To the Graduate Council:

I am submitting herewith a thesis written by Vernon ''Pete'' Mills, Jr. titled ''Axially Loaded Members Employing Limit State Design for Stactically Determinate Truss.'' I have examined the final copy of this thesis for form and content and recommend it be accepted in partial fulfillment of the requirements for the degree of Master of Science with a major in Mechanical Engineering.

Dr. L. R. Deken, Major Professor

We have read this dissertation
and recommend its acceptance:

Dr. R. C. Engels

Dr. R. J. Schulz

Accepted for the Council:

Associate Vice Chancellor and
Dean of The Graduate School

AXIALLY LOADED MEMBERS EMPLOYING LIMIT STATE DESIGN FOR STACTICALLY DETERMINATE TRUSS

A Thesis

Presented for the

Master of Science

Degree

The University of Tennessee at Knoxville

Vernon Mills, Jr.

December 1995

ACKNOWLEDGMENTS

I would like to thank my major professor Dr. L.R. Deken for his guidance and patience in helping me accomplish this study. I would also like to thank the other committee members, Dr. R.J. Schulz and Dr. R.C. Engels, for their comments and assistance to me in completing the master's degree program. I would like to express my thanks to Sverdrup Technology, Inc., AEDC Group, for allowing me the opportunity to complete this degree through their work study program. I want to thank my family for their patience during rough times necessary for the completion of the course work. I especially want to thank my sons, Shane and Ryan, for challenging me to stay equal to their excellent performance as they are in their studies.

ABSTRACT

This study was undertaken to develop a design code for the design of planar trusses employing the "Load and Resistance Factor Design" (LRFD) philosophy. The design code employs matrix methods of structural analysis combined with the LRFD design philosophy focusing on the limit states of strength for structural integrity. The design aid features a personal computer program which evaluates a planar truss system to determine truss member forces, system deflections, restraint loading, and member size based on the failure mode criteria. The modes of failure investigated for axially loaded members include tension, compression, and buckling. The design code has the ability to optimize each truss member by minimization of member weight. Program stress and deflection analysis results are consistent with ANSYS®, a commercial finite element analysis program, thus benchmarking the accuracy of program solutions.

The computer program member sizing technique is consistent with the engineering philosophy of the American Institute of Steel Construction (AISC) publication employing the ultimate strength analysis techniques described by the LRFD manual.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF ABBREVIATIONS

A	=	area of cross-section, $in.^2$
[A]	=	Statics matrix
A_e	=	effective net area of tension member, $in.^2$
A_g	=	gross area of cross-section, $in.^2$
AISC	=	American Institute of Steel Construction
ANSI	=	American National Standards Institute
ANSYS	=	commercial finite element analysis software
ASD	=	Allowable Stress Design
b_f	=	Flange width-thickness ratio
C_c	=	slenderness ratio of columns for ASD
D	=	Dead load, kips
DX	=	Node or joints restraint in x direction
DY	=	Node or joints restraint in y direction
E	=	Earthquake load, kips
E	=	Modulus of elasticity, ksi
e	=	elongation, inches
F	=	internal member load, kips
[F]	=	internal member load matrix
F_{cr}	=	critical stress in compression, ksi
F_y	=	yield stress, ksi
F_u	=	ultimate tensile stress of structural steel, ksi
G_a	=	Joint rigidity for a beam-column connection in a frame

h_c = unsupported web height, inches

I_b = Beam moment of inertia, in.^{4}

I_c = Column moment of inertia, in.^{4}

K = effective length factor

[K] = stiffness matrix

L = member length, inches

L = live load, kips

L_r = roof live load, kips

L_b = laterally unbraced beam length, inches

L_c = laterally unbraced column length, inches

l = x direction cosine

LRFD = Load and Resistance Factor Design

m = y direction cosine

n = numerical label for component of load and deformation

[P] = external load matrix, kips

P_n = nominal strength of axially loaded members

P_u = ultimate design strength of axially loaded members

PX = joint external load in x direction of truss

PY = joint external load in y direction of truss

Q = nominal load, kips

r = radius of gyration, inches

R = Load due to pooling or ponding from precipitation, kips

R_n = nominal resistance, kips

S = snow load, kips

$[S]$ = stiffness matrix

W = wind load, kips

x_i = x coordinate, inches

$[X]$ = deformation matrix, inches

y_i = y coordinate, inches

β = λ_c^2

γ = load factor

λ_c = slenderness parameter for columns

λ_p = maximum slenderness ratio for compact members

λ_r = maximum slenderness ratio for non-compact members

$\varnothing$ = resistance factor; strength reduction factor

$\varnothing_c$ = resistance reduction factor for compression member

$\varnothing_t$ = resistance reduction factor for tension member

CHAPTER 1

INTRODUCTION

This thesis presents the development of a design code for a planar truss to predict both failure mode and member size optimization. The code resulted from an investigation combining matrix methods for structural analysis and the ''limit state design philosophy'' which were integrated into a single computer program for a planar truss system. The limit state philosophy is described by the American Institute of Steel Construction in the LRFD manual.[1]

Two matrix methods of structural analysis techniques considered for this study included the "Method of Joints" and the "Displacement Method."[2] The ''Method of Joints" is ideally suited for statically determinate trusses because no initial cross-sectional properties of the truss are required. Analysis is based on member geometry and the solution of matrix equations representing the equilibrium of forces for the truss. The "Displacement Method" requires the use of a member stiffness matrix ([S]) and requires an initial knowledge of cross-sectional properties of the member. In this design code, the cross-sectional area is determined and optimized based on truss member internal loading. Therefore, this study selected the "Method of Joints" for the structural analysis of trusses, thus limiting the analysis to stactically determinate planar trusses.

In September 1986, the American Institute of Steel Construction published a new approach to the design of structures. The publication "Load and Resistance Factor Design Manual" (LRFD)[1] provides a guide for the design of structural steel members. This design approach presents an improved method for structural analysis over the commonly used allowable strength design method, because it provides a more uniform safety factor on structural design load capacity. This approach is based on limit states of strength which define the ultimate capacity of a member under load.

The LRFD specification defines the limit state as the design condition where members are proportioned so structural stability is maintained when exposed to all factored loads.[1] Stability is defined as the ability of a member to carry all factored loads without collapsing whether the cross-section is yielded or not. Two types of limit state behavior defined by LRFD are strength and serviceability. The limit state of serviceability ensures functional performance in design of the structural members to include limitations imposed by the designer on structural behavior such as deflections, drift, corrosion, and vibrations. The LRFD specification rationalizes the focus on the limit states of strength, ''The LRFD specifications, like other structural codes, focuses on the limit states of strength because of the overriding consideration of public safety for life, limb and property of human beings. This does not mean that limit states of serviceability are not important to the designer, who must equally insure functional performance and economy of design. However, these latter considerations permit more exercise of judgment on the part of designers. Minimum considerations of public safety, on the other hand, are not matters of individual judgment and, therefore, specifications dwell more on the limit states of strength than on the limit states of serviceability.'' The major concern is a safety issue concerning structural integrity of a member, in particular, the structural member's maximum load carrying capacity. Examples of limit states of strength include member maximum strength, buckling, fatigue, and fracture. Strength is a measure of ability of a member to carry the loading and has units of kips or ksi based on the application. Buckling is the premature collapse of a member under compressive loads before the ultimate capacity of the cross-section is reached. Fracture is the failure of the cross-section under stress. Fatigue is the failure of a member under cyclic loading.

The basis of the LRFD design for strength criterion is the specification of a required strength ($\varnothing R_n$) at some critical combination of various types of structural loading. The design strength is the summation of nominal load effects (Q) multiplied by corresponding load factors (γ). The algebraic sum of the γQ for all loading must be less than or equal to a member's design strength. Load factors provide a safety factor for a given type of loading since it accounts for probable variations in the defined nominal loads. Design strength is nominal strength or resistance (R_n) multiplied by a corresponding resistance factor ($\varnothing$). The resistance factor defines a reduction in capacity to ensure a safety factor against the possibility that the true resistance of a member may be less than the design due to impurities in material and quality of fabrication. The general format for design strength is as follows:

$$\Sigma\ \gamma Q \quad \leq \quad \varnothing R_n$$

where γ = load factor corresponding to load Q

Q = nominal load, kips

γQ = required resistance load, kips

R_n = nominal resistance load of member, kips

$\varnothing$ = resistance factor corresponding to Rn

$\varnothing R_n$ = design strength, kips

The required design strength is based on loads, load factors, and load combinations as define by the LRFD manual or other load requirements. The details of the load factors, load combinations, and LRFD design strength are provided in Chapter 2.

This thesis objective is to develop a design code to analyza planar truss employing the ''limit state philosophy.'' This study developed a code to determine member loading using the matrix ''Method of Joints'' approach for truss analysis, then determined optimum member sizes for failure modes including tension, compression, and buckling under the externally applied loading.

CHAPTER 2

DESIGN FOR STRENGTH

It is necessary to discuss some general provisions of the LRFD specifications to help understand its concept. This chapter discusses general provisions including loads, load combinations, and design strength.

The application of load factors with load combinations is a new approach to the traditional allowable strength design method used in the design of structural members. Design loads for many applications are given by ANSI A58.1 "Minimum Design Loads for Buildings and Other Structures"(4) and include dead, live, wind, snow, and earthquake. The required strength of a structure and its elements must be determined from some appropriate critical, weighted combination of various types of loads, called factored loads. The abbreviated definition of the LRFD types of loads and their symbols, as defined by the LRFD code, are summarized as follows:

D = dead load due the weight of structural elements and permanent features on the structure.

L = live load due to occupancy and moveable equipment.

L_r = roof live load.
W = wind load.

S = snow load.

E = earthquake load.

R = load due to initial rainwater or ice, exclusive of any combination that results from pooling or ponding of precipitation.

In this study, the investigation of load combinations on structural integrity is based on the largest value of all combinations used in the determination of the required design strength. The development of probability load criteria by Galambos, Ellingwood, MacGgreger, and Cornell(5) led to the factored load combinations of the 1982 ANSI Standard. The ANSI Standard was developed for use in design with all materials. It is reasonable to assume the probability of various loads acting in combinations as well as the probability of overload with certain types of loads, should be unrelated to the material of which a structure is built. With this concept in mind, the 1986 LRFD specifications adopted the ANSI factored load combinations.(5) The factored load combinations defined by LRFD for strength design are as follows:(1)

1. 1.4 D
2. 1.2 D + 1.6 L + 0.5 (Lr or S or R)
3. 1.2 D + 1.6 (Lr or S or R) + 0.5 (L or 0.8 W)
4. 1.2 D + 1.3 W + 0.5 L + 0.5 (Lr or S or R)

5. $1.2\ D \pm 1.5\ E + (0.5\ L$ or $0.2\ S)$

6. $0.9\ D \pm (1.3\ W$ or $1.5\ E)$

Other requirements and loading may require other factors and load combinations be defined for design.

A simple application illustrating the use of load factors and load combinations is shown in Figure 2.1.* For discussion of the evaluation of load and load combination on the structural response, it is assumed the total dead weight of the truss system in Figure 2.1 at joint 3 is .52 kips. A snow load of 9.87 kips and wind load of 8 kips are applied loads at joint 3. The LRFD load combinations for the loading in Figure 2.1 (kips) are:

1. $1.4D = 1.4(.52) = .728$
2. $1.2D+1.6L+.5S = 1.2(.52)+ 0 +.5(9.87) = 5.6$
3. $1.2D+1.6S+.8W = 1.2(.52)+1.6(9.87)+.8(8) = 22.8$
4. $1.2D+1.3W+.5L+.5S = 1.2(.52)+1.3(8)+0+.5(8) = 15$

Based on these load combinations, the largest value is determined by load combination equation 3. Therefore, the planar truss must be designed to have a minimum load capacity of 22.83 kips.

The factored load combination approach as described above can be applied under both elastic or

---

* All Figures and Tables are located in Appendix A.

plastic structural behavior in member design. Elastic behavior is when working stresses in a member are less than or equal to the yield stress of the material. This theory is based on the assumption of a linear stress/strain relationship to the point of attainment of the material yield point. Plastic behavior occurs when stresses greater than the yield stress are permitted on the cross section of a member. A limitation which governs plastic analysis on steels is that the yield stress shall not exceed 65 ksi.(1) This yield stress limitation ensures that a joint in a beam member can achieve the necessary rotation to develop the full plastic moment capacity of the member cross section. In addition to factored load combinations, all members are subject to the resistance limitation requirements for the various modes of failure including tension, compression, and buckling.

CHAPTER 3

MODES OF FAILURE

The design code developed during this study addresses three modes of failure for truss members. The failure modes are tension, compression, and buckling of axially loaded structural members. Each type of failure must meet different but specific design requirements for acceptance under the limit state LRFD specifications.

In tension failure, it is assumed the stress is distributed uniformly over the cross-sectional area of the member. Tension failure occurs when tensile stresses reach uniform yielding of the member's cross-section. The LRFD imposes a limit to the slenderness ratio of truss members to be less than or equal to 300. The slenderness ratio is defined as the length (L) of a truss member divided by the radius of gyration (r) about the critical axis of cross-section for bending. Length is the distance between lateral supports of the structural member. This slenderness ratio limitation is required for tension members which may be subjected to a compressive force when wind or earthquake loads are present.

The investigation of truss member end connections is critical when designing for tension failure. End connections include; bolting, riveting, or welding of the cross-sectional elements. These connections may reduce the load carrying capacity of the member at the connection. This design code allows for reduced or effective net areas (A_e) for end connections introduced by the construction process. The use of reduced end connection areas ensures fracture on the net cross-sectional area will not govern a tension failure over yielding of the gross cross-sectional area of the member. The design tensile strength is the lower value of either yielding on the gross cross-sectional area or fracture on the net cross-sectional end connection area for tension members.

Compression failure of truss members is handled similarly to tension failure, with the additional constraint that a limiting slenderness ratio shall not exceed a value of 200. The slenderness ratio is equal to the effective length factor (K) multiplied by L and divided by radius of gyration (r) about the cross-sectional axis of buckling. Effective length factor (K), when multiplied by the member length, will equal the distance between any two inflection points in an equivalent member at locations other than its ends. Therefore, KL is the equivalent length of a column with pinned ends, whereas K is equal to 1.0 for axially loaded (pinned-pinned) truss members. The development of effective length factors for frames is discussed in

Chapter 5 for the case where the design code is used to design single axially loaded members.

Compression failure of members can occur when they are subjected to axial loading, and the resulting compressive stress, distributed uniformly over the cross-sectional area of the member, exceeds a failure stress. Compression failure occurs by total yielding of the cross-sectional area of the member. An example of compression failure is a short column yielding under axial loads.

Buckling failure, a sudden bending of axially loaded columns, occurs as a result of column instability under load. This mode of failure may occur elastically or inelastically. Elastic buckling is a condition which the cross-sectional elements are in an elastic state of stress when unstable bending behavior occurs. Inelastic buckling is a condition which a portion of the cross-sectional element yields (stress beyond the proportional limit), resulting in an unstable deflection or bending behavior of the column.

Compression members are classified by LRFD[1] into categories based on cross-sectional slenderness properties, called compact and non-compact sections, and slender compression elements. The element cross-section classifications are based on element width to thickness ratios (such as half of the flange width divided by the flange thickness) of the column. For a truss member to be classified as a compact section, the flanges must be continuously connected to the web or webs, and the width-thickness ratios, $b_f/2t_f$, (where b_f is the flange width and t_f is the flange thickness) and web width-thickness ratio, h_c/t_w, (where h_c is the web width and t_w is the web thickness) of its compression elements must not exceed λp. The parameter λ_p is maximum slenderness ratio to be classified as a compact element as define in LRFD specification, Table B5.1.[1] If the member qualifies as a compact section, the member can develop the full plastic stress distribution without element buckling and can generate a joint rotation of approximately 3 (plastic hinge) before buckling. The division between compact and non-compact is defined by the values of λp.

Non-compact sections are defined by λ_r which is also a slenderness ratio for the flange and web section of a structural member. This classification (non-compact) occurs when width-thickness ratios of the compressive members exceed λ_p but does not exceed λ_r as defined by the LRFD specification, Table B5.1.[1] Non-compact sections can develop partial yielding of the cross-sectional elements before buckling; therefore, inelastic local buckling at strain hardening levels of stress will occur before full plastic distribution of stress is achieved. The formula for λ_r is defined for the flanges of wide flange sections as follows:[1]

$\lambda_r = 95/SQRT(F_y)$

and λ_r for webs is defined as: [1]

$\lambda_r = 253/SQRT(F_y)$

where λ_r is upper limit for width-thickness ratios for non-compact sections. F_y is the yield stress of the material in ksi. The value of λ_r specifies the division between non-compact sections and slender compression elements. Both flange and web slenderness ratios must meet the minimum upper limit for cross-section elements to qualify as non-compact sections.

Elements of member cross-sections are classified as slender compression elements when the width-thickness ratio exceeds λ_r. This type of member buckles elastically due to the slender elements of the cross-section before the yield stress is achieved. Element cross-sections with slender compression elements require the application of a reduction factor to prevent buckling of the member or its cross-sectional elements. The requirements for this type of member is discussed in LRFD Commentary, Appendix B.[1]

This chapter discussed the modes of failure and the specific requirements incorporated in the analysis of a structural design. The design code developed as part of this study limits the analysis to compact and non-compact sections having a width-thickness ratio less than λ_r for compression elements. In designing members for pure compression, the LRFD sets the slenderness requirements to the limitation of λ_r. (The restriction on λ_p applies to combined flexure and compression loading.) It is the intent of this design code to use standard commercial structural shapes when setting the upper limit to non-compact cross-sections. By limiting the slenderness ratio to the upper limit λ_r for the flange and web, it ensures the design will meet width-thickness ratio requirements and prevent premature buckling.

CHAPTER 4

TRUSS DESIGN CODE OVERVIEW

The design code developed in this study includes a FORTRAN program for truss analysis to determine the internal forces acting on truss members, member sizes, truss deflections, and restraint loads. The member forces and deflections are determined utilizing a matrix analysis method. The member size is determined from a force equilibrium analysis using factored structural loads specified by LRFD. A feature of this design process is the ability to select a standard structural shape and optimize the member size by minimizing weight. Another feature is it provides a design tool for analyzing single axially loaded members in tension or compression and determines its optimum member size. Further discussion of this feature is provided in Chapter 5. The single member option also provides the capability to select an effective length factor (K) other than the default 1.0 for truss members. The determination of the effective length factor (K) shall be in accordance with Chapter C of the LRFD manual.[1]

Truss member force and joint displacement solutions are obtained from node coordinates, defining the geometry of the truss, element connectivity, defining the truss members, and the applied loading.

This study employed the "Method of Joints"[(2)] to solve for internal forces using the following matrix equation:

$$[P] = [A][F]$$

where $[P]$ = external loads matrix, kips

$[A]$ = statics matrix

$[F]$ = internal member loads matrix, kips

The matrix [A] is defined as the statics matrix and is based on the equilibrium conditions of static forces at the truss joints. The internal forces are unknown, but can determined by multiplying the above equation by the inverse of [A] to give the following equation:

$$[F] = [A]^{-1}\ [P] \qquad \text{Equation (4-1)}$$

Since the inverse of matrix [A] is required, the design method is limited to statically determinate trusses, which results in matrix [A] being a square N x N matrix, where N is the number of truss elements. The Gauss-Jordan method is used to solve the linear system of equations represented by Equation 4-1.

The statics matrix is derived by writing the equilibrium force equations for force components at each joint for a truss system. The equilibrium equations are developed by writing the equilibrium conditions between the external loads and the internal member forces for each joint. Applying this process to the truss system in Figure

4.1, the equilibrium conditions for joints A, B, C are as follows:

For joint A (Figure 4.2)

$$P1 = F1\cos\theta_1 + F2\cos\theta_2$$

$$P2 = F1\sin\theta_1 + F2\sin\theta_2$$

For joint B (Figure 4.3)

$$P3 = F1\cos\theta_1 + F3\cos\theta_3$$

$$P4 = F1\sin\theta_1 + F3\sin\theta_3$$

For joint C (Figure 4.4)

$$P5 = F2\cos\theta_2 + F3\cos\theta_3$$

$$P6 = F2\sin\theta_2 + F3\sin\theta_3$$

where x (external force) components are odd numbered and y components are even numbered.

Joints having an end restraint condition will result in applied loads being reacted by the support and not by the truss. For this condition, no force components are contributed to the statics matrix. For the three member truss in Figure 4.5, the applied loads P1, P2, and P4 are constrained; therefore, they will contribute zero force components, and are not consider in the statics matrix. The matrix [A] for this simple truss is formally written as follows:

$$[A] = \begin{array}{c} \\ P3 \\ P5 \\ P6 \end{array} \begin{array}{c} \begin{array}{ccc} F1 & F2 & F3 \end{array} \\ \begin{bmatrix} \cos\theta_1 & 0 & \cos\theta_3 \\ 0 & \cos\theta_2 & \cos\theta_3 \\ 0 & \sin\theta_2 & \sin\theta_3 \end{bmatrix} \end{array}$$

The terms of the statics matrix are essentially the directional cosines of the members. The values of the direction cosines for the truss elements are determined from the beginning and ending nodes defining the element connectivity. The direction cosines are then assigned to the proper position in the matrix. Calculation of the direction cosines requires the determination of member length (L) defined by the equation:

$$L = SQRT((X_2 - X_1)^2 + (Y_2 - Y_1)^2)$$

where L = member length, in.

X_i = x-coordinate at ends of member, in.

Y_i = y-coordinate at ends of member, in.

The directional cosines are calculated from:

$$l = (X_2 - X_1)/L$$

$$m = (y_2 - y_1)/L$$

where l = x direction cosine

m = y direction cosine

Figure 4.6 is diagram of a generalized element used in the determination of the equilibrium conditions at each joint. The determination of the statics matrix for the three element truss in Figure 4.3 can be demonstrated with Element 2 of the truss. The element beginning node is 3 and ending node is 1. For this element, the direction cosines are $l = 0.6$ and $m = 0.8$. Therefore A(2,2) is equal to .6. Similarly for the y direction, the matrix position A(3,2) is equal to .8 for Element 2. The ending node 1 is defined as a constrained joint in both x and y direction and that value is eliminated from the statics matrix. After the statics matrix is developed, the code employs the Gauss-Jordan elimination method to solve the equations for the internal forces of each truss member. The internal force for the element is the required design ultimate load used to determine the required design capacity of the truss member. The design code is based on the assumption all loads are factored loads and the required critical load combinations are specified by the user prior to the analysis.

Once the solution for the internal loading is complete, the design code determines the proper mode of failure for the design calculations for each truss element. This is accomplished by testing the positive or negative value of the element internal loading (F). Positive values of F result in the member being analyzed for tension failure, while negative values of F result in an evaluation of the member for compression and buckling failure.

For tension failure, the program routes the element and the associated design parameters to a subroutine optimizing the member size for tension design requirements. Tension member design requirements include slenderness ratio limitations and criteria to the type of failure (yielding or fracture). The code contains an internal database of structural geometric properties that include member size, weight, cross-sectional area, flange width-thickness ratio, radius of gyration about both axes of cross-section, and web width-thickness ratio. The optimum member is the smallest element size, (for example, the minimum member weight for structure), which meets all design criteria and carries the required design strength. The design tensile strength of tension members is the lower value obtained from the limit states of yielding on the gross cross sectional area or the effective net cross sectional area multiplied by a predefined tensile reduction factor ($\varnothing_t$). The gross cross-sectional area is defined as the total area of all cross-sectional elements of a structural member. The effective net cross-sectional area is based on end connection conditions, and how loads are transmitted from the connection to the cross-section of the member. Construction details associated with bolting, riveting, or welding must be evaluated to determine the effective area. In the present study, it is assumed any element connection would maintain an effective net area equal to approximately 75 percent of the gross cross-sectional area for a selected structure shape. The design code could easily be modified to allow the user to input the effective net cross-sectional area parameters. The effective net area determination should be in accordance with LRFD specification, "Design Requirements," Chapter B., Section B3.[1] The limit equations for yielding and fracture are as follows:

a. For yielding on the gross cross-section:

$\varnothing_t = 0.9$

$P_n = F_y A_g$

b. For fracture in the effective net cross-section:

$\varnothing_t = 0.75$

$P_n = F_u A_e$

where $\varnothing_t$ = resistance factor corresponding to type of failure (yielding or fracture)

A_e = effective net area in.^{2}

A_g = gross area of the member, in.^{2}

F_y = specified minimum yield strength of material, ksi

F_u = specified minimum tensile strength of material, ksi

P_n = nominal axial strength, kips

In the design code, it is assumed that $A_e = .75 A_g$. Final design tensile strength or load, P_u, is determined by the smaller of the values of yielding or fracture loads. Final yielding and fracture design tensile strength is defined as follows:

$P_u = \varnothing_t P_n$

If a member force is negative, it is evaluated for compression yielding or buckling failure requirements.

Considerations for compression failure includes column slenderness ratio and cross section element slenderness requirements. The optimum member size is selected in a similar manner as described for tension failure. The optimization is accomplished by the selection of the lightest member for the required load capacity.

The slenderness ratio for compression members is subjected to the limitation:

$$K L / r \leq 200$$

where K = effective length factor

L = unbraced element length, in.

r = smaller radius of gyration of the member cross-section, inches

Additionally, the compressive truss member shall meet the cross-section element slenderness requirements, λ_r, for the flange and web as defined by LFRD specification.

Plastic analysis is permitted for compression members provided the column slenderness parameter (λ_c) does not exceed the value 1.5 K, where λ_c is defined as follows:

$$\lambda_c = (K L / r \pi) \text{ SQRT}(F_y / E)$$

where F_y = specified yield stress, ksi

E = modulus of elasticity, ksi

K = effective length factor

L = unbraced length of member, inches

r = smallest radius of gyration of cross-section, inches

The equation to determine critical stress (F_{cr}) for inelastic buckling, when λ_c is less than or equal to 1.5, for compressive truss members which meet the LRFD section element slenderness requirements, λ_r, is as follows:

$$F_{cr} = (0.658^{\beta})\ F_y$$

where F_{cr} = critical stress, ksi

$\beta = \lambda_c^2$

F_y = specified yield stress, ksi

For λ_c greater than 1.5, the critical stress (F_{cr}) is calculated from the following equation:

$$F_{cr} = (0.877 / \lambda_c^2)\ F_y$$

The nominal compressive design strength is given by the equation:

$$P_n = A_g\ F_{cr}$$

where P_n = nominal compressive design strength, kips

The ultimate design strength required for the member is:

$$P_u = \varnothing_c\ P_n$$

where $\varnothing_c$ = 0.85 (resistance factor for compression members). Once the member design process is complete the truss joint translations can be determined.

Equations describing element deflections are derived from equilibrium equations, compatibility equations, and Hooke's law for the analysis of a two-dimensional truss. This requires the determination of the statics matrix [A] and a deformation matrix $[A]^T$. The statics matrix defines the relationship between internal member forces and external loads for each joint in the truss. Hooke's law is used to define a matrix of element stiffnesses [S] for the truss. The member internal forces and joint deflections are then determined from the statics matrix, elements stiffness matrix, and the deformation matrix. The matrix [S] requires the area of each truss member and can only be determined once all members are sized. However, ''Method of Joints'' technique does not require member cross-sectional areas to determine internal forces. The method chosen for the design code greatly simplifies what would otherwise be an iterative analysis/design process for two dimensional trusses.

The design code determines the cross-sectional area based on the type of failure predicted for each truss member. This area is then used in the determination of the element stiffness matrix [S]. Hooke's law expresses the relationship between internal forces and elongations of truss members as:

$$F = (A\ E\ /\ L)\ e$$

where F = internal force, kips

A = cross-sectional area, in.^{2}

E = modulus of elasticity, ksi

L = member length, inches

e = member elongation, inches

The element stiffness is the A E/L term in the above equation. This equation in matrix notation for the truss is written as:

$$[F] = [S]\ [e]$$

where [S] is defined as a diagonal matrix of the element stiffnesses.

$$[S] = \begin{bmatrix} A_1E_1/L_1 & 0 & 0 & 0 \\ 0 & A_2E_2/L_2 & 0 & 0 \\ ***** & ***** & ***** & ***** \\ 0 & 0 & ***** & A_nE_n/L_n \end{bmatrix}$$

The equilibrium expression and the compatibility expression for the truss are defined as:

a. Equilibrium equation

$$[P] = [A]\ [F]$$

b. Compatibility equation

$$[e] = [B]\ [X] = [A]^T\ [X]$$

where [B] is defined as the deformation matrix and is equal to the transpose of the statics matrix [A]. The deformation matrix [X] is the matrix of joint deflections. The external joint deflections can be determined from the expression:

$$[F] = [S][e]$$

Substituting the compatibility equation for [e] produces;

$$[F] = [S][A]^T\ [X]$$

If this expression is substituted into the equilibrium equation for [F], the external loads are defined as:

$$[P] = [A][S][A]^T[X]$$

or as:

$$[P] = [ASA^T][X]$$

where the matrix $[ASA^T]$ is defined as the truss external stiffness matrix [K].

The joint deflections [X] are determined by the multiplication:

$$[P] = [K][X]$$

$$[K]^{-1}[P] = [K]^{-1}[K][X]$$

This produces the deflection equation used in the program:

$$[X] = [K]^{-1}\,[P]$$

The final step in the truss analysis is the determination of end constraint forces or reactions. These forces are obtained by equilibrium conditions at the constrained joints, and are simply the summation of internal forces of all elements at the constrained joint. The determination of restraint forces is obtained by the same technique used for the development of the statics matrix [A]. The restraint forces are solved by employing the static equilibrium equations at the restrained joint for both x and y directions as required.

An overview of the techniques used in the design code for planar truss analysis and member sizing has been presented in this chapter. The design analysis includes the determination of: required internal forces; ultimate strength for tension and compression members; deflections; and restraint forces. The analysis results from the code are saved in a file labeled TRUSS.OUT and include member loads, member sizes with associated properties, deflections, and reactions.

CHAPTER 5

METHOD FOR TREATING SINGULAR COLUMNS

An option provided in the design code permits the selection of a single axially-loaded member for design and analysis. This option is introduced to the user during the initialization of the program. The user is required to specify if a truss system or single column design option is desired. In the single column design option, a member is evaluated for tension or compression failure in the same manner as described for truss analysis except the member data is entered by the user interactively. The required inputs include effective length, applied axial loading, and the cross-sectional axis of bending to be used in the evaluation. A compression member requires an additional input of the effective length factor (K). The member sizing calculations are accomplished by the code in the same manner as described for each mode of failure for truss members.

For tension failure, the program determines the design tensile strength and utilizes the same database of geometric properties to select the optimum column size. The results of the tensile design calculations are printed to the screen.

As stated for compression failure, the code requires the user to enter an effective length factor (K) for the column. This permits the use of effective length factor for columns with end conditions that are not pinned-pinned. The K factors assumed for the truss were for the pinned-pinned column end condition. Examples of other end conditions for a column include rotation and translation fixed, rotation free and translation fixed, rotation fixed and translation free, and finally rotation and translation free. The LRFD specifications in, Chapter C, "Frames and other structures"[1], provides recommendations for the use of K values of columns with various end conditions. The effective length factor can also be approximated using the following equation: [3]

$$K = (3\ G_a + 1.4) / (3\ G_a + 2.0)$$

where G_a = the top joint rigidity[5] value for a column in a frame. G_a is determined by:

$$G_a = (\Sigma\ I_c / L_c) / (\Sigma\ I_b / L_b)$$

where Σ = summation

I_c = moment of inertia of all columns at the joint, $(\text{inches})^4$

L_c = length of all columns at the joint, inches

I_b = moment of inertia of all beams at the joint, $(\text{inches})^4$

L_b = length of all beams at the joint, inches

Figure 5.1 provides an example of the nomenclature for the determination of K for columns in a frame system. G_a for column AB in Figure 5.1 is as follows:

$$Ga = (I_c / L_c) / ((I_{b1} + I_{b2}) / (L_{b1} + L_{b2}))$$

Using this method, the approximation of K is accurate to within 2 percent as compared to Newton's method of successive approximations for the exact solution[3] of the effective length factor K. This approximation is a convenient tool for the determination of K in a framed system. The design code uses the entered K value in place of 1.0 as assumed in the truss system option where the effective length factor was based on columns with hinged ends. The program determines the ultimate compressive design strength and selects the optimum structural member size to carry the required load.

In summary, the single member analysis option provides more flexibility in the use of effective length factors, which gives the user the ability to evaluate single axially loaded members of a frame system. The results printed to the screen include member size and load capacity.

CHAPTER 6

MEMBER SIZE OPTIMIZATION

As previously discussed, the design code selects the optimum structural size in the design analysis for its particular mode of failure. This approach ensures a design that satisfies all the necessary maximum capacity requirements necessary for a member while minimizing its weight. This approach establishes the best balance between size and stress. For example, an element of a truss system has many selections possible for a feasible structural shape and size to meet the required design strength. If the restriction is to select the lightest weight element capable of carrying the required ultimate design load, then the truss element is optimized for its intended usage. This study recognizes there are techniques available to determine a generalized structure shape that can carry the required maximum capacity of a member. However, the design optimizations of the present study uses a techique of searching among several standard structural shapes for a shape that minimizes element mass. That is, the optimum design process incorporated in the computer code is based on selecting a standard structural shape to carry the required design capacity that limits the member to a minimum weight. This optimum definition minimizes the cost of the truss since industry purchases structural shapes by weight. Therefore, the design code is a valuable practical tool for use by industrial design groups in general.

As described previously, this design code has the ability to optimize axially loaded members using standard structural shapes based on geometric properties published in the LRFD manual. This is accomplished by incorporating an internal database of structure shapes which the code accesses during the member sizing process. The required geometric properties included in the database are member size, shape, weight, cross-sectional area, flange-thickness ratio, radius of gyration about both axis of the cross-section, and web-thickness ratio. The data base is arranged with the cross-sectional areas in ascending order. Since the area (assuming constant cross sections) relates to weight, the member sizing process starts with the smallest area and attempts to verify if each member can achieve the required design capacity. If the required design capacity cannot be achieved, the next member size in the database is investigated. This process provides a simple iterative technique for ensuring the minimum weight of structural members is selected.

CHAPTER 7

DESIGN CODE PREPARATION

The design code requires input data including node coordinates, element connectivity, end constraints, and external applied loads to analyze a truss system. The code is written in FORTRAN and utilizes an internal data file (NODE.DAT) defining node numbers, x and y coordinates of nodes, and element connectivity. After the code determines lengths and direction cosines, the end constraints and applied loads are interactively entered by the user during the execution of the code. It is necessary for the user to understand the node coordinate numbering system and its relationship to the input data to construct and analyze a truss system.

For a truss system, the code requires a coordinate system be established which includes labeling of node numbers, labeling of the element numbers, end constraints, applied loads, and deflections. Figure 7.1 illustrates a truss system that has six nodes and nine elements. The code employs an internal load/deflection numbering system that is based on the node numbers. For the x and y direction of loads and deflections, the numbering system establishes the x direction being odd integers and the y direction being even integers. The calculated value n for a node is for load designation P(n) and for deflection designation DELTA(n) where n is the distinct numerical label for each load or deflection at a particular node. The numerical labeling for the load and deflection numbering system is established as follows:

a. Numerical label for the x direction components

$$n = 2(\text{Node Number}) - 1$$

b. Numerical label for the y direction components

$$n = 2(\text{Node Number})$$

For example, Node number 6, as illustrated in Figure 7.1 would result in the following numerical labeling of the load and deflections:

For applied loads in the x direction

$$P(n) = P([(2)(6)]-1) = P(11)$$

For applied loads in the y direction

$$P(n) = P([(2)(6)]) = P(12)$$

The deflections are defined similarly: DELTA(11) is in the x direction and DELTA(12) is in the y direction.

The code requires an internal file, NODE.DAT, be developed to provide the data for the construction of the truss model. The truss configuration requires node data be established which includes node number, x coordinate, and y coordinate for each node. Each node data point is entered on one line per node with a blank between each entry. Each entry is in free format with nodes being an integer and the decimal coordinates having units of inches. All node data must be entered before entering the element connectivity information in the internal data file. The required data entries for the element connectivity includes element number, beginning node, and ending node. It is important the element connectivity data follows the nodal data so the code continues to read the input data in the proper sequence. The element connectivity information establishes the element number and then node connectivity by establishing the higher node number as the beginning node and the ending node as the smaller node number. The element connectivity data requires the element number be followed by a beginning node number and ending node number with one line per element and each entry separated with a blank space. The following is an example of node and element connectivity representing the input data to NODE.DAT. The list includes the node, the x and y coordinates, as nodal information in the first six entries, and element connectivity information for the last nine entries which include element numbers, its beginning node and ending node. The input data is as follows for truss in Figure 7.1:

```
1 0 0
2 180 0
3 360 0
4 540 0
5 180 240
6 360 240
1 2 1
2 3 2
3 4 3
4 5 1
5 5 2
```

6 6 2
7 6 5
8 6 3
9 6 4

Entering data for constrained nodes is accomplished interactively. The code fixes the directional end condition at the appropriate node. This is illustrated for the truss in Figure 7.1, where node 1 has fixed conditions in both directions (x and y) and node 4 has fixed condition for y direction only. The appropriate entry for node 1 (upper case letters) is as follows:

1,DX,DY

and the entry for node 4 is as follows:

4, ,DY

This defines the necessary end constrains for each joint. The design code reads the node, then the direction of the fixed constraint designated by DX and DY separated by commas. The blank space for node 4 designates the x direction is free.

The applied load data is interactively entered in a similar manner as end constraints. The user enters the node number, load direction and the applied loads in units of kips. The code establishes a matrix of the x and y components of the N applied forces for each node. The code internally establishes a zero load for the x and y components of nodes not entered during execution of the applied loads entry. This is illustrated for the truss in

Figure 7.1; node 5 has x and y applied load components and node 6 has a y applied load component only. The loads are defined in units of kips. The appropriate entry for the node (uppercase letters) is as follows:

```
5,PX,30,PY,-70
```

and entry for node 6 is as follows:

```
6, , ,PY,-60
```

The code reads the data in order of node, the direction, and the applied load. Labels PX and PY are designating the coordinate direction of the load followed by its numerical amplitude with algebraic sign indicating direction of load. The applied loading for node 5 reads: a 30 kips load is being applied on the truss in the positive x direction, and 70 kips load is being applied in negative y direction. At node 6, the blank space after the node number separated by commas indicates that no load is applied for the x component of the node. The only load being applied on the node is a negative y direction of 60 kips. The force and deflection vector components shown in Figure 7.1 are positive.

It is recommended the described steps be followed to be sure the entered data is consistent with the truss configuration and in the order required by the code. These steps are intended to help prepare and check the required input data for a truss system and therefore, reduce input errors. A summary of the recommended steps are as follows:

1. Sketch the truss configuration. Establish the coordinate system by labeling joint nodes and member elements as illustrated in Figure 7.1.
2. Prepare the file NODE.DAT to establish the node coordinates and element connectivity for the truss configuration. The node data must have three entrees per line, the node number, x and y coordinates. A free format style of input can be used for the data. It is important to enter coordinates in units of inches because of design analysis requirements. (**NOTE:** Ensure all node data has been entered before continuing with input of the element connectivity.) Prepare the element connectivity for the truss system. The element data has three entries per line, the element number, the beginning node, and the ending node in a free format input. The code assumes these data entries are integers numbers. The order of node input for each element is the higher value node being the beginning node and lower value being the ending node.

After the proper truss model data input file (NODE.DAT) has been established, then the program can be executed. All additional input data must be interactively entered during execution.

This chapter described the preparation of the truss model input data and the requirements for the proper execution of the design code. The proper units for lengths and loads employed by the code were defined. Figure 7.1 provided a general truss configuration and illustrated the required numbering system for all data used by the code. Internal data file NODE.DAT is required to execute the design code and is followed with interactive inputs of end constraints and applied loads during execution.

CHAPTER 8

PROGRAM EXECUTION

The operating sequence of the main program is illustrated by the flowchart in Figures 8.1 - 8.4. To execute the design code, the user chooses between ''truss analysis'' or ''single member'' options. The requirements for each were discussed in previous chapters.

Once the program and data files are saved on the computer, the execution of the design code is initiated by typing LRFD. The user is then requested to choose the design option by entering either ''TR'' or ''SC.'' ''TR'' begins analysis of a truss system, and ''SC'' begins analysis of a single member. For the truss system, the code responds requiring interactive input for the number of nodes and the number of elements. The next interactive input is to define which cross-sectional axis to use in the design of all truss elements. Selection of ''1'' tells the code to evaluate the member about the strong axis, and ''2'' to evaluate about the weak axis of the member. The code selects the appropriate radius of gyration from the geometric properties database as required. Strong axis is defined as the largest radius of gyration, and the weak axis is the smallest. The code then determines the lengths and direction cosines for the truss configuration.

Once the truss end constraints are defined, the code determines the statics matrix [A], then solves for the internal loads in each truss element using Gauss-Jordan elimination.

Once the internal forces are computed, the code calculates failure loads and determines the member optimum structural size. The geometric properties established for the elements are used to determine the element stiffness matrix for the truss. The design code then determines the truss external stiffness matrix and deflections for each node in the truss.

At this point the only unsolved variables are the end restraint loads (reactions). They are determined by summing the internal member loads at the constrained joints and setting the unbalanced forces equal to the restraint forces. For the truss system, this completes the solution process for analysis and design of the truss members and output is sent to a file.

The truss analysis report is saved as an internal file called TRUSS.OUT. The truss analysis report includes input data for nodes, node coordinates, element connectivity, applied loads, and lengths of elements. Analysis results in the report include internal member forces, selected member size, member cross-sectional area, and radius of gyration. The report provides under separate heading the solutions for deformations and reactions for the truss.

The single member sequence will require the selection of either tension(TN) or compression failure (CP) mode for design requirements. This design option requires interactive inputs for length, applied loads, and cross-sectional axis to be used in design for both types of failure. The compression design option requires the additional input of the effective length factor as discussed in Chapter 5. The results printed to the screen include applied load, member size, and member design capacity.

CHAPTER 9

VALIDATION OF DESIGN CODE

For validation of the design code, three truss systems were evaluated: a three member truss system; a four member cantilever truss system; and a nine member truss system. The design code predicted internal loads, deformations, and restrained reactions were verified for accuracy against similar results obtained from a general purpose structural finite element code. The optimum size selection (design analysis) was verified by manual calculation. A 19 element truss system was also evaluated to verify the maximum model size (that was minimized for validation purposes) that could be solved by the design code. The maximum model size which can be solve by the design cose can easily be increased by changing all subroutine dimension statement to the disired size.

Verification of the predicted internal loads, deformations, and restraint reactions were made by using a commercial finite element code known as ANSYS® distributed by Swanson Analysis Systems, Inc.[6] A comparison of the design code and ANSYS results for the three truss systems are shown in Tables 9.1 - 9.3. The corresponding truss configurations are shown in Figure 9.1 - 9.3. As shown in the tables, excellent agreement was obtained between the design code and ANSYS for all three test cases. Documentation of the ANSYS solutions and the design code solutions are shown in Appendix A. Included in Appendix A are the manual calculations made to check the optimum size selection process made by the design code.

The validation and verification process shows the design code is accurate in determining loads, deformations, and selecting the optimum member size. This conclusion was supported by the ANSYS analysis and manual calculations for each truss system.

CHAPTER 10

CONCLUSIONS AND RECOMMENDATIONS

In conclusion, the design code developed in this study was based on a new design approach, known as limit states of strength, for the design of truss systems. The code conforms to AISC LRFD specifications for tension, compression, and buckling modes of truss element failure using this limit states concept. This study developed a two-dimensional design code for truss systems that may be used to size truss or single axially loaded members. Future work is planned to expand the design code to analyze three-dimensional truss systems. Presently, the internal geometric properties database is limited to wide flange structural shapes which were used to verify the design code. The geometric properties database, however, may be expanded to utilize other structural shapes and built-up sections if the appropriate data is added by the user.

The employment of the LRFD philosophy is an advantage for designers because it is based on a rationale that provides more flexibility of options to the user for structural design. The LRFD philosophy permits more economical use of material in analyzing load combinations and designing truss configurations. In Appendix B, the data presented demonstrates that by using LRFD philosophy the load carrying capacity of a member is much greater than that allowed by the ASD method because LRFD allows more of the reserve strength of the member cross-section to be taken into account in strength determination. For example, as shown in Appendix B, a sixteen foot standard wide flange W8 X 48 pound per foot column by ASD design, has a design allowable capacity of 254.6 kips compared to the LRFD design allowable capacity of 287.5 kips.

The present study developed a design code for two-dimensional truss structures optimizing structural members or elements for minimum weight while ensuring an acceptable margin of safety against structural member failure. The code provides for selection of structural shapes for axially loaded members of statically determinate, two-dimensional truss systems. The code provides a summary of design and analysis results for the truss including element loading, member design incorporating the LRFD limit state philosophy, deformations, restraint reactions, and the optimization of the truss member sizes. This study accomplished the objective of providing a design code for planar truss systems. Additionally, a feature is provided to select an optimum structural size of a member which adheres to the limit state design philosophy as defined by AISC in the LRFD manual.(1)

In conclusion, it is recommended for future work that the two-dimensional code be expanded to a three-dimensional truss systems. A three-dimensional truss system will require the z coordinate be included with the x and y coordinate data. This will require some revisions be made to the input of nodal data.

Another future enhancement to the design code can be accomplished by expanding the internal geometric properties data base. This will require the design code be revised to recognize additional properties such as S shapes, angles, and built-up sections. The revision to the database will require inputs of size, shape symbols, weight per foot, area, cross-sectional element width-thickness ratios, and radius of gyration about each cross-sectional axis for each member.

BIBLIOGRAPHY

BIBLIOGRAPHY

AISC. Load and Resistance Factor Design Specification for Structural Steel Buildings, Chicago,IL: American Institue of Steel Construction, September 1, 1986

AISC. Manual of Steel construction, Allowable Stress Design, 9th ed. Chicago, IL: American Institue of Steel Construction, 1989

Wang, Chu-Kia. Matrix Methods of Structural Analysis. Scranton: International Textbook Company, 1970

Kirby, P.A. and Nethercot, D. A. Design for Structural Stability. New York: Halsted Press Book, 1979

Beaufait, Fred W., Rowan,Jr., William H., Hoadley, Peter G., Hackett, Robert M. Computer Methods of Structural Analysis. Nashville: Beaufait/Rowan/Hoadley/Hackett, 1975

Emmons, Howard W. and Budiansky, Bernard. Optimization Methods for engineering Design. Reading, Massachusetts: Addison-Wesley Publishing Company, 1971

Salmon, Charles G. and Johnson, John E. Steel Structures; Design and Behavior. New York: Harper and Row, Publishers, 1989

Pierre Dumonteil, ''Simple Equations for Effective Length Factors,'' Engineering Journal, AISC, (Third Quarter, 1992)

William E. Moore II, ''Discussion - Simple Equations for Effective Length Factors,'' Engineering Journal, AISC, 10, 1, (First Quarter, 1993),37

Guide to the Preparation of Theses and Dissertations. Knoxville: The University of Tennessee, 1991

ANSI. American National Standards Minimum Design Loads for Buildings and Other Structures, A58.1-1982, New York: American National Standards Institue,Inc.,1982

ANSYS User's Manual Revision 5.0, Swanson Analysis Systems, Inc., Houston,PA: Swanson Analysis Systems,inc., 1992

LIST OF REFERENCES

LIST OF REFERENCES

(1) AISC. Load and Resistance Factor Design Specification for Structural Steel Buildings, Chicago,IL: American Institue of Steel Construction, September 1, 1986

(2) Wang, Chu-Kia. Matrix Methods of Structural Analysis. Scranton: International Textbook Company, 1970

(3) William E. Moore II, ''Discussion - Simple Equations for Effective Length Factors,'' Engineering Journal, AISC, 10, 1, (First Quarter, 1993),37

(4) ANSI. American National Standards Minimum Design Loads for Buildings and Other Structures, A58.1-1982, New York: American National Standards Institue,Inc.,1982

(5) Salmon, Charles G. and Johnson, John E. Steel Structures; Design and Behavior. New York: Harper and Row, Publishers, 1989

(6) ANSYS User's Manual Revision 5.0, Swanson Analysis Systems, Inc., Houston,PA: Swanson Analysis Systems,Inc., 1992

APPENDIX A

FIGURES AND TABLES

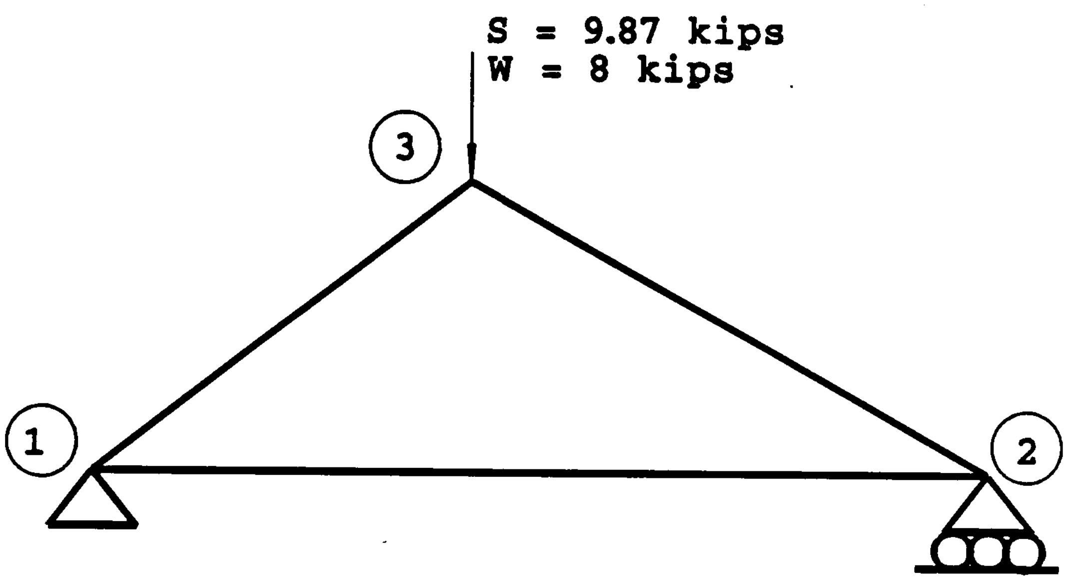


Figure 2.1 LRFD Truss Loading Diagram

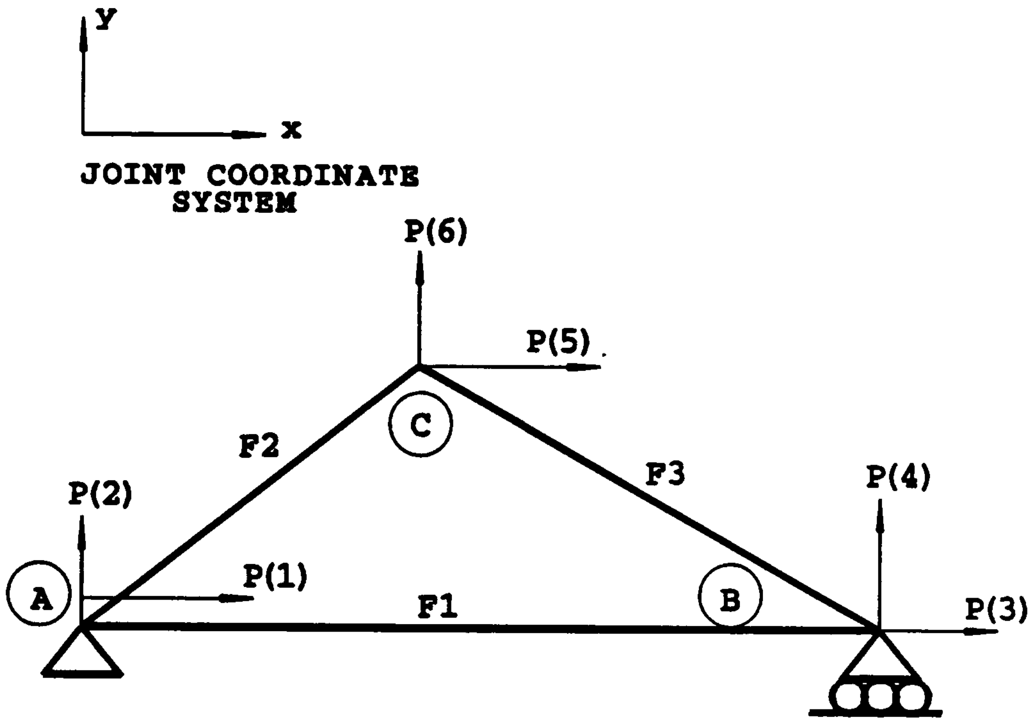


Figure 4.1 Truss Internal-External Force Relationship

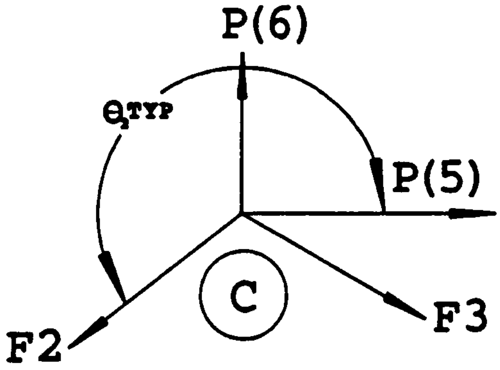


Figure 4.4 Joint C Static Equilibrum Diagram

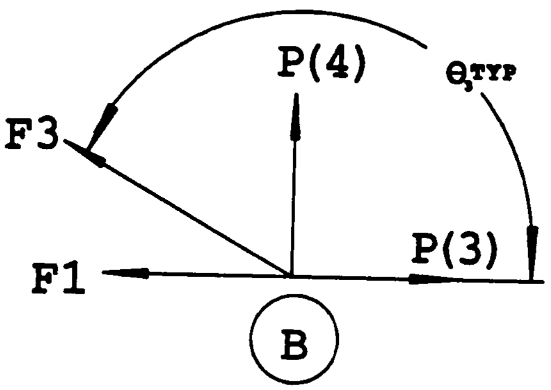


Figure 4.3 Joint B Static Equilibrum Diagram

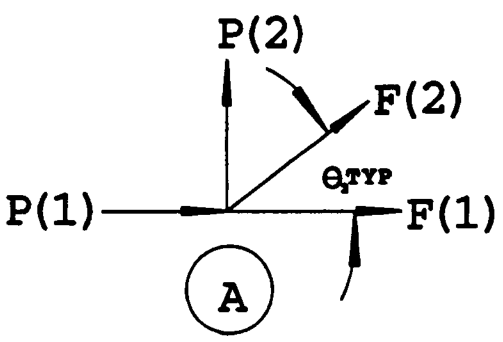


Figure 4.2 Joint A Static Equilibrum Diagram

delta(5) = Joint Deflection

P(5) = External Load

2 -Node Number

1 -Element Number

R's are reaction forces

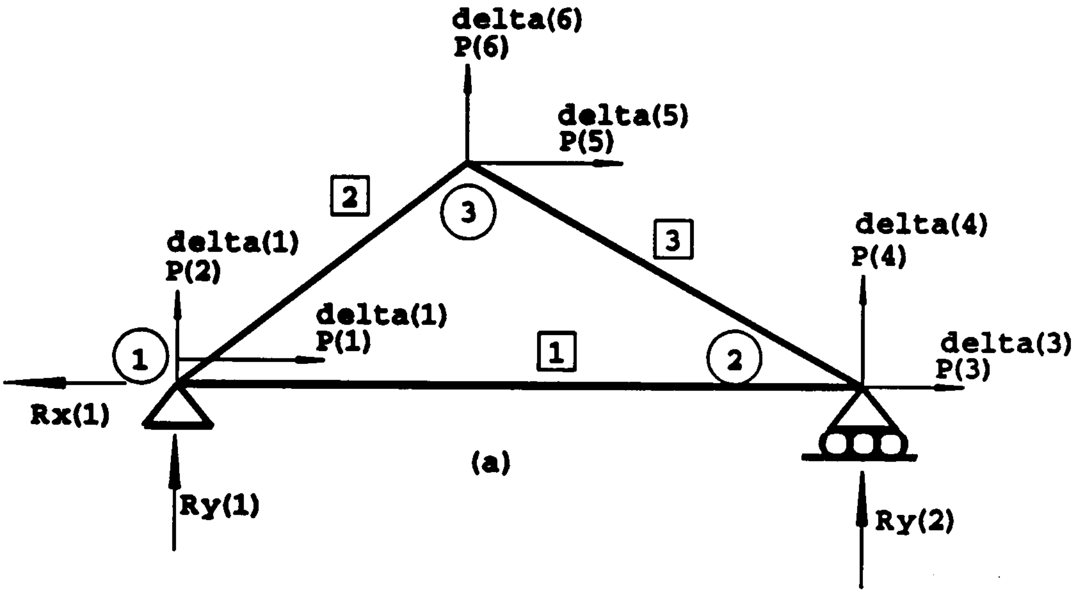


Figure 4.5 Typical Three Element Truss

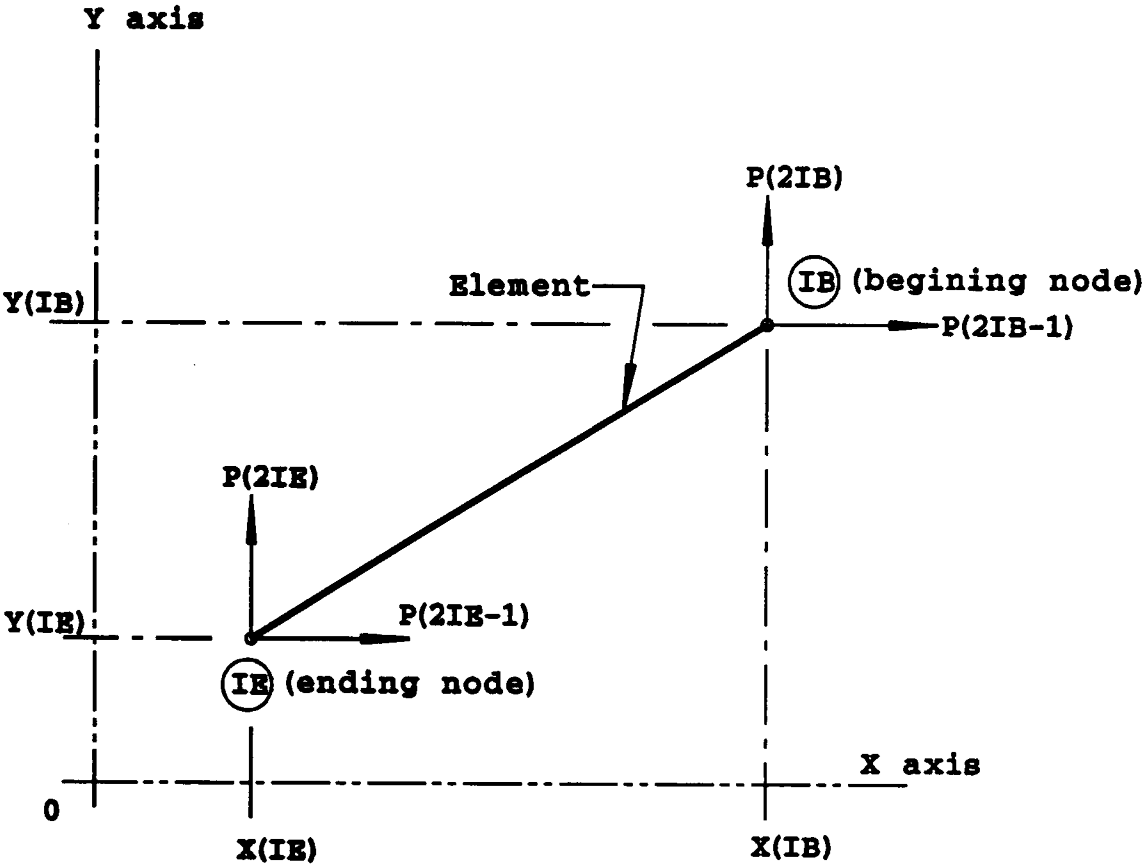


Figure 4.6 Generalized Element Configuration

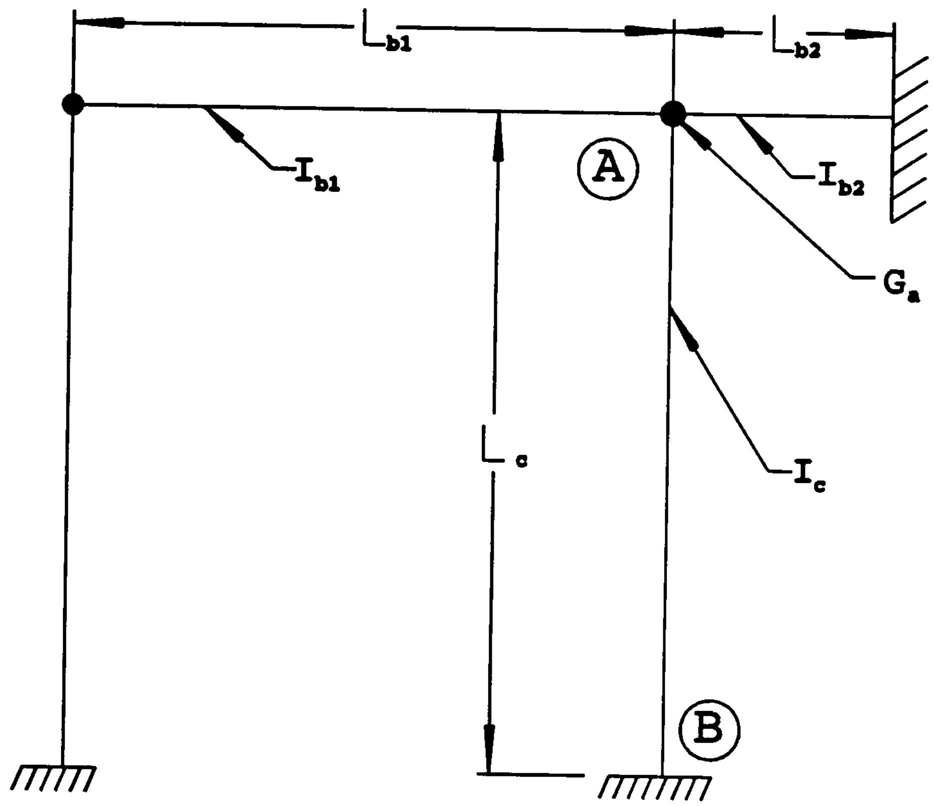


Figure 5.1 Frame Nomenclature for K Factor

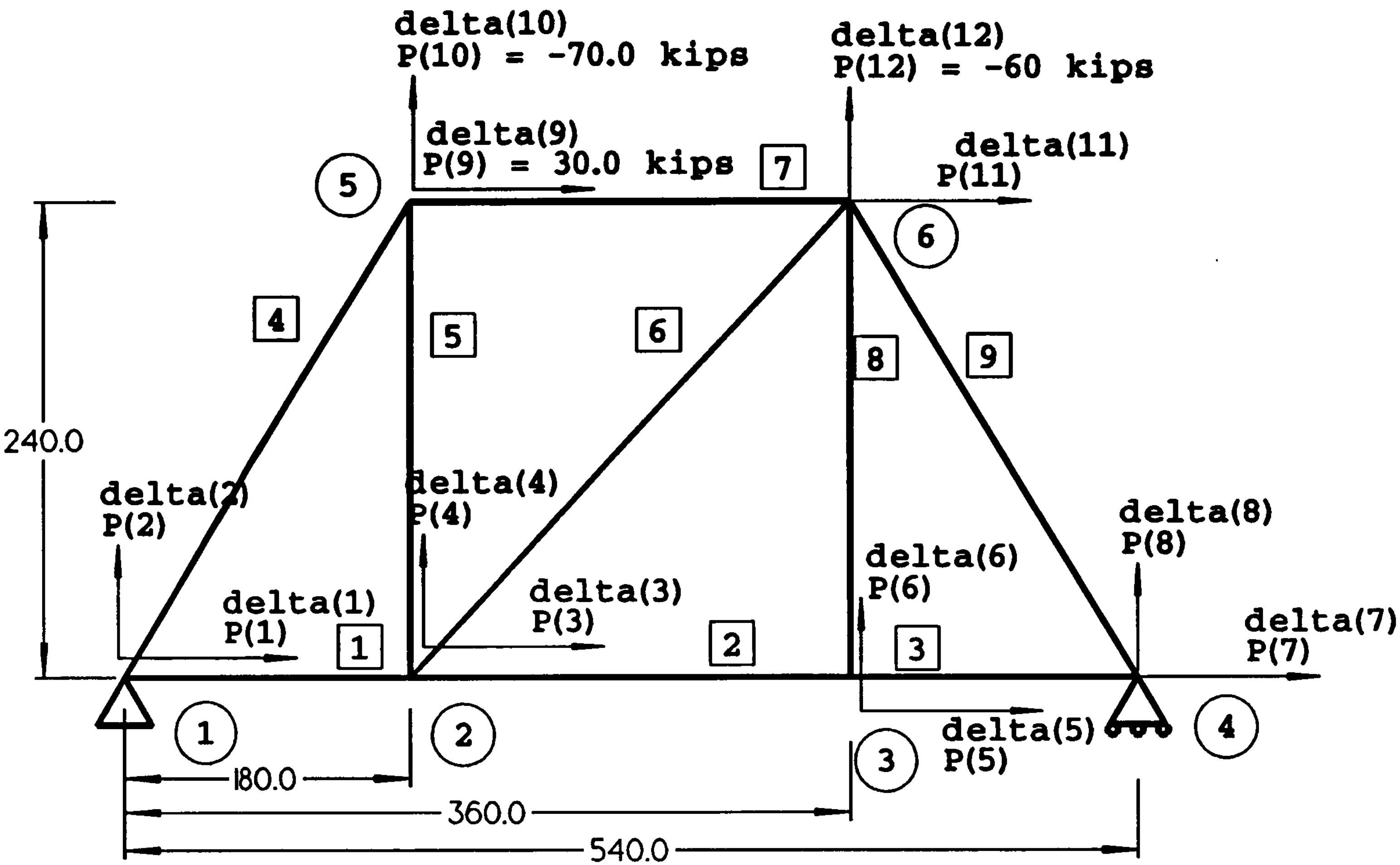


Figure 7.1 Numbering System for Truss Analysis

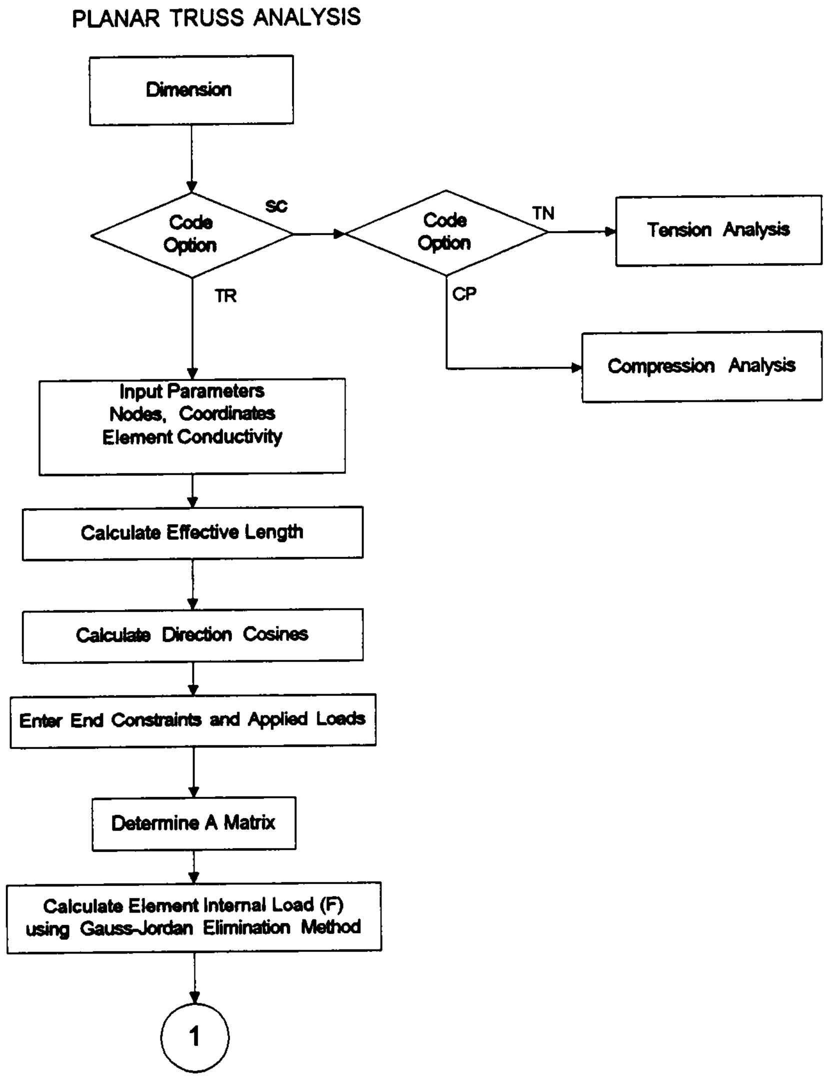


Figure 8.1 Design Code Flow Chart

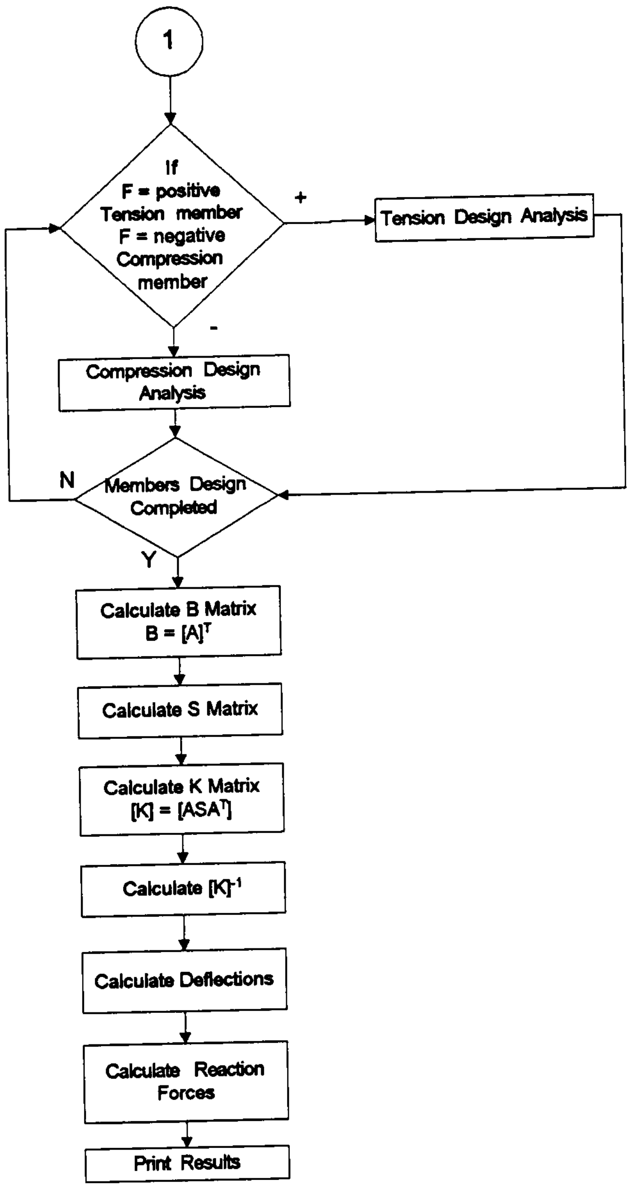


Figure 8.2 Design Code Flow Chart (continue)

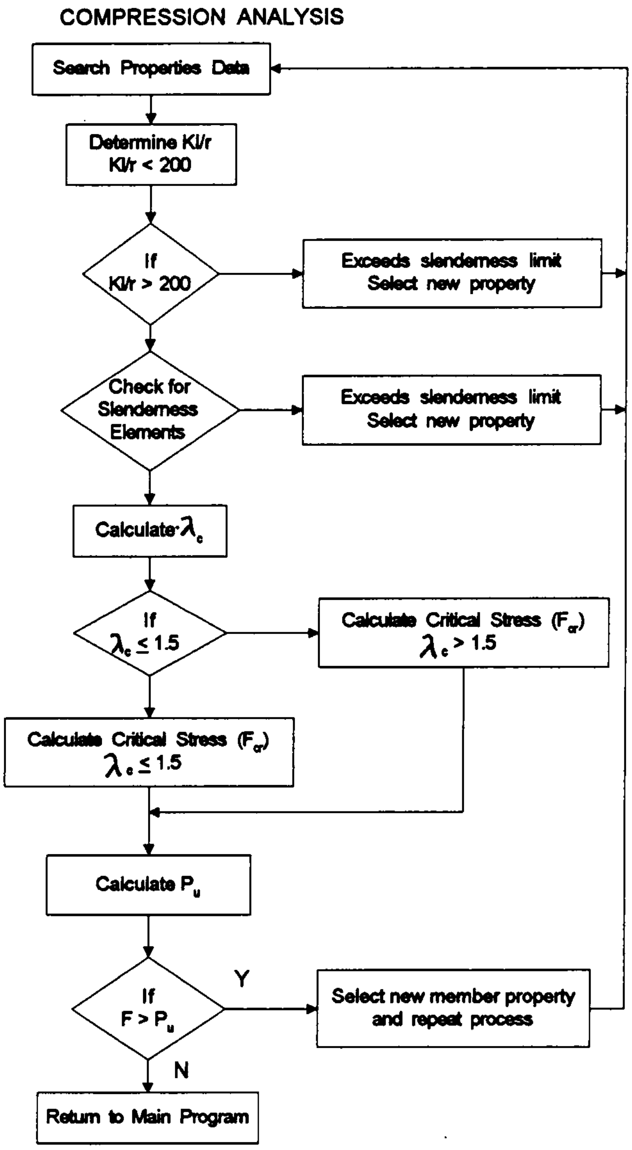


Figure 8.3 Tension Failure Design Flow Chart

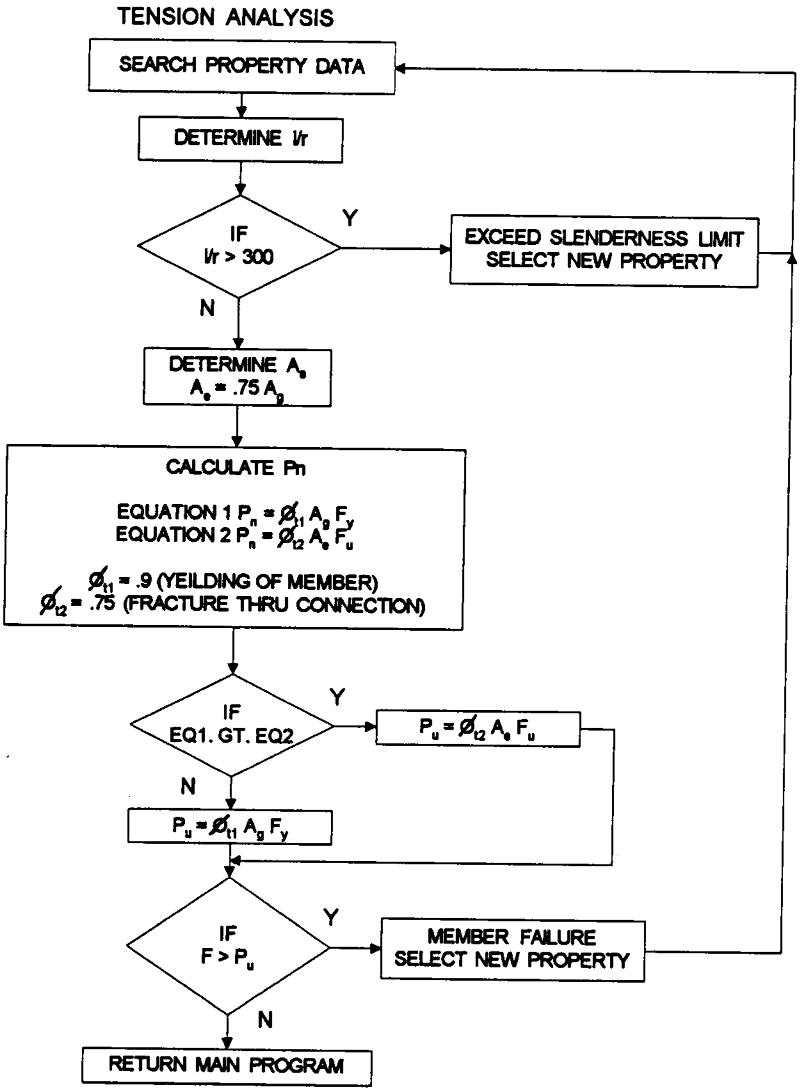


Figure 8.4 Compression Failure Design Flow Chart

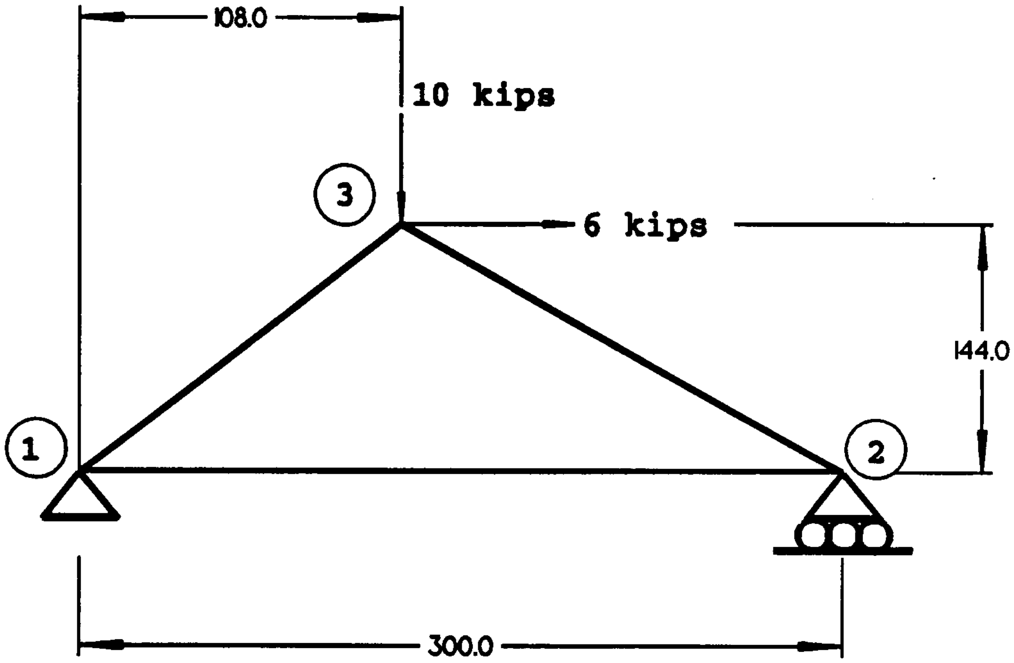


Figure 9.1 Case 1 - Simple Truss

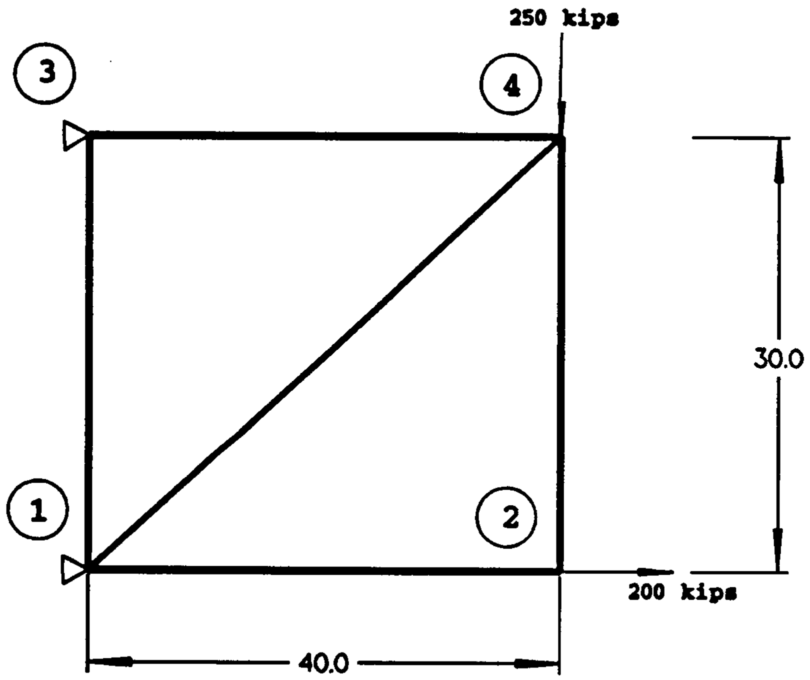


Figure 9.2 Case 2 - Cantilever Truss

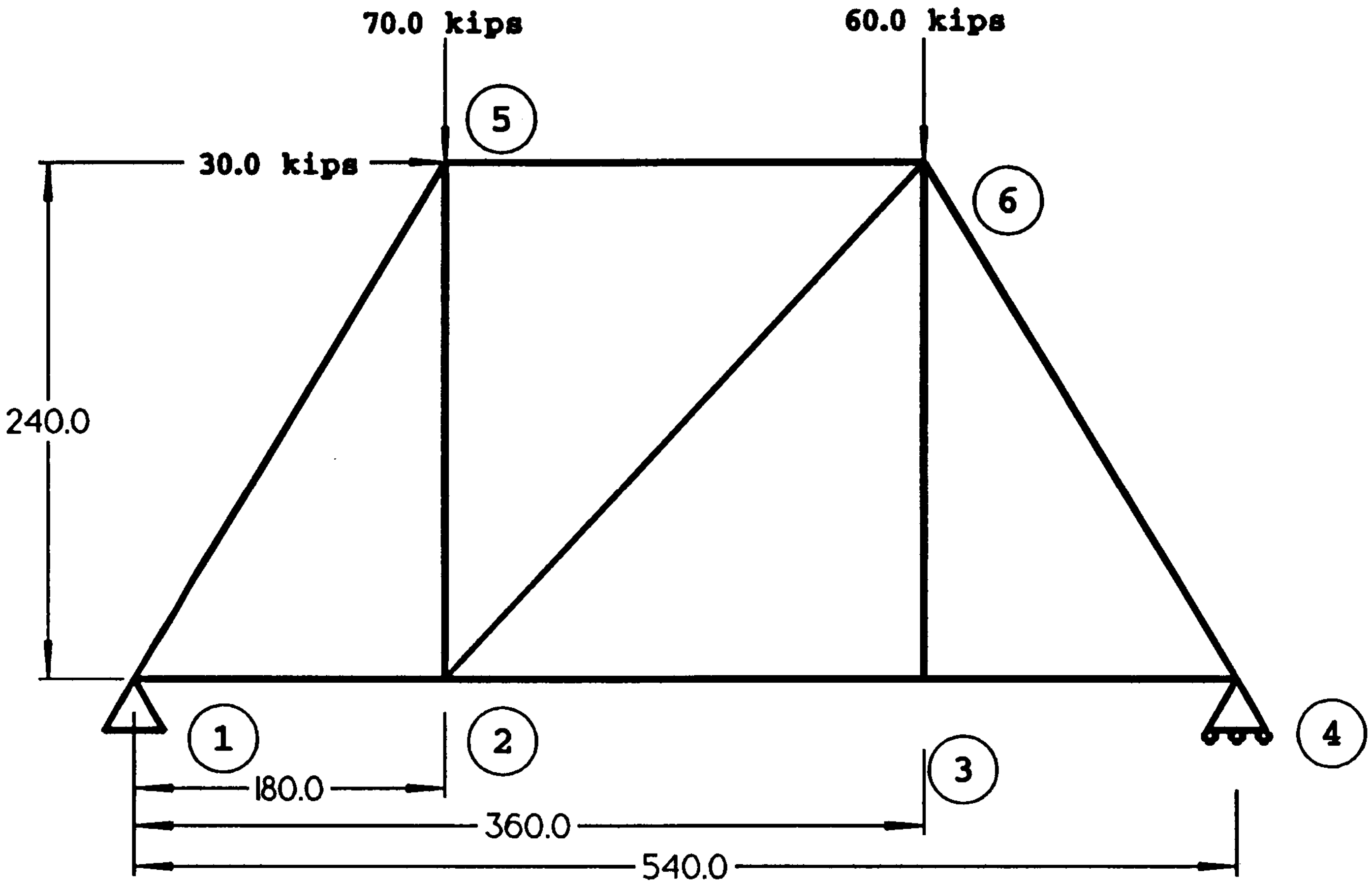


Figure 9.3 Case 3 - Single Panel Truss

Table 9.1 CASE 1 - ANSYS vs Design Code Comparision
3 Element Truss

Element	Internal Load (kips)		Node	Deflections (inches)				Restraint Loads (kips)			
				X - Direction		Y - Direction		X - Direction		Y - Direction	
	ANSYS	Code		ANSYS	Code	ANSYS	Code	ANSYS	Code	ANSYS	Code
1	8.64	8.64	1	0.0	0.0	0.0	0.0	-6.0	-6.0	3.52	3.52
2	-4.40	-4.40	2	0.03335	0.03335	0.0	0.0	0.0	0.0	6.48	6.48
3	-10.80	-10.80	3	0.04191	0.04191	-0.04417	-0.04417	0.0	0.0	0.0	0.0

Table 9.2 CASE 2 - ANSYS vs Design Code Comparision
4 Element Truss

Element	Internal Load (kips)		Node	Deflections (inches)				Restraint Loads (kips)			
				X - Direction		Y - Direction		X - Direction		Y - Direction	
	ANSYS	Code		ANSYS	Code	ANSYS	Code	ANSYS	Code	ANSYS	Code
1	200	200	1	0.0	0.0	0.0	0.0	133.3	133.3	250.0	250.0
2	-416	-416	2	0.042571	0.04257	-0.26372	-0.26371	0.0	0.0	0.0	0.0
3	0.0	0.0	3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
4	333.3	333.3	4	0.078952	0.07095	-0.26372	-0.26371	-333.3	-333.3	0.0	0.0

Table 9.3 CASE 3 - ANSYS vs Design Code Comparision
9 Element Truss

Element	Internal Load (kips)		Node	Deflections (inches)				Restraint Loads (kips)			
				X - Direction		Y - Direction		X - Direction		Y - Direction	
	ANSYS	Code		ANSYS	Code	ANSYS	Code	ANSYS	Code	ANSYS	Code
1	70.0	70.0	1	0	0.0	0	0.0	-30.0	-30.0	53.33	53.33
2	57.5	57.5	2	0.16212	0.16212	-0.55719	-0.55719	0.0	0.0	0.0	0.0
3	57.5	57.5	3	0.29529	0.29529	-0.53754	-0.53754	0.0	0.0	0.0	0.0
4	-66.67	-66.67	4	0.42846	0.42846	0.0	0.0	0.0	0.0	76.667	76.667
5	-16.67	-16.67	5	0.41673	0.41673	-0.60379	-0.60379	0.0	0.0	0.0	0.0
6	20.83	20.83	6	0.26995	0.26994	-0.53754	-0.53754	0.0	0.0	0.0	0.0
7	-70.0	-70.0									
8	0.0	0.0									
9	-95.83	-95.83									

APPENDIX B

DESIGN CODE VALIDATION

////////////////LRFD TRUSS ANALYSIS REPORT//////////////////

TRUSS INPUT DATA EFFECTIVE LENGTH FACTOR (K) IS EQUAL TO 1.0

COORDINATE DATA

NODE	X(in.)	Y(in.)
1	.00	.00
2	300.00	.00
3	108.00	144.00

ELEMENT CONNECTIVITY

ELEMENT	BEGINNING NODE	ENDING NODE	LENGTH
1	2	1	300.00
2	3	1	180.00
3	3	2	240.00

EXTERNAL APLLIED LOADS

LOAD NUMBER	LOAD(kips)
1	.00
2	.00
3	.00
4	.00
5	6.00
6	-10.00

TRUSS ANALYSIS RESULTS

INTERNAL MEMBER AXIAL LOAD

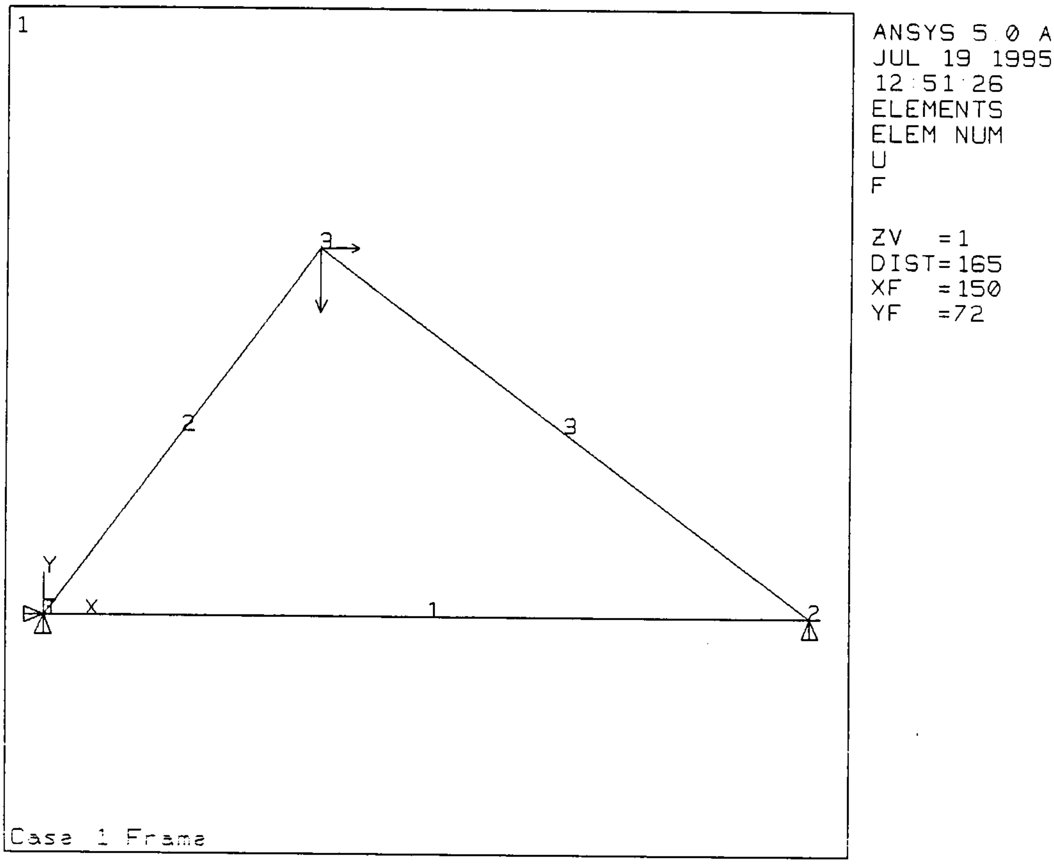
ELEMENT	LOAD(kips)	MEMBER SIZE	AREA(sq. in.)	RADIUS OF GYRATION(in.)	
1	8.6400	6 W 9.0	2.68	2.5	
2	-4.4000	6 W 9.0	2.68	2.5	
3	-10.8000	6 W 9.0	2.68	2.5	

DEFORMATION(in.)

DELTA(1) = .000000000000
DELTA(2) = .000000000000
DELTA(3) = .033350490551
DELTA(4) = .000000000000
DELTA(5) = .041910448389
DELTA(6) = -.044170871263

RESTRAINT LOADS(kips)

RX(1) = -6.0000
RY(1) = 3.5200
RY(2) = 6.4800

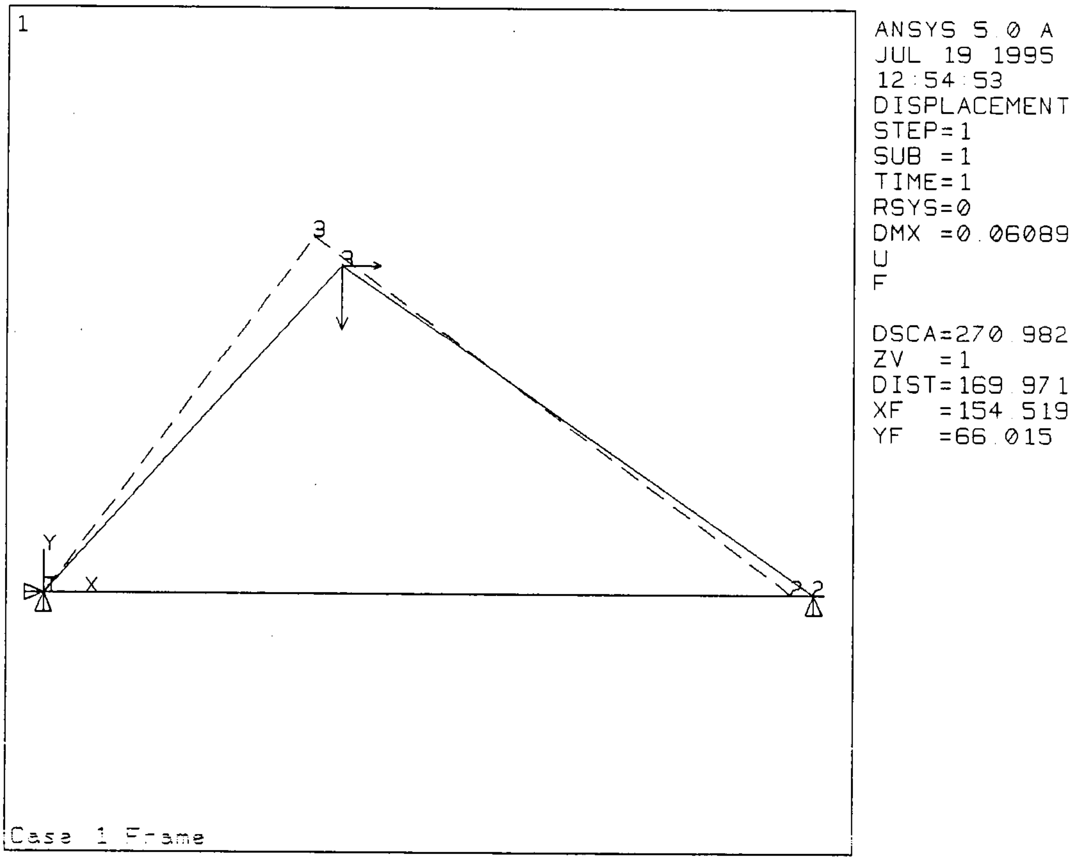

1
ANSYS 5.0 A
JUL 19 1995
12:51:26
ELEMENTS
ELEM NUM
U
F
ZV =1
DIST=165
XF =150
YF =72
3
2
3
Y
X
1
2
Case 1 Frame

```
                                         EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
--------------------------------------------------------------------------------
----------------------------------------------------------------------
Freeing printer buffer #1 at segment 7303
PRINT MADE. Print time was 52.00 seconds

 PREP7:
nlist,all

 LIST ALL SELECTED NODES.   DSYS=  0

   NODE         X           Y           Z          THXY    THYZ    THZX
      1        0.          0.          0.          0.00    0.00    0.00
      2     300.00         0.          0.          0.00    0.00    0.00
      3     108.00      144.00         0.          0.00    0.00    0.00

 PREP7:
elist,all

 LIST ALL SELECTED ELEMENTS.  (LIST NODES)

    ELEM MAT TYP REL ESY        NODES

                                         EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
       1   1   1   1   0      1      2
       2   1   1   1   0      1      3
       3   1   1   1   0      2      3

 PREP7:
rlist,all

 LIST REAL SETS         1 TO        1 BY        1

 REAL CONSTANT SET         1  ITEMS    1 TO    6
    2.6800          0.           0.           0.           0.           0.

 PREP7:
mplist,all

 LIST MATERIALS      1 TO     1 BY     1
  PROPERTY= ALL

 PROPERTY TABLE EX     MAT=    1  NUM. POINTS=  1
   TEMPERATURE      DATA    TEMPERATURE      DATA     TEMPERATURE      DATA
        0.      0.29000E+08

 PROPERTY TABLE NUXY  MAT=    1  NUM. POINTS=  1
   TEMPERATURE      DATA    TEMPERATURE      DATA     TEMPERATURE      DATA

                                         EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
        0.      0.29000

 PREP7:
flist,all

 LIST NODAL FORCES FOR SELECTED NODES        1 TO        3 BY        1
```

```
 CURRENTLY SELECTED NODAL LOAD SET= FX    FY    FZ

    NODE  LABEL      REAL           IMAG
       3  FX      6000.00000            0.
       3  FY     -10000.0000            0.

 PREP7:
etlist,all

 LIST ELEMENT TYPES FROM     1 TO     1 BY     1

 ELEMENT TYPE     1 IS LINK8         3-D SPAR  ( OR TRUSS )          INOPR
  KEYOPT(1-12)=    0  0  0    0  0  0    0  0  0    0  0  0          0

 CURRENT NODAL DOF SET IS  UX     UY     UZ
  THREE-DIMENSIONAL MODEL

 PREP7:
```

```
                                             EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
*** NOTE ***                                  CP=       1.698   TIME= 12:53:51
Present time 0 is less than or equal to the previous time.
 Time will default to 1.

                        L O A D   S T E P   O P T I O N S

   LOAD STEP NUMBER. . . . . . . . . . . . . . . . .       1
   TIME AT END OF THE LOAD STEP. . . . . . . . . . .  1.0000
   NUMBER OF SUBSTEPS. . . . . . . . . . . . . . . .       1
   STEP CHANGE BOUNDARY CONDITIONS . . . . . . . . .      NO
   PRINT OUTPUT CONTROLS . . . . . . . . . . . . . .NO PRINTOUT
   DATABASE OUTPUT CONTROLS. . . . . . . . . . . . .ALL DATA WRITTEN
                                                    FOR THE LAST SUBSTEP


Range of element maximum matrix coefficients in global coordinates
Maximum= 276337.778 at element 2.
Minimum= 207253.333 at element 3.

  *** ELEMENT MATRIX FORMULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

    1       3  LINK8          0.040    0.013

                                             EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
Time at end of element matrix formulation CP= 1.83257098.

Estimated number of active DOF= 6.
Maximum wavefront= 38.

Time at end of matrix triangularization CP= 1.87905808.
Equation solver maximum pivot= 466320 at node 2 UX.
Equation solver minimum pivot= 270580.741 at node 3 UX.

  *** ELEMENT RESULT CALCULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

    1       3  LINK8          0.008    0.003

  *** NODAL LOAD CALCULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

    1       3  LINK8          0.004    0.001
*** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
*** TIME =    1.00000         TIME INC =    1.00000      NEW TRIANG MATRIX


  *** PROBLEM STATISTICS
 ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =      3

                                             EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
    1       3  LINK8          0.008    0.003

  *** NODAL LOAD CALCULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

    1       3  LINK8          0.004    0.001
```

```
*** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
*** TIME =   1.00000          TIME INC =   1.00000      NEW TRIANG MATRIX


 *** PROBLEM STATISTICS
 ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =      3
 R.M.S. WAVEFRONT SIZE =     2.2


*** ANSYS BINARY FILE STATISTICS
 BUFFER SIZE USED= 16384
       0.063 MB WRITTEN ON ELEMENT MATRIX FILE: case1.emat
       0.063 MB WRITTEN ON ELEMENT SAVED DATA FILE: case1.esav
       0.063 MB WRITTEN ON TRIANGULARIZED MATRIX FILE: case1.tri
       0.063 MB WRITTEN ON RESULTS FILE: case1.rst

SOLU_LS2:
```


1
ANSYS 5.0 A
JUL 19 1995
12:54:53
DISPLACEMENT
STEP=1
SUB =1
TIME=1
RSYS=0
DMX =0.06089
U
F
DSCA=270.982
ZV =1
DIST=169.971
XF =154.519
YF =66.015
Y
X
1
2
3
Case 1 Frame

```
                                          EDIT:Alt-c   HELP:Alt-h   EXIT:Alt-x    1
PRINT MADE. Print time was 52.00 seconds

 POST1:
prrsol,f

 PRINT F    REACTION SOLUTIONS PER NODE

Case 1 Frame

  ***** POST1 TOTAL REACTION SOLUTION LISTING *****

  LOAD STEP=      1  SUBSTEP=      1
   TIME=    1.0000      LOAD CASE=   0

  THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

   NODE       FX           FY            FZ
      1   -6000.0       3520.0
      2                 6480.0

 TOTAL VALUES
 VALUE    -6000.0       10000.          0.

 POST1:

                                          EDIT:Alt-c   HELP:Alt-h   EXIT:Alt-x    1
prnsol,u,comp

 PRINT U    NODAL SOLUTION PER NODE

Case 1 Frame

  ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

  LOAD STEP=      1  SUBSTEP=      1
   TIME=    1.0000      LOAD CASE=   0

  THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

   NODE       UX          UY          UZ          USUM
      1       0.          0.          0.          0.
      2   0.33350E-01     0.          0.       0.33350E-01
      3   0.41910E-01-0.44171E-01     0.       0.60890E-01

 MAXIMUM VALUES
 NODE          3           3          0            3
 VALUE    0.41910E-01-0.44171E-01     0.       0.60890E-01

 POST1:
```

```
pretab,mforce,saxial

 PRINT ELEMENT TABLE ITEMS PER ELEMENT

Case 1 Frame

  ***** POST1 ELEMENT TABLE LISTING *****

   STAT     CURRENT     CURRENT
   ELEM     MFORCE      SAXIAL
      1   8640.0       3223.9
      2  -4400.0      -1641.8
      3  -10800.      -4029.9

 MINIMUM VALUES
 ELEM          3           3
 VALUE   -10800.      -4029.9

 MAXIMUM VALUES
 ELEM          1           1
 VALUE    8640.0       3223.9

 POST1:
```

//////////////////////LRFD TRUSS ANALYSIS REPORT/////////////////

TRUSS INPUT DATA EFFECTIVE LENGTH FACTOR (K) IS EQUAL TO 1.0

COORDINATE DATA

NODE	X(in.)	Y(in.)
1	.00	.00
2	40.00	.00
3	.00	30.00
4	40.00	30.00

ELEMENT CONNECTIVITY

ELEMENT	BEGINNING NODE	ENDING NODE	LENTH
1	2	1	40.00
2	4	1	50.00
3	4	2	30.00
4	4	3	40.00

EXTERNAL APLLIED LOADS

LOAD NUMBER	LOAD(kips)
1	.00
2	.00
3	200.00
4	.00
5	.00
6	.00
7	.00
8	-250.00

TRUSS ANALYSIS RESULTS

INTERNAL MEMBER AXIAL LOAD

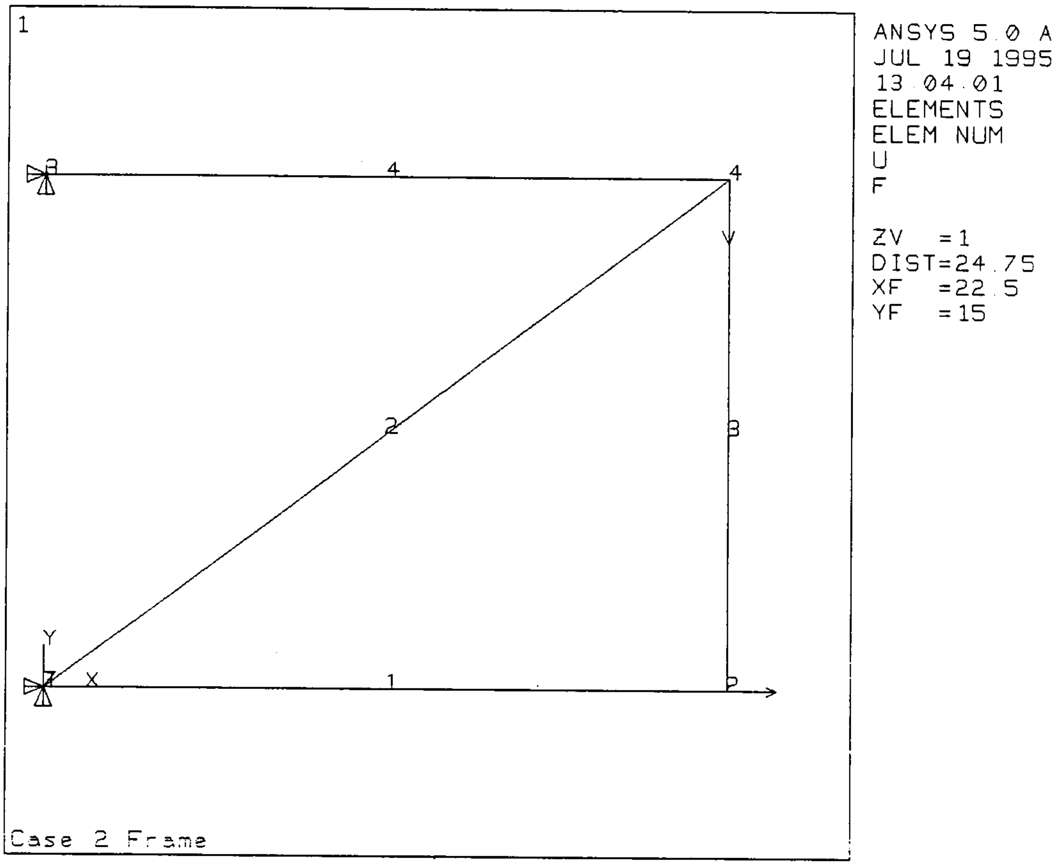
ELEMENT	LOAD(kips)	MEMBER SIZE	AREA(sq. in.)	RADIUS OF GYRATION(in.)
1	200.0000	12 W 22.0	6.48	4.9
2	-416.6667	8 W 24.0	7.08	3.4
3	.0000	6 W 9.0	2.68	2.5
4	333.3333	12 W 22.0	6.48	4.9

DEFORMATION(in.)

DELTA(1) = .000000000000
DELTA(2) = .000000000000
DELTA(3) = .042571307858
DELTA(4) = -.263715548403
DELTA(5) = .000000000000
DELTA(6) = .000000000000
DELTA(7) = .070952161935
DELTA(8) = -.263715542799

RESTRAINT LOADS(kips)

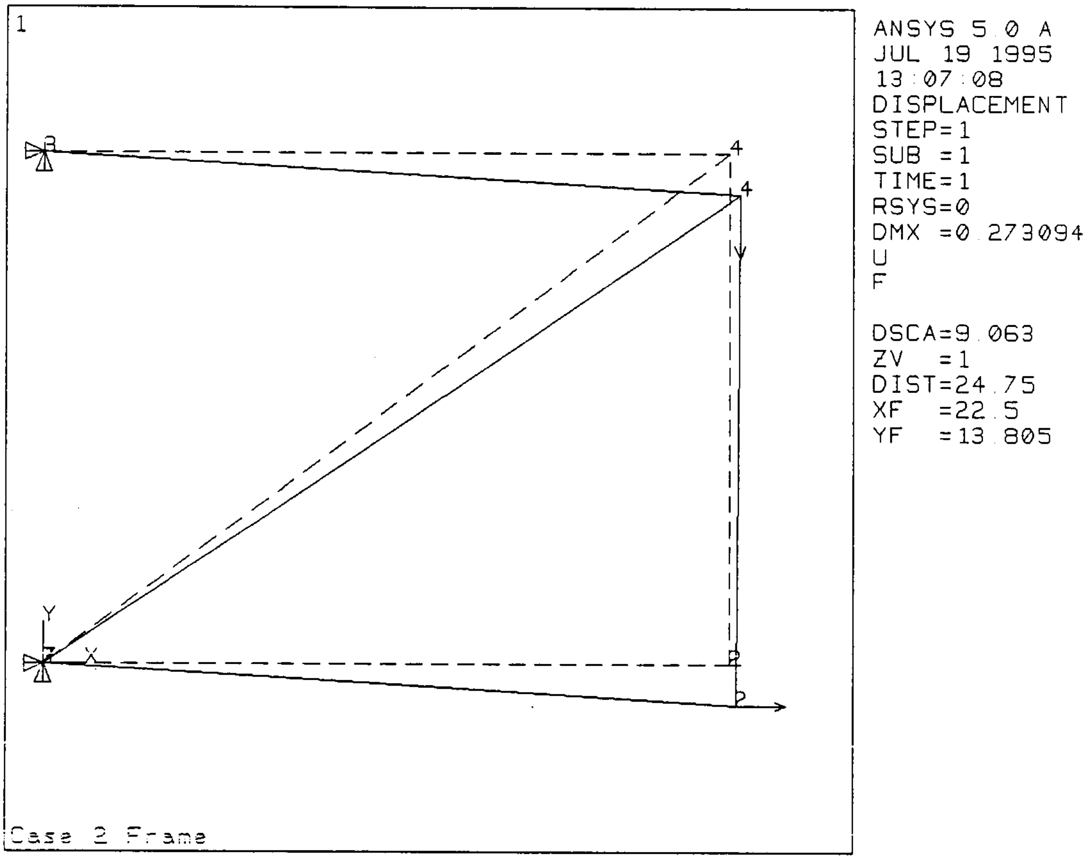
RX(1) = 133.3333
RY(1) = 250.0000
RX(3) = -333.3333


1
4
4
2
3
Y
X
1
Case 2 Frame
ANSYS 5.0 A
JUL 19 1995
13.04.01
ELEMENTS
ELEM NUM
U
F
ZV =1
DIST=24.75
XF =22.5
YF =15

```
                                          EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
 LIST ALL SELECTED NODES.   DSYS=  0

   NODE        X           Y            Z          THXY    THYZ    THZX
      1        0.          0.          0.          0.00    0.00    0.00
      2     40.000         0.          0.          0.00    0.00    0.00
      3        0.       30.000         0.          0.00    0.00    0.00
      4     40.000      30.000         0.          0.00    0.00    0.00
      5     45.000         0.          0.          0.00    0.00    0.00

 PREP7:
elist,all

 LIST ALL SELECTED ELEMENTS.  (LIST NODES)

    ELEM MAT TYP REL ESY       NODES

      1   1   1   1   0      1      2
      2   1   1   2   0      1      4
      3   1   1   1   0      2      4
      4   1   1   1   0      3      4

 PREP7:
rlist,all

                                          EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
 LIST REAL SETS          1 TO       2 BY       1

 REAL CONSTANT SET         1  ITEMS    1 TO    6
    6.4800            0.            0.           0.           0.           0.

 REAL CONSTANT SET         2  ITEMS    1 TO    6
    7.0800            0.            0.           0.           0.           0.

 PREP7:
mplist,all

 LIST MATERIALS       1 TO     1 BY    1
  PROPERTY= ALL

 PROPERTY TABLE EX     MAT=    1  NUM. POINTS=  1
   TEMPERATURE      DATA    TEMPERATURE      DATA     TEMPERATURE      DATA
         0.     0.29000E+08

 PROPERTY TABLE NUXY  MAT=    1  NUM. POINTS=  1
   TEMPERATURE      DATA    TEMPERATURE      DATA     TEMPERATURE      DATA
         0.     0.29000

 PREP7:
flist,all

                                          EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
 LIST NODAL FORCES FOR SELECTED NODES         1 TO         5 BY         1
 CURRENTLY SELECTED NODAL LOAD SET= FX    FY   FZ

   NODE  LABEL      REAL            IMAG
     2   FX      200000.000           0.
```

```
      4  FY      -250000.000              0.

 PREP7:
dlist,all

 LIST CONSTRAINTS FOR SELECTED NODES        1 TO        5 BY        1
 CURRENTLY SELECTED DOF SET= UX   UY   UZ

     NODE  LABEL      REAL              IMAG
        1  UX            0.                0.
        1  UY            0.                0.
        1  UZ            0.                0.
        3  UX            0.                0.
        3  UY            0.                0.
        3  UZ            0.                0.

 PREP7:


                                         EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1

 SOLU_LS1:
solve

 *****  ANSYS SOLVE    COMMAND  *****

                    S O L U T I O N   O P T I O N S

   PROBLEM DIMENSIONALITY. . . . . . . . . . . . .3-D
   DEGREES OF FREEDOM. . . . . . UX   UY   UZ
   ANALYSIS TYPE . . . . . . . . . . . . . . . . .STATIC (STEADY-STATE)

 *** NOTE ***                            CP=       1.974   TIME= 13:06:27
 Present time 0 is less than or equal to the previous time.
  Time will default to 1.

                    L O A D   S T E P   O P T I O N S

   LOAD STEP NUMBER. . . . . . . . . . . . . . . .     1
   TIME AT END OF THE LOAD STEP. . . . . . . . . .  1.0000
   NUMBER OF SUBSTEPS. . . . . . . . . . . . . . .     1
   STEP CHANGE BOUNDARY CONDITIONS . . . . . . . .    NO
   PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
   DATABASE OUTPUT CONTROLS. . . . . . . . . . . .ALL DATA WRITTEN

                                         EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
                                                   FOR THE LAST SUBSTEP


 Range of element maximum matrix coefficients in global coordinates
 Maximum= 6264000 at element 3.
 Minimum= 2628096 at element 2.

   *** ELEMENT MATRIX FORMULATION TIMES
  TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

     1       4  LINK8         0.050    0.013
 Time at end of element matrix formulation CP= 2.13388711.
```

```
Estimated number of active DOF= 6.
Maximum wavefront= 38.

Time at end of matrix triangularization CP= 2.20579702.
Equation solver maximum pivot= 7326096 at node 4 UX.
Equation solver minimum pivot= 947990.88 at node 4 UY.

  *** ELEMENT RESULT CALCULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

                                    EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1
    1       4  LINK8          0.010     0.002

  *** NODAL LOAD CALCULATION TIMES
 TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

    1       4  LINK8          0.004     0.001
*** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
*** TIME =    1.00000          TIME INC =    1.00000      NEW TRIANG MATRIX

  *** PROBLEM STATISTICS
 ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =      4
 R.M.S. WAVEFRONT SIZE =     2.7

*** ANSYS BINARY FILE STATISTICS
 BUFFER SIZE USED= 16384
       0.063 MB WRITTEN ON ELEMENT MATRIX FILE: case2.emat
       0.063 MB WRITTEN ON ELEMENT SAVED DATA FILE: case2.esav
       0.063 MB WRITTEN ON TRIANGULARIZED MATRIX FILE: case2.tri
       0.063 MB WRITTEN ON RESULTS FILE: case2.rst

SOLU_LS2:
```


1
ANSYS 5.0 A
JUL 19 1995
13:07:08
DISPLACEMENT
STEP=1
SUB =1
TIME=1
RSYS=0
DMX =0.273094
U
F
DSCA=9.063
ZV =1
DIST=24.75
XF =22.5
YF =13.805
Y
X
3
4
4
2
2
Case 2 Frame

```
                                        EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1

 POST1:
prrsol,f

 PRINT F    REACTION SOLUTIONS PER NODE

Case 2 Frame

  ***** POST1 TOTAL REACTION SOLUTION LISTING *****

  LOAD STEP=     1  SUBSTEP=     1
   TIME=    1.0000       LOAD CASE=   0

  THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

   NODE       FX          FY          FZ
      1  0.13333E+06 0.25000E+06      0.
      3 -0.33333E+06      0.          0.

 TOTAL VALUES
 VALUE  -0.20000E+06 0.25000E+06      0.

 POST1:
prnsol,u,comp
                                        EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1

 PRINT U    NODAL SOLUTION PER NODE

Case 2 Frame

  ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

  LOAD STEP=     1  SUBSTEP=     1
   TIME=    1.0000       LOAD CASE=   0

  THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

   NODE       UX          UY          UZ          USUM
      1      0.          0.          0.           0.
      2  0.42571E-01-0.26372          0.      0.26713
      3      0.          0.          0.           0.
      4  0.70952E-01-0.26372          0.      0.27309

 MAXIMUM VALUES
 NODE         4           4           0           4
 VALUE   0.70952E-01-0.26372          0.      0.27309

 POST1:
```

```
 PRINT ELEMENT TABLE ITEMS PER ELEMENT

Case 2 Frame

  ***** POST1 ELEMENT TABLE LISTING *****

   STAT    CURRENT     CURRENT
   ELEM    MFORCE      SAXIAL
      1  0.20000E+06  30864.
      2 -0.41667E+06 -58851.
      3 -0.34772E-09-0.53661E-10
      4  0.33333E+06  51440.

 MINIMUM VALUES
 ELEM         2           2
 VALUE  -0.41667E+06 -58851.

 MAXIMUM VALUES
 ELEM         4           4
 VALUE   0.33333E+06  51440.

 POST1:
```

//////////////////LRFD TRUSS ANALYSIS REPORT/////////////////////
TRUSS INPUT DATA EFFECTIVE LENGTH FACTOR (K) IS EQUAL TO 1.0

COORDINATE DATA

NODE	X(in.)	Y(in.)
1	.00	.00
2	180.00	.00
3	360.00	.00
4	540.00	.00
5	180.00	240.00
6	360.00	240.00

ELEMENT CONNECTIVITY

ELEMENT	BEGINNING NODE	ENDING NODE	LENTH
1	2	1	180.00
2	3	2	180.00
3	4	3	180.00
4	5	1	300.00
5	5	2	240.00
6	6	2	300.00
7	6	5	180.00
8	6	3	240.00
9	6	4	300.00

EXTERNAL APLLIED LOADS

LOAD NUMBER	LOAD(kips)
1	.00
2	.00
3	.00
4	.00
5	.00
6	.00
7	.00
8	.00
9	30.00
10	-70.00
11	.00
12	-60.00

TRUSS ANALYSIS RESULTS
INTERNAL MEMBER AXIAL LOAD

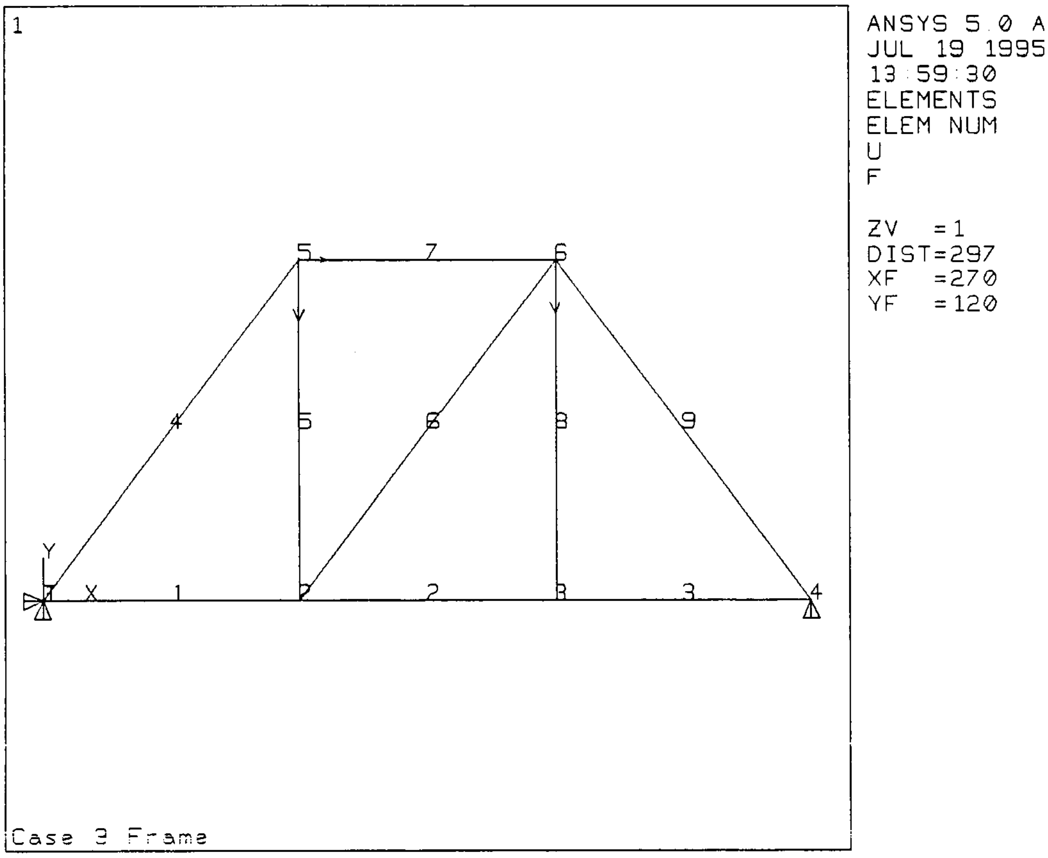
ELEMENT	LOAD(kips)	MEMBER SIZE	AREA(sq. in.)	RADIUS OF GYRATION(in.)
1	70.0000	6 W 9.0	2.68	2.5
2	57.5000	6 W 9.0	2.68	2.5
3	57.5000	6 W 9.0	2.68	2.5
4	-66.6667	8 W 13.0	3.84	3.2
5	-16.6667	8 W 10.0	2.96	3.2
6	20.8333	6 W 9.0	2.68	2.5
7	-70.0000	8 W 13.0	3.84	3.2
8	.0000	6 W 9.0	2.68	2.5
9	-95.8333	10 W 15.0	4.41	3.9

DEFORMATION(in.)

```
DELTA( 1) =   .000000000000
DELTA( 2) =   .000000000000
DELTA( 3) =   .162120425138
DELTA( 4) =  -.557191146641
DELTA( 5) =   .295290758754
DELTA( 6) =  -.537541113650
DELTA( 7) =   .428461078046
DELTA( 8) =   .000000000000
DELTA( 9) =   .416733253967
DELTA(10) =  -.603789457664
DELTA(11) =   .269948555584
DELTA(12) =  -.537541107546
```

RESTRAINT LOADS(kips)

```
RX( 1) =   -30.0000
RY( 1) =    53.3333
RY( 4) =    76.6667
```


1
ANSYS 5.0 A
JUL 19 1995
13:59:30
ELEMENTS
ELEM NUM
U
F
ZV =1
DIST=297
XF =270
YF =120
Case 3 Frame

```
------------------------------------------------------------------------
Freeing printer buffer #1 at segment 77b6
PRINT MADE. Print time was 52.00 seconds

 PREP7:
nlist,all

 LIST ALL SELECTED NODES.   DSYS=  0

   NODE        X            Y            Z          THXY    THYZ    THZX
      1       0.           0.           0.          0.00    0.00    0.00
      2     180.00         0.           0.          0.00    0.00    0.00
      3     360.00         0.           0.          0.00    0.00    0.00
      4     540.00         0.           0.          0.00    0.00    0.00
      5     180.00       240.00         0.          0.00    0.00    0.00
      6     360.00       240.00         0.          0.00    0.00    0.00

 PREP7:
elist,all

 LIST ALL SELECTED ELEMENTS.  (LIST NODES)

    ELEM MAT TYP REL ESY       NODES
```



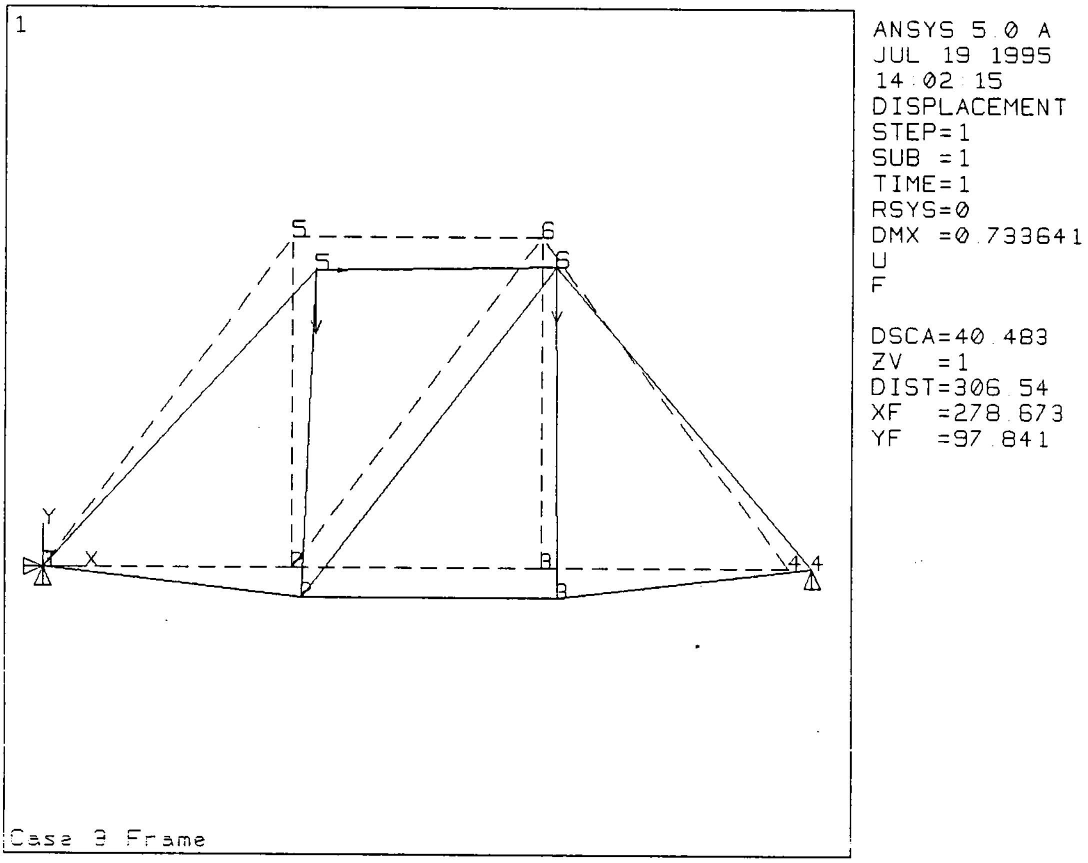
```
      1   1   1   1   0       1     2
      2   1   1   1   0       2     3
      3   1   1   1   0       3     4
      4   1   1   2   0       1     5
      5   1   1   2   0       2     5
      6   1   1   1   0       2     6
      7   1   1   2   0       5     6
      8   1   1   1   0       3     6
      9   1   1   2   0       4     6

 PREP7:
rlist,all

 LIST REAL SETS        1 TO      2 BY      1

 REAL CONSTANT SET        1  ITEMS    1 TO    6
    2.6800          0.            0.            0.            0.            0.

 REAL CONSTANT SET        2  ITEMS    1 TO    6
    2.9600          0.            0.            0.            0.            0.

 PREP7:
mplist,all
```



```
 LIST MATERIALS     1 TO     2 BY     1
  PROPERTY= ALL

 PROPERTY TABLE EX    MAT=    1  NUM. POINTS=  1
   TEMPERATURE      DATA    TEMPERATURE     DATA    TEMPERATURE     DATA
         0.      0.29000E+08
```

```
 PROPERTY TABLE NUXY  MAT=    2  NUM. POINTS=  1
   TEMPERATURE       DATA     TEMPERATURE      DATA     TEMPERATURE      DATA
         0.      0.29000

 PREP7:
flist,all

 LIST NODAL FORCES FOR SELECTED NODES        1 TO         6 BY         1
 CURRENTLY SELECTED NODAL LOAD SET= FX    FY    FZ

    NODE  LABEL      REAL             IMAG
       5  FX       30000.0000                0.
       5  FY      -70000.0000                0.
       6  FY      -60000.0000                0.

 PREP7:
dlist,all

                                             EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1
flist,all

 LIST NODAL FORCES FOR SELECTED NODES        1 TO         6 BY         1
 CURRENTLY SELECTED NODAL LOAD SET= FX    FY    FZ

    NODE  LABEL      REAL             IMAG
       5  FX       30000.0000                0.
       5  FY      -70000.0000                0.
       6  FY      -60000.0000                0.

 PREP7:
dlist,all

 LIST CONSTRAINTS FOR SELECTED NODES        1 TO         6 BY         1
 CURRENTLY SELECTED DOF SET= UX    UY    UZ

     NODE  LABEL      REAL             IMAG
        1  UX              0.               0.
        1  UY              0.               0.
        1  UZ              0.               0.
        4  UY              0.               0.

 PREP7:


                                             EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x    1

 SOLU_LS1:
solve

 *****  ANSYS SOLVE    COMMAND  *****

                        S O L U T I O N   O P T I O N S

   PROBLEM DIMENSIONALITY. . . . . . . . . . . . . .3-D
   DEGREES OF FREEDOM. . . . . . UX    UY   UZ
   ANALYSIS TYPE . . . . . . . . . . . . . . . . . .STATIC (STEADY-STATE)

 *** NOTE ***                               CP=       1.928   TIME= 14:01:35
 Present time 0 is less than or equal to the previous time.
```

```
Time will default to 1.

                    L O A D   S T E P   O P T I O N S

  LOAD STEP NUMBER. . . . . . . . . . . . . . . .        1
  TIME AT END OF THE LOAD STEP. . . . . . . . . .   1.0000
  NUMBER OF SUBSTEPS. . . . . . . . . . . . . . .        1
  STEP CHANGE BOUNDARY CONDITIONS . . . . . . . .       NO
  PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
  DATABASE OUTPUT CONTROLS. . . . . . . . . . . .ALL DATA WRITTEN

                              EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1
                                          FOR THE LAST SUBSTEP


Range of element maximum matrix coefficients in global coordinates
Maximum= 476888.889 at element 7.
Minimum= 165802.667 at element 6.

  *** ELEMENT MATRIX FORMULATION TIMES
 TYPE  NUMBER   ENAME       TOTAL CP  AVE CP

    1       9  LINK8          0.050    0.006
Time at end of element matrix formulation CP= 2.09391001.

Estimated number of active DOF= 14.
Maximum wavefront= 41.

Time at end of matrix triangularization CP= 2.14748409.
Equation solver maximum pivot= 639887.914 at node 6 UX.
Equation solver minimum pivot= 157747.591 at node 2 UY.

  *** ELEMENT RESULT CALCULATION TIMES
 TYPE  NUMBER   ENAME       TOTAL CP  AVE CP


                              EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1
    1       9  LINK8          0.018    0.002

  *** NODAL LOAD CALCULATION TIMES
 TYPE  NUMBER   ENAME       TOTAL CP  AVE CP

    1       9  LINK8          0.006    0.001
*** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
*** TIME =   1.00000          TIME INC =   1.00000       NEW TRIANG MATRIX


  *** PROBLEM STATISTICS
 ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =      9
 R.M.S. WAVEFRONT SIZE =      5.6


*** ANSYS BINARY FILE STATISTICS
 BUFFER SIZE USED= 16384
       0.063 MB WRITTEN ON ELEMENT MATRIX FILE: case3.emat
       0.063 MB WRITTEN ON ELEMENT SAVED DATA FILE: case3.esav
       0.063 MB WRITTEN ON TRIANGULARIZED MATRIX FILE: case3.tri
       0.063 MB WRITTEN ON RESULTS FILE: case3.rst
```


1
ANSYS 5.0 A
JUL 19 1995
14:02:15
DISPLACEMENT
STEP=1
SUB =1
TIME=1
RSYS=0
DMX =0.733641
U
F
DSCA=40.483
ZV =1
DIST=306.54
XF =278.673
YF =97.841
Case 3 Frame

```
                                          EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1
prrsol,f

 PRINT F    REACTION SOLUTIONS PER NODE

Case 3 Frame

  ***** POST1 TOTAL REACTION SOLUTION LISTING *****

  LOAD STEP=      1  SUBSTEP=     1
   TIME=    1.0000       LOAD CASE=   0

  THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

   NODE       FX            FY            FZ
      1  -30000.       53333.          0.
      4                76667.

 TOTAL VALUES
 VALUE   -30000.     0.13000E+06       0.

 POST1:
prnsol,u,comp

 PRINT U    NODAL SOLUTION PER NODE
                                          EDIT:Alt-c  HELP:Alt-h  EXIT:Alt-x   1

Case 3 Frame

  ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

  LOAD STEP=      1  SUBSTEP=     1
   TIME=    1.0000       LOAD CASE=   0

  THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

   NODE       UX          UY           UZ          USUM
      1      0.          0.          0.          0.
      2  0.16212    -0.55719         0.      0.58030
      3  0.29529    -0.53754         0.      0.61331
      4  0.42846         0.          0.      0.42846
      5  0.41673    -0.60379         0.      0.73364
      6  0.26995    -0.53754         0.      0.60152

 MAXIMUM VALUES
 NODE          4           5          0           5
 VALUE   0.42846    -0.60379         0.      0.73364

 POST1:
```

EDIT:Alt-c HELP:Alt-h EXIT:Alt-x 1

```
 ***** POST1 ELEMENT TABLE LISTING *****

   STAT     CURRENT      CURRENT
   ELEM     MFORCE       SAXIAL
      1   70000.        26119.
      2   57500.        21455.
      3   57500.        21455.
      4  -66667.       -22523.
      5  -16667.       -5630.6
      6   20833.        7773.6
      7  -70000.       -23649.
      8 -0.35953E-10-0.13415E-10
      9  -95833.       -32376.

 MINIMUM VALUES
 ELEM          9             9
 VALUE   -95833.       -32376.

 MAXIMUM VALUES
 ELEM          1             1
 VALUE    70000.        26119.

 POST1:
```

OPTIMIZING TECHINQUE

Column Parameters are as follows:

Length(L) = 300.0 inches F_y = 36 ksi

Factored Internal Load (F) = 66.7 kips

E = 29,000 ksi K = 1.0

Compresion Member the case where $\lambda_c < 1.5$

Checking code begining with a stanard wide flange, W6 X 9, where A_g = 2.68 in^2, r = 2.47 inches, $b_t/2t_f$ = 9.2, h_c/t_w = 29.2

λ_r(flange)=95/SQRT(F_y)=95/SQRT(36)=15.83 > $b_t/2t_f$ = 9.2

λ_r(web)=253/SQRT(F_y)=253/SQRT(36)=42.16 > h_c/t_w = 29.2

$\lambda_c < 1.5$ where λ_c = $(KL/r\pi)$SQRT(F_y/E) = 1.36

Therefore: $F_{cr} = [.658^{\beta}]F_y = [.658^{(1.849)}]\ 36 = 16.59$ ksi

where $\beta = \lambda_c^2 = (1.36)^2 = 1.849$

$P_u = \varnothing_c\ F_{cr}\ A_g = .85(16.59)(2.68) = 37.81$ kips < 66.7 kips

F > P_u therefore; code indexes to member size wide flange W8 X 13. For wide flange W8 X 13 properties A_g = 3.84 in^2, r = 3.21 inches, $b_t/2t_f$ = 1.37, h_c/t_w = 29.9

λ_r(flange)=95/SQRT(F_y)=95/SQRT(36)=15.83 > $b_t/2t_f$ = 1.37

λ_r(web)=253/SQRT(F_y)=253/SQRT(36)=42.16 > h_c/t_w = 29.9

$\lambda_c < 1.5$ where λ_c = $(KL/r\pi)$SQRT(F_y/E) = 1.04

Therefore: Fcr = $[.658^{\beta}]F_y = [.658^{(1.082)}]\ 36 = 22.8$ ksi where $\beta = \lambda_c^2 = (1.04)^2 = 1.082$

$P_u = \varnothing_c F_{cr} A_g = .85(22.8)(3.84) = 74.70$ kips > 66.7 kips

$F < P_u$, therefore code conludes as optimum size and prints

W8 X 13 for the optimum member size.

Compresion Member the case where $\lambda_c > 1.5$, L =492.0

inches, checking code begins with W6 X 16 where $A_g = 2.68$

in2, r = 2.47 inches, $b_t/2t_f = 5.0$, $h_c/t_w = 19.1$

λ_r(flange)=95/SQRT(F_y)=95/SQRT(36)=15.83 > $b_t/2t_f = 5.0$

λ_r(web)=253/SQRT(F_y)=253/SQRT(36)=42.16 > $h_c/t_w = 19.1$

λ_c >1.5 where λ_c = (KL/rπ)SQRT(F_y/E) = 2.108

Therefore: $F_{cr} = [.877/\lambda_c^2] F_y = [.877/4.44]\ 36 = 7.10$ ksi

where $\beta = \lambda_c^2 = (2.108)^2 = 4.44$

$P_u = \varnothing_c F_{cr} A_g = .85(7.10)(4.74) = 28.62$ kips < 66.7 kips

$F > P_u$ therefore; code indexes to next member size wide

flange W10 X 17. For wide flange W10 X 17 properties

$A_g = 4.99$ in^2, r = 4.05 inches, $b_t/2t_f = .844$, $h_c/t_w = 36.9$

λ_r(flange)=95/SQRT(F_y)=95/SQRT(36)=15.83 > $b_t/2t_f = .844$

λ_r(web)=253/SQRT(F_y)=253/SQRT(36)=42.16 > $h_c/t_w = 36.9$

λ_c < 1.5 where λ_c = (KL/rπ)SQRT(F_y/E) =1.36

Therefore: Fcr = $[.658^{\beta}] F_y = [.658^{(1.849)}]\ 36 = 16.59$ ksi

where $\beta = \lambda_c^2 = (1.36)^2 = 1.849$

$P_u = \varnothing_c \ F_{cr} \ A_g = .85(16.59)(4.99) = 70.4$ kips < 66.7 kips

$F > P_u$ therefore; member is optimized with W10 X 17

LRFD - ASD Comparison

Load capcatiy of 16 foot truss member with the member size of W8 x 48 #/ft beam being evaluated about the strong axis of the member for A36 steel.

For a W8 x 48 #/ft:

$A_g = 14.1$ in.^{2}	$b_t/2t_f = 5.9$
$r = 3.61$ inches	$h_c/t_w = 15.8$
$Fy = 36$ ksi	$E = 29000$ ksi

ASD Load Capacity

$C_c = SQRT(\ 2\pi^2 \ E/F_y) = SQRT(2\pi^2(29000/36)) = 126.0$

$KL/r = 1.0\ (192)/3.61 = 53.18 \quad (53.18 \leq 126)$

$$\text{Therefore:} F_a = \frac{[\ 1-\ (KL/r)^2/\ 2C_c^2]F_y}{5/3 + 3(Kl/r)/8C_c - (KL/r)^3/8C_c^3}$$

F_a **= 32.79/1.8158 = 18.06 ksi**

F $= F_a A_g = 18.06(14.1) = 254.6$ kips

Hence, the ASD method produces a maximum allowable load of 254.6 kips.

LRFD Load Capacity

$KL/r = 53.18 < 200$

$\lambda_r(\text{flange}) = 95/SQRT(F_y) = 95/SQRT(36) = 35 > b_t/2t_f = 5.9$

$\lambda_r(\text{webb}) = 253/SQRT(F_y) = 253/SQRT(36) = 42.16 > h_c/t_w = 15.8$

$\lambda_c < 1.5$ where $\lambda_c = (KL/r\pi)\,SQRT(F_y/E) = .596$

Therefore: $Fcr = [.658^{\beta}]F_y = [.658^{(.355)}]\ 36 = 31.02$ ksi

where $\beta = \lambda_c^2 = (.596)^2 = .355$

$P_u = \varnothing_c\ F_{cr}\ A_g = .85(31.02)(14.1) = 371.76$ kips

Assuming live load (*L*) is equal to .3 D results:

$P_u = 1.2\ D + 1.6\ L = 1.68\ D = 371.6$; therefore, D = 221.2 and *L* = .3 D = 66.34. The allowable load applied to the column is the sum of the D and *L*. This value equals 287.54 kips. This demonstrates that the LRFD philosophy provides higher load carring capacity than the ASD method with LRFD having the capcity of 287.54 kips as to ASD 254.6 kips.

APPENDIX C

DESIGN CODE

```
C       PROGRAM LRFD ANALYSIS
C       THIS PROGRAM DETERMINES THE LRFD REQURIEMENTS FOR
        STATIC
C       DETERMINATE TRUSS, DETERMINES ELEMENT CONNECTION OF
        THE NODES,
C       DIRECTIONAL COSINES FOR MATRIX-A, EFFECTIVE LENGTHS
OF
        EACH
C       ELEMENT.

C       READ INPUT FOR NODE AND THE CARTESIAN
COORDINATES.....

        REAL*8
  ATT,A,F,L,M,LE,AN,IEQ,MA,SUM,AT,PLOAD,WT,RCOMPARE,AREA,
+ASAT,SAT,DEF,KINV,P,RX,RY,MWT,CX,CY,XSECT,GYR,AXIS,DEFL,
+CNODE,PXLOAD,PYLOAD,APLOAD
        CHARACTER*1 ISHAPE
        DIMENSION
CNODE(20,2),IELEM(20,2),LE(20),MA(20,20),S(20,20),
+IEQ(40),AN(20,40),NODE(20),NMEMB(20),L(20),M(20),A(20,20)
,
      +AT(20,20),F(20),ASAT(20,20),SAT(20,20),DEF(20),KINV(
      20 ,20),P(20),
      +RX(20),RY(20),IMSM(20),ISHPE(20),MWT(20),CX(20),CY(2
      0) ,XSECT(20), +GYR(20),DEFL(40),APLOAD(40)

        COMMON/CARVAR/ATT(20)
        COMMON/IO/NF(25)

        NF(5) =5
        NF(6) =6

C        OPEN(8,FILE='MATRIXP.DAT')
        OPEN(11,FILE='NODE.DAT',STATUS='OLD')
C        OPEN(12,FILE='ELEMENT.DAT',STATUS='OLD')
        OPEN(14,FILE='TRUSS.OUT')
C        OPEN(15,FILE='EXTLOAD.DAT')

C        DETEMINE TYPE OF
PROBLEM...........................
WRITE(*,*) '  LRFD AXIALLY LOADED MEMBER ANALYSIS'
WRITE(*,*) '  ENTER TYPE OF PROJECT, TR FOR TRUSS OR SC
FOR SINGLE COLUMN'
        CALL CARD
        IP=ATT(1)
        IF(IP.EQ.3331) GOTO 5
        IF(IP.EQ.3216) GOTO 6
```

```
    5  WRITE(*,*) 'ENTER NUMBER OF NODAL POINTS "NP" AND
THE NUMBER OF                                    ELEMENTS
"NE"'
      CALL CARD
      NP=ATT(1)
      NE=ATT(2)
101   FORMAT(2I2)
  WRITE(*,*) 'ENTER THE AXIS FOR WHICH THE MEMBER IS TO BE
                              EVALUATE
     +D, (1) X-AXIS, (2) Y-AXIS'
      CALL CARD
      ISTRAX=ATT(1)

C     READ INPUT X AND Y COORDINATES AND LABEL
NODES....................

       WRITE(*,*) 'INPUT DATA NODES AND COORDINATES'
      DO 10 I = 1,NP
       NF(5) = 11
       CALL CARD
       NODE(I) = ATT(1)
       CNODE(I,1) = ATT(2)
       CNODE(I,2) = ATT(3)
    WRITE(*,*) NODE(I),CNODE(I,1),CNODE(I,2) 10
   CONTINUE
      WRITE(*,*) 'NODE CONNECTION TO ELEMENT' DO 20 I =
        1,NE
          NF(5)=11
          CALL CARD
          NMEMB(I) = ATT(1)
          IELEM(I,1) = ATT(2) IELEM(I,2) = ATT(3)
      WRITE(*,*) NMEMB(I),IELEM(I,1),IELEM(I,2)
   20   CONTINUE
      NF(5) = 5
      WRITE(*,*) 'EFFECTIVE LENGTH'
      DO 30 K = 1,NE
       IB = IELEM(K,1)
       IE = IELEM(K,2)
       LE(K) = SQRT(((CNODE(IB,1)-CNODE(IE,1))**2.) + +
     ((CNODE(IB,2)-CNODE(IE,2))**2.))
C      WRITE(*,*) K,LE(K)
C      WRITE(13,*) LE(K)
   30 CONTINUE

      DO 40 J = 1,NE
      IB = IELEM(J,1)
      IE = IELEM(J,2)
      L(J) = (CNODE(IB,1)-
      CNODE(IE,1))/LE(J) M(J) = (CNODE(IB,2)-
      CNODE(IE,2))/LE(J)
C      WRITE(*,*) J,L(J),M(J)
C      WRITE(14,*) L(J),M(J)
```

```
   40 CONTINUE

      DO 102 J =1,40
  102 IEQ(J) = 0.0

  103 CONTINUE
      WRITE(*,*) 'ENTER THE RESTRAINT CONDITION;
     NODE,DX,DY (NOTE: LEAVE + BLANK IF RESTRAINT IS FREE.
     EXAMPLE: 1, ,DY) TO END TYPE; ENDR'
      CALL CARD
      NODER = ATT(1)
      IF(NODER.EQ.18271731) GOTO 104
      IDIRX = ATT(2)
      IF(IDIRX.EQ.1737) IEQ(2*NODER-1) = 1.
      IDIRY = ATT(3)
      IF(IDIRY.EQ.1738) IEQ(2*NODER) = 1.
      GO TO 103

  104 CONTINUE

C     DETERMINE EXTERNAL LOAD
MATRIX....................................

      DO 2001 JJ=1,2*NP
 2001 P(JJ) = 0.0

 2002 CONTINUE

      WRITE(*,*) '  ENTER THE EXTERNAL LOADING CONDITION;
     NODE, PX, LOAD +, PY, LOAD. (NOTE: LEAVE A BLANK
     SPACE IF NO LOADING IS PRESENT) T +O END TYPE: END'

      CALL CARD
      NODEP = ATT(1)
      IF(NODEP.EQ.182717) GO TO 2003
      IPDIRX = ATT(2)
      PXLOAD = ATT(3)
      IF(IPDIRX.EQ.2937) APLOAD(2*NODEP-1) = PXLOAD IPDIRY
      = ATT(4)
      PYLOAD = ATT(5)
      IF(IPDIRY.EQ.2938) APLOAD(2*NODEP) =
PYLOAD
      GO TO 2002
 2003 CONTINUE
       INDXP = 0
       DO 9900 I=1,2*NP IF(IEQ(I).EQ.1.0) GOTO 9900 INDXP
       = INDXP + 1
       P(INDXP) = APLOAD(I)
       WRITE(*,*) P(INDXP)
 9900  CONTINUE

       DO 44 I =1,2*NP IF(IEQ(I).EQ.1.)THEN WRITE(*,105) I
```

```
105 FORMAT(2X,'IEQ(',I2,')=',3X,'CONSTRAINT JOINT') ELSE
       WRITE(*,106) I
      106 FORMAT(2X,'IEQ(',I2,')= ',3X,'FREE JOINT')
       ENDIF
   44 CONTINUE

      DO 45 I =1,NE
      DO 45 J =1,2*NP
   45 AN(I,J) = 0.0

      DO 90 J = 1,NE
      IBN = IELEM(J,1)
      IBE = IELEM(J,2)
      IVT = 0
      IF((IEQ(2*(IBN)-1)).EQ.1.) GOTO
      80 AN(J,2*(IBN)-1) = L(J)
   80 IF(IEQ(2*(IBN)).EQ.1.) GOTO 85 AN(J,2*(IBN)) = M(J)
   85 IF((IEQ(2*(IBE)-1)).EQ.1.0) GOTO
      88 AN(J,2*(IBE)-1) = -L(J)
  88 IF(IEQ(2*(IBE)).EQ.1.0) GOTO 90 AN(J,2*(IBE)) = -M(J)
      IVT = IVT + 1
      IF(IVT.GT.1) GOTO 90
   90 CONTINUE

      NF(5) = 5

C      WRITE(*,*) 'THE COS vs ELEMENT MATRIX'
C      DO 120 I =1,NE
C      DO 120 J =1,(2*NP)
C      WRITE(15,*) AN(I,J)
C 120 WRITE(*,130) I,J,AN(I,J)
C 130
FORMAT(2X,'COLUMN',I2,1X,'ROW',I2,1X,F20.2)

      INDEX = 0

      DO 160 IROW =1,2*NP
      SUM = 0.0
       DO 150 ICOL = 1,NE
               SUM = SUM+ABS(AN(ICOL,IROW))
  150  CONTINUE

      IF(SUM.NE.0.0) GOTO 140 IF(SUM.EQ.0.0) GOTO 160
  140  INDEX = INDEX+1
       DO 165 KK =1,NE
       MA(INDEX,KK) = AN(KK,IROW)
  165 CONTINUE
  160 CONTINUE
C      WRITE(*,*) '  MATRIX A'
C      DO 170 I = 1,NE
C      DO 170 J = 1,NE
C      WRITE(*,130) I,J,MA(I,J)
```

```
C 170 CONTINUE

      DO 200 I =1,NE
      DO 200 J =1,NE
      AT(I,J) = MA(J,I)
C      WRITE(*,130) J,I,AT(J,I)
  200 CONTINUE


      EMOD = 29000.0

      DO 201 I=1,NE
      DO 201 J=1,NE
      A(I,J) = MA(I,J)
  201 CONTINUE

C      DO 900 I =1,2*NP
C      WRITE(8,895) P(I)
C  895 FORMAT(F10.2)
C  900 CONTINUE

      CALL TRUSS(A,F,P,NE)

C     DETERMINE THE  STIFFNESS MATRIXS.......

      DO 230 I =1,NE
      DO 230 J =1,NE
      S(I,J) = 0.0
  230 CONTINUE

      DO 195 K =1,NE
      IAMS = 6
      IF(F(K).GE.0.0) GOTO 190 IF(F(K).LT.0.0) GOTO 191

190 CALL OPTEN
      (IAMS,F,LE,ISTRAX,IMS,ISHAPE,WT,AREA,AXIS,RCOMPARE)
      GO TO 192

                                        191 CALL
  OPCOMP(IAMS,F,LE,ISTRAX,IMS,ISHAPE,WT,AREA,AXIS,PLOAD)

  192    S(K,K) = EMOD*AREA/LE(K)

      IMSM(K) = IMS
      ISHPE(K) = ISHAPE
      MWT(K) = WT
      XSECT(K) = AREA
      GYR(K) = AXIS
C      WRITE(*,*) 'CHECKING VARIBLES'
C      DO 193 I =1,NE
C      WRITE(*,*) I,IMSM(I),ISHPE(I),MWT(I)
```

```
C  193 CONTINUE

  195 CONTINUE


C       WRITE(*,*) '    STIFFNESS MATRIX S'
C       DO 241 I =1,NE
C       WRITE(*,130) I,I,S(I,I)
C  241 CONTINUE

C      MULTIPLY S MATRIX BY AT MATRIX.....

       DO 320 I=1,NE
        DO 330 J =1,NE
        SUM = 0.0
         DO  340 K =1,NE
                 SUM = SUM + S(I,K)*AT(K,J)
  340    CONTINUE
        SAT(I,J) = SUM
  330   CONTINUE
  320  CONTINUE

C      MULTIPY MA MATRIX BY SAT MATRIX........

       DO 420 I=1,NE
        DO 430 J=1,NE
        SUM = 0.0
         DO 440 K=1,NE
         SUM = SUM + MA(I,K)*SAT(K,J) 440
       CONTINUE
       ASAT(I,J) = SUM
  430  CONTINUE
  420  CONTINUE

C       WRITE(*,*) '    K MATRIX'
C       DO 441 I=1,NE
C       DO 441 J=1,NE
C       WRITE(*,442) I,J,ASAT(I,J)
C  442 FORMAT(2X,'COLUMN',I2,1X,'ROW',I2,1X,F20.4)

C  441 CONTINUE

C      INVERSE THE ASAT MATRIX FOR DEFLECTION CALCULATION

       CALL INVER (ASAT,NE,KINV)

C       WRITE(*,*) 'K INVERSE MATRIX'
C       DO 450 I =1,NE
C       DO 450 J =1,NE
C       WRITE(*,135) I,J,KINV(I,J)
C 135  FORMAT(2X,'COLUMN',I2,1X,'ROW',I2,1X,F20.17)
```

```
C 450 CONTINUE

C     CALCULATED
DEFLECTION.............................................

      DO 460 KK=1,NE
       SUM = 0.0
         DO 470 I=1,NE
         SUM = SUM + KINV(KK,I)*P(I)
  470    CONTINUE
         DEF(KK) = SUM
  460 CONTINUE

C     CALCULATED THE RESTRAINT
FORCES..................................

       DO 480 K =1,NP
       IF(IEQ(2*K-1).NE.1.0) GOTO 590
       IX = 1
       SUM =0.0

         DO 500 I =1,NE
         IB = IELEM(I,1)
         IE = IELEM(I,2)
         IF(IX.NE.1) GOTO 480 IF(IB.NE.K) GOTO 520
  540    CX(I) = (CNODE(IE,1)-
CNODE(IB,1))/LE(I)
                  SUM = SUM + F(I)*CX(I)
         GO TO 500
  520    IF(IE.NE.K) GOTO 500
         GO TO 540
  500    CONTINUE
      RX(K) = SUM

  590 IF(IEQ(2*K).NE.1.) GOTO 480
       IY = 2
       SUM =0.0
      DO 550 NN =1,NE
       IB = IELEM(NN,1)
       IE = IELEM(NN,2)
       IF(IY.NE.2) GOTO 590 IF(IB.NE.K) GOTO 560
575 CY(NN) = (CNODE(IE,2)-CNODE(IB,2))/LE(NN)
      SUM = SUM + F(NN)*CY(NN)
      GO TO 550
  560 IF(IE.NE.K) GOTO 550
      GO TO 575
  550 CONTINUE
       RY(K) = SUM
  480 CONTINUE
```

```
      GO TO 249


    6 WRITE(*,*) '  ENTER TYPE OF ANALYSIS TE FOR TENSION
     AND CP FOR CO +MPRESSION MEMBERS'

      CALL CARD
      ITYPE = ATT(1)
      IF(ITYPE.EQ.3318) GOTO 180
      IF(ITYPE.EQ.1629) GOTO 181

  180 CALL TEN (IMS,ISHAPE,WT,PLOAD)
C      WRITE(*,*) '  MEMBER SIZE,  COLUMN CAPACITY (KIPS)'
C      WRITE(*,183) IMS,ISHAPE,WT,PLOAD
      GO TO 999


  181 CALL COMPRESS (ISTRAX,IMS,ISHAPE,WT,PLOAD)
C      WRITE(*,*) '  MEMBER SIZE,  COLUMN CAPACITY (KIPS)'
C      WRITE(*,183) IMS,ISHAPE,WT,PLOAD
C 183 FORMAT(2X,A2,A1,F5.1,3X,F20.4)
      GO TO 999

  249 NF(5) =14
      WRITE(14,*)'/////////////////////LRFD TRUSS ANALYSIS
     REPORT/////// +///////////////'
      WRITE(14,599)
  599 FORMAT(15X,'TRUSS INPUT DATA',/)
      WRITE(14,*)'   EFFECTIVE LENGTH FACTOR (K) IS EQUAL
      TO 1.0' WRITE(14,*)'
      '
WRITE(6,*)'     COORDINATE DATA' WRITE(14,*)'
COORDINATE DATA' WRITE(14,*)'  NODE          X(in.)
Y(in.)'

      DO 601 KK =1,NP
      WRITE(14,600) NODE(KK),CNODE(KK,1),CNODE(KK,2)
  600 FORMAT(4X,I2,10X,F6.2,10X,F6.2)
  601 CONTINUE
      WRITE(14,*)'                ' WRITE(14,*)  '
      ELEMENT CONNECTIVITY'
      WRITE(14,*) '  ELEMENT     BEGINNING NODE
     ENDING NODE LENGTH'
      DO 602 JJ= 1,NE
      WRITE(14,603)
  NMEMB(JJ),IELEM(JJ,1),IELEM(JJ,2),LE(JJ) 603
  FORMAT(4X,I2,15X,I2,15X,I2,8X,F8.2)
  602 CONTINUE
      WRITE(14,605)
  605 FORMAT(15X,'EXTERNAL APLLIED LOADS',/) WRITE(14,*)'
      LOAD NUMBER           LOAD(kips)' DO 606 KK =1,2*NP
      WRITE(14,607) KK,APLOAD(KK)
```

```
  607 FORMAT(4X,I2,17X,F10.2)
  606 CONTINUE
                        WRITE(14,*)'
                                  '
      WRITE(14,604)
  604 FORMAT(15X,'TRUSS ANALYSIS RESULTS',/)
      WRITE(14,*) '          INTERNAL MEMBER AXIAL LOAD'
      WRITE(14,*) '  ELEMENT    LOAD(kips)    MEMBER SIZE
     AREA(sq. in.) +  RADIUS OF GYRATION(in.)'
      DO 250 I =1,NE
                       WRITE(14,251)
      I,F(I),IMSM(I),ISHPE(I),MWT(I),XSECT(I),GYR(I)
  251
FORMAT(2X,I2,F20.4,5X,I2,1X,A1,F5.1,8X,F5.2,10X,F5.1)
  250 CONTINUE
      WRITE(14,*)'
      ' WRITE(14,*)'              DEFORMATION(in.)'
      INDX = 0
      DO 700 J =1,2*NP
      IF(IEQ(J).NE.1.0) GOTO 710
      DEFL(J) = 0.0
      GO TO 700
  710 INDX = INDX + 1
      DEFL(J) = DEF(INDX)
  700 CONTINUE

      DO 256 I =1,2*NP
      WRITE(14,252) I,DEFL(I)
         252 FORMAT(2X,'DELTA(',I2,') =',F15.12)
  256 CONTINUE
      WRITE(14,*)'
      WRITE(14,*) '            RESTRAINT LOADS(kips)'
      DO 254 K =1,NP
                IF(RX(K).EQ.0.0) GOTO 257
      WRITE(14,253) K,RX(K)
  257 IF(RY(K).EQ.0.0) GOTO 254 WRITE(14,255) K,RY(K)
  253 FORMAT(2X,'RX(',I2,') =',F20.4) 255
   FORMAT(2X,'RY(',I2,') =',F20.4)
  254 CONTINUE


  999 STOP
      END


                SUBROUTINE TRUSS(A,F,P,NE)
C     PROGRAM TRUSS ANALYSIS

C     THIS PROGRAM IS DESIGN AIDE FOR SOLUTION TO A PLANE
TRUSS. THE
```

```
C      TRUSS IS A 2-DIMENSIONAL. THIS PROGRAM SHALL SOLVE
THE
INTERNAL
C      LOADS, THEN OPTIUMIZE THE ELEMENT TO THE LRFD
SPECIFICATON.
C      SOLVE THE INTERNAL
LOADS(F).....................................
       REAL*8 ATT,A,P,F,INDEX
       DIMENSION A(20,20), P(20), F(20), INDEX(20,2)
       COMMON/CARVAR/ATT(20)
       COMMON/IO/NF(25)

C       OPEN(7,FILE='MATRIXA.DAT',STATUS='OLD')
C       OPEN(8,FILE='MATRIXP.DAT',STATUS='OLD')
C       OPEN(9,FILE='MATRINVA.OUT',STATUS='NEW')

C      READ THE LOADING CONDITION
.....................................

C       NF(5)=8
C       DO 300 IR = 1,NE
C       CALL CARD
C       P(IR)=ATT(1)
C 300   WRITE(*,301) IR,P(IR)
C 301   FORMAT(2X,'P(',I2,') =',F20.4)

C      PRINT THE INPUT
DATA.............................................

C       WRITE(*,103)
C103   FORMAT(3X,'TRUSS ANALYSIS BY THE METHOD OF
JOINTS',/)
C       WRITE(*,104)
C104   FORMAT (2X,'THE MATRIX A',/)
C       DO 105 IR=1,NE
C       DO 105 IC=1,NE
C105   WRITE(*,106) IR,IC,A(IR,IC)
C106   FORMAT (4X,'ROW',1X,I2,1X,'COLUMN',1X,I2,F10.4)

       WRITE(*,107)
 107   FORMAT (3X,'THE MATRIX P',/)
       DO 108 I=1,NE
 108   WRITE(*,155) I,P(I)
    155   FORMAT(2X,'P',I2,1X,'EXTERNAL LOAD',1X,F20.4)

C      DETERMINE THE INVERSE OF THE MATRIX
A.............................

       DO 109 I=1,NE
 109   INDEX (I,1) = 0
       II = 0
 110   AMAX = -1.
```

```
      DO 111 I =1,NE
      IF (INDEX(I,1)) 111,112,111
  112 DO 113 J=1,NE
      IF (INDEX(J,1)) 113,114,113
  114 TEMP = DABS(A(I,J))
      IF(TEMP-AMAX) 113,113,115
  115 IROW = I
      ICOL = J
      AMAX = TEMP
  113 CONTINUE
  111 CONTINUE
      IF (AMAX) 226,116,117
  117 INDEX(ICOL,1) = IROW
      IF (IROW-ICOL) 120,119,120
  120 DO 121 J = 1,NE
      TEMP = A(IROW,J)
      A(IROW,J) =A(ICOL,J)
  121 A(ICOL,J) = TEMP
      II = II + 1
      INDEX (II,2) = ICOL
  119 PIVOT = A(ICOL,ICOL) A(ICOL,ICOL) = 1.0
      PIVOT =1./PIVOT
      DO 122 J = 1,NE
  122 A(ICOL,J) = A(ICOL,J)*PIVOT
      DO 123 I = 1,NE
      IF (I-ICOL) 124,123,124
  124 TEMP = A(I,ICOL)
      A(I,ICOL) = 0.0
      DO 125 J = 1,NE
           125  A(I,J) = A(I,J) - A(ICOL,J)*TEMP
  123 CONTINUE
      GO TO 110
  126 ICOL = INDEX(II,2)
      IROW = INDEX(ICOL,1)
      DO 127 I = 1,NE
      TEMP = A(I,IROW)
      A(I,IROW) = A(I,ICOL)
  127 A(I,ICOL) = TEMP
      II = II-1
  226 IF (II) 126,129,126

C     PRINT MATRIX A INVERSE

C      WRITE(*,*)  'MATRIX A INVERSE'
C      DO 231 I=1,NE
C      DO 231 J=1,NE
C      WRITE(*,230) I,J,A(I,J)
C 230 FORMAT(2X,'ROW',1X,I2,1X,'COLUMN',1X,I2,1X,F10.4) C
231 CONTINUE

C     DETERMINE THE INTERNAL LOADS (F) MATRIX
```

```
 129   DO 130 I = 1,NE

       F(I) = 0.0
       DO 130 K = 1,NE
 130   F(I) = F(I) + A(I,K)*P(K)

       WRITE(*,131)
 131   FORMAT(4X,'THE MATRIX F',/)
       DO 132 I = 1,NE
 132   WRITE(*,156) I,F(I)
 156   FORMAT(2X,'ELEMENT INTERNAL LOAD F',I2,1X,F20.4)
       GO TO 134

C      INDENTIFY UNSTABLE STATIC MATRIX
A.................................

 116    WRITE(*,133)
 133    FORMAT (2X,'ZERO PIVOT'/)

 134    RETURN
        END

       SUBROUTINE
      OPTEN(IAMS,F,LE,ISTRAX,IMS,ISHAPE,WT,RAREAG,RAXIS,RCO
      MP +ARE)
C      PROGRAM TENSION ANALYSIS
C      THIS PROGRAM IS TO DETERMINE THE COLUMN STRENGTH FOR
FOR TENSION
C      MEMBERS OF A PLANAR TRUSS.

       REAL*8 ATT,F,LE,WT,RAREAG,RCOMPARE,RAXIS
       CHARACTER*1 IW,ISHAPE
       COMMON/CARVAR/ATT(20)
       COMMON/IO/NF(25)

       DATA IW/'W'/ OPEN(1,FILE='GEODATA.DAT',STATUS='OLD')

       REWIND 1

       NF(5)=5
       NF(6)=6

       ICT = 0

       WRITE(*,*) '       BEGINNING TENSION ANALYSIS'
 1200 CONTINUE
          NF(5)=1
         CALL CARD
         IMS=ATT(1)
          IF(IMS.NE.182717) GOTO 1201 WRITE(*,1202)
     1202   FORMAT(2X,'END OF DATA FILE WAS REACHED WITH
                              NO SOLUTION')
```

```
         GOTO 1100

 1201    ISP=ATT(2)
         IF(ISP.EQ.36) ISHAPE = IW
         WT=ATT(3)
         RAREAG=ATT(4)
         RFLRATIO=ATT(5)
         RADX=ATT(6)
         RADY=ATT(7)
         NF(5)=5

       IF(ICT.NE.0) GOTO 1203
       IF(IAMS.NE.IMS) GOTO 1200
 1203  ICT = ICT + 1

       IF(ISTRAX.EQ.1) THEN
          RAXIS = RADX
          ELSE
          RAXIS = RADY
          ENDIF

       REMOD = 29000.0
       RFY = 36.
       RFU = 58.
       RYPHI = .9
       RFPHI = .75

C      MS IS MEMBER SIZE, AREAG IS THE GROSS AREA,
C      FLRATIO IS THE COMPACT SECTION CRITERIA (B/T),
C      RADX AND RADY IS THE RADIUS OF GYRATION ABOUT THEIR
RESPECTIVE
C      AXIS.


C      CALCULATE SLENDERNESS
RATIO......................................

       RSLENDR = LE/RAXIS

         IF (RSLENDR.GT.300.) THEN
         WRITE(*,400)
  400    FORMAT(5X,'DESIGN EXCEEDS SLENDERNESS LIMIT SELECT
NEW
      +STRUCTURAL MEMBER')
         ELSE

C      CALCULATE THE ALLOWABLE
LOAD...................................
         RAREAE = .75*RAREAG
         WRITE(*,*) RAREAE,RAREAG
         RYLOAD = (RAREAG*RFY)*RYPHI
         RFLOAD = (RAREAE*RFU)*RFPHI
```

```
          ENDIF

C         WRITE(*,*) RYLOAD, RFLOAD
          IF(RYLOAD.LT.RFLOAD) THEN
          WRITE(*,1000)
      1000    FORMAT(3X,'YIELD EQUATION CONTROLS',/)
C          WRITE(*,1001) RYLOAD
C1001     FORMAT(28X,F20.2)
          RCOMPARE = RYLOAD
          ELSE
          WRITE(*,1002)
 1002 FORMAT(3X,'FRACTURE EQUATION CONTROLS',/)
C          WRITE(*,1003) RFLOAD
C1003     FORMAT(35X,F20.2)
          RCOMPARE = RFLOAD
          ENDIF
C       WRITE(*,*) RCOMPARE,F

      IF(RCOMPARE.LT.F) THEN
      WRITE(*,800)
800 FORMAT(1X,'NOT ABLE TO SUPPORT LOADING, SELECT NEW
      MEMBER SIZE',/) ELSE
      WRITE(*,5000)
  5000 FORMAT(6X,'ELEMENT',22X,'MEMBER',10X,'COLUMN LOAD
                         (kips)',/)
      WRITE(*,9990) F,IMS,ISHAPE,WT,RCOMPARE
 9990 FORMAT(2X,F20.4,16X,I2,1X,A1,1X,F5.2,10X,F20.2)
      ENDIF

      IF(RCOMPARE.LT.F) GOTO 1200

 1100 RETURN
      END

                       SUBROUTINE OPCOMP
     (IAMS,F,LE,ISTRAX,IMS,ISHAPE,WT,AREAG,AXIS,PLOAD
     +)
C     PROGRAM COMPRESSION ANALYSIS

C     THIS PROGRAM IS TO DETERMINE THE COLUMN STRENGTH FOR
FOR A
COMPRESSION
C     MEMBERS FOR A PLANAR TRUSS INCORPORATING THE LRFD
SPECIFICATION


                              REAL*8
LE,F,RADX,RADY,PI,EMOD,FY,FU,K,SLRATIO,AXIS,LAMDAR,LAMDAC,
     +FCR,AREAG,FLRATIO,PLOAD,SLIMIT,ATT,WT,WRATIO,LAMDARW

      CHARACTER*1 IW,ISHAPE
```

```
      COMMON/CARVAR/ATT(20)
      COMMON/IO/NF(25)

      DATA IW/'W'/

      OPEN(1,FILE='GEODATA.DAT',STATUS='OLD')

      REWIND 1
      NF(5) = 5
      NF(6) = 6

      PI = 3.141593
      EMOD =29000.
      FY = 36.
      FU = 58.
      K = 1.0
      WRITE(*,*)' BEGINNING COMPRESSION ANALYSIS'
 1200 CONTINUE

      NF(5)=1
      CALL CARD
      IMS=ATT(1)
      IF(IMS.NE.182717) GOTO 1201 WRITE(*,1202)
    1202 FORMAT(2X,'END OF DATA FILE WAS REACHED WITH NO
                                   SOLUTION')
      GOTO 1100
 1201 ISP=ATT(2)
       IF(ISP.EQ.36) ISHAPE = IW
      WT=ATT(3)
      AREAG=ATT(4)
      FLRATIO=ATT(5)
      RADX=ATT(6)
      RADY=ATT(7)
      WRATIO=ATT(8)

      NF(5) =5

      IF(ICT.NE.0) GOTO 1203
                IF(IAMS.NE.IMS) GOTO 1200
 1203 ICT =ICT + 1

       IF(ISTRAX.EQ.1) THEN
       AXIS = RADX
       ELSE
       AXIS = RADY

       ENDIF

C     CALCULATE SLENDERNESS RATIO

SLRATIO = (K*LE)/AXIS
```

```
      SLIMIT = 200.0

      IF(SLRATIO.LE.SLIMIT) THEN

C     CALCULATE LAMDAR TO CHECK THE FLANGE

RATIO............................
        LAMDAR = 95.0/SQRT(FY)
C        WRITE(*,4500) LAMDAR
C4500    FORMAT(3X,'LAMDAR = 'F20.4,/)
        ELSE
        WRITE(*,4000)
 4000 FORMAT(5X,'DESIGN EXCEEDS SLENDERNESS LIMIT, SELECT
                    NEW STRUCTURAL
     + MEMBER')
       ENDIF

        IF(SLRATIO.GT.SLIMIT) GOTO 1200

        LAMDAC = ((K*LE)/(AXIS*PI))*SQRT(FY/EMOD)
C        WRITE(*,*) LAMDAC
        IF(FLRATIO.GT.LAMDAR) THEN
        WRITE(*,4400)
 4400 FORMAT(5X,'DESIGN EXCEEDS COMPACT SECTION LIMIT, SELECT
                       NEW MEMBER
     +',/)
        ELSE

C     CALCULATE LAMDAC TO CHECK STRENGTH
CRITERIA............................
        LAMDAC = LAMDAC
        ENDIF

        IF(FLRATIO.GT.LAMDAR) GOTO 1200

        LAMDARW = 253.0/SQRT(FY)
        IF(WRATIO.GT.LAMDARW) THEN
        WRITE(*,4410)
  4410 FORMAT(5X,'DESIGN EXCEEDS NON-COMPACT SECTION LIMIT,
                      SELECT NEW M
     +EMBER',/)
        ELSE
C     CALCULATE LAMDAC TO CHECK STRENGTH
CRITERIA............................
        LAMDAC = LAMDAC
        ENDIF

        IF(WRATIO.GT.LAMDARW) GOTO 1200
C     CHECK THE LAMDAC TO LRFD LIMIT...................
        IF(LAMDAC.LE.(1.5)) THEN
        FCR = ((.658)**(LAMDAC**2.0))*FY
```

```
          ELSE
          FCR = (.877/(LAMDAC**2.0))*FY
          ENDIF

C        WRITE(*,4600) FCR
C 4600 FORMAT(3X,'FCR (PSI) =',F20.4,/)

C       CALCULATE LOAD CAPCITY OF THE
MEMBER....................

        PLOAD = AREAG*FCR*.85

        IF(PLOAD.LT.F) THEN
        WRITE(*,8000)
    8000 FORMAT(1X,'MEMBER NOT OPTIMUMIZE, SELECT NEW MEMBER
                              SIZE')
        ELSE
        WRITE(*,5000)
       5000 FORMAT(3X,'MEMBER',3X,'COLUMN CAPACITY LOAD
                           (KIPS)',/)
        WRITE(*,9990) IMS,ISHAPE,WT,PLOAD
   9990 FORMAT(3X,I2,A1,F5.1,6X,F20.2)
        ENDIF

          IF(PLOAD.LE.F) GOTO 1200

   1100 RETURN
        END

        SUBROUTINE TEN (IMS,ISHAPE,WT,RCOMPARE)
C       PROGRAM TENSION ANALYSIS
C       THIS PROGRAM IS TO DETERMINE THE COLUMN STRENGTH FOR
TENSION
C       MEMBERS OF A PLANAR TRUSS.

        REAL*8 ATT,WT,RCOMPARE

        CHARACTER*1 IW,ISHAPE

        COMMON/CARVAR/ATT(20)
        COMMON/IO/NF(25)

        DATA IW/'W'/
        OPEN(1,FILE='GEODATA.DAT',STATUS='OLD')
        REWIND 1
        NF(5)=5
        NF(6)=6

         WRITE(*,*) 'ENTER LOADING FE(kips) =' C
       READ(*,200) RFE
          CALL CARD
```

```
        RFE=ATT(1)

     WRITE(*,*) 'ENTER EFFECTIVE LENTGH OF THE COLUMN
LE(in.) ='

C       READ(*,200)  RLE
        CALL CARD
        RLE=ATT(1)
  200 FORMAT(F10.2)

      WRITE(*,*) 'ENTER THE AXIS FOR WHICH THE MEMBER IS
     TO BE EVALUATE +D, (1) X-AXIS, (2) Y-AXIS'
C       READ(*,300) ISTRAX
        CALL CARD
        ISTRAX=ATT(1)
  300 FORMAT(I1)

900 WRITE(*,*) 'ENTER STRUCTURAL MEMBER SIZE MS' ICT = 0
C       READ(*,100) AMS
        CALL CARD
        IAMS=ATT(1)
  100 FORMAT(A)


      WRITE (*,*) RFE,RLE,ISTRAX,IAMS

 1200 CONTINUE


         NF(5)=1
        CALL CARD
        IMS=ATT(1)
        IF(IMS.NE.182717) GOTO 1201 WRITE(*,1202)
     1202   FORMAT(2X,'END OF DATA FILE WAS REACHED WITH
                          NO SOLUTION')
        GOTO 1100

 1201   ISP=ATT(2)
        IF(ISP.EQ.36) ISHAPE =IW
        WT =ATT(3)
        RAREAG=ATT(3)
        RFLRATIO=ATT(4)
        RADX=ATT(5)
        RADY=ATT(6)

C       READ(1,50) MS,RAREAG,RFLRATIO,RADX,RADY,ICOUNT
     50
         FORMAT(1X,A8,1X,F6.2,1X,F5.2,1X,F5.2,1X,F5.2,1X,I
         3) NF(5)=5

      IF(ICT.NE.0) GOTO 1203
      IF(IAMS.NE.IMS) GOTO 1200
```

```
 1203 ICT = ICT + 1

      WRITE(*,*) IMS,ISHAPE,WT,RAREAG,RFLRATIO,RADX,RADY
                   IF(ISTRAX.EQ.1) THEN
         RAXIS = RADX
         ELSE
         RAXIS = RADY
         ENDIF


      REMOD = 29000.0
      RFY = 36.
      RFU = 58.
      RYPHI = .9
      RFPHI = .75

C     MS IS MEMBER SIZE, AREAG IS THE GROSS AREA,
C     FLRATIO IS THE COMPACT SECTION CRITERIA (B/T),
C     RADX AND RADY IS THE RADIUS OF GYRATION ABOUT THEIR
RESPECTIVE
C     AXIS.


C     CALCULATE SLENDERNESS
RATIO......................................

      RSLENDR = RLE/RAXIS
      WRITE(*,*) RSLENDR,RAXIS
        IF (RSLENDR.GT.300.) THEN
        WRITE(*,400)
  400   FORMAT(5X,'DESIGN EXCEEDS SLENDERNESS LIMIT SELECT
NEW
     +STRUCTURAL MEMBER')
        ELSE

C     CALCULATE THE ALLOWABLE
LOAD....................................

        RAREAE = .75*RAREAG

        RYLOAD = (RAREAG*RFY)*RYPHI
        RFLOAD = (RAREAE*RFU)*RFPHI
        ENDIF

        WRITE(*,*) RYLOAD, RFLOAD

         IF(RYLOAD.LT.RFLOAD) THEN
         WRITE(*,1000)
       1000    FORMAT(3X,'YIELD EQUATION CONTROLS,    MAX
                             COLUMN LOAD(kips)',/)
         WRITE(*,1001) RYLOAD
 1001    FORMAT(28X,F20.2)
```

```
          RCOMPARE = RYLOAD
          ELSE
          WRITE(*,1002)
1002 FORMAT(3X,'FRACTURE EQUATION CONTROLS,    JOINT
          LOADING(kips)',/) WRITE(*,1003) RFLOAD
 1003     FORMAT(35X,F20.2)
          RCOMPARE = RFLOAD
          ENDIF
      WRITE(*,*) RCOMPARE,RFE
      IF(RCOMPARE.LT.RFE) THEN
      WRITE(*,800)
800 FORMAT(1X,'NOT ABLE TO SUPPORT LOADING, SELECT NEW
      MEMBER SIZE',/) ELSE
      WRITE(*,5000)
 5000 FORMAT(2X,'MEMBER',3X,'COLUMN LOAD (kips)',/)
      WRITE(*,9990)
IMS,ISHAPE,WT,RCOMPARE
 9990 FORMAT(2X,I2,A1,F5.1,4X,F20.2)
      ENDIF
      IF(RCOMPARE.LT.RFE) GOTO 1200

 1100 RETURN
      END

      SUBROUTINE COMPRESS (ISTRAX,IMS,ISHAPE,WT,PLOAD)
C     PROGRAM COMPRESSION ANALYSIS

C     THIS PROGRAM IS TO DETERMINE THE COLUMN STRENGTH FOR
FOR A
COMPRESSION
C     MEMBERS FOR A PLANAR TRUSS INCORPORATING THE LRFD
SPECIFICATION


     REAL*8
      LE,FE,RADX,RADY,PI,EMOD,FY,FU,SLRATIO,AXIS,LAMDAR,LAM
      DAC, +FCR,AREAG,FLRATIO,PLOAD,SLIMIT,ATT,WT,KFACT

      CHARACTER*1 IW,ISHAPE

      COMMON/CARVAR/ATT(20)
      COMMON/IO/NF(25)

      DATA IW/'W'/

      OPEN(1,FILE='GEODATA.DAT',STATUS='OLD')

      REWIND 1
      NF(5) = 5
      NF(6) = 6
      IAMS = 6
```

```
      WRITE(*,*)'ENTER EFFECTIVE LENGTH OF THE COLUMN IN
      INCHES LE =' CALL CARD
      LE=ATT(1)
C     READ(*,2000) LE

      WRITE(*,*)' ENTER EFFICTIVE LENGTH FACTOR K ='
      CALL CARD
      KFACT = ATT(1)

      WRITE(*,*)'ENTER MEMBER LOADING IN KIPS, FE ='
      CALL CARD
      FE =ATT(1)
C     READ(*,2000) FE
 2000 FORMAT(F10.2)

C      WRITE(*,*)'ENTER THE AXIS FOR WHICH MEMBER IS TO BE
EVALUATED,
C     +(1) X-AXIS,(2) Y-AXIS'
C      CALL CARD
C      ISTRAX = ATT(1)
C      READ(*,3000) ISTRAX

 3000 FORMAT(I1)


      PI = 3.141593
      EMOD =29000.
      FY = 36.
      FU = 58.

C9000 WRITE(*,*)'ENTER STRUCTURAL MEMBER SIZE MS =' C
     ICT = 0
C      CALL CARD
C      IAMS = ATT(1)
C      READ(*,1000) AMS
C 1000 FORMAT(A)


 1200 CONTINUE

      NF(5)=1
      CALL CARD
      IMS=ATT(1)
      IF(IMS.NE.182717) GOTO 1201 WRITE(*,1202)
    1202 FORMAT(2X,'END OF DATA FILE WAS REACHED WITH NO
                          SOLUTION')
      GOTO 1100
 1201 ISP = ATT(2)
      IF (ISP.EQ.36) ISHAPE = IW
      WT = ATT(3)
      AREAG=ATT(4)
      FLRATIO=ATT(5)
```

```
      RADX=ATT(6)
      RADY=ATT(7)


C     READ(1,50)
MS,AREAG,FLRATIO,RADX,RADY,ICOUNT
                                     50
       FORMAT(1X,A8,1X,F6.2,1X,F5.2,1X,F5.2,1X,F5.2)

      NF(5) =5

      IF(ICT.NE.0) GOTO 1203
      IF(IAMS.NE.IMS) GOTO 1200
 1203 ICT =ICT + 1

C      WRITE(*,*) IMS,ISHAPE,WT,AREAG,FLRATIO,RADX,RADY
       IF(ISTRAX.EQ.1) THEN
       AXIS = RADX
       ELSE
       AXIS = RADY

       ENDIF

C     CALCULATE SLENDERNESS RATIO

      SLRATIO = (KFACT*LE)/AXIS

C      WRITE(*,*) SLRATIO

      SLIMIT = 200.0

      IF(SLRATIO.LE.SLIMIT) THEN

C     CALCULATE LAMDAR TO CHECK THE FLANGE

RATIO............................
        LAMDAR = 141.0/SQRT(FY-10.0)
        WRITE(*,4500) LAMDAR
 4500   FORMAT(3X,'LAMDAR = 'F20.4,/)
        ELSE
        WRITE(*,4000)
4000 FORMAT(5X,'DESIGN EXCEEDS SLENDERNESS LIMIT, SELECT NEW
                              STRUCTURAL
     + MEMBER')
       ENDIF

        IF(SLRATIO.GT.SLIMIT) GOTO 1200

        LAMDAC = ((KFACT*LE)/(AXIS*PI))*SQRT(FY/EMOD)
        WRITE(*,*) LAMDAC
        IF(FLRATIO.GT.LAMDAR) THEN
        WRITE(*,4400)
```

```
 4400 FORMAT(5X,'DESIGN EXCEEDS COMPACT SECTION LIMIT, SELECT
                              NEW MEMBER
     +',/)
        ELSE

C     CALCULATE LAMDAC TO CHECK STRENGTH
CRITERIA............................
        LAMDAC = LAMDAC
        ENDIF


        IF(FLRATIO.GT.LAMDAR) GOTO 1200

C     CHECK THE LAMDAC TO LRFD
LIMIT.....................................

        IF(LAMDAC.LE.(1.5)) THEN
        FCR = ((.658)**(LAMDAC**2.0))*FY
        ELSE
        FCR = (.877/(LAMDAC**2.0))*FY
        ENDIF

C      WRITE(*,4600) FCR
C4600 FORMAT(3X,'FCR (PSI) =',F20.4,/)

C     CALCULATE LOAD CAPCITY OF THE
MEMBER................................

      PLOAD = AREAG*FCR*.85

      IF(PLOAD.LT.FE) THEN
      WRITE(*,8000)
8000 FORMAT(1X,'MEMBER NOT OPTIMUMIZE, SELECT NEW MEMBER
     SIZE') ELSE
     WRITE(*,5005) KFACT
5005 FORMAT(2X,'EFFECTIVE LENGTH FACTOR K =',F11.5,/)
     WRITE(*,5000)
5000 FORMAT(3X,'MEMBER',3X,'COLUMN LOAD CAPACITY (KIPS)',/)
     WRITE(*,9990) IMS,ISHAPE,WT,PLOAD
9990 FORMAT(3X,I2,A1,F4.1,6X,F20.2)
     ENDIF

      IF(PLOAD.LE.FE) GOTO 1200
1100 RETURN

                              END
```

VITA

Vernon ''Pete'' Mills, Jr. was born in Cincinnati, Ohio on Decemeber 4, 1952. He attended the Covington (Kentucky) 10th District Elementry Schools and graduated from Covington Holmes High School in June 1971. In the fall of 1971, he entered University of Cincinnati and earned a Bachelor of Science in Mechanical Engineering Technology through evening program in August 1981. He entered University of Tennessee Space Institute in June, 1991 and in December 1995 received a Masters of Science Degree in Mechanical Engineering.

He is presently employed as Senior Lead Designer at Sverdrup Technology, Inc.