DRAIN-2DX USER GUIDE

by

RAKESH ALLAHABADI
Postgraduate Research Engineer

and

GRAHAM H. POWELL
Professor of Civil Engineering

March 1988

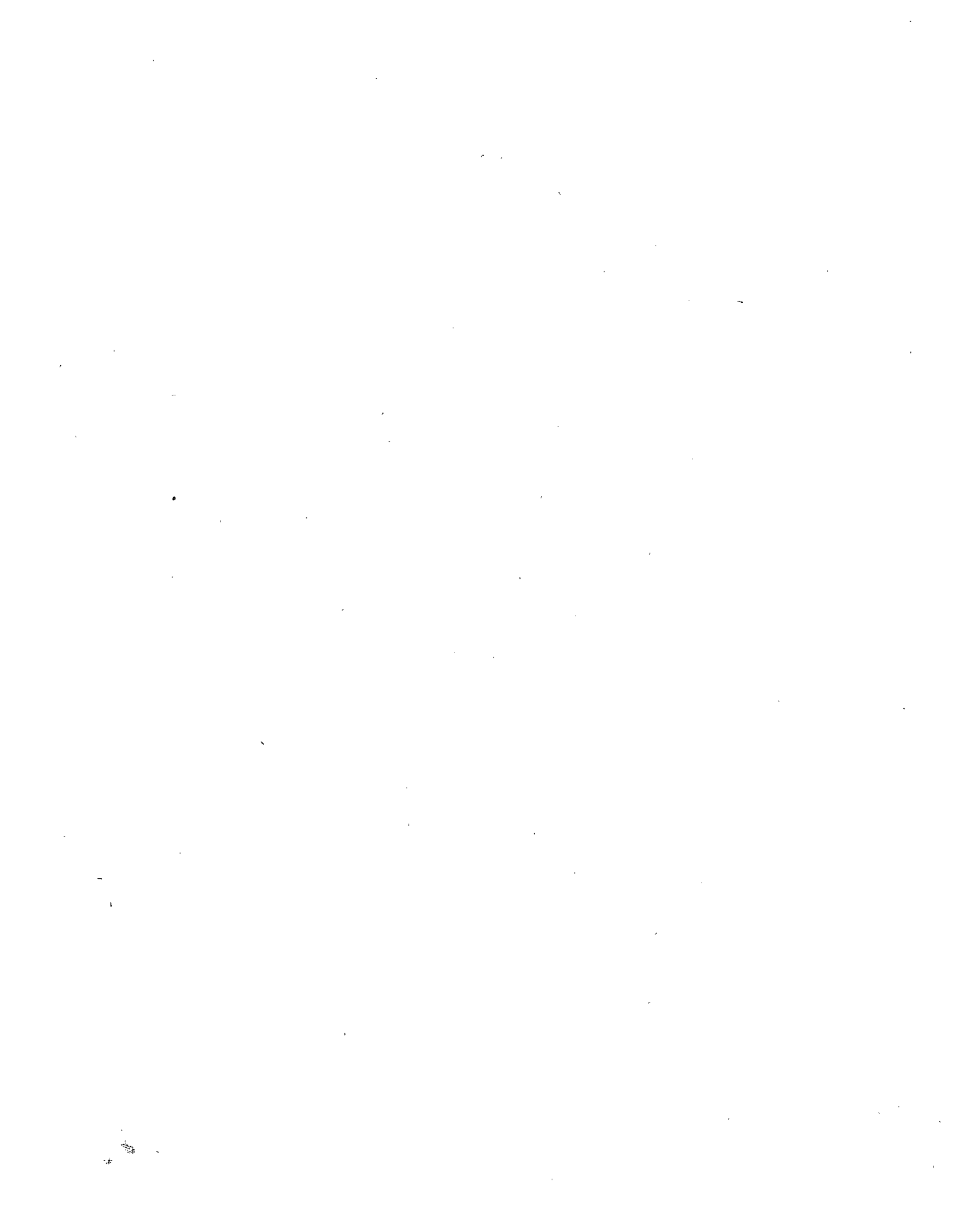50272-101

REPORT DOCUMENTATION PAGE	1. REPORT NO. NSF/ENG-88043	2.	3. PB91-212530

4. Title and Subtitle

DRAIN-2DX User Guide

5. Report Date
March 1988

6.

7. Author(s)

R. Allahabadi and G.H. Powell

8. Performing Organization Rept. No.

UCB/EERC-88/06

9. Performing Organization Name and Address

Earthquake Engineering Research Center
University of California, Berkeley
1301 S 46th St.
Richmond, CA 94804

10. Project/Task/Work Unit No.

11. Contract(C) or Grant(G) No.

(C)

(G) CEE 8414905

12. Sponsoring Organization Name and Address

National Science Foundation
1800 G. St. NW
Washington, DC 20550

13. Type of Report & Period Covered

14.

15. Supplementary Notes

16. Abstract (Limit: 200 words)

DRAIN-2DX, a computer program for the analysis of inelastic 2-dimensional structures under static and dynamic loading, with particular emphasis on seismic response, is an extension of, and supersedes, earlier versions of the DRAIN-2D program. Many desirable features, lacking in DRAIN-2D because of its simplicity, have been added to DRAIN-2DX. Nonlinear static, as well as dynamic, analyses can be performed. More sophisticated dynamic step-by-step solution strategies may be specified: the time step can be varied automatically and corrections can be applied to compensate for errors in force equilibrium and energy balance. Energy balance computation can be performed, and detailed logs of energy and equilibrium errors can be obtained. Static and dynamic loads can be applied in any sequence. The structure state can be saved permanently at the end of any analyses. A new analysis can then begin from any previously saved state. Dynamic analyses can be carried out for ground accelerations (all supports moving in phase), ground displacements (supports may move out-of-phase), specified external forces (e.g. wind) and initial velocities (corresponding to an initial impulse). Mode shapes and frequencies can be calculated, and linear response spectrum analyses can be performed. Cross sections can be specified through a structure, and the resultant normal, shear and overturning effects on these sections can be computed. Post-processing files can be produced, and an interactive post-processing program has been developed to permit tabulation of results from these files.

17. Document Analysis   a. Descriptors

b. Identifiers/Open-Ended Terms

c. COSATI Field/Group

18. Availability Statement	19. Security Class (This Report)	21. No. of Pages
Release Unlimited	unclassified	253
	20. Security Class (This Page)	22. Price
	unclassified	

(See ANSI-Z39.18)   See Instructions on Reverse

OPTIONAL FORM 272 (4-77)
(Formerly NTIS-35)
Department of Commerce

DRAIN-2DX USER GUIDE

SUMMARY

DRAIN-2DX is a computer program for the analysis of inelastic two-dimensional structures under static and dynamic loading, with particular emphasis on seismic response. The program is an extension of, and supersedes, earlier versions of the DRAIN-2D program. The original program has been a valuable inelastic analysis tool for many years, largely because of its simplicity. However, this simplicity has also been a weakness, because the program lacks many desirable features. DRAIN-2DX seeks to correct this weakness, by providing the following capabilities.

(1) Nonlinear static, as well as dynamic, analyses can be performed.

(2) More sophisticated dynamic step-by-step solution strategies may be specified. In particular, (a) the time step can be varied automatically (this is valuable for analyses involving gap closure), and (b) corrections can be applied to compensate for errors in force equilibrium and energy balance.

(3) Energy balance computation can be performed, and detailed logs of energy and equilibrium errors can be obtained.

(4) Static and dynamic loads can be applied in any sequence.

(5) The structure state can be saved permanently at the end of any analyses. A new analysis can then begin from any previously saved state.

(6) Dynamic analyses can be carried out for ground accelerations (all supports moving in phase), ground displacements (supports may move out-of-phase), specified external forces (e.g. wind) and initial velocities (corresponding to an initial impulse).

(7) Mode shapes and frequencies can be calculated, and linear response spectrum analyses can be performed.

(8) Cross sections can be specified through a structure, and the resultant normal, shear and overturning effects on these sections can be computed.

(9)   Post-processing files can be produced, and an interactive post-processing program has been
developed to permit tabulation of results from these files.

This report is in four sections, as follows:

(1) Section A: User guide for the DRAIN-ANAL analysis program, excluding element user
guides.

(2) Section E: Theory and user guides for a number of elements. More elements will be added
in the future.

(3) Section P: User guide for the DRAIN-POST post-processing program. This program has
limited features, and extensions will be added in the future.

(4) Section X: Instructions for the addition of new elements to DRAIN-ANAL and DRAIN-
POST.

Additional information is contained in the DRAIN-2DX theoretical documentation, which is a
separate report.

DRAIN-2DX is more complex to use than DRAIN-2D. Nevertheless, the basic concept of the
original program is unchanged. The input data is also very similar, and data files prepared for
DRAIN-2D can be used for DRAIN-2DX with only minor changes.

The extended program still has many limitations. The most important of these are the follow-
ing.

(1)   The program is limited to 2D frames. It may be noted, however, that full 3D behavior is much
more complex to model, and vastly more expensive computationally.

(2)   The input data is still in the form of a formatted file (although free form input may be
accepted, depending on the FORTRAN compiler). An interactive preprocessor could be
developed, but this has not yet been done.

(3)   The elements in the element library are simple ones, essentially the same as those developed
for DRAIN-2D. There is a need for more sophisticated elements.

(4)   The post-processing program has no graphics capability. Graphs (such as time histories of ele-

ment response) can easily be produced from the DRAIN-POST tables using standard plotting programs. However, the plotting of deflected shapes will require additional program development.

ACKNOWLEDGEMENTS

SECTION A

DRAIN-ANAL INPUT DATA

DRAIN-2DX USER GUIDE

SECTION A

DRAIN-ANAL INPUT DATA

Table of Contents

1. INTRODUCTION

1.1. GENERAL

DRAIN-2DX is a general purpose computer program for static and dynamic analysis of inelastic plane structures. The program consists of two modules, namely DRAIN-ANAL and DRAIN-POST. DRAIN-ANAL is the analysis module, which consists of a base program and a library of elements. It is executed in batch mode, and produces results files that can be processed by DRAIN-POST. DRAIN-POST is the postprocessing module, which is executed interactively.

This section is a User Guide for DRAIN-ANAL. The User Guide for DRAIN-POST is contained in a separate section.

1.2. REPORT LAYOUT

Chapter 2 briefly reviews the features of DRAIN-ANAL. More complete details are published in separate theoretical documentation. Chapter 3 describes the permanent files which are used and created by the program. Chapter 4 contains instruction for preparation of input data files.

2. SUMMARY OF MODELING AND ANALYSIS PROCEDURES

2.1. STRUCTURE MODEL

The structure is modeled as a 2D assemblage of nonlinear elements connected at nodes. Nodes are identified by number and must be numbered in sequence. Each node can have up to three displacement degrees of freedom, namely horizontal (X), vertical (Y) and rotational (R). Provision is made for degrees of freedom to be deleted or combined, so that the total number of unknowns may be much less than three times the number of nodes. Any degree of freedom can be specified to be completely restrained (i.e to have a zero displacement relative to the ground), in which case no degree of freedom number is assigned. Alternatively, the translational or rotational displacements of any group of nodes may be specified to have identical values, in which case the same degree of freedom number is assigned to all of these displacements.

The behavior of a structural element is determined by its element type number. Refer to the element user guides for the available element types. In any model, the elements must be divided into element groups (currently a maximum of 20 groups is permitted). All elements in a group must be of same type, and all elements of single type will often be included in a single group. However, elements of same type may be divided into separate groups, if desired. Elements must be numbered sequentially within each group, and are identified by group and element number.

The structure mass is assumed to be lumped at the nodes, so that the mass matrix is diagonal. X, Y and R inertias can be defined separately at each node. Stiffness and/or mass dependent viscous damping can be included. Mass dependent damping introduces dampers at the nodes, with damping coefficients equal to αM, where M is the X, Y or R inertia of the node, and α is a factor which can be varied from node to node. Stiffness dependent damping introduces dampers in parallel with the elements, with damping matrices equal to βK_0, where K_0 is the initial elastic stiffness matrix of the element, and β is a factor which can be varied from element group to element group.

A support may be introduced at a supported node by fixing the node relative to the ground (i.e. by deleting degrees of freedom) or by declaring that the node has special spring support elements.

Spring supports must be used at points where ground displacement are to be introduced. They may be used at other support points. Spring support elements are introduced automatically (i.e. there is no corresponding element group). The springs are linear, and essentially rigid.

2.2. ANALYSIS OPTIONS

Analyses of a number of different types can be performed, as follows.

(1) **Static Gravity:** Linear static analysis for any combination of element and nodal loads. The structure must remain linear.

(2) **Other Static:** Nonlinear static analysis for any combination of nodal loads. This option will typically be used to perform lateral load analyses.

(3) **Ground Acceleration:** Nonlinear dynamic analysis for ground motions defined by X, Y and/or R accelerograms.

(4) **Return to Static:** At the end of a dynamic analysis, the structure will generally still be moving, and hence will not be in static equilibrium. This special analysis returns the structure to a static equilibrium state.

(5) **Mode Shapes:** Calculation of mode shapes and frequencies, in the initial elastic state or any later state.

(6) **Response Spectrum:** Linear dynamic analysis for specified X and/or Y response spectra, using mode shapes and frequencies in the initial elastic state only.

(7) **Initial Velocity (Impulse):** Nonlinear dynamic analysis for specified initial nodal velocities. This option can be used to compute response following an impact or to evaluate energy absorption capacity.

(8) **Ground Displacement:** Nonlinear dynamic analysis for ground motions defined by X, Y and/or R displacement records at support points, including different (out of phase) motions at different supports.

(9) **Dynamic Nodal Load:** Nonlinear dynamic analysis for dynamic forces defined by X, Y and/or

R force records on the nodes.

A structure will typically be analyzed for several different loadings, carried out in a number of analysis sessions. In any analysis session, analyses can be performed for any number of *analysis segments*, each segment corresponding to a static load increment or the application of a dynamic load for a period of time. The analysis segments are numbered in sequence, with the unstressed state assigned a segment number of 0. The structure state at the end of any analysis segment can be saved on a permanent file, if desired.

In any analysis session, the initial state for each analysis segment **except the first** is always the state at the end of the preceding segment. The initial state for the **first** analysis segment can be **any previously saved state** (identified by its analysis segment number), not necessarily the latest state. This feature allows unlimited flexibility in choosing load sequences. In particular, static and dynamic segments can be inter-mixed. However, a static gravity segment **must** begin from the unstressed state.

2.3. LOADS

Seven different load types can be specified, as follows:

(1) **Element Loads:** Element loads are defined in terms of *element load patterns*, each of which consists of some pattern of loads applied on the elements rather than nodes. As many patterns as desired may be defined, and new patterns may be added at the beginning of any analysis session. The element loads which make up any pattern are defined element group by element group. The types of load (gravity, prestress, etc.) which are allowed depend on the element type. Some element types do not include provisions for element loads. Element loads can be applied only in gravity analysis segments. The loading for any gravity load segment can be any combination of the available element load patterns.

(2) **Static Nodal Loads:** Static nodal loads are defined in terms of *nodal load patterns*, each of which consists of some pattern of X, Y and/or R nodal loads. As many patterns as desired may be defined, and new patterns may be added at the beginning of any analysis session.

Static nodal loads can be applied in gravity and/or other static analysis segments. The loading for any analysis segment can be any linear combination of the available patterns.

(3) **Ground Accelerograms:** Ground accelerograms can be specified for the X, Y and/or R directions. As many accelerograms as desired may be defined, but a maximum of three can be used in any analysis segment.

(4) **Acceleration Response Spectra:** As many response spectra as desired may be defined, but a maximum of two can be used in any analysis segment.

(5) **Initial Velocities:** Initial velocities are defined in terms of *nodal velocity patterns*, each of which consists of some pattern of X, Y and/or R nodal velocities. As many patterns as desired may be defined, and new patterns may be added at the beginning of any analysis session. The initial velocities for any analysis segment can be any linear combination of the available patterns.

(6) **Ground Displacement Records:** Ground displacement records can be specified for the X, Y and/or R directions. As many records as desired may be defined, but a maximum of three can be used in any analysis segment. It is important to note that ground displacements can be introduced only through spring supports. Out-of-phase effects can be produced either by specifying different records at different support points or by specifying different time delays at different supports. In this second procedure, each displacement record defines a wave train which moves past the structure.

(7) **Dynamic Force Records:** Each dynamic force record is defined by (a) a nodal load pattern of "unit" magnitude, and (b) a record defining how the magnitude varies with time. The nodal load pattern must be a previously defined static load pattern. As many dynamic force records as desired may be defined, but a maximum of three can be used in any analysis segment.

2.4. ANALYSIS PROCEDURE

An analysis segment will typically involve several load steps for a static analysis, and several time steps for a dynamic analysis.

In a static analysis the program selects the load step size automatically, by determining when the next stiffness change (*event*) will occur, and ending the step at that event. The structure stiffness is then reformed, and an analysis is performed for the next step. The analysis segment ends when all of the load has been applied, or when a specified displacement is reached.

In a dynamic analysis the time step may be specified to be constant or variable. If the time step can vary, upper and lower error tolerances to control the step must be specified. The program computes an error measure in each step. If this measure exceeds the upper tolerance in any step, the time step is reduced and the step is repeated. If the measure is less than the lower tolerance for some specified number of steps (typically 2), the time step is increased in the following step. The time step decrease and increase factors can be specified. They will typically be 0.5 and 2.0, respectively.

Other options are also available for dynamic analysis, including (a) event computations within time steps (in which case each step may be divided into substeps) and (b) corrections at the step ends to improve energy balance and/or equilibrium. For details on the dynamic step-by-step strategy, refer to the DRAIN-2DX theoretical documentation.

2.5. SECOND ORDER ANALYSIS

Second order $(P - \Delta)$ effects can be considered if desired. This is done by adding a geometric stiffness matrix to the tangent stiffness matrix for each element, and accounting for second order effects in the resisting force computation. The geometric stiffness is changed at each event in static analysis. It can be kept constant for dynamic analysis, or alternatively allowed to change. For details on second order effects, refer to the DRAIN-2DX theoretical documentation.

2.6. ENERGY BALANCE

Energy calculations can be performed, for both static and dynamic analyses. This calculation accounts for external work on the nodes, elastic-plastic work on the elements, kinetic energy, and damping work. If there is a significant energy unbalance, the analysis results are likely to be inaccurate.

2.7. STRUCTURE SECTIONS

Any number of plane sections may be specified through the structure. their resultants (tangential force, normal force and moment about the center of section) are calculated. This feature can be used, for example, to calculate story shear and overturning moment.

2.8. RESULTS PRINTOUT

Results for selected nodal displacements, element responses and structure section forces can be written to an output file if desired. This file can then be printed. Mode shapes, frequencies and the results of response spectrum analyses are automatically written to the output file.

2.9. POST PROCESSING

As described in the preceding section, results print files can be produced during the analysis if desired. However, results will usually be obtained by executing the DRAIN-POST program, using post-processing files. For the postprocessing procedure, refer to the DRAIN-POST user guide.

3. PERMANENT FILES

A number of permanent files are created for each problem. All files have names of the form YYYYYY.ZZZZ, where YYYYYY is the problem name and ZZZZ is a four character extension. The extensions have the following meanings.

ACCN: A **binary** file containing ground acceleration records.

DISP: A **binary** file containing ground displacement records.

ECHO: A **text** file containing an echo of the input data for the last analysis session. If an echo file exists at the beginning of any analysis session, execution will not proceed. This is to avoid inadvertent over-writing.

FORC: A **binary** file containing dynamic force records.

ELOD: A **binary** file containing element load patterns.

GEOM: A **binary** file containing the X,Y coordinates of all nodes, the node numbers for all elements, and other data. This file is used by DRAIN-POST.

LIST: A **text** file describing all analysis segments which have been run, and all structure states which have been saved, since the current problem was begun.

MODE: A **binary** file containing mode shapes and frequencies.

OUTP: A **text** file containing the results printout for the last analysis session. As with ECHO, an OUTP file must not exist at the beginning of any analysis session. The OUTP file contains an event log, unbalanced load information, and optional node, element, etc. results.

RXXX: A **binary** post-processing file containing results for analysis segment XXX. These files are used for post processing of results by DRAIN-POST. Note that a separate file is produced for each analysis segment.

SLOG: A **text** file containing a log of the last session. This log consists of energy balance and unbalanced load information. As with ECHO, an SLOG file must not exist at the beginning of any analysis session.

SPEC: A **binary** file containing earthquake response spectra.

STAT: A **binary** file containing static nodal load patterns.

SXXX: A **binary** file containing the structure state at the end of analysis segment XXX. These files are used for restarting.

VELO: A **binary** file containing initial velocity patterns.

4. INPUT DATA FILE

The following sections describe the required input data. Except as noted, units must be consistent throughout. This is a guide for the DRAIN-ANAL base program only. User guides for the elements are contained in separate sections.

The input data file name **must** be DRAIN.INP (in upper case letters). An input file can consist of the following sections:

A. PROBLEM INITIATION: Specifies a problem name and the initial state for the analysis session.

B. NODE DEFINITION: Defines nodal coordinates, degrees of freedom, nodal masses, and mass proportional damping coefficients.

C. ELEMENT DEFINITION: Defines element geometry and properties, and stiffness proportional damping factors.

D. LOAD DEFINITION: Defines static element load patterns, static nodal load patterns, ground accelerograms, etc. New loads can be added in each analysis session, if desired.

E. STRUCTURE SECTION DEFINITION: Defines structure sections. These can be changed in each analysis session, if desired.

F. RESULTS PRINT SPECIFICATION: Specifies the results that are to be written to the OUTP file. These can be changed in each analysis session, if desired.

G. ANALYSIS SPECIFICATIONS: Specifies the analyses (analysis segments) to be performed in the current analysis session.

In the following sections, input items are of type A, I or R. Type A is character data (Fortran "A" conversion), type I is integer, and type R is real (Fortran "F" or "E" conversion). This guide assumes formated input. Some versions of the program may allow "free-form" input.

A. PROBLEM INITIATION

A1. PROBLEM STATUS

One line.

Columns	Notes	Variable	Data
1-6(A)	A1	PROBID	Problem name (1 to 6 characters).
8-10(A)	A2	STATUS	The word "new" or "old".
15(I)	A3	KDATA	Data checking code: (a) 1: data checking only. (b) 0: execute. (c) -1: execute if the element data can all be held in memory, otherwise data checking only.
17-64(A)			Problem title (max. 48 characters).

A2. RESTART OPTION

One line. Omit if STATUS (Section A1) is "new".

Columns	Notes	Variable	Data
1-5(I)	A4	NROPT	Restart state number. (a) 0: Start from unstressed state. (b) n: Start from state at end of analysis segment "n".
10(I)	A5	NLOD	Load pattern indicator. (a) 1: New load patterns will be added. Include Section D. (b) 0: No new load patterns. Omit Section D.
15(I)	A6	NSECT	Structure sections indicator. (a) 1: New sections specification. Include section E. (b) 0: Sections specification unchanged. Omit section E.
20(I)	A7	NOUT	Results print indicator. (a) 1: New print specification. Include section F. (b) 0: Print specification unchanged. Omit Section F.

B. NODE DEFINITION

Omit entire Section B if STATUS (Section A1) is "old".

B1.   CONTROL INFORMATION

One line.

Columns	Notes	Variable	Data
1-5(I)		NNODS	Total number of nodes.
6-10(I)		NCNNOD	Number of "control nodes" for which coordinates are specified directly. See Section B2.
11-15(I)		NCDNOD	Number of coordinate generation commands. See Section B3.
16-20(I)		NCDDOF	Number of commands specifying nodes with zero-displacements. See Section B4.
21-25(I)		NCDDIS	Number of commands specifying nodes with identical displacements. See Section B5.
26-30(I)		NCDMS	Number of commands specifying lumped masses. See Section B6.
31-35(I)	B1	NELGR	Number of element groups (max. 20). See section C.
40(I)		KENR	Energy calculation code: (a) 1: perform calculations (b) 0: omit

B2.   CONTROL NODE COORDINATES

NCNNOD lines, one per control node.

Columns	Notes	Variable	Data
1-5(I)	B2		Node number, in any sequence.
6-10(I)			Node number for offset. Default = absolute coordinates.
11-20(R)			X coordinate or offset.
21-30(R)			Y coordinate or offset.

B3. COORDINATE GENERATION COMMANDS

NCDNOD lines, one per command.

Columns	Notes	Variable	Data
1-5(I)	B3		Node at beginning of generation line.
6-10(I)			Node at end of generation line.
11-15(I)			Number of nodes to be generated.
16-20(I)			Node number difference between successive nodes. Default = 1.
21-30(R)			Spacing between nodes: (a) ≥ 1.0: assumed to be actual spacing (b) < 1.0: assumed to be proportion of generation line length (c) 0: equally spaced along generation line (d) < 0: not allowed.

B4. ZERO DISPLACEMENT COMMANDS

NCDDOF lines, one per command.

Columns	Notes	Variable	Data
1-5(I)	B4		Node number, or number of first node in a series of nodes covered by this command.
10(I)	B5	KODZ	Code for X displacement: (a) 0: free. (b) 1: fixed. (c) 2: spring supported (see *Note*).
15(I)			Code for Y displacement.
20(I)			Code for rotation.
21-25(I)			Number of last node in series. Default = single node.
26-30(I)			Node number difference between successive nodes in series. Default = 1.

B5. IDENTICAL DISPLACEMENT COMMANDS

NCDDIS lines, one per command.

Columns	Notes	Variable	Data
5(I)			Displacement type: (a) 1: X displacement (b) 2: Y displacement (c) 3: rotation
6-10(I)	B6		Number of nodes covered by this command (max. 14). See *Note* for more than 14 nodes.
11-80(I)			List of nodes in **increasing numerical order**, in 14I5 format.

B6. LUMPED MASS COMMANDS

NCDMS lines, one per command.

Columns	Notes	Variable	Data
1-5(I)	B7		Node number, or number of first node in a series of nodes covered by this command.
6-15(R)			X mass. May be zero.
16-25(R)			Y mass. May be zero.
26-35(R)			Rotary mass. May be zero.
36-40(I)			Number of last node in series. Default = single node.
41-45(I)			Node number difference between successive nodes in series. Default = 1.
46-55(R)	B8		Modifying factor by which mass values are to be *divided*. Default as follows: (a) First command: Default = 1. (b) Subsequent commands: Default = same as preceding command.
56-65(R)	B9		Mass proportional damping factor (α) for these masses. Default as follows: (a) First command: Default = 0. (b) Subsequent commands: Default = same as preceding command.

C. ELEMENT DEFINITION

NELGR (Section B1) sets of lines, one set per element group. Omit entire Section C if STATUS (Section A1) is "old".

C1. CONTROL INFORMATION

One line.

Columns	Notes	Variable	Data
1-5(I)	C1	KEL	Element type number.
6-10(I)		NMEM	Number of elements in group.
11-15(I)	C2	KEVE	Event calculation code. (a) 1: calculate event factors for elements in this group. (b) 0: suppress event factor calculation.
16-20(I)	C3	KGEM	Second order analysis code. (a) 2: allow geometric stiffness to change at each analysis step. (b) 1: allow geometric stiffness to change only for static analysis. (c) 0: ignore second order effects.
21-30(R)	C4		Initial stiffness damping factor (β).
31-78(A)			Group title (max. 48 characters)

C2. ELEMENT DATA

See separate Element User Guides for details. See Note C5 for generation option.

D. LOAD DEFINITION

Omit entire Section D if STATUS (Section A1) is "old" and NLOD (Section A2) is zero.

D1. CONTROL INFORMATION

One line. See Note D1.

Columns	Notes	Variable	Data
1-5(I)	D2	NNGPAT	Number of static element load patterns.
6-10(I)	D3	NNSPAT	Number of static nodal load patterns.
11-15(I)	D4	NNAREC	Number of ground acceleration records.
16-20(I)	D5	NNSPEC	Number of acceleration response spectra.
21-25(I)	D6	NNVPAT	Number of initial velocity patterns.
26-30(I)	D7	NNDREC	Number of ground displacement records.
31-35(I)	D8	NNDFOR	Number of dynamic force records.

D2. STATIC ELEMENT LOAD PATTERNS

NNGPAT sets of lines, one set per load pattern.

D2(a). Pattern ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Pattern name (1 to 4 characters). Must be unique.
6-10(I)		NGRPL	Number of element groups which are affected.
11-58(A)			Pattern title (max. 48 characters).

D2(b). Element Loads

NGRPL subsets of lines, one subset per affected element group.

D2(b)(i). Control Information

Columns	Notes	Variable	Data
1-5(I)	D10	NGR	Element group number, in any sequence.
6-10(I)	D11	NLOD	Number of element load sets (max. 40)

D2(b)(ii). Element Load Data

The element load set data depends on the element type. See separate Element User Guides for details.

D3. STATIC NODAL LOAD PATTERNS

NNSPAT sets of lines, one set per load pattern.

D3(a). Pattern ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Pattern name (1 to 4 characters). Must be unique.
6-10(I)		NCDLD	Number of commands specifying nodal loads.
11-58(A)			Pattern title (max. 48 characters).

D3(b). Nodal Loads

NCDLD lines, one per command.

Columns	Notes	Variable	Data
1-5(I)	D12		Node number, or number of first node in a series of nodes covered by this command.
6-15(R)			X load (the same on all nodes in the series).
16-25(R)			Y load. Note that the Y direction will normally be positive upwards.
26-35(R)			Moment load (right-hand screw rule about Z axis; hence, counter-clockwise positive as normally viewed).
36-40(I)			Number of last node in series. Default = single node.
41-45(I)			Node number difference between successive nodes in series. Default = 1.

D4. GROUND ACCELERATION RECORDS

NNAREC sets of lines, one set per acceleration record.

D4(a).   Record ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Record name (1 to 4 characters). Must be unique.
6-10(I)		NAVAL	Number of acceleration values.
11-58(A)			Record title (max. 48 characters).

D4(b).   Control Information

One line.

Columns	Notes	Variable	Data
1-10(R)		DT	Time interval. (a) 0.: NAVAL time-acceleration pairs (time followed by acceleration) (b) < 0.: NAVAL acceleration-time pairs (acceleration followed by time) (c) > 0.: NAVAL acceleration values only, assumed at constant interval DT.
11-20(R)	D13		Time scale factor. Default = 1.0
21-30(R)	D14		Acceleration scale factor. Default = 1.0
35(I)			Print code: (a) 0: print record as input (b) 1: print record as scaled (c) 2: no print
37-60(A)			Input format (e.g. (4(f8.5,f12.5))). Default = (8f10.0)
62-73(A)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.

D4(c). Acceleration Values

Enough lines to specify NAVAL accelerations, in the above format.

Columns	Notes	Variable	Data
1-80(R)			Time-acceleration pairs or acceleration values.

D5. ACCELERATION RESPONSE SPECTRA

NNSPEC sets of lines, one set per spectrum.

D5(a).   Spectrum ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Spectrum name (1 to 4 characters). Must be unique.
6-10(I)		NPAT	Number of spectral acceleration values.
11-58(A)			Spectrum title (max. 48 characters).

D5(b). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			Period-frequency code. (a) 0: period-acceleration pairs. (b) 1: frequency-acceleration pairs.
11-15(R)		DT	Period (frequency) interval. (a) 0.: NPAT period(frequency)-acceleration pairs (period(frequency) followed by acceleration). (b) < 0.: NPAT acceleration-period(frequency) pairs (acceleration followed by period(frequency)). (c) > 0.: NPAT acceleration values only, assumed at constant period interval DT.
16-25(R)	D14		Acceleration scale factor. Default = 1.0
30(I)			Print code: (a) 0: print spectrum as input (b) 1: print spectrum as scaled (c) 2: no print
32-55(A)			Input format (e.g. (4(f8.5,f12.5))). Default = (8f10.0)
57-68(A)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.

D5(c). Spectrum Values

Enough lines to specify NPAT period(frequency)-acceleration pairs, in the above format.

Columns	Notes	Variable	Data
1-80(R)			Period(frequency)-acceleration pairs or acceleration values.

D6. INITIAL VELOCITY PATTERNS

NNVPAT sets of lines, one set per load pattern.

D6(a). Pattern ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Pattern name (1 to 4 characters). Must be unique.
6-10(I)		NCDVL	Number of commands specifying nodal velocities.
11-58(A)			Pattern title (max. 48 characters).

D6(b). Nodal Velocities

NCDVL lines, one per command.

Columns	Notes	Variable	Data
1-5(I)	D15		Node number, or number of first node in a series of nodes covered by this command.
6-15(R)			X velocity (the same on all nodes in the series).
16-25(R)			Y velocity. Note that the Y direction will normally be positive upwards.
26-35(R)			Angular velocity (right-hand screw rule about Z axis; hence, counter-clockwise positive as normally viewed).
36-40(I)			Number of last node in series. Default = single node.
41-45(I)			Node number difference between successive nodes in series. Default = 1.

D7. GROUND DISPLACEMENT RECORDS

NNDREC sets of lines, one set per displacement record.

D7(a). Record ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Record name (1 to 4 characters). Must be unique.
6-10(I)		NDVAL	Number of displacement values.
11-58(A)			Record title (0 to 48 characters).

D7(b). Control Information

One line.

Columns	Notes	Variable	Data
1-10(R)		DT	Time interval. (a) 0.: NDVAL time-displacement pairs (time followed by displacement) (b) < 0.: NDVAL displacement-time pairs (displacement followed by time) (c) > 0.: NDVAL displacement values only, assumed at constant interval DT.
11-20(R)			Time scale factor. Default = 1.0
21-30(R)			Displacement scale factor. Default = 1.0
35(I)			Print code: (a) 0: print record as input (b) 1: print record as scaled (c) 2: no print
37-60(A)			Input format (e.g. (4(f8.5,f12.5))). Default = (8f10.0)
62-73(A)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.

D7(c). Displacement Values

Enough lines to specify NDVAL displacements, in the above format.

Columns	Notes	Variable	Data
1-80(R)			Time-displacement pairs or displacement values.

D8. DYNAMIC FORCE RECORDS

NNDFOR sets of lines, one set per record. Each record defines a variation of magnitude (load factor) with time. The load pattern to which the load factor is to be applied is specified later.

D8(a). Record ID

One line.

Columns	Notes	Variable	Data
1-4(A)	D9		Record name (1 to 4 characters). Must be unique.
6-10(I)		NFVAL	Number of magnitude values.
11-58(A)			Record title (0 to 48 characters).

D8(b). Control Information

One line.

Columns	Notes	Variable	Data
1-10(R)		DT	Time interval: (a) 0.: NFVAL time-magnitude pairs (time followed by magnitude) (b) < 0.: NFVAL magnitude-time pairs (magnitude followed by time) (c) > 0.: NFVAL magnitude values only, assumed at constant interval DT.
11-20(R)			Time scale factor. Default = 1.0
21-30(R)			Magnitude scale factor. Default = 1.0
35(I)			Print code: (a) 0: print record as input (b) 1: print record as scaled (c) 2: no print
37-60(A)			Input format (e.g. (4(f8.5,f12.5))). Default = (8f10.0)
62-73(A)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.

D8(c). Magnitude Values

Enough lines to specify NFVAL magnitudes, in the above format.

Columns	Notes	Variable	Data
1-80(R)	D16		Time-magnitude pairs or magnitude values.

E.   STRUCTURE SECTION DEFINITION

Omit entire Section E if STATUS (Section A1) is "old" and NSECT (Section A2) is zero. See

Note E1 for an explanation of structure sections.

E1.   CONTROL INFORMATION

One line.

Columns	Notes	Variable	Data
1-5(I)		NNSSEC	Number of structure sections (max. 20). Default = none.
6-10(I)	E2	NMXCUT	Maximum number of elements cut by any section.
11-15(I)		NNTRNS	Number of different force transformations. See Section E3.

E2.   SECTION DEFINITION

NNSSEC sets of lines, one set per structure section.

E2(a).   Control Information

One line per section.

Columns	Notes	Variable	Data
1-5(I)	E1	NCUT	Number of elements cut by this section. Must not exceed NMXCUT.
6-15(R)			Inclination angle of the section (counter-clockwise from X axis to section axis, in degrees)
16-63(A)			Section title (max. 48 characters).

E2(b). Cut Elements

NCUT subsets of lines, one subset for each element cut by the section.

Columns	Notes	Variable	Data
1-5(I)			Element group number.
6-10(I)			Element number in its group.
11-15(I)			Force transformation number for this element. (See Section E3).
16-25(R)			Distance along section axis from cut point in element to center of section.

E3. FORCE TRANSFORMATIONS

NNTRNS sets of lines, one set for each force transformation.

E3(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)	E3	NCOL	No. of columns in force transformation matrix (no. of rows is always 3)

E3(b). Force Transformation

NCOL lines, one line for each column of the force transformation matrix.

Columns	Notes	Variable	Data
1-10(R)	E4		Transformation coefficient for X-force.
11-20(R)			Transformation coefficient for Y-force.
21-30(R)			Transformation coefficient for Moment.

F. RESULTS PRINT SPECIFICATION

Omit entire Section F if STATUS (Section A1) is "old" and NOUT (Section A2) is zero. This section defines the results which are to be written to the OUTP file. The printout may be bulky. Hence, it will be usual to print results for only a few nodes or elements (or none), and to obtain printed or plotted results using DRAIN-POST.

F1. CONTROL INFORMATION

One line.

Columns	Notes	Variable	Data
1-5(I)		NNHOUT	Number of nodes for X-displacement printout. See Section F2. Default = none; -1 = all nodes; -2 = no change from previous specification.
6-10(I)		NNVOUT	Number of nodes for Y-displacement printout. See Section F3. Default = none; -1 = all nodes; -2 = no change from previous specification.
11-15(I)		NNROUT	Number of nodes for rotation printout. See Section F4. Default = none; -1 = all nodes; -2 = no change from previous specification.
16-20(I)	F1	NNHR	Number of pairs of nodes for relative X-displacement printout. See Section F5. Default = none; -2 = no change from previous specification.
21-25(I)		NNVR	Number of pairs of nodes for relative Y-displacement printout. See Section F6. Default = none; -2 = no change from previous specification.
26-30(I)		NNELTH	Number of elements for which printout are required. Default = none; -1 = all elements; -2 = no change from previous specification.
31-35(I)		NNPSEC	Number of structure sections (max. 20). See section E7. Default = none; -1 = all sections; -2 = no change from previous specification.

F2. NODES FOR X-DISPLACEMENT

As many lines as needed to specify NNHOUT nodes. Omit if NNHOUT is ≤ 0.

Columns	Notes	Variable	Data
1-80(I)			Node numbers. Up to 16 nodes in 16I5 format.

F3. NODES FOR Y-DISPLACEMENT

As many lines as needed to specify NNVOUT nodes. Omit if NNVOUT is ≤ 0.

Columns	Notes	Variable	Data
1-80(I)			Node numbers. Up to 16 nodes in 16I5 format.

F4. NODES FOR ROTATION

As many lines as needed to specify NNROUT nodes. Omit if NNROUT ≤ 0.

Columns	Notes	Variable	Data
1-80(I)			Node numbers. Up to 16 nodes in 16I5 format.

F5. PAIRS OF NODES FOR RELATIVE X-DISPLACEMENT

As many lines as needed to specify NNHR pairs of nodes. Omit if NNHR ≤ 0.

Columns	Notes	Variable	Data
1-80(I)			Pairs of nodes. Up to 8 pairs in 8(2I5) format.

F6. PAIRS OF NODES FOR RELATIVE Y-DISPLACEMENT

As many lines as needed to specify NNVR pairs of nodes. Omit if NNVR ≤ 0.

Columns	Notes	Variable	Data
1-80(I)			Pairs of nodes. Up to 8 pairs in 8(2I5) format.

F7. ELEMENTS

As many lines as needed to specify NNELTH elements. Omit if NNELTH $\leq$ 0.

Columns	Notes	Variable	Data
1-5(I)		NGR	Element group number. (a) Positive: elements are listed as follows. (b) Negative: all elements in this group.
6-80(I)			Element numbers. Up to 15 elements in 15I5 format. Omit if NGR is negative.

F8. STRUCTURE SECTIONS

As many lines as needed to specify NNPSEC sections. Omit if NNPSEC $\leq$ 0.

Columns	Notes	Variable	Data
1-80(I)			Section numbers. Up to 16 sections in 16I5 format.

G. ANALYSIS SPECIFICATION

As many sets of lines as desired. Each set defines a new "Analysis Segment". Any number of analysis segments can be considered in a single analysis session. See Sections G2, etc. for restrictions on the sequencing of analysis segments.

G1. CONTROL INFORMATION

One line.

Columns	Notes	Variable	Data
1-4(A)		ANAL	Analysis type for the next analysis segment. One of the following. (a) **grav** = static gravity. See Section G2. (b) **stat** = other static. See Section G3. (c) **accn** = new ground acceleration. See Section G4. (d) **accr** = resume ground acceleration. See Section G4. (e) **rest** = return to static following a dynamic analysis. See Section G5. (f) **eign** = mode shapes and frequencies. See Section G6. (g) **spec** = response spectrum. See Section G7. (h) **veln** = new initial velocity. See Section G8. (i) **velr** = resume initial velocity. See Section G8. (j) **disn** = new ground displacement. See Section G9. (k) **disr** = resume ground displacement. See Section G9. (l) **forn** = new dynamic force. See Section G10. (m) **forr** = resume dynamic force. See Section G10. (n) **stop** = stop the analysis session.
10(I)	G1	KSOA	Second order analysis code: (a) 0 = ignore second order effects. (b) 1 = consider second order effects.
11-74			Title for analysis segment (max. 48 characters).

This line is followed by data for the analysis type specified. See the following sections.

G2. STATIC GRAVITY

Include only if ANAL (section G1) = **grav**. The analysis always begins with the unstressed state. NROPT (Section A2) must be zero, and this must be the first analysis segment of the current analysis session. The analysis is performed in a single step, and the behavior must be linear.

G2(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)		NELOD	No. of static element load patterns to be applied.
6-10(I)		NNDLD	No. of static nodal load patterns to be applied.
15(I)	G2	ISAVE	Structure state save code (SXXX file). (a) 1: save (at end of analysis segment). (b) 0: do not save.
20(I)	G3		Post processing file code (RXXX file). (a) 1: save results for post processing (one set only, at end of analysis segment). (b) 0: do not save.
25(I)	G4	IOUT	Code for results selected in Section F (OUTP file). (a) 1: print results. (b) 0: do not print.

G2(b). Element Load Patterns

As many lines as needed to specify NELOD patterns.

Columns	Notes	Variable	Data
1-75	G5		Pattern name - pattern multiplier pairs. Up to 5 pairs per line, in (5(a4,1x,f10.0)) format.

G2(c). Nodal Load Patterns

As many lines as required to specify NNDLD patterns.

Columns	Notes	Variable	Data
1-75	G5		Pattern name - pattern multiplier pairs. Up to 5 pairs per line, in (5(a4,1x,f10.0)) format.

G3. OTHER STATIC

Include only if ANAL (Section G1) = stat. The loading for the analysis segment will usually be applied in a number of load steps, controlled by load increment and/or displacement increment. Within each step, an event-to-event solution strategy is used automatically, dividing each step into substeps at each event.

G3(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)		NNDLD	No. of static nodal load patterns to be applied.
6-10(I)	G2	ISAVE	Load step interval for saving structure state. (a) > 0: save state every ISAVE load steps and at end of analysis segment. (b) 0: do not save.
15(I)	G3		Post processing file code. (a) 1: save results for post processing (results are saved every event). (b) 0: do not save.
16-20(I)	G4	IOUT	Load step interval for results selected in Section F. (a) > 0: print every IOUT load steps and at end of analysis segment. (b) 0: do not print (c) -1: print every event.
21-25(I)	G6	IENV	Load step interval for envelopes. (a) > 0: print every IENV load steps and at end of analysis segment. (b) 0: print at end of analysis segment only. (c) < 0: do not print.

G3(b). Nodal Load Patterns

As many lines as required to specify NNDLD patterns.

Columns	Notes	Variable	Data
1-75	G5		Pattern name - pattern multiplier pairs. Up to 5 pairs per line, in (5(a4,1x,f10.0)) format.

G3(c). Step Size Criteria

The load step size is controlled by either a load factor increment or an optional displacement increment, whichever is more critical. If a displacement control is specified, the increment applies to a specified "controlled displacement", which may be either a single displacement or the difference between two displacements.

One line.

Columns	Notes	Variable	Data
1-10(R)			Maximum load factor increment per load step. Default = 1.0.
11-15(I)	G7		Node 1 for controlled displacement (optional). Default = no displacement control (rest of line is ignored).
16-20(I)			Node 2 for controlled displacement (optional - if specified, node 1 displacement minus node 2 displacement is controlled).
25(I)			Displacement direction (1 = X; 2 = Y; 3 = R). Default = 1.
26-35(R)			Maximum displacement increment per load step for controlled displacement. (a) > 0: must increase progressively. (b) < 0: must decrease progressively. (c) Default = positive, no limit.

G3(d). Analysis Termination Criteria

The analysis segment is complete when (a) a maximum load factor is reached, (b) a maximum displacement is reached, or (c) the number of load steps exceeds a specified limit, whichever occurs first. To avoid waste of computing time if the solution flip-flops, additional limits may be placed on the number of events in any step and/or on the number of successive direction changes in the controlled displacement.

One line.

Columns	Notes	Variable	Data
1-10(R)			Maximum load factor. Default = 1.0.
11-20(R)			Maximum absolute value of controlled displacement. Default = no limit.
21-25(I)			Maximum number of load steps. Default = no limit.
26-30(I)			Maximum number of events in any load step (analysis quits if exceeded). Default = 10.
31-35(I)	G8		Maximum number of successive direction changes for load increment (analysis quits if exceeded). Default = 3. Applies only if displacement control is specified.
36-45(R)			Maximum absolute translation (X or Y) at any node. Default = no limit.
46-55(R)			Maximum absolute rotation at any node. Default = no limit.

G4. GROUND ACCELERATION

Include only if ANAL (Section G1) = **accn** or **accr**. The analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

G4(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)	G2	ISAVE	Step interval for saving structure state. (a) > 0: save state every ISAVE time steps and at end of analysis segment. (b) 0: do not save.
10(I)	G3		Post processing file code. (a) 1: save results for post processing (results are saved every time step). (b) 0: do not save.
11-15(I)	G4	IOUT	Step interval for results selected in Section F. (a) > 0: print every IOUT time steps, unless TOUT governs, and at end of analysis segment. (b) Default: ignored.
16-25(R)		TOUT	Time interval for results selected in Section F. (a) > 0.: print at this interval, unless IOUT governs. (b) Default: ignored.
26-30(I)	G6	IENV	Step interval for envelopes. (a) > 0: print every IENV time steps, unless TENV governs, and at end of analysis segment. (b) 0: print at end of of analysis segment only. (c) < 0: do not print.
31-40(R)		TENV	Time interval for envelopes. (a) > 0: print at this interval, unless IENV governs. (b) Default: print controlled by IENV.

G4(a). Control Information (continued)

Columns	Notes	Variable	Data
41-50(R)	G9		Multiplier for β_K damping. Default = 1.0
51-60(I)	G9		Multiplier for α_M damping. Default = 1.0

G4(b). Ground Acceleration Records

One line. Omit if ANAL (Section G1) = accr.

Columns	Notes	Variable	Data
1-4(A)			Name of X acceleration record. Default = none.
6-15(R)			X acceleration multiplier. Default = 1.0
16-19(A)			Name of Y acceleration record. Default = none.
21-30(R)			Y acceleration multiplier. Default = 1.0
31-34(A)			Name of R acceleration record. Default = none.
36-45(R)			R acceleration multiplier. Default = 1.0 Counterclockwise acceleration is positive.
46-55(R)	G10		X coordinate of center of rotation (used only if rotational acceleration record is specified).
56-65(R)			Y coordinate of center of rotation.

G4(c). Analysis Type and Termination Criteria

One line.

Columns	Notes	Variable	Data
5(I)	G11	KAUTO	Time step code: (a) 1: variable time step. (b) 0: constant time step.
10(I)	G12	KEVNT	Event calculation code: (a) 1: consider events within time steps. (b) 0: ignore events.
11-15(I)	G12	MAXEV	Maximum number of events allowed in a time step. (a) n: stop execution if there are more than n events in any step. (b) -n: stop checking for events in current step, but continue execution. (c) Default = no limit.
16-20(I)	G13	KENRC	Velocity correction code: (a) 1: modify velocities (to satisfy energy balance). (b) 0: do not modify velocities. KENRC is ignored if KEVNT = 0.
21-25(I)	G13	KEQBC	Acceleration correction code: (a) 1: modify accelerations (to improve equilibrium). (b) 0: do not modify accelerations.
26-35(R)		DT	Time step: (a) If KAUTO = 0: time step (constant throughout). (b) If KAUTO = 1: initial time step.
36-40(I)		NSTEPS	Maximum number of steps (analysis quits). Default = no limit.
41-50(R)		TIMAX	Maximum time for which response is to be calculated. Default = no limit.
51-60(R)			Displacement (X or Y) at any node at which collapse may be assumed (analysis quits). Default = no limit.
61-70(R)			Rotation at any node at which collapse may be assumed (analysis quits). Default = no limit.

G4(d). Additional Control Information For Variable Time Step

One line. Omit if KAUTO = 0 (Section G4(c)). See Note G11.

Columns	Notes	Variable	Data
1-5(R)		DTMAX	Maximum allowable time step. Default = no limit.
6-10(R)		DTMIN	Minimum allowable time step. Default = 0.
11-15(R)		DTRED	Time step decrease factor. Default = 0.5
16-20(R)		DTINC	Time step increase factor. Default = 2.0
21-25(I)		IICONT	Number of steps before time step will be increased. Default = 2 .
26-35(R)		TOLHIS	Upper static force (mean equilibrium) error tolerance. No default.
36-45(R)		TOLLOS	Lower static force error tolerance. No default.
46-55(R)		TOLHII	Upper inertia force (equivalent impulse) error tolerance. No default.
56-65(R)		TOLLOI	Lower inertia force error tolerance. No default.
66-75(R)		TOLMX.	Maximum static force error tolerance. Analysis quits if exceeded. Default = no limit.

G5. RETURN TO STATIC

Include only if ANAL (section G1) = rest. The preceding analysis segment must have been a dynamic analysis. The analysis is performed in a single load step to bring the structure to a static equilibrium state.

One line.

Columns	Notes	Variable	Data
1-5(I)	G2	ISAVE	Structure state save code (SXXX file). (a) 1: save (at end of analysis segment). (b) 0: do not save.
10(I)	G3		Post processing file code (RXXX file). (a) 1: save results for post processing (one set only, at end of analysis segment). (b) 0: do not save.
11-15(I)	G4	IOUT	Code for results selected in section F (OUTP file). (a) 1: print results (at end of analysis segment). (b) 0: do not print. (b) -1: print every event.
16-20(I)			Maximum number of events in any load step (analysis quits if exceeded). Default = no limit.

G6. MODE SHAPES AND FREQUENCIES

Include only if ANAL (section G1) = **eign**. Frequencies and mode shapes may be calculated for any state of the structure. However. only mode shapes calculated for the unstressed state may be used in response spectrum analyses.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVEC	Maximum number of mode shapes to be calculated. Default = controlled by TMIN.
10(I)		TMIN	Shortest significant period. Mode shapes with periods < TMIN are not calculated. Default = controlled by NVEC.

G7. RESPONSE SPECTRUM

Include only if ANAL (Section G1) = **spec**. This analysis segment can be used to perform linear dynamic analysis. Only mode shapes and frequencies calculated for the unstressed state can be used, and these must have been calculated prior to this analysis segment. Square root of sum of squares (SRSS) combination is used to combine the modal responses.

One line.

Columns	Notes	Variable	Data
1-4(A)			Name of X response spectrum. Default = none.
6-15(R)			X spectrum multiplier. Default = 1.0
16-19(A)			Name of Y response spectrum. Default = none.
21-30(R)			Y spectrum multiplier. Default = 1.0

G8. INITIAL VELOCITY

Include only if ANAL (Section G1) = **veln** or **velr**. The analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

G8(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)		NIVELP	No. of initial velocity patterns to be used.
6-15(R)		EKIN	Initial kinetic energy (optional). Default = patterns as specified in Section G8(b) are used directly.
16-20(I)	G2	ISAVE	Step interval for saving structure state. (a) > 0: save state every ISAVE time steps and at end of analysis segment. (b) 0: do not save.
25(I)	G3		Post processing file code. (a) 1: save results for post processing (results are saved every time step). (b) 0: do not save.
26-30(I)	G4	IOUT	Step interval for results selected in Section F. (a) > 0: print every IOUT time steps, unless TOUT governs, and at end of analysis segment. (b) Default: ignored.
31-40(R)		TOUT	Time interval for results selected in Section F. (a) > 0.: print at this interval, unless IOUT governs. (b) Default: ignored.
41-45(I)	G6	IENV	Step interval for envelopes. (a) > 0: print every IENV time steps, unless TENV governs, and at end of analysis segment. (b) 0: print at end of of analysis segment only. (c) < 0: do not print.

G8(a). Control Information (continued)

Columns	Notes	Variable	Data
46-55(R)		TENV	Time interval for envelopes. (a) > 0: print at this interval, unless IENV governs. (b) Default: print controlled by IENV.
56-65(R)	G9		Multiplier for β_K damping. Default = 1.0
66-75(I)	G9		Multiplier for α_M damping. Default = 1.0

G8(b). Initial Velocity Patterns

As many lines as needed to specify NIVELP patterns. If EKIN = 0, these patterns are combined and used directly. If EKIN > 0, the patterns are combined, then all velocities are scaled to give the specified initial kinetic energy. Omit if ANAL (Section G1) is **velr**.

Columns	Notes	Variable	Data
1-75			Pattern name - pattern multiplier pairs. Up to 5 pairs per line, in (5(a4,1x,f10.0)) format.

G8(c). Analysis Type and Termination Criteria

One line.

Columns	Notes	Variable	Data
5(I)	G11	KAUTO	Time step code: (a) 1: variable time step. (b) 0: constant time step.
10(I)	G12	KEVNT	Event calculation code: (a) 1: consider events within time steps. (b) 0: ignore events.
11-15(I)	G12	MAXEV	Maximum number of events allowed in a time step. (a) n: stop execution if there are more than n events in any step. (b) -n: stop checking for events in current step, but continue execution. (c) Default = no limit.
16-20(I)	G13	KENRC	Velocity correction code: (a) 1: modify velocities (to satisfy energy balance). (b) 0: do not modify velocities. KENRC is ignored if KEVNT = 0.
21-25(I)	G13	KEQBC	Acceleration correction code: (a) 1: modify accelerations (to improve equilibrium). (b) 0: do not modify accelerations.
26-35(R)		DT	Time step: (a) If KAUTO = 0: time step (constant throughout). (b) If KAUTO = 1: initial time step.
36-40(I)		NSTEPS	Maximum number of steps (analysis quits). Default = no limit.
41-50(R)		TIMAX	Maximum time for which response is to be calculated. Default = no limit.
51-60(R)			Displacement (X or Y) at any node at which collapse may be assumed (analysis quits). Default = no limit.
61-70(R)			Rotation at any node at which collapse may be assumed (analysis quits). Default = no limit.

G8(d). Additional Control Information For Variable Time Step

One line. Omit if KAUTO = 0 (Section G8(c)). See Note G11.

Columns	Notes	Variable	Data
1-5(R)		DTMAX	Maximum allowable time step. Default = no limit.
6-10(R)		DTMIN	Minimum allowable time step. Default = 0.
11-15(R)		DTRED	Time step decrease factor. Default = 0.5
16-20(R)		DTINC	Time step increase factor. Default = 2.0
21-25(I)		IICONT	Number of steps before time step will be increased. Default = 2 .
26-35(R)		TOLHIS	Upper static force (mean equilibrium) error tolerance. No default.
36-45(R)		TOLLOS	Lower static force error tolerance. No default.
46-55(R)		TOLHII	Upper inertia force (equivalent impulse) error tolerance. No default.
56-65(R)		TOLLOI	Lower inertia force error tolerance. No default.
66-75(R)		TOLMX	Maximum static force error tolerance. Analysis quits if exceeded. Default = no limit.

G9. GROUND DISPLACEMENT

Include only if ANAL (Section G1) = **disn** or **disr**. The analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

G9(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)		NIDISP	No. of ground displacement records to be applied (max. 3).
6-10(I)	G2	ISAVE	Step interval for saving structure state. (a) > 0: save state every ISAVE time steps and at end of analysis segment. (b) 0: do not save.
15(I)	G3		Post processing file code. (a) 1: save results for post processing (results are saved every time step). (b) 0: do not save.
16-20(I)	G4	IOUT	Step interval for results selected in Section F. (a) > 0: print every IOUT time steps, unless TOUT governs, and at end of analysis segment. (b) Default: ignored.
21-30(R)		TOUT	Time interval for results selected in Section F. (a) > 0.: print at this interval, unless IOUT governs. (b) Default: ignored.
31-35(I)	G6	IENV	Step interval for envelopes. (a) > 0: print every IENV time steps, unless TENV governs, and at end of analysis segment. (b) 0: print at end of of analysis segment only. (c) < 0: do not print.
36-45(R)		TENV	Time interval for envelopes. (a) > 0: print at this interval, unless IENV governs. (b) Default: print controlled by IENV.

(continued on next page)

G9(a). Control Information (continued)

Columns	Notes	Variable	Data
41-50(R)	G9		Multiplier for β_K damping. Default = 1.0
51-60(I)	G9		Multiplier for α_M damping. Default = 1.0

G9(b). Ground Displacement Records

NIDISP pairs of lines, one per record. Omit if ANAL (Section G1) = **disr**.

G9(b)(i). Line 1

Columns	Notes	Variable	Data
1-4(A)			Displacement record name.
10(I)			Displacement direction (1 = X, 2 = Y, 3 = R).
11-20(R)			Displacement scale factor.
21-60(I)			The numbers of up to 8 nodes at which this displacement record is to be applied, in 8I5 format. Important: the value of KODZ, Section B4, **must** be 2 for the specified displacement direction at all of these nodes.

G9(b)(ii). Line 2

Columns	Notes	Variable	Data
1-80(R)			Up to 8 time delay values, one for each node in the preceding line. Must be ≥ 0. This much time elapses before the node begins to be affected by the record.

G9(c). Analysis Type and Termination Criteria

One line.

Columns	Notes	Variable	Data
5(I)	G11	KAUTO	Time step code: (a) 1: variable time step. (b) 0: constant time step.
10(I)	G12	KEVNT	Event calculation code: (a) 1: consider events within time steps. (b) 0: ignore events.
11-15(I)	G12	MAXEV	Maximum number of events allowed in a time step. (a) n: stop execution if there are more than n events in any step. (b) -n: stop checking for events in current step, but continue execution. (c) Default: no limit.
16-20(I)	G13	KENRC	Velocity correction code: (a) 1: modify velocities (to satisfy energy balance). (b) 0: do not modify velocities. KENRC is ignored if KEVNT = 0.
21-25(I)	G13	KEQBC	Acceleration correction code: (a) 1: modify accelerations (to improve equilibrium). (b) 0: do not modify accelerations.
26-35(R)		DT	Time step: (a) If KAUTO = 0: time step (constant throughout). (b) If KAUTO = 1: initial time step.
36-40(I)		NSTEPS	Maximum number of steps (analysis quits). Default = no limit.
41-50(R)		TIMAX	Maximum time for which response is to be calculated. Default = no limit.
51-60(R)			Displacement (X or Y) at any node at which collapse may be assumed (analysis quits). Default = no limit.
61-70(R)			Rotation at any node at which collapse may be assumed (analysis quits). Default = no limit.

G9(d). Additional Control Information For Variable Time Step

One line. Omit if KAUTO = 0 (Section G9(c)). See Note G11.

Columns	Notes	Variable	Data
1-5(R)		DTMAX	Maximum allowable time step. Default = no limit.
6-10(R)		DTMIN	Minimum allowable time step. Default = 0.
11-15(R)		DTRED	Time step decrease factor. Default = 0.5
16-20(R)		DTINC	Time step increase factor. Default = 2.0
21-25(I)		IICONT	Number of steps before time step will be increased. Default = 2 .
26-35(R)		TOLHIS	Upper static force (mean equilibrium) error tolerance. No default.
36-45(R)		TOLLOS	Lower static force error tolerance. No default.
46-55(R)		TOLHII	Upper inertia force (equivalent impulse) error tolerance. No default.
56-65(R)		TOLLOI	Lower inertia force error tolerance. No default.
66-75(R)		TOLMX	Maximum static force error tolerance. Analysis quits if exceeded. Default = no limit.

G10. DYNAMIC NODAL LOADS

Include only if ANAL (Section G1) = **forn** or **forr**. The analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

G10(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)		NDLOD	No. of dynamic force records to be applied (max. 3).
6-10(I)	G2	ISAVE	Step interval for saving structure state. (a) > 0: save state every ISAVE time steps and at end of analysis segment. (b) 0: do not save.
15(I)	G3		Post processing file code. (a) 1: save results for post processing (results are saved every time step). (b) 0: do not save.
16-20(I)	G4	IOUT	Step interval for results selected in Section F. (a) > 0: print every IOUT time steps, unless TOUT governs, and at end of analysis segment. (b) Default: ignored.
21-30(R)		TOUT	Time interval for results selected in Section F. (a) > 0.: print at this interval, unless IOUT governs. (b) Default: ignored.
31-35(I)	G6	IENV	Step interval for envelopes. (a) > 0: print every IENV time steps, unless TENV governs, and at end of analysis segment. (b) 0: print at end of of analysis segment only. (c) < 0: do not print.
36-45(R)		TENV	Time interval for envelopes. (a) > 0: print at this interval, unless IENV governs. (b) Default: print controlled by IENV.

G10(a). Control Information (continued)

Columns	Notes	Variable	Data
41-50(R)	G9		Multiplier for β_K damping. Default = 1.0
51-60(I)	G9		Multiplier for α_M damping. Default = 1.0

G10(b). Dynamic Force Records

As many lines as needed to specify NDLOD records. Up to 4 nodal load patterns may be combined to form a "unit" loading for each record. The record then defines the load magnitude variation with time. Omit if ANAL (Section G1) = **forr**.

Columns	Notes	Variable	Data
1-4(A)			Dynamic force record name.
6-15(R)			Multiplier for dynamic force record values. Default = 1.0.
16-75(A,R)			Static nodal load pattern name and multiplier pairs. Up to 4 pairs in (4(a4,1x,f10.0)) format.

G10(c). Analysis Type and Termination Criteria

One line.

Columns	Notes	Variable	Data
5(I)	G11	KAUTO	Time step code: (a) 1: variable time step. (b) 0: constant time step.
10(I)	G12	KEVNT	Event calculation code: (a) 1: consider events within time steps. (b) 0: ignore events.
11-15(I)	G12	MAXEV	Maximum number of events allowed in a time step. (a) n: stop execution if there are more than n events in any step. (b) -n: stop checking for events in current step, but continue execution. (c) Default = no limit.
16-20(I)	G13	KENRC	Velocity correction code: (a) 1: modify velocities (to satisfy energy balance). (b) 0: do not modify velocities. KENRC is ignored if KEVNT = 0.
21-25(I)	G13	KEQBC	Acceleration correction code: (a) 1: modify accelerations (to improve equilibrium). (b) 0: do not modify accelerations.
26-35(R)		DT	Time step: (a) If KAUTO = 0: time step (constant throughout). (b) If KAUTO = 1: initial time step.
36-40(I)		NSTEPS	Maximum number of steps (analysis quits). Default = no limit.
41-50(R)		TIMAX	Maximum time for which response is to be calculated. Default = no limit.
51-60(R)			Displacement (X or Y) at any node at which collapse may be assumed (analysis quits). Default = no limit.
61-70(R)			Rotation at any node at which collapse may be assumed (analysis quits). Default = no limit.

G10(d). Additional Control Information For Variable Time Step

One line. Omit if KAUTO = 0 (Section G10(c)). See Note G11.

Columns	Notes	Variable	Data
1-5(R)		DTMAX	Maximum allowable time step. Default = no limit.
6-10(R)		DTMIN	Minimum allowable time step. Default = 0.
11-15(R)		DTRED	Time step decrease factor. Default = 0.5
16-20(R)		DTINC	Time step increase factor. Default = 2.0
21-25(I)		IICONT	Number of steps before time step will be increased. Default = 2 .
26-35(R)		TOLHIS	Upper static force (mean equilibrium) error tolerance. No default.
36-45(R)		TOLLOS	Lower static force error tolerance. No default.
46-55(R)		TOLHII	Upper inertia force (equivalent impulse) error tolerance. No default.
56-65(R)		TOLLOI	Lower inertia force error tolerance. No default.
66-75(R)		TOLMX	Maximum static force error tolerance. Analysis quits if exceeded. Default = no limit.

H. DRAIN-ANAL USER GUIDE NOTES

A1.   Each problem must be assigned a unique, 1-6 character name. Several permanent files are created for each problem, with file names of the form YYYYYY.ZZZZ, where YYYYYY is the problem name, and ZZZZ is a 4-character extension identifying the type of data in the file. See Chapter 3 for a list of the extensions.

A2.   When a structure model is first defined, the status is "new". Input data sections B through F must be included. If an existing model is to be analyzed, the status is "old". Input data sections B and C must be omitted. Sections D, E and F may or may not be required, depending on the restart indicators in section A2.

A3.   The option KDATA = -1 is to avoid inefficient execution if there is not enough memory for "in-core" execution. During execution of the program, data is stored for each element to monitor its nonlinear behavior. If the data for all elements can be stored in memory, then the program will execute most rapidly. If there is insufficient memory, the data is blocked, and the blocks must be recalled from disk for each step of the analysis. The additional I/O cost can be substantial, and execution time can be greatly increased. If KDATA = -1 and the program does not execute, it will be necessary to change the following two statements in the MAIN program.

COMMON L(N)

NTST = N

where N = number of integer words of memory to be allocated. On a virtual memory system, it will usually be most efficient to specify NTST sufficiently large to ensure "in-core" execution (i.e. virtual memory paging to disk will usually be more efficient than FORTRAN input-output to disk).

A4.   An analysis session can include several analysis segments. The initial state for each analysis segment except the first is always the state at the end of the preceding segment. For a session with STATUS = "new", the initial state for the first analysis segment is always the unstressed state. For a session with status = "old", the initial state for the first analysis segment can be

any previously saved state (identified by its analysis segment number), not necessarily the latest state. The default initial state is the unstressed state (segment number = 0).

A5. If desired, new load patterns can be defined. These patterns are **added** to any previously defined patterns. Both the new patterns and the previously defined patterns can be used in future analysis segments. Once defined, a pattern can not be deleted.

A6. If desired, new structure sections can be defined. These sections will **replace** any previously defined sections. If the sections are not changed, the most recently defined sections are used.

A7. If desired, the print specification can be changed for the new analysis session. If the specification is not changed, the most recent specification applies.

B1. For input and output, the elements must be divided into groups. All elements in any group must be of the same type, and typically all elements of any type will be included in a single group. However, elements of the same type may be divided into separate groups if desired (for example, to separate the elements for output purposes, to allow different amounts of damping to be specified, etc.).

B2. For "control nodes", absolute coordinates or offsets must be defined, in the global X, Y system. The coordinates of any remaining nodes may be generated (using straight line generation commands, Section B3).

B3. The number of nodes generated by each command may be one or more. For each command, the coordinates of the nodes at the beginning and at the end of the generation line must have been previously defined, either as control nodes or by previous straight line generation.

B4. Each node of the structure may have up to three degrees of freedom, namely X displacement, Y displacement, and rotation. Initially, the program assumes that all three degrees of freedom are present at all nodes (code = 0). If this assumption is correct, Section B4 should be omitted. In most cases, however, either (a) certain nodes will be fixed in certain directions or (b) it may be reasonable to assume zero displacement. Section B4 is then required, to specify a nonzero code for all fixed degrees of freedom.

B5.   A code = 1 means that the displacement magnitude is identically zero (the displacement is not

a structure degree of freedom). A code = 2 means that the displacement is restrained by a

very stiff support spring (which is added automatically) but is not identically zero. The

difference is significant only for **dynamic ground displacement analyses** (disn or disr

analysis type, Section G). For these analyses, code = 1 means zero **absolute** displacement,

whereas code = 2 means essentially zero displacement **relative to the ground**. Support

points through which ground displacement will be introduced into the structure must have

code = 2, not code = 1.

B6.   It may be desired to specify that certain nodes displace identically in certain directions. Ident-

ical displacements may be specified by the commands of this section. Up to 14 nodes can be

covered by one line. If more than 14 nodes are to be assigned identical displacements, use

two or more lines, with the nodes in increasing numerical order in each line, and with the

smallest numbered node common to all lines.

B7.   The structure mass is assumed to be lumped at the nodes. The masses can generally be input

with only a few commands. Masses can be specified separately for X translation, Y transla-

tion and rotation. If a node appears more than once in these commands, the mass is the **sum**

of those specified in the separate lines. If certain nodes are constrained to have an identical

displacement, the mass associated with the displacement will be the sum of the masses

specified for the individual nodes.

B8.   The masses are required in units of **mass**, not **weight**. The modifying factor can be used to

convert from weight units to mass units (e.g. for lengths in inch units, the factor would be

386.4). Note that the factor needs to be included in only the first line.

B9.   If desired, a different damping factor (α) can be specified for each node. Typically, the same

factor will be used for all nodes, and this factor then needs to be specified in only the first

line. If a node appears more than once in these commands, the α value is the **sum** of those

specified on the separate lines. If certain nodes are constrained to have an identical displace-

ment, the αM value associated with the displacement will be the **sum** of those for the

individual nodes.

C1.   Each element type in the element library is identified by a type number. For example, element type 1 is a truss element and element type 2 is a beam column element. See the separate element user guides for details. All elements in a group must be of same type. Element groups may be arranged in any convenient order. Within any group, the elements must be numbered in sequence, beginning with 1.

C2.   An event-to-event solution strategy is automatically used for static analysis, and can be specified as an option for dynamic analysis. In this strategy, the structure stiffness is updated every time there is a significant change in stiffness for any element (i.e. an "event"). The option KEVE = 0 suppresses the event calculations for all elements in this group, for both static and dynamic analyses. In general this should be done only for elements which are specified to be very strong in order to force them to remain linear. Significant computer time can be saved, since event factors are not calculated for these elements.

C3.   Second order effects will typically be negligible in girders, but may be important for columns. These effects are included in the analysis by adding geometric stiffnesses. The option KGEM = 1 allows the geometric stiffness to change progressively during static analysis but keeps it constant for dynamic analysis. This will often be accurate for building frames with second order effects in only the columns, since the story geometric stiffness depends on the sum of the axial forces in all columns at the story, which is constant if there is no vertical earthquake. Selecting KGEM = 1 can save substantial computing time. The option KGEM = 2 allows the geometric stiffness to change progressively during both static and dynamic analyses. If desired, second order effects can be suppressed, even though KGEM is nonzero for some or all element groups (see KSOA, Section G1).

C4.   Each element has a constant viscous damping matrix equal to βK_0, where K_0 is the initial elastic stiffness matrix of the element (not including geometric stiffness). The value of β is same for all elements in any group, but can vary from group to group (e.g to allow different damping to be specified for different materials). Earlier versions of DRAIN have allowed

viscous damping based on the **tangent** stiffness matrix. This is no longer allowed in DRAIN-2DX.

C5. The elements in each group must be specified in increasing numerical order, starting with 1. One data line may be provided for each element, in which case element data will not be generated. Alternatively, the lines for groups of element may be omitted, in which case the data for each missing group is usually generated as follows. Some elements may be different. If so, this will be noted in the element user guide.

1) All elements are assigned the same stiffnesses, strengths, etc. as the element **preceding** the missing group.

2) The node numbers for each missing element are obtained by adding the specified node number increment to the node numbers for the preceding element. The node number increment is that specified with the element **preceding** the missing group.

In the echo print of element data, generated elements are identified by asterisks.

D1. Any number of new load patterns, of any type, can be defined in any analysis session. New patterns are **added** to any previously defined patterns. Once defined, a pattern can not be deleted. The same applies for dynamic records and response spectra. The actual loadings for each analysis segment are defined later (Section G).

D2. Element loads will typically account for gravity loads (dead or live) acting along element lengths. However, they may also account for thermal effects, or any other effects which originate within elements. Element loads can be applied only in "static gravity" analysis segments. These must begin with the unstressed state, and nonlinear behavior is not permitted.

D3. Static nodal loads can account for gravity and/or lateral forces. There are no restrictions on the sequences in which static nodal loads can be applied, or on whether or not the structure remains linear.

D4. Ground acceleration records are used for dynamic analyses with all supports moving in phase. There are no restrictions on the sequences in which ground accelerations are applied, or on whether or not the structure remains linear.

D5. Acceleration spectra can be used to perform linear analyses only. Mode shapes and frequencies must first have been calculated (see Section G6) for the **unstressed** structure state.

D6. Initial velocity patterns can be used to test the energy absorbing capacity and/or impact response of a structure. Initial velocities corresponding to a known kinetic energy or a set of known impulses can be defined. With the velocities as initial conditions, a dynamic analysis can be performed. The analysis must begin from a static, but not necessarily unstressed, state.

D7. Ground displacement records are used for dynamic analyses with supports moving out of phase. There are no restrictions on the sequences in which ground displacements are applied, or on whether or not the structure remains linear. In-phase support motions can be considered as a special case.

D8. Dynamic force records are used for dynamic analyses with time varying forces on the nodes. There are no restrictions on the sequences in which dynamic forces are applied, or on whether or not the structure remains linear.

D9. A load pattern is identified by its type (e.g. static element load) and its 4-character name. The name must be unique within its load pattern type, but may be re-used if desired for a different type.

D10. Because the nature of the loading and its effect on an element depend on the element type, different input data is needed for each element type. Some element types do not have provision for element loads. Refer to the element user guides for details.

D11. An element "load set" will typically correspond to some distribution of load along an element (e.g. uniform load over the length of a beam element). The element load **pattern** is defined by specifying which load sets act on which elements. In any pattern, several load sets may act on a single element, and the load set magnitudes may be scaled. Refer to the element user guides for details.

D12. The static nodal loads for any pattern can usually be input with only a few lines. Any node may appear in more than one line, in which case, the loads on the node are the sums of those

specified in the separate lines. If certain nodes are constrained to have identical displacements, the loads associated with those displacements will be sums of the loads specified for the individual nodes.

D13. The input times are **multiplied** by this factor (f_t). The effect is to increase the ground velocities by f_t, to increase the ground displacements by f_t^2, and to alter the frequency content of the earthquake. Time should not be scaled without carefully considering the influence on the ground motion.

D14. The input accelerations are **multiplied** by this factor. The factor will normally be used to convert from multiples of gravity to acceleration units. Further scaling can be specified when a time history analysis is actually performed.

D15. If a node appears in more than one data line, the velocities for the node will be the sums of those specified in the separate lines. If certain nodes are constrained to have identical displacements, the velocities associated with those displacements will be sums of the velocities specified for the individual nodes.

D16. If a node appears in more than one data line, the magnitudes for the node will be the sums of those specified in the separate lines. If certain nodes are constrained to have identical displacements, the magnitudes associated with those displacements will be sums of the magnitudes specified for the individual nodes.

E1. The procedure for defining a structure section is as follows:

(1) Draw a straight line through the structure. Identify the elements which are cut by the line and the locations of the cuts. Identify an axis along the section, and determine the angle from the X axis to this section axis.

(2) Define the "center" of the section. The section moment will be the moment about this point.

(3) Calculate the distance from each element cut to the center of the section, measured along the section axis. All distances are measured **from** the element **to** the center of the

section, positive in the positive direction of the section axis.

(4) At each element cut there will be, for any state of the structure, a set of element forces. The section normal force, shear force and moment are obtained by (a) transforming these element forces to the section center, and (b) summing over all elements. The forces at a cut in an element depend on the element end forces (or the nodal forces for a solid element) and the location of the cut in the element. Once the forces at the cut have been calculated, the program automatically transfers them to the section center and sums them to get section forces. However, the forces at the cut are **not** determined automatically. Instead, a force transformation which gives the forces at the cut in terms of the forces at the element ends, must be **input** (see Section E3(b)). The size of the transformation matrix for any element is 3*NDOF, where NDOF is the number of element degrees of freedom (e.g., for a beam-column element the size of the transformation is 3*6). Usually the transformation will be the same for several elements, and only a small number of transformations will need to be input.

E2. This is required for memory allocation. The value of NCUT, Section E2(a), must not exceed this value for any section.

E3. NCOL must equal the number of degrees of freedom (stiffness matrix size) for the element type to which the transformation applies. The program does **not** check for consistency. If the transformation size is incorrect, the section forces will be incorrect.

E4. The transformation must transform nodal forces (X force, Y force and moment) acting on the element to forces (X,Y,R) acting on the element section at the cut. Elements will typically be cut at nodes, in which case the transformation coefficients will be either unity or zero. Since there are two sides to each element cut, the signs of the transformation coefficients will depend on which side is assumed to be retained and which is removed.

F1. The relative displacement is the displacement of the first node of each pair minus the displacement of the second node. Take care with story drifts, because the relative horizontal displacement between two floors may not be a true measure of damage-inducing drift (some of it

may be due to column shortening and extension). To obtain a better measure, insert a dummy shear panel in each floor, and monitor the shear deformation.

G1. This code is considered only if the analysis segment **begins from the unstressed state**. For analysis segments which begin from any other state, the code of the preceding segment is automatically used (i.e. once turned on, second order effects can not be turned off, and vice versa). Second order effects are considered only for element groups with KGEM greater than zero (see Section C1).

G2. The structure state may be saved at the end of any analysis segment. The state is saved in a file with a name of the form YYYYYY.SZZZ, where YYYYYY is the problem name, S indicates a state file, and ZZZ is the analysis segment number. Analysis segments are numbered in sequence, regardless of whether or not they are saved.

For nonlinear static and dynamic analyses, the state may also be saved at loads or times within an analysis segment. If this is done, a new file is not created each time the state is saved. Instead a new file is created the **first** time that the state is saved, but at each **subsequent** time that file is rewound and over-written. Hence, if the analysis segment runs to completion, the file will contain the state at the end of segment. If, however, execution is terminated for some reason before the segment is completed, the file will contain the most recently saved state.

G3. A file with a name of the form YYYYYY.RZZZ may be created for post processing of the results by DRAIN-POST, where YYYYYY is the problem name, R indicates a post processing result file,' and ZZZ is the analysis segment number. This file contains results for all nodes, all elements and all structure sections. For dynamic analysis, results are saved every time step. Hence, the file can be very large. It may be necessary to run (and post process) a large problem in several short analysis segments.

G4. Results for selected nodal displacements, elements and structure sections (see Section F) may be written to the OUTP file for each analysis segment. The results are written every IOUT load or time steps. To obtain results at the end of the analysis segment only, input a very

large value for IOUT.

G5. Each pattern is multiplied by its multiplier, and the patterns are then combined. The combined patterns define a loading with a **load factor equal to one.** For a static gravity analysis the loading can be any combination of element and nodal load patterns, and the full loading is applied in a single step. For other static analyses the loading can be a combination of nodal load patterns only, and the loading will generally be applied in steps. The load factor increment in each step will be governed by one or more criteria (see Section G3), and the final load factor will not necessarily equal one.

G6. Envelope values include maximum positive and negative values for all node displacements, selected response quantities for all elements (varies with element type) and all structure section forces. The time or load factor at which the maximum occurred is also output for each value.

G7. Displacement control is particularly useful if the structure becomes very flexible, and is essential if the structure is to be loaded into the post buckling range. If displacement control is not used and the structure becomes unstable, a positive load increment may cause a negative displacement, leading to a to flip-flop condition (successive unloading-reloading). The controlled displacement can be used to force the structure to continue displacing in the positive (or negative) direction, even though the load decreases. This type of control can be very effective, but will not always work. Hence, additional flip-flop controls (Section G3(d)) are desirable to avoid waste of computing time. If there is no displacement control, the analysis will stop if the structure becomes unstable.

G8. Even if a displacement control is specified, the solution can flip-flop. This will be indicated by successive changes in the sign of the load increment required to make the controlled displacement increase progressively. This flip-flop limit can avoid waste of computing time.

G9. The stiffness proportional and mass proportional damping factors may be either increased or decreased. The multipliers apply for all elements and masses. This feature can be used to study the effect of damping on the structure response without redefining the problem.

G10. The center of rotation is the point around which the foundation rotates (assuming rigid body rotation at the foundation level). This is required to determine the effective dynamic loads on the structure due to ground rotation.

G11. Four dynamic solution schemes are available, namely constant time step with and without event calculation, and variable time step with and without event calculation. In the variable time step scheme, the time step is controlled by upper and lower tolerances on the "static force" and "inertia force" errors (see DRAIN-2DX theory report). If either TOLHIS or TOLHII (see Section G4(d)) is exceeded in a step, the time step is decreased by multiplying the current time step by DTRED, and the step is **repeated**. The time step is not reduced below a specified minimum value (DTMIN). If both errors are less than their lower tolerances for IICONT consecutive steps, the time step is multiplied by DTINC (for the **next** time step). The time step is not increased above a specified maximum value (DTMAX). DTMAX must be chosen to ensure that ground accelerations or other dynamic loads are accurately represented. See DRAIN-2DX theory report for guidance in selecting tolerances.

G12. Events are not necessarily calculated for all element groups (see KEVE, Section C1). In a solution scheme with event calculations, a large number of events in a single time step may indicate a flip-flop situation. If MAXEV is exceeded, the analysis may be terminated or it may be continued, omitting calculation of any further events. Termination is usually recommended, since a large number of events in a time step usually indicates too long a time step or other problems.

G13. In a step-by-step dynamic analysis, energy balance and/or equilibrium may not be satisfied at the end of a time step. Energy balance can be satisfied by modifying the velocities at the end of a time step, and equilibrium can be satisfied by modifying the accelerations. These corrections will usually improve the accuracy. For more details, see the DRAIN-2DX theory report.

SECTION E

ELEMENT THEORY AND INPUT DATA

DRAIN-2DX USER GUIDE

SECTION E

ELEMENT THEORY AND INPUT DATA

Table of Contents

SUMMARY

Several simple elements were available for the original DRAIN-2D, and were described in the report "General Purpose Computer Program for Inelastic Dynamic Response of Plane Structures" by A.E. Kanaan and G.H. Powell (EERC 73-6, Earthquake Engineering Research Center, University of California, Berkeley, 1973). The following of these elements have been modified for DRAIN-2DX:

Type 01: Truss Element.

Type 02: Beam-Column Element.

Type 03: Infill Panel Element.

Type 04: Semi-Rigid Connection Element.

Type 07: Beam Element with Degrading Stiffness.

The last of these was described in part 1 of the Ph.D. thesis "A Contribution to the Analysis of Concrete Structures Under Cyclic Loading" by R.W. Litton (Dept. of Civil Engineering, University of California, Berkeley, 1975).

Several element types for modeling panels and joints in large panel buildings were described in the report: "Inelastic Seismic Analysis of Large Panel Buildings" by V. Schricker and G. H. Powell (UCB/EERC-80/38, Earthquake Engineering Research Center, University of California, Berkeley, 1980). These elements were implemented in a modified version of the original DRAIN-2D. The following ones have been modified for DRAIN-2DX:

Type 05: Gap-Friction Joint Element.

Type 06: Modified Beam Panel Element.

Type 09: Link Element.

Type 10: Key Joint Element.

Type 11: Degrading Friction Joint Element.

Type 12: Shear Friction Joint Element.

This section describes the theory for these eleven elements, using material from the above reports. A few other elements have been developed for DRAIN-2D over the years, but have not yet been modified for DRAIN-2DX.

It may be noted that it was not intended that the original DRAIN-2D elements would be sufficient for all purposes. Rather, it was intended for DRAIN to be a base program, to which researchers and practitioners could add elements to suit their own purposes (e.g. the Schricker large panel elements). The elements in the original program were very simple ones, and it is recognized that they are deficient in may respects. In particular, more sophisticated elements are needed for modeling reinforced concrete beams, columns, walls and connections. In a similar vein, DRAIN-2DX is not intended to be a complete program, but a base program which will need additions to its element library.

When new elements are developed, and also when DRAIN is used to perform analyses, it is important to remember that the accuracy with which a mathematical model can represent the nonlinear response of a structure depends on several properties of the model. These include stiffness, strength, energy absorption, and cyclic loading characteristics. Tangent stiffness is the most fundamental property, since if tangent stiffness is modeled accurately, then all other properties are automatically accurate. However, experience suggests that if the strength and energy absorption properties are modeled with reasonable accuracy, then sufficiently accurate results can be obtained for many engineering purposes, even if the tangent stiffness values are not accurate. It should also be recognized that the true nonlinear properties of structural members are rarely known with any degree of accuracy, so that truly accurate modeling is rarely possible. That is, there is no substitute for engineering judgement.

DRAIN-2DX USER GUIDE

ELEMENT THEORY

TRUSS ELEMENT (TYPE 01)

E01.1 GENERAL CHARACTERISTICS

Truss elements may be oriented arbitrarily in the XY plane, but can transmit axial load only. Two alternative modes of inelastic behavior may be specified, namely (a) yielding in both tension and compression (Fig. E01.1a) and (b) yielding in tension but elastic buckling in compression (Fig. E01.1b). Strain hardening effects are included by dividing each element into two parallel components, one elastic and one inelastic (Fig. E01.2). Element loads and second order effects can be included.

E01.2 ELEMENT DEFORMATIONS

The only element deformation is axial extension, v. The displacement transformation relating increments of deformation and displacement (Fig. E01.3) is

$$dv = \left[\begin{array}{cccc} \dfrac{-X}{L} & \dfrac{-Y}{L} & \dfrac{X}{L} & \dfrac{Y}{L} \end{array} \right] \left\{ \begin{array}{c} dr_1 \\ dr_2 \\ dr_3 \\ dr_4 \end{array} \right\} \tag{E01.1}$$

or

$$dv = \mathbf{a} \ \mathbf{dr} \tag{E01.2}$$

where X, Y and L are the projections and element length in the undeformed state.

The measure of inelastic deformation is the extension beyond yield of the inelastic component of the element.

E01.3 STATIC ELASTO-PLASTIC STIFFNESS

The static elasto-plastic stiffness in terms of deformation is

$$dS = \frac{E_T \, A}{L} \, dv \tag{E01.3}$$

or

$$dS = \mathbf{k}_{\text{ep}} \, dv \tag{E01.4}$$

in which E_T = tangent modulus in current state, A = cross section area, and L = undeformed length. Hence the elasto-plastic stiffness in terms of nodal displacements is

$$\mathbf{K}_{\text{ep}} = \mathbf{a}^T \, \mathbf{k}_{\text{ep}} \, \mathbf{a} \tag{E01.5}$$

E01.4 GEOMETRIC STIFFNESS

The geometric stiffness in terms of rigid body rotation (Fig. E01.4) is

$$dM_G = \frac{S}{L} \, d\theta \tag{E01.6}$$

in which S = current axial force and L = undeformed length. The transformation relating rigid body rotation and nodal displacement is

$$dv = \left[\begin{array}{cccc} \dfrac{-Y}{L} & \dfrac{X}{L} & \dfrac{Y}{L} & \dfrac{-X}{L} \end{array} \right] \, \mathbf{dr} \tag{E01.7}$$

or

$$dv = \mathbf{a}_\theta \, \mathbf{dr} \tag{E01.8}$$

Hence the geometric stiffness is

$$\mathbf{K}_G = \mathbf{a}_\theta^T \, \frac{S}{L} \, \mathbf{a}_\theta \tag{E01.9}$$

E01.5 DYNAMIC STIFFNESS

If stiffness dependent (βK) damping is specified, then in effect a viscous damping component is added in parallel with the elasto-plastic component. The axial force in this damping component is

$$S_d = \beta \frac{EA}{L} \, \dot{v} \tag{E01.10}$$

in which $\dot{v}$ = rate of change of extension and E = elastic modulus.

For dynamic analysis, the damping component contributes to both the left hand side

(effective stiffness) and right hand side (effective load) of the step-by-step equilibrium equations. The equations are given in the DRAIN-2DX theoretical documentation, and are not repeated here.

A damping geometric stiffness is not considered (i.e., the geometric stiffness is based only on the elasto-plastic axial force, S, not on $S + S_d$).

E01.6 RESISTING FORCE

The element resisting force is used to calculate equilibrium unbalance (refer to the DRAIN-2DX theoretical documentation). This resisting force is

$$\mathbf{R}^1 = \mathbf{a}^T (S + S_d) + \mathbf{a}_\theta^T \frac{S}{L} \mathbf{a}_\theta \ r \qquad \text{(E01.11)}$$

where the term containing $[\frac{S}{L}]$ applies only when second order effects are included. Note that since the value of S used in Eqn. (E01.11) is at the end of a step, it will generally be different from that used for $\mathbf{K}_G$, which is at the beginning of the step. Hence, the second order effect can cause an equilibrium unbalance even if $\mathbf{K}_{ep}$ is constant.

E01.7 ELEMENT LOADS

Static loads applied along the lengths of truss elements may be taken into account by specifying end clamping forces as shown in Fig. E01.5. These forces are those which must act *on the element ends* to prevent end displacements. The fixed end forces for any element contribute to the static loads (gravity load cases only) on the nodes to which the element connects. Element loads do not produce element initial forces (i.e. in effect, the element loads are simply lumped at the element ends).

E01.8 ACCUMULATED PLASTIC DEFORMATIONS

The computer response includes both maximum and "accumulated" plastic deformations. The accumulated values are defined by Fig. E01.6.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.01

TRUSS ELEMENT (TYPE 01)

See Fig. E01.1 through E01.6 for element behavior and properties.

C2.01(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See Section C2.01(b).

C2.01(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Young's modulus.
16-25(R)			Strain hardening ratio, as a **proportion** of Young's modulus.
26-35(R)			Cross section area.
36-45(R)			Yield stress in tension.
46-55(R)			Yield stress or elastic buckling stress in compression. See Fig. E01.1
60(I)			Code for compression behavior. (a) 1: Element buckles elastically in compression. (b) 0: Element yields in compression without buckling.

C2.01(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included.**

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element at end i.
11-15(I)			Node number at element at end j.
16-20(I)			Node number increment for element generation. Default = 1.
21-25(I)			Stiffness type number.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION D2(b)(ii).01

ELEMENT LOAD DATA FOR TRUSS ELEMENT (TYPE 01)

D2(b)(ii).01(a). Load Sets

NLOD lines (see section D2(b)(i)), one line per element load set. See Fig. E01.5.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
6-10(I)			Coordinate code. (a) 0: Forces are in local (element) coordinates. (b) 1: Forces are in global (structure) coordinates.
11-20(R)			Clamping force P_i.
21-30(R)			Clamping force V_i.
31-40(R)			Clamping force P_j.
41-50(R)			Clamping force V_j.

D2(b)(ii).01(b). Loaded Elements and Load Set Scale Factors

As many lines as needed. Terminate with a blank line.

Columns	Notes	Variable	Data
1-5(I)			No. of first element in series.
6-10(I)			No. of last element in series. Default = single element.
11-15(I)			Element no. increment. Default = 1.
16-20(I)			Load set number.
21-30(R)			Load set scale factor.
31-45(I-R)			Optional second load set no. and scale factor.
46-60(I-R)			Optional third load set no. and scale factor.
61-75(I-R)			Optional fourth load set no. and scale factor.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

TRUSS ELEMENT (TYPE 01)

Item	Description
1	Node number at end I.
2	Node number at end J.
3	Yield code (1: yielded; 0: not yielded).
4	Axial force, tension positive.
5	Total axial extension.
6	Accumulated positive plastic extension.
7	Accumulated negative plastic extension.
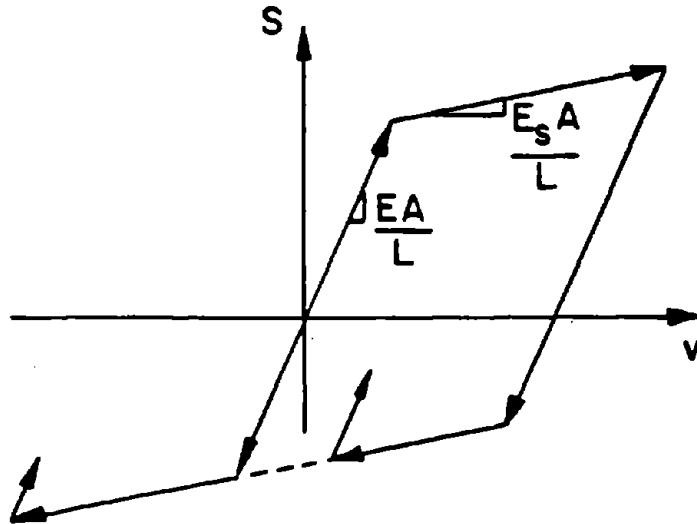
(a) YIELD IN TENSION AND COMPRESSION
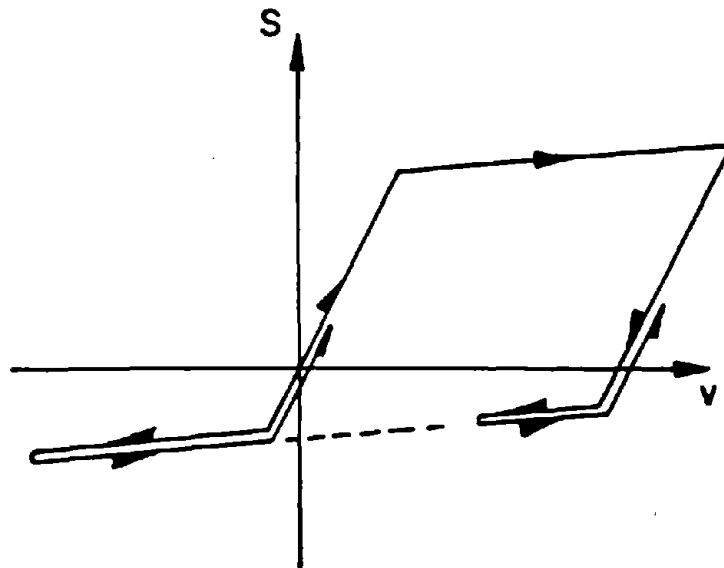
(b) YIELD IN TENSION, BUCKLING IN COMPRESSION

FIG. E01.1 TRUSS ELEMENT BEHAVIOR
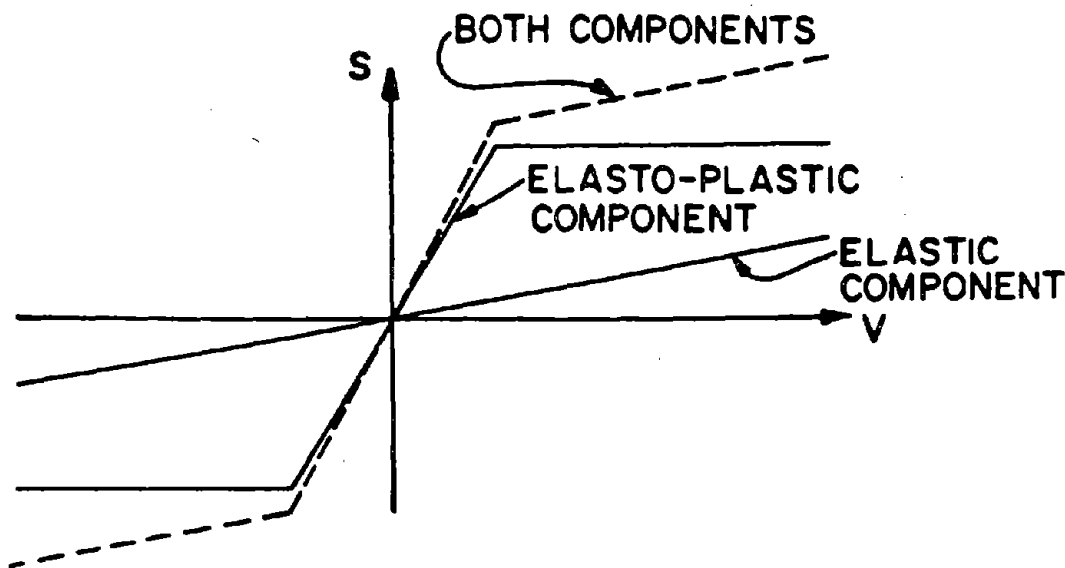
FIG. E01.2 DECOMPOSITION OF BILINEAR
RELATIONSHIP INTO TWO COMPONENTS
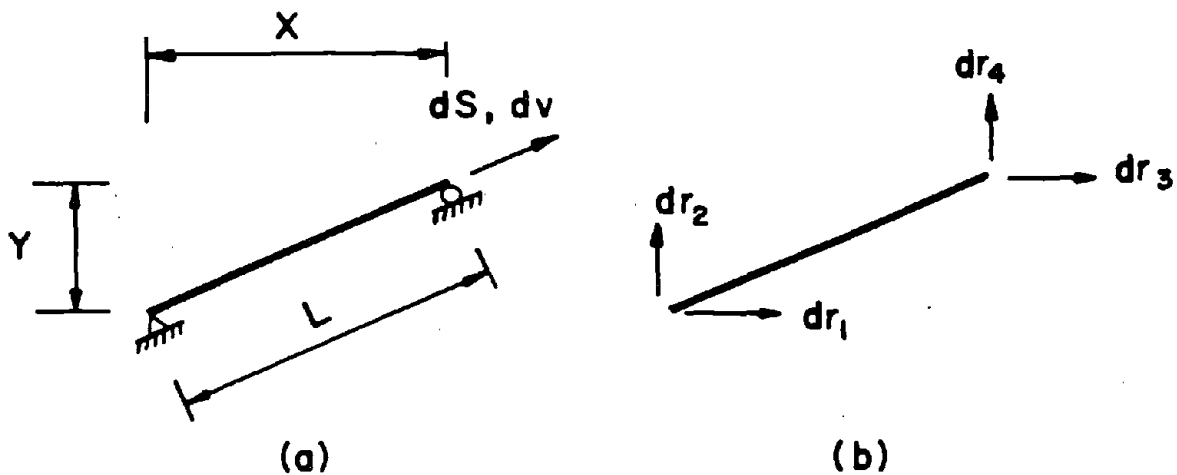
FIG. E01.3 DEFORMATIONS AND DISPLACEMENTS
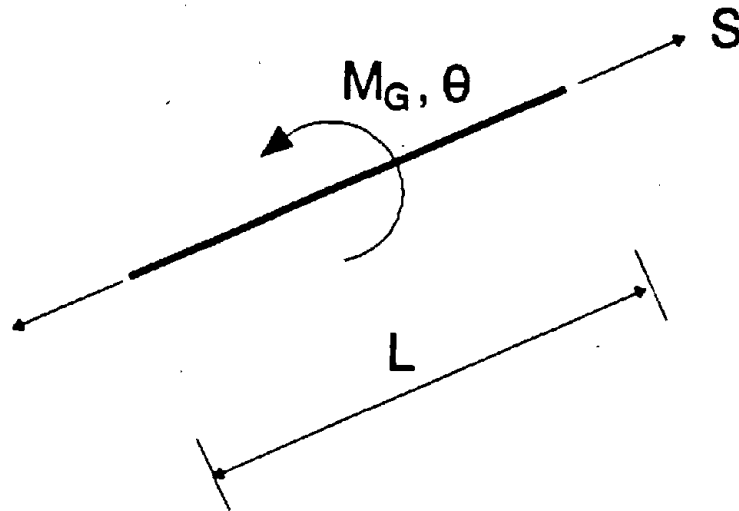
FIG. E01.4 TERMS FOR ROTATIONAL GEOMETRIC STIFFNESS

(a) CODE = 0   (b) CODE = 1
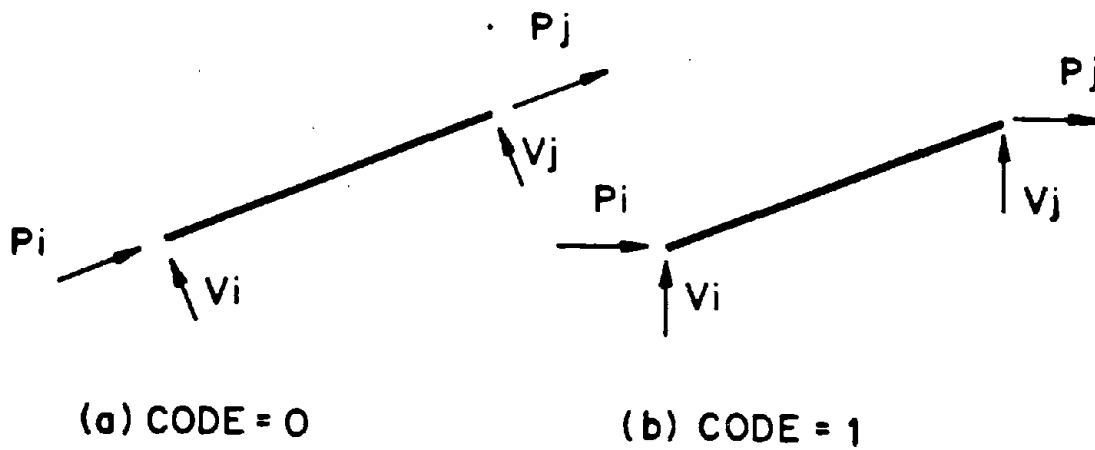
FIG. E01.5 END CLAMPING AND INITIAL FORCES
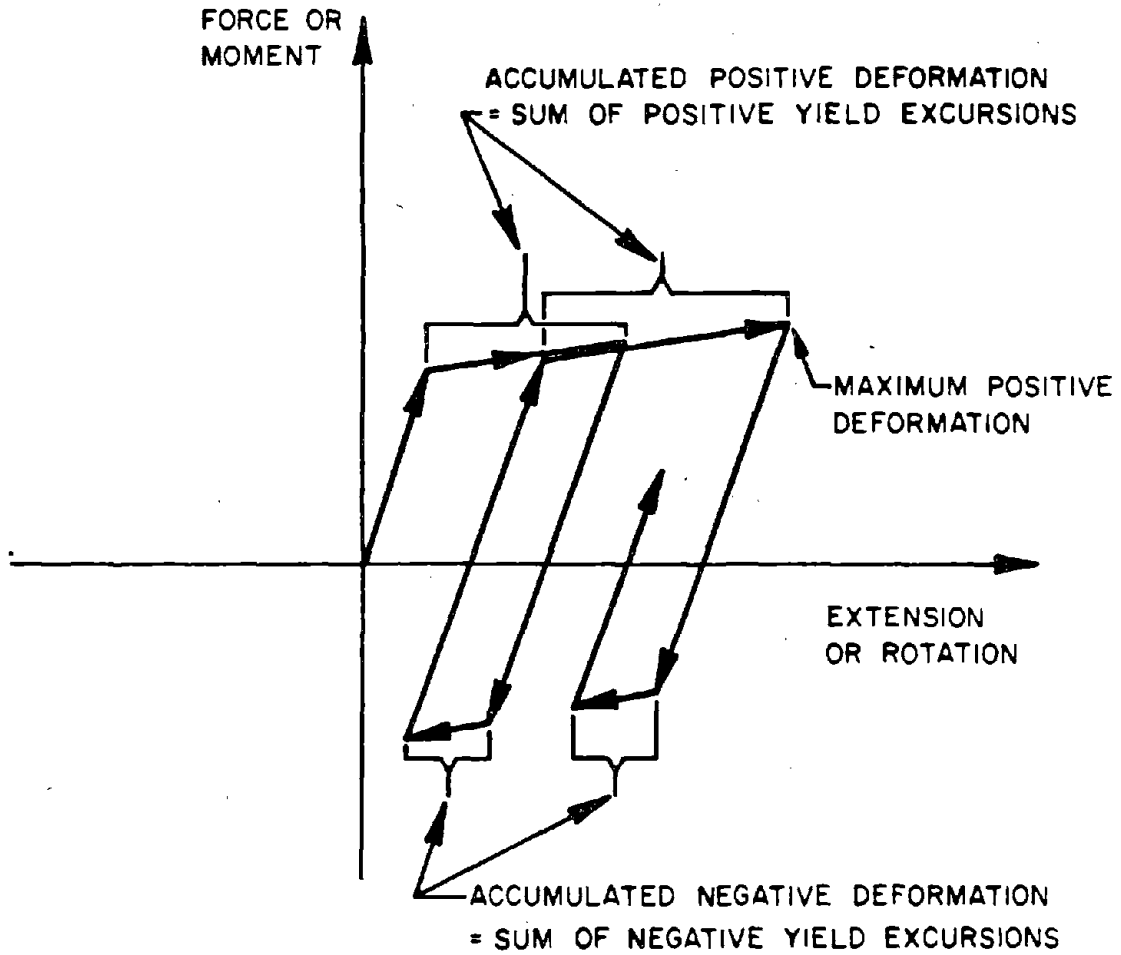
FIG. E01.6 PROCEDURE FOR COMPUTATION OF ACCUMULATED PLASTIC DEFORMATIONS

DRAIN-2DX USER GUIDE

ELEMENT THEORY

BEAM-COLUMN ELEMENT (TYPE 02)

E02.1 GENERAL CHARACTERISTICS

Beam column elements may be oriented arbitrarily in the XY plane. The elements possess flexural and axial stiffness. Elements of variable cross section can be considered by specifying appropriate flexural stiffness coefficients. Flexural shear deformations and the effects of eccentric end connections can be taken into account. Yielding may take place only in concentrated plastic hinges at the element ends. Strain hardening is approximated by assuming that the element consists of elastic and inelastic components in parallel. The hinges in the inelastic component yield under constant moment, but the moment in the elastic component may continue to increase.

With this strain hardening model, if the bending moment on the element is constant, and if the element is of uniform strength, then the moment-rotation relationship for the element will have the same shape as its moment-curvature relationship (Fig. E02.1a). This follows because curvature and rotation in this case are directly proportional. If, however, the bending moment or strength vary, then the curvatures and rotations are no longer proportional, and the moment-rotation and moment-curvature variations may be quite different (Fig. E02.1b). With the parallel component model, a moment-rotation relationship is, in effect, being specified. Care must be taken in relating this to the moment-curvature relationship.

The yield moments may be specified to be different at the two element ends, and for positive and negative bending. The interaction between axial force and moment in producing yield may be taken into account approximately.

Static loads applied along any element length may be taken into account by specifying fixed end force values. Second order effects may be approximated by including a simple

geometric stiffness based on the element axial force.

E02.2 ELEMENT DEFORMATIONS

A beam-column element has three modes of deformation, namely axial extension, flexural rotation at end i, and flexural rotation at end j. The displacement transformation relating increments of deformation and displacement (Fig. E02.2) is

$$
\begin{Bmatrix} dv_1 \\ dv_2 \\ dv_3 \end{Bmatrix} = \begin{bmatrix} \dfrac{-X}{L} & \dfrac{-Y}{L} & 0 & \dfrac{X}{L} & \dfrac{Y}{L} & 0 \\ \dfrac{-Y}{L^2} & \dfrac{X}{L^2} & 1 & \dfrac{Y}{L^2} & \dfrac{-X}{L^2} & 0 \\ \dfrac{-Y}{L^2} & \dfrac{X}{L^2} & 0 & \dfrac{Y}{L^2} & \dfrac{-X}{L^2} & 1 \end{bmatrix} \begin{Bmatrix} dr_1 \\ dr_2 \\ dr_3 \\ dr_4 \\ dr_5 \\ dr_6 \end{Bmatrix} \tag{E02.1}
$$

or

$$
\mathbf{dv} = \mathbf{a} \; \mathbf{dr} \tag{E02.2}
$$

Where X, Y and L are the projections and element length in the undeformed state.

A plastic hinge forms when the moment in the inelastic component of the element reaches its yield moment. A hinge is then introduced into this component, the elastic component remaining unchanged. The measure of flexural plastic deformation is the plastic hinge rotation.

For any increments of total flexural rotation, dv_2 and dv_3, the corresponding increments of plastic hinge rotation, dv_{p2} and dv_{p3}, are given by

$$
\begin{Bmatrix} dv_{p2} \\ dv_{p3} \end{Bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{Bmatrix} dv_2 \\ dv_3 \end{Bmatrix} \tag{E02.3}
$$

in which A, B, C and D are given in Table E02.1. Unloading occurs at a hinge when the increment in hinge rotation is opposite in sign to the bending moment.

Inelastic axial deformation is assumed not to occur. Hence, only an approximate procedure for considering interaction effects is included, as explained in the following section. This procedure is not theoretically sound, but may be reasonable for many practical applica-

tions.

E02.3 INTERACTION SURFACES

This section applies to the inelastic component only. Yield interaction surface of three types may be specified for the ends of this component, as follows.

(1) Beam type (shape code = 1, Fig. E02.3a). This type of surface should be specified where axial forces are small or are ignored. Yielding is affected by bending moment only.

(2) Steel column type (shape code = 2, Fig. E02.3b). This type of surface is intended for use with steel columns.

(3) Concrete column type (shape code = 3, Fig. E02.3c). This type of surface is intended for use with concrete columns.

For any combination of axial force and bending moment within a yield surface, the cross section is assumed to be elastic. If the force-moment combination lies on or outside the surface, a plastic hinge is introduced. Combinations outside the yield surface are permitted only temporarily, being compensated for by applying corrective loads in the succeeding step. This procedure is not strictly correct because the axial and flexural deformations interact after yield, and it is therefore wrong to assume that only the flexural stiffness changes whereas the axial stiffness remains unchanged.

If a force-moment combination goes from the elastic range to beyond the yield surface in any time or load step, an equilibrium correction is made as shown in Fig. E02.4a. Because the axial stiffness is assumed to remain unchanged, in subsequent steps the force-moment combination at a plastic hinge will generally move away from the yield surface within any time step, as shown in Fig. E02.4b. An equilibrium correction, as shown, is therefore made.

The axial force in an element with a column-type interaction surface can, in reality, never exceed the yield value for zero moment. However, because of the computational pro-

cedure which is used, axial forces in excess of yield can be computed. For axial forces in excess of yield, the yield moments are assumed to be zero. The printed results from the program should be examined carefully and interpreted with caution. If axial forces approaching or exceeding yield are computed for a column, the results are probably incorrect, and severe column damage is probably implied.

E02.4 STATIC ELASTO-PLASTIC STIFFNESS

The element is considered as the sum of an elastic component and an inelastic component. The element actions and deformations are shown in Fig. E02.2a. The axial stiffness is constant, and is given by

$$dS_1 = \frac{EA}{L} \, dv_1 \qquad \text{(E02.4)}$$

in which E = elastic modulus, and A = effective uniform cross sectional area. The flexural stiffness in the elastic range is given by

$$\begin{Bmatrix} dS_2 \\ dS_3 \end{Bmatrix} = \frac{EI}{L} \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ij} & k_{jj} \end{bmatrix} \begin{Bmatrix} dv_2 \\ dv_3 \end{Bmatrix} \qquad \text{(E02.5)}$$

in which I = reference moment of inertia; and k_{ii}, k_{ij}, k_{jj} are coefficients which depend on the cross section variation. For a uniform element, I = actual moment of inertia, $k_{ii} = k_{jj} = 4$ and $k_{ij} = 2$. The coefficients must be specified by the program user, and may, if desired, account for such effects as shear deformations and nonrigid end connections, as well as cross section variations.

After one or more hinges form, the coefficients for the inelastic component change to k_{ii}', k_{ij}' and k_{jj}', as follows

$$k_{ii}' = k_{ii} \, (1-A) \; - \; k_{ij} \, C \qquad \text{(E02.6)}$$

$$k_{ij}' = k_{ij} \, (1-D) \; - \; k_{ii} \, B \qquad \text{(E02.7)}$$

$$k_{jj}' = k_{jj} \, (1-D) \; - \; k_{ij} \, B \qquad \text{(E02.8)}$$

in which A, B, C and D are defined in Table E02.1.

The elasto-plastic stiffness in terms of node displacements is

$$\mathbf{K}_{ep} = \mathbf{a}^T \mathbf{k}_{ep} \mathbf{a} \tag{E02.9}$$

where $\mathbf{K}_{ep}$ is the sum of the stiffness for the elastic and inelastic components.

E02.5 GEOMETRIC STIFFNESS

The geometric stiffness which is used is the same as for the truss bar element. This is not the exact geometric stiffness for a beam column element, but is sufficiently accurate for taking into account P-delta effects in building frames.

E02.6 DYNAMIC STIFFNESS

As for the truss bar element, if βk damping is specified, a viscous damping element is added in parallel with the elastic component. This component contributes both axial and flexural stiffness during dynamic analysis, and develops both axial and flexural damping resistance. As for the truss bar, the geometric stiffness is based on the elastic-plastic axial force only.

E02.7 RESISTING FORCE

The element resisting force is

$$\mathbf{R}^1 = \mathbf{a}^T(\mathbf{S} + \mathbf{S}_d) + \mathbf{a}_\theta^T \frac{S_1}{L} \mathbf{a}_\theta r \tag{E02.10}$$

where $\mathbf{S}$ and $\mathbf{S}_d$ are the elasto-plastic and damping actions, respectively, as S_1 is the elasto-plastic axial force. As for the truss bar, the geometric stiffness is based on S_1 at the beginning of a step, and the resisting force on the value at the end of the step, so that second order effects can produce equilibrium unbalances even for a "linear" structure.

E02.8 ELEMENT LOADS

Static loads applied along the lengths of beam column elements may be taken into account by specifying end clamping forces as shown in Fig. E02.5. These forces are those which must act *on the element ends* to prevent end displacement.

The fixed end forces for any element contribute to the static loads on the nodes to which the element connects. Frequently, the live load reduction factor permitted for a column in a building will exceed that for the beams it supports, because columns support

tributary loads from several floors. Hence, if the full live load fixed end shears for each beam are applied at the structure nodes, the accumulated loads on the the columns may be unnecessarily large. This can be taken into account by means of live load reduction factors for the fixed end forces, which are used as follows.

For initialization of the element shear and axial forces, the full specified fixed end forces are used. However, for computation of the static loads on the nodes connected to the element, the fixed end shear and axial forces due to live load (but not the moments) are first multiplied by the specified reduction factor. The forces producing axial loads in the columns can therefore be reduced to account for differences in the permissible live load reductions for beams and columns, yet the shear forces computed for the beams will still be correct.

E02.9 SHEAR DEFORMATIONS

If desired, effective flexural shear areas may be specified. The program then modifies the flexural stiffness to account for the additional shear deformations. The fixed end forces are not changed. Hence if shear deformations are important, the specified fixed end force patterns should take these deformations into account.

E02.10 END ECCENTRICITY

Plastic hinges in frames and coupled frame-shear-wall structures will form near the joint faces rather than at the theoretical joint centerlines. This effect can be approximated by postulating rigid and infinitely strong connecting links between the nodes (which are located at the joint centerlines) and the element ends, as shown in Fig. E02.6. The displacement transformation relating the node displacements, (dr_n) , with those at the element ends is

$$
\begin{Bmatrix} dr_1 \\ dr_2 \\ dr_3 \\ dr_4 \\ dr_5 \\ dr_6 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & -Y_i & 0 & 0 & 0 \\ 0 & 1 & X_i & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & -Y_j \\ 0 & 0 & 0 & 0 & 1 & X_j \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} dr_{1n} \\ dr_{2n} \\ dr_{3n} \\ dr_{4n} \\ dr_{5n} \\ dr_{6n} \end{Bmatrix} \qquad \text{(E02.11)}
$$

This transformation has been incorporated into the calculation of the element stiffnesses and deformations. If end eccentricities are specified, the stiffness coefficients in Eqn. E02.5 must apply to that part of the element between the joint faces, ignoring the joint region. Similarly, the fixed end forces are those applying at the joint faces. The end eccentricity effects are taken into account in transferring the fixed end forces to the nodes (i.e the moment loads are augmented by couples created by the fixed end shears and axial forces). Any specified live load reduction factors are applied to the fixed end shear and axial forces *before* they are transferred from the joint forces to the nodes. End eccentricities do not affect the geometric stiffness. This stiffness is based on the element length between nodes, not between joint faces.

TABLE E02.1

COEFFICIENTS FOR PLASTIC HINGE ROTATIONS

Yield Condition	A	B	C	D
Elastic ends	0	0	0	0
Plastic hinge at end i only	1	$\dfrac{k_{ij}}{k_{ii}}$	0	0
Plastic hinge at end j only	0	0	$\dfrac{k_{ij}}{k_{jj}}$	1
Plastic hinges at both ends i and j	1	0	0	1

Coefficients k_{ii}, k_{ij}, and k_{jj} are defined by Eq. E02.5.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.02

BEAM-COLUMN ELEMENT (TYPE 02)

See Fig. E02.1 through Fig. E02.6 for element behavior and properties.

C2.02(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.02(b).
6-10(I)			No. of end eccentricity types (max 15). See section C2.02(c).
11-15(I)			No. of yield surfaces for cross sections (max. 40) See section C2.02(d).

C2.02(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Young's modulus.
16-25(R)			Strain hardening ratio, as a **proportion** of Young's modulus.
26-35(R)			Cross sectional area.
26-45(R)			Moment of inertia.
46-50(R)			Flexural stiffness factor k_{ii}.
51-55(R)			Flexural stiffness factor k_{jj}.
56-60(R)			Flexural stiffness factor k_{ij}.
61-70(R)			Shear area. Leave blank if shear deformations are to be ignored, or if shear deformations have already been taken into account in computing the flexural stiffness factors.
71-80(R)			Poisson's ratio (used for computing shear modulus, and used only if shear area is nonzero).

C2.02(c). End Eccentricities

One line for each end eccentricity. Omit if there are no end eccentricities. See Fig. E02.6 for explanation. All eccentricities are measured **from** the node to the element end.

Columns	Notes	Variable	Data
1-5(I)			End eccentricity number, in sequence beginning with 1.
6-15(R)		X_i = X eccentricity at end i.
16-25(R)			X_j = X eccentricity at end j.
26-35(R)			Y_i = Y eccentricity at end i.
26-45(R)			Y_j = Y eccentricity at end j.

C2.02(d). Cross Section Yield Surfaces

One card for each yield surface. See Fig. E02.3 for explanation.

Columns	Notes	Variable	Data
1-5(I)			Yield surface number, in sequence beginning with 1.
10(I)			Yield surface shape code, as follows. (a) 1: beam type, without P-M interaction. (b) 2: steel I-beam type. (c) 3: reinforced concrete column type.
11-20(R)			Positive yield moment, M_{y+} (counter clockwise).
21-30(R)			Negative yield moment, M_{y-} (clockwise).
31-40(R)			Compression yield force, P_{yc}. Leave blank if shape code = 1.
41-50(R)			Tension yield force, P_{yt}. Leave blank if shape code = 1.
51-55(R)			M coordinate of balance point A, as a proportion of M_{y+}. Leave blank if shape code = 1.
56-60(R)			P coordinate of balance point A, as a proportion of P_{yc}. Leave blank if shape code = 1.
61-65(R)			M coordinate of balance point B, as a proportion of M_{y-}. Leave blank if shape code = 1.
66-70(R)			P coordinate of balance point B, as a proportion of P_{yc}. Leave blank if shape code = 1.

C2.02(e). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements must be included.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element at end i.
11-15(I)			Node number at element at end j.
16-20(I)			Node number increment for element generation. Default = 1.
21-25(I)			Stiffness type number.
26-30(I)			End eccentricity number. Default = no end eccentricity.
31-35(I)			Yield surface number at end i.
36-40(I)			Yield surface number at end j.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION D2(b)(ii).02

ELEMENT LOAD DATA FOR BEAM-COLUMN ELEMENT (TYPE 02)

D2(b)(ii).02(a). Load Sets

NLOD lines (see Section D2(b)(i)), one line per element load set. See Fig. E02.5.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
6-10(I)			Coordinate code. (a) 0: Forces are in local (element) coordinates. (b) 1: Forces are in global (structure) coordinates.
11-20(R)			Live load reduction factor.
21-30(R)			Clamping force P_i.
31-40(R)			Clamping force V_i.
41-50(R)			Clamping moment M_i.
51-60(R)			Clamping force P_j.
61-70(R)			Clamping force V_j.
71-80(R)			Clamping moment M_j.

D2(b)(ii).02(b). Loaded Elements and Load Set Scale Factors

As many as lines needed. Terminate with a blank line.

Columns	Notes	Variable	Data
1-5(I)			No. of first element in series.
6-10(I)			No. of last element in series. Default = single element.
11-15(I)			Element no. increment. Default = 1.
16-20(I)			Load set number.
21-30(R)			Load set scale factor.
31-45(I-R)			Optional second load set no. and scale factor.
46-60(I-R)			Optional third load set no. and scale factor.
61-75(I-R)			Optional fourth load set no. and scale factor.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

BEAM-COLUMN ELEMENT (TYPE 02)

Item	Description
1	Bending moment at end I.
2	Bending moment at end J.
3	Shear force at end I.
4	Shear force at end J.
5	Axial force at end I.
6	Axial force at end J.
7	Current plastic hinge rotation at end I.
8	Current plastic hinge rotation at end J.
9	Accumulated positive plastic hinge rotation at end I.
10	Accumulated positive plastic hinge rotation at end J.
11	Accumulated negative plastic hinge rotation at end I.
12	Accumulated negative plastic hinge rotation at end J.
13	Yield code at end I (1: hinge; 0: no hinge).
14	Yield code at end J (1: hinge; 0: no hinge).
15	Node number at end I.
16	Node number at end J.
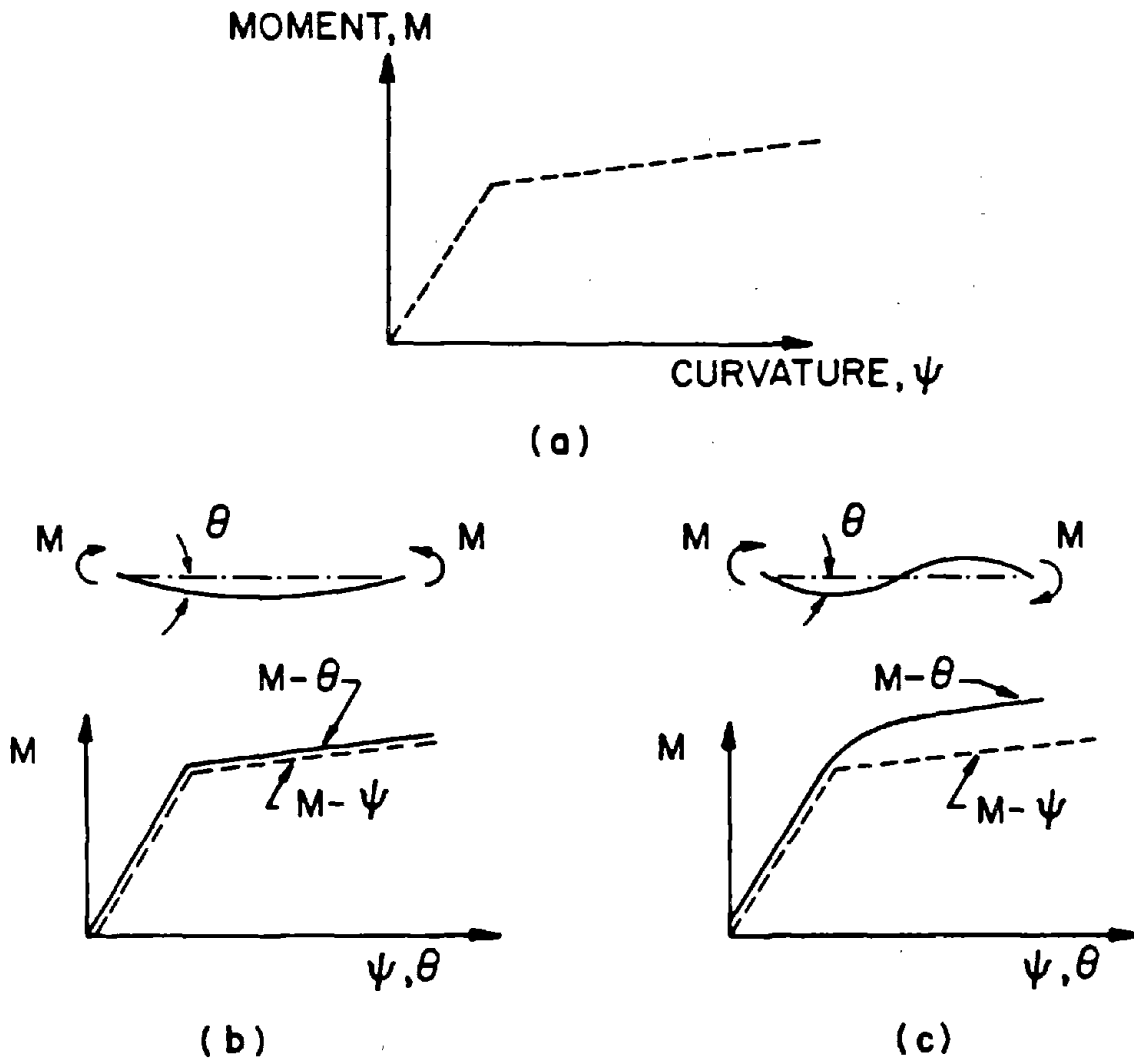
FIG. E02.1 MOMENT – CURVATURE AND MOMENT –
ROTATION RELATIONSHIP
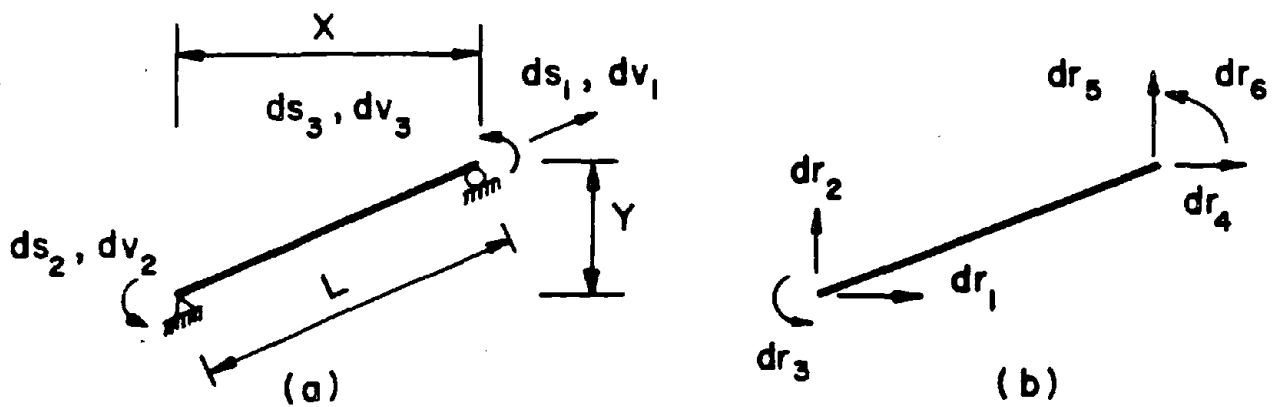
FIG. E02.2 DEFORMATIONS AND DISPLACEMENTS
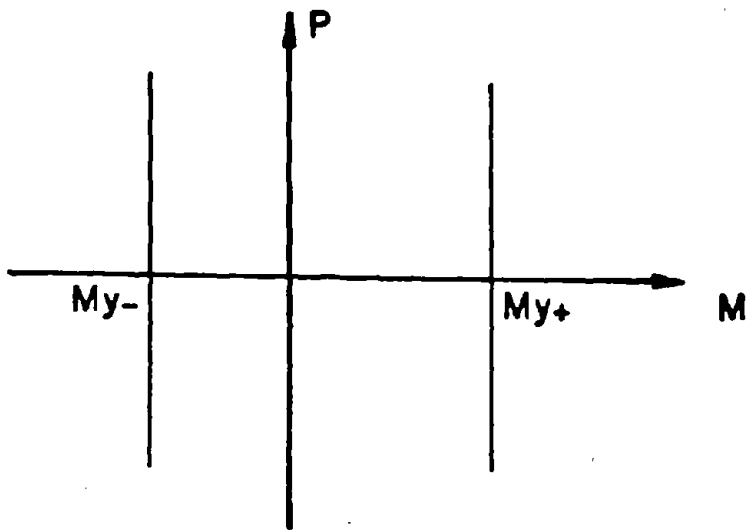
(a)   SHAPE CODE = 1
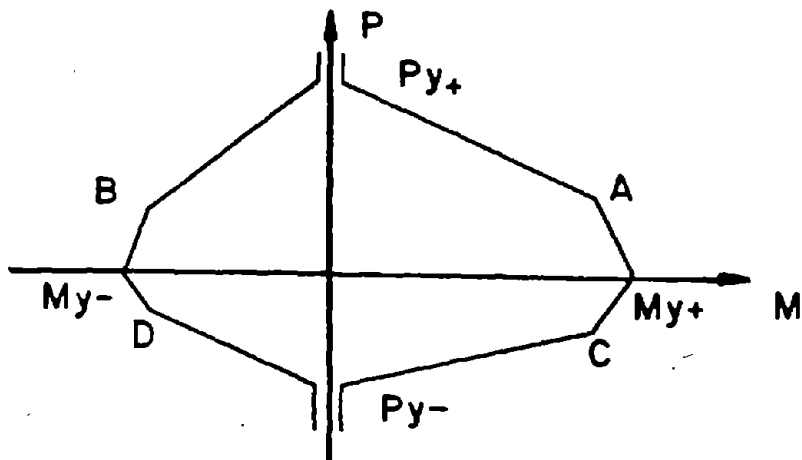
(b)   SHAPE CODE = 2
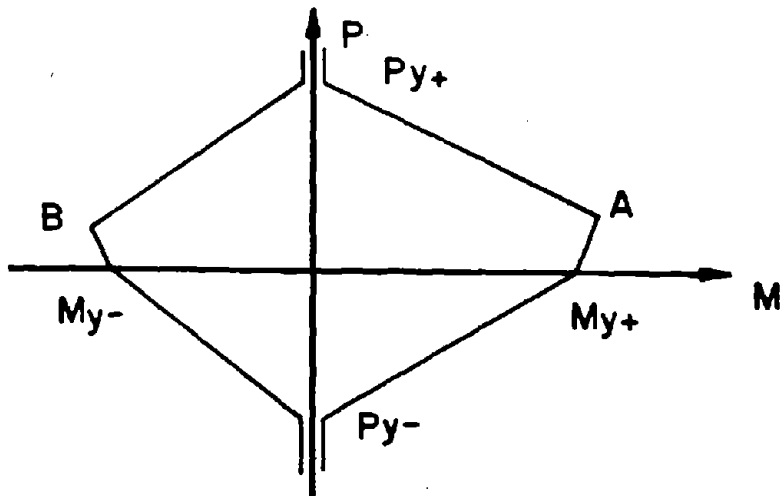
(c)   SHAPE CODE = 3

FIG. E02.3 YIELD INTERACTION SURFACES
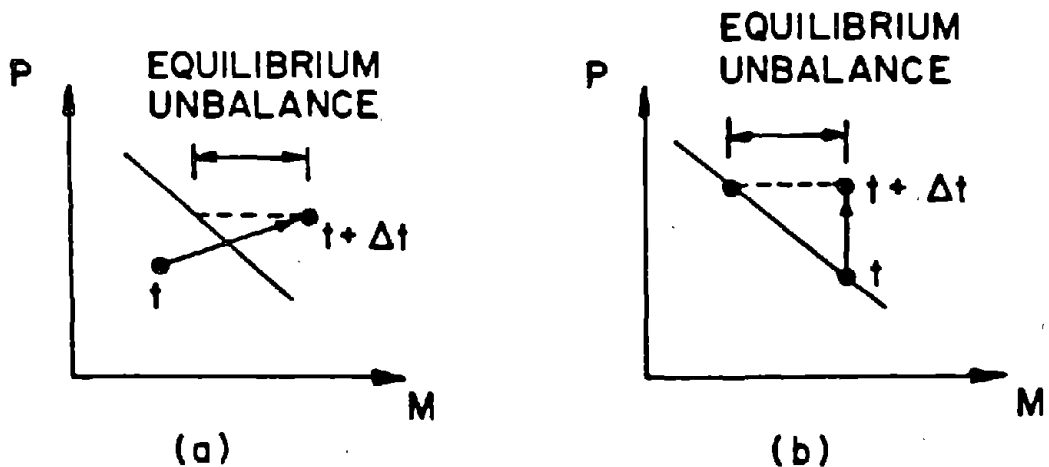
FIG. E02.4 EQUILIBRIUM CORRECTION FOR YIELD
SURFACE OVERSHOOT

(a) CODE = 0

(b) CODE = 1
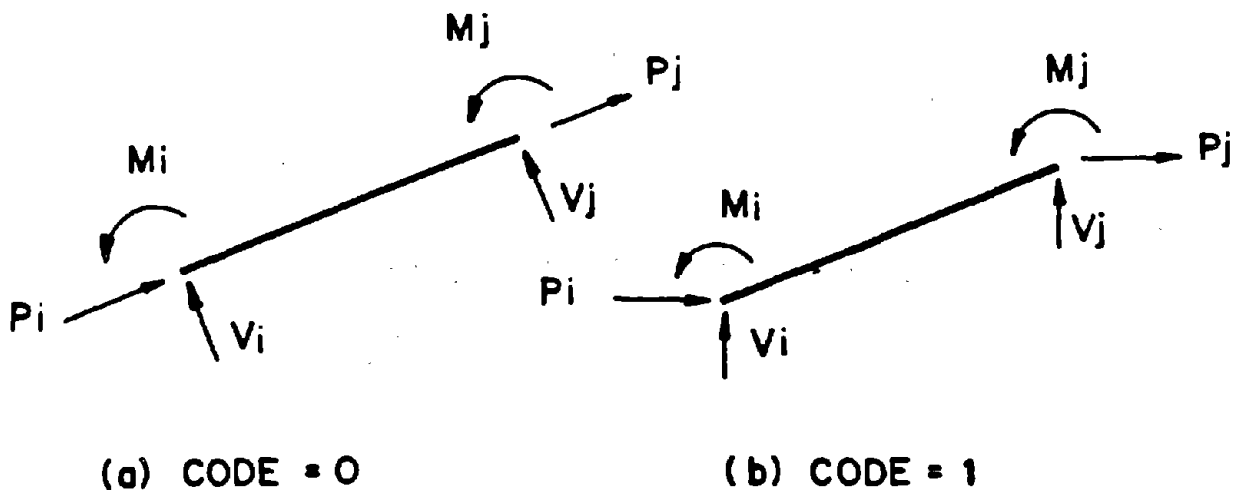
FIG. E02.5 END CLAMPING AND INITIAL FORCES
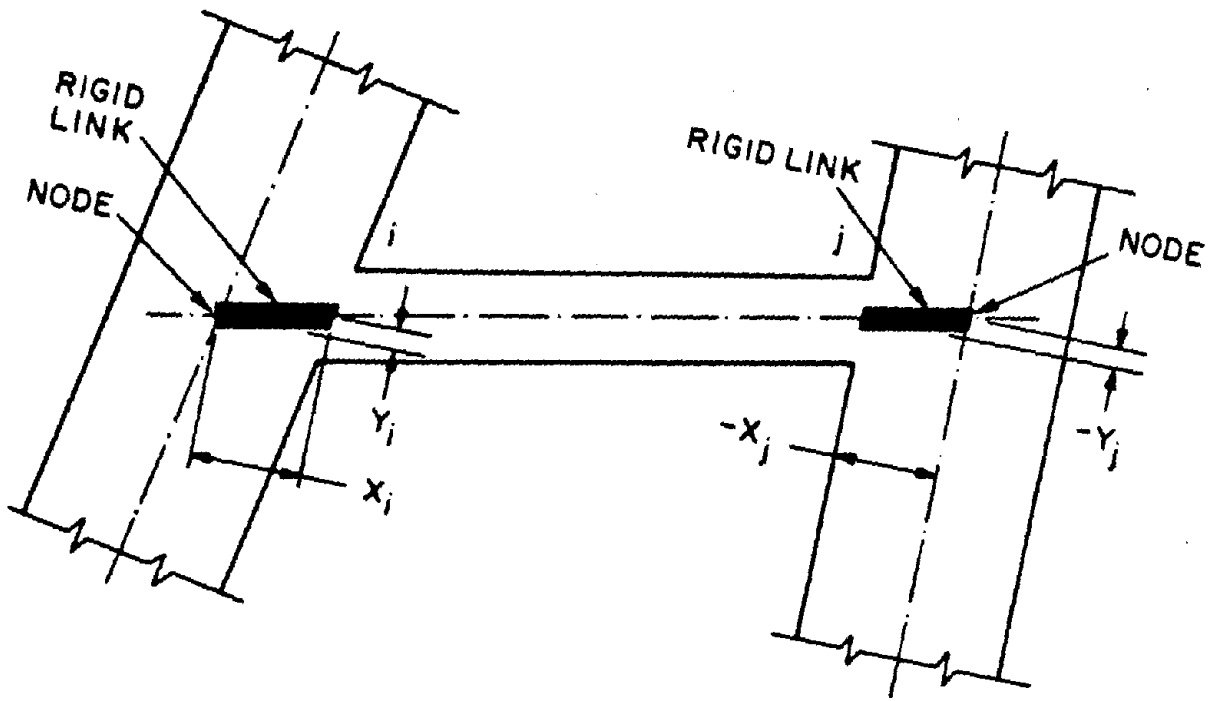
FIG. E02.6 END ECCENTRICITIES

DRAIN-2DX USER GUIDE

ELEMENT THEORY

INFILL PANEL ELEMENT (TYPE 03)

E03.1 GENERAL CHARACTERISTICS

Infill panel elements are intended to permit approximate consideration of infill panels of masonry and similar materials. They may also be used to construct inelastic shear beams.

The element is assumed to have shear stiffness only, in the XY coordinate system. Hence, it provides resistance, through shear deformation, to relative horizontal and/or vertical displacement of the nodes it connects. The relationship between shear stress and shear strain may be inelastic, as shown in Fig. E03.1. If sudden failure takes place, the forces being resisted by the element immediately prior to failure will be transferred to the remaining structure, essentially as a shock loading.

Infill panel elements will most commonly be rectangular. However, provisions have been included to permit variations from a rectangular shape without violating equilibrium. Any variations from a rectangular shape should normally be small, otherwise the assumption that the element has only shear stiffness may become grossly incorrect.

E03.2 ELEMENT DEFORMATIONS

The shear strain in XY coordinates at the element center is the only deformation considered. The element is treated an an isoparametric finite element, and the increment in shear strain, γ_{xy}, is related to the increments of horizontal and vertical displacement by a displacement transformation of the usual form, namely

$$dv = d\gamma_{xy} = a \ dr \qquad (E03.1)$$

in which $dr = <dr_1,.....,dr_8>^T$ as in Fig. E03.2, and a depends on the element shape. The details of the transformation are lengthy, and are not repeated here. They may be found in standard finite element texts. The inelastic deformation is the amount of shear strain

beyond yield in the elasto-plastic component. Second order effects are not taken into account.

It is important to note that infill panel elements have shear stiffness only. If these elements are used without a surrounding frame, it is essential that the displacement degrees of freedom permitted do not allow modes of deformation for which there is no shear strain at the element center, otherwise the structure stiffness matrix will be singular.

E03.3 STATIC ELASTO-PLASTIC STIFFNESS

The tangent stress-strain relationship is

$$d\tau_{xy} = G_T \ d\gamma_{xy} \tag{E03.2}$$

in which $d\tau_{xy}$ = increment of shear stress, and G_T = tangent shear modulus. Hence, the element stiffness matrix can be formulated by standard procedures as

$$\mathbf{K}_{ep} = \int_V \mathbf{a}^T \ G_T \ \mathbf{a} \ dV \tag{E03.3}$$

in which V = volume of the element. The integration is performed by one-point Guass quadrature.

E03.4 OTHER PROPERTIES

As for other elements, βK damping has the effect of adding a viscous element (affected by shear strain rate only) in parallel with the elasto-plastic component. There are no provisions for geometric stiffness or element loads.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.03

INFILL PANEL ELEMENT (TYPE 03)

See Fig. E03.1 for element behavior and properties.

C2.03(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.03(b).

C2.03(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Elastic shear modulus.
16-25(R)			Strain hardening shear modulus, as a **proportion** of elastic shear modulus.
26-35(R)			Average panel thickness.
36-45(R)			Yield stress in shear.
46-55(R)			Failure strain, as a **proportion** of yield strain. This must be greater than 1.0.
60(I)			Failure code. (a) 1: Brittle failure. (b) 0: Ductile failure.

C2.03(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be** **included**.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element point I.
11-15(I)			Node number at element point J.
16-20(I)			Node number at element point K.
21-25(I)			Node number at element point L.
26-30(I)			Node number increment for element generation. Default = 1.
31-35(I)			Stiffness type number.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

INFILL PANEL ELEMENT (TYPE 03)

Item	Description
1	Yield Code (1: yielded; 0: not yielded).
2	Shear stress.
3	Accumulated positive plastic strain.
4	Accumulated negative plastic strain.
5	Node number at point I.
6	Node number at point J.
7	Node number at point K.
8	Node number at point L.
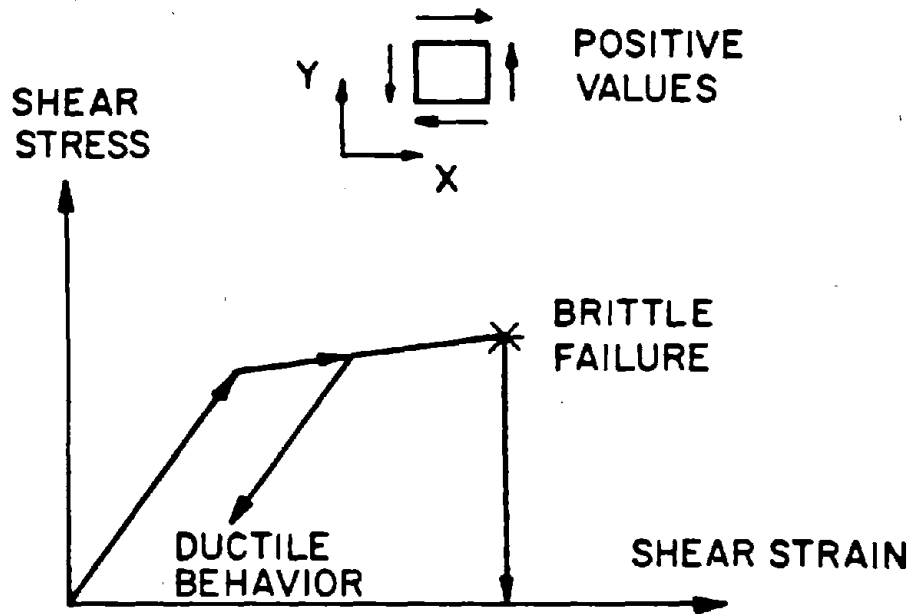
FIG. E03.1 STRESS – STRAIN RELATIONSHIP FOR INFILL PANEL

DRAIN-2DX USER GUIDE

ELEMENT THEORY

' SEMI-RIGID CONNECTION ELEMENT (TYPE 04)

E04.1 GENERAL CHARACTERISTICS

Semi-rigid connection elements may be used to represent connections in which angle changes occur between connected beams or columns. In particular, connection elements may be used to represent deformable panel zones in steel building frames.

Each element is connected to two nodes, and is influenced by only the relative rotational displacement between nodes. Normally, the translational displacements of the two nodes should be constrained to be identical, in which case these nodes must have identical coordinates, otherwise equilibrium will not be satisfied. The relationship between the moment transmitted by the element and the relative rotation between the connected nodes may be inelastic, as shown in Fig. E04.1.

E04.2 ELEMENT DEFORMATION

The relative rotation between the connected nodes is related to the node rotations as follows:

$$d\phi = \begin{bmatrix} 1 & -1 \end{bmatrix} \begin{Bmatrix} d\theta_i \\ d\theta_j \end{Bmatrix} \tag{E04.1}$$

or

$$d\theta = \mathbf{a} \ \mathbf{dr} \tag{E04.2}$$

in which $d\phi$ = increment of relative rotation (element deformation); and $d\theta_i$, $d\theta_j$ = increments of rotation of the connected nodes. The inelastic deformation is the amount of relative rotation beyond yield.

E04.3 STATIC ELASTO-PLASTIC STIFFNESS

The tangent stiffness relationship is

$$dM = k_T \, d\phi \tag{E04.3}$$

The positive sign convention is shown in Fig. E04.1. Hence, in terms of node rotations the stiffness, K_{ep}, is given by

$$K_{ep} = a^T \, k_T \, a \tag{E04.4}$$

E04.4 OTHER PROPERTIES

As for other elements, βK damping has the effect of adding a (rotational) damping element in parallel with the elasto-plastic element. There are no provisions for geometric stiffness or element loads.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.04

SEMI-RIGID CONNECTION ELEMENT (TYPE 04)

See Fig. E04.1 for element behavior and properties.

C2.04(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.04(b).

C2.04(b). Stiffness Types
One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Initial rotational stiffness (moment per radian).
16-25(R)			Strain hardening rotational stiffness, as a **proportion** of initial rotational stiffness.
26-35(R)			Positive yield moment.
36-45(R)			Negative yield moment.

C2.04(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included.**

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element end I.
11-15(I)			Node number at element end J.
16-20(I)			Node number increment for element generation. Default = 1.
21-25(I)			Stiffness type number.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

SEMI-RIGID CONNECTION ELEMENT (TYPE 04)

Item	Description
1	Node number at end I.
2	Node number at end J.
3	Yield code for element (1: yielded; 0: not yielded).
4	Bending Moment.
5	Total rotational deformation.
6	Accumulated positive plastic rotational deformation.
7	Accumulated negative plastic rotational deformation.
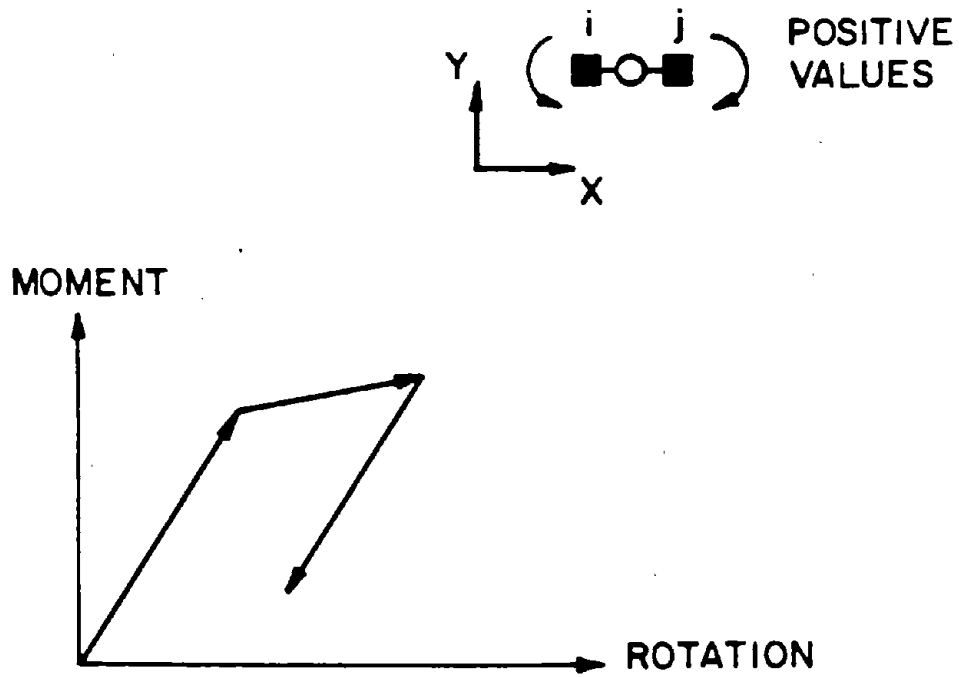
FIG. E04.1 MOMENT – ROTATION RELATIONSHIP FOR
SEMI – RIGID CONNECTION

DRAIN-2DX USER GUIDE

ELEMENT THEORY

GAP-FRICTION JOINT ELEMENT (TYPE 05)

E05.1 GENERAL CHARACTERISTICS

E05.1.1 Gap Behavior

Consider, first, a gap element with zero friction. Such an element consists of a spring with zero length, placed normal to the joint surface. A finite stiffness is assigned to the element in compression (typically very large) and a zero stiffness is assigned in tension. Hence, the element allows a gap to develop between the connected panels.

The gap force-deformation relationship is as shown in Fig. E05.1. For a horizontal joint, the element provides this relationship between vertical force and relative vertical displacement. For a vertical joint, the relationship is between horizontal force and displacement. The relationship allows for nonlinear behavior in compression (joint crushing), with the joint bearing surfaces yielding as the normal compressive force increases. The element has options to unload elastically or inelastically, as shown.

Compressive deformation is assumed to be positive. The element may be preloaded to represent gravity and/or posttensioning effects. Separation occurs when any added tension force exceeds the preload.

Elements may be placed in horizontal or vertical joints. An element may connect four nodes as shown in Fig. E05.2, in which case the distance between the connected lines must be zero. Alternatively, an element may connect two nodes directly, in which case the nodes must have identical coordinates.

Three-node connectivity may also be specified if desired, as shown in Fig. E05.3. This figure illustrates a horizontal joint between two adjacent panels, one idealized with a single modified beam element and the other with several rectangular finite elements. The

joint is modeled with five elements, each connecting one node in the lower panel to the two nodes on the upper panel. This modeling can be used to enforce an essentially straight upper edge on the lower panel, in order to satisfy displacement compatibility between the panels.

A real joint will actually be of finite thickness, with finite compressive stiffness. It is desirable to model the deformability of the joint as accurately as possible, by calculating the compressive stiffness from the actual joint geometry and mechanical properties.

If event calculations are not specified, substantial unbalances can occur when a gap closes, especially if the time step is long or the element stiffness is high. When gap elements are used, the element stiffnesses should generally be made as low as possible, the time step should be short (or should be variable), and the results should be examined carefully to ensure that oscillation or divergence of results does not occur following gap closure.

E05.1.2 Panel Tilting

If a horizontal joint develops a gap, the upper panel tilts relative to the lower panel. When modified beam elements are used for the panels, it will be natural to place a gap element at each end of the joint (Fig. E05.4a). If a gap opens, the assumption is then that the panels pivot about the corner point, as shown.

In an actual structure, joint opening is likely to take place progressively, rather than suddenly, and the assumption that plane joint sections remain plane may be substantially incorrect. To consider the true mechanism of joint opening, it is necessary to use a finite element panel model, with several elements across the joint. Distortion of the joint plane, and progressive joint opening, can then be modeled. However, this type of idealization is more expensive computationally.

When the modified beam panel element is used, the error in assuming a rigid joint plane can be partially corrected either by moving the assumed pivot points or by modeling

the joint with several gap elements. The pivot points can be moved by specifying two gap elements located within the joint rather than at the corners, as shown in Fig, E05.4b. A less sudden joint opening can also be obtained by specifying several gap elements along the joint, as shown in Fig. E05.4c. In this case, however, the gap elements must be made relatively flexible in compression, otherwise tilting will occur essentially about one corner, and all gap elements will open at essentially the same time.

E05.1.3 Combination With Friction

For the friction joint element (type 11), it is assumed that the bearing force on the joint is constant. In an actual joint, the bearing stress may vary with time and with position in the joint, because of bending of the walls and vertical shaking of the building. Hence, the shear strength will also vary. This is taken into account in the gap-friction element. If a nonzero friction coefficient is specified, the frictional resistance varies as the gap force changes. However, a constant friction coefficient is assumed (i.e. no degradation).

The complete gap-friction element combines a gap element with a friction element, and adjusts the friction element so that its shear strength at any time is equal to the compressive force on the gap element multiplied by the coefficient of friction. The shear strength becomes zero if gap opening occurs. In compression, the shear resistance is assumed to increase in direct proportion to the bearing force.

Frictional slip under varying bearing force is a complex process. The behavior is particularly complex if shear movement is occurring with an open gap and the gap suddenly closes. The procedure used to determine the state of an element at the end of any time step is not exact, but is believed to be reasonable. The logic of the procedure is shown in Fig. E05.5.

In this procedure, the state of the friction element is first found using the friction strength at the beginning of the step. The state of the gap element is then found, and the friction strength is updated, based on the new bearing force. Finally, the state of the friction

element is modified, taking account of the new bearing force. Five different cases can be identified, as shown in Fig. E05.5. The figure shows the assumption made for each case.

E05.2 STATIC ELASTO-PLASTIC STIFFNESS

The element combines a gap element with a friction element. A four-node gap element has four displacement degrees of freedom, as shown in Fig. E05.2, and one extensional mode of deformation. The gap deformation is given by

$$q = <-\frac{b_1}{L_1} \quad -\frac{a_1}{L_1} \quad \frac{b_2}{L_2} \quad \frac{a_2}{L_2}> \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \end{Bmatrix} \tag{E05.1}$$

where $L_1 = a_1 + b_1$ and $L_2 = a_2 + b_2$; or

$$q = a\,r \tag{E05.2}$$

Hence, the gap stiffness matrix is

$$K = a^T k a = k \begin{bmatrix} (\frac{b_1}{L_1})^2 & \frac{a_1 b_1}{L_1^2} & \frac{b_1 b_2}{L_1 L_2} & \frac{a_2 b_1}{L_1 L_2} \\ . & (\frac{a_1}{L_1})^2 & \frac{a_1 b_2}{L_1 L_2} & \frac{a_1 a_2}{L_1 L_2} \\ . & \text{Symmetric} & (\frac{b_2}{L_2})^2 & \frac{a_2 b_2}{L_2^2} \\ . & . & . & (\frac{a_2}{L_2})^2 \end{bmatrix} \tag{E05.3}$$

where k is the tangent stiffness of the element (k_1, k_2, k_3, k_4, or zero, Fig. E05.1). A two-node element is obtained by setting $b_1 = b_2 = 0$, and a three-node element by setting either $b_1 = 0$, or $b_2 = 0$.

The stiffness of the friction element is obtained in essentially the same way, except that the displacements are parallel, not normal, to the joint surface.

E05.3 OTHER PROPERTIES

There is no provision for second order effects in the element, or for element loads. Also, regardless of what value is specified for the stiffness dependent (βK) damping

coefficient, β, this value is assumed to be zero (i.e. no viscous damping). This is a hang-over from the original DRAIN-2D program, in which only a single value of β could be specified, the same for all elements (in DRAIN-2DX, a different value can be specified for each element group). Because the initial (elastic) stiffness of a joint element is likely to be high, then the damping coefficient, βK, is also likely to be high. Hence, it was felt that the viscous forces and energy absorption would be unrealistically large. Further, it seemed, intuitively, that the energy absorption in a joint should be entirely hysteretic, and that it should be unnecessary to account for miscellaneous losses. For these reasons, βK was assumed to be zero in the DRAIN-2D elements. The assumption has been retained in DRAIN-2DX.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.05

GAP FRICTION JOINT ELEMENT (TYPE 05)

See Fig. E05.1 through E05.4 for element behavior and properties.

C2.05(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.05(b).

C2.05(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Displacement limit U1.
16-25(R)			Displacement limit U2.
26-35(R)			Stiffness k_1.
36-45(R)			Stiffness k_2.
46-55(R)			Stiffness k_3.
56-65(R)			Stiffness k_4.
66-70(R)			Friction coefficient.
71-80(R)			Shear stiffness k_s.

C2.05(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included**.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
10(I)			Element orientation code (a) 0: Horizontal joint. (b) 1: Vertical joint.
11-15(I)			Node number at element point I.
16-20(I)			Node number at element point J.
21-25(I)			Node number at element point K.
26-30(I)			Node number at element point L.
31-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Stiffness type number.
41-50(R)			Initial bearing force.
55(I)			Unloading code for shear component.
56-65(R)			Location ratio a1/L1.
66-75(R)			Location ratio a2/L2.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

GAP FRICTION JOINT ELEMENT (TYPE 05)

Item	Description
1	Node number at point I.
2	Node number at point J.
3	Node number at point K.
4	Node number at point L.
5	Normal force.
6	Normal deformation.
7	Line number for normal force.
8	Shear force.
9	Shear deformation.
10	Shear yield code (0: elastic ; 1: yielding).
11	Accumulated positive slip.
12	Accumulated negative slip.
13	Accumulated gap opening.
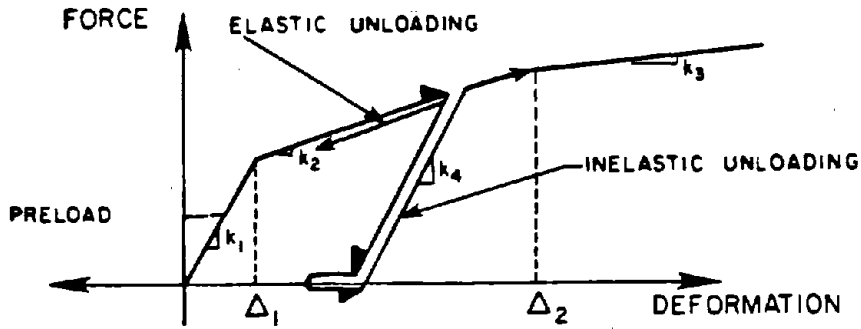
FIG. E05.1 GAP ELEMENT FORCE – DEFORMATION RELATIONSHIP

(a) HORIZONTAL JOINT

(b) VERTICAL JOINT
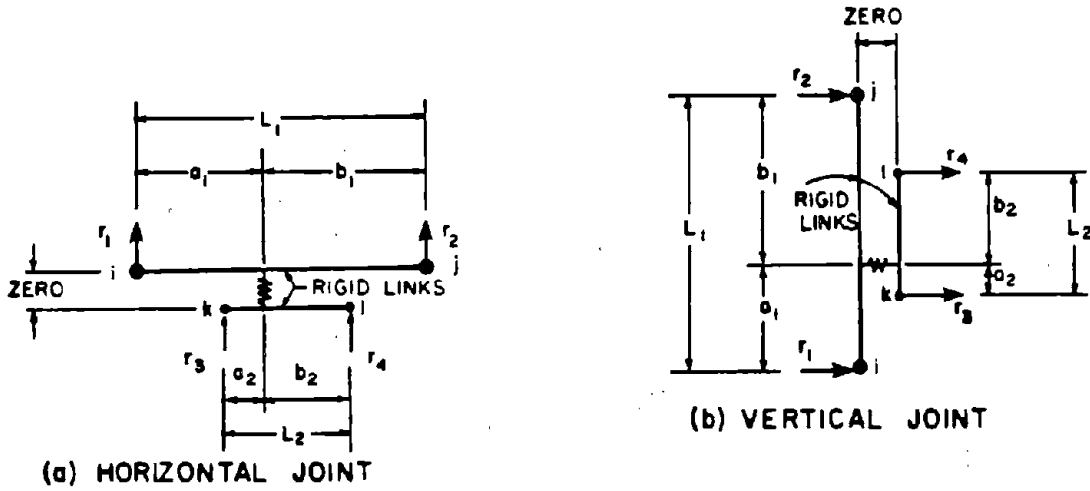
FIG. E05.2 GAP ELEMENT DEGREES OF FREEDOM
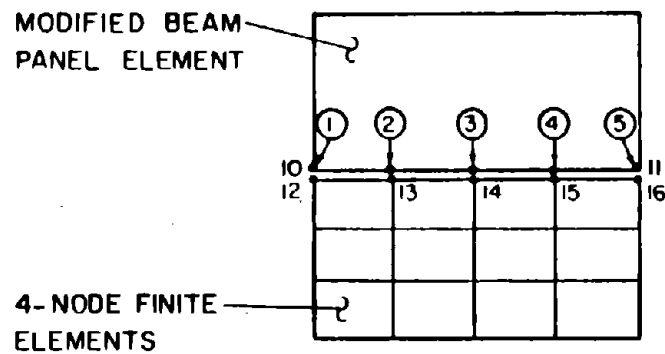
MODIFIED BEAM PANEL ELEMENT

① ② ③ ④ ⑤

10 11
12 13 14 15 16

4-NODE FINITE ELEMENTS

JOINT ELEMENT NO.	ODE NUMBERS			
	i	j	k	l
1	10	10	12	12
2	10	11	13	13
3	10	11	14	14
4	10	11	15	15
5	11	11	16	16

FIG. E05.3 MATCHING OF BEAM AND FINITE ELEMENT PANEL MODELS

CENTER OF ROTATION

CENTER OF ROTATION

GAP ELEMENT (PIVOT POINT)

(a) 2 GAP ELEMENT PER JOINT   **(b) CHANGE OF PIVOT POINT**   **(c) 7 GAP ELEMENT PER JOINT**
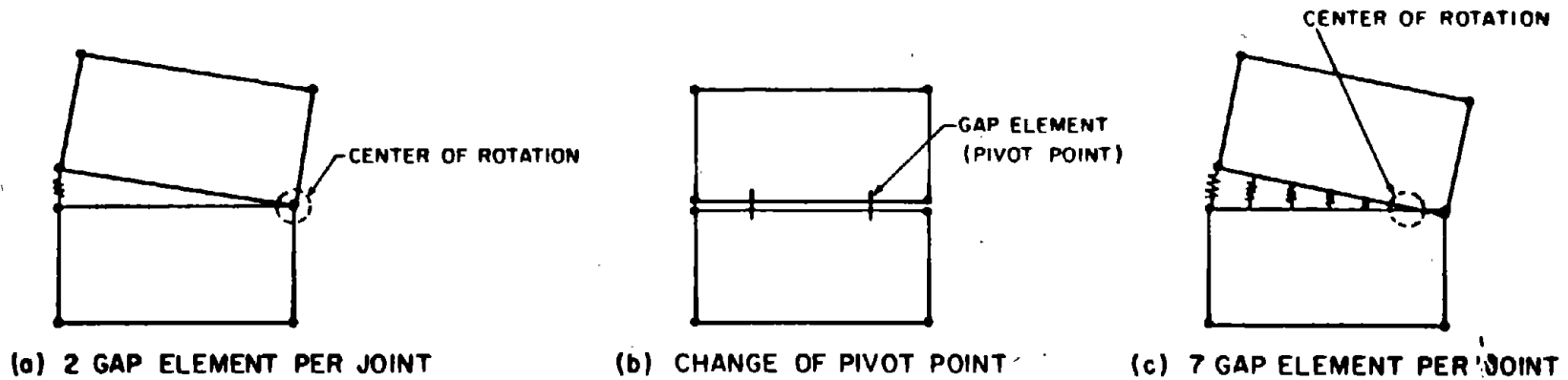
FIG. E05.4 PANEL TILTING

VARIABLES:

KODY = Slip Code (0 = no slip, 1 = slip)

F_s = Shear Force

F_n = Normal Force

$F_y = \mu F_n$ = Shear Strength

Δu = Shear Deformation Increment

Calculate F_s based on F_n at $t - \Delta t$.

Set KODY

Calculate F_n. Hence new $F_y = \mu F_n$.

No ———— $ABS(F_s) \leqslant F_y$ ———— Yes

No —— KODY = 0 —— Yes

KODY = 1 —— Yes

No $F_s = F_y$

Yes —— Δu and F_s same sign —— No

shear force increases   shear force decreases

KODY = 1

$F_s = F_s * ABS\left(\dfrac{F_y}{F_s}\right)$

OUT
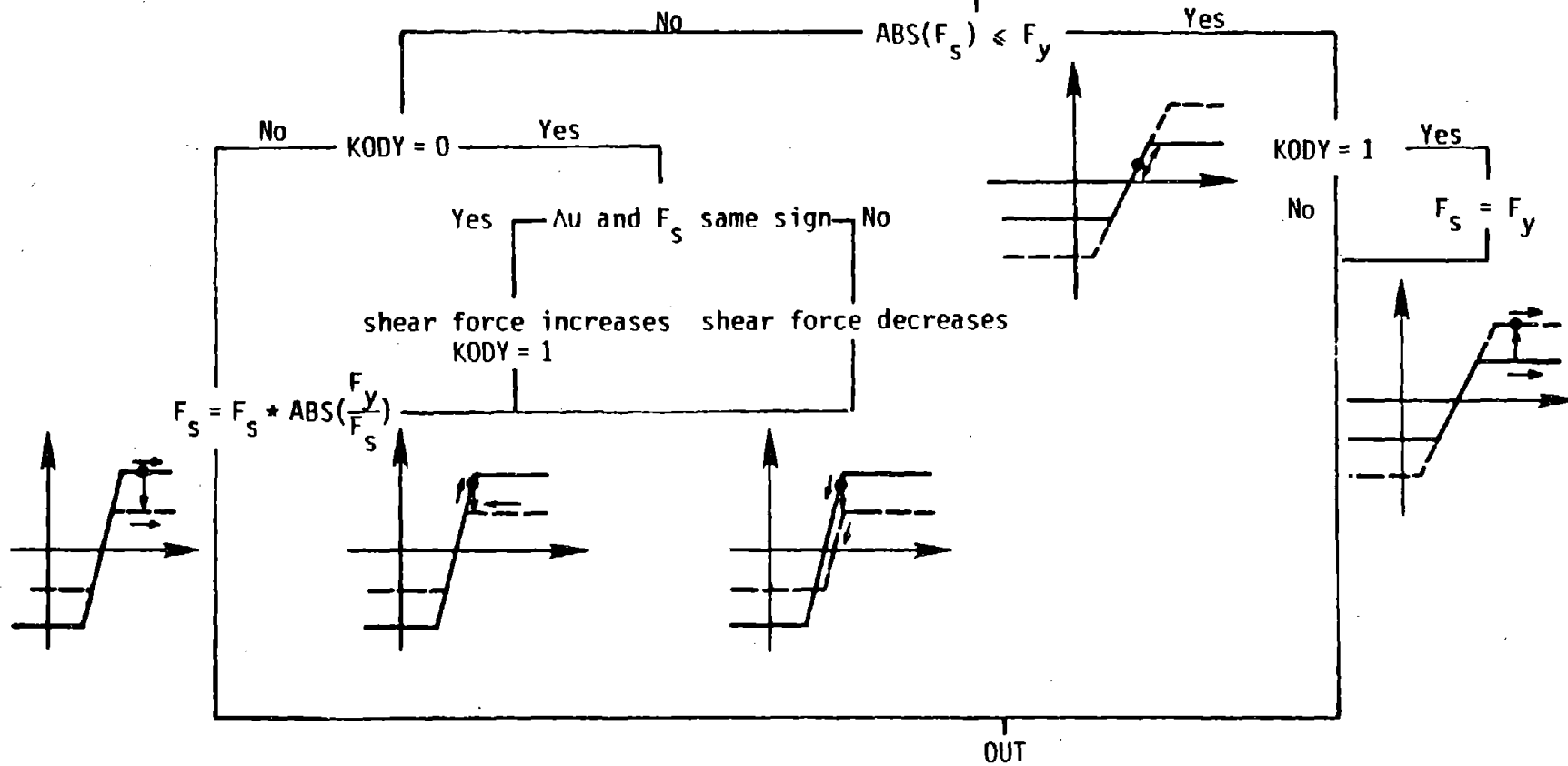
FIG. E05.5 LOGIC FOR GAP – FRICTION ELEMENT

DRAIN-2DX USER GUIDE

ELEMENT THEORY

MODIFIED BEAM PANEL ELEMENT (TYPE 06)

E06.1 GENERAL CHARACTERISTICS

In many analyses of large panel buildings, it will be reasonable to idealize each build-
ing panel as a single structural element in which the overall extensional, flexural, and shear
stiffnesses of the panel are modeled. The modified beam model provides this type of ideali-
zation.

Fig. E06.1a shows a large panel with an opening. An effective centroidal axis can be
found, such that an axial force applied along this axis produces no bending. The panel is
idealized as shown in Fig. E06.1b, with four nodes and eight degrees of freedom. It consists
of a beam element (with nonzero axial, flexural, and shear stiffnesses), placed along the
effective centroidal axis and connected to the nodes by flexurally rigid links. These links
enforce a plane section condition along the top and bottom edges of the panel.

The panel is assumed to have three uncoupled primary modes of deformation, as
shown in Fig. E06.2a. The element stiffness is defined by its rigidities in extension
(effective EA), bending (effective EI), and shear (effective GA), respectively. The element
has two secondary modes of deformation (Fig. E06.2b) for which zero stiffnesses are
assumed. These deformation modes plus three rigid body modes (Fig. E06.2c) make up the
eight degrees of freedom of the element. It should be noted that the third mode of deforma-
tion in Fig. E06.2a is shown as pure shear with no flexure. This feature of the element is
considered further in Section E06.2.

Panels will typically be arranged in complete structure as indicated in Fig. E06.3.
Horizontal joint elements will connect nodes on horizontal planes (e.g. nodes 5,6,9,12) and
vertical joint elements will connect nodes on vertical planes (e.g. nodes 2,3,6,7). The verti-
cal displacements of pairs of nodes on horizontal planes (e.g. pairs (5,9) , (6,12)) may be

made equal, indicating rigid vertical connection, or may be connected by elements which allow separation when tension develops. The same applies for horizontal displacements at pairs of nodes on vertical planes.

It should be noted that the top and bottom edges of the panel are assumed to remain straight, whereas the vertical edges are allowed to bend. This may be important for multi-bay walls, because it allows incompatibility at vertical panel edges, for example as illustrated in Fig. E06.4. The compatibility is improved if each wall panel is modeled by several elements, as shown in Fig. E06.5. It should also be noted that rotational displacements of the nodes are not restrained by either the panel or joint elements, and must usually be constrained to be zero.

The mass of each panel must be lumped at its nodes. This permits a reasonable representation of the translational inertia (both vertical and horizontal) of the panel, but overestimates its rotational inertia. This is an inherent error of this panel model, but should not be serious in most cases. If it is believed that the rotational inertia will substantially affect the dynamic response, each panel should be divided into several elements to provide a more accurate representation of the mass distribution in the panel.

E06.2 STATIC ELASTIC STIFFNESS

E06.2.1 Deformations and Actions

The displacement degrees of freedom are r_1 through r_8 as shown in Fig. E06.1b. The primary modes of deformation , as shown in Fig. E06.2a, involve (1) vertical extension; (2) uniform bending; and (3) "racking" in shear. These three modes of deformation are believed to model the most important deformation characteristics of a typical panel. The stiffnesses associated with each mode (i.e., effective extensional, flexural, and shear stiffnesses) must be determined by experiment or by separate calculations, and entered as data to the computer program. These effective stiffnesses must take account of doors, windows, thickness variations, etc.

Panel edges ij and kl (Fig. E06.1) are assumed to remain straight (analogous to plane sections remaining plane in simple beam theory). Compatibility violations between adjacent panels due to bending of lines ik and jl are assumed to be acceptable (Fig. E06.4). It will typically be specified that $r_1 = r_2$ and $r_3 = r_4$ (i.e., no extension of lines ij and kl), using identical displacement commands. These displacements need not be made equal if the analyst desires otherwise, but it should be noted that the element has zero stiffness for extension of ij and kl.

The stiffness matrix in terms of the three primary modes of deformation is

$$\mathbf{k}_d = \begin{bmatrix} \dfrac{EA_c}{h} & \cdot & \cdot \\ \cdot & \dfrac{EI_c}{h} & \cdot \\ \cdot & \cdot & \dfrac{GA_c'}{h} \end{bmatrix} \qquad (E06.1)$$

in which A_c = effective area for vertical extension; I_c = effective moment of inertia for symmetrical bending; A_c' = effective shear area for racking; E = Young's modulus; G = shear modulus; and h = panel height.

These stiffnesses are referred to the effective centroidal axis (Fig. E06.1). The extensional and symmetrical bending modes are uncoupled because of the way in which this axis is defined. The racking mode is assumed to be uncoupled from the other modes, although this is not necessarily correct.

For single bay walls, the racking deformation will be associated with both shear force and bending moment in the wall. The stiffness $\dfrac{GA_c'}{h}$ is an effective stiffness, which must account for both shearing and flexural deformations. For a beam deformed as shown in Fig. E06.6 (antisymmetrical bending plus shear), the flexibility is

$$f = \frac{h^3}{12EI} + \frac{h}{GA} \qquad (E06.2)$$

in which I = actual moment of inertia and $A^{\cdot}$ = actual shear area. The third stiffness term in matrix k_d should therefore be

$$\frac{GA_e^{\cdot}}{h} = \frac{1}{f} \tag{E06.3}$$

For a beam with a rectangular section, a depth (i.e., panel width) d, and Poisson's ratio 0.15, the ratio of shear to bending flexibilities is

$$\frac{(\dfrac{h}{GA^{\cdot}})}{(\dfrac{h^3}{12EI})} = 2.3(\frac{d}{h})^2 \tag{E06.4}$$

For a panel with d/h = 3, this ratio is 20.7, indicating that shear flexibility dominates, and the antisymmetrical bending flexibility can be ignored. In narrower panels, $A_e^{\cdot}$ should be calculated from Eq. E06.3, accounting for the bending flexibility.

For multi-bay panels, racking deformation will be associated with more complex loadings, involving not only bending and shear from the panels above and below, but also shear forces from other panels on either side. In this case, the effective racking stiffness will more closely approach that based on shear deformations only.

The discussion in this section emphasizes that this is a simplified panel model which may not always be sufficiently accurate.

E06.2.2 Stiffness Matrix

The uncoupled element actions, Q, and deformations, q, are shown in Fig. E06.7. The vectors Q and q are conjugate (that is, $\frac{1}{2}Q^{T}q$ = strain energy). The basic stiffness relationship is

$$\begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = k_d \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} \tag{E06.5}$$

where k_d is given by Eq. E06.1.

The relationship between the nodal displacements, r, and element deformations, q, is

$$\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 & 0 & \dfrac{b}{d} & \dfrac{a}{d} & -\dfrac{b}{d} & -\dfrac{a}{d} \\[2mm] 0 & 0 & 0 & 0 & -\dfrac{1}{2d} & \dfrac{1}{2d} & \dfrac{1}{2d} & -\dfrac{1}{2d} \\[2mm] \dfrac{b}{d} & \dfrac{a}{d} & -\dfrac{b}{d} & -\dfrac{a}{d} & -\dfrac{h}{2d} & \dfrac{h}{2d} & -\dfrac{h}{2d} & \dfrac{h}{2d} \end{bmatrix} \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \\ r_5 \\ r_6 \\ r_7 \\ r_8 \end{Bmatrix} \qquad \text{(E06.6a)}$$

or

$$q = a\ r \qquad\qquad \text{(E06.6b)}$$

Hence, the (8 x 8) element stiffness matrix, K, is given by

$$K = a^T\ k_J\ a \qquad\qquad \text{(E06.7)}$$

The computer program calculates top and bottom moments, shear force and axial force in the element, as shown in Fig. E06.8. These actions are defined as follows

$$M_{top} = -(Q_2 + \frac{1}{2}hQ_3)$$

$$M_{bottom} = Q_2 - \frac{1}{2}hQ_3$$

$$F_{shear} = Q_3$$

$$F_{axial} = Q_1$$

E06.3 OTHER PARAMETERS

The dynamic stiffness is determined in the usual way. There are no provisions for second order effects or element loads.

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.06

MODIFIED BEAM PANEL ELEMENT (TYPE 06)

See Fig. E06.1 through E06.8 for element behavior and properties.

C2.06(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.06(b).

C2.06(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Effective EA for vertical extension.
16-25(R)			Effective EI for symmetrical bending.
26-35(R)			Effective GA' for shear racking.
36-45(R)			Distance from panel center to effective centroidal axis, plus or minus, as a **proportion** of panel width (i.e. range is -0.5 to +0.5). Default = 0.

C2.06(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included**.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element point I.
11-15(I)			Node number at element point J.
16-20(I)			Node number at element point K.
21-25(I)			Node number at element point L.
26-30(I)			Node number increment for element generation. Default = 1.
31-35(I)			Stiffness type number.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

MODIFIED BEAM PANEL ELEMENT (TYPE 06)

Item	Description
1	Axial force.
2	Top moment.
3	Bottom moment.
4	Shear force.
5	Total extension.
6	Total rotation.
7	Total shear deformation.
8	Node number at point I.
9	Node number at point J.
10	Node number at point K.
11	Node number at point L.

EFFECTIVE CENTROIDAL AXIS

(a) ACTUAL PANEL
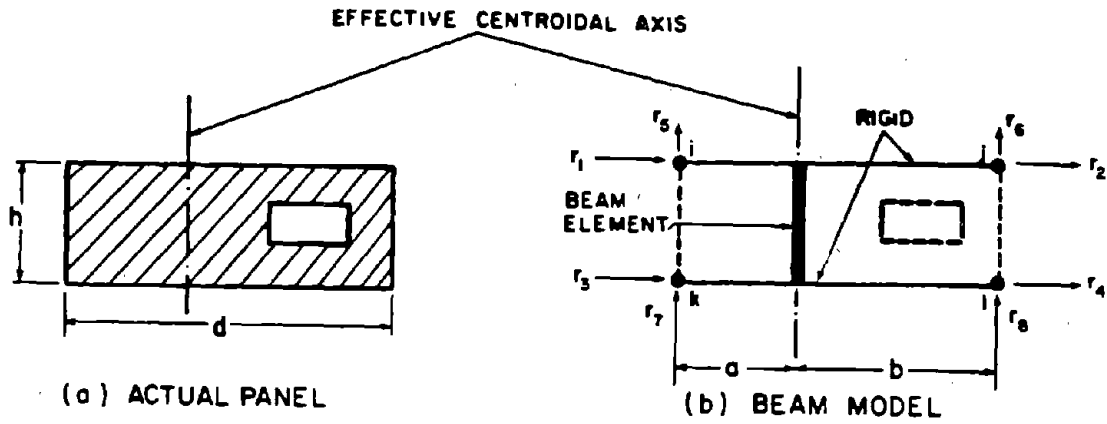
(b) BEAM MODEL

FIG. E06.1 PANEL ELEMENT: MODIFIED BEAM MODEL

(a) PRIMARY MODES OF DEFORMATION

(b) SECONDARY MODES OF DEFORMATION

(c) RIGID BODY MODES
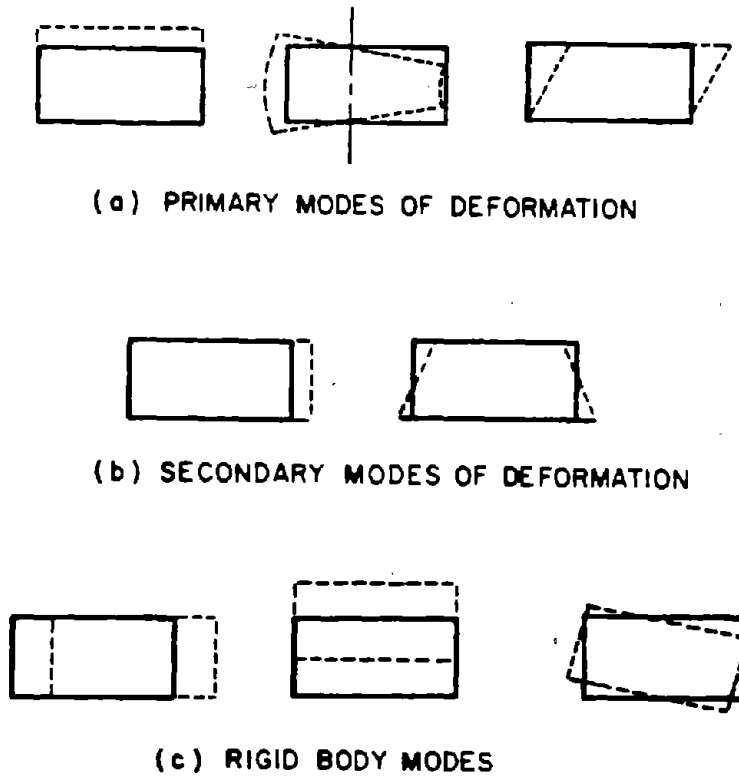
FIG. E06.2 DEFORMATION MODES
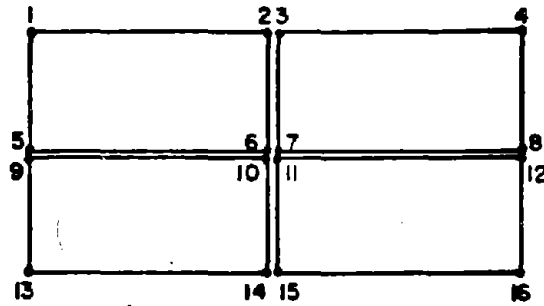
FIG. E06.3 TYPICAL ARRANGEMENT OF PANELS
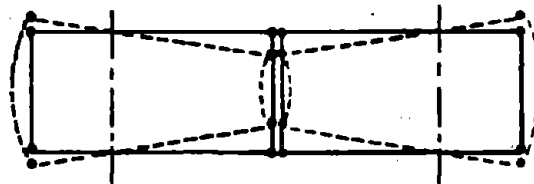
FIG. E06.4 INCOMPATIBILITY ALONG VERTICAL CONNECTIONS

FIG. E06.5 SEVERAL ELEMENTS PER PANEL IMPROVES COMPATIBILITY AT VERTICAL EDGES

(a) ANTISYMMETRIC BENDING   (b) PURE SHEAR
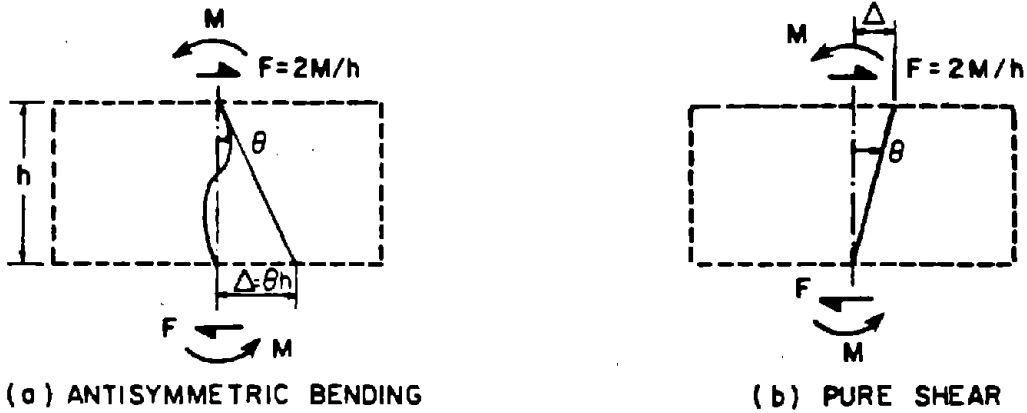
FIG. E06.6 BEAM DEFORMATIONS
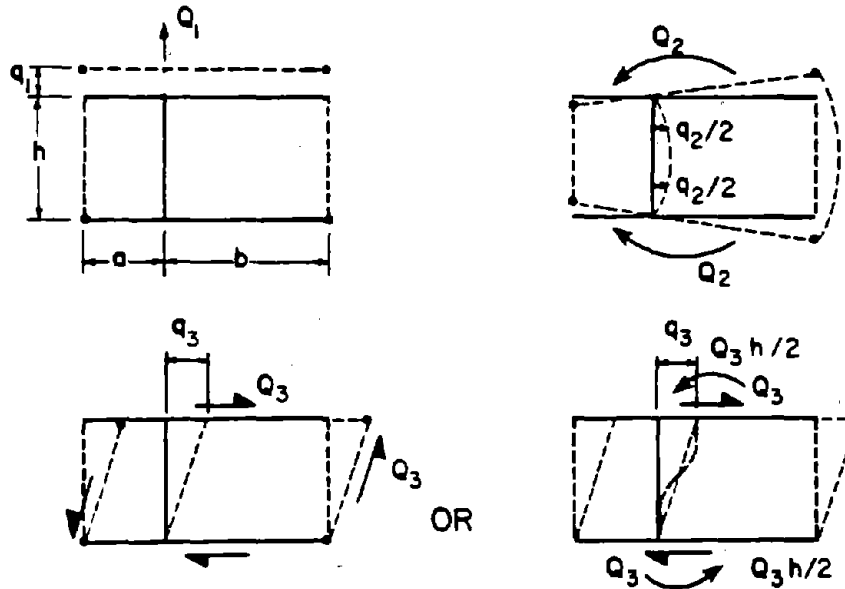
FIG. E06.7 POSITIVE ACTIONS AND DEFORMATIONS FOR PANEL ELEMENT
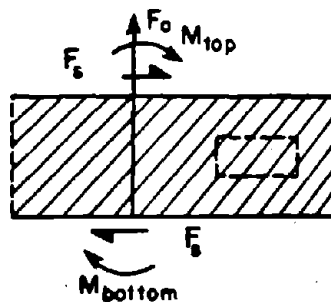
FIG. E06.8 OUTPUT ACTIONS FOR PANEL ELEMENT

DRAIN-2DX USER GUIDE

ELEMENT THEORY

BEAM ELEMENT WITH DEGRADING STIFFNESS (TYPE 07)

E07.1 GENERAL CHARACTERISTICS

The beam element with degrading stiffness is formulated to model reinforced concrete beams, which characteristically exhibit degrading flexural stiffness properties when subjected to cyclic loads. As is the case with the beam-column element (type 02), the element possesses flexural and axial stiffness, and elements of variable cross section can be considered by specifying appropriate flexural stiffness coefficients. Flexural shear deformations and the effects of eccentric end connections can be taken into account.

Yielding can take place only in concentrated plastic hinges at the ends. Strain hardening and degrading flexural stiffness are approximated by assuming that the element consists of a linear elastic beam element with non-linear rotational springs at each end, as shown in Fig. E07.1. All plastic deformation effects, including the effects of degrading stiffness, are introduced by means of the moment-rotation relationships for the hinge springs. Note that this "series" model is fundamentally different from "parallel" model used for element type 02.

The moment-rotation relationship for each hinge is an extended version of Takeda's model, which has the behavior illustrated in Fig. E07.2. The basic relationship is in the form of a bilinear curve, with an initial stiffness and a subsequent strain hardening stiffness which are characteristic of monotonic loading conditions. The degrading stiffness of the hinges is introduced when reversed loading is applied. Fig. E07.2 also indicates rules which are followed for small amplitude oscillations. The numbers on the legs of the relationship are yield codes, which are printed by the computer program.

The extensions which have been made to the Takeda model are shown in Fig. E07.3. These include (1) a reduction of the unloading stiffness, by an amount which depends on

the largest previous hinge rotation, and (2) incorporation of a variable reloading stiffness, which is larger than that of the Takeda model and also depends on the past rotation history. To a large extent these extensions reflect behavior observed in practice. However, some of the features assumed for the extended model, particularly those associated with small amplitude oscillations, have been selected on the basis of engineering judgement, and on the need to avoid illogical or inconsistent patterns of hinge behavior.

The unloading stiffness, K_u, depends on the maximum hinge rotation, and is controlled by the input parameter α. This parameter controls the unloading stiffness by locating the "recovery point", R_{rec}, as shown in Fig. E07.3(a). It must be non-negative, and might typically lie between zero and 0.4. Regardless of the value of α, the unloading slope is never permitted to be less the reloading slope, otherwise a hysteresis loop with a negative area could be produced.

The reloading stiffness, K_l, also depends on the maximum rotation, and is governed by the input parameter β, as shown in Fig. E07.3(b). The parameter β must be non-negative, and might typically lie between zero and 0.6.

The small amplitude relations which have been assumed are very similar to those of the Takeda model. However, it has been necessary to formulate additional rules to compute the reloading stiffness. In the state depicted in Fig. E07.3(c), the reversal point, R_{rev}, lies inside the positive recovery point, R_{rec}^+, because the inelastic excursion did not reach the previous maximum, point B^+. In this situation, it is unlikely that the negative reloading slope would be directed towards point A^-, which would be the reloading point defined in terms of the parameter β and the previous maximum point B^-. If reloading toward point A^- were assumed, the reloading slope would tend to be excessively high, and could even be negative if point R_{rev} were to lie to the left of point A^-. Accordingly, for this type of situation reloading to a point X is assumed, where X lies between points A and B, at a location which depends on the location of R_{rev} between the limiting recovery points R_{rec}^- and R_{rec}^+.

The specific relationship which is assumed for negative reloading is

$$\frac{AX}{AB} = \left(\frac{R_{rec}^{+} - R_{rev}}{R_{rec}^{+} - R_{rec}^{-}} \right)^{N} \tag{E07.1}$$

with a similar relationship for positive reloading. For the case $N=1$, the point X will be positioned between A and B in the same proportion as R_{rev} is positioned between R_{rec}^{+} and R_{rec}^{-}. For the case $N>1$, the point X will be weighted more heavily towards point A, and for the case $N<1$ more heavily towards point B. It is anticipated that a value $N=1$ will be used, but the possibility of specifying some other value is included for greater flexibility. If the factor β is equal to zero, reloading will always take place towards point B, as in the Takeda model, regardless of the value of N.

For a series of small amplitude cycles, as illustrated in Fig. E07.3(d), the assumed behavior is identical to that of the Takeda model, except that the reloading stiffness, from point C to line AB, is based on the position of the most recent reversal point.

The yield moments may be specified to be different at the two element ends, and for positive and negative bending. There is no interaction between axial force and bending moment in producing yield.

Static loads applied within any element length may be taken into account by specifying fixed end force values. Second order effects may be approximated by including a simple geometric stiffness based on the element axial force.

E07.2 ELEMENT DEFORMATIONS

In contrast to the beam-column element, the degrading stiffness element has two additional deformation modes, namely an additional rotational deformation associated with each of the two rotational springs, as shown in Fig. E07.4. These additional degrees of freedom are condensed out at the element level, and do not appear as structure degrees of freedom. The resulting deformation measures for the element are the same as those of the beam-column element, and the displacement transformation is also the same. Within the element,

however, the deformations divide into hinge rotations and deformations of the elastic flexural member.

E07.3 ELEMENT PROPERTIES

In the analysis of reinforced concrete frame structures, difficulties inevitably arise in assigning cross section and material properties. The nonlinear material behavior of the concrete, the nonhomogeneous composition of the cross section, and the presence of a varying cross section, due to cracking and to changing steel areas, all combine to make it difficult to assign accurate stiffness values. Considerable experience and experimentation will be needed before the element properties can be specified with confidence.

An effective flexural stiffness (EI), which might typically be the EI value of the cracked section, must be specified for the linear elastic line element. An axial stiffness (EA) and an effective shear stiffness (GA') must also be specified.

Two options are available for assigning stiffness properties to the hinge springs. The first option is a default option, which provides for automatic generation of the spring properties as follows. If the values for spring stiffness are input blank or zero, the program automatically assigns a very large initial elastic stiffness to the spring (10^8 * EI), so that each hinge is essentially rigid up to yield. The program then assigns a spring strain hardening ratio such that the strain hardening ratio for the $P-\delta$ relationship of a cantilever beam, as shown Fig. E07.5, equals the input value of the strain hardening ratio.

The second option is more general, but requires knowledge that will not always be available. If relationships between tip load, tip displacement and tip rotation for a cantilever are available, the following equations can be used to obtain the effective EI, initial spring stiffness and spring strain hardening ratio (Fig. E07.5).

$$EI = \frac{M_y L}{3(2\theta_y - 2\frac{\delta_y}{L})} \qquad (E07.2)$$

$$K_{sp} = \frac{M_y L}{(\delta_y - M_y \frac{L^2}{3EI})} \tag{E07.3}$$

$$p = \frac{(P_u L - M_y) L}{K_{sp} (\delta_u - \delta_y - (P_u L - M_y) \frac{L^2}{3EI})} \tag{E07.4}$$

in which K_{sp} = initial spring stiffness; p = spring strain hardening ratio; M_y = yield moment; δ_y = displacement of cantilever tip at yield; θ_y = rotation of cantilever tip at yield; P_u, δ_u = any corresponding tip load and deflection along the strain-hardening part of the $P - \delta$ relationship; and L = cantilever length. An idealized cantilever with these properties will follow a specified $P - \delta$ relationship as shown in Fig. E07.5(d), and a $P - \theta$ relationship as shown in Fig. E07.5(e). The strain hardening ratio of the $P - \theta$ relationship will not necessarily be equal to that observed in a test. If this second option is used, the computed EI and spring properties should be examined to ensure that they are physically reasonable.

Spring properties based on a cantilever $P - \delta$ relationship will reproduce a specified $M - \theta$ relationship for a beam bent in reverse double curvature, as indicated in Fig. E07.5. For other bending moment variations, the idealized model will not necessarily have the same moment-rotation relationship as an actual beam, and the results of any analysis must be interpreted with caution. If the bending moment variation is complex, and if it is particularly important that accurate results be obtained, the following idealization procedure may be used. This procedure makes use of the second option for specifying the hinge spring properties. Its disadvantage is that it requires each beam of the actual structure to be subdivided into several beam elements for analysis, and hence that the computational cost will be increased. the procedure has the advantage, however, that the moment-curvature relationship for the beam can be considered directly, rather than some less direct moment-rotation relationship.

The moment-rotation and moment-curvature relationships for a beam will be of the same form provided the beam is subjected to a constant bending moment. If any complete

beam is subdivided into short elements, as indicated in Fig. E07.6(a), so that the variation in bending moment over the length of any yielding element is small, then each yielding element is subjected to nearly constant moment, and it is reasonable to assume similar forms for the moment-curvature and moment-rotation relationships. The moment-rotation relationship corresponding to bilinear beam element properties can be obtained from the following equations.

$$EI = \frac{M_y}{\theta_y} \qquad (E07.5)$$

$$K_{sp} = 10^8 EI \qquad (E07.6)$$

$$P = \frac{2(M_u - M_y)}{K_{sp} (\theta_u - M_u/EI)} \qquad (E07.7)$$

in which M_u, θ_u are any corresponding points on the strain hardening part of the $M-\theta$ relationship.

E07.4 STATIC ELASTO-PLASTIC STIFFNESS

The element is considered as a linear elastic beam element with nonlinear rotational springs at each end. The element actions and deformations are shown in Fig. E07.4. The axial stiffness is constant and is given by

$$dS_1 = \frac{EA}{L} dv_1 \qquad (E07.8)$$

in which EA is the effective axial stiffness of the cross section. The elastic flexural stiffness of the elastic beam element is given by

$$\begin{Bmatrix} dS_4 \\ dS_5 \end{Bmatrix} = \frac{EI}{L} \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ij} & k_{jj} \end{bmatrix} \begin{Bmatrix} dv_4 \\ dv_5 \end{Bmatrix} \qquad (E07.9)$$

in which EI is the effective flexural stiffness of the cross section, and k_{ii}, k_{jj}, k_{ij} are input coefficients which depend on the cross section variation. For a uniform element, $k_{ii} = k_{jj} = 4.0$, and $k_{ij} = 2.0$. These coefficients must be specified by the program user, and may, if desired, account for such effects as shear deformations and nonrigid end connections as well as cross section variations.

To form the complete element stiffness, the elastic element stiffness is inverted to obtain its flexibility, and modified to account for flexural shear deformations. The current hinge flexibility values are then added to give the element flexibility matrix as

$$
\mathbf{f}_e = \begin{bmatrix} f_{ii} + \dfrac{1}{K_{sp_i}} & f_{ij} \\[2ex] f_{ij} & f_{jj} + \dfrac{1}{K_{sp_j}} \end{bmatrix}
\tag{E07.10}
$$

This flexibility matrix is then inverted to form the elasto-plastic stiffness, $[K_{ep}]$.

As for the beam-column element, the stiffness in terms of global nodal displacements is obtained as

$$
\mathbf{K}_{ep} = \mathbf{a}^T \, \mathbf{k}_{ep} \, \mathbf{a}
\tag{E07.11}
$$

E07.5 OTHER PROPERTIES

The remaining properties are essentially as for the beam-column element (type 02).

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.07

BEAM ELEMENT WITH DEGRADING STIFFNESS (TYPE 07)

See Fig. E07.1 through E07.7 for element behavior and properties.

C2.07(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See Section C2.07(b).
6-10(I)			No. of eccentricity types (max. 40). See Section C2.07(c).
11-15(I)			No. of yield surfaces (max. 40). See Section C2.07(d).

C2.07(b). Stiffness Types

Two lines for each stiffness type.

Line 1: Beam properties

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			EI, reference flexural stiffness.
16-25(R)			EA, reference axial stiffness.
26-35(R)			GA', effective shear stiffness. If blank or zero, shear deformation are ignored.
36-40(R)			Flexural stiffness factor k_{ii}.
41-45(R)			Flexural stiffness factor k_{jj}.
46-50(R)			Flexural stiffness factor k_{ij}.
51-60(R)			Strain hardening ratio for inelastic flexure at end I. If a non-zero hinge stiffness is specified for end I on line 2, columns 6-15, then this strain hardening ratio will apply directly to the hinge moment-rotation relationship. Otherwise, this ratio will apply to the overall beam end moment-rotation relationship or cantilever $P-\Delta$ relationship.
61-70(R)			Strain hardening ratio for inelastic flexure at end J. As for end I, a zero or non-zero hinge stiffness for end J on line 2, columns 6-15, will control whether this ratio is applied to the hinge alone or to the beam as a whole.

Line 2: Hinge Properties

Leave blank to obtain Takeda Model.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1 and corresponding to the stiffness type number on **line 1**.
6-15(R)			Hinge stiffness at end I. Leave blank or zero if hinge properties are to be determined by the program (these properties are marked by "*" in the output). If blank or zero, the following field for end J must also be blank or zero. If non-zero, the following field must also be nonzero.
16-25(R)			Hinge stiffness at end J.
26-35(R)			α_i, unloading parameter for end I. Leave blank or zero for unloading according to Takeda model.
36-45(R)			α_j, unloading parameter for end J. Leave blank or zero for unloading according to Takeda model.
46-55(R)			β_i, loading parameter for end I. Leave blank or zero for unloading according to Takeda model.
56-65(R)			β_j, loading parameter for end J. Leave blank or zero for unloading according to Takeda model.
66-75(R)			N, loading exponent.

C2.07(c). End Eccentricities

One line for each end eccentricity, as for beam-column element. All eccentricities are measured **from** the node **to** the element end.

Columns	Notes	Variable	Data
1-5(I)			End eccentricity number, in sequence beginning with 1.
6-15(R)			X_i = X eccentricity at end I.
16-25(R)			X_j = X eccentricity at end J.
26-35(R)			Y_i = Y eccentricity at end I.
36-45(R)			Y_j = Y eccentricity at end J.

C2.07(d). Cross Section Yield Surfaces

One line for each yield surface.

Columns	Notes	Variable	Data
1-5(I)			Yield surface number, in sequence beginning with 1.
6-15(R)			Positive yield moment, M_{y+}.
16-25(R)			Negative yield moment, M_{y-}.

C2.07(e). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and 'ast elements **must be included**.

Columns	Notes	Variable	Data
1-5(I)	C.5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element end I.
11-15(I)			Node number at element end J.
16-20(I)			Node number increment for element generation. Default = 1.
21-25(I)			Stiffness type number.
26-30(I)			End eccentricity number. Default = no end eccentricity.
31-35(I)			Yield surface number at end I.
36-40(I)			Yield surface number at end J.

DRAIN-ANAL USER GUIDE

INPUT DATE SECTION D2(b)(ii).07

ELEMENT LOAD DATA FOR BEAM ELEMENT WITH DEGRADING STIFFNESS (TYPE 07)

D2(b)(ii).07(a). Element Load Sets

NLOD lines (see Section D2(b)(i)), one line per element load set.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
10(I)			Coordinate Code. (a) 0: Forces are in local (element) coordinates. (b) 1: Forces are in global (structure) coordinates.
11-20(R)			Live load reduction factor.
21-30(R)			Clamping force, F_i.
31-40(R)			Clamping force, V_i.
41-50(R)			Clamping moment, M_i.
51-60(R)			Clamping force, F_j.
61-70(R)			Clamping force, V_j.
71-80(R)			Clamping moment, M_j.

D2(b)(ii).07(b). Loaded Elements and Load Set Scale Factors

As many lines as needed. Terminate with a blank line.

Columns	Notes	Variable	Data
1-5(I)			Number of first element in a series.
6-10(I)			Number of last element in a series.
11-15(I)			Element number increment. Default = 1.
16-20(I)			Load set number.
21-30(R)			Load set scale factor.
31-45			Optional second load set number and scale factor.
46-60			Optional third load set number and scale factor.
61-75			Optional fourth load set number and scale factor.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

BEAM ELEMENT WITH DEGRADING STIFFNESS (TYPE 07)

Item	Description
1	Node number at end I.
2	Yield code at end I (1: hinge; 0: no hinge).
3	Bending moment at end I.
4	Shear force at end I.
5	Axial force at end I.
6	Current hinge rotation at end I.
7	Accumulated primary positive plastic hinge rotation at end I.
8	Accumulated primary negative plastic hinge rotation at end I.
9	Accumulated secondary positive plastic hinge rotation at end I.
10	Accumulated secondary negative plastic hinge rotation at end I.
11	Node number at end J.
12	Yield code at end J (1: hinge, 0: no hinge).
13	Bending moment at end J.
14	Shear force at end J.
15	Axial force at end J.
16	Current hinge rotation at end J.
17	Accumulated primary positive plastic hinge rotation at end J.
18	Accumulated primary negative plastic hinge rotation at end J.
19	Accumulated secondary positive plastic hinge rotation at end J.
20	Accumulated secondary negative plastic hinge rotation at end J.
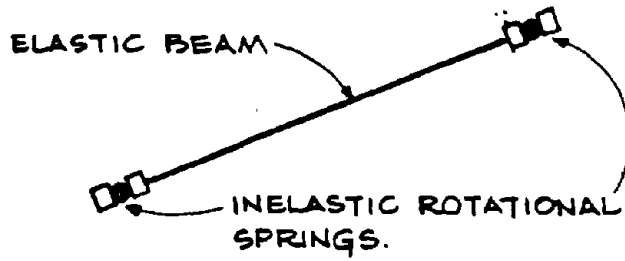
FIG. E07.1 ELEMENT IDEALIZATION

Numbers indicate yield code
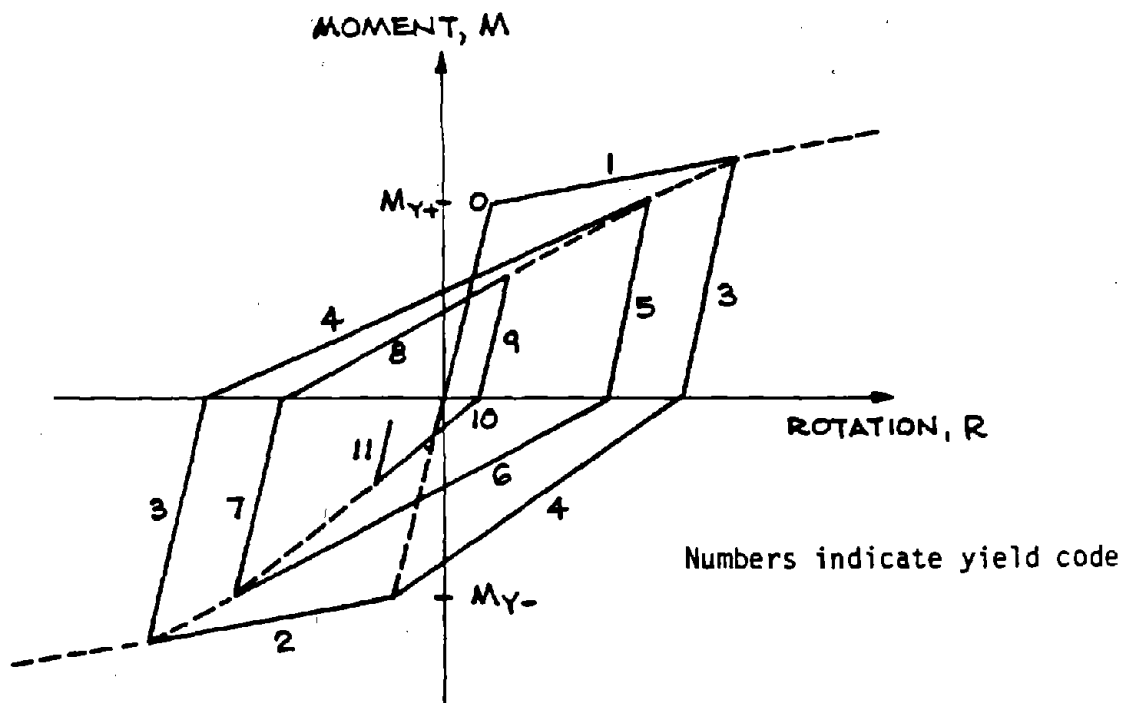
FIG. E07.2 HINGE MOMENT – ROTATION RELATIONSHIP
FOR TAKEDA MODEL
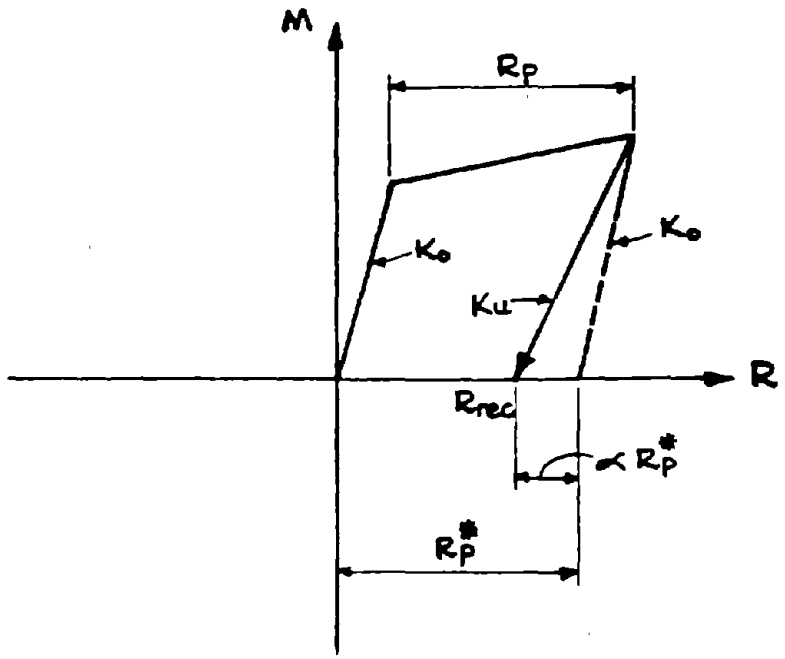
(a) Unloading Stiffness
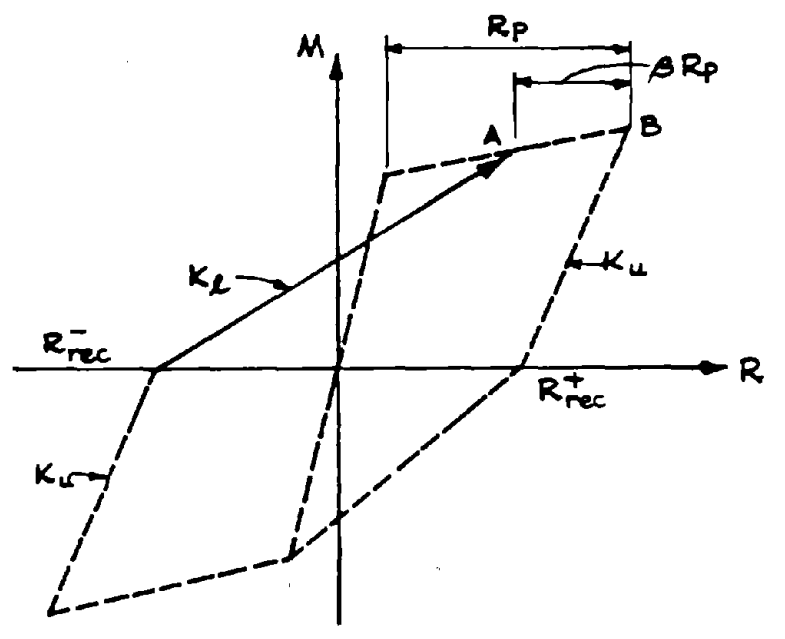
(b) Reloading Stiffness

FIG. E07.3 HINGE MOMENT – ROTATION RELATIONSHIP FOR EXTENDED TAKEDA MODEL
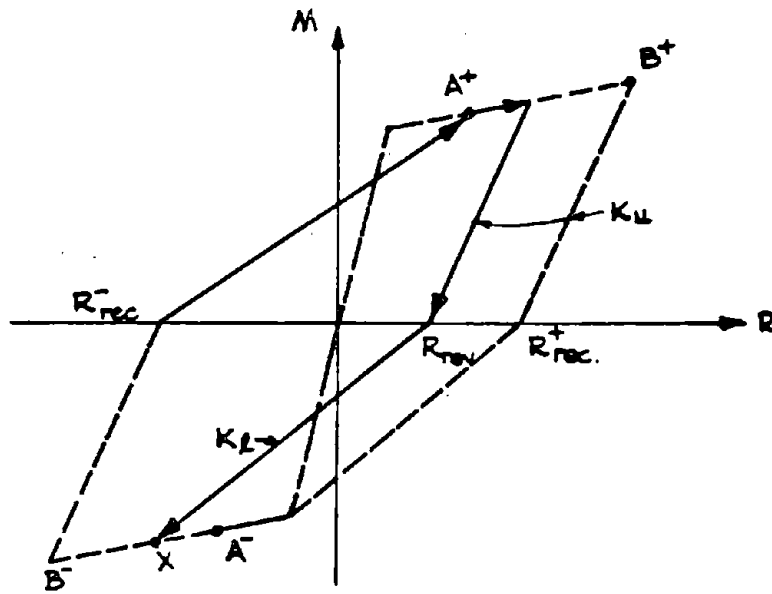
(c)   Reloading After Small Yield Excursion
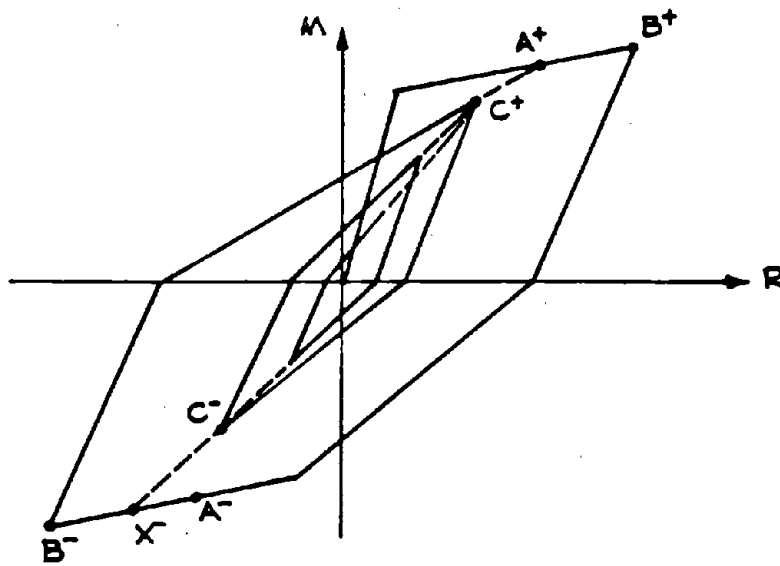
(d)   Small Amplitude Behavior

FIG. E07.3 (Continued)
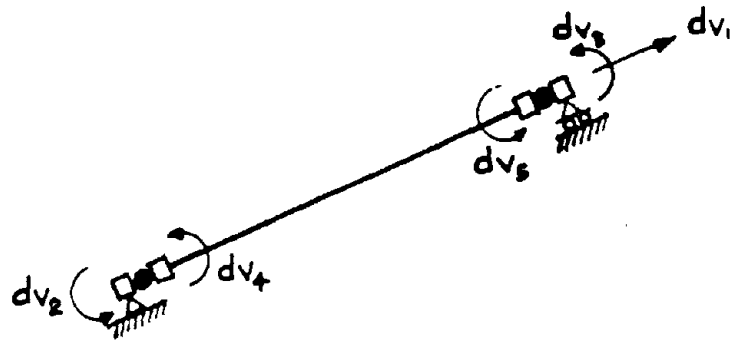
FIG. E07.4 ELEMENT DEFORMATIONS

(a) Beam

(b) Equivalent Cantilever

(c) Idealized Cantilever

(d) P- δ Relationship

(e) P-θ Relationship
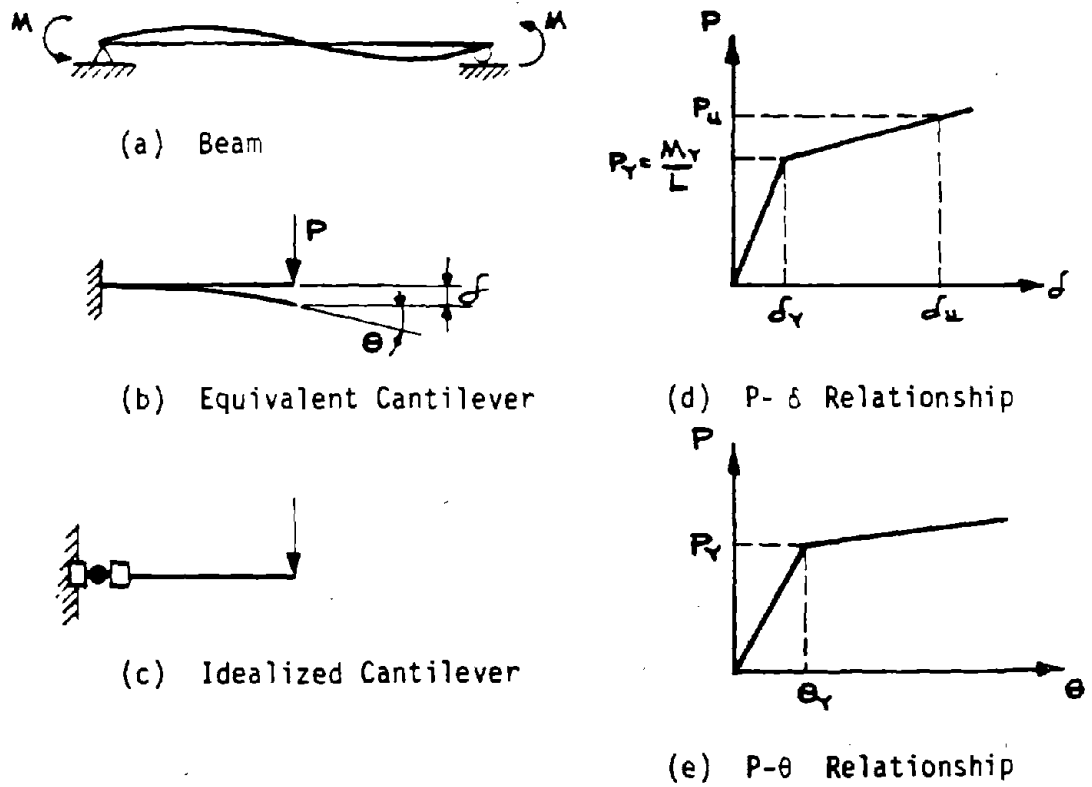
FIG. E07.5 REPRESENTATION OF CANTILEVER BEHAVIOR
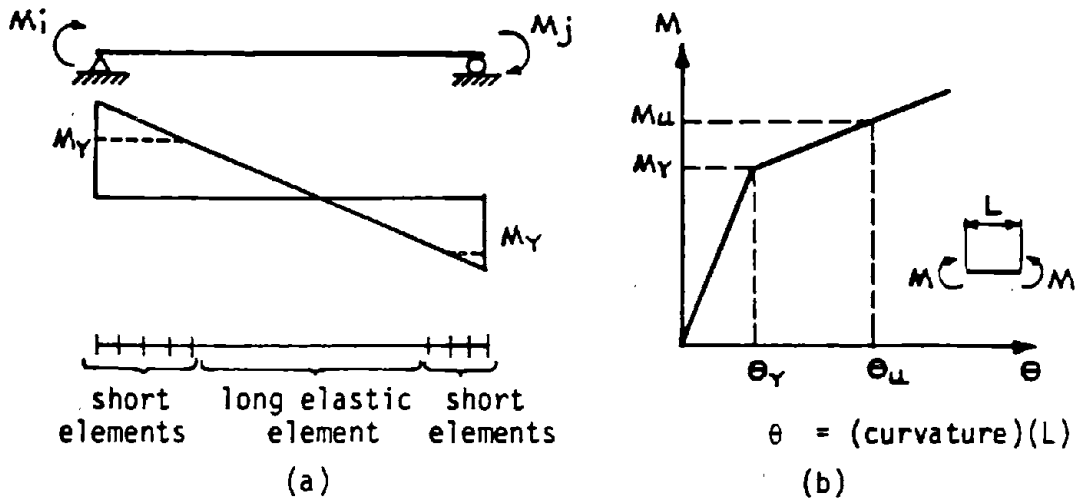
FIG. E07.6 SUBDIVISION OF BEAM INTO SHORT ELEMENTS
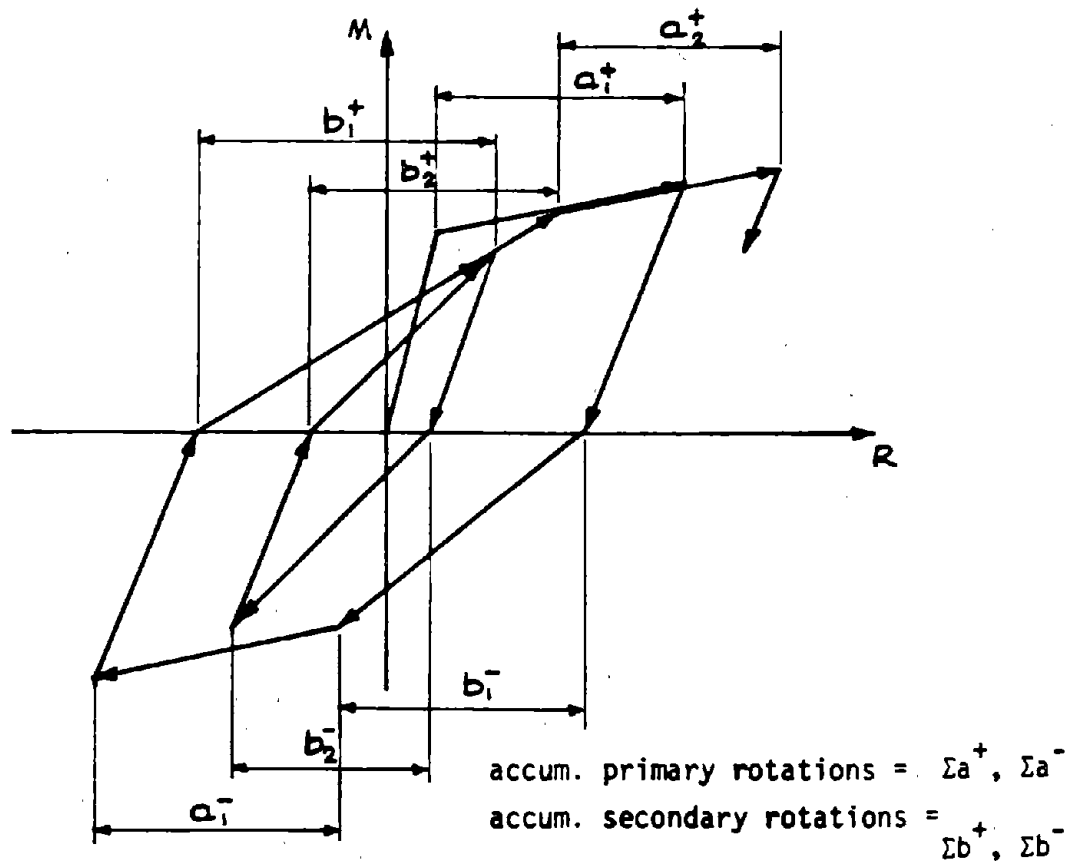
FIG. E07.7 DEFINITION OF ACCUMULATED ROTATIONS

DRAIN-2DX USER GUIDE

ELEMENT THEORY

LINK ELEMENT (TYPE 09)

E09.1 GENERAL CHARACTERISTICS

E09.1.1 Basic Properties

The link element is a uniaxial element with finite length, which may be arbitrarily oriented (i.e. not necessarily horizontal or vertical). The element has zero stiffness in tension and a finite stiffness in compression.

The force-displacement relationship is as shown in Fig. E09.1. Either one of two unloading paths for the element, namely elastic or inelastic, may be specified. For the inelastic option, unloading takes place parallel to the initial elastic stiffness k_1. The element can be preloaded to a specified compression force if desired, or alternatively can be prestrained in tension to give a specified initial clearance. The element can thus function as a prestressed bearing element or as an element with an initial gap.

Each link element must be connected to two nodes, which must not have identical coordinates. For large panel applications, link elements will usually, be placed in either horizontal or vertical joints, connecting nodes on adjacent panels.

E09.1.2 Link Element Use to Model a Stop

A stop in a joint will permit a certain amount of slip and will then develop resistance to constrain further slip. Such behavior could be produced by a set of unbonded dowels projecting vertically through a horizontal joint, and placed in slots or clearance holes. A stop can be modeled by placing a pair of link elements, with initial clearances, between panels as shown in Fig. E09.2a. Fig. E09.2b shows the resulting force-deformation characteristic (assuming elastic behavior).

E09.1.3 Other Combination of Link Elements

A pair of link elements can also be used to obtain the behavior shown in Fig. E09.3. In this case, the elements have no initial gap or preload, and are specified to be elastic-plastic in compression. More complex behavior can also be obtained (Fig. E09.4) by specifying a multi-linear relationship in compression for each link element (in this case, $k_2 = 0$).

E09.2 STATIC ELASTO-PLASTIC STIFFNESS

The link element has four displacement degrees of freedom and one deformation degree of freedom, as shown in Fig. E09.5. The relationship between nodal displacement and element deformation is

$$q = < \cos\theta \quad \sin\theta \quad -\cos\theta \quad -\sin\theta > \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \end{Bmatrix} \qquad (E09.1)$$

or

$$q = a \ r \qquad (E09.2)$$

Hence, the elasto-plastic stiffness matrix is

$$K = a^T k a \qquad (E09.3)$$

where k is the element tangent stiffness (i.e., k_1, k_2, k_3, or zero).

E09.3 OTHER PROPERTIES

No provision is made for second order effects or element loads. Also, viscous damping is assumed to be zero, regardless of the value of βK (see discussion for the gap-friction element).

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.09

LINK ELEMENT (TYPE 09)

See Fig. E09.1 through Fig. E09.5 for element behavior and properties.

C2.09(a). Control Information
One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.09(b).

C2.09(b). Stiffness Types
One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
10(I)			Unloading code. (a) 0: Inelastic unloading. (b) 1: Elastic unloading.
11-20(I)			Displacement limit u_1.
21-30(R)			Displacement limit u_2.
31-40(R)			Stiffness k_1.
41-50(R)			Stiffness k_2.
51-60(R)			Stiffness k_3.

C2.09(c). Element Generation Commands

One line for each generation command. the first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included**.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(I)			Node number at element end I.
11-15(I)			Node number at element end J.
16-20(I)			Stiffness type number.
21-30(R)			Initial bearing force or gap. (a) < 0.0 Initial gap. (b) > 0.0 Initial bearing force.
35(I)			Node number increment for element generation. Default=1

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

LINK ELEMENT (TYPE 09)

Item	Description
1	Node number at end I.
2	Node number at end J.
3	Line number.
4	Bearing force.
5	Bearing deformation.
6	Accumulated gap opening.
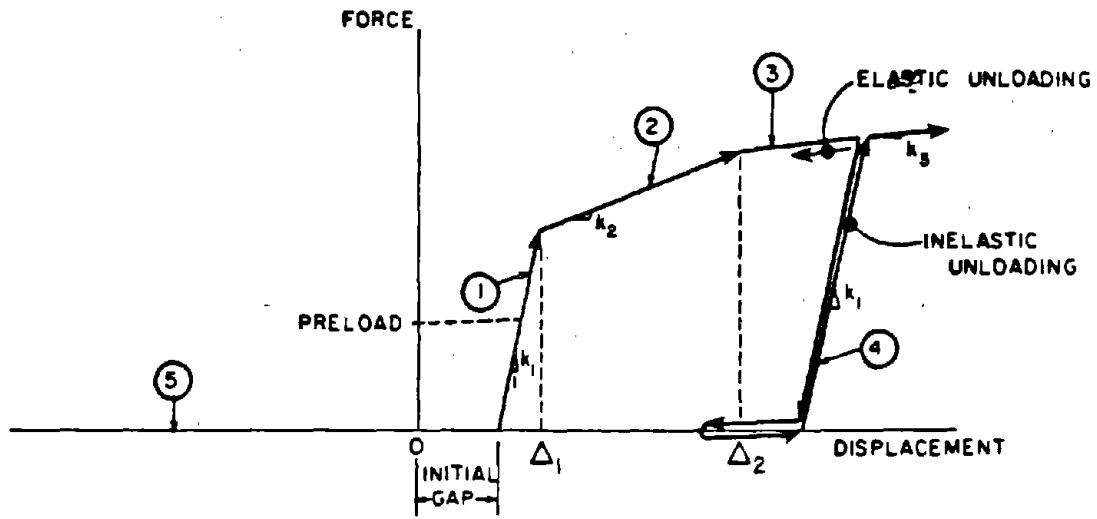
FIG. E09.1 LINK ELEMENT: FORCE – DISPLACEMENT RELATIONSHIP

(a) JOINT MODELLING

(b) JOINT CHARACTERISTIC
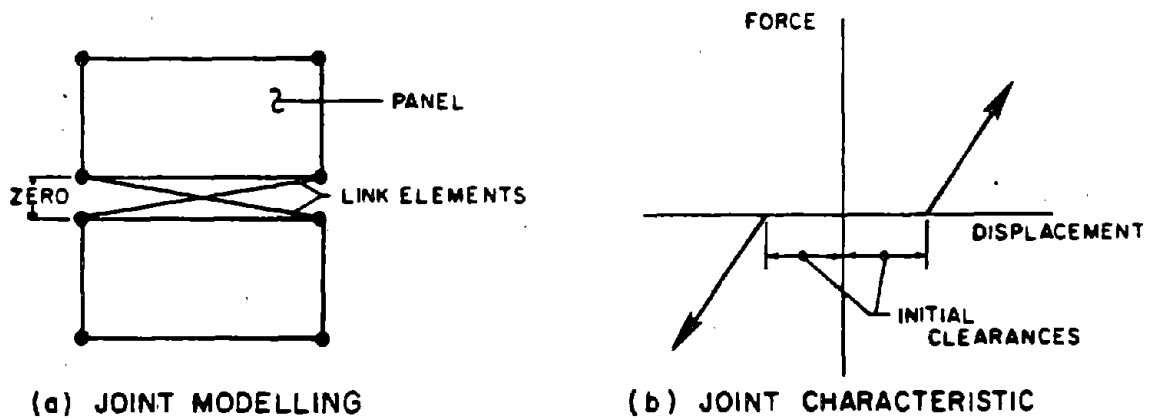
FIG. E09.2 STOP ELEMENT IN A HORIZONTAL JOINT
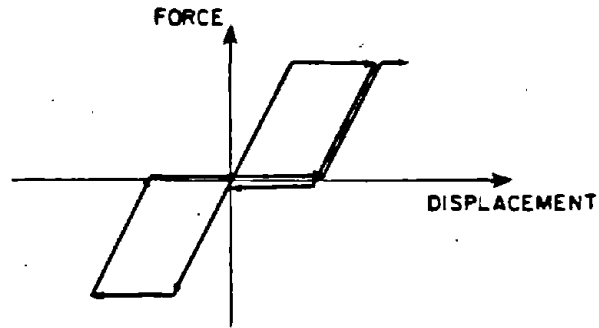
FIG. E09.3 TWO LINK ELEMENT WITH $K_2 = 0$ AND $\triangle_2 = \infty$
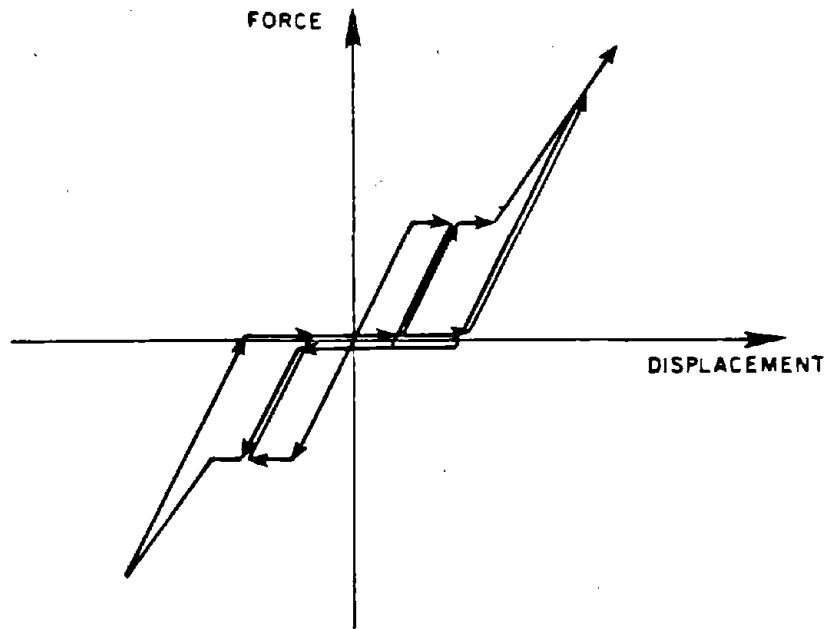
FIG. E09.4 TWO LINK ELEMENTS WITH $K_2 = 0$ AND $K_3 < K_1$

(a) POSITIVE
ACTION - DEFORMATION

(b) DISPLACEMENT
D.O.F
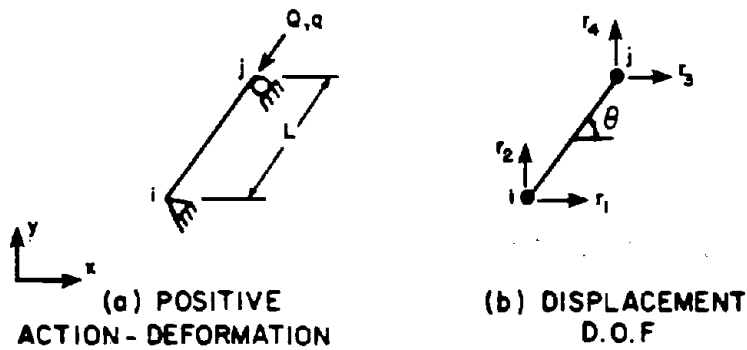
FIG. E09.5 DEGREES OF FREEDOM FOR LINK ELEMENT

DRAIN-2DX USER GUIDE

ELEMENT THEORY

KEY JOINT ELEMENT (TYPE 10)

E10.1 GENERAL CHARACTERISTICS

The key element allows modeling of mechanical keys in horizontal or vertical joints. The element can also be used to model connections which have initial adhesion or strong bond.

Each key element can be connected to either two or four nodes on adjacent panels. Elements may be specified at horizontal and/or vertical joints.

The relationship between shear force and shear displacement between the connected nodes is as shown in Fig, E10.1. The force-displacement relationship is assumed to be initially elastic (line 1, Fig. E10.1). The initial stiffness will usually be specified to be large. At a specified yield strength, the element yields at constant force (line 2). Beyond a specified yield displacement the element loses strength along line 3, simulating failure of the key. When the strength of the element reaches zero, total failure is assumed and the behavior follows line 5. After failure, no further shear resistance is provided by the element.

The failure may be either brittle or ductile, as shown in Fig. E10.2. A brittle failure is characterized by a short yield plateau and subsequent steep decline along line 3. A ductile failure is characterized by a longer plateau and slow decline.

In the absence of detailed experimental results, unloading has been assumed to take place along line 4. This assumes partial key failure, with consequent loss of stiffness.

An actual keyed joint will not lose strength completely after failure of the key, but will retain frictional strength. A keyed joint will therefore usually be modeled using a key element in parallel with a friction element. Key elements can also be placed in parallel with other element types, to obtain complex degrading strength characteristics.

E10.2 NEGATIVE STIFFNESS

It should be noted that the tangent stiffness associated with line 3 (Fig. E10.1) is negative. Thus, diagonal terms in the element stiffness matrix become negative, and if line 3 is steep, the structure tangent stiffness matrix may no longer remain positive definite. This may cause computational difficulties, and hence large negative slopes for line 3 should be used with caution.

E10.3 STATIC ELASTO-PLASTIC STIFFNESS

The four-node element has four displacement degrees of freedom and the two-node element has two, as shown in Fig. E10.3. The element has one deformation degree of freedom, namely the average relative shear displacement across the joint. The deformation-displacement transformation for a four-node element is

$$q = \frac{1}{2} <1 \quad 1 \quad -1 \quad -1> \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \end{Bmatrix} \tag{E10.1}$$

or

$$q = a\,r \tag{E10.2}$$

The basic action-deformation relationship is

$$Q = kq \tag{E10.3}$$

in which k is the current tangent stiffness of the element. Hence, the element stiffness matrix is

$$K = a^T k a \tag{E10.4}$$

The relationship for the two-node element is the same, except that

$$q = <1 \quad -1> \begin{Bmatrix} r_1 \\ r_2 \end{Bmatrix} \tag{E10.5}$$

E10.4 OTHER PROPERTIES

No provision is made for second order effects or element loads. Also, viscous damping is assumed to be zero, regardless of the value of βK (see discussion for the gap-friction element).

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.10

KEY JOINT ELEMENT (TYPE 10)

See Fig. E10.1 through Fig. E10.3 for element behavior and properties.

C2.10(a). Control Information
One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.10(b).

C2.10(b). Stiffness Types
One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Displacement limit u_1.
16-25(R)			Displacement limit u_2.
26-35(R)			Displacement limit u_3.
36-45(R)			Yield strength, F_{yp}.

C2.10(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included.**

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-10(1)	.		Node number at element end I.
11-15(I)			Node number at element end J.
16-20(1)			Node number increment for element generation. Default = 1.
21-25(1)			Stiffness type number.
30(I)			Horizontal or vertical joint code. (a) 0: Horizontal joint. (b) 1: Vertical joint.

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

KEY JOINT ELEMENT (TYPE 10)

Item	Description
1	Node number at end I.
2	Node number at end J.
3	Line number.
4	Shear force.
5	Shear deformation.
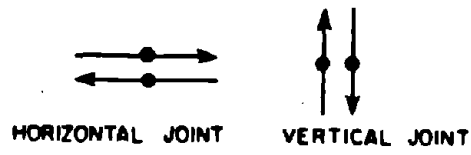
(a) POSITIVE DEFORMATION
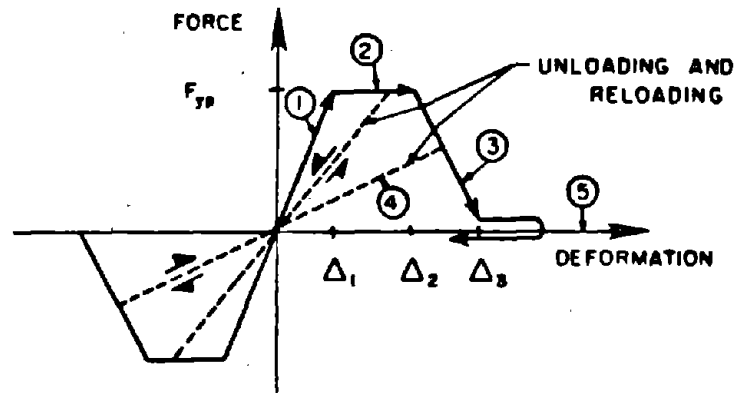
(b) FORCE- DEFORMATION RELATIONSHIP

FIG. E10.1 CHARACTERISTICS OF KEY ELEMENT
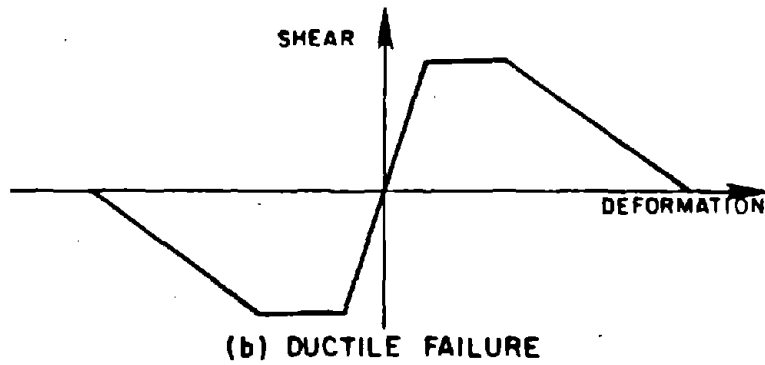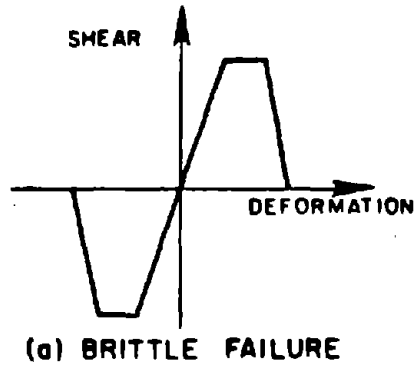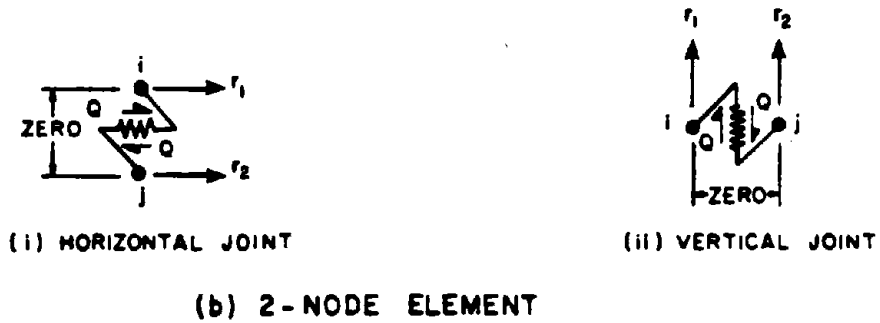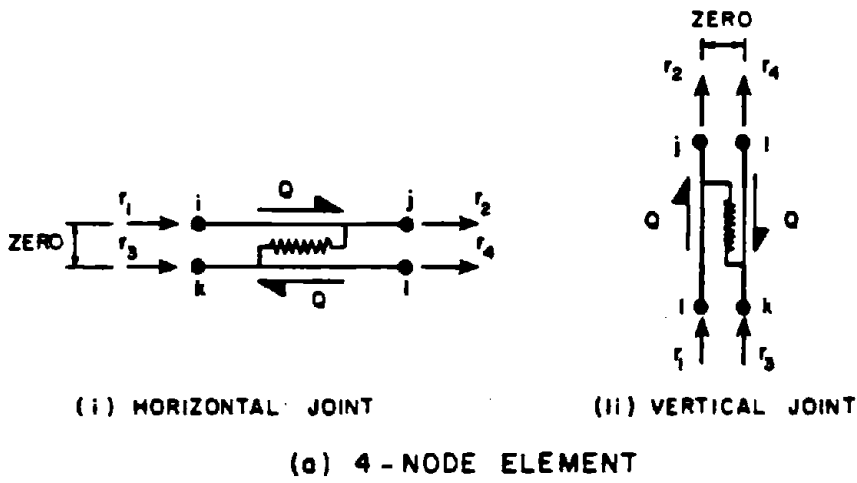
FIG. E10.2 BRITTLE AND DUCTILE CHARACTERISTICS

(i) HORIZONTAL JOINT   (ii) VERTICAL JOINT

(a) 4 - NODE ELEMENT

(i) HORIZONTAL JOINT   (ii) VERTICAL JOINT

(b) 2 - NODE ELEMENT

FIG. E10.3 KEY ELEMENT DEGREES OF FREEDOM

DRAIN-2DX USER GUIDE

ELEMENT THEORY

DEGRADING FRICTION JOINT ELEMENT (TYPE 11)

E11.1 GENERAL CHARACTERISTICS

If a joint is unreinforced or has very little reinforcing, its behavior in shear is dominated by friction. If the normal force on the joint is essentially constant, it may be reasonable to assume a simple elastic-plastic shear-slip relationship. The friction element can be specified to have a non-degrading friction strength equal to μF_n, where μ is the coefficient of friction and F_n is the compressive force on the joint, both assumed to remain constant. Alternatively, the coefficient of friction, μ, can be specified to degrade.

Each friction element must be connected to four nodes, two on one panel and two in an adjacent panel. For a horizontal joint, the nodes must be as shown in Fig. E11.1a, and for a vertical joint as shown in Fig. E11.1b. The element restrains relative sliding between the panels, with a force-slip relationship as shown in Fig. E11.2. The initial shear stiffness, k_s, will typically be large. The stiffness after slip commences, k_h, will typically be small or zero. The unloading stiffness is equal to the initial stiffness. If desired, the element can be used in parallel with other elements to obtain more complex behavior. The element (joint) thickness must be zero, as indicated in Fig. E11.1, otherwise the solution will violate equilibrium.

Because the stiffness k_s will typically be large, the joint stiffness increases a great deal when a slipping element reverses. This can lead to numerical stability problems if event calculations are not performed. The problem is illustrated in Fig. E11.3a, where it can be seen that the state determination procedure can lead to a large unbalanced force. To avoid this problem, an option is provided to assume behavior as shown in Fig. E11.3b on unloading. With this option, the element stiffness is assumed to change at the end of the step, and there is no unbalanced force.

E11.2 STATIC ELASTO-PLASTIC STIFFNESS

The element has four displacement degrees of freedom, as shown in Fig. E11.4, and one deformation degree of freedom. The element deformation is given by

$$q = \frac{1}{2}(r_1 + r_2) - \frac{1}{2}(r_3 + r_4) \qquad \text{(E11.1)}$$

which is the average slip across the joint, or

$$q = \mathbf{a}\,\mathbf{r} \qquad \text{(E11.2)}$$

where the displacement transformation matrix is

$$\mathbf{a} = \frac{1}{2}<1 \quad 1 \quad -1 \quad -1> \qquad \text{(E11.3)}$$

The basic stiffness relationship between the element action, Q, and deformation, q, is

$$Q = kq \qquad \text{(E11.4)}$$

in which k is the tangent shear stiffness(k_s or k_n). Hence, the tangent stiffness matrix is

$$\mathbf{K} = \mathbf{a}^T \mathbf{k}\,\mathbf{a} = \frac{k}{4}\begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \qquad \text{(E11.5)}$$

E11.3 DEGRADING FRICTION

The behavior of a friction joint may be influenced by polishing of the joint surfaces under cyclic loading, with a progressive reduction in the coefficient of friction. The element has an option to allow for decrease of the friction force as the joint accumulates slip. The reduction may be caused by polishing of the joint or by the accumulation of sand and aggregate particles in the joint, which tend to roll and facilitate slippage.

A multi-linear relationship is assumed between the friction coefficient and the accumulated joint slip (sum of the absolute values of the slip excursions), as shown in Fig. E11.5. This type of relationship permits modeling of either sudden loss or gradual degradation of joint strength. The stiffnesses k_s and k_h are assumed not to be affected by strength degradation.

The element has a basic bilinear force-slip relationship, which is decomposed into two components, one elastic and one inelastic, as shown in Fig. E11.2. The degrading strength feature is applied to the inelastic component. In any analysis step, the friction strength is first assumed to be constant, equal to the strength at the beginning of the step. The slip (if any), and hence the accumulated slip, are calculated. The element strength is then obtained at the end of the step, using the specified degrading characteristic (Fig. E11.1). Because of this adjustment there is an equilibrium error at the end of the step. This implies a shear-slip relationship as shown in Fig. E11.3.

E11.4 OTHER PROPERTIES

No provision is made for second order effects or element loads. Also, viscous damping is assumed to be zero, regardless of the value of βK (see discussion for the gap-friction element).

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.11

DEGRADING FRICTION JOINT ELEMENT (TYPE 11)

See Fig. E11.1 through E11.7 for element behavior and properties.

C2.11(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.11(b).

C2.11(b). Stiffness Types

One line for each stiffness type.

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Initial shear stiffness k_s.
16-25(R)			Strain hardening shear stiffness k_h, as a **proportion** of initial shear stiffness k_s. Must be < 1.0.
26-35(R)			Shear strength, F_{s0}.
36-45(R)	·		Shear force F_{s1}, for friction degradation.
46-55(R)			Shear force F_{s2}, for friction degradation..
56-65(R)			Accumulated slip S_1, for friction degradation.
66-75(R)			Accumulated slip S_2, for friction degradation..

C2.11(c). Element Generation Commands

One line for each generation command. The first element can be assigns any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements **must be included.**

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
10(I)			Element orientation code (a) 0: Horizontal joint. (b) 1: Vertical joint.
11-15(I)			Node I.
16-20(I)			Node J. May be same as I.
21-25(I)			Node K. Must have same coordinates as node I
26-30(I)			Node L. May be same as K. Must have same coordinates as node J
31-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Stiffness type number.
45(I)			Unloading code for shear component. (a) 0: unloading as in Fig. E11.3a (b) 1: unloading as in Fig. E11.3b

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

DEGRADING SHEAR JOINT ELEMENT (TYPE 11)

Item	Description
1	Node number at point I.
2	Node number at point J.
3	Node number at point K.
4	Node number at point L.
5	Shear yield code (0: elastic ; 1: yielded).
6	Shear force.
7	Current yield strength.
8	Shear deformation.
9	Accumulated positive shear deformation.
10	Accumulated negative shear deformation.

(a) HORIZONTAL JOINT

(b) VERTICAL JOINT
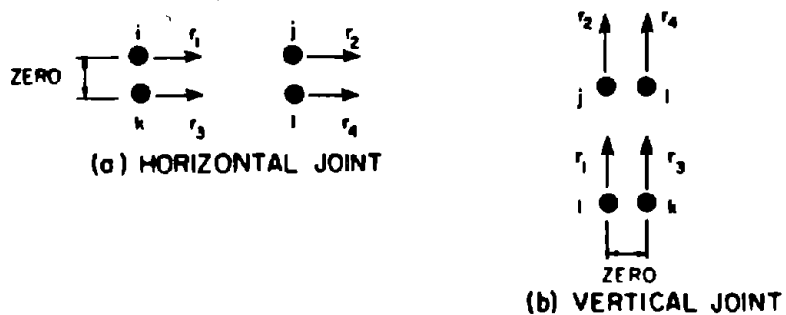
FIG. E11.1 ELEMENT DEGREES OF FREEDOM
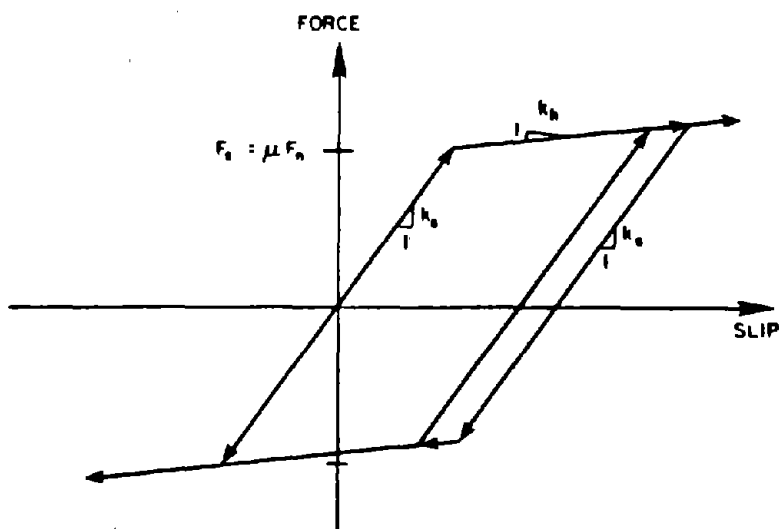
FIG. E11.2 FORCE – SLIP RELATIONSHIP

(a) PROCEDURE 1   (b) PROCEDURE 2
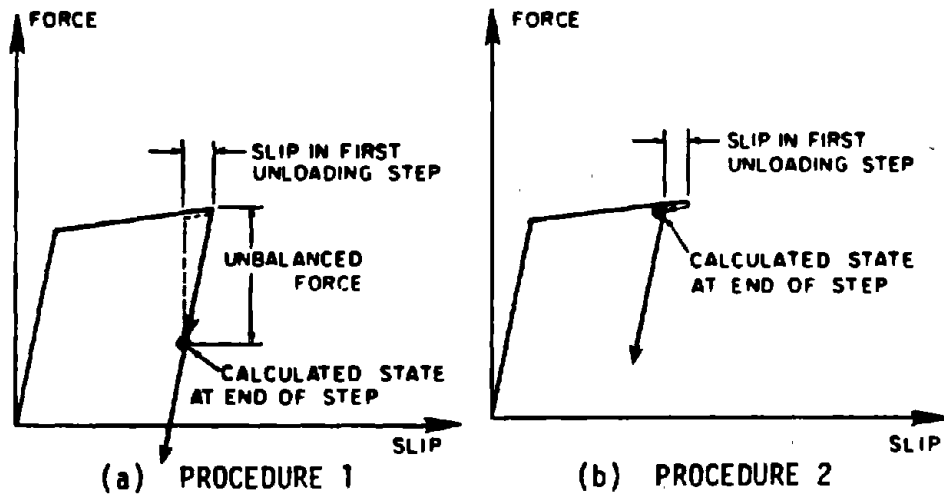
FIG. E11.3 STATE DETERMINATION PROCEDURES

(a) HORIZONTAL JOINT   (b) VERTICAL JOINT
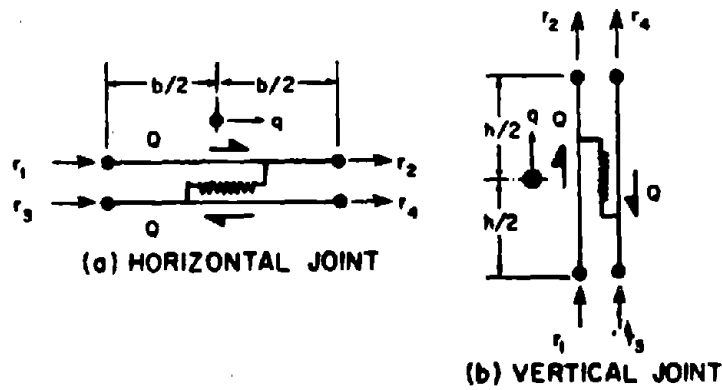
FIG. E11.4 POSITIVE SHEAR FORCE AND DEFORMATION

FRICTION FORCE

F_{s_0} = INITIAL FRICTION FORCE CORRESPONDING TO COEFFICIENT
OF FRICTION μ_0

F_{s_1} = FORCE CORRESPONDING TO μ_1

F_{s_2} = FORCE CORRESPONDING TO μ_2

F_{s_0}

F_{s_1}

F_{s_2}

S_1 S_2 TOTAL ACCUMULATED SLIP
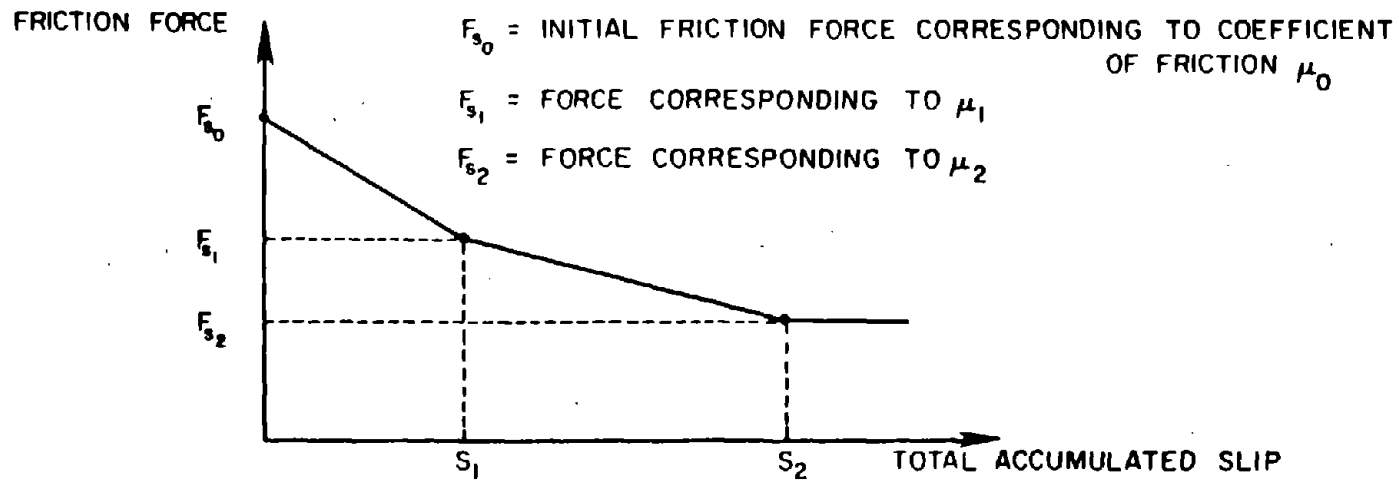
FIG. E11.5 DEGRADING STRENGTH CHARACTERISTIC

SHEAR

F_{s_0}

k_s

Δ SLIP

$=$

$F_{s_0}(1-\frac{k_h}{k_s})$

$k_s - k_h$

Δ

ELASTIC- PLASTIC COMPONENT

$+$

k_h

ELASTIC COMPONENT
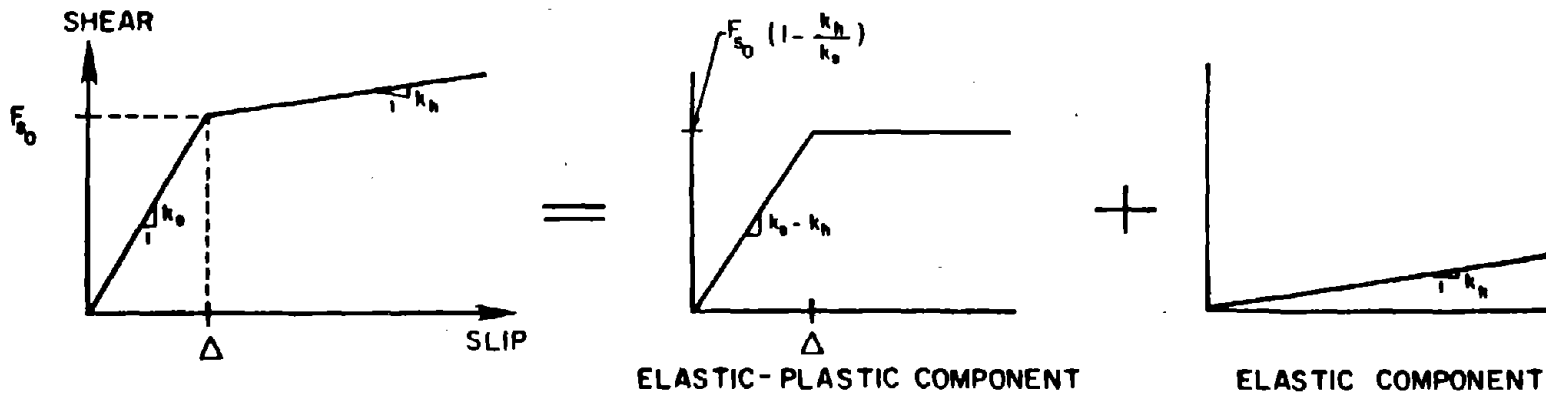
FIG. E11.6 DECOMPOSITION OF SHEAR – SLIP
RELATIONSHIP
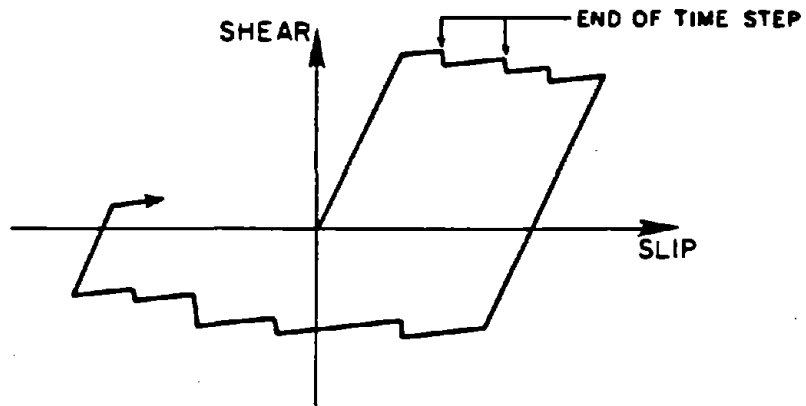
FIG. E11.7 IMPLIED SHEAR – SLIP RELATIONSHIP

DRAIN-2DX USER GUIDE

ELEMENT THEORY

SHEAR FRICTION JOINT ELEMENT (TYPE 12)

E12.1 GENERAL CHARACTERISTICS

E12.1.1 Basic Properties

Joints which are crossed by reinforcing bars typically exhibit "shear friction" behavior characterized by: (a) "pinched" hysteresis loops, with stiffening behavior near the end of each cycle and a high unloading stiffness, and (b) degradation of stiffness and strength as the number of load cycles increases. The shear friction element can be used to model joints with these characteristics.

Each element must be connected to four nodes, two on one panel and two on the adjacent panel, as shown in Fig. E12.1. The element shear deformation is the relative displacement between the connected panels, assumed to be the average displacement of nodes i, j minus the average displacement of nodes k, l.

The element consists of two components in parallel, with force-displacement relationships as shown in Fig. E12.2. Component 1 has an elastic-plastic relationship (Fig. E12.2a), to account for simple friction due to applied compressive load on the joint. Component 2 accounts for the "shear friction" mechanism, developing a "pinched" hysteresis loop (Fig. E12.2b) after several loading cycles. This loop begins with a low stiffness region (line 1), representing the state in which the reinforcement exerts little or no clamping action on the joint. As slip increases, clamping action develops and the stiffness increases (line 2). After further deformation, the reinforcement yields and the clamping action reaches a limiting value. The strength on this line may be constant or may be specified to increase with displacement (stiffness k_3). Component 2 unloads first along line 4, with a large stiffness, and then along line 5, with a small or zero stiffness. Lines 1 and 5 are both assumed to pass through the origin (no residual slip under zero shear). Under reversed loading the behavior

is similar.

E12.1.2 Variation of Friction Resistance

The strength of Component 1 (simple friction) is assumed to be constant, with no strength degradation due joint polishing and no allowance for variation of the bearing force. If joint polishing is important, it can be accounted for by specifying zero strength for Component 1, and placing the shear friction element in parallel with a friction element with degrading strength. Similarly, if variation of bearing force is important, the shear friction element may be placed in parallel with a combined gap-friction element.

E12.1.3 Stiffness Degradation

The stiffness and strength of reinforced joints both tend to degrade under cyclic loading. The element accounts for degradation effects in the hysteresis loop of Component 2 (shear friction component). The procedure allows for reduction in the stiffness of line 1 (k_1, Fig. E12.2b) and decrease in the maximum strength of the component (F_2, Fig. E12.2b). The procedure for stiffness degradation is as follows.

It is assumed that under cyclic loading with constant shear force the stiffness k_1 decreases, while the force F_1 (Fig. E12.2b) remains unchanged. That is, there is an increase in the displacement at which the stiffness changes from k_1 to k_2 (Δ_1, Fig. E12.2b). The change in Δ_1 is assumed to be based on the maximum deformation in the k_2 and k_3 regions (S_{2m}, Fig. E12.3, measured from the end of the current line 1). Degradation of stiffness k_1 is obtained by adding a proportion of S_{2m} to the deformation Δ_1, to obtain an increased Δ_1 value. The degradation is controlled by a factor α ($\alpha > 0$), such that $\Delta_1' = \Delta_1 + \alpha S_{2m}$. The new stiffness k_1 is then given by

$$k_1' = \frac{\Delta_1}{(\Delta_1 + \alpha S_{2m})} k_1 \tag{12.1}$$

When α is specified to be zero, no degradation occurs, and k_1 and Δ_1 remain constant.

The degradation process is implemented during load reversal. The increase in Δ_1 begins when the cycle moves from line 4 to line 5, and is completed when the cycle returns to the origin. In between these two points, Δ_1 in increased linearly. This gradual degradation is necessary because a complete loop is not necessarily obtained under arbitrary loading. If Δ_1 were increased immediately the cycle moved to line 5, incorrect results would be obtained for partial cycling.

The degradation of k_1 affects both directions because lines 1 and -1 are assumed to have the same slopes. Hence, deformation S_{2m} in the positive direction affects the negative part of the hysteresis loop, and vice versa. The stiffness k_1 is not allowed to become less than k_5. If k_1 reduces to k_5, then Δ_1 is increased as above, but F_1 is also increased, so that k_1 remains constant. The stiffnesses k_2, k_3, k_4, and k_5 all remain constant.

With this procedure, the force-displacement relationship stabilizes in the second cycle for cycling under constant load. This is because S_{2m} is measured from the current value of Δ_1, and hence $S_{2m} = 0$ for the second cycle. The loop will similarly stabilize in the second cycle for constant displacement cycling.

E12.1.4 Strength Degradation

A tri-linear "strength envelope", as shown in Fig. E12.4, is assumed to define the strength degradation. The strength, F_2, is decreased as Δ_1 increases, as shown. For any current value of k_1, the intersection of line 2 with the specified strength envelope is found. This defines a new displacement limit Δ_2, and a new strength F_2. With this procedure the hysteresis loop for constant displacement cycling stabilizes in the second cycle, but does not stabilize for constant load cycling. The multi-linear representation of the strength envelope permits modeling of either sudden loss or gradual degradation of strength.

E12.1.5 Examples

Figure E12.5a shows example properties for the simple friction and shear friction

components. Figure E12.5b shows the force-displacement relationship for the resulting element under four displacement-controlled cycles. The behavior is qualitatively similar to that observed in experiments. Figure E12.6b shows loops for similar cycling, but with the simple friction component specified to have zero strength, and a friction element with degrading strength placed in parallel with the shear friction component. The properties of the friction element with degrading strength are shown in Fig. E12.6a. In this case the loops show a progressive narrowing as the amount of cycling increases.

E12.2 STATIC ELASTO-PLASTIC STIFFNESS

The element has four displacement degrees of freedom, as shown in Fig. E12.1, and one deformation degree of freedom. The element deformation is given by

$$ q = \frac{1}{2} <1 \quad 1 \quad -1 \quad -1> \begin{Bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \end{Bmatrix} \tag{E12.2} $$

or

$$ q = \mathbf{a} \; \mathbf{r} \tag{E12.3} $$

where **a** is the displacement transformation. Hence, the element tangent stiffness matrix is

$$ \mathbf{K} = \mathbf{a}^T \mathbf{k} \mathbf{a} = \frac{k}{4} \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \tag{E12.4} $$

where k is the tangent stiffness at any time, equal to the combined stiffnesses of the simple friction and shear friction components.

E12.3 OTHER PROPERTIES

No provision is made for second order effects or element loads. Also, viscous damping is assumed to be zero, regardless of the value of βK (see discussion for the gap-friction element).

DRAIN-ANAL USER GUIDE

INPUT DATA SECTION C2.12

SHEAR FRICTION JOINT ELEMENT (TYPE 12)

See Fig. E12.1 through Fig. E12.6 for element behavior and properties.

C2.12(a). Control Information

One line.

Columns	Notes	Variable	Data
1-5(I)			No. of stiffness types (max. 40). See section C2.12(b).

C2.12(b). Stiffness Types

Two lines for each stiffness type.

Line 1

Columns	Notes	Variable	Data
1-5(I)			Stiffness type number, in sequence beginning with 1.
6-15(R)			Stiffness k_0.
16-25(R)			Stiffness k_1.
26-35(R)			Stiffness k_2.
36-45(R)			Stiffness k_3.
46-55(R)			Stiffness k_4.
56-65(R)			Stiffness k_5.

Line 2

Columns	Notes	Variable	Data
1-10(R)			Slip Force, F_f.
11-20(R)			Displacement limit u_1.
21-30(R)			Displacement limit u_2.
31-40(R)			Force limit P_1.
41-50(R)			Force limit P_2.
51-60(R)			Slip limit S_1.
61-70(R)			Slip limit S_2.

C2.12(c). Element Generation Commands

One line for each generation command. The first element can be assigned any number. Subsequent elements must be defined in numerical sequence. Lines for the first and last elements must be included.

Columns	Notes	Variable	Data
1-5(I)	C5		Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
10(I)			Horizontal or vertical joint code. (a) 0: Horizontal joint. (b) 1: Vertical joint.
11-15(I)			Node number at element point I.
16-20(I)			Node number at element point J.
21-25(I)			Node number at element point K.
26-30(I)			Node number at element point L.
31-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Stiffness type number.
41-50(R)			Degradation factor ($\alpha \geq 0.$).

DRAIN-POST USER GUIDE

OUTPUT ITEMS FOR POSTPROCESSING

SHEAR FRICTION JOINT ELEMENT (TYPE 12)

Item	Description
1	Node number at point I.
2	Node number at point J.
3	Node number at point K.
4	Node number at point L.
5	Slip code (1: yes; 0: no)
6	Line number.
7	Elastic-perfectly plastic shear force component.
8	Friction shear force.
9	Total shear force.
10	Shear deformation.
11	Accumulated positive slip.
12	Accumulated negative slip.
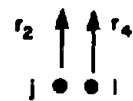
(a) HORIZONTAL JOINT   (b) VERTICAL JOINT

FIG. E12.1 DEGREES OF FREEDOM FOR SHEAR FRICTION
ELEMENT

(a) SIMPLE FRICTION
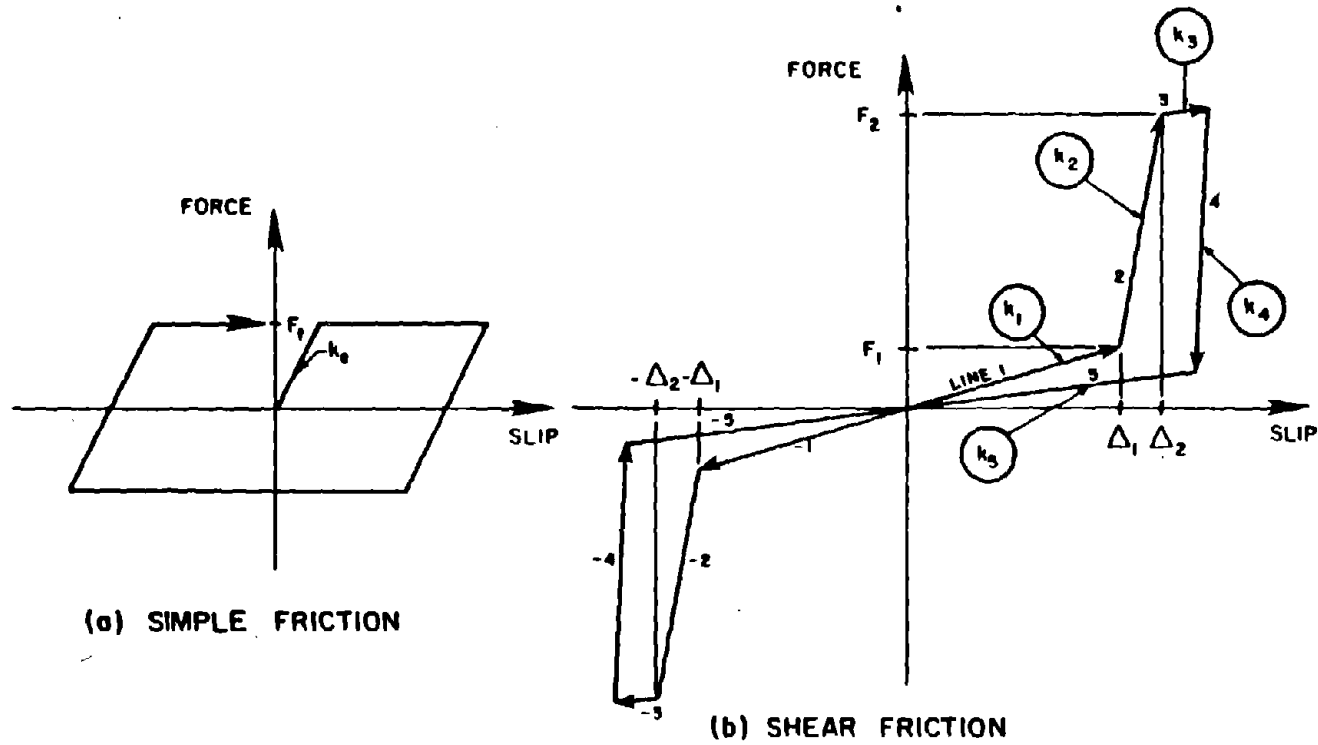
(b) SHEAR FRICTION

FIG. E12.2 SHEAR – SLIP RELATIONSHIPS FOR SHEAR FRICTION ELEMENT
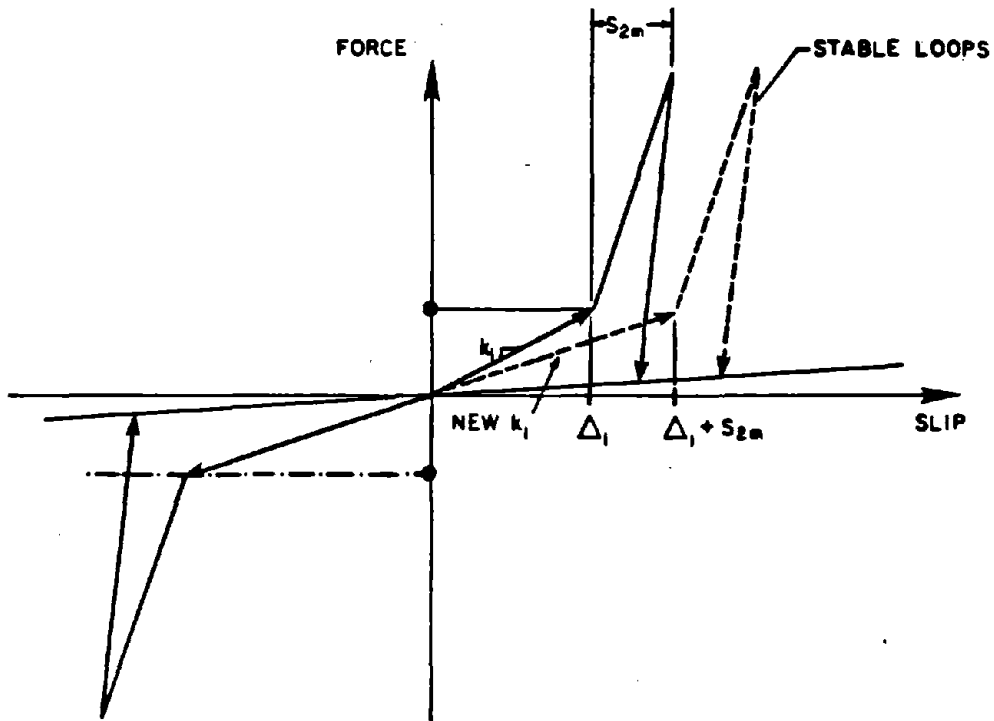
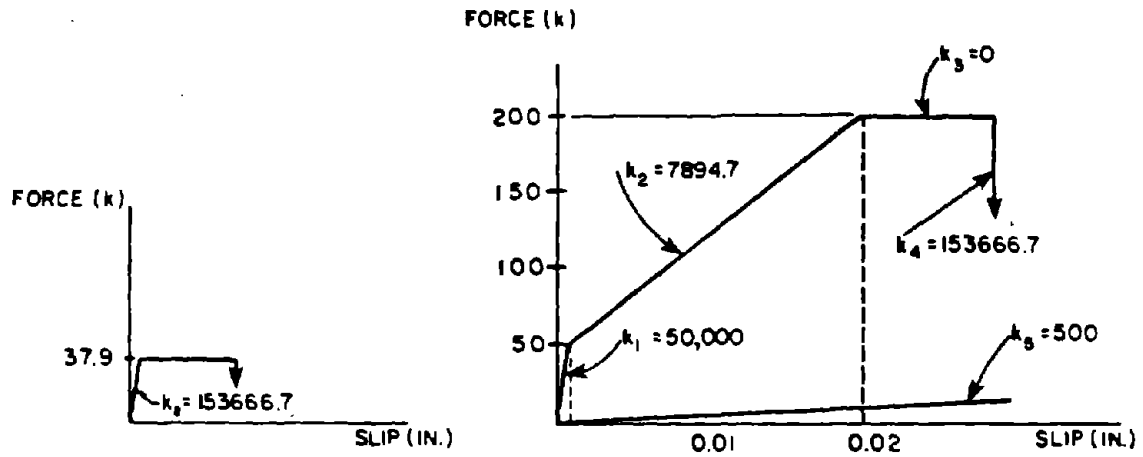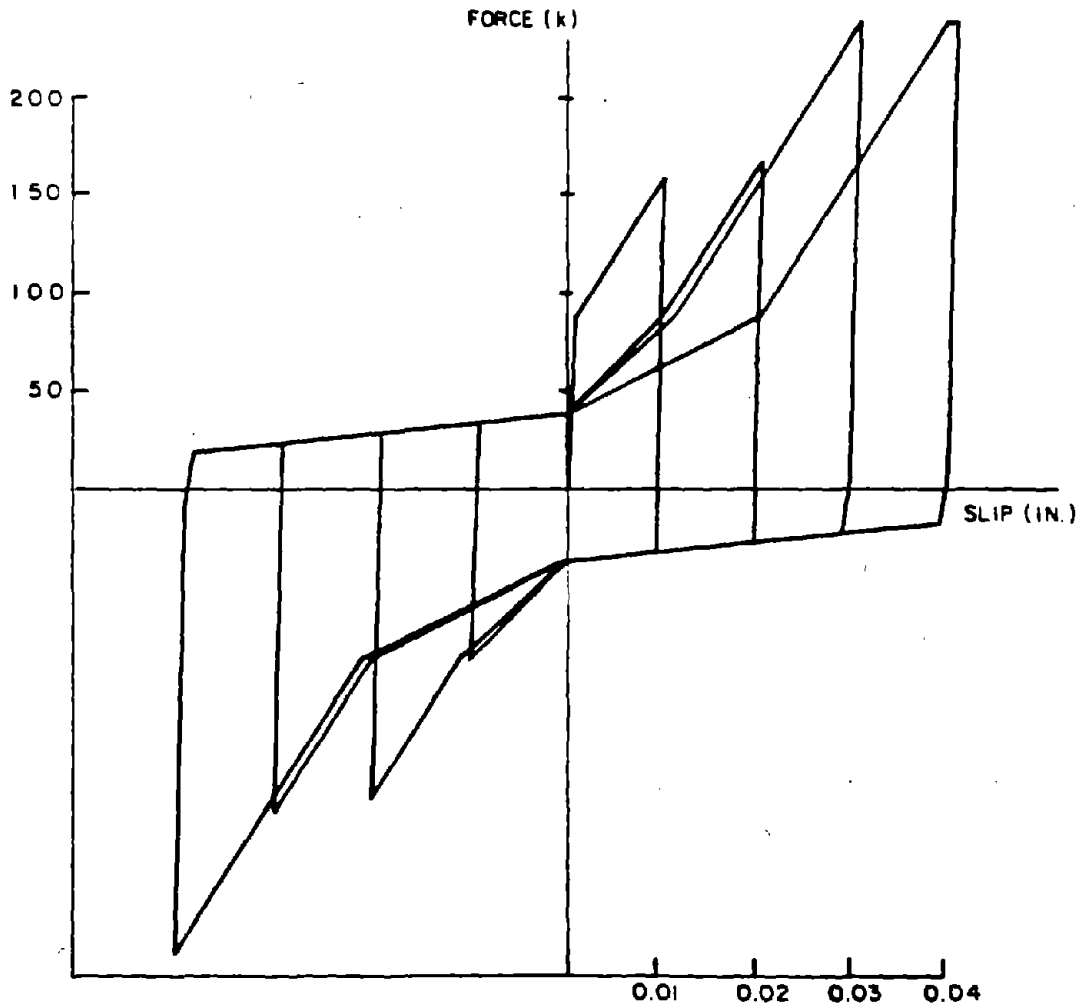FIG. E12.3 STABILIZED HYSTERESIS LOOPS AFTER STIFFNESS DEGRADATION
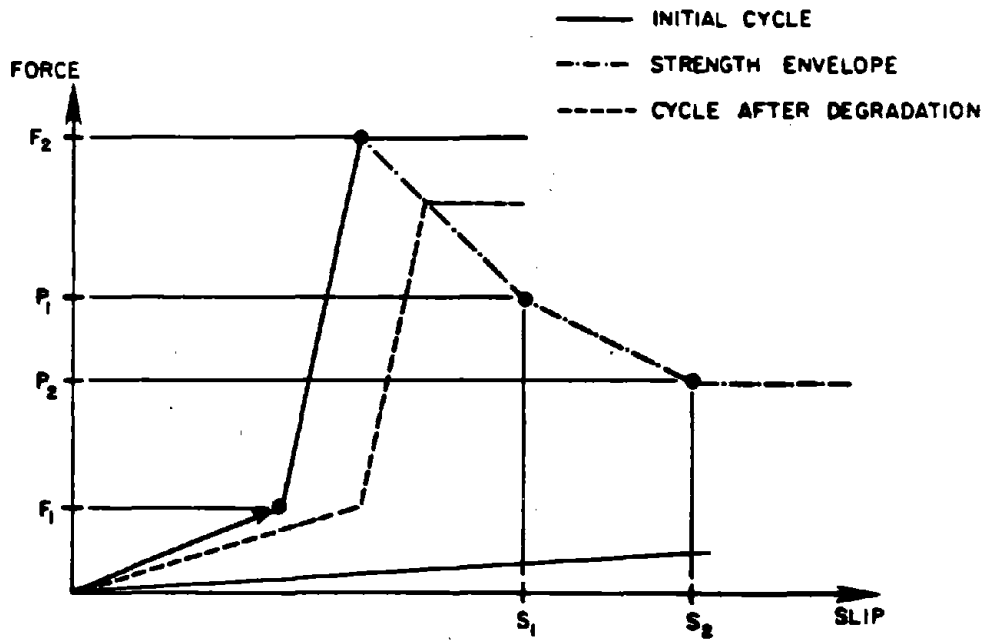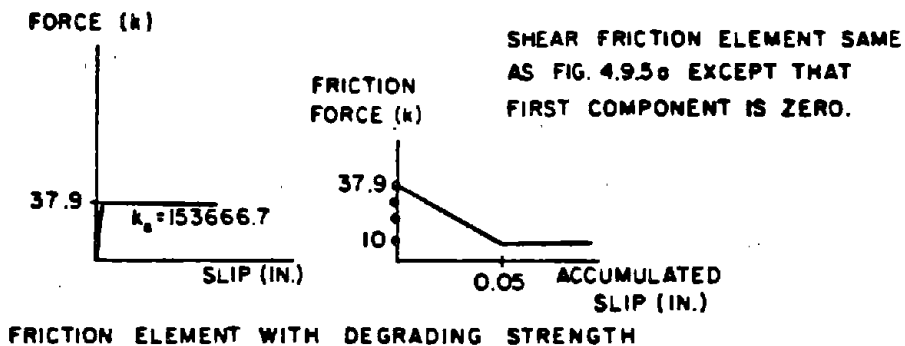

FIG. E12.4 STRENGTH ENVELOPE

FORCE (k)

(a) PROPERTIES OF TWO COMPONENTS

(b) COMPLETE ELEMENT

FIG. E12.5 CYCLIC BEHAVIOR OF SHEAR FRICTION ELEMENT

FORCE (k)

37.9 $k_s = 153666.7$

SLIP (IN.)

FRICTION ELEMENT WITH DEGRADING STRENGTH

FRICTION FORCE (k)

SHEAR FRICTION ELEMENT SAME AS FIG. 4.9.5a EXCEPT THAT FIRST COMPONENT IS ZERO.

37.9

10

0.05 **ACCUMULATED SLIP (IN.)**

(a) PROPERTIES OF TWO ELEMENTS IN PARALLEL

FORCE (k)

200
150
100
50

SLIP (IN.)

0.01 **0.02** **0.03** **0.04**
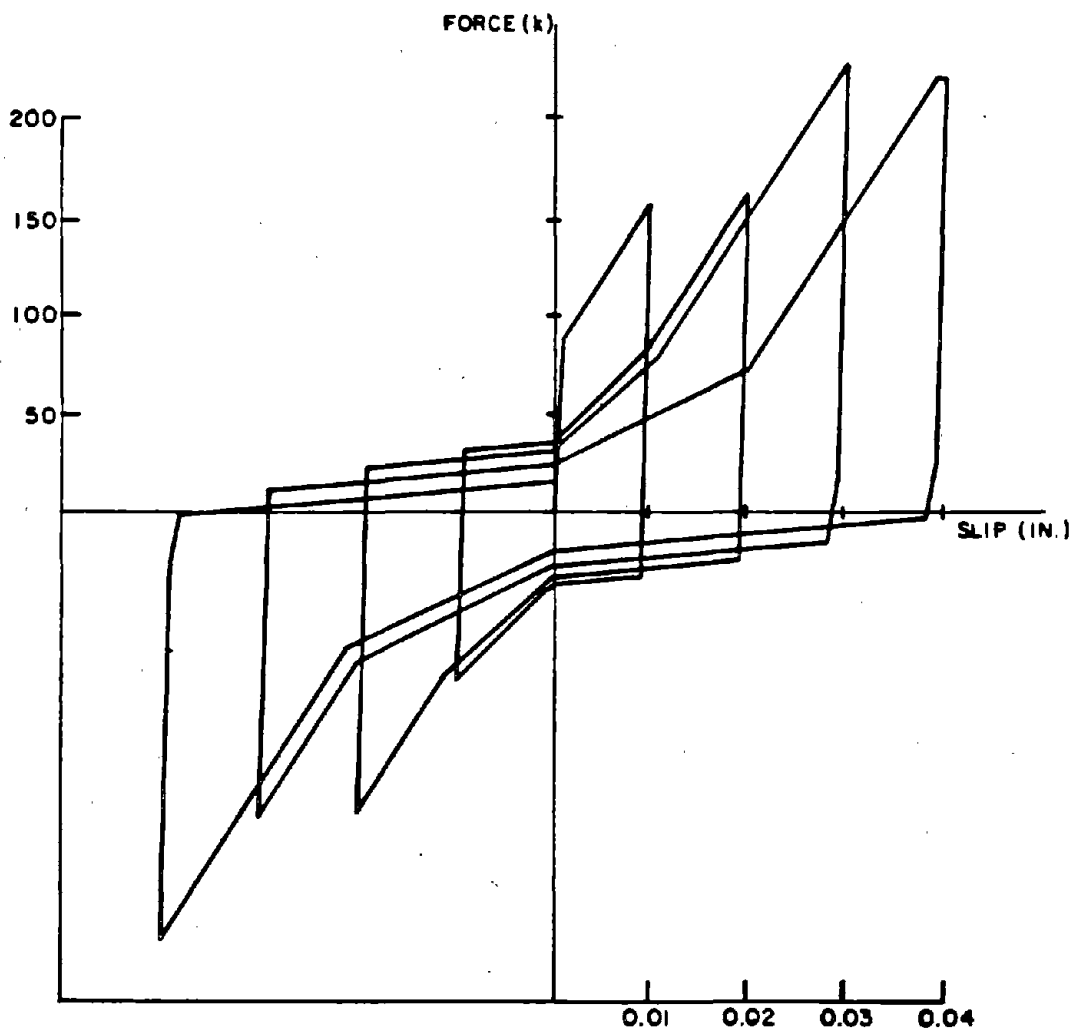
(b) COMPLETE ELEMENT

FIG. E12.6 SHEAR FRICTION BEHAVIOR WITH PINCHING EFFECT

SECTION P

DRAIN-POST INPUT DATA

DRAIN-2DX USER GUIDE

SECTION P

DRAIN-POST INPUT DATA

Table of Contents

67

1. GENERAL FEATURES

1.1. PURPOSE

DRAIN-ANAL produces a separate post-processing file for each analysis segment in which the post-processing file code is set to 1. Each file name has the form YYYYYY.RZZZ, where YYYYYY is the problem name, R indicates a post-processing results file, and ZZZ is the analysis segment number. Each file contains one results set for each load or time step in the analysis segment. A results set contains the displacements for all nodes, the output results for all elements, and the forces for all structure sections. DRAIN-POST allows the DRAIN user to select results files, extract results from them, and organize those results in tables. Up to 8 files can be combined (concatenated) if desired, before results are extracted. DRAIN-POST does not currently include any capabilities for plotting results graphs, although it is expected that this capability will be added in the future. In the meantime, users must write the results to an output file (using the PRINT options), and plot from this file using their own plot programs. Also, DRAIN-POST can not currently be used to draw deflected shapes. Data from which deflected shapes can be plotted is included in the post-processing files, and it is expected that a shape drawing capability will be added in the future.

1.2. OPERATIONS

1.2.1 General

DRAIN-POST can be used in either TABLE or PRINT mode. In TABLE mode, tables of node, element and section results can be extracted from the post-processing files, and operations on the tables can be performed. In PRINT mode, results can be written to a formatted output file.

1.2.2 TABLE Mode

If results histories are to be tabulated and plotted, DRAIN-POST must first be used to extract a table of results from one or more post-processing files. Column 0 of the table is always time (for dynamic analysis) or load factor (for static analysis), and is automatically set up for each new table. The other columns of the table can contain any desired result quantities. Once it has been set up, a

table can be processed in a number of ways.

1.2.3 PRINT Mode

In PRINT mode, results for selected elements, nodes and/or sections can be extracted from one or more post-processing files, and written to an output file. The current table can also be output.

1.2.4 DPOST.LOG

A log of each post processing session is written to a file named DPOST.LOG, to allow review of the session if desired.

2. DRAIN-POST COMMANDS

DRAIN-POST first requests the problem name and the analysis segment numbers (maximum 8). The corresponding post-processing files are concatenated, and DRAIN-POST commands can then be given. The commands are as follows.

TABLE: Produce a new result history table. This must be done before any MAXVAL, DISPLAY, SCALE or SHIFT commands are given. See Section 3.

MAXVAL: Determine maximum and minimum values, with the times or load factors at which they occurred, for all or selected columns in the current table. Display and also write to the DPOST.LOG file. See Section 4.

DISPLAY: Display all or part of the current table. The displayed data is **not** written to the DPOST.LOG file. See Section 4.

SCALE: Scale (multiply by a factor) selected rows of any column in the current table. See Section 4.

SHIFT: Shift (add a constant to) selected rows of any column in the current table. See Section 4.

PRINT: Write specified result histories on a specified output file (enter PRINT mode). See Section 5.

QUIT: Exit from DRAIN-POST.

3. TABLE COMMAND

3.1. CREATION OF A TABLE

3.1.1. Data Types

A new table is begun by selecting the TABLE command. Each column of the table contains the response history for a particular **data item**. The DRAIN-POST user specifies the data item for each column by defining its results type (node, element, section or combined), followed by other data as follows.

3.1.2. Node Data

For node data, the node number, result type, and displacement direction must be specified. The node number is that assigned in Section B of the DRAIN-ANAL input. The result type can be displacement, velocity or acceleration. The direction can be global X, global Y, rotation, or any inclined direction. If an inclined direction is required, the user must specify its projections on the global X,Y axes.

3.1.3. Element Data

For element data, the element group, element number and "output item" number must be specified. The group and element numbers are those assigned in Section C of the DRAIN-ANAL input. The output items depend on the element type. Refer to the element user guides for the output items for each element type.

3.1.4. Structure Section Data

For a structure section, the section number, result type, and force type must be specified. The section number is that assigned in Section E of the DRAIN-ANAL input. The result type is either elastic force or damping force. The force type is normal force, shear force, or moment (at the center of the structure section).

3.1.5. Column Sum

A new column can be defined by adding together two or more previously defined columns. A possible application is when a plane surface is modeled using several gap-friction elements. The total impact force on the section at any time is the sum of the forces for the separate elements. This total force can be obtained by setting up a column for each element and then combining the columns. A second application could be to obtain story shear by summing the shear forces for several columns (this could also be done by defining a structure section).

3.1.6.. Column Difference

A new column can be defined by subtracting one column from another. A possible application is to get story drift by subtracting the horizontal displacement at the bottom of the story from that at the top.

4. OPERATIONS ON A TABLE

4.1. MAXVAL

The MAXVAL command can be used to get the maximum and minimum values in any column or columns. A list of column numbers is requested, to identify the columns for which maximum values are required (default is all columns). The maximum and minimum values with corresponding times or load factors are displayed, and written to the DPOST.LOG file.

4.2. DISPLAY

The display command can be used to scan a table, by displaying all or part of it on the terminal. The time/ load column plus up to five other columns can be displayed at a time. All rows may be displayed, or the rows for a specified range of times or load factors. The display is not written to the DPOST.LOG file. DRAIN-POST requests the column numbers, the starting time/ load factor (default = first time/ load factor), and the ending time/ load factor (default = last time/ load factor).

4.3. SCALE

The SCALE command can be used to multiply all or part of any column by a scale factor. DRAIN-POST requests the column number, the scale factor, and the beginning and ending rows in the column (default = entire column). Scaling may be necessary before a plot is produced.

4.4. SHIFT

The SHIFT command can be used to add a constant to all or part of any column. DRAIN-POST requests the column number, the value of the constant, and the beginning and ending rows in the column (default = entire column).

One case where a shift may be needed is when two or more results files are combined, since the load factor at the beginning of each analysis segment is always zero. The shift command can be used to add on the initial load factors, to get the correct total values.

5. PRINT COMMAND

5.1. GENERAL

The PRINT command causes the program to enter the PRINT mode. The PRINT mode has its own series of commands, as follows.

TABLE: Print the current table (if one exists).

ELEM: Print element results. For the element items which are printed for each element type, refer to the element user guides.

NODE: Print nodal results (displacement, velocity or acceleration).

SECT: Print structure section results (normal force, shear force and moment at center of section).

QUIT: Exit from PRINT mode.

For the ELEM, NODE and SECT commands, results are printed for all the times or load factors in the currently selected post-processing files. It is not necessary for a current table to exist.

5.2. REQUIRED DATA

On entering the PRINT mode, a file name (1-6 characters) for the printed output is requested. The default is the previous file name for this session. New results are always appended to the file.

Additional data is then required as follows:

ELEM a) Element group number.

b) Element numbers in group (maximum of 10 elements if listed, all elements if defaulted).

NODE a) Node numbers (maximum of 10 nodes if listed, all nodes if defaulted).

b) Result type: d (displacement), v (velocity), or a (acceleration).

c) Result direction: x (global X), y (global Y), or r (rotation).

SECT a) Structure section numbers (maximum of 10 if listed, all sections if defaulted).
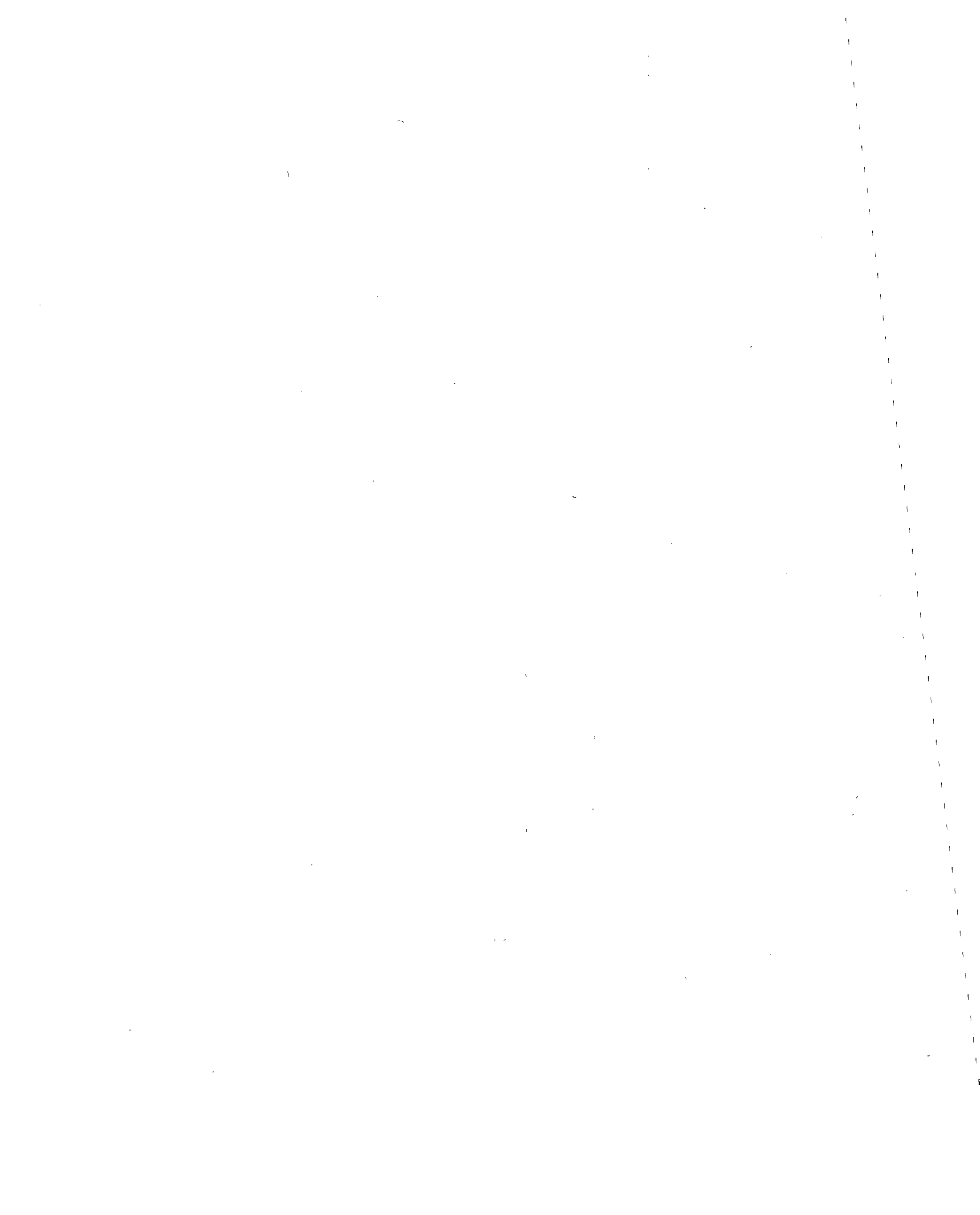
SECTION X

ADDITION OF NEW ELEMENTS

DRAIN-2DX USER GUIDE

SECTION X

ADDITION OF NEW ELEMENTS

Table of Contents

1. INTRODUCTION

DRAIN-2DX consists of two programs. The first is the analysis program, DRAIN-ANAL, which analyses a structure and produces certain permanent files. The second is the post processing program, DRAIN-POST, which uses the files produced by DRAIN-ANAL to produce results in tabular format. These tables can then be used to plot results.

DRAIN-ANAL consists of a series of "base" subroutines which perform the following tasks.

1) Read and print the structure and loading data.

2) Carry out a variety of bookkeeping operations

3) Assemble the structure stiffness and loading.

4) Determine the response of the structure.

5) Print results and produce files that can be processed by the post-processor.

The base subroutines are combined with element subroutines of a variety of types to produce the complete program. To add new elements, it is not necessary to make major program changes, but only to develop a small number of new subroutines, with no modification to the base program.

The post processing program is executed interactively to produce tables and graphs from the results files. Each time a new element is added to DRAIN-ANAL, additions must also be made to DRAIN-POST.

A new element type can be added by developing ten new subroutines for DRAIN-ANAL and two new subroutines for DRAIN-POST. The procedures which must be observed are described in this section.

For DRAIN-ANAL, the subroutines must be named INELXX, ELODXX, GLODXX, STIFXX, FACTXX, RESPXX, ENPRXX, THPRXX, FLINXX, and PLINXX, where the "XX" at the end of the subroutine name must be a two digit number which identifies the element type. For example, for element type 2 the subroutines must be INEL02, STIF02, etc. The subroutines may call other subprograms as needed. The names of such subprograms should end with the element type number. For example, a recommended procedure is for RESPXX to call subroutines ELASXX, DAMPXX, and

SAVEXX for particular phases of the element response calculation. Similarly, for DRAIN-POST, the two subroutines must be named RHHDXX and RHPRX

The purposes of the twelve subroutines for any element are as follows:

(1) INELXX: Input and initialize element data.

(2) ELODXX: Input element load pattern data.

(3) GLODXX: Set up gravity load vectors and initialize element forces, using element load patterns.

(4) STIFXX: Calculate element stiffness. This is the "linearization" phase.

(5) FACTXX: Calculate element event factor for event-to-event analysis.

(6) RESPXX: Calculate element deformations, actions and response quantities. This is the "state determination" phase.

(7) ENPRXX: Print element envelope results.

(8) THPRXX: Print element current state.

(9) FLINXX: Calculate element actions and deformations for response spectrum analysis.

(10) PLINXX: Print element actions and deformations for response spectrum analysis.

(11) RHHDXX: Print element results heading in DRAIN-POST.

(12) RHPRXX: Print element results in DRAIN-POST.

The specific computations to be performed in each subroutine are described in later sections. First, however, it is necessary to describe certain variables which are used to control the computation and to transfer data to and from the subroutines. These variables are described in the following sections.

2. KEY VARIABLES

2.1. General

A number of variables are used to control the computation. These can be grouped into three

categories, as follows:

(1)   Overall structure variables.

(2)   Element group variables.

(3)   Individual element variables.

The variables are defined in this section. Specific procedures for using them are described later.

2.2. Overall Structure Variables

Structure variables are defined for the structure as a whole, and have the same value for all element groups. The values of these variables are set by the base program.

ID: Two-dimensional array of size NNODS,3. ID(I,1), ID(I,2), ID(I,3) contain the structure degree of freedom (equation) numbers corresponding to the X-displacement, Y-displacement and rotation, respectively, at node I.

KDATA: Indicator for input data error.

(a) KDATA = -2: input data error, execution is suppressed.

(b) KDATA = -1: execute only if single element block.

(c) KDATA = 0: execute.

(d) KDATA = 1: input data checking only. No execution.

KENR: Indicator for calculating element energies.

(a) KENR = 0: no.

(b) KENR = 1: elastic-plastic only (static).

(c) KENR = 2: elastic-plastic and damping (dynamic).

KRESIS: Indicator for calculating element resisting forces.

(a) KRESIS = 1: elastic-plastic only (static).

(b) KRESIS = 2: elastic-plastic and damping (dynamic).

KSAVE: Indicator for saving and/or printing of element results in current step.

(a) KSAVE = 1: yes.

(b) KSAVE = 0: no.

KSTEP: Current step number in step by step analysis (static or dynamic).

KSTT: Indicator for forming total stiffness or change in stiffness.

    (a) KSTT = 1: Form total stiffness.

    (b) KSTT = 0: Form change in stiffness.

KTYPE: Indicator for type of element stiffness.

    (a) KTYPE = 1: elastic-plastic stiffness only.

    (b) KTYPE = 0: elastic-plastic plus geometric stiffness.

    (c) KTYPE = -1: geometric stiffness only.

NNODS: Number of nodes in structure.

PROBID: Six character problem name, specified by the analyst. All files created by the program have file names of the form YYYYYY.ZZZZ where YYYYYY is the problem name and ZZZZ is a four character extension.

TIM: Time at beginning of current step for dynamic analysis.

X,Y: One-dimensional arrays of dimension NNODS. X(I), Y(I) contain the X,Y coordinates, respectively, of node I.

2.3. Element Group Variables

Group variables are defined for each element group, and have the same value for all elements in the group. Except as noted, the values of these variables are set by the base program.

BETO: Initial stiffness damping factor. Each element has a damping matrix equal to BETO*K0 , where K0 = initial elastic stiffness of element.

EFLIN: One dimensional array of size NFLIN. Element results for response spectrum analysis. Set in subroutine FLINX

ELFACT: Two dimensional array of size NLOD,NMEM, set up for each new element load pattern

that is defined. Each column correspond to an element in the current group, and each row to an element load set (see array SETLOD). Term I,J stores the scale factor by which load set I is to be multiplied for element J, to define the load pattern. Set in subroutine ELODX

IELNOD: Two dimensional array of size NMEM,NELNOD. Row I contains the node numbers for element I in current group. Set in subroutine INELX

IGRC: Number of current element group.

KEVE: Event calculation code.

(a) KEVE = 1: Event factors are to be calculated for elements in current group.

(b) KEVE = 0: Event factors are not required.

KGEM: Second order analysis code.

(a) KGEM = 0: Ignore second order calculations.

(b) KGEM = 1: Perform second order calculations, but keep geometric stiffness constant during dynamic analysis.

(c) KGEM = 2: Perform second order calculations and allow geometric stiffness to change during dynamic analysis.

NDOF: Number of element degrees of freedom (rows in element stiffness matrix) for elements in current group. Set in subroutine INELX

NELNOD: Number of element nodes for elements in current group. Set in subroutine INELX

NFLIN: Number of element results for response spectrum analysis. Set in subroutine INELX

NINFG: Number of words in group information block (/ INFGR/) for current element group. Set in subroutine INELX

NINFE: Number of words in element information block (/ INFEL/) for each element in current group. Set in subroutine INELX

NINFL: Number of words required to store one element load set for elements in current group. Set in subroutine INELX

NINFT: Number of words in element output block (/ THELM/) for each element in current group. Set in subroutine INELX

NLOD: Number of element load sets for current group and current element load pattern.

NMEM: Number of elements in current group.

SETLOD Two dimensional array of size NINFL,NLOD. Each column stores one element load set. Set in subroutine ELODX

2.4. Individual Element Variables

Element variables are defined for each element individually and can have different values for each element. Except as noted, the values of these variables are set by the base program.

DDISE: One-dimensional array of size NDOF. Increments of nodal displacement (element degrees of freedom for current element in current analysis step). Used in subroutine FACTXX to calculate element event factor and in subroutine RESPXX to update element state.

DISE: One-dimensional array of size NDOF. Total nodal displacements for current element. Used in subroutine RESPXX for calculations affected by second order (P-Δ) effects.

ENED: Change of element damping energy for current displacement increment. Set in subroutine RESPXX if KENR > 1.

ENER: Change of element elastic-plastic energy for current displacement increment. Set in subroutine RESPXX if KENR > 0.

ENSO: Change of element "second order" energy for current displacement increment. Set in subroutine RESPXX if KENR > 0 and KTYPE ≤ 0.

FK: Two-dimensional array of size NDOF*NDOF. Element stiffness matrix (depends on the structure variables KSTT and KTYPE). Set in subroutine STIFX

IMEM: Number of current element within its group. Set in subroutine INELX

KST: Stiffness change code. Set in subroutine RESPXX as follows:

(a) KST = 1: Stiffness for current element has changed in the current step. Subroutine STIFXX will be called to update stiffness for this element.

(b) KST = 0: No change has occurred. Subroutine STIFXX will not be called.

LM: One dimensional array of size NDOF. LM(I) contains the structure degree of freedom number corresponding to element degree of freedom I. Set in subroutine INELX

RDAMP: One-dimensional array of size NDOF. Element resisting force vector due to damping forces. Set in subroutine RESPX

RELAS: One-dimensional array of size NDOF. Element resisting force vector due to elastic-plastic (static) forces. Set in subroutine RESPX

EFAC: Element event factor, which is the proportion of the current displacement increment required to reach the next significant nonlinear event for the current element. If EFAC ≥ 1.0, no event occurs within the current analysis step. If EFAC < 1.0, an event occurs and the base program must subdivide the step. EFAC is calculated only for those groups of elements for which KEVE = 1. Set in subroutine FACTX

VELE: One-dimensional array of size NDOF. Nodal velocities for current element at end of current step in dynamic analysis. Used in subroutine RESPXX to calculate element damping forces.

3. LABELED COMMON BLOCKS

3.1. General

Data needs to be stored for each element, to monitor its nonlinear behavior. This data, plus some of the key variables, are transferred to and from the element subroutines by means of the labeled common blocks /INFGR/ and /INFEL/. There is one

/INFGR/ data block for each element group (containing data common to all elements in the group) plus one /INFEL/ data block for each element (containing data unique to each element). The length of /INFGR/ may be zero. In addition, element results are transferred to results files by means of the labeled common block /THELM/. The values in /THELM/ block must be set each time the results

for an element are saved.

3.2. Group Information Block

The /INFGR/ block contains information common to all elements in the current group. The values in /INFGR/ must be initialized in subroutine INELXX, and once initialized must not be modified. The base program stores the block and makes it available for the current element when needed. The length of /INFGR/ is NINFG 4-byte words. The base program initializes NINFG to zero before calling subroutine INELX If NINFG is not redefined in subroutine INELXX, the base program assumes that there is no /INFGR/ block for the element group. The block may be divided in any way the programmer wishes, up to a maximum of 1000 4-byte words.

3.3. Element Information Block

The /INFEL/ block contains information unique to each element. It must be initialized in sub-routine INELXX, and then progressively updated in RESPXX and (if necessary) in STIFX All of the information to be retained for any element must be contained either in this block or in the /INFGR/ block. The length of /INFEL/ is NINFE 4-byte words, which must be defined in INELXX and is the same for all elements in any group. The block may be divided in any way the programmer wishes, subject to the following restrictions:

(1)   The maximum length currently permitted is 1000 4-byte words.

(2)   The first two variables must be IMEM and LM, respectively.

The block must be initialized for each element in turn in subroutine INELX It is then transferred, by the base program, to a storage area. Each time computations for an element are to be carried out (by GLODXX, STIFXX, FACTXX, RESPXX or ENPRXX), the base program first transfers the block for the current element from the storage area to the /INFEL/ block, then calls the subroutine. When control is returned from RESPXX or STIFXX, the base program transfers the updated information block back to the storage area. The block is **not** transferred back for FACTXX, ENPRX or GLODXX. Hence, any changes made to an /INFEL/ block in these subroutines will **not** be saved.

During the execution of INELXX, first the /INFGR/ block for the element group (if it exists) and then the /INFEL/ blocks for all elements are written to disk. At the end of the input phase, the blocks are brought back into memory and stored compactly in the available memory space. If memory becomes filled, the elements stored constitute an "element block". This block is written to disk and a new block is begun. During subsequent calculation the element blocks are returned to memory as needed and if updated are rewritten to disk. The blocking and I/O operations are all carried out by the base program. If the information for all elements can be held in memory (i.e. only one element block), then there is no I/O of element blocks to disk storage, and considerable execution time may be saved. In order to reduce I/O cost, the analyst may specify that execution of an analysis should not begin unless there is only one element block. This type of blocking will normally be used only on fixed-memory computers. On virtual memory computers it is probably more efficient to request sufficient memory to execute with a single element block.

3.4. Element Results Block

For each element, the results at the current load or time step can be saved for subsequent post processing and/or printing as the analysis proceeds. The data is transferred by means of the labeled common block /THELM/. The length of /THELM/ is NINFT 4-byte words. NINFT must be set in subroutine INELXX, and is the same for all elements in a group. The maximum length currently permitted is 40 4-byte words.

If results for any element are to be saved and/or printed (i.e. KSAVE = 1), these results (consisting of anything that the programmer wishes) must be placed in /THELM/ during execution of subroutine RESPX The base program keeps track of whether element results are to be saved and/or printed. For saving, the base program buffers the /THELM/ blocks in an in-core array, and when the buffer is full writes it to a post processing file. The name of this file is YYYYYY.RZZZ, where YYYYYY is the problem name and ZZZ is the analysis segment number. For printing, the base program buffers the /THELM/ blocks in a second in-core array, and at the end of step writes the buffer to a scratch file. At the end of each analysis segment the results are recalled, rearranged and printed using subroutine THPRX It is recommended that double precision variables be converted to single

precision before placing the results in /THELM/, in order to reduce buffer lengths and disk storage.

3.5. Work Block

Labeled common block /WORK/ provides a work area for use by the programmer. The block currently has a maximum length of 4000 4-byte words, and may be subdivided in any way. It provides a means of grouping local variables, and with some compilers may save storage.

The /WORK/ block should generally not be used to transfer data between subroutines, because it can be used differently for different elements, and the data may therefore be continually changing.

3.6. Tapes Block and File Names

Labeled common block /TAPES/ contains the I/O unit numbers. The variables in this block must not be modified. Unit numbers 1 through 4 and 7 through 14 are all reserved for use by the base program. Units INP (input unit number, = 5) and IOU (output unit number, = 6) may be used in element subroutines for read and write statements. The input file name must be DRAIN.INP.

For a problem with name YYYYYY, a number of permanent files are created with names of the form YYYYYY.ZZZZ, where ZZZZ is a four character extension. All such file names are reserved. The extensions have the following meanings:

ACCN: A binary file containing ground acceleration records.

DISP: A binary file containing ground displacement records.

ECHO: A text file containing an echo of the input data for the last analysis session. If an echo file exists at the beginning of any analysis session, execution will not proceed. This is to avoid inadvertent over-writing.

ELOD: A binary file containing element load patterns.

FORC: A binary file containing dynamic force records.

GEOM: A binary file containing the X,Y coordinates of all nodes, the node numbers for all elements, and other data. This file is used by DRAIN-POST.

LIST: A **text** file describing all analysis segments which have been run, and all structure states which have been saved, since the current problem was begun.

MODE: A **binary** file containing mode shapes and frequencies.

OUTP: A **text** file containing the results printout for the last analysis session. As with ECHO, an OUTP file must not exist at the beginning of any analysis session. The OUTP file contains an event log, unbalanced load information, and optional node, element, etc. results.

RXXX: A **binary** post-processing file containing results for analysis segment XX These files are used for post processing of results by DRAIN-POST. Note that a separate file is produced for each analysis segment.

SLOG: A **text** file containing a log of the last session. This log consists of energy balance and unbalanced load information. As with ECHO, an SLOG file must not exist at the beginning of any analysis session.

SPEC: A **binary** file containing earthquake response spectra.

STAT: A **binary** file containing static nodal load patterns.

SXXX: A **binary** file containing the structure state at the end of analysis segment XX These files are used for restarting.

VELO: A **binary** file containing initial velocity patterns.

For debugging or special purpose postprocessing, a programmer may create additional files. The reserved unit numbers and file names must not be used.

4. ENVELOPE VALUES

Maximum and minimum envelope values for various element response quantities may be stored in the /INFEL/ blocks, and progressively updated. These quantities can then be printed at specified intervals (ENPRXX subroutine). Although it is acceptable to store only the envelope value, it will be common to store also the load or time step number at which the value was reached. These step

numbers begin with zero in each analysis segment (a zero step number thus indicates that the envelope value was reached in some earlier segment). Hence, at the beginning of each new segment, the numbers must be re-initialized to zero. See subroutine RESPXX for the procedure.

5. ARGUMENT TYPES IN ARGUMENT LISTS

Data which is not transferred through the common blocks is transferred through argument lists. The variables transferred through any argument list are of three types, as follows:

(a) Input: These are set by the calling subprogram and must not be modified in the called subprogram.

(b) Output: These must be set in the called subprogram and returned for use by the calling subprogram. They are not set before entry to the called subprogram.

(c) Modify: These are set by the calling subprogram, and may either be left unchanged or modified in the called subprogram, depending on circumstances.

6. SUBROUTINE INELXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine INELXX are as follows:

    SUBROUTINE INELXX (NMEM, NNODS, NDOF, NINFG, NINFE, NINFL, NINFT,
    NFLIN, NELNOD, IELNOD,ID, X, Y, KDATA)

    COMMON /INFEL/ IMEM, LM (. . .

    COMMON /INFGR/ . . .

    COMMON /TAPES/ INP, IOU

    COMMON /WORK/ . . .

    DIMENSION ID(NNODS,3), X(NNODS), Y(NNODS), IELNOD(NMEM,1)

Argument Types

The arguments are of the following types:

(a) Input: NMEM, NNODS, ID, X, Y

(b) Output: NDOF, NINFE, NINFL, NINFT, NFLIN, NELNOD, IELNOD

(c) Modify: NINFG, KDATA

Purpose

INELXX is called once for each element group, in the input phase. Its purpose is to read the input data for all elements in the group, to return values of several control variables, and to initialize the /INFGR/ and /INFEL/ blocks.

Tasks Performed Before Entry

The base program reads the first card of the input data for the element group, and prints the group title. This first card contains the following data:

(a) Element type number.

(b) Number of elements in the group (NMEM).

(c) Event calculation code (KEVE).

(d) Second order analysis code (KGEM).

(e) Initial stiffness damping factor (BETO).

(f) Group title (optional).

The base program initializes NINFG to zero, and then calls INELXX for the corresponding element type.

Tasks to be Performed in INELXX

Subroutine INELXX must read and echo-print the input data for elements in the group, and initialize the /INFGR/ and /INFEL/ blocks. The following tasks must be performed:

(1) Print appropriate headings for the element type.

(2) Set the values of NDOF, NINFG, NINFE, NINFL, NINFT, NFLIN, and NELNOD. The value of NINFG may be left as zero (i.e. no /INFGR/ block).

(3) Set the array IELNOD. Row I in IELNOD stores the node numbers for element I in the group.

(4) Store information common to all elements in the /INFGR/ block. This block must be completed before the call to subroutine FINISH (see below) for the first element in the group. This is because the /INFGR/ block is placed in storage immediately preceding the /INFEL/ block for the first element.

(5) If desired, establish tables of stiffnesses, strengths, etc. for subsequent use in specifying individual elements. The arrays for these tables may be defined within the /WORK/ block, to avoid requesting new storage.

(6) Read the data for each element in the group, including node numbers, stiffnesses, strengths, etc., typically using the tables established in Step 5. Generation options may be included, provided the elements are generated in element number sequence and calls to subroutines BAND and FINISH are made once for each element (see below).

(7) For each element, carry out the following initialization operations:

(a) Set IMEM to the element number within the group (in sequence starting with 1).

(b) Set up the LM array, within /INFEL/. This is done with reference to the element node numbers and the ID array.

(c) Establish, within /INFEL/ , appropriate variables to permit the state of the element to be monitored. Initialize these variables to correspond to the unstressed state of the structure. Among other things these will include variables defining current element actions, which will be initialized to zero and then progressively updated as the analysis proceeds.

(d) If the element type has an option for element loads, define variables in /INFEL/ to store initial element actions, in addition to the current values, and initialize to zero. Separate initial and current values are needed for calculation of the element elastic-plastic energies (ENER) and resisting force vectors (RELAS). If a gravity load analysis is performed,

both the initial and current actions will be set, in subroutine GLODXX, at the beginning of the analysis. The initial actions will then remain constant, while the current actions will be progressively updated.

(e)   Save information on the element size and orientation in space, so that subsequent lineari-zation and state determination calculations can be carried out. One way is to store a displacement transformation matrix relating nodal displacements to element deformations. Alternatively, the X,Y coordinates of the element nodes may be stored.

(f)   Initialize all other variables in /INFEL/ to appropriate values corresponding to the unstressed state of the structure.

(g)   Call subroutine BAND with the statement

            CALL BAND

to permit the base program to establish information on the form of the structure stiffness matrix. This call may be made any time after LM has been defined and must be made once for each element.

(h)   Call subroutine FINISH with the statement

            CALL FINISH

This call must be made after the /INFEL/ block for the element has been fully initialized, and must be made once for each element.

(8)   Do appropriate input data checking. If there are errors set KDATA to -2.

7. SUBROUTINE ELODXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine ELODXX are as follows:

SUBROUTINE ELODXX (NMEM, NINFL, NLOD, SETLOD, ELFACT, KDATA)

COMMON /TAPES/ INP, IOU

DIMENSION SETLOD (NINFL, NLOD), ELFACT (NLOD, NMEM)

Argument Types

The arguments are of following types:

(a) Input: NMEM, NINFL, NLOD

(b) Modify: SETLOD, ELFACT, KDATA

Purpose

Subroutine ELODXX is called once for each element **group**, each time an element load pattern is input. Its purpose is to store element load sets (SETLOD array) and load set scale factors (ELFACT array) for the load pattern. These arrays are used by subroutine GLODXX to construct load vectors for gravity load analysis.

Tasks Performed Before Entry

For each element group which is affected by the load pattern, the base program reads one line of input data. This line contains the number of element load sets (NLOD) for the group. The base program then initializes SETLOD and ELFACT to zero, and calls ELODXX for the corresponding element type.

Tasks to be Performed in ELODXX

Each column of array SETLOD will store one load set (typically a fixed end force set). It is suggested that the first term of each column be a key to identify the type of load set. For example this key could identify whether fixed end forces are in local (element) or global (structure) coordinates.

Each column of array ELFACT corresponds to an element in the current group. Term I,J in ELFACT is the scale factor by which load set I is to be multiplied for element J. If the factor is zero, the load set does not affect the element.

The following tasks must be performed:

(1)   Print appropriate headings for the load sets.

(2)   Read NLOD load sets, store in SETLOD, and echo print.

(3)   Print appropriate headings for the load set scale factors.

(4)   Read in all nonzero scale factors, identifying both the element and the load set, and using generation options as appropriate. Store in ELFACT and echo print.

(5)   If an element has no element loads (i.e. all scale factors are zero for that element), set ELFACT(1) for the element to 999999. This will save some time in later computation.

(6)   Do appropriate input checking. If there are errors set KDATA to -2.

8.   SUBROUTINE STIFXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine STIFXX are as follows:

SUBROUTINE STIFXX (KSTT, KTYPE, NDOF, FK)

COMMON /INFGR/

COMMON /INFEL/ IMEM, LM(..

COMMON /WORK/ .....

DIMENSION FK (NDOF, NDOF)

Argument Types

The arguments are of the following types:

(a) Input: KSTT, KTYPE, NDOF

(b) Output: FK

Purpose

Subroutine STIFXX is called for an element whenever its stiffness changes. Only those elements with stiffness changes are affected (for initial formation of the stiffness this will be all

elements; for subsequent changes typically only a few elements). Its purpose is to form the element total stiffness or change in stiffness, and return it in array FK.

Tasks Performed Before Entry

The base program sets up /INFGR/ for the current element group and /INFEL/ for the current element before calling STIFX The variables KSTT and KTYPE are also set, to specify the required stiffness type.

Tasks to be Performed in STIFXX

STIFXX must have the capability to form both the total stiffness and the change in stiffness since the last time the stiffness was formed. Depending on the values of the variables KSTT and KTYPE, it may be required to return the elastic-plastic stiffness only, the elastic-plastic plus geometric stiffness or the geometric stiffness only.

The following tasks must be performed:

(1)   KSTT = 0: Form the change in stiffness since the last time subroutine STIFXX was called for the element.

> (a) KTYPE = 1: Elastic-plastic stiffness only. This will be done if second order effects are being ignored, or if they are being considered but the structure geometric stiffness is assumed to be constant.

> (b) KTYPE = 0: Elastic-plastic plus geometric stiffness. This will be done if second order effects are being considered and the structure geometric stiffness progressively changes.

> (c) KTYPE = -1: Geometric stiffness only. This will be be done if the structure geometric stiffness progressively changes but there is no change in the structure elastic-plastic stiffness.

(2)   KSTT = 1: Form the total stiffness, in the current element state.

> (a) KTYPE = 1: Elastic-plastic stiffness only. This will be done when a complete structure stiffness is being formed and second order effects are being ignored.

> (b) KTYPE = 0: Elastic-plastic plus geometric stiffness. This will be done when a

complete structure stiffness is being formed and second order effects are being considered.

(3) Update variables in the /INFEL/ block so that stiffness changes can be computed when STIFXX is next called. For simple elements it may be possible to store some indicators which identify the state when STIFXX was last called, and to calculate stiffness changes by comparing these "previous" indicators with the "current" values (which must be progressively updated in subroutine RESPXX). For more complex elements it may be necessary to store the total stiffness, and to get the stiffness change by subtraction.

9. SUBROUTINE GLODXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine GLODXX are as follows:

SUBROUTINE GLODXX (NINFL, NLOD, SETLOD, ELFACT, PMULT)

COMMON /INFEL/ IMEM, LM(..

COMMON /WORK/ ..

DIMENSION SETLOD (NINFL, NLOD), ELFACT (NLOD)

DIMENSION ELFOR (...

Argument Types

The arguments are of the following types:

(a) Input: NINFL, NLOD, SETLOD, ELFACT, PMULT

Array ELFOR is a local array.

Purpose

Subroutine GLODXX is called at the beginning of a gravity load analysis, once for each element and each element load pattern which contribute to the gravity load. Its purpose is to initialize

the element forces for the gravity load analysis, and to set up equivalent nodal loads for assembly into the gravity load vector.

Tasks Performed Before Entry

The following tasks are performed by the base program before subroutine GLODXX is called:

(1)   Multipliers for the element load patterns are read. The value for the current pattern is PMULT in the argument list.

(2)   The element load sets (SETLOD array) and load set scale factors (ELFACT array) are set up for the current load pattern and element group. The complete SETLOD array (all load set data for the current load pattern) plus one column of ELFACT (load set scale factors for the current element) are sent through the argument list. See section 2 for a description of these arrays.

Tasks to be Performed in GLODXX

The following tasks must be performed:

(1)   If all load set scale factors for the element are zero (ELFACT(1) = 999999.), simply return.

(2)   Using the SETLOD and ELFACT arrays, combine the element load data (considering all load sets with nonzero ELFACT values), and set up equivalent nodal forces values (i.e. forces in equilibrium with the element loads). These nodal force values must be forces in the **global** coordinate system acting **on the element**. For a beam-type element they will be conventional fixed-end forces, acting **on the element**.

(3)   Multiply the nodal force values by the load pattern multiplier (PMULT), and set up in the local array ELFOR(NDOF), where NDOF = number of element degrees of freedom. Then call subroutine SFORCE as follows:

CALL SFORCE (ELFOR)

Subroutine SFORCE changes the sign of ELFOR, then assembles it directly into the gravity load vector.

(4)   The /INFEL/ block must contain variables storing, separately, the current and initial element

actions. The initial values define the state of the element in the fixed end condition. These variables must have been initialized to zero in subroutine INELX Update these element actions by adding to them the contributions from the current load pattern. Addition, rather than replacement, is necessary because there may be several load patterns contributing to the gravity load analysis. The current and initial element actions will be equal at this stage. The initial actions remain constant, while the current actions are progressively updated during the analysis. The initial actions are needed for calculating the element elastic-plastic energy (ENER) and resisting force vector (RELAS). See the DRAIN-2DX theory manual for the procedure.

10. SUBROUTINE FACTXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine FACTXX are as follows:

SUBROUTINE FACTXX (NDOF, EFAC, DDISE)

COMMON /INFGR/ ...

COMMON /INFEL/ IMEM, LM (...

DIMENSION DDISE (NDOF)

Argument Types

The arguments are of the following types:

(a) Input: NDOF, DDISE

(b) Modify: EFAC

Purpose

Subroutine FACTXX is called once for each element in each static analysis substep, and for each dynamic analysis substep if the event-to-event solution strategy is specified. The purpose of the subroutine is to calculate the element event factor (EFAC), based on the element displacement increment DDISE.

Tasks Performed Before Entry

The following tasks are performed by the base program before subroutine FACTXX is called:

(1) The /INFGR/ and /INFEL/ blocks are set up for for the current group and element.

(2) Nodal displacement increments for the current element are set up in array DDISE.

(3) EFAC is set to 1.0.

Task to be Performed in FACTXX

Determine the proportion of DDISE required to reach the next significant nonlinear "event". This is the element event factor. If it is less than 1.0, set EFAC = element event factor.

11. SUBROUTINE RESPXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine RESPXX are as follows:

SUBROUTINE RESPXX (KRESIS, KSAVE, KGEM, KSTEP, NDOF, KST, KENR, ENER,

ENED, ENSO. TIM, BETO, RELAS, RDAMP, DDISE, DISE, VELE)

COMMON /INFGR/ ...

COMMON /INFEL/ IMEM, LM (...

COMMON /TAPES/ INP, IOU

COMMON /THELM/ ...

COMMON /WORK/ ..

DIMENSION RELAS (NDOF), RDAMP(NDOF), DDISE (NDOF), DISE (NDOF), VELE

(NDOF)

Argument Types

The arguments are of the following types:

(a) Input: KRESIS, KSAVE, KGEM, NDOF, KENR, TIM, BETO, DDISE , DISE, VELE

(b) Output: ENER, ENED, ENSO, RELAS, RDAMP

(c) Modify: KSTEP, KST

Purpose

Subroutine RESPXX is called once for each element in each analysis substep. Its purpose is to update the element state, form the static and damping resisting forces, perform energy calculations, update response quantities, and put element results in /THELM/ for saving or printing.

Tasks Performed Before Entry

The following tasks are performed by the base program before subroutine RESPXX is called:

(1) KRESIS is set to define the type of resisting force to be calculated. The options are as follows:

       (a) KRESIS = 1: elastic-plastic only (static).

       (b) KRESIS = 2: elastic-plastic and damping (dynamic).

(2) KST is set to zero.

(3) KGEM is set to indicate whether second order calculations are to be performed.

(4) KSAVE is set to indicate whether element results are to be saved and/or printed.

(5) Displacement increments, total displacements and velocities are set up in arrays DDISE, DISE and VELE, respectively.

(6) The /INFGR/ and /INFEL/ blocks are set up for the current group and element.

Tasks to be Performed in RESPXX

The following tasks must be performed:

(1) If KSTEP = -1, this is the first entry to RESPXX for the current analysis segment. Set any envelope step numbers or times to 0, and re-set KSTEP = 1.

(2)   Update the state for the element. Typically the following steps will be followed:

(a)   Using DDISE, determine increments of element deformations. Hence, by tracing the non-linear action-deformation relationship, determine increments of element actions. For a complex element this can be a lengthy computation. Update the element deformations, actions, etc. as appropriate.

(b)   If KENR = 1, calculate the **increment** of element elastic-plastic work, ENER.

(c)   If KENR > 0 and KGEM > 0, calculate the **increment** of element second order work, ENSO. See the DRAIN-2DX theory manual for the procedure.

(d)   Compare the current state with the state when the stiffness matrix was last formed (i.e. when KST was last set to > 0). A commonly used procedure will be to assign values to some state indicators storing previous and current values. If the state of the element changes, so that its stiffness must be modified for the next step, set KST to 1. If KST is set to = 1, subroutine STIFXX will be called for the element. Do not replace previous state indicators with the current ones - this must be done in STIFXX.

(e)   If KRESIS = 1, form the elastic-plastic resisting force vector (RELAS), in global coordinates. RELAS must include "second order" effects if KGEM > 0. This task may be performed by calling a subroutine. If so, the subroutine name ELASXX is recommended.

(f)   If KRESIS = 2, form the element damping force vector (RDAMP). If KENR = 2, calculate the **increment** of element damping work (ENED), equal to the work done by the average damping forces moving through the deformation increments. These tasks may be performed by calling a subroutine. If so, the subroutine name DAMPXX is recommended.

(g)   Calculate values of response quantities for the element.

(h)   Update the envelope values as necessary and the step numbers or times at which they occur. Note that the step number should be relative to the beginning of the current analysis segment.

(i)   If KSAVE = 1, put the results for the element in /THELM/. Convert double precision variables to them in /THELM/, to save disk storage. This task may be performed by calling a subroutine. If so, the subroutine name SAVEXX is recommended.

12. SUBROUTINE ENPRXX

Beginning Statements

The SUBROUTINE and COMMON statements for subroutine ENPRXX are as follows:

SUBROUTINE ENPRXX (NFOUTP)

COMMON /INFEL/ IMEM, LM(...

COMMON /INFGR/ . . .

Argument Types

The argument NFOUTP is of input type.

Purpose

Subroutine ENPRXX prints, on unit NFOUTP, envelope values for the current element. ENPRXX is called from the base program for all elements at step intervals defined by the input data. In addition, the subroutine is automatically called for all elements at the end of each analysis segment.

Tasks Performed Before Entry

The /INFGR/ and /INFEL/ blocks are set up for the current group and element.

Tasks to be Performed in ENPRXX

The following tasks must be performed:

(1)   Print an appropriate heading (only for the first element in the group, i.e., if IMEM = 1)

(2)   Print the envelope values.

13. SUBROUTINE THPRXX

Beginning Statements

The SUBROUTINE and COMMON statements for the subroutine THPRXX are as follows:

SUBROUTINE THPRXX(NS, IMEM, KSTEP, KHD, TIM, NFOUTP)

COMMON /THELM/ . . .

Argument Types

The arguments are of following types:

(a) Input: NS, IMEM, KSTEP, KHD, TIM, NFOUTP

Purpose

Subroutine THPRXX is called by the base program to print, on unit NFOUTP, element results at the end of an analysis segment. The subroutine is called once for each set of results that has been saved.

Tasks Performed Before Entry

The base program sets up the results for the current element and analysis step in the /THELM/ block. The /INFGR/ and /INFEL/ blocks are not set up. Hence, all data to be printed must be in the /THELM/ block (or in the argument list).

Tasks to be Performed in THPRXX

The following tasks must be performed:

(1)   If KHD = 1 (which identifies the first set of results for the first element in an element group), print a table heading for the results. It is suggested that the first column of the table be the element number (IMEM), the second column be the step number (KSTEP) and the third column (TIM) be time (for dynamic analysis) or substep load factor (for static analysis).

(2)   If NS = 1 (which identifies the first set of results for each element), print the element number and the results. Otherwise, omit the element number (i.e. leave the first column blank), and

print the results only.

14. SUBROUTINE FLINXX

Beginning Statements

The SUBROUTINE, COMMON, and DIMENSION statements for subroutine FLINXX are as follows:

SUBROUTINE FLINXX (NDOF, NFLIN, RELAS, EFLIN, DDISE)

COMMON /INFGR/ ...

COMMON /INFEL/ IMEM, LM(...

COMMON /TAPES/ INP, IOU

COMMON /WORK/ ...

DIMENSION RELAS (NDOF), EFLIN (NFLIN), DDISE (NDOF)

Argument Types

The arguments are of following types:

(a) Input: NDOF, NFLIN, DDISE

(b) Output: RELAS, EFLIN

Purpose

Subroutine FLINXX is called once for each mode and each element during response spectrum analysis. Its purpose is to form element results and static resisting forces, assuming linear behavior. The element results are used to set up the results output for the analysis. The resisting forces are used to calculate the forces on structure sections.

Tasks Performed Before Entry

The following two tasks are performed by the base program before subroutine FLINXX is called.

(1) Displacements (corresponding to a mode shape) are set up in array DDISE for the current element.

(2) The /INFGR/ and /INFEL/ blocks for the **unstressed** state are set up for the current group and element.

Tasks to be Performed in FLINXX

The following tasks must be performed.

(1) Determine element deformations using DDISE.

(2) Calculate **linear** element actions corresponding to the deformations in task 1

(3) Set up the static resisting force vector (RELAS) in global coordinates.

(4) Set up in EFLIN, the element results which are to be combined (using SRSS combination) and output. Since this is a linear analysis, the results are likely to be different from those saved in /THELM/. Note also, that EFLIN is a double precision array.

15. SUBROUTINE PLINXX

Beginning Statements

The SUBROUTINE, and DIMENSION statements for subroutine PLINXX are as follows:

SUBROUTINE PLINXX (IMEM, NFLIN, EFLIN, NFOUTP)

DIMENSION EFLIN (NFLIN)

Argument Types

The arguments are of the following types:

(a) Input: IMEM, NFLIN, EFLIN, NFOUTP

Purpose

Subroutine PLINXX is called to print, on unit NFOUTP, the element results for response spectrum analyses. It is called once for each element.

Tasks Performed Before Entry

The base program sets up the results for the current element in array EFLIN.

Tasks to be Performed in PLINXX

The following tasks must be performed.

(1)   If the element number (IMEM) is 1, print a table heading for the results. It is suggested that the first column of the table be the element number.

(2)   Print the results.

16. SUBROUTINE RHHDXX

Beginning Statement

The SUBROUTINE statement for subroutine RHHDXX is as follows:

SUBROUTINE RHHDXX (NFOUTP, KRSLT)

Argument Types

The arguments NFOUTP and KRSLT are of input type. NFOUT is the output unit number (formatted). KRSLT is an indicator for the type of results to be printed (KRSLT = 1 : element actions and deformations; currently KRSLT is always 1).

Purpose

Subroutine RHHDXX is called by DRAIN-POST to print element results headings.

Tasks to be Performed in RHHDXX

Write the appropriate table heading, based on KRSLT, on unit NFOUTP. It is suggested that the first column be time (for dynamic analysis) or load factor (for static analysis).

17. SUBROUTINE RHPRXX

Beginning Statements

The SUBROUTINE and COMMON statements for subroutine RHPRXX are as follows:

SUBROUTINE RHPRXX (TIME, NFOUTP, KRSLT)

COMMON /THELM/ . . .

Argument Types

The arguments TIME. NFOUTP, and KRSLT are of input type. The argument TIME is time (for dynamic analysis), or load factor (for static analysis). NFOUTP and KRSLT are as for subroutine RHHDX

Purpose

Subroutine RHPRXX is called by DRAIN-POST to print element results. The subroutine is called once for each TIME value.

Tasks Performed Before Entry

DRAIN-POST sets up the results for the current element and analysis step in the /THELM/ block. These results must have been saved in /THELM/ during a DRAIN analysis.

Tasks to be Performed in RHPRXX

Print the element results. corresponding to the heading printed in Subroutine RHHDX

EARTHQUAKE ENGINEERING RESEARCH CENTER REPORT SERIES

UCB/EERC-80/01 "Earthquake Response of Concrete Gravity Dams Including Hydrodynamic and Foundation Interaction Effects," by Chopra, A.K., Chakrabarti, P. and Gupta, S., January 1980, (AD-A087297)A10.

UCB/EERC-80/02 "Rocking Response of Rigid Blocks to Earthquakes," by Yim, C.S., Chopra, A.K. and Penzien, J., January 1980, (PB80 166 002)A04.

UCB/EERC-80/03 "Optimum Inelastic Design of Seismic-Resistant Reinforced Concrete Frame Structures," by Zagajeski, S.W. and Bertero, V.V., January 1980, (PB80 164 635)A06.

UCB/EERC-80/04 "Effects of Amount and Arrangement of Wall-Panel Reinforcement on Hysteretic Behavior of Reinforced Concrete Walls," by Iliya, R. and Bertero, V.V., February 1980, (PB81 122 525)A09.

UCB/EERC-80/05 "Shaking Table Research on Concrete Dam Models," by Niwa, A. and Clough, R.W., September 1980, (PB81 122 368)A06.

UCB/EERC-80/06 "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 1a): Piping with Energy Absorbing Restrainers: Parameter Study on Small Systems," by Powell, G.H., Oughourlian, C. and Simons, J., June 1980.

UCB/EERC-80/07 "Inelastic Torsional Response of Structures Subjected to Earthquake Ground Motions," by Yamazaki, Y., April 1980, (PB81 122 327)A08.

UCB/EERC-80/08 "Study of X-Braced Steel Frame Structures under Earthquake Simulation," by Ghanaat, Y., April 1980, (PB81 122 335)A11.

UCB/EERC-80/09 "Hybrid Modelling of Soil-Structure Interaction," by Gupta, S., Lin, T.W. and Penzien, J., May 1980, (PB81 122 319)A07.

UCB/EERC-80/10 "General Applicability of a Nonlinear Model of a One Story Steel Frame," by Sveinsson, B.I. and McNiven, H.D., May 1980, (PB81 124 877)A06.

UCB/EERC-80/11 "A Green-Function Method for Wave Interaction with a Submerged Body," by Kioka, W., April 1980, (PB81 122 269)A07.

UCB/EERC-80/12 "Hydrodynamic Pressure and Added Mass for Axisymmetric Bodies," by Nilrat, F., May 1980, (PB81 122 343)A08.

UCB/EERC-80/13 "Treatment of Non-Linear Drag Forces Acting on Offshore Platforms," by Dao, B.V. and Penzien, J., May 1980, (PB81 153 413)A07.

UCB/EERC-80/14 "2D Plane/Axisymmetric Solid Element (Type 3-Elastic or Elastic-Perfectly Plastic)for the ANSR-II Program," by Mondkar, D.P. and Powell, G.H., July 1980, (PB81 122 350)A03.

UCB/EERC-80/15 "A Response Spectrum Method for Random Vibrations," by Der Kiureghian, A., June 1981, (PB81 122 301)A03.

UCB/EERC-80/16 "Cyclic Inelastic Buckling of Tubular Steel Braces," by Zayas, V.A., Popov, E.P. and Martin, S.A., June 1981, (PB81 124 885)A10.

UCB/EERC-80/17 "Dynamic Response of Simple Arch Dams Including Hydrodynamic Interaction," by Porter, C.S. and Chopra, A.K., July 1981, (PB81 124 000)A13.

UCB/EERC-80/18 "Experimental Testing of a Friction Damped Aseismic Base Isolation System with Fail-Safe Characteristics," by Kelly, J.M., Beucke, K.E. and Skinner, M.S., July 1980, (PB81 148 595)A04.

UCB/EERC-80/19 "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol.1B): Stochastic Seismic Analyses of Nuclear Power Plant Structures and Piping Systems Subjected to Multiple Supported Excitations," by Lee, M.C. and Penzien, J., June 1980, (PB82 201 872)A08.

UCB/EERC-80/20 "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 1C): Numerical Method for Dynamic Substructure Analysis," by Dickens, J.M. and Wilson, E.L., June 1980.

UCB/EERC-80/21 "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 2): Development and Testing of Restraints for Nuclear Piping Systems," by Kelly, J.M. and Skinner, M.S., June 1980.

UCB/EERC-80/22 "3D Solid Element (Type 4-Elastic or Elastic-Perfectly-Plastic) for the ANSR-II Program," by Mondkar, D.P. and Powell, G.H., July 1980, (PB81 123 242)A03.

UCB/EERC-80/23 "Gap-Friction Element (Type 5) for the Ansr-II Program," by Mondkar, D.P. and Powell, G.H., July 1980, (PB81 122 285)A03.

UCB/EERC-80/24 "U-Bar Restraint Element (Type 11) for the ANSR-II Program," by Oughourlian, C. and Powell, G.H., July 1980, (PB81 122 293)A03.

UCB/EERC-80/25 "Testing of a Natural Rubber Base Isolation System by an Explosively Simulated Earthquake," by Kelly, J.M., August 1980, (PB81 201 360)A04.

UCB/EERC-80/26 "Input Identification from Structural Vibrational Response," by Hu, Y., August 1980, (PB81 152 308)A05.

UCB/EERC-80/27 "Cyclic Inelastic Behavior of Steel Offshore Structures," by Zayas, V.A., Mahin, S.A. and Popov, E.P., August 1980, (PB81 196 180)A15.

UCB/EERC-80/28 "Shaking Table Testing of a Reinforced Concrete Frame with Biaxial Response," by Oliva, M.G., October 1980, (PB81 154 304)A10.

UCB/EERC-80/29 "Dynamic Properties of a Twelve-Story Prefabricated Panel Building," by Bouwkamp, J.G., Kollegger, J.P. and Stephen, R.M., October 1980, (PB82 138 777)A07.

UCB/EERC-80/30 "Dynamic Properties of an Eight-Story Prefabricated Panel Building," by Bouwkamp, J.G., Kollegger, J.P. and Stephen, R.M., October 1980, (PB81 200 313)A05.

UCB/EERC-80/31 "Predictive Dynamic Response of Panel Type Structures under Earthquakes," by Kollegger, J.P. and Bouwkamp, J.G., October 1980, (PB81 152 316)A04.

UCB/EERC-80/32 "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 3): Testing of Commercial Steels in Low-Cycle Torsional Fatique," by Spanner, P., Parker, E.R., Jongewaard, E. and Dory, M., 1980.

UCB/EERC-80/33   "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 4): Shaking Table Tests of Piping Systems with Energy-Absorbing Restrainers," by Stiemer, S.F. and Godden, W.G., September 1980, (PB82 201 880)A05.

UCB/EERC-80/34   "The Design of Steel Energy-Absorbing Restrainers and their Incorporation into Nuclear Power Plants for Enhanced Safety (Vol 5): Summary Report," by Spencer, P., 1980.

UCB/EERC-80/35   "Experimental Testing of an Energy-Absorbing Base Isolation System," by Kelly, J.M., Skinner, M.S. and Beucke, K.E., October 1980, (PB81 154 072)A04.

UCB/EERC-80/36   "Simulating and Analyzing Artificial Non-Stationary Earth Ground Motions," by Nau, R.F., Oliver, R.M. and Pister, K.S., October 1980, (PB81 153 397)A04.

UCB/EERC-80/37   "Earthquake Engineering at Berkeley - 1980," by , September 1980, (PB81 205 674)A09.

UCB/EERC-80/38   "Inelastic Seismic Analysis of Large Panel Buildings," by Schricker, V. and Powell, G.H., September 1980, (PB81 154 338)A13.

UCB/EERC-80/39   "Dynamic Response of Embankment, Concrete-Gavity and Arch Dams Including Hydrodynamic Interation," by Hall, J.F. and Chopra, A.K., October 1980, (PB81 152 324)A11.

UCB/EERC-80/40   "Inelastic Buckling of Steel Struts under Cyclic Load Reversal.," by Black, R.G., Wenger, W.A. and Popov, E.P., October 1980, (PB81 154 312)A08.

UCB/EERC-80/41   "Influence of Site Characteristics on Buildings Damage during the October 3,1974 Lima Earthquake," by Repetto, P., Arango, I. and Seed, H.B., September 1980, (PB81 161 739)A05.

UCB/EERC-80/42   "Evaluation of a Shaking Table Test Program on Response Behavior of a Two Story Reinforced Concrete Frame," by Blondet, J.M., Clough, R.W. and Mahin, S.A., December 1980, (PB82 196 544)A11.

UCB/EERC-80/43   "Modelling of Soil-Structure Interaction by Finite and Infinite Elements," by Medina, F., December 1980, (PB81 229 270)A04.

UCB/EERC-81/01   "Control of Seismic Response of Piping Systems and Other Structures by Base Isolation," by Kelly, J.M., January 1981, (PB81 200 735)A05.

UCB/EERC-81/02   "OPTNSR- An Interactive Software System for Optimal Design of Statically and Dynamically Loaded Structures with Nonlinear Response," by Bhatti, M.A., Ciampi, V. and Pister, K.S., January 1981, (PB81 218 851)A09.

UCB/EERC-81/03   "Analysis of Local Variations in Free Field Seismic Ground Motions," by Chen, J.-C., Lysmer, J. and Seed, H.B., January 1981, (AD-A099508)A13.

UCB/EERC-81/04   "Inelastic Structural Modeling of Braced Offshore Platforms for Seismic Loading," by Zayas, V.A., Shing, P.-S.B., Mahin, S.A. and Popov, E.P., January 1981, (PB82 138 777)A07.

UCB/EERC-81/05   "Dynamic Response of Light Equipment in Structures," by Der Kiureghian, A., Sackman, J.L. and Nour-Omid, B., April 1981, (PB81 218 497)A04.

UCB/EERC-81/06   "Preliminary Experimental Investigation of a Broad Base Liquid Storage Tank," by Bouwkamp, J.G., Kollegger, J.P. and Stephen, R.M., May 1981, (PB82 140 385)A03.

UCB/EERC-81/07   "The Seismic Resistant Design of Reinforced Concrete Coupled Structural Walls," by Aktan, A.E. and Bertero, V.V., June 1981, (PB82 113 358)A11.

UCB/EERC-81/08   "Unassigned," by Unassigned, 1981.

UCB/EERC-81/09   "Experimental Behavior of a Spatial Piping System with Steel Energy Absorbers Subjected to a Simulated Differential Seismic Input," by Stiemer, S.F., Godden, W.G. and Kelly, J.M., July 1981, (PB82 201 898)A04.

UCB/EERC-81/10   "Evaluation of Seismic Design Provisions for Masonry in the United States," by Sveinsson, B.I., Mayes, R.L. and McNiven, H.D., August 1981, (PB82 166 075)A08.

UCB/EERC-81/11   "Two-Dimensional Hybrid Modelling of Soil-Structure Interaction," by Tzong, T.-J., Gupta, S. and Penzien. J., August 1981, (PB82 142 118)A04.

UCB/EERC-81/12   "Studies on Effects of Infills in Seismic Resistant R/C Construction," by Brokken, S. and Bertero, V.V., October 1981, (PB82 166 190)A09.

UCB/EERC-81/13   "Linear Models to Predict the Nonlinear Seismic Behavior of a One-Story Steel Frame," by Valdimarsson, H., Shah, A.H. and McNiven, H.D., September 1981, (PB82 138 793)A07.

UCB/EERC-81/14   "TLUSH: A Computer Program for the Three-Dimensional Dynamic Analysis of Earth Dams," by Kagawa, T., Mejia, L.H., Seed, H.B. and Lysmer, J., September 1981, (PB82 139 940)A06.

UCB/EERC-81/15   "Three Dimensional Dynamic Response Analysis of Earth Dams," by Mejia, L.H. and Seed, H.B., September 1981, (PB82 137 274)A12.

UCB/EERC-81/16   "Experimental Study of Lead and Elastomeric Dampers for Base Isolation Systems," by Kelly, J.M. and Hodder, S.B., October 1981, (PB82 166 182)A05.

UCB/EERC-81/17   "The Influence of Base Isolation on the Seismic Response of Light Secondary Equipment," by Kelly, J.M., April 1981, (PB82 255 266)A04.

UCB/EERC-81/18   "Studies on Evaluation of Shaking Table Response Analysis Procedures," by Blondet, J. M., November 1981, (PB82 197 278)A10.

UCB/EERC-81/19   "DELIGHT.STRUCT: A Computer-Aided Design Environment for Structural Engineering," by Balling, R.J., Pister, K.S. and Polak, E., December 1981, (PB82 218 496)A07.

UCB/EERC-81/20   "Optimal Design of Seismic-Resistant Planar Steel Frames," by Balling, R.J., Ciampi, V. and Pister, K.S., December 1981, (PB82 220 179)A07.

UCB/EERC-82/01   "Dynamic Behavior of Ground for Seismic Analysis of Lifeline Systems," by Sato, T. and Der Kiureghian, A., January 1982, (PB82 218 926)A05.

UCB/EERC-82/02   "Shaking Table Tests of a Tubular Steel-Frame Model," by Ghanaat, Y. and Clough, R.W., January 1982, (PB82 220 161)A07.

UCB/EERC-82/03 "Behavior of a Piping System under Seismic Excitation: Experimental Investigations of a Spatial Piping System supported by Mechanical Shock Arrestors," by Schneider, S., Lee, H.-M. and Godden, W. G., May 1982, (PB83 172 544)A09.

UCB/EERC-82/04 "New Approaches for the Dynamic Analysis of Large Structural Systems," by Wilson, E.L., June 1982, (PB83 148 080)A05.

UCB/EERC-82/05 "Model Study of Effects of Damage on the Vibration Properties of Steel Offshore Platforms," by Shahrivar, F. and Bouwkamp, J.G., June 1982, (PB83 148 742)A10.

UCB/EERC-82/06 "States of the Art and Pratice in the Optimum Seismic Design and Analytical Response Prediction of R/C Frame Wall Structures," by Aktan, A.E. and Bertero, V.V., July 1982, (PB83 147 736)A05.

UCB/EERC-82/07 "Further Study of the Earthquake Response of a Broad Cylindrical Liquid-Storage Tank Model," by Manos, G.C. and Clough, R.W., July 1982, (PB83 147 744)A11.

UCB/EERC-82/08 "An Evaluation of the Design and Analytical Seismic Response of a Seven Story Reinforced Concrete Frame," by Charney, F.A. and Bertero, V.V., July 1982, (PB83 157 628)A09.

UCB/EERC-82/09 "Fluid-Structure Interactions: Added Mass Computations for Incompressible Fluid," by Kuo, J.S.-H., August 1982, (PB83 156 281)A07.

UCB/EERC-82/10 "Joint-Opening Nonlinear Mechanism: Interface Smeared Crack Model," by Kuo, J.S.-H., August 1982, (PB83 149 195)A05.

UCB/EERC-82/11 "Dynamic Response Analysis of Techi Dam," by Clough, R.W., Stephen, R.M. and Kuo, J.S.-H., August 1982, (PB83 147 496)A06.

UCB/EERC-82/12 "Prediction of the Seismic Response of R/C Frame-Coupled Wall Structures," by Aktan, A.E., Bertero, V.V. and Piazzo, M., August 1982, (PB83 149 203)A09.

UCB/EERC-82/13 "Preliminary Report on the Smart 1 Strong Motion Array in Taiwan," by Bolt, B.A., Loh, C.H., Penzien, J. and Tsai, Y.B., August 1982, (PB83 159 400)A10.

UCB/EERC-82/14 "Shaking-Table Studies of an Eccentrically X-Braced Steel Structure," by Yang, M.S., September 1982, (PB83 260 778)A12.

UCB/EERC-82/15 "The Performance of Stairways in Earthquakes," by Roha, C., Axley, J.W. and Bertero, V.V., September 1982, (PB83 157 693)A07.

UCB/EERC-82/16 "The Behavior of Submerged Multiple Bodies in Earthquakes," by Liao, W.-G., September 1982, (PB83 158 709)A07.

UCB/EERC-82/17 "Effects of Concrete Types and Loading Conditions on Local Bond-Slip Relationships," by Cowell, A.D., Popov, E.P. and Bertero, V.V., September 1982, (PB83 153 577)A04.

UCB/EERC-82/18 "Mechanical Behavior of Shear Wall Vertical Boundary Members: An Experimental Investigation," by Wagner, M.T. and Bertero, V.V., October 1982, (PB83 159 764)A05.

UCB/EERC-82/19 "Experimental Studies of Multi-support Seismic Loading on Piping Systems," by Kelly, J.M. and Cowell, A.D., November 1982.

UCB/EERC-82/20 "Generalized Plastic Hinge Concepts for 3D Beam-Column Elements," by Chen, P. F.-S. and Powell, G.H., November 1982, (PB83 247 981)A13.

UCB/EERC-82/21 "ANSR-II: General Computer Program for Nonlinear Structural Analysis," by Oughourlian, C.V. and Powell, G.H., November 1982, (PB83 251 330)A12.

UCB/EERC-82/22 "Solution Strategies for Statically Loaded Nonlinear Structures," by Simons, J.W. and Powell, G.H., November 1982, (PB83 197 970)A06.

UCB/EERC-82/23 "Analytical Model of Deformed Bar Anchorages under Generalized Excitations," by Ciampi, V., Eligehausen, R., Bertero, V.V. and Popov, E.P., November 1982, (PB83 169 532)A06.

UCB/EERC-82/24 "A Mathematical Model for the Response of Masonry Walls to Dynamic Excitations," by Sucuoglu, H., Mengi, Y. and McNiven, H.D., November 1982, (PB83 169 011)A07.

UCB/EERC-82/25 "Earthquake Response Considerations of Broad Liquid Storage Tanks," by Cambra, F.J., November 1982, (PB83 251 215)A09.

UCB/EERC-82/26 "Computational Models for Cyclic Plasticity, Rate Dependence and Creep," by Mosaddad, B. and Powell, G.H., November 1982, (PB83 245 829)A08.

UCB/EERC-82/27 "Inelastic Analysis of Piping and Tubular Structures," by Mahasuverachai, M. and Powell, G.H., November 1982, (PB83 249 987)A07.

UCB/EERC-83/01 "The Economic Feasibility of Seismic Rehabilitation of Buildings by Base Isolation," by Kelly, J.M., January 1983, (PB83 197 988)A05.

UCB/EERC-83/02 "Seismic Moment Connections for Moment-Resisting Steel Frames.," by Popov, E.P., January 1983, (PB83 195 412)A04.

UCB/EERC-83/03 "Design of Links and Beam-to-Column Connections for Eccentrically Braced Steel Frames," by Popov, E.P. and Malley, J.O., January 1983, (PB83 194 811)A04.

UCB/EERC-83/04 "Numerical Techniques for the Evaluation of Soil-Structure Interaction Effects in the Time Domain," by Bayo, E. and Wilson, E.L., February 1983, (PB83 245 605)A09.

UCB/EERC-83/05 "A Transducer for Measuring the Internal Forces in the Columns of a Frame-Wall Reinforced Concrete Structure," by Sause, R. and Bertero, V.V., May 1983, (PB84 119 494)A06.

UCB/EERC-83/06 "Dynamic Interactions Between Floating Ice and Offshore Structures," by Croteau, P., May 1983, (PB84 119 486)A16.

UCB/EERC-83/07 "Dynamic Analysis of Multiply Tuned and Arbitrarily Supported Secondary Systems," by Igusa, T. and Der Kiureghian, A., July 1983, (PB84 118 272)A11.

UCB/EERC-83/08 "A Laboratory Study of Submerged Multi-body Systems in Earthquakes," by Ansari, G.R., June 1983, (PB83 261 842)A17.

UCB/EERC-83/09 "Effects of Transient Foundation Uplift on Earthquake Response of Structures," by Yim, C.-S. and Chopra, A.K., June 1983, (PB83 261 396)A07.

UCB/EERC-83/10 "Optimal Design of Friction-Braced Frames under Seismic Loading," by Austin, M.A. and Pister, K.S., June 1983, (PB84 119 288)A06.

UCB/EERC-83/11 "Shaking Table Study of Single-Story Masonry Houses: Dynamic Performance under Three Component Seismic Input and Recommendations," by Manos, G.C., Clough, R.W. and Mayes, R.L., July 1983, (UCB/EERC-83/11)A08.

UCB/EERC-83/12 "Experimental Error Propagation in Pseudodynamic Testing," by Shiing, P.B. and Mahin, S.A., June 1983, (PB84 119 270)A09.

UCB/EERC-83/13 "Experimental and Analytical Predictions of the Mechanical Characteristics of a 1/5-scale Model of a 7-story R/C Frame-Wall Building Structure," by Aktan, A.E., Bertero, V.V., Chowdhury, A.A. and Nagashima, T., June 1983, (PB84 119 213)A07.

UCB/EERC-83/14 "Shaking Table Tests of Large-Panel Precast Concrete Building System Assemblages," by Oliva, M.G. and Clough, R.W., June 1983, (PB86 110 210/AS)A11.

UCB/EERC-83/15 "Seismic Behavior of Active Beam Links in Eccentrically Braced Frames," by Hjelmstad, K.D. and Popov, E.P., July 1983, (PB84 119 676)A09.

UCB/EERC-83/16 "System Identification of Structures with Joint Rotation," by Dimsdale, J.S., July 1983, (PB84 192 210)A06.

UCB/EERC-83/17 "Construction of Inelastic Response Spectra for Single-Degree-of-Freedom Systems," by Mahin, S. and Lin, J., June 1983, (PB84 208 834)A05.

UCB/EERC-83/18 "Interactive Computer Analysis Methods for Predicting the Inelastic Cyclic Behaviour of Structural Sections," by Kaba, S. and Mahin, S., July 1983, (PB84 192 012)A06.

UCB/EERC-83/19 "Effects of Bond Deterioration on Hysteretic Behavior of Reinforced Concrete Joints," by Filippou, F.C., Popov, E.P. and Bertero, V.V., August 1983, (PB84 192 020)A10.

UCB/EERC-83/20 "Analytical and Experimental Correlation of Large-Panel Precast Building System Performance," by Oliva, M.G., Clough, R.W., Velkov, M. and Gavrilovic, P., November 1983.

UCB/EERC-83/21 "Mechanical Characteristics of Materials Used in a 1/5 Scale Model of a 7-Story Reinforced Concrete Test Structure," by Bertero, V.V., Aktan, A.E., Harris, H.G. and Chowdhury, A.A., October 1983, (PB84 193 697)A05.

UCB/EERC-83/22 "Hybrid Modelling of Soil-Structure Interaction in Layered Media," by Tzong, T.-J. and Penzien, J., October 1983, (PB84 192 178)A08.

UCB/EERC-83/23 "Local Bond Stress-Slip Relationships of Deformed Bars under Generalized Excitations," by Eligehausen, R., Popov, E.P. and Bertero, V.V., October 1983, (PB84 192 848)A09.

UCB/EERC-83/24 "Design Considerations for Shear Links in Eccentrically Braced Frames," by Malley, J.O. and Popov, E.P., November 1983, (PB84 192 186)A07.

UCB/EERC-84/01 "Pseudodynamic Test Method for Seismic Performance Evaluation: Theory and Implementation," by Shing, P.-S.B. and Mahin, S.A., January 1984, (PB84 190 644)A08.

UCB/EERC-84/02 "Dynamic Response Behavior of Kiang Hong Dian Dam," by Clough, R.W., Chang, K.-T., Chen, H.-Q. and Stephen, R.M., April 1984, (PB84 209 402)A08.

UCB/EERC-84/03 "Refined Modelling of Reinforced Concrete Columns for Seismic Analysis," by Kaba, S.A. and Mahin, S.A., April 1984, (PB84 234 384)A06.

UCB/EERC-84/04 "A New Floor Response Spectrum Method for Seismic Analysis of Multiply Supported Secondary Systems," by Asfura, A. and Der Kiureghian, A., June 1984, (PB84 239 417)A06.

UCB/EERC-84/05 "Earthquake Simulation Tests and Associated Studies of a 1/5th-scale Model of a 7-Story R/C Frame-Wall Test Structure," by Bertero, V.V., Aktan, A.E., Charney, F.A. and Sause, R., June 1984, (PB84 239 409)A09.

UCB/EERC-84/06 "R/C Structural Walls: Seismic Design for Shear," by Aktan, A.E. and Bertero, V.V., 1984.

UCB/EERC-84/07 "Behavior of Interior and Exterior Flat-Plate Connections subjected to Inelastic Load Reversals," by Zee, H.L. and Moehle, J.P., August 1984, (PB86 117 629/AS)A07.

UCB/EERC-84/08 "Experimental Study of the Seismic Behavior of a Two-Story Flat-Plate Structure," by Moehle, J.P. and Diebold, J.W., August 1984, (PB86 122 553/AS)A12.

UCB/EERC-84/09 "Phenomenological Modeling of Steel Braces under Cyclic Loading," by Ikeda, K., Mahin, S.A. and Dermitzakis, S.N., May 1984, (PB86 132 198/AS)A08.

UCB/EERC-84/10 "Earthquake Analysis and Response of Concrete Gravity Dams," by Fenves, G. and Chopra, A.K., August 1984, (PB85 193 902/AS)A11.

UCB/EERC-84/11 "EAGD-84: A Computer Program for Earthquake Analysis of Concrete Gravity Dams," by Fenves, G. and Chopra, A.K., August 1984, (PB85 193 613/AS)A05.

UCB/EERC-84/12 "A Refined Physical Theory Model for Predicting the Seismic Behavior of Braced Steel Frames," by Ikeda, K. and Mahin, S.A., July 1984, (PB85 191 450/AS)A09.

UCB/EERC-84/13 "Earthquake Engineering Research at Berkeley - 1984," by , August 1984, (PB85 197 341/AS)A10.

UCB/EERC-84/14 "Moduli and Damping Factors for Dynamic Analyses of Cohesionless Soils," by Seed, H.B., Wong, R.T., Idriss, I.M. and Tokimatsu, K., September 1984, (PB85 191 468/AS)A04.

UCB/EERC-84/15 "The Influence of SPT Procedures in Soil Liquefaction Resistance Evaluations," by Seed, H.B., Tokimatsu, K., Harder, L.F. and Chung, R.M., October 1984, (PB85 191 732/AS)A04.

UCB/EERC-84/16 "Simplified Procedures for the Evaluation of Settlements in Sands Due to Earthquake Shaking," by Tokimatsu, K. and Seed, H.B., October 1984, (PB85 197 887/AS)A03.

UCB/EERC-84/17 "Evaluation of Energy Absorption Characteristics of Bridges under Seismic Conditions," by Imbsen, R.A. and Penzien, J., November 1984.

UCB/EERC-84/18 "Structure-Foundation Interactions under Dynamic Loads," by Liu, W.D. and Penzien, J., November 1984, (PB87 124 889/AS)A11.

UCB/EERC-84/19 "Seismic Modelling of Deep Foundations," by Chen, C.-H. and Penzien, J., November 1984, (PB87 124 798/AS)A07.

UCB/EERC-84/20 "Dynamic Response Behavior of Quan Shui Dam," by Clough, R.W., Chang, K.-T., Chen, H.-Q., Stephen, R.M., Ghanaat, Y. and Qi, J.-H., November 1984, (PB86 115177/AS)A07.

UCB/EERC-85/01 "Simplified Methods of Analysis for Earthquake Resistant Design of Buildings," by Cruz, E.F. and Chopra, A.K., February 1985, (PB86 112299/AS)A12.

UCB/EERC-85/02 "Estimation of Seismic Wave Coherency and Rupture Velocity using the SMART 1 Strong-Motion Array Recordings," by Abrahamson, N.A., March 1985, (PB86 214 343)A07.

UCB/EERC-85/03 "Dynamic Properties of a Thirty Story Condominium Tower Building," by Stephen, R.M., Wilson, E.L. and Stander, N., April 1985, (PB86 118965/AS)A06.

UCB/EERC-85/04 "Development of Substructuring Techniques for On-Line Computer Controlled Seismic Performance Testing," by Dermitzakis, S. and Mahin, S., February 1985, (PB86 132941/AS)A08.

UCB/EERC-85/05 "A Simple Model for Reinforcing Bar Anchorages under Cyclic Excitations," by Filippou, F.C., March 1985, (PB86 112 919/AS)A05.

UCB/EERC-85/06 "Racking Behavior of Wood-framed Gypsum Panels under Dynamic Load," by Oliva, M.G., June 1985.

UCB/EERC-85/07 "Earthquake Analysis and Response of Concrete Arch Dams," by Fok, K.-L. and Chopra, A.K., June 1985, (PB86 139672/AS)A10.

UCB/EERC-85/08 "Effect of Inelastic Behavior on the Analysis and Design of Earthquake Resistant Structures," by Lin, J.P. and Mahin, S.A., June 1985, (PB86 135340/AS)A08.

UCB/EERC-85/09 "Earthquake Simulator Testing of a Base-Isolated Bridge Deck," by Kelly, J.M., Buckle, I.G. and Tsai, H.-C., January 1986, (PB87 124 152/AS)A06.

UCB/EERC-85/10 "Simplified Analysis for Earthquake Resistant Design of Concrete Gravity Dams," by Fenves, G. and Chopra, A.K., June 1986, (PB87 124 160/AS)A08.

UCB/EERC-85/11 "Dynamic Interaction Effects in Arch Dams," by Clough, R.W., Chang, K.-T., Chen, H.-Q. and Ghanaat, Y., October 1985, (PB86 135027/AS)A05.

UCB/EERC-85/12 "Dynamic Response of Long Valley Dam in the Mammoth Lake Earthquake Series of May 25-27, 1980," by Lai, S. and Seed, H.B., November 1985, (PB86 142304/AS)A05.

UCB/EERC-85/13 "A Methodology for Computer-Aided Design of Earthquake-Resistant Steel Structures," by Austin, M.A., Pister, K.S. and Mahin, S.A., December 1985, (PB86 159480/AS)A10 .

UCB/EERC-85/14 "Response of Tension-Leg Platforms to Vertical Seismic Excitations," by Liou, G.-S., Penzien, J. and Yeung, R.W., December 1985, (PB87 124 871/AS)A08.

UCB/EERC-85/15 "Cyclic Loading Tests of Masonry Single Piers: Volume 4 - Additional Tests with Height to Width Ratio of 1," by Sveinsson, B., McNiven, H.D. and Sucuoglu, H., December 1985.

UCB/EERC-85/16 "An Experimental Program for Studying the Dynamic Response of a Steel Frame with a Variety of Infill Partitions," by Yanev, B. and McNiven, H.D., December 1985.

UCB/EERC-86/01 "A Study of Seismically Resistant Eccentrically Braced Steel Frame Systems," by Kasai, K. and Popov, E.P., January 1986, (PB87 124 178/AS)A14.

UCB/EERC-86/02 "Design Problems in Soil Liquefaction," by Seed, H.B., February 1986, (PB87 124 186/AS)A03.

UCB/EERC-86/03 "Implications of Recent Earthquakes and Research on Earthquake-Resistant Design and Construction of Buildings," by Bertero, V.V., March 1986, (PB87 124 194/AS)A05.

UCB/EERC-86/04 "The Use of Load Dependent Vectors for Dynamic and Earthquake Analyses," by Leger, P., Wilson, E.L. and Clough, R.W., March 1986, (PB87 124 202/AS)A12.

UCB/EERC-86/05 "Two Beam-To-Column Web Connections," by Tsai, K.-C. and Popov, E.P., April 1986, (PB87 124 301/AS)A04.

UCB/EERC-86/06 "Determination of Penetration Resistance for Coarse-Grained Soils using the Becker Hammer Drill," by Harder, L.F. and Seed, H.B., May 1986, (PB87 124 210/AS)A07.

UCB/EERC-86/07 "A Mathematical Model for Predicting the Nonlinear Response of Unreinforced Masonry Walls to In-Plane Earthquake Excitations," by Mengi, Y. and McNiven, H.D., May 1986, (PB87 124 780/AS)A06.

UCB/EERC-86/08 "The 19 September 1985 Mexico Earthquake: Building Behavior," by Bertero, V.V., July 1986.

UCB/EERC-86/09 "EACD-3D: A Computer Program for Three-Dimensional Earthquake Analysis of Concrete Dams," by Fok, K.-L., Hall, J.F. and Chopra, A.K., July 1986, (PB87 124 228/AS)A08.

UCB/EERC-86/10 "Earthquake Simulation Tests and Associated Studies of a 0.3-Scale Model of a Six-Story Concentrically Braced Steel Structure," by Uang, C.-M. and Bertero, V.V., December 1986, (PB87 163 564/AS)A17.

UCB/EERC-86/11 "Mechanical Characteristics of Base Isolation Bearings for a Bridge Deck Model Test," by Kelly, J.M., Buckle, I.G. and Koh, C.-G., 1987.

UCB/EERC-86/12 "Effects of Axial Load on Elastomeric Isolation Bearings," by Koh, C.-G. and Kelly, J.M., 1987.

UCB/EERC-87/01 "The FPS Earthquake Resisting System: Experimental Report," by Zayas, V.A., Low, S.S. and Mahin, S.A., June 1987.

UCB/EERC-87/02 "Earthquake Simulator Tests and Associated Studies of a 0.3-Scale Model of a Six-Story Eccentrically Braced Steel Structure," by Whittaker, A., Uang, C.-M. and Bertero, V.V., July 1987.

UCB/EERC-87/03 "A Displacement Control and Uplift Restraint Device for Base-Isolated Structures," by Kelly, J.M., Griffith, M.C. and Aiken, I.G., April 1987.

UCB/EERC-87/04 "Earthquake Simulator Testing of a Combined Sliding Bearing and Rubber Bearing Isolation System," by Kelly, J.M. and Chalhoub, M.S., 1987.

UCB/EERC-87/05 "Three-Dimensional Inelastic Analysis of Reinforced Concrete Frame-Wall Structures," by Moazzami, S. and Bertero, V.V., May 1987.

UCB/EERC-87/06 "Experiments on Eccentrically Braced Frames with Composite Floors," by Ricles, J. and Popov, E., June 1987.

UCB/EERC-87/07 "Dynamic Analysis of Seismically Resistant Eccentrically Braced Frames," by Ricles, J. and Popov, E., June 1987.

UCB/EERC-87/08 "Undrained Cyclic Triaxial Testing of Gravels-The Effect of Membrane Compliance," by Evans, M.D. and Seed, H.B., July 1987.

UCB/EERC-87/09 "Hybrid Solution Techniques for Generalized Pseudo-Dynamic Testing," by Thewalt, C. and Mahin, S.A., July 1987.

UCB/EERC-87/10 "Investigation of Ultimate Behavior of AISC Group 4 and 5 Heavy Steel Rolled-Section Splices with Full and Partial Penetration Butt Welds," by Bruneau, M. and Mahin, S.A., July 1987.

UCB/EERC-87/11   "Residual Strength of Sand from Dam Failures in the Chilean Earthquake of March 3, 1985," by De Alba. P., Seed, H.B., Retamal, E. and Seed, R.B., September 1987.

UCB/EERC-87/12   "Inelastic Seismic Response of Structures with Mass or Stiffness Eccentricities in Plan," by Bruneau, M. and Mahin, S.A., September 1987.

UCB/EERC-87/13   "CSTRUCT: An Interactive Computer Environment for the Design and Analysis of Earthquake Resistant Steel Structures," by Austin, M.A., Mahin, S.A. and Pister, K.S., September 1987.

UCB/EERC-87/14   "Experimental Study of Reinforced Concrete Columns Subjected to Multi-Axial Loading," by Low, S.S. and Moehle, J.P., September 1987.

UCB/EERC-87/15   "Relationships between Soil Conditions and Earthquake Ground Motions in Mexico City in the Earthquake of Sept. 19, 1985," by Seed, H.B., Romo, M.P., Sun, J., Jaime, A. and Lysmer, J., October 1987.

UCB/EERC-87/16   "Experimental Study of Seismic Response of R. C. Setback Buildings," by Shahrooz, B.M. and Moehle, J.P., October 1987.

UCB/EERC-87/17   "Three Dimensional Aspects of the Behavior of R. C. Structures Subjected to Earthquakes," by Pantazopoulou, S.J. and Moehle, J.P., October 1987.

UCB/EERC-87/18   "Design Procedures for R-FBI Bearings," by Mostaghel, N. and Kelly, J.M., November 1987.

UCB/EERC-87/19   "Analytical Models for Predicting the Lateral Response of R C Shear Walls: Evaluation of their Reliability," by Vulcano, A. and Bertero, V.V., November 1987.

UCB/EERC-87/20   "Earthquake Response of Torsionally-Coupled Buildings," by Hejal, R. and Chopra, A.K., December 1987.

UCB/EERC-87/21   "Dynamic Reservoir Interaction with Monticello Dam," by Clough, R.W., Ghanaat, Y. and Qiu, X-F., December 1987.

UCB/EERC-87/22   "Strength Evaluation of Coarse-Grained Soils," by Siddiqi, F.H., Seed, R.B., Chan, C.K., Seed, H.B. and Pyke, R.M., December 1987.

UCB/EERC-88/01   "Seismic Behavior of Concentrically Braced Steel Frames," by Khatib, I., Mahin, S.A. and Pister, K.S., January 1988.

UCB/EERC-88/02   "Experimental Evaluation of Seismic Isolation of Medium-Rise Structures Subject to Uplift," by Griffith, M.C., Kelly, J.M., Coveney, V.A. and Koh, C.G., January 1988.

UCB/EERC-88/03   "Cyclic Behavior of Steel Double Angle Connections," by Astaneh-Asl, A. and Nader, M.N., January 1988.

UCB/EERC-88/04   "Re-evaluation of the Slide in the Lower San Fernando Dam in the Earthquake of Feb. 9, 1971," by Seed, H.B., Seed, R.B., Harder, L.F. and Jong, H.-L., April 1988.

UCB/EERC-88/05   "Experimental Evaluation of Seismic Isolation of a Nine-Story Braced Steel Frame Subject to Uplift," by Griffith, M.C., Kelly, J.M. and Aiken, I.D., May 1988.

UCB/EERC-88/06   "DRAIN-2DX User Guide," by Allahabadi, R. and Powell, G.H., March 1988.