Title

Drain-2DX element description and user guide for element type 01, type 02, type 04, type 06, type 09, and type 15: version 1.10

Author

Powell, Graham

Publication Date

1993-12-01

REPORT NO.
UCB/SEMM-93/18

STRUCTURAL ENGINEERING
MECHANICS AND MATERIALS

DRAIN-2DX ELEMENT DESCRIPTION AND USER GUIDE FOR ELEMENT TYPE01, TYPE02, TYPE04, TYPE06, TYPE09, and TYPE15

VERSION 1.10

BY

G. H. POWELL

DECEMBER 1993

DEPARTMENT OF CIVIL ENGINEERING
UNIVERSITY OF CALIFORNIA
BERKELEY, CALIFORNIA

INELASTIC TRUSS BAR ELEMENT (TYPE 01) FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E01.1 PURPOSE, FEATURES AND LIMITATIONS

E01.1.1 PURPOSE

This is a simple inelastic bar element. It can be used for truss bars, simple columns, and nonlinear supports springs.

E01.1.2 ELEMENT MODEL

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 (1) yielding in both tension and compression, as shown in Figure E01.2(a), and (2) yielding in tension but elastic buckling in compression as shown in Figure E02.1(b)). Strain hardening effects are included by dividing each element into two parallel components, one elastic and one elastic-perfectly-plastic, as shown in Figure E01.3.

P-Δ effects can be considered.

Static loads applied along the element length, or initial forces due to other causes, can be taken into account by specifying fixed end forces.

E01.1.3 VISCOUS DAMPING

If βK damping is specified, a linear viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the initial (elastic) stiffness of the element.

The stiffness of the viscous element remains constant for any dynamic analysis, even if the basic element yields. However, the amount of viscous damping can be changed if the structure is in a static state, using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E01.1.4 OVERSHOOT TOLERANCE

If event-to-event analysis is to be used, an overshoot tolerance must be specified. This is a tolerance on the element yield force.

An "event" corresponds to a change in stiffness of an element, due to yield, inelastic unloading, etc.. If event-to-event analysis is used, the structure stiffness is reformed at each event. It is usually wise to use event-to-event analysis.

Consider the case where the event is element yield. If a zero value is input for the overshoot tolerance, the event factor is calculated so that the most critical element just yields. If a nonzero value is input, the event factor is chosen so that the force or moment in the element is its yield value plus the tolerance. That is, the element is allowed to "overshoot" beyond its nominal yield value. As a result, there will be an equilibrium unbalance at the event, and the analysis will be less accurate. However, the number of events (stiffness reformulations) may be reduced, because a number of elements may yield in a single analysis substep. In general, a small overshoot tolerance will give a more accurate analysis, but will require more execution time.

The amount of overshoot can be controlled in two ways, first by specifying an overshoot tolerance as part of the element properties, and second by specifying "event overshoot scale factors" with the "F" option in the *PARAMETERS input section. If no overshoot scale factors are input, these factors default to 1.0, and the overshoot tolerances input with the element properties are used. If overshoot scale factors are input, the overshoot tolerances are scaled by these factors. Separate overshoot scale factors can be input for static and dynamic analyses, and for each element group. The overshoot tolerances can thus be changed at any time, by changing the overshoot scale factors. One way to define overshoot tolerances is to specify a unit value with the element properties, and then control the actual value with overshoot scale factors.

E01.1.5 ELEMENT LOADS

Static loads applied along the lengths of an element, or element initial forces, can be taken into account by specifying fixed end forces as shown in Figure E01.4. These are the forces that must act ***on the element ends*** to prevent end displacement.

E01.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E01.1 and E01.2 for element geometry and properties.

E01.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NPROP	No. of property types (min. 1, max. 40). See Section E01.2.2.

E01.2.2 Property Types

NPROP lines, one line per property type.

Columns	Notes	Variable	Data
1-5(I)			Property type number, in sequence beginning with 1.
6-15(R)			Young's modulus, E.
16-25(R)			Strain hardening ratio, Eh/E. Must be > 0 and < 1.
26-35(R)			Cross section area, A.
36-45(R)			Yield stress in tension, Syt.
46-55(R)			Yield stress or buckling stress in compression, Syc.
60(I)			Buckling code, as follows. 0 = yields in compression without buckling. 1 = buckles elastically in compression.
61-70(R)			Force overshoot tolerance.

E01.2.3 Element Generation Commands.

One line for each command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number or number of first element in a sequentially numbered series of elements to be generated by this command.
6-15(I)			Node number at end I..
16-25(I)			Node number at end J.
26-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Property type number.

E01.3 INPUT DATA FOR *ELEMENTLOAD

E01.3.1 Load Sets

NLOD lines (see Element Group line of *ELEMENTLOAD section), one line per element load set.

See Figure E01.4 for sign convention.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
6-10(R)			Coordinate code. 0 = forces are in local (element) coordinates. 1 = forces are in global (structure)coordinates.
11-20(R)			Force Pi.
21-30(R)			Force Vi.
31-40(R)			Force Pj.
41-50(R)			Force Vj.

E01.3.2 Loaded Elements and Load Set Scale Factors

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

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

E01.4 INTERPRETATION OF RESULTS

E01.4.1 SIGN CONVENTION

Tension force and axial extension are positive.

Accumulated plastic deformations are calculated as shown in figure E01.5.

E01.4.2 EVENT CODES

In an event-to-event analysis, the element that governs the event is identified in the .ECH file, with a code that shows the type of event. The event types are as follows.

Code	Event type
1	Tension yield.
2	Compression yield.
3	Buckling.
4	Unloading from tension yield.
5	Unloading from compression yield.
6	Unloading from buckling.

E01.4.3 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E01.4.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E01.4.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (8 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine SAVE01 in the ANAL01.FOR source code file.

Item	Description
1	Static force.
2	Viscous force.
3	Deformation.
4	Accumulated positive plastic deformation.
5	Accumulated negative plastic deformation.
6	Node number at end I.
7	Node number at end J.
8	Yield code (0 = not yielded, 1 = yielded or buckled).

E01.4.6 USER OUTPUT (.USR FILE)

A sample subroutine (source code file USER01.FOR) is included to illustrate how the user output option might be used.

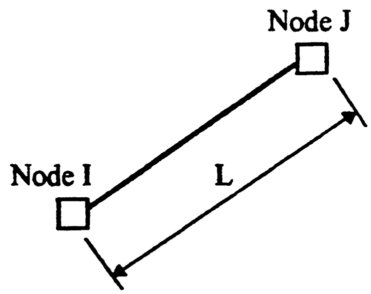


FIGURE E01.1 ELEMENT GEOMETRY

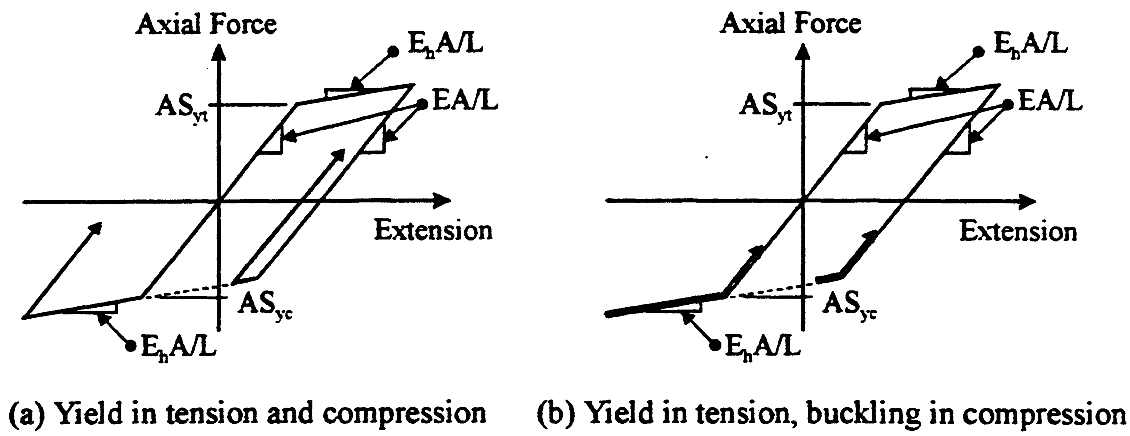


FIGURE E01.2 ELEMENT BEHAVIOR

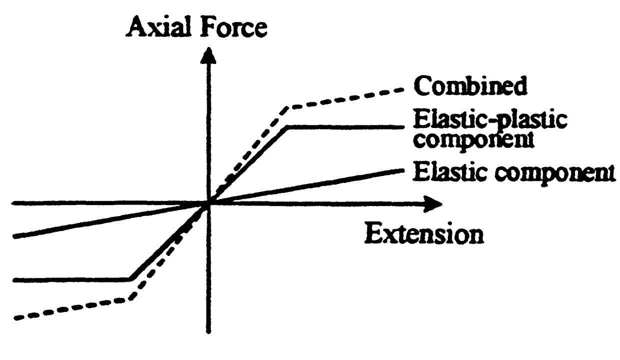
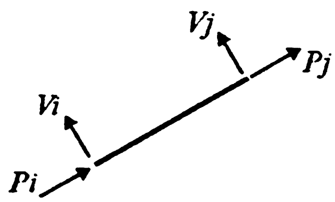
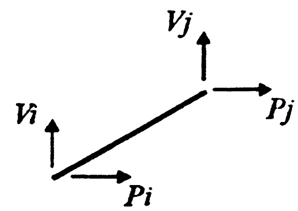


FIGURE E01.3 PARALLEL COMPONENTS



(a) Code = 0 (local)



(b) Code =1 (global)

FIGURE E01.4 FIXED END FORCES

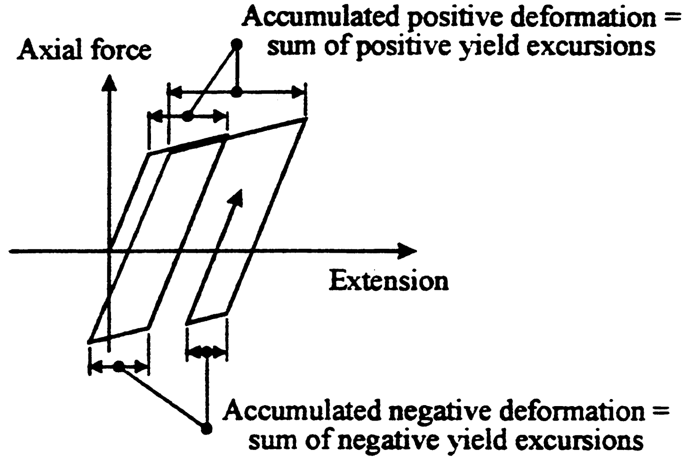


FIGURE E01.5 ACCUMULATED PLASTIC DEFORMATIONS

PLASTIC HINGE BEAM-COLUMN ELEMENT (TYPE 02) FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E02.1 PURPOSE, FEATURES AND LIMITATIONS

E02.1.1 PURPOSE

This is a simple inelastic element for modelling beams and beam-columns of steel and reinforced concrete type. The element has serious limitations, as noted later. In particular, it is not theoretically correct for beam-columns with P-M interaction. If P-M interaction effects are important, consider using element type 15 rather than this element.

E02.1.2 ELEMENT MODEL

The element geometry is shown in Figure E02.1. An element consists essentially of an elastic beam, two rigid-plastic hinges at the ends of this beam, and optional rigid end zones.

Elements of variable cross section can be considered by specifying appropriate flexural stiffness coefficients for the elastic beam, as shown in Figure E02.2. Elastic shear deformations can be included by specifying an effective shear area.

Yielding takes place only in the plastic hinges. The hinge yield moments can be specified to be different at the two element ends, and for positive and negative bending. The effect of axial force on bending strength is taken into account by specifying P-M yield surfaces as shown in Figure E02.3. However, the plastic hinges are assumed to yield only in bending, with no inelastic axial deformation. This corresponds to plastic flow along the M direction only, not along the normal to the yield surface, which is not theoretically correct.

Strain hardening in bending is modelled by assuming that the element consists of elastic and inelastic components in parallel, as indicated in Figure E02.4. Plastic hinges that yield at constant moment form in the inelastic component. The moments in the elastic component continues to increase, simulating strain hardening.

P-Δ effects can be considered.

Static loads applied along the element length, or initial forces due to other causes, can be taken into account by specifying fixed end forces.

E02.1.3 VISCOUS DAMPING

If βK damping is specified, a viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the initial (elastic) stiffness of the element.

The stiffness of the viscous element remains constant for any dynamic analysis, even if the basic element yields. However, the amount of viscous damping can be changed if the structure is in a static state, using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E02.1.4 OVERSHOOT TOLERANCE

If event-to-event analysis is to be used, an overshoot tolerance must be specified. This is a tolerance on the yield moment at a plastic hinge.

An "event" corresponds to a change in stiffness of an element, due to yield, inelastic unloading, etc. If event-to-event analysis is used, the structure stiffness is reformed at each event. It is usually wise to use event-to-event analysis.

Consider the case where the event is element yield. If a zero value is input for the overshoot tolerance, the event factor is calculated so that the most critical element just yields. If a nonzero value is input, the event factor is chosen so that the moment in the element is its yield value plus the tolerance. That is, the element is allowed to "overshoot" beyond its nominal yield value. As a result, there will be an equilibrium unbalance at the event, and the analysis will be less accurate. However, the number of events (stiffness reformulations) may be reduced, because a number of elements may yield in a single analysis substep. In general, a small overshoot tolerance will give a more accurate analysis, but will require more execution time.

The amount of overshoot can be controlled in two ways, first by specifying an overshoot tolerance as part of the element properties, and second by specifying "event overshoot scale factors" with the "F" option in the *PARAMETERS input section. If no overshoot scale factors are input, these factors default to 1.0, and the overshoot tolerances input with the element properties are used. If overshoot scale factors are input, the overshoot tolerances are scaled by these factors. Separate overshoot scale factors can be input for static and dynamic analyses, and for each element group. The overshoot tolerances can thus be changed at any time, by changing the overshoot scale factors. One way to define overshoot tolerances is to specify a unit value with the element properties, and then control the actual value with overshoot scale factors.

E02.1.5 ELEMENT LOADS

Static loads applied along the lengths of an element can be taken into account by specifying fixed end forces as shown in Figure E02.5. These are the forces that 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. Often 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 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 the element shear and axial forces, the full specified fixed end forces are used. However, for 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 are thus reduced, and the forces acting on the beam ends are still correct.

If rigid end zones are present, the fixed end forces are at the ends of the deformable part of the element, not at the nodes. Rigid end zone effects are taken into account in transferring the fixed end forces to the nodes (i.e., the moment loads on the nodes are augmented by couples created by the fixed end shears and axial forces).The live load reduction factors are applied to the fixed end shear and axial forces before they are transferred to the nodes.

E02.1.6 MAJOR LIMITATIONS

Strain Hardening

Strain hardening is modelled by placing an elastic component in parallel with an inelastic component, as shown in Figure E02.4. Care must be taken in choosing both the element strength and the strain hardening ratio.

To illustrate this, consider a beam with uniform stiffness and strength, and a moment-curvature relationship as shown in Figure E02.6(a). If the bending moment on the beam is constant, the moment-rotation relationship at an element end has the same shape as the moment-curvature relationship, as shown in Figure E02.6(b). This is because curvature and rotation in this case are directly proportional. If, however, the bending moment varies, for example as shown in Figure E02.6(c), then curvatures and rotations are no longer proportional, and the moment-rotation and moment-curvature relationships for the actual beam are no longer the same. However, if the same element properties are specified as for Figure E02.6(b), these relationships are still the same for the parallel component model. That is, if the element properties are chosen to give the correct behavior for uniform bending, the behavior may not be correct for nonuniform bending.

Yield Surfaces and Axial Force Behavior

When a plastic hinge forms, it is assumed to yield only in bending, at a constant moment. If a yield surface of steel or reinforced concrete type is specified, the yield moment is calculated allowing for the magnitude of the axial force. The hinge is then assumed to rotate under constant moment, as indicated in Figure E02.7(a). This means that the P-M point can move outside the yield surface. The error is corrected in the next analysis step, by recalculating the yield moment and applying an equilibrium correction as shown. This also means that the element is assumed to be elastic for axial forces, and that there is no axial growth or shortening associated with yield at the hinges. This is not correct. For an element that is more correct, both theoretically and physically, consider element type 15.

An equilibrium correction is also applied when a new hinge forms, as shown in Figure E02.7(b).

P-Δ Effects

P-Δ effects are based on the geometric stiffness for a truss bar element, not a beam element (i.e., assuming a straight line displaced shape, not a cubic or some other curve). This is not exact for an *elastic* beam-column element. However, the cubic shape is not exact, but should be sufficiently accurate for P-Δ effects in most building frames. Greater accuracy can be obtained by dividing a member into several elements.

For P-Δ effects in elements with rigid end zones, a further approximation is made. It is assumed that P-Δ effects are produced by a truss bar extending directly from node to node, and that the axial force in this bar is the axial force in the deformable part of the element. This approximation is shown in Figure E02.8. If this approximation is not acceptable, model the rigid end zones as separate elements, with large stiffnesses (but not too large -- say about 100 times stiffer than the deformable part of the element).

For dynamic analysis, the geometric stiffness is based on the static axial force only, not on the sum of the static and viscous (βK) forces. This is not strictly correct, but avoids rapid changes in axial force.

E02.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E02.1 through E02.8 for element behavior and properties.

E02.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NSTIF	Number of stiffness types (max. 40). See section E02.2.2).
6-10(I)		NECC	Number of rigid end zone types (max 15). See section E02.2.3.
11-15(I)		NSURF	Number of yield surfaces for cross sections (max. 40) See section E02.2.4.

E02.2.2 Stiffness Types

NSTIF lines, one 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)			Cross section moment of inertia.
46-50(R)			Flexural stiffness factor *kii*.
51-55(R)			Flexural stiffness factor *kjj*.
56-60(R)			Flexural stiffness factor *kij*.
61-70(R)			Shear area. Leave blank if shear deformations are to be ignored, or if shear deformation effects are included in the flexural stiffness factors.
71-75(R)			Poisson's ratio (used for computing shear modulus, and used only if shear area is nonzero).
76-80(R)			Moment overshoot tolerance. Default = very small.

E02.2.3 Rigid End Zone Types

NECC lines, one for each rigid end zone type.

Omit if there are no rigid zones. See Figure E02.1 for explanation.
All rigid zone projections are in the global X,Y directions, measured ***from*** the node ***to*** the element end.

Columns	Notes	Variable	Data
1-5(I)			Rigid end zone type number, in sequence beginning with 1.
6-15(R)			*Xi* = X projection at end I.
16-25(R)			*Xj* = X projection at end J.
26-35(R)			*Yi* = Y projection at end I.
36-45(R)			*Yj* = Y projection at end J.

E02.2.4 Yield Surface Types

NSURF lines, one for each yield surface type.

See Figure E02.3. Note the sign convention for yield moments.

Columns	Notes	Variable	Data
1-5(I)			Yield surface type number, in sequence beginning with 1.
6-10(R)			Yield surface shape code, as follows. 1 = beam type, without P-M interaction. 2 = steel I-beam type. 3 = reinforced concrete column type.
11-20(R)			Positive yield moment, *My+*.
21-30(R)			Negative yield moment, *My-*.
31-40(R)			Compression yield force, *Pyc*. Leave blank if shape code = 1.
41-50(R)			Tension yield force, *Pyt*. Leave blank if shape code = 1.
51-55(R)			M/My+ for point A (i.e., M at A as a proportion of *My+*). Leave blank if shape code = 1.
56-60(R)			P/Pyc for point A. Leave blank if shape code = 1.
61-65(R)			M/My- for point B. Leave blank if shape code = 1.
66-70(R)			P/Pyc for point B. Leave blank if shape code = 1.

E02.2.5 Element Generation Commands.

As many lines as needed, one line per command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number, or number of first element in asequentially numbered series of elements to be generated by thiscommand.
6-15(I)			Node number at end I.
16-25(I)			Node number at end J.
26-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Stiffness type number.
41-45(I)			Rigid end zone type number. Default = no rigid zone.
46-50(I)			Yield surface type number at end I.
51-55(I)			Yield surface type number at end J.

E02.3 INPUT DATA FOR *ELEMENTLOAD

E02.3.1 Load Sets

NLOD lines (see Element Group line of *ELEMENTLOAD section), one line per element load set.

See Figure E02.5 for sign convention.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
6-10(R)			Coordinate code. 0 = forces are in local (element) coordinates. 1 = forces are in global (structure)coordinates.
11-20(R)			Live load reduction factor.
21-30(R)			Force Pi.
31-40(R)			Force Vi.
41-50(R)			Moment Mi.
51-60(R)			Force Pj.
61-70(R)			Force Vj.
71-80(R)			Moment Mj.

E02.3.2 Loaded Elements and Load Set Scale Factors

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

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

E02.4 INTERPRETATION OF RESULTS

E02.4.1 SIGN CONVENTIONS

The sign conventions for element force and plastic hinge rotation output is shown in Figure E02.9. Accumulated plastic hinge rotations are calculated as shown in figure E02.10.

Note that the sign convention for moment results is different from that used to define the yield surfaces.

E02.4.2 EVENT CODES

In an event-to-event analysis, the element that governs the event is identified in the .ECH file, with a code that shows the type of event. The event types are as follows.

Code	Event type
11	New hinge at end i, positive yield.
12	New hinge at end i, negative yield.
10	Hinge unloads at end i.
21	New hinge at end j, positive yield.
22	New hinge at end j, negative yield.
20	Hinge unloads at end j.

E02.4.3 ENVELOPE OUTPUT (.OUT AND .EXX FILES)

To be added.

E02.4.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

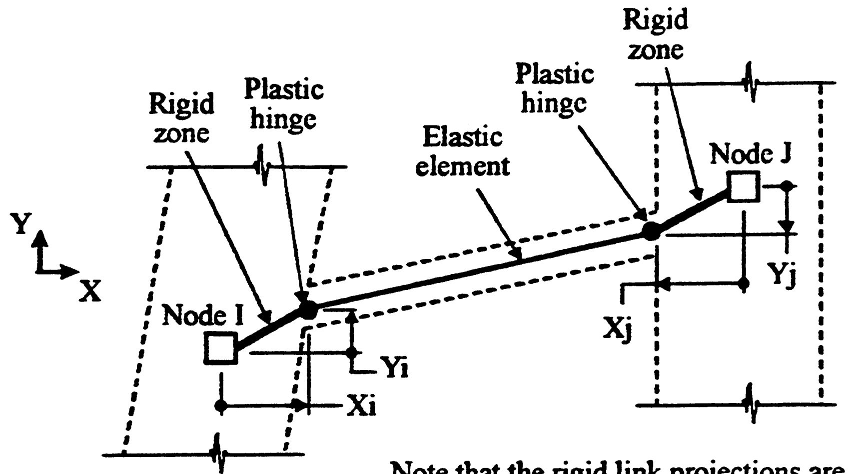
E02.4.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (16 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine SAVE02 in the ANAL02.FOR source code file.

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.

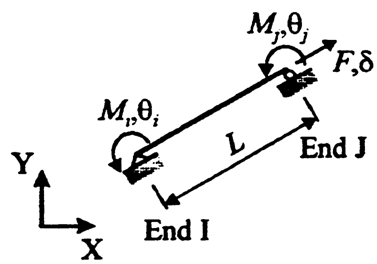
E02.4.6 USER OUTPUT (.USR FILE)

A sample subroutine (source code file USER02.FOR) is included to illustrate how the user output option might be used. This subroutine writes only to a formatted .USR file. The following items are output: current yield code, bending moment, shear force, axial force, plastic hinge rotation and accumulated plastic hinge rotations at element ends I and J.



Note that the rigid link projections are measured from the node to the element end. In this figure, Xj and Yj have negative values.

FIGURE E02.1 ELEMENT GEOMETRY



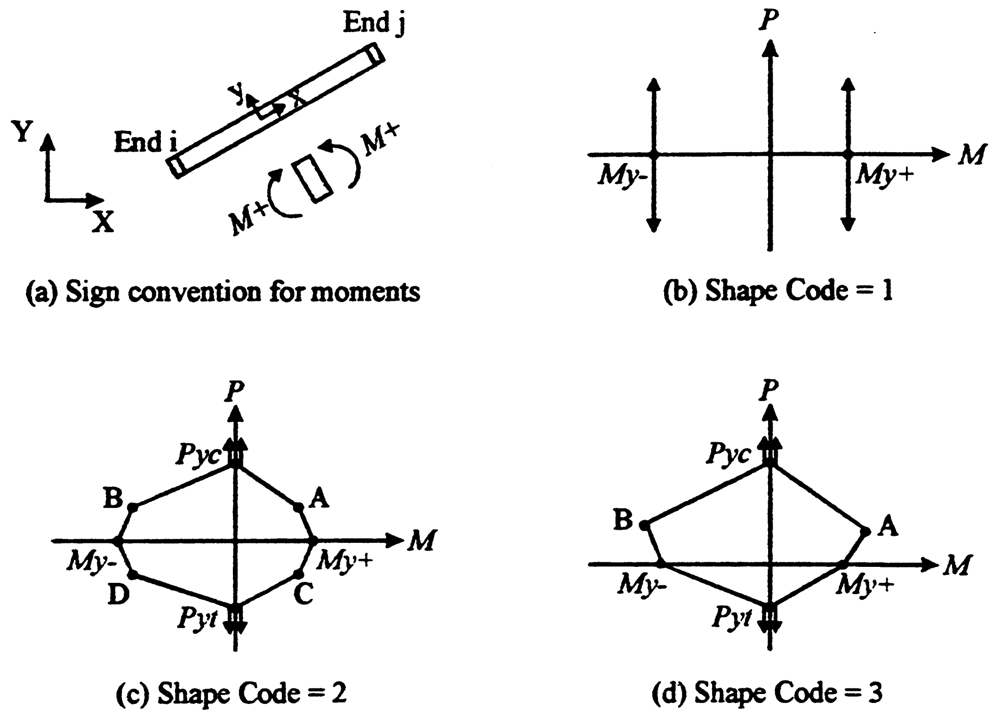
(a) Element actions and deformations

$$\begin{bmatrix} M_i \\ M_j \end{bmatrix} = \frac{EI}{L}\begin{bmatrix} k_{ii} & k_{ij} \\ k_{ij} & k_{jj} \end{bmatrix}\begin{bmatrix} \theta_i \\ \theta_j \end{bmatrix} \; ; \; F = \frac{EA}{L}\delta$$

I = reference moment of inertia
A = effective area

(b) Element stiffness relationships

FIGURE E02.2 ELASTIC ELEMENT STIFFNESS



(a) Sign convention for moments

(b) Shape Code = 1

(c) Shape Code = 2

(d) Shape Code = 3

For Shape Code = 2:
P/Pyt for C = P/Pyc for A and M/My+ for C = M/My+ for A
P/Pyt for D = P/Pyc for B and M/My- for D = M/My- for B

FIGURE E02.3 YIELD SURFACES, AND SIGN CONVENTION FOR YIELD SURFACE MOMENTS

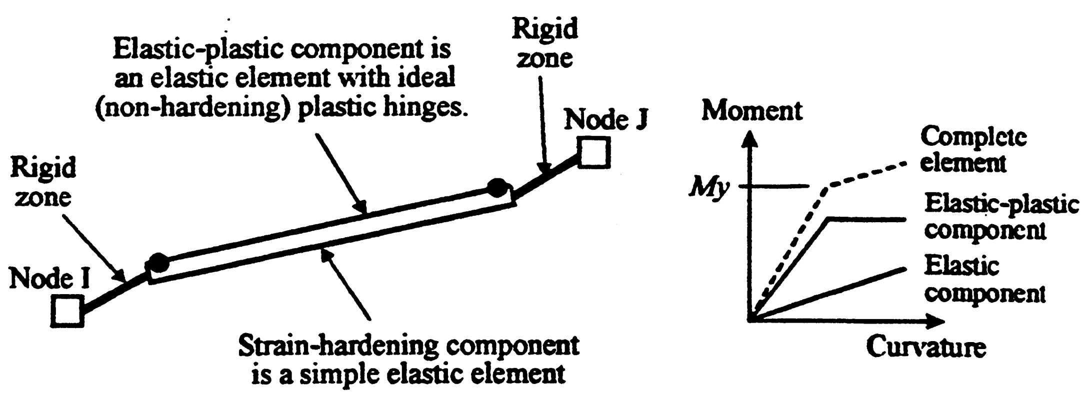
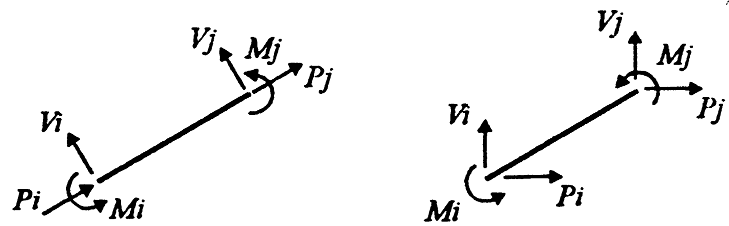


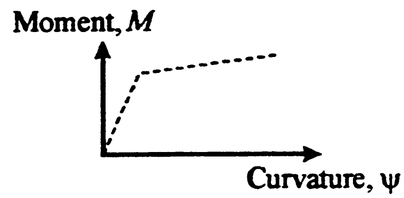
FIGURE E02.4 PARALLEL COMPONENTS



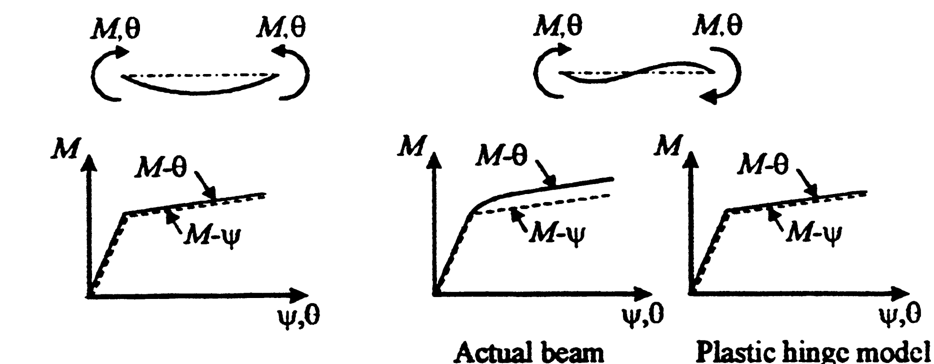
(a) Code = 0 (local)

(b) Code =1 (global)

FIGURE E02.5 FIXED END FORCES



(a) M-ψ Relationship



(b) M-ψ and M-θ relationships for uniform bending. Actual beam and plastic hinge model are the same.

(c) M-ψ and M-θ relationships for nonuniform bending. Actual beam and plastic hinge model are different.

FIGURE E02.6 MOMENT-CURVATURE AND MOMENT-ROTATION RELATIONSHIPS

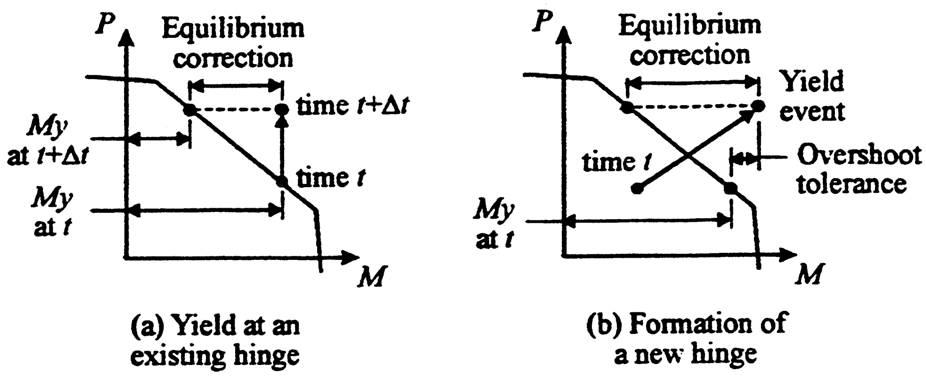


FIGURE E02.7 EQUILIBRIUM CORRECTIONS FOR YIELD SURFACE OVERSHOOT

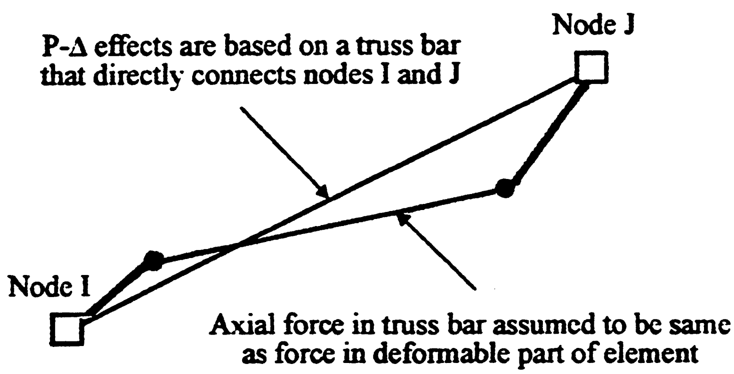
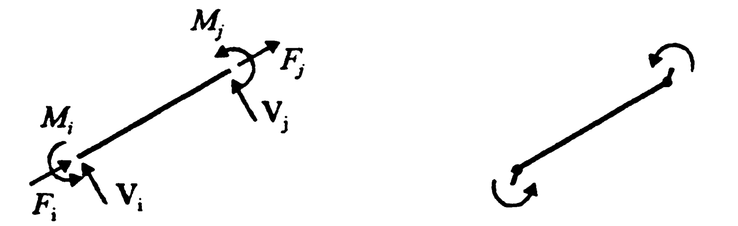


FIGURE E02.8 ASSUMPTION FOR P-Δ EFFECTS



(a) Element end forces (b) Plastic hinge rotations

FIGURE E02.9 SIGN CONVENTIONS FOR RESULTS

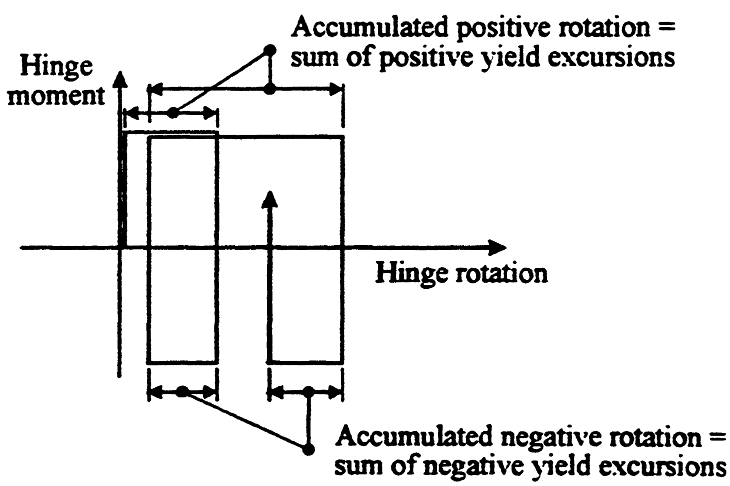


FIGURE E02.10 ACCUMULATED HINGE ROTATIONS

SIMPLE CONNECTION ELEMENT (TYPE 04) FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E04.1 PURPOSE, FEATURES AND LIMITATIONS

E04.1.1 PURPOSE

This is a simple inelastic element for modelling structural connections with rotational and/or translational flexibility.

E04.1.1 ELEMENT MODEL

The element connects two nodes which must have identical coordinates (i.e. this is a ***zero-length*** element). An element can connect either the rotational displacements of the nodes or the translational displacements. Positive actions (moments or forces) and deformations are shown in Figure E04.1. For a translational connection the element can connect horizontal displacements or vertical displacements, but not inclined displacements.

The element can be specified to behave elastically or inelastically, as shown in Figure E04.2. Complex modes of behavior can be obtained by placing two or more elements in parallel.

There is no provision for second order (P-Δ) effects, for element loads, or for initial forces.

E04.1.2 VISCOUS DAMPING

If βK damping is specified, a linear viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the initial (elastic) stiffness of the element.

The stiffness of the viscous element remains constant for any dynamic analysis, even if the basic element yields. However, the amount of viscous damping can be changed if the structure is in a static state. using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If the initial stiffness is large, as in a connection that is nearly rigid before it yields, the βK damping stiffness will be large, and hence large amounts of viscous energy may be absorbed after yield. This may not be a correct model, and it may be wise to specify zero β values for connection elements, and to use other element types to obtain viscous damping.

Some connections absorb energy by viscous action rather than by hysteresis. Such connections can be modelled by specifying a very small value of K and a very large value of β, so that βK is the required damping stiffness. The element behaves as a linear dashpot, with a constant damping stiffness. Nonlinear rate dependence can not be modelled.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E04.1.3 OVERSHOOT TOLERANCE

If event-to-event analysis is to be used, an overshoot tolerance must be specified. This is a tolerance on the element yield force or moment.

An "event" corresponds to a change in stiffness of an element, due to yield, inelastic unloading, or gap closure. If event-to-event analysis is used, the structure stiffness is reformed at each event. It is usually wise to use event-to-event analysis.

Consider the case where the event is element yield. If a zero value is input for the overshoot tolerance, the event factor is calculated so that the most critical element just yields. If a nonzero value is input, the event factor is chosen so that the force or moment in the element is its yield value plus the tolerance. That is, the element is allowed to "overshoot" beyond its nominal yield value. As a result, there will be an equilibrium unbalance at the event, and the analysis will be less accurate. However, the number of events (stiffness reformulations) may be reduced, because a number of elements may yield in a single analysis substep. In general, a small overshoot tolerance will give a more accurate analysis, but will require more execution time.

The amount of overshoot can be controlled in two ways, first by specifying an overshoot tolerance as part of the element properties, and second by specifying "event overshoot scale factors" with the "F" option in the *PARAMETERS input section. If no overshoot scale factors are input, these factors default to 1.0, and the overshoot tolerances input with the element properties are used. If overshoot scale factors are input, the overshoot tolerances are scaled by these factors. Separate overshoot scale factors can be input for static and dynamic analyses, and for each element group. The overshoot tolerances can thus be changed at any time, by changing the overshoot scale factors. One way to define overshoot tolerances is to specify a unit value with the element properties, and then control the actual value with overshoot scale factors.

E04.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E04.1 and E04.2 for element behavior and properties.

E04.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NPROP	No. of property types (min. 1, max. 40).

E04.2.2 Property Types

NPROP lines, one for each property type.

See Figure E04.1 for the sign convention. Note that positive deformation is the displacement of node J relative to (i.e., minus) the displacement of node I. Positive actions are in the same directions as positive deformations.

Columns	Notes	Variable	Data
1-5(I)			Property type number, in sequence beginning with 1.
6-15(R)			Initial stiffness, *k1* (for rotation, moment per radian).
16-25(R)			Strain hardening ratio, *k2/k1*. Must be < 1.
26-35(R)			Positive yield force or moment, *Fy+* or *My+*.
36-45(R)			Negative yield force or moment, *Fy-* or *My-*.
46-55(R)			Overshoot tolerance (force or moment value).
56-60(I)			Direction code, as follows. 1 = X translation. 2 = Y translation. 3 = Rotation.
61-65(I)			Elasticity code, as follows. 0 = Unload inelastically. 1 = Unload elastically. 2 = Unload inelastically with gap.

E04.2.3 Element Generation Commands.

As many lines as needed, one line per command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-15(I)			Node number at element end I.
16-25(I)			Node number at element end J.
26-35(I)			Node number increment for element generation. Default = 1
36-40(I)			Property type number. Default = same as preceding element.

E04.3 INTERPRETATION OF RESULTS

E04.3.1 SIGN CONVENTIONS

The sign conventions for element actions and deformations are shown in Figure E04.1. Accumulated plastic hinge rotations are calculated as shown in figure E04.3.

E04.3.2 EVENT CODES

In an event-to-event analysis, the element that governs the event is identified in the .ECH file, with a code that shows the type of event. The event types are as follows.

Code	Event type
1	Yielding.
-1	Unloading.
2	Gap opens.
-2	Gap closes.

E04.3.3 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E04.3.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

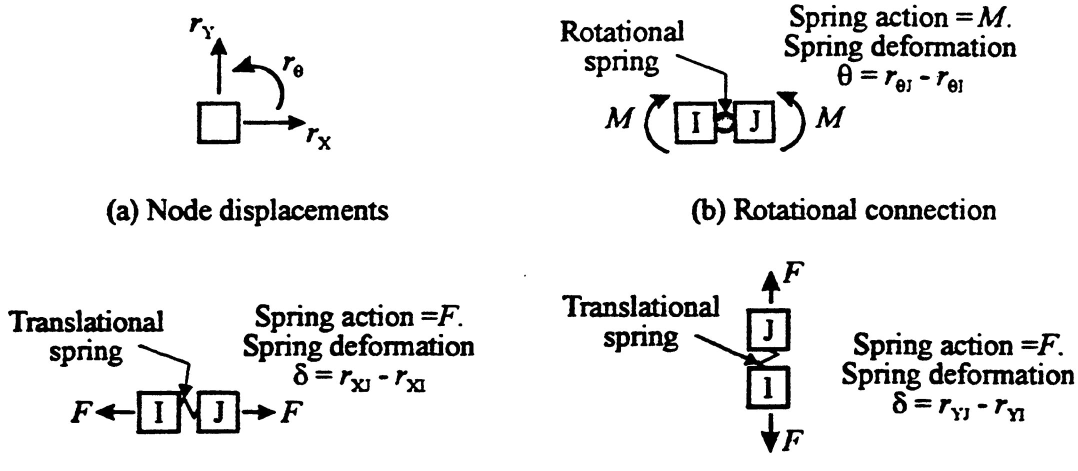
E04.3.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (9 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine RESP04 in the ANAL04.FOR source code file.

Item	Description
1	Static force or moment.
2	Viscous force or moment.
3	Total deformation.
4	Accumulated positive plastic deformation (sum of all positive excursions with yield code = 1).
5	Accumulated negative plastic deformation (sum of all negative excursions with yield code = 1).
6	Node number at end I.
7	Node number at end J.
8	Direction code (1 = X, 2 = Y, 3 = R).
9	Yield code (0 = not yielded; 1 = yielded; 2 = gap open).

E04.3.6 USER OUTPUT (.USR FILE)

There is no user output subroutine (source code file USER04.FOR) for this element.



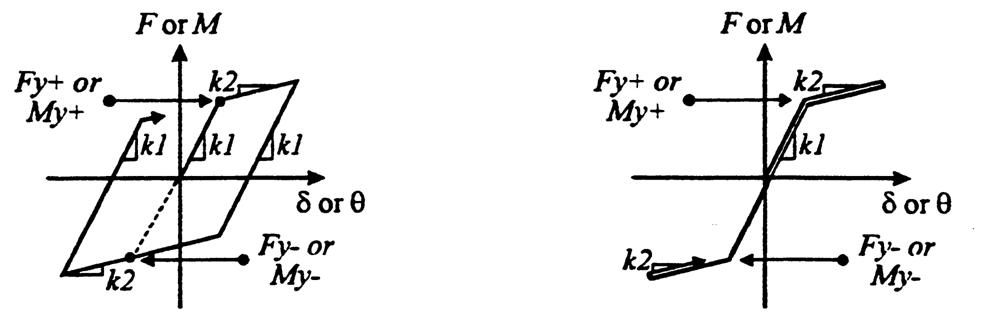
(a) Node displacements

(b) Rotational connection

(c) X translational connection

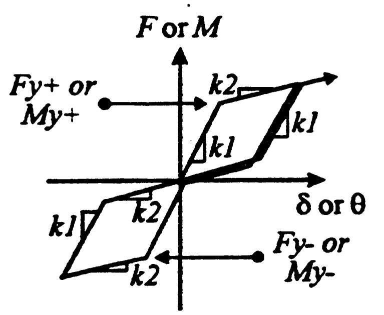
(d) Y translational connection

FIGURE E04.1 CONNECTION TYPES



(a) Inelastic unloading
(elasticity code = 0)

(b) Elastic unloading
(elasticity code = 1)



(c) Inelastic unloading with gap
(elasticity code = 2)

k1 = initial stiffness.
k2/k1 = strain hardening ratio.

Yield code is as follows:
= 0 when k = k1;
= 1 when k = k2 and element is yielding:
= 2 when k = k2 and gap is open.

FIGURE E04.2 BEHAVIOR OPTIONS

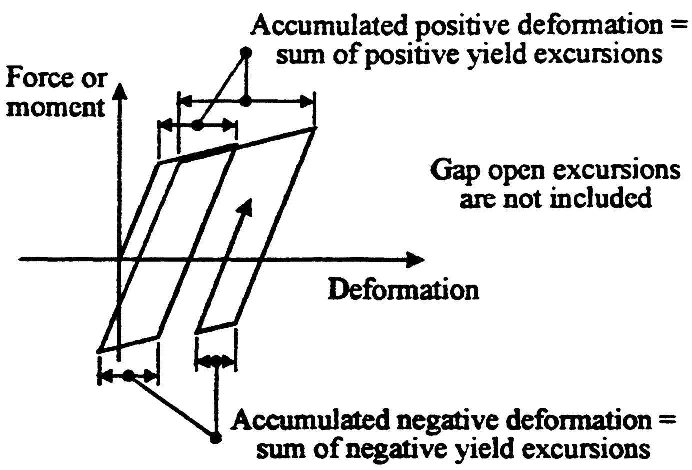


FIGURE E04.3 ACCUMULATED PLASTIC DEFORMATIONS

ELASTIC PANEL ELEMENT (TYPE 06)
FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E06.1 PURPOSE, FEATURES AND LIMITATIONS

E06.1.1 PURPOSE

This is a simple elastic (linear) element for modelling rectangular panels with extensional, bending and/or shear stiffness.

E06.1.2 ELEMENT MODEL

In analyses of buildings with structural panels it will often be reasonable to idealize each panel as a single elastic element in which the overall extensional, flexural, and shear stiffnesses of the panel are modeled. This element provides this type of idealization.

Figure E06.1 shows a large panel with an opening. In the vertical direction an effective centroidal axis can be found, such that an axial force applied along this axis produces no bending (and, correspondingly, a bending moment produces no axial deformation). A similar effective centroidal axis can be found in the horizontal direction. The extensional and flexural stiffnesses of the panel must be specified as effective EA and EI values along these axes, where E = Young's modulus, A = effective cross section area, and I = effective cross section moment of inertia. In addition, the shear stiffness of the panel must be specified. (NOTE - see warning 1.)

A panel is idealized as shown in Figure E06.2, with four nodes and eight displacement degrees of freedom. These provide for five deformation modes, as shown in Figure E06.3, plus three rigid body modes.

The five deformation modes are assumed to be uncoupled, with stiffnesses for vertical extension (effective vertical EA), vertical bending (effective vertical EI), horizontal extension (effective horizontal EA), horizontal bending (effective horizontal EI), and shear. The shear stiffness is defined in terms of shear strain and shear force per unit edge length (effective Gt, where G = shear modulus and t = effective panel thickness). These stiffnesses must be determined by experiment or by separate calculations, taking into account openings, stiffening ribs, thickness variations, etc.

E02.1.3 VISCOUS DAMPING

If βK damping is specified, a linear viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the stiffness of the element.

The stiffness of the viscous element remains constant for any dynamic analysis. However, the amount of viscous damping can be changed if the structure is in a static state, using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E06.1.4 WARNINGS ON USE

Warning 1: Incorrect Theory

The computer code is currently based on a theory that is correct only if the effective centroidal axes pass through the center of the panel. For other panel geometries the results will not be correct.

Warning 2: Panel Mass

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. Note, however, that the panel edges do not remain straight (in effect the element is a plane stress finite element with one-point shear quadrature). Hence, a panel modelled with several elements may be too flexible.

Warning 3: Rotational Displacements

Rotational displacements of the nodes are not restrained by panel elements, and may have to be restrained using *RESTRAINT commands.

E062 INPUT DATA FOR *ELEMENTGROUP

See Figures E06.1 through E06.3 for element geometry and properties.

E06.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NPROP	No. of property types (min. 1, max. 40).

E06.2.2 Property Types

NPROP lines, one for each property type.

Columns	Notes	Variable	Data
1-5(I)			Property type number, in sequence beginning with 1.
6-15(R)			Effective EA for vertical extension (EA of horizontal section).
16-25(R)			Effective EI for vertical bending (EI of horizontal section).
26-35(R)			Effective EA for horizontal extension (EA of vertical section).
36-45(R)			Effective EI for horizontal bending (EI of vertical section).
46-55(R)			Effective Gt for shear racking.
56-65(I)			Distance from panel centerline to effective vertical centroidal axis, plus or minus, as a proportion of panel width (i.e. range is -0.5 to +0.5, - to left, + to right). Default = 0.
66-75(I)			Distance from panel midheight to effective horizontal centroidal axis, plus or minus, as a proportion of panel height (i.e. range is -0.5 to +0.5, - down , + up). Default = 0.

E06.2.3 Element Generation Commands.

As many lines as needed, one line per command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-15(I)			Node I (top left).
16-25(I)			Node J (top right).
26-35(I)			Node K (bottom left).
36-45(I)			Node L (bottom right).
46-55(I)			Node number increment for element generation. Default = 1.
56-60(I)			Property type number. Default = same as preceding element.

E06.3 INTERPRETATION OF RESULTS

E06.3.1 SIGN CONVENTIONS

The sign conventions for element actions and deformations are shown in the following tables.

E06.3.2 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E06.3.3 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E06.3.4 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (22 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine RESP06in the ANAL06.FOR source code file.

Item	Description
1	Vertical axial force (tension +).
2	Vertical bending moment (tension at right edge +).
3	Horizontal axial force (tension +).
4	Horizontal bending moment (tension at bottom edge +).
5	Shear force per unit edge length (to right at top +).
6	Not used.
7	Vertical extension.
8	Rotation of top edge relative to bottom (contercloackwise +).
9	Horizontal extension.
10	Rotation of right edge relative to left (conterclockwise +).
11	Shear strain.
12	Rigid body rotation (counterclockwise +).
13-18	As for 1-6, but viscous damping forces and moments.
19	Node number at point I.
20	Node number at point J.
21	Node number at point K.
22	Node number at point L.

E06.3.5 USER OUTPUT (.USR FILE)

There is no user output subroutine (source code file USER06.FOR) for this element.

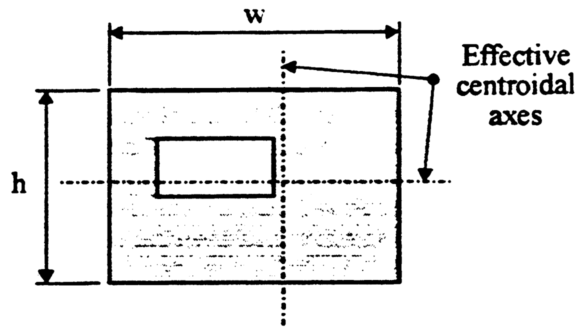


FIGURE E06.1 PANEL GEOMETRY

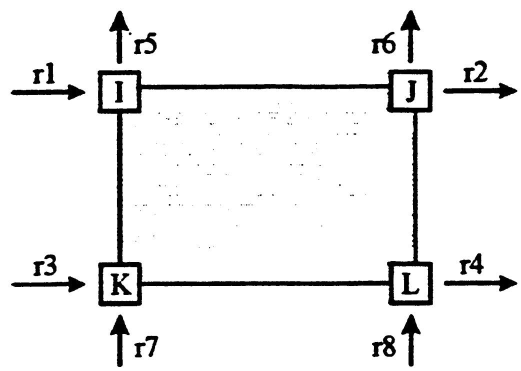


FIGURE E06.2 ELEMENT MODEL AND DEGREES OF FREEDOM

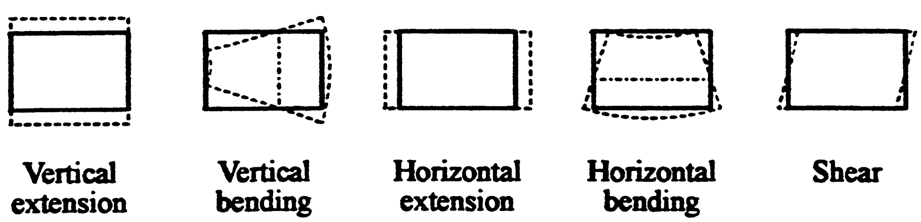


FIGURE E06.3 DEFORMATION MODES

COMPRESSION/TENSION LINK ELEMENT (TYPE 09) FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E09.1 PURPOSE, FEATURES AND LIMITATIONS

E09.1.1 PURPOSE

This is a simple inelastic bar element that resists only axial force. It can be used to model (a) a cable prestressed in tension, (b) a cable with initial slack, (c) a bearing element prestressed in compression, or (d) a bearing element with an initial gap.

E09.1.2 ELEMENT MODEL

The element has finite length and an arbitrary orientation. It resists axial force only, and can be specified to act in tension (tension force and extension are positive) or in compression (compression forceand shortening are positive). A tension element has finite stiffness in tension and goes slack in compression. A compression element has finite stiffness in compression and a gap opens in tension.

The force-deformation relationship is as shown in Fig. E09.1. Either one of two unloading paths, namely elastic or inelastic, can be specified. An element can be preloaded to a specified positive force if desired, or alternatively can be prestrained to a specified negative deformation. Complex modes of behavior can be obtained by placing two or more elements in parallel.

There is no provision for second order (P-Δ) effects or for element loads.

E09.1.3 VISCOUS DAMPING

If βK damping is specified, a linear viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the initial stiffness of the element.

Note: If an element acts in tension and has initial slack, or if it acts in compression and has an initial gap, the initial stiffness is zero, and hence βK is zero. If the element does not have initial slack or an initial gap, βK is nonzero.

The stiffness of the viscous element remains constant for any dynamic analysis, even if the basic element yields. However, the amount of viscous damping can be changed if the structure is in a static state, using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E09.1.4 OVERSHOOT TOLERANCE

If event-to-event analysis is to be used, an overshoot tolerance must be specified. This is a tolerance on the element yield force.

An "event" corresponds to a change in stiffness of an element, due to yield, inelastic unloading, gap closure, etc.. If event-to-event analysis is used, the structure stiffness is reformed at each event. It is usually wise to use event-to-event analysis.

Consider the case where the event is element yield. If a zero value is input for the overshoot tolerance, the event factor is calculated so that the most critical element just yields. If a nonzero value is input, the event factor is chosen so that the force or moment in the element is its yield value plus the tolerance. That is, the element is allowed to "overshoot" beyond its nominal yield value. As a result, there will be an equilibrium unbalance at the event, and the analysis will be less accurate. However, the number of events (stiffness reformulations) may be reduced, because a number of elements may yield in a single analysis substep. In general, a small overshoot tolerance will give a more accurate analysis, but will require more execution time.

The amount of overshoot can be controlled in two ways, first by specifying an overshoot tolerance as part of the element properties, and second by specifying "event overshoot scale factors" with the "F" option in the *PARAMETERS input section. If no overshoot scale factors are input, these factors default to 1.0, and the overshoot tolerances input with the element properties are used. If overshoot scale factors are input, the overshoot tolerances are scaled by these factors. Separate overshoot scale factors can be input for static and dynamic analyses, and for each element group. The overshoot tolerances can thus be changed at any time, by changing the overshoot scale factors. One way to define overshoot tolerances is to specify a unit value with the element properties, and then control the actual value with overshoot scale factors.

E09.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E09.1 and E09.2 for element geometry and properties.

E09.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NPROP	No. of property types (min. 1, max. 40). See section E09.2.2.

E09.2.2 Property Types

NPROP lines, one for each property type. See Figure E09.2.

Columns	Notes	Variable	Data
1-5(I)			Property type number, in sequence beginning with 1.
10(I)			Property code, as follows. +1: Acts in tension, unloads inelastically. +2: Acts in tension, unloads elastically. -1: Acts in compression, unloads inelastically. -2: Acts in compression, unloads elastically.
11-20(I)			Displacement limit u1.
21-30(R)			Displacement limit u2.
31-40(R)			Stiffness k1.
41-50(R)			Stiffness k2.
51-60(R)			Stiffness k3.
61-70(R)			Unloading stiffness k4. Default = k1.
71-80(R)			Force overshoot tolerance..

E09.2.3 Element Generation Commands.

As many lines as needed, one line per command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-15(I)			Node number at element end I.
16-25(I)			Node number at element end J.
26-35(I)			Node number increment for element generation. Default = 1
36-40(I)			Property type number. Default = same as preceding element.
41-50(R)			Initial force or deformation. < 0.0 = initial deformation (amount of slack if a tension element, size of gap if a compression element). > 0.0 = initial force (tension if a tension element, compression if a compression element).

E09.3 INTERPRETATION OF RESULTS

E09.3.1 SIGN CONVENTION

For a tension element, tension and extension are positive. For a compression element, compression and shortening are positive.

E09.3.2 EVENT CODES

In an event-to-event analysis, the element that governs the event is identified in the .ECH file, with a code that shows the type of event. The event types are as follows.

Code	Event type
1	Stiffness change, increasing force.
-1	Stiffness change, decreasing force.
2	Goes slack (if a tension element) or gap opens (if a compression element).
-2	Re-tightens or gap closes.

E09.3.3 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E09.3.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E09.3.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (7 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine RESP09 in the ANAL09.FOR source code file.

Item	Description
1	Static force.
2	Viscous force.
3	Deformation.
4	Accumulated inelastic deformation (sum of all positive :excursions on lines 2 and 3 if element is inelastic)
5	Node number at end I.
6	Node number at end J.
7	Line number (0,1,2,3 or 4).

E09.3.6 USER OUTPUT (.USR FILE)

There is no user output subroutine (source code file USER09.FOR) for this element.

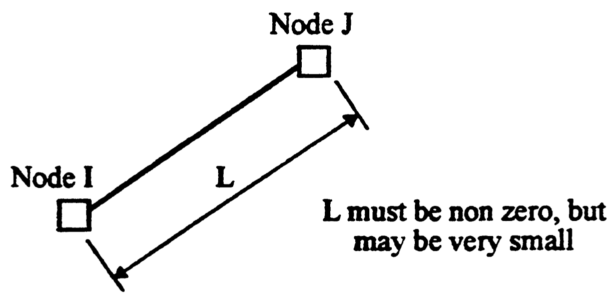
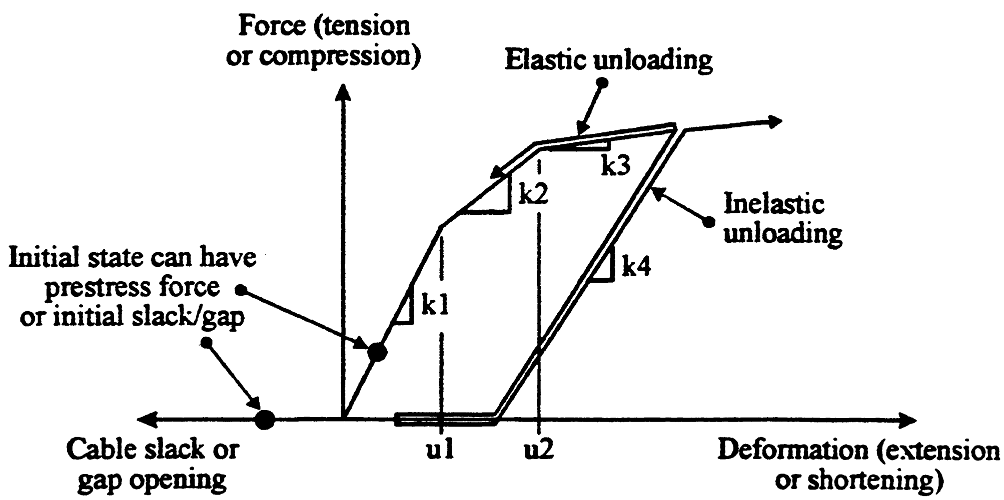


FIGURE E09.1 ELEMENT GEOMETRY



Note: k2, k3 may be > k1. k4 should be >= max(k1,k2,k3).

FIGURE E09.2 ELEMENT PROPERTIES

FIBER BEAM-COLUMN ELEMENT (TYPE 15) FOR DRAIN-2DX

VERSION 1.10
DECEMBER 1993

ELEMENT DESCRIPTION AND USER GUIDE

E15.1 PURPOSE, FEATURES AND LIMITATIONS

E15.1.1 PURPOSE

This is an inelastic element for modelling beams and beam-columns. It can be used to model steel, reinforced concrete, or composite steel-concrete members. The element has a wide variety of options, and can be used in many different ways. It can be used to model a single cross section of a beam or column, a single beam or column member, or beams and columns in a larger structure.

The material models account for yield of steel, including strain hardening, for cracking and crushing of concrete, including post-crushing strength loss if desired, and for "tension stiffening" of concrete. Flexible nonlinear connections can be considered at the element ends, to model semi-rigid steel connections or bond slip effects in concrete joints.

Shear deformations can be included, but the behavior in shear is assumed to be elastic, and it is not currently possible to consider nonlinear shear effects.

E15.1.2 ELEMENT MODEL

Figure E15.1 shows the element model. The deformable part of the element is divided into a number of segments. The behavior is monitored at the center cross section (or "slice") in each segment. The cross section properties are assumed to be constant within each segment, but can vary from segment to segment.

Each cross section is either elastic or is divided into a number of fibers. The fibers can have nonlinear longitudinal stress-strain relationships of concrete or steel type. Figures E15.2 and E15.3 show the properties for concrete and steel materials. The use of fibers to model cross sections accounts rationally for P-M interaction (axial force plus bending).

The element model within the length of the deformable region is of "distributed plasticity" type, accounting for the spread of inelastic behavior both over the cross sections and along the member length. This is in contrast to a "lumped plasticity" model, where the inelastic behavior is concentrated in zero-length plastic "hinges".

The accuracy of the model increases with the number of segments along the element length, the number of fibers in each cross section, and the number of points on each material stress-strain curve. However, the computational cost also increases.

Connection hinges can be specified at the element ends, to model deformations that occur in beam-to-column or column-to-footing connections. These are zero-length fiber hinges. For concrete members the fiber properties can be chosen to model effects such as bond slip within the connection and crack opening at the connection face. For steel members the fiber properties can be chosen to model the deformations of semi-rigid framed connections. Fiber properties of "pullout" and "gap" types can be specified, as shown in Figures E15.4 through E15.6. Pullout fibers can be used to model behavior of a variety of types, particularly including reinforcing bar pullout with bond slip. Gap fibers are intended mainly to model gap opening effects at column faces.

Plastic hinges can be modelled as a special case, using either connection hinges or short segments.

The element is assumed to be elastic in shear.

P-Δ effects can be considered.

E15.1.3 ASSUMPTIONS AND LIMITATIONS

It is important to recognize that the element is based on many simplifying assumptions, and that it does not capture a number of potentially important aspects of beam-column behavior. This is particularly true for reinforced concrete members. The main assumptions and limitations are as follows.

(1) Plane sections are assumed to remain plane. This means that within the body of the element, bond slip is assumed to be zero for reinforced concrete members, and full composite action is assumed for composite steel-concrete members (the connection hinges, however, can account for bond slip in connections).

(2) Shear deformations can be included, but the shear behavior is assumed to be elastic, based on a specified shear modulus and effective shear area. An extension of the element that includes inelastic shear deformations, with P-M-V interaction, is being developed.

(3) There is currently no provision for prestressing or for initial stresses. Hence, prestressed concrete members can not be considered, and composite steel-concrete members are assumed to be fully shored until the concrete has hardened. An extension of the element to allow initial stresses is being developed.

(4) The model assumes constant slice properties over each segment, based on the properties of the monitored slice at the segment center. The computed behavior of the element can be sensitive to the number of segments that are specified, and to the segment lengths. In finite element terms this is a "low order" element. A higher order element has been considered, with linear property variation over each segment, based on monitored slices at the segment ends. The lower order element was chosen mainly because it has more stable behavior if negative material moduli are specified. An option to allow the higher order assumption may be added in the future, with a recommendation that it be used only if all material moduli are positive.

(5) There is no provision for element loads (i.e., loads applied within the length of an element, rather than at the nodes).

E15.1.4 VISCOUS DAMPING

If βK damping is specified, a linear viscous damping element is added in parallel with the basic element. The viscous element stiffness is β multiplied by the initial (elastic) stiffness of the element. This stiffness assumes that all fibers of steel type are elastic, and that all fibers of concrete and gap type are uncracked.

The stiffness of the viscous element remains constant for any dynamic analysis, even as the basic element cracks and yields. However, the amount of viscous damping can be changed if the structure is in a static state, using the "VS" and/or "VE" options in the *PARAMETERS input section. These allow the β values to be changed for subsequent dynamic analyses.

If mode shapes and frequencies are calculated (*MODE analysis), the proportions of critical damping implied by the current β values are shown for each mode in the .OUT file. These proportions should be checked, to make sure that they are reasonable.

The amounts of energy absorbed by the viscous damping elements in each element group are shown in the .SLO (solution log) file. These values should be checked to make sure that they are reasonable. The .SLO file should also be checked to make sure that there is an energy balance. If there is a large difference between the external and internal energies, the analysis results may be inaccurate.

E02.1.5 OVERSHOOT TOLERANCES

If event-to-event analysis is to be used, overshoot tolerances must be specified. These are input as tolerances on the yield strengths of the fibers.

An "event" corresponds to a change in stiffness of an element, due to yield, inelastic unloading, etc. of a fiber. If event-to-event analysis is used, the structure stiffness is reformed at each event. It is usually wise to use event-to-event analysis.

Consider the case where the event is fiber yield. If a zero value is input for the overshoot tolerance, the event factor is calculated so that the most critical fiber in the most critical element just yields. If a nonzero value is input, the event factor is chosen so that the stress in the critical fiber is its yield value plus the tolerance. That is, the fiber is allowed to "overshoot" beyond its nominal yield value. As a result, there will be an equilibrium unbalance at the event, and the analysis will be less accurate. However, the number of events (stiffness reformulations) may be reduced, because a number of fibers, and possibly a number of elements, may yield in a single analysis substep. In general, a small overshoot tolerance will give a more accurate analysis, but will require more execution time. This is especially true if the elements have large numbers of fibers.

The amount of overshoot can be controlled in two ways, first by specifying an overshoot tolerance as part of the element properties, and second by specifying "event overshoot scale factors" with the "F" option in the *PARAMETERS input section. If no overshoot scale factors are input, these factors default to 1.0, and the overshoot tolerances input with the element properties are used. If overshoot scale factors are input. the overshoot tolerances are scaled by these factors (the same factor for all fibers in any element group). Separate overshoot scale factors can be input for static and dynamic analyses, and for each element group. The overshoot tolerances can thus be changed at any time, by changing the overshoot scale factors. One way to define overshoot tolerances is to specify unit values with the element properties, and then control the actual values with overshoot scale factors.

E15.1.6 WARNINGS ON USE

Warning E15.1. Element Complexity

This is a complex element, and it can be used in many different ways. At present there is not much available experience, and guidelines for effective use of the element have not yet been developed. Since it is much more difficult to model nonlinear members than linear members, users must be cautious.

Before using the element as part of a large analysis model, it is strongly recommended that users proceed as follows.

(1) Create models with single segments, and hence single cross sections, and study the effects of using (a) different numbers and locations of fibers, and (b) different material stress-strain curves. Analyze the models to calculate cross section behavior (e.g., moment-curvature relationships and ultimate moments under different axial forces). Compare the results against the expected behavior and against available experimental results. Do not proceed until "correct" cross section behavior has been obtained.

(2) Create models of single members (i.e., single beams and columns, possibly but not neccssarily using single elements), and study the effects of using different segment lengths and numbers of segments. If connection hinges are to be used, also study the effects of the number of connection fibers and the fiber properties. This is particularly important because the connection hinges are substantially empirical, and hence must be carefully calibrated. Analyze the models to calculate the behavior of the

member under end loads. Compare the results against the expected behavior and against available experimental results. Do not proceed until "correct" member behavior has been obtained.

(3) The goal of steps (1) and (2) is to obtain member models that are as simple as possible, while providing sufficient accuracy for practical purposes. Proceed with modelling of the complete structure only when you are satisfied that the member models are sound..

Pullout fibers in connection hinges can have complex degrading behavior. If degrading behavior is to be specified, it is wise to set up a very simple model with a connection hinge, and to subject it to imposed cyclic deformations to confirm that the required degrading behavior is obtained.

Warning E15.2. Numbers Of Fibers And Segments

There are no limits specified for the number of fibers in a cross section or the number of segments in an element. There are, however, limits on the maximum memory and disk storage that can be occupied by the data for any one element. If these limits are exceeded, an error message will be printed in the .ECH file, and the program will not execute. The limits are set for a maximum of roughly 200 total fibers in any one element. There are also limits on the number of points allowed on the stress-strain curve for any steel or concrete material. These limits can all be increased, if necessary, by changing a few lines in the source code.

It may be appropriate to specify a large number of fibers for a small analysis model consisting of one or two elements. However, if several such elements are specified as part of a large model, the execution time may be long, especially for a dynamic analysis. The recommended way of using this element is as follows.

(1) Specify only a small number of fiber slices, with small numbers of fibers, for elements in large analysis models, where the goal is usually to calculate structure displacements. It is not usually necessary to use a refined model to calculate displacements.

(2) Specify large numbers of fibers only for small models, where the goal is to obtain detailed information on damage.

Warning E15.3. Grouping Of Elements

In any element group, the amount of storage allocated for each element is based on the largest element in the group. Hence, if an element group consists of several elements with a small number of fibers plus one or two elements with a large number, there will be a lot of wasted storage. To avoid this, place the elements with large numbers of fibers in one group, and the elements with fewer fibers in a separate group..

Warning E15.4. Negative Material Moduli

The strengths of concrete fibers can be specified to decrease after a maximum strength is reached. If this is done, the material tangent modulus becomes negative, and it is possible for the stiffness of a slice or a complete element also to become negative. If this happens, the element, and possibly the structure, becomes unstable, and it may not be possible to obtain a solution. Difficulties can arise at the element level and/or at the structure level.

At the element level, the forces in the element must redistribute if the element becomes unstable. If the strength loss is rapid (i.e., if the tangent moduli for a number of fibers have large negative values), the solution may fail to converge. If so, a message will be printed in the .ECH file and execution will stop. If this happens, try one or both of the following.

(1) Specify a less rapid rate of strength loss for the concrete materials. This is the recommended method.

(2) Turn on the "anti-flip-flop" logic by using the *PARAMETERS,E input data option (set the first integer parameter for the element group to 1). This can help, but is not foolproof.

At the structure level, if the structure becomes unstable and a ***load controlled*** static analysis has been specified, then no solution is possible and the analysis will flip-flop (yield and unload at the same point in successive analysis substeps). To obtain a solution, a ***displacement controlled*** analysis must be used (this is usually a good idea anyway). By its nature, a ***dynamic analysis*** is essentially load controlled. Hence, flip-flopping can occur if the effective structure stiffness becomes negative. However, the effective stiffness depends on the time step, and it is possible that a solution can be obtained by specifying a smaller time step.

To avoid wasting computer time if the analysis flip-flops or otherwise fails to converge, ***be sure to specify limits on the number of flip-flops and the number of events in any load or time step*** (see MAXEV and MAXFP in the *STAT input data, and MAXEV in the *PARAMETERS,DC input data). If these limits are exceeded the analysis will end reasonably gracefully.

Also, for dynamic analysis it is usually wise to perform the analysis in a number of time segments, and to examine the results at the end of each segment before resuming, to ensure that the analysis is proceeding without error. Be sure to check both the energy balance and the unbalanced loads (in the .SLO file).

E15.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E15.1 through E15.6 for element geometry and fiber properties. See Chapter E15.1 for a description of the element features and limitations, and for warnings on use of the element.

E15.2.1. Control Information.

One line

Columns	Notes	Variable	Data
1-5(I)		NCMAT	No. of concrete material types for fiber cross sections (may be 0). See Section E15.2.2(a).
6-10(I)		NSMAT	No. of steel material types for fiber cross sections (may be 0). See Section E15.2.2(b).
11-15(I)		NFSEC	No. of fiber cross section types (may be 0). See Section E15.2.3(a).
16-20(I)		NESEC	No. of elastic cross section types (may be 0). See Section E15.2.3(b).
21-25(I)		NPMAT	No. of pullout property types for connection hinges (may be 0). See Section E15.2.4(a).
26-30(I)		NGMAT	No. of gap property types for connection hinges (may be 0). See Section E15.2.4(b).
31-35(I)		NCHIN	No. of connection hinge types (may be 0). See Section E15.2.5.
36-40(I)		NRIGZ	No. of rigid end zone types (may be 0). See Section E15.2.6.
41-45(I)		NETYP	No. of element geometry types (must be >0). See Section E15.2.7.

E15.2.2(a). Concrete Material Properties.

NCMAT sets of lines. Omit if NCMAT = 0.
Each set consists of one control line plus one line per stress-strain point. See Figure E15.2
Concrete material types are numbered in input sequence.

E15.2.2(a)(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NCOM	No. of stress-strain points for compression (max. 5, may be 0).
6-10(I)		NTEN	No. of stress-strain points for tension (max. 2, may be 0).
11-20(R)			Unloading factor, FU (0 <= FU <= 1). FU = 0 means no stiffness degradation on unloading.
21-30(R)			Stress overshoot tolerance (for event factor calculation).

E15.2.2(a)(ii). Stress-Strain Points for Compression: NCOM lines. Omit if NCOM = 0.

Columns	Notes	Variable	Data
1-10(R)			Stress (S1C, S2C, etc.). Must be >0.
11-20(R)			Corresponding strain (E1C, E2C, etc.). Must be > 0.

E15.2.2(a)(iii). Stress-Strain Points for Tension: NTEN lines. Omit if NTEN = 0.

Columns	Notes	Variable	Data
1-10(R)			Stress (S1T, etc.). Must be > 0.
11-20(R)			Corresponding strain (E1T, etc.). Must be > 0.

E15.2.2(b). Steel Material Properties.

NSMAT sets of lines. Omit if NSMAT = 0.
Each set consists of one control line plus one line per stress-strain point. See Figure E15.3
Steel material types are numbered in input sequence.

E15.2.2(b)(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NPTS	No. of stress-strain points (min. 1, max. 5).
6-15(R)			Stress overshoot tolerance (for event factor calculation).

E15.2.2(b)(ii). Stress-Strain Points: NPTS lines.

Columns	Notes	Variable	Data
1-10(R)			Stress (S1, S2, etc.). Must be >0.
11-20(R)			Corresponding strain (E1, E2, etc.). Must be > 0.

E15.2.3(a). Fiber Cross Section Types.

NFSEC sets. Omit if NFSEC = 0.
Each set consists of one control line plus one line per fiber.
Fiber section types are numbered in input sequence.

The element local y axis should be a principal axes of the cross section, otherwise this is not a 2D element. The cross section centroid need ***not*** be at the x axis.

E15.2.3(a)(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NFIBS	No. of fibers (min. 2, not both with same y coordinate).
6-15(R)			Shear A' or GA'. Default = no shear deformation.
16-35(R)			Shear modulus, G. Default = 1 (i.e., GA' is specified directly).

E15.2.3(a)(ii). Fibers: NFIBS lines, one per fiber, in any order.

Columns	Notes	Variable	Data
1-10(R)			Fiber y coordinate.
11-20(R)			Fiber Area.
23(C)			"C" if concrete, "S" if steel.
24-25(I)			Material number of this type. Example: "S03" in columns 23-25 = steel material type 3.

E15.2.3(b). Elastic Cross Section Types.

NESEC lines. Omit if NESEC = 0.
Elastic section types are numbered in input sequence.

The element local y axis should be a principal axes of the cross section, otherwise this is not a 2D element. The cross section centroid ***must*** be at the x axis.

Columns	Notes	Variable	Data
1-10(R)			Flexural I or EI..
11-20(R)			Axial A or EA.
21-30(R)			Shear A' or GA'. Default = no shear deformation.
31-40(R)			Young's modulus, E. Default = 1 (i.e., EA and EI are specified directly).
41-50(R)			Shear modulus, G. Default = 1 (i.e., GA' is specified directly).

E15.2.4(a). Pullout Properties for Connection Hinge Fibers

NPMAT lines or pairs of lines. Omit if NPMAT = 0.
One line for a simple non-degrading material. Two lines for a material with stiffness degradation, strength degradation, and/or pinching behavior.
Pullout property types are numbered in input sequence.

E15.2.4(a)(i). Basic Properties

See Figure E15.4. Note that moduli are in terms of stress and *displacement*, not stress and *strain*.

Columns	Notes	Variable	Data
1-10(R)			Modulus K1. Must be >0.
11-20(R)			Modulus K2. Must be < K1 and >0.
21-30(R)			Modulus K3. Must be < K2 and >0.
31-40(R)			Yield stress S1T in tension Must be > 0..
41-50(R)			Yield stress S2T in tension. Must be > S1T.
51-60(R)			Yield stress S1C in compression. Must be > 0.
61-70(R)			Yield stress S2C in compression. Must be > S1C.
71-75(R)			Stress overshoot tolerance (for event factor calculation).
80(I)		IDGD	Degradation indicator (blank, 0 or 1). If blank or 0, material does not degrade. Omit line E15.2.4(a)(ii). If 1, material degrades. Include line E15.2.4(a)(ii).

E15.2.4(a)(ii). Degradation Parameters

Omit if degradation indicator, IDGD, is blank or 0. See Figure E15.5.

Columns	Notes	Variable	Data
1-10(R)			Stiffness degradation factor, SDF. Between 0 and 1, 0 = no degradation.
11-20(R)			Tension strength degradation factor, STDF. Between 0 and 1, 0 = no degradation. Saturated strain in compression must also be specified.
21-30(R)			Compression strength degradation factor, SCDF. Between 0 and 1, 0 = no degradation. Saturated strain in tension must also be specified.
31-40(R)			Saturated displacement in compression, SC (accumulated plastic displacement in compression for full strength loss in tension). Must be > 0.
41-50(R)			Saturated displacement in tension, ST (accumulated plastic displacement in tension for full strength loss in compression). Must be > 0.
51-60(R)			Pinch factor, PF. Between 0 and 1, 0 = no pinching.
61-70(R)			Pinch strength factor, PSF. Between 0 and 1, 0 = no strength loss. Omit if pinch factor is 0.
71-80(R)			Pinch plateau factor, PPF. Between 0 and 1, 0 = no plateau. Omit if pinch factor is 0.

E15.2.4(b). Gap Properties for Connection Hinge Fibers

NGMAT lines. Omit if NGMAT = 0.
Gap property types are numbered in input sequence.

See Figure E15.6. Note that moduli are in terms of stress and *displacement*, not stress and *strain*.

Columns	Notes	Variable	Data
1-10(R)			Crushing stress SC1. Must be > 0.
11-20(R)			Crushing stress SC2. Must be > SC1.
21-30(R)			Modulus K1. Must be > 0.
31-40(R)			Modulus K2. Must be > 0 and < K1.
41-50(R)			Modulus K3. Must be > 0 and < K2
51-60(R)			Unloading factor, FU (0 <= FU <= 1). FU = 0 means no stiffness degradation on unloading.
61-70(R)			Stress overshoot tolerance (for event factor calculation).

E15.2.5. Connection Hinge Types.

NCHIN sets. Omit if NCHIN = 0.
Each set consists of one control line plus one line per fiber.
Connection hinge types are numbered in input sequence.

The element local y axis should be a principal axes of the hinge. otherwise this is not a 2D element. The effective hinge centroid need ***not*** be at the x axis.

E15.2.5(i). Control Line.

Columns	Notes	Variables	Data
1-5(I)		NFIBH	No. of fibers (min. 2, not both with same y coordinate).

E15.2.5(ii). Fibers. NFIBH lines, one per fiber, in any order

Columns	Notes	Variable	Data
1-10(R)			Fiber y coordinate.
11-20(R)			Fiber Area.
23(C)			"P" if a bar pullout fiber, "G" if gap fiber.
24-25(I)			Property number of this type. E.g., "P03" in columns 23-25 = pullout property type 3.

E15.2.6. Rigid End Zone Types.

NRIGZ lines. Omit if NRIGZ = 0.
Rigid zone types are numbered in input sequence.
See Section E15.2.7(i) for how directions are assigned to rigid zones.

Columns	Notes	Variable	Data
1-10(R)			Global X projection of rigid zone.
11-20(R)			Global Y projection of rigid zone.

E15.2.7. Element Geometry Types.

NETYP sets. Each set has one control line plus one line per segment.
Element geometry types are numbered in input sequence.

E15.2.7(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NSEG	Number of segments.
6-10(I)			Type number forconnection hinge at end i. Default = none.
11-15(I)			Type number for connection hinge at end j. Default = none.
16-20(I)			Rigid zone type no. at end i. Default = none. If +, projections are from node to element end. If - , projections are from element end to node.
21-25(I)			Rigid zone type no. at end j. Default = none. If +, projections are from node to element end. If - , projections are from element end to node.

E15.2.7(ii). Segments: NSEG lines, in sequence from end i to end j.

Columns	Notes	Variable	Data
1-10(R)			Segment length, as a *proportion of element length*. Total for all segments must sum to 1.
13(C)			"F" if a fiber section, "E" if an elastic section.
14-15(I)			Fiber or elastic section type number. E.g., "F03" in columns 13-15 = fiber section type 3.

E15.2.8. Element Generation Commands.

One line for each command.

Elements must be numbered in sequence beginning with 1.
Lines for the first and last elements must be provided. Intermediate elements may be generated.

Columns	Notes	Variable	Data
1-5(I)			Element number, or number of first element in a sequentially numbered series of elements to be generated by this command.
6-15(I)			Node number at element end i.
16-25(I)			Node number at element end j.
26-35(I)			Node number increment for element generation. Default = 1.
36-40(I)			Element geometry type number. Default = same as preceding element.

E15.3 INTERPRETATION OF RESULTS

E15.3.1 SIGN CONVENTIONS

The sign conventions for element end forces and moments are shown in Figure E15.7(a) The sign conventions for slice and hinge axial forces and bending moments are shown in Figure E15.7(b) Note that the conventions for element end moments and slice/hinge moments are different. The sign convention for fiber stresses and strains is tension positive.

E15.3.2 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E15.3.3 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E15.3.4 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (34 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine SAVE15 in the ANAL15.FOR source code file.

Item	Description
1	Node I.
2	Node J.
3-10	Output list for up to 8 hinges and fiber slices (hinges, if any, followed by fiber slices): -1 = connection hinge at end i. -2 = connection hinge at end j. n = fiber slice in segment n. 0 = no more in list.
11	Static axial force .
12	Static bending moment at end i.
13	Static bending moment at end j.
14	Static shear force.
15-18	Viscous values.
19	Slice axial strain or hinge axial deformation for first slice or hinge in list.
20	Slice curvature or hinge rotation for first slice or hinge in list.
21-22	Ditto for second slice or hinge in list.
23-34	Ditto for up to 8 slices or hinges.

E15.3.5 USER OUTPUT (.USR FILE)

There is no user output subroutine (source code file USER15.FOR) for this element.

Since there are many results items that could be output, and only a few of these are output in the .OUT file, it is probable that user output will be useful for this element type.

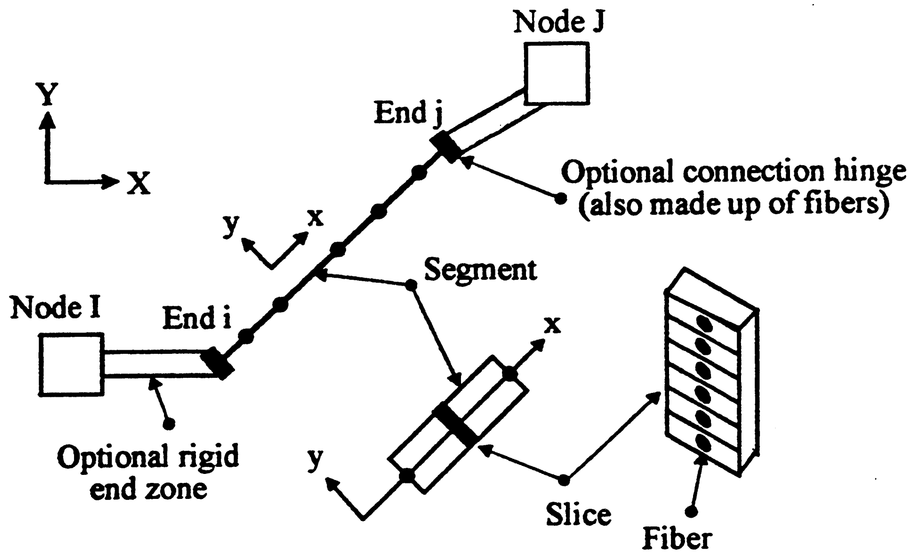


FIGURE E15.1 ELEMENT MODEL

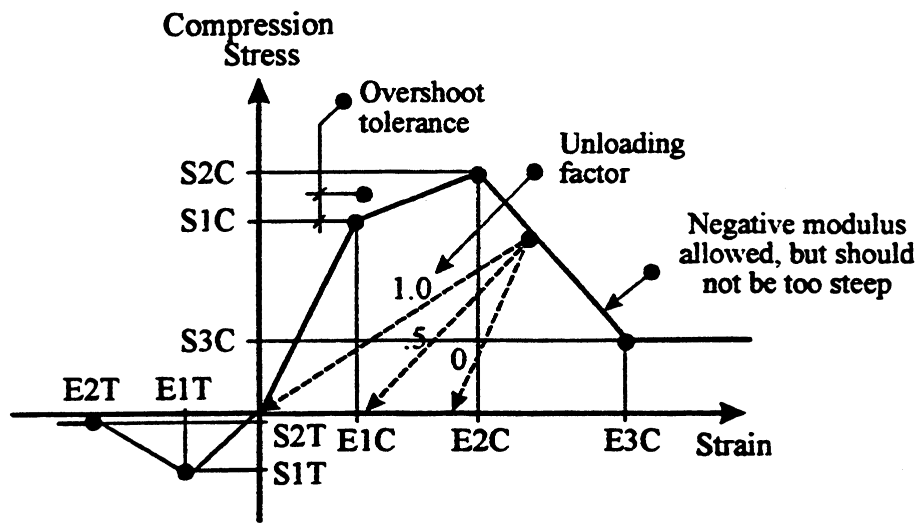


FIGURE E15.2 CONCRETE MATERIAL PROPERTIES

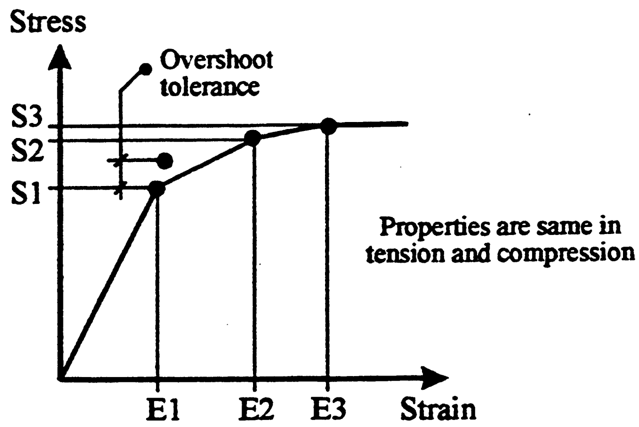


FIGURE E15.3 STEEL MATERIAL PROPERTIES

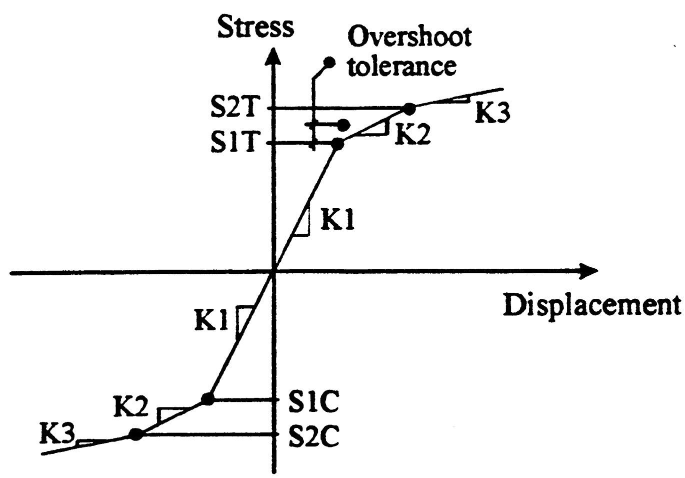
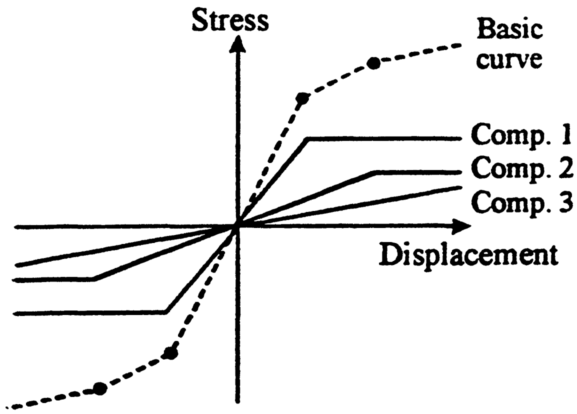
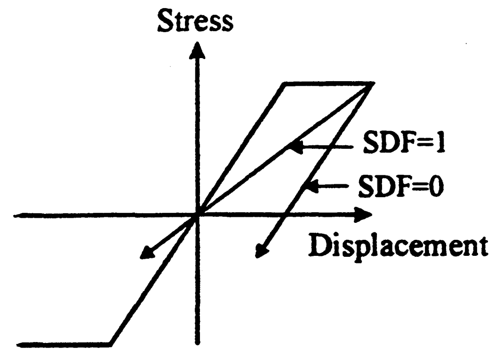


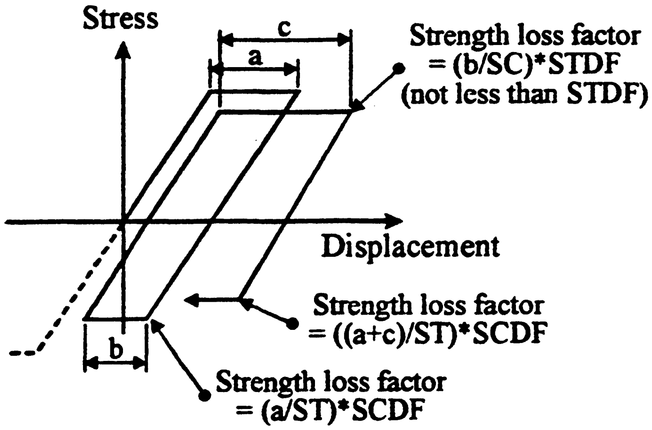
FIGURE E15.4 PULLOUT FIBER BASIC PROPERTIES



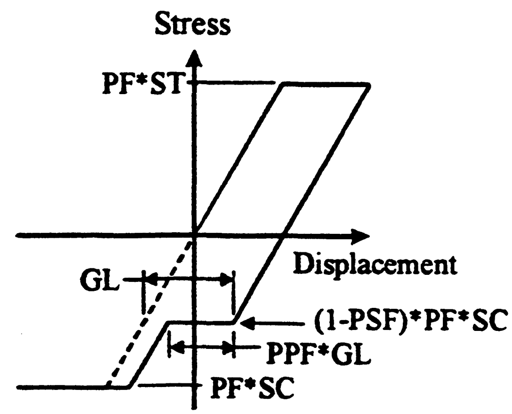
(a) Basic trilinear curve is first decomposed into three parallel components



(b) Stiffness degradation factor applies to both elastic-plastic components



(c) Strength loss in each component depends on strength degradation factor (STDF or SCDF) and ratio of accumulated plastic displacement to saturated displcement (ST or SC)



(d) Pinch factor (PF) divides each component into pinching and non-pinching parts. Pinch strength factor (PSF) and plateau factor (PPF) are then applied.

FIGURE E15.5 PULLOUT FIBER DEGRADATION PROPERTIES

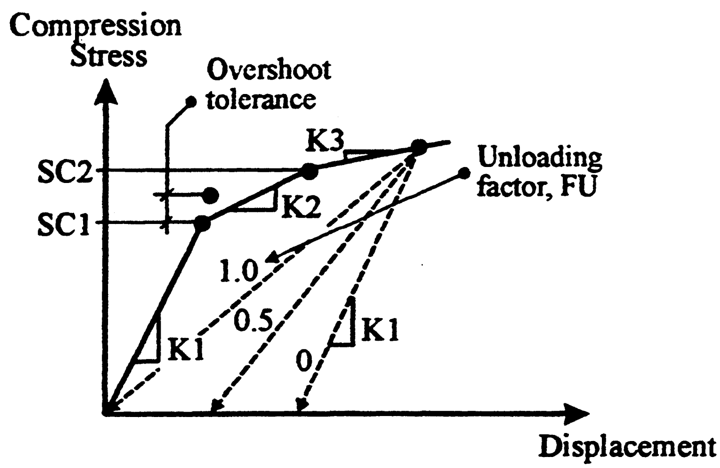


FIGURE E15.6 GAP FIBER PROPERTIES

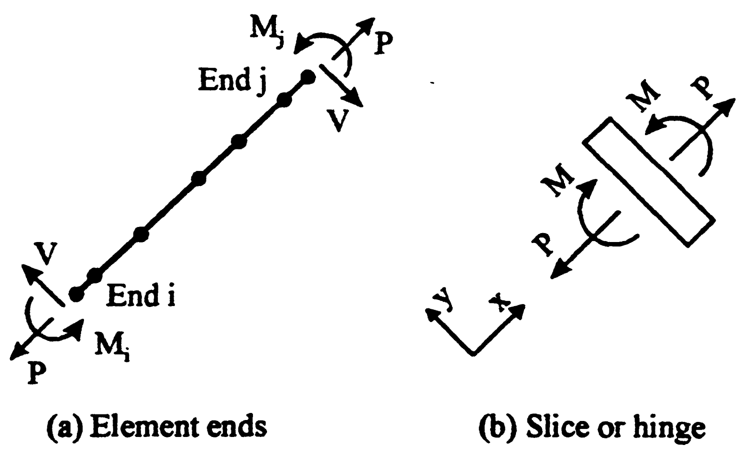


FIGURE E15.7 SIGN CONVENTIONS