

REPORT NO.
CE/299-561

STRUCTURAL ENGINEERING AND
STRUCTURAL MECHANICS

ULARC

COMPUTER PROGRAM FOR SMALL
DISPLACEMENT ELASTO PLASTIC
ANALYSIS OF PLANE STEEL AND
REINFORCED CONCRETE FRAMES

by
ARUN SUDHAKAR

A Computer Program
Distributed by
National Information Service for
Earthquake Engineering

DECEMBER 1972

DEPARTMENT OF CIVIL ENGINEERING
UNIVERSITY OF CALIFORNIA
BERKELEY, CALIFORNIA

STRUCTURAL ENGINEERING

AND

STRUCTURAL MECHANICS

GRADUATE STUDENT REPORT

Computer Program for Small Displacement
Elasto Plastic Analysis of Plane Steel
and Reinforced Concrete Frames

by

ARUN SUDHAKAR

#561

Quarter Fall Year 1972

R. K. Klein

John C. Paulay 12/14/72
Supervisor

University of California
Berkeley, California

INTRODUCTION

An ideal way of designing structures is by "Limit State" approach.^(1,8) The method examines the structural behaviour under all possible load conditions from service load to ultimate load. This approach incorporates the three important conditions that govern design solutions: (a) adequate strength to sustain ultimate loads (b) adequate ductility at all critical sections and (c) adequate serviceability at working loads. Ultimate strength design, working stress design are particular cases of this limit state approach. Where safety is the controlling factor in design, ultimate strength is more rational method of designing concrete structures than working stress design. One problem, however, is to determine the collapse load at which the structure develops into a mechanism. This is often tedious and time consuming when computations are done by hand. With the advent of computers, however, such tedious computations can be avoided and collapse loads can be computed quickly and easily. In working stress design, since serviceability is the governing factor, the designer is interested in the structural behaviour at service loads which are stipulated in the ACI code.

Giving due thought to these design problems the objective was to write a program for collapse load analysis of plane steel and concrete frames.

This report elaborates on the above-mentioned problems and discusses the application assumptions, limitations and applicability of the computer program "ULARC" for the small displacements Elasto-Plastic analysis of plane frames.

The program computes the node displacements, member forces, support reactions, plastic hinge rotations and rigid-plastic collapse loads for plane frames of arbitrary shape, subjected to static joint loads and support settlements the program gives results at the end of each cycle of loading.

To demonstrate the use of the program and its effectiveness in design of concrete and steel structures, three examples have been given.

Suggestions are made for further work to extend the applicability of the program in the design of concrete structures.

DESIGN OF STRUCTURES:

2.

For many years, the basis for structural design has been the allowable stress concept. The design stress was taken to be some fraction of the ultimate stress in case of concrete, and yield stress in case of steel, depending on the factor of safety used. The analysis methods used were always linear elastic. However, the important consideration in an engineering structure is whether the structure will carry the intended loads or perform its intended function. Moreover, assuming that the stress in the structure should never exceed the elastic limit is not really reasonable. In fact, in most structures, local plastic flow will occur at points of discontinuity in the geometry and at stress raisers, and further residual stresses may already be present in some parts of the structure even before the load is applied.

In these design procedures, no attempt is made to determine the stresses and strains in the structure, but rather, what is sought is the load carrying capacity or limiting load at which the structure will collapse. This type of analysis is called collapse load analysis and the load at collapse is called the ultimate load.

For many design problems where safety is the governing

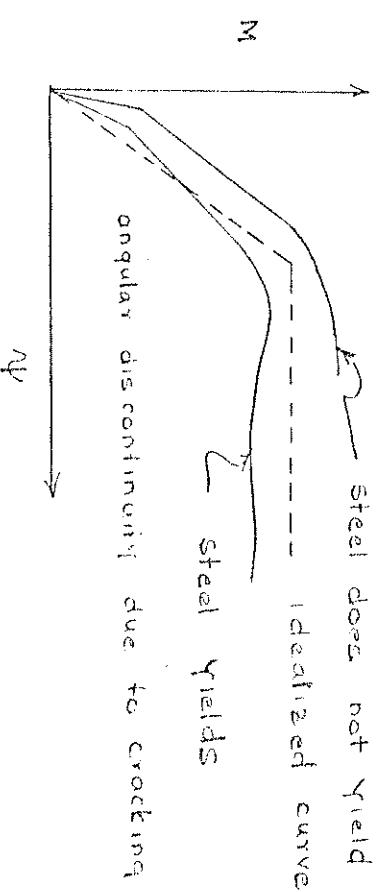
factor, this type of approach makes more sense than a design based on elastic analysis. However, problems of buckling, fatigue and fracture may not always be resolved on the basis of simple ultimate load design and a more involved analysis may need to be carried out.

Ultimate load design recognizes the possibilities of redistribution of forces and moments within a structure subject to high loads, due to inelastic behaviour of the component members. It takes into account the fact that due to the inelastic behaviour of steel and concrete the stresses in the steel and concrete are not proportional to the applied moments and forces at a section, and that a section does not fail when the stress in either the steel or the concrete alone reaches its yield stress or ultimate stress respectively.

Designing structures, assuming linear elastic behaviour, to resist the worst combinations of moments and forces existing at a section sometimes leads to over design. Recognition of the possibility of redistribution of moments and forces within a structure at ultimate load, through the use of ultimate load analysis procedures, enables this over design to be reduced, leading to greater economy.

A reasonably clear understanding of inelastic behaviour of concrete structures is essential for designing structures using ultimate load design procedures.

The moment-average curvature relationship for a concrete structure, depending on the type and percentage of steel, is shown in the figure below. The limiting moment at which the curvature increases rapidly is usually referred to yield moment.



Redistribution of moments occurs in the structure due to reduction in stiffness of a member when the bending moment at any critical section reaches the yield moment of the section. Some redistribution of moments may occur before yield of the reinforcing steel. This is because the moment curvature relationship is not linear up to the yield moment (as seen in the figure) because of tension cracking of the concrete and the curvilinear shape of the concrete stress-strain curve.

In steel because of its large yield strain capacity, large inelastic rotations are feasible. In concrete, however, the inelastic rotations are limited to a low value. For moment rotation relationships to be satisfied, the inelastic rotations at critical sections should not exceed the rotational capacities at those sections.

In actual design practice, the moment curvature diagram is taken as an idealized elasto-plastic relation consisting of two straight lines (shown by dotted lines in the figure). Structures designed by ultimate load method should meet the following requirements:

- a) Limit Equilibrium: the structure should resist any arrangements of the loads up to the specified ultimate load.
 - b) The inelastic rotations should not exceed the rotational capacities of the region.
 - c) The behaviour of the structure under service loads should be satisfactory in regard to the magnitude of deflections, crack widths, maximum stresses and safety against yield of critical sections. That is, the structure should be serviceable at working loads.
 - d) The total cost of the structure should be minimized.
- In addition, the design should include:

e) Determination of the amount of inelastic behaviour to be expected (i.e. computation of "Ductility demand" on the structure)

f) Selection of material properties detailing procedures, and construction practices to ensure that the ductility of the final structure exceeds the ductility demand.

"Ductility demand" is usually related to such measures as displacements, hinge rotations and curvature, although the most fundamental measure is probably the total strain in the material.

For simple structures, displacement provides a suitable measure of ductility demand. This is usually expressed as a "ductility ratio", which is the ratio of maximum displacement to displacement at yield. In reinforced concrete, structures curvature is a suitable measure of ductility demands, conveniently expressed as a "Ductility ratio" obtained by dividing the maximum curvature by the yield curvature.

Curvatures can be computed directly when methods of analysis based on "Distributed plasticity" (explained later) are used. In Elasto-plastic analysis based on lumped plasticity assumptions, however, the plastic deformations are obtained as hinge rotations. Hence, an important problem in analyses

at this type is to relate the plastic hinge rotations to curvature to obtain a meaningful measure of ductility demands. In lumped plasticity analyses, the plastic deformation is concentrated at the hinges (that is, plastic deformations are distributed over zero length). This infinite curvature which is obviously not correct: Analyses based on distributed plasticity overcomes this difficulty. However, the methods are complex and more time consuming computationally.

Another way to get around this problem is to assume that the inelastic curvature γ_{ip}^{inel} occurs over a length of member l_p , sometimes referred to as the effective hinging length. An empirical expression by Baker (21) gives the value of l_p as $l_p = k_1 k_2 k_3 (\bar{z}/d)^4 d$

where \bar{z} = distance from hinging section to point of contraflexure.

d = effective depth of cracked section, or total depth of an uncracked section.

$k_1 = 0.70$ for mild steel, 0.9 /cold worked steel.

$$k_2 = (1 + 0.5 P/P_u)$$

P = Actual axial load at ultimate

P_u = Axial load capacity of section

k_3 = Varies linearly from 0.6 when $f_c' = 5000$ psi to 0.9 when $f_c' = 1600$ psi.

This expression was developed to ensure that the inelastic rotation θ_p as computed from the equation $\theta_p = \gamma \psi_p l_p$ would be within safe limits.

However, Mattock (22) proposed new expressions for computing the inelastic rotations. These were based on moment rotation tests by Mattock and Corley (23) at the P.C.R. laboratories. Based on these tests the following expression was used to compute the inelastic rotations.

$$I_p = 0.5d + 0.05z$$

d - effective depth of the cracked section or total depth of an unworked section.

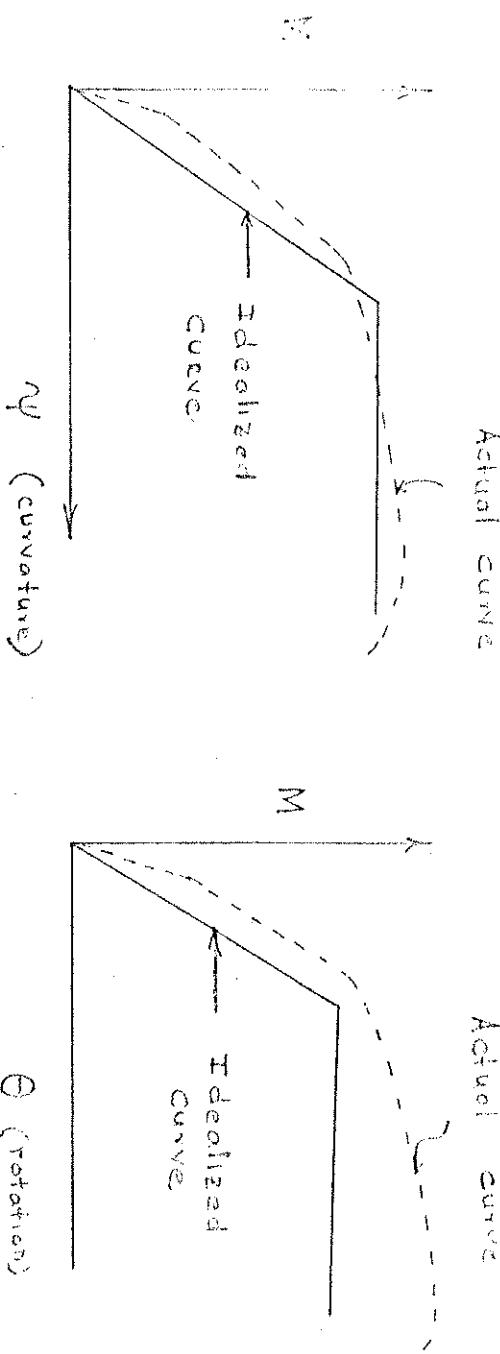
z - distance from hinging section to point of contra-flexure.

This expression gives values close to those given by Baker's expression when $K_1 K_2 K_3 = 0.7$.

These expressions were used to compute the inelastic rotations occurring at ultimate strength for all beams in the P.C.R. tests. The computed inelastic rotations were compared with the measured inelastic rotations. The expressions for computing the inelastic rotations gave a conservative estimate of the total inelastic rotations. For details of the discussion of the test results see reference 19.

IDERALIZATION OF PLASTIC BEHAVIOUR:

For simplicity, the moment curvature relationship is assumed to consist of two straight lines as shown below:

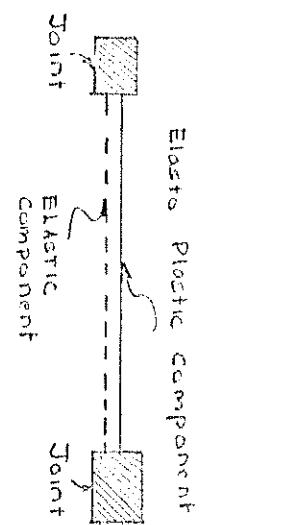


The simplest yield assumption for a frame member is that an ideal plastic hinge is produced when the bending moment reaches the yield moment. The plastic deformation is, therefore, assumed to be concentrated at a point. This is the lumped plasticity assumption, which corresponds to an assumption of elastic, perfectly plastic material and a flexural shape factor of 1.00. In actual practice, however, the shape factor will be greater than 1.0 for strain hardening materials which corresponds to plastic deformation distributed over a finite length, of the member analysis procedures based on distributed plasticity.

through more complex and time consuming, will yield more meaningful information about the ductibility demands on the structure.

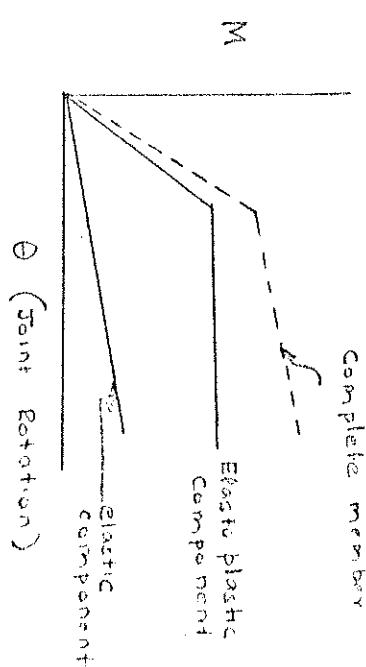
STRUCTURE IDEALIZATION:

A two component model is assumed for each member. This means that each member is actually two component members in parallel; one completely elastic and the other elastic, perfectly plastic as shown below. Plastic hinges may form in the elasto-plastic component. The elastic member does not yield. Thus, a plastic hinge is modeled by an elastic member which remains elastic throughout the analysis and the elasto-plastic member with a hinge at that plastic moment section.

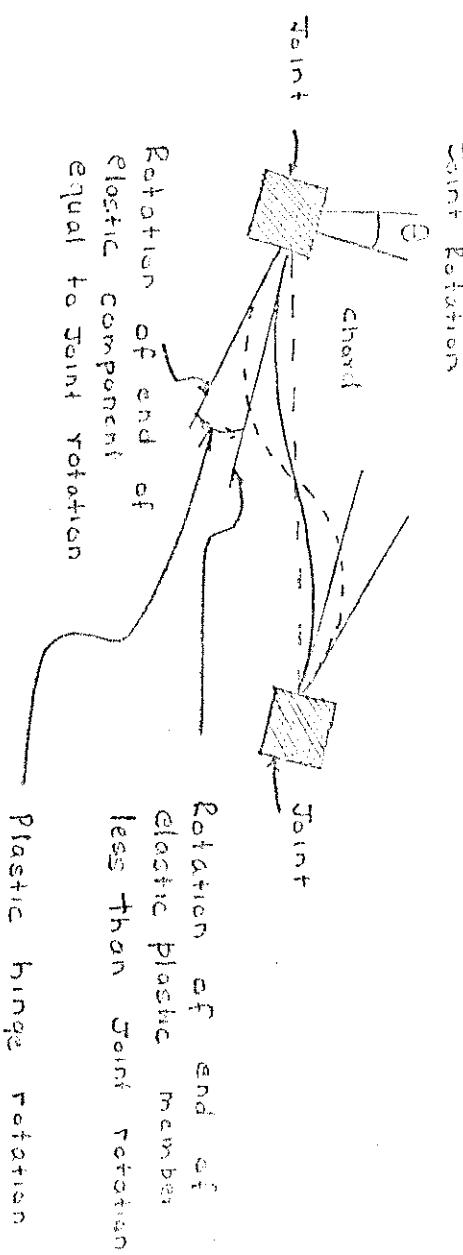


TWO COMPONENT MODEL

The plastic deformation is assumed to be concentrated in the plastic hinges. This type of model is very convenient computationally for two main reasons. First, the stiffness of an element changes suddenly rather than progressively, is very simple to compute and remains constant until a new hinge forms or an old one unloads. Second, it is simple to compute plastic hinge rotations, and hence to obtain a



measure of ductility demand on the member. It is also simple to determine when reversal of hinge rotation occurs.

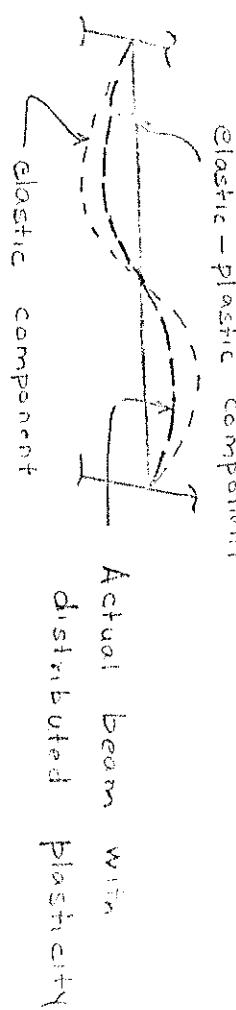


Plastic Hinge Rotation

To compute the curvature ductility ratio for defining the ductility demand one approach is as recommended by Anderson and Bertero (24), the two component model. The curvature used to compute the elastic moment at a point in the curvature of the elastic component of the two component model. Thus, in the two component model, the ductility ratio computed in the elastic component divided by the yield curvature. However, this may be questionable since in distributed plasticity the actual curvature in a member may be larger than in the elastic component of the two

component model. However, the actual curvature ductility demand can be estimated from the model computed average ductility. For details see Anderson and Bertero, Reference

24.



In a statically indeterminate structure, the stiffness of a member at any particular section will be considerably reduced when the bending moment at that section reaches the yield moment of the section. This reduction in stiffness results in a re-distribution of the bending moment in the structure. This is easily modeled in the elasto-plastic method by changing the stiffness of the member which yields. Thus, the structure stiffness is modified whenever a plastic hinge forms.

Yielding is considered to be concentrated at the points of maximum moment within the yielding regions and the members between the points of yield are assumed to behave elastically. It is assumed that prior to failure of the structure, sufficient plastic hinges will form to make all or part of the structure statically determinate but stable.

The collapse load is reached when the structure forms a mechanism, producing deformation of the structure under constant load.

The program detects collapse when the displacement at any node of the frame exceeds a large arbitrarily chosen value (10000). If a structure forms, a mechanism, the stiffness matrix will be singular. This problem is avoided in the program by introducing of a very flexible elastic member in parallel with each elastic-plastic member. This elastic plastic member in the program has a stiffness of

$$\frac{EI}{L} \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ji} & k_{jj} \end{bmatrix}$$

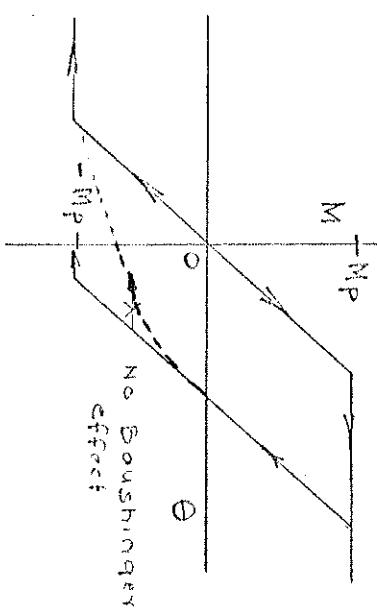
Assumed elastic member

$$10^{-8} \frac{EI}{L} \begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}$$

guarantees no singularity, yet so flexible that does not influence behaviour of frame. It also permits false mechanism to be detected (that is mechanisms which have enough hinges to form mechanism but in which one or more of these hinges must close as the mechanism deforms).

ASSUMPTIONS AND LIMITATIONS:

- (1) Material stress strain relation is idealized as elastic perfectly plastic. The effects of strain hardening are basically neglected but can be included approximately by considering parallel elements, as explained subsequently.
- (2) The Baushinger effect is neglected



- (3) Yield is assumed to occur at one load over the section depth. For most structural I sections this is a good assumption since shape factors are usually about 1.12. However, for concrete shape factors are usually greater and this assumption is a bit on the conservative side.
- (4) Material is assumed to be infinitely ductile.
- (5) Visco-elastic effects such as creep are neglected, this is reasonable assumption for structural steel, but creep may be significant in concrete structures under long time loading.

- (5) The program is restricted to two dimensional structures yielding in flexure only.
- (7) Member properties are assumed to be concentrated along their axes. Joints are represented by points on the member axes. Plastic hinges may form at member ends immediately adjacent to joints.
- (8) Plastic hinge zone spreading are not modelled (i.e. lumped plasticity is assumed)
- (9) P- Δ effects are not included.
- (10) Column interaction effects are neglected.
- (11) Since the program is two-dimensional, the effects of torsion in members due to framing-in beam members or from eccentric loading on the section axes are neglected.
- (12) Column buckling and lateral torsion buckling of beams are neglected.
- (13) In the case of beam-column panel joints, the panel joints are assumed to remain rigid while yield may occur in any of the members framing into the joint. If the joint is not made sufficiently strong and stiff to ensure this, joint deformations may greatly change the behaviour of the structure.
- (14) Shear and axial forces, bond effects and instability phenomena are not critical.

(15) Redistribution of moments due to tension cracking which results in changes in relative stiffness between various parts of the structure, are neglected.

(16) Continuity of the structure at ultimate load for concrete structures may or may not be preserved since the material is assumed to have infinite ductility.

(17) Strength and stiffness degradation is not considered. This may become important in reinforced concrete members when subjected to several cycles of reversed loading beyond yield. In the present analysis, however, a stable hysteresis loop is assumed.

SYNOPSIS OF THE PROGRAM:

The program "ULARC" has nine subroutines

- (a) **INPUT**
 - Node coordinates, member properties and member location data are read in this subroutine. Generation of the above-mentioned items is possible. (See INPUT DATA)
- (b) **MATERIAL**
 - Subroutine used in automatic generation of the node coordinates.
- (c) **LOAD**
 - Joint loads, support displacements, are input and appropriate indicators input dependency on whether the analysis is required up to (a) collapse, (b) a specified load factor (c) a limiting displacement (for details see INPUT DATA)
- (d) **BAND C & STIF**
 - These subroutines are used to formulate the stiffness of the structure.
- (e) **OPSOI**
 - Equation solving subroutine.
- (f) **FORCE**
 - Subroutine computes node displacement, member forces, support reactions, hinge rotations, and collapse loads.
- (g) **OUTPUT**
 - Prints the above computed quantities.

STRUCTURE DEFINITION

The frame is defined by a series of nodes (joints) connected by one dimensional members (elements) possessing both flexure & axial stiffness. The nodes must be numbered in sequence. This numbering should preferably be chosen to minimize the largest node number difference within the members. But this is not essential. The members must also be numbered in sequence in any convenient manner.

A coordinate system, X, Y, with axis-Y 90 degrees counter-clockwise from axis-X, is established to define the node locations and load directions.

Supported joints are assumed to be restrained by elastic springs, which may be parallel or inclined to the X, Y axes. A rotational spring stiffness and two translational stiffness along a pair of axes A and B must be specified, where axis B is 90 degrees counter-clockwise from axis A. In addition, the angle between axes X and A must be specified for inclined supports. Realistic spring stiffnesses should be assigned, taking into account the fact that real supports cannot be completely rigid. An essentially rigid support can be obtained by assigning a very large stiffness (e.g. 10^{10} K/in) yielding may take place by the formation of localized plastic hinges at the member ends. The moment curvature

relationship at a plastic hinge is assumed to be rigid perfectly plastic that is (shape factor = 1.0) strain hardening effects may be approximated by placing two or more members of appropriate strengths and stiffness in parallel.

Members of nonuniform stiffness may be considered by specifying end stiffeners and carry over factors. The plastic moments for any member may be different at the two ends of the member and for positive and negative bending.

External loads may be specified to act the nodes only. Supported displacements may be specified at any supported nodes. The displacement is applied not directly at the node, but at the base of the spring, or springs supporting the node. If the support spring stiffeners is very large in comparison with the structure, the displacement at the supported node will essentially equal the specified displacement. However, if the spring stiffness is low, the node displacement may be substantially different from the specified displacement.

METHOD OF NON-LINEAR ANALYSIS:

The load-displacement relationship for the idealized structure is piecewise linear between "events" where an event corresponds to the formation of a new plastic hinge or unloading at an existing hinge. Beginning with the structure in its initial unloaded state, a load is applied and an analysis by the direct stiffness method is carried out. That proportion of the load increment required to produce a new event is determined. The results are then scaled linearly so that the state of the structure corresponds to the occurrence of the event. A new structure stiffness, accounting for any new or unloaded hinges, is then formed, a further load increment is applied and the process is repeated. Within any load case, the sequence terminates when one of the following occurs:

- a) A collapse mechanism is produced.
- b) The full specified loading is applied.
- c) A specified maximum displacement is reached. The analysis then proceeds to the next appropriate load case.

RESTRICTIONS:

Dimension statements limit the program to frames with no more than 50 nodes, 10 supported nodes, 15 different member shapes, 20 different cross-section strengths and 100 members. In addition, the storage occupied by the structure stiffness matrix may not exceed 3000 locations. The capacity, however, can easily be expanded.

INPUT DATA

The following sequence of punched cards defines the problem. Units must be consistent throughout.

1. **NAME (12A6)** - One card.
2. **CONTROL INFORMATION (1215)** - One card.
cols 1 - 5: Number of nodes (max. 50).
- 6-10: Number of "Control" nodes, for which coordinates are specified directly. See Section 3.
- 11 - 15: Number of node coordinate generation commands (no limit). See Section 4.
- 16 - 20: Number of supported nodes (max. 10) See Section 5.
- 21 - 25: Number of different member stiffness types (max. 15)
See Section 6.
- 26 - 30: Number of different cross-section strength types (max. 20). See Section 7.
- 31 - 35: Number of members (max. 100).
- 36 - 40: Number of generation commands for member locations.
See Section 8.
- 41 - 45: Number of generation commands for member stiffnesses.
See Section 9.
- 46 - 50: Number of generation commands for member strengths.
See Section 10.

54 - 55: Number of load cases (no limit). See Section 11.

56 - 60: Maximum number of analysis cycles permitted within any load case. It is possible, although rare, for the solution to get caught in unending cycles of loading and unloading at plastic hinges. To avoid waste of computer time, execution will be terminated if the number of separate analyses for any loading case exceeds this number. A recommended value is approximately 1.5 times the number of plastic hinges required to cause collapse. If in doubt, specify a very large number.

3. CONTROL NODE COORDINATES (15, 2F10.0) - One card for each control node.

cols 1 - 5: Node number, in any sequence.

6 - 15: X coordinate.

16 - 25: Y coordinate.

4. RODD COORDINATE GENERATION (415, F10.0) - One card for each generation command. Omit if there are no generation commands. See Note below for automatic generation feature.

cols 1 - 5: Number of node at beginning of generation line.

6 - 10: Number of node at end of generation line.

11 - 15: Number of nodes to be generated along the line.

16 - 20: Node number difference (constant) between successive generated nodes, and also between the first generated node and the node at the beginning of the generation line.

21 - 50: Distance between successive generated nodes. If greater than or equal to 1.0, assumed to be an actual distance. If less than 1.0, assumed to be a proportion of the distance between the beginning and end nodes. If zero or blank, the nodes are automatically spaced uniformly along the generation line.

Note: Only straight line generation is permitted. The number of nodes generated by each command may be one or any larger number. The coordinates of the two nodes at the beginning and end of the generation line must have been previously defined, by direct specification or by previous straight line generation. If these coordinates have not been defined, a warning message is printed.

It is not necessary to provide generation commands for nodes which are (a) sequentially numbered between the beginning and end nodes of any straight line, and (b) equally spaced along that line. After all generation commands have been executed,

the coordinates for each group of unspecified nodes are automatically generated assuming sequential numbering and equal spacing along lines joining the specified nodes immediately preceding and following the group.

5. SUPPORTED NODES (15, 4E10.0) - One card for each supported node
- Cols 1 - 5: Node number, in any sequence.
- 6 - 15: Translational support stiffness along Z axis.
- 16 - 25: Translational support stiffness along B axis.
- 26 - 35: Rotational support stiffness.
- 36 - 45: Angle (degrees) between X axis and A axis, measured counter-clockwise from the X axis.
- Note: Any of the support stiffnesses may be zero if desired.
6. MEMBER STIFFNESS TYPES (15, 6F10.0, 215) - One card for each different type. See Note below.
- Cols 1 - 5: Stiffness type number.
- 6 - 15: Average cross sectional area.
- 16 - 25: Reference value of flexural moment of inertia.
- 26 - 35: Young's modulus of elasticity.
- 36 - 45: Stiffness factor K_{ii}.
- 46 - 55: Stiffness factor K_{jj}.
- 56 - 65: Stiffness factor K_{ij}.
- 66 - 70: Cross section strength type at end i.

66 - 70: optional. If left blank, no specific cross section strength is associated with this member stiffness type, and a section strength must be specified in section 10. If the number of a cross section strength type is punched, the specification of member stiffnesses in section 9 serves also to define the cross section strength, and this need not be specified separately.

71 - 75: cross section strength type at end j. This is optional, as for end i.

Note: The flexural stiffness matrix of any member is of the form

$$\frac{EI_r}{L} \begin{bmatrix} K_{ii} & K_{ij} \\ K_{ji} & K_{jj} \end{bmatrix}$$

in which E = Young's modulus; I_r = reference moment of inertia;

L = member length; and K_{ii} , K_{jj} are stiffness factors.

For a member of uniform stiffness, $K_{ii} = K_{jj} = 4.0$, $K_{ij} = 2.0$.

For a uniform member with a real hinge at end j, $K_{ii} = 3.0$,

$K_{ij} = K_{ji} = 0$.

For tapered members or members in which shear deformations are important, appropriate stiffness coefficients must be determined. It should be noted that the collapse load of a rigid-plastic frame depends only on the member strengths, and not on their stiffness properties.

7. CROSS SECTION STRENGTH TYPES (15, 4F10.0) - One card for each different cross section strength, defined by the ultimate moment capacity of the section.

Cols 1 - 5: Strength type number.

6 - 15: Ultimate moment for clockwise moments acting on the member end.

16 - 25: Ultimate moment for counter-clockwise moments acting on the member end.

26 - 35: Limiting plastic hinge rotation capacity for clockwise moments acting on the member end (radians).

See Note below.

36 - 45: Limiting plastic hinge rotation capacity for counter-clockwise moments acting on the member end (radians).

Note: Specifications of limiting hinge rotation capacities is optional. If hinge rotations are not of concern, leave blank or punch a large number. If the specified rotation at any hinge is exceeded, the rotation is marked by an asterisk in the computer printout. This helps to identify sections at which excessive plastic deformations are taking place.

8. MEMBER LOCATION GENERATION (515) - One card for each generation command.

cols

1 - 5: Member number, or number of first member in a sequentially numbered series of members to be generated by this command.

6 - 10: Number of node at one end of this member (node i).

11 - 15: Number of node at the other end of member (node j).

16 - 20: Number of last member in series. If this command covers only a single member, leave blank.

21 - 25: Node number difference. For each successive member in the series, the numbers of nodes i and j are obtained by adding this number to those for the previous member. If this command covers only a single member, leave blank.

9. MEMBER STIFFNESS GENERATION (415) - One card for each generation command. Note that if cross section strength types have been associated with member stiffness types (Section 6), these commands may also serve to generate some or all of the member strengths.

cols 1 - 5: Member, number, or number of first member in a series of members with identical stiffnesses. The members in the series must be regularly, but not necessarily sequentially numbered.

6 - 10: Stiffness type number. The stiffness coefficients will correspond to the i,j ends as defined in Section 6.

11 - 15: Number of last member in series. If this command covers only a single member, leave blank.

16 - 20: Member number difference, n. If the first member in the series is m, the members in the series are m, m + n, m + 2n, etc. If this command covers only a single member, leave blank.

10. MEMBER STRENGTH GENERATION (515) - One card for each generation command. This number may be zero if all strengths have been defined with the member stiffnesses in section 9. The strength generation commands must be used to specify strengths not previously specified by Section 9. The commands may also be used, if desired, to change any strengths previously specified by Section 9.

cols 1 - 5: Member number, or number of first member in a series of members with identical strengths.

6 - 10: Strength type for cross section at end of member.

11 - 15: Strength type for cross section at ends of member.

16 - 20: Number of last member in series. This command covers only a single member, leave blank.

21 - 25: Member number difference, n, as defined in Section 9.

11. LOADINGS - One set of cards, as follows, for each load case.

11a. Control Information (315, 2F10.0, 45A1) - One card.

cols 1 - 5: Number of generation commands for applied external loads on nodes. See Section 11b.

6 - 10: Number of generation commands for nodes with specified support displacements.

See Section 11c.

11 - 15: Load application code. If zero or blank, the loading is to be added to the load acting at the end of the previous load case. If 1, the structure is reinitialized to an unloaded state, and the loading is then applied.

16 - 25: Load factor. The loads specified in

sections lib and lic are scaled by this factor to define the loads to be applied to the structure. For example, if working loads are specified in

Sections lib and lic, then the load factor might be a typical design load factor. If load is to be applied until the frame collapses, enter a large number (e.g. 100).

26 - 35: Limiting displacement. If the absolute value of horizontal or vertical displacement at any node exceeds this value, the results are scaled to reduce the displacement to this value and the load case is then terminated. A limiting displacement may be specified if the limit of usefulness of the structure is reached prior to complete collapse, or if cyclic loading of the structure between specified displacement limits is to be applied. If the analysis is to be continued to collapse, leave blank.

36 - 30: Load case title, to be printed with output.

Note: If the limiting displacement is reached, the program moves to the next load case. If collapse occurs, the program moves to the next load case for which the load application code is equal to 1.

LIB. Nodal Load Generation (15, 3F10.0,215) - One card for each load generation command.

Cols 1 - 5: Node number, or number of first node in a series of nodes to which identical loads are to be added.

6 - 15: Applied external load acting in direction of X axis.

16 - 25: Applied external load acting in direction of Y axis.

26 - 35: Applied external moment, clockwise positive.

36 - 40: Number of last node in series. If this command covers only a single node, leave blank.

41 - 45: Node number difference, n, as defined in Section 9.

Note: Any node may appear in several generation commands. The total loads applied at such nodes will be the sum of those specified in the different commands.

11c. Support Displacement Generation (25, 3210.0, 215) - One card for each generation command.

cols 1 - 5: Node number, or number of first node in a

series of nodes to which identical support displacements are to be added.

6 - 15: Support base displacement in direction of x axis.

16 - 25: Support base displacement in direction of y axis.

26 - 35: Support base rotation (degrees, clockwise positive).

36 - 40: Number of last node in series. If this command covers only a single node, leave blank.

41 - 45: Node number difference, n, as defined in Section 9.

Note: Any node may appear in several generation commands. The total displacements applied at such nodes will be the sum of those specified in the different commands.

12. NEXT PROBLEM

The data for a new problem, starting at Section 1, may be added if desired. Any number of problems may be solved in a single computer run, provided there are no fatal errors

present in the data. Any such error will lead to termination of execution within the problem for which it occurs. Add two blank cards after the last problem.

Applications:

(1) Plane frames of arbitrary shape subjected to static joint loads and support settlements (can be in inclined), can be analyzed since $P\Delta$ effects are not considered, the program is applicable to low-rise frames of steel or reinforced concrete. The results of the analysis can be used effectively in the design of structures as will be shown later

(2) Members of non-uniform stiffness can be considered by specifying end stiffness and carry over factors.

$$\text{Prismatic member } \frac{EI}{L} \begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix} \quad \text{Fixed ends}$$

$$\text{Prismatic member } \frac{EI}{L} \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix} \quad \text{Hinged ends}$$

$$\text{Prismatic member } \frac{EI}{L} \begin{bmatrix} 0 & 0 \\ 0 & 3 \end{bmatrix} \quad \begin{array}{l} \text{One end fixed} \\ \text{Other hinged} \end{array}$$

$$\text{Variable cross-section } \frac{EI}{L} \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ji} & k_{jj} \end{bmatrix}$$

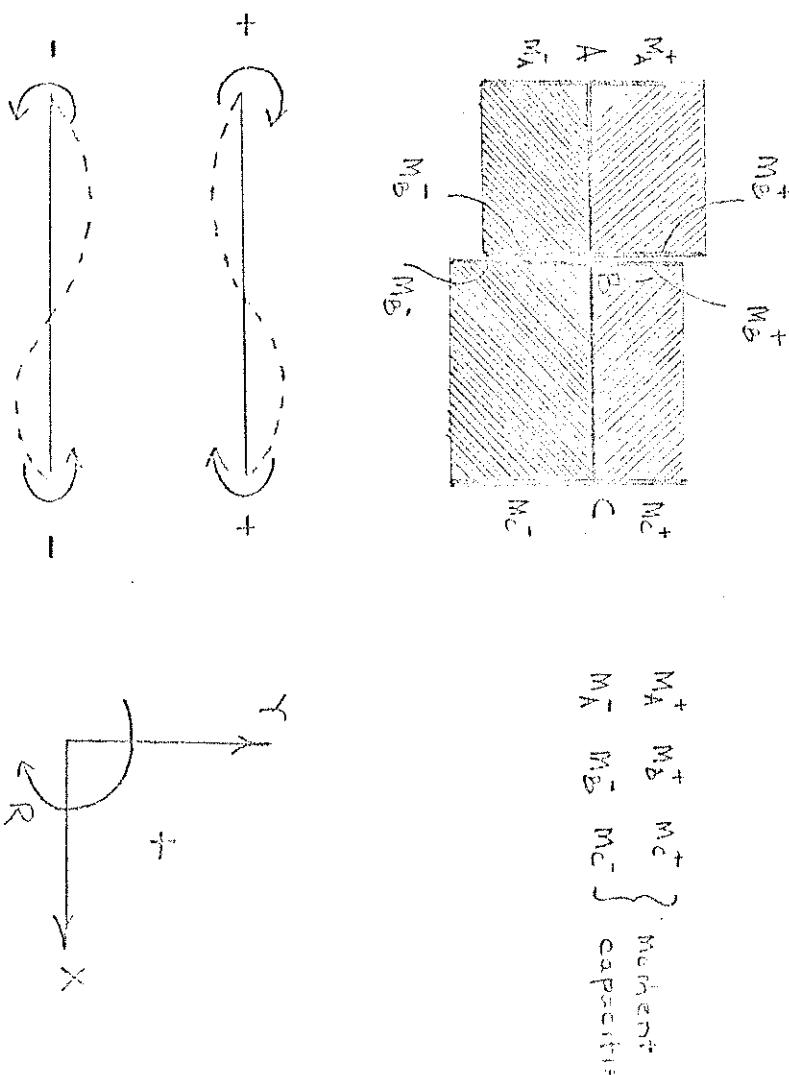
Note SEE INPUT DATA

Where

K_{ii} , K_{ij} , K_{ji} , stiffness factors are input. (See input data)

A small elastic member parallel to the main member having a very low stiffness $\frac{EI}{L} \times 10^{-8} \begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}$ has been added to prevent singularity of the structure stiffness and thus avoid the instability caused by the singular matrix.

- (3) The moment capacities for any member may be different at the both ends of the member and for positive and negative bending

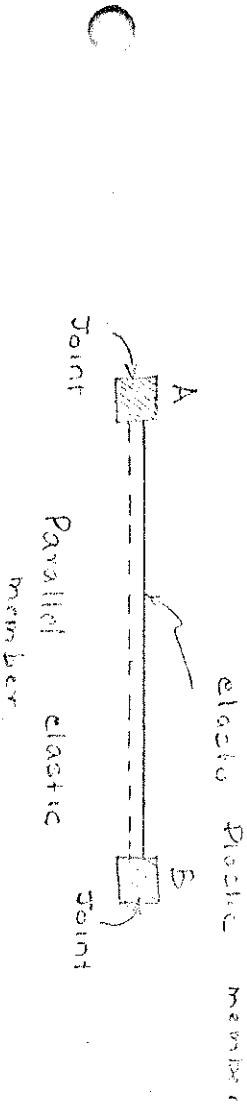


(4) Several distinct load patterns may be applied in succession up to specified load factors. Non-proportional loading including revised loading is permitted. This is specially useful in practice where for example the gravity load is to be kept fixed and lateral load increased till collapse occurs or it reaches a specified load factor. This can be easily done by two load cases. In the first load case, gravity loads only are applied and a load factor of 1.00 is specified with loading code as zero. In the second load case, lateral loads are applied and with loading code still zero, the load factor is applied up to which the analysis is to be done. A limiting displacement can be handled similarly.

The structure can be loaded from one side laterally and an analysis carried out to a limiting displacement or specified load factor, and in the next load case the loading may be reversed. In this case hinges which were formed during the first loading case will generally unload. Uniform loading must be idealized as several joint loads at equal intervals. The interval can be reduced for greater accuracy. Because of the generation option available this can be done easily by changing the increment number on the first joint.

(5) Since non-proportional loading is admissible and since the program keeps track of incremental and total plastic hinge rotations at the end of each load cycle, studies can be made of cyclic loading problems where shakedown incremental collapse, or alternating plasticity are involved.

(6) Strain hardening of the material can be easily introduced by using an elastic parallel element in the two component model of the structure.



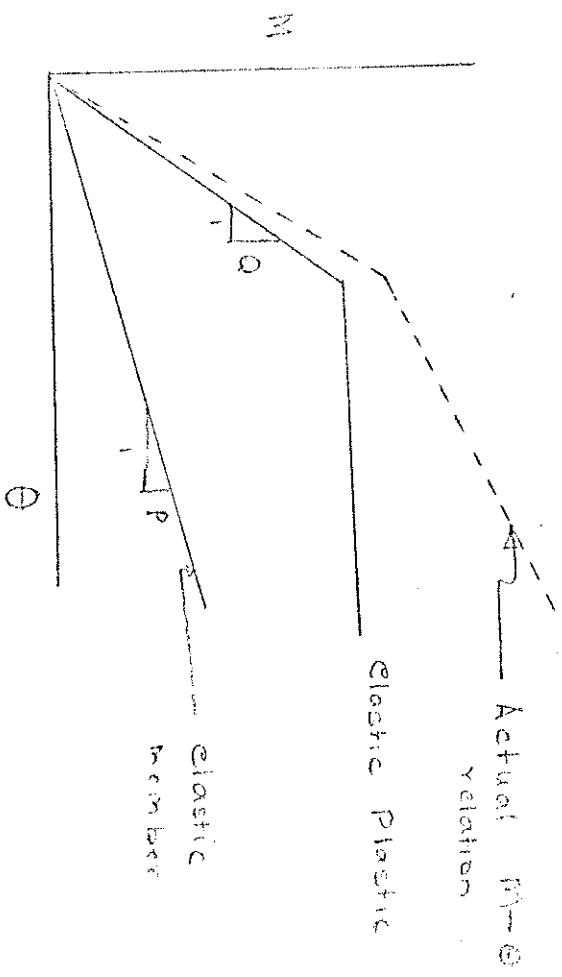
The way this elastic member will be introduced is by specifying this as extra member in parallel, that is, connected between the same joints. The moment capacity of this member at both ends will be high enough relative to that of the other member having the same location so that it does not yield at the end of the analysis; i.e. remains elastic. The stiffness factors would be as follows:

$$\frac{EI}{L} \begin{bmatrix} 4(p) & 2(p) \\ 2(p) & 4(p) \end{bmatrix} \quad p \text{ is \% strain hardening}$$

The stiffness of the other elasto plastic member will be

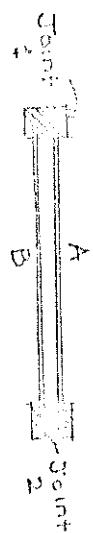
$$\frac{EI}{L} \begin{bmatrix} k_{11} @ k_{12} @ \\ @ Q = (1-p) \\ k_{21} @ k_{22} @ \end{bmatrix}$$

The effect of this would approximate the moment rotation curve of a strain hardened material as shown.



Strain Hardening

The above method of introducing strain hardening in the member is best explained by an example.



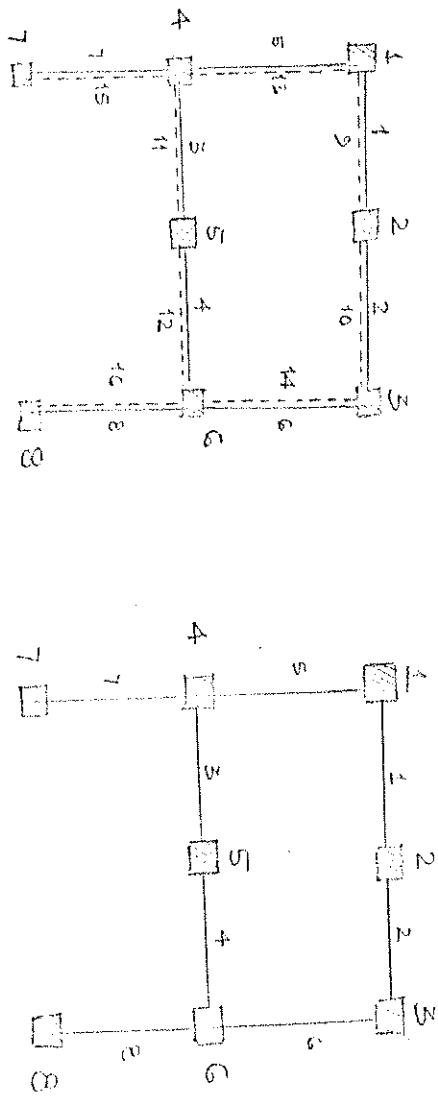
Normally, one would specify member (A) between joints.

To introduce strain hardening an extra member (B) is specified between the same joints that is in member location

generation there would be 2 members with the same connectivity.

In other words, between any two joints where originally

there was only one member, there would now be two members.



STRAIN HARDENING
TOTAL MEMBERS = 16
JOINTS = 8

NO STRAIN HARDENING
TOTAL MEMBERS = 6
JOINTS = 6

The dotted members are the extra members in parallel to the original members. The stiffness of these members will be

$$\frac{EI}{L} \begin{bmatrix} k_{ijP} & k_{ijP} \\ k_{ijP} & k_{ijP} \end{bmatrix}$$

P = Percentage of strain hardening

Moment capacities of these members should be relatively higher than the other members; that is, if moment capacity of members

(1) to (8) are M_p , the moment capacities of members (9) to

(16) should be about $1.5 M_p$. (Note 1.5 is an arbitrary

figure chosen, actually any factor can be taken such that the moment capacities of members (9) to (16) do not exceed during

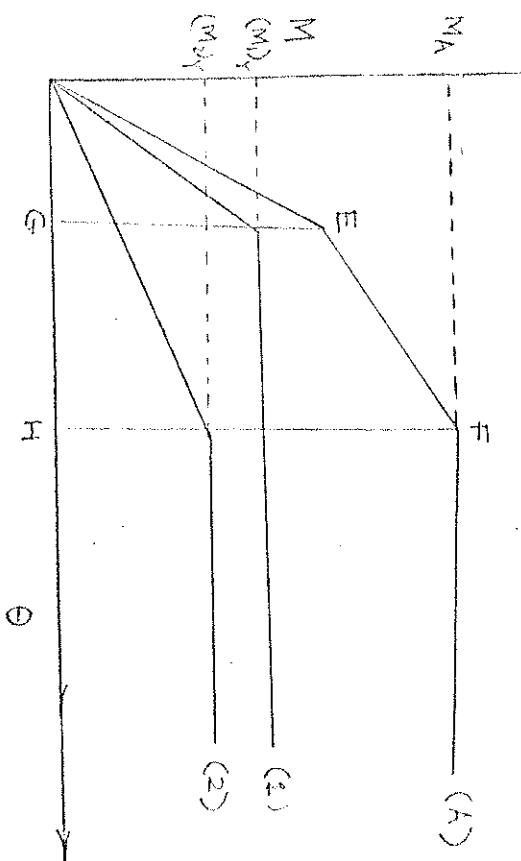
the analysis; that is, they do not yield.)

The flexure stiffness of members (1) to (8) will be

$$\frac{EI}{L} \begin{bmatrix} k_{ijQ} & k_{ijQ} \\ k_{ijQ} & k_{ijQ} \end{bmatrix} Q = (1 - P)$$

Stiffness factors k_{ijQ} , k_{ijQ} , k_{ijQ} are input depending on the type of cross-section of the member and its end conditions (See input data).

Further suppose one was interested in modeling a structure having the following moment-rotation curve shown as (A)



This could be modelled as two members (1) and (2) in parallel having a certain moment capacities and stiffness type. These (see input data) can be easily found knowing that moment on Curve A at any instant should be moment of curve (1) plus moment of curve (2); that is

$$M_A = M_1 + M_2$$

Before the discontinuity at E the moment will be

$$M_A = (M_1)_Y e^{\tau} (M_2)_Y \quad (\text{both elastic})$$

Between E and F the moment

$$M_A = (M_2)_Y + (M_2)_Y \quad (1) \text{ plastic; } (2) \text{ elastic}$$

Beyond F the moment will be

$$M_A = (M_1)_Y + (M_2)_Y \quad (\text{both plastic})$$

The stiffness of the member should be such that member (1) remains elastic up to and plastic beyond. Member (2) must remain elastic up to 'H' and plastic beyond that point. Thus, with this in mind, the stiffness and moment capacities should be selected to satisfy the above requirement. Since a curve has two discontinuities this can probably be done by fit and trial with hand computation. However, in/complicated curve where there may be four or five discontinuities this would become cumbersome. However, this can be avoided by writing a separate short computer program to satisfy the requirements and come out with a particular stiffness type and moment capacities of the members.

APPLICATION TO ANALYSIS OF ULTIMATE LOAD DESIGN.

The design requirements and the problems associated with design were indicated in the earlier discussion. Here it will be shown how the program results can be used in design.

The program computes node, displacements_{member}, member support reactions, plastic hinge rotations and plastic collapse loads and prints them after each cycle of loading each time an increment of load is applied to form a plastic hinge or reach a specified load factor or a limit displacement, constitutes a cycle. Since the results are known for each cycle of loading, a plot of story shear vs displacement can be made. From this plot, the displacement at working load can be computed. This can give a good indication of the serviceability of the structure. Another way to investigate the behaviour of the structure at working load is to specify load factors, so that the program can give the results at those loads.

In design, it has to be insured that the "Ductile capacity" exceeds the "Ductile demand" of the structure. The problem is to compute the 'Ductile demand' of the structure. As mentioned earlier, there are several measures for ductility

demand. For reinforced concrete structures curvature is a suitable measure of ductility demand. However, as noted, the program gives plastic hinge rotations, and the question is how to relate their hinge rotations in lumped plasticity analysis to curvatures to give a more meaningful measure of ductility demand. The approach recommended by Anderson and Bertero (24), for a two-component, strain hardening model, is to compute a "curvature ductility ratio" as the ratio of the curvature in the elastic component divided by the yield curvature. The method has been indicated earlier. A detailed discussion of the method will be found in reference (20).

Another empirical approach is to assume an effective hinging length, 'l_p', over which the inelastic curvature extends. Then the inelastic rotation $\epsilon_p = \psi_p l_p$

$$\text{Thus } \psi_p = \frac{\epsilon_p}{l_p}$$

As indicated before l_p is given by the empirical expressions by Baker (21) and Mattock (22). Now for this approach is applicable to lumped plasticity analysis is still questionable.

In steel structures, however, hinge rotation may be used in preference to curvatures, primarily, because of convenience.

Since it has been assumed that the material is infinitely ductile, the program does not stop when the inelastic plastic rotations exceed the rotation capacities of the section. Instead three asterisks (***) are printed alongside the plastic hinge rotations indicating the rotation capacities of the section has been exceeded. In actual design practice in such cases, the design of the cross-section concerned is revised by altering the reinforced steel at that section. With the

section changed the analysis can be repeated using the program and again checked to see that the inelastic rotation does not exceed the rotation capacity. At the end of the analysis the distribution of moments can be compared with the assumed distribution at the start of the design to get the sections of the members.

Permissible deflections of the structure according to the code can easily be checked. From the results of the program, however, the requirements of crack width cannot be checked.

BARRIER WORK

- (1) Plastic deformation of panel joints may be significant and this should be considered. This may be accounted for by adding an elasto-plastic joint from which joint deformations may be estimated.
- (2) Axial yield has not been considered and further investigation is needed to incorporate it. This will permit the analysis of cable stayed structures.
- (3) Large displacements effects ($P-\Delta$) should be considered and column interaction effects incorporated.
- (4) The program does not formally compute the "curvature ductility" using Anderson and Bertero's method. This could be computed.
- (5) The only way at present to deal with a uniformly distributed load is to treat it as a number of point loads spaced at equal intervals. Some better procedure might be possible.

REFERENCES

15. V. Kazinczy, "Kisérletek betonágyú-tartókkal" (Experiments with clamped girders), Betonszámle, v. 2, p. 63 (1914).
- 2E. Maier-Leibnitz, "Vergleichende Versuchsanwendung", Preliminary Publication, International Association for Bridge and Structural Engineering, 2nd Congress, Berlin (1936).
- 3J. P. Baker, M. R. Horne, and J. Heyman, "The Steel Skeleton", Volume III: Plastic Behaviour and Design, Cambridge University Press (1956).
- 4B. G. Neal, "The Plastic Methods of Structural Analysis", John Wiley & Sons, Inc., New York (1956).
- 5W. Prager & P.G. Hodge Jr., "Theory of Perfectly Plastic Solids", John Wiley and Sons, Inc., New York (1951).
- 6P.S. Symonds, "A Review of Methods for the Plastic Analysis of Rigid Frames of Ductile Metal", DAM Report AII-S6, Brown University, Providence (1950).
- 7P. G. Hodge, Jr., "Plastic Analysis of Structures", McGraw Hill (1959).
- 8L. S. Beedle, "Plastic Design of Steel Frames", John Wiley & Sons, Inc., New York (1958).
- 9R. K. Livesley, "The Application of Computers to Problems Involving Plasticity", Symposium, University of Southampton (1959).
- 10R. K. Livesley, "Matrix Methods of Structural Analysis", New York: Pergamon Press, Macmillan (1954).
- 11R. S. Jenkins, "Matrix Methods in Structural Mechanics", University of Nottingham (1961).
- 12J. N. Argyris & S. Kelsey, "Energy Theorems and Structural Analysis", London: Butterworths Scientific Publications (1960).

REFERENCES (contd)

135. M. Nagyvis, "Recent Advances in Matrix Methods of Structural Analysis," Pergamon Press, Oxford (1964).
142. R. Clough, H. J. Turner, H. C. Martin, and L. S. Popov, "Stiffness and Deflection Analysis of Complex Structures," Journal of Aeronautical Science, v. 23, no. 9 (September 1956), pp. 805-823.
152. R. Clough, "The Finite Element Method in Plane Stress Analysis," Proceedings of the 2nd Conference on Electronic Computations, ASCE (September 1960), pp. 345-378.
162. W. Wheaton, "Rigid Frames-plastic Collapse Analysis," Graduate Student Research Report No. 414, Dept. of Civil Engineering, University of California, Berkeley, California, Spring 1969.
- 17C. Orr, "Blasto-plastic-analysis of plane frames" Graduate student research report (478) Dept. of Civil Engineering, University of California, Berkeley, California.
- 18D. B. Crierson, "Deformational analysis of Elastic-plastic frames," Paper in the Journal of the Structural Division, Proceeding of the ASCE Oct. 1972.
- 19Report of ACI-ACCE, Committee 428 Limit Design
- 20G. H. Powell, "Notes on Inelastic dynamic analyses of tall buildings presented for the earthquake resistant design of engineering structures," June 1972.
- 21Baker A. E. J. "The Inelastic Space Frame" Concrete Publications, London 1967.
- 22Mattock, R. R. "Rotational Capacity of Trussing Sections in Reinforced Concrete Beams," Proc. of the International Symposium on Flexural Mechanics of Reinforced Concrete, Miami, Florida, Nov. 1964, pp. 143-150. ASCE-1965-50 and ACI SP-12.

REFERENCES (Contd)

23Corley, W. G. "Rotational Capacity of Reinforced Concrete Beams", Journal of Structural Division ASCE, Proc. Paper 4939, Vol. 93, ST5, pp. 121-146. Oct. 1965.

24Anderson, J.C. and V. V. Bertero, "Seismic Behavior of Multi-story frames, designed by different Philosophies", Report No FERC 69-11 Earthquake Engineering Research Center, University of California, Berkeley, Oct. 1969.

25"Ultimate Load Design of Concrete Structures", A Report of the Institution of Civil Engineers" Committee on ultimate load design of concrete structures Proc. I.C.E. Vol. 21, 22, 399-442, Feb. 1962, London.

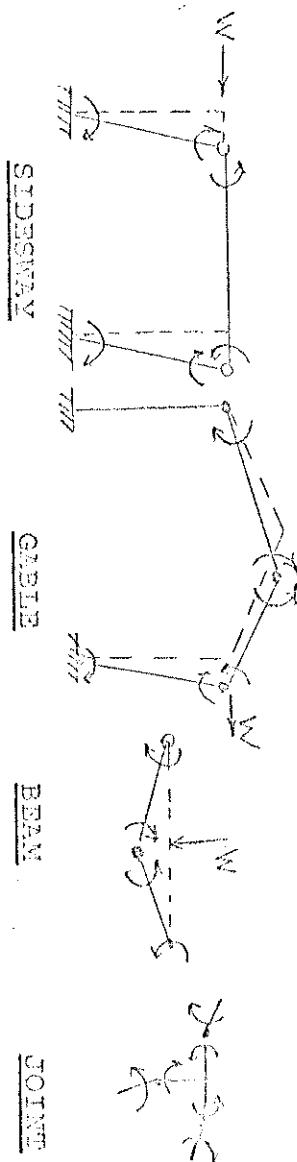
26Baker A.L.L., "The ultimate load theory applied to Design of Reinforced and Prestressed Concrete Frames", Concrete Publications. London 1956.

27Mendelson, A. "Plasticity Theory and Application".

28Sayer, M. A., "Flexure Mechanics of Reinforced Concrete" ACI-ASCE SP-12 1965 pp. 2-28.

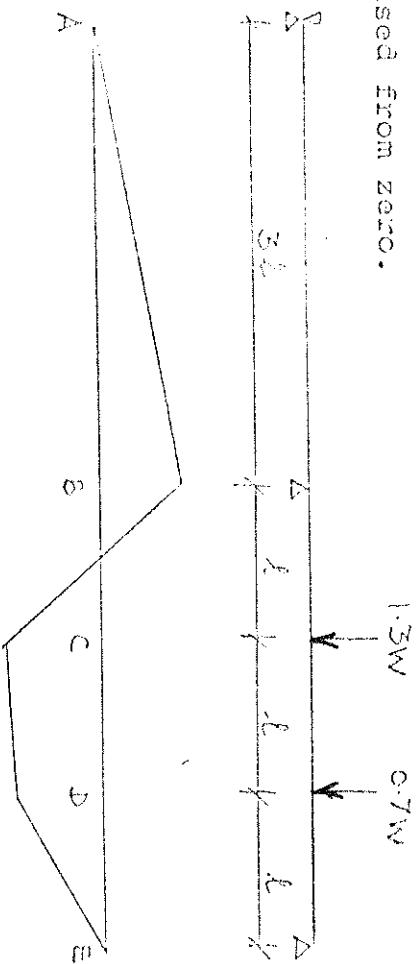
ADMISSIBLE MECHANISMS

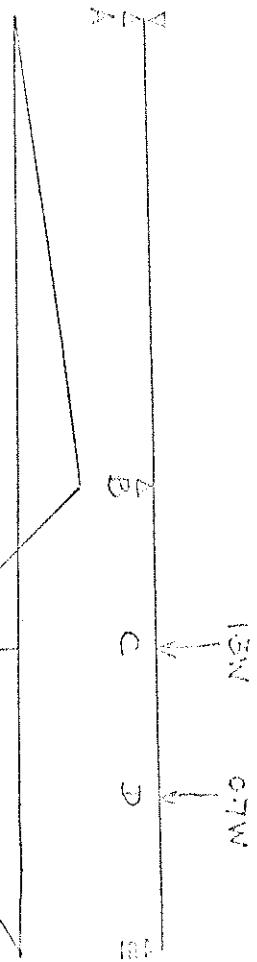
Four distinct types of independent elementary configurations mechanisms may be distinguished: sidesway, beam, gable, and joint. These are illustrated below:



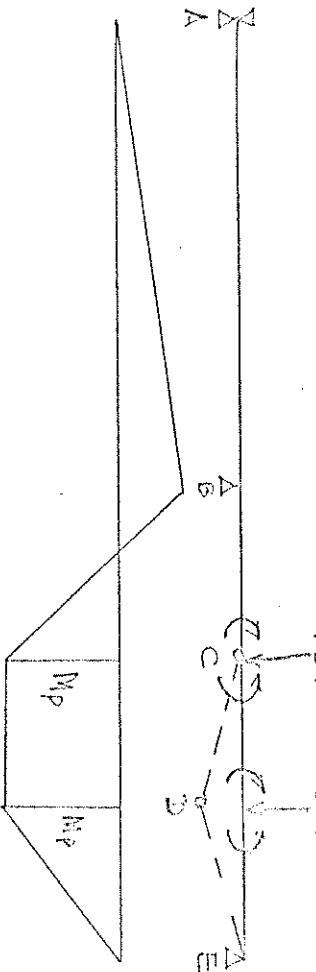
A mechanism is defined as existing in a structure when there are sufficient plastic hinges in part of the structure to permit an infinitesimal motion of that part. Further, a mechanism is said to be kinematically admissible if the signs of the moments at the mechanism plastic hinges are consistent with the mechanism motion. This is seen to be the case with the elementary mechanisms shown above.

To show that this need not always be the case, consider the following continuous beam. The loading is steadily increased from zero.

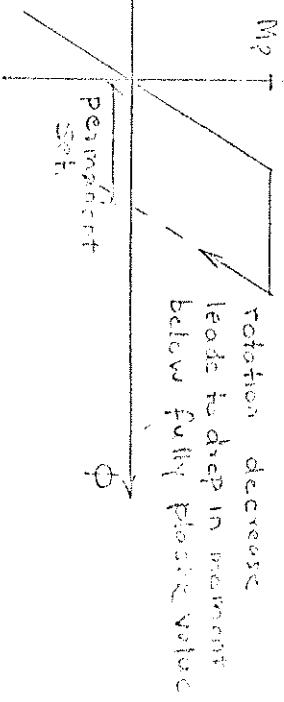


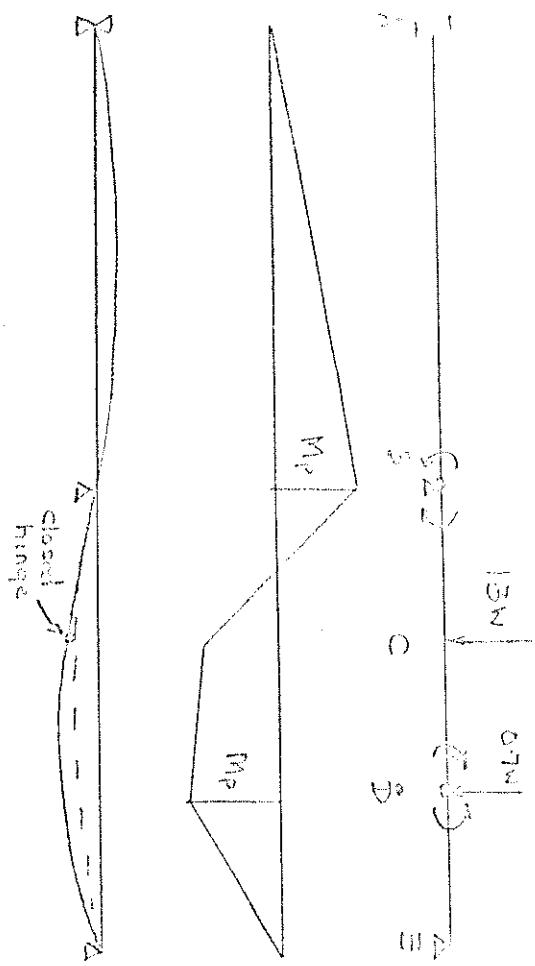


The first plastic hinge to form is at C. A subsequent increase in load leads to formation of a plastic hinge at D.



It might appear that CDE is a beam mechanism. Considerations of the plastic hinge moments show, however, that the moment at C will oppose any downward mechanism rotation of CD, and so collapse cannot occur. In fact, rotation of the hinge at D causes the hinge rotation at C to decrease, the moment at C to fall below the fully plastic value, and the hinge at C "closes", although a permanent plastic set has been induced in the beam at C. The moment-curvature history at C may be traced on the $M-\phi$ plot:





Finally, a hinge forms at section B. Since the plastic hinge at C has now closed, the beam mechanism BDC is seen to be kinematically admissible. The structure will carry no further load, and collapse occurs.

STRUCTURAL INTEGRITY, COLLAPSE, AND INTRINSIC PLASTICITY

It has been shown that when a plastic hinge closes, a permanent set, or deformation, is induced in the member. Now, plastic hinge closure, which is caused by reversal of plastic hinge rotation, occurs now and again under monotonic loading; on the other hand, it is an inevitable feature of cyclic loading, where deflection reversals are commonplace. Now, collapse of a structure is defined by the build-up of excessive deflections; might it not be possible for a structure under cyclic loading to accumulate permanent plastic hinge deformations in loading/unloading hinges, which would lead to large deflections, and render the structure un-serviceable? It turns out that, within a narrow band of cyclic loading, this is the case. Three distinct cases may be distinguished:

- (i) When a loading is cyclically applied and removed, and the resulting plastic deformation increments on successive loading cycles converge to a limit of zero then, once this limiting deformation is attained, no further plastic deformation will occur, the structure will behave elastically, and is said to have shaken down. The load is termed the shake-down load

load for the structure.

(ii) It is observed that, when a slightly larger load is applied and removed, the plastic deformation increment remains constant or even diverge. This is termed incremental collapse. The incremental collapse load is less than the plastic collapse load, and so, although incremental collapse is more likely to occur in practice, an adequate safety factor should be provided against its occurrence.

(iii) When structure loading is actually reversed, sufficient hinges and plastic flow may develop in a structure to render it useless, although the plastic collapse load is never reached. This phenomenon is termed alternating plasticity.

* This is a reproduction from G. Ciz's report (17).

APPENDIX

The direct stiffness method for the matrix analysis of structures is a modified version of the displacement method, transforming all individual member actions to a structural coordinate system. In this way, a structure stiffness matrix may easily be assembled from its component element stiffness by summation of interacting member stiffness factors.

For each element, in localized coordinates, the member end forces (s) are related to end displacements (v) by the element stiffness matrix (k):

$$[s] = [k][v]$$

$$k = \begin{bmatrix} \frac{AE}{L} & 0 & 0 \\ 0 & k_{11}\frac{EI}{L} & k_{12}\frac{EI}{L} \\ 0 & k_{21}\frac{EI}{L} & k_{22}\frac{EI}{L} \end{bmatrix}$$

$K_{11}, K_{12}, K_{21}, K_{22}$, are stiffness factors

for the flexural stiffness and depend on the type of the member. Member and displacements (v) are related to the structure displacements (r) by the displacement transformation

$$\text{matrix (a)} \quad [v] = [o] [r] = [o] [\bar{v}]$$

v - local coordinate system

\bar{v} - structure coordinate system

The matrix (a) may take one of the several forms according to the member end conditions, whether there is there is a hinge or no

Now, transforming the member end forces $[F_S]$ into the structure coordinate system:

$$\begin{aligned}
 [R] &= [S] = [a]^T \cdot [S] \\
 &= [a]^T \cdot [K] \cdot [-v] \\
 &= [a]^T \cdot [K] \cdot [a] \cdot [M] \\
 &= [K] \cdot [r]
 \end{aligned}$$

The triple matrix product $[K] = [a]^T \cdot [K] \cdot [a]$ is called the structure stiffness for that member. The $[K]$ matrices are evaluated for each member of the structure, and a banded stiffness matrix for the whole structure, $[K]$, is assembled by storing the $[K]$ matrices in determined locations in $[K]$.

For the entire structure,

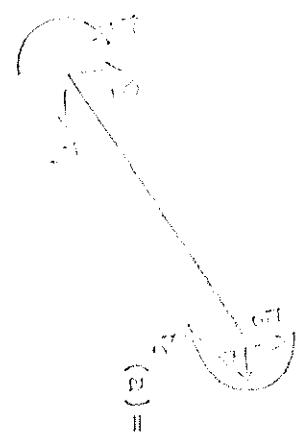
$$[R] = [S] = [K] \cdot [r] = [K] \cdot [-v]$$

From the input load data $[r]$, the structure joint deflections r may be found by solving $[r] = [K] \cdot [r]$ using equation solution techniques such as Prof. E. L. Wilson's subroutine SYMSOL for banded systems of equations.

Having found $[r]$, the internal member end deflections $[-v]$ and end forces $[F]$ may be found from:

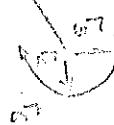
$$[v] = [a] \cdot [r]$$

$$\text{and } [S] = [K] \cdot [v] = [K] \cdot [a] \cdot [r]$$



$$(a) = \begin{bmatrix} -\cos\alpha & -\sin\alpha & 0 & \cos\alpha & \sin\alpha & 0 \\ \sin\alpha/L & -\cos\alpha/L & 1 & -\sin\alpha/L & \cos\alpha/L & 0 \\ \sin\alpha/L & -\cos\alpha/L & 0 & -\sin\alpha/L & \cos\alpha/L & 0 \end{bmatrix}$$

(1). Element without hinge-Displacement transformation matrix.



$$(a) = \begin{bmatrix} -\cos\alpha & -\sin\alpha & 0 & \cos\alpha & \sin\alpha & 0 \\ -\sin\alpha/2L & \cos\alpha/2L & 0 & \sin\alpha/2L & -\cos\alpha/2L & 0 \\ \sin\alpha/2L & -\cos\alpha/2L & 0 & -\sin\alpha/2L & \cos\alpha/2L & 0 \end{bmatrix}$$

(2). Hinge at T-end-Displacement transformation Matrix.

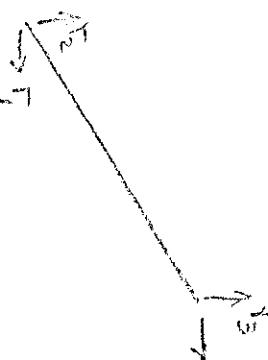
C



(a)

$$\begin{bmatrix} -\cos\alpha & -\sin\alpha & 0 & \cos\alpha & \sin\alpha & 0 \\ \sin\alpha/L & -\cos\alpha/L & 1 & -\sin\alpha/L & \cos\alpha/L & 0 \\ -\sin\alpha/2L + \cos\alpha/2L & \sin\alpha/2L - \cos\alpha/2L & 0 & -\sin\alpha/2L + \cos\alpha/2L & \sin\alpha/2L - \cos\alpha/2L & 0 \end{bmatrix}$$

(3). Hinge at J-end-Displacement transformation Matrix.



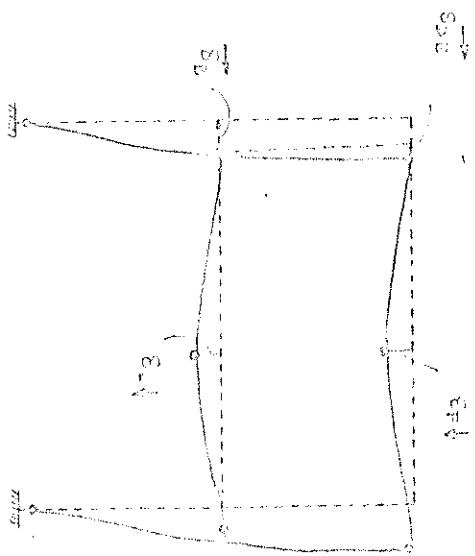
(a)

$$(a) = \begin{bmatrix} -\cos\alpha & -\sin\alpha & 0 & \cos\alpha & \sin\alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

(4). Hinges at both ends-Displacement transformation Matrix.

(EXAMPLES)

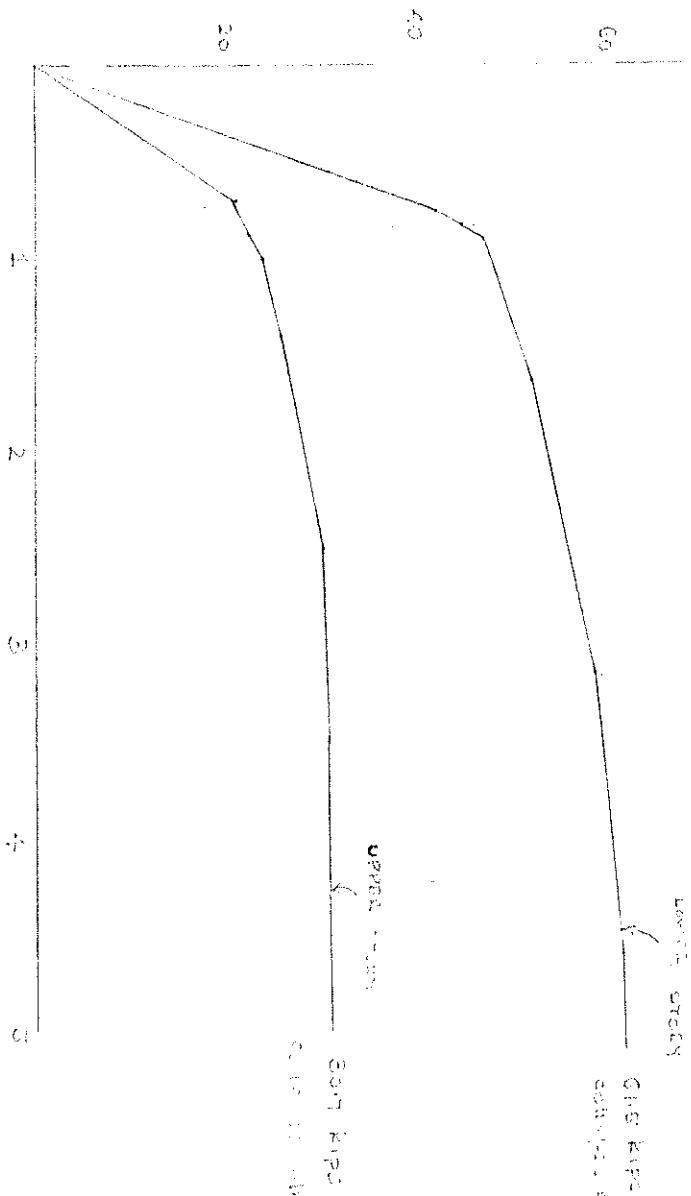
THIS PAGE ILLEGIBLE - ORIGINAL NO LONGER AVAILABLE



DEFLECTION CURVE AT INCIPIENT COLLAPSE

DISPLACEMENTS (S)	
ULARC	HODGE
\bar{e}_e	4.500
\bar{e}_d	3.536
e_p	4.000
e_u	2.662

REMARKS: DESIGN CONSIDERATIONS (CONT)



STORY CLEAR (ft)

12

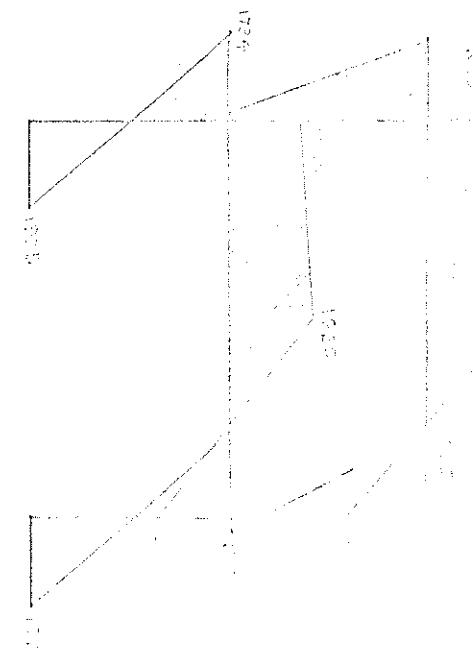
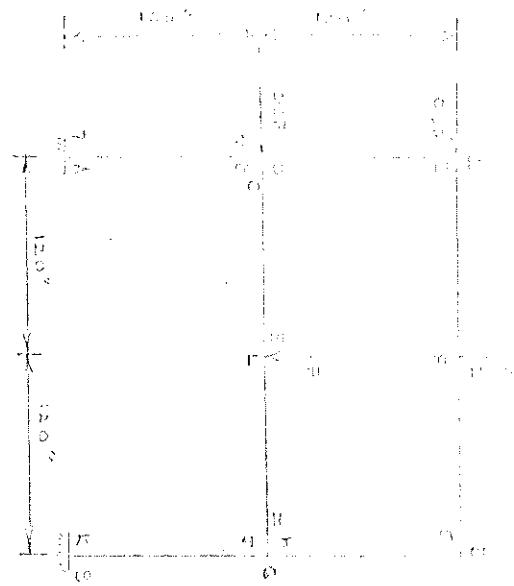
60

70

$$\text{Collapsible Load Factor } F = 3.4 \cdot 1.2 \cdot 1.15$$

$\left(\frac{F_L}{M_s}\right)$ LOAD FACTOR

Existing moment diagram for column



C

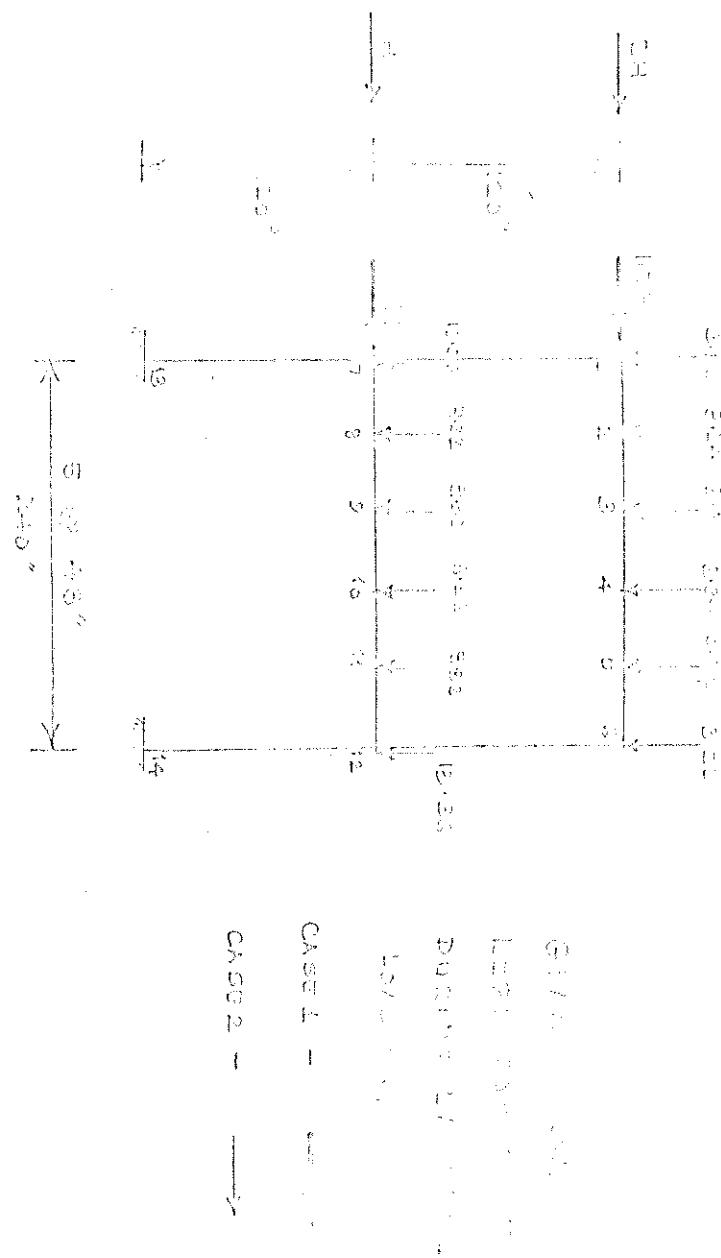
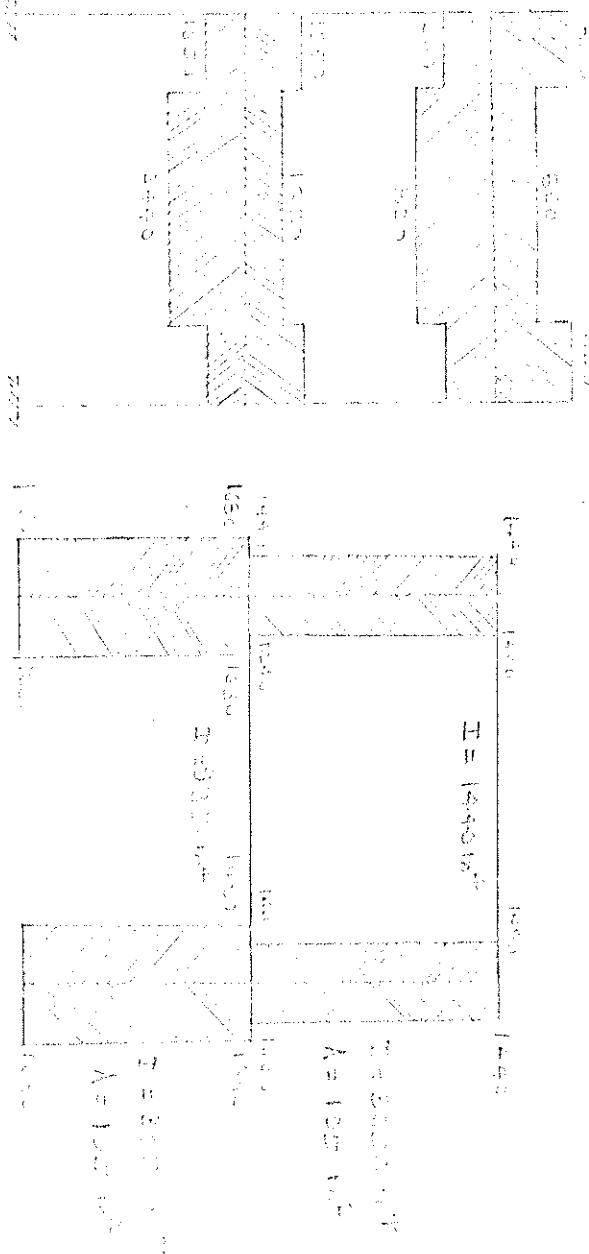
THIS PAGE ILLEGIBLE - ORIGINAL NO LONGER AVAILABLE

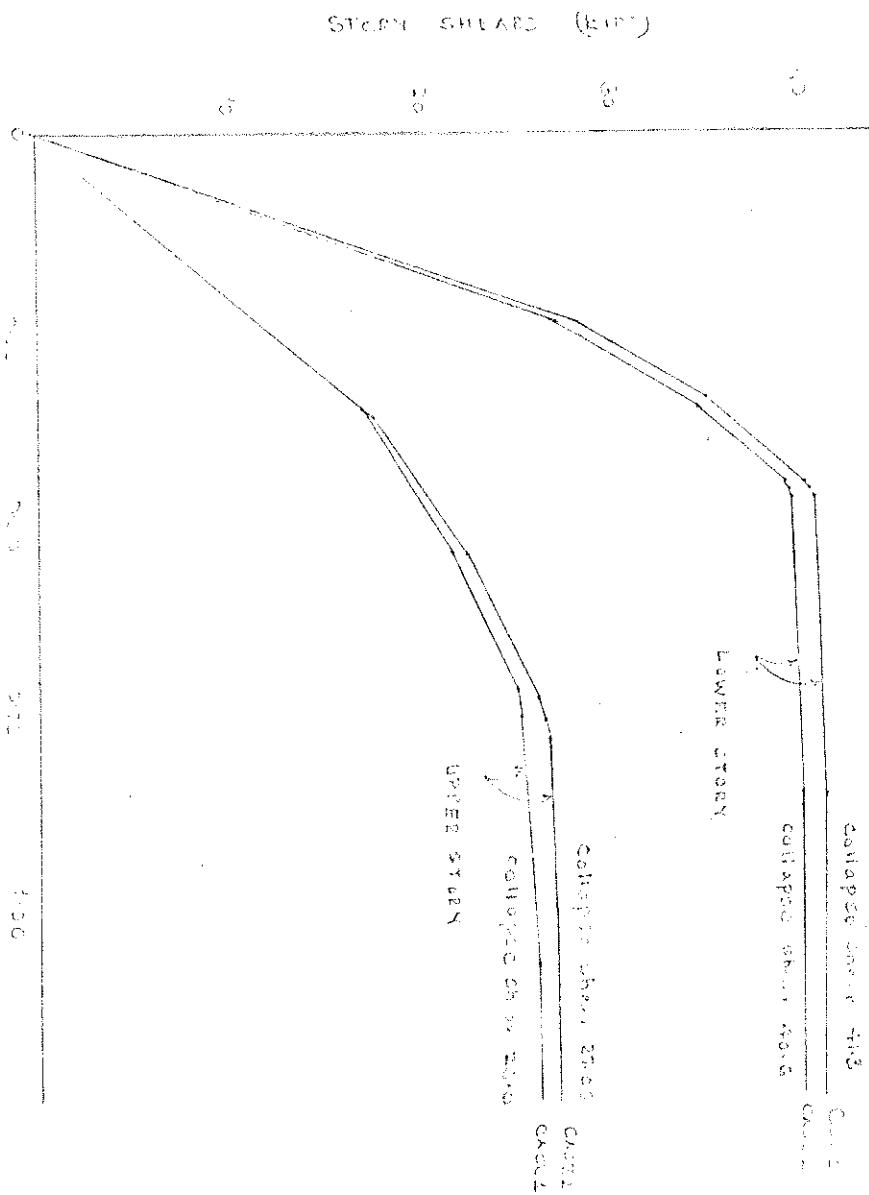
THEORY OF THE ELECTRONIC STATE
OF THE HYDROGEN ATOM

TRANSITION PROBABILITIES
FOR SPINLESS STATES

W. HEITLER
LONDON

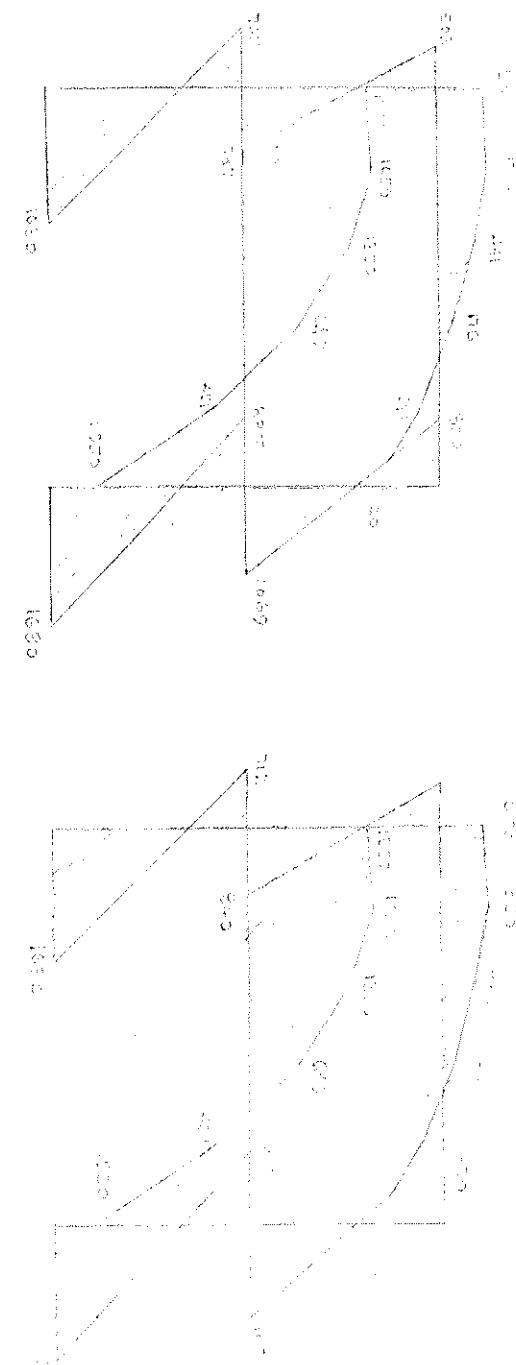
OXFORD UNIVERSITY PRESS



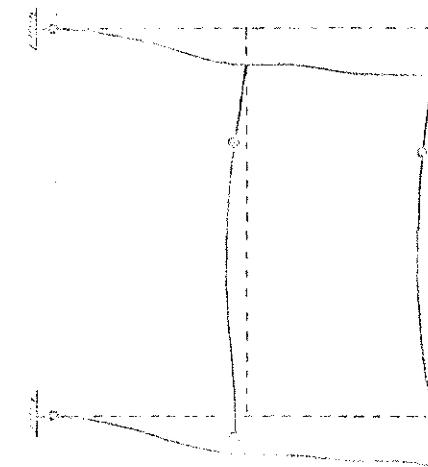


COMPARISON OF THE DISTRIBUTION AND DENSITY OF BENTHIC INVERTEBRATES IN THE BAY OF BISCAY

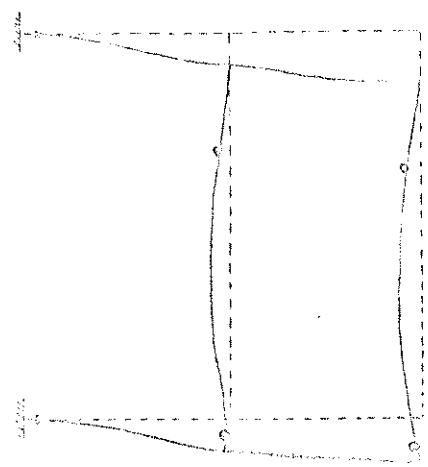
卷之三



DEMONSTRATION PROFILE AT INCREASING CONCENTRATION



CASE 4

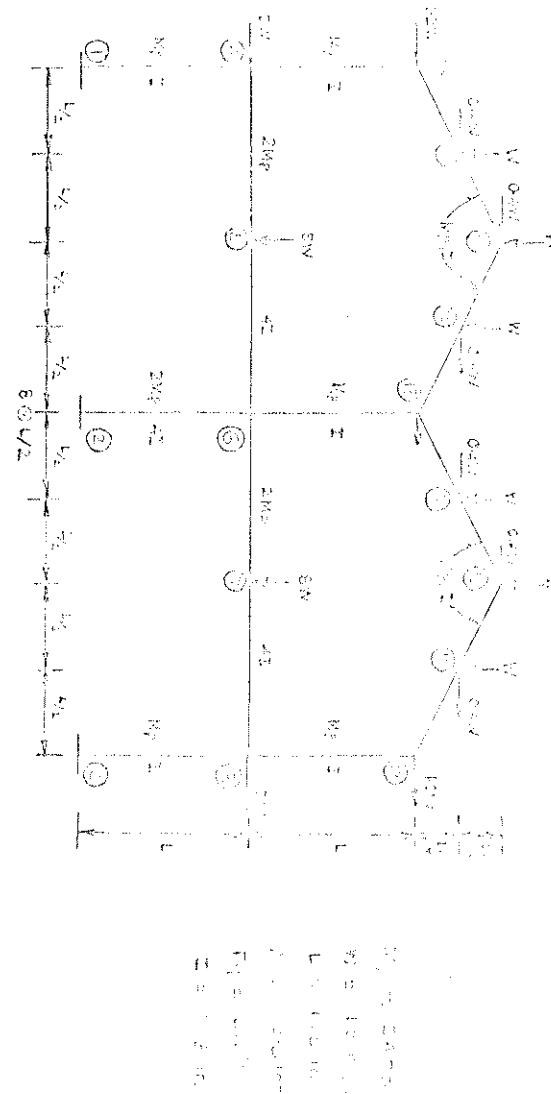
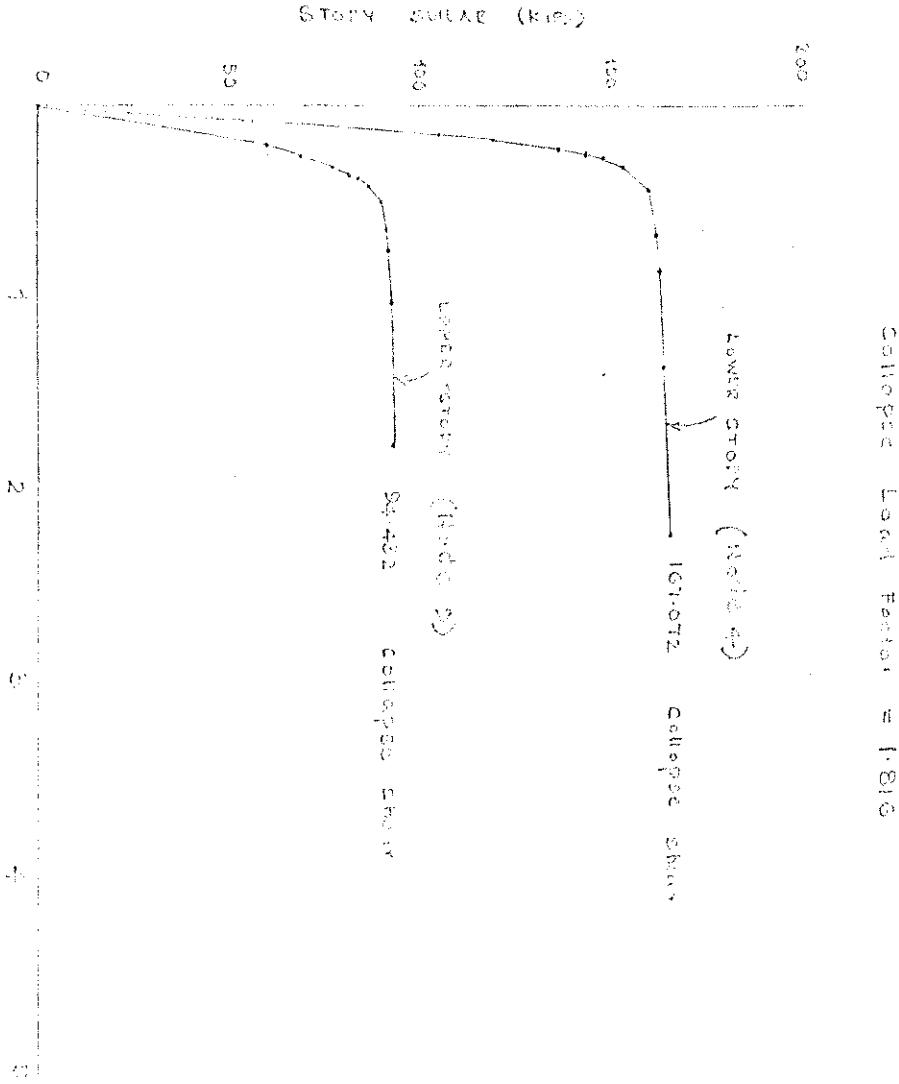


CASE 4

DEMONSTRATION PROFILE AT INCREASING CONCENTRATION

THIS PAGE ILLEGIBLE - ORIGINAL PAGE NO LONGER EXISTS

STRUCTURAL DESIGN OF TOWER



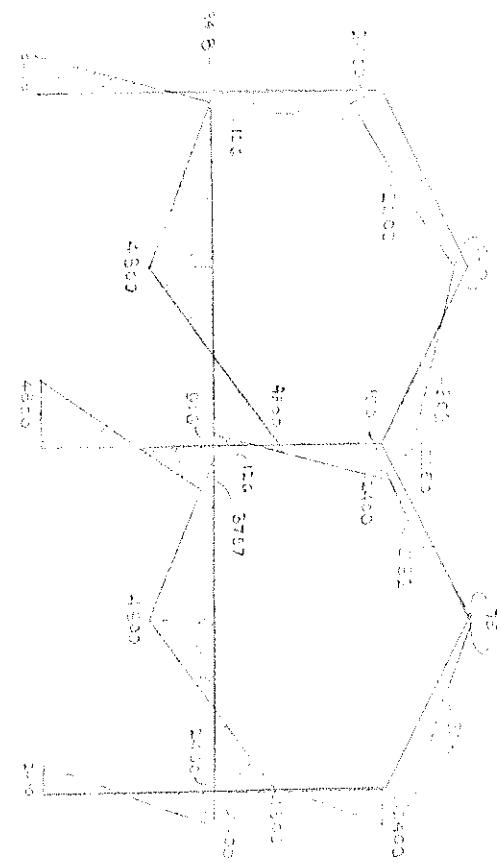
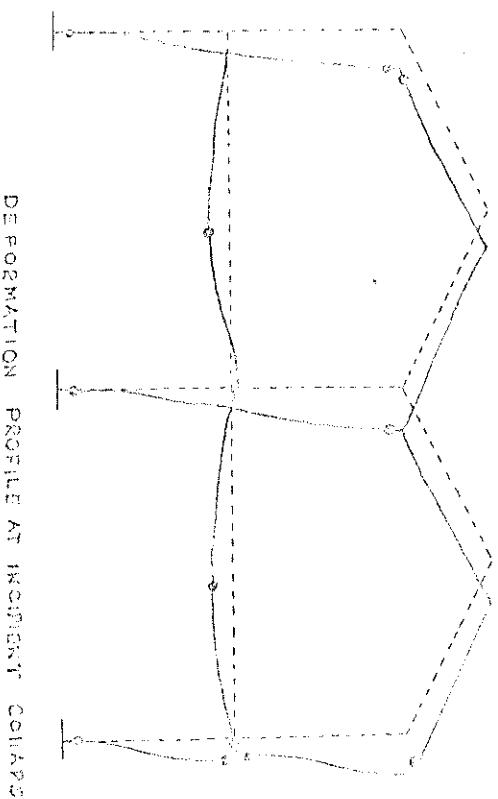


FIGURE 12-3. ENGINEERS' DISTANCE CONVERSION

(Kilometers)



DEFORMATION PROFILE AT HIGHEST COLLAPSE



NATIONAL INFORMATION SERVICE

EARTHQUAKE ENGINEERING

COMPUTER PROGRAM APPLICATIONS

IDENTIFICATION

ULARC: Small displacements elasto-plastic analysis of plane frames

Programmed: A. Sudhakar, G. H. Powell, G. Orr, R. Wheaton

PURPOSE

The program computes the node displacements, member forces, support reactions, plastic hinge rotations and rigid-plastic collapse loads for plane frames of arbitrary shape subjected to static joint loads and support settlements. The program is applicable to low-rise frames of steel or reinforced concrete. Large displacement ($P-\Delta$) effects are ignored. Non-proportional loading, including reversed loading, is permitted. The members may be of nonuniform stiffness and strength.

RESTRICTIONS

Dimension statements limit the program to frames with no more than 50 nodes, 10 supported nodes, 15 different member shapes, 20 different cross section strengths and 100 members. In addition, the storage occupied by the structure stiffness matrix may not exceed 3000 locations. The capacity can easily be expanded.

STRUCTURAL IDEALIZATION

The frame is defined by a series of nodes (joints) connected by one dimensional members (elements) possessing both flexural and axial stiffness. The nodes must be numbered in sequence. This numbering should preferably be chosen to minimize the largest node number difference within the members, but this is not essential. The members must also be numbered in sequence, in any convenient manner.

A coordinate system, X, Y, with axis Y 90 degrees counter clockwise from axis X, is established to define the node locations and load directions.

Supported joints are assumed to be restrained by elastic springs, which may be parallel or inclined to the X, Y axes. A rotational spring stiffness and two translational stiffnesses along a pair of axes A and B must be specified, where axis B is 90 degrees counter-clockwise from axis A. In addition, the angle between axes X and A must be specified for inclined supports. Realistic spring stiffnesses should be assigned, taking into account the fact that real supports cannot be completely rigid. An essentially

Rigid support can be obtained by assigning a very large stiffness (e.g. 10^{10} k/in).

Yielding may take place by the formation of localized plastic hinges at the member ends. The moment-curvature relationship at a plastic hinge is assumed to be of rigid perfectly plastic type (that is, shape factor = 1.0). Strain hardening effects may be approximated by placing two or more members of appropriate strengths and stiffnesses in parallel.

Members of nonuniform stiffness may be considered by specifying end stiffness and carry-over factors. The plastic moments for any member may be different at the two ends of the member and for positive and negative bending.

External loads may be specified to act at the nodes only. Support displacements may be specified at any supported nodes. The displacement is applied not directly at the node, but at the base of the spring, or springs, supporting the node. If the support spring stiffness is very large in comparison with the structure, the displacement at the supported node will essentially equal the specified displacement. However, if the spring stiffness is low, the node displacement may be substantially different from the specified displacement.

METHOD OF NONLINEAR ANALYSIS

The load-displacement relationship for the idealized structure is piecewise linear between "events", where an event corresponds to the formation of a new plastic hinge or unloading at an existing hinge. Beginning with the structure in its initial unloaded state, a load increment is applied and an analysis by the direct stiffness method is carried out. That proportion of the load increment required to produce a new event is determined. The results are then scaled linearly so that the state of the structure corresponds to the occurrence of the event. A new structure stiffness, accounting for any new or unloaded hinges, is then formed, a further load increment is applied, and the process is repeated. Within any load case, the sequence terminates when one of the following occurs:

- (a) A collapse mechanism is produced;
- (b) The full specified loading is applied;
- (c) A specified maximum displacement is reached. The analysis then proceeds to the next appropriate load case.

INPUT DATA

The following sequence of punched cards defines the problem. Units must be consistent throughout.

1. TITLE (12A6) - One card.
Cols 1 - 72: Problem title, to be printed with output
2. CONTROL INFORMATION (1215) - One card.
Cols 1 - 5: Number of nodes (max. 50).

- 6 - 10: Number of "Control" nodes, for which coordinates are specified directly. See Section 3.
- 11 - 15: Number of node coordinate generation commands (no limit). See Section 4.
- 16 - 20: Number of supported nodes (max. 10) See Section 5.
- 21 - 25: Number of different member stiffness types (max. 15) See Section 6.
- 26 - 30: Number of different cross section strength types (max. 20). See Section 7.
- 31 - 35: Number of members (max. 100).
- 36 - 40: Number of generation commands for member locations.
- 41 - 45: Number of generation commands for member stiffnesses. See Section 9.
- 46 - 50: Number of generation commands for member strengths. See Section 10.
- 51 - 55: Number of load cases (no limit). See Section 11.
- 56 - 60: Maximum number of analysis cycles permitted within any load case. It is possible, although rare, for the solution to get caught in unending cycles of loading and unloading at plastic hinges. To avoid waste of computer time, execution will be terminated if the number of separate analyses for any loading case exceeds this number. A recommended value is approximately 1.5 times the number of plastic hinges required to cause collapse. If in doubt, specify a very large number.
3. CONTROL NODE COORDINATES (15, 2F10.0) - One card for each control node.
- Col 1 - 5: Node number, in any sequence.
- 6 - 15: X coordinate.
- 16 - 25: Y coordinate.
4. NODE COORDINATE GENERATION (415, F10.0) - One card for each generation command. Omit if there are no generation commands. See Note below for automatic generation feature.
- Col 1 - 5: Number of node at beginning of generation line.
- 6 - 10: Number of node at end of generation line.
- 11 - 15: Number of nodes to be generated along the line.
- 16 - 20: Node number difference (constant) between successive generated nodes, and also between the first generated node and the node at the beginning of the generation line.
- 21 - 30: Distance between successive generated nodes. If greater than or equal to 1.0, assumed to be an actual distance. If less than 1.0, assumed to be a proportion of the distance between the beginning and end nodes. If zero or blank, the nodes are automatically spaced uniformly along the generation line.
- Note:** Only straight line generation is permitted. The number of nodes generated by each command may be one or any larger number. The coordinates of the two nodes at the beginning and end of the generation line must have been

previously defined, by direct specification or by previous straight line generation. If these coordinates have not been defined, a warning message is printed.

It is not necessary to provide generation commands for nodes which are (a) sequentially numbered between the beginning and end nodes of any straight line, and (b) equally spaced along that line. After all generation commands have been executed, the coordinates for each group of unspecified nodes are automatically generated assuming sequential numbering and equal spacing along lines joining the specified nodes immediately preceding and following the group.

5. SUPPORTED NODES (15, 4E10.0) - One card for each supported node.
 Cols
 1 - 5: Node number, in any sequence.
 6 - 15: Translational support stiffness along A axis.
 16 - 25: Translational support stiffness along B axis.
 26 - 35: Rotational support stiffness.
 36 - 45: Angle (degrees) between X axis and A axis, measured counter-clockwise from the X axis.

Note: Any of the support stiffnesses may be zero if desired.

6. MEMBER STIFFNESS TYPES (15, 6F10.0, 215) - One card for each different type. See Note below.
 Cols
 1 - 5: Stiffness type number.
 6 - 15: Average cross sectional area.
 16 - 25: Reference value of flexural moment of inertia
 26 - 35: Young's modulus of elasticity.
 36 - 45: Stiffness factor K_{ii}.
 46 - 55: Stiffness factor K_{jj}.
 56 - 65: Stiffness factor K_{ij}.
 66 - 70: Cross section strength type at end i. This is optional. If left blank, no specific cross section strength is associated with this member stiffness type, and a section strength must be specified in Section 10. If the number of a cross section strength type is punched, the specification of member stiffnesses in section 9 serves also to define the cross section strength, and this need not be specified separately.
 71 - 75: Cross section strength type at end j. This is optional, as for end i.

Note: The flexural stiffness matrix of any member is of the form

$$\frac{EI_r}{L} \begin{bmatrix} K_{ii} & K_{ij} \\ K_{ji} & K_{jj} \end{bmatrix}$$

in which E = Young's modulus; I_r = reference moment of inertia; L = member length; and K_{ii}, K_{jj} and K_{ij} are stiffness factors.

For a member of uniform stiffness, K_{ii} = K_{jj} = 4.0, K_{ij} = 2.0. For a uniform member with a real hinge at end j, K_{ii} = 3.0, K_{jj} = K_{ij} = 0.

For tapered members or members in which shear deformations are important, appropriate stiffness coefficients must be determined. It should be noted that the collapse load of a rigid-plastic frame depends only on the member strengths, and not on their stiffness properties.

7. CROSS SECTION STRENGTH TYPES (I5, 4F10.0) - One card for each different cross section strength, defined by the ultimate moment capacity of the section.

- | | | |
|------|----------|---|
| Cols | 1 - 5: | Strength type number. |
| | 6 - 15: | Ultimate moment for clockwise moments acting on the member end. |
| | 16 - 25: | Ultimate moment for counter-clockwise moments acting on the member end. |
| | 26 - 35: | Limiting plastic hinge rotation capacity for clockwise moments acting on the member end (radians).
See Note below. |
| | 36 - 45: | Limiting plastic hinge rotation capacity for counter-clockwise moments acting on the member end (radians). |

Note: Specification of limiting hinge rotation capacities is optional. If hinge rotations are not of concern, leave blank or punch a large number. If the specified rotation at any hinge is exceeded, the rotation is marked by an asterisk in the computer printout. This helps to identify sections at which excessive plastic deformations are taking place.

8. MEMBER LOCATION GENERATION (515) - One card for each generation command.

- | | | |
|------|----------|---|
| Cols | 1 - 5: | Member number, or number of first membr in a sequentially numbered series of members to be generated by this command. |
| | 6 - 10: | Number of node at one end of this member (node i). |
| | 11 - 15: | Number of node at the other end of member (node j). |
| | 16 - 20: | Number of last member in series. If this command covers only a single member, leave blank. |
| | 21 - 25: | Node number difference. For each successive member in the series, the numbers of nodes i and j are obtained by adding this number to those for the previous member. If this command covers only a single member, leave blank. |
| | 6 - 10: | Member number, or number of first membr in a series of members with identical stiffnesses. The members in the series must be regularly, but not necessarily sequentially, numbered. |
| | 6 - 10: | Stiffness type number. The stiffness coefficients will correspond to the i,j ends as defined in Section 6. |
| | 11 - 15: | Number of last member in series. If this command covers only a single member, leave blank. |
| | 16 - 20: | Member number difference, n. If the first member in |

the series is m , the members in the series are $m, m+n, m+2n$, etc. If this command covers only a single member, leave blank.

10.

MEMBER STRENGTH GENERATION (515) - One card for each generation command. This number may be zero if all strengths have been defined with the member stiffnesses in section 9. The strength generation commands must be used to specify strengths not previously specified by Section 9. The commands may also be used, if desired, to change any strengths previously specified by Section 9.

cols

1 - 5:

Member number, or number of first member in a series of members with identical strengths.

6 - 10:

Strength type for cross section at end i of member.

11 - 15:

Strength type for cross section at end j of member.

16 - 20:

Number of last member in series. If this command covers only a single member, leave blank.

21 - 25:

Member number difference, n, as defined in Section 9.

11. LOADINGS - One set of cards, as follows, for each load case.

11a. Control Information (315, 2F10.0,45A1) - One card.

cols

1 - 5:

Number of generation commands for applied external loads on nodes. See Section 11b.

6 - 10:

Number of generation commands for nodes with specified support displacements. See Section 11c.

15: Load application code. If zero or blank, the loading

is to be added to the loads acting at the end of the previous load case. If 1, the structure is reinitialized to an unloaded state, and the loading is then applied.

Load factor. The loads specified in sections 11b and 11c are scaled by this factor to define the

loads to be applied to the structure. For example, if working loads are specified in Sections 11b and 11c, then the load factor might be a typical design

load factor. If load is to be applied until the frame collapses, enter a large number (e.g. 100). Limiting displacement. If the absolute value of

horizontal or vertical displacement at any node exceeds this value, the results are scaled to reduce the displacement to this value, and the load case is then terminated. A limiting displacement may be

specified if the limit of usefulness of the structure is reached prior to complete collapse, or if cyclic loading of the structure between specified displacement limits is to be applied. If the analysis is to be continued to collapse, leave blank.

36 - 80:

Load case title, to be printed with output.

Note: If the limiting displacement is reached, the program moves to the next load case. If collapse occurs, the program moves to the next load case for which the load application code is equal to 1.

11b. Nodal Load Generation (15, 3F10.0,215) - One card for each load

generation command.

- | | | |
|----------|--|---|
| Cols | 1 - 5: | Node number, or number of first node in a series of nodes to which identical loads are to be added. |
| 6 - 15: | Applied external load acting in direction of X axis. | |
| 16 - 25: | Applied external load acting in direction of Y axis. | |
| 26 - 35: | Applied external moment, clockwise positive. | |
| 36 - 40: | Number of last node in series. If this command covers only a single node, leave blank. | |
| 41 - 45: | Node number difference, n, as defined in Section 9. | |

Note: Any node may appear in several generation commands. The total loads applied at such nodes will be the sum of those specified in the different commands.

11c. Support Displacement Generation (15, 3F10.0, 215) - One card for each generation command.

- | | | |
|----------|--|---|
| Cols | 1 - 5: | Node number, or number of first node in a series of nodes to which identical support displacements are to be added. |
| 6 - 15: | Support base displacement in direction of X axis. | |
| 16 - 25: | Support base displacement in direction of Y axis. | |
| 26 - 35: | Support base rotation (degrees, clockwise positive). | |
| 36 - 40: | Number of last node in series. If this command covers only a single node, leave blank. | |
| 41 - 45: | Node number difference, n, as defined in Section 9. | |

Note: Any node may appear in several generation commands. The total displacements applied at such nodes will be the sum of those specified in the different commands.

12. NEXT PROBLEM

The data for a new problem, starting at Section 1, may be added if desired. Any number of problems may be solved in a single computer run, provided there are no fatal errors present in the data. Any such error will lead to termination of execution within the problem for which it occurs. Add two blank cards after the last problem.


```

C
3      TH    40.   64.4H   I   4X4H   J   3X9H
4      3X7H   J   1/1
5      {15.6X,13.5X,13.4X,16.0X,14.6X,14.1}
RETURN
END

```

卷之三

PAGE 6

PAGE 8


```

      SUBROUTINE FORMA (A,IMJ,IMJ,COSA,SINA,ELL)
      DIMENSION A(3,6)
      C
      C   FORM TRANSFORMATION MATRIX-- NO HINGE AT I AND J END
      C
      A(1,1)=COSA
      A(1,2)=-SINA
      A(1,3)=0.
      A(1,4)=0.
      A(1,5)=SINA
      A(1,6)=-COSA
      A(2,1)=SINA
      A(2,2)=COSA
      A(2,3)=0.
      A(2,4)=0.
      A(2,5)=-SINA
      A(2,6)=COSA
      A(3,1)=0.
      A(3,2)=0.
      A(3,3)=SINA/ELL
      A(3,4)=-COSA/ELL
      A(3,5)=COSA/ELL
      A(3,6)=-SINA/ELL
      C
      C   FORM TRANSFORMATION MATRIX--HINGE AT I END ONLY
      A(2,1)=+SINA/12.*ELL
      A(2,2)=+COSA/12.*ELL
      A(2,3)=0.
      A(2,4)=+SINA/(12.*ELL)
      A(2,5)=-COSA/(12.*ELL)
      A(2,6)=0.
      C
      C   FORM TRANSFORMATION MATRIX--HINGE AT J END ONLY
      A(1,1)=+SINA/12.*ELL
      A(1,2)=+COSA/12.*ELL
      A(1,3)=0.
      A(1,4)=0.
      A(1,5)=+SINA/12.*ELL
      A(1,6)=0.
      C
      C   FORM TRANSFORMATION MATRIX---HINGE AT I AND J ENDS
      20 DO 30 I=1,4
      30 DO 30 J=1,6
      30 A(I,J)=0.
      A(1,1)=+COSA
      C
      C   SUPPORT REACTIONS
      C
      C   PRINT 80
      80 FORMAT(//,34H SUPPORT REACTIONS AT END OF CYCLE/,/
      1          5H NODE,6X,7H-FORCE,6X,7H-MOMENT,/
      2          12X,7H-FORCE,6X,7H-FORCE,/)
      C
      DO 90 N=1,NSJS
      NJS(J,N)
      CC(COSS(N))
      SS(SINS(N))
      FF=SFRA(INI)*SS+SFRR(INI)*CC
      FF=SFRA(INI)*SS+SFRR(INI)*CC
      90 PRINT 100, NJS, SFRA(INI), SFRR(INI), FF, FF
      100 FORMAT(14+1X,3F13.3,1X,2F13.3)
      C
      RETURN
      END

```

PAGE 22

PAGE 24

U L A R C
Microcomputer Version
(October 1986)

INPUT DATA

1-CONTROL INFORMATION followed by separator "SYSTEM"

Format:

N1 = N1, N2, N3, ..., N12

where

- N1: Number of nodes
- N2: Number of "Control" nodes, for which coordinates are specified directly.
- N3: Number of node coordinate generation commands
- N4: Number of supported nodes (max. 10)
- N5: Number of different member stiffness types (max. 15)
- N6: Number of different cross-section strength types (max. 20)
- N7: Number of generation commands for member locations
- N8: Number of generation commands for member stiffness
- N9: Number of generation commands for member strengths
- N10: Number of generation commands for member strengths
- N11: Number of load cases
- N12: Maximum number of analysis cycles permitted within any load case. It is possible, although rare, for the solution to get caught in unending cycles of loading and unloading at plastic hinges. To avoid waste of computer time, execution will be terminated if the number of separate analyses for any loading case exceeds this number. A recommended value is approximately 1.5 times the number of plastic hinges required to cause collapse. If in doubt, specify a very large number.

2-CONTROL NODE COORDINATES introduced by separator "JOINT". One line for each node.

Format:

N1 X = X1 Y = Y1

where

- N1: Node number, in any sequence
- X1: X Coordinate
- Y1: Y Coordinate

3. NODE COORDINATE GENERATION introduced by separator "NODE". Unit if there are no generation commands. One line for each generation command. See note below for automatic generation feature.

Format:

N1 = N11,N12,N13,N14 P = P1

where

N1: Number of node at beginning of generation line.

N2: Number of node at end of generation line.

N3: Number of nodes to be generated along the line.

N4: Node number difference(constant) between successive generated nodes, and also between the first generated node and the node at the beginning of the generation line.

P1: Distance between successive generated nodes. If greater than or equal to 1.0, assumed to be an actual distance. If less than 1.0, assumed to be a proportion of the distance between the beginning and end nodes. If zero, the nodes are automatically spaced uniformly along the generation line.

Note: Only straight line generation is permitted. The number of nodes generated by each command may be one or any large number. The coordinates of the two nodes at the beginning and end of the generation line must have been previously defined, by direct specification or by previous straight line generation. If these coordinates have not been defined, a warning message is printed.

It is not necessary to provide generation commands for nodes which are (a) sequentially numbered between the beginning and end nodes of any straight line, and (b) equally spaced along that line. After all generation commands have been executed, the coordinates for each group of unspecified nodes are automatically generated assuming sequential numbering and equal spacing along lines joining the specified node immediately preceding and following the group.

4. SUPPORTED NODES INFORMATION followed by separator "SUPPORT". One line for each support-node.

Format:

K = K1 S = S1 Q = Q1 V = V1 R = R1

where

K1: Node number, in any sequence.

S1: Translational support stiffness along A axis.

Q1: Translational support stiffness along B axis.

V1: Rotational support stiffness.

R1: Angle (degrees) between X axis and A axis, measured counter-clockwise from the X axis.

Note: Any of the support stiffnesses may be zero if desired.

5 - MEMBER STIFFNESS TYPES followed by separator "STIFFNESS". One line for each type.

Format:
K=K1 A=A1 X=X1 E=E1 S=S1 V=V1 R=R1 M=M1 N=N1

where

K1: Stiffness type number
A1: Average cross sectional area
X1: Reference value of flexural moment of inertia
E1: Young's modulus of elasticity
S1: Stiffness factor Kij
V1: Stiffness factor Kij
R1: Stiffness factor Kij
M1: Cross section strength type at end i. This is optional. If left blank, no specific cross section strength is associated with this member stiffness type, and a section strength must be specified in Section "GENERATION". If the member of a cross section strength type is punched, the specification of member stiffnesses in section "FRAMES" serves also to define the cross section strength, and this need not be specified separately.
M2: Cross section strength type at end j. This is optional, as for end i.

Note: The flexural stiffness matrix of any member is of the form

$$\frac{E \cdot I_r}{L} \begin{pmatrix} K_{ii} & K_{ij} \\ K_{ji} & K_{jj} \end{pmatrix}$$

in which E=Young's modulus; Ir = reference moment of inertia; L = member length; and Kii, Kij, Kjj are stiffness factors.

For a member of uniform stiffness, Kii = Kjj = 4.0, Kij = 2.0. For a uniform member with a real hinge at end j, Kii = 3.0, Kjj = Kij = 0. For tapered members or members in which shear deformations are important, appropriate stiffness coefficients must be determined. It should be noted that the collapse load of a rigid-plastic frame depends only on the member strengths, and not on their stiffness properties.

6. CROSS SECTION STRENGTH TYPES followed by separator "SECTION". One line
for each different gross section strength defined by the ultimate
moment capacity of the section.

Format:

A = A1 B = B1 C = C1 D = D1 E = E1

where:

A1: Strength type number.
B1: Ultimate moment for clockwise moments acting on the member end.
C1: Ultimate moment for counter-clockwise moments acting on the
member end.
D1: Limiting plastic hinge rotation capacity for clockwise moments
acting on the member end. (radians)
E1: Limiting plastic hinge rotation capacity for counter-clockwise
moments acting on the member end. (radians)

Note: Specification of limiting hinge rotation capacities is optional.
If hinge rotations are not of concern, leave zero or punch a large
number. If the specified rotation at any hinge is exceeded, the rotation
is marked by an asterisk in the computer printout. This helps to
identify sections at which excessive plastic deformations are taking
place.

7. MEMBER LOCATION GENERATION introduced by separator "MEMBER". One line for
each member generation command:

Format:

F = F1 G = G1, G2, G3, G4

where:

F1: Member number, or number of first member in a sequentially
numbered series of members to be generated by this command.
G1: Number of node at one end of this member (node i).
G2: Number of node at the other end of this member (node j).
G3: Number of last member in series. If this command covers only a
single member, leave zero.
G4: Node number difference. For each successive member in the
series, the numbers of nodes i and j are obtained by adding
this number to those for the previous member. If this command
covers only a single member, leave zero.

D MEMBER STIFFNESS GENERATION informed by section "FRAMES". Note that if CROSSES SECTION STRENGTH TYPES have been associated with member stiffness types (section 5), these commands may also serve to generate some or all of the member strengths. One line for each command.

Format:

T = TI S = S1,S2,S3

where

TI = Stiffness type number. The stiffness coefficients will correspond to the i,j ends as defined in section "STIFFNESS".

S1 = Member number or number of first member in a series of members with identical stiffnesses. The members in the series must be regularly, but not necessarily sequentially numbered.

S2 = Number of last member in series. If this command covers only a single member, leave zero.

S3 = Member number difference, n. If the first member in the series is n, the members in the series are n, n + n, n + 2n, etc. If this command covers only a single member, leave zero.

G MEMBER STRENGTH GENERATION followed by separator "GENERATION", one line for each generation command. This number may be zero if all strengths have been defined with the member stiffnesses in section B. The strength generation commands must be used to specify strengths not previously specified by section B. The commands may be also be used, if desired, to change any strengths previously specified by section G.

Format:

W = W1,W2 W = W1,W2,W3

where

W1 = Strength type for cross section at end 1.

W2 = Strength type for cross section at end 2.

W1,W2,W3 have the same meanings as S1,S2,S3 respectively.

10.=LOADING INFLUENCING SEPARATORS "CONTROL", "LOADS" AND "DISPLACEMENT".

10E= CONTROL.

Format:

I1 = I1,I2,I3 I = I1,I2

where

I1= Number of generation commands for applied external loads on nodes. See section 10b.

I2= Number of generation commands for nodes with specified support displacements. See section 10c.

I3= Load application code. If zero, the loading is to be added to the loads acting at the end of previous load case. If 1, the structure is reinitialized to an unloaded state, then the loading is applied.

I4= Load factor. The loads specified in section 10b and 10c are scaled by this factor to define the loads to be applied to the structure. For example, if working loads are specified in section 10b and 10c, then the load factors might be a typical design load factor. If load is to be applied until the frame collapses enter a large number (e.g. 100).

I5= Limiting displacement. If the absolute value of horizontal or vertical displacement at any node exceeds this value, the results are scaled to reduce the displacement to this value and the load case is then terminated. A limiting displacement may be specified if the limit of usefulness of the structure is reached prior to complete collapse, or if cyclic loading of the structure between specified displacement limits is to be applied. If the analysis is to be continued to collapse, leave zero.

Note: If the limiting displacement is reached, the program moves to the next load case. If collapse occurs, the program moves to the next load case for which the load application code is equal to 1.

10F= NODEAL LOAD GENERATING IN SECTION "LOADS". One line for each generation command.

Format:

I = I1,I2,I3 R = R1,R2,R3

where

I1= Node number, or number of first node in a series of nodes to which identical loads are to be applied.

I2= Number of last node in series. If this command covers only a single node, leave zero.

I3= Node number difference, as defined in section B.

R1= Applied external load acting in direction of X axis.

R2= Applied external load acting in direction of Y axis.

R3= Applied external moment, clockwise positive.

Note: Any node key appear in several generation commands. The total loads applied at such nodes will be the sum of those specified in the different commands.

LOC-SUPPORT DISPLACEMENT GENERATION in section "DISPLACEMENT". One line
for each generation command.

Format:

D = D1,D2,D3 G = G1,G2,G3

where

D1: Node number, or number of first node in a series of nodes to
which identical support displacements are to be added.

D2: Number of last node in series. If this command covers only a
single node, leave zero.

D3: Node difference, n, as defined in section 8.

G1: Support base displacement in direction of X axis.

G2: Support base displacement in direction of Y axis.

G3: Support base rotation (degrees, clockwise positive).

Note: Any node may appear in several generation commands. The total
displacement applied at such nodes will be the sum of those specified
in the different commands.