Title

DRAIN-3DX: Element Description and User Guide for Element Type 01, Type 04, Type 05, Type 08, Type 09, Type 15 and Type 17 Version 1.10

Authors

Powell, Graham
Campbell, Scott

Publication Date

1994-08-01

REPORT NO.
UCB/SEMM-94/08

STRUCTURAL ENGINEERING
MECHANICS AND MATERIALS

DRAIN-3DX ELEMENT DESCRIPTION AND USER GUIDE FOR ELEMENT TYPE01, TYPE04, TYPE05, TYPE08, TYPE09, TYPE15, and TYPE17

VERSION 1.10

BY

G. H. POWELL
AND
S. CAMPBELL

AUGUST 1994

DEPARTMENT OF CIVIL ENGINEERING
UNIVERSITY OF CALIFORNIA
BERKELEY, CALIFORNIA

INELASTIC TRUSS BAR ELEMENT (TYPE 01) FOR DRAIN-3DX

VERSION 1.10
MARCH 1994

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 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. These are forces that must act ***on the element ends*** to prevent end displacement.

Columns	Notes	Variable	Data
1-5(I)			Load set number, in sequence beginning with 1.
6-10(I)		KCOOR	Coordinate code. 0 = forces are in local (element) coordinates. Only axial forces (P) can be specified. 1 = forces are in global (X, Y, Z) coordinates..
11-20(R)			Force Pi or Xi.
21-30(R)			Force Pj or Xj.
31-40(R)			Force Yi.
41-50(R)			Force Yj.
51-60(R)			Force Zi
61-70(R)			Force Zj.

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.

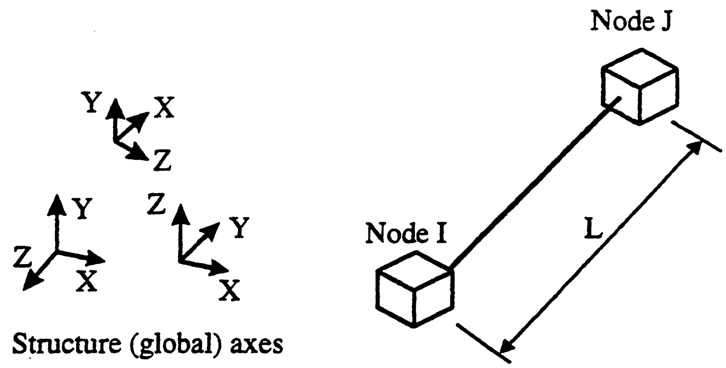


FIGURE E01.1 ELEMENT GEOMETRY

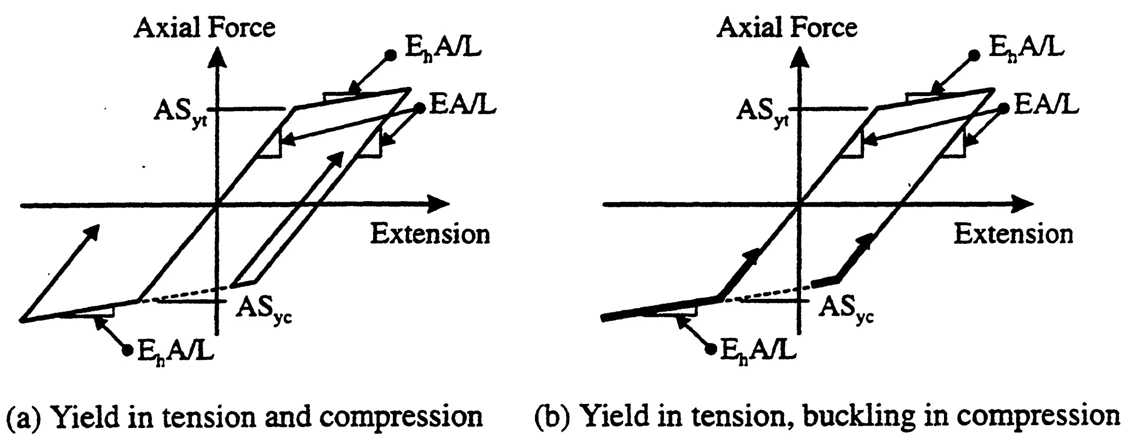


FIGURE E01.2 ELEMENT BEHAVIOR

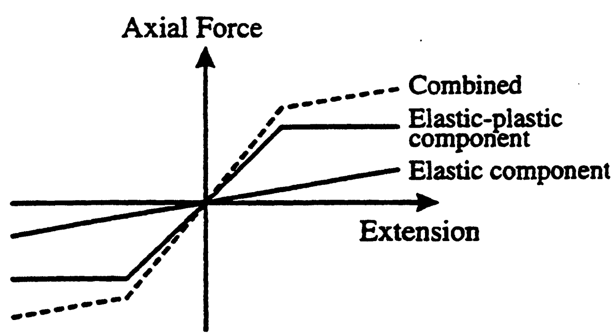


FIGURE E01.3 PARALLEL COMPONENTS

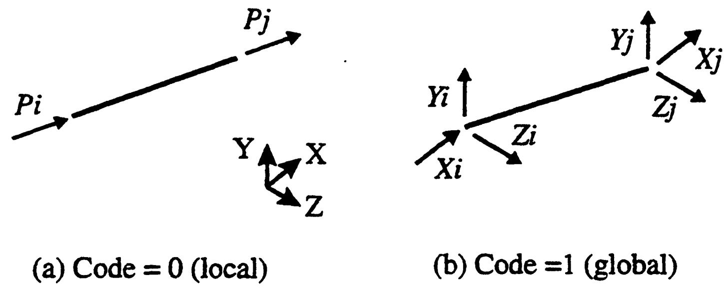


FIGURE E01.4 FIXED END FORCES

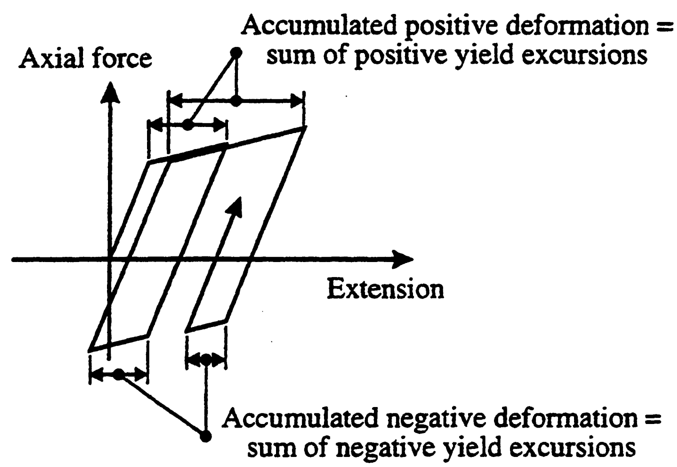


FIGURE E01.5 ACCUMULATED PLASTIC DEFORMATIONS

SIMPLE CONNECTION ELEMENT (TYPE 04) FOR DRAIN-3DX

VERSION 1.10
MARCH 1994

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).

The element properties are defined in a local coordinate system, x,y,z, as shown in Figure E04.1(a). These axes can be oriented arbitrarily in space. The orientation is specified as follows.

(1) Projections of the local x axis with respect to the global X,Y, and Z axes must be input. These projections can be the projections of a unit length of x axis, in which case they are the direction cosines. However, they can be the projections of any length.
(2) Projections or direction cosines of a local y' axis with respect to the global X,Y, and Z axes must be input, where y' is any axis in the local x-y plane, as shown in Figure E04.1(b). As a special case y' may be parallel to the y axis. However, y' must not be parallel or near parallel to x.

Given these projections, the program calculates the direction cosines of the local x,y,z axes.

An element connects the nodes either translationally or rotationally along *one* of the local axes (this is the *element axis*). To connect the nodes translationally *or* rotationally along all three local axes, three elements are required. To connect the nodes translationally *and* rotationally along all three local axes, six elements are required.

Since an element has zero length, care must be taken in defining the positive and negative actions and deformations. Positive actions are in the same directions as positive deformations. Positive deformations are as follows.

(1) Positive translational deformation is when Node I moves along the element axis relative to Node J. Examples are shown in Figure E04.2(a).
(2) Positive rotational deformation is when Node I has a positive rotation about the element axis relative to Node J, where positive rotation is according to the "right hand screw" rule.. Examples are shown in Figure E04.2(b).

An element can be specified to behave elastically or inelastically, as shown in Figure E04.3. 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 through E04.3 for element geometry 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).
6-10(I)		NDIR	No. of direction (orientation) types (min. 1, max. 20)

E04.2.2 Property Types

NPROP lines, one for each property type.

See Figure E04.2 for sign convention. See Figure E04.3 for element behavior and properties.

Specify realistic stiffnesses. Astronomically large values (e.g., 10^{10}) are unrealistic and can cause numerical sensitivity problems.

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).
60(I)		KFM	Force/moment code, as follows. 1 = This is a force (translational) connection. 2 = This is a moment (rotational) connection.
65(I)			Elasticity code, as follows. 0 = Unload inelastically. 1 = Unload elastically. 2 = Unload inelastically with gap.

E04.2.3 Direction (Orientation) Types

NDIR lines, one for each direction type. See Figure E04.1.

Columns	Notes	Variable	Data
1-5(I)			Direction type number, in sequence beginning with 1.
6-15(R)			X projection or direction cosine of element x-axis.
16-25(R)			Y projection or direction cosine of element x-axis.
26-35(R)			Z projection or direction cosine of element x-axis.
36-45(R)			X projection or direction cosine of element y'-axis.
46-55(R)			Y projection or direction cosine of element y'-axis.
56-60(I)			Z projection or direction cosine of element y'-axis.

E04.2.4 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.
16-25(I)			Node J.
26-35(I)			Node number increment for element generation. Default = 1
36-40(I)			Property type number. Default = same as preceding element.
41-45(I)			Direction type number. Default = same as preceding element.
50(I)			Element axis (1 = x, 2 = y, 3 = z).

E04.3 INTERPRETATION OF RESULTS

E04.3.1 SIGN CONVENTIONS

The sign conventions for element actions and deformations are shown in Figure E04.2. Accumulated plastic deformations are calculated as shown in figure E04.4.

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.

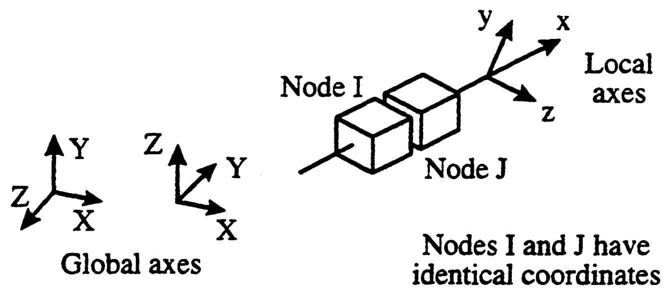
E04.3.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

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.

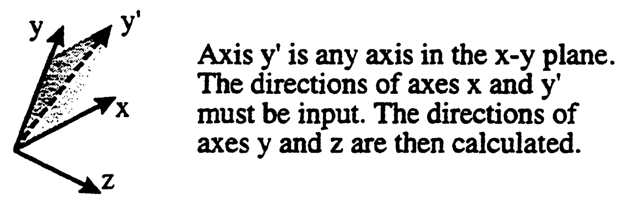
Item	Description
1	Static force or moment.
2	Viscous force or moment.
3	Total deformation.
4	Accumulated positive plastic deformation (sum of all positive excursions with yield code = 1).
5	Accumulated negative plastic deformation (sum of all negative excursions with yield code = 1).
6	Node number at end I.
7	Node number at end J.
8	Direction code (1 = translational x, 2 = y, 3 = z, 4 = rotational x, 5 = y, 6 = z).
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.

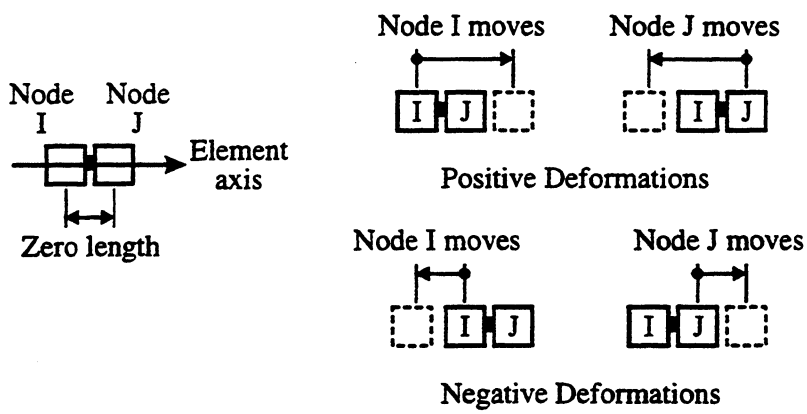


(a) Nodes and Axes

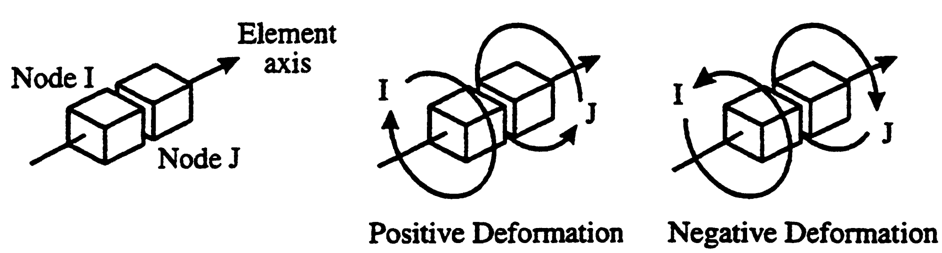


(b) Procedure for Orienting Axes

FIGURE E04.1 ELEMENT GEOMETRY

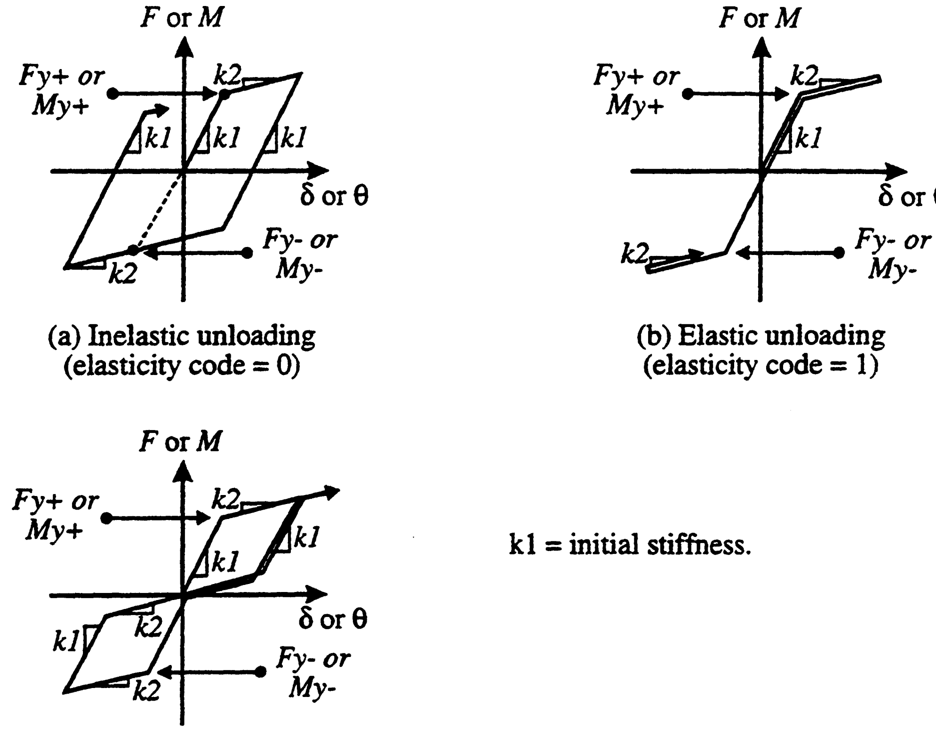


(a) Translational Element



(b) Rotational Element

FIGURE E04.2 SIGN CONVENTION



(a) Inelastic unloading
(elasticity code = 0)

(b) Elastic unloading
(elasticity code = 1)

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

k1 = initial stiffness.

FIGURE E04.3 BEHAVIOR OPTIONS

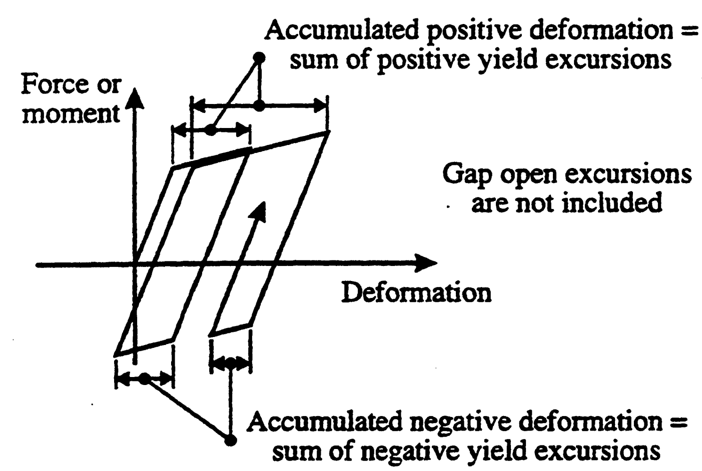


FIGURE E04.4 ACCUMULATED PLASTIC DEFORMATIONS

FRICTION BEARING ELEMENT (TYPE 05) FOR DRAIN-3DX

VERSION 1.10
MARCH 1994

ELEMENT DESCRIPTION AND USER GUIDE

E05.1 PURPOSE, FEATURES AND LIMITATIONS

E05.1.1 PURPOSE

This is a relatively simple inelastic element for modelling supports that resist normal force in bearing and lateral force through friction.

E05.1.2 ELEMENT MODEL

The element connects two nodes (Nodes I and J) which must have identical coordinates (i.e. this is a ***zero-length*** element).

The element properties are defined in a local coordinate system, x,y,z, as shown in Figure E05.1(a). These axes can be oriented arbitrarily in space. The orientation is specified as follows.

(1) Projections of the local x axis with respect to the global X,Y, and Z axes must be input. These projections can be the projections of a unit length of the x axis, in which case they are the direction cosines. However, they can be the projections of any length.
(2) Projections or direction cosines of a local y' axis with respect to the global X,Y, and Z axes must be input, where y' is any axis in the local x-y plane, as shown in Figure E05.1(b). As a special case y' may be parallel to the y axis. However, y' must not be parallel or near parallel to x.

The element consists of a bearing (normal) component and a friction (lateral) component. For bearing, Node I pushes on Node J in the positive x direction of the element, as shown in Figure E05.2(a). The x axis can have any orientation. For a typical bearing, the x axis is vertical down if the part of the structure represented by Node I sits on top of the part represented by Node J, and vertical up if the reverse is the case. The stiffnesses of the bearing component can be different in compression and tension, as shown in Figure E05.3(a). The stiffness in tension can be very small, but is not allowed to be exactly zero.

Friction (lateral) forces act in the element y-z plane. The direction of slip is in the direction of the resultant friction force. The sign convention for friction deformations and forces is shown in Figure E05.2(b). The friction behavior is shown in Figure E05.3(b). Essentially friction slip occurs when the resultant friction force exceeds the bearing or tension force multiplied by a specified friction coefficient. However, the friction force will not be exactly equal to the nominal slip force, but will be within a specified tolerance of the nominal value. This tolerance is introduced mainly for computational reasons, to avoid unnecessary stiffness change events.

Theoretically, slip is governed by a "slip surface"., which is a circle. For the model, however, the slip surface is octagonal, as shown in Figure E05.4. The slip force in the model than thus be slightly larger or smaller than the theoretical slip force, even if the friction coefficient tolerance is very small. Since it is unlikely that the friction coefficient will be known precisely, the difference should not be of much practical importance.

There is no provision for element loads. However, an initial compression or tension force can be specified for the bearing component.

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

E05.1.3 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, in both the bearing and lateral directions.

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 bearing that is nearly rigid before it slips, the βK damping stiffness will be large, and hence large amounts of viscous energy may be absorbed after slip occurs. This may not be a correct model. It is recommended that a zero β value be specified for elements of this type, and that other element types be used to obtain viscous damping.

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.

E05.1.4 OVERSHOOT TOLERANCE

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

An "event" corresponds to a change in stiffness of an element, due to slip, unloading from a slipping condition, gap opening or closing, element yield, 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 (considering all elements of all types). 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.

E05.2 INPUT DATA FOR *ELEMENTGROUP

See Figures. E05.1 through E05.3 for element geometry and properties.

E05.2.1 Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NPROP	No. of property types (min. 1, max. 20).
6-10(I)		NDIR	No. of direction (orientation) types (min. 1, max. 20)

E05.2.2 Property Types

NPROP lines, one for each property type.

See Figure E05.2 for sign convention. See Figure E05.3 for element properties.

Specify realistic stiffnesses. Astronomically large values (e.g., 10^{10}) are unrealistic and can cause numerical sensitivity problems.

Columns	Notes	Variable	Data
1-5(I)			Property type number, in sequence beginning with 1.
6-15(R)			Bearing stiffness, KB.
16-25(R)			Tension stiffness, KT. Specify a very small value if bearing can lift off in tension
26-35(R)			Pre-slip lateral stiffness, K1, the same along both lateral axes..
36-45(R)			Post-slip lateral stiffness, K2, the same along both lateral axes..
46-55(R)			Friction coefficient, μ.
56-65(R)			Force overshoot tolerance. Used for both bearing uplift and lateral slip.
66-75(R)			Friction coefficient tolerance. Suggested value 0.05. With this value, slip occurs if the slip force is between 0.95 and 1.05 of the nominal value.

E05.2.3 Direction (Orientation) Types

NDIR lines, one for each direction type. See Figure E05.1.

Columns	Notes	Variable	Data
1-5(I)			Direction type number, in sequence beginning with 1.
6-15(R)			X projection or direction cosine of element x-axis.
16-25(R)			Y projection or direction cosine of element x-axis.
26-35(R)			Z projection or direction cosine of element x-axis.
36-45(R)			X projection or direction cosine of element y'-axis.
46-55(R)			Y projection or direction cosine of element y'-axis.
56-60(I)			Z projection or direction cosine of element y'-axis.

E05.2.4 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.

Strictly speaking, Nodes I and J should have identical coordinates. A warning (not a fatal error) is printed if this is not the case.

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 of Node I.
16-25(I)			Node number of Node J.
26-35(I)			Node number increment for element generation. Default = 1
36-40(I)			Property type number. Default = same as preceding element.
41-45(I)			Direction type number. Default = same as preceding element.
46-55(R)			Initial bearing force (negative value = tension force). Default = 0.

E05.3 INTERPRETATION OF RESULTS

E05.3.1 SIGN CONVENTIONS

The sign conventions for element actions and deformations are shown in Figure E05.1. Accumulated slip is calculated as shown in figure E05.3.

E05.3.2 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

E05.3.3 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E05.3.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

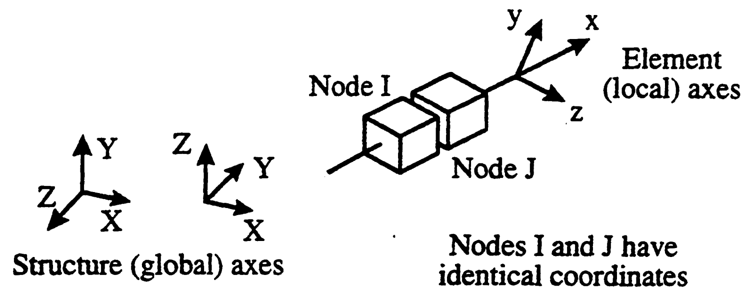
E05.3.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

The following items (9 4-byte words) are output for each element in the .RXX file. To change these output items, see subroutine RESP05 in the ANAL05.FOR source code file.

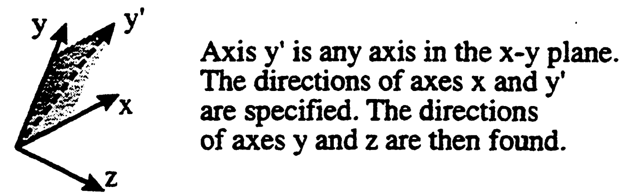
Item	Description

E05.3.6 USER OUTPUT (.USR FILE)

There is no user output subroutine (source code file USER05.FOR) for this element.

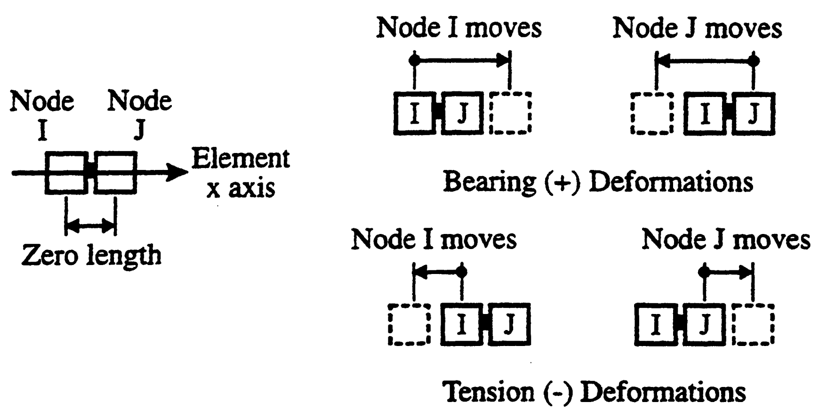


(a) Nodes and Axes

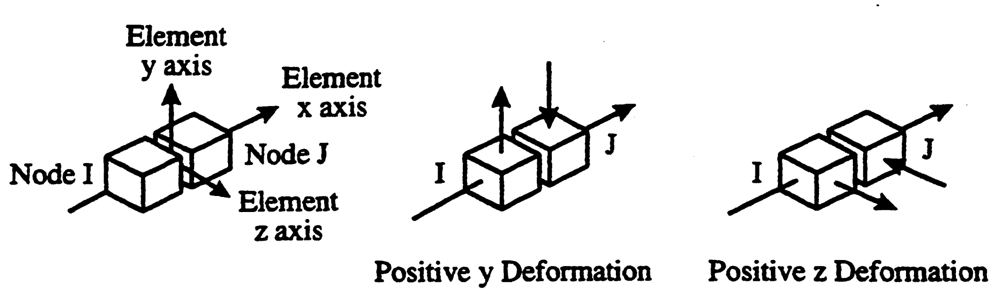


(b) Procedure for Orienting Axes

FIGURE E05.1. ELEMENT GEOMETRY

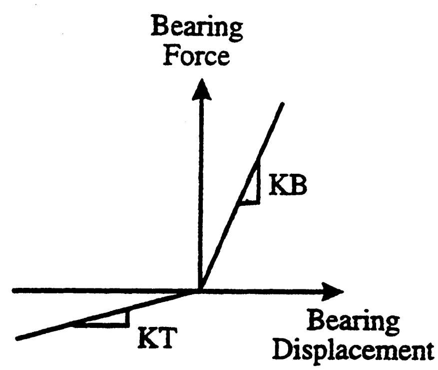


(a) Bearing Component

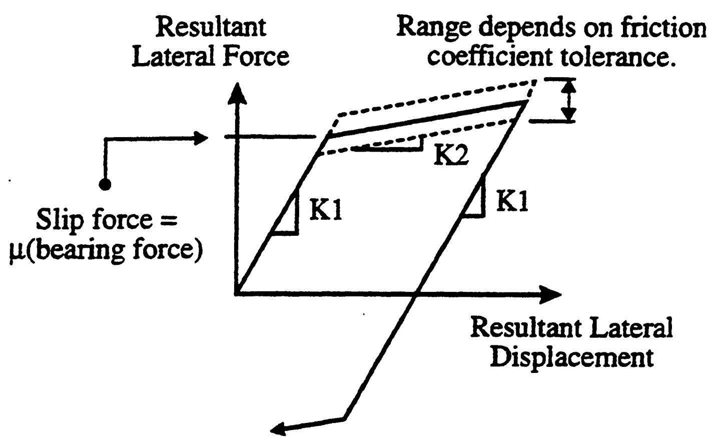


(b) Friction Component

FIGURE E05.2 SIGN CONVENTION



(a) Bearing Component



(b) Friction Component

FIGURE E05.3. ELEMENT PROPERTIES

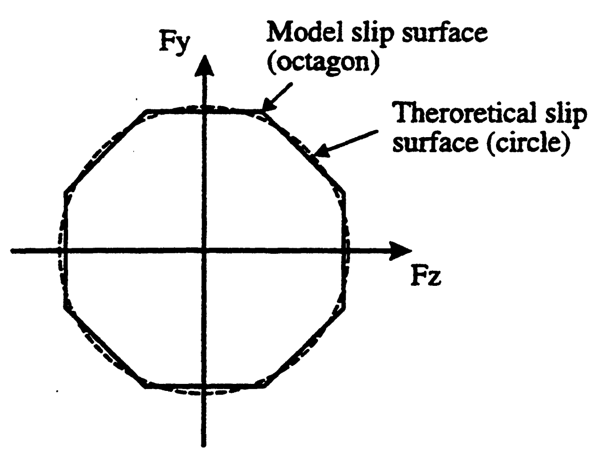


FIGURE E05.4. SLIP SURFACE

FIBER HINGE BEAM-COLUMN ELEMENT (TYPE 08)
FOR DRAIN-3DX AND DRAIN-BUILDING

VERSION 1.10
APRIL 1994

ELEMENT DESCRIPTION AND USER GUIDE

E08.1 PURPOSE, FEATURES, AND LIMITATIONS

E08.1.1 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 main limitation of the element is that it is assumes lumped plasticity (generalized plastic hinges). The element has a wide variety of options, and can be used in many different ways.

E08.1.2 ELEMENT MODEL

The element consists of an elastic beam with a number of inelastic hinges, as shown in Figure E08.1. The beam accounts for elastic flexural, axial, torsional, and shear deformations along the element length. The hinges account for inelastic axial, flexural, and shear deformations. These inelastic deformations will actually be distributed over zones of finite length. However, the hinges are assumed to have zero length (i.e., the element is based on lumped plasticity approximations).

Hinges of three types can be specified, namely P-M hinges, shear hinges, and connection hinges. The P-M hinges account for inelastic axial and flexural deformations within the body of the element, including P-M interaction effects. The shear hinges account for both elastic and inelastic shear deformations. These hinges do not account for P-V or M-V interaction. The connection hinges account for local axial and flexural deformations due to effects such as bond slip in connections at the element ends. These hinges account for P-M interaction.

Each P-M hinge is made up of rigid-plastic-strain-hardening fibers. A fiber can be of steel or concrete type, as shown in Figure E08.2. There is no hinge deformation until enough fibers yield to create a hinge mechanism. The deformation of this mechanism may be purely flexural (a simple plastic hinge), purely axial (which is unlikely), or mixed axial and flexural (a "generalized" plastic hinge), depending on the axial force and moment values at the hinge. The location and properties of the fibers determine the hinge behavior.

There can be up to three P-M hinge locations along the length of an element, at any chosen locations (not necessarily at the element ends). At each location there can be a single hinge or a pair of hinges. If a single hinge is used its properties will usually be chosen to capture the ultimate strength of the cross section, and the fiber strain hardening moduli will usually be very small. That is, this will be a plastic hinge with little or no strain hardening. If a pair of hinges is used, one will usually be a "weak" hinge and one a "strong" hinge. The weaker hinge will be chosen to capture the first significant yield of the cross section, and the stronger hinge to capture the ultimate strength. The fibers of the first yield hinge will have significant strain hardening. As the axial force and bending moments increase, the yield hinge forms a mechanism first. This hinge hardens as it deforms. After sufficient hardening the ultimate strength is reached and the ultimate strength hinge also forms a mechanism. If a single hinge is used the beam moment-rotation relationship is, in effect, bilinear (elastic-perfectly plastic if the fibers have very small strain hardening). If a pair of hinges is used the beam moment-rotation relationship is, in effect, trilinear.

There can be up to two connection hinges, one at each element end, to model deformations that occur in beam-to-column or column-to-footing connections. Each hinge is made up of a number of fibers, each of which is elastic-plastic-strain-hardening. A fiber can be of bar pullout (steel) or gap (concrete) type, as shown in Figures E08.3 and E08.4. For reinforced concrete connections, the bar pullout fibers capture deformations of the reinforcing steel in the joint region (including bond slip), and will usually be relatively flexible. The concrete fibers capture compressive deformations in the joint region, and will usually be very stiff. Pullout and/or gap fibers can also be used to model semi-rigid connections in steel structures.

If both connection hinges and P-M hinges are used, the connection hinges will usually capture only the deformations in the end connections, and the P-M hinges will usually capture the inelastic deformations within the element. A connection hinge can, however, be used without P-M hinges. In this case the connection hinge properties must be chosen to capture both the joint deformations and the deformations within the element.

There can be up to two shear hinges, one for shear along the element y axis and one for the z axis. The behavior of a shear hinge is specified directly, in terms of shear force and shear deformation, as shown in Figure E08.5. The shear hinges are not composed of fibers.

The element is assumed to be elastic in torsion.

P-Δ effects can be included if desired.

E08.1.3 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, account for bond slip in connections).

2. Shear deformations are included but P-M-V interaction is ignored.

3. The behavior in torsion is assumed to be elastic, based on a specified shear modulus and effective torsional inertia.

4. There is currently no provision for prestressing or for initial stresses. Therefore, prestressed concrete members can not be considered, and composite steel-concrete members are assumed to be fully shored until the concrete has hardened.

5. All inelastic behavior is considered to be lumped into the hinges. Since a limited number of hinges are allowed the hinge locations and properties must be carefully chosen to approximate the correct beam-column behavior.

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

E08.1.4 INITIAL STIFFNESS AND VISCOUS DAMPING

Each element has a constant viscous damping matrix equal to $\beta \underline{K}_0$, where $\underline{K}_0$ is the initial stiffness matrix of the element (not including a geometric stiffness). The value of β is the same for all elements in the group.

The initial stiffness matrix is calculated from the elastic beam stiffness and the connection and shear hinge stiffnesses. The P-M hinges are initially rigid and do not contribute to the initial stiffness matrix. If a conection hinge has gap fibers, these fibers are assumed to be initially in compression.

E08.1.5 WARNINGS ON USE

Warning E08.1 Numbers of Fibers

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

1. Create models with single hinges 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 necessarily using single elements), and study the effects of using different numbers, locations, and tributary lengths of hinges. If bond 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 bond hinges are substantially emperical and must be carefully calibrated. 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 modeling of the complete structure only when you are satisfied that the member models are sound.

Warning E08.2 Numbers of Fibers

It is possible to specify elements with large numbers of fibers. 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 is likely to be extremely long, especially for a dynamic analysis. The recommended way of using this element is as follows.

(1) Specify only a small number of hinges, 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 E08.3 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 E08.4 Convergence

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 the *PARAMETERS part of the input file). If these limits are exceeded the analysis will end reasonably gracefully.

It is possible that there are combinations of axial and bending deformations for which the fiber hinges are unable to converge in the state determination phase. In this phase, if convergence has not been obtained after 50 iterations for any element, the program writes an error message in the .ECH file, and continues to execute. Large load unbalances may result. These should be temporary.

For dynamic analysis it is usually wise to perform the analysis in a number of analysis (time) segments, and to examine the results at the end of each segment before resuming, to ensure that the analysis is proceeding correctly. See the *ACCR and *DISR analysis options for resuming an analysis.

E08.1.6 TO INCREASE CAPACITY

In Section E08.2.1, there are limits on such values as the number of steel materials. Since the element has been used mainly for research purposes, several of these limits are quite restrictive, and probably impractical for realistic structures. The limits can be changed, as noted in the file INFGR08.H. It will be necessary to edit this file and file PARM08.H, and to recompile the program.

E08.2 INPUT DATA FOR *ELEMENTGROUP

See Section E08.1 for a description of the element features and limitations, and for warnings on use of the element.

E08.2.1. Control Information

One line. See Section E08.1.6 to change maximum allowable values..

Columns	Notes	Variable	Data
1-5(I)		NSPMF	No. of different steel material definitions for fibers in P-M hinges (min. 0, max. 5). See Section E08..22(a).
6-10(I)		NCPMF	No. of different concrete material definitions for fibers in P-M hinges (min. 0, max. 5). See Section E08.2.2(b).
11-15(I)		NPMHIN	No. of different P-M hinge definitions (min. 0, max. 6). See Section E08.2.3.
16-20(I)		NSBF	No. of different bar pullout material definitions for fibers in bond hinges (min. 0, max. 5). See Section E08.2.4(a).
21-25(I)		NCBF	No. of different gap (concrete) material definitions for fibers in bond hinges (min. 0, max. 5). See Section E08.2.4(b).
26-30(I)		NCHIN	No. of different connection hinge definitions (min. 0, max. 6). See Section E08.2.5.
31-35(I)		NVHIN	No. of different shear hinge definitions (min. 0, max. 6). See Section E08.2.6.
36-40(I)		NBEAM	No. of different elastic beam definitions (min. 1, max. 8). See Section E08.2.7.
41-45(I)		NRIG	No. of different rigid end zone definitions (min. 0, max. 8). See Section E08.2.8.
46-50(I)		NETYP	No. of different element geometry definitions (min. 1, max. 10). See Section E08.2.9.

E08.2.2(a). Steel Material Properties

NSPMF lines. Omit if NSPMF = 0. See Figure E08.2(a).
Steel material types are numbered in input sequence.

Note that the hardening modulus can be specified in terms of stress and *strain* or stress and *displacement*, depending on whether or not a hinge tributary length is specified (see Section E08.3).

Columns	Notes	Variable	Data
1-10(R)		SY	Yield stress. Must be positive.
11-20(R)		YMH	Hardening modulus. Must be positive.
21-30(R)			Stress overshoot tolerance (for event factor calculation).

E08.2.2(b). Concrete Material Properties

NCPMF lines. Omit if NCPMF = 0. See Figure E08.2(b).
Concrete material types are numbered in input sequence.

Note that the hardening modulus can be specified in terms of stress and *strain* or stress and *displacement*, depending on whether or not a hinge tributary length is specified (see Section E08.3).

Columns	Notes	Variable	Data
1-10(R)		SC	Crushing stress. Must be positive.
11-20(R)		YMH	Hardening modulus. Must be positive.
21-30(R)		FU	Unloading factor ($0 \leq FU \leq 1$). FU = 0 means no stiffness degradation upon unloading.
31-40(R)			Stress overshoot tolerance (for event factor calculation).

E08.2.3. P-M Hinge Types

NPMHIN sets. Omit if NPMHIN = 0.
Each set consists of one control line plus one line per fiber.
P-M hinge types are numbered in input sequence.

E08.2.3(i). Control Line

Columns	Notes	Variable	Data
1-5(I)		NFIBS	No. of fibers (min. 3, not all on the same straight line.).
6-15(R)			Optional tributary length, as follows. 0 or blank: No tributary length. Fiber moduli are stress/displacement. Positive: Proportion of element length. Fiber moduli are stress/strain. Negative: Absolute length. Fiber moduli are stress/strain.

E08.2.3(ii). Fibers

NFIBS lines, one per fiber, in any order.

Columns	Notes	Variable	Data
1-10(R)			Fiber y coordinate.
11-20(R)			Fiber z coordinate.
21-30(R)			Fiber area.
34(C)			Fiber material type. "S" if steel, "C" if concrete.
35(I)			Fiber material definition number.

E08.2.4(a). Bar Pullout Material Properties

NSBF lines or pairs of lines. Omit if NSBF = 0.
One line for a simple non-degrading material. Two lines for a material with stiffness degradation, strength degradation, and/or pinching behavior.

Pullout material types are numbered in input sequence.

E08.2.4(a)(i). Basic Properties

See Figures E08.3(a) and (c). Note that moduli are in terms of stress and deformation, not stress and strain.

Columns	Notes	Variable	Data
1-10(R)		YM1	Elastic modulus. Must be > 0.
11-20(R)		YM2	1st hardening modulus. 0 < YM2 < YM1.
21-30(R)		YM3	2nd hardening modulus. 0 < YM3 < YM2.
31-40(R)		S1T	Tensile yield stress. Must be > 0.
41-50(R)		S2T	2nd tensile yield stress. S2T > S1T.
51-60(R)		S1C	Compressive yield stress. Must be > 0.
61-70(R)		S2C	2nd compressive yield stress. S2C > 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 E08.2.4(a)(ii). If 1, material degrades. Include line E08.2.4(a)(ii).

E08.2.4(a)(ii). Degradation Parameters

Omit if the degradation indicator, IDGD, is blank or 0.

The basic material is first divided into parallel elastic-perfectly-plastic components. Each component is then divided into degrading and non-degrading parts, depending on the degradation factors. The strength degrading part loses compressive strength based on the accumulated tensile plastic deformation, and vice versa. Strength loss varies linearly with accumulated deformation, up to the maximum loss. Each component is also divided into pinching and non-pinching parts. The pinching behavior is shown in Figure E08.3(b).

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

08.2.4(b). Gap Material Properties

NCBF lines. Omit if NCBF = 0. See Figure E08.4.
Note that moduli are in terms of stress and deformation, not stress and strain.

Gap material types are numbered in input sequence.

Columns	Notes	Variable	Data
1-10(R)		SC1	Crushing stress 1. Must be positive
11-20(R)		SC2	Crushing stress 2. Must be > SC1.
21-30(R)		K1	Elastic modulus. Must be positive.
31-40(R)		K2	Hardening modulus. 0 < K2 < K1.
41-50(R)		K3	Plastic modulus. 0 < K3 < K2.
51-60(R)		FU	Unloading factor (0 ≤ FU ≤ 1). FU = 0 means no stiffness degradation upon unloading.
61-70(R)			Stress overshoot tolerance (for event factor calculation).

E08.2.5. 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.

E08.2.5(i). Control Line

Columns	Notes	Variable	Data
1-5(I)		NFIBS	No. of fibers (min. 3).

E08.2.5(ii). Fibers

NFIBS lines, one per fiber, in any order.

Columns	Notes	Variable	Data
1-10(R)			Fiber y coordinate.
11-20(R)			Fiber z coordinate.
21-30(R)			Fiber area.
34(C)			Fiber material type. "S" if steel (bar pullout), "C" if concrete (gap).
35(I)			Fiber material definition number.

E08.2.6. Shear Hinge Types

NVHIN lines or pairs of lines. Omit if NVHIN = 0.
One line for a simple non-degrading material. Two lines for a material with stiffness degradation, strength degradation, and/or pinching behavior.
Shear hinge deformations are added to any shear deformations in the elastic beam.

Shear hinge types are numbered in input sequence.

E08.2.6(i). Basic Properties

See Figures E08.5(a) and (c). Note that the stiffnesses are in terms of hinge force and deformation.

Columns	Notes	Variable	Data
1-10(R)		K1	Elastic stiffness. Must be > 0.
11-20(R)		K2	First hardening stiffness. 0 < K2 < K1.
21-30(R)		K3	Second hardening stiffness. 0 < K3 < K2.
31-40(R)		F1P	First positive yield force. Must be > 0.
41-50(R)		F2P	Second positive yield force. F2P > F1P.
51-60(R)		F1N	First negative yield force. Must be > 0.
61-70(R)		F2N	Second negative yield force. F2N > F1N.
71-75(R)			Force 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 E08.2.6(ii). If 1, material degrades. Include line E08.2.6(ii).

E08.2.6(ii). Degradation Parameters

Omit if degradation indicator, IDGD, is blank or 0.

The strength degradation and pinching are the same as for the pullout material. The basic material is first divided into parallel elastic-perfectly-plastic components. Each component is then divided into degrading and non-degrading parts, depending on the degradation factors. The strength degrading part loses compressive strength based on the accumulated tensile plastic deformation, and vice versa. Strength loss varies linearly with accumulated deformation, up to the maximum loss. Each component is also divided into pinching and non-pinching parts. The pinching behavior is shown in Figure E08.5(b).

Columns	Notes	Variable	Data
1-10(R)			Stiffness degradation factor (between 0 and 1, 0 = no degradation).
11-20(R)			Positive strength degradation factor (between 0 and 1, 0 = no degradation). Saturated deformation in negative direction must also be specified.
21-30(R)			Negative strength degradation factor (between 0 and 1, 0 = no degradation). Saturated deformation in positive direction must also be specified.
31-40(R)			Saturated deformation in negative direction (accumulated plastic deformation in negative direction for full strength loss in positive direction). Must be negative.
41-50(R)			Saturated deformation in positive direction (accumulated plastic deformation in positive direction for full strength loss in negative direction). Must be positive.
51-60(R)			Pinch factor (between 0 and 1, 0 = no pinching).
61-70(R)		PSF	Pinch strength factor. Omit if pinch factor is 0.
71-80(R)		PPF	Pinch plateau factor. Omit if pinch factor is 0.

E08.2.7. Elastic Beam Types

NBEAM sets. Omit if NBEAM = 0.
Each set consists of 3 lines.
Elastic beam types are numbered in input sequence.

E08.2.7(i). Material Properties

Columns	Notes	Variable	Data
1-10(R)			Young's Modulus. Must be > 0.
11-20(R)			Shear modulus (used for torsional and shear stiffness). Must be > 0

E.08.2.7(ii). Cross Section Properties

Columns	Notes	Variable	Data
1-10(R)			Torsional inertia. Must be > 0.
11-20(R)			Bending inertia about local y axis (must be a principal axis). Must be > 0.
21-30(R)			Bending inertia about local z axis. Must be > 0.
31-40(R)			Cross section area. Must be > 0.
41-50(R)			Shear area for shear along local y axis. Leave blank for no elastic shear deformation.
51-60(R)			Shear area for shear along local z axis. Leave blank for no elastic shear deformation.

E08.2.7(iii). Stiffness Factors

The 2x2 bending stiffness matrix about a principal axis is:

$$\frac{EI}{L}\begin{bmatrix} c_{ii} & c_{ij} \\ c_{ij} & c_{jj} \end{bmatrix}.$$

For a uniform section $c_{ii} = c_{jj} = 4$ and $c_{ij} = 2$. For a uniform beam with a moment release at end j $c_{ii} = 3$ and $c_{jj} = c_{ij} = 0$. Other values of these factors can be used for a release at i, a tapered beam, etc. If the entire line is blank, a uniform beam with no releases is assumed.

Columns	Notes	Variable	Data
1-10(R)			c_{ii} for axis yy.
11-20(R)			c_{jj} for axis yy.
21-30(R)			c_{ij} for axis yy.
31-40(R)			c_{ii} for axis zz.
41-50(R)			c_{jj} for axis zz.
51-60(R)			c_{ij} for axis zz.

E08.2.8. Rigid End Zone Types

NRIG lines. Omit if NRIG = 0.
Specify global X, Y, Z projections, from the node to the end of the deformable element, or vice-versa. See Section E08.2.9 for assigning signs to the projections.
Rigid zone types are numbered in input sequence.

Columns	Notes	Variable	Data
1-10(R)			Global X projection of rigid zone.
11-20(R)			Global Y projection of rigid zone.
21-30(R)			Global Z projection of rigid zone.

E08.2.9. Element Geometry Types

NETYP sets.
Each set consists of two lines.
Element geometry types are numbered in input sequence.

E08.2.9(i). Basic Data

Columns	Notes	Variable	Data
1-5(I)			Elastic beam property number.
6-10(I)			Type no. of bond hinge at end i. Default = none.
11-15(I)			Type no. of bond hinge at end j. Default = none.
16-20(I)			Type no. of shear hinge for y direction shear. Default = none.
21-25(I)			Type no. of shear hinge for z direction shear. Default = none.
26-30(I)			Type no. (+ or -) of rigid end zone no. at end i. Default = none. If +, X,Y,Z projections are from node to element end. If -, X,Y,Z projections are from element end to node.
31-35(I)			Type no. (+ or -) of rigid end zone no. at end j. Default = none. If +, X,Y,Z projections are from node to element end. If -, X,Y,Z projections are from element end to node.

E08.2.9(ii). P-M Hinges

Columns	Notes	Variable	Data
1-5(I)			Type no. of P-M yield hinge 1. Default = none.
6-10(I)			Type no. of P-M ultimate hinge 1. Default = none.
11-20(R)			Location of P-M hinge 1 *as a percentage of element length.*
21-40			Repeat for P-M hinge 2, if any.
41-60			Repeat for P-M hinge 3, if any.

E08.2.10. 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.
6-10(I)			Element geometry type number.
11-20(I)			Node I..
21-30(I)			Node J.
31-40(I)			Node number increment for element generation. Default = 1.
41-50(I)			Node K, to orient element y and z axes.

NOTE ON NODE NUMBERING FOR INTERFLOOR ELEMENTS IN DRAIN-BUILDING

In DRAIN-BUILDING, an element which is part of an interfloor can connect to a node in Floor 1 of the interfloor, and/or a node in Floor 2 of the interfloor, and/or a node in the interfloor itself. The node location is indicated as follows.

- For an interfloor node, specify the node number.
- For a node in Floor 1, place a "-" immediately after the node number. For example, if the node is node number 123456 in Floor 1, specify 123456-.
- For a node in Floor 2, place a "+" immediately after the node number. For example, if the node is node number 123456 in Floor 2, specify 123456+.

This applies for Node I and Node J, but not for the node number increment or for Node K. Node K must be a node in the interfloor.

E08.3 INTERPRETATION OF RESULTS

This section is to be added. For a detailed description of the element, including theory and recommended modeling procedures, see Scott Campbell's Ph.D. thesis, dated August 1994.

E08.3.1 SIGN CONVENTIONS

To be added.

E08.3.2 EVENT CODES

To be added.

E08.3.3 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E08.3.4 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E08.3.5 TIME HISTORY POST-PROCESSING (.RXX FILE)

To be added.

E08.3.6 USER OUTPUT (.USR FILE)

To be added.

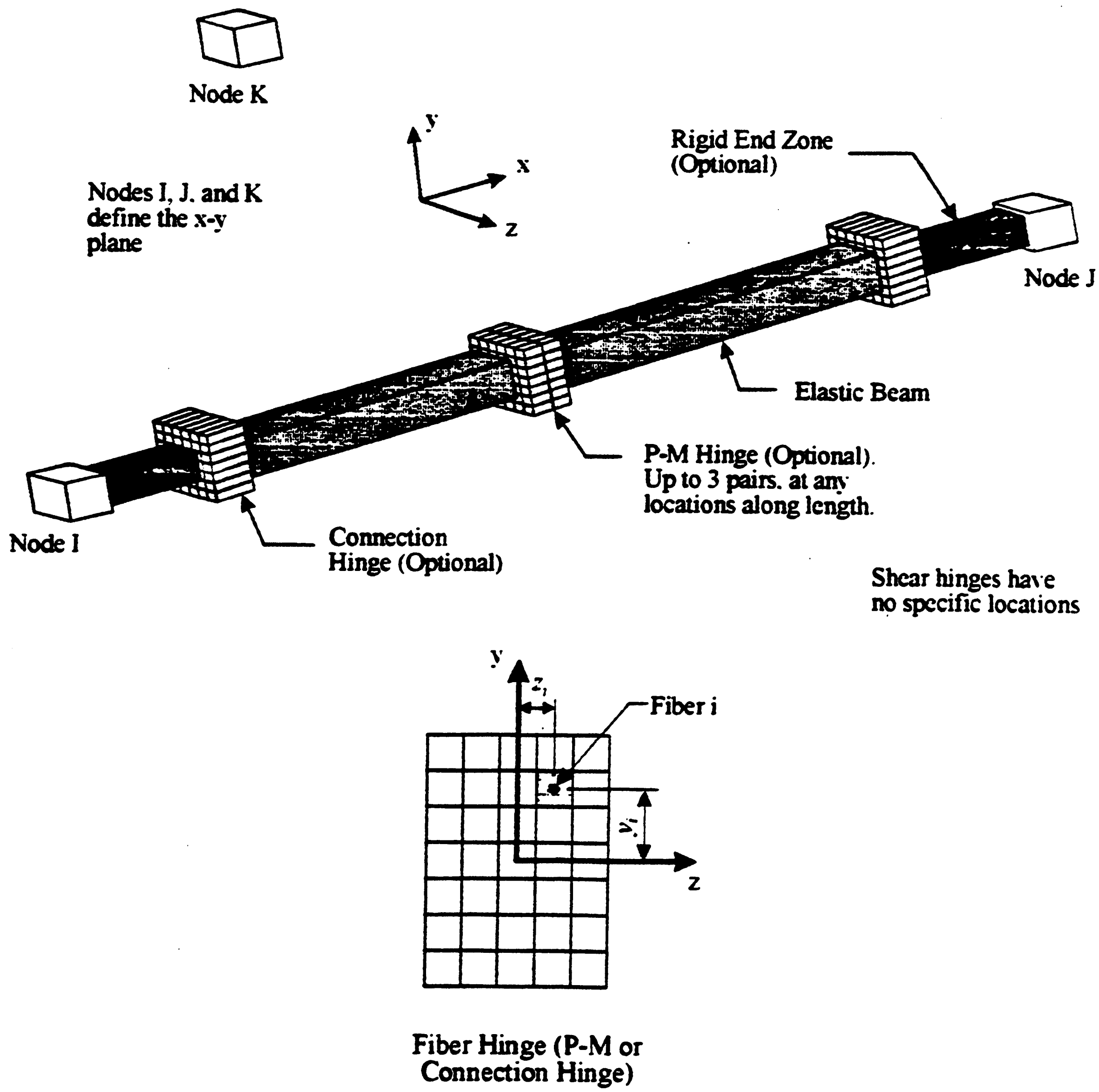


FIG. E08.1 ELEMENT GEOMETRY

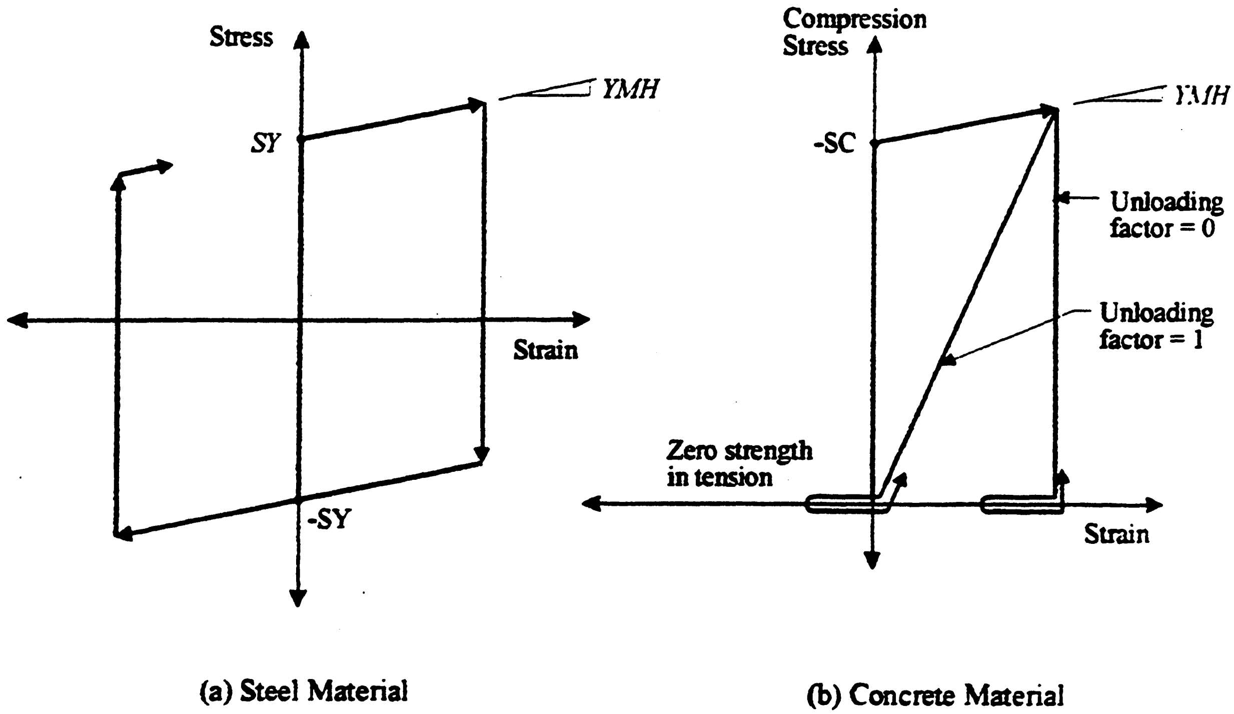
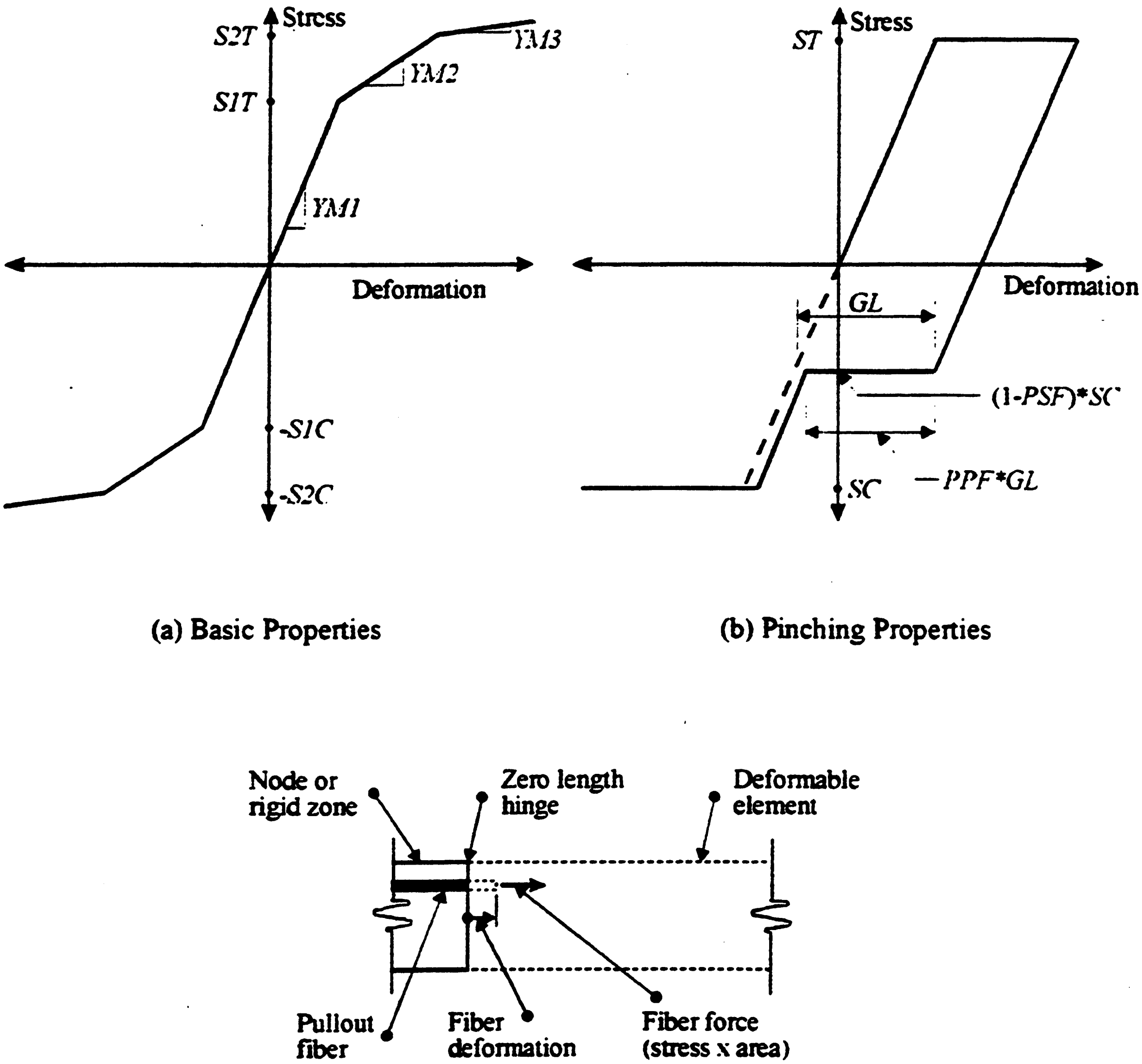


FIG. E08.2 MATERIALS FOR P-M HINGES



(a) Basic Properties

(b) Pinching Properties

(c) Fiber Deformation

FIG. E08.3 PULLOUT MATERIAL

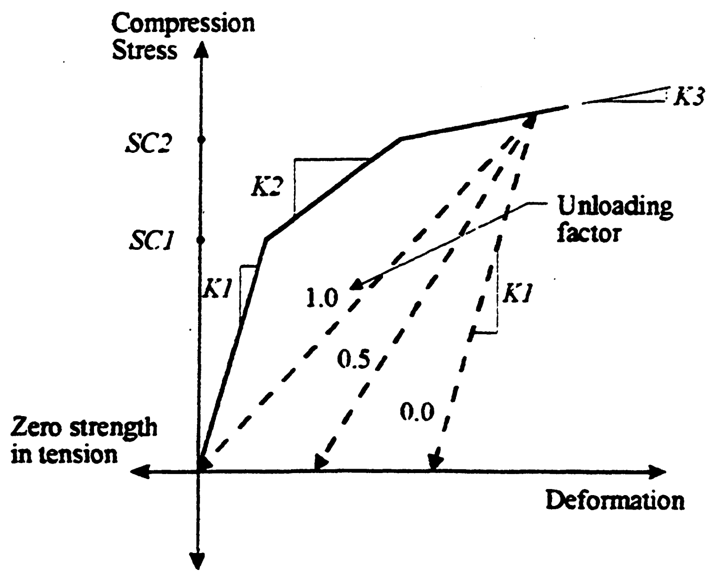
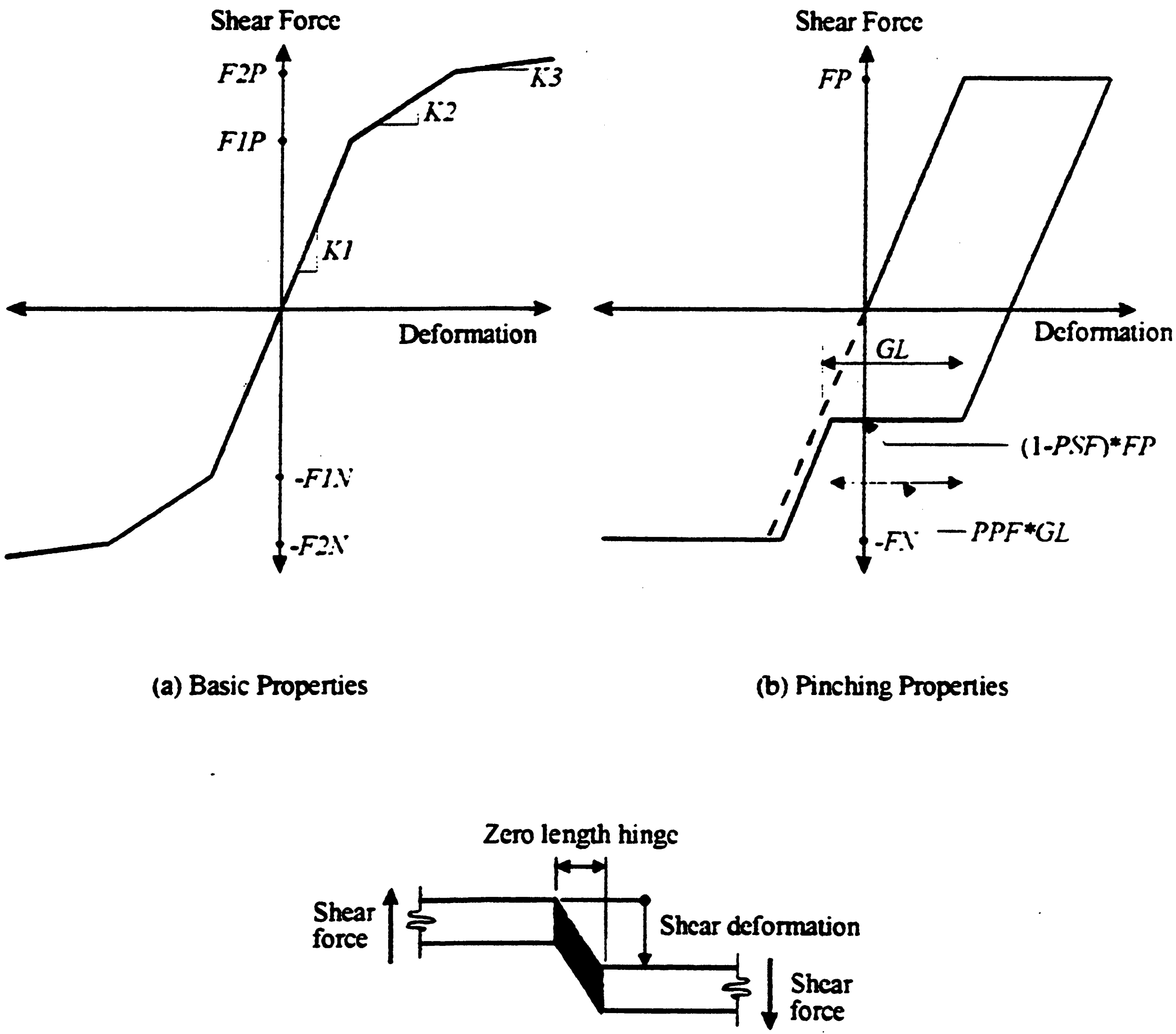


FIG. E08.4 GAP MATERIAL



(a) Basic Properties

(b) Pinching Properties

(c) Hinge Deformation

FIG. E08.5 SHEAR HINGE PROPERTIES

DRAIN-3DX USER GUIDE

FIBER HINGE BEAM-COLUMN ELEMENT (TYPE 08)
VERSION 2.00, JUNE 1993

BAR PULLOUT MODEL
PRELIMINARY DOCUMENTAION

Purpose

The bar pullout fiber is intended to model the behavior of reinforcing bars embedded in concrete. The model combines the deformations of the bar and the bond slip within the connection. The pullout fibers are assembled, along with concrete fibers, into "bond" hinges. These hinges account for the additional end deformation due to bar pullout and crack opening.

Technical Details

This section explains the inner workings of the bar pullout material model. Included are descriptions of the decomposition process, the stiffness and strength degradation, and the use of the pinching parameters. The monotonic stress-displacement curve consists of trilinear tensile and compressive portions. The stiffness is the same in both tension and compression while the strength may vary. In practice, the trilinear curve is decomposed into two bilinear curves and an elastic curve acting in parallel as shown in Figure 2. The degradation parameters control the behavior of the two bilinear curves.

The stiffness degradation factor controls the unloading/reloading stiffness of the bilinear curves. A value between zero and one is specified for the degradation. No degradation of stiffness occurs if a value of zero is input (unload at initial stiffness) while a value of one causes the curve to unload along a line passing through the point where the curve last crossed the zero stress axis (see Figure 3). Values between zero and one cause a linear interpolation of the stiffness between the two extremes.

The strength degradation is based on the accumulated plastic deformation of the bilinear fiber. Tensile plastic displacement causes compressive strength degradation and vice versa. A value between zero and one is input for the strength degradation. This factor determines the residual strength by specifying the fraction of the initial yield strength that is lost by degradation. Additionally, a saturated displacement value is input. Once this value of accumulated plastic displacement is reached the fiber is assumed to have fully degraded. The strength degrades linearly between zero loss before yield to full loss at residual. This is done separately for each bilinear fiber.

Each bilinear subfiber is divided into pinching and non-pinching portions used in parallel. The pinching behavior of the is controlled by three parameters. The first is the pinching factor that determines how much of the fiber strength is carried by the pinching and non-pinching portions. A value between zero and one is input with zero signifying no pinching and one indicating a fiber that has only pinching behavior. Values between zero and one provide a combined behavior with the specified fraction of the yield strength in the pinching portion.

Once the basic pinching curve behavior has been established the pinched strength can be determined. The pinch strength factor controls the value of stress at which pinching starts (see Figure 4). The pinch strength factor specifies the fraction of the current yield stress remaining when pinching begins. A value between zero and one must be input. A pinching strength factor of zero indicates no pinching.

When pinching begins, the fiber will deform with no additional stress until the plateau is traversed. The pinch plateau factor determines the length of the plateau. A value of one indicates that the plateau should extend until it meets the last unloading curve (see Figure 5). A value of zero indicates no pinching.

Examples

A series of plots showing the hysteresis curves of a fiber with varying pinching and unloading parameters are enclosed. The fiber had the following properties for the monotonic curve.

YM1 = 30.0E+06
YM2 = 10.0E+06
YM3 = 10.0E+05
S1T = 2.0E+05
S2T = 3.0E+05

The strengths are symmetric in tension and compression. Zero strength degradation was assumed to better illustrate the effects of the other parameters. The enclosed curves are not meant to model a specific material but are included to demonstrate the range of behavior that can be obtained from the bar pullout model.

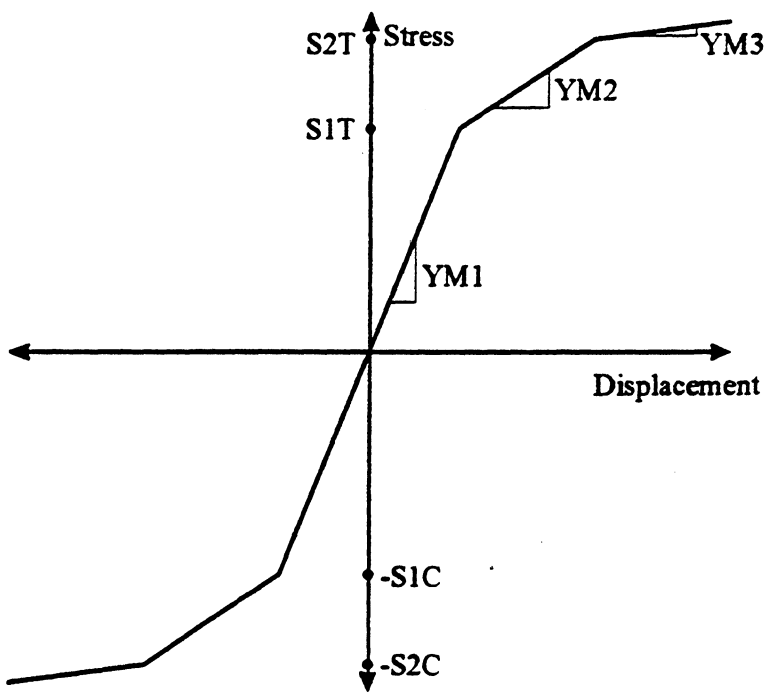


Figure 1. Bar Pullout Material Stress-Displacement Curve.

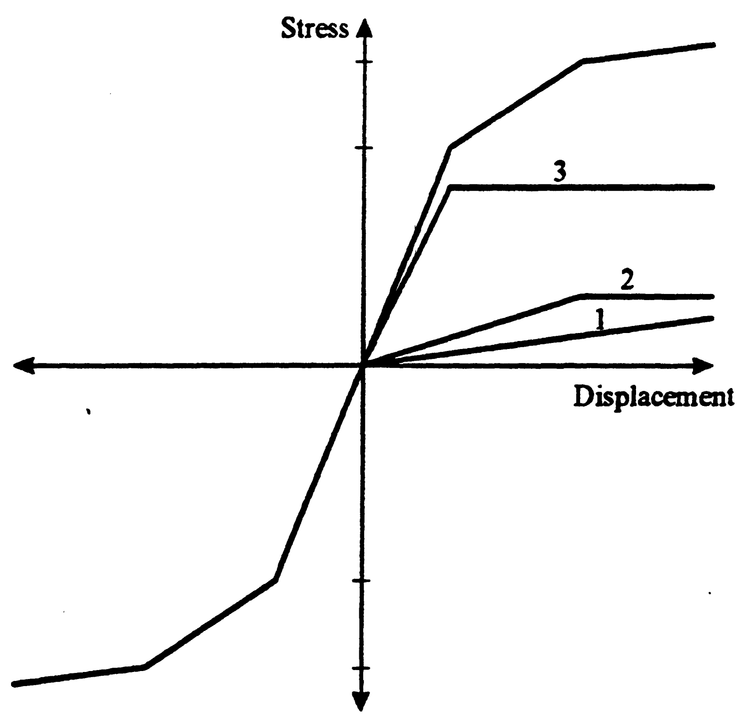


Figure 2. Decomposition of Trilinear Stress-Strain Curve.

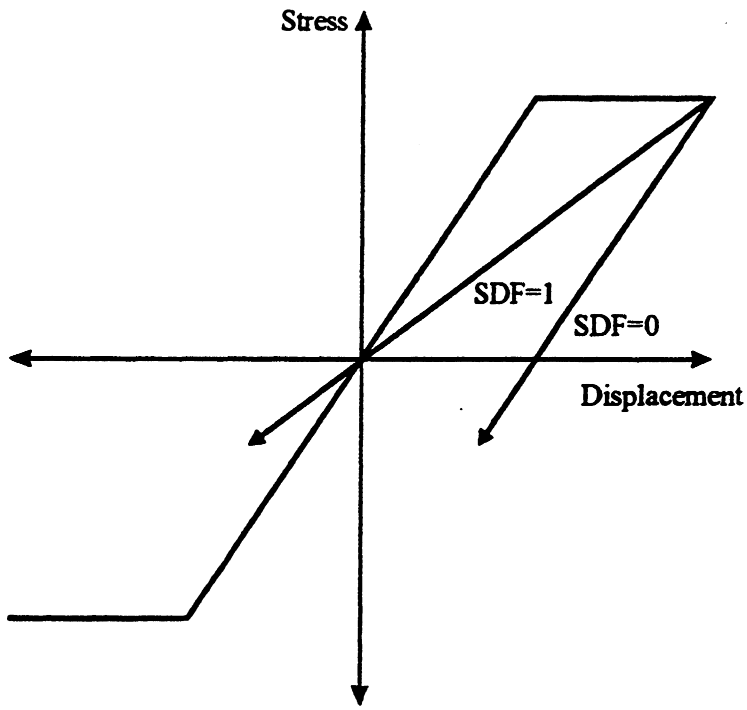


Figure 3. Stiffness Degradation Behavior.

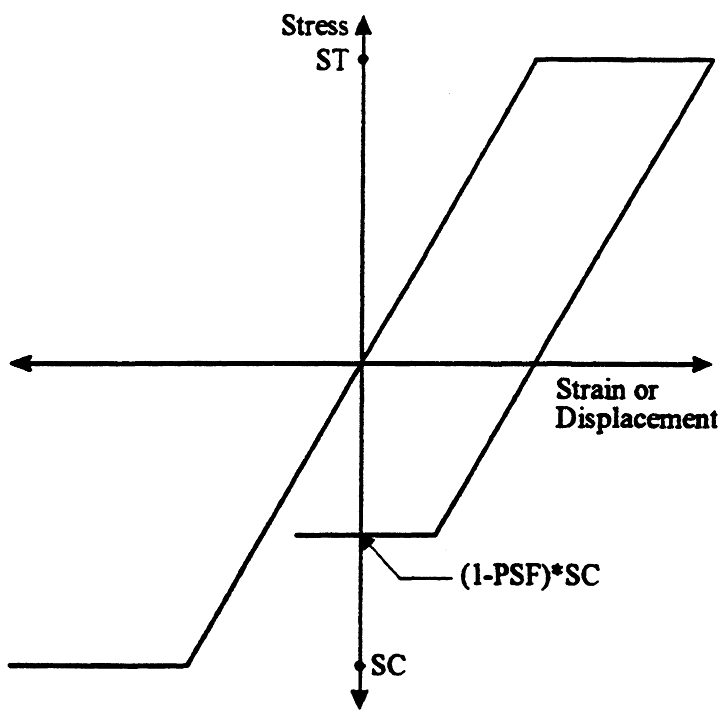


Figure 4. Pinch Strength Factor.

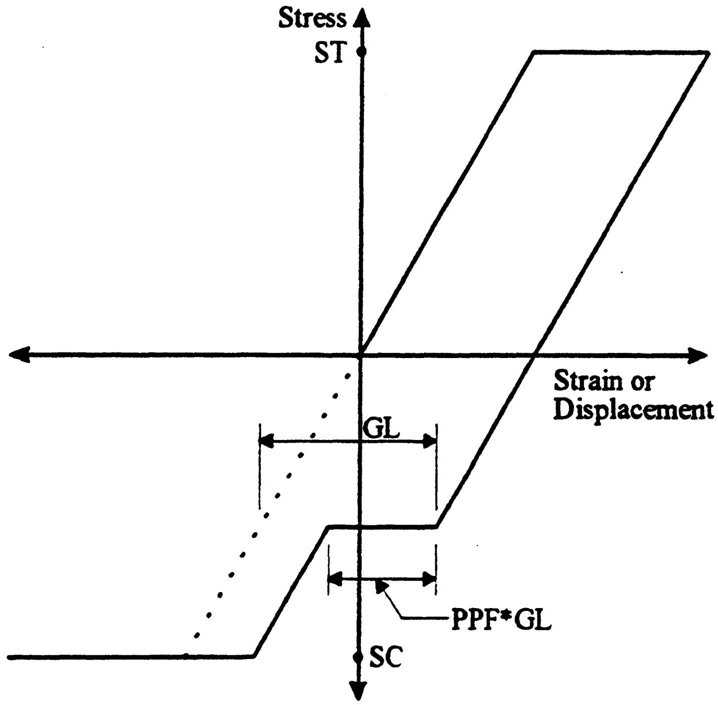
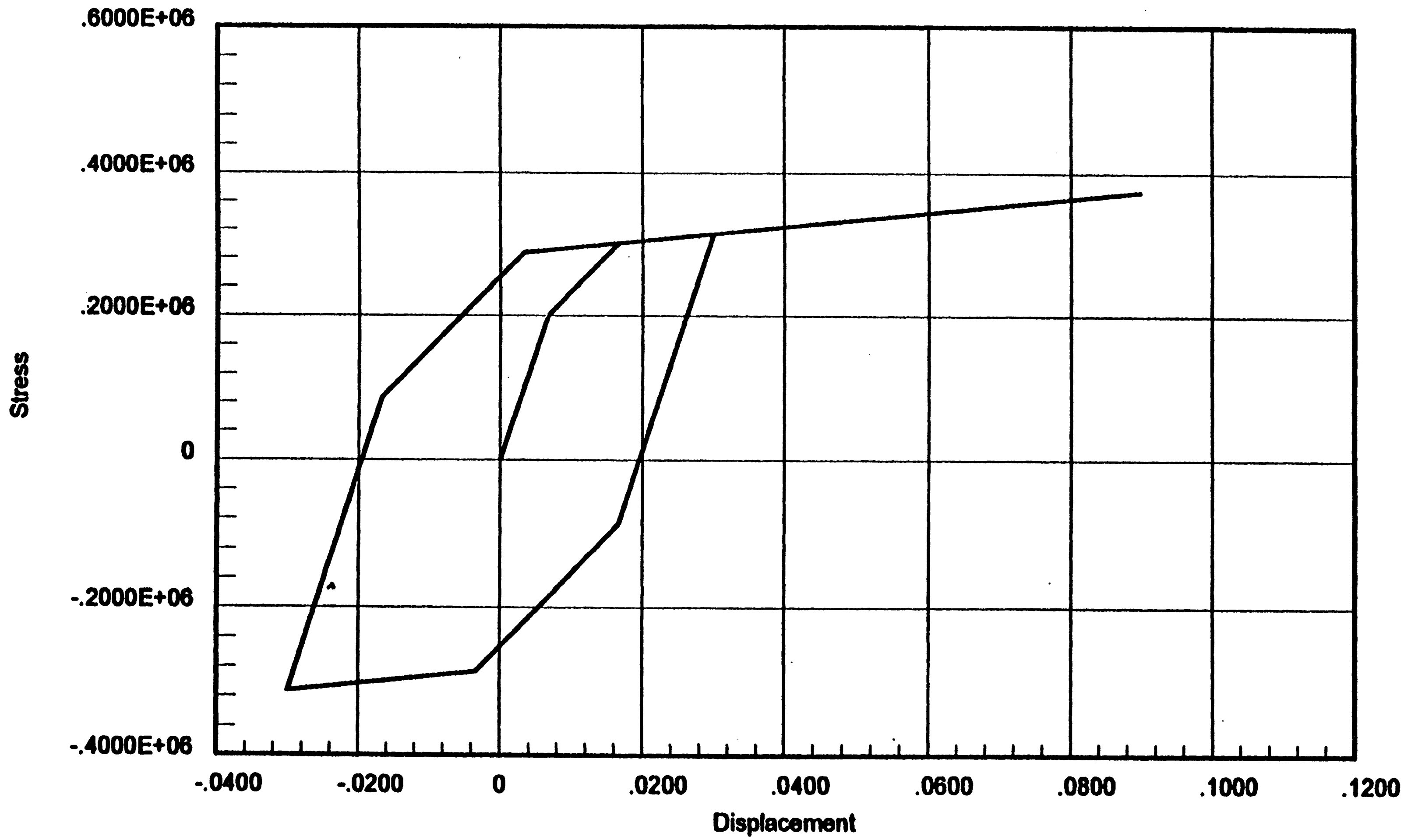
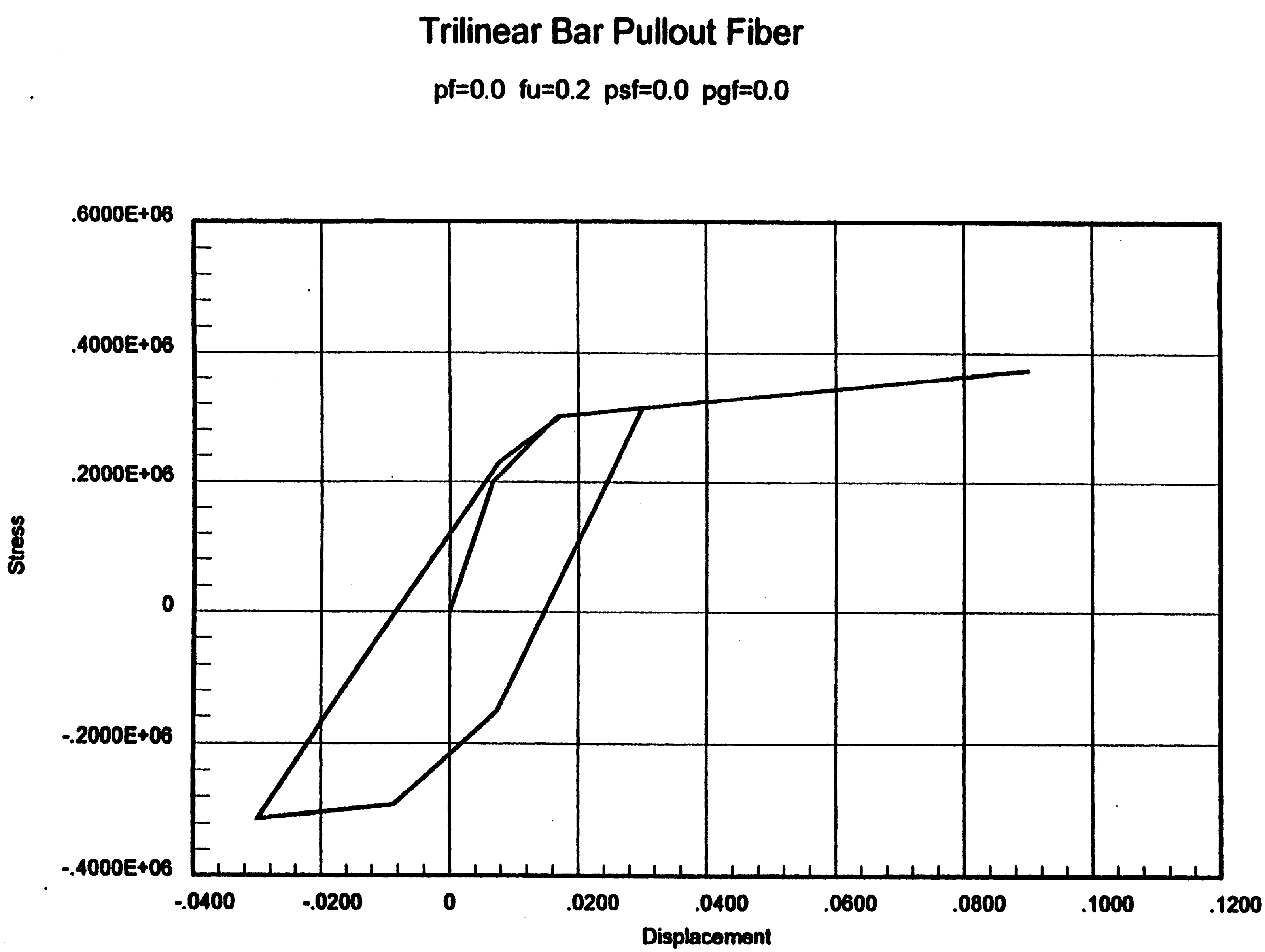


Figure 5. Pinch Plateau Factor.

Trilinear Bar Pullout Fiber

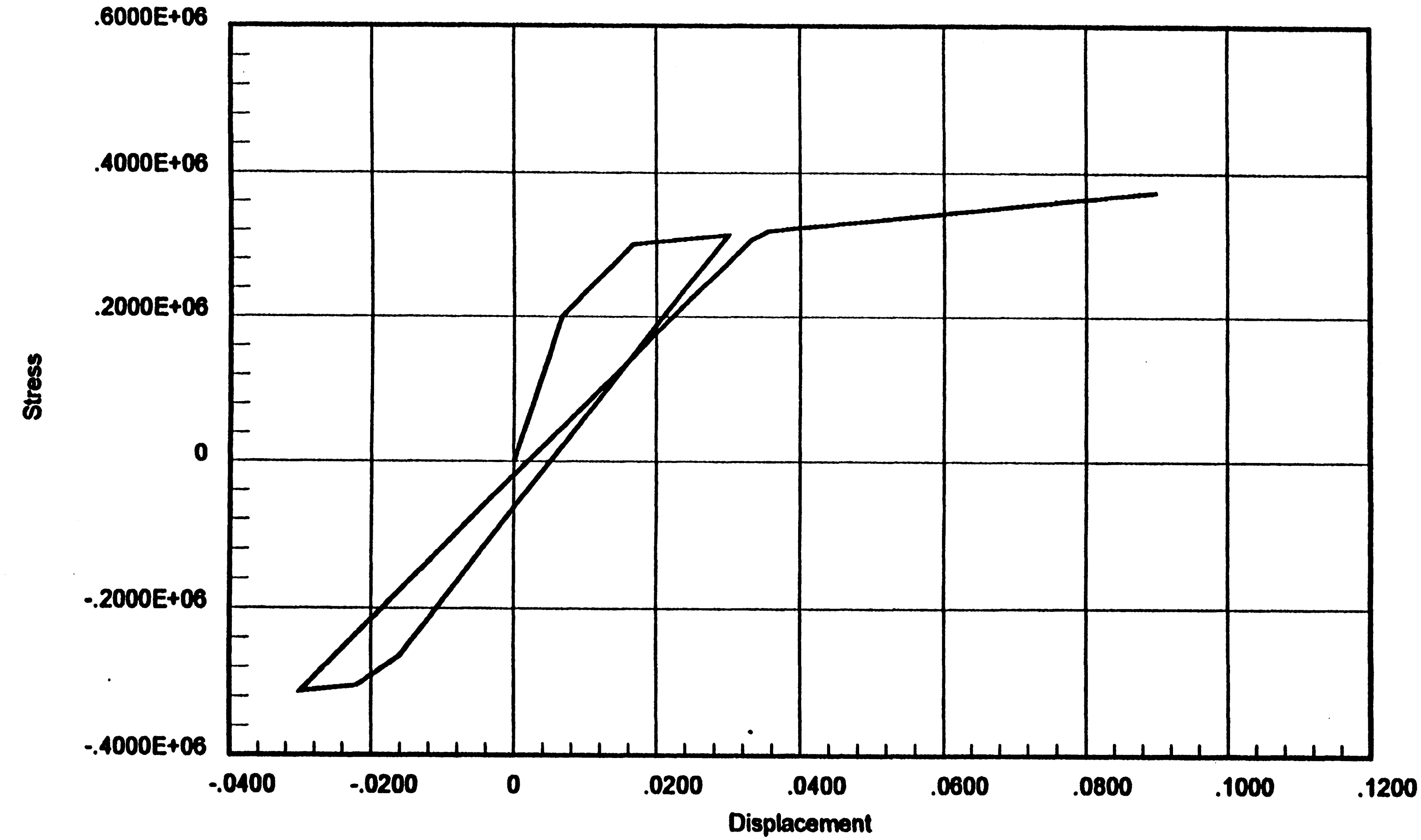
pf=0.0 fu=0.0 psf=0.0 pgf=0.0




Trilinear Bar Pullout Fiber
pf=0.0 fu=0.2 psf=0.0 pgf=0.0
.6000E+06
.4000E+06
.2000E+06
0
-.2000E+06
-.4000E+06
Stress
-.0400
-.0200
0
.0200
.0400
.0600
.0800
.1000
.1200
Displacement

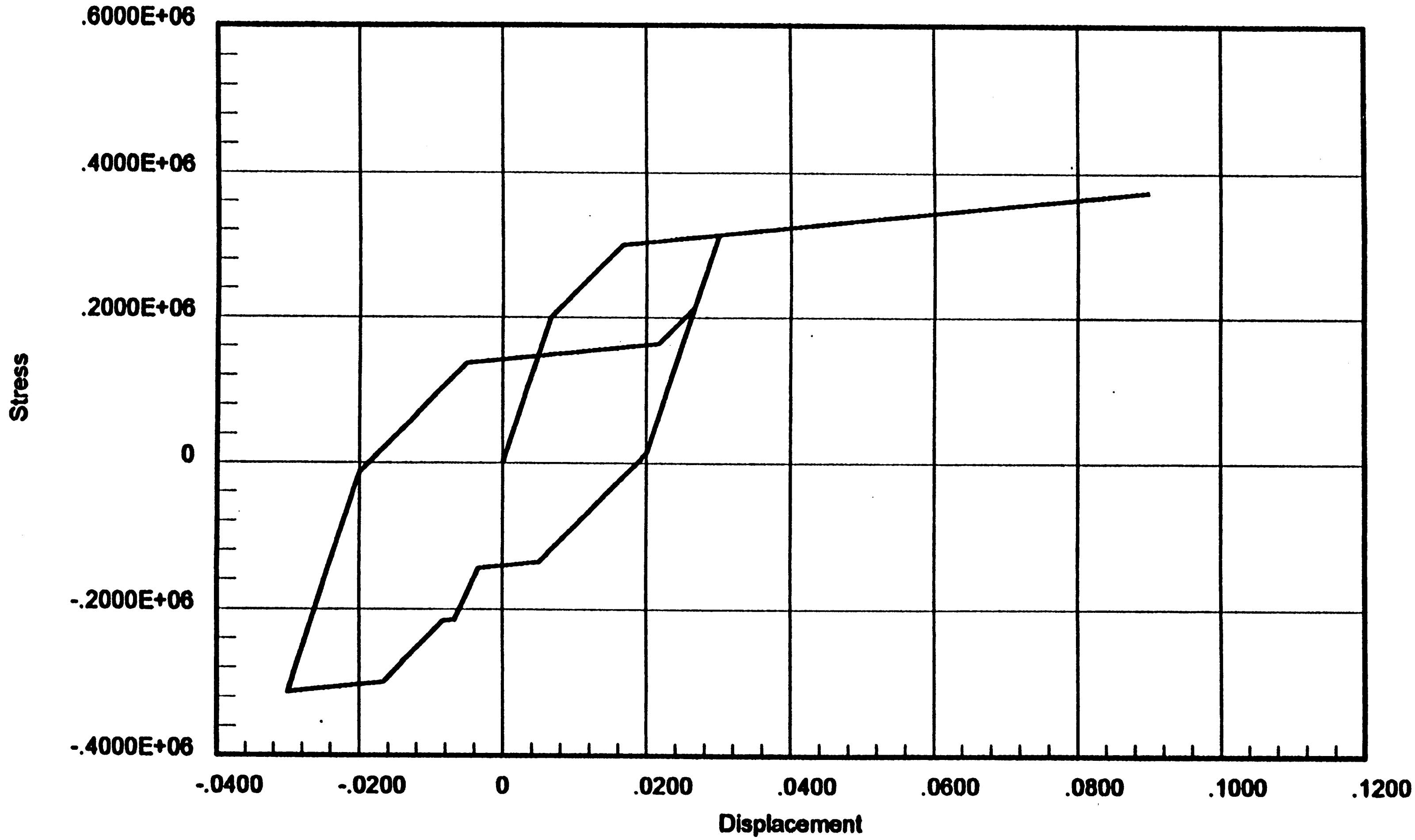
Trilinear Bar Pullout Fiber

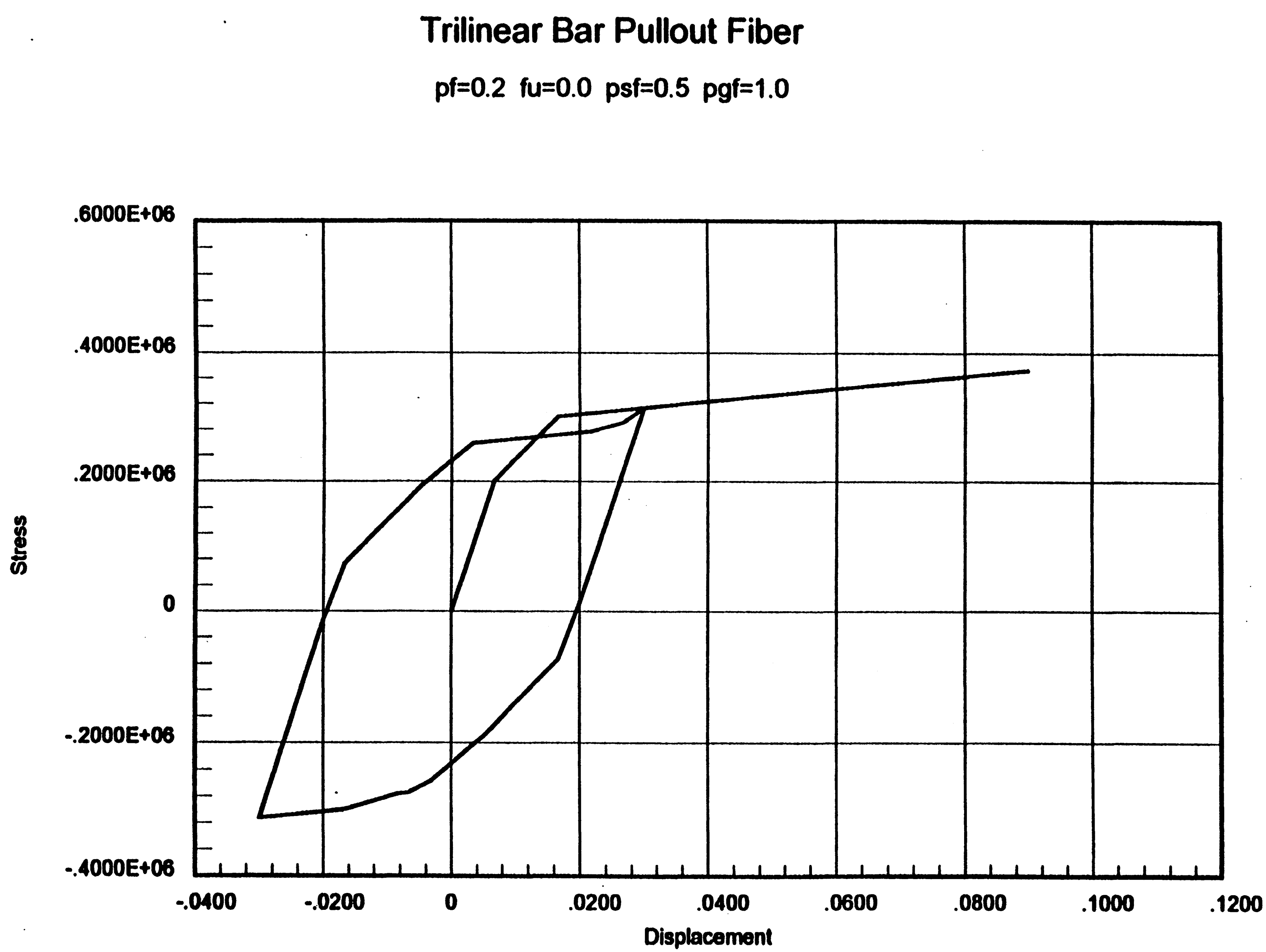
pf=0.0 fu=0.7 psf=0.0 pgf=0.0



Trilinear Bar Pullout Fiber

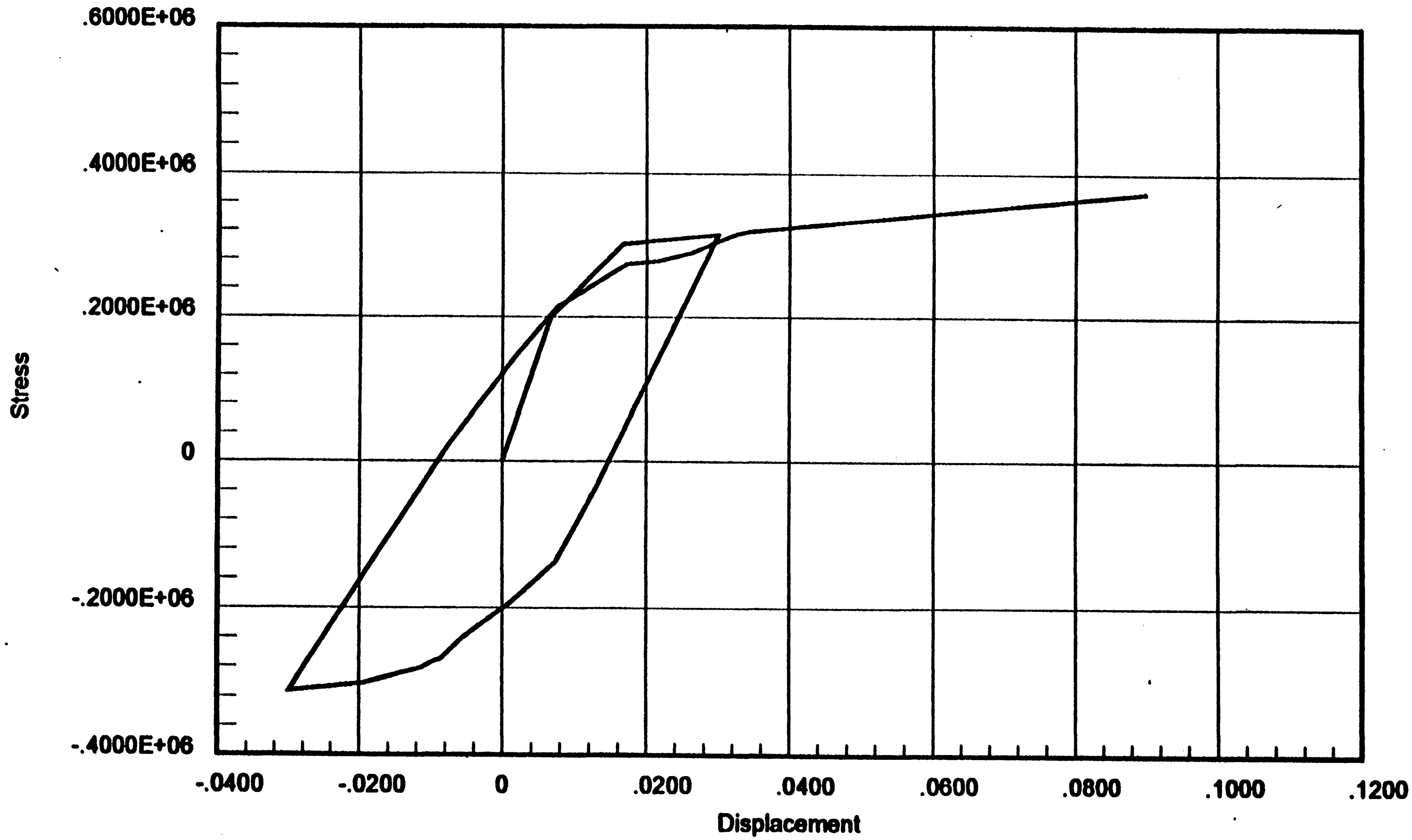
pf=1.0 fu=0.0 psf=0.5 pgf=1.0




Trilinear Bar Pullout Fiber
pf=0.2 fu=0.0 psf=0.5 pgf=1.0
.6000E+06
.4000E+06
.2000E+06
0
-.2000E+06
-.4000E+06
Stress
-.0400
-.0200
0
.0200
.0400
.0600
.0800
.1000
.1200
Displacement

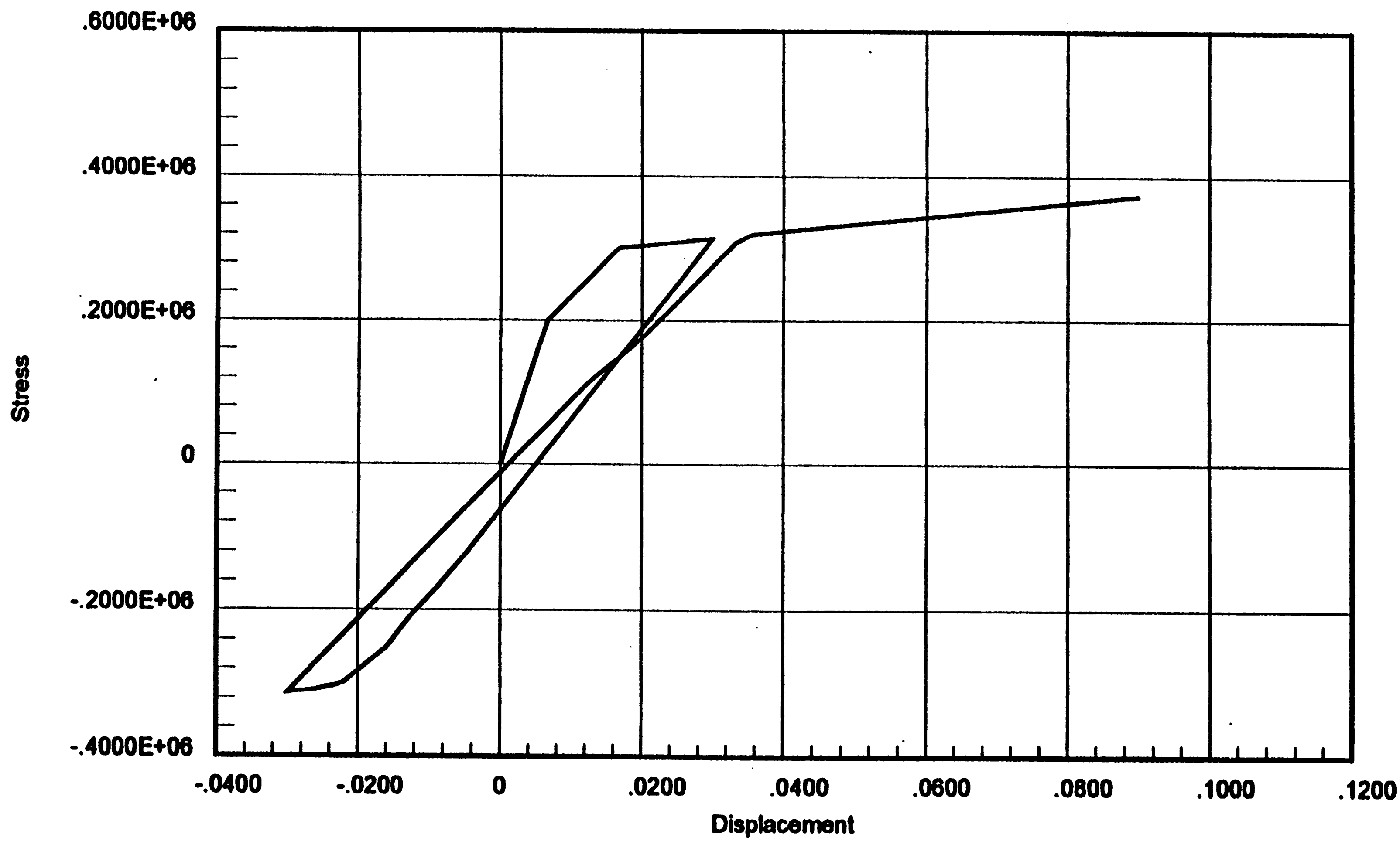
Trilinear Bar Pullout Fiber

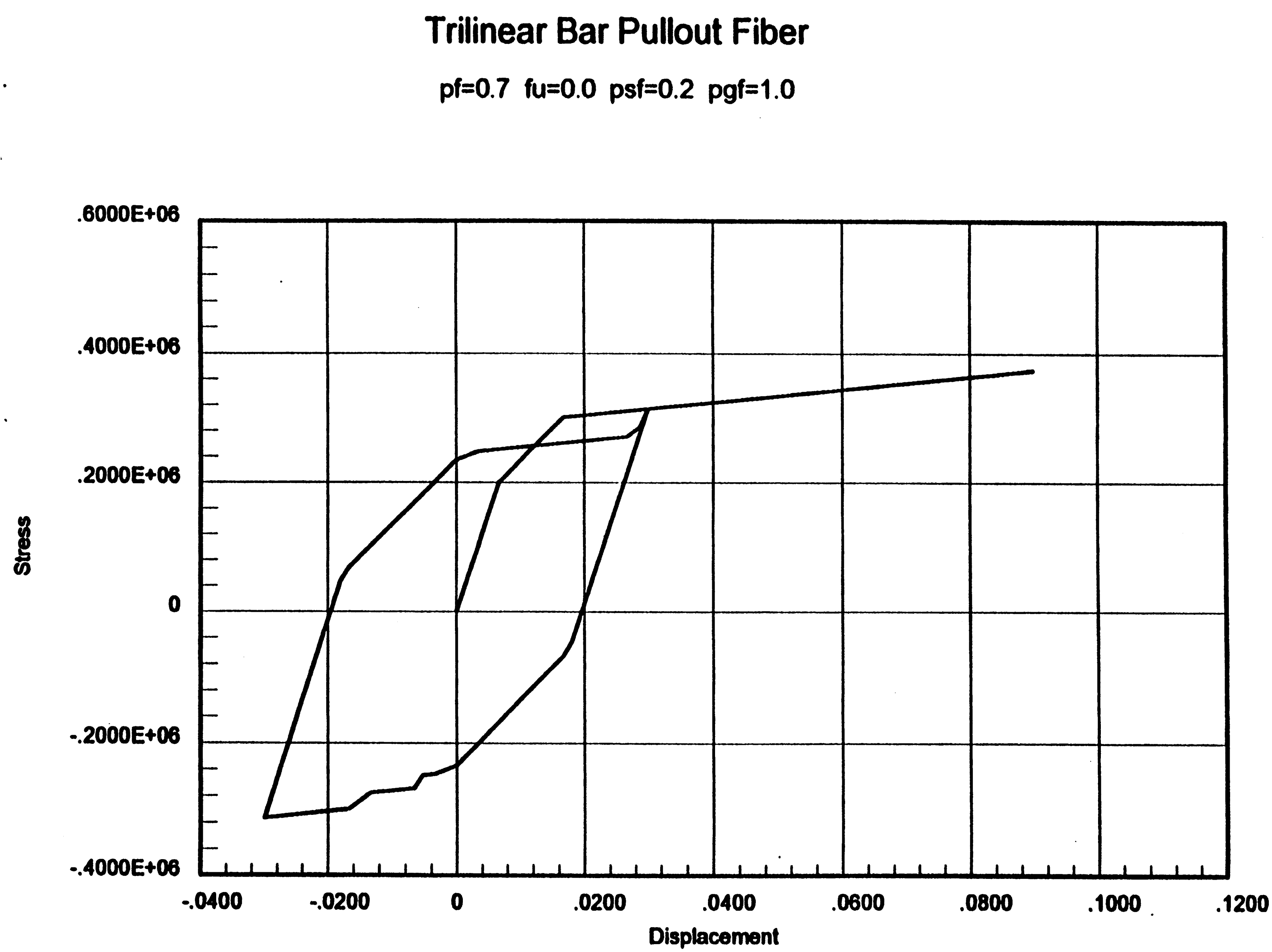
pf=0.2 fu=0.2 psf=0.5 pgf=1.0

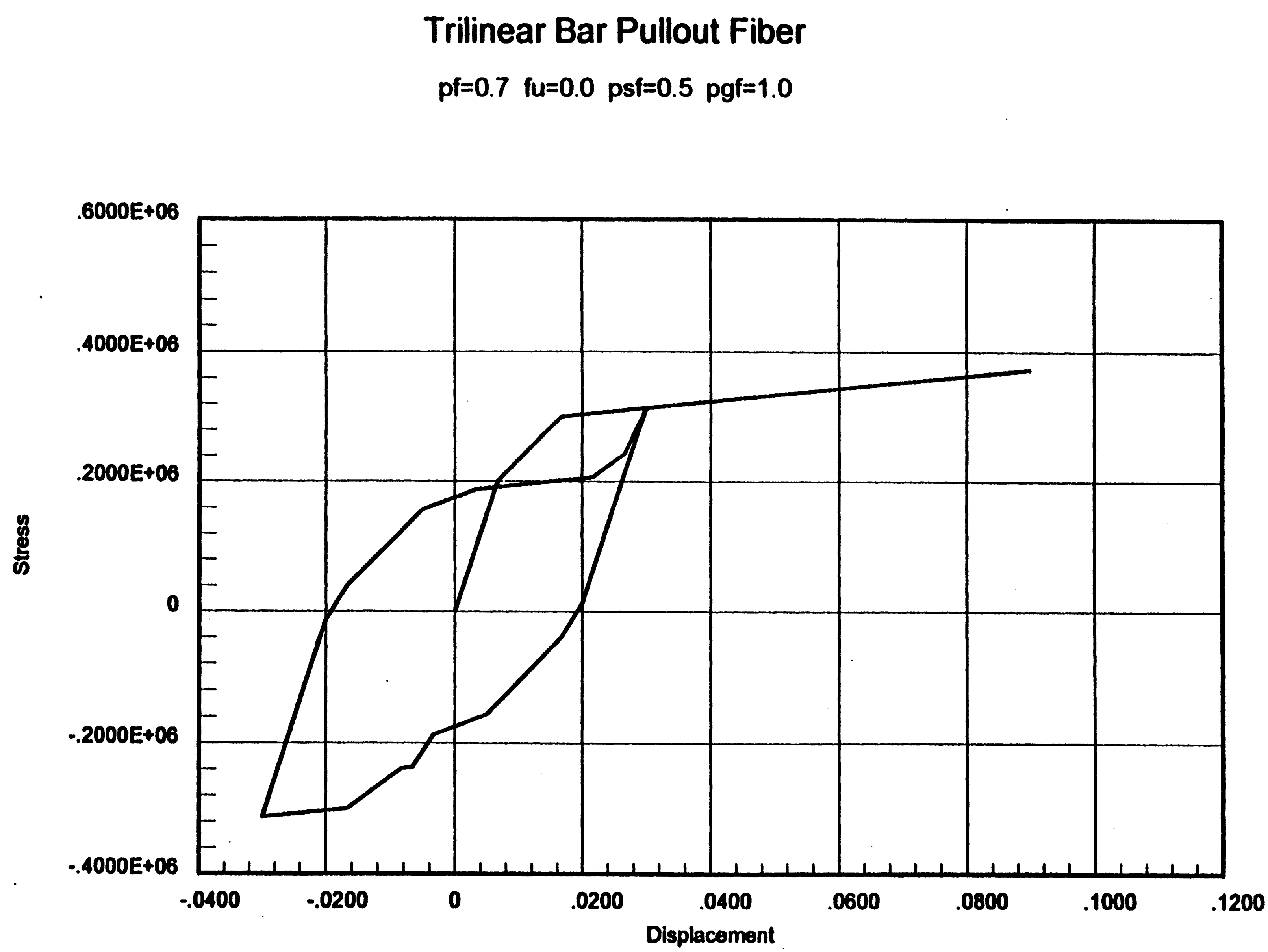


Trilinear Bar Pullout Fiber

pf=0.2 fu=0.7 psf=0.5 pgf=1.0



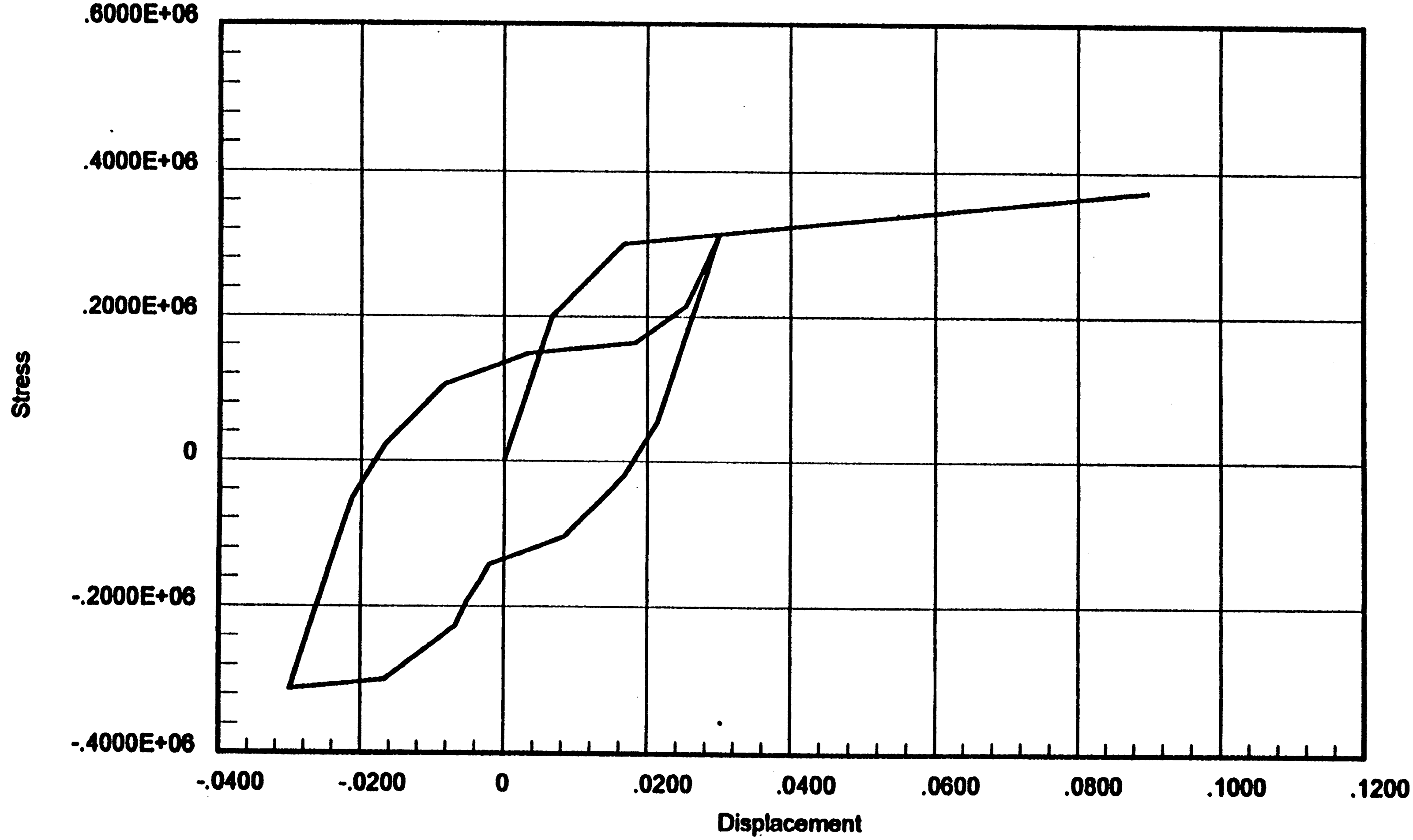

Trilinear Bar Pullout Fiber
pf=0.7 fu=0.0 psf=0.2 pgf=1.0
.6000E+06
.4000E+06
.2000E+06
0
-.2000E+06
-.4000E+06
Stress
-.0400
-.0200
0
.0200
.0400
.0600
.0800
.1000
.1200
Displacement

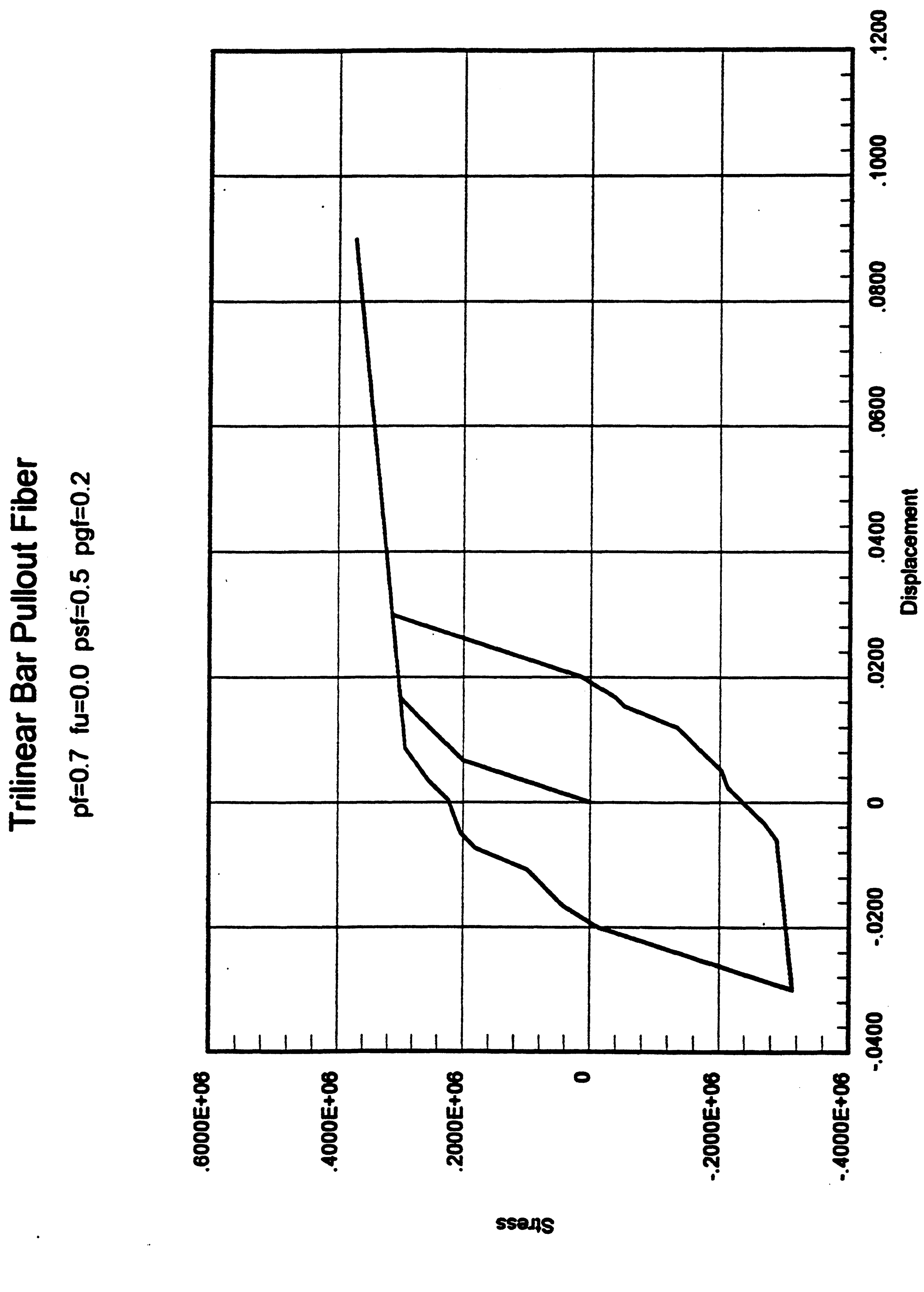


Trilinear Bar Pullout Fiber
pf=0.7 fu=0.0 psf=0.5 pgf=1.0
.6000E+06
.4000E+06
.2000E+06
0
-.2000E+06
-.4000E+06
Stress
-.0400
-.0200
0
.0200
.0400
.0600
.0800
.1000
.1200
Displacement

Trilinear Bar Pullout Fiber

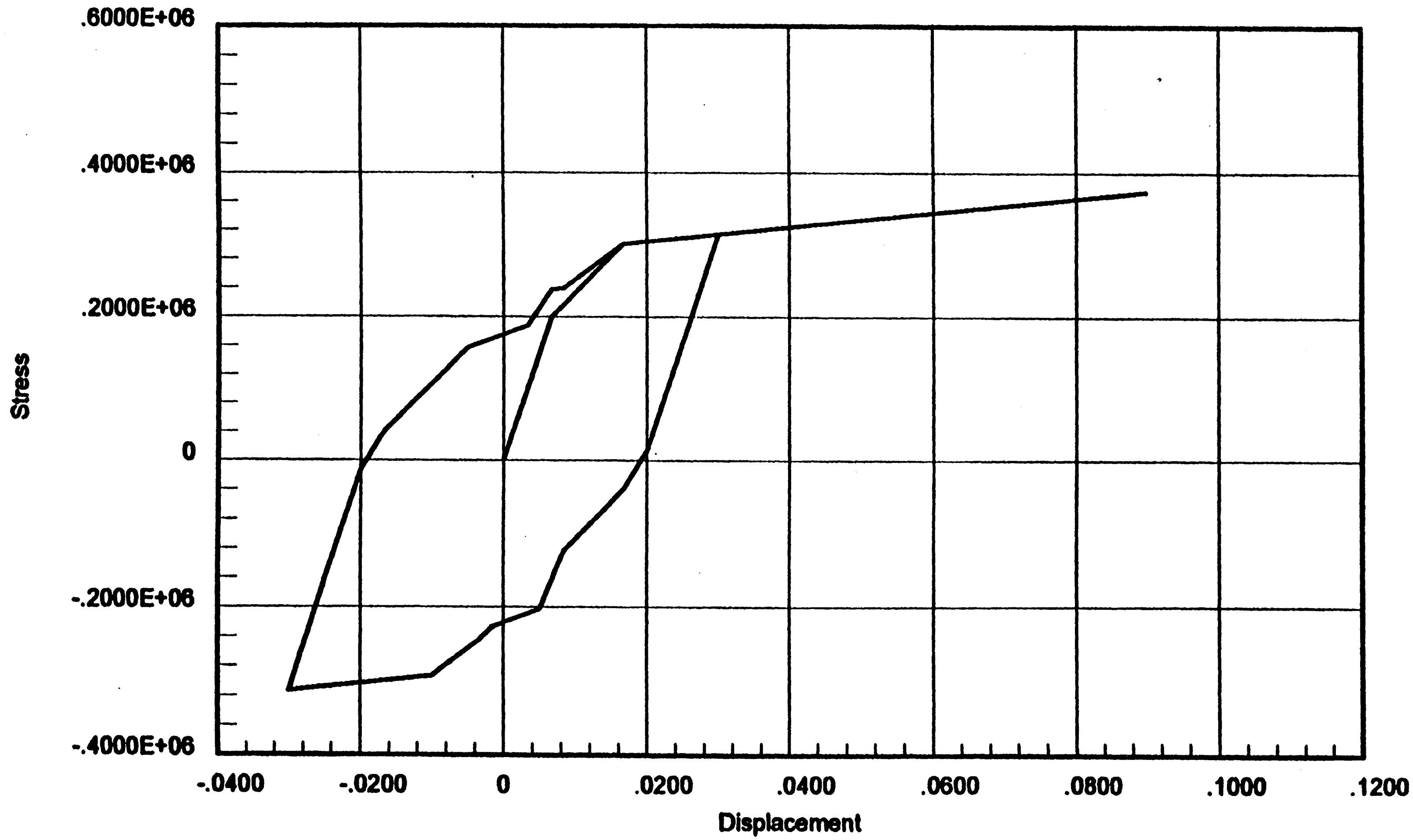
pf=0.7 fu=0.0 psf=0.7 pgf=1.0




Trilinear Bar Pullout Fiber
pf=0.7 fu=0.0 psf=0.5 pgf=0.2
Stress
.6000E+06
.4000E+06
.2000E+06
0
-.2000E+06
-.4000E+06
Displacement
-.0400
-.0200
0
.0200
.0400
.0600
.0800
.1000
.1200

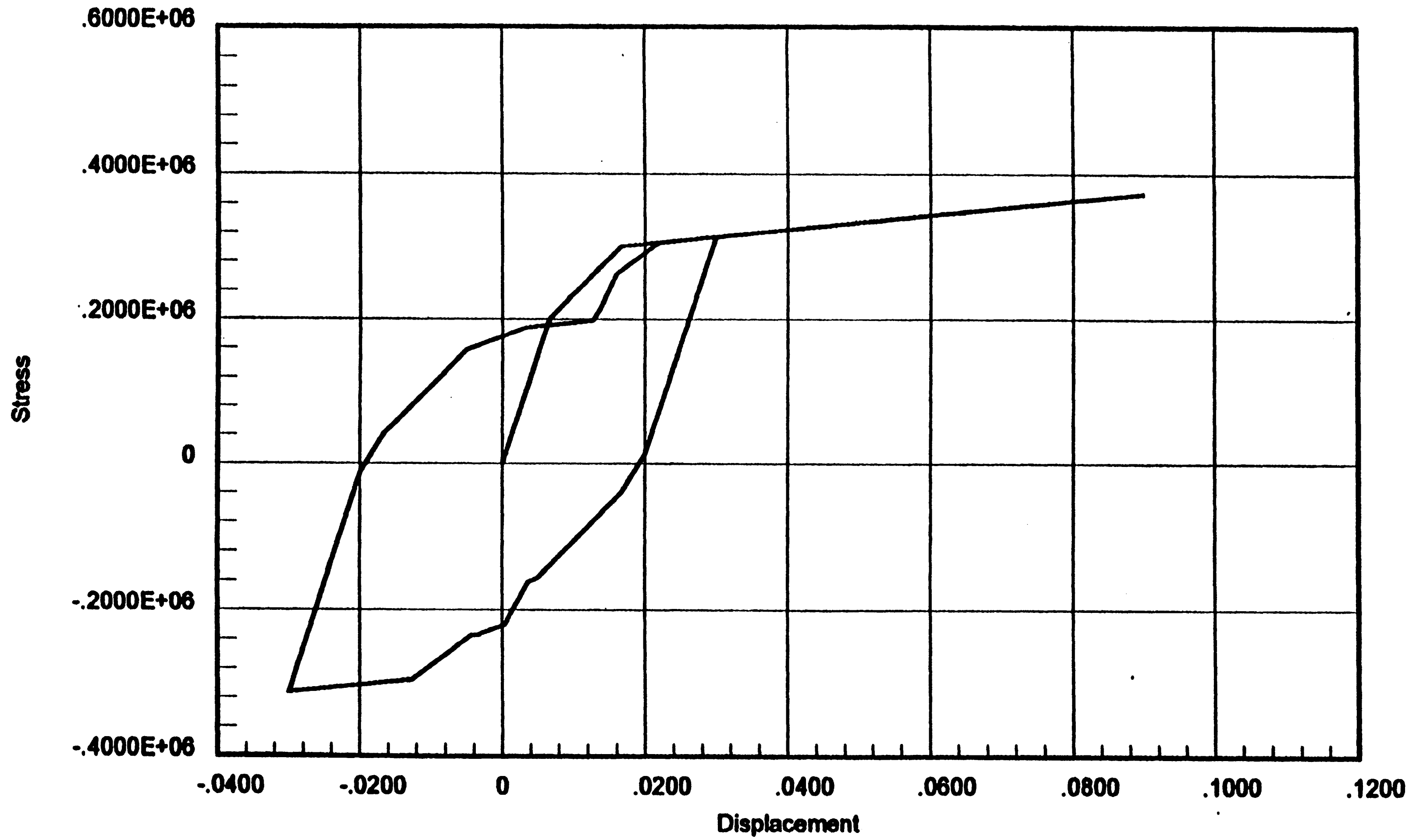
Trilinear Bar Pullout Fiber

pf=0.7 fu=0.0 psf=0.5 pgf=0.5



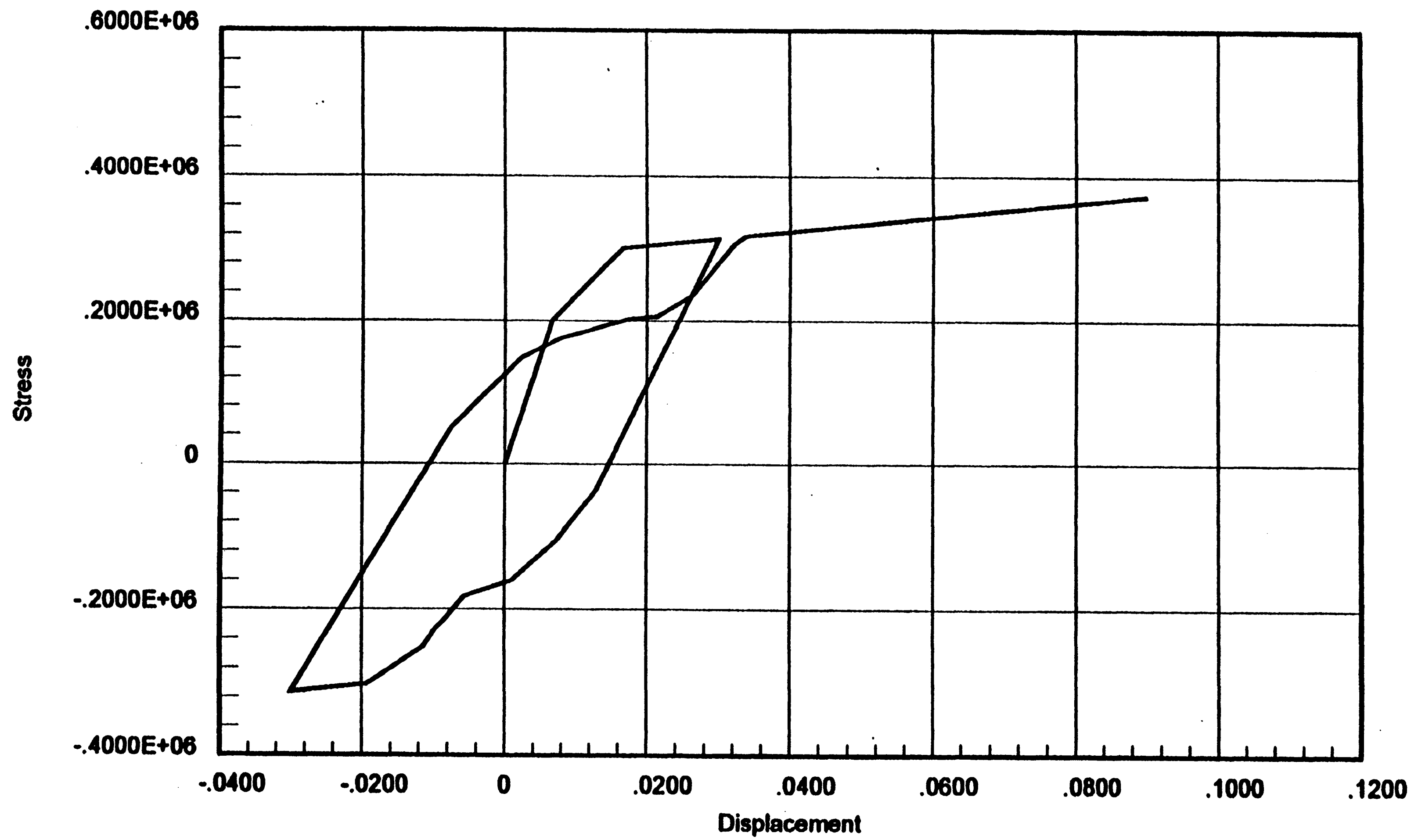
Trilinear Bar Pullout Fiber

pf=0.7 fu=0.0 psf=0.5 pgf=0.7



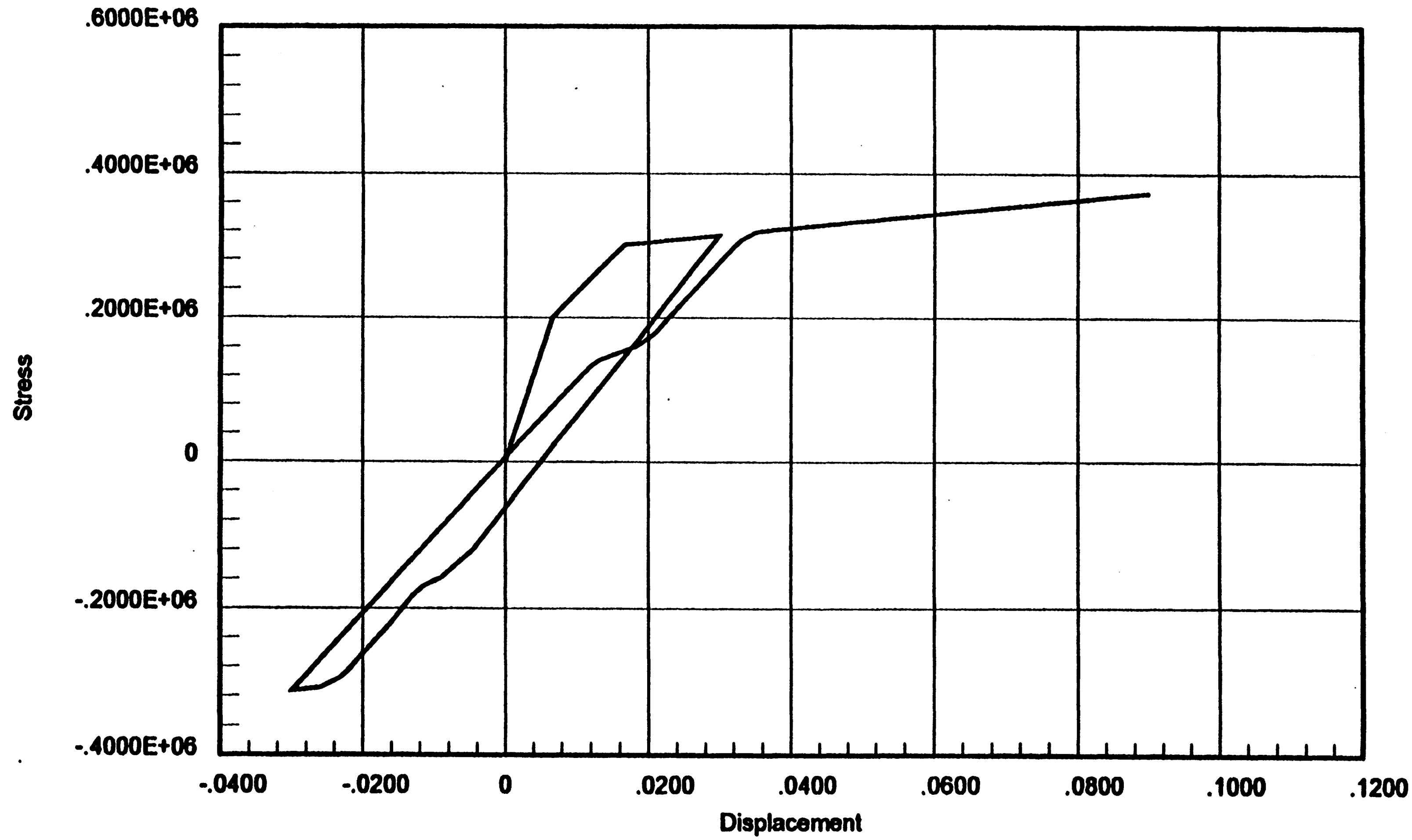
Trilinear Bar Pullout Fiber

pf=0.7 fu=0.2 psf=0.5 pgf=1.0



Trilinear Bar Pullout Fiber

pf=0.7 fu=0.7 psf=0.5 pgf=1.0



COMPRESSION/TENSION LINK ELEMENT (TYPE 09) FOR DRAIN-3DX

VERSION 1.10
MARCH 1994

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, as shown in Figure E09.1. It resists axial force only, and can be specified to act in tension (tension force and extension are positive) or in compression (compression force and 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. E09.2. 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 regardless of the value of β.. 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. 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, in the .OUT file, for each mode. 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 I.
6	Node J.
7	Line number (0,1,2,3 or 4, where 0 = slack or open gap, 1 - 4 = k1 - k4).

E09.3.6 USER OUTPUT (.USR FILE)

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

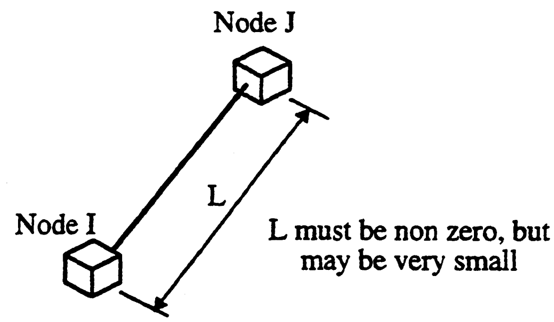
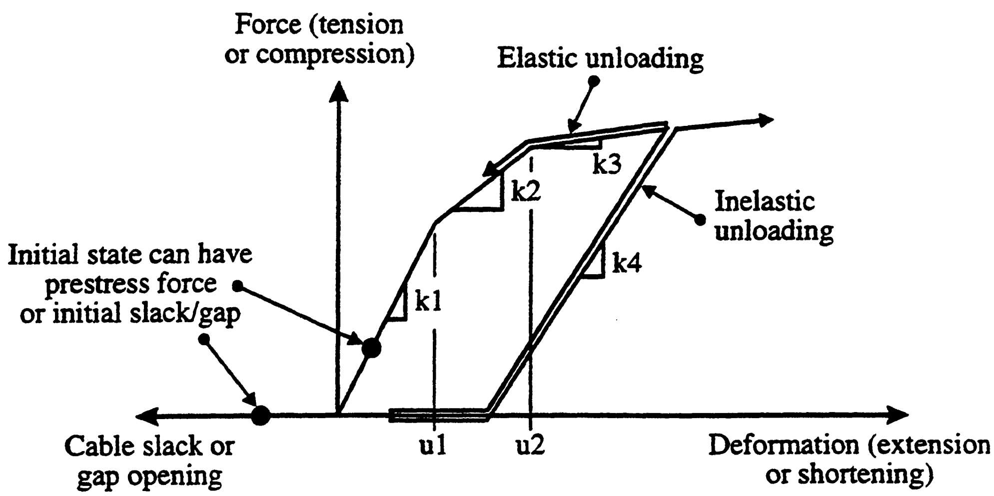


FIGURE E09.1 ELEMENT GEOMETRY



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

FIGURE E09.2 ELEMENT PROPERTIES

FIBER BEAM-COLUMN ELEMENT (TYPE 15) FOR DRAIN-3DX AND DRAIN-BUILDING

VERSION 1.10
JUNE 1994

ELEMENT DESCRIPTION AND USER GUIDE

E15.1 PURPOSE, FEATURES AND LIMITATIONS

E15.1.1 PURPOSE

The element can be used to model steel, reinforced concrete, or composite steel-concrete beams and columns. 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.

E15.1.2 ELEMENT MODEL

Figures E15.1, E15.2 and E15.3 show 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 stress-strain relationships of concrete or steel type. Figures E15.4 and E15.5 show the assumed properties for concrete and steel materials.

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 framing angles.

The element is assumed to be elastic in shear and torsion.

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.

The element model within the length of the deformable region is essentially 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". Plastic hinges can be modelled as a special case if desired.

The use of fibers to model cross sections accounts rationally for P-M-M interaction (axial force plus biaxial bending).

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 can increase dramatically.

If connection hinges are used to model connection deformations, these deformations are lumped at the ends of the deformable region. The fiber model allows for semi-rational modelling of connection behavior (as distinct from completely empirical modelling).

P-Δ effects can be included if desired.

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, 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) The behavior in torsion is assumed to be elastic, based on a specified shear modulus and effective torsional inertia.

(4) 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.

(5) 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.

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

E15.1.4 INITIAL STIFFNESS AND 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. The initial stiffness is calculated assuming that all concrete and gap fibers are in compression.

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.

E15.1.5 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 necessarily 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..

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 7000 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 is likely to be extremely 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. The element includes logic to do this, but if the strength loss is rapid (i.e., if the tangent moduli for a number of fibers have large negative values), this logic may not work, and the solution may not converge. If this happens a message will be printed in the .ECH file and execution will stop. If this happens, try specifying a less rapid rate of strength loss for the concrete materials.

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 USER GUIDE

See Chapter E15.1 for a description of the element features and limitations, and for warnings on use of the element. See Chapter E15.3 for results interpretation.

UG.15.1. 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 UG.15.2(a).
6-10(I)		NSMAT	No. of steel material types for fiber cross sections (may be 0). See Section UG.15.2(b).
11-15(I)		NFSEC	No. of fiber cross section types (may be 0). See Section UG.15.3(a).
16-20(I)		NESEC	No. of elastic cross section types (may be 0). See Section UG.15.3(b).
21-25(I)		NPMAT	No. of pullout property types for connection hinges (may be 0). See Section UG.15.4(a).
26-30(I)		NGMAT	No. of gap property types for connection hinges (may be 0). See Section UG.15.4(b).
31-35(I)		NCHIN	No. of connection hinge types (may be 0). See Section UG.15.5.
36-40(I)		NRIGZ	No. of rigid end zone types (may be 0). See Section UG.15.6.
41-45(I)		NETYP	No. of element geometry types (must be >0). See Section UG.15.7.

UG.15.2(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 E15.4
Concrete material types are numbered in input sequence.

UG.15.2(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).

UG.15.2(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.

UG.15.2(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.

UG.15.2(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 E15.5
Steel material types are numbered in input sequence.

UG.15.2(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).

UG.15.2(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.

UG.15.3(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 and z axes need not be the principal axes of the cross section, and the section centroid need not be at the x axis. However, the section shear center is assumed to be at the x axis.

UG.15.3(a)(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NFIBS	No. of fibers (min. 3, not all along one line).
6-15(R)			Torsional J or GJ. The element is assumed to be elastic in torsion.
16-25(R)			Shear A' or GA' for shear forces in local y direction. Default = no shear deformation.
26-35(R)			Shear A' or GA' for shear forces in local z direction. Default = no shear deformation.
36-45(R)			Shear modulus, G. Default = 1 (i.e., GJ and GA' are specified directly).

UG.15.3(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 z coordinate.
21-30(R)			Fiber Area.
33(C)			"C" if concrete, "S" if steel.
34-35(I)			Material number of this type. Example: "S03" in columns 33-35 = steel material type 3.

UG.15.3(b). Elastic Cross Section Types.

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

The element local y and z axes must be the cross section principal axes, and the section centroid and shear center must be at the x axis.

Columns	Notes	Variable	Data
1-10(R)			Torsional J or GJ.
11-20(R)			Flexural I or EI about local y axis (must be a principal axis).
21-30(R)			Flexural I or EI about local z axis.
31-40(R)			Axial A or EA.
41-50(R)			Shear A' or GA' for shear forces in local y direction. Default = no shear deformation.
51-60(R)			Shear A' or GA' for shear forces in local z direction. Default = no shear deformation.
61-70(R)			Young's modulus, E. Default = 1 (i.e., EA and EI are specified directly).
71-80(R)			Shear modulus, G. Default = 1 (i.e., GJ and GA' are specified directly).

UG.15.4(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. See Figure E15.6 for basic properties. See separate documentation for degradation parameters.
Pullout property types are numbered in input sequence.

Note that moduli are in terms of stress and *displacement*, not stress and *strain*.

UG.15.4(a)(i). Basic Properties

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 UG.15.4(a)(ii). If 1, material degrades. Include line UG.15.4(a)(ii).

UG.15.4(a)(ii). Degradation Parameters

Omit if degradation indicator, IDGD, is blank or 0.

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

UG.15.4(b). Gap Properties for Connection Hinge Fibers

NGMAT lines. Omit if NGMAT = 0. See Figure E15.7.
Gap property types are numbered in input sequence.

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).

UG.15.5. 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.

UG.15.5(i). Control Line.

Columns	Notes	Variables	Data
1-5(I)		NFIBH	No. of fibers (min. 3, not all along one line).

UG.15.5(ii). Fibers. NFIBH lines, one per fiber, in any order

Columns	Notes	Variable	Data
1-10(R)			Fiber y coordinate.
11-20(R)			Fiber z coordinate.
21-30(R)			Fiber Area.
33(C)			"P" if a bar pullout fiber, "G" if gap fiber.
34-35(I)			Property number of this type. E.g., "P03" in columns 33-35 = pullout property type 3.

UG.15.6. Rigid End Zone Types.

NRIGZ lines. Omit if NRIGZ = 0.
Rigid zone types are numbered in input sequence.
See Section UG.15.7(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.
21-30(R)			Global Z projection of rigid zone.

UG.15.7. Element Geometry Types.

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

UG.15.7(i). Control Line.

Columns	Notes	Variable	Data
1-5(I)		NSEG	Number of segments.
6-10(I)			Type number for connection 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 element 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 element end j. Default = none. If +, projections are from node to element end. If - , projections are from element end to node.

UG.15.7(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.

UG.15.8. 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.
36-45(I)			Node number of Point K, to orient local y axis. See Figure E15.1. Default = same as preceding element. If elements are generated, Point K is the same for all elements in the series.
46-50(I)			Element geometry type number. Default = same as preceding element.

NOTE ON NODE NUMBERING FOR INTERFLOOR ELEMENTS IN DRAIN-BUILDING

In DRAIN-BUILDING, an element which is part of an interfloor can connect to a node in Floor 1 of the interfloor, and/or a node in Floor 2 of the interfloor, and/or a node in the interfloor itself. The node location is indicated as follows.

- For an interfloor node, specify the node number.
- For a node in Floor 1, place a "-" immediately after the node number. For example, if the node is node number 123456 in Floor 1, specify 123456-.
- For a node in Floor 2, place a "+" immediately after the node number. For example, if the node is node number 123456 in Floor 2, specify 123456+.

This applies for Node I and Node J, but not for the node number increment.

E15.3 INTERPRETATION OF RESULTS. USER OUTPUT

E15.3.1 ENVELOPE OUTPUT (.OUT AND .E** FILES)

To be added.

E15.3.2 TIME HISTORY PRINTOUT (.OUT FILE)

To be added.

E15.3.3 TIME HISTORY POST-PROCESSING (.RXX FILE)

To be added.

E15.3.4 USER OUTPUT (.USR FILE)

To be added.

E15.4 THEORY

To be added.

E15.5 EXAMPLES, APPLICATIONS AND GUIDELINES

To be added.

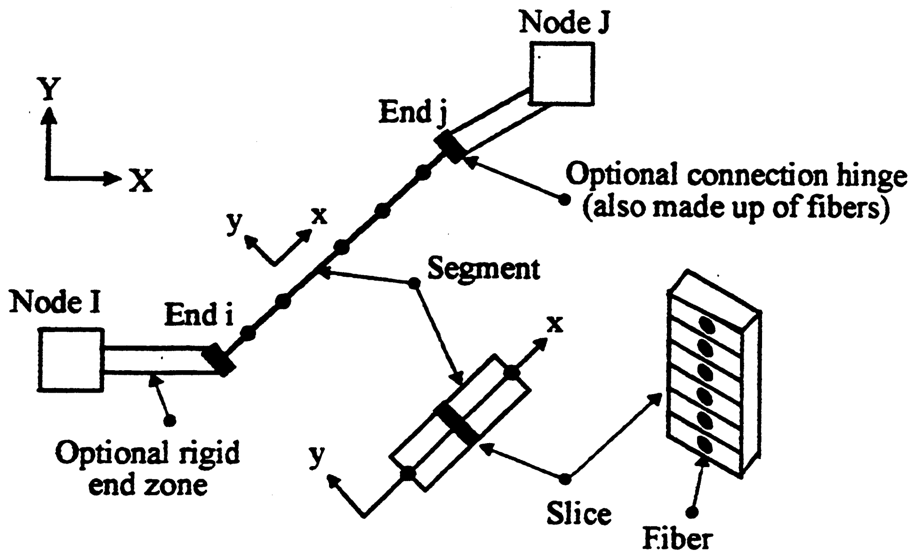


FIGURE E15.1 ELEMENT MODEL

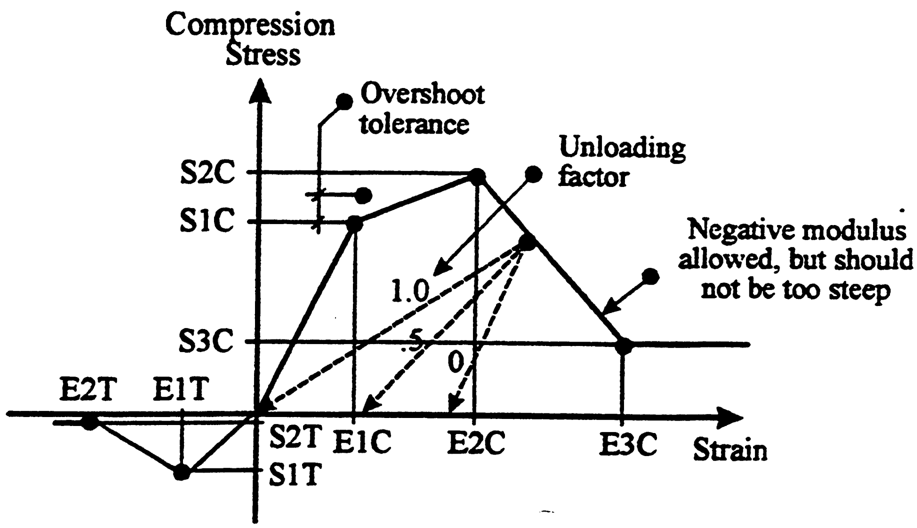


FIGURE E15.2 CONCRETE MATERIAL PROPERTIES

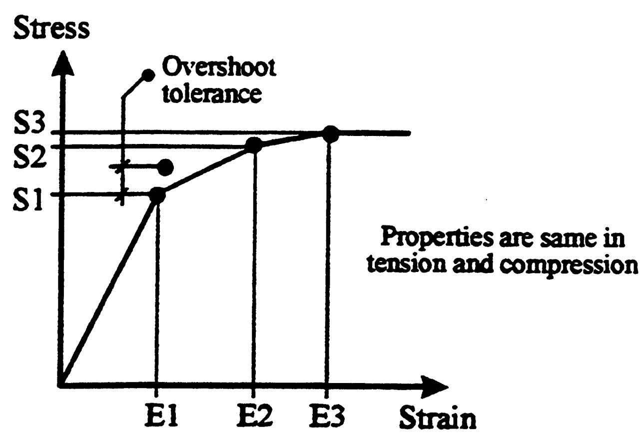


FIGURE E15.3 STEEL MATERIAL PROPERTIES

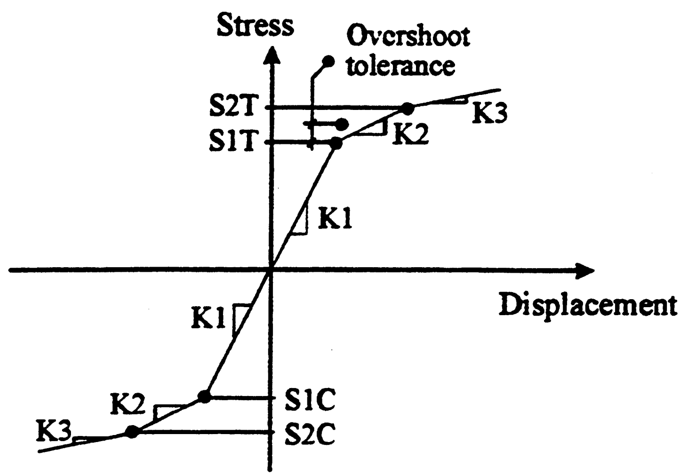
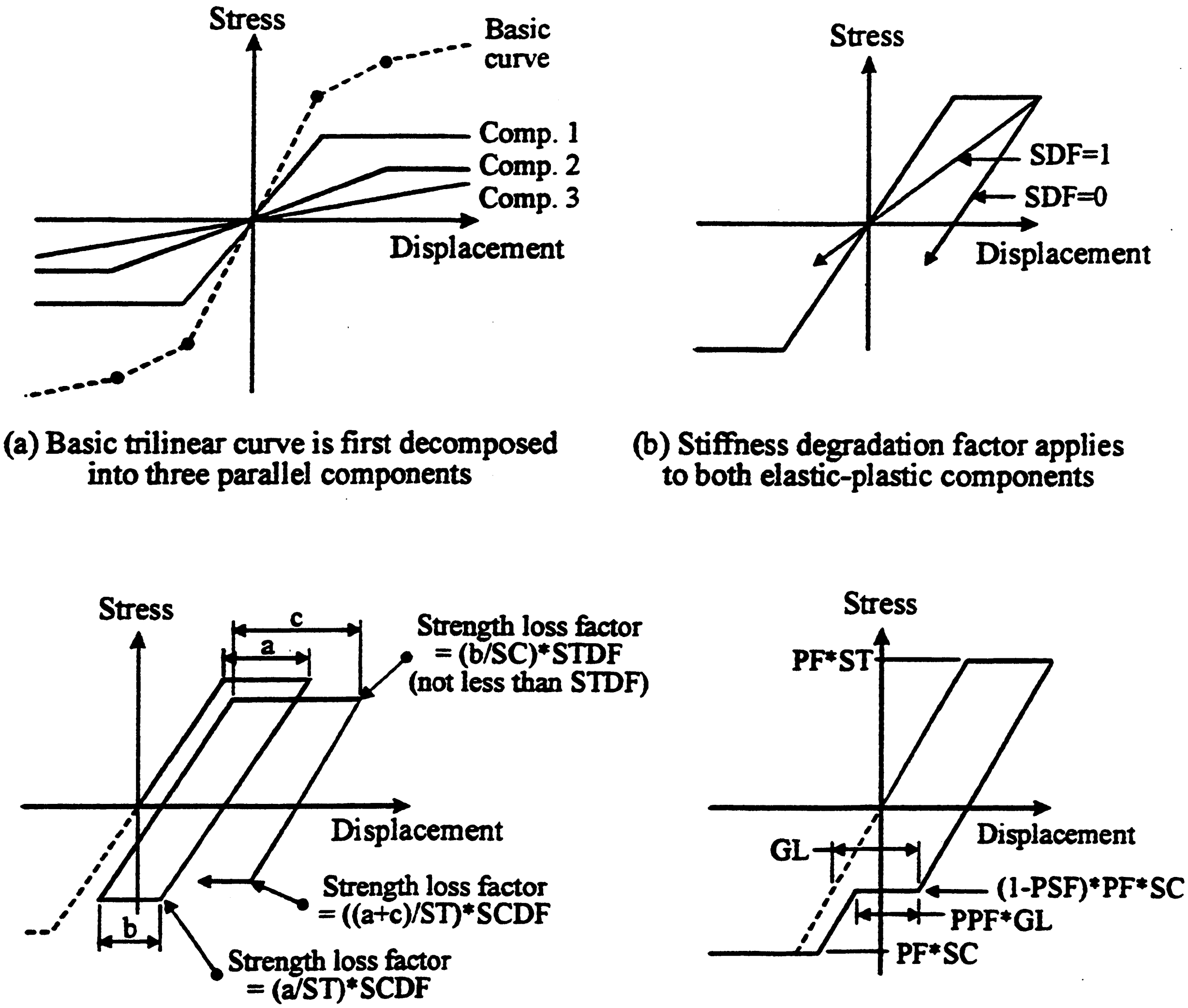


FIGURE E15.4 PULLOUT FIBER BASIC PROPERTIES



(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

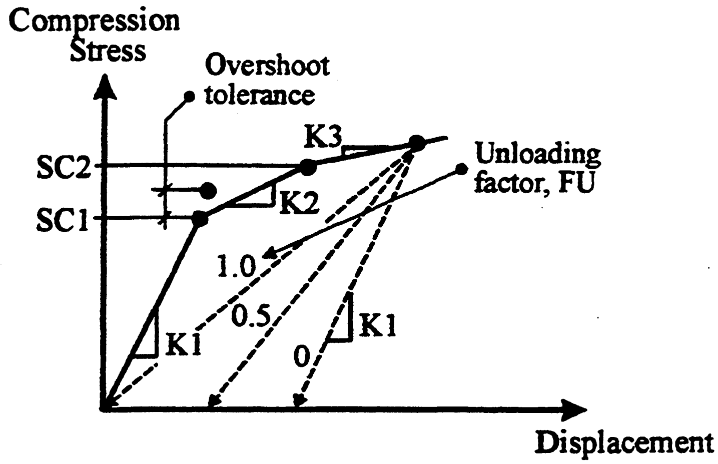
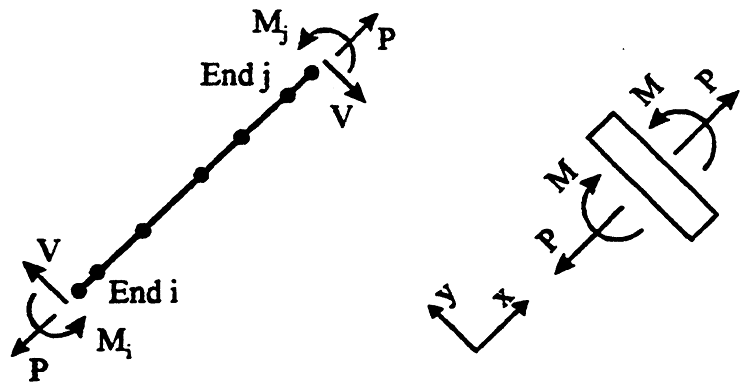


FIGURE E15.6 GAP FIBER PROPERTIES



(a) Element ends

(b) Slice or hinge

FIGURE E15.7 SIGN CONVENTIONS

ELASTIC BEAM-COLUMN ELEMENT (TYPE 17) FOR DRAIN-3DX AND DRAIN-BLDG

VERSION 1.10
MARCH 1994

ELEMENT DESCRIPTION AND USER GUIDE

E17.1 PURPOSE, FEATURES AND LIMITATIONS

E17.1.1 PURPOSE

This is a simple linear beam-column element.

E17.1.2 ELEMENT MODEL

The element model is shown in Figure E17.1.

Element loads can be included.

P-Δ effects can not be included.

E17.1.3 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 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, in the .OUT file, for each mode. 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.

E17.1.4 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 E17.2. Fixed end forces can also be used to account for thermal expansion effects, and, for an element that is a pipe, the effects of internal pressure. The fixed end forces 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. For building design, the live load reduction factor permitted for a column will often exceed that for the beams that 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 force reduction factors for the fixed end forces, which are used as follows.

For the shear and axial forces shown in the element results, the full specified fixed end forces are used. However, for static loads on the nodes that connect to the element, the fixed end shear and axial forces (but not the moments) are 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 force reduction factors are applied to the fixed end shear and axial forces before they are transferred to the nodes.

E17.2 INPUT DATA FOR *ELEMENTGROUP

Figure E17.1 shows the element model.

E17.2.1. Control Information

One line

Columns	Notes	Variable	Data
1-5(I)		NMAT	No. of different materials (max 10). See Section E17.2.2.
6-10(I)		NSECT	No. of different cross sections (max. 10). See Section E17.2.3.
11-15(I)		NSTIF	No. of different stiffness factor sets (max. 10, may be zero). See Section 17.2.4.
16-20(I)		NRIG	No. of rigid end zone types (max. 10). See Section E17.2.5.

E17.2.2. Material Properties

NMAT lines, one line per material.

Columns	Notes	Variable	Data
1-5(I)			Material number (in sequence starting with 1).
6-15(R)			Young's modulus.
16-25(R)			Shear modulus (used for torsional and shear stiffnesses).

E17.2.3. Cross Section Properties.

NSECT lines, one line per section.

Columns	Notes	Variables	Data
1-5(I)			Section number (in sequence starting with 1)
6-15(R)			Torsional inertia.
16-25(R)			Bending inertia about local y axis (must be a principal axis).
26-35(R)			Bending inertia about local z axis.
36-45(R)			Cross section area.
46-55(R)			Shear area for shear along local y axis. Default = no shear deformation.
56-65(R)			Shear area for shear along local z axis. Default = no shear deformation.

E17.2.4. Stiffness Factors.

NSTIF lines, one line per section. Omit if NSTIF = 0.

The 2x2 bending stiffness matrix is:

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

If the stiffness factor set for any element is 0 or blank (see Section E17.2.6), the default values $c_{ii} = c_{jj} = 4$ and $c_{ij} = 2$ are used (i.e., uniform section).

Columns	Notes	Variable	Data
1-5(I)			Stiffness factor set no. (in sequence starting with 1).
6-15(R)			c_{ii}.
16-25(R)			c_{jj}.
26-35(R)			c_{ij}.

E17.2.5. Rigid End Zones.

NRIG lines. Omit if NRIG = 0. See Section E17.2.6 for use of rigid zones.

Columns	Notes	Variable	Data
1-5(I)			Rigid zone number (in sequence starting with 1)
6-15(R)			Global X projection of rigid zone.
16-25(R)			Global Y projection of rigid zone.
26-35(R)			Global Z projection of rigid zone.

E17.6. 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. See following note for interfloor elements in DRAIN-BUILDING.
16-25(I)			Node number at element end J.
26-35(I)			Node number increment for element generation. Default = 1.
36-45(I)			Node number for Point K to orient element y and z axes.
46-50(I)			Material number.
51-55(I)			Cross section number.
56-60(I)			Stiffness factor set number for bending about y axis. (0 = uniform section).
61-65(I)			Stiffness factor set number for bending about z axis. (0 = uniform section).
66-70(I)			Rigid zone number at end I: Zero = none. Positive = offsets are from node to element. Negative = offsets are from element to node.
71-75			Rigid end number at end J (0, + or -).
77(I)			End release code for y axis bending at end I. 0 = no release. 1 = release.
78(I)			End release code for y axis bending at end J.
79(I)			End release code for z axis bending at end I.
80(I)			End release code for z axis bending at end J.

NOTE ON NODE NUMBERING FOR INTERFLOOR ELEMENTS IN DRAIN-BUILDING

In DRAIN-BUILDING, an element which is part of an interfloor can connect to a node in Floor 1 of the interfloor, and/or a node in Floor 2 of the interfloor, and/or a node in the interfloor itself. The node location is indicated as follows.

- For an interfloor node, specify the node number.
- For a node in Floor 1, place a "-" immediately after the node number. For example, if the node is node number 123456 in Floor 1, specify 123456-.
- For a node in Floor 2, place a "+" immediately after the node number. For example, if the node is node number 123456 in Floor 2, specify 123456+.

This applies for Node I, Node J and Node K, but not for the node number increment.

E17.3 INPUT DATA FOR *ELEMENTLOAD

E17.3.1 Load Sets

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

See Figure E17.2 for sign convention. These are forces that must act ***on the element ends*** to prevent end displacement.

Line 1

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 reduction factor.
21-30(R)			Force Pi.
31-40(R)			Force Vyi.
41-50(R)			Force Vzi.
51-60(R)			Moment Ti.
61-70(R)			Moment Myi.
71-80(R)			Moment Mzi.

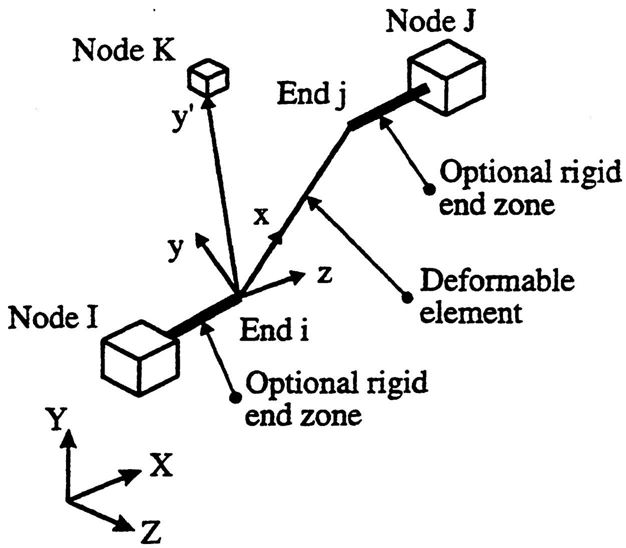
Line 2

Columns	Notes	Variable	Data
1-20			Leave blank.
21-30(R)			Force Pj.
31-40(R)			Force Vyj.
41-50(R)			Force Vzj.
51-60(R)			Moment Tj.
61-70(R)			Moment Myj.
71-80(R)			Moment Mzj.

E17.3.2 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.



Note on Node K:

Local x axis is from end i to end j.

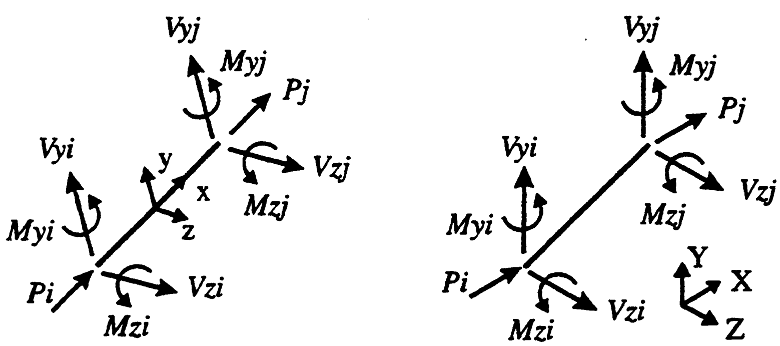
Local y' axis is from end i to node K.

Local x and y' axes define xy plane.

Local y axis is normal to x in xy plane.

Local z axis is normal to both x and y.

FIGURE E17.1 OVERALL ELEMENT GEOMETRY



(a) Code = 0 (local) **(b) Code =1 (global)**

FIGURE E17.2 FIXED END FORCES