Computer-Aided Analysis and Design of Steel Frames

VINCENT MEHRINGER, GEORGE PIERSON and JAMES G. ORBISON

ABSTRACT

A computer package has been developed for the analysis and design of steel plane frames. The package will, with the stiffness method, analyze a plane frame and select the least-weight wide-flange section for each member within the frame. The two modules of the package, the analysis and design, may be used independently of, or in conjunction with, one another. When used together, they interact to establish an analysis/design iteration sequence by which the least weight design is obtained. Within the analysis module, graphical displays of the structure allow for visual verification of structural data as well as analytical results. The design module minimizes data entry by making design condition assumptions, all of which can be altered. By calculating internally various design parameters, such as effective column length factors, a fast, easy design session is permitted.

The package is presently being used in upper-level structural analysis and steel design courses. A student using the package has the benefit of conceptualizing the entire analysis and design process rather than the fractionized procedure often presented in texts. To enhance further the student's understanding of the process, the package saves all values used in the final analysis and steel section selection for verification.

INTRODUCTION

A computer package has been developed for the elastic analysis and design of steel plane frames. The package will analyze a plane frame and select the least-weight wide flange section for each member within the frame. The analysis and design modules of the package may be used independently of, or in conjunction with, one another. When used together, the two modules interact to establish an automatic analysis/design iteration sequence whereby the least weight design is obtained.

Included with the analysis portion of the package are many graphics displays. Displays of the structure and its response to load permit visual verification of structural data as well as analytic results. The interaction between the two packages forms an integral part of the design process as the analytic results are used in the design module. The design module simplifies data entry by making design assumptions, all of which the analyst can alter. Additionally, the design module calculates many design parameters such as the effective column length factor.

CONFIGURATION

The package, written in standard FORTRAN 77, is presently operating on a Honeywell mainframe computer. In its current form (limited to structural models with a maximum of 100 nodes and 200 elements), 128K bytes of core memory are required. Interactive graphics capabilities are provided by using a proprietary graphics programming language developed at Bucknell University. This language is essentially a two-dimensional subset of the CORE graphics language, but has the capability of writing graphics segments to file for display at a later time. The package operates on any of several terminals and hardcopy output devices.*

ANALYSIS MODULE

The analysis portion of the package uses the stiffness method to determine displacements and internal forces

Vincent Mehringer is a Civil Engineering Graduate Student, Bucknell University, Lewisburg, PA (presently with 3D-Eye, Inc., Ithaca, NY).

George Pierson is a Civil Engineering Graduate Student, University of California (Berkeley), Berkeley, CA (presently with Camp Dresser & McKee, Inc., New York, NY).

James G. Orbison is Assistant Professor of Civil Engineering, Bucknell University, Lewisburg, PA.

**The analysis and design program will be made available to educational institutions upon request. Send inquiries to: Prof. James G. Orbison, Dept. of Civil Eng., Bucknell Univ., Lewisburg, PA. 17837.*

in plane frames. Linear elastic behavior is assumed, and connections are rigid by default. However, frictionless hinges (pinned or simple connections) can be modeled as well.

This section of the package allows the analyst to describe, examine (both graphically and numerically) and revise the model and loading. If the frame is to be designed using the design module, communication files are created automatically. Incremental analyses, where the loading and/or the frame geometry may be altered in steps, can also be performed.

Preprocessing

A previously created data file may be used to supply the required input data, or it may be developed interactively. Automatic geometry generation for rectangular frames is provided by the package. If the frame consists of one or more floors containing bays of uniform width, the program will internally generate nodal coordinates, supports, member numbers and nodal numbers. An option allows the user to number the nodes floor-by-floor or column-by-column, so as to minimize the stiffness matrix bandwidth. Bay spacing, story height and the number of bays and stories are the only required entries.

For irregular frames, several other features exist which speed the input process. Automatic constants allow member properties to be carried forward from one member to subsequently created members. Hence, common member properties need only be entered once.

Throughout preprocessing, the user is given an opportunity to display, inspect and revise the structure. To supplement the graphic display, two columns are provided for numerical output. Hence, the display screen serves as the sole and sufficient source of information. A typical display of a frame with the support conditions, loads and tabulated load data is shown in Fig. 1.

Revisions to all input data can be accomplished easily. For example, a support could be changed from a pin to a roller, or a nodal coordinate altered. Further, nodes and members can be added to or removed from the frame. Any renumbering of nodes and/or members is carried out automatically.

Analysis Capabilities

At present, a linear elastic frame consisting of up to 100 nodes and 200 members may be analyzed. An optional modeling technique available is a fully incremental frame analysis. This method allows one to enter a partial frame,

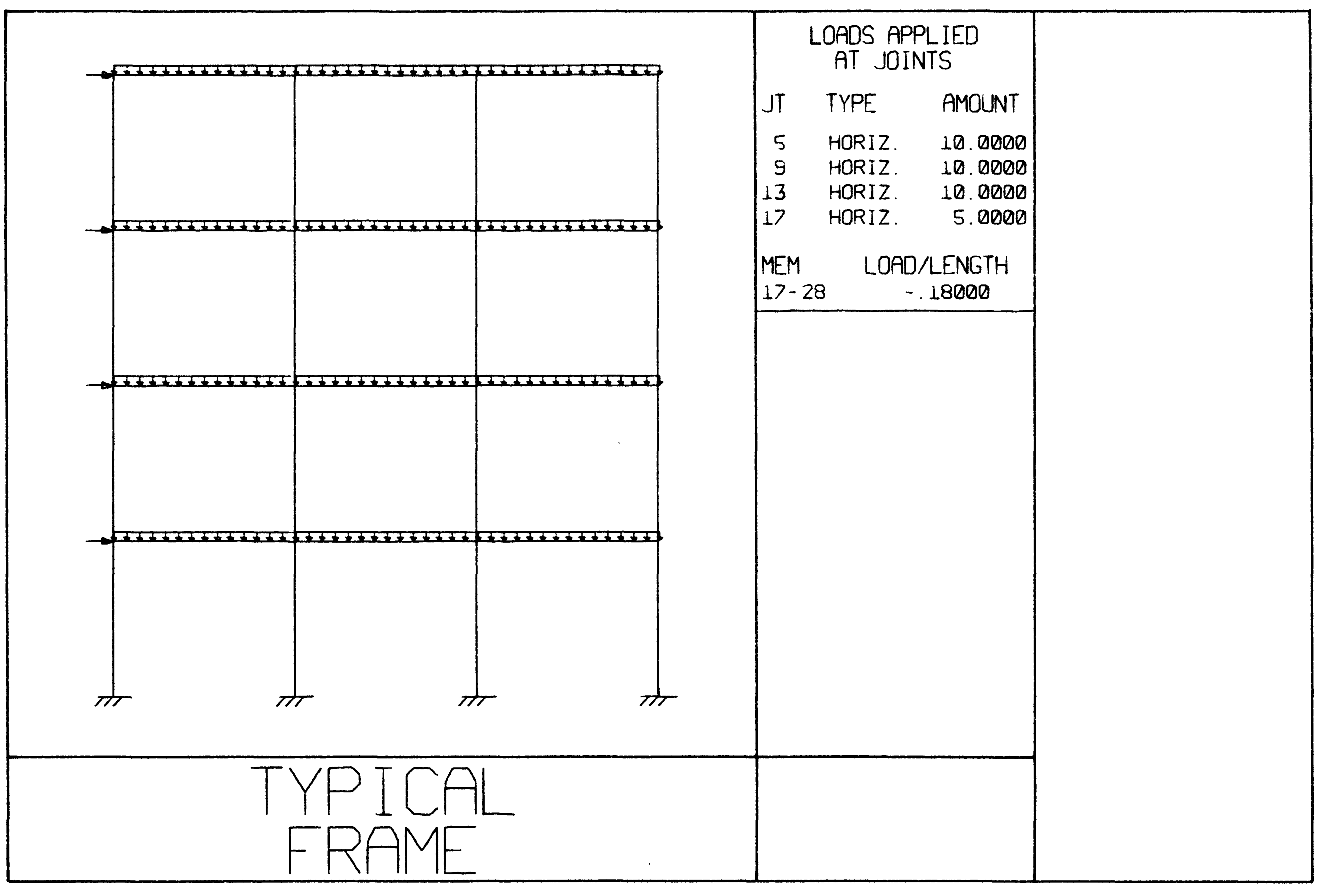


Fig. 1. Typical plot of a frame display

then analyze/design with whatever loading conditions are encountered. Another increment (additional stories for example) can be added and the expanded structure analyzed accounting for the previous geometric and loading conditions. In addition, loads can be applied (or removed) in each increment. Thus the analyst can approximate closely the actual loading and construction sequence if desired.

Though the normal beam-column element only provides for rigid connections, the program allows the analyst to place a frictionless hinge (pin) at one or both ends of any member. Hinges are modeled not as separate elements, but by modification of the stiffness matrix of the beam-column element.

Postprocessing

Analytic data (such as shear and moment diagrams) can be displayed. Thus, one can grasp quickly structural response of the frame to different loading conditions. Other displays available include joint and member numbering, the deflected shape, support conditions, loads, inflection points and graphical load and geometry alterations.

The stiffness method applied to plane frames determines horizontal, vertical and rotational deflections only at the nodes. Therefore, an expression was developed which describes for display the correct deflected shape of each member along its length. Given that concentrated loads may exist only at the member ends, that the member may be uniformly loaded perpendicular to its axis, that displacements of the two end nodes are known, and that values of shear and moment are known at each member end, deflections at intermediate points along the member can be calculated.

The relationship:

$$EI\frac{d^4v}{dx^4} = w \tag{1}$$

is integrated four times, and boundary conditions on V, M, θ and v at the ith node ($x = 0$) are applied. The intermediate deflection $v(x)$ is then given by:

$$v(x) = \frac{1}{EI}\left[\frac{wx^4}{24} + \frac{Vx^3}{6} - \frac{Mx^2}{2}\right] + x\theta_i \tag{2}$$

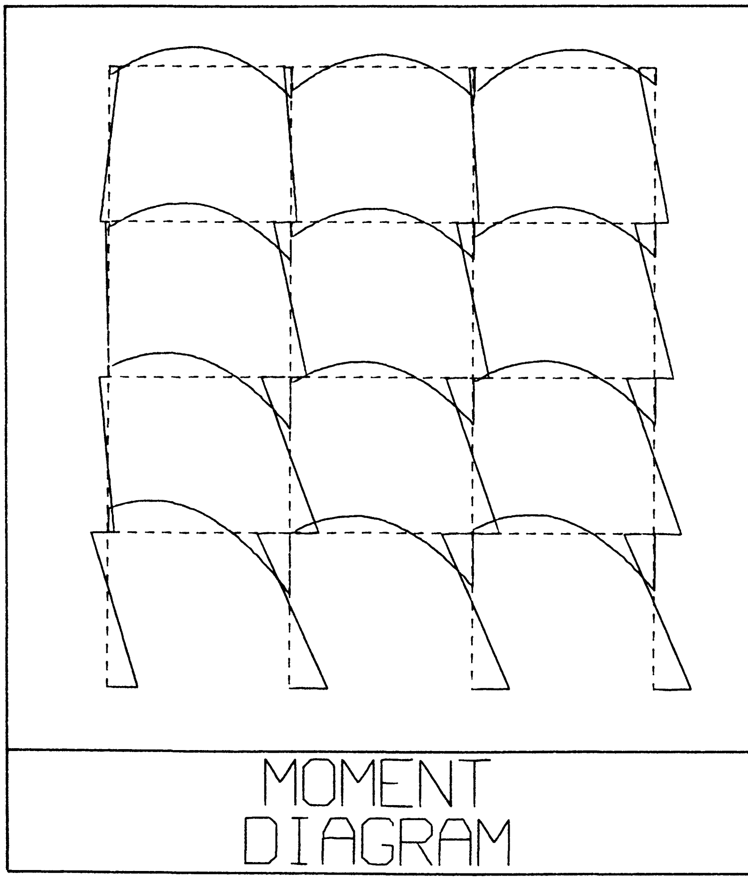
where:

w = uniform load, per unit length, in positive y direction

V = V_1, the end shear at $x = 0$

M = M_1, the end moment at $x = 0$

θ_i = member end rotation at $x = 0$



MOMENT AT MEMBER ENDS

MEM	J-JT	K-JT
1	-703.84	404.42
2	-149.95	223.84
3	-2.17	80.13
4	210.03	-213.08
5	-890.50	778.93
6	-662.56	686.06
7	-376.62	416.38
8	-139.03	187.36
9	-870.28	739.26
10	-614.68	648.88
11	-358.83	406.88
12	-108.68	128.28
13	-860.35	719.77
14	-610.92	670.13
15	-426.10	494.90
16	-304.30	411.58
17	554.37	-1435.03
17	734.51	
18	226.00	-1192.32
18	556.54	
19	-129.90	-901.35
19	425.70	
20	-213.08	-730.83
20	446.66	
21	6.45	-1239.50
21	391.28	
22	-129.65	-1085.22
22	355.98	
23	-345.93	-854.84
23	317.60	
24	-543.48	-706.12
24	277.04	
25	114.43	-1330.69
25	436.90	
26	-77.51	-1096.23
26	385.20	
27	-339.28	-799.20
27	345.45	
28	-577.84	-411.58
28	407.21	

Fig. 2. Moment diagram plot display of frame in Fig. 1

The deflection given by Eq. 2 is perpendicular to the original member axis. A magnification or scaling factor can be applied directly to this deflection for display purposes. Also, the smoothness of the plot can be altered by specifying the number of intermediate points along each member where $v(x)$ is calculated. Note that Eq. 2 accounts for the rigid body rotation of the member; rigid body translations must be considered separately.

The moment diagram display for the frame in Fig. 1 is shown in Fig. 2, and the deflected shape of this frame, with displacements magnified by a factor of 15 (and with the inflection points displayed), is shown in Fig. 3.

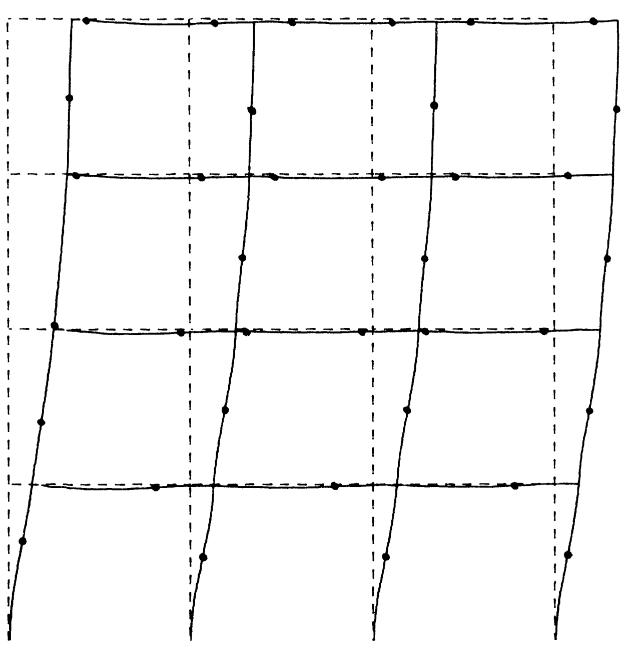
DESIGN MODULE

The design module of this package has been developed to enhance the design efficiency of the engineer by eliminating virtually all computational effort inherent in section selection. Further, the design program was developed to minimize the numerical input required of the analyst and, in all instances, to provide the easiest and most user-friendly design session possible.

Capabilities

The package uses the working stress design method in accordance with Part 1 of the American Institute of Steel Construction Specifications.[1] The selection of steel sections draws from a data base of the 187 available wide flange sections from which all section properties are obtained. The design module of the package is separated into five different sections, each designing the members applicable to that section. Design for shear is included when applicable. The five member types considered are beams, columns, tension members, beam-columns and zero-force members.

Parameters necessary for design include lateral bracing conditions, section depth limitations, bending axis, steel type, sidesway condition and wind or seismic loadings. The analyst may impose continuous or discrete lateral bracing for bending about either axis for any or all members. Discrete lateral bracing, however, may only occur at nodes. This limitation is imposed for computational efficiency. The analyst has the option of limiting the depth of sections, and may also specify the steel type to be used in the design. The depth limitation and steel type used may be uniform throughout the frame, or different for individual members. The sidesway classification (permitted or inhibited) and the section bending axis also may be specified. These too may be different for individual members. Finally, any lateral loads specified are



INFLECTION POINT LOCATIONS

MEM	DIST TO J-JT
1	107.965
2	68.198
3	4.479
4	84.387
5	90.681
6	83.519
7	80.737
8	72.416
9	91.919
10	82.699
11	79.666
12	77.970
13	92.562
14	81.071
15	78.651
16	72.262
17	135.078
18	139.239
19	9.796
19	147.346
20	15.171
20	156.065
21	131.326
22	10.565
22	136.348
23	26.459
23	145.268
24	40.001
24	150.963
25	129.531
26	6.281
26	137.124
27	25.270
27	149.179
28	37.354
28	171.883

DEFLECTED STR AND INFL POINTS

INTERVALS 10

DEFLECTIONS AMPLIFIED 15.000

Fig. 3. Plot display of the deflected shape of frame in Fig. 1

assumed to be wind or seismic loads, in which case allowable stresses are automatically increased by one-third. The analyst may change this assumption or, in the event that no lateral loads exist, specify that existing loads be treated as wind or seismic loads and allowable stresses increased by one-third.

For efficiency, the design module calculates internally all parameters required for design. These include C_b (the moment gradient coefficient), C_m (axial force-bending interaction coefficient), and most importantly the effective column length factor K.

Input Data Defaults

The design module employs several assumptions to reduce user input. These assumptions are not attempts at engineering judgment, as the engineer has the option to change any or all of the assumptions. The assumptions, as they appear in the program, are listed below:

1. Point lateral bracing (out of the plane of the frame) at all nodes,
2. No depth-of-section limitation,
3. No continuous bracing on any member,
4. All members are of steel type ASTM A36,
5. All vertical and inclined members are classified as "sidesway permitted" and all horizontal members are "sidesway prevented,"
6. All members in flexure will undergo strong axis bending, and
7. Lateral loads (if any) are assumed to be wind or seismic loads.

Continuous Members

A member is defined as being continuous if it has an internal node (included for the placing of a point load in the analysis or for discrete lateral bracing in the design) and is to exist as a single wide-flange section. Continuous members are comprised of what are termed individual members. Individual members are those appearing in the analysis model regardless of how the design is affected. Thus, an extra node added so that discrete lateral bracing may be placed at its location separates a continuous member into two individual members. The individual members appear in the analysis and are combined into the appropriate continuous member in the design.

Four criteria are used to identify continuous members from individual members. First, for any two members to be continuous they must share a common node. Second, for two individual members to be considered continuous their angles of inclination must be equal. Third, the common node must not have more than these two individual members framing into it. The final criterion states that for a member to be continuous with another member it must have at least one node which is totally unrestrained. This restricts the design of what is traditionally termed a continuous beam (i.e. a beam continuous over several supports), but is necessary for the efficient determination of design lengths of the continuous members. However, this limitation is surmountable and will be discussed later.

The identification of continuous members from individual members is an important capability of the design program. The frame transformation process frees the engineer of the task of piecing together the frame from smaller components. By automating the transformation process, an uninterrupted analysis and design sequence flow is realized.

Effective Column Length Factor

A major facet of frame design is the calculation of effective column length factors, or K values. The design program includes the nontrivial capability of internally calculating effective lengths from existing data. The algorithm used is that recommended by the AISC *Manual of Steel Construction*. The nomographs used by AISC were developed from the following equations for the effective length factor of a member:[2]

Sidesway permitted:

$$\frac{\pi^2 G_t G_b}{4K^2} + \frac{G_t + G_b}{2}\left[1 - \frac{\pi}{K\tan(\pi/K)} + \frac{2K\tan(\pi/2K)}{\pi}\right] = 1.0 \quad (3)$$

Sidesway prevented:

$$\frac{\dfrac{\pi^2 G_t G_b}{K} - 36}{6\,(G_t + G_b)} - \frac{\pi}{K\tan(\pi/K)} = 1.0 \quad (4)$$

where:

$G_t = \Sigma\,[(I_c/L_c)/(I_g/L_g)]_t$

$G_b = \Sigma\,[(I_c/L_c)/I_g/L_g)]_b$

[I_c, L_c, I_g and L_g are the moments of inertia and lengths of the columns and girders, respectively, framing into the top (t) and bottom (b) of the member]

The method of bisection is used to compute the effective length factor from Eqs. 3 or 4, as applicable. The effective lengths of members for out-of-plane buckling are assumed to equal the actual out-of-plane unbraced lengths.

The design module has the capability, as does the analysis module, to act as a stand-alone program, thus permitting design of individual members. Here, since no information on the other members framing into the individual member exists, calculation of an effective length by the previously stated algorithm is impossible. The data needed for the calculation of an approximate K value is obtained from a user-defined relative degree of rotational restraint ranging from 0.0 to 1.0. A value of 0.0 would correspond to zero rotational restraint (a pinned connection) and a value of 1.0 would correspond to total rotational fixity. The following equation was developed in this work to calculate G_t and G_b for determination of K:

$$G_t \text{ or } G_b = 10(1 - R_s) \quad (5)$$

R_s = the degree of rotational restraint of the respective joint

This equation was developed through examination of the nomographs resulting from Eqs. 3 and 4. For a sidesway inhibited frame, the effective column length resulting from these empirical equations for G_t and G_b is exactly that obtained from similar conditions when calculating G_t and G_b as specified for Eqs. 3 and 4. For a sidesway permitted condition, however, there is a limitation on the resulting effective column length. From the Equation 5 for G_t and G_b and Eqs. 3 and 4, it can be seen the maximum effective column length factor will not exceed 3.0. However, since the AISC Specification states the maximum value for G "may be taken as '10' for practical design,"[1] a value of 10 for both G_t and G_b results in a K value of 3.0, the maximum for these equations. Once the values for G_t and G_b are calculated internally, K is determined as before.

A final note on the capabilities of the design program is in order. The design program can accept a user-input wide flange section to be checked for adequacy. An existing section may be entered for a member which is part of a frame and that member will be checked for adequacy under the existing conditions. The remaining members in the frame, however, will not be checked, but rather will be designed. If a section is checked for adequacy and determined to be inadequate, the appropriate, least weight wide flange section will be chosen for it. As alluded to previously, this capability also allows the engineer to design structures such as beams continuous over several supports. The design may be completed and the section chosen for any span checked for adequacy with regard to the other spans.

PROGRAM INTERACTION

Interaction between the two programs is often essential for design optimization. Member properties are not known before design and therefore must be assumed. After analysis and completion of a design based on that analysis, the properties of the selected sections are passed back to the analysis module for re-analysis. Once the second analysis is complete, the new results are transferred to the design module to be used in the second design iteration. The analysis/design process continues automatically until design convergence is obtained. Convergence is said to occur when sections selected for all members of the frame remain unchanged between two analysis/design iterations. This procedure produces a least-weight frame design using wide flange sections for the given geometry and loading.

To efficiently obtain the most accurate design possible, the member weights of the selected wide-flange sections are accounted for internally. After the completion of the first design, the member weights are added to the loads acting upon the frame. Sample design results of the frame shown in Fig. 1 are shown in Table 1. For this example, gravity and wind loads were considered, sidesway is permitted, full lateral support of the girders was assumed, A36 steel employed and strong axis bending was assumed throughout. Four analysis/design iterations were performed to obtain the final section selections (note, however, the number of iterations required will vary, from a minimum of two, depending upon the initial estimate of the section properties).

The complete analysis/design session, from the start of the data entry through generation of final results, required less than 10 minutes at the terminal. The effective length factor K for each member given in Table 1 is that corresponding to the governing (maximum) slenderness ratio. Members 1 through 16 are columns, and the out-of-plane or weak axis slenderness ratio controls with K

Table 1. Partial Listing of Final Design Results for the Frame in Fig. 1

Original Member	Section Chosen	f_b (ksi)	f_a (ksi)	C_m	C_b	K
1	W14×30	8.68	6.03	2.03	0.85	1.000
2	W16×26	2.98	5.57	2.30	0.85	1.000
3	W8×18	2.15	5.83	2.13	0.85	1.000
4	W14×22	9.89	2.48	2.30	0.85	1.000
5	W18×55	12.42	9.38	2.26	0.85	1.000
6	W21×50	9.00	7.70	2.30	0.85	1.000
7	W14×30	10.53	8.60	2.30	0.85	1.000
8	W8×21	11.16	6.16	2.30	0.85	1.000
9	W18×55	11.89	9.24	2.30	0.85	1.000
10	W18×50	8.38	7.69	2.30	0.85	1.000
11	W14×30	9.51	8.37	2.30	0.85	1.000
12	W16×26	3.43	4.92	2.30	0.85	1.000
13	W21×50	14.71	5.80	2.30	0.85	1.000
14	W14×30	11.39	6.89	2.30	0.85	1.000
15	W14×30	12.93	4.38	2.30	0.85	1.000
16	W14×22	17.64	2.76	2.30	0.85	1.000
17	W16×31	29.10	0.92	0.00	0.00	0.664
18	W14×30	28.79	1.04	0.00	0.00	0.688
19	W16×26	24.75	0.88	0.00	0.00	0.781
20	W14×22	25.63	1.22	0.00	0.00	0.832
21	W14×30	28.56	0.90	0.00	0.00	0.626
22	W14×30	29.11	0.45	0.00	0.00	0.626
23	W14×22	28.76	0.63	0.00	0.00	0.740
24	W14×22	24.74	0.93	0.00	0.00	0.839
25	W14×30	28.08	0.80	0.00	0.00	0.610
26	W12×26	28.80	0.08	0.00	*	*
27	W14×22	29.01	0.20	0.00	0.00	0.672
28	W12×19	27.51	0.86	0.00	0.00	0.731

equal to unity. Members 17 through 28 are the girders, fully braced in the out-of-plane direction. The K values listed hence correspond to the in-plane (strong axis) slenderness ratio. Member 26 is the only member subjected to a tensile axial force. K and C_b values thus do not apply to the design of that member.

SUMMARY

The computer package developed for the analysis and design of steel plane frame structures has application in engineering education. The user-friendly attributes of the package make it easy to use by students learning analysis and steel design processes. The package allows the student to check all results by hand. Final values (including all design parameters) used in both the analysis and design modules may be written to an output file.

The package is presently being used in upper-level analysis and steel design courses. The speed of the package enables the student to quickly determine the response of a design to many different loading combinations, thus providing more realistic problems than those often found in textbooks. The students also can more fully appreciate the interaction between, and interdependence of, analysis and design that is not otherwise apparent from separate analysis or member design problems.

Use of this package enhances the student's understanding of structural behavior and realistic design problems.

ACKNOWLEDGEMENTS

The authors would like to thank the Bethlehem Steel Corporation for their support of the development of this package. Also, the authors would like to thank Francine Solari for her invaluable assistance in the development of the design module.

REFERENCES

1. *American Institute of Steel Construction, Inc.* Manual of Steel Construction Eighth Edition, *Chicago, Ill., 1980.*
2. *T. V. Galambos* Structural Members and Frames *Prentice-Hall, Inc., Englewood Cliffs, N.J., 1968.*