ADAPTATION OF SAP IV TO A THREE DIMENSIONAL SPACE TRUSS

by

John W. Grove, Jr.

Submitted in Partial Fulfillment of the Requirements
for the Degree of
Master of Science in Engineering
in the
Civil Engineering
Program

Paul X Bellini 6/13/80
Advisor Date

Leon Rand 6/16/80
Dean of the Graduate School Date

YOUNGSTOWN STATE UNIVERSITY
May, 1980

ABSTRACT

ADAPTATION OF SAP IV TO A THREE DIMENSIONAL SPACE TRUSS

John W. Grove, Jr.
Master of Science in Engineering
Youngstown State University, 1980

The purpose of this thesis is to investigate some of the limits and capabilities of the user's structural analysis program known by the acronym, SAP IV. The program, developed at the University of California, Berkeley, was sponsored by grants from the National Science Foundation.

The effective use of this program as a structural analysis tool requires a knowledge of three scientific disciplines -- structural mechanics, numerical analysis and computer science. A modern matrix analysis background in structural mechanics is a basic necessity for the interpretation and utilization of matrix stiffness method which forms the basis of the program's solution techniques.

This thesis is a basic overview of the SAP IV program, a summary of the procedures required to input a problem into the program, the application of the program to the solution of a three dimensional space truss, and the physical interpretation of the computer output.

ACKNOWLEDGEMENTS

The author wishes to acknowledge with deep appreciation the valuable professional help and encouragement given to him by Dr. Paul X. Bellini toward the completion of this thesis in his role as both thesis advisor and friend.

The author also wishes to thank his review committee, Dr. John N. Cernica and Dr. Gus Mavrigian, for giving their valuable time toward the completion of the requirements of this work.

The author further wishes to thank the United States Steel Corporation for its financial support during the course of his master's studies.

Sincere appreciation is given to my wife, Alberta, and my daughter, Debra, for their key-punching help, for the typing of this thesis, and for their continued patience and understanding.

TABLE OF CONTENTS

LIST OF SYMBOLS

SYMBOL	DEFINITION
$[K]$	Stiffness matrix
$[M]$	Mass matrix
$[C]$	Damping matrix
$\{u\}$	Column vector of nodal displacements
$\{\dot{u}\}$	Column vector of nodal velocities
$\{\ddot{u}\}$	Column vector of nodal accelerations
$\{r\}$	Column vector of generalized nodal loads
(-)	Compression
(+)	Tension
X,Y,Z	Axes
M℄	Moment at centerline
w	Weight per lineal foot
E	Young's modulus of Elasticity
I	Second moment of area

LIST OF FIGURES

LIST OF TABLES

CHAPTER I

INTRODUCTION

1.1 Historical Background

Structural Analysis Program (SAP) IV is a general three-dimensional, linear, static and dynamic, finite element structural analysis program.

The computer code was developed by Bathe, Wilson, and Peterson at the University of California, Berkeley, under a grant from the National Science Foundation.(1)* The program is available to the public at a price covering duplication and mailing.

SAP was originally published in September, 1970. In 1971 an improved static analysis program, namely SOLID SAP, or SAP II, was completed. SAP III, a new static and dynamic analysis program, was released in 1972. The SAP IV program, aside from minor improvements to SAP III, has the additional analysis capability of out-of-core direct integration for time history analysis.

The current program version of SAP IV for the static and dynamic analysis of linear structural systems

---

*Number in parenthesis corresponds to literature cited in the Bibliography.

is the result of many years of research and development. Coded in standard Fortran IV, it operates without modifications on the CDC 6400, 6600 and 7600 computers.

Of prime importance in the development of any user's program is the ease with which it may be modified, extended, and updated in order to avoid obsolescence. SAP is specifically coded with this intent in mind. As new structural stiffness elements are developed, new, more efficient numerical procedures become available, and new computer machinery is produced, the SAP program is capable of accepting these modifications with only a minimum amount of alteration.

The SAP program is uniquely designed so that it may be modified by the user. Options such as Calcomp plot, as well as additions to the structural element library may be made with little difficulty.

Programming techniques, utilized within the code to produce optimum allocation of high and low speed storage, produce a highly efficient processing of the numerical techniques utilized in the program as well as an effective computer processing of numerical results.

Although SAP IV has the capacity to analyze very large three-dimensional systems, it loses no efficiency in the solution of smaller problems. The program is particularly usable on small size computers because smaller special purpose programs can easily be assembled from the complete program using only those subroutines which are

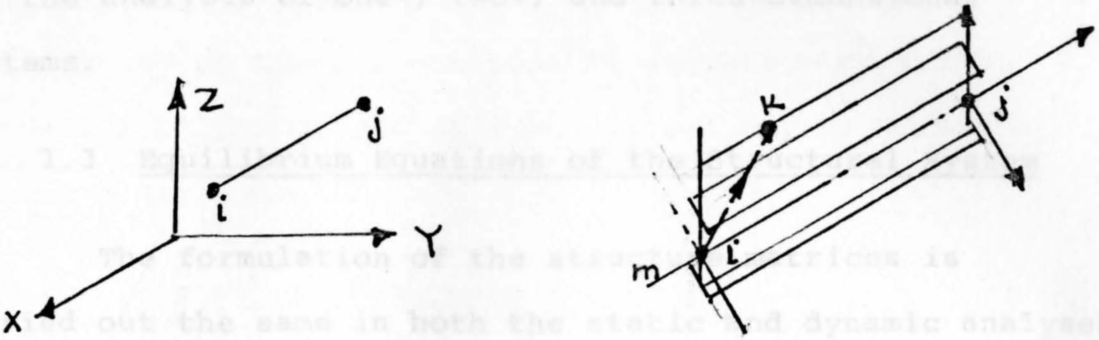
actually needed. It has proven to be a very efficient and flexible analysis tool.

1.2 Program Element Library

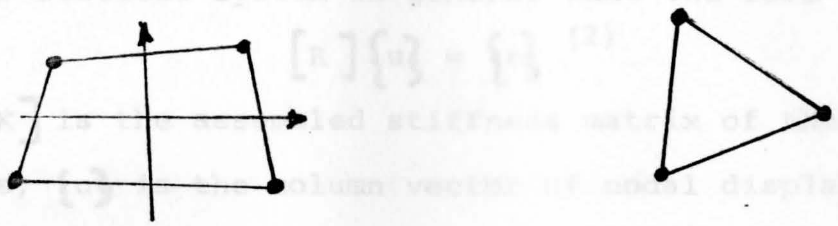
The following element types (See Figure 1) are presently contained in the program. Combinations of a number of these elements may be contained in the structural systems to be analyzed.

(a) three-dimensional truss element,
(b) three-dimensional beam element,
(c) plane stress and plane strain element,
(d) two-dimensional axisymmetric solid,
(e) three-dimensional solid,
(f) thick shell element,
(g) thin plate or thin shell element,
(h) boundary element,
(i) pipe element (tangent and bend).

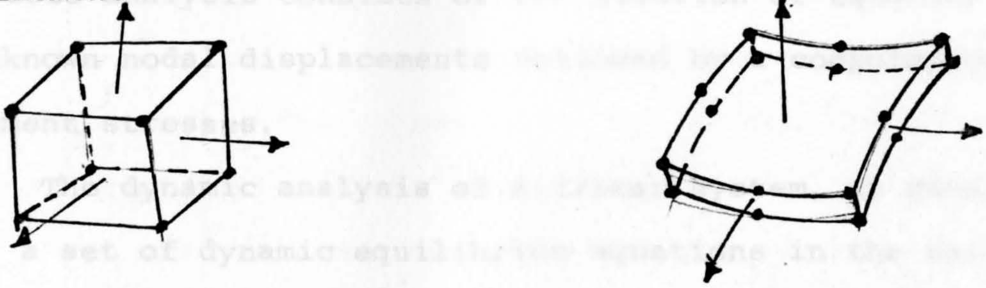
All of the nine structural elements shown may be used in a static or dynamic analysis. The number of nodal points and the number of members used will naturally determine the capacity of this program or any other program. The program restrictions on the number of elements used, along with the number of load cases or the order and band width of the stiffness matrix is almost unlimited. Since each nodal point may have from zero to six degrees of freedom and the element stiffness and mass matrices are assembled in condensed form, the program is efficient



a. Truss Element b. Beam Element

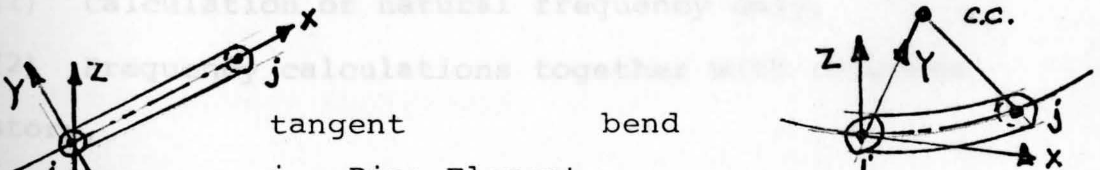


c. Plane Stress & Strain Element d. Axisymmetric Solid Element



e. Three Dimensional Solid Element f. Thick Shell Element

g. Thin Shell Element h. Boundary Element



i. Pipe Element

Figure 1. Element Types -- SAP IV (1)

for the analysis of one-, two-, and three-dimensional systems.

1.3 Equilibrium Equations of the Structural System

The formulation of the structure matrices is carried out the same in both the static and dynamic analyses. The development of the nodal equilibrium equations for a linear statical system in general take the form

$$[K]\{u\} = \{r\}^{(2)} \qquad (1)$$

where $[K]$ is the assembled stiffness matrix of the structure elements, $\{u\}$ is the column vector of nodal displacements, and $\{r\}$ is the column vector of generalized nodal loads. The static analysis consists of the solution of Equation (1) for unknown nodal displacements followed by a computation of element stresses.

The dynamic analysis of a linear system, in general, yields a set of dynamic equilibrium equations in the matrix form

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{r(t)\}^{(2)} \qquad (2)$$

where $[M]$ and $[C]$ are the mass and damping matrices, respectively, and $\{\ddot{u}\}$, $\{\dot{u}\}$, and $\{u\}$ are the nodal accelerations, velocities, and displacements, respectively. In the dynamic analysis solution the following results are obtainable:

(1) Calculation of natural frequency only,

(2) Frequency calculations together with response history,

(3) Frequency calculations followed by response spectrum

analysis,

(4) Response history analysis by direct integration.

Any or all of the preceding four choices may be determined for any dynamic problem.

1.4 Element to Structure Matrices

The structure stiffness matrix $[K]$ is formed by direct addition of the element stiffness matrices, that is,

$$[K] = \sum_{m} [K_m]^{(2)} \qquad (3)$$

where $[K_m]$ is the stiffness matrix of the m'th element and m is the total number of elements. The matrix $[K_m]$ is the same order as $[K]$ and it possesses nonzero terms which pertain to the element degrees of freedom (all other terms zero). The summation of the element matrices into the structure stiffness matrix is performed efficiently by use of identification arrays which relate element degrees of freedom to structure degrees of freedom.

In the program the structure stiffness matrix and a diagonal mass matrix are assembled, which means a lumped mass analysis is assumed. As before, the total structural mass matrix is the sum of the individual element mass matrices along with the concentrated masses which are specified at selected degrees of freedom.

1.5 Boundary Conditions

For the condition of a zero displacement component, the corresponding equation is deleted in the structure equilibrium Equations (1) or (2), and the corresponding element stiffness and mass terms are disregarded. If a nonzero displacement, component x, is specified at a structure degree of freedom i, that is $u_i = x$, the equation

$$ku_i = kx^{(1)} \tag{4}$$

is added to Equations (1) or (2) where $k \gg k_{ii}$, which uniquely forces a solution of Equation (1) or (2) to yield $u_i = x$. From a physical standpoint, this is analogous to applying a spring of large stiffness k at degree of freedom i, and specifying a structure node load which because of the relatively flexible structure at this node, induces the required displacement x.

1.6 Program Organization

The calculation of the structure stiffness matrix and mass matrix is produced in three distinct phases:

(1) The node point data is read and interpreted by the program. The active degrees of freedom ($u_i = 0$) are established at each node.

(2) The element stiffness and mass matrices are calculated together with their connection arrays to the structure nodes, and are stored in sequence on tape.

(3) The element stiffness and mass matrices are combined to form the structure stiffness and mass matrices, and are stored in block form on tape.

These three operations are the basic steps required regardless of the type of element used. A flowchart of the process is shown in Figure 2.

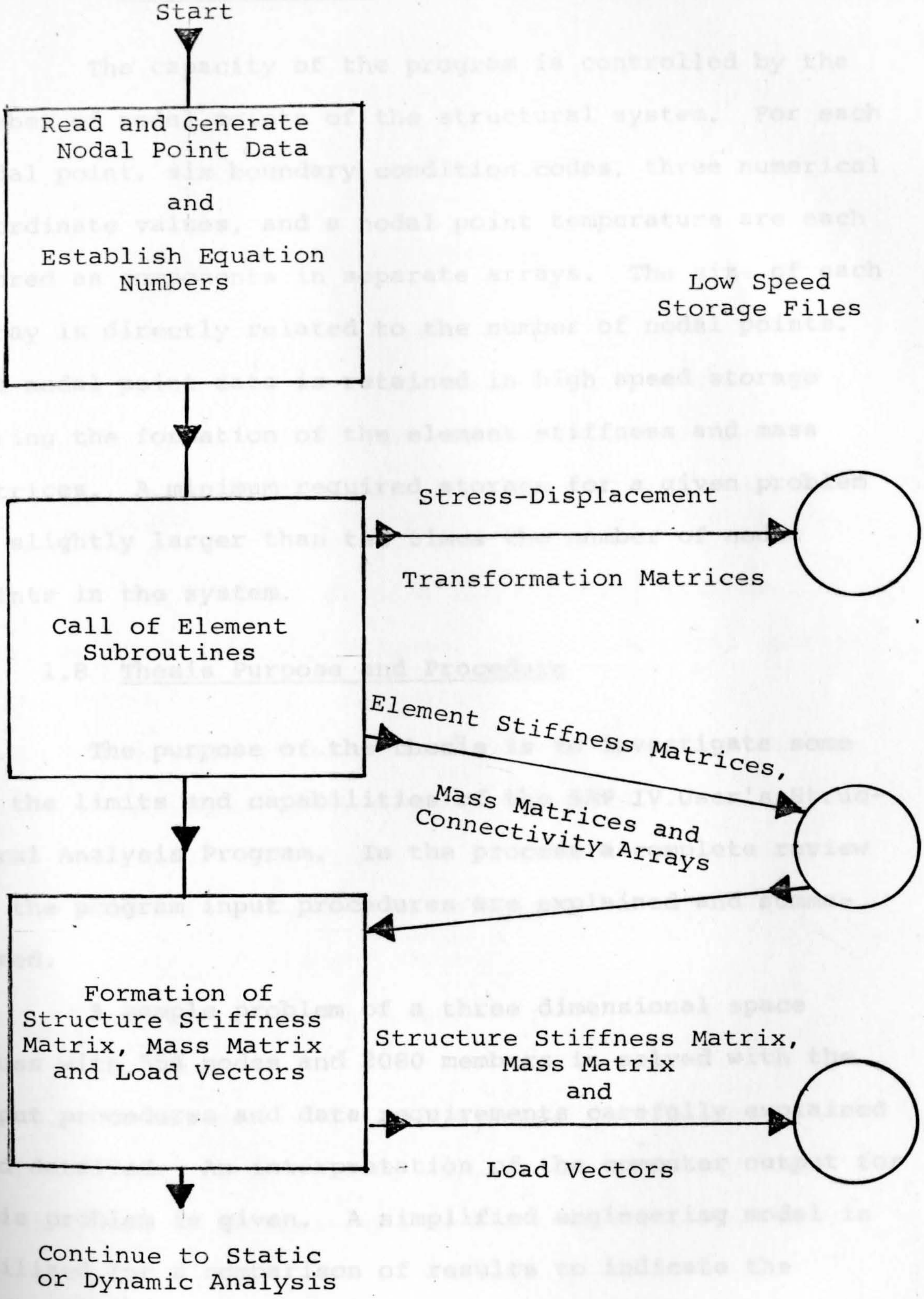


Figure 2. Flowchart for Calculation of Structure Stiffness Matrix and Mass Matrix (1)

1.7 Program Capacity

The capacity of the program is controlled by the number of nodal points of the structural system. For each nodal point, six boundary condition codes, three numerical coordinate values, and a nodal point temperature are each stored as components in separate arrays. The size of each array is directly related to the number of nodal points. All nodal point data is retained in high speed storage during the formation of the element stiffness and mass matrices. A minimum required storage for a given problem is slightly larger than ten times the number of nodal points in the system.

1.8 Thesis Purpose and Procedure

The purpose of the thesis is to investigate some of the limits and capabilities of the SAP IV User's Structural Analysis Program. In the process a complete review of the program input procedures are explained and summarized.

A sample problem of a three dimensional space truss with 554 nodes and 2080 members is solved with the input procedures and data requirements carefully explained and detailed. An interpretation of the computer output for this problem is given. A simplified engineering model is utilized for a comparison of results to indicate the order of magnitude of the computer solutions.

CHAPTER II

PROGRAM INPUT PROCEDURES

2.1 General Program Information

The computer user's program SAP IV is currently stored in the Youngstown State University Computer Library. One needs only to address the program by its code name "SAPIV" to execute the solution of a structural mechanics problem.

A complete set of data for a given structural model is input into the program. This data input consists of the following basic information.

1. Basic geometrical details of the structural system including boundary conditions,
2. Type and total number of library finite elements,
3. Geometric and material properties of each element,
4. Geometry of element connectivity,
5. Values of externally applied loads and/or mass distribution.

The program compiles the data, formulates the structural stiffness matrix, applies the boundary conditions, and efficiently solves the resulting algebraic equations.

For statical problems solutions are obtained for nodal forces and moments, nodal displacements and rotations, and element stresses.

For dynamical problems a variety of solution types are available. These include:

(1) Frequency calculations only,

(2) Frequency calculations followed by response history analysis,

(3) Frequency calculations followed by response spectrum analysis,

(4) Response history analysis by direct integration.

2.2 Data Input to SAP IV

The data input to SAP IV consists of the following ordered information:

(1) Heading card that contains the job name,

(2) Master Control Card indicating
  (a) Total number of nodal points
  (b) Number of element groups
  (c) Number of separate load cases
  (d) Analysis type: static or dynamic; (For a dynamical problem the solution type is specified, see Section 2.1),
  (e) Execution mode: problem solution, or data check only

(3) Node point data indicating
  (a) Coordinate system type: Rectangular cartesian or cylindrical polar coordinates

(b) Node number and node boundary condition: free (loads allowed), fixed (no load allowed)

(c) Numerical coordinates (x, y, z) of the node point

(d) Node increment number for automatic mesh generation

(e) Nodal temperature

(4) Element Data Cards as follows:

(a) Control card indicating element type, total number of elements, and the number of material property cards

(b) Material property cards indicating modulus of Elasticity, Poisson's Ratio, mass density, weight density, and the coefficient of thermal expansion

(c) Element property cards indicating thickness, axial area, shear area, second moment of area, and polar moment of inertia

NOTE: For some simple elements the latter two information sets are contained on one data set since only a small amount of information is necessary.

(d) Element load factor cards specifying the fraction of gravity (in each of the three global directions) to be added to each element load case

(e) Element data cards (one card per element) specifying element number, associated node points, material property number, zero reference temperature, element increment number for automatic element generation.

(5) Concentrated load/mass data

(a) In a static analysis one card is required for each node ("N") having concentrated forces or moments.

(b) Node loadings must be defined in increasing node number and only the loaded nodes are required as input.

(c) In the case of a dynamic analysis, structure load cases have no significance; however, the program does expect to read data. In place of concentrated loads, lumped mass coefficients for the nodal degrees of freedom may be input for all or some of the nodes.

(d) The program terminates reading load or mass data when a blank card is encountered.

(e) In the special case of static analysis with no concentrated load, only one (1) blank card is required.

(f) Similarly, in a dynamic analysis for which the mass matrix is not to be used by any entries in this section, only one (1) blank card is required.

(6) Element Load Multipliers

(a) One card must be given for each static structure load case requested on the master control card. The cards must reference load case numbers in ascending order. Four element load sets (A, B, C, D), if created during the processing of element data are combined with any concentrated loads specified in Section 5 above for the structure load cases.

(b) For dynamic analysis options one blank card is supplied in this section.

2.3 Automatic Data Generation

The program has internal nodal point generation and element generation capabilities.

(1) Node point generation for a set of equally spaced nodes between two node points N_1 and N_2 is automatically produced within the program. The first, second, and last card in the line set is input with appropriate node numbers, boundary conditions, and coordinate geometry. The third card contains a number "1" in column 70.

(2) Element number and connectivity generation for a set of elements having similar material and geometric properties is an available internal option. The first card in the sequence is input with appropriate element number and node connectivity points.

An integer number k is input on the card, its column position depending on element type. The element numbers are sequenced by the integer "1" from the starting element number to one minus the element number on the next data input card. The nodal numbers are sequenced from element to element by the addition of integer parameter k.

(3) No data generation procedures are available for node load and/or mass point data. An individual data card must be entered for each node loaded.

2.4 Boundary Elements

The boundary element is used to compute support reactions, provide linear elastic supports to nodes, and to constrain nodal displacements to specified values (i.e. support settlements).

(1) The boundary elements, with direction given by a single axis through a specified node point, "N", is defined by a linear extensional spring along the axis and a linear rotational spring about the axis.

(2) By specifying a zero displacement of node "N" on the element data input card, the spring force is automatically equal to the support reaction.

(3) There is no limit to the number of boundary elements applied at any node; boundary elements have no effect on the size of the stiffness matrix.

(4) If the boundary element is aligned with the global displacement directions only the corresponding diagonal element of the stiffness matrix is modified and no matrix ill-conditioning results. When the boundary element couples the degrees of freedom, large off-diagonal elements are introduced into the stiffness matrix which may produce severe ill-conditioning and which may cause solution difficulties.

2.5 Data Check Run

For the analysis of large structures, those containing a significant number of nodes and elements, it is of prime importance to be able to check the data which is read and generated by the program. This insures the CPU time is not wasted in needless computation.

For this purpose an option is available in which the program reads, generates, and prints all data. This data is generated on low speed storage and may be copied onto a physical tape to plot finite element mesh if a Calcomp plot program and plotter is available.

CHAPTER III

APPLICATION OF SAP IV TO A THREE DIMENSIONAL SPACE TRUSS

3.1 Sample Problem Overview

As an illustration of the use of SAP IV, the problem of the analysis of a 100' by 65' space truss having 554 node points and 2080 members is performed in detail. The determination of safe working stresses within the truss members for various support boundary conditions is investigated. All truss members are assumed to be pin-connected, with a constant cross-section and modulus of Elasticity. The members are of standard 1½" closed square tubing with a wall thickness of .105", an area of 0.6064$''^2$, and a modulus of Elasticity of 30,000 ksi. A uniform loading of 30 lbs/ft.^{2} is specified on the top chord of the space truss.

The three separate boundary support conditions are considered as follows:

Case I - 8 pinned, top chord connections, symmetrically supported (See Figure 3)

Case II - same as Case I except that the connections are all roller-supported. The structure is supported in the vertical (or Z direction) only. (See Figure 3)

Case III - 16 pinned supports on the bottom chords. (See Figure 4).

PLAN VIEW OF SPACE TRUSS

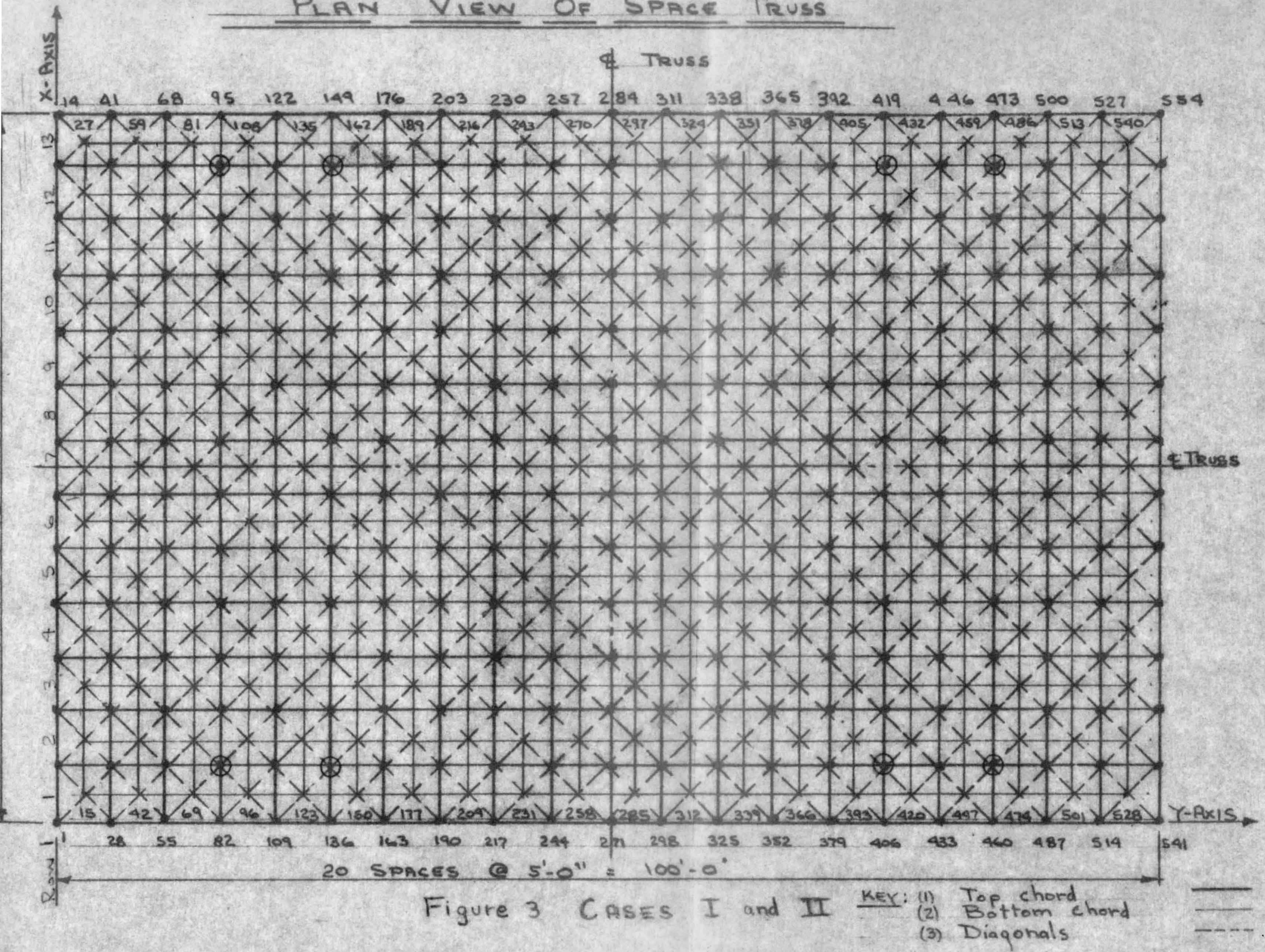


Figure 3 CASES I and II

ONE QUARTER OF SPACE TRUSS SHOWN IN THREE DIMENSIONS

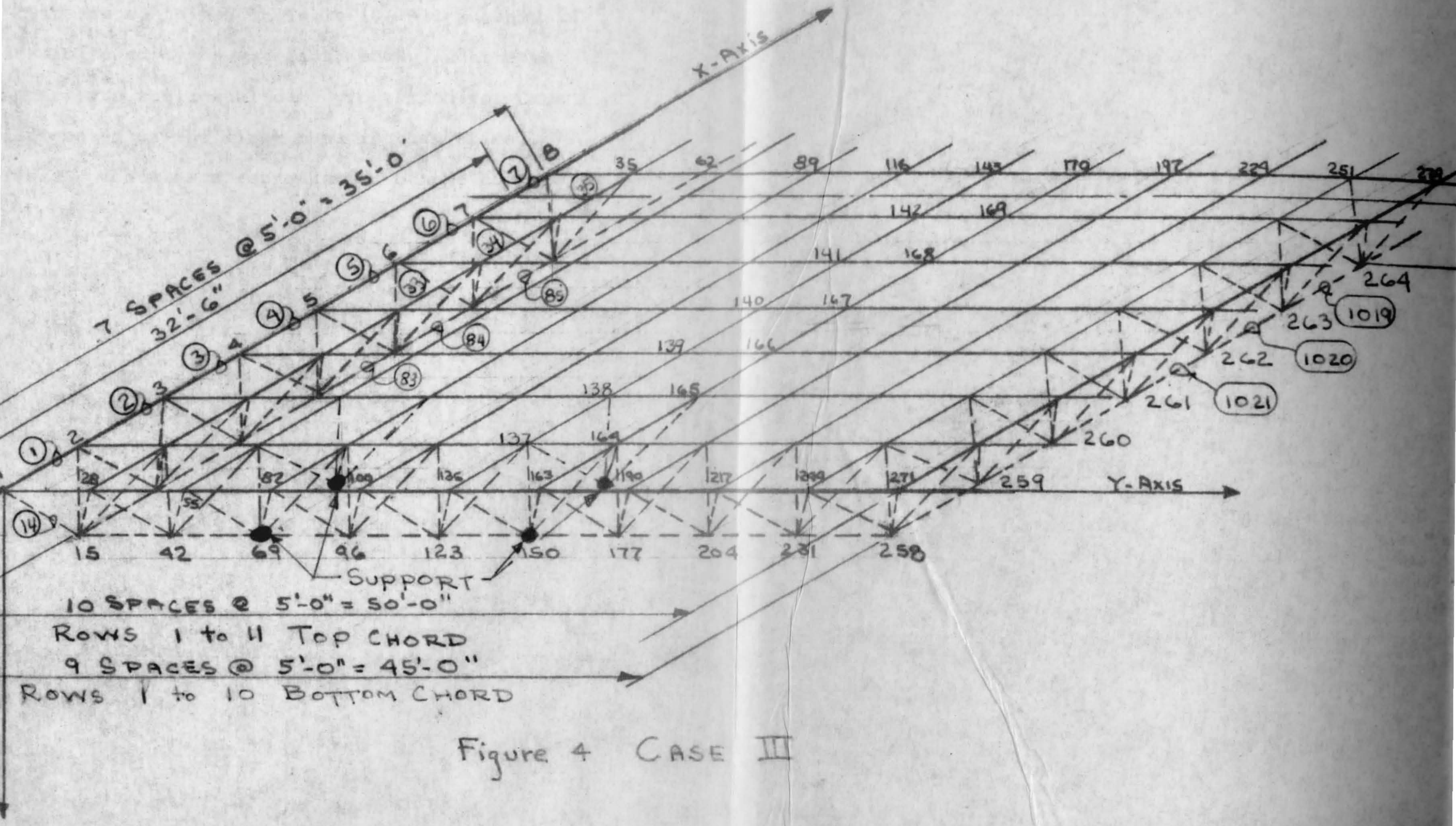


Figure 4 CASE III

3.2 Execution and Master Control Card

In order to call for SAP IV, one must use a standard execution card having the following format, beginning with Column 1: //b̸EXECb̸SAPIV

The first card the computer reads after calling SAP IV is the Master Control Card. The format and data for this card are contained in Table 1. All numbers in parentheses followed by an asterisk, shown under each column number, are the actual data programmed for Case I and are designated on the first card in the series.

Table 2 contains a reproduction of the computer printout.

TABLE 1

MASTER CONTROL CARD FORMAT (1)

	COLUMNS	VARIABLE	DESCRIPTION
(a)	1 - 5 (554)*	NUMNP	Total number of nodal points (joints) in the model
(b)	6 - 10 (1)*	NELTYP	Number of element groups
(c)	11 - 15 (1)*	LL	Number of structure load cases GE.1; static analysis EQ.0; dynamic analysis
(d)	16 - 20 (0)*	NF	Number of frequencies to be found in the eigenvalue solution; EQ.0; static analysis GE.1; dynamic analysis
(e)	21 - 25 (0)*	NDYN	Analysis type code: EQ.0; static analysis EQ.1; eigenvalue/vector solution EQ.2; forced dynamic reponse by mode superposition EQ.3; response spectrum analysis EQ.4; direct step-by-step integration
(f)	26 - 30 (1 programmed first)* (0 programmed second)*	Modex	Program exection mode: EQ.0; problem solution EQ.1; data check only
(g)	31 - 35 (0)* (Note: NAD is principally a program testing parameter and should normally be left blank.)	NAD	Total number of vectors to be used in a SUBSPACE ITERATION solution for eigenvalue/vectors: EQ.0; default set to: MIN 2*NF, NF + 8

TABLE 1 CONTINUED

MASTER CONTROL CARD FORMAT(1)

COLUMNS	VARIABLE	DESCRIPTION
(h) 36 - 40 (0)*	KEQB	Number of degrees of freedom (equation) per block of storage: EQ.0; calculated automatically by the program

TABLE 2

MASTER CARD CONTROL INFORMATION
AS REPRODUCED BY COMPUTER PRINTOUT

```
NUMBER OF NODAL POINTS          =          554
NUMBER OF ELEMENT TYPES         =            1
NUMBER OF LOAD CASES            =            1
NUMBER OF FREQUENCIES           =            0
ANALYSIS CODE (NDYN)            =            0
  EQ.0,  STATIC
  EQ.1,  MODAL EXTRACTION
  EQ.2,  FORCED RESPONSE
  EQ.3,  RESPONSE SPECTRUM
  EQ.4,  DIRECT INTEGRATION
SOLUTION MODE (MODEX)           =            0
  EQ.0,  EXECUTION
  EQ.1,  DATA CHECK
NUMBER OF SUBSPACE
ITERATION VECTORS (NAD)         =            0
EQUATIONS PER BLOCK             =            0
TAPE10 SAVE FLAG (N10SV)        =            0
```

3.3 Node Point Numbering and Boundary Conditions

Table 3 summarizes the format for reading in the node numbers and their boundary conditions and, as before, the numbers shown in parentheses with an asterisk are the numbers on the first card printed.

Table 4 lists the actual printout of nodal numbers, their boundary conditions, and the geometric coordinates as read into the computer. Take, for example, nodes 1 through 14, which appear on Figure 3 of the space truss, the first top chord row (Row 1) nodes. A short explanation of Figure 3 is helpful and necessary.

The top chords are shown with solid lines, the bottom chords and diagonals are shown as dotted lines. The nodes beginning with node 1 are numbered from 1 to 14, starting with the first row (Row 1) and increasing in order along the X-axis to node 14. Figure 4 is drawn in three dimension to better illustrate this and shows only ¼ of the entire space truss for Case III. Proceeding to the bottom chord, first row, the nodes are numbered from 15 through 27 (i.e. 13 nodes). A progressive numbering system is continued, alternating from top chord to bottom chord, ending with a final node and a total node count of 554, as the progression follows along the Y-axis. The principal reason for this numbering system in this short direction is to keep the band width of the structure stiffness matrix minimal, the advantage of which is explained later in more detail.

TABLE 3

NODAL POINT DATA FORMAT (1)

COLUMNS	VARIABLE	DESCRIPTION
(a) 1 (blank)*	CT	Symbol describing coordination system for this node: EQ. ; (blank) cartesian (X,Y,Z) EQ.C; cylindrical (R,Y,θ)
(b) 2 - 5 (1)*	N	Node Number
(c) 6 - 10 (0)*	IX(N,1)	X-translation boundary condition code
11 - 15 (0)*	IX(N,2)	Y-translation boundary condition code
16 - 20 (0)*	IX(N,3)	Z-translation boundary condition code
21 - 25 (1)*	IX(N,4)	X-rotation boundary condition code
26 - 30 (1)*	IX(N,5)	Y-rotation boundary condition code
31 -35 (1)*	IX(N,6)	Z-rotation boundary condition code EQ.0; free (load allowed) EQ.1; fixed (no load allowed) GT.1; master node number (beam nodes only)
(d) 36 - 45 (0)*	X(N)	X (or R) - ordinate
46 - 55 (0)*	Y(N)	Y - ordinate
56 - 65	Z(N)	Z (or θ) - ordinate (degrees)
(e) 66 - 70 (0)*	KN	Node number increment
(f) 71 - 80 (0.0)*	T(N)	Nodal temperature

TABLE 4

NODAL POINT INPUT DATA AS REPRODUCED BY COMPUTER

NODE	BOUNDARY CONDITION CODES						NODAL POINT COORDINATES				
NUMBER	X	Y	Z	XX	YY	ZZ	X	Y	Z	? No	T
1	0	0	0	1	1	1	0.0	0.0	0.0	0	0.0
2	0	0	0	1	1	1	5.000	0.0	0.0	0	0.0
14	0	0	0	1	1	1	65.000	0.0	0.0	1	0.0
15	0	0	0	1	1	1	2.500	2.500	3.700	0	0.0
16	0	0	0	1	1	1	7.500	2.500	3.700	0	0.0
27	0	0	0	1	1	1	62.500	2.500	3.700	1	0.0
(Node numbers 28 to 81 have been omitted.)											
82	0	0	0	1	1	1	0.0	15.000	0.0	0	0.0
83	1	1	1	1	1	1	5.000	15.000	0.0	0	0.0
84	0	0	0	1	1	1	10.000	15.000	0.0	0	0.0
93	0	0	0	1	1	1	55.000	15.000	0.0	1	0.0
94	1	1	1	1	1	1	60.000	15.000	0.0	0	0.0
95	0	0	0	1	1	1	65.000	15.000	0.0	0	0.0
(Node numbers 96 to 527 have been omitted.)											
528	0	0	0	1	1	1	2.500	97.500	3.700	0	0.0
529	0	0	0	1	1	1	7.500	97.500	3.700	0	0.0
540	0	0	0	1	1	1	62.500	97.500	3.700	1	0.0
541	0	0	0	1	1	1	0.0	100.000	0.0	0	0.0
542	0	0	0	1	1	1	5.000	100.000	0.0	0	0.0
554	0	0	0	1	1	1	65.000	100.000	0.0	1	0.0

Looking again at nodes 1 through 14 (Figure 3 and Table 4) we see that the entire row of fourteen nodes is represented by only three cards. The 1 in column 70 for node 14 initiates the node generation subroutine in the program which automatically generates nodes 3 through 13 with coordinates equally spaced along the line. Not only does the computer automatically produce intermediate node data, but it also generates a printout to allow checking. This printout is shown in Table 5. As explained previously, the 0's for the X, Y and Z boundary condition codes inform the program that the node is free to translate in the X, Y and Z directions. The 1's in the XX, YY, and ZZ boundary condition codes inform the program that there are no moments present in these directions. It should be noted that the node generation scheme may be used only if the boundary conditions at each node point on the line are identical. All that remains is to supply the program with the remaining nodes and their nodal point co-ordinates, which is shown in Table 4, and, as before, this printout is shown in Table 5.

Since Table 5 is the generated output from the input shown in Table 4 for Case I, the differences for Case II and Case III will be explained.

Comparing Case I to Case II, the only differences occur in the support boundary conditions. Case I is restrained in the X, Y, and Z directions at node 83 which is defined by 1's. Since Case II is only restrained

TABLE 5

FULL DATA OUTPUT FOR NODAL POINTS AS GENERATED BY COMPUTER

NODE NUMBER	BOUNDARY CONDITION CODES X	Y	Z	XX	YY	ZZ	NODAL POINT COORDINATES X	Y	Z	T
1	0	0	0	1	1	1	0.0	0.0	0.0	0.0
2	0	0	0	1	1	1	5.000	0.0	0.0	0.0
3	0	0	0	1	1	1	10.000	0.0	0.0	0.0
4	0	0	0	1	1	1	15.000	0.0	0.0	0.0
5	0	0	0	1	1	1	20.000	0.0	0.0	0.0
6	0	0	0	1	1	1	25.000	0.0	0.0	0.0
7	0	0	0	1	1	1	30.000	0.0	0.0	0.0
8	0	0	0	1	1	1	35.000	0.0	0.0	0.0
9	0	0	0	1	1	1	40.000	0.0	0.0	0.0
10	0	0	0	1	1	1	45.000	0.0	0.0	0.0
11	0	0	0	1	1	1	50.000	0.0	0.0	0.0
12	0	0	0	1	1	1	55.000	0.0	0.0	0.0
13	0	0	0	1	1	1	60.000	0.0	0.0	0.0
14	0	0	0	1	1	1	65.000	0.0	0.0	0.0
15	0	0	0	1	1	1	2.500	2.500	3.700	0.0
16	0	0	0	1	1	1	7.500	2.500	3.700	0.0
17	0	0	0	1	1	1	12.500	2.500	3.700	0.0
18	0	0	0	1	1	1	17.500	2.500	3.700	0.0
19	0	0	0	1	1	1	22.500	2.500	3.700	0.0
20	0	0	0	1	1	1	27.500	2.500	3.700	0.0
21	0	0	0	1	1	1	32.500	2.500	3.700	0.0
22	0	0	0	1	1	1	37.500	2.500	3.700	0.0
23	0	0	0	1	1	1	42.500	2.500	3.700	0.0
24	0	0	0	1	1	1	47.500	2.500	3.700	0.0
25	0	0	0	1	1	1	52.500	2.500	3.700	0.0
26	0	0	0	1	1	1	57.500	2.500	3.700	0.0
27	0	0	0	1	1	1	62.500	2.500	3.700	0.0

(Node numbers 28 to 81 have been omitted.)

TABLE 5 CONTINUED

FULL DATA OUTPUT FOR NODAL POINTS AS GENERATED BY COMPUTER

NODE NUMBER	BOUNDARY CONDITION CODES X	Y	Z	XX	YY	ZZ	NODAL POINT COORDINATES X	Y	Z	T
82	0	0	0	1	1	1	0.0	15.000	0.0	0.0
83	1	1	1	1	1	1	5.000	15.000	0.0	0.0
84	0	0	0	1	1	1	10.000	15.000	0.0	0.0
85	0	0	0	1	1	1	15.000	15.000	0.0	0.0
86	0	0	0	1	1	1	20.000	15.000	0.0	0.0
87	0	0	0	1	1	1	25.000	15.000	0.0	0.0
88	0	0	0	1	1	1	30.000	15.000	0.0	0.0
89	0	0	0	1	1	1	35.000	15.000	0.0	0.0
90	0	0	0	1	1	1	40.000	15.000	0.0	0.0
91	0	0	0	1	1	1	45.000	15.000	0.0	0.0
92	0	0	0	1	1	1	50.000	15.000	0.0	0.0
93	0	0	0	1	1	1	55.000	15.000	0.0	0.0
94	1	1	1	1	1	1	60.000	15.000	0.0	0.0
95	0	0	0	1	1	1	65.000	15.000	0.0	0.0

(Node numbers 96 to 554 have been omitted.)

in the Z direction, node 83 (Table 5) shows 0's in the X and Y directions and 1 in the Z direction. This condition allows translation in the plane of the truss. These changes involved only 8 cards, one card for each support point, that is nodes 83, 94, 137, 148, 407, 418, 461 and 472.

For Case III, the supports were doubled in number and moved from the top chord to the bottom chord and are all pinned as in Case I. This support condition involves changes in only 32 node point cards.

3.4 Equation Numbering Scheme

After reading in all the node information, the program automatically computes and prints the equation numbers as illustrated by Table 6 for Case I only. An equation number is assigned to each degree of freedom designated as a "0" on the node data card. Those specified as "1" are deleted from the computation process. It is interesting to note that the last node (554) ends with the following line of equation numbers for Case I:

554	1636	1637	1638

and in Case II they end with

554	1652	1653	1654

which means a higher number of degrees of freedom are present for Case II. The difference between the total number of equations in the two cases is 16 and should be. Case II is allowed two additional degrees of freedom at each support point (i.e. eight roller supports), hence, the total number of the equations must end 16 numbers higher. For Case III the last equation is

554	1612	1613	1614

Again comparing this with Case I, we see that the total number is 24 less which is also accountable since there are 8 more connections and 3 degrees of freedom less at each support point.

Taking Case I to illustrate the total equation numbering system, we start with the number of degrees of

TABLE 6

AUTOMATIC EQUATION GENERATIONS AS PRINTED OUT BY COMPUTER

N	X	Y	Z	XX	YY	ZZ
1	1	2	3	0	0	0
2	4	5	6	0	0	0
3	7	8	9	0	0	0
4	10	11	12	0	0	0
5	13	14	15	0	0	0
6	16	17	18	0	0	0
7	19	20	21	0	0	0
8	22	23	24	0	0	0
9	25	26	27	0	0	0
10	28	29	30	0	0	0
11	31	32	33	0	0	0
12	34	35	36	0	0	0
13	37	38	39	0	0	0
14	40	41	42	0	0	0
15	43	44	45	0	0	0
16	46	47	48	0	0	0
17	49	50	51	0	0	0
18	52	53	54	0	0	0
19	55	56	57	0	0	0
20	58	59	60	0	0	0
21	61	62	63	0	0	0
22	64	65	66	0	0	0
23	67	68	69	0	0	0
24	70	71	72	0	0	0
25	73	74	75	0	0	0
26	76	77	78	0	0	0
27	79	80	81	0	0	0

(Node numbers 28 to 81 have been omitted.)

TABLE 6 CONTINUED

AUTOMATIC EQUATION GENERATIONS AS PRINTED OUT BY COMPUTER

```
N            X          Y          Z          XX           YY           ZZ

82          244        245        246          0            0            0
83            0          0          0          0            0            0
84          247        248        249          0            0            0
85          250        251        252          0            0            0
86          253        254        255          0            0            0
87          256        257        258          0            0            0
88          259        260        261          0            0            0
89          262        263        264          0            0            0
90          265        266        267          0            0            0
91          268        269        270          0            0            0
92          271        272        273          0            0            0
93          274        275        276          0            0            0
94            0          0          0          0            0            0
95          277        278        279          0            0            0
             (Node numbers 96 to 553 have been omitted.)
554        1636       1637       1638          0            0            0
```

freedom at each node (3), multiply by the number of nodes (554), then subtract the number of restraints (3) times the number of supports (8). This totals to

$$3 \times 554 - 3 \times 8 = 1638$$

which is the last number in the automatic equation numbering system (Table 6). This latter information is explained neither carefully nor in a detailed manner in the program manual.

3.5 Element Data Input

After the automatic generation of the equation numbers, the computer is now ready for the element data which consists of the following items (See Table 7):

A - Control Card

B - Material Property Cards

C - Element Load Factors (4 cards)

D - Element Data Cards

The Control Card (A) supplies the program with three important pieces of information: the type of element to be used (See Figure 1), the total number of truss elements, and the number of material cards to be used.

The Material Property Card (B) indicates Young's modulus of the material and the cross section area. Only one card is required since these properties remain constant in all elements. Some information on this card is not required when solving a space truss, such as the mass density and the weight density, and node temperature. Mass density is used only when a dynamic solution is required.

The Element Load Factor Cards (C) are used for ease of loading in any other direction such as taking part of the vertical load and applying it in the horizontal direction for wind load which has not been considered in this problem analysis. Element Data Cards are discussed in Section 3.6.

TABLE 7

FORMAT OF ELEMENT DATA FOR TYPE 1 (THREE-DIMENSIONAL TRUSS ELEMENT)(1)

A. Control Card

Columns 1 - 5 (1)*	The number 1
6 - 10 (2080)*	Total number of truss elements
11 - 15 (1)*	Number of material property cards

B. Material Property Cards

There need be as many of the following cards as are necessary to define the properties listed below for each element in the structure.

Columns 1 - 5 (1)*	Material identification number
6 - 15 (30,000 ksi)*	Modulus of elasticity
16 - 25 (.000065 in./°F)*	Coefficient of thermal expansion
26 - 35 (0)*	Mass density (used to calculate mass matrix)
36 - 45 (.6064)*	Cross-sectional area
46 - 55	Weight density (used to calculate gravity loads)

C. Element Load Factor - Four Cards

Three cards specifying the fraction of gravity (in each of the three global coordinate directions to be added to each element load case.

Card 1: Multiplier of gravity load in th +X direction

Columns 1 - 10 (1)*	Element load case A
11 - 20 (0)*	Element load case B
21 - 30 (0)*	Element load case C
31 - 40 (0)*	Element load case D

TABLE 7 CONTINUED

FORMAT OF ELEMENT DATA FOR TYPE 1
(THREE-DIMENSIONAL TRUSS ELEMENT) (1).

Card 2: As above gravity in the +Y direction
(All 0's)*

Card 3: As above for gravity in the +Z direction
(All 0's)*

Card 4: This indicates the fraction of the thermal load to be added to each of the element load cases.
(All 0's)*

D. Element Data Cards

One card per element in increasing numerical order starting with one.

Columns	1 - 5	Element number
	(1)*	
	6 - 10	Node number I
	(1)*	
	11 - 15	Node number J
	(2)*	
	16 - 20	Material property number
	(1)*	
	21 - 30	Reference temperature for zero stress
	(0)*	
	31 - 35	Optional parameter k used for automatic generation of element data.
	(1)*	

( )* indicates actual numbers used in program

3.6 Element Data Cards

The Element Data Cards (D) which require much time for compilation encompass a large amount of data input. This information physically connects the entire structure together as a continuous structural unit.

It is advantageous at this point to utilize the element generation subroutine available within the program. A typical set of data input is shown in Table 8 which utilizes this scheme. The number "1" in column 35 activates the element generation process increasing the element number N by unity and simultaneously increasing the element end node numbers I and J by a value "1". Consider, for example, Member 1 which connects node 1 to node 2. The program generates its own member numbers automatically increasing by a value one up to the member number on the next data card (i.e. 14 in this case). This system of automatic member numbering greatly reduces the number of cards required as input data.

Use of the element generation subroutine reduces the number of data input cards from 2080 (one per member) to approximately 140 cards, which is a tremendous savings in time necessary to produce the data cards. A typical computer printout of the results of Table 8 is shown in part in Table 9.

The element generation scheme for the three dimensional space truss is utilized most efficiently and effectively

TABLE 8

TYPICAL ELEMENT DATA CARDS

N	I	J	TYPE	TEMP.	COL. 35
1	1	2	1	0.0	1
14	1	28	1	0.0	1
28	1	15	1	0.0	1
41	15	2	1	0.0	1
67	15	29	1	0.0	1
80	15	16	1	0.0	1
92	15	42	1	0.0	1

TABLE 9

TYPICAL ELEMENT DATA CARDS AS PRINTED AND GENERATED BY THE COMPUTER

N	I	J	TYPE	TEMP	BAND
1	1	2	1	0.0	6
2	2	3	1	0.0	6
3	3	4	1	0.0	6
4	4	5	1	0.0	6
5	5	6	1	0.0	6
6	6	7	1	0.0	6
7	7	8	1	0.0	6
8	8	9	1	0.0	6
9	9	10	1	0.0	6
10	10	11	1	0.0	6
11	11	12	1	0.0	6
12	12	13	1	0.0	6
13	13	14	1	0.0	6
14	1	28	1	0.0	84
15	2	29	1	0.0	84
16	3	30	1	0.0	84
17	4	31	1	0.0	84
18	5	32	1	0.0	84
19	6	33	1	0.0	84
20	7	34	1	0.0	84
21	8	35	1	0.0	84
22	9	36	1	0.0	84
23	10	37	1	0.0	84
24	11	38	1	0.0	84
25	12	39	1	0.0	84
26	13	40	1	0.0	84
27	14	41	1	0.0	84
28	1	15	1	0.0	45
29	15	2	1	0.0	42
30	2	16	1	0.0	45
31	16	3	1	0.0	42
32	3	17	1	0.0	45
33	17	4	1	0.0	42
34	4	18	1	0.0	45
35	18	5	1	0.0	42
36	5	19	1	0.0	45
37	19	6	1	0.0	42
38	6	20	1	0.0	45
39	20	7	1	0.0	42
40	7	21	1	0.0	45
41	21	8	1	0.0	42
42	8	22	1	0.0	45
43	22	9	1	0.0	42
44	9	23	1	0.0	45
45	23	10	1	0.0	42

utilizing the following order of element definitions:

1. Top chord elements along X-axis
2. Top chord elements along Y-axis
3. Diagonal elements - Southwest (single line)
4. Diagonal Elements - Northwest and Southeast (double line)
5. Diagonal elements - Northeast (single line)
6. Bottom chord elements along X-axis
7. Bottom chord elements along Y-axis

This processing of partial element numbering, if followed in each column of members starting with the first column (See Figure 3) and ending with the twentieth column (moving down the Y-axis), will account for a total of 2080 members.

The printout in Table 9 includes the parameter "Band" which designates the bandwidth of the structural stiffness generated by the element numbering system. The program calculates the bandwidth in a subroutine operation using the following equation:

$$3\left[(I-J)+1\right] = \text{bandwidth}^{(3)} \tag{5}$$

where I and J are the node numbers at the ends of the element, and the value 3 is the number of degrees of freedom at the nodes. It should be noted that as the bandwidth increases the amount of CPU time for the numerical solution of the problem also increases. Therefore, it is imperative that the numbering system for the nodal points be such that the bandwidth of the structural stiffness matrix is effectively minimized. As a typical example, consider Member 28. It follows that $3\left[(15 - 1) + 1\right] = 45$ which is the band width.

It should be noted (See Figure 3) that the node numbering scheme procedes in the smaller geometrical direction (i.e. the side of length sixty-five (65) feet). This directional operation is the single most effective technique for minimizing bandwidth.

For the space truss shown in Figure 3 the maximum value of bandwidth obtained is eighty-four (84), and for the Case I problem sixteen hundred and thirty-eight (1638) equations.

3.7 Load/Mass Data

The remaining input to the computer is the load/mass data for which the format appears in Table 10 for node number 1. There must be a single card for every loaded node.

Table 11 is the actual printout of the input load data. Since the space truss is only loaded in the Z direction with a uniform load of 30#/sq. ft. on the top chord, the table consists of top chord loading only and each 5'-0" square area would have (30 x 25) ÷ 1000 = .7500 kips applied at each interior node. The end and side nodes have a percentage less depending on the area of influence of the node. For example, node 1 has an area of ¼ of a full area or a load of .1875 kips which is the number listed in Table 11 for node 1 under the Z axis force.

There is no node load generation scheme available within the program. Hence, each of the node loadings is assigned a single card. For Z direction loading, the loading cards constitute fifty percent (50%) of the total number of data input cards.

TABLE 10

CONCENTRATED LOAD/MASS DATA (1)

NODE	COLUMNS	VARIABLE	ENTRY
(1)	1 - 5 (1)*	N	Nodal point number
(2)	6 - 10 (1)*	L	Structure load case number; GE.1; static analysis EQ.0; dynamic analysis
	11 - 20 (0)*	FX (N,L)	X-direction force (or translational mass coefficient)
	21 - 30 (0)*	FY (N,L)	Y-direction force (or translational mass coefficient
	31 - 40 (0.1875)*	FZ (N,L)	Z-direction force (or translational mass coefficient)
	41 - 50 (0)*	MX (N,L)	X-axis moment (or rotational inertia)
	51 - 60 (0)*	MY (N,L)	Y-axis moment (or rotational inertia)
	61 - 70 (0)*	MZ (N,L)	Z-axis moment (or rotational inertia)

( )* Indicates the actual data used on the first card.

TABLE 11

TYPICAL LOAD OUTPUT AS PRINTED BY THE COMPUTER

```
NODE     LOAD   X-AXIS     Y-AXIS     Z-AXIS              X-AXIS     Y-AXIS     Z-AXIS
NUMBER   CASE   FORCE      FORCE      FORCE               MOMENT     MOMENT     MOMENT

  1       1      0.0        0.0        0.18750D 00         0.0        0.0        0.0
  2       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  3       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  4       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  5       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  6       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  7       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  8       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
  9       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 10       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 11       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 12       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 13       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 14       1      0.0        0.0        0.18750D 00         0.0        0.0        0.0
 28       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
 29       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 30       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 31       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 32       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 33       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 34       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 35       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 36       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 37       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 38       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 39       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 40       1      0.0        0.0        0.75000D 00         0.0        0.0        0.0
 41       1      0.0        0.0        0.37500D 00         0.0        0.0        0.0
```

3.8 Data Input Summary

After accumulating all the necessary data card input as described in Sections 3.2 through 3.8, the program has a subroutine which prints the entire data input record including all intermediate node and element generation data. This printout is activated by a "1" in column 30 of the master control card (See Table 1). Upon scrutinizing, reviewing, and debugging all input data, a "0" is entered into the same column 30 which designates the numerical results are to be calculated and printed.

The results of the computer output for Cases I, II, and III are discussed in Chapter 4.

3.9 Computer Output

With all of the preceding information in the computer and the data checks run and printed, (a 1 in column 30 on the Master Control Card), the Master Control Card is then inserted again with a 0 in column 30 which allows the computer to numerically solve the problem. (See Table 1).

Table 12 shows a partial computer printout for Case III which includes the most important truss deflections as computed and printed by the computer, including node number and associated translations in the X, Y, and Z directions.

Figure 5 also shows a plan view of the translations (XY plane). Note that the arrows point the directions of translation for the different nodes.

Table 13 also shows a partial computer printout for Case III, showing the largest stresses and forces. The table as presented is in the same form as the actual printout and shows the member number, the stress, and the force in the member along with the conventional signs for tension (+) and compression (-). The only change to this table from the actual printed output is the addition of the column entitled Nodes. This column is added for ease of location and identification of the different members. This table should be used in conjunction with Figure 5 in order to locate a given stressed member.

TABLE 12

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED BY THE COMPUTER FOR CASE III

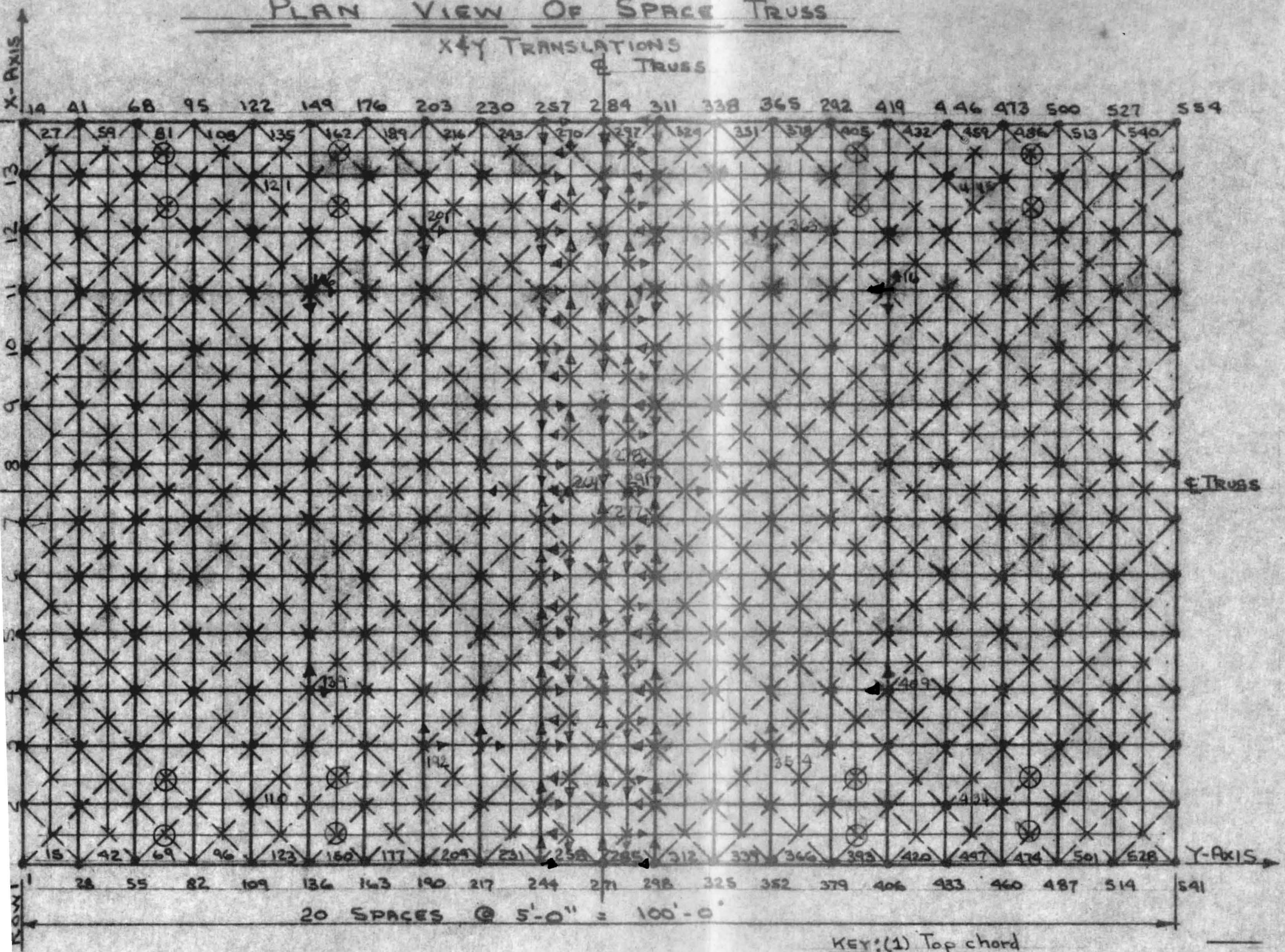
NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
1	1	0.35165D-02	-0.60347D-03	0.13021D-02	0.0	0.0	0.0
2	1	0.35513D-02	-0.23369D-02	0.12156D-01	0.0	0.0	0.0
3	1	0.33738D-02	-0.27361D-02	0.22292D-01	0.0	0.0	0.0
4	1	0.29688D-02	-0.14520D-02	0.30703D-01	0.0	0.0	0.0
5	1	0.23527D-02	-0.36156D-03	0.38085D-01	0.0	0.0	0.0
6	1	0.15198D-03	0.40570D-03	0.43704D-01	0.0	0.0	0.0
7	1	0.52620D-03	0.80457D-03	0.46746D-01	0.0	0.0	0.0
8	1	-0.52620D-03	0.80457D-03	0.46746D-01	0.0	0.0	0.0
9	1	-0.15198D-02	0.40570D-03	0.43704D-01	0.0	0.0	0.0
10	1	-0.23527D-02	-0.36156D-03	0.38085D-01	0.0	0.0	0.0
11	1	-0.29688D-02	-0.14520D-02	0.30703D-01	0.0	0.0	0.0
12	1	-0.33738D-02	-0.27361D-02	0.22292D-01	0.0	0.0	0.0
13	1	-0.35513D-02	-0.23369D-02	0.12156D-01	0.0	0.0	0.0
14	1	-0.35165D-02	-0.60347D-03	0.13021D-02	0.0	0.0	0.0

(Node numbers 15 to 108 have been omitted.)

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
109	1	0.52257D-02	0.48960D-03	-0.78386D-02	0.0	0.0	0.0
110	1	0.52953D-02	0.48499D-03	-0.83242D-04	0.0	0.0	0.0
111	1	0.63407D-02	0.66217D-03	0.10285D-01	0.0	0.0	0.0
112	1	0.68946D-02	0.10009D-02	0.25731D-01	0.0	0.0	0.0
113	1	0.60502D-02	0.13354D-02	0.42054D-01	0.0	0.0	0.0
114	1	0.40717D-02	0.15584D-02	0.54931D-01	0.0	0.0	0.0
115	1	0.14287D-02	0.16652D-02	0.61918D-01	0.0	0.0	0.0
116	1	-0.14287D-02	0.16652D-02	0.61918D-01	0.0	0.0	0.0
117	1	-0.40717D-02	0.15584D-02	0.54931D-01	0.0	0.0	0.0
118	1	-0.60502D-02	0.13354D-02	0.42054D-01	0.0	0.0	0.0
119	1	-0.68946D-02	0.10009D-02	0.25731D-01	0.0	0.0	0.0
120	1	-0.63407D-02	0.66217D-03	0.10285D-01	0.0	0.0	0.0

PLAN VIEW OF SPACE TRUSS

X & Y TRANSLATIONS



KEY: (1) Top chord
(2) Bottom chord
(3) Diagonals
(4) Direction of Translation

Figure 5 CASE III

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED
BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
121	1	-0.52953D-02	0.48499D-03	-0.83242D-04	0.0	0.0	0.0
122	1	-0.52257D-02	0.48960D-03	-0.78386D-02	0.0	0.0	0.0
(Node numbers 123 to 138 have been omitted.)							
139	1	0.81598D-02	0.20013D-02	0.29155D-01	0.0	0.0	0.0
(Node numbers 140 to 145 have been omitted.)							
146	1	-0.81598D-02	0.20013D-02	0.29155D-01	0.0	0.0	0.0
(Node numbers 147 to 189 have been omitted.)							
190	1	0.61069D-02	0.23576D-02	0.32630D-02	0.0	0.0	0.0
191	1	0.61766D-02	0.45230D-02	0.15916D-01	0.0	0.0	0.0
192	1	0.66911D-02	0.49556D-02	0.30193D-01	0.0	0.0	0.0
193	1	0.65247D-02	0.36550D-02	0.46630D-01	0.0	0.0	0.0
194	1	0.53818D-02	0.27320D-02	0.61884D-01	0.0	0.0	0.0
195	1	0.35106D-02	0.21402D-02	0.73300D-01	0.0	0.0	0.0
196	1	0.12161D-02	0.18464D-02	0.79362D-01	0.0	0.0	0.0
197	1	-0.12161D-02	0.18464D-02	0.79362D-01	0.0	0.0	0.0
198	1	-0.35106D-02	0.21402D-02	0.73300D-01	0.0	0.0	0.0
199	1	-0.53818D-02	0.27320D-02	0.61884D-01	0.0	0.0	0.0
200	1	-0.65247D-02	0.36550D-02	0.46630D-01	0.0	0.0	0.0
201	1	-0.66911D-02	0.49556D-02	0.30193D-01	0.0	0.0	0.0
202	1	-0.61766D-02	0.45230D-02	0.15916D-01	0.0	0.0	0.0
203	1	-0.61069D-02	0.23576D-02	0.32630D-02	0.0	0.0	0.0
(Node numbers 204 to 236 have been omitted.)							

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
237	1	-0.62419D-16	-0.11054D-02	0.86743D-01	0.0	0.0	0.0
		(Node numbers 238 to 243 have been omitted.)					
244	1	0.54349D-02	0.12696D-02	0.19082D-01	0.0	0.0	0.0
245	1	0.55045D-02	0.19694D-02	0.34081D-01	0.0	0.0	0.0
246	1	0.52850D-02	0.20101D-02	0.48982D-01	0.0	0.0	0.0
247	1	0.46934D-02	0.16233D-02	0.62758D-01	0.0	0.0	0.0
248	1	0.37279D-02	0.12350D-02	0.74437D-01	0.0	0.0	0.0
249	1	0.24092D-02	0.93523D-03	0.83008D-01	0.0	0.0	0.0
250	1	0.83416D-03	0.77482D-03	0.87558D-01	0.0	0.0	0.0
251	1	-0.83416D-03	0.77482D-03	0.87558D-01	0.0	0.0	0.0
252	1	-0.24092D-02	0.93523D-03	0.83008D-01	0.0	0.0	0.0
253	1	-0.37279D-02	0.12350D-02	0.74437D-01	0.0	0.0	0.0
254	1	-0.46934D-02	0.16233D-02	0.62758D-01	0.0	0.0	0.0
255	1	-0.52850D-02	0.20101D-02	0.48982D-01	0.0	0.0	0.0
256	1	-0.55045D-02	0.19694D-02	0.34081D-01	0.0	0.0	0.0
257	1	-0.54349D-02	0.12696D-02	0.19082D-01	0.0	0.0	0.0
258	1	-0.56147D-02	-0.82923D-03	0.28430D-01	0.0	0.0	0.0
259	1	-0.53646D-02	-0.76361D-03	0.43586D-01	0.0	0.0	0.0
260	1	-0.48073D-02	-0.75644D-03	0.57896D-01	0.0	0.0	0.0
261	1	-0.39775D-02	-0.66547D-03	0.70427D-01	0.0	0.0	0.0
262	1	-0.28676D-02	-0.53009D-03	0.80332D-01	0.0	0.0	0.0
263	1	-0.15082D-02	-0.42552D-03	0.86731D-01	0.0	0.0	0.0
264	1	-0.46147D-16	-0.38761D-03	0.88949D-01	0.0	0.0	0.0
265	1	-0.15082D-02	-0.42552D-03	0.86731D-01	0.0	0.0	0.0
266	1	0.28676D-02	0.53009D-03	0.80332D-01	0.0	0.0	0.0

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED
BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
267	1	0.39775D-02	-0.66547D-03	0.70427D-01	0.0	0.0	0.0
268	1	0.48073D-02	-0.75644D-03	0.57896D-01	0.0	0.0	0.0
269	1	0.53646D-02	-0.76361D-03	0.43586D-01	0.0	0.0	0.0
270	1	0.56147D-02	-0.82923D-03	0.28430D-01	0.0	0.0	0.0
271	1	0.52804D-02	-0.50686D-16	0.21729D-01	0.0	0.0	0.0
272	1	0.53500D-02	-0.29490D-16	0.36731D-01	0.0	0.0	0.0
273	1	0.50996D-02	0.52042D-17	0.51544D-01	0.0	0.0	0.0
274	1	0.44761D-02	0.35562D-16	0.65023D-01	0.0	0.0	0.0
275	1	0.35175D-02	0.37947D-17	0.76259D-01	0.0	0.0	0.0
276	1	0.22591D-02	-0.68522D-16	0.84414D-01	0.0	0.0	0.0
277	1	0.78030D-03	-0.11189D-15	0.88725D-01	0.0	0.0	0.0
278	1	-0.78030D-03	-0.10842D-15	0.88725D-01	0.0	0.0	0.0
279	1	-0.22591D-02	-0.87604D-16	0.84414D-01	0.0	0.0	0.0
280	1	-0.35175D-02	-0.37297D-16	0.76259D-01	0.0	0.0	0.0
281	1	-0.44761D-02	-0.19949D-16	0.65023D-01	0.0	0.0	0.0
282	1	-0.50996D-02	-0.26888D-16	0.51544D-01	0.0	0.0	0.0
283	1	-0.53500D-02	-0.38164D-16	0.36731D-01	0.0	0.0	0.0
284	1	-0.52804D-02	-0.22280D-16	0.21729D-01	0.0	0.0	0.0
285	1	-0.56147D-02	0.82923D-03	0.28430D-01	0.0	0.0	0.0
286	1	-0.53646D-02	0.76361D-03	0.43586D-01	0.0	0.0	0.0
287	1	-0.48073D-02	0.75644D-03	0.57896D-01	0.0	0.0	0.0
288	1	-0.39775D-02	0.66547D-03	0.70427D-01	0.0	0.0	0.0
289	1	-0.28676D-02	0.53009D-03	0.80332D-01	0.0	0.0	0.0
290	1	-0.15082D-02	0.42552D-03	0.86731D-01	0.0	0.0	0.0
291	1	-0.29077D-16	0.38761D-03	0.88949D-01	0.0	0.0	0.0
292	1	0.15082D-02	0.42552D-03	0.86731D-01	0.0	0.0	0.0
293	1	0.28676D-02	0.53009D-03	0.80332D-01	0.0	0.0	0.0
294	1	0.39775D-02	0.66547D-03	0.70427D-01	0.0	0.0	0.0

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED
BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
295	1	0.48073D-02	0.75644D-03	0.57896D-01	0.0	0.0	0.0
296	1	0.53646D-02	0.76361D-03	0.43586D-01	0.0	0.0	0.0
297	1	0.56147D-02	0.82923D-03	0.28430D-01	0.0	0.0	0.0
298	1	0.54349D-02	-0.12696D-02	0.19082D-01	0.0	0.0	0.0
299	1	0.55045D-02	-0.19694D-02	0.34081D-01	0.0	0.0	0.0
300	1	0.52850D-02	-0.20101D-02	0.48982D-01	0.0	0.0	0.0
301	1	0.46934D-02	-0.16233D-02	0.62758D-01	0.0	0.0	0.0
302	1	0.37279D-02	-0.12350D-02	0.74437D-01	0.0	0.0	0.0
303	1	0.24092D-02	-0.93523D-03	0.83008D-01	0.0	0.0	0.0
304	1	0.83416D-03	-0.77482D-03	0.87558D-01	0.0	0.0	0.0
305	1	-0.83416D-03	-0.77482D-03	0.87558D-01	0.0	0.0	0.0
306	1	-0.24092D-02	-0.93523D-03	0.83008D-01	0.0	0.0	0.0
307	1	-0.37279D-02	-0.12350D-02	0.74437D-01	0.0	0.0	0.0
308	1	-0.46934D-02	-0.16233D-02	0.62758D-01	0.0	0.0	0.0
309	1	-0.52850D-02	-0.20101D-02	0.48982D-01	0.0	0.0	0.0
310	1	-0.55045D-02	-0.19694D-02	0.34081D-01	0.0	0.0	0.0
311	1	-0.54349D-02	-0.12696D-02	0.19082D-01	0.0	0.0	0.0
(Node numbers 312 to 317 have been omitted.)							
318	1	-0.19597D-16	0.11054D-02	0.86743D-01	0.0	0.0	0.0
(Node numbers 319 to 351 have been omitted.)							
352	1	0.61069D-02	-0.23576D-02	0.32630D-02	0.0	0.0	0.0
353	1	0.61766D-02	-0.45230D-02	0.15196D-01	0.0	0.0	0.0
354	1	0.66911D-02	-0.49556D-02	0.30193D-01	0.0	0.0	0.0
355	1	0.65247D-02	-0.36550D-02	0.46630D-01	0.0	0.0	0.0

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
356	1	0.53818D-02	-0.27320D-02	0.61884D-01	0.0	0.0	0.0
357	1	0.35106D-02	-0.21402D-02	0.73300D-01	0.0	0.0	0.0
358	1	0.12161D-02	-0.18464D-02	0.79362D-01	0.0	0.0	0.0
359	1	-0.12161D-02	-0.18464D-02	0.79362D-01	0.0	0.0	0.0
360	1	-0.35106D-02	-0.21402D-02	0.73300D-01	0.0	0.0	0.0
361	1	-0.53818D-02	-0.27320D-02	0.61884D-01	0.0	0.0	0.0
362	1	-0.65247D-02	-0.36550D-02	0.46630D-01	0.0	0.0	0.0
363	1	-0.66911D-02	-0.49556D-02	0.30193D-01	0.0	0.0	0.0
364	1	-0.61766D-02	-0.45230D-02	0.15916D-01	0.0	0.0	0.0
365	1	-0.61069D-02	-0.23576D-02	0.32630D-02	0.0	0.0	0.0
		(Node numbers 366 to 408 have been omitted.)					
409	1	0.81598D-02	-0.20013D-02	0.29155D-01	0.0	0.0	0.0
		(Node numbers 410 to 415 have been omitted.)					
416	1	-0.81598D-02	-0.20013D-02	0.29155D-01	0.0	0.0	0.0
		(Node numbers 417 to 432 have been omitted.)					
433	1	0.52257D-02	-0.48960D-03	-0.78386D-02	0.0	0.0	0.0
434	1	0.52953D-02	-0.48499D-03	-0.83242D-04	0.0	0.0	0.0
435	1	0.63407D-02	-0.66217D-03	0.10285D-01	0.0	0.0	0.0
436	1	0.68946D-02	-0.10009D-02	0.25731D-01	0.0	0.0	0.0
437	1	0.60502D-02	-0.13354D-02	0.42054D-01	0.0	0.0	0.0
438	1	0.40717D-02	-0.15584D-02	0.54931D-01	0.0	0.0	0.0
439	1	0.14287D-02	-0.16652D-02	0.61918D-01	0.0	0.0	0.0
440	1	-0.14287D-02	-0.16652D-02	0.61918D-01	0.0	0.0	0.0

TABLE 12 CONTINUED

PARTIAL NODE DISPLACEMENTS AND ROTATIONS AS CALCULATED AND PRINTED BY THE COMPUTER FOR CASE III

NODE NUMBER	LOAD CASE	X-TRANSLATION	Y-TRANSLATION	Z-TRANSLATION	X-ROTATION	Y-ROTATION	Z-ROTATION
441	1	-0.40717D-02	-0.15584D-02	0.54931D-01	0.0	0.0	0.0
442	1	-0.60502D-02	-0.13354D-02	0.42054D-01	0.0	0.0	0.0
443	1	-0.68946D-02	-0.10009D-02	0.25731D-01	0.0	0.0	0.0
444	1	-0.63407D-02	-0.66217D-03	0.10285D-01	0.0	0.0	0.0
445	1	-0.52953D-02	-0.48499D-03	-0.83242D-04	0.0	0.0	0.0
446	1	-0.52257D-02	-0.48960D-03	-0.78386D-02	0.0	0.0	0.0
(Node numbers 447 to 540 have been omitted.)							
541	1	0.35165D-02	0.60347D-03	0.13021D-02	0.0	0.0	0.0
542	1	0.35513D-02	0.23369D-02	0.12156D-01	0.0	0.0	0.0
543	1	0.33738D-02	0.27361D-02	0.22292D-01	0.0	0.0	0.0
544	1	0.29688D-02	0.14520D-02	0.30703D-01	0.0	0.0	0.0
545	1	0.23527D-02	0.36156D-03	0.38085D-01	0.0	0.0	0.0
546	1	0.15198D-02	-0.40570D-03	0.43704D-01	0.0	0.0	0.0
547	1	0.52620D-03	-0.80457D-03	0.46746D-01	0.0	0.0	0.0
548	1	-0.52620D-03	-0.80457D-03	0.46746D-01	0.0	0.0	0.0
549	1	-0.15198D-02	-0.40570D-03	0.43704D-01	0.0	0.0	0.0
550	1	-0.23527D-02	0.36156D-03	0.38085D-01	0.0	0.0	0.0
551	1	-0.29688D-02	0.14520D-02	0.30703D-01	0.0	0.0	0.0
552	1	-0.33738D-02	0.27361D-02	0.22292D-01	0.0	0.0	0.0
553	1	-0.35513D-02	0.23369D-02	0.12156D-01	0.0	0.0	0.0
554	1	-0.35165D-02	0.60347D-03	0.13021D-02	0.0	0.0	0.0

TABLE 13

STRESSES AND FORCES AS CALCULATED AND PRINTED BY THE COMPUTER FOR CASE III

MEMBER	LOAD	STRESS (ksi)	FORCE (kips)	NODES
1	1	0.21	0.127	1 to 2
2	1	-1.07	-0.646	2 to 3
3	1	-2.43	-1.474	3 to 4
4	1	-3.70	-2.242	4 to 5
5	1	-5.00	-3.030	5 to 6
6	1	-5.96	-3.615	6 to 7
7	1	-6.31	-3.829	7 to 8
8	1	-5.96	-3.615	8 to 9
9	1	-5.00	-3.030	9 to 10
10	1	-3.70	-2.242	10 to 11
11	1	-2.43	-1.474	11 to 12
12	1	-1.07	-0.646	12 to 13
13	1	0.21	0.127	13 to 14
14	1	0.21	0.127	1 to 28
15	1	0.42	0.253	2 to 29
16	1	0.42	0.253	3 to 30
17	1	0.42	0.253	4 to 31
18	1	0.42	0.253	5 to 32
19	1	0.42	0.253	6 to 33
20	1	0.42	0.253	7 to 34
21	1	0.42	0.253	8 to 35
22	1	0.42	0.253	9 to 36
23	1	0.42	0.253	10 to 37
24	1	0.42	0.253	11 to 38
25	1	0.42	0.253	12 to 39
26	1	0.42	0.253	13 to 40
27	1	0.21	0.127	14 to 41
28	1	-0.43	-0.259	1 to 15

(Members 29 to 235 have been omitted.)

MEMBER	LOAD	STRESS (ksi)	FORCE (kips)	NODES
236	1	1.58	0.960	55 to 69
237	1	7.31	4.433	69 to 56
238	1	-14.15	-8.578	70 to 57
239	1	-19.78	-11.993	71 to 58

(Members 240 to 261 have been omitted.)

MEMBER	LOAD	STRESS (ksi)	FORCE (kips)	NODES
262	1	6.21	3.764	94 to 81
263	1	10.00	6.069	69 to 83
264	1	-13.59	-8.242	70 to 84
265	1	-17.15	-10.400	71 to 85

(Members 266 to 547 have been omitted.)

MEMBER	LOAD	STRESS (ksi)	FORCE (kips)	NODES
548	1	4.89	2.965	136 to 150
549	1	-10.47	6.347	150 to 137
550	1	-18.65	-11.308	151 to 138
551	1	-23.89	-14.484	152 to 139

TABLE 13 CONTINUED

STRESSES AMD FORCES AS CALCULATED AND PRINTED
BY THE COMPUTER FOR CASE III

MEMBER	LOAD	STRESS (ksi)	FORCE (kips)	NODES
574	1	0.41	0.248	175 to 162
575	1	8.64	5.240	150 to 164
576	1	-22.51	-13.649	151 to 165
577	1	-31.00	-18.800	152 to 166
(Members 578 to 845 have been omitted.)				
846	1	-5.28	-3.203	217 to 244
847	1	-9.80	-5.943	218 to 245
848	1	-10.60	-6.427	219 to 246
(Members 849 to 923 have been omitted.)				
924	1	8.15	4.945	231 to 258
925	1	8.17	4.954	232 to 259
926	1	8.35	5.066	233 to 260
927	1	7.26	4.400	234 to 261
928	1	5.76	3.495	235 to 262
929	1	4.49	2.842	236 to 263
930	1	4.31	2.611	237 to 264
(Members 931 to 949 have been omitted.)				
950	1	-7.62	-4.619	244 to 271
951	1	-11.82	-7.165	245 to 272
952	1	-12.06	-7.313	246 to 273
953	1	-9.74	-5.906	247 to 274
954	1	-7.41	-4.493	248 to 275
955	1	-5.61	-3.403	249 to 276
956	1	-4.65	-2.819	250 to 277
(Members 957 to 1027 have been omitted.)				
1028	1	9.95	6.034	258 to 285
1029	1	9.16	5.556	259 to 286
1030	1	9.08	5.504	260 to 287
1031	1	7.99	4.842	261 to 288
1032	1	6.36	3.857	262 to 289
1033	1	5.11	3.096	263 to 290
1034	1	4.65	2.820	264 to 291
1035	1	5.11	3.096	265 to 292
1036	1	6.36	3.857	266 to 293
1037	1	7.99	4.842	267 to 294
1038	1	9.08	5.504	268 to 295
1039	1	9.16	5.556	269 to 296
1040	1	9.95	6.034	270 to 297

CHAPTER IV

COMPUTER OUTPUT INTERPRETATION

4.1 Space Truss Deflections

In any structural analysis the most important results are generally the points of maximum deflection, maximum moment, maximum shear, and maximum stress.(4) Most naturally, since we are working with a three dimensional space truss, we are mainly interested in the maximum deflections and maximum stresses (tension or compression) of the members. Since Case III yielded the lowest stresses, it is the only case considered here.

Since this program calculated the deflections (or translations) in the X, Y, and Z directions, the maximum deflections, as noted in Table 12, are shown on Figure 5, with small arrows pointing in the direction of deflection giving a general picture of the mode shape in the X and Y plane, and Table 12 is a summary of only the most important deflections taken from the computer printout for Case III.

Looking at Figure 5 it is interesting to note that the top chord translations near the two centerlines (X and Y) all point toward these centerlines regardless of the quadrant in which they appear, if the space truss is divided into four quadrants around the centerline. Of particular note are nodes 244 through 257 and nodes 298 through 311 for the top chord schematic on Figure 5. In contrast, the bottom chord translations always point away from their respective centerlines regardless of the quadrant in which they appear. Particularly note bottom chord nodes 258 through 270 and node 285 through node 297 on Figure 5.

Looking at the centerline translations on the bottom chord, note that the deflections along the X centerline all point away from the Y centerline and are symmetrical. The X deflection is zero along this line which most naturally should be true since the structure is supported and loaded symmetrically (See nodes 264 and 291 in Table 12).

As for the top chord translations along the Y centerline, the X translations along the Y centerline all point towards the X centerline or the center of the structure, while the Y translations are zero which we would also expect because of the bending of the symmetrical structure (See nodes 271 through 284 in Table 12).

Noting Table 12, the maximum Z deflection which was considered plus down and negative up, occurs on the bottom chord at nodes 264 and 291 which are the closest bottom nodes to both centerlines. The two top chord nodes which are closest to the centerline of the structure are 277 and 278, and have almost the same Z deflections, being $.88949 \times 10^{-1}$ ft. (bottom chord) vs. $.88725 \times 10^{-1}$ ft. (top chord) (See Table 12). The maximum negative Z deflection or uplift occurs at nodes 109, 122, 433 and 446 which is also symmetrical and about 10 times less than the downward or positive deflection. It is interesting to note that the maximum uplift occurs between the supports at all four symmetrical points which could be expected as in the analogy with a cantilever beam.

The maximum translations or deflections in the X and Y directions are not as easily predicted. However, from Table 12 it is seen that nodes 139, 146, 409 and 416 produce the maximum X translation (0.81598×10^{-2} ft.). Nodes 139 and 146 have equal but opposite deflection in the X direction as do nodes 409 and 416. The maximum deflection in the Y direction also occurs at four symmetrical points or at nodes 192, 202, 354 and 363. (0.49556×10^{-2} ft.). Again note that nodes 192 and 202 have equal but opposite deflection in the Y direction as do nodes 354 and 363.

4.2 Space Truss Member Stress

After the translation or deflection investigation, the next step in any truss calculation is to analyze the member stresses. In our case all that is necessary is to pick out the maximum stresses from the computer printout given in the abbreviated form in Table 13, showing only the maximum stresses near the boundary supports and some of the most important center members where maximum stresses would be expected.

From Table 13 it is obvious that the largest stresses for Case III occur in the diagonals at or near the supports. The largest stress encountered (31.0 ksi compression), shown by diagonal heavy double lines on Figure 6, occurred

PLAN VIEW OF SPACE TRUSS

HEAVY DARK LINES INDICATING OVERSTRESSED MEMBERS

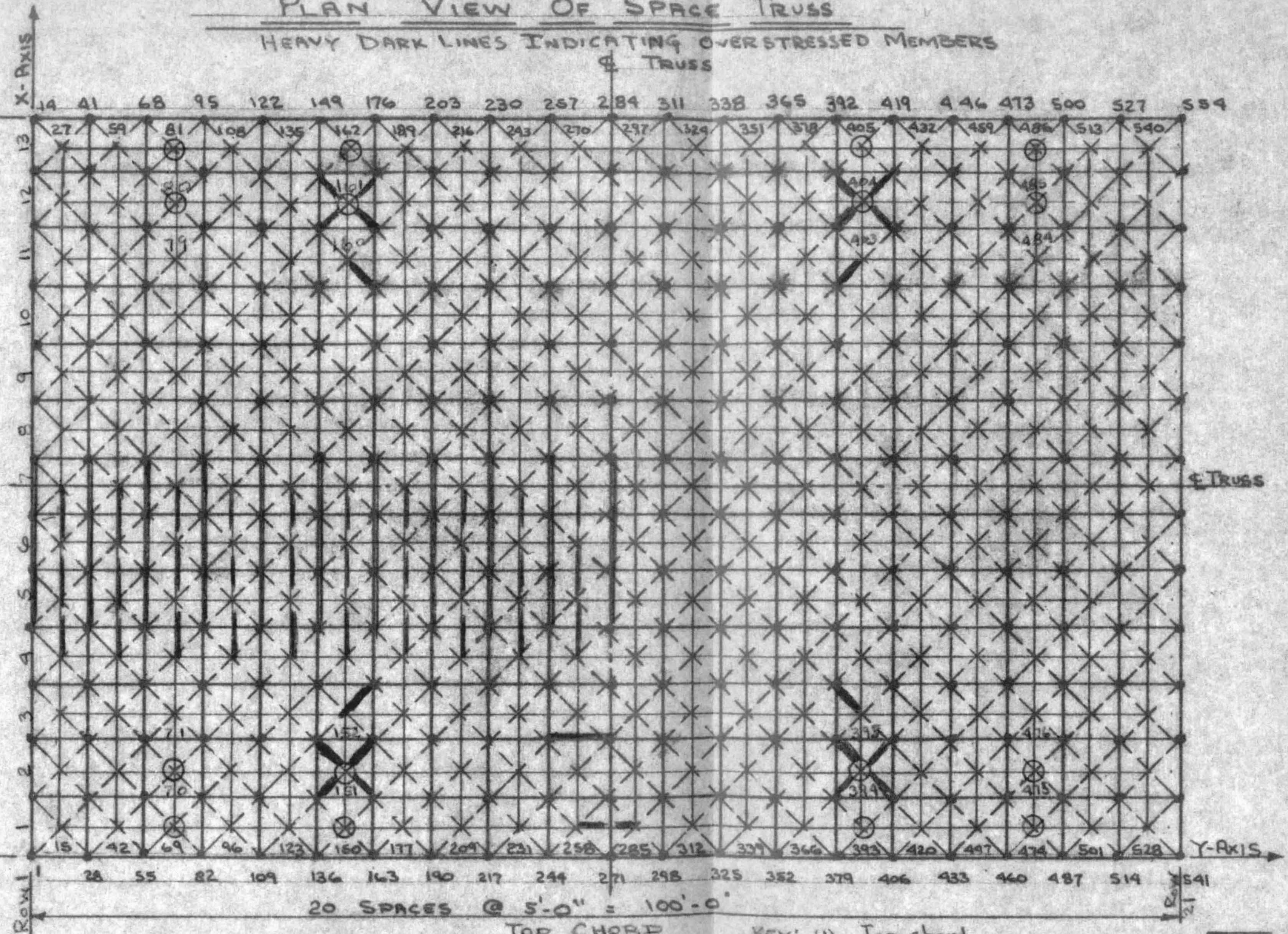


Figure 6 Case III

KEY: (1) Top chord
(2) Bottom chord
(3) Diagonals

one bay or square in towards the center of the space truss from the four inner supports. The member numbers are 579 (node 152 to node 166), 594 (node 160 to node 173), 1489 (node 395 to node 382) and member number 1504 (node 403 to node 389), which are really the same member in this symmetrical structure. Also shown by heavy diagonal lines on Figure 6 are the overstressed diagonals at the four inner supports. Not only are these members overstressed, they are also in compression, which naturally by the L/r ratio requires a reduction of their allowable stress. This condition requires the substitution of a larger member (in cross section area) in these locations. A true design of these members is not the aim of this thesis. The intent is only to compare results of different support cases.

Since, as expected, the maximum stresses occur in the center of the structure, these members are also included in Table 13. Members 950 to 956 are the center members on the top chord paralleling the Y axis, and as shown (Figure 6) where the maximum stress is 12.06 ksi in compression. The bottom chords in the same area as shown in Table 13 (member numbers 1028 to 1040) which are in tension only, reach a stress of 9.95 ksi in member 1028 (node 258 to node 285), the first member in from the Y axis on the bottom chord, which is unexpected.

4.3 Comparison of Member Stresses for Cases I, II, and III

Tables 14 and 15 are compiled to show the relative stresses obtained in the various cases for the same members, (shown in heavy lines on Figure 6). It is obvious that Case III yields much lower stresses in both the top and bottom chords. In fact, Case III yielded much lower stresses in most all of the members which was expected because of the location and distribution of boundary supports.

Practically the entire structure would require larger members for Cases I and II as compared with Case III. In comparison to Case I, Case II gives some lower stresses in the lower chord members but not enough to make any appreciable difference considering that the top chord members in Case II are much higher than those in Case I. Therefore Case I yields lower stress conditions than Case II.

Case III on the other hand yields lower stress conditions than either Case I or Case II which is predictable because of the additional constraining supports.

TABLE 14

MAJOR TOP CHORD STRESSES (ksi) FOR ALL CASES FOR ¼ OF SPACE TRUSS

ROW	MEMBER	CASE I	CASE II	CASE III	NODES
1	5	-7.11	6.78	5.00	5 to 6
	6	-8.40	-8.10	-5.96	6 to 7
	7	-8.88	-8.58	-6.31	7 to 8
2	109	-9.75	-16.48	-7.24	32 to 33
	110	-11.77	-19.10	-9.22	33 to 34
	111	-12.47	-20.00	-9.89	34 to 35
3	213	-10.54	-21.12	-10.25	59 to 60
	214	-13.10	-23.69	-12.69	60 to 61
	215	-13.97	-24.55	-13.50	61 to 62
4	317	-9.69	-24.86	-11.50	86 to 87
	318	-13.17	-27.15	-14.80	87 to 88
	319	-14.22	-27.87	-15.89	88 to 89
5	421	-6.33	-26.88	-11.87	113 to 114
	422	-11.37	-28.87	-15.86	114 to 115
	423	-14.20	-29.48	-17.14	115 to 116
6	525	-10.90	-26.72	-14.13	140 to 141
	526	-14.12	-28.54	-17.28	141 to 142
	527	-15.12	-29.11	-18.29	142 to 143
7	629	-8.87	-23.85	-14.23	167 to 168
	630	-12.87	-26.81	-16.70	168 to 169
	631	-15.99	-26.10	-17.48	169 to 170

TABLE 14 CONTINUED

MAJOR TOP CHORD STRESSES (ksi) FOR ALL CASES FOR ¼ OF SPACE TRUSS

ROW	MEMBER	CASE I	CASE II	CASE III	NODES
8	733	-12.94	-19.98	-11.23	194 to 195
	734	-15.18	-22.64	-13.77	195 to 196
	735	-15.94	-23.52	-14.59	196 to 197
9	837	-11.92	-16.60	-9.08	221 to 222
	838	-14.06	-19.34	-11.15	222 to 223
	839	-14.08	-20.28	-11.86	223 to 224
10	941	-10.98	-14.41	-7.91	248 to 249
	942	-12.96	-17.05	-9.45	249 to 250
	943	-13.65	-17.99	-10.01	250 to 251
11	1045	-10.62	-13.66	-7.55	275 to 276
	1046	-12.52	-16.24	-8.87	276 to 277
	1047	-13.20	-17.17	-9.37	277 to 278

TABLE 15

MAJOR BOTTOM CHORD STRESSES (ksi) FOR ALL CASES FOR ¼ OF SPACE TRUSS

ROW	MEMBER	CASE I	CASE II	CASE III	NODES
1	83	14.60	16.79	4.13	18 to 19
	84	17.70	20.70	6.05	19 to 20
	85	19.54	22.86	7.15	20 to 21
2	187	17.49	18.89	4.19	45 to 46
	188	21.25	22.62	7.11	46 to 47
	189	23.19	24.53	8.60	47 to 48
3	291	22.81	21.38	3.34	72 to 73
	292	26.42	24.86	7.72	73 to 74
	293	28.18	26.52	9.82	74 to 75
4	395	25.59	23.23	4.16	99 to 100
	396	29.52	26.43	9.01	100 to 101
	397	31.32	27.86	11.40	101 to 102
5	499	26.91	23.96	4.78	126 to 127
	500	30.43	26.79	9.65	127 to 128
	501	32.00	28.04	11.99	128 to 129
6	603	25.19	22.71	4.74	153 to 154
	604	28.23	25.59	9.38	154 to 155
	605	29.65	26.91	11.52	155 to 156
7	707	19.13	19.74	6.15	180 to 181
	708	22.81	23.13	9.62	181 to 182
	709	24.61	24.75	11.28	182 to 183

TABLE 15 CONTINUED

MAJOR BOTTOM CHORD STRESSES (ksi) FOR ALL CASES FOR ¼ OF SPACE TRUSS

ROW	MEMBER	CASE I	CASE II	CASE III	NODES
8	811	15.32	16.93	6.29	207 to 208
	812	18.63	20.43	9.13	208 to 209
	813	20.36	22.22	10.58	209 to 210
9	915	13.01	14.84	6.49	234 to 235
	916	15.94	18.17	8.51	235 to 236
	917	17.47	19.98	9.65	236 to 237
10	1019	12.06	13.75	6.66	261 to 262
	1020	14.61	16.92	8.16	262 to 263
	1021	16.02	18.68	9.05	263 to 264

4.4 Beam Model Analysis

One alternate method to obtain a reasonable check of the finite element analysis is to model the three dimensional space truss as a simple two dimensional beam in the long direction. In addition, it is assumed that the simple beam is supported by reactions placed between existing truss supports as shown in Figure 7.

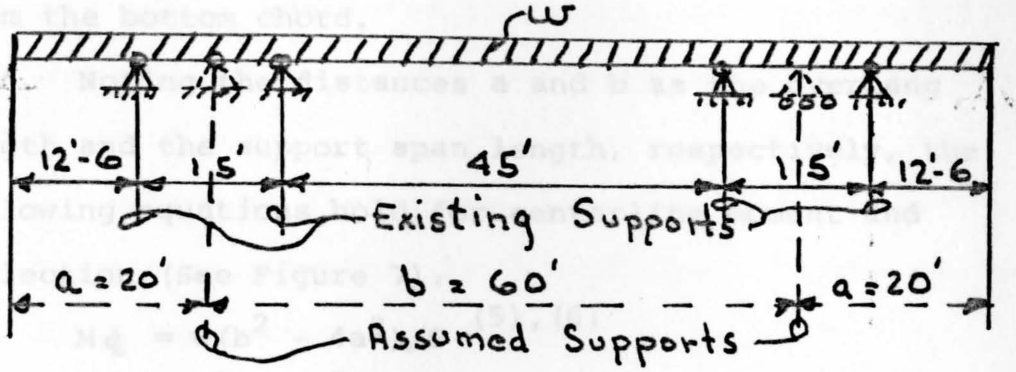


Figure 7. Simple Beam Model

The uniform load "w" placed on the beam is calculated as $w = 30\#/\text{ft.}^2 \times 65' \div 1000 = 1.95$ kips/ft.

The second moment of area (moment of inertia) of the beam model is calculated using the upper and lower chord members of the truss as shown in Figure 8.

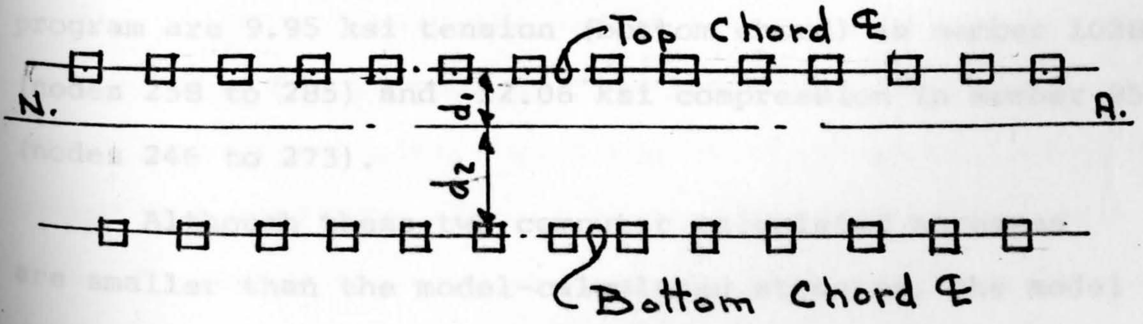


Figure 8. Space Truss Cross Section

The neutral axis of the cross section is first calculated and the transfer axis theorem is incorporated to produce the value of I (second moment of area). Noting the cross sectional area of each chord member is 0.6064 sq. inches and the distance between centerline of top and bottom chord is 45 inches, it follows that the combined moment of inertia for the space truss was computed to be 7,791 in.^{4} with the neutral axis 21.38" from the top chord and 23.02" from the bottom chord.

Noting the distances a and b as the overhang length and the support span length, respectively, the following equations hold for centerline moment and deflection (See Figure 7):

$$M_{℄} = w(b^2 - 4a^2)/8 \; ^{(5),(6)} \tag{6}$$

$$\Delta_{℄} = b^2 w(24a^2 - 5b^2)/384\,EI \tag{7}$$

Computing the member tension and compression forces, respectively, utilizing the results of Equation 6 yields stresses of 13.89 ksi (bottom chord) and -12.91 ksi (top chord). By comparison, the largest stresses found in this same area, the top and bottom chords parallel to the Y axis at the centerline of the truss, as given by the program are 9.95 ksi tension (bottom chord) in member 1028 (nodes 258 to 285) and -12.06 ksi compression in member 952 (nodes 246 to 273).

Although these two computer calculated stresses are smaller than the model-calculated stresses, the model stresses do lend a certain degree of acceptance to the computer analysis.

The maximum model-calculated deflection obtained by Equation (7) is 0.5644 inches or 0.0470 ft. compared to 0.0887 ft. as obtained by the computer analysis (in the Z direction). This maximum Z translation by the computer analysis shown in Table 12 occurs at nodes 277 and 278 at the center of the structure. Again, the deflection, even though roughly calculated and approximately one-half the computer analysis, realistically compares with the computer solution.

Looking at the entire structure we see that the space truss is structurally indeterminate to the 466 degree for Case III, computed as follows:

The degree of indeterminacy = (No. of Members) + (3) (No. of Reactions) - (3) (No. of Joints or Nodes

= (2080) + (3) (16) - (3) (554)

= 466

It is obvious that any hand-calculated solution of a space truss of this magnitude and indeterminacy would be tremendously time-consuming if not impossible. SAP IV, on the other hand, requires time and effort, but within reasonable time limits. Even though the computer solution requires significant time to prepare the input data and about 8 minutes of computer core time for every try or run, the comparatively effortless mathematical solutions are well worth the effort.

CHAPTER V

DISCUSSION AND CONCLUSIONS

5.1 Discussion

As shown in Chapter IV approximately seventy percent of the 2080 members of the space truss were overstressed in both Case I and Case II. Case III, with bottom chord support points, was then introduced with double the number of supports in an attempt to reduce the number of overstressed members. This support case resulted in a reduction to approximately 1% (20 members -- 5 at each internal support) of the 2080 member space truss being overstressed. Although the design of the columns was not a consideration in this thesis, one other advantage of the bottom-supported space truss (Case III) is the reduction of the L/r ratio of the support columns due to the depth factor of the space truss itself.

To further study the advantages and disadvantages of different support cases, it is recommended that additional cases be investigated with top chord supports equal in number as in Case III. This would give a good comparison of the two support conditions. It should be noted that top-chord supports induce tension in the adjacent diagonal members and bottom-chord supports produce compression in these members.

The solution of a space truss of this size (2080 members and 554 nodes) as determined by any other method other than with a computer program analysis would be an immensely time-consuming job, especially with the high degree of static indeterminacy which is inherent in the complex geometry. Even after an accurate computer solution is obtained, however, one must consider the degree of conservativeness of the calculated member forces (stresses). In this solution it is assumed that the truss members are pin-connected. Thus the solutions are basically conservative because we ignore the actual resistance of the truss in bending and the span end moments due to the natural rigidity of all bolted connections, which in turn reduces the deflections.

It is further recommended that this problem be analyzed for different loading conditions (namely snow load and wind load) as in any normal truss problem. A dynamic analysis to obtain the lowest three natural frequencies is appropriate. The SAP IV program possesses the ability to investigate these additional loading conditions with great efficiency.

Cases I, II, and III were run with all members having the same cross-sectional area. A word of caution in designing, which is encountered in this type of solution, is appropriate at this point. There is a tendency to increase the member sizes for overstressed members without rerunning the program with the larger member areas in the program. Since any geometric size change would change the stiffness matrix, the more-stiff members would have a tendency to increase the stress levels in the less-stiff members. Since the actual design of the truss was not the aim of this thesis, this solution was not rerun for the overstressed members. It would be an interesting observation for future work.

Even though SAP IV has been updated and SAP V is now available, one addition to the printout tables, for ease of analysis, would be the inclusion of the node numbers to the printout table for stress as was added to Table 13. This addition eliminates the need to leaf back and forth through the printout tables to analyze the solutions.

5.2 Conclusions

This 2080 member space truss with 554 pinned nodes, 466 degrees of indeterminacy and different support conditions consumed a considerable amount of this user's time. Any amateur, as this user was, should be prepared to spend between 15 to 20 hours a week for about 9 months to go through the task of using SAP IV.

It is anticipated that this thesis work will greatly reduce the amount of preparation time necessary for a beginning user to efficiently and effectively utilize the program. It is generally recommended that relatively small problems be used for initiation purposes before large problems of the size considered in this thesis be attempted.

Once all data input debugging process was accomplished, the different support conditions were then easily read into the computer. One additional suggestion when debugging the input data is to have an independent party review the input data, as we all have a tendency to read over our own mistakes. It takes only one typing error to destabilize the structure and to give the user an error output, which for this particular problem utilizes 8 minutes of C.P.U. time and about ½ hour of printout time for 7000 lines of numerical output.

The updating and modifications of SAP IV to SAP V (7) was performed in 1978. The following additions were incorporated:

(1) A pre- and postprocessor graphics subroutine which activates a Calcomp or SC 4020 plotter was added. This subroutine plots the undeformed and deformed structure and the vibration mode shapes of a structure.

(2) A bandwidth minimization subroutine was added to the program. The user numbers the node points of the structure in any manner he wishes. He then indicates, by an input parameter to the computer, the bandwidth minimization is desired. The bandwidth minimization is then activated and the node points are renumbered to give bandwidth minimization. The structural problem is then

solved using the new node point numbering system.

(3) A general finite element was added to the program to enable the user to include the effect of structural components which cannot be determined from the finite elements in the SAP IV program.

(4) A geometric stiffness matrix for plate and beam elements was added to SAP IV to consider the effect of external loading on the natural frequencies of the system. Buckling can be obtained by increasing the loading until a zero frequency value is obtained.

(5) A frequency response analysis was added to the program in which mode shapes and frequencies computed for dynamic analysis can be used to calculate the response to steady state variable frequency sinusoidal base motion.

(6) A response spectrum analysis option which combines modal shapes for the dynamic response analysis to satisfy the N.R.C. Regulatory Guide Requirement 1.92 was also added to the program.

BIBLIOGRAPHY

1. Bathe, K.; Wilson, E.; and Peterson, F. SAP IV: A Structural Analysis Program for Static and Dynamic Response of Linear Systems. University of California, College of Engineering, Berkeley, California, April 1974.

2. Clough, Ray W., and Penzien, Joseph. Dynamics of Structures. New York: McGraw-Hill Book Company, Inc., 1975.

3. Ural, Oktay. Finite Element Method: Basic Concepts and Applications. New York: Intext Press, Inc., 1973.

4. Timoshenko, S.P., and Young, D.H. Theory of Structures. New York: McGraw-Hill Book Company, Inc., 1965.

5. Cernica, John N. Strength of Materials. New York: Holt, Rinehart and Winston, Inc., 1966.

6. Roark, Raymond J. Formulas for Stress and Strain. New York: McGraw-Hill Book Company, Inc., 1954.

7. Miura, H., and Imam, H. SAP V: A General Three Dimensional Linear, Static, and Dynamic Finite Element Structural Analysis Program. Structural Mechanics Software Series II, University Press of Virginia, University of Virginia, 1978.