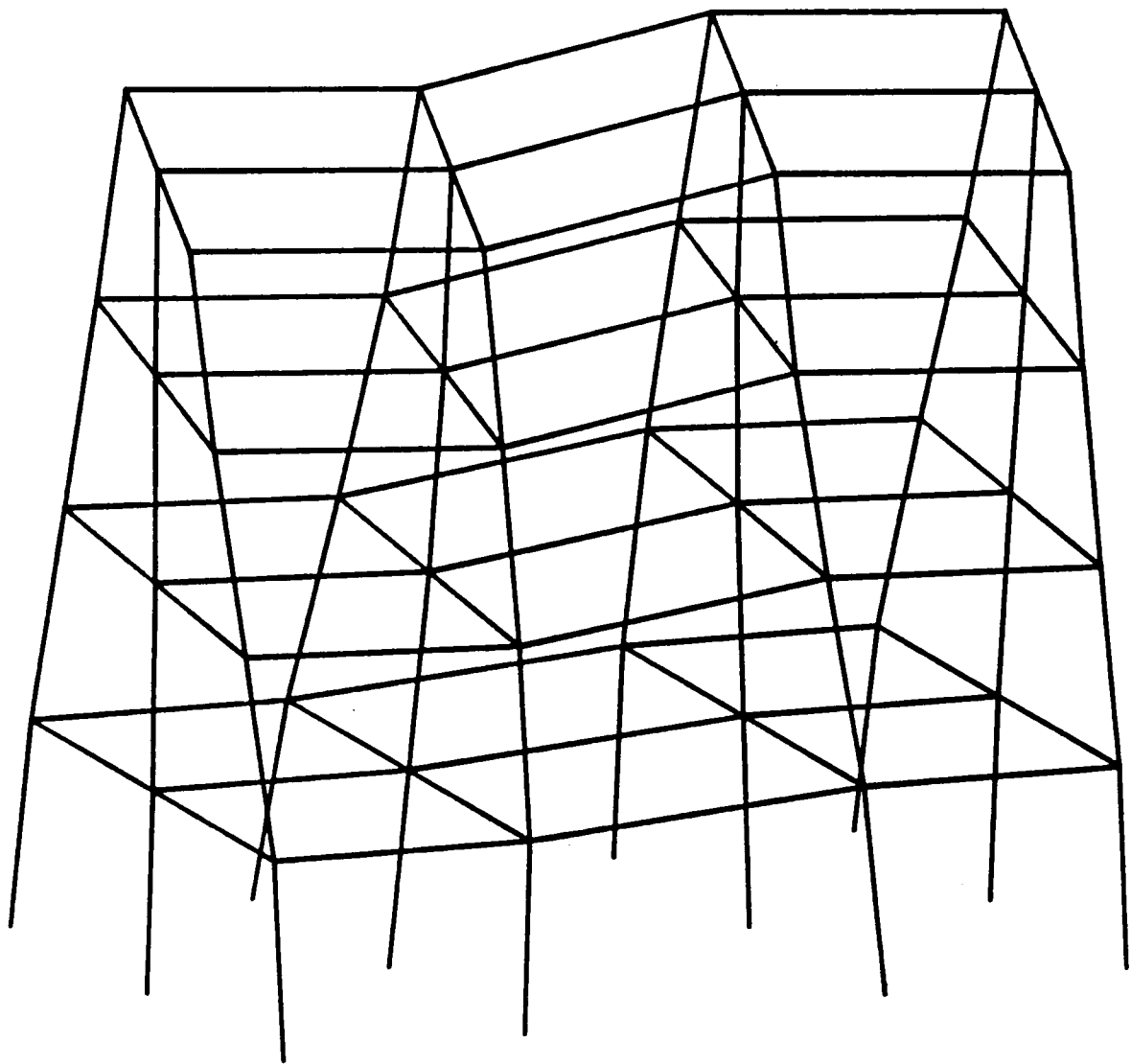


FIGURE 5-8. Mode # 4 [unbraced]

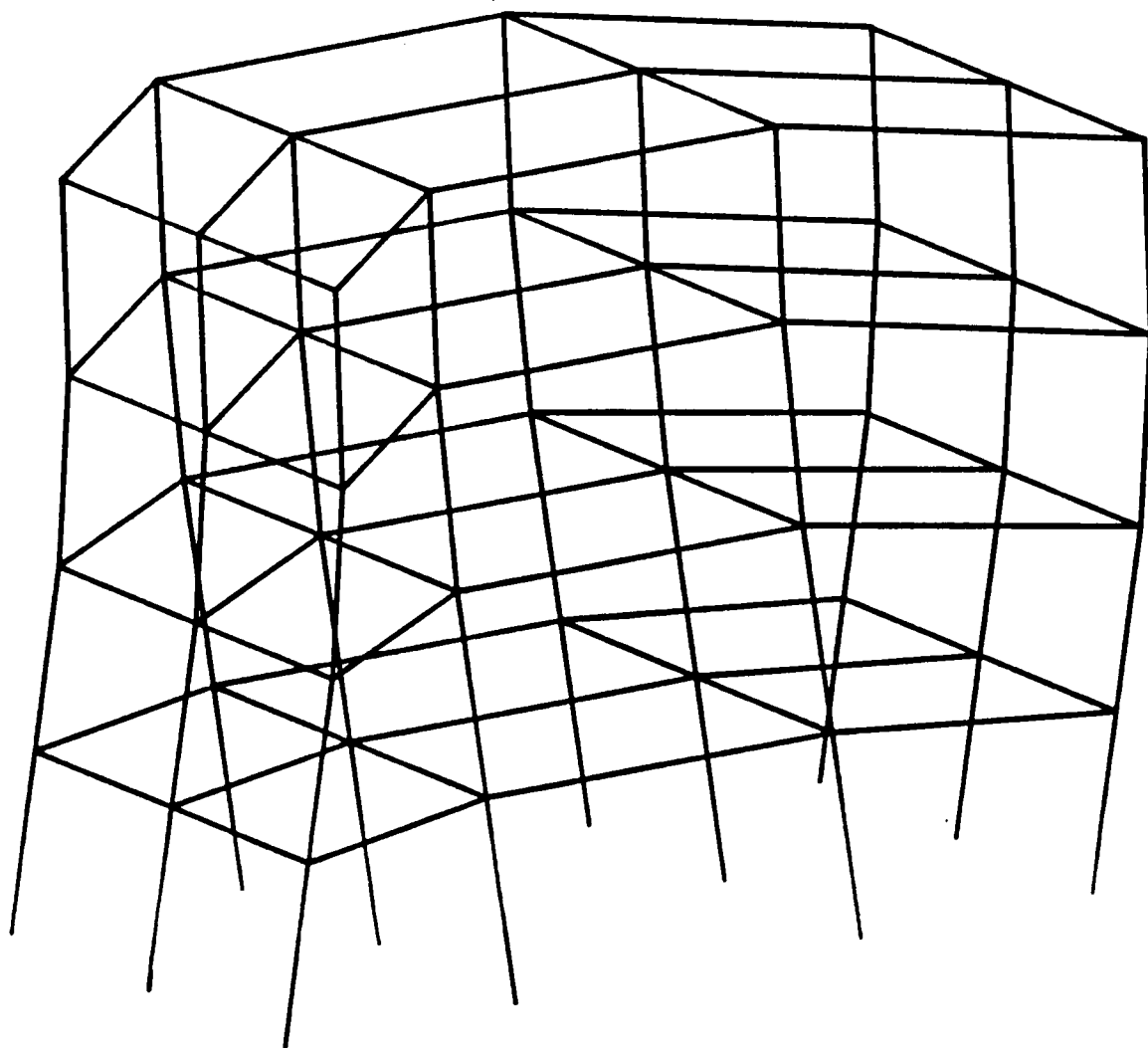


FIGURE 5-9. *Mode # 5 [unbraced]*

PLOT FORMAT SIZE CHAR 3
 PLOT PROJ MODE SHAPE NUM 1 MAG FR 0.2 R X 15 Y 30
 \$
 \$ ACTUAL STRUCTURE SHAPE BRACED
 \$
 \$ SEE FIGURE 5-2
 \$
 RESTORE 'SPACE2'
 PLOT DEVICE PLOTTER NEUTRAL LENGTH 7 WIDTH 5
 PLOT FORMAT OMIT JOINT NAMES
 PLOT FORMAT TITLE 'BRACED STRUCTURE'
 PLOT FORMAT SIZE CHAR 3
 PLOT PROJ R X 15 Y 30
 PLOT FORMAT OMIT MEM NAMES
 PLOT PROJ R X 15 Y 30
 \$
 \$ DEFLECTED SHAPE DUE TO WIND (BRACED)
 \$
 \$ SEE FIGURE 5-4 B
 \$
 PLOT FORMAT OMIT ALL NAMES
 PLOT PROJ DEFORMED STRUCT LOAD ALL MAG FRA 0.2 R X 15 Y 30
 \$
 \$ ACTUAL SHAPE (UNBRACED)
 \$
 \$ SEE FIGURE 5-1 A and B
 \$
 PLOT FORMAT OMIT JOINT NAMES
 PLOT PROJ R X 15 Y 30
 \$
 \$ DEFLECTED SHAPE DUE TO WIND (UNBRACED)
 \$
 \$ SEE FIGURE 5-4 A
 \$
 PLOT FORMAT OMIT ALL NAMES
 PLOT PROJ DEFORM STRUC LOADING ALL
 \$
 \$ MODE SHAPES FOR BRACED STRUCTURE
 \$
 \$ SEE FIGURES 5-10 TO 5-12
 \$
 RESTORE 'SPACE4'
 PLOT FORMAT OMIT ALL NAMES
 PLOT PROJ MODE SHAPE NUM 1 R X 15 Y 30
 FINISH

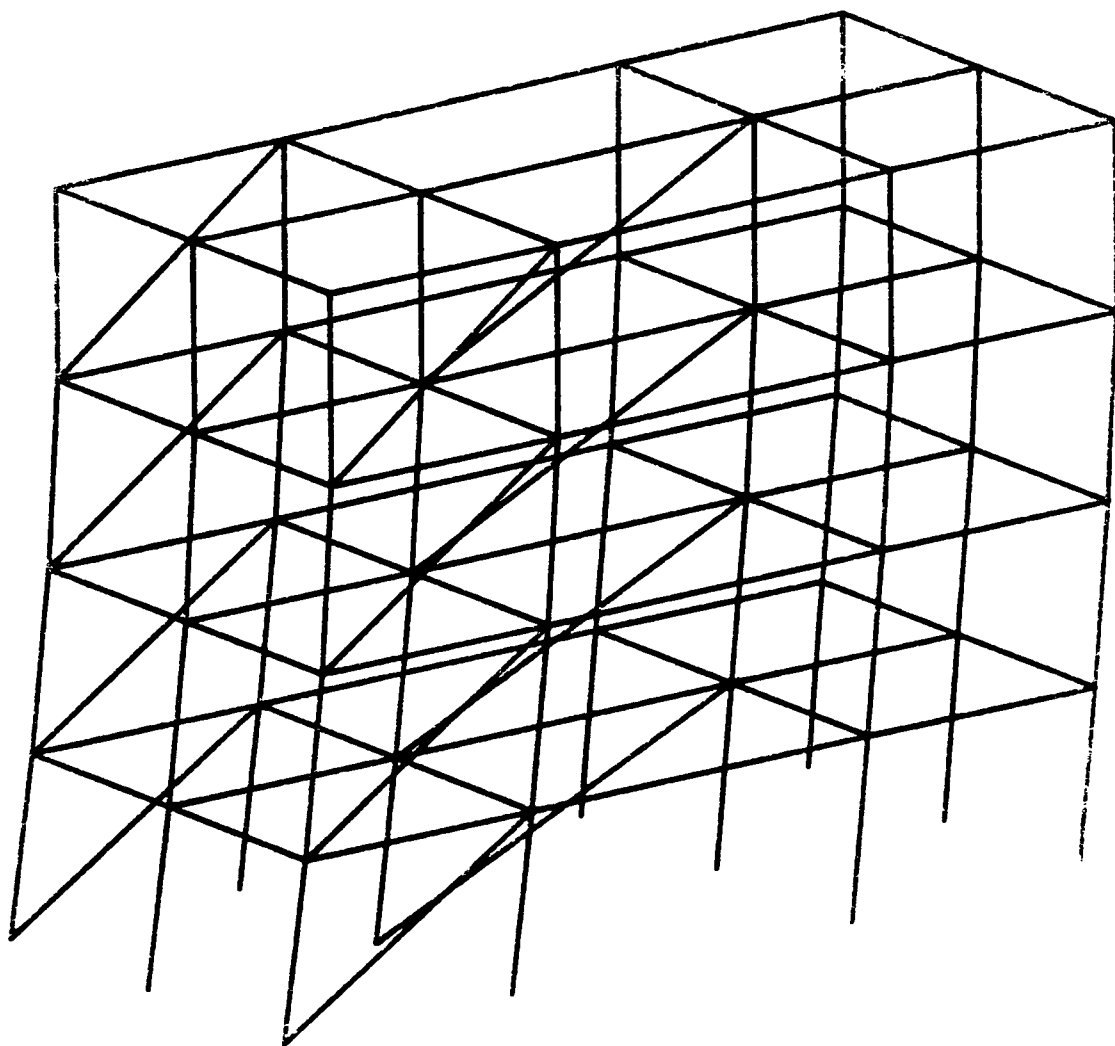


FIGURE 5-10. *Mode #1 [Braced]*

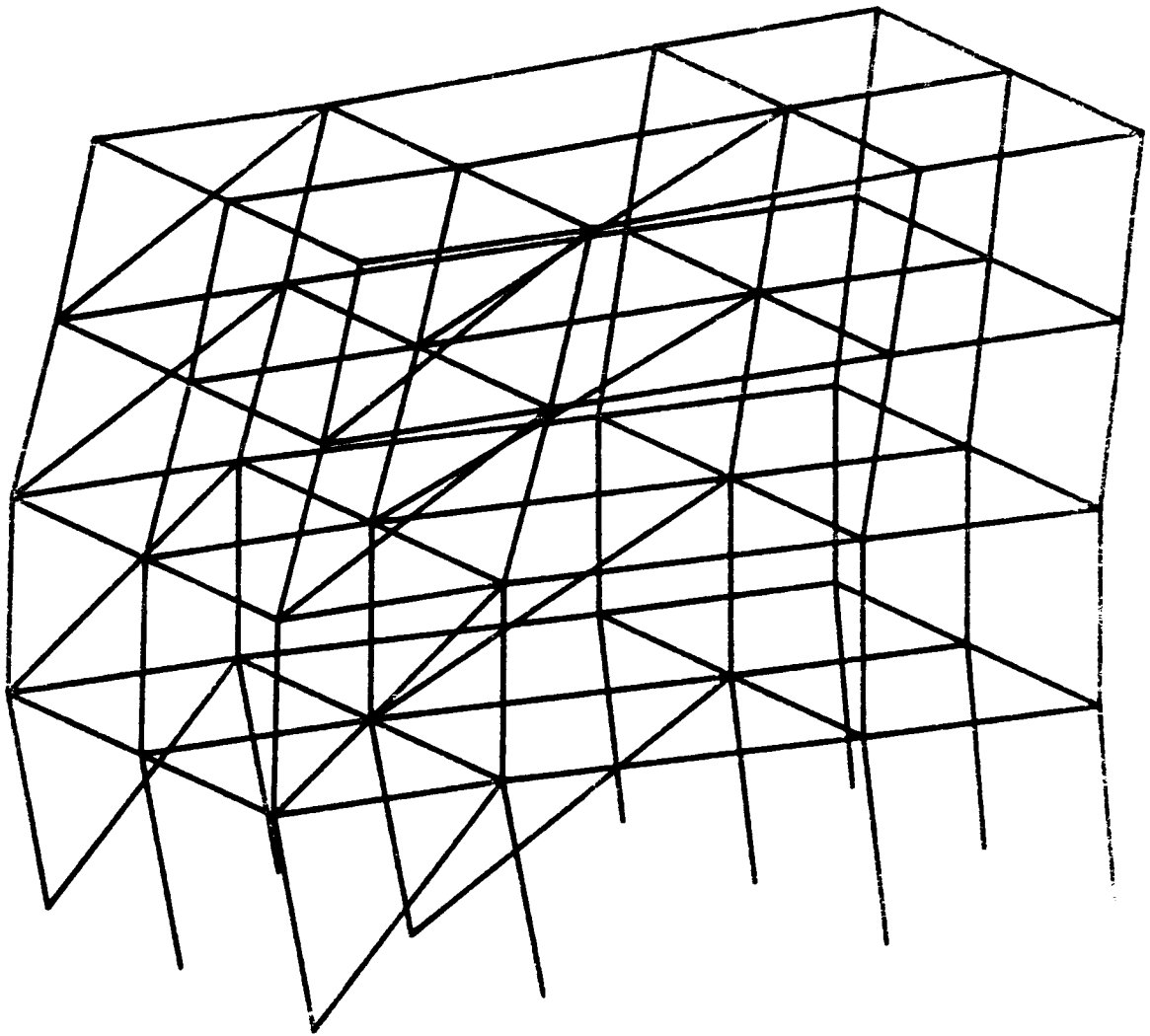


FIGURE 5-11. Mode #3 [Braced]

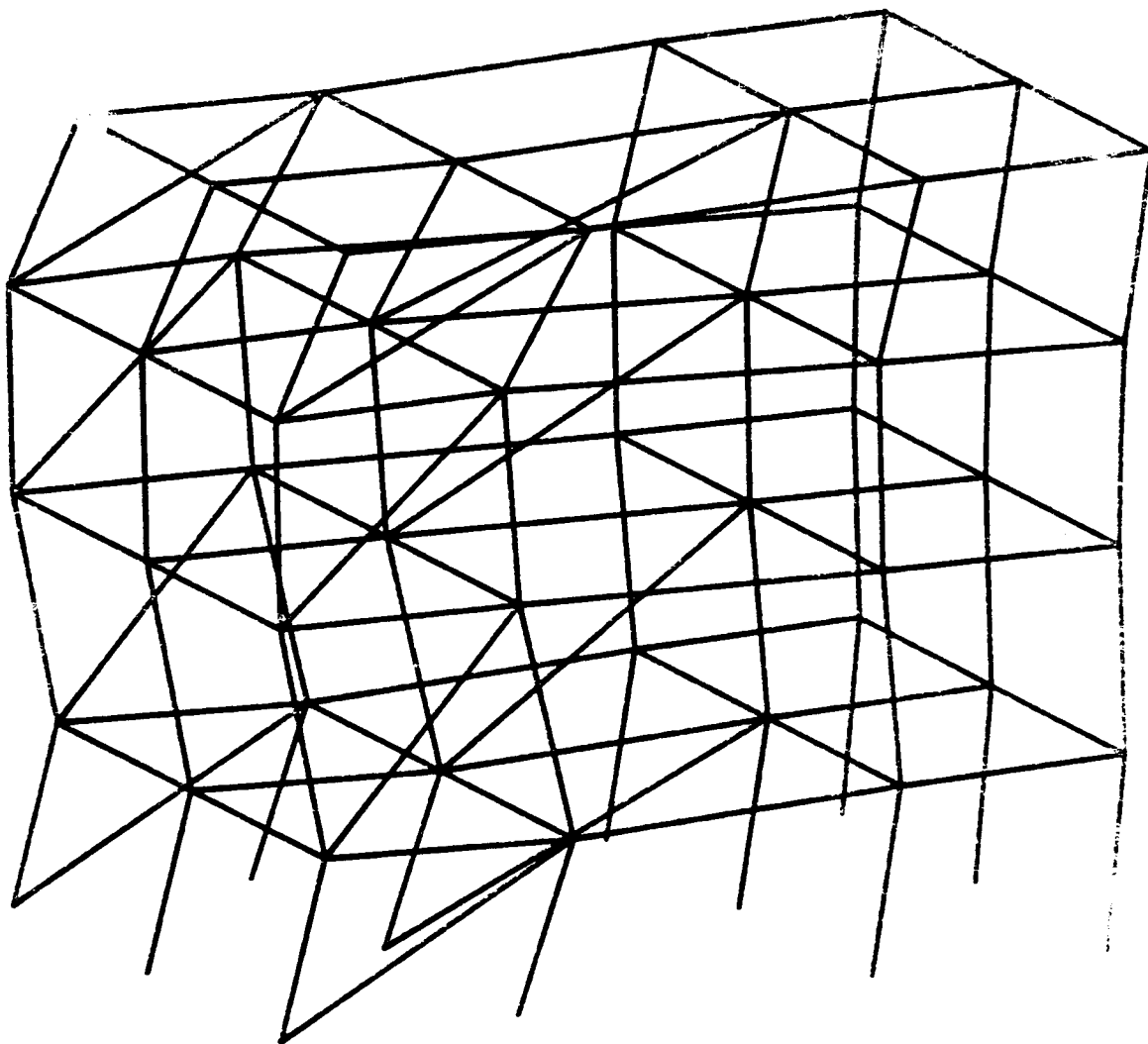


FIGURE 5-12. Mode #5[Braced]

STRUCTURAL GEOMETRY: As is evident from the symmetrical nature of this frame, generation of the joints and elements is the expedient method of geometry input. Generation of the joints begins at joint one located on the bottom floor of the left (windward) side of the structure, and continues in the negative Z-direction in both numbering and dimensional increments of 20 (See FIGURE 5-1b). This process is repeated four subsequent times in the positive Y-direction (vertical) with numbering increments of four and dimensional increments of 15 feet. Similarly it is repeated once in the X-direction with numbering and dimensional increments of 1 and 20 feet respectively. Due to the dimensional nonuniformity of the structure (i.e., three bays at 20, 30, 20 feet respectively) the methodology for the generation of joints must be repeated, as above, for the third bay with joint three as the point of origin (ID).

Once the generation of joints has been established, it is then necessary to begin the generation of members starting with the Z-direction beams on the left, roof section. Two members are generated starting with member one and incrementing the members one and the joints 20 from 17 to 37. It is important to note that following the from statement within the generate command, the beginning joint plus the increment must add up to the ending joint. The procedure is repeated three additional times in the negative Y-direction, starting at member 2, with a joint number increment of -4. As before it is repeated in the positive X-direction beginning at member 8 with a joint number increment of one. It should be noted that the joints have been previously specified, hence the direction of member generation need not be stipulated. For instance the repetition of members one and two must proceed in the negative Y-direction because a joint increment of -4 has been entered as such and the members only serve to supply the connectivity. This entire procedure is then repeated for the end bay starting at member 17.

After the Z-direction beams have been created, the adjacent, perpendicular beams in the X-direction must be formed similarly. Generation begins at member 33 and is repeated three times in the negative

Y-direction and two times in the negative Z-direction. The technique is iterated two additional times, first for the middle bay and finally for the end.

Four columns, starting at number 6, are then generated in the negative Y-direction with joint increments of -4 progressing from 17 down to the origin at 1. This is repeated twice in the negative Z-direction (joint increment of 20) and once in the positive X-direction (joint increment of 1). Finally the right-most bay columns are generated identically starting at member 77.

After the structure, in its entirety, has been input, the user is advised to examine the structure on the scope device for the purpose of eliminating any generation errors. This procedure of generation and examination should be adhered to for any complex structure due to the increasing difficulty in determining errors as the structure becomes progressively larger.

After determining that the structure is error free it is necessary to input the bracing. This operation is carried out manually, via member incidences, due to the erratic joint numbering required for the bracing connectivity. It was decided that the wind load would be applied from left to right on all windward, external columns. Hence bracing is placed on the left-front, center, and left-rear bays.

STRUCTURAL COMPONENT CHARACTERISTICS: As before the enumeration of supports, releases, constants and member properties is crucial for the accurate communication of structural behavior. In this example all ground joints were classified as supports with complete fixity against axial, shear, moment, and torsionally induced forces. All connectivity within a braced structure, other than column to column, is by pins, hence it is necessary to release all moments originally fixed at the start and end of all beams and bracing members. Space Frame type structures, as mentioned earlier, have degrees of freedom consisting of three forces and three moments, thus the release of moments Y, and Z must be made to produce idealized pin connections. The release of torsional moment

resistance as developed about a member's longitudinal axis is prohibitive as detailed in section 1.10.11 of the AISC code. Thus fixity about the local X-axis is left intact. It should be noted, however, that the release of moment Y will have little or no significance due to the primary load (wind) causing rotation solely about the Z-axis.

One of the primary goals of this analysis is to allow GTSTRUDL to select the sizing of members necessary to resist the live load. Therefore the specification of constants will be covered by the material steel command and likewise beams, columns, and bracing will be picked arbitrarily from the 1978 AISC code and later changed as GTSTRUDL deems appropriate.

LOADING: The loads placed on this structure were as dictated in the Standard Building Code for wind loads in the Knoxville area as follows:

1) **SPEED = 85 MPH** (FIGURE 1205.1)

2) **WIND PRESSURE**

<u>HEIGHT(FT)</u>	<u>q (PSF)</u>
0 TO 30	14.5
31 TO 50	20.0
51 TO 60	24.0

3) **FACTORS**

For a rectangular, prismatic structure:

FACTOR = 1.3 = +0.8 (windward), 0.5 (leeward)

4) WINDWARD AND LEEWARD LOADS

<u>HEIGHT(FT)</u>	<u>WINDWARD(K/FT)</u>	<u>LEEWARD(K/FT)</u>
0 TO 30	0.232	0.145
31 TO 50	0.320	0.200
51 TO 60	0.384	0.240

NOTE: Windward and leeward loads are applied
in the positive X-direction.

5) ROOF LOADS

<u>LENGTH(FT)</u>	<u>MAGNITUDE(K/FT)</u>
0 TO 23.3	0.480
23.3 TO 46.7	0.360

NOTE: Roof loads are applied in the positive Y-direction.

It should also be noted that the above calculated loads are assuming a 20 foot bent spacing which is the tributary area for the center columns only. The tributary area for the corner columns is ten feet, thus the wind load sustainable by these columns need only be half that of the center columns.

The format for the loading is very similar for all sections. For instance members 71, 72, 103, 104 of the windward, end sections are loaded uniformly in the global X-direction from the start to the end of the member. Note that the inclusion of LA and LB was needed only for the top members due to the load change at the one third section, however, it was included for

all for the purpose of continuity and understanding. The second applied loading condition is that of the beam weight. This loading command is considered fully independent and requires a new loading number as well as a specified direction, usually in the negative Y-direction. Finally both loadings are activated by the load list command so that the selection of members will consider both wind load and dead weight.

ANALYSIS: The analyses discussed previously in the bounds of this text have consisted of simple, linear static analysis for the purpose of determining moments, shears, rotations etc. However the determination of these quantities in the above example is secondary, with the primary motivation being the GTSTRUDL member size calculations. The select command requests that all members be designed using the general analysis in conjunction with the 1978 AISC code. The GENERAL selection procedure as described in TABLE 5-1 begins checking code compliance with the smallest profile in the table of available W shapes, hence this method can become very costly with large numbers of available shapes to check. The other selection procedures search for sections solely on the basis of minimum section modulus or area requirements before full compliance checking is initiated.

After the SELECT command has completed the task of search and selection, it is often expedient to utilize the TAKE command for the purpose of smoothing the design. For instance in the above example the TAKE command has requested that all same floor beams be smoothed according to the member with the largest section modulus. Likewise the columns, due to high axial compression, are smoothed according to the member with the largest cross sectional area. Though redundant, the beams and columns are then checked as such using the 1978 AISC.

Finally the STEEL TAKEOFF command produces a listing of the member type, length, density, weight, and volume individually as well as the cumulative values for the whole structure. As far as member properties are concerned, after the completion of these commands has transpired, a whole

TABLE 5-1. GTSTRUDL Selection Procedures

Procedure	Code	Ordering	Description
AXIAL	69AISC 78AISC API DBLANG W69AISC W78AISC	AX	Intended for use with members controlled by axial forces. Profiles must meet minimum area requirements before code and constraint checks are made.
BEAM	69AISC 78AISC API DBLANG W69AISC W78AISC	AXSY AXSZ	Intended for use with members controlled by bending moments. Profiles must meet minimum section modulus requirements before code and constraint checks are made.
COMBINED	69AISC 78AISC API DBLANG W69AISC W78AISC	AX AXSY AXSZ	Intended for use with members controlled by the combined actions of axial forces with bending moments. Profiles must meet minimum area and section modulus requirements before code and constraint checks are made.
TTWR	TOWER	AX	Intended for use with members in transmission towers. Profiles must meet minimum area requirements before code and constraint checks are made.
GENERAL	69AISC 78AISC TOWER API DBLANG W69AISC W78AISC	AX	Intended for use with all codes, this procedure places no minimum requirements on a profile before performing code and constraint checks.

new structure has been developed, thus it is necessary to perform another stiffness analysis to find the reactions, forces and displacements.

DATA BASE MANAGEMENT: This section consists of the restoration of the original structural geometry so that modifications may be made for the reanalysis without bracing. To accomplish this the bracing members must be deleted as well as the member releases. As discussed earlier, in a braced structure, complete moment resisting joints are unnecessary due to the structure's inherent ability to resist sidesway, however when the bracing members are removed this ability is lost and moment resistance must be replaced. Finally a stiffness analysis is required before every selection process, obviously for the purpose of establishing the new moments, shears etc. necessary to make an accurate selection. After the analysis has occurred an identical member selection process is implemented as before.

Following a braced reanalysis, it is desired to perform a dynamic analyses on both structures, braced and unbraced. For the purpose of continuity, the braced structure is analyzed first by restoring SPACE2 previously saved after the first member selection. Because a simple dynamic analysis without the effect of loading is the intent, the deletions mode is entered and loads one and two are inactivated.

Due the size of the structure, it is advisable to reduce the size of the eigenproblem by performing an automatic static condensation for the purpose of eliminating inconsequential kinematic degrees of freedom. When requested GTSTRUDL selects for the purpose of inclusion in the elimination process the degrees of freedom having the smallest stiffness (translational or rotational) to mass ratios. Note this operation may be performed by the aforementioned automatic procedure or by the explicit naming of joints.

After static condensation has been activated it is necessary to stipulate the direction(s) in which dynamic translation will be generated. It is the intent of the above example to make eigenvalue and eigenvector

comparisons for each of the two structural types; hence it is necessary, in order to understand the dynamic and translational characteristics of each structure, to allow for translation in all three directions. It should be noted that the term eigenvalue refers to the square of the natural circular frequency used in the calculation of each relative modal amplitude or eigenvector. Eigenvectors are relative translational effects and are the quantities used in the creation of graphical mode shapes. (see FIGURES 5-5 TO 5-12)

After the generation of dynamic translation it is necessary to specify inertial parameters for analysis. Several options are available to the user and are explained in GTSTRUDL VOL. 3, however for this example the lumped command was used, specifying that for translation the mass of each inertial member will be halved and located at the joints serving as connectivity between it and its adjacent members.

Finally a reduced band dynamic analysis is requested with a listing of the eigenvectors for the first mode, followed by the plotting of the first five mode shapes. It is then desired to reanalyze the structure with the bracing in tact. It is important to realize that the restoration of SPACE3 and subsequent analysis as listed, can only occur by rerunning the program after the first modal analysis has been deleted. This is due to GTSTRUDL'S inability to run two successive dynamic analyses in which generation of translation has been requested.

GRAPHICAL OUTPUT: For the purpose of expediency the mode shapes for the unbraced structure are plotted immediately following the structures analysis and member selection. This plotting sequence was selected so that an unnecessary and time consuming restoration of a previous analysis need not be implemented. As before the plotter is initialized with the specification of desired neutral length and width of the plot area. The plot of the first mode shape with a magnification of 1.2 times the actual structure size. The subsequent modes were plotted interactively using identical commands.

The braced structure topology with members and joints listed individually is plotted in Figures 5-2a and 5-2b. The view for each of these plots consists of a rotation of 15 degrees in the X-direction and 30 degrees in the Y-direction with character size of 3 (third largest out of four). The wind loaded deflected shape is then plotted an overlayed upon the original structure. Finally a similar restoration and plotting sequence is carried out for the braced structure.

LIST OF REFERENCES

LIST OF REFERENCES

1. American Institute of Steel Construction, Inc., Manual of Steel Construction, 1980.
2. Biggs, John M., Structural Dynamics, McGraw-Hill, New York, 1964, Chapter 6.
3. GTICES Systems Laboratory, "GTSTRUDL User's Manual VOL. 1, 2, 3," Georgia Institute of Technology Civil Engineering Department, Atlanta, GA., 1982.
4. McCormac, Jack C., Structural Analysis, Harper and Row, New York, 1984, Chapters 5, 6, 7, 8.
5. Control Data, NOS 2 User's Guide, 1986.

VITA

Robert Craig Henderson was born in Denver, Colorado on October 12, 1963. He attended elementary and high school in Memphis and Knoxville, Tennessee respectively and was graduated from Central High school in June 1981. The following September he entered the University of Tennessee, and worked as a COOP student with the Department of Transportation for five quarters before receiving a Bachelor of Science degree in Civil Engineering in August 1986. In the fall of 1986 he accepted a graduate assistantship involving structural research at the University of Tennessee which culminated in a Master's of Science degree in Structural Engineering.

The author is of the Christian faith, an avid tennis player and member of American Society of Civil Engineers and Chi Epsilon honor society. It is the author's intention to pursue a PhD degree subsequent to completion of the masters program.