DEVELOPMENT OF A GENERALIZED STRUCTURAL ANALYSIS SYSTEM

A Thesis
Presented to the
Department of Civil Engineering Science
Brigham Young University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

by
Steven E. Bensley
August 1967
DEVELOPMENT OF A GENERALIZED STRUCTURAL ANALYSIS SYSTEM

A Thesis
Presented to the
Department of Civil Engineering Science
Brigham Young University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

by
Steven E. Benzley
August 1967
This thesis, by Steven E. Benzley is accepted in its present form by the Department of Civil Engineering Science of Brigham Young University as satisfying the thesis requirements for the degree of Master of Science.

May 27, 1967
Date

[Signature]
Chairman, Advisory Committee

[Signature]
Member, Advisory Committee

[Signature]
Chairman, Major Department
DEDICATION

To my wife, Karen
ACKNOWLEDGMENT

The author wishes to express his gratitude to Dr. Henry N. Christiansen for his advice and guidance in this project and the preparation of this thesis.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPROVALS</td>
<td>ii</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENT</td>
<td>iv</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF SYMBOLS</td>
<td>vii</td>
</tr>
<tr>
<td>CHAPTER</td>
<td></td>
</tr>
<tr>
<td>I. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>II. DEVELOPMENT OF THE STIFFNESS METHOD</td>
<td>3</td>
</tr>
<tr>
<td>III. SYSTEM OPERATIONS</td>
<td>7</td>
</tr>
<tr>
<td>Link I</td>
<td>7</td>
</tr>
<tr>
<td>Link II</td>
<td>13</td>
</tr>
<tr>
<td>Link III</td>
<td>15</td>
</tr>
<tr>
<td>IV. REQUIREMENTS OF THE USER SUPPLIED</td>
<td>18</td>
</tr>
<tr>
<td>SUBROUTINES</td>
<td></td>
</tr>
<tr>
<td>V. INPUT REQUIREMENTS</td>
<td>24</td>
</tr>
<tr>
<td>Method of Input</td>
<td>24</td>
</tr>
<tr>
<td>Data Deck</td>
<td>24</td>
</tr>
<tr>
<td>VI. EXAMPLE PROBLEM</td>
<td>31</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>36</td>
</tr>
<tr>
<td>APPENDIX</td>
<td>37</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Block Diagram of Link I</td>
<td>8</td>
</tr>
<tr>
<td>2. Block Diagram of Link II</td>
<td>14</td>
</tr>
<tr>
<td>3. Block Diagram of Link III</td>
<td>16</td>
</tr>
<tr>
<td>4. (a) Sample Deck Setup 1</td>
<td>25</td>
</tr>
<tr>
<td>(b) Sample Deck Setup 2</td>
<td>26</td>
</tr>
<tr>
<td>5. (a) Data Card Formats</td>
<td>28</td>
</tr>
<tr>
<td>(b) Data Card Formats</td>
<td>29</td>
</tr>
<tr>
<td>(c) Data Card Formats</td>
<td>30</td>
</tr>
<tr>
<td>6. Example Problem</td>
<td>31</td>
</tr>
<tr>
<td>7. Listing of Data</td>
<td>33</td>
</tr>
<tr>
<td>8. (a) Example Problem Output</td>
<td>34</td>
</tr>
<tr>
<td>(b) Example Problem Output</td>
<td>35</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS

The following symbols have been adapted for use in this thesis:

C = Displacement transformation matrix
D = Applied displacement vector
E = Young's modulus
F = Constrained nodal forces
K = Stiffness matrix
q = Generalized displacements
Q = Generalized forces
u = Constrained displacements
U = Work
γr = Poisson's ratio
CHAPTER I

INTRODUCTION

The purpose of this thesis is the development of a generalized structural analysis system. The procedure developed is programed (in Fortran IV) for use on a general purpose digital computer (IBM 7040) and provides a basic system of matrix operations compatible with an assortment of structural elements. The direct stiffness approach is the method used by the system. This report does not include the discussion of a stiffness matrix generation routine for any particular element, but does set forth the requirements for the inclusion of new (user supplied) element subroutines.

The system consists of three links. Link I reads nodal coordinates, material properties, displacement constraints and loading systems. It calls the user supplied element subroutine, develops transformation matrices and outputs the transformed stiffness matrix. Link II sums the stiffness and loading matrices, solves equations and outputs displacements. Link III performs the inverse transformations and solves for element stresses and/or strains. A grid generation link may be added in the future.

The following system documentation is divided into three parts:

1. A technical discussion of the required matrix operations associated with the direct stiffness method.
2. An explanation of the system which includes:
   a) Overall discussion of the system
   b) Requirements of user supplied subroutines
   c) Input data requirements

3. Discussion of a rather general example problem
CHAPTER II

DEVELOPMENT OF THE STIFFNESS METHOD

The stiffness method of structural analysis is based upon force equilibrium equations relating the nodal displacements and the applied nodal forces. In matrix form these equations are:

\[
[K][u] = [F]
\]

(1)

where

\[
[u] = \text{nodal displacement vector (displacements and rotations)}
\]
\[
[F] = \text{applied nodal force vector}
\]
\[
[K] = \text{a square, symmetric matrix of stiffness coefficients called the stiffness matrix}
\]

The jth column of \([K]\) is seen to be the set of applied loads required to obtain a unit value of the jth displacement component.

The overall stiffness matrix \([K]\) is in terms of all n coordinates of the structure. The displacement vector \([u]\), consisting of n components, is normally constrained by boundary conditions on the structure and often is further constrained by conditions of symmetry. Each boundary or symmetry condition yields an equation of constraint which reduces by one the number of independent degrees of freedom of the system. The total set of nodal displacement components is then related to a reduced set of independent components in a linear manner. This relationship is called a linear transformation and takes the form
\[ [u] = [C][q] + [D] \]  

(2)

where

\([u]\) = constrained nodal displacement vector (displacements and rotations)

\([q]\) = generalized nodal displacement vector (displacements and rotations)

\([C]\) = transformation matrix relating the \(q_j\)'s to the \(u_i\)'s

\([D]\) = a vector listing applied displacements

The corresponding force transformation is developed by considering the work done by the set of applied forces during their application. Work is defined as \(\int F \cdot ds\). Since (in the elastic region) stress is linearly related to strain, the total work done by the applied loads is equal to \(1/2\) the sum of the products of the final magnitude of the applied load and corresponding displacement. In matrix form this is expressed by

\[ U = \frac{1}{2}[F]^T[u] \]  

(3)

where

\(U\) = work done by external forces

\([F]\) = applied force vector

\([u]\) = displacement vector

Introducing the displacement transformation

\[ [u] = [C][q] + [D] \]

into equation 3,

\[ U = \frac{1}{2}([F]^T[C][q] + [F]^T[D]) \]
\([F]^T[D]\) is the work done by the external reactions corresponding to the non-zero displacement components of \([D]\). The scalar \(\frac{1}{2}[F]^T[C][q]\) is the work done by the generalized forces \([Q]\) acting through the generalized displacements \([q]\). Therefore

\[
[Q]^T = [F]^T[C]
\]
or

\[
[Q] = [C]^T[F]
\] (4)

It is significant that \([Q]\) does not represent the total force system, only that part corresponding to the generalized displacements \([q]\).

The development of the equilibrium equations in the generalized system involves both the force and displacement transformations.

Starting with equilibrium equations (1)

\[
[K][u] = [F]
\]

and introducing the displacement transformation

\[
[u] = [C][q] + [D]
\]

the equilibrium equations become

\[
[K][C][q] + [K][D] = [F]
\] (5)

which is a representation of the same equilibrium equations in terms of the specified displacements \([D]\) and the generalized displacements \([q]\).

Premultiplication of the above equation by \([C]^T\) gives

\[
[C]^T[K][C][q] + [C]^T[K][D] = [C]^T[F]
\]

The right hand side of these equations is seen to be the generalized forces \([Q]\) in terms of \([F]\), therefore
\([C]^P[K][C][q] + [C]^P[K][D] = [Q]\)

\([C]^P[K][C]\) is the stiffness matrix in the generalized coordinate system, \([C]^P[K][D]\) is the representation of the forces in the generalized system required to produce the displacements \([D]\) in the constrained system. Thus

\([C]^P[K][C][q] = [Q] - [C]^P[K][D]\)

are the equilibrium equations in the generalized system.
CHAPTER III

SYSTEM OPERATIONS

This chapter explains in general terms the operation of the three links of the system. The requirements of the user supplied subroutines and the information available to the subroutines are discussed in detail in Chapter IV. Chapter V explains the data input format. For one interested in using the system and element subroutines already supplied, an understanding of Chapter V is all that is necessary.

The system is composed of three links. Each link is run separately by the computer with necessary information being passed from one link to another by means of binary tapes. The following discussion explains the operation of each link.

LINK I

A block diagram showing the operation of link I is seen in figure 1. The system initially reads, writes, and stores the necessary data to define the problem. This consists of the system parameters, the problem title and identification, material properties, nodal coordinates, displacement transformations, and concentrated and distributed loads. Each of these items is discussed below. A discussion of the storage locations of the above coefficients is found in Chapter IV.

The system parameters are the displacement direction numbers
BLOCK DIAGRAM OF LINK I

START

READ AND WRITE
(a) Parameters
(b) Problem title cards
(c) Material properties
(d) Nodal coordinates
(e) Displacement transformations
(f) Concentrated loads
(g) Distributed loads
(h) Loading system definitions

Read element data card

is there an element?
No → Exit link I
Yes → Call proper element subroutine

ELEMENT SUBROUTINE
Forms stiffness and loading matrices

Form element transformation matrices
Transforms stiffness and loading matrices

Output equations (tape)

Figure 1
defining the degrees of freedom of the structure, the number of title cards used, and ILOD, the number of loading systems.

Material properties necessary for certain element subroutines are supplied by the system. The material may be isotropic or orthotropic. In the case of an isotropic material, the coefficients available are the modulus of elasticity, Poisson's ratio, density, coefficient of expansion and elasticity factors. Elasticity factors are members of the E matrix where

\[
[e] = [E][T].
\]

where

\[
e = \text{strains caused by the stresses}
\]

\[
T = \text{stresses}
\]

\[
E = \text{elasticity matrix}
\]

i.e.

\[
\begin{bmatrix}
e_{11} \\
e_{22} \\
e_{33} \\
2e_{12} \\
2e_{13} \\
2e_{23}
\end{bmatrix}
= \begin{bmatrix}
E_{11} & E_{12} & E_{13} & 0 & 0 & 0 \\
E_{12} & E_{22} & E_{23} & 0 & 0 & 0 \\
E_{13} & E_{23} & E_{33} & 0 & 0 & 0 \\
0 & 0 & 0 & G_{12} & 0 & 0 \\
0 & 0 & 0 & 0 & G_{13} & 0 \\
0 & 0 & 0 & 0 & 0 & G_{23}
\end{bmatrix}
\begin{bmatrix}
T_{11} \\
T_{22} \\
T_{23} \\
T_{12} \\
T_{13} \\
T_{23}
\end{bmatrix}
\]

1 See displacement transformation section of this chapter for identification of displacement direction numbers.

2 A prime (') by a symbol indicates the symbol as found in the program.
where

$$E_{ij} = \begin{cases} \frac{1}{E} & i = j \\ \nu/E & i \neq j \end{cases}$$

where $E$ = modulus of elasticity
$\nu$ = Poisson's ratio

$$G_{ij} = 2(1 + \nu)/E$$

For an orthotropic material the coefficients supplied are the density, the coefficients of thermal expansion in the $X_1$, $X_2$, $X_3$ directions and the members of the elasticity matrix. In the case of an orthotropic material the $E_{ij}$'s and $G_{ij}$'s are no longer functions of a single modulus of elasticity or Poisson’s ratio.

**Loading systems definitions** describe the loading systems applied to the structure. Six systems are allowed. A system is a combination of applied nodal or distributed forces, specified element temperature changes, and a definition of the inertial field.

**Nodal coordinates** are the rectangular cartesian coordinates ($X_i$) of the nodes.

**Concentrated nodal loads** are read into the system as forces in the $X_1$, $X_2$, $X_3$ directions and moments about the $X_1$, $X_2$, $X_3$ axes.

**Distributed loads** are defined as uniform loadings in the $X_1$, $X_2$, $X_3$ coordinate directions along the length of a beam element, over the surface of a plate element, or on a specified surface of a three dimensional element.

**Displacement transformations** must be of the form

$$[u] = [C'][q] + [D']$$
where

\[ [u] = \text{constrained displacements} \]
\[ [q] = \text{generalized displacements} \]
\[ [C'] = \text{matrix of relating coefficients between u and q} \]
\[ [D'] = \text{list of applied displacements} \]

The necessary numbers in forming these transformations are the row numbers (equation numbers) and relating coefficients.

Row numbers order the set of equilibrium equations. One row number is required for each degree of freedom of the structure. The equations begin numbering at node 1, taking the equilibrium equations in the following order:

1. Displacement in X1 direction
2. Displacement in X2 direction
3. Displacement in X3 direction
4. Rotation about the X1 axis
5. Rotation about the X2 axis
6. Rotation about the X3 axis

where the numbers 1-6 are the displacement direction numbers. Therefore, the node and direction numbers of a displacement are necessary to define a row number. For example, the row number of the displacement in the X2 direction at node 2 is 8.* These row numbers are specifically referred to as unreduced row numbers because they refer to constrained (unreduced) nodal equilibrium equations.

* Identifying equation for row numbers is \( RN = N \times 6 - 6 + D \)
\( N = \text{node number} \quad RN = \text{row number} \quad D = \text{displacement direction number} \)
Constraints may be of two forms:

1. The relating of one displacement to one or more other displacements

2. Defining the value of a displacement

A constraint condition relating displacements is a row of the \([C']\) transformation matrix. Thus a member of the constrained coordinate system, \(u_k\), may be related to generalized coordinates, \(q_j\), in the following manner:

\[
u_k = c_{k1}q_1 + c_{k2}q_2 + \cdots + c_{ki}q_i + \cdots + c_{kn}q_n
\]

where

\(c_{ki}\)’s are the members of the kth row of the \([C']\) matrix
n is limited to 35

In defining this type of constraint, one data card\(^3\) is required for each non-zero \(q_i\). Each data card will include the following information in the order shown:

1. Node and displacement direction number of constrained coordinate, \((u_k)\)

2. Node and displacement direction number of generalized coordinate, \((q_i)\)

3. Related coefficient, \((c_{ki})\)

When defining an applied displacement, i.e., a member of the \([D']\) matrix, only the node and displacement direction number and applied displacement value need be entered.

The element data card identifies\(^4\) the type of element being

\(^3\) See figure 5a in Chapter V for data card format.

\(^4\) A list of identifying numbers and elements is found in appendix.
entered, and supplies the element temperature, node numbers, and panel number.

After an element card is read, the system transfers control to the proper element subroutine. The subroutine then supplies the system with the stiffness matrix for the element in the overall coordinate system, the unreduced row numbers, and the loading vector for the element. Control then returns to the system.

The transformation matrices \([C']\) and \([D']\) are then formed along with the corresponding reduced row numbers, where the reduced row numbers are the ordered set of equation numbers of the generalized \([q]\) system. The transformations are applied to the element stiffness matrix and loading vector. The generalized stiffness matrix and loading vector are then output on tape. The system continues reading element cards and outputting transformed equations until all elements have been processed. The system then exits link I.

**LINK II**

A block diagram showing the basic operation of link II is seen in figure 2. A discussion of this diagram follows.

Link II initially reads the following parameters from binary tape:

1. Number of equilibrium equations of the structure
2. Number of elements in the structure
3. Bandwidth\(^5\) of total stiffness matrix

---

\(^5\) Bandwidth is a term applied to square matrices with all non-zero factors near the diagonal. It is the maximum number of columns, existing between non-zero components of a row, in the matrix.
BLOCK DIAGRAM OF LINK II

1. Read problem parameters (tape)

   READ
   (a) Size of stiffness matrix
   (b) Number of loading systems
   (c) Reduced row numbers
   (d) Element stiffness matrix
   (e) Loading vectors

2. Add element stiffness matrix to overall stiffness matrix

3. Is there another element?
   Yes
   No

4. Solves equations (Choleski)

5. Outputs generalized displacements (tape)

6. Exit link II

Figure 2
4. Number of loading systems

From a tape supplied by link I the following data is read for each element of the structure:

1. Size of element stiffness matrix
2. Number of loading systems
3. Reduced row numbers for element
4. Element stiffness matrix (transformed)
5. Loading vectors

Link II takes the preceding data, one element at a time, and builds the complete set of generalized equilibrium equations. If all of these equations cannot fit in core, tape storage is used.

With the complete set of generalized equations in storage, link II uses the Choleski\(^6\) method for solving simultaneous equations.

**LINK III**

A block diagram showing the operation of link III is seen in figure 3. A discussion of this diagram follows.

The transformation definitions stored on tape in link I are supplied along with the generalized displacement vectors from link II (also on tape). The inverse transformations are applied directly to the generalized displacement vectors forming the constrained displacements. The constrained displacements are printed for each loading system.

Link III reads from tape the element identification number. The system then transfers control to the proper element subroutine.

---

BLOCK DIAGRAM OF LINK III

Read transformation definitions (tape)

Read generalized displacement vectors (tape)

Apply transformations to generalized displacement vectors

Write constrained displacement vectors

Read element type (tape)

Is there an element? No

Exit link III

Yes

Call proper element subroutine

ELEMENT SUBROUTINE

(a) Read tape information
(b) Output stresses and/or strains

Figure 3
The subroutine outputs the element stresses and/or strains. This procedure cycles until all elements of the structure have output their results. The system then exits link III.
CHAPTER IV

REQUIREMENTS OF THE USER SUPPLIED SUBROUTINES

This chapter explains the information that is made available to, and what is required of, user supplied element subroutines by each of the three links of the system. Included is a list of the variables that are kept in COMMON between the system and the subroutines.

The following conditions exist in link I when an element subroutine is called:

1. Material table available
2. Nodal coordinates available
3. Concentrated loads available
4. Distributed, thermal, and acceleration loading definitions available
5. Tape 0 positioned for output of element information

The following information must be generated by the subroutine:

1. A stiffness matrix in X1, X2, X3 coordinate system
2. A loading vector (i.e., concentrated loads should be added by the subroutine)
3. Unreduced row numbers which are generated from the equation

\[ RN = N \times 6 - 6 + D \]

where

RN = Row number

N = Node number
D = Direction Number

The subroutine must output the description of each element. This requires the writing of a descriptive title block and the parameters which identify the element. These parameters are to include the element node numbers, panel number, temperature, and material. Other values, which will vary between elements, may be output if necessary to describe the element. The output of these parameters should be limited to one line (130 spaces). The system will keep in COMMON, in location IELE', the identification number of the last element subroutine used. The subroutine is to test on this number. If the identification number is not that of the element subroutine, both the title block and parameters are to be output. If the identification number is that of the subroutine, no title block should be output.

The subroutine has tape unit 0 available to store the necessary information (stress matrix) to be used in link III.

Each link I element subroutine is required to have the following COMMON statement. X(1000,3), TURN(24), TABLE(20,17), XXX(35,35), NF(50), F(50,6), ITYPE, XMAT, TEMP, DATA(1000), L(9), DEL(24,6), ILCD, IELE, DL(50,3), NDL(50,4), NDLS(50), NFS(50), SYS(6,6).

The quantities in this statement that are supplied by the system are:

1. X(1000,3) - storage for the nodal cartesian coordinates.

---

7 See Chapter III under displacement transformations for definition of direction number.

8 The representation (1000,3) specifies a two dimensional array (matrix) with 1000 rows and 3 columns. Only one number enclosed in parenthesis specifies a one dimensional array (vector).
The coordinate values for the X1, X2, X3 axes are placed in columns 1, 2, 3 respectively of the X matrix with the row of X being the particular node number.

2. **TABLE (20,17)** - storage for the material properties. Twenty entries into this array are possible. The 17 values for each entry, according to column location, are:
   
a) Material name \((A4)^9\)
   b) Material temperature \((E)\)
   c) \(E_{11}\) \((E)\)
   d) \(E_{12}\) \((E)\)
   e) \(E_{13}\) \((E)\)
   f) \(E_{22}\) \((E)\)
   g) \(E_{33}\) \((E)\)
   h) \(G_{12}\) \((E)\)
   i) \(G_{13}\) \((E)\)
   j) \(G_{23}\) \((E)\)
   k) Modulus of elasticity \((E)\)
   l) Poisson's ratio \((E)\)
   m) Density \((E)\)
   n) Coefficient of thermal expansion in X1 direction \((E)\)
   o) Coefficient of thermal expansion in X2 direction \((E)\)
   p) Coefficient of thermal expansion in X3 direction \((E)\)

3. \(F(50,6)\), \(NF(50)\), and \(NF(50)\) are storage locations for applied nodal forces. The node number is listed in the vector \(NF\), the loading system number \(^{10}\) (six systems allowed) is stored in the corresponding

---

9 Quantities in parenthesis indicate FORTRAN format designation in which variable is entered.

* \(E_{11}, E_{12}, \ldots, G_{23}\) are values as defined by E matrix (Chapter III).

10 The loading system number refers to the particular loading vector \(DEL(24,6)\).
row of NPS. The forces in the X1, X2, and X3 directions and moments about the X1, X2, and X3 axes are stored in column 1 thru 6 respectively of F. The element subroutine, after once placing a concentrated nodal force in the correct loading vector for the element, should set that value of the F matrix equal to zero. The system transformations will allow for the loading of all other elements in that direction at that node.

4. DL(50,3), NDL(50,4) and NDLS(50) are storage locations for distributed loading definitions. The associated node numbers are listed in NDL and the loading system number is stored in the corresponding row of NDLs. The distributed loading definitions in the X1, X2, and X3 directions are stored in the corresponding row of DL in columns 1 thru 3 respectively.

5. ITYPE, XMAT, TEMP, and L(9) are the quantities read on the element identification card. ITYPE is the element identification number. XMAT is the material type and TEMP is the temperature of the element. L(9) contains the element nodal coordinates in L(1) thru L(8) and the panel number in L(9).

6. ILOD is the number of loading systems.

7. IELE is the element identification number of the last element read by the system.

8. SYS(6,6) defines the loading systems. Each column identifies a separate system. The ith loading system is the following combination of loads:
   
   a) SYS(1,i) times the concentrated loads defined in loading system i.
   
   b) SYS(2,i) times the distributed load definitions of loading system i.
   
   c) SYS(3,i) times the thermal loads as defined.
   
   d) SYS(4,i) times the gravity loading in the X1 direction.
   
   e) SYS(5,i) times the gravity loading in the X2 direction.
   
   f) SYS(6,i) times the gravity loading in the X3 direction.

   where SYS(j,i) may be zero as required.

   The quantities of the COMMON statement that are formed by the

   See Appendix for list of identification numbers.
subroutines are:

1. XXX(35,35) – the stiffness matrix of the element in the X1, X2, X3 coordinate system.

2. IURN(24) – a list of unreduced row numbers for the element. The numbers are to come from the equation:

\[ R = N \times 6 - 6 + D \]

where

- \( R \) = unreduced row numbers
- \( N \) = node number
- \( D \) = displacement direction number

3. DEL(24,6) – the loading vectors for the element with six loading systems possible, and ILOD systems used. The loading vectors are to be formed by the subroutine by taking the combination of loads as specified by SYS(6,ILOD).

Link II sums and solves the equilibrium equations generated in link I. The system requires nothing of the subroutines in this routine.

In link III the system supplies the following information when the subroutine is called.

1. The constrained displacement vectors corresponding to the ILOD loading systems.

2. Tape unit 0 in position to read element stress information as output by the element subroutine in link I.

The following is required by the subroutine:

1. Output of a suitable title block

2. Output of stresses and/or strains of element

Each link III element subroutine is to have the following COMMON statement, TOTV(1000,6), ITYPE, IELE. TOTV contains the set of constrained displacements. The row location of a specific displacement is the unreduced row number of the displacement and the column location is the loading system number. ITYPE and IELE are the same as were defined in link I.
Whenever an element subroutine is added, the system is to be entered and a modification made in subroutine SEARCH. This subroutine is used in links I and III. An example of this modification is as follows:

Element type 23 is to be added to the system. A listing of SEARCH is shown below. The statements which are to be added are underlined.

```
SUBROUTINE SEARCH
COMMON (as defined for particular link)
IF(ITYPE.EQ.21) GO TO 721
IF(ITYPE.EQ.26) GO TO 726
IF(ITYPE.EQ.23) GO TO 723
WRITE(6,100) ITYPE
100 FORMAT(1HO,41HSYSTEM HAS NO CAPABILITY FOR ELEMENT TYPE, I5)
CALL EXIT
721 CALL ELE21
   M4* = 2
   RETURN
726 CALL ELE26
   M4 = 12
   RETURN
723 CALL ELE23
   M4 = 6
   RETURN
END
```

The name to be given an element subroutine is indicated in the above example (i.e., ELEITYPE, where ITYPE is the element identification number). It is the responsibility of the user to recompile the system and have it placed on the Systems Library tape in the Brigham Young University computer center whenever a new element subroutine is added.

* M4 is the size of the stiffness matrix formed by the subroutine.
CHAPTER V

INPUT REQUIREMENTS

The following discussion explains the methods of using the system and the data deck setup.

METHOD OF INPUT

The system may be accessed in one of two ways:

1. If no user supplied subroutine is to be added, the system may be entered by calling for CESAS from the Systems Library tape.

2. If a subroutine is to be supplied, a binary deck of the system is to be used.

In either case, the FORTRAN system control cards are required.

Sample deck setups for cases 1 and 2 are shown in figures 4a and 4b respectively.

DATA DECK

The following discussion of data cards will be by card type. Each type is defined as to coefficient, field definition, and format in figures 5a, 5b, and 5c.

The first entry is a card type A. This card contains system parameters. Title cards are the next to be entered. As many title cards as have been specified on card type one are to be used. There are no restrictions as to what may be placed on these cards, with the exception of a $ in column 1.

The following card types are placed in the data deck in blocks.
$IBSYS
BLANK CARD
ETC.
ELEMENT CARDS
CARD TYPE J
ELEMENT CARDS
CARD TYPE J
BLANK CARD
CARD TYPE I
BLANK CARD
CARD TYPE H
BLANK CARD
CARD TYPE G
BLANK CARD
CARD TYPE E OR F
BLANK CARD
CARD TYPE D
BLANK CARD
CARD TYPE B OR C1,C2,C3
TITLE CARDS
CARD TYPE A

$EXECUTE
$JOB

Figure 4a
Figure 4b
Each block is terminated by a blank card. The blocks are to be put in the data deck in the order shown.

<table>
<thead>
<tr>
<th>Block No.</th>
<th>Meaning</th>
<th>Card Types</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Materials Properties</td>
<td>B or C_1, C_2, C_3^*</td>
</tr>
<tr>
<td>2</td>
<td>Nodal Coordinates</td>
<td>D</td>
</tr>
<tr>
<td>3</td>
<td>Constraint Conditions^12</td>
<td>E or F</td>
</tr>
<tr>
<td>4</td>
<td>Concentrated Loads</td>
<td>G</td>
</tr>
<tr>
<td>5</td>
<td>Distributed Loads</td>
<td>H</td>
</tr>
<tr>
<td>6</td>
<td>Loading System Definitions</td>
<td>I</td>
</tr>
</tbody>
</table>

Element identification cards are the next to be entered. Each element must have one card type J. This card may be followed by other entries depending upon element type. For information about these other entries, refer to the discussion on the element type in question.

A blank card signifies the end of data.

^* Card types subscripted mean that more than one card is necessary for an entry.

^12 For a complete description of constraint condition possibilities, see Displacement transformations in Chapter III.
<table>
<thead>
<tr>
<th>CARD TYPE</th>
<th>COLUMNS</th>
<th>FORMAT</th>
<th>MEANING IN PROGRAM</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1 - 5</td>
<td>I</td>
<td>Number of loading systems</td>
</tr>
<tr>
<td></td>
<td>6 - 10</td>
<td>I</td>
<td>Number of title cards</td>
</tr>
<tr>
<td></td>
<td>11 - 15</td>
<td>I</td>
<td>Degrees of freedom existing in constrained structure (displacement direction numbers)</td>
</tr>
<tr>
<td>A</td>
<td>24 - 25</td>
<td>I</td>
<td>Material abbreviation</td>
</tr>
<tr>
<td></td>
<td>26 - 30</td>
<td>I</td>
<td>Material temperature</td>
</tr>
<tr>
<td></td>
<td>31 - 35</td>
<td>I</td>
<td>Modulus of elasticity</td>
</tr>
<tr>
<td></td>
<td>36 - 40</td>
<td>I</td>
<td>Poisson's ratio</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>Density (unit weight)</td>
</tr>
<tr>
<td></td>
<td>51 - 60</td>
<td>E</td>
<td>Coefficient of thermal expansion</td>
</tr>
<tr>
<td>B</td>
<td>1</td>
<td>I</td>
<td>1 (one identifies an orthotropic material)</td>
</tr>
<tr>
<td></td>
<td>2 - 5</td>
<td>A</td>
<td>Material abbreviation</td>
</tr>
<tr>
<td></td>
<td>6 - 20</td>
<td>E</td>
<td>Material temperature</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>Density (unit weight)</td>
</tr>
<tr>
<td>C1</td>
<td>1 - 10</td>
<td>E</td>
<td>E_{11} Members of E matrix</td>
</tr>
<tr>
<td></td>
<td>11 - 20</td>
<td>E</td>
<td>E_{12}</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>E_{13} as defined in Chapter III</td>
</tr>
<tr>
<td></td>
<td>31 - 40</td>
<td>E</td>
<td>E_{22}</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>E_{23}</td>
</tr>
<tr>
<td></td>
<td>51 - 60</td>
<td>E</td>
<td>E_{33}</td>
</tr>
<tr>
<td>C2</td>
<td>1 - 10</td>
<td>E</td>
<td>G_{12} Members of E matrix</td>
</tr>
<tr>
<td></td>
<td>11 - 20</td>
<td>E</td>
<td>G_{13}</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>G_{23}</td>
</tr>
<tr>
<td></td>
<td>31 - 40</td>
<td>E</td>
<td>Coefficient of thermal expansion X1 direction</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>Coefficient of thermal expansion X2 direction</td>
</tr>
<tr>
<td></td>
<td>51 - 60</td>
<td>E</td>
<td>Coefficient of thermal expansion X3 direction</td>
</tr>
</tbody>
</table>

Figure 5a
<table>
<thead>
<tr>
<th>CARD TYPE</th>
<th>COLUMNS</th>
<th>FORMAT</th>
<th>MEANING IN PROGRAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>1 - 10</td>
<td>I</td>
<td>Node number</td>
</tr>
<tr>
<td></td>
<td>11 - 20</td>
<td>E</td>
<td>X1 Coordinate</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>X2 Coordinate</td>
</tr>
<tr>
<td></td>
<td>31 - 40</td>
<td>E</td>
<td>X3 coordinate</td>
</tr>
<tr>
<td>E</td>
<td>1 - 5</td>
<td>I</td>
<td>Node number of constrained coordinate</td>
</tr>
<tr>
<td></td>
<td>6 - 10</td>
<td>I</td>
<td>Displacement direction number of constrained coordinate</td>
</tr>
<tr>
<td></td>
<td>11 - 15</td>
<td>I</td>
<td>Node number of generalized coordinate</td>
</tr>
<tr>
<td></td>
<td>16 - 20</td>
<td>I</td>
<td>Displacement direction number of generalized coordinate</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>Related coefficient</td>
</tr>
<tr>
<td>E</td>
<td>1 - 5</td>
<td>I</td>
<td>Node number of applied displacement</td>
</tr>
<tr>
<td></td>
<td>6 - 10</td>
<td>I</td>
<td>Displacement direction number of applied displacement</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>Applied displacement</td>
</tr>
<tr>
<td>G</td>
<td>1 - 5</td>
<td>I</td>
<td>Node number</td>
</tr>
<tr>
<td></td>
<td>6 - 10</td>
<td>I</td>
<td>Loading system number</td>
</tr>
<tr>
<td></td>
<td>11 - 20</td>
<td>E</td>
<td>Force in X1 direction</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>E</td>
<td>Force in X2 direction</td>
</tr>
<tr>
<td></td>
<td>31 - 40</td>
<td>E</td>
<td>Force in X3 direction</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>Moment about X1 axis</td>
</tr>
<tr>
<td></td>
<td>51 - 60</td>
<td>E</td>
<td>Moment about X2 axis</td>
</tr>
<tr>
<td></td>
<td>61 - 70</td>
<td>E</td>
<td>Moment about X3 axis</td>
</tr>
<tr>
<td>H</td>
<td>1 - 5</td>
<td>I</td>
<td>Associated node</td>
</tr>
<tr>
<td></td>
<td>6 - 10</td>
<td>I</td>
<td>Associated node</td>
</tr>
<tr>
<td></td>
<td>11 - 15</td>
<td>I</td>
<td>Associated node</td>
</tr>
<tr>
<td></td>
<td>15 - 20</td>
<td>I</td>
<td>Associated node</td>
</tr>
<tr>
<td></td>
<td>21 - 30</td>
<td>I</td>
<td>Loading system number</td>
</tr>
<tr>
<td></td>
<td>31 - 40</td>
<td>E</td>
<td>Distributed loading in X1 direction</td>
</tr>
<tr>
<td></td>
<td>41 - 50</td>
<td>E</td>
<td>Distributed loading in X2 direction</td>
</tr>
<tr>
<td></td>
<td>51 - 60</td>
<td>E</td>
<td>Distributed loading in X3 direction</td>
</tr>
</tbody>
</table>

Figure 5b
<table>
<thead>
<tr>
<th>CARD TYPE</th>
<th>COLUMNS</th>
<th>FORMAT</th>
<th>MEANING IN PROGRAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 - 10</td>
<td>I</td>
<td>Loading system number</td>
<td></td>
</tr>
<tr>
<td>11 - 20</td>
<td>E</td>
<td>Factor to be applied to concentrated loads</td>
<td></td>
</tr>
<tr>
<td>21 - 30</td>
<td>E</td>
<td>Factor to be applied to distributed loads</td>
<td></td>
</tr>
<tr>
<td>31 - 40</td>
<td>E</td>
<td>Factor to be applied to temperature loads</td>
<td></td>
</tr>
<tr>
<td>41 - 50</td>
<td>E</td>
<td>g level in X1 direction</td>
<td></td>
</tr>
<tr>
<td>51 - 60</td>
<td>E</td>
<td>g level in X2 direction</td>
<td></td>
</tr>
<tr>
<td>61 - 70</td>
<td>E</td>
<td>g level in X3 direction</td>
<td></td>
</tr>
<tr>
<td>1 - 5</td>
<td>I</td>
<td>Element identification number</td>
<td></td>
</tr>
<tr>
<td>7 - 10</td>
<td>A</td>
<td>Material abbreviation</td>
<td></td>
</tr>
<tr>
<td>11 - 20</td>
<td>E</td>
<td>Material temperature</td>
<td></td>
</tr>
<tr>
<td>21 - 35</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>26 - 30</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>31 - 35</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>36 - 40</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>41 - 45</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>46 - 50</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>51 - 55</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>56 - 60</td>
<td>I</td>
<td>Node number of element</td>
<td></td>
</tr>
<tr>
<td>65 - 70</td>
<td>I</td>
<td>Panel number</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5c
CHAPTER V

EXAMPLE PROBLEM

The following problem is used to demonstrate the system. Three loading systems are used in the example.

Material - Steel
Temperature - 70°F
Modulus of Elasticity - 30(10)^6
Poisson's ratio - .33
Displacement direction numbers - 1, 2, 6

Figure 6

The spring constant of elements 1 and 2 are 5.0 and 7.0 respectively. It is assumed that the stiffness of the beam elements (3 and 4) are much greater than those of the springs. This assumption allows a rather general demonstration of the transformation capabilities of the system.

The following lists explain the constraint conditions and loading systems that are applied to the structure of figure 6.
CONSTRANT CONDITIONS

Boundary Conditions

\begin{align*}
u_{11} &= 0.0 \\
u_{16} &= 0.0 \\
u_{22} &= 0.0 \\
u_{26} &= 0.0 \\
u_{42} &= 0.0
\end{align*}

Symmetry Conditions

\begin{align*}
u_{12} &= 1.00u_{32} \\
u_{21} &= 1.00u_{31} \\
u_{36} &= -0.01u_{32} \\
u_{41} &= 1.00u_{31} \\
u_{46} &= -0.01u_{32} \\
u_{51} &= 1.00u_{31} \\
u_{52} &= -1.00u_{32} \\
u_{56} &= -0.01u_{32}
\end{align*}

LOADING SYSTEMS

\begin{align*}
\text{number} & & \mathbf{P}_{51} & = 0.0 & \mathbf{P}_{52} & = 10.0 \\
1 & & \mathbf{P}_{51} & = 20.0 & \mathbf{P}_{52} & = 0.0 \\
2 & & \mathbf{P}_{51} & = -10.0 & \mathbf{P}_{52} & = 30.0
\end{align*}

A listing of the data cards required for this problem is seen in figure 7.

The system output is shown in figures 8a and 8b.
LISTING OF INPUT DATA

3 1 1 2 6
EXAMPLE PROBLEM FOR THESIS SPRING AND 2-D BEAM SYSTEM
STEL 70 29.0E06 .333 .290 6.2E-6

1 0.0 10.0
2 10.0 .0
3 10.0 10.0
4 110.0 10.0
5 210.0 10.0

1 2 3 2 1.0
1 6
5 2 3 2 -1.0
5 1 3 1 1.0
3 6 3 2 -.01
2 1 3 1 1.0
2 2
2 6
4 1 3 1 1.0
4 2
4 6 3 2 -.01
5 6 3 2 -.01
1 1

5 1 10.0
5 2 20.0
5 3 -10.0 30.0

1 1.
2 1.
3 1.

21 STEL 70.0 1 2 1
5.0
21 STEL 70.0 2 3 2
7.0
23 STEL 70.0 3 4 3
7.5 100.0 1000.0
23 STEL 70.0 4 5 4
7.5 100.0 1000.0

Figure 7
## Example Problem for Thesis: Spring and 2-C Beam System

### Material Properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Temp</th>
<th>Modulus of Elasticity</th>
<th>Poisson's Ratio</th>
<th>Density</th>
<th>Coefficients of Expansion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>T0°C</td>
<td>2.90 × 10^5 N/m²</td>
<td>3.13 × 10^-1</td>
<td>2.90 × 10^3 kg/m³</td>
<td>α1 = 6.20 × 10^-6, α2 = 6.20 × 10^-6</td>
</tr>
</tbody>
</table>

### Elastic Coefficients

<table>
<thead>
<tr>
<th>Material</th>
<th>Temp</th>
<th>E11</th>
<th>E12</th>
<th>E13</th>
<th>E22</th>
<th>E23</th>
<th>E33</th>
<th>G12</th>
<th>G13</th>
<th>G23</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>T0°C</td>
<td>3.44 × 10^8</td>
<td>-1.14 × 10^8</td>
<td>-1.14 × 10^8</td>
<td>3.44 × 10^8</td>
<td>-1.14 × 10^8</td>
<td>3.44 × 10^8</td>
<td>9.19 × 10^8</td>
<td>9.19 × 10^8</td>
<td>9.19 × 10^8</td>
</tr>
</tbody>
</table>

**Note:** 0.0 in E and Nu column indicates an orthotropic material.

### Nodal Coordinates

<table>
<thead>
<tr>
<th>Node Number</th>
<th>X Coordinate</th>
<th>Y Coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>1.00 × 10^4</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>1.00 × 10^4</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>1.00 × 10^4</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>2.10 × 10^4</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### Constraint Conditions

<table>
<thead>
<tr>
<th>Node Dir</th>
<th>Node Dir</th>
<th>Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>1.00 × 10^4</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>1.00 × 10^4</td>
</tr>
<tr>
<td>1</td>
<td>4</td>
<td>1.00 × 10^4</td>
</tr>
<tr>
<td>1</td>
<td>5</td>
<td>1.00 × 10^4</td>
</tr>
</tbody>
</table>

### Concentrated Loads

<table>
<thead>
<tr>
<th>Node Number</th>
<th>Loading System</th>
<th>X Force</th>
<th>Y Force</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>2.00 × 10^4</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>-1.00 × 10^4</td>
<td>3.00 × 10^4</td>
</tr>
</tbody>
</table>

### Distributed Loads

<table>
<thead>
<tr>
<th>Associated Nodes</th>
<th>Loading System</th>
<th>X Load</th>
<th>Y Load</th>
<th>Z Load</th>
</tr>
</thead>
</table>

### Components of Loading Systems

<table>
<thead>
<tr>
<th>Loading Number</th>
<th>Concentrated Loads</th>
<th>Distributed Loads</th>
<th>Temperature Changes</th>
<th>Acceleration and Gravity Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>AS DEFINED</td>
<td>1.000</td>
<td>AS DEFINED = 0.0</td>
<td>X1 DIRECTION = 0.0 DIRECTION = 0.0</td>
</tr>
<tr>
<td>2</td>
<td>AS DEFINED</td>
<td>1.000</td>
<td>AS DEFINED = 0.0</td>
<td>-6.0 DIRECTION = 0.0</td>
</tr>
<tr>
<td>3</td>
<td>AS DEFINED</td>
<td>1.000</td>
<td>AS DEFINED = 0.0</td>
<td>-6.0 DIRECTION = 0.0</td>
</tr>
</tbody>
</table>

Figure 8a
### Displacements

<table>
<thead>
<tr>
<th>Node Number</th>
<th>Loading System</th>
<th>$x_1$</th>
<th>Deflections $x_2$</th>
<th>$x_3$</th>
<th>Rotations $x_1$</th>
<th>$x_2$</th>
<th>$x_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>-0.</td>
<td>-1.4286E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>-0.</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>1</td>
<td>3</td>
<td>-0.</td>
<td>-4.2857E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>-0.</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>-4.0000E+00</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>-2.0000E+00</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>-0.</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.</td>
<td>-1.4286E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>1.4286E-02</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>-4.0000E+00</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>-2.0000E+00</td>
<td>-4.2857E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>1.4286E-02</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>-4.0000E+00</td>
<td>-0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>-2.0000E+00</td>
<td>-4.2857E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-0.</td>
<td>1.4286E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>1.4286E-02</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>4.0000E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>-2.0000E+00</td>
<td>4.2857E+00</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>4.2857E-02</td>
</tr>
</tbody>
</table>

Figure 8b
REFERENCES


User's Manual. Brigham Young University Computer Research Center
APPENDIX

ELEMENT IDENTIFICATION

An element is identified by a two digit number XY

where
\[ X = \text{number of nodes of element} \]
\[ Y = \text{degrees of freedom at each node} \]

<table>
<thead>
<tr>
<th>NAME</th>
<th>SKETCH</th>
<th>IDENTIFICATION NUMBER</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D Beam</td>
<td><img src="image" alt="2-D Beam Sketch" /></td>
<td>23</td>
</tr>
<tr>
<td>2-D Plate</td>
<td><img src="image" alt="2-D Plate Sketch" /></td>
<td>32</td>
</tr>
<tr>
<td>2-D Plate</td>
<td><img src="image" alt="2-D Plate Sketch" /></td>
<td>34</td>
</tr>
<tr>
<td>(bending)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3-D Beam</td>
<td><img src="image" alt="3-D Beam Sketch" /></td>
<td>26</td>
</tr>
</tbody>
</table>
Hexahedron

Truss member
ABSTRACT

A development of a generalized structural analysis system programmed for use on a general purpose digital computer (IBM 7040) is presented. The direct stiffness method is the analysis procedure used.

The system accepts stiffness matrices for various structural elements, sums these matrices, solves the resulting equilibrium equations, and outputs the nodal displacements. The individual element stiffness matrices are developed by user supplied subroutines. The requirements of these subroutines are set forth in the thesis.

The system proves very adequate in analyzing structures consisting of an assemblage of one or more different types structural elements. The displacement transformation capabilities allow a complete treatment of the boundary and symmetry conditions placed on the structure.
Approved:

Henry N. Christian
Chairman, Advisory Committee

Kenneth W. Karren
Member, Advisory Committee

Cliff L. Boston
Chairman, Major Department