TDS

Structural Analysis User's Manual

Tripod Data Systems, Inc.

TDS - 48

STRUCTURAL ANALYSIS

USER'S MANUAL

The structural engineering software in this package was developed by Charles I. Dinsmore P.E. , Mr. Dinsmore was resposible for the development of the HP-41 Structural Design Solutions book and was also involved in the software development of the HP-41 structures pack.

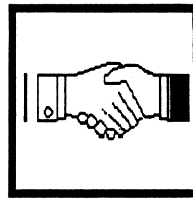
Thanks must be given to Mr. Patrick Ryan P.E. for his technical advice. Mr. Ryan and Edward Lebert P.E. took part in the quality assurance review of this product.

TABLE OF CONTENTS

CONTENTS

1

INTRODUCTION



WELCOME TO THE TDS STRUCTURAL ANALYSIS CARD

The TDS-48 is an integrated circuit (IC) Card for the HP-48SX Scientific Expandable Handheld Calculator.

With the introduction of the HP-48SX, Hewlett-Packard has provided a hand-held computer that can have the impact for the Engineering market in the 90's similar to the impact of the HP-41CX in the 80's. The HP-48SX is the "spiritual" successor of the HP-41CX in that it is designed in the vertical format; it accepts ROM and RAM plug-in cards; and, it allows for data exchange with other computing devices. However, the HP-48SX is the "logical" successor to the HP-28S, in that its programming language is a superset of the HP-28S version of object-oriented RPN. Programs written to run on the HP-28S, once keyed into the HP-48SX, will run compatibly with some minor exceptions. Programs written to run on the HP-41 will not run on the HP-48SX unless converted by using the HP 41-CV Emulator Application Card.

The HP-48SX has significantly more computational capability than previous HP scientific calculator products. Indeed, it is unlikely that any single user will require or need to learn all of the features of the machine. By using the TDS-48 Structural Analysis Card in conjunction with your HP-48SX, you will be able to take advantage of all the hardware features of the calculator; but also, you will have the programs to accomplish the routine calculations in structural engineering.

The HP-48SX Structural Analysis Card was developed with the practicing structural engineer in mind. Today, even with the abundance of desk top computers, there are times that one may not be available to solve routine problems. Most engineering offices don't provide every engineer with his own computer; so, you may have to wait your turn to complete a less complex task. Routine problems for a structural engineer include analysis of beams, frames, and trusses and the design of concrete and steel members. These problems are usually of moderate size and can be solved with a small computing device such as the HP-48SX and the Structural Analysis Card. Students in Engineering will find this product to be a very useful tool. They can solve problems and check solutions. It can be used as an aid to understanding engineering theory. The design of concrete and steel members in practice is code-intensive and must comply with both individual specifications and with the appropriate engineering theory. The concrete and steel programs contained in this card also consider code requirements that will help the student in the learning process.

SYSTEM CONFIGURATION

The minimum configuration required for use of the TDS-48 Structural Analysis Card is the following :

1. 1 HP-48SX Scientific Expandable Calculator.
2. The TDS-48 Structural Analysis Card.
3. 1 82240B Infrared or other Printer.

INSTALLING YOUR STRUCTURAL ANALYSIS CARD

Installation of your Structural Applications Card and the associated RAM cards is simple and straight forward. However, be certain to follow these installation instructions exactly as they are presented. It is assumed that you have read the Users Manual for the calculator and have some knowledge of how it works. For instructions not explained here, refer to the HP-48SX Users Manual.

Be certain your HP-48SX contains three AAA alkaline battery cells and is turned off.

1. Turn your HP-48SX OFF **[->] [ON]**

2. If any other cards are plugged into the HP-48SX and the RAM is merged into the system, be sure to free memory in the port you will use for the Structural Analysis Card prior to this procedure.

3. Insert the Structural Analysis Card in the selected Port.

4. Insert a RAM card in an open port for added memory (optional).

5. Turn the HP-48SX **[ON]**.

6. Merge the installed RAM card into the system.

7. To run the Structural Analysis Card, press the ALPHA key twice **[α] [α]**; type the word **STRUCT**; and, press **[ENTER]**.

INTRODUCTION

The following MAIN MENU Screen for the Structural Analysis Card will appear :

```
  <Select [G] to [K]>
[G] General beam
[H] Plane Frame
[I] Steel solver
[J] Concrete column
[K] Delete data dir.



|    |    |    |    |    |EXIT|
```

Once the software is activated, the Card will be in control of the system until you intentionally return to the main operating system of the HP-48SX. Turning the calculator off and then on will retain the present screen and status of all data when the calculator was turned off.

EXITING THE CARD

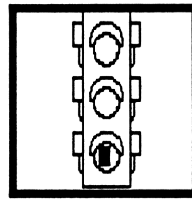
If you wish to exit the Card and return to the main operating system of the 48SX, press **[EXIT]**. Some data variables and some global files will remain in the HOME directory. You can purge these if you desire. They are automatically loaded when the Structural Analysis Card is initialized. To do this, press the **[<-]** key and then the **[{ }]** key. The **{}**-brackets will appear in the command line. Then press the variables in the directory you wish to purge; press **[ENTER]** to place the list on the stack; Press **[<-] [PURGE]** .

HOW TO LEARN THE SYSTEM

The best way to learn the Structural Analysis Card is to just sit down and use it. You will find the user interface to be very intuitive and easy to master. This is because TDS programs utilize a "MENU-and-SCREEN" user interface system. The screens make use of the HP-48SX's softkeys. The six keys across the top of its keyboard are defined separately for each program. The definitions appear at the bottom row of the display screen. After learning the rules of the road, just press some keys and begin.

2

GETTING STARTED

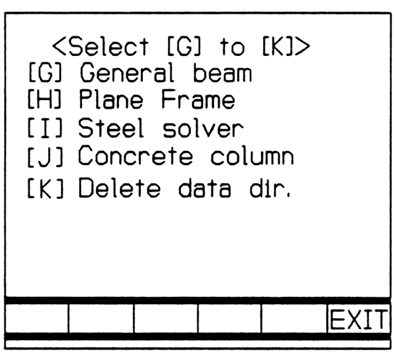


BEFORE YOU START

Before you start, you should be certain that you have installed the RAM card and the Structural Analysis Card correctly. See Chapter One : Introduction.

THE RULES OF THE ROAD

Press the **[ON]** key to turn the calculator ON. You will see the basic system operation screen; or else, the contents left in the calculator at its last use if not cleared. Press **[α] [α] STRUCT [ENTER]**. The Structural Analysis Card has now taken control of your HP-48SX. You will see the Structural Analysis Main Menu screen shown below.



At this point, you may select the application of your choice: General Beam, Plane Frame, Steel Solver, or Concrete Column. You may press **[EXIT]** at any time to return to the previous screen or to the HP-48SX operating system. The six boxes at the bottom of the display screen are called "soft" key labels. They identify the functions of the six keys on the top row of the keyboard. Pressing any one of these keys will activate the function shown in the box above that key. The soft-key functions will change, depending on the particular screen that is being used and the problem that is being solved.

MENUS VS. SCREENS (What's the difference?)

The TDS Structural Analysis Card makes the HP-48SX an intuitive use machine. Much progress can be made in mastering the system by pressing keys and seeing what happens in response. However, full understanding of the machine requires that a few simple concepts be well understood. One is the difference between a **MENU** and a **SCREEN**. A menu is a display that is characterized by a list of functions or operations which may be selected by choosing one of the alphabetic keys listed down the left hand column of the display. Menus do not use the active soft keys. The **[EXIT]** key is always displayed above the **[F]** key on the right of the top row of keys. Pressing one of the alpha keys shown in the **MENU** display will present another MENU (with more alpha-choices and an **[EXIT]** key) or a SCREEN.

The **[EXIT]** key will always return to the SCREEN or MENU location occupied prior to the current SCREEN or MENU. Thus, MENUS are arranged in tree format. By selecting a sequence of alpha keys, you progress through the application.

NOTE: *In the HP-48SX the top row of keys are used for the alpha keys* **A-F**, *as well as the soft keys. For this reason, all menu labels begin with the letter* **[G]**. *Since there is no ambiguity in MENUS between menu selection keys and softkeys, it is not necessary to press the* **[α]** *key prior to making a menu screen selection.*

GLOBAL TOP-ROW KEYBOARD FUNCTIONS

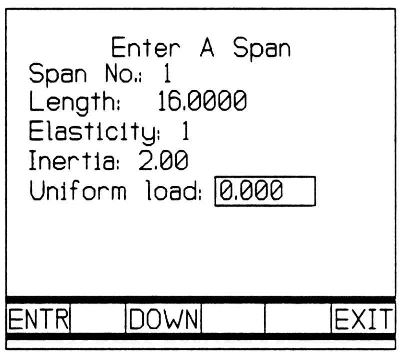
In addition to the six softkeys whose functions change depending on the screen that is active, there are three Global Keys that you access with the gold shift key and one of three other keys in the top row. They are the keystrokes **[<-] [D] [<-] [A]** and **[<-] [F]**. These keys will execute a **PRINT SCREEN** function; a function that will return to the **HP-48SX SYSTEM** with the data field value placed on the stack; and an **EXIT FUNCTION**. If the **[<-] [A]** key were pressed and the system mode is activated, you can return to the program mode using the **[<-] [ON]** key. The value on the stack will be returned to the data field location in the Structural Analysis program accessed prior to moving to the HP-48 system. The **EXIT** function will return to the Main Menu screen at any time.

SCREENS

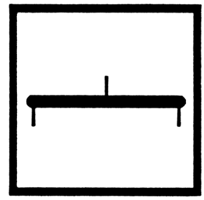
From the MAIN MENU, the screen for any application is accessed by pressing the appropriate alpha key for that application. Recall that the applications are defined by the alpha keys **G-J.** Each screen that contains the MENU functions also contains the softkeys that activate either data entry or execution of a solution. The screen shown below is the data entry screen for the geometry of a beam in the

General Beam application program.

Data entry lines are accessed by using the vertical cursor keys to move up or down through the screen. Use the right cursor key to toggle an input option or a setup requirement. As the cursor key is pressed, the appropriate piece of data required for entry appears in reverse video. When the input values are to your satisfaction, press the appropriate function key displayed at the bottom of the screen.



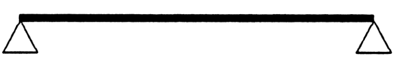
3



GENERAL BEAM ANALYSIS

GENERAL DESCRIPTION:

The General Beam Analysis program will do an elastic analysis of single-span or multiple-span beams. During calculation, a list is created to access the data for each individual span. This allows the program to store the calculated end moments of a continuous beam solution and perform a single span analysis. The individual span analysis may be at a single point or at an user-specified increment. The output will be displayed or printed. An individual span analysis will produce the results of shear, moment, rotation, and displacement.



SIMPLE BEAM

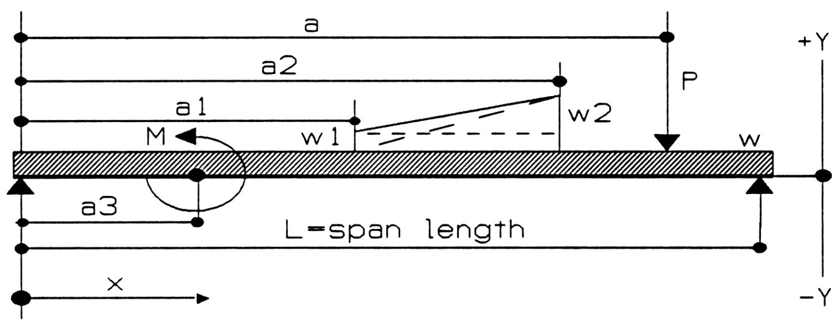
FIXED END BEAM

PROPPED CANTILEVER
fixed left or right

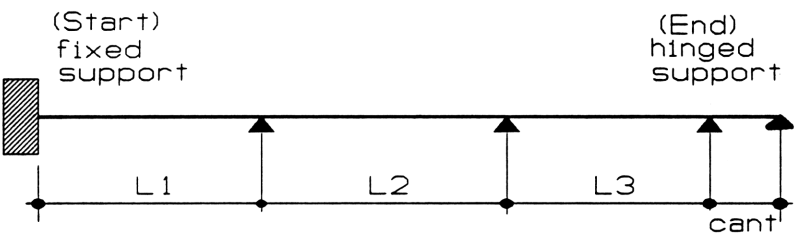
CANTILEVER
always fixed right

Beam loading may include any combination of uniformly distributed load, concentrated point loads, partial distributed loads or applied moments. Trapezoidal loads may be specified as partial distributed loads, varying linear loads, or triangular loads located anywhere on the span.

The following beam diagram shows span geometry and applicable load conditions.



GEOMETRY AND LOADING DIAGRAM



CONTINUOUS BEAM

Using the display screens, geometry and loading for each span are entered. The sign convention used for bending moment is clockwise positive. External and internal signs are indicated in the following table.

BEAM SIGN CONVENTION			
ACTION	VARIABLE		SIGN
Deflection	D	↑	+
Rotation	R	↺	+
Int moment	M	↻ ↺	+
Shear	V	↑ ↓	+
Ext load	P or W	↓	+
Ext moment	M	↻	+

DESCRIPTION OF INPUT:

The following list describes the data required for the General Beam Analysis solution.

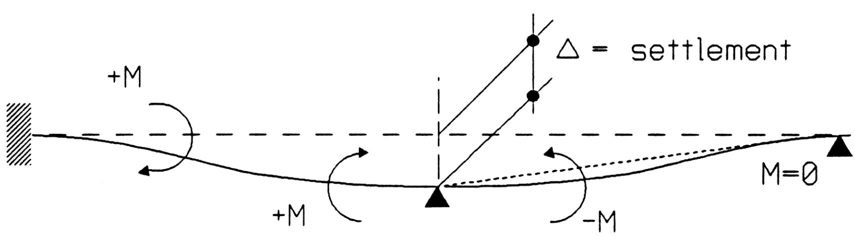
1. Number of spans.
2. Span length, modulus of elasticity, and moment of inertia.
3. Uniform load, w. Enter zero if none.
4. Number of concentrated loads, if any.
5. Location, 'a', and value, 'P', for each concentrated load.
6. Number of distributed loads, if any.
7. Beginning location, 'a1', and beginning value, 'w1'. End

location, 'a2', and end value, 'w2'.

8. Number of applied moments, if any.
9. Location, 'a3', and value, 'M', for each applied moment.
10. Beginning and end support conditions.

Two special conditions may be represented in a multiple span beam system: cantilever spans and support settlements.

For a cantilever span, the cantilever moment must be included as an applied moment at a support. A right-hand cantilever with normal positive loads is a positive moment; a left-hand cantilever with normal positive load is a negative moment; and, each support is considered hinged. Similarly for support settlement, the moments generated by the settlements must be entered as applied moments at the supports. The following diagram and equation explains how the moments are generated.



SUPPORT SETTLEMENT

Where $M=6*E*I*D/L^2$ is for a fixed condition; and $M=3*E*I*D/L^2$ is for an hinged condition.

If more than one settlement occurs, the relative difference must be considered. After data has been entered, the user can edit all data for each span. Select the [H] menu key from the General Beam Analysis Menu Screen to activate the Edit mode. When the user is satisfied that all data is correct, the solution for the problem may be executed.

DESCRIPTION OF OUTPUT:

1. End moments for each span in a continuous beam solution.
2. Individual span analysis, with the following results for each span as selected :
   a. Shear at selected points or incremented points.
   b. Moment at selected points or incremented points.
   c. Rotation at selected points or incremented points.
   d. Deflection at selected points or incremented points.

The user has the option to edit data for a rerun or to begin a new problem by selecting the proper menu keys.

METHOD OF COMPUTATION:

A continuous beam solution uses the Flexibility Method for solving an indeterminate structure. In the Flexibility method, a number of static redundancies must be removed from the indeterminate structure producing a statically determinate structure. The basic procedure is described as follows.

1. The geometry, properties of each member, and loadings on each span are entered.

2. The releases on the structure are determined.

3. Using the simple beam equations, the end rotations for the applied loadings on each span are computed.

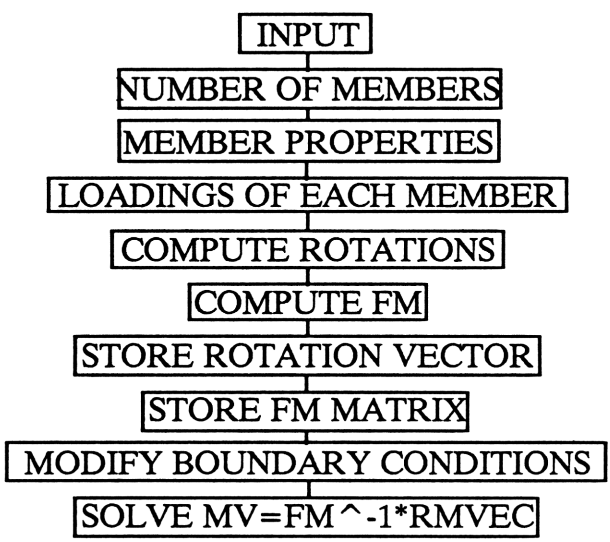
4. The ‘fm’ matrix is determined for each member.

5. The rotations are assembled to form a rotation vector.

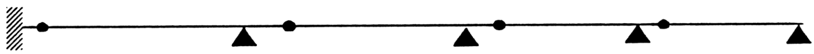
The "fm" matrices are assembled to form an "FM" matrix.

6. The rotation vector and ‘FM’ matrix are modified for the boundary conditions.

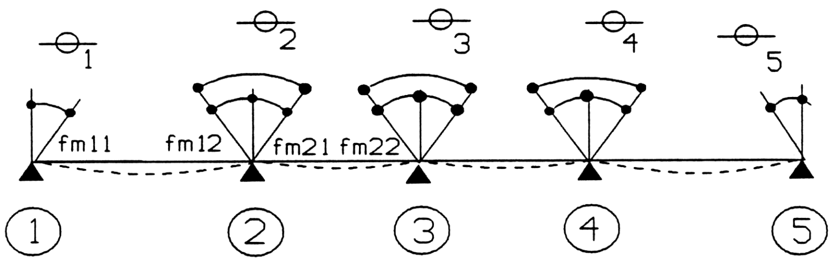
7. The compatibility equation RMVEC+FM*MV=0 is solved where $MV=FM^{-1}*RMVEC$. The general flow diagram of the procedure described above is :



Boundary conditions are considered after all matrices are generated. The diagram shows an assumed release structure.



HINGED RELEASES



DISPLACEMENT OF RELEASED STRUCTURE

SUPERPOSITION OF fm MATRICES TO FORM FM MATRIX

fm^{1}_{11}	fm^{1}_{12}			
fm^{1}_{21}	fm^{1}_{22} fm^{2}_{22}	fm^{2}_{23}		
	fm^{2}_{32}	fm^{2}_{33} fm^{3}_{33}	fm^{3}_{34}	
		fm^{3}_{43}	fm^{3}_{44} fm^{4}_{44}	fm^{4}_{45}
			fm^{4}_{54}	fm^{4}_{55}

For single span analysis, the general equations were derived for programming using the method of singularity functions. They provide an efficient algorithm for computation of only what is required at a single point. To derive the final equations, the load system must be integrated to get shear, moment, slope, and deflection expressions. A brief description follows.

The elastic behavior of a beam may be defined by the four basic relationships of mechanics. Beams may be single-span, overhanging, or continuous. A single-span beam is easily solved by statics and is, therefore, called "statically determinate". Continuous beams, on the other hand, are statically indeterminate because there are more reaction components than the total number of independent equations of statics available. The excess is defined as the degree of indeterminacy. When the continuous beam has been solved, the unknowns are identified. Then, the beam can be considered statically determinate. At this point, any span can be isolated as a free body; and, the values of shear, moment, rotation, and deflection may be computed for that span.

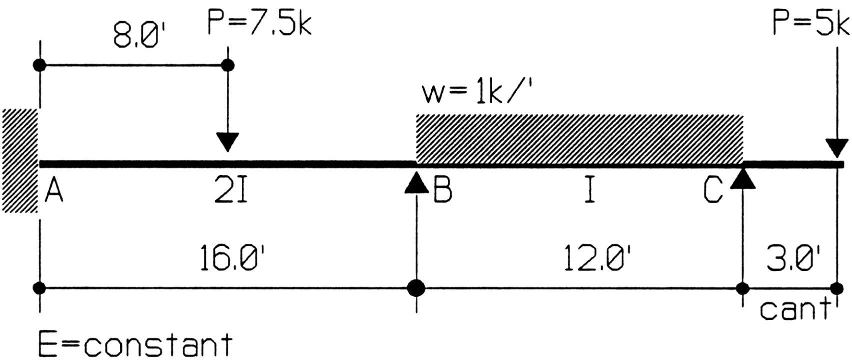
BASIC ELASTIC RELATIONSHIPS

LOAD : $w=d^4y/dx^4$ SHEAR : $V=EId^3y/dx^3$

MOMENT : $M=EId^2y/dx^2$ ROTATION : $R=EIdy/dx$

DEFLECTION : $D=EIy$

The following sketch shows a typical beam problem that can be analyzed using this program. The following instructions are of a general nature describing the procedure of data entry and the resulting screen displays on the HP-48SX. Trying out this problem and using the display screens as described will get you started. This will also be the procedure for the other examples.



CONTINUOUS BEAM WITH CANTILEVER
EXAMPLE PROBLEM

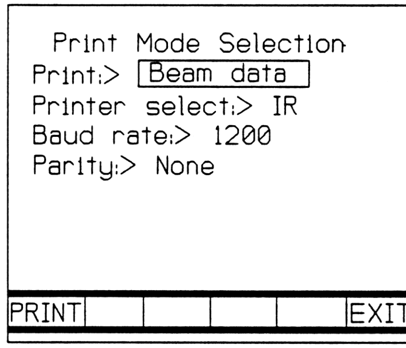
It is assumed that you are at the General Beam program menu.

If you will be using the printer, select the print mode first by pressing the **[K]** key. Use the vertical cursor key to move up or down between data-entry lines. When a data-entry type is followed

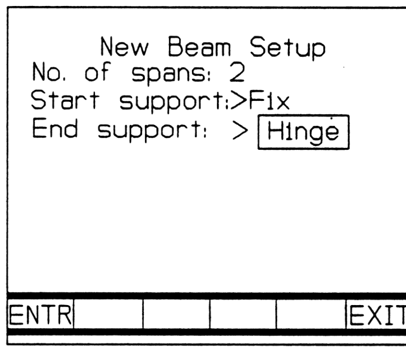
```
        General beam
[G] New beam setup
[H] Edit beam data
[I] Cont beam analysis
[J] Span analysis
[K] Select print mode

                          EXIT
```

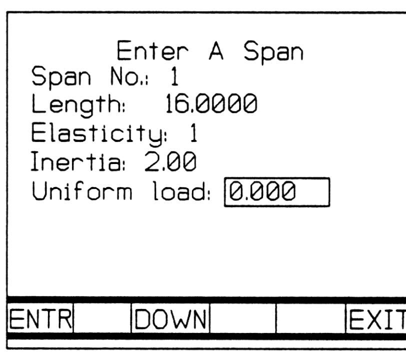
by >, you have several options for that line. Use the horizontal cursor to select the appropriate entry. Line 1 of the Print Mode Screen indicates the type of data to be printed. In line 2, select the type of printer to be used; in lines 3 and 4, select the appropriate baud rate and parity when using a serial printer. You will return to this screen to print out your data. Press **[EXIT]** to return to the General Beam Menu. Press the **[G]** key to display the New Beam Setup screen.



Using the vertical cursor keys, enter the number of spans and the support conditions. The current support conditions are chosen by using the right cursor key. When the data is correct, press **[ENTR]**. The display shows the Span Data Screen.



Span data is also entered by using the cursor keys to locate the input line. When the data is correct, press **[ENTR]**.



NOTE: You use the **[DOWN]** key only during the **edit** mode. The display shows the Span Load Screen.

```
     Enter Span Loads
Span No.: 1
No. of conc. loads: 1
No. of dist. loads: 0
No. of conc. moms: [0]

ENTR                        EXIT
```

The next data to be entered defines the beam loadings. You must tell the program how many of what type loads you have in your problem, using **[ENTR]**.

Data for each of the three types of loadings will be needed if they exist; that is, concentrated loads; distributed and linear loads; and, concentrated moments. This procedure will be repeated for each span.

```
  Cont. Beam Analysis
Span No.: 1
Mom left: -16.500
Mom right: 12.000

SOLVE PRINT DOWN UP         EXIT
```

If you are analyzing a *single span with hinges on each end* or if you are analyzing a *cantilever span*, you may go directly to the span analysis using the **[J]** key. If you are analyzing a **continuous beam** (or a single span beam, such as a propped cantilever or a beam fixed at both ends), you must do a Cont beam analysis by pressing the **[I]** key. When the

```
      Span Analysis
Span No.: 1
>X   :0.000
Shear:           4.031
Moment:          -16.500
Rotation:        2.000E-10
Deflection:      0.000

SOLVE PRINT                 EXIT
```

resulting Cont beam analysis screen (NOTE: the first display shows no results until **[SOLVE]** is pressed) is displayed press **[SOLVE]** to see the resulting moments at the supports of each span. For each additional span, press the **[DOWN]** key.

To do analysis on each span, press the **[J]** key. Any span may be selected for analysis. Results may be obtained for any point on the beam or an increment may be specified with a printer. You may alternate as desired from one span to the other. The data may be edited for any span and reanalyzed. If indeterminate problem data are edited, the problem must be reanalyzed. Use the **[I]** key from the General Beam Analysis Screen.

To go to another program in this card, **it is suggested that you use the [K]** key at the Main Menu to **delete the data from the present directory.**

REFERENCES:

1. Roark, Raymond J.;Young, Warren C.; "Formulas For Stress and Strain" ; McGraw Hill Book Co.;1975.

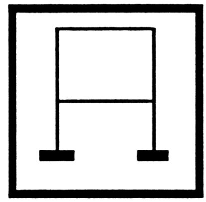
2. Nash, William A.; Schaums Outline Series “Strength of Materials” ; Method of Singularity Functions; 1972.

3. Hsieh, Yuan Yu; "Elementary Theory of Structures."

4. M. D. Vanderbilt; "Matrix Structural Analysis" ; Quantum Press; 1974.

4

PLANE FRAME ANALYSIS



GENERAL DESCRIPTION

The purpose of this program is to solve rigid frames, trusses and indeterminate problems commonly encountered in everyday practice. It is not intended to replace frame analysis programs on desk top computers that solve large problems in a relatively short time. The program can, however, solve small to moderately-sized problems when no PC is available. Each problem should be considered independently as to the most efficient way to obtain a solution.

The plane frame program uses the stiffness method of analysis to provide a solution for most types of structural plane frames and trusses. Compared to larger commercial frame analysis packages, this program is structured for smaller capacity machines. This means the user should be significantly concerned about structural stability and completely understand the model being used for analysis. Complete error checking is not provided by the program. Special care is needed in preparing the input data. The advantage of this type of program structure is the capability of solving larger types of problems on a small computing device. It is important that good engineering judgment is used during model development. It is recommended that the user have a background in or is a student studying the mechanics of engineering.

PROGRAM REQUIREMENTS:

In addition to the requirements stated in the introductory chapter, *it is necessary to have a printer while using this program.* Due to the volume of output, the user cannot be expected to write down the solution manually.

LIMITATIONS:

The program is subject to the following limitations:

1. Member section properties must be entered according to their actual orientation.

2. Spring constants may not be specified at supports, but can be modeled using substitute members.

3. Variable section beams may be analyzed only when the whole member is broken up into several members, and the average areas and moments of inertia of the segmented sections are used.

4. Output format is established by the program and is not selective.

5. Units are established using the American System of kips (kilopounds) and feet for length units. The section properties are in inch units.

PROCEDURE:

Each time the frame program is to be run, the following procedure should followed.

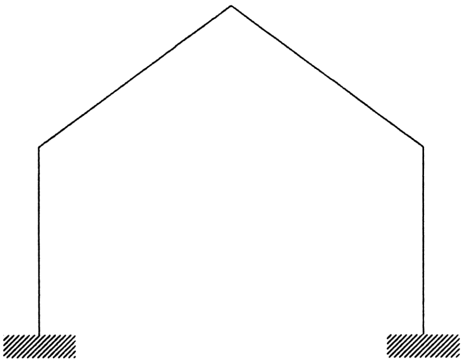
A sketch of the frame model should be made; number each joint; and, compute the coordinates related to an horizontal 'X' and a vertical 'Y' axis. Member input is by member number and member incidence. The joint specifications and coordinates correspond to the structure global axes. Each member should be given a number, and its section properties should be established. Support conditions should be identified as to whether they are hinged, fixed, or hinged with a roller (allowing either vertical or horizontal translation).

Determine loads and load cases. Only the **LOAD** and **ANALYSIS** routines will be required for each solution after the structure stiffness matrix has been assembled and factored. Additional sketches may be required to keep track of the individual loading conditions. Each joint load should be located to correspond to its joint number and global direction. Joint loads not oriented in the global direction will have to be entered according to their global force components. Joint loads normal to a member may be entered as a concentrated member load at the member end framing into the required joint.

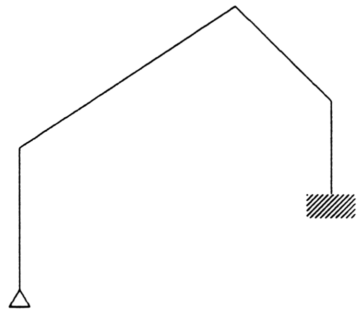
Member loads may be entered as concentrated loads or distributed loads. The concentrated loads may be specified in their local or global direction. See the sketch for description of local and global loading. Distributed loads may be entered as local, global, or global-projected.

DATA PREPARATION:

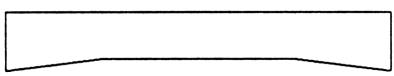
The following sketches show some of the typical frame problems that can be solved with this program.



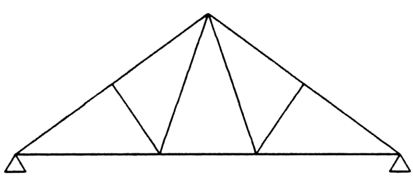
GABLE FRAME



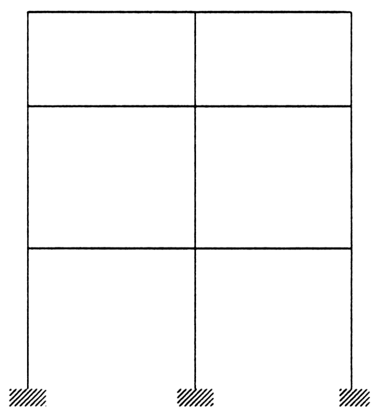
SKEWED FRAME



VARIABLE BEAM



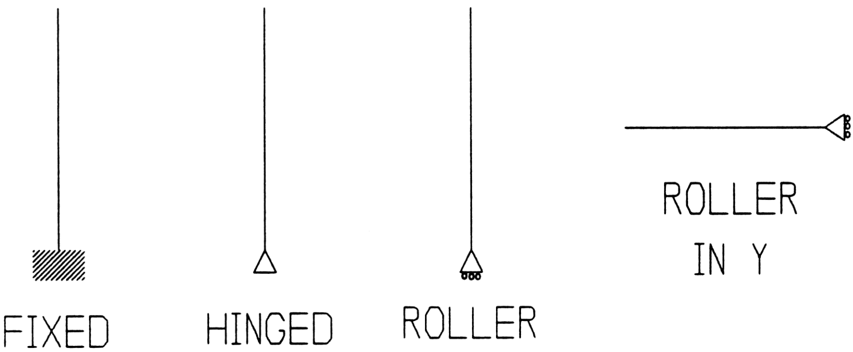
TRUSS



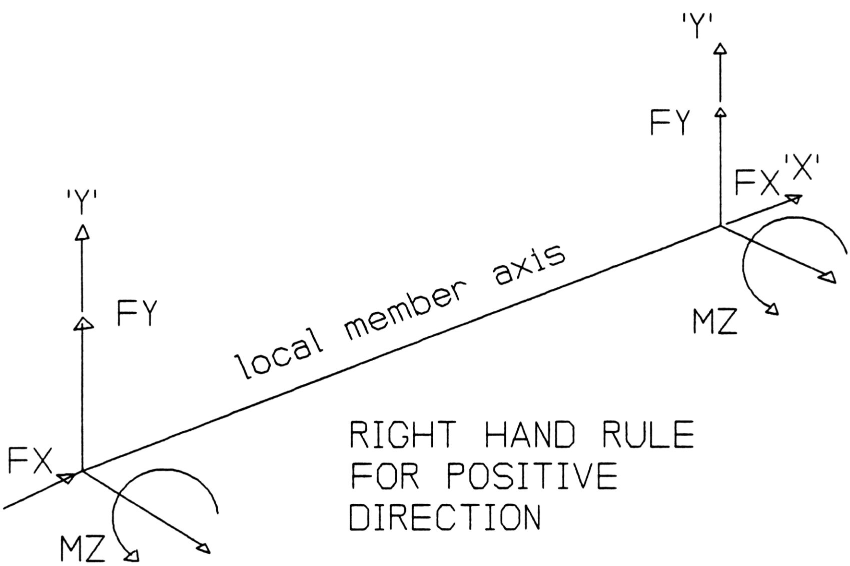
MULTISTORY FRAME

The following sketches show the various supports that may be used in this program. The indices required to indicate joint restraint are shown.

PLANE FRAME

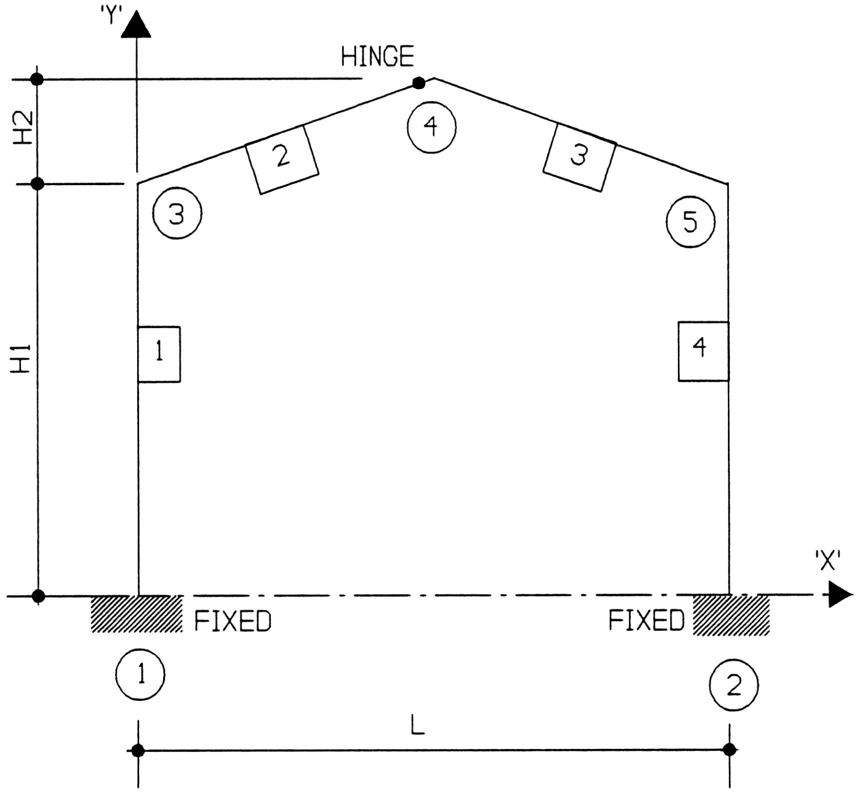


The following sketch shows the sign convention used in the program. The sign convention is the right-hand rule used in most commercial structural analysis programs.



The following is a set of steps suggested for data preparation. It considers only the simplest type of problem.

The process involves the description of the problem data and the operations to be accomplished on this data. The description must include the following:



TYPICAL FRAME MODEL

1. The locations of the joints of the structure (joint coordinates) must be supplied. You must also specify which of them are supported and the manner of their support.

2. Specification of the joints to which each member is

connected (member incidence) must be included.

3. Specification of the types of connections of the members (member releases) are needed.

4. The loads which are applied to the structure (member loads and joint loads) must be described.

5. Member properties and elastic constant must be given, as well as the density of the material being used.

6. A stiffness analysis must be performed.

7. Displacements, member forces and reactions are computed, and the results are displayed.

The first data entered are the joint numbers and the joint coordinates corresponding to the global structure axis. Next, you must identify the structure supports at specified joints. Then, enter member information which describes member number and incidence. *Member incidence* is defined as the member start joint number 'A' and the member end joint 'B'. The following section properties are then entered: AREA (in^2); MOMENT OF INERTIA (in^4); MODULUS OF ELASTICITY (p.s.i. expressed in 10^6 units); and DENSITY (p.c.f.). At this point, the program will allow member releases to be specified. Member release may be hinged left, hinged right, or hinged both ends. The hinge indicates moment release only. Force release for axial load or shear is not accommodated by the program. Member release is used when discontinuity is desired in the structure. Care must be taken when specifying member release at the intersection of several members. Releasing all member ends at a single joint may cause instability in the structure and will probably cause a singularity in the structure matrix. It will also display a "not positive definite error" in the execution of the program. Using the member release feature, truss structures can be modeled. To model an intermediate hinge within a member, divide the member into two members and use a member release at the end required.

At this point, the program begins to assemble the structure stiffness matrix using a condensation routine to minimize the amount of memory used. After assembly and condensation, the stiffness matrix is factored to prepare for the solution portion of the analysis. When this procedure is complete, loadings may be entered.

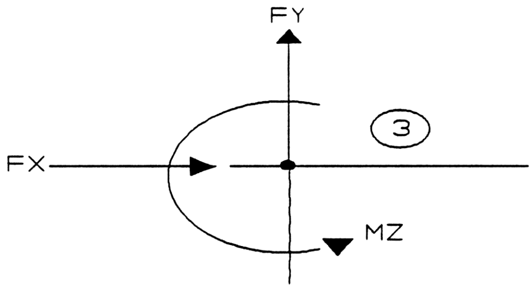
LOADINGS:

The first loadings entered are the joint loads. Joint loads have a direct correspondence to global references. The sign convention follows the right-hand rule explicitly and is shown in the sketch below. In it :

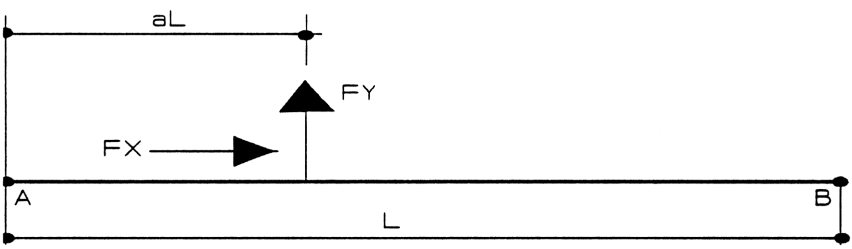
Fx is the force in the global 'X' direction; Fy is the force in the global 'Y' direction; and, Mz is the applied joint moment. For joint loads not corresponding to the global system, calculate the global components of force and apply them in the appropriate direction.

Member loads consist of concentrated loads; distributed loads; partial distributed loads; trapezoidal loads; linear loads; or, combinations of these. Other types of loads may be specified by computing the fixed end actions and applying them as joint loads. Fixed end actions entered as joint loads should be joint loads of opposite sign. All fixed end loads entered in this manner should be added algebraically to all joint loads at the joint. All actions must be considered; i.e., axial load, shear, and moment. The following sketch shows a typical member, all possible loads, and how these loads are specified for input to the program. All directions for loadings on the sketch are shown in the positive direction according to the right-hand rule.

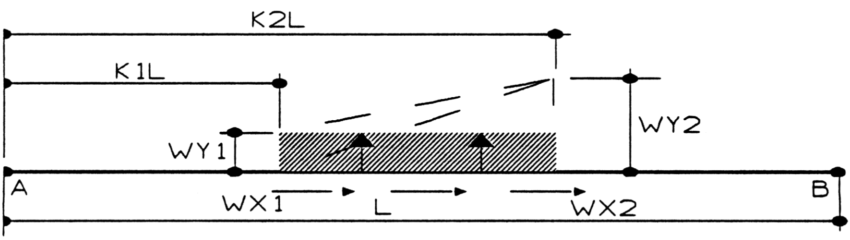
Member loads are established in various ways. Generally, the building codes identify live loads, snow loads, and wind



TYPICAL JOINT LOAD



CONCENTRATED LOADS



MEMBER DISTRIBUTED LOADING

loads on the horizontal and vertical projections. This will usually occur on sloped members. Floor loads are usually identified as transverse loads which are oriented normal to the member. Dead loads on sloped members are always specified as gravity loads acting vertically on the member slope. The program allows entry of concentrated loads and distributed loads in local, global or projected-global orientation. The following sketches will clarify how to specify these different loading conditions.

DESCRIPTION OF OUTPUT:

The output for the frame analysis program consists of the following:

Joint coordinates: feet and decimals of a foot.

Member area: square inches.

Moment of inertia: inches to the fourth power.

Elasticity: p.s.i. in 10 to the sixth power (29000000 p.s.i. is entered as 29.0).

Density: pounds per cubic foot.

Joint loads and forces: kips.

Moments: foot-kips.

Member loads, concentrated loads: in kips; distributed loads in kips per foot.

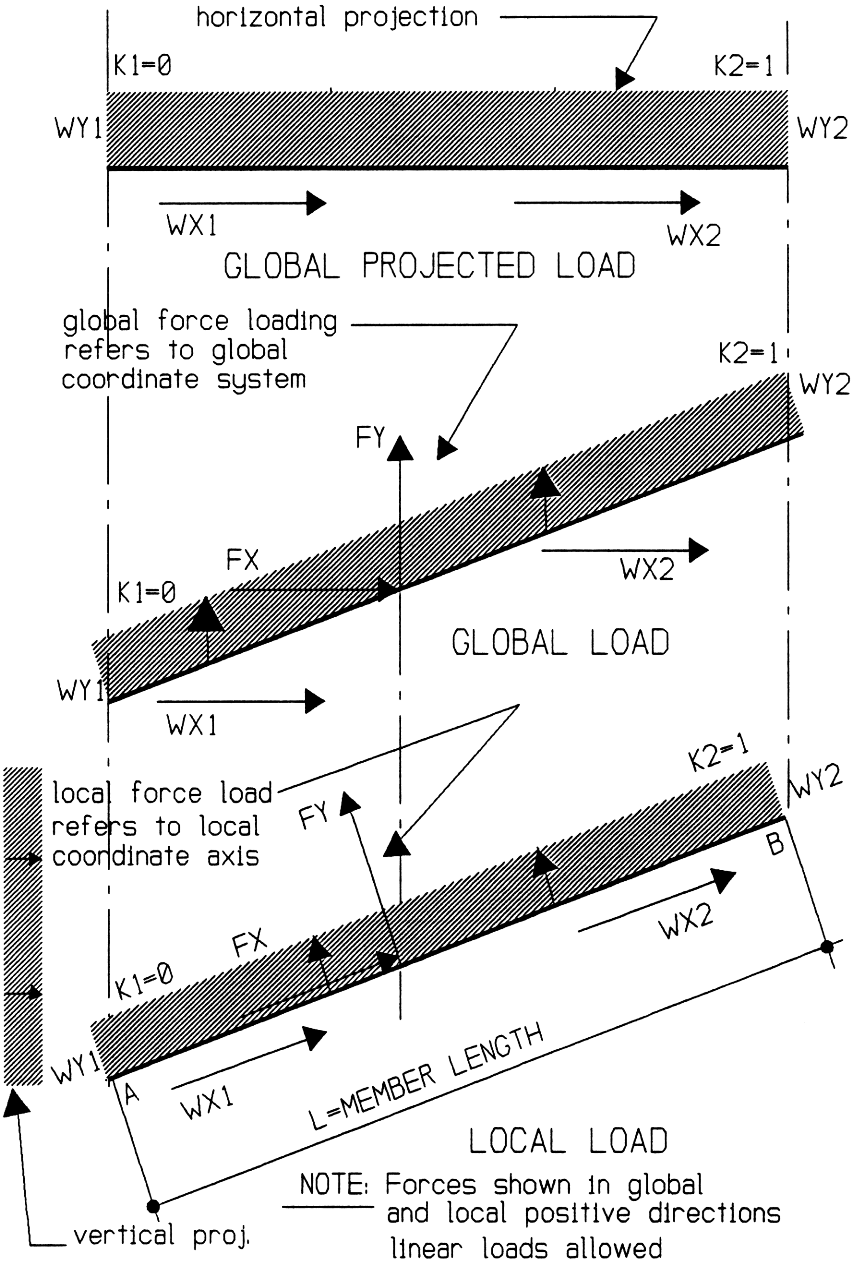
Joint displacements: inches.

Rotations: in radians.

Member forces: kips; moments: in foot-kips.

Reaction forces: in kips; moments : in foot-kips.

PLANE FRAME



horizontal projection
K1=0
K2=1
WY1
WY2
WX1
WX2
GLOBAL PROJECTED LOAD
global force loading
refers to global
coordinate system
K2=1
WY2
FY
FX
K1=0
WX2
GLOBAL LOAD
WY1
WX1
K2=1
WY2
local force load
refers to local
coordinate axis
FY
B
WX2
FX
K1=0
WY1
A
WX1
L=MEMBER LENGTH
LOCAL LOAD
NOTE: Forces shown in global
and local positive directions
linear loads allowed
vertical proj.

GENERAL AND MATHEMATICAL DESCRIPTION:

The stiffness method selects nodal displacements as unknowns. In a plane frame, each node has three degrees of freedom: an horizontal translation in the 'X' direction; a vertical displacement in the 'Y' direction; and, a rotation. The number of equations will be three times the number of nodes. At the supports, one unknown will be deleted for each translation or rotation that is restrained from movement. The general steps to a solution are as follows:

1. The stiffness matrix for each member is determined.

2. If any member releases are specified, the local member stiffness matrix is modified.

3. The member stiffness matrices are assembled into the global structure stiffness matrix.

4. The applied member loads are processed.

5. The load vector is assembled. The global stiffness matrix and the load vector are condensed to account for joint releases.

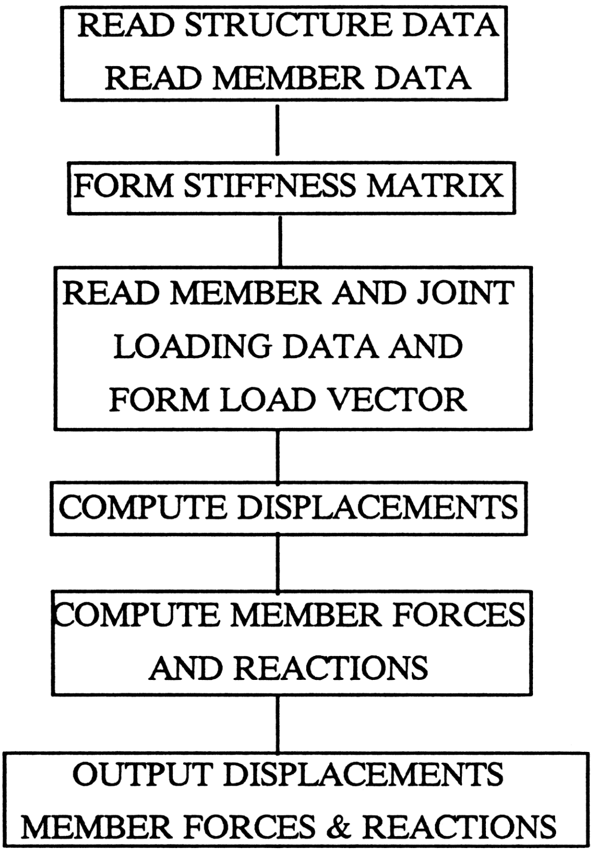
6. The governing joint equilibrium equations are solved for joint displacements.

7. The member distortions and member forces are computed.

8. Support reactions are computed.

The following items of information are necessary for the operation of the program:

FLOW CHART



When using the member release function, care must be taken not to release all members framing into a joint. If all members are released, it is possible to have joint instability. This will be indicated by an error created by the condition of a singular matrix or by an internal error message related to positive definiteness of the structure matrix.

Once the structure matrix has been assembled and factored, many load cases may be run. Press the load menu key for the new load case. When data entry is complete, perform the analysis.

If an error is made while entering data, cursor to the spot and enter the correct data. If all data has been entered and then an error is detected, use the edit routines on the Plane Frame Menu screen to replace the erroneous data.

It is suggested that the sample problem should be run to get a feel of the program.

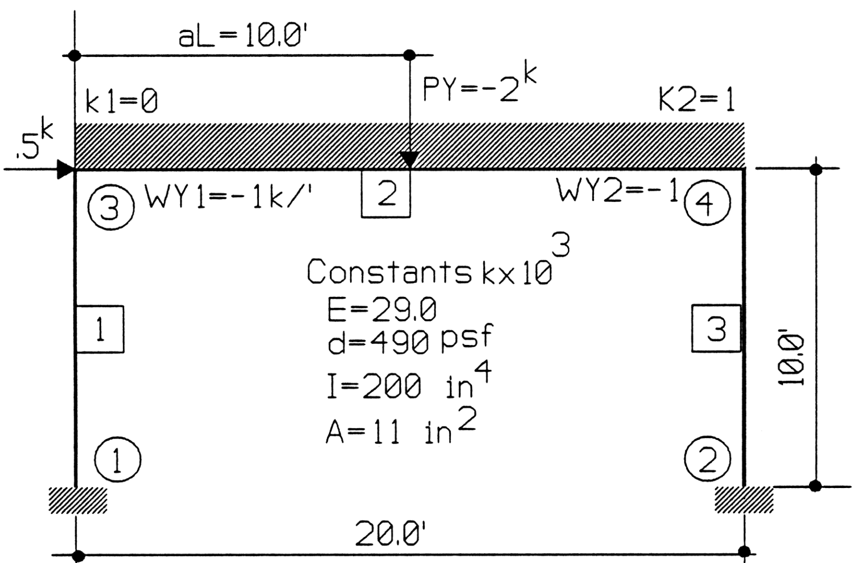
If a matrix that is not positive definite is detected, an error is reported; the run is terminated.

The efficiency of the program is dependent on the joint numbering sequence. The program calculates the matrix band-width. The smallest band-width is the most efficient. The band-width is calculated by taking the numerical difference between the start joint number and the end joint number; add 1; multiply by 3; subtract the number of support restraints, if they exist, at the member end. The equation, 3*((Ji-Jk)+1)-Sr (for each member), may be quickly calculated in each preliminary model to find the most efficient band-width. Member numbers do not necessarily have to correspond directly to the joint numbers.

Execution time increases by a power of two. Twenty or more members may take some time to assemble and factor the structure matrix. Member releases will reduce the time. The program uses a well-known condensation routine to reduce the memory required to perform the solution. Once the structure matrix has been assembled and factored, the remaining solution process is directed to the loadings and the solving of the displacements. Once the matrix has been assembled and factored, more than one load case can be solved.

The following sketch shows a typical frame problem that can be analyzed using this program.

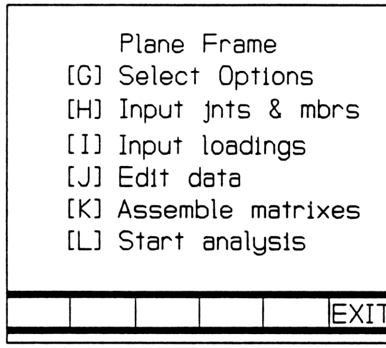
The following general instructions describe the procedure of data entry and the resulting screen displays on the HP-48SX.

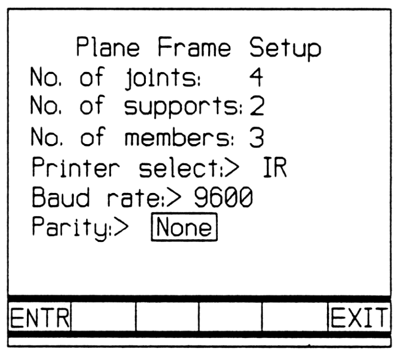


EXAMPLE FRAME MODEL

Trying out this problem and using the display screens together with the explanations should get you started.

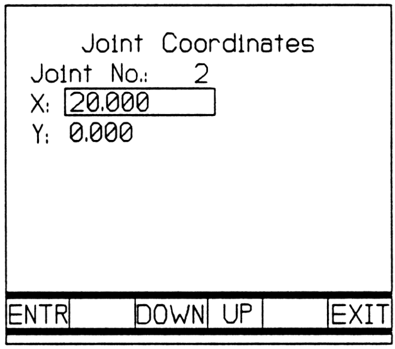
It is assumed that you are at the Plane Frame program menu. Press the **[G]** key: Select Options: to access the Plane Frame Setup screen. You will be using a printer; so, select the printer type. You can select an IR or serial printer. If you are using a serial printer, you will have to specify the baud rate and parity. This screen also specifies the No. of joints; No. of supports; and the No. of members required for the problem. Using the vertical cursor keys, enter the number of joints, the number of supports, and the number of members. When the data is correct, press the **[ENTR]** menu key. The general sequence in this program will be to enter the required data as requested in each screen. When one screen is complete, press **[ENTR]**. You will be returned to the Plane Frame Menu in order to access additional screens. The screens are selected by pressing the appropriate alpha-key.

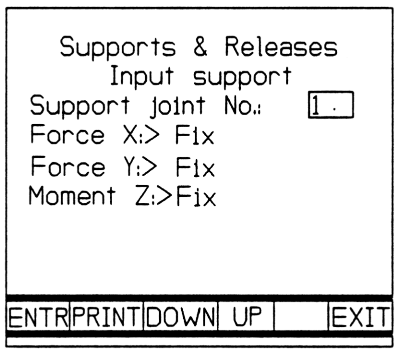


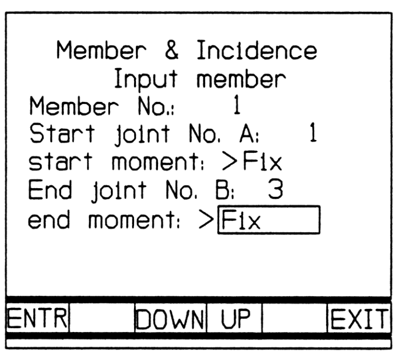


Now, you will need to enter appropriate data for joints and members. Begin with coordinates for all joints. From the Plane Frame Menu, press **[H]**: Input Joints & Members. In the Joint Coordinates Screen, enter data for each joint; press **[ENTR]** when each joint coordinate screen is complete. When data for the total number of joints indicated in the

Plane Frame Setup has been entered, the system automatically displays the Support & Releases Screen. The data required for this section will be the joint number for the support and it's support condition; that is, whether it is fixed or released for the 'X' direction, 'Y' direction or the rotation. These will be selected by using the right arrow cursor key as a toggle. Data must be entered for each support according to the number of supports indicated in the Plane Frame Setup. After you have entered data for the last support and have pressed **[ENTR]**, the next screen displayed is for Member & Incidence Input. Enter the member number. Use the down arrow key and enter the member start joint number; specify whether that end is restrained or released. The release is specified by using the right arrow cursor key as a toggle. Use the down arrow key and enter the member end joint number; specify whether that end is restrained or released. The release is specified by







using the right arrow cursor key as a toggle. Press the **[ENTR]** soft-key when complete. When the member incidence has been specified, the member properties screen will appear. In this screen, you will enter the member area, moment of inertia, the modulus of elasticity, and the material density.

```
   Member & Incidence
Member properties
member No.:   1
Area:         11.000
Inertia:      200.000
Elasticity:   29.000
Dencity pcf: [490.000 ]

ENTR|    |    |    |    |EXIT
```

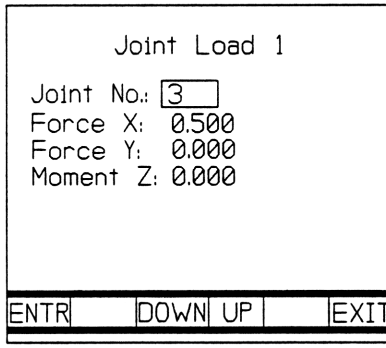
When the member data has been entered, the program will return to the Main Menu screen. At this point, you have two options. The first load case may be entered by pressing the **[I]** key; or, you can press the assemble matrixes **[K]** key. If the **[K]** key is pressed first, all the initial data will be printed out, and the assembling of the matrixes will occur. The loadings must then be entered by pressing the **[I]** key before pressing the Start Analysis key **[L]**; otherwise, an error will occur. In that case, the program will put the Main Menu back on the screen and exit the Plane Frame Program.

When the **[I]** key is pressed to enter loadings, all the screens will automatically appear as the data is entered. The ENTER LOADINGS screen will appear first. Enter the load case number. Use the down-arrow key to input the number of joint loads; the number of members with con-

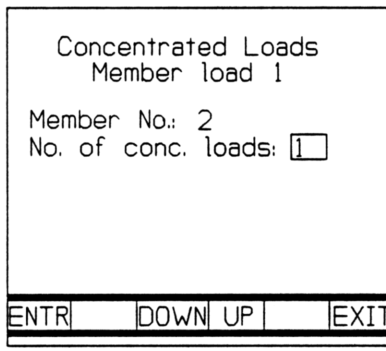
```
      Enter Loadings
       Load case:[1]
No. of joint loads:    1
No. of members with
 concentrated loads: 1
No. of members with
 distributed loads:  1

ENTR|    |    |    |    |EXIT
```

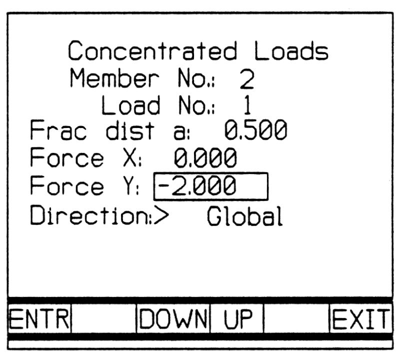
centrated loads; and, the number of members with distributed loads. Press **[ENTR]**.



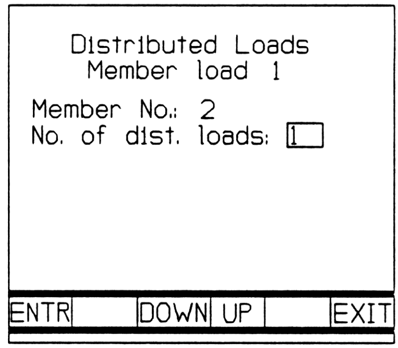
If joint loads are required, enter the joint number. Use the down-arrow key to input the force 'X', force 'Y', and moment 'Z'. Press **[ENTR]** to complete the screen. Do this for all joint loads specified.



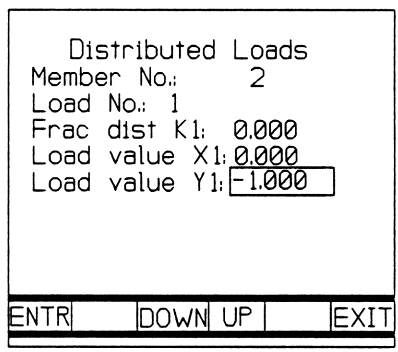
When joint loads are complete, enter the member number for concentrated loads and the number of concentrated loads on the member. Enter the fractional distance of the member, the force 'X', and the force 'Y'. Then, enter the direction of the load, either local or global. Press **[ENTR]**. Do this for all concentrated loads on the member and all members with concentrated loads.



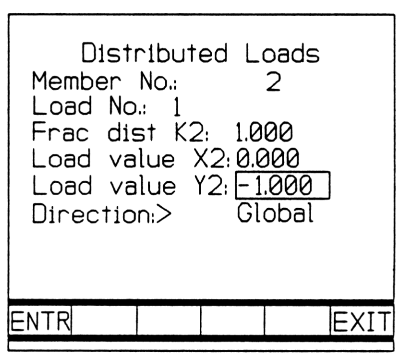
For distributed loads, enter the member number and the number of distributed loads. Enter the fractional start distance of the distributed load; start value in the 'X' direction; and, the start value in the 'Y' direction. Then, press **[ENTR]**. Enter the fractional end distance of the distributed load; end value in the 'X' direction; and, the end value in the 'Y' direction. Then, press **[ENTR]**. When all loadings have been entered, the program returns to the Plane Frame menu screen.



Depending on the option selected above, either the **[K]** key to assemble the matrixes or the **[L]** key to start the analysis is pressed.



After the first solution, other load cases may be entered without having to re-enter the joint and member data. If the load case is the second load case, enter the loading using the **[I]** key. When complete, press the Start Analysis key **[L]** for the second solution. Since the structure matrixes have already been assembled and factored, there is no need to repeat this step. Many load cases can be solved in this manner. The first solution will take the most time; all other solutions will be faster.



All frame geometry and loadings may be edited using the edit menu key **[J]**. If the joint data or any other member data is modified, the **[K]** key must be pressed to assemble the matrixes for the new structure model. If loads are modified, this is not necessary.

To go to another program in this card, it is suggested that you use the **[K]** key at the Main Menu to delete the data from the present directory.

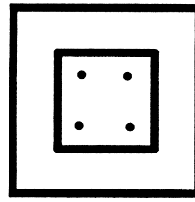
REFERENCES:

1. William Weaver Jr. and John Gere. ; "Matrix Analysis of Framed Structures"; 1975.

2. William Weaver Jr.; "Computer Programs for Structural Analysis"; 1975.

3. M. D. Vanderbilt; "Matrix Structural Analysis"; Quantum Press; 1974.

4. Crawley and Dillon; "Steel Buildings Analysis and Design"; John Wiley & Sons; 1977.



5

CONCRETE COLUMN ANALYSIS

This program computes the ultimate moment capacity of a concrete section with a given axial load, compression or tension. The section may contain reinforcing bars, prestressing strands, or both. The method of analysis is based on the Ultimate Strength Method. The program generates a moment interaction diagram for a given factored axial load.

PROGRAM DESCRIPTION:

This program will compute the ultimate moment capacity of a concrete section of any arbitrary shape. It may contain concrete areas of different strengths or voids. Four areas are allowed to be specified. The section may contain mild reinforcing or 270 k.s.i. low relaxation prestressing strands. If stress relieved strands are being used, a small error will be introduced because of the relative stress/strain diagram differences. The interaction data is generated by rotating the neutral axis between 0 degrees and 180 degrees in increments specified by the user.

The relationship between concrete compressive stress distribution and concrete strain is assumed to be rectangular. The concrete stress is taken as 0.85fc' uniformly distributed over an equivalent compression zone. That zone is bounded by the edges of the cross section and a straight line located parallel to the neutral axis at a distance, a=B1*c, from the fiber of maximum compressive strain.

CONCRETE COLUMN

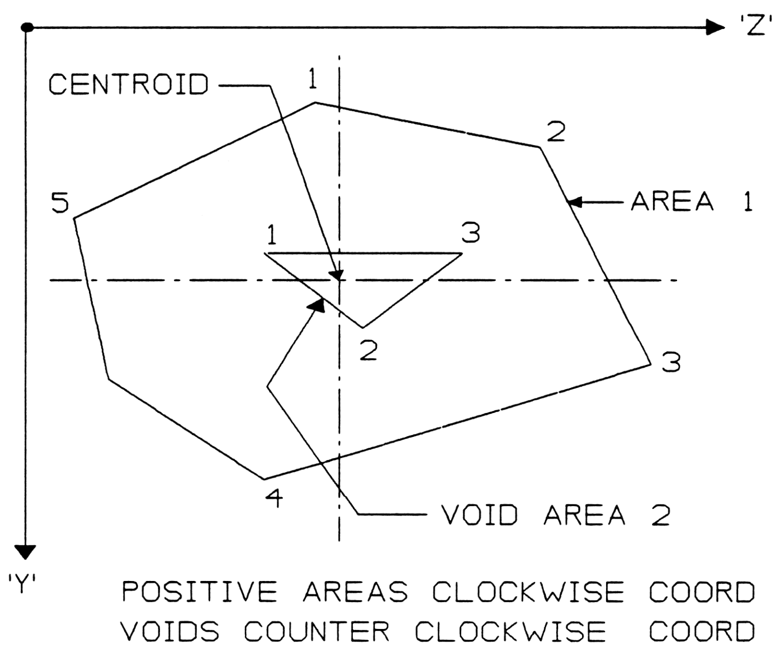
The maximum concrete strain is assumed to be 0.003, with a linear relationship for strain distribution across the section. It is assumed that the concrete has no strength in tension and that a perfect bond exists between concrete and reinforcing. The stress in the prestressing strand is determined from the calculated strain and follows the prescribed stress strain diagram for 270 k.s.i. "low relaxation" steel strands. Only short columns are considered. Moment magnification for slender columns and secondary effects on columns should be considered when comparing output results to the design condition. Computation continues until equilibrium is found.

The user must also be aware of certain conditions that may develop when using prestress in concrete columns. If no mild reinforcing is provided and the strands are distributed around the whole section, it is possible that equilibrium may not be found for a specified axial load. The strands will always contain tension. However, the available concrete contained in the stress block will not develop the required force after the total tension is subtracted from the concrete compression. The column analysis may find a solution for a different force, either greater or lesser than the force where equilibrium could not be found. The user should always provide mild reinforcing to account for this possibility. The program is designed to find the moment capacity for a specific force. If this force is subject to the described situation, the program will produce an error message: "EQUIL NOT FOUND".

The concrete section is defined by entering the coordinates of the vertices of the particular areas of the section. The coordinates may be of any arbitrary coordinate system. The vertices are numbered clockwise around the section. A void is numbered counterclockwise and is given the same strength as the parent material. The program automatically calculates the centroid of the section and transforms the input coordinates to the centroid. The ultimate moment capacity is calculated about the centroid of the section. The reinforcing coordinates are entered using the same coordinate system. The area of each bar or strand and the stress level for the prestressing strands are also entered.

The following sketch shows the sign convention used. The 'Z' axis is to the right, and the 'Y' axis is down. This represents a right-hand system where the 'X' axis runs along the longitudinal axis of the member. The compression face is always on the top parallel to the 'Z' axis.

Inputs are all positive. However, coordinates may be



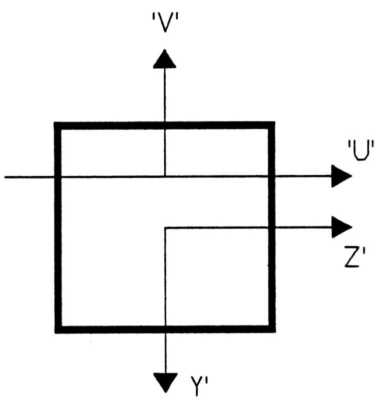
negative. Axial loads are entered positive for compression and negative for tension. Units are in kips, inches and degrees.

APPLICATION OF PHI FACTOR:

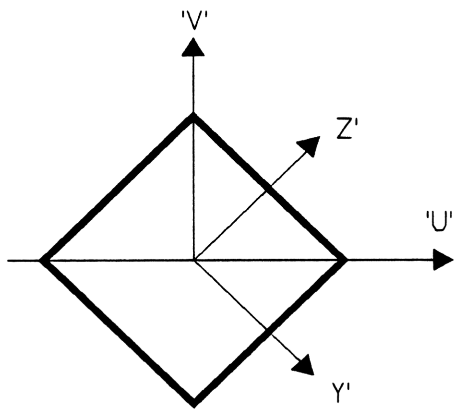
The user has the option of selecting phi (capacity reduction factor) equal to one (1) or having the phi factor calculated by the program. The program uses the American Concrete Institute specifications. The phi factor, if calculated, is based on 0.7 for axial loads greater than 0.1*fc'*Ag. Phi is increased linearly from 0.7 to 0.9 when Pu is between 0.1*fc'*Ag and zero. Whether phi equals one or is computed, its application is as follows: The input value Pu is divided by phi to obtain Pn (the nominal force). Using the nominal force to find equilibrium, the moment capacities are calculated and then multiplied by phi and output.

METHOD OF CALCULATION:

The program mechanics keep the neutral axis horizontal and rotate the section and input axis counterclockwise at the user-specified angle-increment. Using this orientation, the compression face is always at the top. At each specified angle position, the location of the neutral axis is first found at the approximate balance point Cb. Then, the balance load, Pb, is computed and compared to the input value, Pn. If Pn is greater, the neutral axis must be incremented; if the value of Pn is less, then the neutral axis must be decreased. At each position of neutral axis, the axial forces on the section due to strain in the concrete and the steel are summed and compared to the input axial load. If equilibrium is found, the moments are summed about the 'u' and 'v' axes and translated to the 'Z' and 'Y' axes. The angle is then incremented. The process is repeated until the interaction diagram data has been completed. If the input value lies between Pmax and Tmax, a solution should always be found. The program computes the values of Pmax and Tmax. If the input value exceeds these values, an error message is displayed. Pmax is defined as 80% of the maximum calculated capacity at pure concentric load, as specified by most codes for reinforced concrete.



INITIAL ORIENTATION



INCREMENTED ORIENTATION

The following is a list of items required for entry:

1. Problem Title.

2. Number of concrete areas.

3. Number of points defining each area and concrete strength.

4. 'Z' and 'Y' coordinates for each point of each area.

5. Number of prestressing tendons.

6. 'Z' and 'Y' coordinates for each tendon; area Aps and fps for each tendon.

7. Number of reinforcing bars and yield strength.

8. 'Z' and 'Y' coordinates for each bar area; As of bar or bar group.

9. Factored ultimate compression load Pu or tension load Tu.

The following units are used by the program:

Concentrated load, **Pu**, is in **kips** (*1000 pounds=1 kip*). **Length** and **coordinate** units are in **inches.**

Areas are designated in **square inches.**

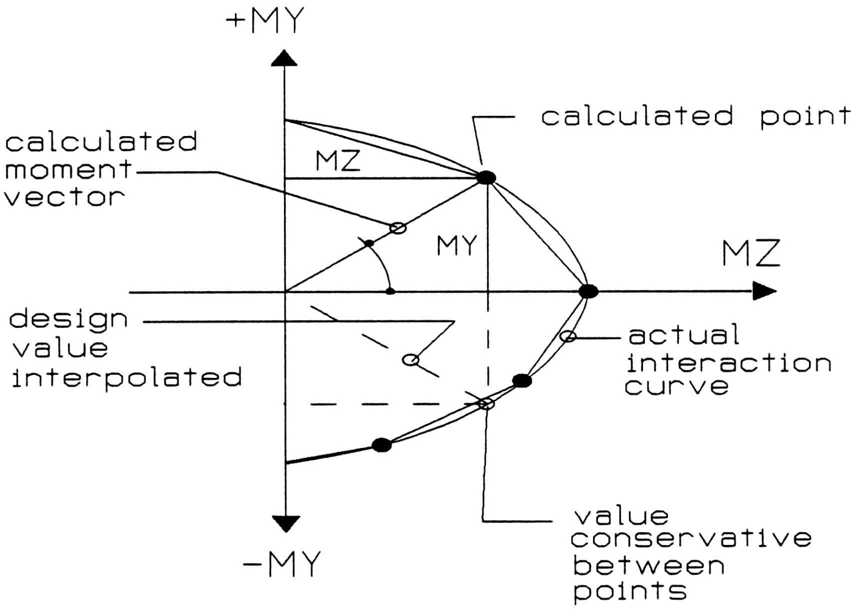
Moments are in **inch-kips.**

Stress in **kips per square inch.**

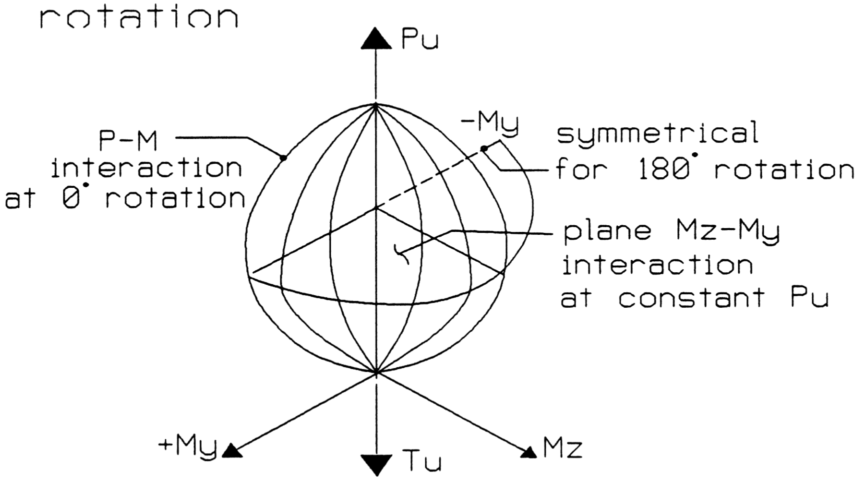
The following sketches show the horizontal interaction diagram for moments Mz and My at a given axial load and a three-dimension diagram.

At this point, several features of this program should be mentioned. It can be used as a general purpose concrete section solver. Concrete beams of any arbitrary shape can be analyzed using Pu=0. Reinforcing can be located anywhere in the section. Prestressed concrete beams can be analyzed using Pu equal to the negative prestress force (essentially Pu=0); or, they can be analyzed as a beam column specifying Pu=0. Moment capacities of concrete piles or hollow concrete sections can be determined.

The following sketch shows a typical column problem that can be analyzed using this program. The following general



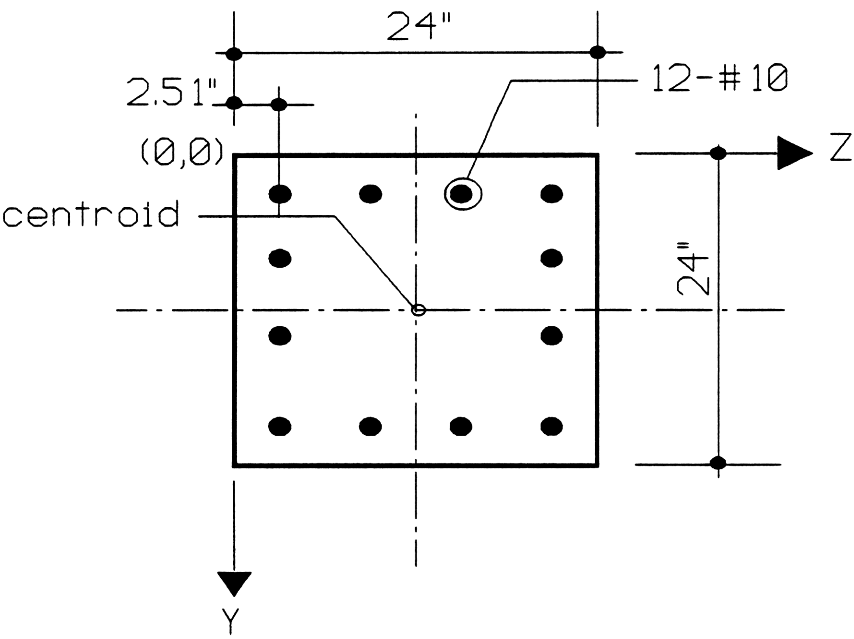
Plane MZ-MY interaction at constant Pu @ 180° rotation



THREE DIMENSION INTERACTION DIAGRAM

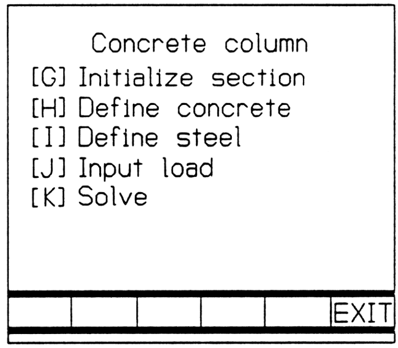
instructions describe the procedure of input and the resulting screen displays on the HP-48SX. Trying out this problem and using the display screens together with the explanations should get you started.

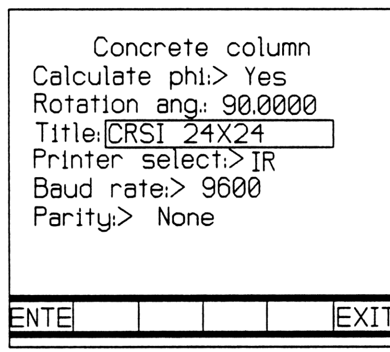
It is assumed that you are at the Concrete Column program screen. Select the **[G]** key to begin. The first option is a toggle selection for the value of the phi factor. The phi



EXAMPLE PROBLEM

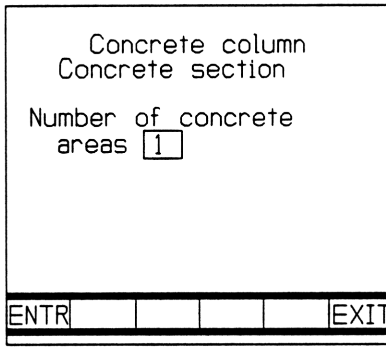
factor (strength reduction factor) may be calculated using the A.C.I. specifications or may be selected to be the value of one. If the value of one is selected, the value of phi may be hand-calculated for use in codes with other requirements. Press the down-arrow cursor key to select the rotation angle.



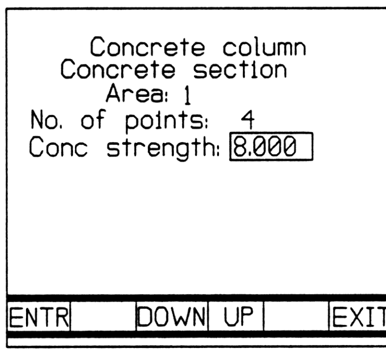


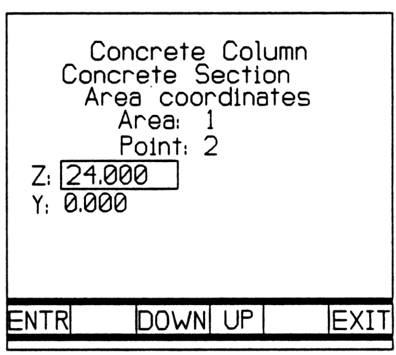
The rotation angle is a selection for the increment value in even-degrees. The value selected must be evenly divisible into 180 degrees. Selecting a value of zero (0) will calculate one value as a point on a uniaxial interaction diagram for the specified load. This increment determines the number of points that will be calculated on the horizontal interaction diagram. Cursor down to enter the title to the problem. Press the alpha key one time and type in the name or description desired. Press the alpha key when complete and cursor down to select printer specifications. You can then select IR or serial printer. If you are using a serial printer, you will have to specify the baud rate and parity if they are required. Press **[ENTE]** to return to the main screen for the next data-entry area.

Press the **[H]** key to define the concrete section. Enter the number of concrete areas and press **[ENTR]**. The next data will be the number of points to describe the section and the strength of that area. When complete, press **[ENTR]**. The next screens to appear require data for the coordinate description of all the points for the area. These two screens will be repeated for each of the concrete areas specified. An area may be specified as a void by entering the coordinates of the void in a counterclockwise direction. When all the concrete areas have been defined, the program returns to the main screen for the description of the mild reinforcing and the prestress reinforcing.



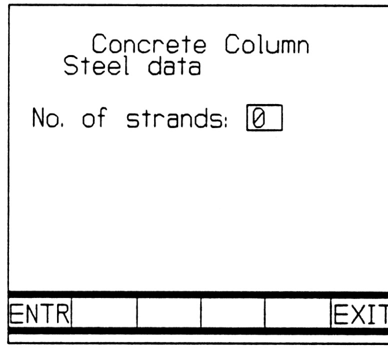
Press the **[I]** key to begin the steel definition. The first data specifies the number of prestressing strands, if any are required. Enter zero if none are being used. Press **[ENTR]**. The next screen to appear will allow the input of data and coordinates for each strand. Use the cursor keys to move from value to value until the data is complete and totally correct. If mild reinforcing is being used, the screen will appear after the selection of the prestress steel specifications. The screen will allow the input of the quantity of bars and the required yield stress specification. Press **[ENTR]** when complete. The next screen will allow the

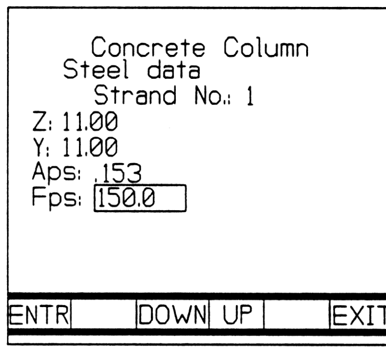


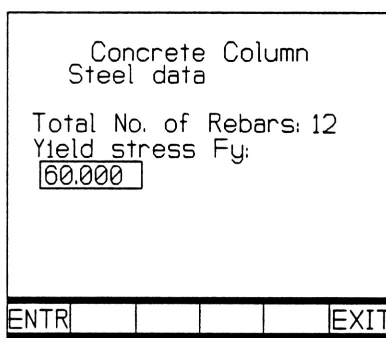


entry of rebar coordinates. Specify the area of each bar. Use the cursor keys to enter the data. When complete, press **[ENTR]**. To enter the specified design load, press the **[J]** key. When complete, press **[ENTR]**.

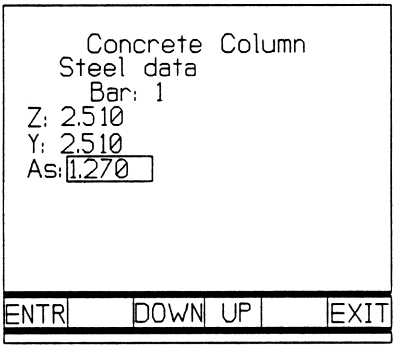
To begin the solution, press the **[K]** key. The solution will be printed on the printer, if one were specified; or, it will be displayed on the HP-48SX screen.

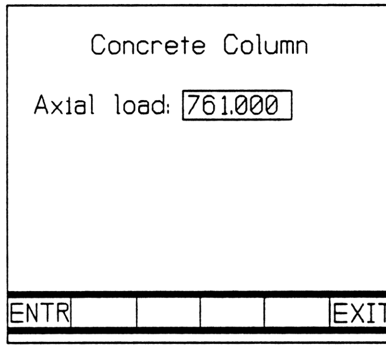






CONCRETE COLUMN





REFERENCES:

1. Wang and Salmon; Reinforced Concrete Design, 3rd Edition; John Wiley and Son.

2. Building Code Requirements for Reinforced Concrete A.C.I. 318-89.

3. C.R.S.I. Concrete Design Manual 1978, 1980, 1982, 1984.

4. Bressler, Boris; Design Criteria for Reinforced Concrete Columns under Axial Load and Biaxial Bending; A.C.I. Journal Proceedings V57N5 1960.

5. Meek, John L.; Ultimate Strength of Columns with Biaxially Eccentric Loads; A.C.I. Journal Proceedings V60N8 Aug.1963.

6. Mattock, A. H.; Kriz, L. B. Hognestead; Rectangular Concrete Stress Distribution in Ultimate Design; A.C.I. Journal Proceedings V57 1961.

7. Furlong, Richard W.; Ultimate Strength of Square Columns under Biaxially Eccentric Loads; A.C.I. Journal Proceedings V57 Mar. 1961.

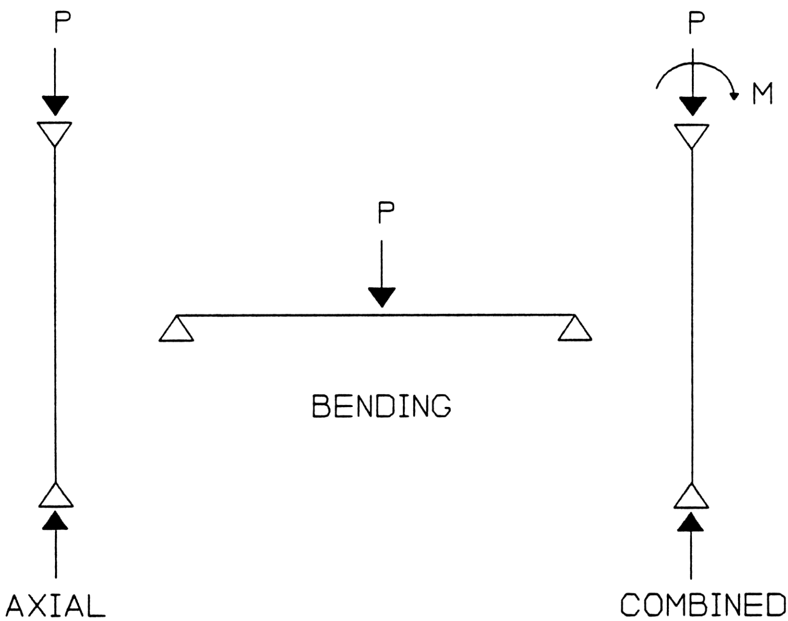
8. Heimdahl, Peter D.; Bianchi, Albert C.; Ultimate Strength of Biaxially Eccentrically Loaded Concrete Columns Rein-

forced with High Strength Steel; A.C.I. SP-50 1975. 9. Furlong, Richard W.; Concrete Columns Under Biaxial Eccentric Thrust; A.C.I. Journal Proceedings V76N10 1979.

9. Quasi-Newton Method for Reinforced Concrete Column Analysis and Design; Richard Yen; A.S.C.E. Journal of Structural Engineering; Vol. 117, No. 3; March 1991.

6

STEEL COLUMN AND BEAM SOLUTIONS



GENERAL DESCRIPTION

This program provides a ready solution for the analysis of structural steel beams and columns of W,I,H,S, and M section shapes. The section properties may be obtained from the manuals referenced below. The properties are entered into the program at the display prompts. Solutions may be in the form of Allowable Stress Design **(ASD)** or Load Resistance Factor Design **(LRFD)**. The Allowable Stress Design of steel members has been the traditional method used by structural engineers for many years. The method uses stresses determined from service load conditions within the elastic range of the material. The Load Resistance Factor Design of steel members is the "limits state design" approach and has gained acceptance in the United States during the 1980's. The LRFD method uses the mode of failure such as yielding, buckling, or fracture at factored service load conditions to determine the adequacy of the members selected for use in a structure. The steel solution module of this application package uses the following code references: The Ninth Edition of the " A.I.S.C. Specifications for the Design, Fabrication and Erection of Structural Steel for Buildings" (in the ASD portion of the solution); The First Edition of A.I.S.C. " Load Resistance and Factor Design" manual (in the LRFD portion of the solution).

ALLOWABLE STRESS DESIGN

AXIAL SOLUTION:

1. Allowable axial stress, Fa, is calculated.

2. Actual axial stress, fa, is calculated.

3. Allowable concentric load, Pa, is calculated.

BENDING SOLUTION:

1. Allowable bending stress is calculated considering compactness and the limiting values of the unsupported length between compression flange bracing points about the X axis,

Fbx.

2. Allowable bending stress is calculated considering compactness and the limiting values for the Y axis, Fby.

3. The actual bending stress is calculated and expressed in terms of unity for bending about one or two axes. For biaxial bending, the equation for interaction is computed.

COMBINED STRESS SOLUTION:

1. The allowable axial stress is calculated.

2. The actual axial stress is calculated.

3. The allowable bending stress is calculated, considering compactness and the limiting values of the unsupported length between compression flange bracing points about the X axis, Fbx.

4. The allowable bending stress is calculated, considering compactness and the limiting values for the Y axis, Fby.

5. Actual bending stress fbx and fby are calculated.

6. Solution of the interaction equations for combined stresses of compression and bending about the 'X' axis and the 'Y' axis is performed.

7. A check of allowable width and thickness ratios is performed.

Included in this section are tables for determining the values of Kx, Ky, Cmx, and Cmy. The value of K is a coefficient which determines the effective column length as dependant on the end condition of the column. The value of Cm is a moment reduction factor as defined in the A.I.S.C. specifications. The value Cb is a coefficient used in determining the allowable bending stress.

DETERMINATION OF Cmx AND Cmy:

1. For compression members in frames subjected to joint translation (sideways), Cm=0.85.

2. For compression members in frames braced against joint translation and not subject to transverse loading between their supports in the plane of bending:

Cm=.6-.4*M1/M2, but not less than 0.4.

Where M1/M2 is the ratio of the smaller to the larger moments at the ends of the member that is unbraced in the plane of bending. M1/M2 is positive when the member is bent in reverse curvature and negative when bent in single curvature.

3. For compression members in frames braced against joint translation in the plane of bending and subjected to transverse loading between supports, the value of Cm may be determined by rational analysis. However, in lieu of such analysis, the following values may be used:

a.) For members with constrained ends, Cm=0.85.

b.) For members with unrestrained ends, Cm=1.0.

DETERMINATION OF Cb:

The following equation is used in determining the value of Cb: $Cb=1.75+1.05*(M1/M2)+0.3*(M1/M2)^2$

but not more than 2.3 (Cb can be conservatively taken as unity). M1 is the smaller and M2 is the larger bending moment at the ends of the unbraced length taken about the strong axis of the member. M1/M2, the ratio of end moments, is positive when of the same sign and negative when of opposite sign. When the bending moment at any point within the unbraced length is greater than the end moments, the value of Cb shall be taken as unity. For frames braced against joint translation, Cb=1.

ASD SECTION PROPERTIES:

A= Area of section (in^2).

d= Depth of section (in).

tw= Thickness of web plate (in).

bf= Width of flange (in).

tf= Thickness of flange (in).

Sx= Section modulus 'X' axis (in^3).

rx= Radius of gyration 'X' axis (in).

Sy= Section modulus 'Y' axis (in^3).

ry= Radius of gyration 'Y' axis (in).

rt= Radius of gyration of a section comprised of the compression flange plus one third of the compression web area (in).

The moments of inertia are not required for the solution.

LOAD AND RESISTANCE FACTOR DESIGN

OF COMPACT SECTIONS

AXIAL SOLUTION:

1. The critical axial stress, Fcr, is calculated. Local buckling, flexural torsional buckling and unbraced length are considered.

2. The critical axial force capacity of the member,

phi*Ag*Fcr, is calculated.

BENDING SOLUTION:

1. The maximum moment capacity, phi*Mnx, considering local buckling, unbraced length of the compression flange and the compactness of the section used.

2. The maximum moment capacity, phi*Mny, considering local buckling, unbraced length of the compression flange and the compactness of the section used.

3. The interaction equation for biaxial bending.

COMBINED STRESS SOLUTION:

1. The critical axial stress, Fcr, considering local buckling, flexural torsional buckling and unbraced length.

2. The critical axial force capacity, phi*Ag*Fcr, of the member.

3. The maximum moment capacity, phi*Mnx, considering local buckling, unbraced length of the compression flange and the compactness of the section used.

4. The maximum moment capacity, phi*Mny, considering local buckling, unbraced length of the compression flange and the compactness of the section used.

5. The interaction equation for axial load and biaxial bending.

LRFD SECTION PROPERTIES:

A= Area of section (in^2).

bf= Width of flange (in).

tf= Thickness of flange (in).

hc/tw=Web height divided by web thickness.

Ix= Moment of inertia 'X' axis (in^4).

Sx= Section modulus 'X' axis (in^3).

rx= Radius of gyration 'X' axis (in).

Iy= Moment of inertia 'Y' axis (in^4).

Sy= Section modulus 'Y' axis (in^3).

ry= Radius of gyration 'Y' axis (in).

Zx= Plastic modulus 'X' axis (in^3).

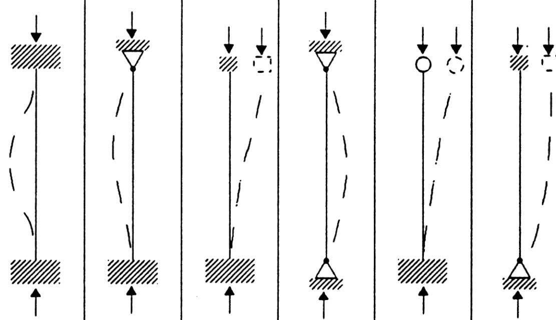
Zy= Plastic modulus 'Y' axis (in^3).

J= Torsional constant.

Cw= Warping constant.

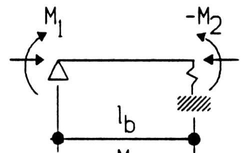
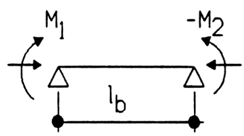
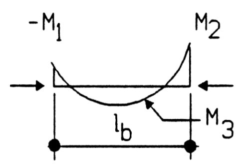
See the Allowable Stress Design section for the determination of the effective length factor Kx, Ky and Cb used in LRFD solution.

The A.I.S.C. specifications are explicit in determining the requirements for slenderness and for second order effects when members are in braced or moment frames. Therefore, the design ultimate moments input into the program should include the appropriate magnification factors due to these conditions. It is suggested that the A.I.S.C. L.R.F.D. design specifications be referenced carefully while using this program.

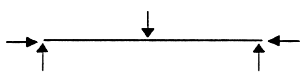
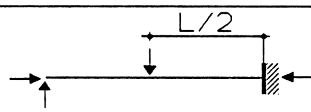
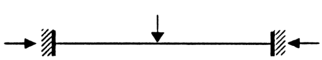
	(a)	(b)	(c)	(d)	(e)	(f)
Buckled shape of column is shown by dashed line						
Theoretical K value	0.5	0.7	1.0	1.0	2.0	2.0
Recommended design value	0.65	0.8	1.2	1.0	2.10	2.0
End condition code		Rotation fixed, translation fixed Rotation free, translation fixed Rotation fixed,translation free Rotation free, translation free				

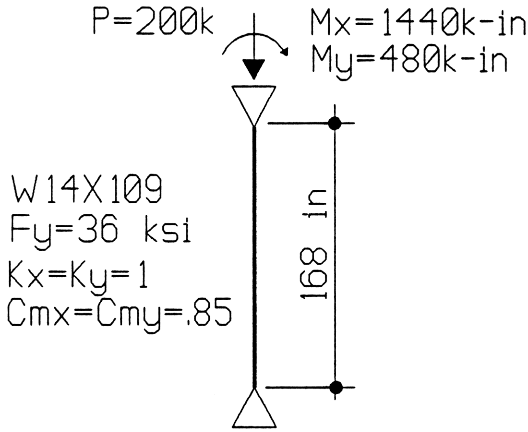
VALUES OF Kx AND Ky

Cm TABLE-1

Cat	Loading conditions ($f_a > 0.15F_a$)	fb	Cm	Remarks
A	Computed moments maximum at end; joint translation not prevented	$\frac{M_2}{S}$	0.85	 $M_1 < M_2$ $\frac{M_1}{M_2}$ negative as shown check both formulas H1-1 and H1-2
B	Computed moments maximum at end; no transverse loading; joint translation prevented	$\frac{M_2}{S}$	$(0.6 \pm 0.4\frac{M1}{M2})$ but not less 0.4	 Check both formulas H1-1 and H1-2
C	Transverse loading; translation prevented	$\frac{M_2}{S}$ form H1-2 $\frac{M_3}{S}$ form H1-1	$1 + \psi\frac{f_a}{F'_e}$	 Check both formulas H1-1 and H1-2

VALUES OF Cm TABLE 2

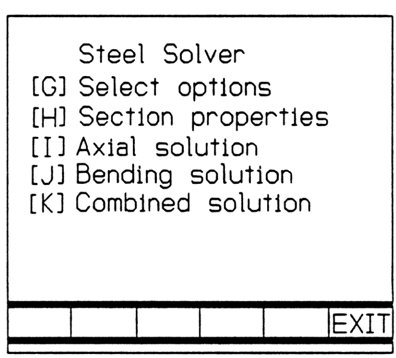
Case	ψ	Cm
	0	1.0
	-0.4	$1-0.4\frac{f_a}{F'_e}$
	-0.4	$1-0.4\frac{f_a}{F'_e}$
	-0.2	$1-0.2\frac{f_a}{F'_e}$
	-0.3	$1-0.3\frac{f_a}{F'_e}$
	-0.2	$1-0.2\frac{f_a}{F'_e}$



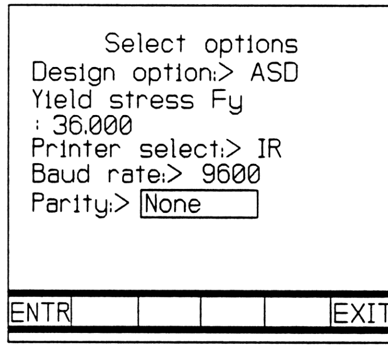
ASD EXAMPLE

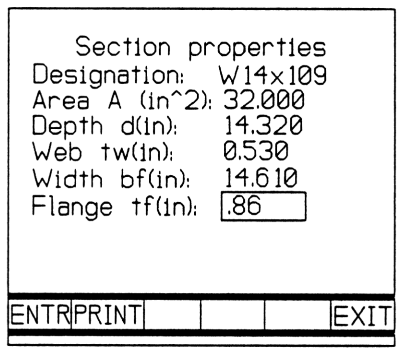
The following sketches show a typical column problem that can be analyzed using this program.

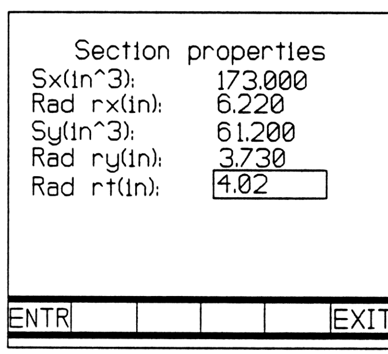
The following instructions are of a general nature describing the procedure of data-entry and the resulting screen displays on the HP-48. Trying out this problem and using the display screens together with the explanations should get you started.



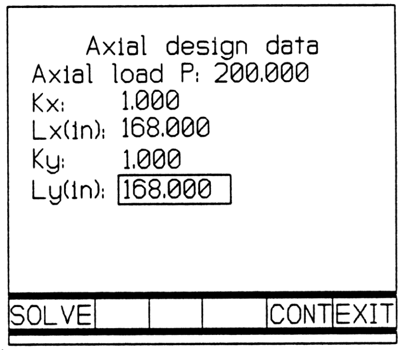
It is assumed that you are at the Steel Solver program screen. Press **[G]** to select options and begin. First, select the ASD option; enter the yield stress; and, select printer options. Press the **[EXIT]** soft-key to return to the main Steel Solver menu screen. Press **[H]** to enter the section properties of the steel section to be analyzed. Use the down arrow key to enter the section properties for the first screen and press **[ENTR]**. Complete the entry of section properties in the second properties screen and press **[ENTR]**. When the section properties data are complete, the Steel Solver Menu will be displayed for other selections. If the solution required is an axial solution only, press the **[I]** menu key. If the solution required is a bending solution, press the **[J]** menu key. If the solution required is a combined solution, press the **[K]** menu key. The example problem is a combined solution. Press the **[K]** menu key. When the combined solution is selected, an axial solution and a bending solution is required. The program automatically executes these routines sequentially. This procedure will help to understand

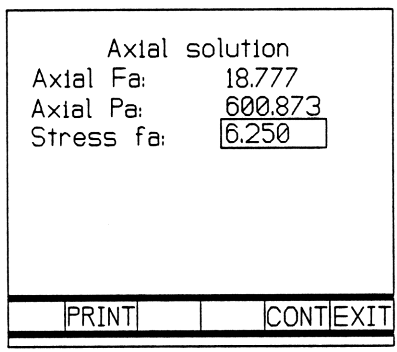


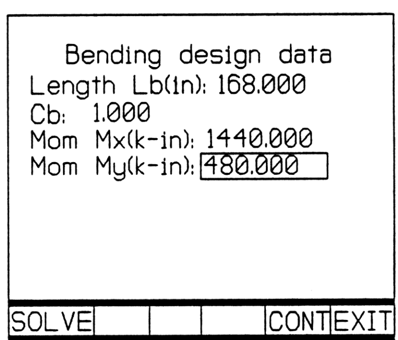




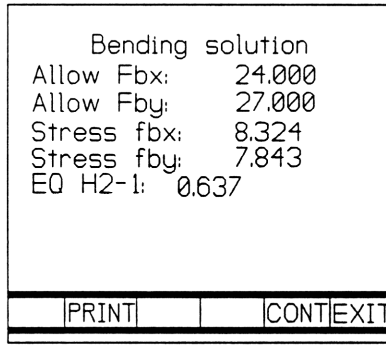
the axial solution and the bending solution portions of the program. There are two softkey options in the Axial Design Data screen and the Bending Design Data screen. One option is the **[SOLVE]** softkey; the other is the **[CONT]** softkey. Press the **[CONT]** softkey to continue a combined analysis. Press the **[SOLVE]** softkey if the solution is an axial-only or bending-only solution. Press the **[CONT]** softkey to continue the example combined solution. The Axial Design Data screen will be displayed. Use the down-arrow cursor key and enter the required information. When complete, press the **[CONT]** softkey. The Axial Solution screen with the solution results is displayed. If hard copy of the results is required, press the **[PRINT]** softkey. Press the **[CONT]** softkey to continue the combined solution. The next screen to be displayed is the Bending Design Data screen. Use the down-arrow cursor key and enter the required

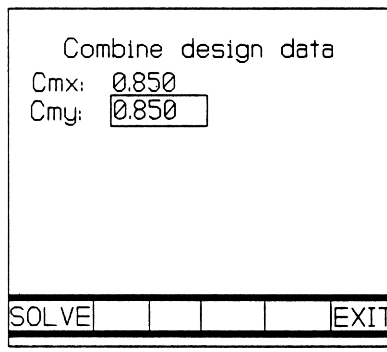


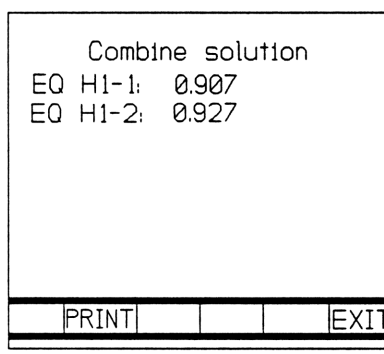




information. When complete, press the **[CONT]** soft-key. The Bending Solution screen with the solution results. Press the **[PRINT]** softkey if hard copy of the results is required. Press the **[CONT]** softkey to continue the combined solution. The next screen to be displayed is the Combine Design Data screen. Use the down-arrow cursor key and enter the required information. When complete, press the **[SOLVE]** softkey. The Combine Solution screen with the solution results is displayed. Press the **[PRINT]** softkey if hard copy of the results is required. Press the **[EXIT]** softkey to return to the main Steel Solver Menu screen.



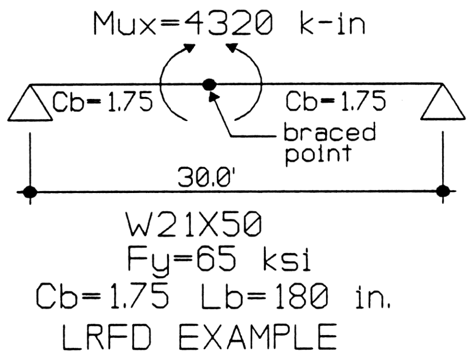




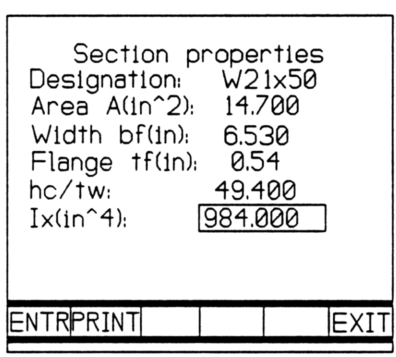
The following sketch shows a

typical beam problem that can be analyzed using the LRFD portion of this program.

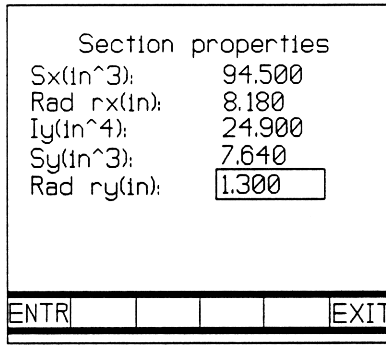
It is assumed that you are in the Steel Solver program

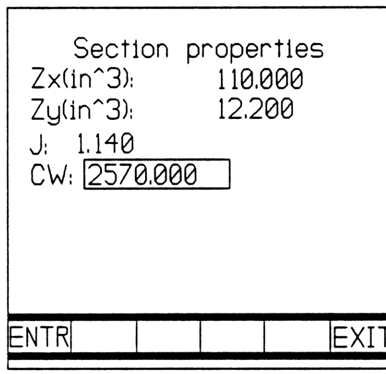


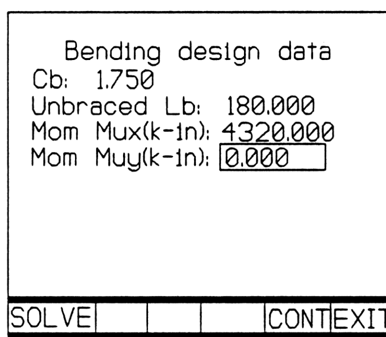
screen. Press the **[G]** key to select options and begin. First, select the LRFD option; enter the yield stress; and, select printer options. Press the **[EXIT]** soft key to return to the main Steel Solver Menu screen. Press the **[H]** menu key to enter the section properties of the steel section to be analyzed. Use the down-arrow key to enter the section properties for the first screen and press **[ENTR]**. Complete the section properties of the second properties screen and press **[ENTR]**. Complete the section properties of the third properties screen and press **[ENTR]**. When the section properties data is complete, the main Steel Solver Menu screen will be displayed for other selections. If the solution



required is an axial solution only, press **[I]**. If the solution required is a bending solution, press **[J]**. If the solution required is a combined solution, press **[K]**. The example problem is a bending solution. Press the **[J]** menu key. The next screen to be displayed is the Bending Design Data screen. Use the down-arrow cursor key and enter the required data. When complete, press the **[SOLVE]** soft-key. The Bending Solution screen with the solution results will be displayed. Press the **[PRINT]** soft-key if hard copy of the results is required. Press **[EXIT]** to return to the main Steel Solver Menu screen.







OPERATING LIMITS AND WARNINGS:

1. The user should be familiar with the A.I.S.C. specifications

```
      Bending solution
ø Mnx:  4391.148
ø Mny:   713.000
EQ= 0.984

     PRINT            CONT EXIT
```

for determining allowable stresses in design and with the complexity of the conditions involved.

2. The program is not valid for A514 steel or box type members, angles, channels, or tee's.

3. Flanges shall be continuously connected to the web.

4. If the slenderness ratio, KL/r, is > 200, a message is displayed.

5. The program assumes non-stiffened edges for compression flanges.

6. When the maximum width thickness ratios are exceeded, a message is displayed and the program is halted.

7. When Lb is exceeded or Lu is exceeded, a message is displayed and the program computes the allowable stresses accordingly.

REFERENCES:

1. Charles G. Salmon, John E. Johnson "Design of Steel Structures", Third Edition, Harper and Row, N.Y.

2. Johnson, Jen-Lin, Galambos "Basic Steel Design"; Second Edition; Prentice-Hall, 1980.

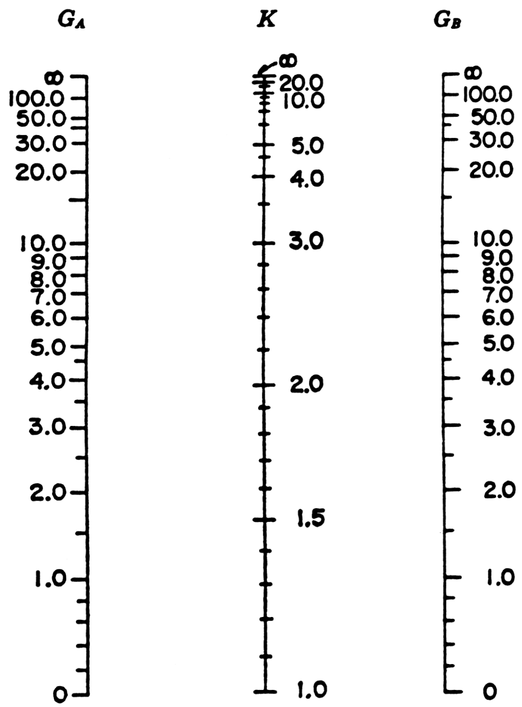
3. Crawley, Dillon, "Steel Building Analysis and Design" John Wiley and Son, 1977.

4. A.I.S.C. Specifications for the Design, Fabrication and Erection of Steel Buildings. Ninth Edition.

5. A.I.S.C. Load Resistance Factor Design manual, First Edition.

6. J.C.Smith, "Structural Steel Design: LRFD Fundamentals", John Wiley and Son, N.Y., 1988.

APPENDIX A



Sidesway Uninhibited

The subscripts A and B refer to the joints at the two ends of the column section being considered. G is defined as

$$G = \frac{\sum \frac{I_c}{L_c}}{\sum \frac{I_g}{L_g}}$$

in which Σ indicates a summation of all members rigidly connected to that joint and lying in the plane in which buckling of the column is being considered. I_c is the moment of inertia and L_c the unsupported length of a column section, and I_g is the moment of inertia and L_g the unsupported length of a girder or other restraining member. I_c and I_g are taken about axes perpendicular to the plane of buckling being considered.

For column ends supported by but not rigidly connected to a footing or foundation, G is theoretically infinity, but, unless actually designed as a true friction free pin, may be taken as "10" for practical designs. If the column end is rigidly attached to a properly designed footing, G may be taken as 1.0. Smaller values may be used if justified by analysis.

This Nomograph was duplicated from

The Design Manual of Steel Constriction by the

American Institute of Steel Construction, Ninth Edition.

APPENDIX B

EXAMPLE SOLUTION

```
GENERAL BEAM ANALYSIS

SPAN: 1

LENGTH: 16.000
ELASTICITY: 1
INERTIA: 2.000
UNIFORM load: 0.000

CONCENTRATED LOAD: 1
DIST a: 8.000
VALUE P: 7.500

SPAN: 2

LENGTH: 12.000
ELASTICITY: 1
INERTIA: 1.000
UNIFORM load: 1.000

CONCENTRATED MOMENT: 1
DIST a3: 12.000
MOMENT: 15.000

SUPPORT MOMENTS

SPAN 1
Moment left=-16.5000
Moment right=12.0000

SPAN 2
Moment left=-12.0000
Moment right=0.0000

SPAN 1

At X=0
Shear = 4.031
Moment = -16.500
Rotation = 2.000E-10
Deflection = 0.000

SPAN 1

At X=16
Shear = -3.469
Moment = -12.000
Rotation = 6.000
Deflection = 0.000

SPAN 2

At X=0
Shear = 5.750
Moment = -12.000
Rotation = 6.000
Deflection = 0.000

SPAN 2

At X=12
Shear = -6.250
Moment = -15.000
Rotation = -12.000
Deflection = 0.000
```

APPENDIX B

EXAMPLE SOLUTION

```
PLANE FRAME ANALYSIS

JOINT COORD AND SUPPORTS

JOINT 1
X=0.0000
Y=0.0000
SUPPORT FX RESTRAINED
SUPPORT FY RESTRAINED
SUPPORT MZ RESTRAINED

JOINT 2
X=20.0000
Y=0.0000
SUPPORT FX RESTRAINED
SUPPORT FY RESTRAINED
SUPPORT MZ RESTRAINED

JOINT 3
X=0.0000
Y=10.0000

JOINT 4
X=20.0000
Y=10.0000

MEMBER DATA

MEMBER 1

JOINT 1 TO 3
LENGTH=10.0000
A=11.0000
IZ=200.0000
E=29.0000

MEMBER 2

JOINT 3 TO 4
LENGTH=20.0000
A=11.0000
IZ=200.0000
E=29.0000

MEMBER 3

JOINT 2 TO 4
LENGTH=10.0000
A=11.0000
IZ=200.0000
E=29.0000

TOT WT=1497.2222 LBS

.OAD CASE 1

JOINT LOADS

JOINT No. 3
FX=0.5000
FY=0.0000
MZ=0.0000

CONCENTRATED LOADS

MEMBER 2
a=0.5000
FX=0.0000
FY=-2.0000
GLOBAL

DISTRIBUTED LOADS

MEMBER 2
K1=0.0000
WX1=0.0000
WY1=-1.0000
K2=1.0000
WX2=0.0000
WY2=-1.0000
GLOBAL

DISPLACEMENTS

JOINT 1
X=0.0000
Y=0.0000
R=0.0000

JOINT 2
X=0.0000
Y=0.0000
R=0.0000

JOINT 3
X=0.0127
Y=-0.0041
R=-0.0020

JOINT 4
X=0.0091
Y=-0.0042
R=0.0018
```

APPENDIX B

EXAMPLE SOLUTION

```
MEMBER FORCES

MEMBER 1
FA=10.9063
VA=-4.3208
MA=-13.5508
FB=-10.9063
VB=4.3208
MB=-29.6570

MEMBER 2
FA=4.8208
VA=10.9063
MA=29.6570
FB=-4.8208
VB=11.0937
MB=-31.5302

MEMBER 3
FA=11.0937
VA=4.8208
MA=16.6776
FB=-11.0937
VB=-4.8208
MB=31.5302


REACTIONS

JOINT 1
FX=4.3208
FY=10.9063
MZ=-13.5508

JOINT 2
FX=-4.8208
FY=11.0937
MZ=16.6776

MEM USED 4340
```

APPENDIX B

EXAMPLE SOLUTION

CONCRETE COLUMN ANALYSIS

CRSI 24X24

Phi ø Calculated

No. of Conc. Areas = 1
Concrete Area 1
No. of Points = 4
F'c (ksi) = 8.000

Pt.	Z in	Y in
1	0.00	0.00
2	24.00	0.00
3	24.00	24.00
4	0.00	24.00

No. of Strands = 0

No. of Rebar = 12
Fy (ksi) = 60

Z in	Y in	Area in2
2.51	2.51	1.27
8.84	2.51	1.27
15.16	2.51	1.27
21.49	2.51	1.27
21.49	8.84	1.27
21.49	15.16	1.27
21.49	21.49	1.27
15.16	21.49	1.27
8.84	21.49	1.27
2.51	21.49	1.27
2.51	15.16	1.27
2.51	8.84	1.27

Pu=761.000

CGz (in) = 12.00
CGy (in) = 12.00
C. Area (in2) = 576.00

ULTIMATE MOMENTS
ABOUT Z AND Y AXES

Pn=1087.14

NA∠=0.00
C=10.97
a=7.13
MZ=10971.58
MY=7.00E-10
Mr=[10971.58 ∠3.66E-12]

NA∠=90.00
C=10.97
a=7.13
MZ=1.40E-9
MY=10971.58
Mr=[10971.58 ∠90.00]

NA∠=180.00
C=10.97
a=7.13
MZ=-10971.58
MY=0.00
Mr=[10971.58 ∠180.00]

APPENDIX B
EXAMPLE SOLUTION

```
ASD SECTION PROPERTIES
W14X109

Area A=32.00
Depth d=14.32
Web tw=0.53
Width bf=14.61
Flange tf=0.86
Sx=173.00
Rad rx=6.22
Sy=61.20
Rad ry=3.73
Rad rt=4.02

AXIAL SOLUTION

P=200.000
Kx=1.000
Lx=168.000
Ky=1.000
Ly=168.000

Allow Fa=18.777
Axial Pa=600.873
stress fa=6.250

BENDING SOLUTION

Lb=168.000
Cb=1.000
Mom Mx=1440.000
Mom My=480.000

Allow Fbx=24.000
Allow FbY=27.000
Stress fbx=8.324
Stress fby=7.843
EQ H2-1=0.637

COMBINED SOLUTION

Cmx=0.850
Cmy=0.850

H1-1,3=0.907
H1-2=0.927
```

```
LRFD SECTION PROPERTIES
W21X50

Area A=14.700
Width bf=6.530
Flange tf=0.540
hc/tw=49.400
Ix=984.000
Sx=94.500
Rad rx=8.180
Iy=24.900
Sy=7.640
Rad ry=1.300
Zx=110.000
Zy=12.200
J=1.140
Cw=2570.000

BENDING SOLUTION

Cb=1.750
Lb=180.000
Mom Mux=4320.000
Mom Muy=0.000

øMnx=4391.148
øMny=713.700
EQ=0.984
```

APPENDIX C

QUICK-START USERS INSTRUCTIONS

GENERAL BEAM ANALYSIS

1. **[α] [α] [STRUCT]** to start the TDS Structural Analysis Program.

2. Press **[G]** to start the General Beam Analysis Program

3. For printer select options, press **[K]**; select IR or baud rate, press **[EXIT]**.

4. Press **[G]** for new beam setup.

a. Enter the number of spans.

b. Enter the start support definition.

c. Enter the end support definition. Choose support conditions with the right-arrow key.

d. A cantilever beam should be specified with a start support as **NONE** and end support as **FIX.**

e. Press **[ENTR]** .

5. Enter the span L, modulus of elasticity E, and moment of inertia I for each span; press **[ENTR]** when completed.

6. Enter the number of concentrated loads, distributed loads, and concentrated moments. Enter the distances and load values as requested.

7 To print beam data, press **[K] [PRINT]**. When printing is complete, press **[EXIT]**.

8. To edit the previous data, press **[H]** and follow the instructions as shown in the screens.

9. For a simple statically determinate beam, press **[J]** (cantilever allowed). For an indeterminate beam or continuous beam, press **[I]** for continuous beam analysis. Press **[SOLVE]** for the solution.

a. To print the end moment support solution, press **[PRINT]**. All supports will be printed.

b. Press **[DOWN]** for the next support display.

c. Press **[EXIT]** to return to the MAIN MENU.

10. To perform an individual span analysis, press **[J]**.

a. Select the span to analyze; select point X; or, toggle with the right arrow key to select an increment output when using a printer.

b. Press **[SOLVE]** and [**PRINT**] as required.

c. Select each point where results are desired. Change from one span to another as required.

11. Press **[EXIT**] to return to the MAIN MENU.

PLANE FRAME ANALYSIS - Printer required for output.

1. From the MAIN MENU, press **[H]** to select Plane Frame Analysis.

2. Press **[G]** to select options from the Plane Frame Analysis Menu.

a. Enter the number of joints.

b. Enter the number of supports.

c. Enter the number of members.

d. Select printer type, baud rate, and parity.

e. Press **[ENTR]** when the data-entry is complete.

3. Press **[H]** to enter joints and members.

a. Enter joint coordinates for each joint; press **[ENTR]**.

b. Enter support joint number and support condition using the right arrow key. Press **[ENTR]**.

c. Enter member number, start joint number, and start restraint, either fixed or hinged. Enter end joint number and end restraint, either fixed or hinged.

d. Enter properties A, I, E, and density.

e. Press **[ENTR]**.

4. Coordinates, supports, and members may be edited by using the **[J]** key.

5. Optional : Enter first load case; edit, if necessary; or, press **[K]** to assemble matrixes.

6. Enter loadings by pressing the **[I]** key. The load case definition is a label only.

7. Press **[K]** to assemble the matrixes.

8. Press **[L]** to start the analysis.

CONCRETE COLUMN ANALYSIS.

1. Press **[J]** to select Concrete Column Analysis.

2. Press **[G]** to initialize the section.

a. Select the phi option.

b. Select the rotation increment angle.

c. Input the problem title.

d. Select the printer options.

e. Press **[ENTR]**.

3. Select printer output if desired : yes or no.

4. Press **[H]** to define the concrete areas.

a. Enter the values requested.

b. Enter coordinates defining the concrete areas. Press **[ENTR]**.

5. Press **[I]** to define provided steel reinforcing.

6. Press **[J]** to define loading. Compression is entered as positive. Tension is entered as negative.

7. Press **[K]** to solve the problem.

THE STEEL SOLVER

1. Press **[I]** to select the Steel Solver.

2. Press **[G]** to select options.

a. Select **ASD** or **LRFD**.

b. Select yield stress.

c. Select printer options.

d. Press **[EXIT]**.

3. Press **[H]** to define the section properties.

4. Press **[PRINT]** to print the section properties.

5. Press **[I]** for axial load only. Press **[SOLVE] [PRINT]**.

6. Press **[J]** for bending solution only. Press **[SOLVE] [PRINT]**.

7. Press **[K]** for a combined solution. Press **[CONT] [PRINT]**.

8. Press **[EXIT]** to return to the MAIN MENU or for a new problem.

TDS

Tripod Data Systems, Inc.

1853 SW Airport Road
P.O. Box 947
Corvallis, Oregon 97339-0947
503/753-9322, 1-800-426-8026