PLANE FRAME ELEMENT ADDITION TO THE MESS FINITE ELEMENT PROGRAM

by

Ronald H. Thompson, Jr.

Thesis submitted to the Faculty of the

Virginia Polytechnic Institute and State University

in partial fulfillment of the requirements for the degree of

Master of Science

in

Mechanical Engineering

APPROVED:

Dr. Charles E. Knight, Chairman

Dr. H. H. Mabie

Dr. Larry D. Mitchell

May, 1985

Blacksburg, Virginia

PLANE FRAME ELEMENT ADDITION TO THE MESS FINITE ELEMENT PROGRAM

by

Ronald H. Thompson, Jr.

Dr. Charles E. Knight, Chairman

Mechanical Engineering

(ABSTRACT)

A plane frame element based on linear, elastic theory is developed and implemented into the MESS finite element program. Post-processed results include nodal displacements, end reactions, maximum tensile and average shear stress, and a deformed geometry plot. The element is tested for accuracy relative to simple beam theory and by comparison with results generated using another finite element program. In both cases agreement to within 6 significant figures was achieved.

Because the intended use is educational, a survey of its benefit as a design aid in undergraduate instruction is included. These benefits are based on test cases from senior design class projects. Results generated using analysis techniques presently available are contrasted with those using the plane frame element. Students' work that was examined contained mistakes resulting from large amounts of hand calculations. Conversely, results generated using the finite element method proved to be easily obtained and to have a higher degree of accuracy. A recommendation for further improvements in program capability is provided at the end of the study.

ACKNOWLEDGEMENTS

The author first wishes to express his thanks to God for seeing him through to the completion of this thesis. At times, things looked pretty grim, but when despair reached the breaking point, His Presence made the difference.

Almost as importantly, unquestioned love from a very dear family made a personal crisis bearable, thus enabling progress on this work to continue. Special words of appreciation must go to Dr. and Mrs. Ron Thompson. Faith may doubt, confidence can waver, but Love conquers all.

The members of Pearisburg Baptist Church continually supplied a strong emotional base of support. In particular, Joy Smith and Mike Journell gave assistance, words of encouragement, and shoulders to cry on when they were needed the most. There is no giving or taking between such friends, for we are all One in the Spirit.

The technical guidance of Dr. Charles E. Knight, Dr. Larry D. Mitchell, and Dr. H. H. Mabie certainly increased the quality and integrity of this work. If there were no committees to guide hapless students through the innumerable pitfalls associated with writing a thesis, how would one ever get finished?

Final mention will be made of one additional person, Karina Ruiz. As always, her ultimate desire was to help, assist, and encourage. The author wishes to express belief in her motives and love. It is finished.

TABLE OF CONTENTS

1.0 INTRODUCTION

The ability to correctly model a physical system using the finite element method is often limited by the types of elements available for use. The modeling of any continuum is limited in accuracy by the basic assumptions made regarding element formulation, i.e., the elemental degrees of freedom and the manner in which the element reacts under load. These are all related to the element stiffness matrix formulation.

The most basic finite element commonly used is the truss element, a one-dimensional bar able to withstand axial forces and deflections only. Other element types slightly more complex yet still easily formulated include the beam, plane frame, the 2-D isoparametric, and the 2-D axisymmetric element. These are often found in basic finite element programs; hence, they are often used for instructional purposes.

An existing finite element program, MESS (Mechanical Engineer'S Stress program), is used for undergraduate instruction in the Computer-Aided Engineering Design lab of the Mechanical Engineering Department of Virgina Polytechnic Institute and State University. The program is based on the program described by Bathe [1].[1] The element library has been expanded to include a 2-D isoparametric and a 2-D axisymmetric element, as well as the original truss element. An interactive, menu-driven data preprocessor has also been developed and is presently in use.

---

[1] Numbers in square brackets refer to similarly numbered references in the List of References.

Postprocessing capabilities currently employed include model plots, element stresses, and graphical plotting of deformed geometry.

This thesis involves the addition of a plane frame beam element to the element library of MESS. Modification of the preprocessor will be necesary such that recognition of an additional element is made and input is prompted accordingly. An additional subroutine containing formulation of the plane frame element will be added to the main program. Existing routines that call the element subroutine, plot deformed geometry, or display results will be changed as necessary.

The element to be developed and implemented in this project will follow the following guidelines: As closely as possible, the element subroutine will parallel the formulation used in the other element routines. This is especially pertinent for the truss element, as the 3-D space truss and 2-D plane frame elements have stiffness matrices of the same dimensions. Thus, ease of interaction with the assembly and solution routines presently in use will be facilitated, so nodal displacements can be computed using existing coding. Inter-element deflections will be calculated using the element shape functions and used for plotting the deformed structural geometry. End reactions (tensile, shear, and moment) will be calculated at each node for all elements, as well as the element stresses (maximum normal and average shear). An output file will be generated which contains all element input data as well as numerical postprocessed results.

Finally, the plane frame element will be evaluated for design use and accuracy. This analysis will be done in three areas, each of which encompasses a different area of application. First, accuracy relative to results predicted by simple beam theory will be analyzed. Secondly, an indeterminant frame structure modeled and analyzed by Meyers [2] will be modeled and the results compared. Finally, structurally optimized frames designed by undergraduate students in ME 4070 Mechanical Design II will be modeled and evaluated. The results will be compared with those predicted by classical analysis.

2.0 LITERATURE SURVEY

Because direct physical argument can successfully formulate simple structural components, provided the material is assumed to be linear and elastic, the modern use of finite elements originally began in the field of matrix structural analysis [3]. The need for analysis of complex structures consisting of one-dimensional elements combined with the comparative ease of formulation lead to rapid increases in finite element use as digital computing techniques evolved in the early 1950's. This resulted in an early formulation and use of the plane frame element.

The plane frame element is a one-dimensional bar element capable of withstanding in-plane forces and torques, as shown in Figure 1. The displacement method is presented by Meyers [2] as a valid method by which to mathematically describe the reaction of the element to load inputs. By using fundamental strength of material concepts from simple beam theory, the element stiffness matrices are derived for the truss and beam elements. The plane frame element is described as simply a superposition of these two elements, this is a valid assumption as long as the interaction between axial and bending modes of member deflection remain small [2]. Similar derivation and results are listed by Kanchi [4].

Since the 1950's other more mathematically based methods of arriving at element stiffness matrices have been developed. As a rule they are more mathematically intensive, but when appropriate assumptions are made, they yield identical stiffness matricies for the truss and beam elements as those derived using simple beam theory. Hence, the limitations

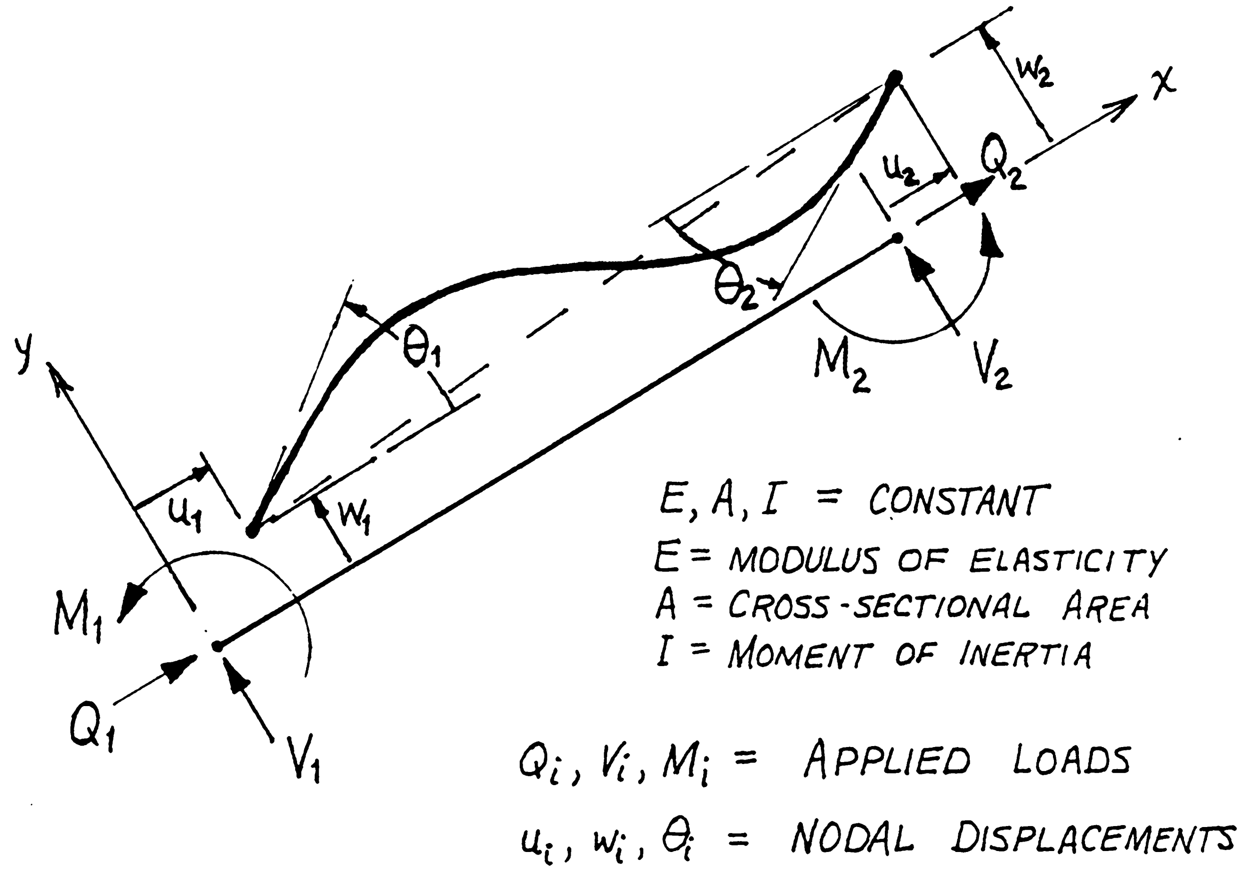


FIGURE 1: PLANE FRAME ELEMENT WITH APPLIED LOADS AND RESULTING NODAL DISPLACEMENTS (AFTER [2])

associated with the most basic formulations of one-dimensional elements are more visible.

The classic finite element method of deriving element equations consists of numerically representing the variational formulation of the governing differential equation. This is done in detail for the beam element by Reddy [5]. Simplifications are made related to linear theory, such as assuming the modulus of elasticity, moment of inertia, and cross-sectional area are constant across the element. When the simplest polynomial that fulfills continuity and end condition requirements is chosen as a displacement shape function, the resulting element stiffness matrix is identical to that obtained using the displacement method. Stiffness matrices for the simple beam, plane frame, space frame, plane truss, space truss, and more complex plate elements are given by Rao [6].

Although certainly a useful tool, the plane frame formulation found using linear elastic assumptions has definite limitations, both in accuracy and application. These are primarily related to material and geometric nonlinearities and the possibility of nonlinear boundary conditions. The sources briefly quoted below provide a more comprehensive understanding of the assumptions involved and the resulting limitations of linear theory as applied to the plane frame element.

Interaction between bending and axial deflection modes is a nonlinear problem that can contribute to inaccuracies that are associated with the plane frame element. Because the deflected slope of the element is assumed small when compared with unity in linear theory, the governing differential equation becomes uncoupled and is reducible to results obtained earlier [5]. However, when not too small, the coupled set of equations must be solved and the stiffness matrix developed accordingly. Butler [7] describes implementation of Green's strain tensor as a way of achieving a nonlinear relationship between strain and displacement.

Geometric nonlinearities can also be introduced by large deflections within the structure that significantly alter the way in which loads are carried. Cook [8] gives a general look at techniques available for treating inaccuracies in finite element models resulting from large deflections. A study of geometric nonlinear behavior for the plane frame is done by Phan [9].

When a material's yield point is reached, plasticity becomes a factor in deflection analysis. Hence, under circumstances of very high stress and low safety factors, a nonlinear stress-strain relationship may result. Two major techniques of solving such material nonlinearity are covered by Cook [8]. The first method involves an incremental procedure which relates increments of stress to increments of strain; the second method is a direct iteration method and relates total stress to total strain. Both attempt approximation of the changing stress-strain curve.

Other limitations of the linear theory plane frame element are listed below:

1. Two-dimensional problems only.

2. The in-plane beam thickness must be small compared to length so shear deformation is negligible.

3. Deflections are small relative to the problem's dimensions.

Only the development of the element stiffness matrix and limitations due to linear formulation have been previously discussed. Also of interest is the way in which the element is implemented, the global structure matrix assembled, the forces applied, and the deflections calculated. Because this thesis dealt with an existing finite element program, it was desired to use the numerical solution methods employed in the existing program. Hence, only the procedures used in it were of interest. They are developed and explained in detail by Bathe and Wilson [1].

The first step involved in the solution phase of the finite element method is calculating nodal displacements, while reaction forces and element stresses are found in the second phase of analysis. Methods of postprocessing results are given in [5] specifically for the beam element and in [2] for the plane frame element. Both of these references find end reactions by reforming each element stiffness matrix and multiplying by the corresponding nodal displacements previously computed. General guidelines for developing a postprocessor are presented by Ford [10]. Methods for implementing interactive graphic capability and the benefits of such postprocessing are also discussed.

3.0 THEORY

3.1 GENERAL DESCRIPTION OF THE FINITE ELEMENT METHOD

In the finite element method, the actual body of matter is represented as an assemblage of subdivisions called finite elements, interconnected at specified joints called nodal points [6]. When applied to structural problems, differential equations that govern deflections inside the continuum are approximated by simple algebraic interpolation functions. These functions are defined in terms of the values of nodal displacements. Generally speaking, the resulting set of equations are then arranged such that the relationship between force input and displacement can be determined.

The solution of a problem by the finite element method follows an orderly, step-by-step process. With reference to structural problems, the following procedure is used:

1. Discretization of the structure

   The structure is appropriately divided into elements. The number, type, and arrangement must be selected.

2. Selection of proper interpolation functions

   The functions chosen must approximate the unknown solution, be computationally simple, and meet end condition and continuity requirements.

3. Derivation of element stiffness matrices

   From the assumed displacement model, the stiffness matrix $[K^e]$ and load vector $\{P^e\}$ are derived using equilibrium conditions or variational principles.

4. Assemblage of element equations to obtain the overall equilibrium equations

   Since the structure is composed of many finite elements, individual element stiffness and load vectors are assembled and the overall equilibrium equations formulated as

   $$[K]\{D\} = \{P\};$$

   where $[K]$ = assembled stiffness matrix

   $\{D\}$ = vector for nodal displacements

   $\{P\}$ = nodal force vector.

5. Solution for the unknown nodal displacements

   The overall equilibrium equations are modified to account for the boundary conditions of the problem. Nodal displacements are then solved using numerical techniques.

6. Computation of element strains and stresses

   From the known nodal displacements, element strains and stresses can be solved using the applicable equations of solid mechanics.

3.2 STIFFNESS MATRIX DERIVATION FOR PLANE FRAME ELEMENT

This project dealt with the development and implementation of a plane frame element; thus, steps #2, #3, and #6 as outlined above were necessary for implementation in the existing code. As described earlier, this element is to be based on linear theory and represented as the superposition of the plane beam and plane truss elements. The formulation also neglects shear deformation. In cases where this is significant, as in short beams with large shear forces, accuracy is limited. The same notation as used in Figure 1 is used in the following development.

3.2.1 BEAM ELEMENT DERIVATION

The finite element method is built upon establishing a relationship between force and displacement. As outlined in Section 3.1, this consists first of deriving individual element stiffness relations in the form

$$[K^e]\{D^e\} = \{P^e\} \tag{3.1}$$

where $[K^e]$ = element stiffness matrix

$\{D^e\}$ = nodal displacement vector

$\{P^e\}$ = nodal force vector

The element equation is then added to the entire structural matrix to form the overall equilibrium equation. The present task is to find $[K^e]$ for the beam element.

The fourth order differential equation

$$EI \frac{d^4w}{dx^4} = 0. \tag{3.2}$$

is the governing equation for the bending of beams, where E is the modulus of elasticity, I is the area moment of inertia, w(x) is the transverse displacement (+ up), and E and I are constant across x. The transverse deflection must satisfy Eq. 3.2 as well as the 4 associated boundary conditions. For a beam, these conditions relate to displacements at each end: w and dw/dx. Hence, Eq. 3.1 will take the following form:

$$\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} \\ K_{21} & K_{22} & K_{23} & K_{24} \\ K_{31} & K_{32} & K_{33} & K_{34} \\ K_{41} & K_{42} & K_{43} & K_{44} \end{bmatrix} \begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \end{Bmatrix} \tag{3.3}$$

where K_{ij} = stiffness matrix coefficients

θ_i = dw/dx = slope at ends (+ counter-clockwise)

V_i = shear force at ends (+ up)

M_i = moment at ends (+ counter-clockwise)

To determine the values of the stiffness matrix coefficients, the unit displacement method will be used. In this approach a unit value of each member-end displacement is applied in separate steps as shown in Figure 2. For each of the 4 cases, Eq. 3.2 is solved for w(x) given the 4 sets

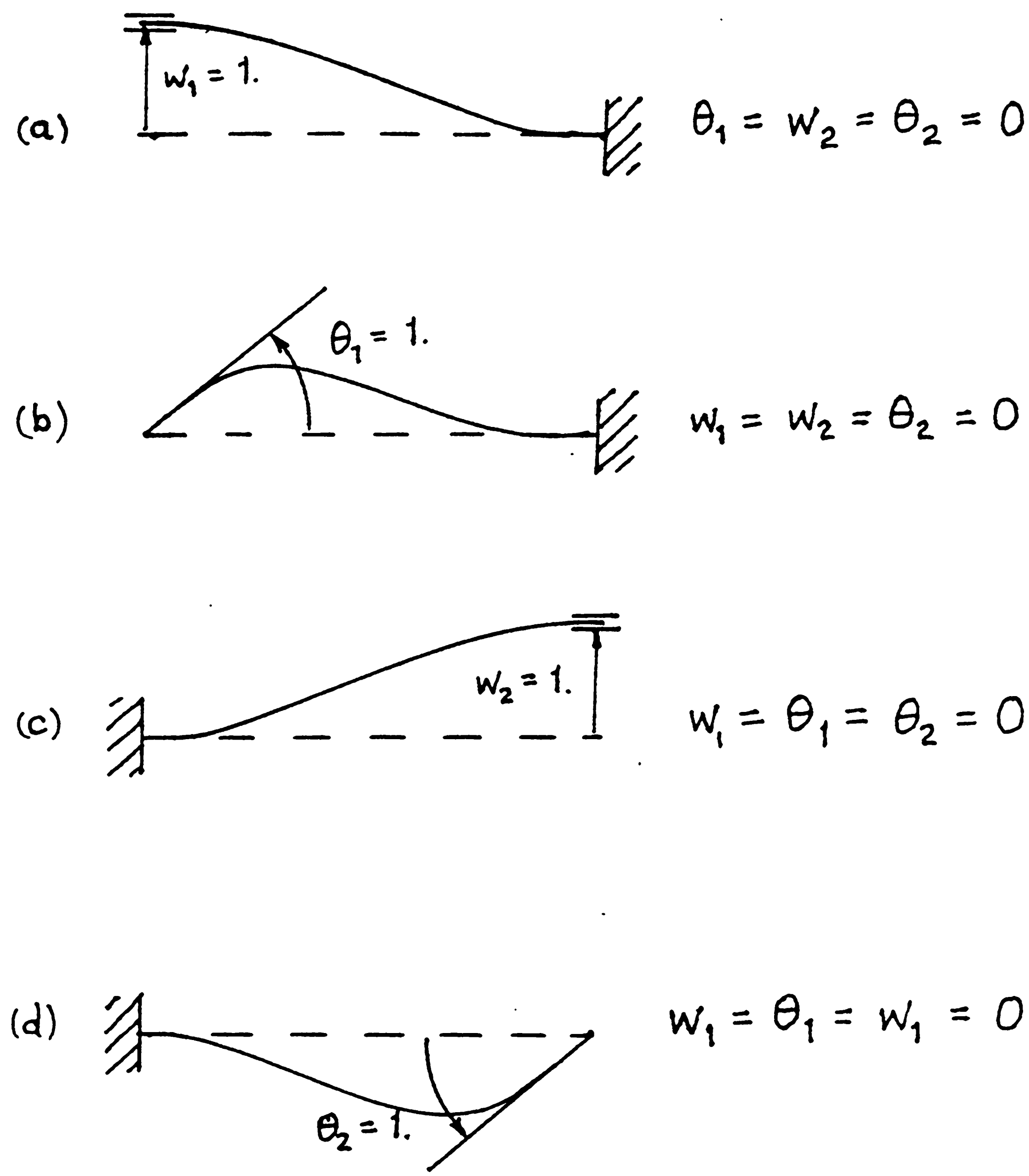


FIGURE 2: UNIT DISPLACEMENTS FOR BEAM ELEMENT

of boundary conditions. Resulting end reactions are calculated using the following relationships between w(x) and V and M [5]:

$$V(x) = \frac{d^3w}{dx^3} EI \tag{3.4}$$

$$M(x) = -\frac{d^2w}{dx^2} EI \tag{3.5}$$

Because a counter-clockwise moment is defined as positive, the negative sign in Eq. 3.5 is necessary to assure positive curvature results from a positive moment.

The solution of Eq. 3.2 for Case A as shown in Figure 2 proceeds as follows:

$$d^4w/d^4x = 0$$

with boundary conditions (a) $w(0)=1$ (b) $w'(0)=0$

(c) $w(L)=0$ (d) $w'(L)=0$

where L = beam length.

(A) Integrate three times:

$$w(x) = c_1x^3 + c_2x^2 + c_3x + c_4$$

(B) Apply boundary conditions:

From (a) and (b), $c_3 = 0$, $c_4 = 0$

From (c) and (d), $w(L) = c_1L^3+c_2L^2+1 = 0$

$w'(L) = 3c_1$

$$\text{so} \quad c_1 = 2/L^3, \quad c_2 = -3/L^2$$

Hence,

$$w(x) = \left(\frac{2}{L^3}\right)x^3 - \left(\frac{2}{L^2}\right)x^2 + 1 \tag{3.6}$$

(C) Find end reactions

$$V_1 = V(0) = \frac{d^3w}{dx^3}EI\bigg|_{x=0} = \frac{12}{L^3}EI$$

$$M_1 = M(0) = -\frac{d^2w}{dx^2}EI\bigg|_{x=0} = [-6(\frac{2}{L^3})(0)+2(\frac{3}{L^2})]EI = \frac{6}{L^2}EI$$

Once given V_1 and M_1, V_2 and M_2 can be obtained from statics:

$\Sigma F=0$ leads to $V_2 = -V_1 = -(12/L^3)EI$

$\Sigma M=0$ @ $x=0$ leads to $M_2 = -M_1 + LV_2$

$=(6/L^2)EI$

Hence, the first row of the stiffness matrix, K_{i1} is known:

$K_{11} = 12EI/L^3 \qquad K_{21} = 6EI/L^2$

$K_{31} = -12EI/L^3 \qquad K_{41} = 6EI/L^2$

This procedure is followed for each of the three remaining conditions. An additional row of the stiffness matrix is found for each set of boundary conditions. The resulting matrix is listed below:

$$[K^e] = \frac{12\,EI}{L}\begin{bmatrix} (6/L^2) & (3/L) & (-6/L^2) & (3/L) \\ (3/L) & 2 & (-3/L) & 1 \\ (-6/L^2) & (-3/L) & (6/L^2) & (-3/L) \\ (3/L) & 1 & (-3/L) & 2 \end{bmatrix} \tag{3.7}$$

Equation 3.6 represents the beam displacement as a function of x given that $w_1 = 1$ and all end displacements = 0. For each of the other 3 sets of boundary conditions shown in Figure 2, a unique displacement function results. These functions are commonly known as element shape functions and are useful in approximating inter-element displacements given nodal displacements. Each of the four interpolation functions corresponding to cases (A)-(D) respectively are listed below:

$$\begin{aligned} N_1 &= 2\left(\frac{x}{L}\right)^3 - 3\left(\frac{x}{L}\right)^2 + 1 \\ N_2 &= -x\left(-\frac{x}{L} + 1\right)^2 \\ N_3 &= -2\left(\frac{x}{L}\right)^3 + 3\left(\frac{x}{L}\right)^2 \\ N_4 &= -x\left[\left(\frac{x}{L}\right)^2 - \frac{x}{L}\right] \end{aligned} \tag{3.8}$$

For a combination of non-zero end displacements, inter-element displacements are expressed as the superposition of each shape function multiplied by the corresponding displacement [5]. Thus,

$$w(x) = N_1 w_1 + N_2 \theta_1 + N_3 w_2 + N_4 \theta_2 \tag{3.9}$$

3.2.2 TRUSS-ELEMENT STIFFNESS MATRIX

The second-order differential equation

$$- EA \frac{d^2u}{d^2x} = 0 \tag{3.10}$$

is the governing equation for the axial deflection of a bar, where u(x) is the longitudinal displacement, A is the cross-sectional area, and E is the modulus of elasticity. If the unit displacement method is applied, two sets of boundary conditions result: u = 1 @ end 1 and u = 0 @ end 2, or u = 0 @ end 1 and u = 1 @ end 2. When Eq. 3.10 is solved using these two sets of boundary conditions, end reactions and stiffness matrix coefficients can be found using the same procedure as in the previous section. When put in the form of Eq. 3.1, the following equation results:

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} \tag{3.11}$$

where $u_1 = u(0)$ $\quad$ $u_2 = u(L)$

Q_1 = force @ x=0 $\quad$ Q_1 = force @ x=L

3.2.3 SUPERPOSITION OF TRUSS AND BEAM ELEMENTS

Because the truss and beam elements are geometrically similar, the two can be combined into one element with three degrees-of-freedom per node: longitudinal deflection u, transverse deflection w, and rotational deflection θ. When the properties A, E, and I are constant within the element and deflections are assumed small, the following equation results from the superposition of Eq. 3.7 and Eq. 3.11:

$$
\frac{2EI}{L}\begin{bmatrix} A/2I & 0 & 0 & -A/2I & 0 & 0 \\ & 6/L^2 & 3/L & 0 & -6/L^2 & 3/L \\ & & 2 & 0 & & 1 \\ & & & A/2I & 0 & 0 \\ & (\text{sym.}) & & & 6/L^2 & -3/L \\ & & & & & 2 \end{bmatrix} \begin{Bmatrix} u_1 \\ w_1 \\ \theta_1 \\ u_2 \\ w_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ V_1 \\ M_1 \\ Q_2 \\ V_2 \\ M_2 \end{Bmatrix} \tag{3.12}
$$

3.3 GLOBAL PLANE FRAME STIFFNESS MATRIX

In most problems of practical interest, it is necessary to find the stiffness matrix of an element relative to some common, global coordinate system. This is accomplished by using a rotation transformation. For a frame element oriented at an angle ϕ from the positive x axis, the components of displacement are changed from the element coordinates x' and y' to global coordinates x and y. As shown in Figure 3, the rotation of coordinate axes changes the reaction force and displacement directions. The modification of the stiffness matrix is related to both of these. Thus, it is necessary to derive the relationship between global forces and element forces as well the relationship between global displacements and element displacements.

From Figure 3 it can be shown that

$$
[\lambda]\{P^e\} = \{P\} \tag{3.13}
$$

where $\{P\} = [\ F_{x1}\ F_{y1}\ M_1\ F_{x2}\ F_{y2}\ M_2]^T$ = force vector in global coordinants

$\{P^e\} = [\ Q_1\ V_1\ M_1\ Q_2\ V_2\ M_2]^T$ = force vector in element coordinates

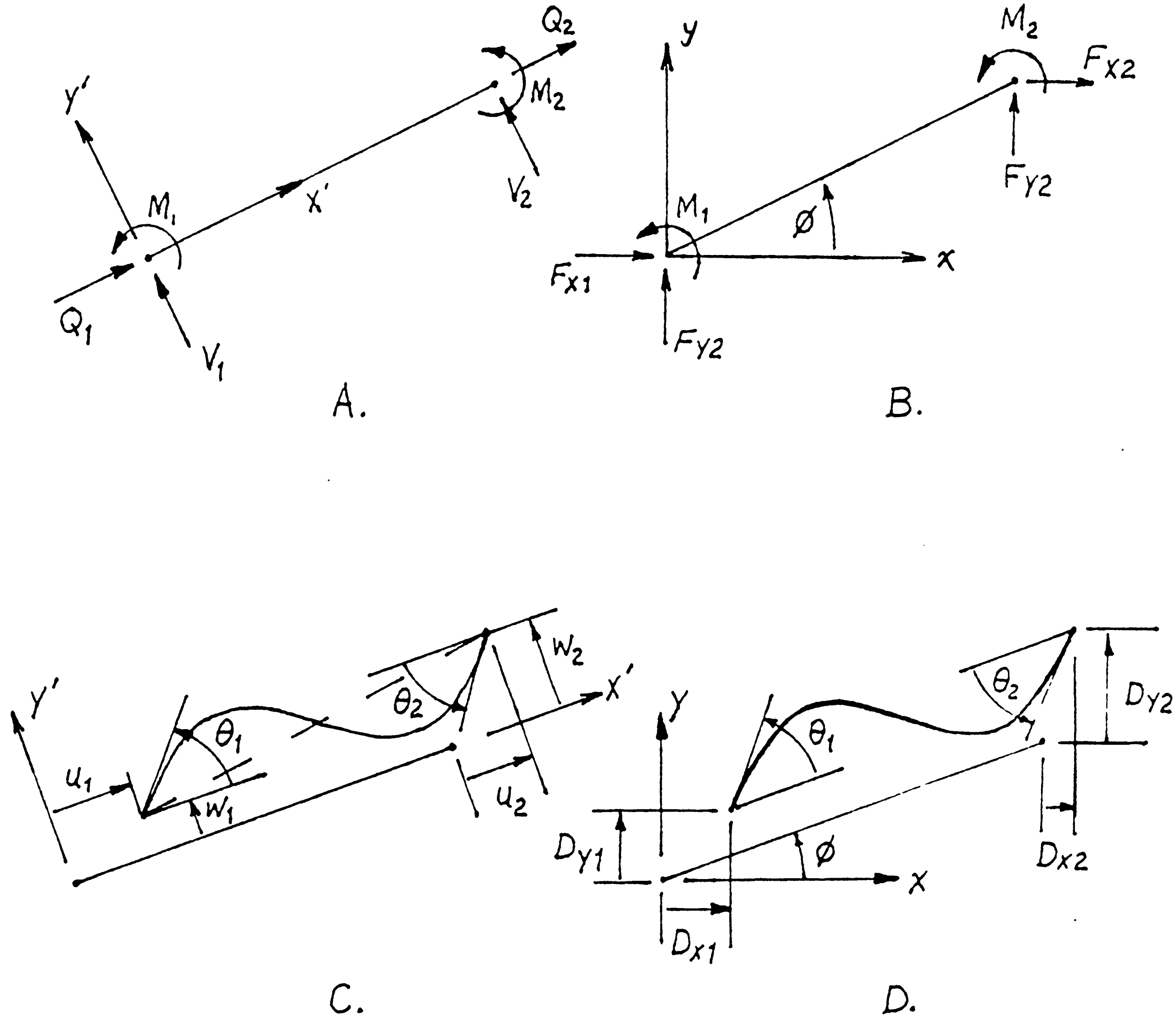


FIGURE 3: MEMBER END FORCES AND DISPLACEMENTS

A. FORCES, LOCAL COORDINATES

B. FORCES, GLOBAL COORDINATES

C. DISPLACEMENTS, LOCAL COORDINATES

D. DISPLACEMENTS, GLOBAL COORDINATES

$$[\lambda] = \begin{bmatrix} \cos\phi & -\sin\phi & 0 & 0 & 0 & 0 \\ \sin\phi & \cos\phi & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\phi & -\sin\phi & 0 \\ 0 & 0 & 0 & \sin\phi & \cos\phi & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{3.14}$$

Also from Figure 3 the relationship between elemental and global displacements is:

$$[\lambda]^T\{D\} = \{D^e\} \tag{3.15}$$

where $\{D\} = [D_{x1}\ D_{y1}\ \theta_1\ D_{x2}\ D_{y2}\ \theta_2]^T$ = displacement vector in global coordinates

$\{D^e\} = [u_1\ w_1\ \theta_1\ u_2\ w_2\ \theta_2]^T$ = displacement vector in element coordinates

This particular equation form is desired for later substitution such that a matrix equation is derived that relates global displacements, the element stiffness matrix, and global forces.

Recall now Eq. 3.1: $[K^e]\{D^e\} = \{P^e\}$. By substituting this into Eq. 3.13, the following relationship is obtained:

$$[\lambda][K^e]\{D^e\} = \{P\}$$

The substitution of Eq. 3.15 into the above equation yields

$$[\lambda][K^e][\lambda]^T\{D\} = \{P\} \tag{3.16}$$

After the matrix multiplication of Eq. 3.16 is done, the following equation results:

$$\frac{2EI}{L}\begin{bmatrix} c_1 & c_2 & c_3 & -c_1 & -c_2 & c_3 \\ & c_5 & c_4 & -c_2 & -c_5 & c_4 \\ & & 2 & -c_3 & -c_4 & 1 \\ & \text{(sym.)} & & c_1 & c_2 & -c_3 \\ & & & & c_5 & -c_4 \\ & & & & & 2 \end{bmatrix} \begin{Bmatrix} D_{x1} \\ D_{y1} \\ \theta_1 \\ D_{x2} \\ D_{y2} \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ M_1 \\ F_{x2} \\ F_{y2} \\ M_2 \end{Bmatrix} \qquad (3.17)$$

where

$$c_1 = \frac{A \cos^2\phi}{2\ I} + \frac{6 \sin^2\phi}{L^2}$$

$$c_2 = \frac{A \sin\phi \cos\phi}{2\ I} - \frac{6 \sin\phi \cos\phi}{L^2}$$

$$c_3 = -\frac{3 \sin\phi}{L}$$

$$c_4 = \frac{3 \cos\phi}{L}$$

$$c_5 = \frac{A \sin^2\phi}{2\ I} + \frac{6 \cos^2\phi}{L^2}$$

The stiffness matrix in this equation is in a form easily applied to a variety of plane frame element sizes, positions, and materials. Thus, a structure consisting of many elements can be correctly modeled by assembling individual element stiffness matrices. This assemblage is accomplished by superimposing the element stiffness matrices such that a large structural matrix is obtained. The resulting matrix has dimension equal to the total number of degrees of freedom in the structure. The matrix equation is given in step #4 of Section 3.1.

3.4 CALCULATION OF END LOADS FROM DISPLACEMENTS

After all elements are assembled and the overall equilibrium equations are determined, the overall load vector is applied. Nodal displacements are solved for using numerical techniques as outlined in step #5 of Section 3.1. The final step consists of finding element end reactions, stresses, and strains by using the known nodal displacements.

The displacements calculated are given in global coordinates; hence, by directly substituting the corresponding values into Eq. 3.17, the element end reactions are determined. However, the resultant forces are also in global coordinates. In order to find element stresses, it is necessary to transform these reactions back to element coordinates. From Figure 3, the relationship between forces in the element coordinate system and global coordinate system can also be expressed as $[\lambda]^T\{P\} = \{P^e\}$. Once end reactions are calculated using this equation, stress computation is performed using classic strength-of-materials equations relating to linear theory. These equations and the method of application are covered in Chapter 4.

4.0 IMPLEMENTATION

As mentioned earlier, this project involved work on an existing software package at Virginia Polytechnic Institute known as MESS (Mechanical Engineer'S Stress program). Program logic and computational techniques are presented by Bathe and Wilson [1].

The core program developed in [1] contained only a space truss element; however, the code was written such that additional elements could easily be added, provided similar algorithms were used to solve for the element stiffness matrices. Accordingly, the program has been in a continual state of upgrading since its introduction at Virginia Polytechnic Institute. Two additional elements, the 2-D, 4-noded isoparametric and axisymmetric elements, were added, as well interactive graphic capability consisting of plotting the element mesh and the deformed mesh. In 1982 Jara-Almonte [11] added a variety of post-processing abilities relative to the axisymmetric and isoparametric elements. His work in implementing these capabilites and the alterations made to existing code is described in his thesis.

The basic requirements of this project consisted of the development of a plane frame element subroutine that would operate in a manner very similar to the existing truss element routine. As both elements have the same geometry, the same connectivity array is used, and the stiffness matrices are of the same dimension. Thus, the formation of the element stiffness matrices and their individual contributions to the global stiffness matrix was identical to the original method used in [11]. However, material properties, stress computation, and deformed mesh plotting would necessarily be different.

The plane frame element routine performs four primary functions:

1. It reads in all element material properties: i.e., Young's modulus, cross-sectional area, moment of inertia, and distance from the centroidal axis to the outer fiber.

2. It loops over all elements to form the connectivity array, calls a subroutine to update column heights, and calls a subroutine to plot the mesh geometry.

3. It loops over all elements to calculate the corresponding stiffness matrix using Eq. 3.17, and calls the subroutine to add the result to the global stiffness matrix.

4. It loops over all elements reforming each stiffness matrix , multiplying it by the corresponding nodal displacement vector. Deformed geometry of the element is found using the element shape functions given by Eq. 3.8, and a plotting routine is called to plot the structural deformed mesh. Stresses are

calculated using the element properties and end reactions in element coordinates. Input data, node displacements, nodal reactions, and stresses are written to an output file.

In order to accomplish the above functions, some modifications to existing code were necessary. For the undeformed mesh geometry, the same routine was used as for the truss element. However, an entirely new routine was necessary for the deformed plot. Conditionals relating to element type were used to channel the problem to the correct subroutines. Other modifications included the expansion of dimension statements to handle the increased number of material properties and the arrays necessary to plot inter-element deflections.

Stress computation was based on classic strength of materials equations applied to elastic beams of constant properties, with element end reactions being the only force inputs. Element orientation and end reactions are shown in Figure 3. The maximum normal stress σ_x is found at both nodes of each element, and is represented as the combination of normal bending stress at the outer fiber and the stress due to axial force. The equation used to calculate this is given below:

$$\sigma_x = \frac{F_x}{A} \pm \frac{M\,c}{I}$$

where F_x = axial force

M = end moment

A = cross-sectional area

c = distance to outer fiber from centroidal axis

I = area moment of inertia

Shear stress is also of interest in beam analysis. However, it varies with respect to the cross-sectional area and is not easily formulated for a variety of shapes. Rather than calculate average shear stress or some general value that could be misinterpreted by undergraduate students, shear force is given as output.

In order to plot the deformed element shapes using the shape functions, transverse and longitudinal deflection was first calculated in element coordinates at interpolation points, then transformed to global coordinates. A total of 20 interpolation points were used per element: one at each endpoint and 18 evenly spaced points between. The displacement values were scaled relative to the maximum displacement over all elements such that the largest displacement will be seen as approximately 1/2" by the program user. The values were then passed to a routine which plots the structure beginning with element one and continuing to the highest element number.

Several modifications were necessary in the preprocessor. These primarily related to input data interpretation and data file formatting. The processor is formatted for very easy, almost bug-free use; hence, nodal, element, force, and material properties input is formatted relative to the element type being used for the analysis. Input prompts and data interpretation loops relative to the plane frame element were added. Conditional statements relating to element type were used to control which prompts were written to the screen and which interpretive loops were executed. The Appendix contains sample output files from MESS using the plane frame element subroutine BEAM relating to the case studies covered in Chapter 5.

5.0 ACCURACY STUDY.

5.1 SIMPLE BEAM THEORY TESTS

Because element formulation was based on linear, elastic theory, displacements predicted by simple beam theory should agree with program output. As a check, several beams having various end conditions and loadings were modeled. Formulas for calculating displacements and end reactions derived using classical strength of materials equations are given by Shigley [12]. The four cases listed below were modeled:

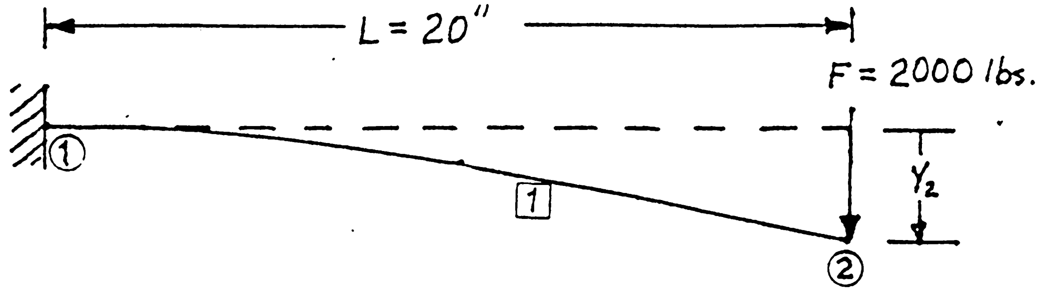
A. Fixed - free, end force

B. simply supported, center force

C. Simply supported, center moment

D. Fixed - fixed, center force

For each condition, displacements and end reactions predicted by classical analysis agreed to 6 digits precision with that predicted by the program. This was as expected, because with simple finite element models consisting of only a few elements, roundoff errors involved in solving the matrix equation are negligible. Parameters used in modeling cases A and B are given in Figure 4, as is the deformed geometry and displacements generated by MESS. Corresponding displacements predicted

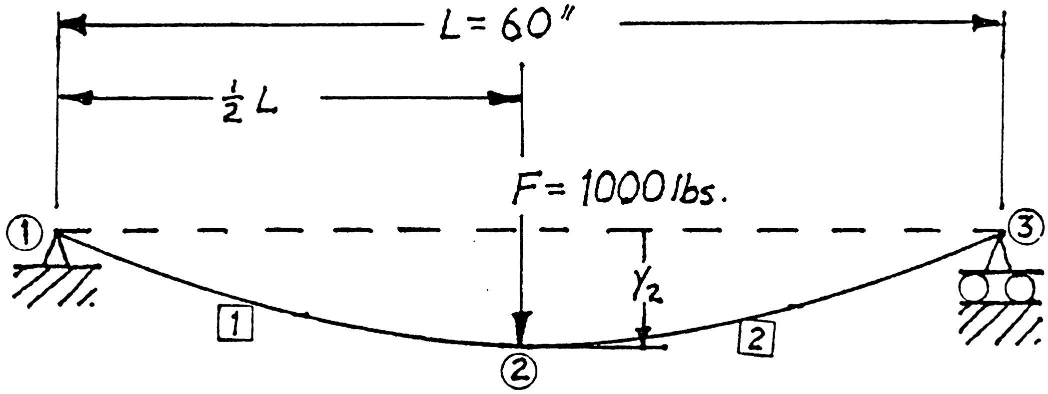


$E = 30\,(10^6)$ PSI; $A = 4.0$ IN.^{2}; $I = 1.33$ IN^4

THEORETICAL: $Y_2 = -0.133675$"

FROM MESS: $Y_2 = -0.133668$"

CASE A



$E = 30\,(10^6)$ PSI; $A = 4.0$ IN^2; $I = 1.33$ IN^4

THEORETICAL: $Y_2 = -0.1127820$"

FROM MESS: $Y_2 = -0.112782$"

CASE B

FIGURE 4: SIMPLE BEAM THEORY

A. FIXED-FREE, END FORCE

B. SIMPLY SUPPORTED, CENTER FORCE

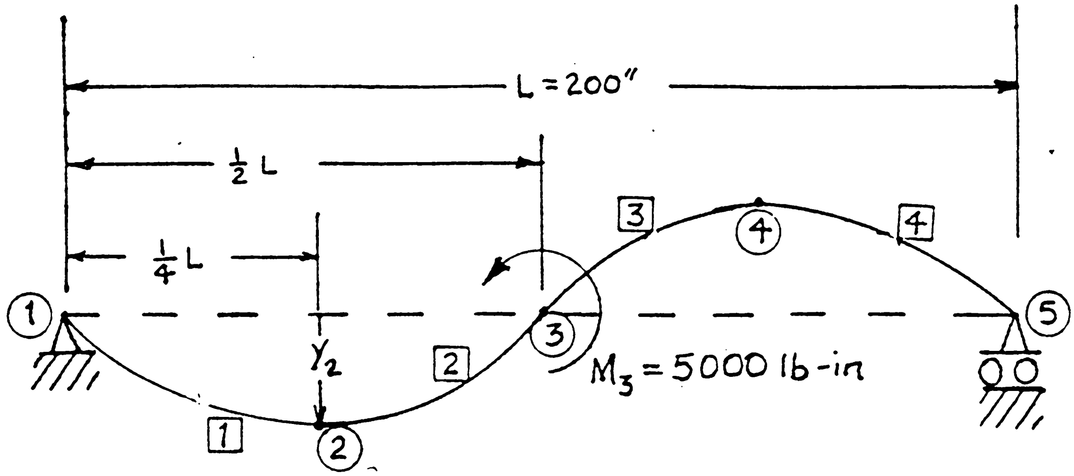
by classical theory are also given. Figure 5 contains the above information for cases C and D.

5.2 INDETERMINANT FRAME ANALYSIS

Meyers [2] has analyzed the frame structure shown in Figure 6 using a plane frame finite element with identical formulation as the one implemented in this project. Hence, as a further check, this frame was modeled and analyzed using MESS. Problem parameters and the node and element numbering schemes used in the finite element model are also shown in Figure 6.

When displacement values predicted by MESS are compared with those listed in [2], there is agreement to within 7 digits of precision, with both listing the maximum displacement as 1.954414 in the x direction at node 2. An element plot and a deformed geometry plot, both generated by MESS, are shown in Figure 7. The complete MESS output file is contained in the Appendix.

The original problem presented in [2] contained a distributed load of 12 kN/m in the x direction on element 1. A force of 100 kN in the x direction was also applied at node 2. In the reference, the distributed load was simply modeled as equal forces of 60 kN at nodes 1 and 2. The accuracy of this approach is questionable for two reasons: first, node 1 is fixed in the x direction, causing that portion of the load to have no effect on the structure; second, a distributed load results in a varying shear force over the length of the element, whereas a point load

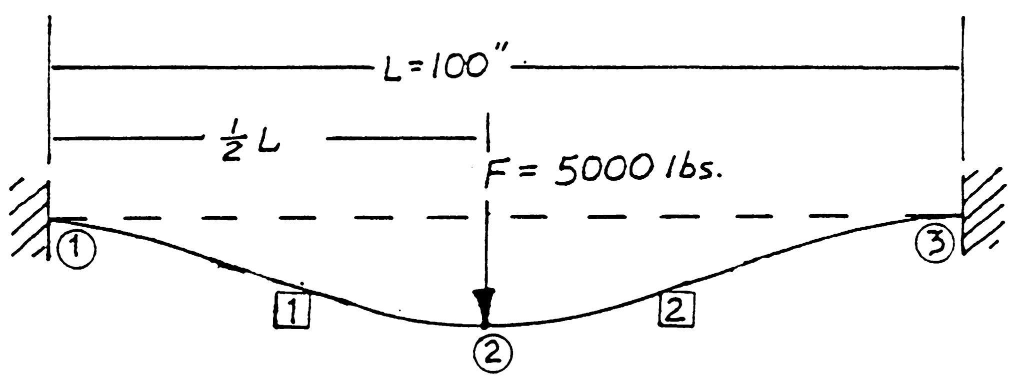


$E = 30(10^6)$ PSI ; $A = 4.0$ IN^2 ; $I = 1.33$ IN^4

THEORETICAL : $Y_2 = -0.3916040''$

FROM MESS : $Y_2 = -0.391604''$

CASE C



$E = 30(10)^6$ PSI ; $A = 4.0$ IN^2 ; $I = 1.33$ IN^4

THEORETICAL : $Y_2 = -0.6526734''$

FROM MESS : $Y_2 = -0.652673''$

CASE D

FIGURE 5 : SIMPLE BEAM THEORY

C. SIMPLY SUPPORTED, CENTER MOMENT

D. FIXED - FIXED, CENTER FORCE

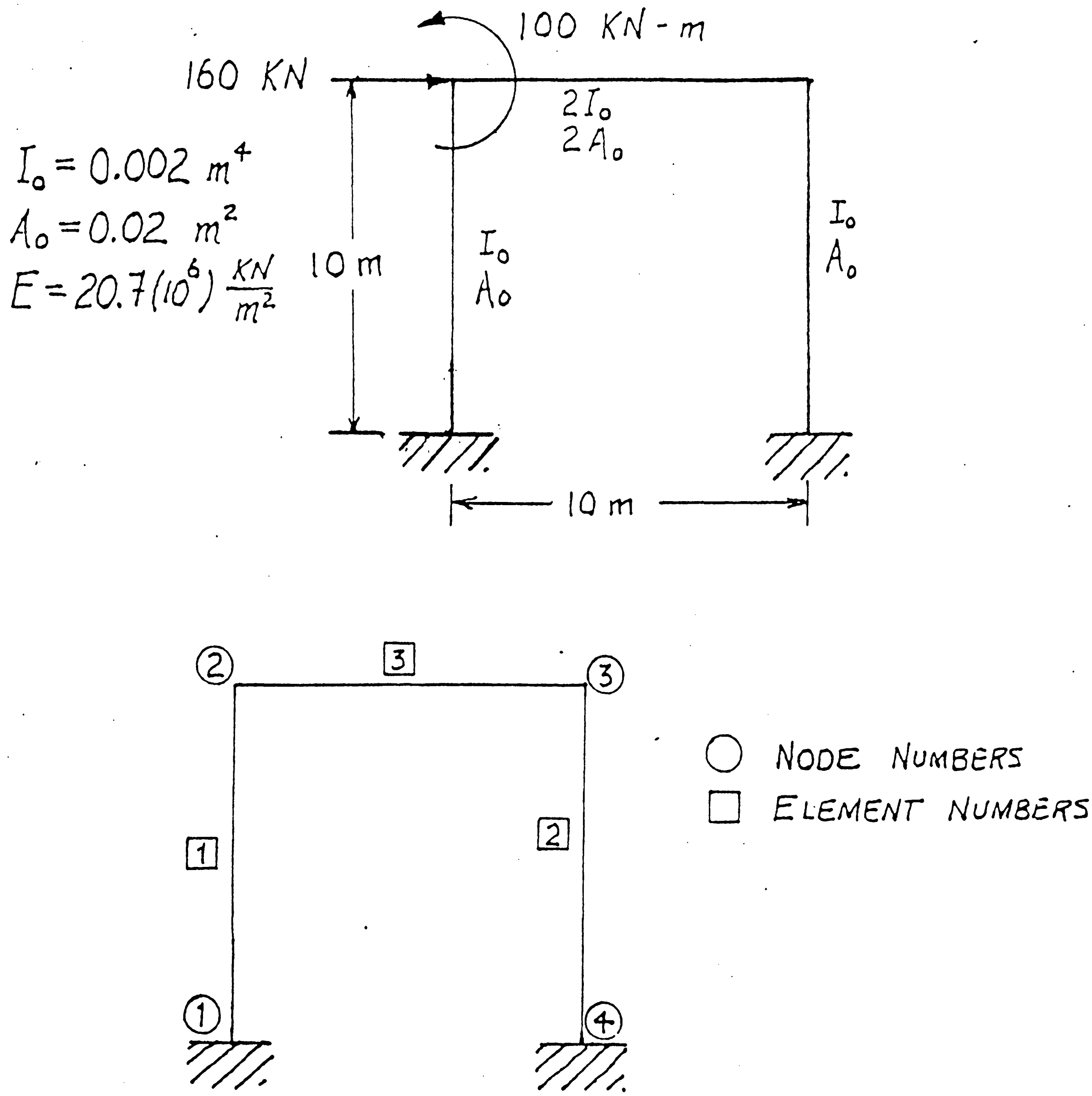


FIGURE 6: INDETERMINANT PORTAL ARCH
A. DIMENSIONS AND LOADING
B. NODE AND ELEMENT NUMBERING
(AFTER [2])

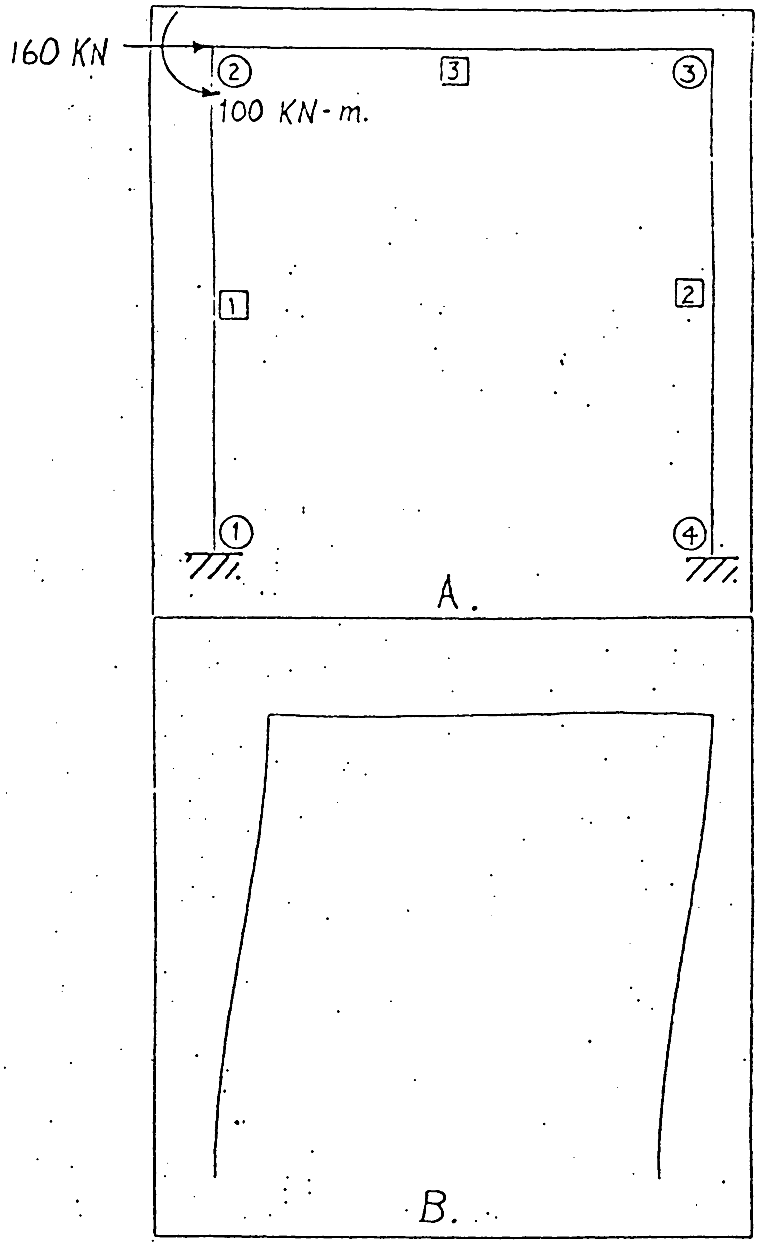


FIGURE 7: MESS OUTPUT FOR INDETERMINANT FRAME
A. ELEMENT PLOT
B. DEFORMED GEOMETRY PLOT

at one end results in a constant shear force. Thus, the physical system has been completely altered. The correct modeling approach approximates the distributed load by using several elements and applying a portion of the load at each additional node. This was accomplished by breaking up element 1 into 10 elements each 1 meter long. When the equivalent loading was applied to the refined model, the deflection at node 2 increased to 1.954901 in the x direction. This represents only a slight increase; thus for this particular loading, minimal error was introduced by the simplifying assumption.

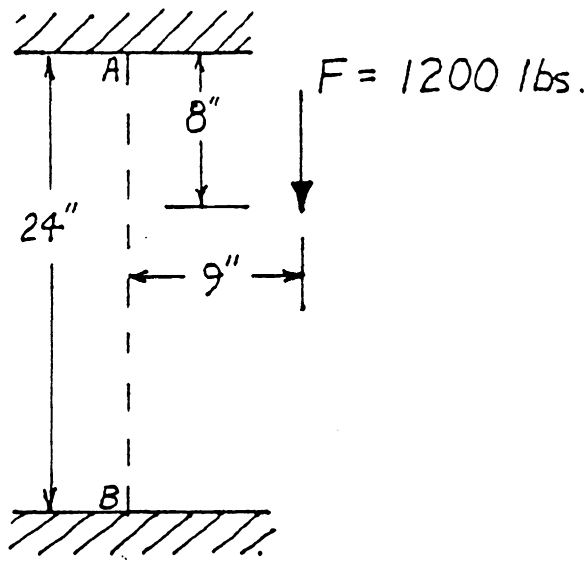
The deformed plot of Figure 7 is very useful in visually depicting the modes of deflection in the structure resulting from the given loading as modeled in [2]. For example, the deformed shape of element 1 shows significant curvature, while element 3 remains relatively linear in shape. Thus, bending forces are more predominant in element 1. Viewed in this manner, the availability of deformed structure plots certainly contribute to the program's value as a design aid.

5.3 COMPARISON OF PROGRAM PREDICTIONS WITH STUDENT WORK

MESS is frequently used in undergraduate instruction as an aid in analysis for design projects. Before the addition of the plane frame element, students had no effective method of easily analyzing even relatively simple indeterminant structures. To survey the benefit and applicability of the new element, three designs submitted for a senior structural optimization project were analyzed using MESS. The project included the design of a steel structure capable of supporting an off-center weight. The design objectives and constraints are given in Figure 8. Analysis results using MESS were compared with those generated by the students using other analysis techniques. The first design to be analyzed is shown in Figure 9; the second design is shown in Figure 10; and the third is shown in Figure 11. Node and element numbering schemes used in MESS are also shown. For ease of comparison, student results will be referenced to the same element numberings used for the finite element analysis.

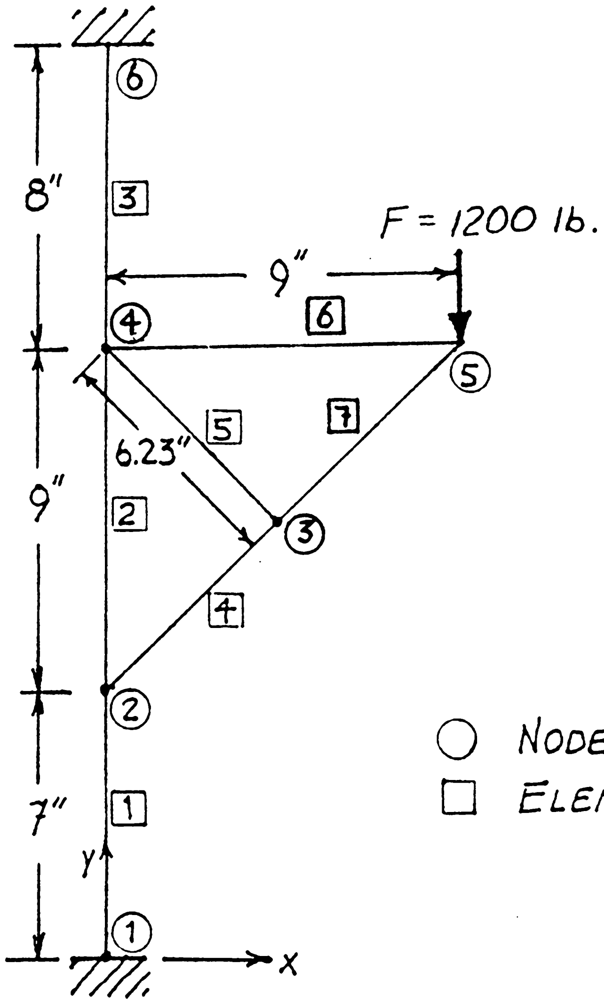
Students submitting the first design chose to analyze the structure using another software package, BEAM II [13]. This program is actually designed to analyze complex loadings of beams and has various capabilities; however, it is limited to one-dimensional analyses. Input quantities required include material and cross-sectional properties, applied forces, and end conditions. Hence, in order to analyze a frame structure, it is necessary to divide it into several one-dimensional segments. If the beam has a non-rigid connection at one or both ends, it is necessary to calculate the equivalent moment and/or linear stiffnesses at that end.

In order to use BEAM II as an analysis tool, the first design was divided into three linear segments. With reference to the finite element notation, the segments are as follows: elements 1, 2, and 3 comprised the first; element 6 the second; and elements 4 and 7 the third. The students assumed element 5 to be a zero force member but included it as a safeguard against buckling. Equivalent moment and linear stiffnesses were calculated where the beams connected to non-rigid supports.



A STEEL STRUCTURE IS TO BE DESIGNED TO TRANSFER THE 1200 lb. LOAD TO SUPPORTS ON CENTERLINE AB. IT IS TO BE AS LIGHT AS POSSIBLE AND MADE OF G10400 STEEL. A SAFETY FACTOR OF 2.0 IS TO BE USED.

FIGURE 8: DIMENSIONS AND CONSTRAINTS FOR SENIOR DESIGN PROJECTS



$E = 30\,(10^6)$ PSI
$I_1 = I_2 = I_3 = 0.141$ IN.^{4}
$A_1 = A_2 = A_3 = 0.844$ IN.^{2}
$c_1 = c_2 = c_3 = 0.75$ IN.
$I_4 = I_7 = 0.00165$ IN^4
$A_4 = A_7 = 0.141$ IN^2
$c_4 = c_7 = 0.188$ IN.
$I_6 = 0.00488$ IN.^{4}
$A_6 = 0.0938$ IN.^{2}
$c_6 = 0.125$ IN.
$I_5 = 0.000326$ IN.^{4}
$A_5 = 0.0625$ IN.^{2}
$c_5 = 0.125$ IN.

○ NODE NUMBERS
□ ELEMENT NUMBERS

FIGURE 9: DESIGN #1 WITH NODE AND ELEMENT NUMBERING

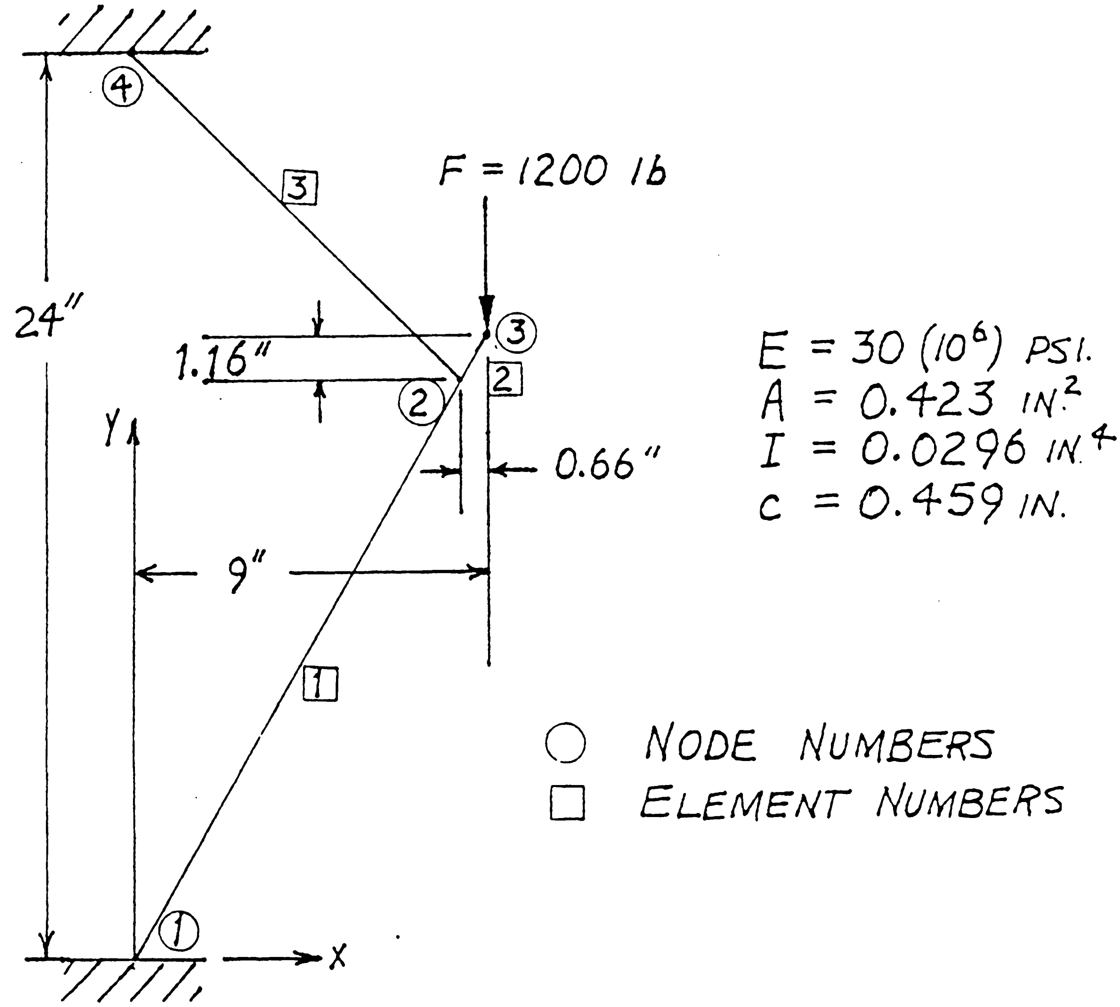


FIGURE 10: DESIGN #2 WITH NODE AND ELEMENT NUMBERING

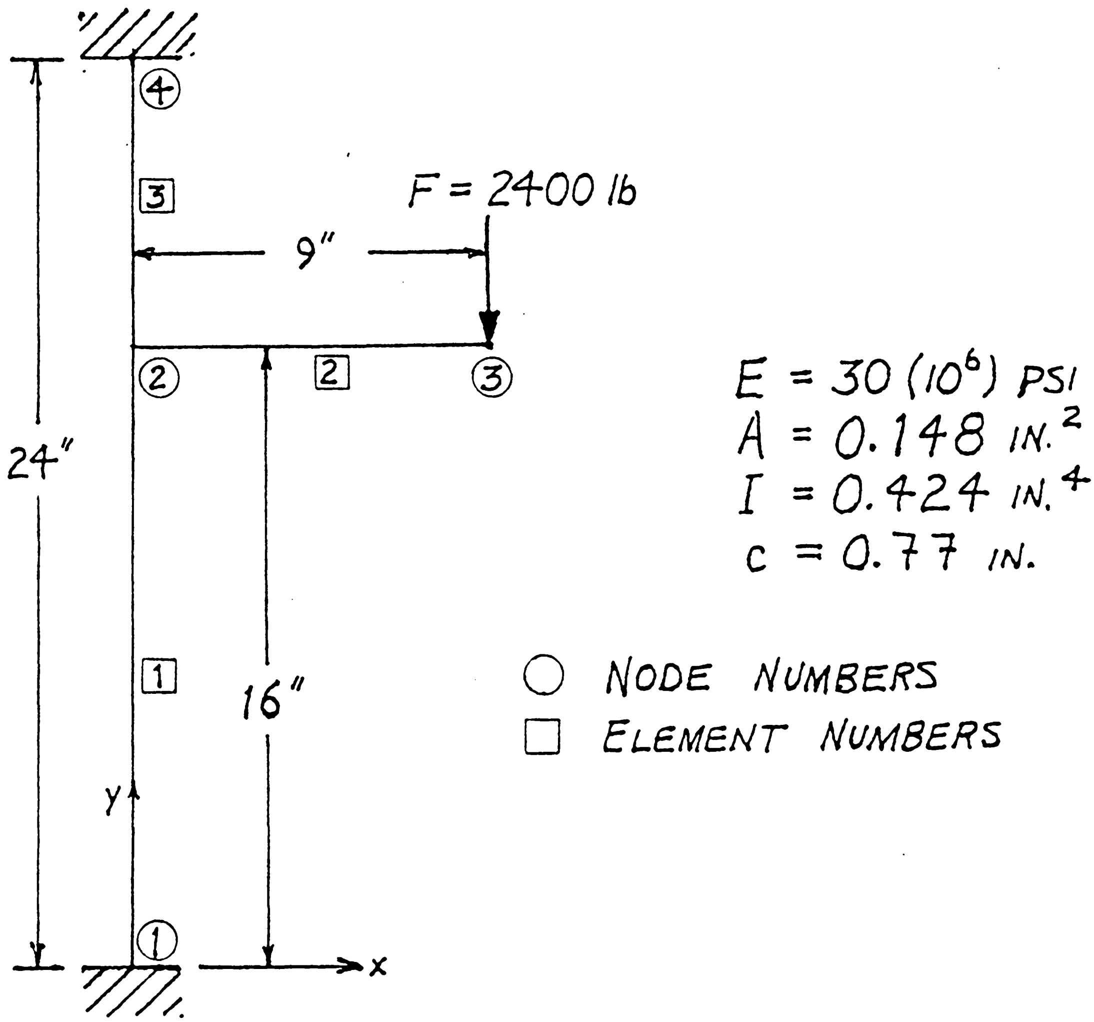


FIGURE 11: DESIGN #3 WITH NODE AND ELEMENT NUMBERING

After analysis using BEAM II, this rather cumbersome approach predicted a maximum stress of 14.3 kpsi in element 3. Correspondingly, MESS predicted a stress of 14.292 kpsi. However, a higher stress of 15.74 kpsi was found in element 6, whereas the student analysis listed the stress there as 13.5 kpsi. When their work was inspected, an error was found in the calculations of the equivalent end moment spring constant of element 6. From inspection of the deformed geometry plot given in Figure 12, it is evident that bending forces are present in element 5. Hence, the assumption that it is a zero-force member is invalid. These two mistakes in the analysis combine to yield inaccurate stress results. The corresponding MESS output file is given in the Appendix.

The second design was analyzed by the student group using classical techniques. With reference to the finite element notation, the students predicted a maximum stress of 33.9 kpsi at element 3. Conversely, MESS found the maximum stress in element 3 as 8.32 kpsi, with a maximum stress of 14.6 kpsi occurring in element 2. Examination of the student's calculations revealed a gross error: a resultant moment was assumed to act in the wrong direction; thus all stress results were invalid. The deformed structure plot in Figure 13 shows that there is significant bending in elements 1 and 3. However, because element 2 is roughly an order of magnitude shorter, inter-element displacements in it are not seen as clearly as in the two larger elements. Obviously, element 2 must entirely bear the moment induced from transferring the force from node 3 to node 2; elements 1 and 3 work together to support this load. Hence, it is

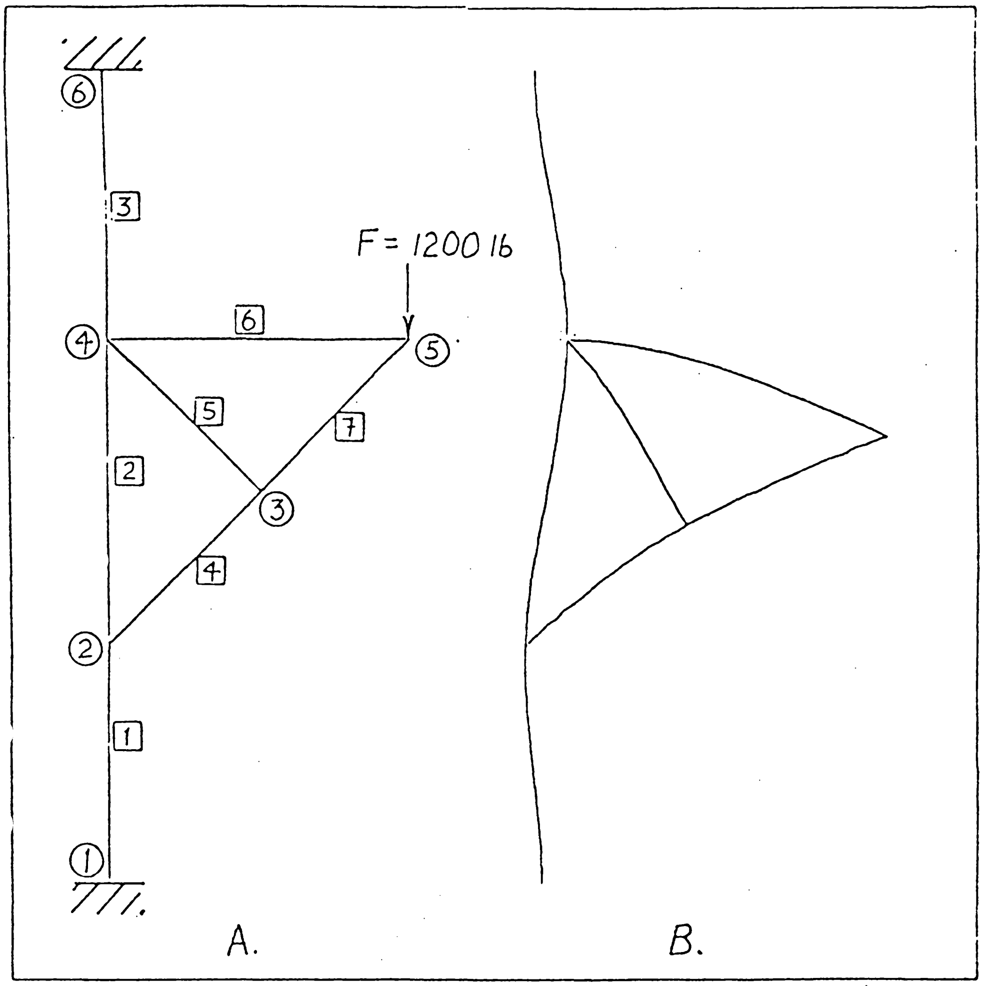


FIGURE 12: MESS OUTPUT FOR
DESIGN #1
A. ELEMENT PLOT
B. DEFORMED GEOMETRY PLOT

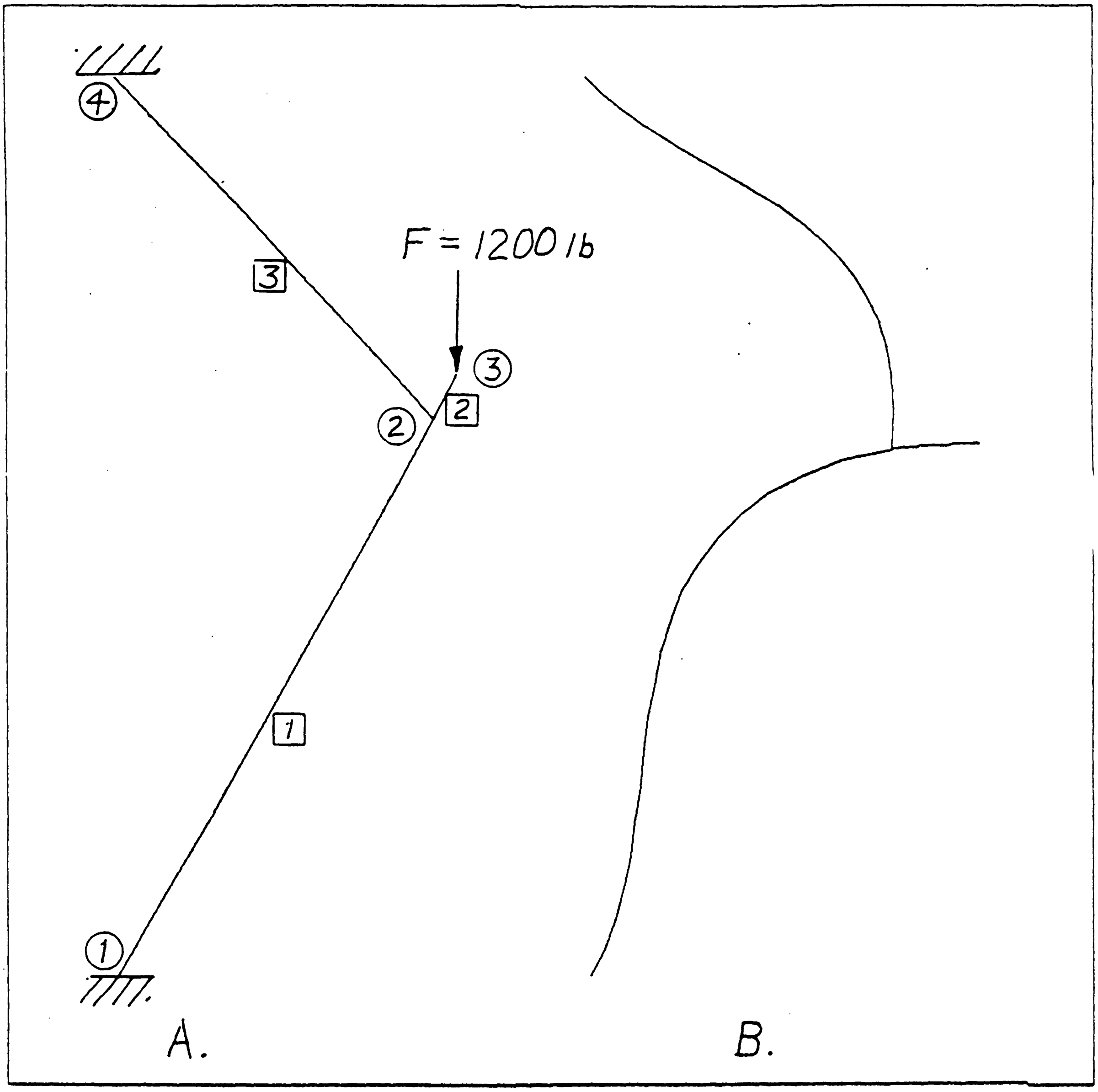


FIGURE 13: MESS OUTPUT FOR DESIGN #2
A. ELEMENT PLOT
B. DEFORMED GEOMETRY PLOT

logical to suspect that the highest stress occurs in element 2, as MESS predicted. The output file generated by MESS is listed in the Appendix.

The final design is the simplest of the three and should yield a close agreement between computer and hand stress calculations. Because the design criteria contained a safety factor of 2.0, this group modeled their structure using twice the design load of 1200 lbs, then designed the structure based on a safety factor of 1. Thus, the finite element model uses an applied force of 2400 lbs in order to directly relate the stress results. The students' hand anaylyis yielded a maximum stress of 39.16 kpsi in element 2, while MESS calculated a stress of 39.24 kpsi. End reactions found by hand agreed closely with those calculated by MESS also. However, the design simplicity results in a key inferiority, as the the deformed geometry shown in Figure 14 illustrates. Relative to element 2, deflections in elements 1 and 3 are small. The resulting stresses are 13.6 kpsi and -18.5 kpsi in elements 1 and 3 respectively, less than half that of element 2. Thus, the material has been used inefficiently because the main vertical support is overdesigned. The MESS output file is listed in the Appendix.

All three designs illustrate a shortcoming of analysis techniques available presently for undergraduates and highlight a positive aspect of the finite element approach. In the first design, the use of a software package not specifically tailored for static frame analysis resulted in error because additional hand calculation work and a simplifying assumption were necessary. Neither error would have been introduced if a software package designed to analyze frame structures had been used. The

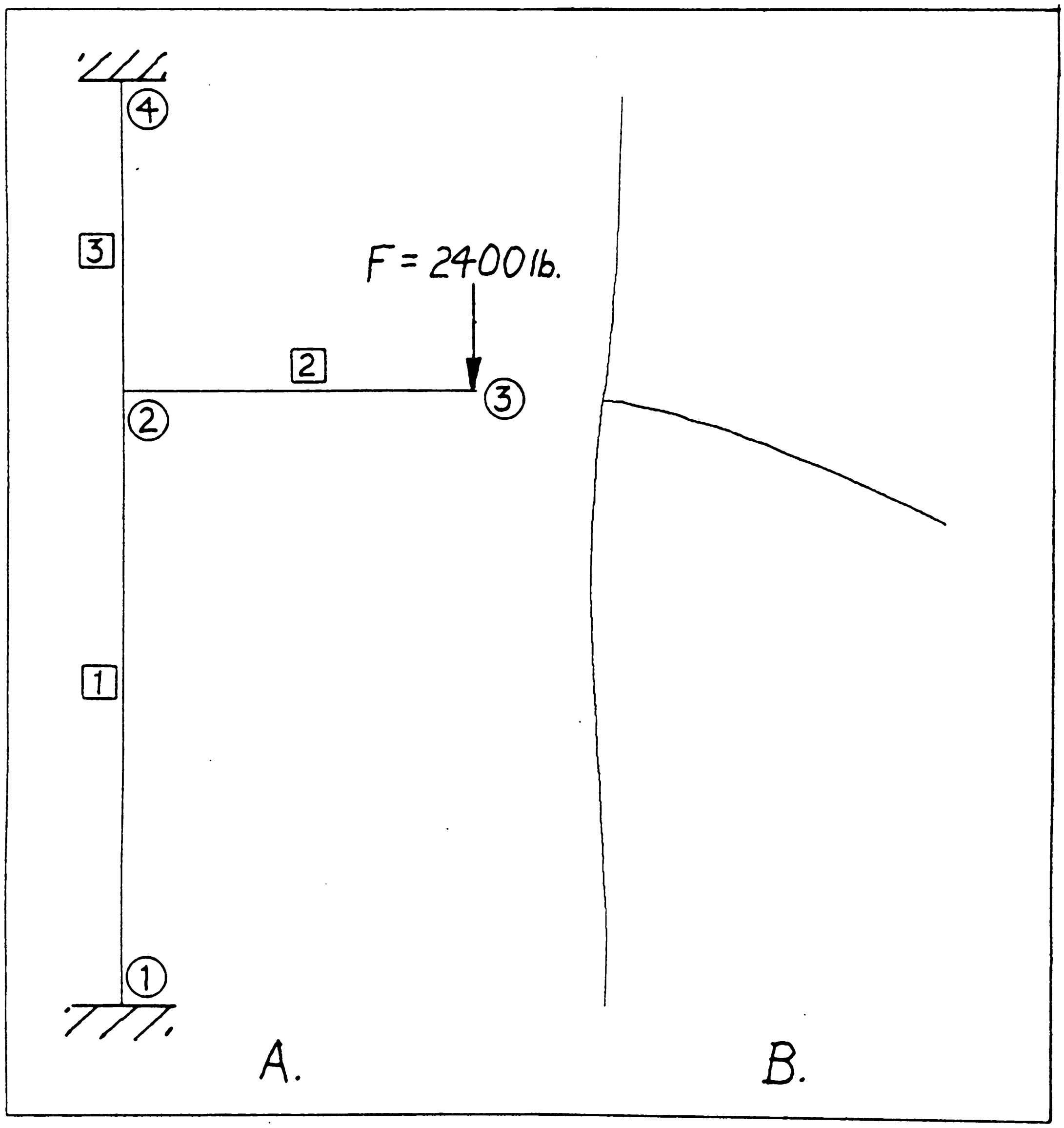


FIGURE 14: MESS OUTPUT FOR DESIGN #3
A. ELEMENT PLOT
B. DEFORMED GEOMETRY PLOT

second design illustrated the complexity of solving an indeterminant structure using hand calculations only. Such a lengthy and involved method easily results in calculation mistakes. The third case shows an inferior design approach to supporting the off-center weight obviously chosen only because the structure can be easily analyzed. Thus, design creativeness and the ability to try various approaches are limited because of an inability to effectively analyze structural response. When used in this context, the frame element now available in the MESS program shifts the burden of analysis to the computer, thus freeing the student to experiment with varied structural approaches.

6.0 CONCLUSIONS AND RECOMMENDATIONS

The plane frame beam element added to MESS was determined to be accurate within the limits of simple beam theory. The element was tested relative to classical theory predictions for transverse deflections of beams with various loadings and constraints. Additionally, an indeterminant frame structure was modeled and analyzed. Displacement results were compared with those predicted by another finite element program containing a plane frame element. In both tests, agreement to within 6 digits of precision was achieved.

The educational benefit related to the addition of the plane frame element was found to be significant. The test cases of designs submitted by ME 4070 Design II students contained calculation mistakes or either represented structures designed on the basis of analysis simplicity, rather than analysis. Capability of deformed structure plotting adds insight to structural behavior; this heightened awareness can result in more efficient designs and encourages inventiveness.

There are several ways in which the capabilities of the plane frame element could be enhanced. A list of possible additions is given below, with the degree of complexity increasing.

1. Write out inter-element displacements if the user wishes.

2. Add capability to model uniform and variable loads across beam length. Presently, if the user wishes to model a uniform load, he must break it into end reactions manually because the program only recognizes nodal forces and moments.

3. Add capability to model internal pins in the structure. Pin joints can presently be modeled, but only if the joint is grounded.

4. Employ the ability to manually scale the deformed structural geometry. The present system of scaling can distort the user's conception of how much deflection is actually present in the structure.

5. Add graphic capability to show applied loads and constraints.

6. Add capability to model variable cross-sectional area and moment of inertia values relative to element length.

LIST OF REFERENCES

1. Bathe, K. J., and Wilson, E. L., Numerical Methods in Finite Element Analysis, Prentice-Hall Incorporated, Englewood Cliffs, New Jersey, 1976.

2. Meyers, v. James, Matrix Analysis of Structures, Harper & Row Publishers, New York, 1983.

3. Davies, Alan J., The Finite Element Method, Clarendon Press, Oxford, 1980.

4. Kanchi, M. B., Matrix Methods of Structural Analysis, John Wiley and Sons, New York, 1981.

5. Reddy, J. N., An Introduction to the Finite Element Method, McGraw Hill, New York, 1983.

6. Rao, S. S. The Finite Element Method in Engineering, Pergamon Press, Oxford, 1982.

7. Butler, Michael J., "A Comparison of Two Models for Geometrically Nonlienar Finite Element Analysis of Plane Frames," M.S. Thesis, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, December 1983.

8. Cook, Robert D., Concepts and Applications of Finite Element Analysis, John Wiley and Sons, New York, 1981.

9. Phan, Hoang V., "Geometric and Material Nonlinear Finite Element Analysis in Geomechanics," M.S. Thesis, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, December 1977.

10. Ford, R., "A Post-Processing System for Finite Elements," Finite Elements in the Commercial Environment, Volume 2, Robinson & Associates, Horton Road, England, 1978.

11. Jara-Almonte, J., "Synthesis of a Near-Optimum Finite Element Mesh," M.S. Thesis, Virginia Polytechnic Institute and State University, Blacksburg, Virginia, December 1982.

12. Shigley, Joseph E., Mechanical Engineering Design, 3rd Edition, McGraw Hill, New York, 1977.

13. Mitchell, Larry D., "Users Guide--BEAM ANALYSIS II," MECC Publication 121.237, Mechanical Engineering Department, Virginia Polytechnic Institute and State Universtiy, Blacksburg, Virginia, August 1982.

APPENDIX

MESS output files for Indeterminant Frame Analysis, Student Design #1, Student Design #2, and Student Design #3

D I S P L A C E M E N T S

NODE	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
1	0.000000	0.000000	0.000000
2	1.954414	0.015550	-0.053150
3	1.945222	-0.015550	-0.082654
4	0.000000	0.000000	0.000000

E N D R E A C T I O N S F O R E L E M E N T G R O U P 1

I N E L E M E N T C O O R D I N A T E S

	END A			END B		
ELEM	TENSILE	SHEAR	MOMENT	TENSILE	SHEAR	MOMENT
1	-.644E+02	0.839E+02	0.441E+03	0.644E+02	-.839E+02	0.397E+03
2	0.644E+02	0.761E+02	0.415E+03	-.644E+02	-.761E+02	0.346E+03
3	0.761E+02	-.644E+02	-.297E+03	-.761E+02	0.644E+02	-.346E+03

M A X I M U M N O R M A L S T R E S S A N D S H E A R F O R C E F O R

E L E M E N T G R O U P 1

	END A		END B	
ELEM	NORMAL STRESS	SHEAR FORCE	NORMAL STRESS	SHEAR FORCE
1	-113585.83	83.89	102583.69	-83.89
2	106907.73	76.11	-89798.29	-76.11
3	57676.29	-64.38	-66837.23	64.38

MESS OUTPUT FILE FOR INDETERMINANT FRAME ANALYSIS FROM (2)

D I S P L A C E M E N T S

NODE	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
1	0.000000	0.000000	0.000000
2	-0.003378	-0.000234	-0.000275
3	-0.000456	-0.006754	-0.001992
4	0.006170	-0.000111	-0.000076
5	0.009993	-0.020805	-0.003143
6	0.000000	0.000000	0.000000

E N D R E A C T I O N S F O R E L E M E N T G R O U P 1

I N E L E M E N T C O O R D I N A T E S

	END A			END B		
ELEM	TENSILE	SHEAR	MOMENT	TENSILE	SHEAR	MOMENT
1	0.848E+03	-.642E+03	-.208E+04	-.848E+03	0.642E+03	-.241E+04
2	-.347E+03	0.555E+03	0.240E+04	0.347E+03	-.555E+03	0.259E+04
3	-.352E+03	-.642E+03	-.261E+04	0.352E+03	0.642E+03	-.253E+04
4	0.169E+04	-.124E+01	0.943E+01	-.169E+04	0.124E+01	-.173E+02
5	-.350E+01	0.128E+01	0.112E+01	0.350E+01	-.128E+01	0.700E+01
6	-.120E+04	0.150E+01	0.117E+02	0.120E+04	-.150E+01	0.174E+01
7	0.169E+04	0.227E+01	0.162E+02	-.169E+04	-.227E+01	-.174E+01

M A X I M U M N O R M A L S T R E S S A N D S H E A R F O R C E F O R

E L E M E N T G R O U P 1

	END A		END B	
ELEM	NORMAL STRESS	SHEAR FORCE	NORMAL STRESS	SHEAR FORCE
1	12075.32	-642.04	-13840.34	642.04
2	-13196.33	554.83	14186.93	-554.83
3	-14292.28	-642.04	13862.32	642.04
4	13069.97	-1.24	-13965.48	1.24
5	-483.86	1.28	2741.27	-1.28

MESS OUTPUT FILE FOR STUDENT DESIGN #1

D I S P L A C E M E N T S

NODE	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
1	0.000000	0.000000	0.000000
2	-0.000367	-0.001173	-0.001664
3	0.001962	-0.002608	-0.002250
4	0.000000	0.000000	0.000000

E N D   R E A C T I O N S   F O R   E L E M E N T   G R O U P   1

I N   E L E M E N T   C O O R D I N A T E S

	END A			END B		
ELEM	TENSILE	SHEAR	MOMENT	TENSILE	SHEAR	MOMENT
1	0.896E+03	-.300E+02	-.169E+03	-.896E+03	0.300E+02	-.342E+03
2	0.105E+04	0.588E+03	0.782E+03	-.105E+04	-.588E+03	-.114E-12
3	-.634E+03	-.518E+02	-.440E+03	0.634E+03	0.518E+02	-.202E+03

M A X I M U M   N O R M A L   S T R E S S   A N D   S H E A R   F O R C E   F O R

E L E M E N T   G R O U P   1

	END A		END B	
ELEM	NORMAL STRESS	SHEAR FORCE	NORMAL STRESS	SHEAR FORCE
1	4736.41	-30.03	-7427.38	30.03
2	14605.50	587.97	-2473.01	-587.97
3	-8322.41	-51.77	4625.36	51.77

MESS OUTPUT FOR STUDENT DESIGN #2

D I S P L A C E M E N T S

NODE	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
1	0.000000	0.000000	0.000000
2	-0.008050	-0.000288	-0.003019
3	-0.008050	-0.073307	-0.010660
4	0.000000	0.000000	0.000000

E N D R E A C T I O N S F O R E L E M E N T G R O U P 1

I N E L E M E N T C O O R D I N A T E S

	END A			END B		
ELEM	TENSILE	SHEAR	MOMENT	TENSILE	SHEAR	MOMENT
1	0.800E+03	-.120E+04	-.720E+04	-.800E+03	0.120E+04	-.120E+05
2	0.000E+00	0.240E+04	0.216E+05	0.000E+00	-.240E+04	-.909E-12
3	-.160E+04	-.120E+04	-.960E+04	0.160E+04	0.120E+04	-.682E-12

M A X I M U M N O R M A L S T R E S S A N D S H E A R F O R C E F O R

E L E M E N T G R O U P 1

	END A		END B	
ELEM	NORMAL STRESS	SHEAR FORCE	NORMAL STRESS	SHEAR FORCE
1	13616.01	-1200.00	-22332.99	1200.00
2	39226.42	2400.00	0.00	-2400.00
3	-18515.04	-1200.00	1081.08	1200.00

MESS OUTPUT FILE FOR STUDENT DESIGN #3

The vita has been removed from
the scanned document