STRUCTURAL ANALYSIS OF BUILDING FRAMES

by

D. Page Cotton Jr.

A Thesis Submitted to the Faculty of the

DEPARTMENT OF CIVIL ENGINEERING
AND ENGINEERING MECHANICS

In Partial Fulfillment of the Requirements
For the Degree of

MASTER OF SCIENCE
WITH A MAJOR IN CIVIL ENGINEERING

In The Graduate College

THE UNIVERSITY OF ARIZONA

1 9 7 0

STATEMENT BY AUTHOR

This thesis has been submitted in partial fulfillment of requirements for an advanced degree at The University of Arizona and is deposited in the University Library to be made available to borrowers under rules of the Library.

Brief quotations from this thesis are allowable without special permission, provided that accurate acknowledgment of source is made. Requests for permission for extended quotation from or reproduction of this manuscript in whole or in part may be granted by the head of the major department or the Dean of the Graduate College when in his judgment the proposed use of the material is in the interests of scholarship. In all other instances, however, permission must be obtained from the author.

SIGNED: [signature]

APPROVAL BY THESIS DIRECTOR

This thesis has been approved on the date shown below:

[signature]
DR. R.M. RICHARD
Professor of Civil Engineering
and Engineering Mechanics

Aug. 18, 1969
Date

ACKNOWLEDGMENT

The Author wishes to express his gratitude to the following for their encouragement and useful criticism which aided in the development and presentation of this thesis:

Dr. Ralph M. Richard, Professor of Civil Engineering, for his overall guidance and his assistance in the theoretical and programing aspects of the thesis.

Wilber D. Birchler, graduate student, for his helpful suggestions in the basic mechanics used in the program.

W. T. Hamlyn, Professional Engineer and Lecturer, for his aid in understanding the practicing engineer's problems and for his help in selecting the example structures.

Dale R. Birtch, Architect and Engineer, for his criticisms of the practical aspects of the program.

Nancy Horwath, English Teacher, for her invaluable assistance in proof reading the manuscript.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF ILLUSTRATIONS--*Continued*

LIST OF TABLES

ABSTRACT

A method to program the structural analysis of building frames was developed in this thesis. Special consideration was given to the problems encountered by the engineer with respect to the building codes. The program has been made encompassing enough to handle practical engineering problems while being short enough to hold down computer expenses.

The direct stiffness method of finite element analysis is used to find nodal displacements and member end forces. Separate dead, live, wind, and seismic load vectors are generated by the program. Member end forces are solved for each loading condition. Maximum and minimum end forces are found for any possible pattern of live loadings, thus giving the engineer a complete frame analysis from which to work.

The program was applied to example problems and the results compared by other methods of analysis. Computer time to run the complete analysis of each example is given and considerable savings in time and expense may be noted. Solutions were obtained using the CDC 6400 computer at the Computer Center, University of Arizona, Tucson, Arizona.

CHAPTER I

INTRODUCTION

The increasing availability of digital computers is making access to programed analysis possible for even the smallest engineering office. To date, the emphasis on computer methods in structural engineering has been largely pioneered by the aero-space industries. Designs can be well defined within rigid optimization requirements, by the computer. Aero-space industries, being less oriented to tradition, are willing to invest time and money for research and development of computer programs. The building industry on the other hand is steeped in tradition as exemplified by the building codes. The need for a highly refined structural solution is not as critical, and few engineers can afford to develop computer programs except for the largest structures. Academic attempts to write a workable program have been so encompassing as to cause it to be uneconomical to run the program. An attempt is made here to develop a program of practical significance. Emphasis is placed on the maximum amount of significant output for minimum amount of computer time. The program is restricted to analysis only, as programs organized for design optimization take an inappropriate amount of time. A workable program to be of use to the small engineering office is herein presented.

CHAPTER II

THE DESIGN PROCESS

The development of a building from its conception to the completion of the construction is a complex process. This development process may be broken down into three major stages: the design stage, the analysis stage, and the construction stage. The prospective owner must first recognize the need for the building and engage the professional personnel necessary to carry out the design. Preliminary schemes are pursued considering the physical and economic limitations, the legal requirements, and the owner's desires. A solution is reached based on a qualitative analysis of the design criteria. In the second stage, the design solution is refined based on a quantitative analysis. The architect coordinates the various efforts making certain all the pieces of the plan fit together and conform to the design solution. Working drawings and specifications are produced in this stage. Finally in the third stage, the contractor constructs the building in accordance with the drawings and specifications under the supervision of the architect and his consultants. The aesthetic qualities, architectural and structural, and the ease of construction will determine the success of the design in the eyes of the building team; however, the real success can be measured only by the satisfaction of those who must use the building.

A computer program which handles the design stages of an architectural problem is impractical. There are too many variables and the time investment for an analysis would be economically unfeasible. Designs of structures such as highway bridges have been successful because of the limited number of criteria to be satisfied. The sole function of a bridge is to carry traffic over some barrier. Other design factors such as span length, soil bearing pressures, and material to be used are limited in number. Programs have been developed to design the structure for some specified optimum; for example, the most economical span length and member sizes as compared to a specified material and set of loads. The design of a building consists of the synthesis of many seemingly unrelated criteria. The function alone can be complex and determinants such as esthetics are difficult to define. The design solution is based on subjective reasoning, not objective reasoning. The digital computer is not the appropriate tool for subjective logic for the intuition of the designer cannot be programed.

The building design evolves as dictated by the owner and molded by the architect. Ideally the consulting engineers should be present during the planning stages to influence the design toward a logical engineering solution. All too often, however, the structural engineer is presented with the architectural plan and faces the problem of providing an adequate structure within the confines of that plan. The engineer's job becomes one of merely sizing the structure already designated by the architect.

Conventional methods of structural analysis are based on a prior knowledge of the stiffness of the structure. The engineer must first complete an approximate analysis for the stresses in the structure to obtain some idea as to the size required for the members. The approximate analysis is made in part on the engineers' experience and intuition, and takes a relatively short amount of time. Preliminary selection of member sizes is made and then a detailed analysis to determine the adequacy of the selection is applied. There are various exact and approximate methods of analysis available. Exact analysis by classical methods such as slope deflection or moment distribution may be used. These methods, though somewhat cumbersome, work quite well for analyzing stresses due to gravity loads. Analysis of stresses due to lateral loads, however, is quite another matter. Classical methods are unacceptably tedious especially in tall buildings. Approximate methods, such as the cantilever or portal methods, may be used and these work reasonably well except for tall buildings or discontinuities such as the top or bottom of the frame or setbacks. An exact elastic analysis is a time consuming process which the computer can be programed to perform with much greater speed and accuracy. The criteria of the stress analysis are all well definable and thus the problem is well suited for programing. The computer can be programed to give exact solutions for any loading condition and programs can be developed to handle a plane frame of the structure or the entire three dimensional frame. Thus, torsional effects on the frame due to lateral loads may be handled.

The engineer's job does not end with the completion of the structural analysis. Some members may prove to be inadequate and will need to be resized. The computer may be used to reanalyze the modified structure. When the member sizes have been finalized, the engineer must design his details and produce the set of specifications and drawings related to the structural portion of the building. During the construction stage, periodic inspections to ensure conformation to the design will be carried out.

Computer programing enters the design process in the analysis stage. Its primary function is to relieve the engineer of the mathematical calculations involved in the structural analysis thus freeing him from much of the tedium of analysis. Hopefully, the time saved could be spent in the preliminary stages of the design helping to coordinate and solidify the architects ideas resulting in an improved overall solution.

CHAPTER III

THE PROGRAM

The Basic Theory

The program is based on a variation of the displacement method of analysis called the Direct Stiffness Method (Turner et al., 1956). The displacement method considers the structure as an assemblage of finite or individual members (See Fig. 1). The connecting points of the members are called nodes and the unknowns are the displacements of those nodes. For the two dimensional or plane case, three displacements for each node are considered: horizontal, vertical, and rotational (See Fig. 2).

Nodal displacements and the member displacements of those members associated with a particular node must be consistant. A compatibility matrix correlates the displacements so that

$$V = B \cdot D \qquad (3\text{-}1)$$

where:

V is the member displacement vector

B is the compatibility matrix

D is the nodal displacement vector

At the same time, the member displacements are related to

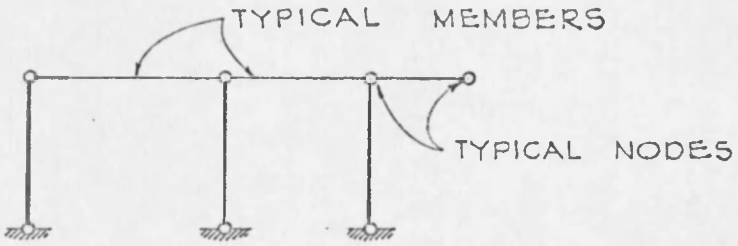


FIG. 1 TYPICAL STRUCTURE

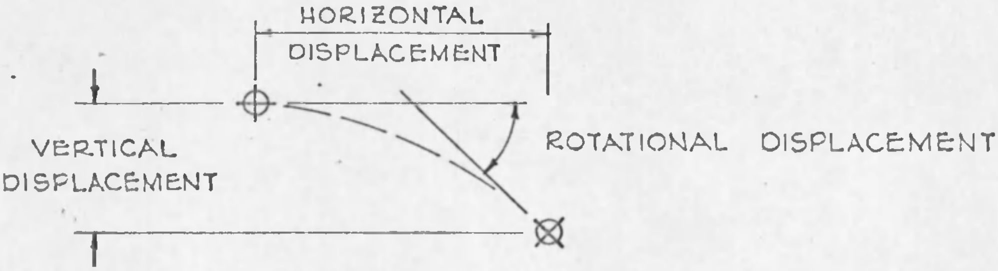


FIG. 2 NODAL DISPLACEMENTS

member forces by a member stiffness matrix such that

$$F = KV \tag{3-2}$$

where:

K is the element stiffness matrix

F is the member forces vector

Equilibrium must also be satisfied when the free body of a typical node is considered. External forces applied to the node must balance the member forces at the node. The forces may be related through a geometry matrix C such that

$$P = CF \tag{3-3}$$

where:

P is the externally applied force vector

C is the equilibrium matrix

The above equations may be combined by back substituting, yielding an equation relating the nodal displacements to the nodal forces as follows:

$$P = [CKB]\ D$$

it can be proven, using the principle of virtual displacement that $C = B^T$ (the transpose of the B matrix), thus

$$P = [B^TKB]\ D$$

or, letting $B^TKB = SK$, the above equation may be written in the form

$$P = SK \cdot D \qquad (3\text{-}4)$$

thus, the nodal forces are related to the nodal displacements by the system matrix SK.

The SK matrix is obtained through a chain of matrix transformations of the individual member stiffnesses. The stiffness component (B^TKD) of each member is found in turn and packed into the larger system stiffness matrix. After each member has been considered, the SK matrix is known; thus, if the nodal loads (P) vector is known, the nodal displacements can be found.

The external P loads are found by analyzing the loading; dead, live, wind, and seismic, that occur on the structure. Loads are either placed directly on the nodes in the directions of the nodal displacements, or, in the case of loads acting on a member itself, are reduced to nodal loads by superposition. In the latter case, the nodes are first considered fixed (held against rotation and translation). The fixed end forces are then calculated and superimposed on the nodes. The nodes are released and the structure is allowed to deform to a position of equilibrium.

The deflections can now easily be found once the load vector P and the stiffness matrix SK are known,

$$D = SK^{-1}P \qquad (3\text{-}5)$$

There are various method (algorithms) for solving the above equation. Gauss Elimination and back substitution have been used here.

The member forces can be found once D is known by combining equations (3-1) and (3-2) so that

$$F = KBD \tag{3-6}$$

The fixed end forces superimposed on the nodes must finally be subtracted from the forces found above and thus, analysis of the structure is completed.

The Computer Program

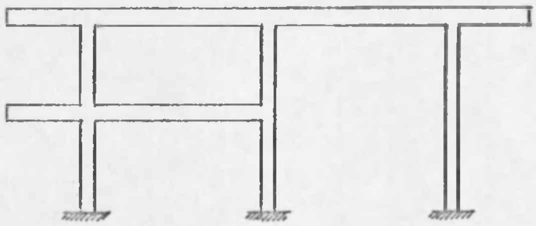
The computer program consists of subroutines, each serving a purpose of its own. In some cases a group of subroutines will form a major element of the program such as assembling the SK matrix (See the flow chart in Appendix A). Each segment of the program is presented here with explanation of the function and methods used. Special information, such as input formats for actual use of the program, are referenced to the Appendix, which may serve as an operators instruction manual.

Input Data

The first step in any program is to read in and store all known data pertinent to the problem. The structural analysis problem requires information as to the geometry of the structure, the stiffness of the structure, and the loading on the structure. The information is read in as follows.

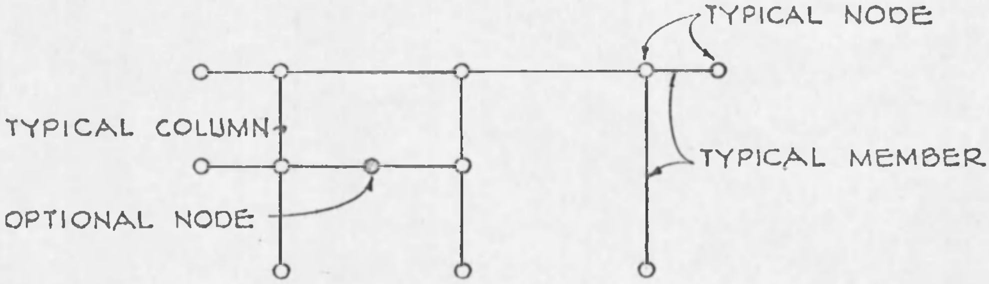
Geometry. The geometry of the structure must be considered first in an idealized form. The number of node points is read in. There is some leeway here for the engineer. Although all the joints connecting members or any free ends of members must be considered as a node, a point along a member could be of interest (the deflections might be useful) and that point can be considered as a node (See Fig. 3). Each node point is numbered. This number along with the x and y coordinates of the node point are read into the computer. The coordinates are measured from an origin located at the lower left-hand corner of the structure. Care should be taken in numbering the nodes to keep node separation to a minimum. Node separation is the greatest difference between the maximum and the minimum node point numbers of nodes connected by any member (See Fig. 4). A usual procedure is to number the nodes from left to right, a floor at a time, from bottom to top. A small node separation will mean a comparatively smaller stiffness matrix, thus conserving storage space in the computer.

Members. The number of members must be read in as an index to member operations. A member is defined by node points and always terminates at a node, thus a beam with a node in the middle is considered as two members (See Fig. 5). The number of members includes all the members, columns and beams, in the structure. A provision indexing the number of columns has been added. These members, being numbered and



ACTUAL SYSTEM

(a)



IDEALIZED STRUCTURE

(b)

FIG. 3 STRUCTURAL IDEALIZATION

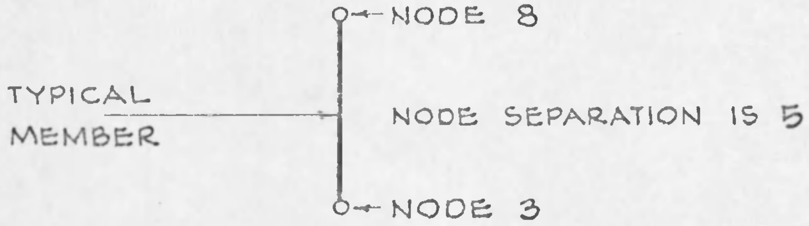
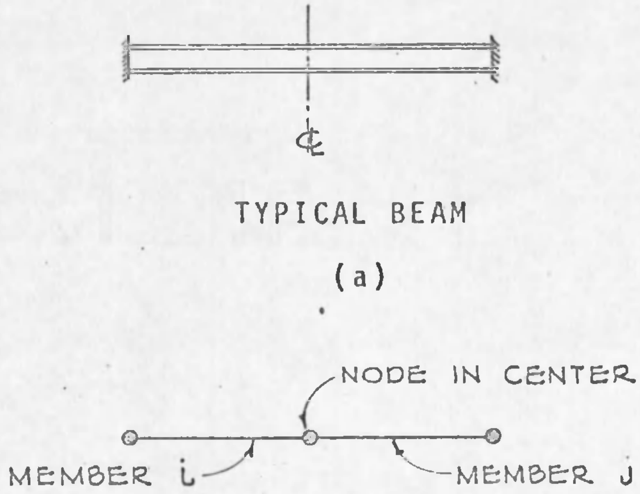


FIG. 4 NODE SEPARATION



TYPICAL BEAM

(a)

IDEALIZED BEAM

(b)

FIG. 5 MEMBER DEFINITION

placed in the beginning of the member information data deck, are not considered to have any associated gravity live loads and thus are omitted when the fixed end live loads are calculated.

A separate data card is made for each member. The data consists of the member number, the nodes associated with the member, the section number which refers to a library of standard shapes, and various dead loading and support information regarding that member (See Appendix for the specifics of the input data). Fixed end loads can be calculated for a variety of loading conditions (See Appendix for load types). For a load type not covered, the fixed end loads may be calculated independently by the engineer and then read into the program under the additional loading types discussed later in this section.

Library. Provisions for a library of data cards for standard section properties have been included in the program. At present only the data cards for the steel shapes listed in the A. I. S. C. Handbook and the U. S. Steel Catalogue have been made up and the example test problems use steel members exclusively. Each section has an assigned library number and the "properties for designing" are listed for that section. A library of concrete shapes could be developed considering either relative stiffness of members or actual properties of the concrete member.

Supports. The member must be supported and hence support points must be read into the program. Estimated deflections of the support may also be considered. Each node is capable of three deflections: horizontal, vertical, and rotational. The nodal deflections are numbered consecutively following the node point numbering. A rotational number is always the node number multiplied by three. The vertical deflection number is always one less than the rotational deflection number for a node and the horizontal deflection number is two less (See Fig. 6). When a support is added to the structure, one or more deflections at a node is restrained. The number of the restrained deflection is read into the program along with any previously calculated or estimated deflection.

FIG. 6 DISPLACEMENT NUMBERING

Other Loads. Loading conditions other than those dead loads on the member have been provided for. A dead loading not covered under the load types provided, or two more independent sets of loads such as wind forces may be applied directly to the nodes. These loads are applied in the same directions as the deflections of the node and are numbered in the same order as the deflections as was explained under the support conditions.

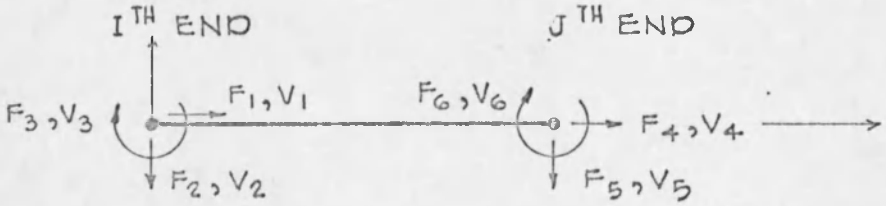
Live loading on the members is considered separately from other loading. Many members in a structure will have the same uniform live load as provided for by the codes. The number of different live loads is read in, then each live load along with all members subject to that live load are read into the computer.

Seismic Constants. Seismic constants as provided for by the code have been considered by allowing input for: the seismic zone, the stiffness due to the building system, and the base shear constant (Uniform Building Code, section 2312).

Formation of the Stiffness Matrix

Once the input data is stored, the stiffness matrix is calculated. As previously stated, the stiffness of the structure is a composite of the stiffness of all the members. Each member is considered in turn. Subroutine SYSTIF (See flow diagram in Appendix) zeros the stiffness matrix, sets the index for a particular member, and then calls subroutine ELSTIF which generates the member stiffness.

Influence coefficients for actions at the ends of a restrained plane frame member due to unit displacements of the ends of the member form the K matrix. The Ith end displacements are shown with the actions due to each of those displacements (See Fig. 7). The Jth end displacements are similar. From this, the element stiffness matrix K can be

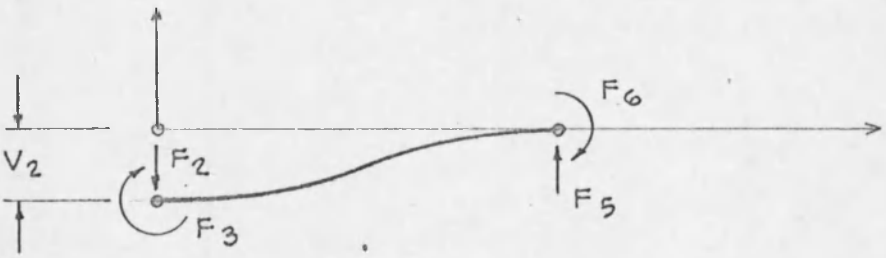


TYPICAL MEMBER WITH DISPLACEMENTS AND END FORCES

(a)

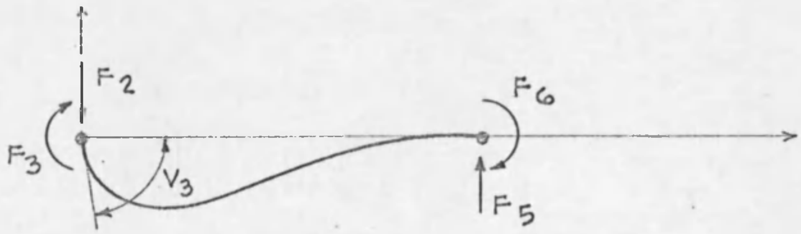
AXIAL DISPLACEMENTS

(b)



LATERAL DISPLACEMENT

(c)



ROTATIONAL DISPLACEMENT

(d)

FIG. 7 MEMBER STIFFNESSES

formed from equation 3-2:

$$
\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} =
\left[\begin{array}{ccc|ccc}
\frac{AE}{L} & o & o & -\frac{AE}{L} & o & o \\
o & \frac{12EI}{L^3} & \frac{6EI}{L^2} & o & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\
o & \frac{6EI}{L^2} & \frac{4EI}{L} & o & -\frac{6EI}{L^2} & \frac{2EI}{L} \\
\hline
-\frac{AE}{L} & o & o & \frac{AE}{L} & o & o \\
o & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & o & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\
o & \frac{6EI}{L^2} & \frac{2EI}{L} & o & -\frac{6EI}{L^2} & \frac{4EI}{L}
\end{array}\right]
\begin{Bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \end{Bmatrix} \tag{3-7}
$$

where:

A = Area of the cross section

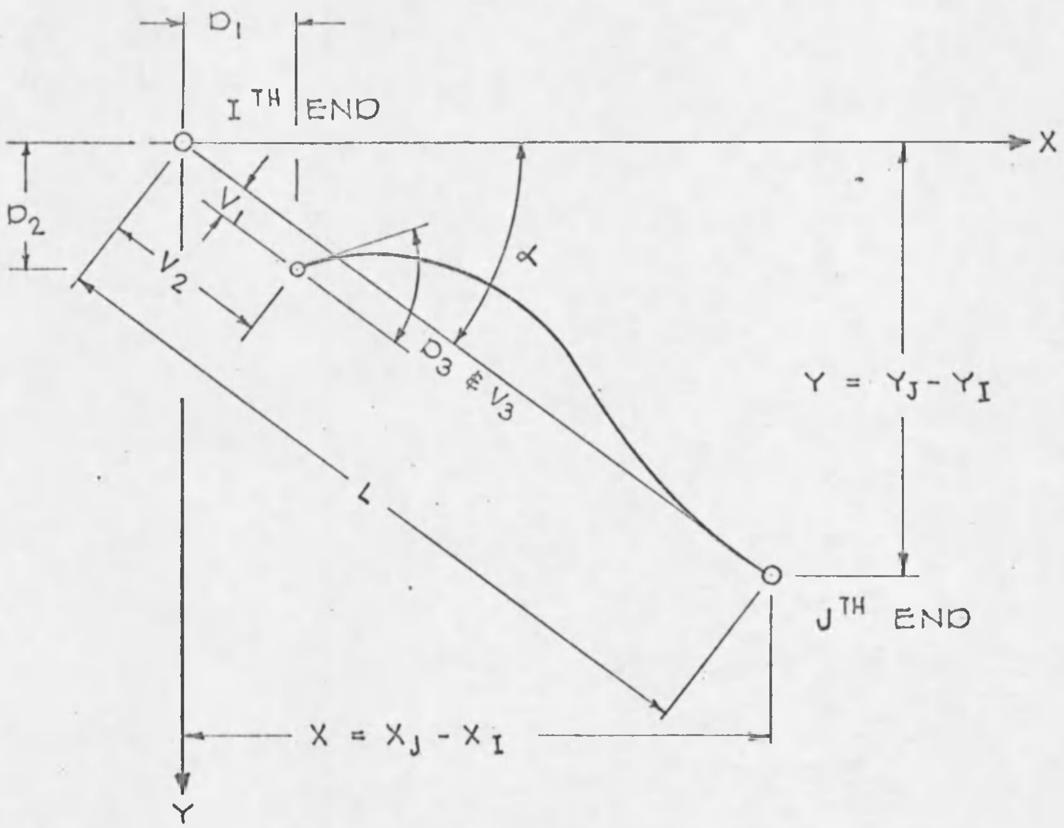
E = Modulus of elasticity

I = Moment of inertia

The compatibility matrix relates the member end displacements to the nodal system displacements as was previously stated (equation 3-1) where

$$V = BD$$

thus, considering the diagram (See Fig. 8), the following may be written



D = SYSTEM DISPLACEMENTS

V = ELEMENT DISPLACEMENTS

L = ELEMENT LENGTH

FIG. 8 COMPATIBILITY RELATIONSHIPS

$$\begin{Bmatrix} V_1 \\ V_2 \\ V_3 \end{Bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha & o \\ -\sin\alpha & \cos\alpha & o \\ o & o & 1 \end{bmatrix} \begin{Bmatrix} D_1 \\ D_2 \\ D_3 \end{Bmatrix} \tag{3-8}$$

Letting $\cos\alpha = x/L$ and

$\sin\alpha = y/L$

and considering the displacements at the Jth end which are similar to those at the Ith end, the total compatibility matrix may be written

$$\begin{Bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \\ V_5 \\ V_6 \end{Bmatrix} = \begin{bmatrix} \frac{x}{L} & \frac{y}{L} & o & o & o & o \\ -\frac{y}{L} & \frac{x}{L} & o & o & o & o \\ o & o & L & o & o & o \\ o & o & o & \frac{x}{L} & \frac{y}{L} & o \\ o & o & o & -\frac{y}{L} & \frac{x}{L} & o \\ o & o & o & o & o & L \end{bmatrix} \begin{Bmatrix} D_1 \\ D_2 \\ D_3 \\ D_4 \\ D_5 \\ D_6 \end{Bmatrix} \tag{3-9}$$

Remembering that the equilibrium matrix is the transpose of the compatibility matrix, the subroutine MXMULT may now be called to obtain B^TKB thus forming the stiffness component of the system due to the member being considered. The coefficients of the six by six matrix formed are placed in their proper position in the system stiffness matrix SK. Proper placement depends on which nodes (hence which displacements) are associated with the member (See Fig. 9). Thus the placement of the member in the structure corresponds to the placement of its stiffness contribution in the total stiffness of the system.

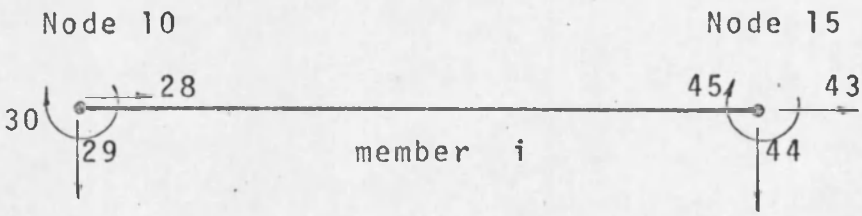
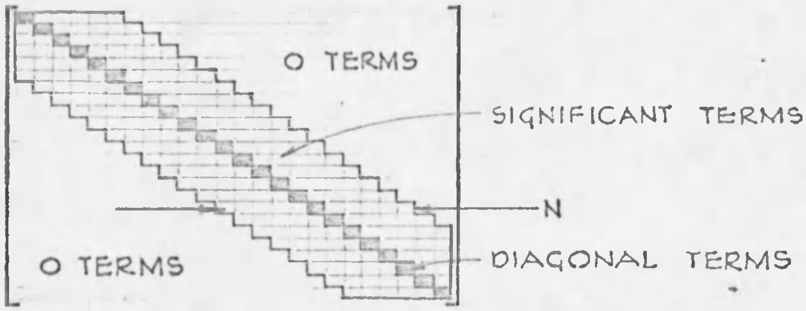


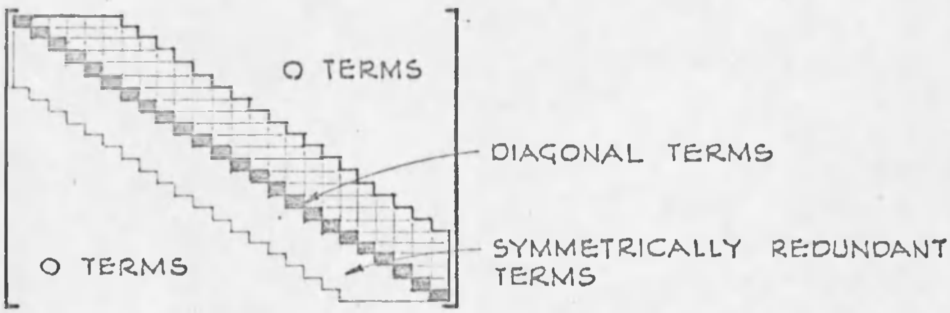
FIG. 9 MEMBER PLACEMENT

Banding. The form of the SK matrix will often be strongly diagonal as can be seen in the Figure 10. The dimension N is dependent on the node separation. The maximum difference between I and J for the member in the structure will have that maximum difference. The matrix will always be symmetrical about the diagonal axis shown for linear systems. Thus to save storage, the full matrix is banded, ignoring the zero terms and the identical terms on the lower side of the diagonal, into the form shown in Figure 10c. The



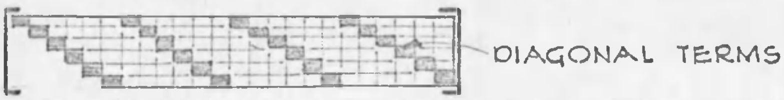
FULL SK MATRIX

(a)



HALF SK MATRIX

(b)



BANDED SK MATRIX

(c)

FIG. 10 MATRIX BANDING

saving of computer space that is realized allows for a much larger structure simulation to be placed in the computer. Banding vectors are generated in a separate subroutine band and the stiffness coefficients are stored directly into the banded configuration.

The stiffness of the structure has now been generated. A final operation is carried out decomposing the SK matrix prior to actually solving the equations for the deflections. This operation will be explained more fully with the equation solving method. The loading vectors must now be generated before continuing. Two subroutines PLOADS and GENLD accomplish this.

Formation of the Load Vectors

The building codes require that a structure be designed to withstand not only the stresses due to the dead load of the building itself, but the stresses due to lateral loads such as wind or earthquake and the maximum stresses due to the critical live loading pattern causing that stress. As the stress allowable varies with both the building code and the loading case, each loading case is considered separately. Six loading vectors are generated to cover the static loadings and a separate method is employed to cover the live gravity loads. A number of different member force combinations are available and those applicable to each loading case are considered.

Dead Loads. The dead load assumed in the design shall consist of the weight of the structure proper and all material permanently fastened thereto or supported thereby (A. I. S. C. specifications). The P_1 vector is reserved for the dead loads and the member forces due to the dead load are printed out under the heading, load condition one. The dead loading on each member must be determined by the engineer and then may either be read in on the member itself (See load types in Appendix A) or the fixed end forces may be calculated by the engineer and read in on the nodes directly. It must be remembered, however, if the latter case is used, the fixed end forces must be subtracted from the computed member forces by the engineer. The load read in directly on the nodes are placed in the correct order in the P_1 vector at the beginning of subroutine PLOADS. Each member is then considered in turn. The index is set for a particular member and subroutine GENLD is called. GENLD considers the orientation of the member, calculates the length of the member, considers the loads on the member including the weight of the member itself, and then calculates the fixed end forces for the member. Those fixed end member forces are saved and the fixed end forces superimposed on the nodes are added to any forces already on the nodes. When each member in the structure has been considered, the total P_1 vector will have been populated.

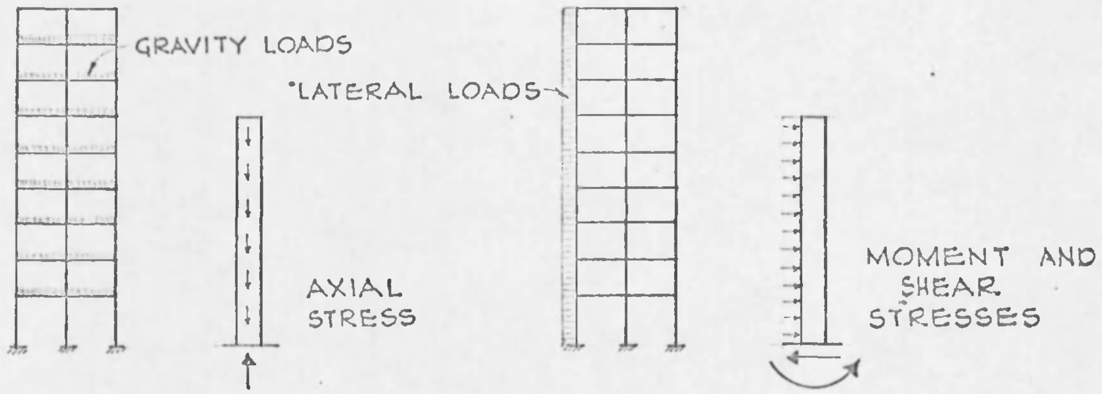
Lateral Loads. Analysis for lateral loads by conventional methods differs from analysis for vertical loads.

If the total structure were considered as a free standing column, the gravity loads would cause only an axial stress in that column, (See Fig. 11a). The structure, however, would act as a cantilever when horizontal loads are applied. Hence a moment and a shear would be created at any horizontal cross section of the building (See Fig. 11b).

Lateral loads such as those due to wind or earthquake are of importance particularly in tall buildings. Here the stresses due to horizontal loads may be critical, especially in the lower members. As wind and earthquake need not be assumed to act simultaneously, separate vectors have been assigned for each. Other considerations in the code such as load reductions make separate consideration of each horizontal load advantageous. Both horizontal loading conditions are analyzed statically as provided for by the present codes.

Wind Loads. Two vectors, P_2 and P_3 have been set aside for wind loads. If the building is symmetric and the wind loading from either side is the same, one vector only need be used. The second vector is provided so winds from either direction may be considered. The wind forces are calculated by the engineer and read in directly on the nodes. The deflections and the member forces due to both sets of wind loads (See Fig. 12) are printed out under the headings load cases two and three, respectively.

Seismic Loads. Every building shall be designed to



AXIAL STRESS

(a)

FLEXURAL STRESS

(b)

FIG. 11 LATERAL LOAD STRESS

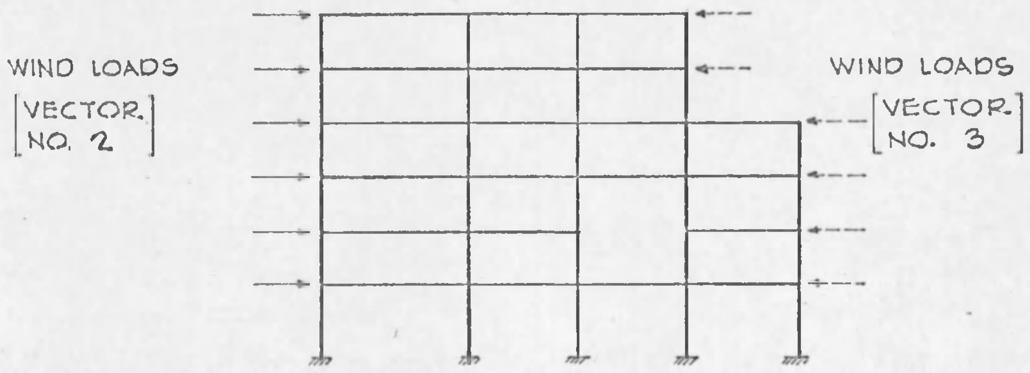


FIG. 12 WIND LOADS

resist stresses produced by lateral forces due to earthquake. Stresses shall be calculated as the effect of a force applied horizontally at each floor or roof level above the foundation. The force shall be assumed to come from any horizontal direction (Uniform Building Code). Vectors P_4 and P_5 have been set aside for the seismic loads coming from either side of the plane frame. The magnitudes of the horizontal loads to be applied at each floor are calculated in accordance with Section 2313 of the Uniform Building Code $F_x = \frac{(ZKCW)\ W_x h_x}{Wh}$. The total dead load per floor is found from the P_1 vector previously calculated. The dead load on a particular floor W_x is multiplied by: the height of that floor h_x, the seismic constants that were read in, and the total sum of the dead loads W. That product is divided by the sum of the floor weights multiplied by their respective heights wh to obtain the force F_x to be applied at the floor under consideration. The force is applied uniformly to the nodes at that floor level. To properly consider the total loads and the horizontal rotational effect of the seismic load on the structure, all of the parallel plane frames of the structure must be considered at the same time. An orthogonal multistory building may be considered as two sets of parallel frames interacting at right angles. Each set of parallel frames being considered as a series of plane frames in a single plane. The individual frames are connected by rigid pinned bars representing the floor system

(See Fig. 13). Thus lateral forces on one frame may be transferred to the next parallel frame through the floor system in proportion to the frame's relative stiffnesses.

A set of operations in subroutine PLOADS locates the nodes associated with each floor and sums the vertical dead loads on those nodes. The manipulations required to calculate F_x are carried out and two load vectors P_4 and P_5 are formed. P_4 consists of the seismic loads to the right and member forces are printed out for the seismic loads.

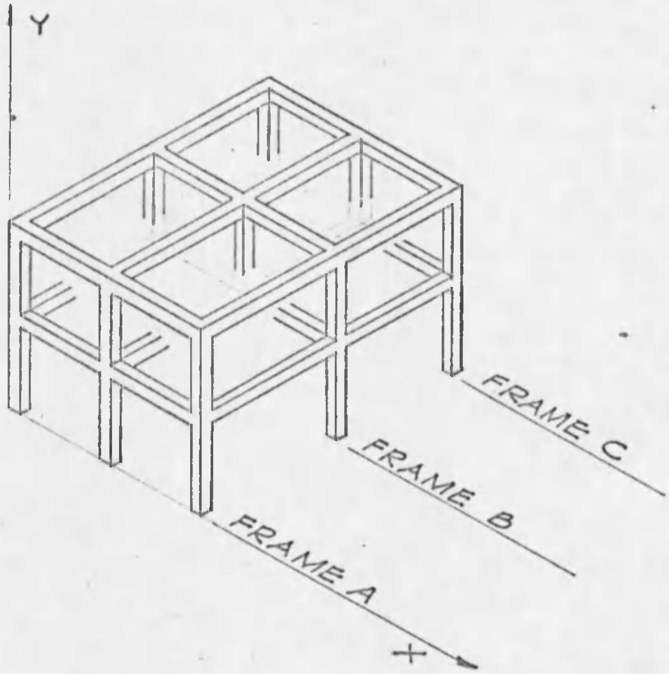
Gravity Loads. A sixth vector is provided which is the sum of the dead loads P_1 and all of the live loads. This would represent all of the gravity forces dead and live on the structure. Although the live load vector is formed in exactly the same manner as the dead load vector; the solution of the maximum or minimum member force due to different live loading patterns is quite different from the solution for member forces due to dead loads. The live loading condition will therefore be explained after the discussion on solving member forces. With the sixth load vector populated, the deflections of the nodes may now be solved.

Solution of the Equations

There are various methods for solving the set of simultaneous equations (equation 3-4)

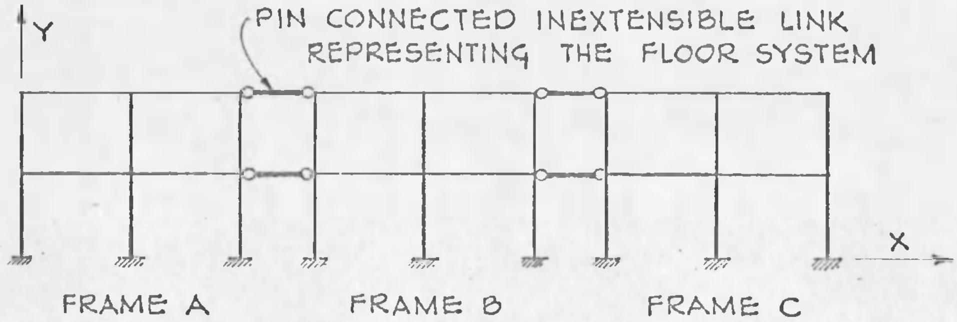
$$P = SK \cdot D$$

for the unknown deflections. One method would be to solve



STRUCTURE

(a)



IDEALIZED STRUCTURE

(b)

FIG. 13 MULTIPLE BAY IDEALIZATION

the equation in the form (equation 3-5)

$$D = SK^{-1}P$$

by inverting the SK matrix. This however takes time and storage space, both of which are at a premium. Another method which is used is Gauss Elimination and back substitution (Wylie, 1966). Given a system of simultaneous linear equations

$$\begin{array}{ccccccc} a_{11}x_1 & + & a_{12}x_2 & + \cdots + & a_{1n}x_n & = & b_1 \\ a_{21}x_1 & + & a_{22}x_2 & + \cdots + & a_{2n}x_n & = & b_2 \\ \vdots & & \vdots & & \vdots & & \vdots \\ a_{m1}x_1 & + & a_{m2}x_2 & + \cdots + & a_{mn}x_n & = & b_m \end{array} ,$$

a procedure consisting of a series of reductions will be applied. For the first reduction of the equations, the first equation is divided by A_{11} and then in turn, multiplied by the first term of each other equation and subtracted from that equation until the Mth equation has been considered. For the second reduction, the first equation is left alone and the second equation is divided by A_{22} and then in turn multiplied by the first term of each new equation and subtracted from that equation. The result of the second reduction is the system

$$
\begin{aligned}
x_1 + A_{12}x_2 + A_{13}x_3 + \ldots + A_{1n}x_n &= B_1 \\
x_2 + A_{23}x_3 + \ldots + A_{2n}x_n &= B_2 \\
a''_{33}x_3 + \ldots + a''_{3n}x_n &= b''_3 \\
\vdots \qquad\qquad & \quad \vdots \\
a''_{m3}x_3 + \ldots + a''_{mn}x_n &= b''_m \ .
\end{aligned}
$$

The series of reductions terminates when the reduction on the M-1 equation has been completed, yielding the following set of equations:

$$
\begin{aligned}
x_1 + A_{12}x_2 + A_{13}x_3 + \ldots + A_{1n}x_n &= B_1 \\
x_2 + A_{23}x_3 + \ldots + A_{2n}x_n &= B_2 \\
x_3 + \ldots + A_{3n}x_n &= B_3 \\
& \ \vdots \\
a'''_{mn}x_n &= b'''_m \ .
\end{aligned}
$$

x_n can be solved by dividing the last equation by a''_{mn}. The rest of the unknowns can now be found by a series of back substitutions. It must be noted that there can be no zero terms on the diagonal (i.e.: $a_{11} \neq o, \ldots . a_{mm} \neq o$), for an equation cannot be divided through by zero. Due to the method of forming the stiffness matrix there can be no zero on its diagonal. A zero on the diagonal could come about only by an inconsistancy in the structure or an error in the input data. If this happens, the program will print out which term is equal to zero and it is up to the engineer to find the error.

It is noticed that the stiffness matrix is destroyed in the process of solving for the deflections. To save having to regenerate the stiffness matrix each time a set of deflections must be solved, the stiffness matrix can be partially decomposed. The diagonal terms are restored to their original value and the deflections may be solved by working on the P vector separately without further destroying the stiffness matrix. The stiffness matrix is decomposed at the end of subroutine SYSTIF. Work on the P vector and the backsubstitution are done in subroutine EQSOLV.

Just prior to solving the equations, the boundary conditions (supported points of known deflection) must be applied. The matrix equations may be written in the form

$$\begin{bmatrix} P_{free} \\ P_{supported} \end{bmatrix} = \begin{bmatrix} SK_{ff} & SK_{fs} \\ SK_{sf} & SK_{ss} \end{bmatrix} \begin{bmatrix} D_{free} \\ D_{supported} \end{bmatrix} \quad (3\text{-}10)$$

where:

P_{free} = loads at the unsupported nodes

$P_{supported}$ = loads at the supported nodes

The equations may be rewritten as follows:

$$D_{free} = SK_{ff}^{-1}P_{free} - SK_{ff}^{-1}SK_{fs}D_s$$

and the last term will drop out if the support deflection vector D_s is null. Rather than rearrange the equations in the program, the support equations are zeroed out. No diagonal term of the stiffness matrix can equal zero therefore the diagonal term is put at one ($a_{jj} = 1$).

$$\text{(support equation)} \quad ox_1 + ox_2 + \ldots + 1x_j + \ldots ox_n = D^1{}_j$$

$D^1{}_j$ is a known deflection replacing the P, value. The above is achieved by multiplying the diagonal term a_{jj} and the D^1 term by the large number 10^{16}. When the equation is divided through by the diagonal term for the Gauss Reduction, the above equation essentially is left. The support deflection x_j therefore will equal the read in deflection $D^1{}_j$ and the deflections of the free nodes are affected only in as much as the support deflects.

<u>The Reactions</u>. The reaction vector $P_{supported}$ is found by solving the complete equation 3-4

$$P = SK\ D$$

where all of the nodal forces are found. The stiffness matrix must be regenerated as it was destroyed in the equation solve process. Any forces applied directly to the supported node must be added to the support reaction for it will not be included in the solution for the deflections. The reactions for each load type except the live loading conditions are printed out along with an equilibrium check.

The Member Forces. Once the deflections have been solved, the member forces can be found from equation 3-6

$$F = KBD$$

K and B are regenerated for each member in turn and multiplied by those node deflections associated with that member. Any fixed end forces on the member must be subtracted from the F forces. The member forces are then printed out and labeled as to which loading condition caused them.

Live Load Considerations

The member forces due to the live loading will now be discussed. The codes state that the arrangement of loads resulting in the highest stresses in the supporting member shall be used in design. The highest axial force possible along with the highest moment possible and its associated axial force (See Fig. 14).

The program analyzes the maximum and minimum force conditions due to any worst possible loading condition. Each live loaded member is considered in turn. A load vector of fixed end forces for the one loaded member is generated in the same manner as the fixed end forces due to the dead loads. The deflections and hence the member forces in all the members in the structure are calculated due to the load on that one member (See Fig. 15a). The member forces are stored in one of two columns depending on the sign of the force. The next member is now considered and the process repeated (See Fig.

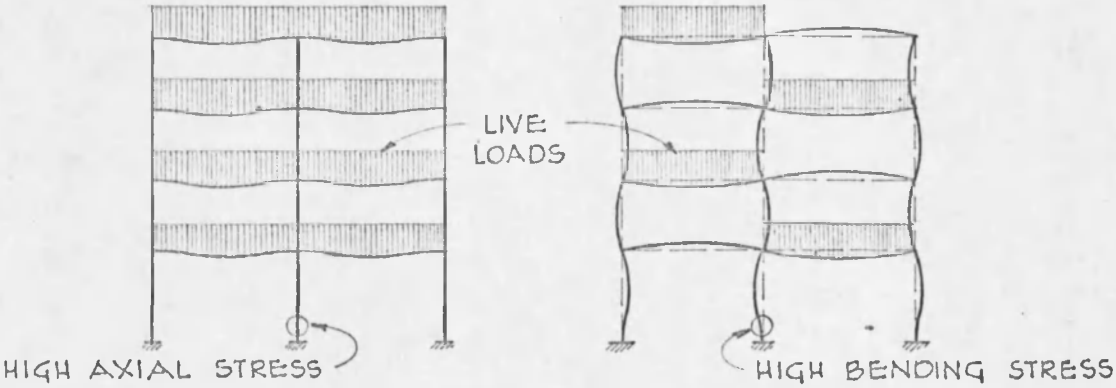
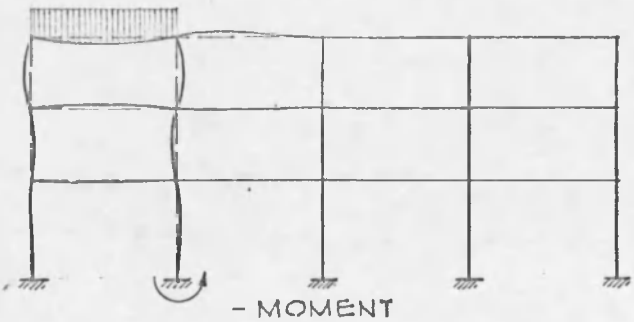
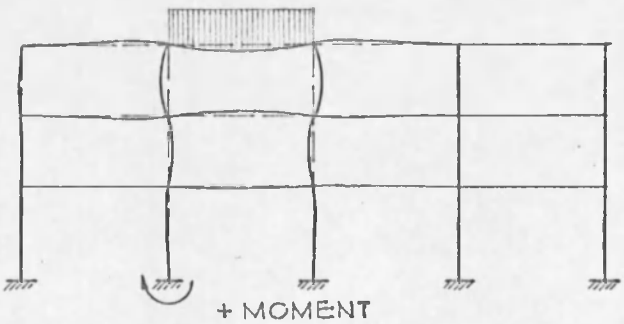


FIG. 14 MAXIMUM LIVE LOAD STRESSES



FIRST CONSIDERATION

(a)



SECOND CONSIDERATION

(b)

FIG. 15 LIVE LOAD STRESS

15b). For each cycle the load vector is generated, the nodal deflections are solved, and the member forces are calculated using the process described before. The member forces are then summed for each load that causes a similar force. Those forces will be maximum after all of the live loaded members have been considered. Other non-maximum forces associated with the maximum forces are also calculated and hence the requirement above is satisfied.

Output Data

The final step in the program is to print out all the required data that has been found. The node deflections are printed out in inches and are labeled as to the node. Reactions are printed out for each loading condition. The total weight of the structural members is printed out along with some other miscellaneous by-products of the seismic considerations (See Appendix for exact data output format). All of the member forces are printed out in kips or foot-kips. A member is listed and then the forces according to the loading condition are printed. Maximum and minimum moments with their associated shears and axial forces for each end of a member are printed out under load cases seven and eight. Maximum and minimum axial forces and shears are similarly printed out under load cases nine through twelve. The answers are left as member forces rather than stresses for convenience. The various forces must be studied and the stress calculated for the worst combination of forces as required by the code.

This process has not been included in the program because of the variation in different codes and changes taking place as newer additions are adopted.

Information Evaluation

The engineer must now evaluate the information supplied by the computer output and check the stresses of each member against the stresses allowed by the code. If a member is undersized a larger section must be used. Oversized members may also be altered. The program can be rerun to check the adequacy of the resized structure. Design of connections and footings may now be carried out in the conventional manner. The program has relieved the engineer of the tedium of the actual stress analysis but has not replaced his judgment.

CHAPTER IV

EXAMPLES

Presented herein is a list of structural problems analyzed by the computer program. Typical structures, with loads, are shown in the figures. Node deflections or member end forces are given in the tables.

The first problem is a beam fixed at both ends with the loading conditions shown (See Fig. 16). The beam was used as the original test problem during the development of the program. Exact solutions in the form of moment and shear diagrams were calculated by hand and compared with the computer output. A few selected comparisons are given in tables (1 and 2).

The remaining problems are examples taken from various textbooks. These problems demonstrate the reliability of the program compared to the other methods of analysis noted in the tables. Member end moments due to lateral loads, gravity loads, and live loading patterns are tabulated and compared. Two factors must be remembered however: the program takes into account the effects of axial deformation and the other methods of analysis used are approximations at best. As an added interest, the computer time used for the complete analysis is given with each problem.

EXAMPLE PROBLEM NO. 1
BEAM FIXED AT BOTH ENDS

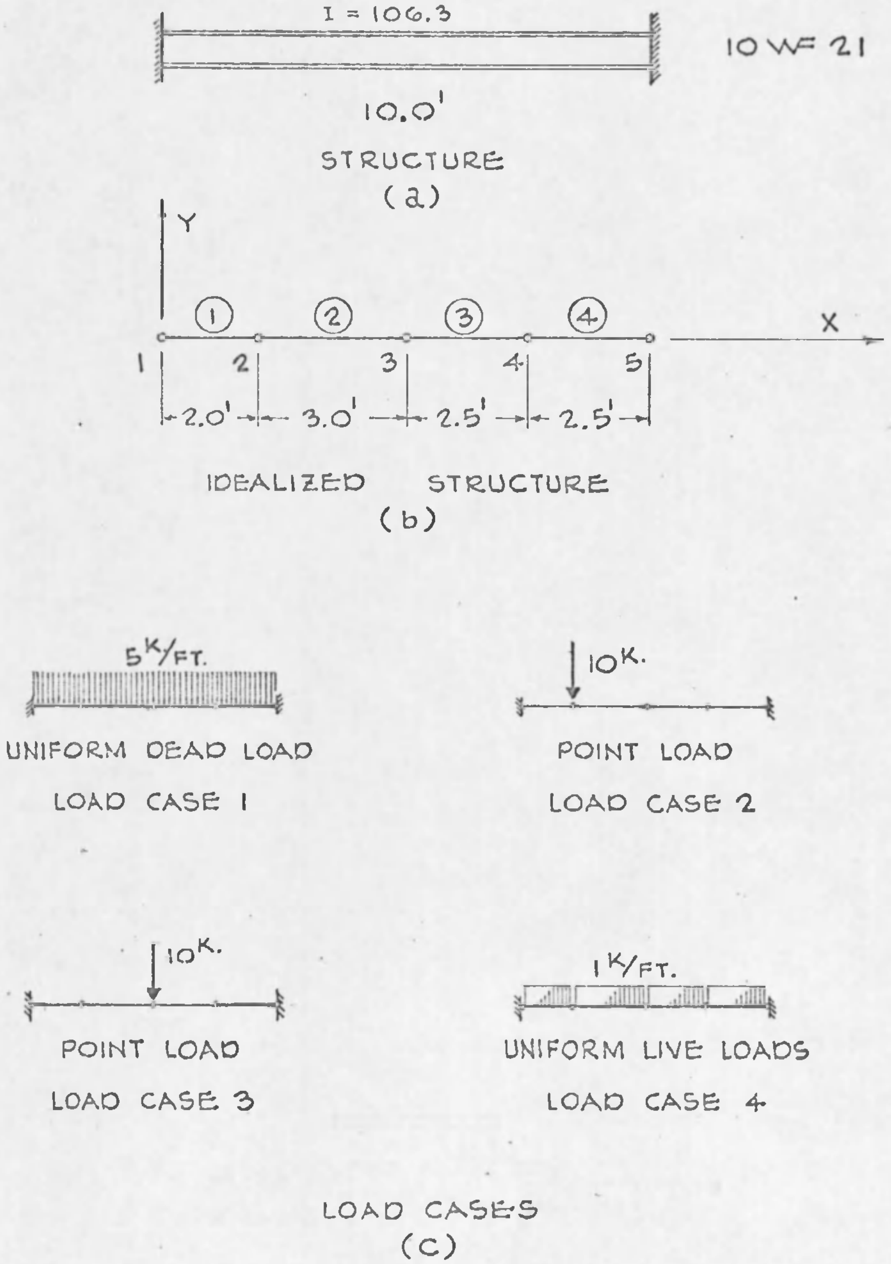


FIG. 16 EXAMPLE 1

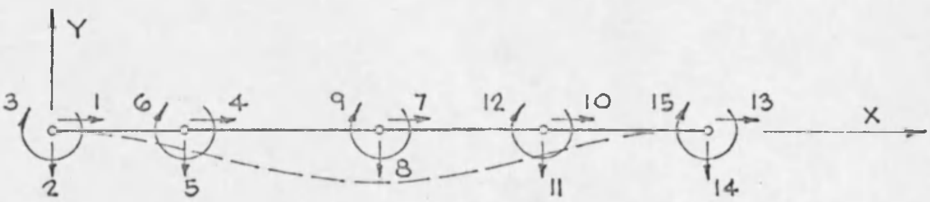


FIG. 17 DEFLECTIONS

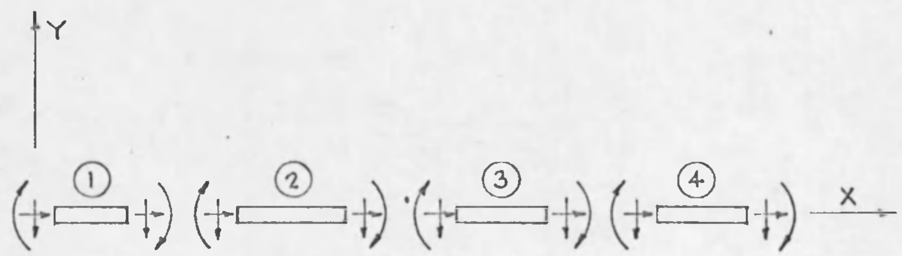


FIG. 18 MEMBER FORCES

SELECTED DEFLECTIONS			
Loading	Def.	Exact Solution	Computer Solution
1	5	.03002	.030021
	8	.07329	.073294
	11	.04122	.041228
2	5	.00765	.007653
	8	.01027	.010277
3	8	.029195	.029195

TABLE 1 DEFLECTIONS Example 1

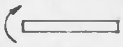
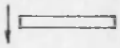
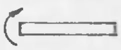
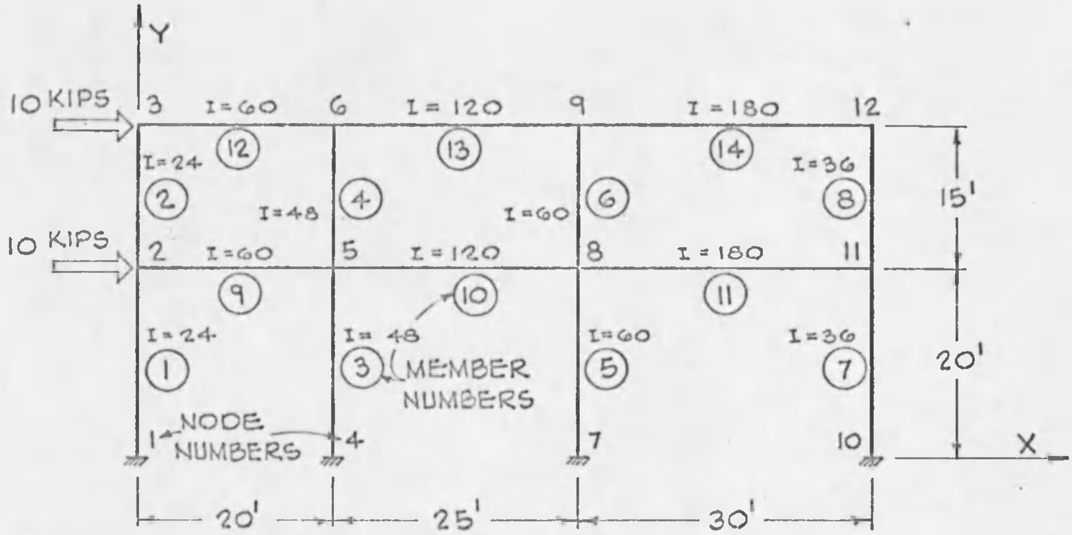
SELECTED MEMBER FORCES			
Load Case	Member-End Force	Exact Solution	Computer Solution
1	 1	-41.84	-41.842
	 2	-20.92	-20.921
	 1	-25.105	-25.105
2	 2	- 2.00	- 2.00
3	 1	+12.50	+12.50
4	max  1	- 8.333	- 8.333
	min  1	+ .3958	+ .39583
	max  1	+ .7291	+ .72916
	min 	+ .072	+ .072
	max 	3.072	3.072

TABLE 2 MEMBER FORCES Example 1

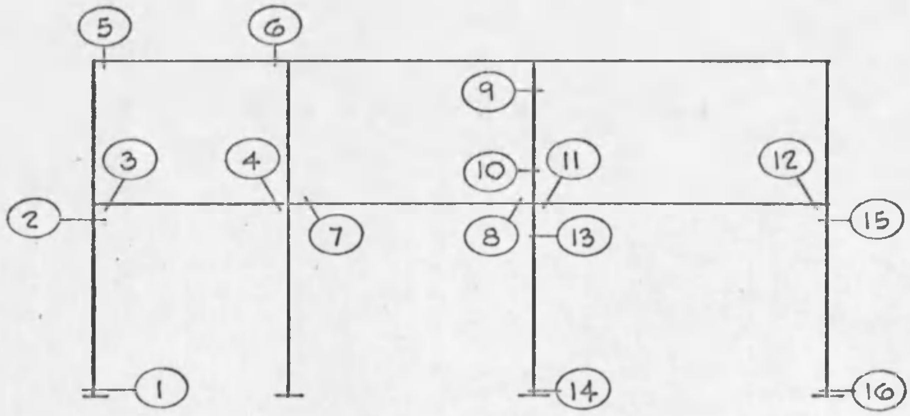
EXAMPLE PROBLEM NO. 2

SEE NORRIS AND WILBUR (1960) P.P. 304-311



STRUCTURE WITH LOADING

(a)



SEE TABLE THREE FOR THE MEMBER MOMENTS NOTED ABOVE

(b)

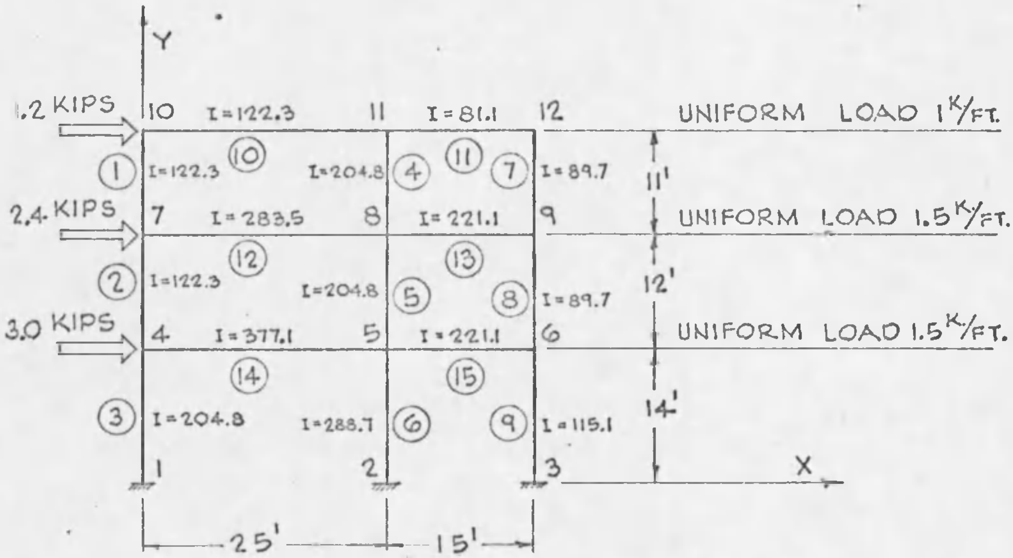
FIG. 19 Example 2

SELECTED MEMBER FORCES				
End Mom.	"Exact" Solution	Portal Method Solution	Cantilever Method Solution	Computer Solution
1	29.60	33.33	22.24	29.68
2	25.10	33.33	22.24	25.15
3	31.60	45.83	30.57	31.69
4	29.50	45.83	30.57	29.47
5	10.60	12.50	8.33	10.57
6	10.00	12.50	8.33	9.97
7	42.50	45.83	54.60	42.63
8	41.30	45.83	54.60	41.51
9	32.40	25.00	29.18	32.43
10	25.20	25.00	29.18	25.15
11	52.20	45.83	52.50	51.96
12	53.60	45.83	52.50	53.59
13	68.30	66.67	77.92	68.32
14	76.80	66.67	77.92	76.74
15	40.10	33.33	38.20	39.97
16	45.60	33.33	38.20	45.50
Computation Time For Complete Analysis				3.83 seconds

Table 3 END MOMENTS Example 2

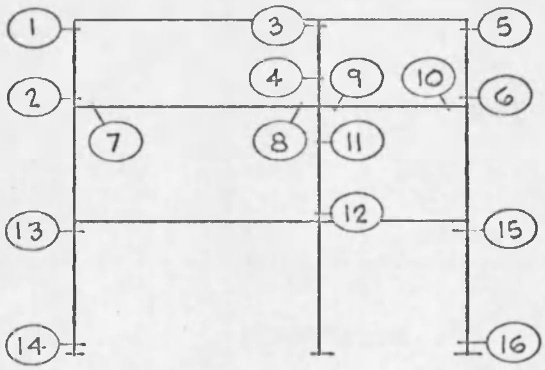
EXAMPLE PROBLEM NO. 3

SEE BULL AND SVED (1964) P.P. 70-86



STRUCTURE WITH LOADING

(a)



SEE TABLE FOUR FOR THE MEMBER MOMENTS NOTED ABOVE

(b)

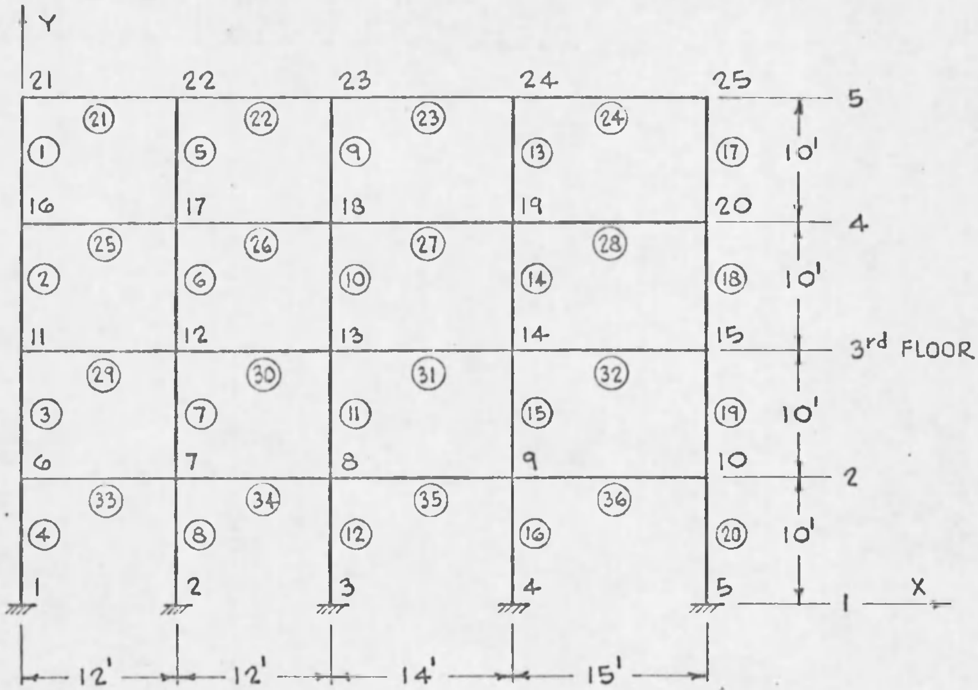
FIG. 20 Example 3

SELECTED MEMBER FORCES				
	Due To Gravity Loads		Due To Lateral Loads	
End Mom.	Moment Distribution Solution	Computer Solution	Moment Distribution Solution	Computer Solution
1	+40.2	+40.94	- 1.84	- 1.95
2	+33.9	+33.63	- 0.91	- 1.04
3	-27.7	-27.76	- 3.79	- 3.87
4	-23.7	-23.75	- 2.63	- 2.69
5	-11.1	-13.14	- 1.97	- 2.01
6	- 9.5	-10.90	- 1.63	- 1.63
7	-60.8	-62.49	+ 6.97	+ 7.11
8	+82.2	+79.71	+ 6.68	+ 6.86
9	-39.9	-36.88	+ 7.93	+ 7.85
10	+16.6	+19.23	+ 7.51	+ 7.49
11	-18.4	-19.08	-11.98	-12.01
12	-20.1	-20.32	- 9.79	- 9.88
13	+26.7	+27.24	-11.71	-11.89
14	+12.2	+12.87	-17.27	-17.47
15	- 6.1	- 7.04	- 8.15	- 8.06
16	- 3.7	- 4.01	-10.52	-10.44
Computation Time For Complete Analysis 4.06 seconds				

Table 4 END MOMENTS Example 3

EXAMPLE PROBLEM NO. 4

SEE MANTELL AND MARRON (1962) P.P. 302-310



GIVEN:

MULTISTORY FRAME

COLUMN K's = 1/2 BEAM K's

DEAD LOAD = 2^{K}/FT.

LIVE LOAD = 3^{K}/FT.

FIND:

MAXIMUM END MOMENTS IN THE BEAMS OF FLOOR 3

FIG. 21 Example 4

MAXIMUM END MOMENTS IN THE BEAMS OF FLOOR 3								
M	29		30		31		32	
1	45.0	72.0	72.0	76.7	76.7	105.1	105.1	70.3
2	31.0	73.5	66.1	74.0	80.6	102.8	109.5	49.5
3	33.0	71.5	68.0	73.6	81.2	99.9	108.8	52.1
4	51.57	69.07	72.64	79.23	90.88	98.60	104.98	77.29

Solutions in Kip-Ft

M - Method of Analysis

1 - Example 10-1 Moment Coefficients

2 - Example 10-2 Simplified Structure-Unit Loads

3 - Example 10-6 Two cycle method of moment distribution

4 - Computer Output

Computation Time For Complete Analysis 13.64 seconds

Table 5 END MOMENTS Example 4

CHAPTER V

POSSIBLE ADDITIONS

A program of this type evolves with use. Many changes and innovations could be made, some to fit individual preferences, some to fit a particular structure, some to fit a particular building code, and some to fit a particular computer. A few ideas for further investigation are listed here.

a. A subroutine to automatically generate the geometry of the structure would eliminate the reading in of each node point with its corresponding x and y coordinates. This would be particularly useful in large, uniform, grid structures. A simple "if" statement could given the operator a choice of input methods or possibly a combination of methods.

b. A stress routine could be added and the output would then be the maximum stress for each member rather than the maximum force in the member.

c. A routine could be added to find the forces at midspan or even the quarter points of each member, by using the known loading condition and the calculated end forces for that member. This can be achieved now by placing node points along the member where the forces are wanted, however the input data is increased and the increase in node points will decrease the size of the structure allowed.

d. A plotting subroutine could be added and the output would include a plot of the deflected structure along with the member forces listed on the member. An alternative would be to draw the moment diagram and/or shear diagram for each member.

e. A joint stiffness factor could be incorporated into the element stiffness matrix. This factor would consider the per cent rotational stiffness of a joint from zero per cent for a pinned connection to one hundred per cent for an ideally fixed connection. Each end of a member would have a factor that would be considered when calculating end actions due to end displacements. Fixed end forces superimposed to the nodes would also be modified.

f. Tape storage is a must if a structure of any size is to be analyzed. When using external storage, the structure is broken down into substructures (See Fig. 22). The stiffness for each substructure is calculated and then stored on tape. Each plane frame of the structure would be considered as a substructure. For analyzing gravity loads, each plane frame is considered separately, but when analyzing lateral loading, all of the frames must be considered together. Further breakdown into substructures would prove difficult due to the method of analyzing the live loading and to the computer time involved in switching from internal

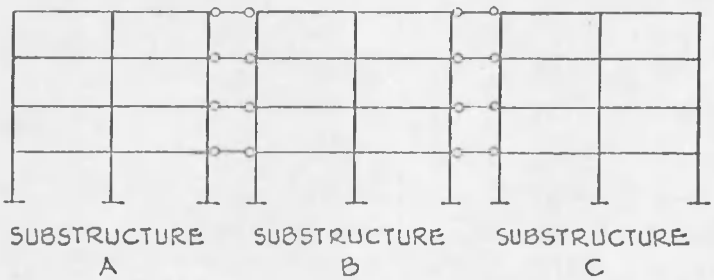


FIG. 22 SUBSTRUCTURES

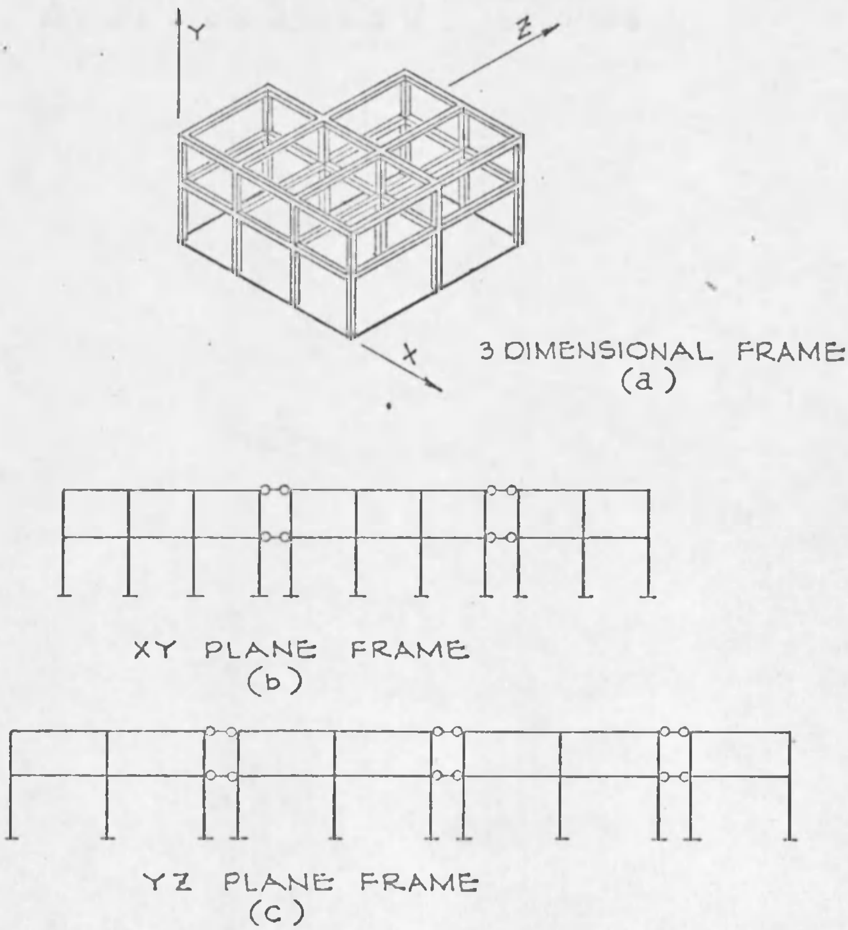


FIG. 23 THREE DIMENSIONAL CONSIDERATION

to external storage and back again.

g. The node points could be read in in their three dimensional locations. The computer would then be programed to select the frames parallel to the xy plane and arrange them in the correct order as a set of plane frames connected by inextensible links (See Fig. 23). The process would be repeated for the yz plane frames. Each orthogonal set of frames would be analyzed separately. Thus the whole building will have been analyzed.

Some of the above ideas must be added to bring the program from the role of an academic answer to the role of a practical answer useful to the architect's office and the affiliated structural engineer. Care, however, must be taken to keep the program simple and to keep computer time to a minimum. The value of any change must balance advantages against cost.

CHAPTER VI

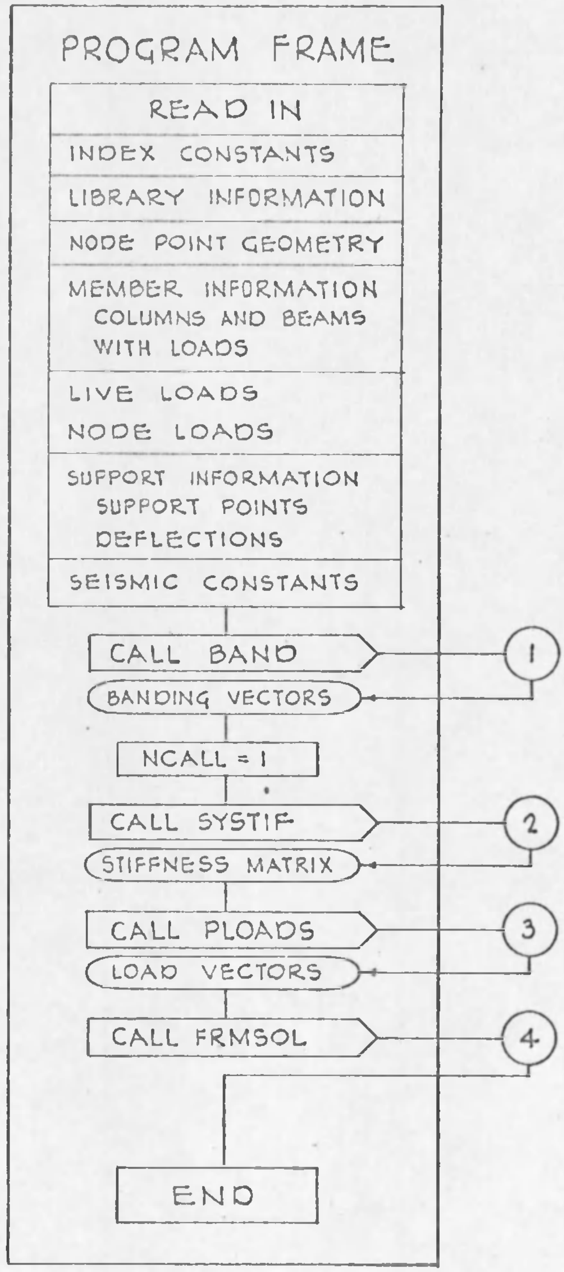
CONCLUSION

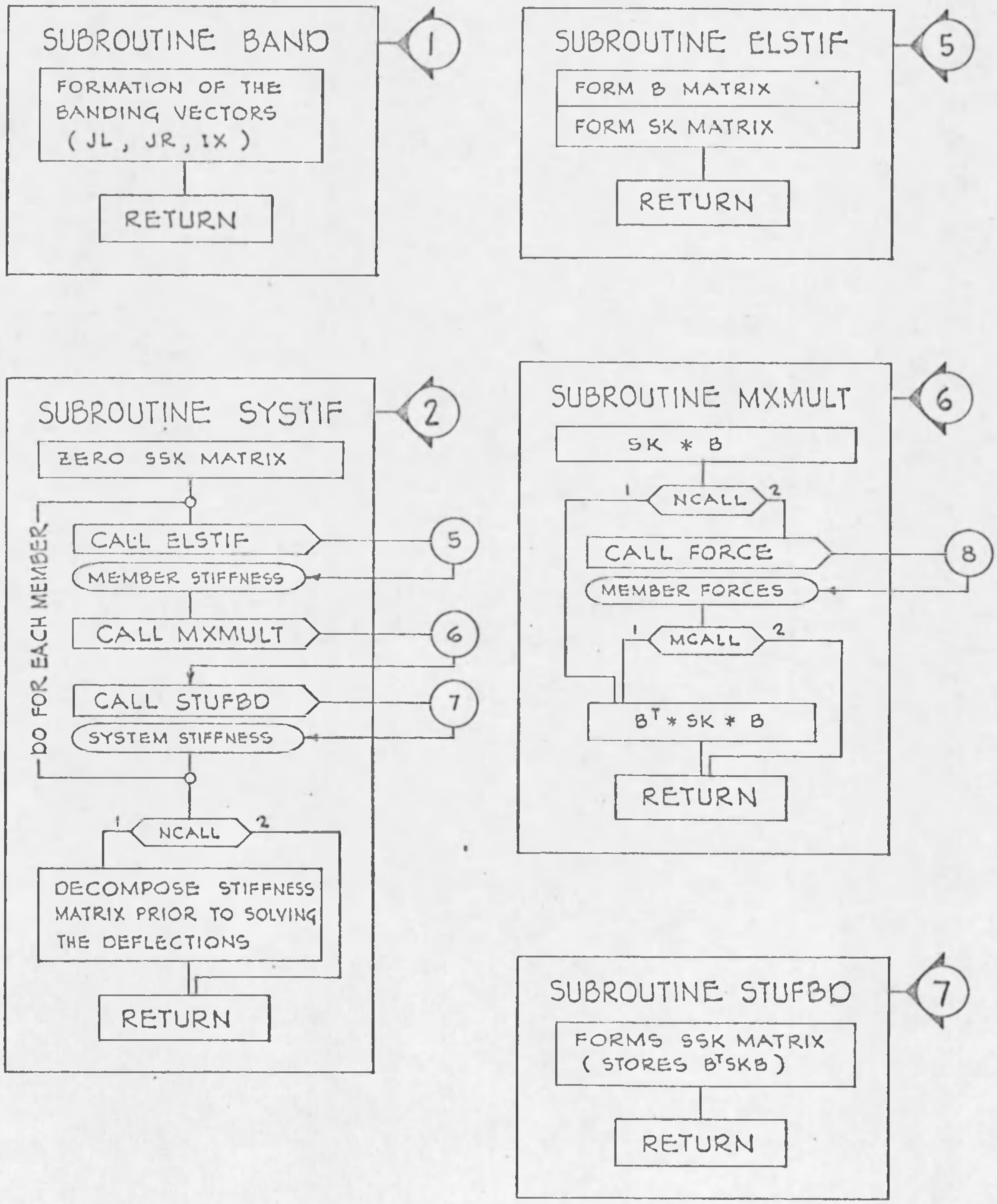
The program presented is not meant to replace the structural engineer but is to facilitate the mathematical calculations of the design analysis. A prime consideration has been to keep the running time of the program as short as possible, yet still cover the areas of analysis necessitated by standard engineering practices or required by the building codes. Gravity and lateral loading conditions have each been considered and a freedom of structural configurations has been allowed for. The program is not practical for small single story frames nor is it designed for nonframed structures. It has been designed to handle the intermediate framed structures, two or more stories in height. Very large structures are impractical due to the computer storage limitations. The program at present is limited to steel members. In time, variations either in the present program or in sister programs allowing for analysis of concrete design or incorporation elements such as shear walls, can be added.

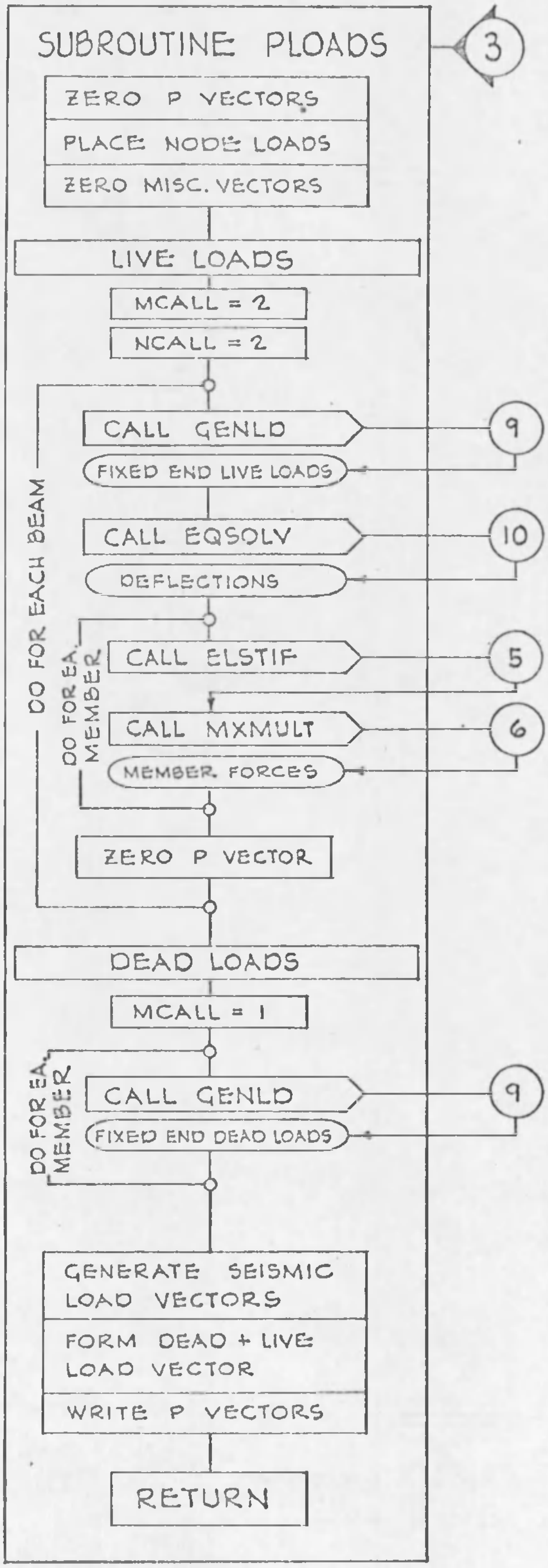
Using longhand methods, an engineer will spend approximately forty hours analyzing the stresses for the single plane frame shown in Example Number 3. This time can be translated to about five hundred dollars in cost of analysis. The computer can be programed to analyze the same frame with the same loads, arriving at more exact answers with no chance of error in about twelve seconds or for less than five dollars in computer costs. This represents a considerable savings. Based on the above rough cost estimate, programs of the type presented here definitely have a practical commercial value.

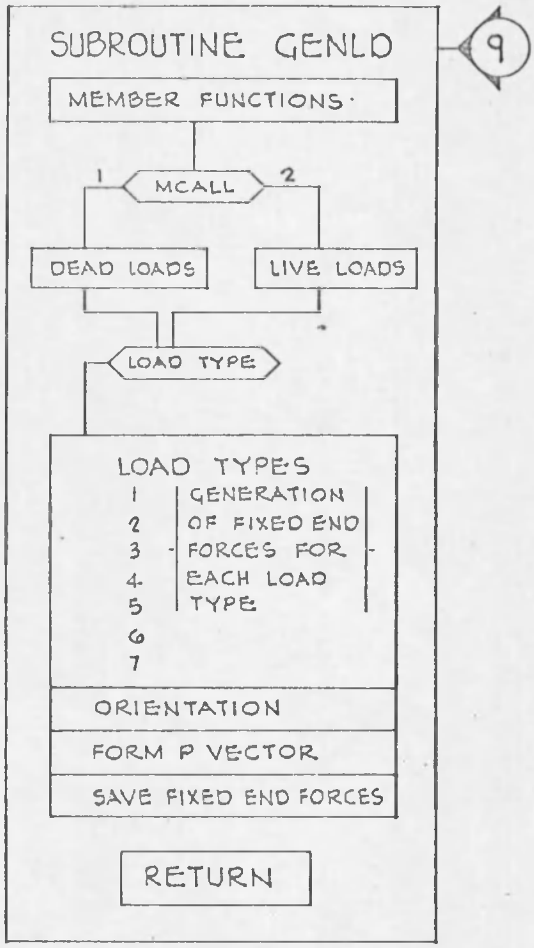
APPENDIX A

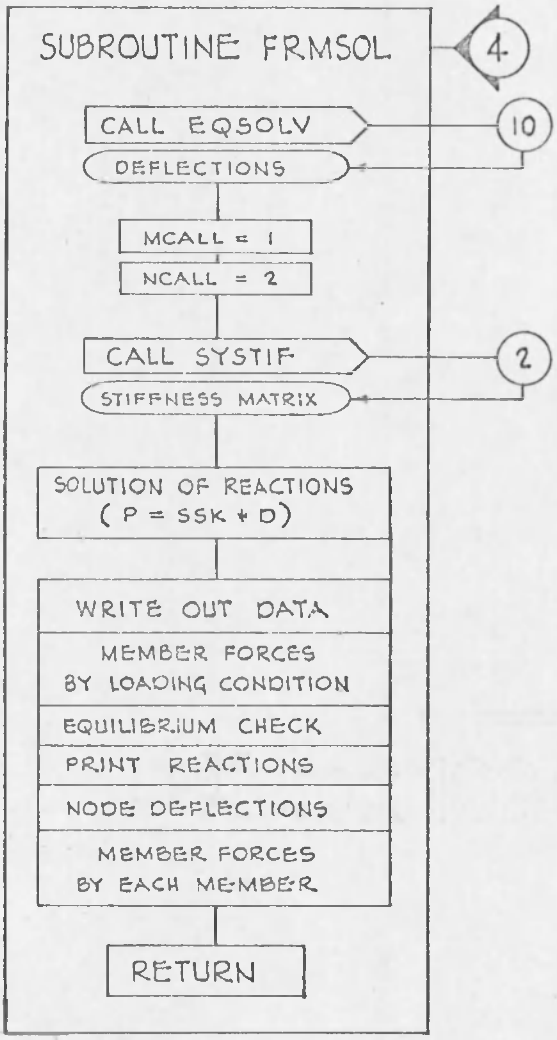
Flow Chart For Computer Analysis

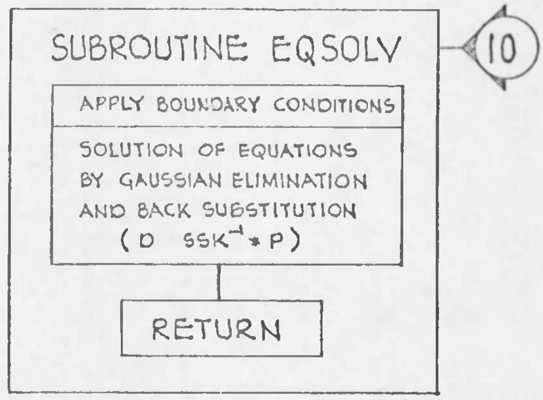


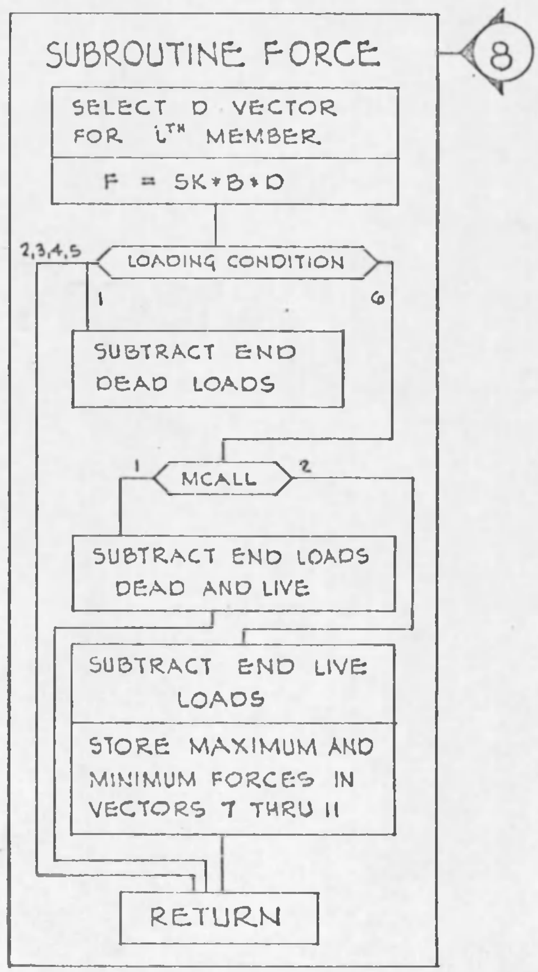

SUBROUTINE BAND
FORMATION OF THE BANDING VECTORS (JL, JR, IX)
RETURN
1
SUBROUTINE ELSTIF
FORM B MATRIX
FORM SK MATRIX
RETURN
5
SUBROUTINE SYSTIF
2
ZERO SSK MATRIX
DO FOR EACH MEMBER
CALL ELSTIF
5
MEMBER STIFFNESS
CALL MXMULT
6
CALL STUFBD
7
SYSTEM STIFFNESS
NCALL
1
2
DECOMPOSE STIFFNESS MATRIX PRIOR TO SOLVING THE DEFLECTIONS
RETURN
SUBROUTINE MXMULT
6
SK * B
NCALL
1
2
CALL FORCE
8
MEMBER FORCES
MCALL
1
2
B^T * SK * B
RETURN
SUBROUTINE STUFBD
7
FORMS SSK MATRIX (STORES B^TSKB)
RETURN


SUBROUTINE PLOADS
3
ZERO P VECTORS
PLACE NODE LOADS
ZERO MISC. VECTORS
LIVE LOADS
MCALL = 2
NCALL = 2
DO FOR EACH BEAM
CALL GENLD
9
FIXED END LIVE LOADS
CALL EQSOLV
10
DEFLECTIONS
DO FOR EA. MEMBER
CALL ELSTIF
5
CALL MXMULT
6
MEMBER FORCES
ZERO P VECTOR
DEAD LOADS
MCALL = 1
DO FOR EA. MEMBER
CALL GENLD
9
FIXED END DEAD LOADS
GENERATE SEISMIC LOAD VECTORS
FORM DEAD + LIVE LOAD VECTOR
WRITE P VECTORS
RETURN


SUBROUTINE GENLD
9
MEMBER FUNCTIONS
1
MCALL
2
DEAD LOADS
LIVE LOADS
LOAD TYPE
LOAD TYPES
1
2
3
4
5
6
7
GENERATION OF FIXED END FORCES FOR EACH LOAD TYPE
ORIENTATION
FORM P VECTOR
SAVE FIXED END FORCES
RETURN


SUBROUTINE FRMSOL
4
CALL EQSOLV
10
DEFLECTIONS
MCALL = 1
NCALL = 2
CALL SYSTIF
2
STIFFNESS MATRIX
SOLUTION OF REACTIONS
(P = SSK * D)
WRITE OUT DATA
MEMBER FORCES
BY LOADING CONDITION
EQUILIBRIUM CHECK
PRINT REACTIONS
NODE DEFLECTIONS
MEMBER FORCES
BY EACH MEMBER
RETURN


SUBROUTINE EQSOLV
10
APPLY BOUNDARY CONDITIONS
SOLUTION OF EQUATIONS
BY GAUSSIAN ELIMINATION
AND BACK SUBSTITUTION
(D SSK-1 * P)
RETURN


SUBROUTINE FORCE
8
SELECT D VECTOR
FOR ith MEMBER
F = SK * B * D
2,3,4,5
LOADING CONDITION
1
6
SUBTRACT END
DEAD LOADS
1
MCALL
2
SUBTRACT END LOADS
DEAD AND LIVE
SUBTRACT END LIVE
LOADS
STORE MAXIMUM AND
MINIMUM FORCES IN
VECTORS 7 THRU 11
RETURN

APPENDIX B

INPUT INFORMATION

SPACE REQUIREMENTS

Size limitations for in core storage must be maintained as follows:

No. of NODE POINTS	100
No. of MEMBERS	200
Maximum NODE SEPARATION	19

Space requirements are dimensioned as follows:

NAME	DEPENDENCE	QUAN.	SPACE/PER	TOTAL SPACES DIMENSIONED
General Indexes	-	16	1	16
Work Arrays	-	4	36	144
Work Vectors	-			9
Stiffness Matrix	Nodes	60	300	18,000
Banding Vectors	Nodes	20	300	6,000
Node Points	Nodes	4	100	400
Library	Library	8	50	400
Supports		7	100	700
Floors		4	30	120
Members	Members	27	200	5,400
Member Forces	Members	72	200	14,420
Miscellaneous		1	100	100
Miscellaneous				2,000
	TOTAL			47,639

The limits of the CDC 6400 is 48,000 spaces.

INPUT DATA FORMATS

General Information

NO. OF NODES	NO. OF COLUMNS	NO. OF MEMBERS	NO. OF SUPPORTS	NODE SEPERATION	NO OF LIBRARY
10	20	30	40	50	60

Node Point Information

NODE PT. NO.	X-COORD (FT)	Y-COORD (FT)
10	20	30

Library of Sections

CD. NO.	SECTION NAME	WEIGHT (LBS/FT)	AREA (SQ.IN.)	DEPTH (IN)	Ixx (IN^4)	IYY (IN^4)
4	12	19	26	33	52 – 59	66 – 73

Member Information

	MEM NO.	P END	Q END	SECT NO	LOAD TYPE	UNIFORM LD. (KIPS/FT)	POINT LD. 1 (KIPS)	POINT LD. 2 (KIPS)	A DIMENSION (FEET)	B DIMENSION (FEET)	END COND. S	L	S	L
COLS	4	8	12	16	20	30	40	50	60	70	72	74	76	78
BEAMS	4	8	12	16	20	30	40	50	60	70	72	74	76	78

Live Load Information

NO. OF DIFFERENT LIVE LOADS — 5

IF ZERO OMIT FOLLOWING CARD

LIVE LOADS (KIPS/FT.)	NO OF MEM.	THE MEMBER NUMBERS												
10	15	20	25	30	35	40	45	50	55	60	65	70	75	80

Loads At The Node Points

NO. OF LDS. VECT. 1	NO. OF LDS. VECT. 2	NO. OF LDS. VECT 3
10	20	30

2 CARDS BELOW FOR EACH LOAD VECTOR
OMIT IF ZERO NO OF LDS

THE LOADED POINTS IN ORDER

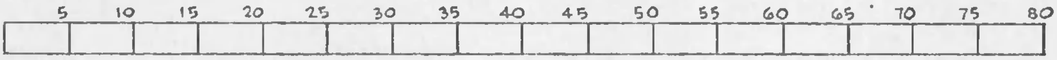
5	10	15	20	25	30	35	40	45	50	55	60	65	70	75	80

THE LOADS CORRESPONDING TO THE POINTS ABOVE

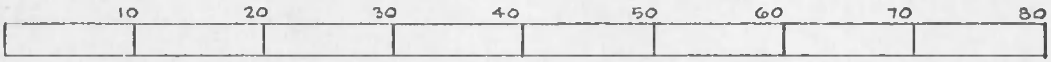
10	20	30	40	50	60	70	80

Support Information

THE SUPPORTED POINTS



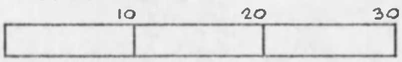
THE DEFLECTIONS ASSOCIATED WITH THE POINTS ABOVE



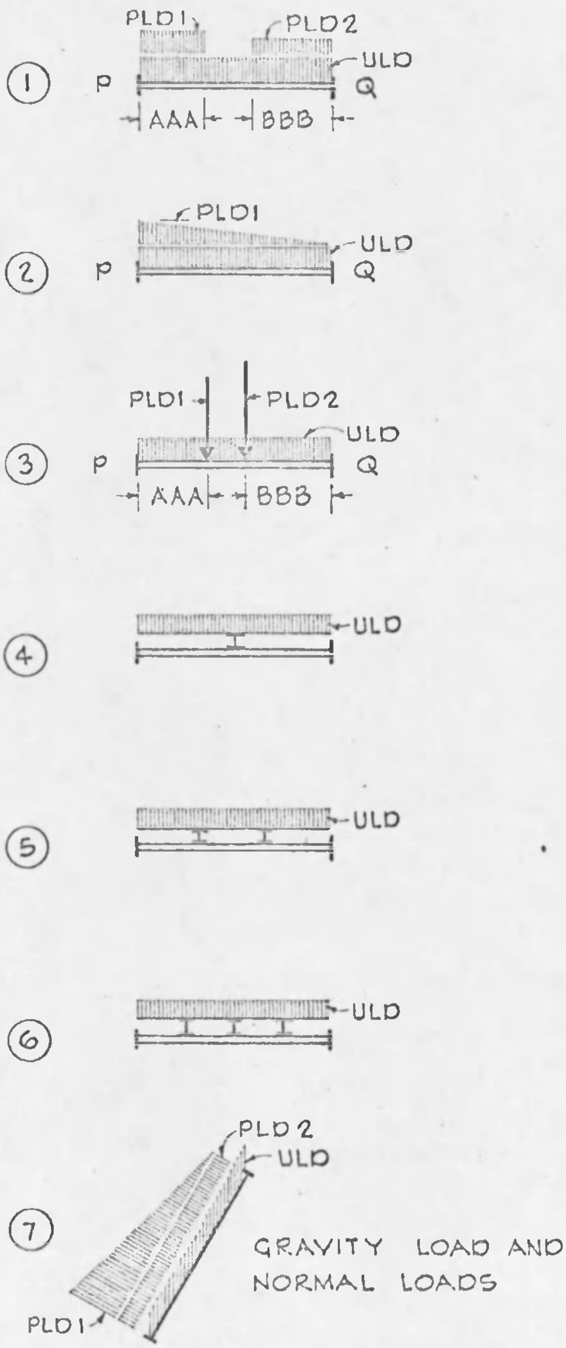
Seismic Constants

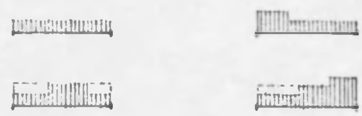
(SEE UNIFORM BUILDING CODE)

ZONE K C

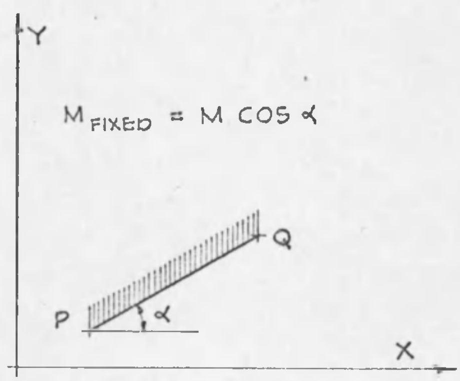


LOAD TYPES





COMBINATIONS OF ①



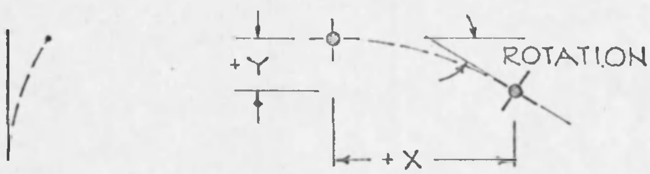
MEMBER ORIENTATION

APPENDIX C

OUTPUT DATA

DEFLECTIONS

FOR EACH NODE POINT



MEMBER END FORCES

FOR EACH MEMBER

P END		Q END	
1	AXIAL FORCE	4	AXIAL FORCE
2	SHEAR	5	SHEAR
3	MOMENT	6	MOMENT

LOADING CONDITIONS

deflections are given for loading conditions one thru six
member forces are given for loading conditions one thru eleven

1 DEAD LOADING - generated from member loads or applied directly to nodes in load vector one

2+ WIND LOADS - applied to nodes in load vector two

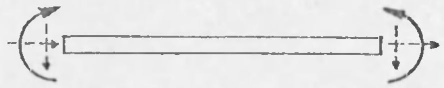
3- WIND LOADS - applied to nodes in load vector three

4+ SEISMIC LOADS - generated, load vector four

5- SEISMIC LOADS - generated, load vector five

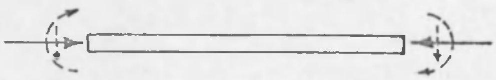
6. GRAVITY LOADS - sum of dead plus live loadings.

7 MAXIMUM MOMENT DUE TO LIVE LOADING
with associated axial force and shear



8 MINIMUM MOMENT DUE TO LIVE LOADING
with associated axial force and shear

9 MAXIMUM AXIAL FORCE - COMPRESSION
with associated shear and moment



10 MAXIMUM SHEAR DUE TO LIVE LOADING

11 MINIMUM SHEAR DUE TO LIVE LOADING

REACTIONS
reactions are given and an equilibrium check is made for loading conditions one thru six

TOTAL WEIGHT
of structural members is given

REFERENCES

American Institute of Steel Construction, Inc. *Manual of Steel Construction*, 6th ed. New York: 1963.

Bull, F. B., and G. Sved. *Moment Distribution in Theory and Practice*, London: Pergamon Press, Ltd., 1964.

Coull, A., and B. Stafford Smith (eds.). *Tall Buildings*, London: Pergamon Press, Ltd., 1967.

Hyman, C. J. (trans.). *Analysis of Multistory Frames*, by Gaspar Kani. New York: Frederick Ungar Publishing Co., 1957.

International Conference of Building Officials. *Uniform Building Code*, Vol. I, 1967.

Mantell, M. I., and J. F. Marron. *Structural Analysis*. New York: The Ronald Press Co., 1962.

Murrill, P. W., and C. L. Smith. *Fortran IV Programming For Engineers and Scientists*, Scranton: International Textbook Co., 1968.

Norris, C. H., and J. B. Wilbur. *Elementary Structural* Analysis, 2nd ed. New York: McGraw-Hill, 1960.

Turner, M. J., R. W. Clough, H. C. Martin, and L. J. Topp. "Stiffness and Deflection Analysis of Complex Structures," *Journal of Aeronautical Sciences*, Vol. 23, No. 9, (September, 1956).

Weaver, W. *Computer Programs For Structural Analysis*, Princeton: D. Van Nostrand Co., 1967.

Wylie, C. R. *Advanced Engineering Matematics*, 3rd ed., New York: McGraw-Hill, 1966.