13th World Conference on Earthquake Engineering
Vancouver, B.C., Canada
August 1-6, 2004
Paper No. 2530

UIUC EXTENSIONS TO THE LIBRARY OF ELEMENTS FOR DRAIN-2DX

Omer O. ERBAY[1], Seung-Yul YUN[2], Douglas A. FOUTCH[3], and Mark ASCHHEIM[4]

SUMMARY

Extensions to the library of elements in the nonlinear structural analysis computer program, DRAIN-2DX, were developed as part of research activities at the University of Illinois at Urbana-Champaign. Two new elements were added: a connection element capable of modeling connection fracture and degradation for steel moment frames and a pinching and stiffness degrading element capable of modeling symmetric and non-symmetric reinforced concrete members. The features and capabilities of each element are discussed and illustrated with examples.

INTRODUCTION

DRAIN-2DX is a general purpose computer program for static and dynamic analysis of plane structures. The program was originally developed by Kanaan and Powell [4] as DRAIN-2D. The code was restructured to its current form, known as DRAIN-2DX, by Allahabadi and Powell [1] and Prakash *et. al.* [8, 9]. Both linear and nonlinear static and dynamic analyses can be performed with DRAIN-2DX. Dynamic loads can be imposed as ground accelerations, ground displacements, and initial velocities. In addition to static and dynamic analyses, DRAIN-2DX can be used to compute mode shapes and periods of vibration, and can be used to and carry out linear response spectrum analyses.

Nonlinear response is captured through a step-by-step solution scheme in which imposed loads (static or dynamic) are incrementally applied to the model of the structure. The load increments can be either variable (events are considered within steps) or fixed (events are not considered within steps). An event is defined as any kind of state change that causes a change in the stiffness of the structure. In the variable load increment case, the program initially utilizes the load increment that is defined by the user and reduces it until the error in calculated response at the end of the analysis step is less than a defined

[1] Senior Engineer, Ph.D., Simpson Gumpertz & Heger, Waltham, MA, USA. Email: OOErbay@sgh.com.
[2] Research Engineer, Caterpillar, South Research Park, University of Illinois at Urbana-Champaign, Urbana, IL, USA. Email: yunseungyul@yahoo.com.
[3] Professor of Civil and Environmental Engineering, University of Illinois at Urbana-Champaign, Urbana, IL, USA. Email: d-foutch@uiuc.edu.
[4] Associate Professor of Civil Engineering, Santa Clara University, Santa Clara, CA, USA. Email: maschheim@scu.edu.

tolerance level. In the fixed load increment case, user specified load increment is used and kept constant throughout the analysis.

The DRAIN-2DX program comes with a built-in element library. The element library contains an inelastic truss bar (Element 01), a simple inelastic beam column element (Element 02), a simple inelastic translational and rotational connection spring (Element 04), an elastic panel element (Element 06), an inelastic link element that can act in compression/tension with initial gap or axial force (Element 09), and a fiber beam-column element for steel and reinforced concrete members (Element 15). In addition to these elements, the main program is structured to allow addition of new elements. Detailed discussion on the incorporation of new elements to DRAIN-2DX can be found in Prakash [8].

Following the guidelines provided in Prakash [8], two new elements were developed as part of the research efforts at the University of Illinois at Urbana-Champaign. The new elements include a connection element capable of modeling connection fracture and degradation for steel moment frames and a pinching and stiffness degrading element capable of modeling symmetric and non-symmetric reinforced concrete members such as Tee beams and walls. In the following sections, the features and the capabilities of each element are discussed and illustrated with examples. The extended version of the element library, the associated FORTRAN source codes, and the executable DRAIN-2DX file can be downloaded from the National Information Service for Earthquake Engineering (NISEE) web site, NISEE [5].

CONNECTION ELEMENT FOR STEEL MOMENT FRAMES

Introduction

The steel frame connection element, Element 10, is a simple inelastic element for modeling structural connections with rotational and translational flexibility. The element was developed as part of the SAC steel project. The connection element that comes with DRAIN-2DX, Element 04 (Powell [7] and Prakash *et. al.* [9]), was taken as the basis and extended to include four additional "elasticity codes." The additional features can represent ductile flexural behavior, connection fracture, and degradation of steel moment frame connections. Element parameters allow some or all of these behaviors to be represented.

Element model

The Element 10 formulation follows the Element 04 formulation. The nodes that are connected by the element must have identical coordinates (i.e. this is a zero-length element). The element can connect either the rotational displacements of the nodes or the translational displacements. Positive actions (moments or forces) and deformations as well as available connection types are shown in Figure 1. For a translational connection, the connected displacement should be aligned with the global coordinate axes (e.g. horizontal or vertical displacements), displacements aligned in non-coordinate directions cannot be connected with the element.

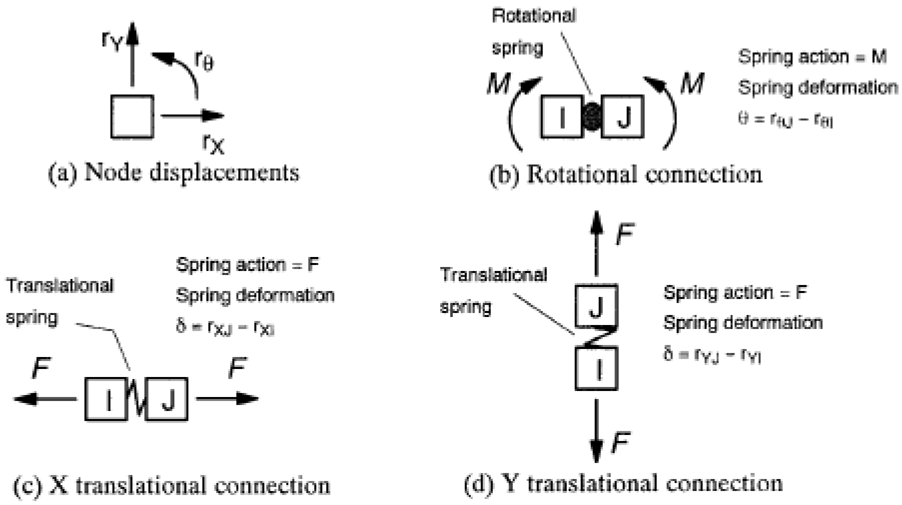


Figure 1 Connection types and sign convention for positive deformations (Figure adapted from Powell [7]).

The Park and Ang [6] damage index model was used to model the effect of low cycle fatigue on fracture of a connection. The damage index expressed below, Equation 1, is a linear combination of the damage caused by excessive deformation and the dissipated hysteretic energy. A positive and a negative fracture deformation under monotonic loading (δ_u^+ and δ_u^-) are used as input. The input parameters β and α are the linear and the power coefficient of dissipated hysteretic energy, respectively.

$$Damage\ Index,\ DI = \frac{\delta_m}{\delta_u} + \frac{\beta}{M_y \delta_u} \int dE \qquad (1)$$

where, δ_m = maximum response deformation under earthquake

δ_u = ultimate deformation capacity under monotonic loading

M_y = calculated yield moment

dE = incremental dissipated hysteretic energy

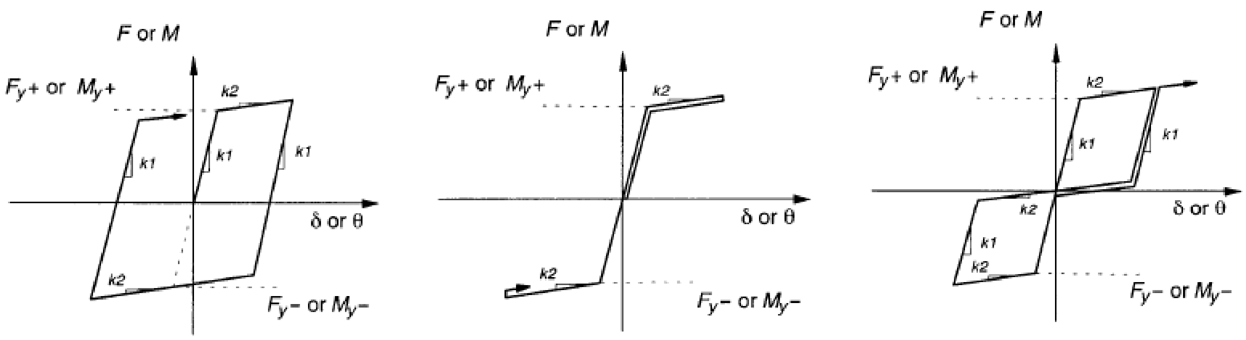
β = non-negative parameter

α = non-negative parameter

The details of the element formulation are beyond the scope of this paper. Interested readers may refer to Shi [10].

Element features

As mentioned earlier, Element 10 is an extended version of Element 04. For this reason, Element 10 includes all the force-deformation relationship models that Element 04 has. These models are summarized in Figure 2 and basically include bilinear inelastic, bilinear elastic, and bilinear inelastic with gap force-deformation relationships.



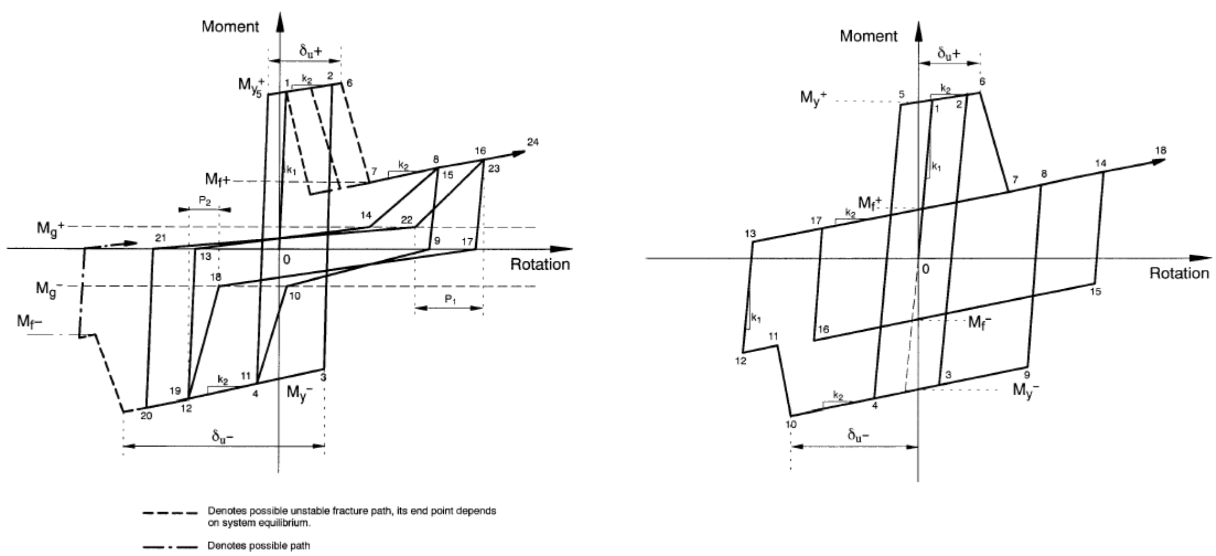
(a) Bilinear inelastic (Elasticity code = 0) | (b) Bilinear elastic (Elasticity code = 1) | (c) Bilinear inelastic with gap (Elasticity code = 2)

Figure 2 Force-deformation relationships in Element 04 (Figures taken from Powell [7]).

In addition to these models, Element 10 also includes four new force-deformation relationships. These relationships include two models to simulate the failure behavior of steel connections, another model to represent the behavior of reinforced concrete connections, and a final model to represent the behavior of panel zones at steel connections. Figure 3 summarizes the features and the hysteresis curves associated with each model.

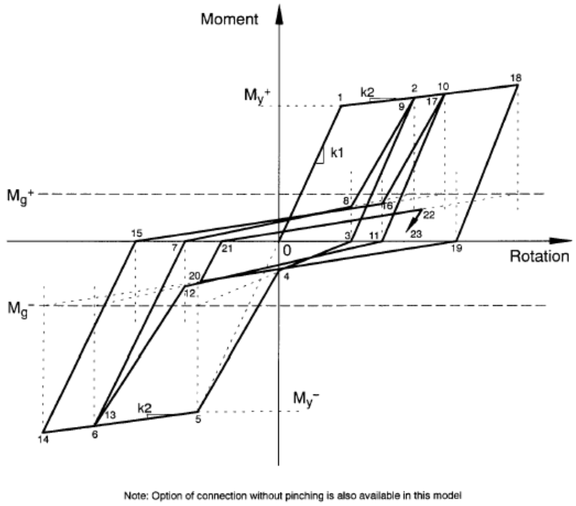
The main difference between the steel connection models in Figure 3a and 3b is that pinching behavior can be represented in the first model. Furthermore the first model can only be used in dynamic analysis whereas the second model can be used in both dynamic and static analyses.

For model simulating fracture in the steel connections, a modification was required in the solution algorithm in order to account for the force-unbalance associated with brittle connection fracture. The modified code is compatible with existing DRAIN-2DX elements.

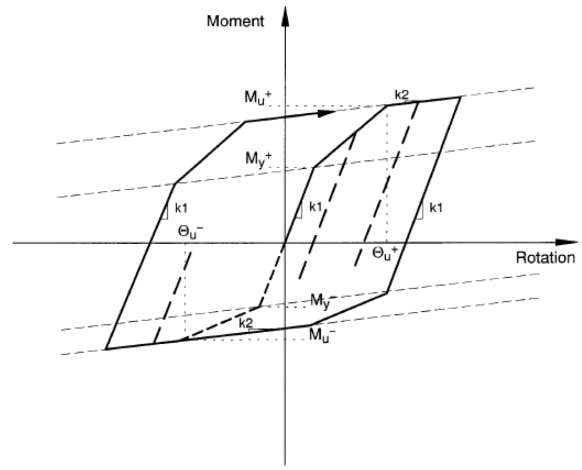


(a) Steel moment connection hysteresis model (Elasticity code = 3) | (b) Steel moment connection hysteresis model (Elasticity code = -3)

Figure 3 Force-deformation relationships implemented in Element 10.



(c) Concrete connection hysteresis model
(Elasticity code = 4)



(d) Steel panel zone tri-linear hysteresis model
(Elasticity code = 5)

Figure 3 Force-deformation relationships implemented in Element 10 (continued).

BEAM ELEMENT FOR SYMMETRIC AND NON-SYMMETRIC REINFORCED CONCRETE MEMBERS

Introduction

The reinforced concrete plastic hinge element, Element 07, uses discrete plastic hinges at the ends of the element to represent the behavior of ductile reinforced concrete beams and walls. The element is capable of representing non-symmetric cross sections (e.g. Tee beams and flanged walls), stiffness degradation (based on Takeda model [11), and pinching. The lack of symmetry of the cross section leads to differences in the elastic cracked section stiffness and strength for positive and negative moment. A special solution scheme is introduced to reformulate the stiffness matrix based on movement of the inflection point for this element.

This element is partially based on the work of Tang and Goel [12], who implemented a stiffness-degrading element in the program Drain-2DM and subsequent modifications made by Hueste and Wight [3].

Element model

The conceptual reinforced concrete member that the element is intended to represent is shown in Figure 4. The geometric features of the element are also shown in the same figure. In simple terms, the element formulation consists of an elastic beam connected in series to rigid-plastic hinges located at either end of the beam. Rigid end zones may be specified, and elastic shear deformations can be included by specifying an effective shear area. Although the element computations are based on rigid-plastic hinges, the input data are based on the moment-curvature response determined for cross sections located at either end of the beam. The flexural stiffnesses at each end of the beam, which may differ for positive and negative moment, are assumed to extend to midspan, exclusive of rigid end offsets, if present.

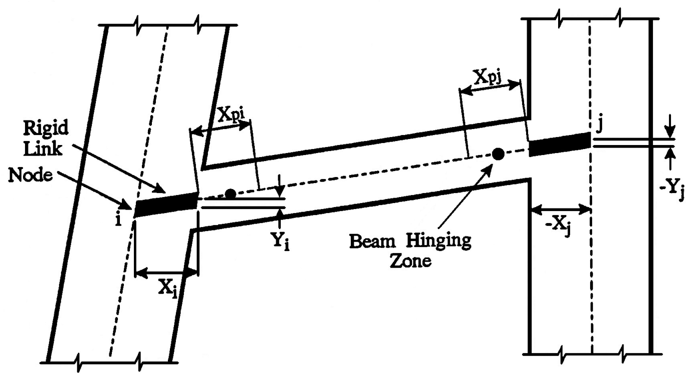


Figure 4 Conceptual reinforced concrete member and the geometric properties of the element (Figure adopted from Hueste and Wight [3]).

Yielding takes place only at the plastic hinges. Each plastic hinge may have a different elastic stiffness and yield strength for positive and negative bending, in order to represent the cracked section stiffness and flexural strength of non-symmertric reinforced concrete members under positive and negative moment. Uncracked behavior is not modeled; rather, the member ends are assumed to be pre-cracked. The plastic hinges are assumed to yield only in bending; inelastic axial deformation (e.g. gap opening) is not modeled. This corresponds to plastic flow along the M direction only, not along the normal to the yield surface of a P-M interaction diagram. The hysteretic behavior of the hinge is based on the Takeda model. The user may specify the exponent of the decay function to mimic the degradation in the unloading stiffness. However, the Takeda rules were not implemented for the small cycles that occur when unloading commences prior to reaching a previous peak. For these small unloading cycles, the response follows the unloading slope calculated for the most recent peak in the response quadrant. Additional changes were made to introduce pinching to the hysteretic behavior. Further information on the details of the element formulation can be found in Erbay and Aschheim [2]

Status checks incorporated in the element were primarily developed for an event-to-event solution strategy. Two overshoot tolerance values are defined to capture two different types of events; each event forces the structure stiffness matrix to be re-formed. Small or zero overshoot tolerances are used for moment overshoot and small overshoot tolerances are used for movement of the inflection point. The response of each element can be tracked accurately using an event-to-event solution strategy; other solution strategies that do not track events may be used but can be expected to be less accurate.

The first type of event concerns the acceptable error in moment associated with overshoot in the non-linear hysteretic response of the plastic hinge. As is in Element 02 (Powell [7]), a change in stiffness can introduce error if the stiffness matrix is not updated. An event is called to prevent the error in the calculated moment from exceeding the specified overshoot tolerance. The change in stiffness associated with each event may be due to yielding, inelastic unloading, etc.

The second type of event concerns movement of the inflection point (IP). Because the flexural stiffnesses for positive and negative moment may differ (Figure 5), even for elastic response, any movement of the inflection point is associated with a change in the stiffness of the element. The element stiffness is reformulated each time the inflection point moves beyond the specified tolerance. The reformulation uses the new location of the IP to identify the segments of the member where the positive or negative member properties are active.

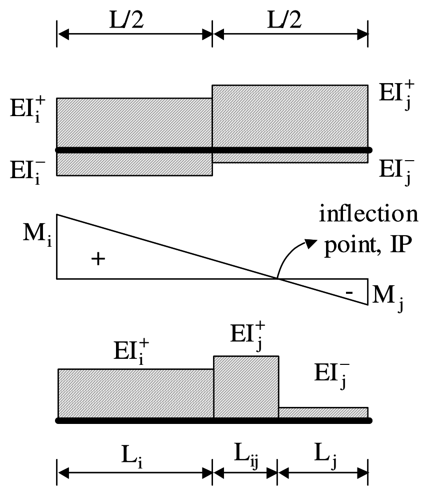


Figure 5 Possible flexural stiffness variation over the length of the element due to unequal positive and negative stiffness values along the element length.

The way the IP tolerance is utilized can be explained as follows: Consider the analysis state where the moment distribution over the member length is as shown in Figure 6a. At this state the location of IP is at point "a". Based on the current stiffness matrix, subsequent steps in the analysis may cause the IP to move to location "b" (Figure 6b). Movement of the IP location changes the lengths of the segments over which the positive or negative member properties are active. This change causes a change in the element stiffness, which therefore changes the element response. If the element stiffness is reformulated based on the IP being at “b”, the calculated moment increments at the member end will differ, in general, as indicated by the dashed line in Figure 6b. The updated moment increments will result in a different moment distribution as well as a different IP location ("*b" in Figure 6c). At the end of each analysis time step, the element subroutines calculate the updated moment increments and compare the normalized error with the calculations based on the initial element stiffness by using Equation 2.

$$NE = max\left(\frac{\left|\Delta^* M_{i,j}^{n+1} - \Delta M_{i,j}^{n+1}\right|}{\left|M_{yi,j}\right|}\right) \tag{2}$$

where NE = the maximum of the normalized error for end i and j, $\Delta M_{i,j}^{n+1}$ = moment increment at end i (or j) by using the stiffness corresponding to IP being at point a, $\Delta^* M_{i,j}^{n+1}$ = moment increment at end i (or j) by using the stiffness corresponding to IP being at point b, and $M_{yi,j}$ = yield moment at the response quadrant (positive or negative) for end i (or j).

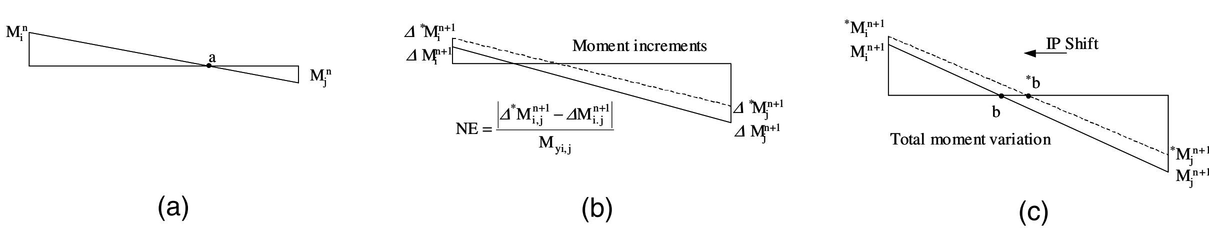


Figure 6 Normalized error calculations in moment increments for checking the event corresponding to excessive movement of the inflection point.

If the error calculated by Equation 1 exceeds the user-specified IP overshoot tolerance, an event is called and the time step is divided in proportion to estimate the deformation at which the error exceeds the tolerance. The change in stiffness is a nonlinear function of the change in inflection point. Therefore, a linear proportioning of the time step only gives a first order approximation to the deformation at which the error exceeds the tolerance. Once the step at which the error exceeds the tolerance is determined, the element stiffness is reformulated using the new IP location at that step. It should be noted that the normalized error in moments is calculated on a step-by-step basis (i.e. using incremental moments associated with the incremental rotations determined by for the entire structure in each analysis step). Because the errors may accumulate over successive steps (particularly when small steps are used), the error (Equation 2) is accumulated over successive steps until the next event is called. Each time the structure stiffness matrix is updated with the current element stiffness, the accumulated error is reset to zero.

The default values for the moment and inflection point tolerances are each 1.0e-5. Guidance on the of the overshoot tolerance for inflection point movement is provided in later sections..

Element features

In Element 07, there are four constants that define the response characteristics of the hysteresis model. These four constants can be adjusted to represent various response modes that can be useful to characterize the response of some reinforced concrete members. The description of each constant is in Table 1 and illustrated in Figure 7. Also provided in Table 1 are the parameter ranges that have been found to provide stable and convergent solutions.

Table 1 Possible ranges of values for the hysteresis model parameters.

Parameter	Description	Recommended range
α^+ and α^-	Post-yield stiffness ratio of the moment-curvature response.	0.001 (almost perfectly plastic) – 0.8 (high post-yield stiffness ratio)
γ^+ and γ^-	Unloading stiffness degradation parameter	0.001 (no reduction) - 0.6 (high degradation)
β^+ and β^-	Stiffness deterioration due to pinching	0.1 (high pinching) – 1.0 (no pinching)
δ^+ and δ^-	Coefficient to determine the recover curvature for pinching	0.1 (high pinching) – 1.0 (no pinching)

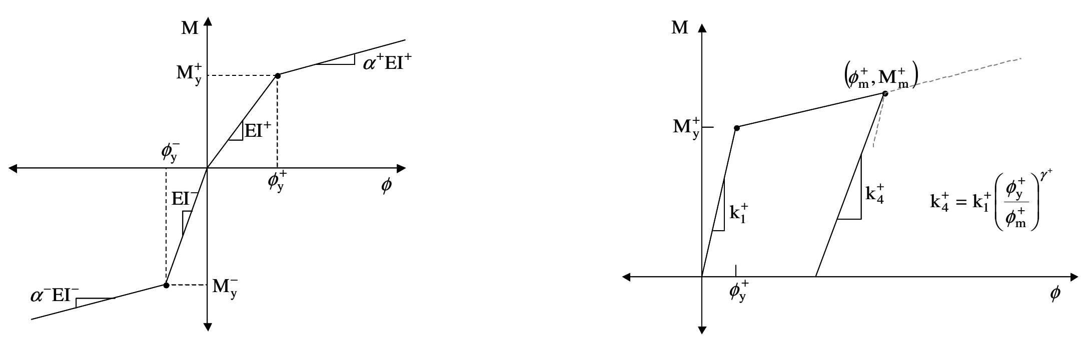


Figure 7 Hysteresis model parameters in Element 07.

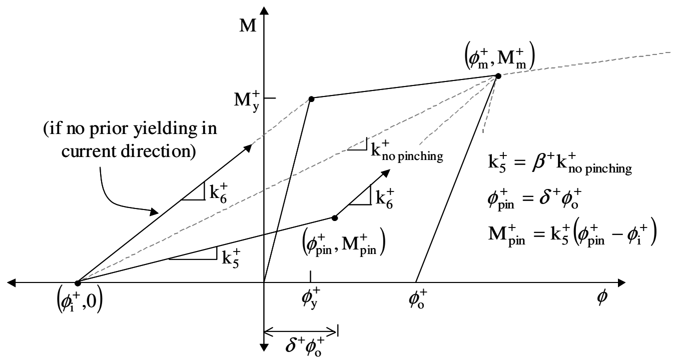


Figure 7 Hysteresis model parameters in Element 07 (continued).

Figure 8 shows some of the possible hysteresis loops that can be obtained using different combinations of model parameters. The data of this figure represent the moment-curvature response at the base of a single degree of freedom oscillator exposed to static positive and negative horizontal load cycles.

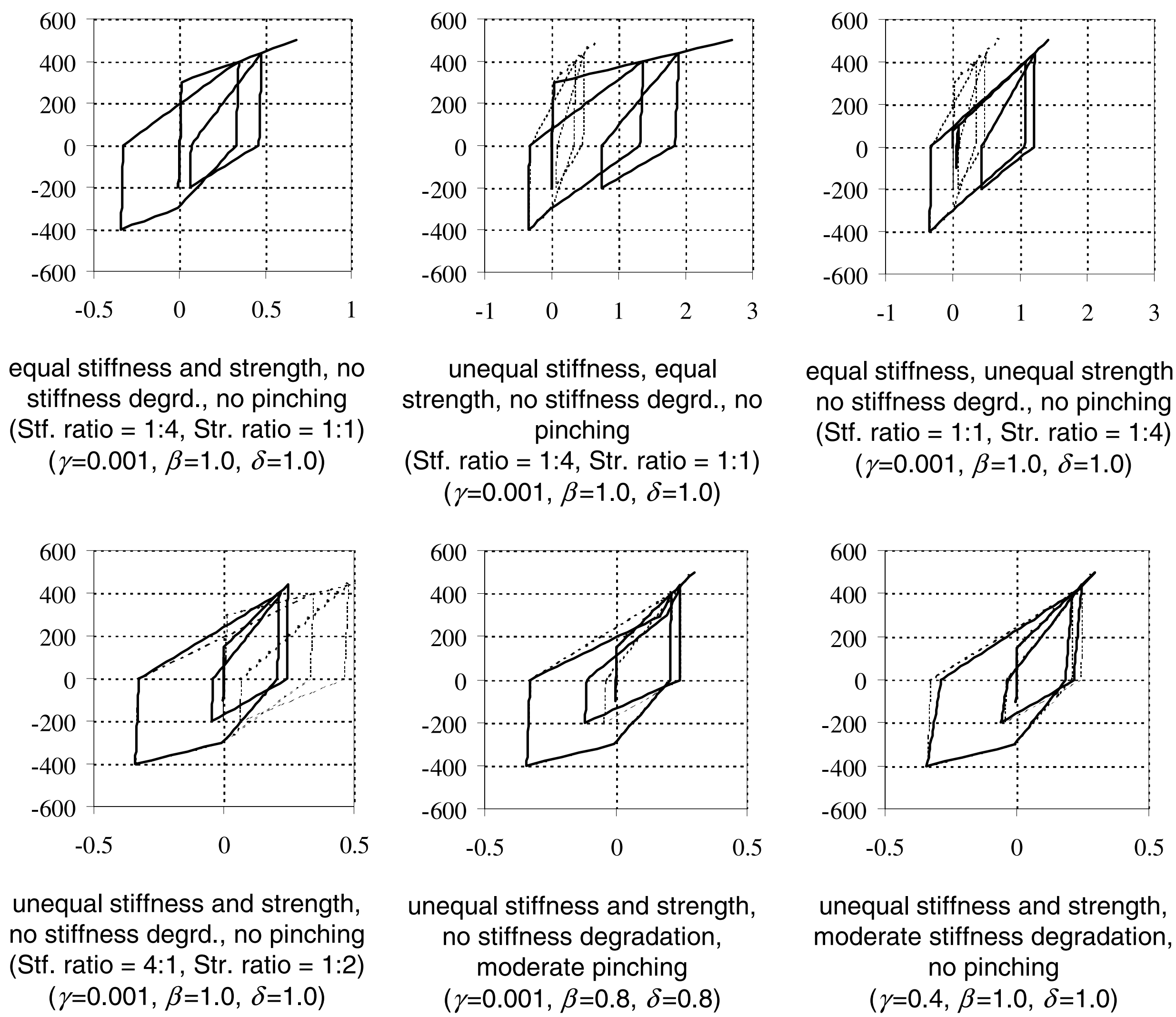
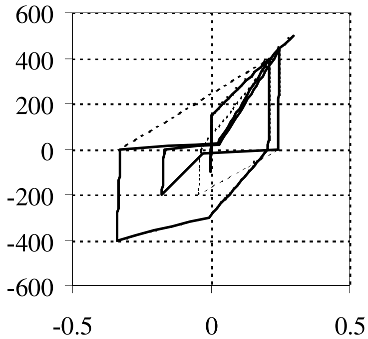
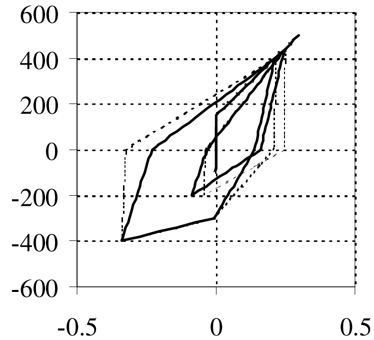


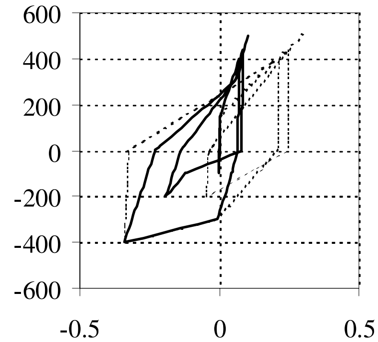
Figure 8. Effects of model parameters on the hysteresis loops.



unequal stiffness and strength, no stiffness degrd., high pinching
(β^{+}=0.1, β^{-}=0.1
δ^{+}=0.1, δ^{-}=0.1)



unequal stiffness and strength, high stiffness degradation., no pinching
(γ^{+}=0.6, γ^{-}=0.6)

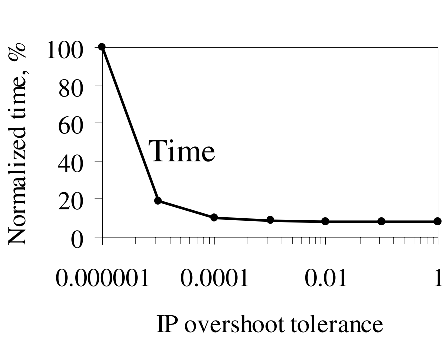


all parameters are unequal
(α^{+}=0.03, α^{-}=0.01,
γ^{+}=0.1, γ^{-}=0.6,
β^{+}=0.8, β^{-}=0.5
δ^{+}=0.8, δ^{-}=0.5)

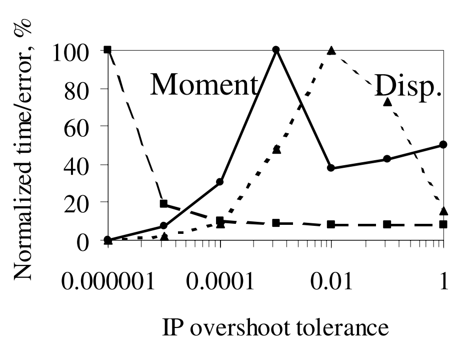
Figure 8. Effects of model parameters on the hysteresis loops (continued).

Guidance on defining tolerance value to for movement of inflection point

As discussed in the previous sections, the overshoot tolerance for inflection point movement is compared to the normalized error in moment increments (calculated by Equation 2) at the ends of the element. As explained in Figure 6, the yield moment in the direction of the response is used to normalize the moment error at the end of the element. The specified values of the yield moments determine whether the normalized error is relatively small or relatively large, and this will influence the effectiveness of the selected overshoot tolerance value. For example, an overshoot tolerance value of 0.001 may be ineffective for the analysis of an element whose yield moment values lie in the range of 100000 and the moments calculated during the analysis are on the order of 10. In contrast, an overshoot tolerance of 0.001 can be too small if the specified yield moments are on the order of 1.0 and the calculated response moments on the order of 10. A very small tolerance will result in many events and a large number of structure stiffness matrix determinations, and therefore will require more computation time.

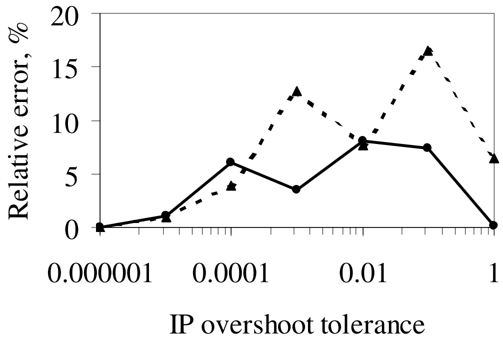


Variation of computation time with changes in IP overshoot tolerance. The time values are relative to the maximum time measured for the smallest overshoot tolerance

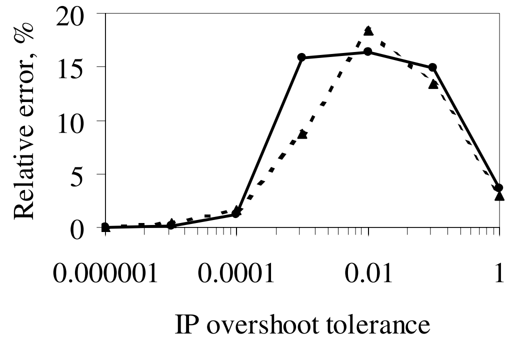


Variation of normalized computation time and normalized error in maximum displacement and moment as a function of IP overshoot tolerance

Figure 9 The effect of inflection point overshoot tolerance level on computation time and estimates of response variables.



Variation of error in maximum positive and negative displacements, in percent, as a function of IP overshoot tolerance



Variation of error in maximum moment estimates, in percent, on either side of the beam as a function of IP overshoot tolerance

Figure 9 The effect of inflection point overshoot tolerance level on computation time and estimates of response variables (continued).

Figure 9 shows the variation of computation time (on the ordinate) and the convergence of the results with respect to different overshoot tolerance values (on the abscissa) for a simple structure subjected to dynamic analysis. The structure consists of a single-bay frame (period = 0.8s) with equal positive and negative yield moments of 300kN-m, preloaded by gravity loads, so that movement of the inflection point occurs under lateral excitation. The variation of the calculated moments is on the order of 100kN-m. In each analysis the same ground motion is used to calculate the response. The ground motion is scaled to result in elastic response. For comparison purposes the analysis results for the lowest overshoot tolerance are assumed to be "exact" in generating the values provided in the plots. The overshoot tolerance for moment is taken as 1.0e-5 and kept constant for all analyses. As can be seen, the results converge to the "exact" value for tolerance values smaller than 1.0e-4, which is about 0.1% of the ratio of the moment variation (100kN-m) and the yield moment (300kN-m). (The apparent convergence of the results for a relatively high tolerance (1.0) may be coincidental and not representative of the convergence characteristics for different structures and ground motion records.) Based on this investigation, a value of 0.05% of the calculated moment and the yield moment ratio can be taken as a basis for selecting the overshoot tolerance for the IP movement.

CONCLUDING REMARKS

Two new elements that were implemented in DRAIN-2DX are introduced. Like for all new elements, first time user's are encouraged to get familiar with the features and capabilities of the elements through simple test analyses. A set of example input files are included with the element files and can be downloaded from the NISEE's web site.

ACKNOWLEDGEMENTS

Element 10 was developed under the SAC Steel Project with funding from the Federal Emergency Management Agency.

REFERENCES

1. Allahabadi, R.; Powell, G. H., “DRAIN-2DX: user guide,” UCB/EERC-88/06, Berkeley: Earthquake Engineering Research Center, University of California, Mar. 1988.

2. Erbay O. O., Aschheim, M. A., “User’s manual for DRAIN-2DX Element 07,” University of Illinois at Urbana-Champaign, Urbana, IL 2003. (Also provided in the downloadable version of DRAIN-2DX from the NISEE’s [5] web site)

3. Hueste, M. B. D. and J. K., Wight, "Nonlinear Punching Shear Failure Model for Interior Slab-Column Connections," Journal of Structural Engineering, ASCE, Vol. 125, No. 9, September 1999, pp. 997-1008.

4. Kanaan, A. E.; Powell, G. H., “General purpose computer program for inelastic dynamic response of plane structures,” UCB/EERC-73/06, Berkeley: Earthquake Engineering Research Center, University of California, Apr. 1973.

5. NISEE, National Information Service for Earthquake Engineering, “Drain-2DX, Static and Dynamic Analysis of Inelastic Plane Structures,” http://nisee.berkeley.edu/software/drain2dx/.

6. Park, Y. J., Ang A. H. S., “Mechanistic seismic damage model for reinforced concrete,” Journal of Structural Engineering, ASCE, Vol. 111, No. 4, 1985.

7. Powell, G. H., “Drain-2DX element description and user guide for element type01, type02, type04, type06, type09, and type15: version 1.10,” UCB/SEMM-1993/18, Berkeley: Department of Civil Engineering, University of California, Dec. 1993.

8. Prakash, V., “Dynamic response analysis of inelastic building structures: the DRAIN series of computer programs,” UCB/SEMM-1992/28, Berkeley: Department of Civil Engineering, University of California, Dec. 1992.

9. Prakash, V.; Powell, G. H.; Campbell, S., “DRAIN-2DX base program description and user guide: version 1.10,” UCB/SEMM-1993/17, Berkeley: Department of Civil Engineering, University of California, Nov. 1993.

10. Shi S., “Evaluation of connection fracture and hysteresis type on the seismic response of steel buildings,” Ph.D. thesis, University of Illinois at Urbana Champaign, Urbana, IL, 1997.

11. Takeda, T., M. A., Sozen, and N. Nielsen, "Reinforced Concrete Response to Simulated Earthquakes," Journal of Structural Division, ASCE, Vol. 96 (ST12), 1970.

12. Tang, X., Goel, S. C., "DRAIN-2DM – Technical notes and user's guide," Research Report UMCE 88-1, Department of Civil Engineering, University of Michigan at Ann Arbor, MI, 1988.