Optimum Design of Steel Pipe Racks

SURESH C. ARYA, EDWARD G. FENG AND GEORGE PINCUS

The availability of user-oriented computer programs has had great impact on engineering analysis and design of structural systems. This computational tool can be useful not only to obtain fast and accurate values of forces and displacements in structural systems, but its usefulness has been extended to the economical (minimum weight) design of structural steel members and frames. The design phase can be made very efficient by following an optimization process when suitable computer programs are available. The optimization process may include a number of design constraints, such as: (a) attainment of overall minimum weight, (b) limiting the selection process to a given range of member sizes, and (c) limiting displacements to certain tolerable values.

Current engineering practice often utilizes the capability of the computer for the analysis phase only, and considers the subsequent design process a professional art to be left to the discretion of the "Design Engineer." There are reasons for this distinction between analysis and design. For example, the application of a design code requires experience and judgement, the selection of member sizes should be practical, and there is usually a match among the selected member sizes. However, some computer programs available today are so versatile that they not only include code design criteria as built-in algorithms, but in addition the input procedure is relatively simple and easy to learn and apply. In fact, many engineers are able to understand the computer code in their first encounter with a coded problem even if lacking in prior computer experience. One user-designed computer program which has achieved this capability is the MIT-developed Integrated Civil Engineering System, Structural Design Language (ICES, STRUDL-II).[1,2] This program is available to the professional and is marketed by many commercial computer service bureaus.

This paper describes the use of the STRUDL II computer program for the optimum design of a steel pipe rack structure commonly used in petrochemical plants. The procedure leads to substantial weight (and cost) savings in most situations. Considering the tonnage of steel used in petrochemical plant pipe racks, the savings potential may be truly remarkable.

STRUCTURAL SYSTEM—PIPE RACK

A two-dimensional frame representing a pipe rack structure is shown in Fig. 1. Even though the example presented here is two-dimensional, most designers will consider three-dimensional action due to forces acting perpendicular to the plane of the pipe rack (temperature expansion and contraction or longitudinal wind and/or earthquake). For simplicity, the example considers plane frame action. Many typical racks will not include all the support conditions

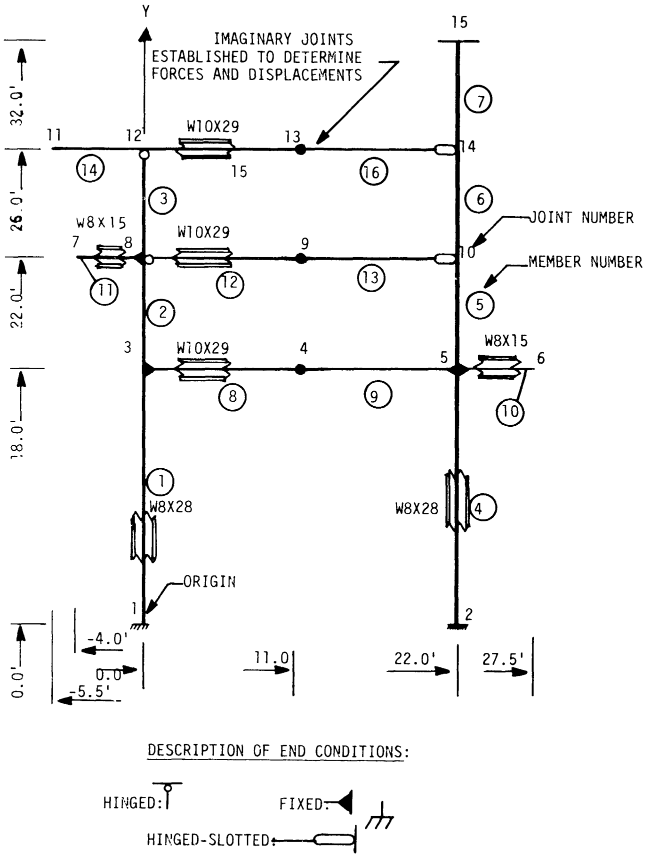


Fig. 1. Typical pipe rack steel frame (computer model input)

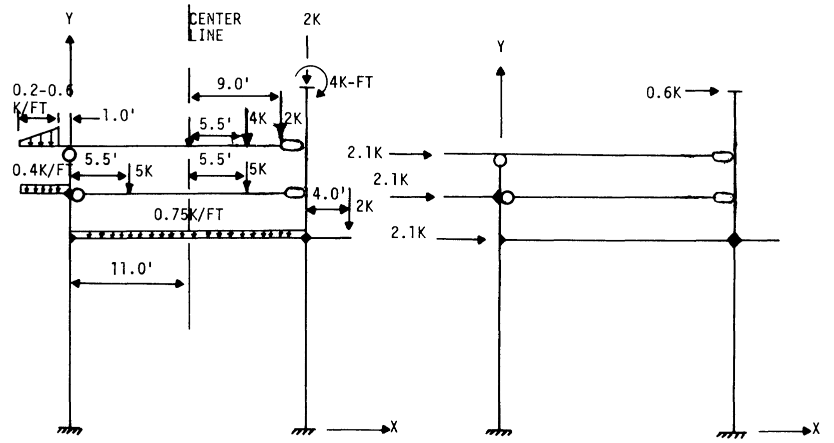
Suresh C. Arya is Principal Design Engineer, CE Lummus Company, Houston, Tex.

Edward G. Feng is Senior Design Engineer, CE Lummus Company, Houston, Tex.

George Pincus is Professor and Chairman, Department of Civil Engineering, University of Houston, Houston, Tex.

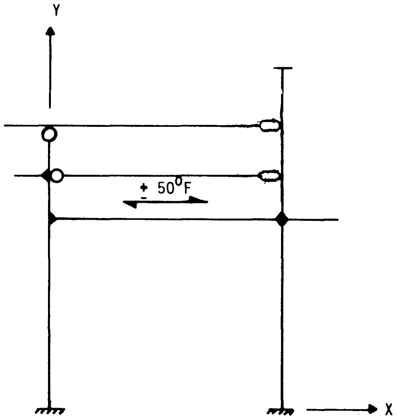
shown in this example and will, therefore, be much simpler to design. The frame includes three levels of beams, and each beam has a cantilever overhang at one end. One column is extended upwards to function as a "tee" support for pipes at a higher elevation. The lower level beam has its ends fixed (moment connection) to the columns. The intermediate level beam has one end hinged and the other end is allowed to slide horizontally to provide for possible expansion (connection to the column is through slotted holes). The upper level beam is continuous over the column and becomes a cantilever overhang on one end, and is allowed to slide at the other end. The bases of the columns are considered fixed. The frame has a 22-ft span and an overall height of 32 ft. The frames are spaced at 25 ft and are connected to adjacent frames by horizontal longitudinal beams at each cross-beam level. These frames are restrained in the longitudinal direction by vertical cross-bracing which is usually placed at a spacing of about 100 ft or at each fourth bay. Only the in-plane design of the pipe rack is considered here, although three-dimensional action can be included with minimal additional effort.

The loads that act on the frame are shown in Fig. 2. Three independent loadings are considered for design purposes. The loadings corresponding to permanent loads, wind, and temperature changes are shown in Figs. 2a through 2c. Temperature change is only considered on the lower level beam, since the upper level beams are free to expand or contract longitudinally. Design loading combinations are indicated in Figs. 3a through 3d. Out-of-plane loadings, such as longitudinal expansion and contraction forces and longitudinal wind/earthquake loads, could be considered as additional loading conditions. Initial sizes for the members are selected through manual analytical procedures or the sizes might be assumed. These member sizes are then used as initial trial sizes in the design optimization process and are indicated in Fig. 1. The selection of member sizes can also be arbitrary, as noted above. Regardless of what initial sizes are used, subsequent program iterations will converge to the *same* final design selection. However, the exact number of iterations will vary, depending on how close the initial trial sizes are to the final computer-designed sizes.



(a) PERMANENT LOADS 'DED+LIVE'

(b) WIND LOAD 'WIND'



(c) AXIAL TEMPERATURE EXPANSION AND CONTRACTION 'TEMP*50F'

Fig. 2. Basic loading conditions

DESIGN PROCEDURE

Analysis and design optimization are accomplished by the computer program during the following steps (see Fig. 4 for program flow chart):

1. Analysis of the frame for the given loading conditions, and combining the results for the design phase.
2. Trial members are checked against the provisions of the AISC Specification for strength adequacy for all loading conditions.
3. Member sizes are selected by the computer program algorithm without applying any constraint condition. The AISC Specification is used.
4. The member sizes selected in step 3 are used to perform a new stiffness analysis, and again member sizes are selected by the computer program algorithm. This step serves the purpose of optimization without applying any constraint condition, i.e., the difference of weights between step 3 and step 4 will be negligible.
5. Size of members which are continuous are made equal for practical reasons; for example, a column is a continuous member of uniform size.
6. Constraint conditions regarding a given range of member dimensions are applied, and a new set of member sizes is obtained using the AISC Specification for all design loading conditions. Subsequently, step 5 is repeated to arrive at practical member sizes.
7. Using the new member sizes selected in step 6, the frame is reanalyzed for all loading conditions. Using these new design forces, the member sizes are again selected such that the limiting deflection constraint condition (which is applied at specific controlling points of the frame) is satisfied. This deflection constraint condition is in addition to the previous constraint condition listed under step 6. Step 5 is again repeated to arrive at practical member sizes.
8. The member sizes obtained from the previous step are used once more to reanalyze the frame and as a

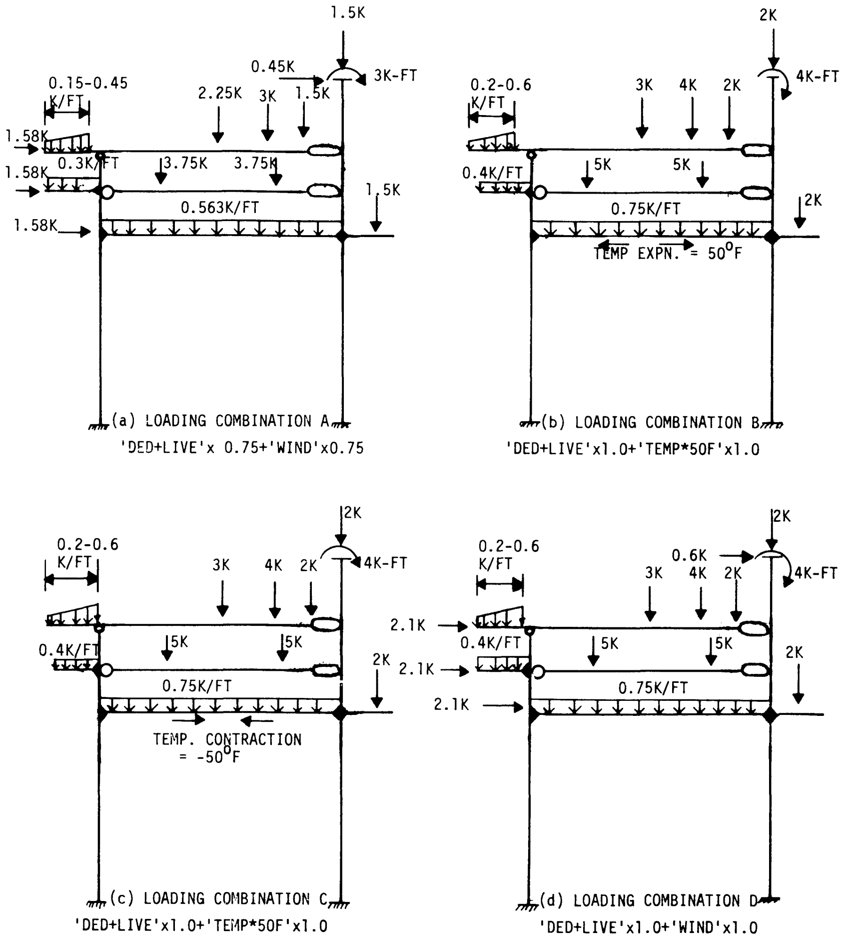


Fig. 3. Loading combinations

final AISC Specification check. This step assures that the member forces obtained from the latest stiffness analysis have not changed significantly, which would require a further change in member sizes.

9. Finally, the member force, the reactions at the supports, and the deflection of the frame are tabulated for subsequent design. These results are normally required for the design of the connections, including the column base details and the foundations, and also to further assure that the required deflection limitations have been achieved.

Summarizing the previous nine steps for the optimum design process:

Steps 1–5 accomplish minimum weight design.

Step 6 accomplishes minimum weight design with a constraint condition on desirable dimensions (width and depths) of members.

Step 7 accomplishes minimum weight design with constraint conditions on member dimensions *and* limiting deflection criteria.

Steps 8–9 document the results of steps 1–2.

COMPUTER MODELING

Computer modeling is a technique of frame idealization such that computer coding may be conveniently applied. The following steps are followed in coding a steel pipe rack for STRUDL-II computer processing:

1. The frame is described by a single line diagram showing the proper end conditions, as given in Fig. 1.
2. The location of the joints is selected. The joints are located at: (a) the support points, (b) the free end of members, (c) the intersection of beams and columns, and (d) the mid-points of beams. Joints may also be

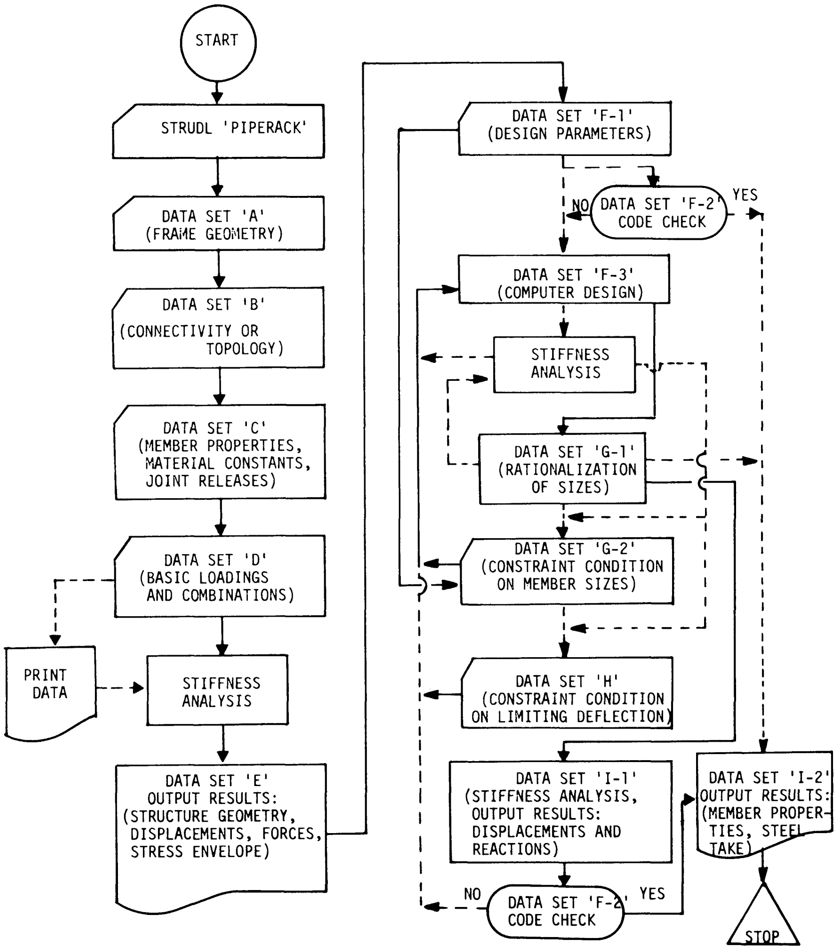


Fig. 4. Computer program input/output flow chart.
(Note: Broken lines indicate optional flow path)

established at any other points where knowledge of displacements and forces is desired. Any joint (usually the left bottom support) is selected as the origin and *X* (horizontal) and *Y* (vertical) global axes are indicated as shown in Fig. 1. The global *Z*-axis is perpendicular to the frame. The coordinates of each joint from the origin are then computed. Joints are numbered in ascending order such that the difference in numbering between joints at the two ends of a member is as small as possible. For pipe racks, the joint numbering scheme is not as critical as in a structure with many joints and, therefore, may be arbitrary.

3. A member always exists between two joints. These members are numbered in sequential order and are indicated within a circle in Fig. 1 in order to create differentiation between joint and member numbers.
4. The loading conditions are shown in Fig. 2. Loads which may act independently are indicated on separate sketches, e.g., as shown in Figs. 2a through 2c. Combined loadings formed from the factored sum of independent loadings are shown in Figs. 3a through 3d.

INPUT TO COMPUTER PROGRAM—EXAMPLE

Input data is taken from Figs. 1 through 3. The data is entered in groups in certain sequential logical order. This procedure is described below using the example problem.

A typical pipe rack frame which is used in the petrochemical industry is shown in Fig. 1, and the basic and design loadings are shown in Figs. 2 and 3, respectively. A constraint condition on deflection is selected, i.e., the maximum vertical deflection in the beams at joints 4, 9, and 15 should not exceed 1.0 in. and the lateral displacement of the frame at joint 3 is limited to 0.75 in. The input data is coded on 80 column sheets, and a free format procedure may be followed, i.e., data may be coded in any column and extra spaces between data are ignored. Each command statement or line of data should occupy a single line. There are nine sets of data, A through I, which are required to code and solve this problem, and these are illustrated in Tables 1–9. These data sets must be preceded by a command STRUDL, which activates the computer program stored in the system. The command statements and the input data which are key punched on computer cards are shown on the left side of Tables 1 through 9 in capital letters. Explanation for each command is given on the right

Table 1. Coordinates of Frame Geometry

Input Data Set 'A'	Explanation
UNIT FEET	Indicates length units
TYPE PLANE FRAME	Indicates structure type
JOINT COORDINATES	
1 0. 0. SUPPORT	Describes the geometry of the frame and points where design information is desired.
2 22. 0. SUPPORT	
3 0. 18.	
4 11. 18.	
5 22. 18.	
6 27.5 18	
7 -4.0 22.	
8 0. 22.	
9 11. 22.	
10 22. 22.	
11 -5.5 26.	
12 0. 26.	
13 11. 26.	
14 22. 26.	
15 22. 32.	
	Y-coordinate (Z-coordinate automatically set to zero)
	X-coordinate
	Joint Number

Table 2. Connectivity of Structure Topology

Input Data Set 'B'	Explanation
MEMBER INCIDENCES	Indicates connection of member between joints
1 1 3	
2 3 8	
3 8 12	
4 2 5	
5 5 10	
6 10 14	
7 14 15	
8 3 4	
9 4 5	
10 5 6	
11 7 8	
12 8 9	
13 9 10	
14 11 12	
15 12 13	
16 13 14	

End joint number (End joint is always further away from origin than start joint)

Start joint number

Member number

Table 3. Member Releases, Properties and Elastic Constants

Input Data Set 'C'	Explanation
MEMBER RELEASES	Release the ends of member to model the desired boundary conditions.
3 13 16 END MOMENT Z	Makes the far end of members 3, 13 and 16 hinged.
12 START MOMENT Z	Makes the near end of member 12 hinged.
13 16 END FORCE X	Makes the far end of members 13 and 16 free to move horizontally (slotted).
UNIT KIPS INCHES	
MEMBER PROPERTIES	The section properties of members follow.
1 TO 7 TABLE 'STEELW' 'W8X28'	Structural shapes assumed for members as noted, e.g., 1 through 7 are W8X28.
8 9 12 13 14 15 16 TABLE 'STEELW' 'W10X29'	
10 11 TABLE 'STEELW' 'W8X15'	
CONSTANTS E 29000. ALL	Material modulus of elasticity.
CTE .0000065 ALL	Coefficient of thermal expansion.
BETA 0. ALL	Orientation of member section, i.e., local member strong axis is parallel to the global Z-axis.
UNITS IN POUNDS	
CONSTANT	
DENSITY .284 ALL	Weight density of material.

Table 4. Basic Loadings and Combinations

Input Data Set 'D'	Explanation
UNITS KIPS FEET FAHRENHEIT	
LOADING 'DED+LIVE' 'PERMANENT LOADS (DEAD+MAX LIVE)'	Describes loading condition due to permanent loads.
DEAD LOAD WITH MOMENTS Y -1.0	Member dead load including fixed end moments.
JOINT LOADS	
13 FORCE Y -3.	Concentrated loads and moments acting at joints, e.g., a force of 3 kips at joint 13 acting down.
15 FORCE Y -2. MOMENT Z 4.	
MEMBER LOADS	Loads acting over the length of the member.
8 9 FOR Y UNIFORM W -.75	Uniform load over full length of members 8 and 9 of intensity 0.75 kips/ft.
11 FOR Y UNIFORM W -.4 LA 0. LB 3.0	Uniform load over partial length i.e., from start to 3 feet from start.
14 FOR Y LINEAR WA -.2 WB 0.6 LA 0. LB 4.5	Linearly varying load over partial length with initial intensity .2 kips/ft. at start to .6 kips/ft. 4.5 ft. from start acting downward (negative y-direction).
10 FOR Y CONCENTRATED P -2.0 L 4.0	Concentrated load applied at a distance from start, e.g., 2 kips on member 10, at 4 ft. from start acting down (negative y-direction).
12 13 FOR Y CON P -5.0 L 5.5	
16 FOR Y CON P -4.0 L 5.0	
16 FOR Y CON P -2.0 L 9.0	
LOADING 'WIND' 'WIND LOAD LATERAL DIRECTION'	Describes another loading condition due to wind.
JOINT LOADS	Loads applied at the joints.
15 FOR X 0.6	A force of 0.6 kips is acting in horizontal direction. on joint 15.
3 8 12 FOR X 2.1	
LOADING 'TEMP*50F' 'TEMP VARIATION RISE 50F'	Describes another loading condition due to temperature.
MEMBER 8 9 TEMP AXIAL 50	Axial temperature change of 50 degrees F applied to beams 8 and 9.
LOADING COMB 'A' 'PERMANENT LOADS+WIND' -	Combines two basic loading conditions for design of member, i.e., 0.75 x 'DED+LIVE' plus 0.75 x 'WIND'. (the end dash indicates continued on next card).
COMBINE 'DED+LIVE' 0.75 'WIND' .075	
LOADING COMB 'B' 'PERMANENT LOADS+TEMP RISE +50F' -	Combines the basic loading condition with temperature expansion of 50 degrees F.
COMBINE 'DED+LIVE' 1.0 'TEMP*50F' 1.0	
LOADING COMB 'C' 'PERMANENT LOADS+TEMP FALL -50F' -	Combines the basic loading condition with temperature contraction of 50 degrees F.
COMBINE 'DED+LIVE' 1.0 'TEMP*50F' -1.0	
LOADING COMB 'AA' 'FULL PERMANENT LOADS+WIND' -	Combines two basic loading conditions for limiting deflection constraint condition, i.e., 1.0 x 'DED+LIVE' plus 1.0 x 'WIND'.
COMBINE 'DED+LIVE' 1.0 'WIND' 1.0	

Table 5. Stiffness Analysis and Output Results

Input Data Set 'E'	Explanation
PRINT DATA ALL	Prints out input data.
STIFFNESS ANALYSIS	This command performs structural analysis of the frame.
PLOT PLANE	Structure geometry is printed out as a check on the input.
UNITS KIPS INCHES	
OUTPUT BY JOINT	
LIST DISPLACEMENTS ALL	Results are printed out for deflections and rotations of joints.
OUTPUT BY LOADS	
LIST FORCES REACTIONS ALL	Results are printed out for forces, moments at joints, and reactions at supports.
OUTPUT BY MEMBERS	
SECTION FRACTION NS 2 0. 1.	
LIST FORCE ENVELOPE ALL MEMBERS	Maximum forces and moments in each member are printed out. Considers all loading conditions, and two sections on each member, the start and the end.
LIST STRESS ENVELOPE ALL MEMBERS	Maximum stresses in each member is printed out similar to above.

side of each table. Each table contains the following information (capital letters are actual commands):

Data Set 'A' and 'B' (*Tables 1 and 2*)—Information is taken from Fig. 1 and includes a description of the frame geometry and topology. Note that for three-dimensional action, TYPE PLANE FRAME, would become TYPE SPACE FRAME and three sets of coordinates ($x,y,z,$) would be listed for each joint.

Data Set 'C' (*Table 3*)—Information on member releases (program assumes all member ends fully fixed, unless modified by this command) and member properties are taken from Fig. 1. The parameters under the command CONSTANTS are for steel material. The value of BETA is taken as zero, since the web of the sections is in the plane of paper (members are oriented such that bending is about their strong axis). BETA becomes 90 degrees when the web of a member is perpendicular to the plane XY.

Data Set 'D' (*Table 4*)—Information for loading conditions is as shown on Fig. 2. Three loading combinations 'A', 'B', 'C' are performed following current design practice, and combination 'AA' is formulated to check for the limiting deflection criteria. Information on loading combinations is obtained from Fig. 3.

Data Set 'E' (*Table 5*)—This data set includes commands for the listing of the stiffness analysis results. The results include the forces in the members, displacements of the free joints, and reactions at the supports for all basic and combined loading conditions. Maximum stress envelope results are also printed out for design purposes.

Data Set 'F' (*Table 6*)—This data set defines the parameters that are required for the design of steel members. Effective and unbraced lengths for columns are defined, but beam and cantilever members are considered to be totally braced along their compression flange, since they are attached to pipes perpendicular to their lengths. A code check on the assumed member sizes is performed for strength adequacy. Subsequently, member sizes are selected by the computer program for the forces generated by the stiffness analysis of data set 'E'. Another cycle of stiffness analysis is performed using the new sizes as input for member properties. Then, results of the stiffness analysis are used to select new member sizes.

The preceding steps accomplish minimum weight optimization using the computer program without applying any constraint condition.

Table 6. Prescribed Design Parameters, AISC Specification Check and Computer Design

Input Data Set 'F-1'	Explanation
LOADING LIST 'A' 'B' 'C' 'DED + LIVE'	Design loading conditions are considered.
PARAMETERS	Design parameters are prescribed.
'CODE' 'SP69' ALL	AISC-Code is used.
'UNLCF' 66.0 8 9 12 13 15 16	Unbraced length of member compression flange, i.e., 66 inches for members 8, 9, 12, 13, 15 and 16.
'FLYD' 36. ALL	Yield strength of material.
'SECNDARY' 1 ALL	Indicates that all members are primary members.
'PRIDTA' 1 ALL	No diagnostic output is desired.
'KZ' 1.2 MEM 1 4	Effective length factor for columns in local Z-axis, i.e., the plane of the frame.
'KZ' 2.0 MEM 2 3 5 6 7	
'KY' 0.80 MEM 3 6 7	Effective length factor for columns in local Y-axis, i.e., perpendicular to the frame.
'KY' 0.65 MEM 1 2 4 5	
'LZ' 240.0 MEM 1 4	Unbraced length against buckling in local Z-axis, i.e., in the plane of the frame.
'LZ' 96.0 MEM 7	
Input Data Set 'F-2'	
CHECK CODE FOR MEMBERS 1 to 16	AISC-Code check is performed on all members.
Input Data Set 'F-3'	
SECTION FR NS 2 0. 1. MEM 1 to 16	Sections are selected for member design considering moments and axial forces at both ends.
SELECT MEM 1 to 16 WITH 'COMBINED'	AISC-Code axial and bending forces interactive formulae (axial load and moment) are used.

Data Set 'G' (Table 7)—The constraint condition on member dimensions are given in this table. A stiffness analysis is performed on the member sizes selected in the previous step. Member sizes are reselected for the new member forces, taking into account the constraint conditions. Segments of continuous members may be selected by the computer program with different sizes. Thus, segments common to a single or continuous member are made equal, based on the largest required moment of inertia for any one segment, since flexural stresses control the design process.

Data Set 'H' (Table 8)—The commands in this data set apply an additional constraint condition on maximum permissible deflections at certain joints (in the vertical direction at mid-span of the beams and in the horizontal direction at the intersection of the first-level beam and column). First, the stiffness analysis is performed on the frame using the member sizes obtained in the preceding table. Redesign of those members which are affected by the constraint condition is performed by applying the constraint condition of limiting deflection. Continuous member sizes are rationalized and section properties are printed.

Data Set 'I' (Table 9)—The commands in this table reanalyze the frame for all the loading conditions, a code check of the members is performed, and a printout of the final deflections at the joints, the support reactions, and the member properties is produced.

Table 7. Rationalization of Sizes and Constraint Condition of Member Sizes

Input Data Set 'G-1'	Explanation
TAKE MEM 1 TO 3 AS LAR 'IZ' of MEM 1 to 3	Same section based on maximum moment of inertia is used on entire length of each column.
TAKE MEM 4 TO 7 AS LAR 'IZ' OF MEM 4 TO 7	
TAKE MEM 14 15 16 AS LAR 'IZ' of MEM 14 15 16	Same section based on maximum moment of inertia is used on entire length of each beam.
TAKE MEM 12 13 AS LAR 'IZ' OF MEM 12 13	
TAKE MEM 8 9 AS LAR 'IZ' OF MEM 8 9	
TAKE MEM 10 11 AS LAR 'IZ' OF MEM 10 11	Same section is used for noncontinuous cantilevers based on maximum moment of inertia.
PRINT MEMBER PROP ALL MEMBERS	Table of member sizes is printed out.
STEEL TAKE OFF	Steel weight is calculated for the current sections.

Input Data Set 'G-2'	
MEMBER CONSTRAINT	Member sizes can be limited to desired range of dimensions.
1 TO 7 CONS 'YD' GE 7.5	Column least depth is indicated, e.g., members 1 through 7 should be at least 7.5 inches deep.
1 TO 7 CONS 'YD' LE 18.0	Columns maximum depth be less or equal to 18 inches.
1 TO 7 CONS 'ZD' GE 5.0	Column flange least width is indicated, e.g., members flange width should be equal or greater than 5.0 inches.
1 TO 7 CONS 'ZD' LE 10.0	Column's maximum flange width be equal or less than 10 inches.
8 9 12 13 15 16 CONS 'YD' GE 9.5	Minimum beam depth of 9.5 in. or less is indicated for members 8, 9, 12, 13, 15 and 16.
8 9 12 13 15 16 CONS 'ZD' GE 5.0	Maximum beam flange width of 5.0 in. or greater is indicated for members 8, 9, 12, 13, 15 and 16.

Table 8. Constraint Condition on Limiting Deflection

Input Data Set 'H'	Explanation
LOADING LIST 'AA'	Only loading combination for which deflection constraint condition is prescribed is considered.
ADDITIONS	Additional design parameter are described.
PARAMETER	
'DEFLECTN' 132.0 8 12 15	Deflection constraint is described on beams, e.g., a maximum deflection of L/132 is desired, which is 1 in.
'DEFLECTN' 288.0 1 4	Deflection constraint is described on columns, e.g., a maximum deflection of L/288 is desired, which is 0.75 in.
'LOADING' 'AA' 1 4 8 12 15	Loading condition is also selected.

Table 9. Stiffness Analysis and Output Results

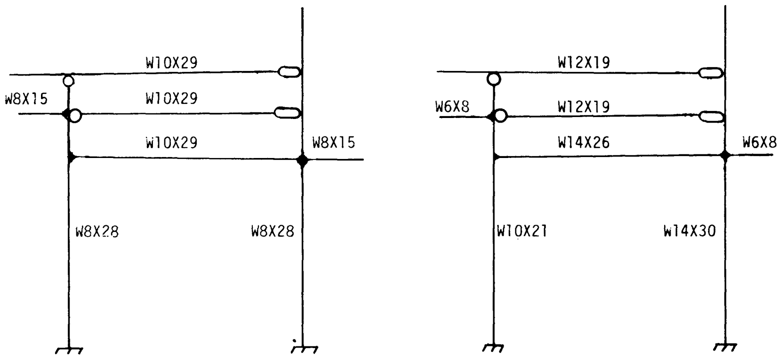
Input Data Set 'I-1'	Explanation
LOADING LIST ALL	All basic and combination loading conditions are considered.
STIFFNESS ANALYSIS	Static analysis is performed on previously selected sections.
LOADING LIST 'A' 'B' 'C' 'AA' 'DED + LIVE'	Desired loading conditions are activated.
LIST REACTIONS DISP ALL	Reactions at supports and deflections at joints for the desired loadings are printed out.
Input Data Set 'I-2'	
PRINT MEMBER PROPERTIES ALL MEMBERS	Final section properties for all members are printed out in tabular form.
STEEL TAKE	Steel weight for the final designed frame is calculated.
FINISH	Termination of program.

DISCUSSION OF RESULTS

A two-dimensional frame of structural steel, commonly called "pipe rack" in the petrochemical industry, was designed using the AISC Specification by manual computation, and is then compared with optimum sizes obtained by using the computer program STRUDL-II. The sizes are selected by the computer program initially without any constraint condition, and again using constraint conditions on member sizes and on limiting deflections. The sizes obtained by the above investigation are shown in Fig. 5b. Comparing the sizes of Fig. 5a (manually designed) with Fig. 5b (designed by the computer program), it may be noted that the former uses the same section (w8x28) for both columns, while the latter uses different sections, i.e., w10x21 and w14x30 for the left and right side columns, respectively. This occurs because: (1) a horizontal sliding hinge connection is used between the two upper-level beams and columns, and (2) the right side column projects above the top-level beam. The beams selected by the computer program are lighter but deeper compared to those selected manually. This indicates that the computer program is organized to select the most economical sections. Comparing the sizes shown in Fig. 5c (designed by the computer program with constraint condition on member sizes as given in Table 7) with the sizes shown on Fig. 5b (design by the computer with no constraints), it may be noted that, except for the beams at the two upper levels, all sizes are the same. The two upper-level beam sections w10x21, are reduced in depth (from 12 to 10 inches), but became slightly heavier (from 19 plf to 21 plf). Thus, having a size constraint condition has a mild effect on the weight, whereas manual vs. computer design has a strong effect on weight. The sizes shown in Fig. 5d are obtained by the computer program with an additional constraint condition of limiting deflection to a maximum value of 1 in. in the vertical direction at mid-span of all beams and 0.75 in. in the horizontal direction in the columns at the first-level beam. Thus, the additional design constraint results in an increase in size of the right side column compared to the previous design (w14x30 to w14x34), and indicates that the previous computer designed sizes did not, in fact, meet the limiting deflection criteria in the horizontal direction. Comparing the deflection results (caused by full dead, live, and wind loading) corresponding to Figs. 5a and 5b and which are plotted in Figs. 6 and 7, respectively, it may be noted that the horizontal deflection at the lower-level beam is 1.60 in. for the manual design and 0.72 in. in the computer design. Thus, according to the prescribed limit of 0.75 in., the manual design is unacceptable, but the computer-designed sizes adequately meet the established criteria. It should be noted that manual design seldom checks for maximum tolerable deflection, due to the fact that hand or handbook formulas or procedures do not provide a simple way of computing deflections. Concerning the vertical deflection in the beams, the maximum value is 0.64 in. for the manual design and 0.94 in. for the computer design, while the prescribed limit is 1.0 in. This indicates that the computer program selects sizes which approach the limiting requirements, while manually selected sizes are overdesigned. Similarly, this is also true in the selection process for the cantilevers. A size of w6x8 is selected by the computer program and a size of w8x15 is selected by the manual procedure. One fact worth clarifying in these figures is the difference between respective horizontal deflection values for the right and left side columns at the two upper-level beams. The reason that the right side column does not deflect as much as the left side column is because the right column is allowed to slide freely in the horizontal direction while the left side column is not free to slide horizontally.

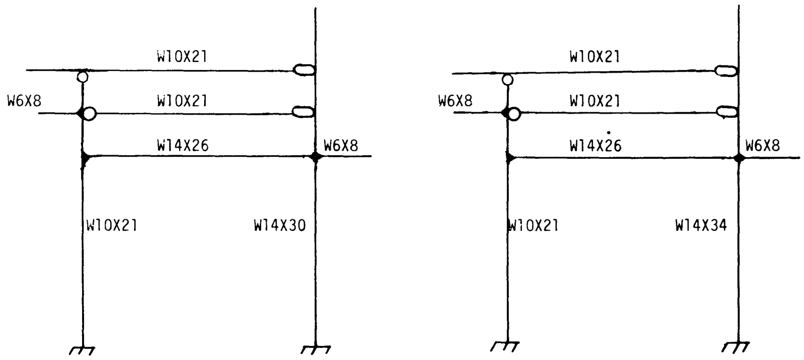
Table 10. Weight Optimization—Comparative Results

DESCRIPTION	WEIGHT OF STEEL - POUNDS			
	MANUAL DESIGN WITHOUT CONSTRAINT CONDITIONS	DESIGN BY COMPUTER PROGRAM		
		NO CONSTRAINT CONDITION	CONSTRAINT ON MEMBER SIZES	CONSTRAINT ON MEMBER SIZES AND DEFLECTION
MEMBER SIZES WITHOUT RATIONALIZATION	-	1st. CYCLE=2,454 2nd. CYCLE=2,569	2,876	3,232
MEMBER SIZES WITH RATIONALIZATION	3,853	3,112	3,215	3,343
PERCENTAGE SAVINGS OVER MANUAL DESIGN	0	19.2	16.6	13.2



(a) MANUAL DESIGN RESULTS

(b) COMPUTER PROGRAM DESIGN RESULTS



(c) COMPUTER PROGRAM DESIGN RESULTS INCLUDING MEMBER SIZE CONSTRAINT

(d) COMPUTER PROGRAM DESIGN INCLUDING MEMBER SIZE AND DEFLECTION CONSTRAINTS

Fig. 5. Design comparison

A weight comparison of the four designs in Fig. 5 is given in Table 10. The upper row of steel weights is for the case when different sizes are selected by the computer program for different segments of a single continuous member. However, the size of a segment which has the largest moment of inertia should be selected as a common size for the remaining segments to achieve a practical design and to reduce fabrication costs. The weight of the members after

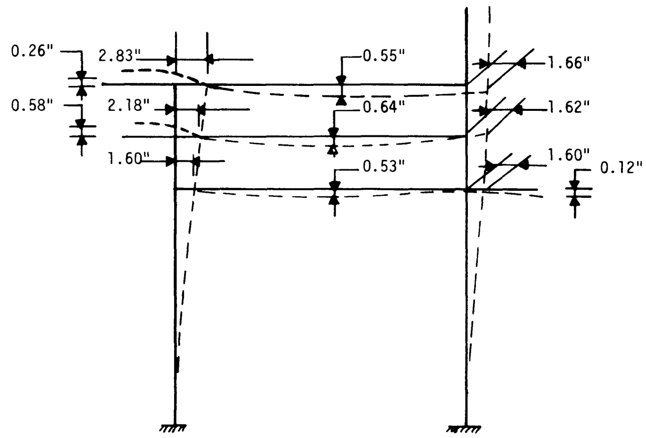


Fig. 6. Deflected shape due to full loading including wind—manual member design

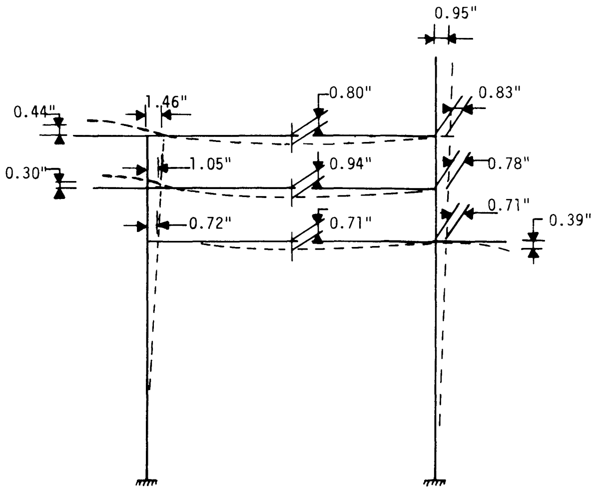


Fig. 7. Deflected shape due to full loading including wind—computer design of members including constraint conditions

this rationalization is performed is given in the second row. The percentage savings of the computer design over the manual design is shown on the bottom row. It may be noted that this percentage is reduced as constraint conditions are added to the design process. With no constraint condition, the weight savings is 19.2%, while the savings reduces to 16.6% when a member size constraint condition is added. However, with constraint conditions on member sizes *and* limiting deflection, the savings is 13.2% over manual design. Thus, as constraint conditions are added, there is a penalty increase represented by additional steel weight.

CONCLUSIONS

The following major conclusions can be made, based on the computerized investigation of the steel pipe rack structure:

1. Member sizes selected by the computer program can result in weight savings of 13–19 percent over manual design. These savings can become substantial, since the frame may be repeated many times in a particular facility.

2. Use of a computer program can save engineering time, once familiarity in data-input and mastery of the commands are achieved. The example problem described in this article was analyzed for a variety of loading conditions, member sizes were selected according to the AISC Specification, and then reanalyzed and code-checked in a single computer run. The amount of computational work performed during a computer design becomes unrealistic for manual design. Thus, using a computer program for this type of structure not only achieves efficiency, but is also cost effective.
3. Design of structural members, when accomplished by a prescribed building code, can become tedious and there is a strong possibility of overlooking important checks in a manual design. For example, deflection checks are often not performed when using manual design. It may be shown that a structure may, in fact, not meet deflection limitations, as was the case in the example problem presented here. On the other hand, in a computer design, sizes are selected by following all the provisions of the code. In addition, computer design is free of arithmetic errors provided that the input data is correct. Thus, use of a computer will result in accurate and efficient analytical design work.
4. Computer input/output is normally given in a tabulated form and can be well documented in design files. Computer output is well organized and easy to understand and can be directly used for subsequent detailing and drafting purposes.

REFERENCES

1. ICES STRUDL II Engineering User's Manual R 69-91 *Volume 1: Frame Analysis, School of Civil Engineering, Massachusetts Institute of Technology, November 1969.*
2. ICES STRUDL II Engineering User's Manual R 70-77 *Volume 2: Additional Design and Analysis Facilities, School of Civil Engineering, Massachusetts Institute of Technology, June 1971.*
3. CE STRUDL User's Manual *Engineering Science Department, Combustion Engineering, Inc., Windsor, Connecticut, July 1975.*
4. Manual of Steel Construction, *Seventh Ed., American Institute of Steel Construction, New York, 1970.*