HybridFEM: A PROGRAM FOR DYNAMIC TIME HISTORY ANALYSIS OF 2D INELASTIC FRAMED STRUCTURES AND REAL-TIME HYBRID SIMULATION

HybridFEM Version 4.2.4 User's Manual

by

Theodore L. Karavasilis

Choung-Yeol Seo

James Ricles

ATLSS Report No. 08-09

December 2008

(Users' Manual Updated June 2010)

ATLSS is a National Center for Engineering Research on Advanced Technology for Large Structural Systems

117 ATLSS Drive
Bethlehem, PA 18015-4729

Phone: (610)758-3525
Fax: (610)758-5902

www.atlss.lehigh.edu
Email: inatl@lehigh.edu

HybridFEM: A PROGRAM FOR DYNAMIC TIME HISTORY ANALYSIS OF 2D INELASTIC FRAMED STRUCTURES AND REAL-TIME HYBRID SIMULATION

HybridFEM Version 4.2.4 User's Manual

by

Theodore L. Karavasilis
Research Associate
ATLSS Center

Choung-Yeol Seo
Research Associate
ATLSS Center

James Ricles
Bruce G. Johnston Professor of Structural Engineering
Department of Civil and Environmental Engineering,
Lehigh University
Deputy Director, ATLSS Center

ATLSS Report No. 08-09

December 2008

(Users' Manual Updated June 2010)

ATLSS is a National Center for Engineering Research on Advanced Technology for Large Structural Systems

117 ATLSS Drive
Bethlehem, PA 18015-4729

Phone: (610)758-3525
Fax: (610)758-5902

www.atlss.lehigh.edu
Email: inatl@lehigh.edu

DISCLAIMER

The program HybridFEM has been thoroughly tested and used. However, the user of the program understands that no warranty is implied by the development team on the accuracy of the program. The user is responsible to understand the assumptions and theoretical background of the program.

SOFTWARE UPDATES AND DOCUMENTATION

Professor James M. Ricles was responsible for the conception of HybridFEM. The program was developed for research purposes and is in constant evolution. For updates on software and documentation, please contact:

James M. Ricles
Bruce G. Johnston Professor of Structural Engineering, ATLSS Center, Lehigh University
jmr5@lehigh.edu
ATLSS Engineering Research Center
Department of Civil and Environmental Engineering
Lehigh University
Bethlehem, PA 18015, U.S.A.

ACKNOWLEDGEMENTS

The research presented in this report was conducted at the Engineering Research Center for Advanced Technology for Large Structural Systems (ATLSS), Department of Civil and Environmental Engineering, Lehigh University, Bethlehem, Pennsylvania. The excellent work of the ATLSS technical and support staff is gratefully recognized.

This research was financially supported by a grant from the Pennsylvania Department of Community and Economic Development through the Pennsylvania Infrastructure Technology Alliance (PITA) program and ATLSS. The authors would like to acknowledge Professor Krawinkler and Dr. Lignos of Stanford University for providing the DRAIN2D state determination algorithm for an element with stiffness and strength deterioration capabilities. The opinions, findings and conclusions expressed in this report are those of the authors and do not necessarily reflect the views of those acknowledged here.

TABLE OF CONTENTS

1. INTRODUCTION

The program HybridFEM-2D runs:

(a) Under MATLAB to produce the time-history response of non-linear two-dimensional framed structures to ground accelerations.

(b) Under MATLAB/Simulink and compatible with Target PC for real-time hybrid simulation of non-linear two-dimensional framed structures.

2. MODELLING OPTIONS

The program has a wide variety of modelling options available to represent the structure. The current version of the program requires the user to work in the metric system with KN, m., and sec.

2.1 GLOBAL COORDINATE SYSTEM

X (horizontal axis) – Y (vertical axis with +Y being upward) – Z (satisfies the right-hand rule)

2.2 MASS

The mass of the structure is modelled by following the consistent mass method. The total mass matrix is formed by assembling the consistent mass matrices of the elements. Using Element Type 4, the user can specify concentrated lumped masses which are appropriately added to the total consistent mass matrix.

2.3 RESTRAINTS

The boundary conditions are specified.

2.4 CONSTRAINTS

The equal dof constraint is specified.

2.5 DAMPING

The damping exhibited by the structure is modelled by the commonly assumed Rayleigh damping. There is the option to specify multiplicative factors that define the contribution of each element to the formation of the assembled total structure Rayleigh damping matrix.

2.6 MATERIALS

TYPE1: Elastic material

TYPE 2: Bilinear elasto-plastic material

TYPE 3: Hysteretic material

TYPE 4: Bouc-Wen material

TYPE 5: Trilinear material

TYPE 6: Stiffness degrading material

2.7 SECTIONS

TYPE1: Wide flange section

2.8 ELEMENTS

TYPE1: Elastic beam-column element

TYPE 2: Elastic spring element – has the experimental element option

TYPE 3: Inelastic beam-column element (parallel theory)

TYPE 4: Dummy column element

TYPE 5: Nonlinear spring element with stiffness and strength degradation capabilities

TYPE 6: Displacement-based beam column element

TYPE 7: Force-based beam column element

TYPE 8: Zerolength element

TYPE 9: 2D planar panel zone element

3. INTEGRATION OF THE EQUATIONS OF MOTION

The unconditionally stable explicit CR algorithm is used to integrate the coupled 2^{nd} order differential equations of motion of the structure [1]. Alternatively, a user can choose Rosenbrock-W algorithm for the numerical integration [2].

4. INPUT FILE FOR HybridFEM-2D

In the following user guide, each line of input data is indicated by a box containing the data items. Below each box there is a table describing the data items. This table specifies whether the data item is an integer "I" or a floating "F" number. If a default integer or floating number value should be assigned to a data item, this will be specified in that table. Additional notes are provided for further explanation. The data items of each line should be separated by blank spaces. The format of the data items is free, e.g., a floating number may or may not have a decimal point and may or may not take a scientific form.

The input file for the structure is described by the following sequence of input lines. No blank lines are permitted. The program provides comment lines for free alphanumeric input (the rule for separation by blank spaces does not apply for these lines).

1. Comment line

Any alphanumeric characters

2. Comment line

Any alphanumeric characters

3. Comment line

Any alphanumeric characters

4. Structure data block line

```
NN NE  NM  NSEC  NDIM  NEN  NDN  ND  NSL NGN
```

NN	:	Number of nodes	I
NE	:	Number of elements	I
NM	:	Number of materials	I
NSEC	:	Number of sections	I
NDIM	:	2 (default)	I
NEN	:	2 (default)	I
NDN	:	3 (default)	I
ND	:	Number of restrained dofs	I
NSL	:	Number of constrained (slaved) dofs	I
NGN	:	Number of dofs with gravity load	I

5. Comment line

Any alphanumeric characters

6. Comment line

Any alphanumeric characters

7. Nodal data block line

NN *lines*

```
NODE   X   Y   Z
```

`NODE`	:	Node ID number	I
`X`	:	Nodal coordinate X	F
`Y`	:	Nodal coordinate Y	F
`Z`	:	Nodal coordinate Z	F

Notes

1: The node ID number can be any integer number. There are no restrictions on the nodes numbering.

2: The free dynamic dofs of the structure are numbered sequentially, i.e., 1, 2, 3, ..., NN*3 – ND – NSL.

8. Comment line

```
Any alphanumeric characters
```

9. Comment line

```
Any alphanumeric characters
```

10. Boundary data block

ND *lines*

```
NODE    UX     UY     THETA
```

`NODE`	:	Node ID number	I
`UX`	:	X direction dof	I
`UY`	:	Y direction dof	I
`THETA`	:	Rotational dof	I

Notes

1: For each node, the user must specify the ID number of the three (UX, UY and THETA) associated dofs. (1 = fixed, 0 = free). Nodes not assigned boundary conditions are assumed to have all of there degrees of freedom free.

11. Comment line

```
Any alphanumeric characters
```

12. Comment line

```
Any alphanumeric characters
```

13. Equal DOF Constraint data block

NSL *lines*

```
MNODE  SNODE    UX     UY     THETA
```

`MNODE`	:	Master Node ID number	I
`SNODE`	:	Slave Node ID number	I
`UX`	:	X direction dof	I
`UY`	:	Y direction dof	I
`THETA`	:	Rotational dof	I

Notes

1: In order to constrain dofs, specify the master and slave node ID numbers and specify the dof to be constrained. (1 = constrained, 0 = free)

14. Comment line

Any alphanumeric characters

15. Comment line

Any alphanumeric characters

16. Material data block

NM ***lines***

```
MATID  MATTYPE {Material properties}
```

`MATID`	:	Material ID number	I
`MATTYPE`	:	Material TYPE	I
`{…}`	:	Material parameters	F

Notes

1: The MAT ID number can be any integer number.

16.1: Material TYPE 1: Elastic material

`E (data1)`	:	Initial stiffness / Youngs modulus	F

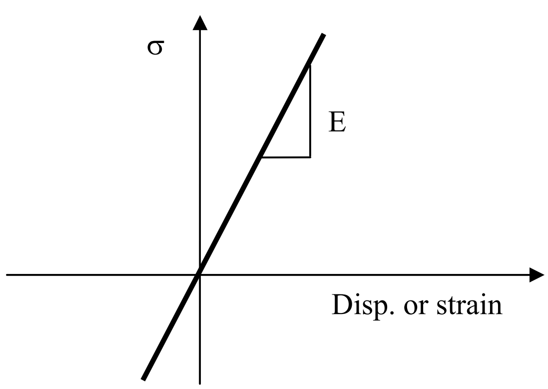


Fig. 1. Elastic behaviour

16.2: Material TYPE 2: Bilinear elasto-plastic material

`E (data1)`	:	Initial modulus / stiffness	F
`SigmaY (data2)`	:	Yield stress / strength	F
`Alpha (data3)`	:	Post-yielding stiffness	F

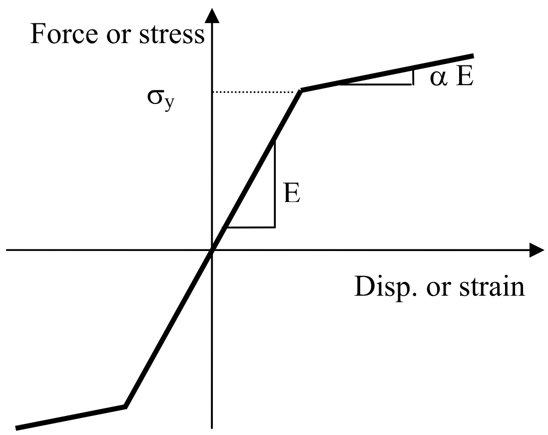


Fig. 2. Bilinear elasto-plastic behaviour

16.3: Material TYPE 3: Hysteretic material

`pinchx (data1)`	:	Pinching factor for deformation	F
`pinchy (data2)`	:	Pinching factor for force	F
`damage1(data3)`	:	Damage due to ductility	F
`damage2(data4)`	:	Damage due to energy	F
`Beta (data5)`	:	power used to determine the degraded unloading stiffness based on ductility	F
`s1p (data6)`	:	Force at *first* point of the envelope in the *positive* direction	F
`e1p (data7)`	:	Deformation at *first* point of the envelope in the *positive* direction	F
`s2p (data8)`	:	Force at *second* point of the envelope in the *positive* direction	F
`e2p (data9)`	:	Deformation at *second* point of the envelope in the *positive* direction	F
`s3p (data10)`	:	Force at *third* point of the envelope in the *positive* direction	F
`e3p (data11)`	:	Deformation at *third* point of the envelope in the *positive* direction	F
`s1n (data12)`	:	Force at *first* point of the envelope in the *negative* direction	F

e1n (data13)	:	Deformation at *first* point of the envelope in the *negative* direction	F
s2n (data14)	:	Force at *second* point of the envelope in the *negative* direction	F
e2n (data15)	:	Deformation at *second* point of the envelope in the *negative* direction	F
s3n (data16)	:	Force at *third* point of the envelope in the *negative* direction	F
e3n (data17)	:	Deformation at *third* point of the envelope in the *negative* direction	F

Notes

1. This material model is equivalent to the OpenSEES Hysteretic material and the detail information can be found in Mazzoni et al [4]

2. Ductiltiy, μ, is calculated during analysis (μ ≥ 1)

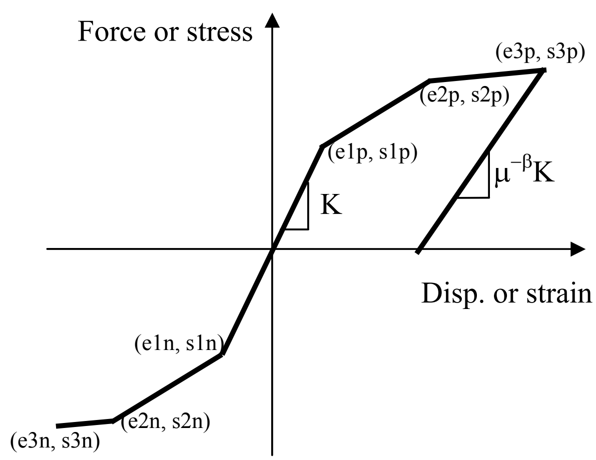


Fig. 3. Hysteretic material envelope curve

16.4: Material TYPE 4: Bouc-Wen material

C (data1)	:	Viscous damping coefficient	F
K1 (data2)	:	Elastic stiffness	F
K2 (data3)	:	Stiffness	F
Alpha (data4)	:	Factor for elastic stiffness in non-hysteretic component (0< Alpha <1)	F

`uy (data5)`	:	Yield displacement
`a (data6)`	:	Power factor for nonlinear viscous damping (=1,linear viscous damping) in non-hysteretic component
`Beta (data7)`	:	Parameter that controls shape of hysteresis loop in a hysteretic component
`Gamma (data8)`	:	Parameter that controls shape of hysteresis loop in a hysteretic component
`N (data9)`	:	Parameter that controls smoothness of transition from linear to nonlinear range for a hysteretic component. As n increases the transition becomes sharper

Notes

1. The restoring force , Q, is modeled by non-hysteretic component and hysteretic component in parallel as shown in Figure 4. Its mathematical is written as

$$Q(x,\dot{x},t) = g(x,\dot{x}) + h(x)$$

where: non-hysteretic component g is written as

$$g(x,\dot{x}) = c \cdot |\dot{x}|^{a} \cdot \left(\frac{\dot{x}}{|\dot{x}|}\right) + \alpha \cdot k \cdot x$$

C is damping coefficient, Hysteretic component is

$$h(x) = (1-\alpha) \cdot k \cdot x \cdot z(t)$$

The state variable z is described as

$$\dot{z}(t) = \frac{1}{u_y}\left[\frac{A_0 \cdot \dot{x} - \nu\left(\gamma \cdot |\dot{x}| \cdot |z|^{n} sign(z) + \beta \cdot \dot{x} \cdot |z|^{n}\right)}{\eta}\right]$$

A_0 is set to be one in the element, ν and η are set to 1 for non-degradation in the element, K is the initial stiffness of the element (=**`K1`** + **`K2`**)

2. The detail information can be found in Wen [5]

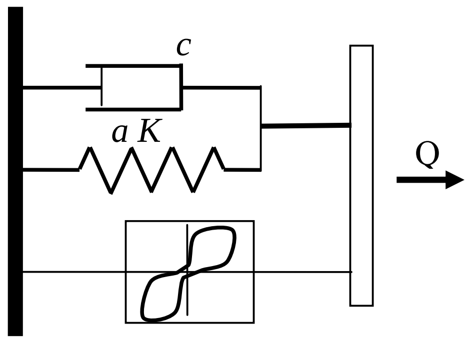


Figure 4. Schematic view of Bouc-Wen material model

16.5: Material TYPE 5: Trilinear material

`Vy (data1)`	:	Yield shear strength	F
`Vu (data2)`	:	Ultimate strength	F
`Dy (data3)`	:	Yield displacement	F
`Du (data4)`	:	Ultimate displacement	F

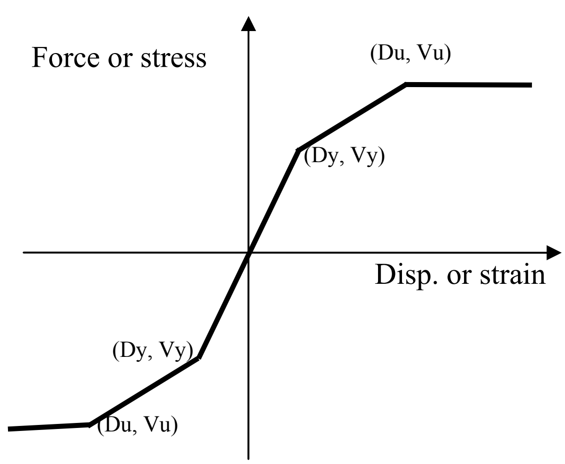


Fig. 5. Trilinear material envelope curve

16.6: Material TYPE 6: Stiffness degrading material

`Kip (data1)`	:	Initial stiffness in positive direction	F
`Fyp (data2)`	:	Positive yield strength	F
`Fup (data3)`	:	Positive ultimate strength	F
`Uup (data4)`	:	Displacement corresponding to Fup	F
`Frp (data5)`	:	Positive residual strength	F
`Urp (data6)`	:	Displacement corresponding to Frn	F
`Kin (data7)`	:	Initial stiffness in negative direction	F
`Fyn (data8)`	:	Negative yield strength	F
`Fun (data9)`	:	Negative ultimate strength	F
`Uun (data10)`	:	Displacement corresponding to Fun	F
`Frn (data11)`	:	Negative residual strength	F
`Urn (data12)`	:	Displacement corresponding to Frn	F

Notes

1. The detail information on the hysteresis rule can be found in Wu [6]

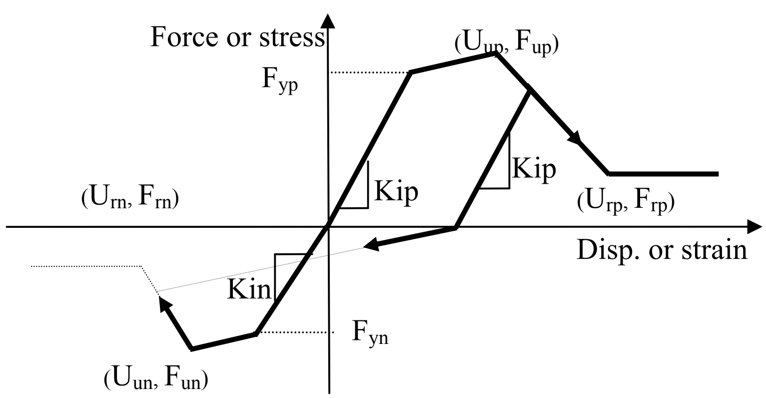


Fig. 6. Stiffness degrading material envelope curve, Material Type 6

17. Comment line

Any alphanumeric characters

18. Comment line

Any alphanumeric characters

19. Section data block

NSEC *lines*

```
SECID  SECTYPE {Section properties}
```

SECID	:	Section ID number	I
SECTYPE	:	Section TYPE	I
{…}	:	Section parameters	F

Notes

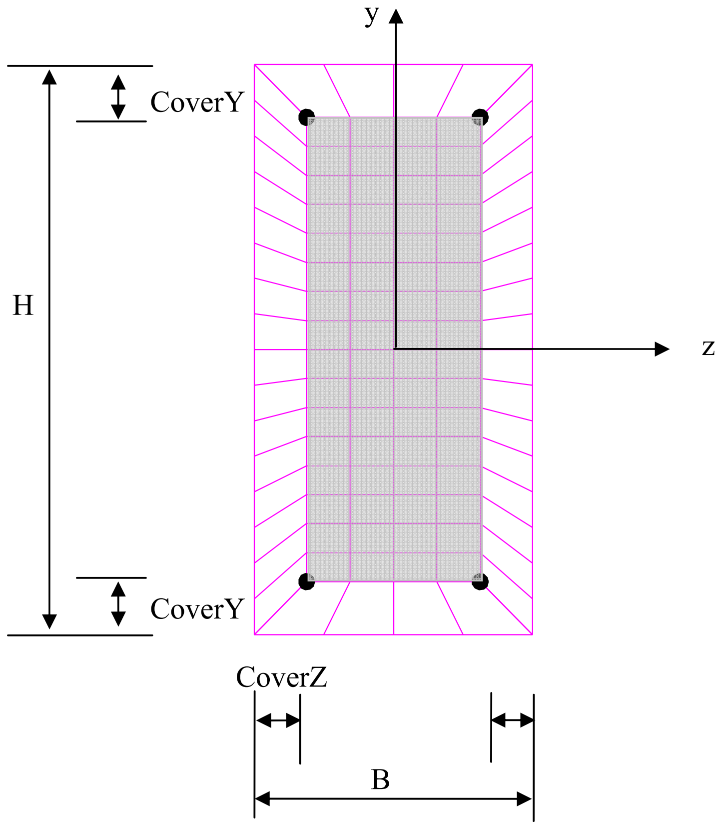
1. The SECID and SECTYPE number can be any integer number. There are no restrictions on section numbering.

19.1: Section TYPE 1: Wide flange section

`d (data1)`	:	Section depth	F
`B (data2)`	:	Section width	F
`tf (data3)`	:	Flange thickness	F
`tw (data4)`	:	Web thickness	F
`NFF (data5)`	:	Number of fibers along flange thickness	F
`NFW (data6)`	:	Number of fibers along web depth	F
`MATID (data7)`	:	Material ID assigned to the section	

19.2: Section TYPE 2: Rectangular RC section

`H (data1)`	:	Section height	F
`B (data2)`	:	Section width	F
`CoverY (data3)`	:	Cover thickness in y-direction	F
`CoverZ (data4)`	:	Cover thickness in z-direction	F
`nfCoreY (data5)`	:	Number of fibers in y direction in the core	I
`nfCoreZ (data6)`	:	Number of fibers in y direction in the core	I
`nfCoverY (data7)`	:	Number of fibers in y direction in the cover	I
`nfCoverZ (data7)`	:	Number of fibers in Z direction in the cover	I
`nLayers (data8)`	:	Number of layers in y direction	I
`As (data 9)`	:	RS bars Section area at the top layer	F
`Nrs (data10)`	:	Number of RS bars at the top layer	I
:		:	
`As (data9+ nLayers)`	:	RS bars Section area at the last layer	F
`Nrs (data10+nLayers)`	:	Number of RS bars at the last layer	I
`matID1 (data11+nLayers)`	:	Concrete material tag in the core (Confined concrete)	I
`matID2 (data12+nLayers)`	:	Concrete material tag in the cover (Unconfined concrete)	I
`matID3 (data13+nLayers)`	:	Reinforced steel bar material tag	I



e.g., nfCoreY = 16, nfCoreZ = 4, nfCoverY = 16, nfCoverZ = 1, nLayers = 2, Nrs = 2

Figure 6. Fiber rectangular reinforced concrete section

20. Comment line

Any alphanumeric characters

21. Comment line

Any alphanumeric characters

22. Element data block

***NE* lines**

```
ELID ELTYPE NODEI NODEJ DAMPK DAMPM {data1, data2,…dataN}
```

`ELID`	:	Element ID number	I
`ELTYPE`	:	Element Type	I
`NODEI`	:	ID number of NODE I	I
`NODEJ`	:	ID number of NODE J	I
`DAMPK`	:	Stiffness proportional damping	F
`DAMPM`	:	Mass proportional damping	F
`{…}`	:	Depends on element type	I/F

Notes

1. The element ID and TYPE numbers are any integer number. There are no restrictions on element numbering.

2. The number of data items {data1 data2 ..... dataN} depends on the element type and are given below:

22.1: Element TYPE 1: Elastic beam-column element

`MAT (data1)`	:	Material ID number	I
`A (data2)`	:	Cross section (axial) area	F
`I (data3)`	:	Moment of inertia	F
`Load (data4)`	:	Element gravity load per unit length	F

Notes

1. The element gravity load per unit length is converted by the program to element mass per unit length according to: Mass= Load /g, where g is the gravity of acceleration. The program assumes that g=9.81 m/sec^2.

2. The damping matrix of each element is calculated as [C] = DampK*A_o*[K] + DampM*A_1*[M], where [K] is the stiffness matrix, [M] is the mass matrix, and A_o and A_1 are the Rayleigh proportional damping factors.

22.2: Element TYPE 2: Elastic spring element

`K (data1)`	:	Stiffness of the spring	F
C_{exp} `(data2)`	:	Damping constant of experimental element	F

Notes

1. This element connects nodes which can have the same coordinates (zero length element) or different coordinates.

2. The element is assumed to provide stiffness only for the horizontal global UX dof of the nodes.

3. C_{exp} is a damping constant which is appropriately added to the total damping matrix of the structure. However, the damping matrix that includes the effect of C_{exp} is used only for the calculation of the integration parameters of the CR algorithm and for extrapolation or the experimental substructure restoring force at the last substep of the "ramping" of the actuator command displacements during a hybrid test. The effect of C_{exp} is not considered in the damping matrix used during the solution of the equations of motion.

4. This element has the ability to represent a physical experimental element in the laboratory. In that case, the stiffness of the element contributes to the initial structure stiffness matrix, however, during the real-time hybrid simulation, the resisting force of the elements comes from the load cells attached to the physical experimental element.

22.3: Element TYPE 3: Plastic hinge inelastic beam-column element (parallel component theory)

`MATID (data1)`	:	Material ID number	I
`A (data2)`	:	Cross section (axial) area	F
`I (data3)`	:	Moment of inertia	F
`Load (data4)`	:	Element gravity load per unit length	F
`Hard (data5)`	:	Hardening	F
A_v `(data6)`	:	Shear area	F
`v (data7)`	:	Poisson's ratio	F
M_{pl} `(data8)`	:	Plastic moment	F
N_{pl} `(data9)`	:	Plastic axial force	F

Notes

1. The element has the same capabilities as the DRAIN-2DX plastic hinge beam column-element (parallel component theory).

2. The element requires a P-M interaction surface to account for inelastic response to account for axial force effect (see Figure 8)

3. Detail information on this element can be found in Prakash et al. **[3]**

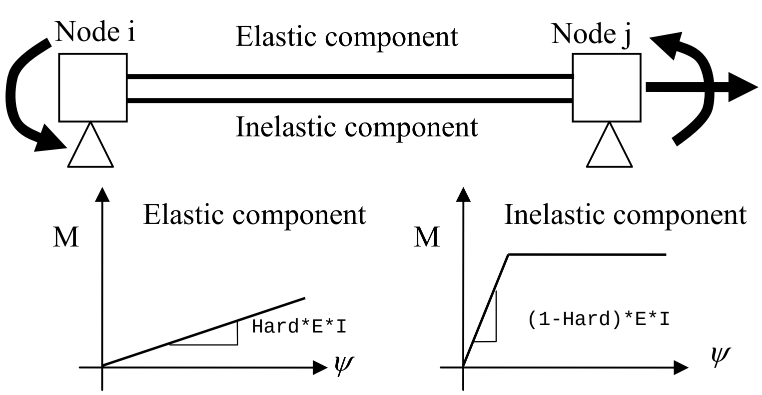


Fig. 7. Schematic view of element Type 3

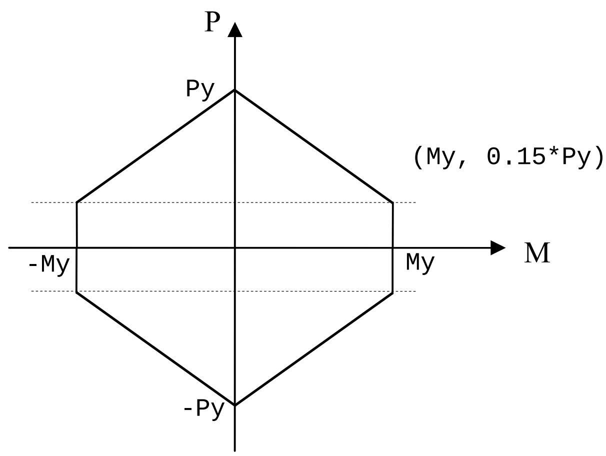


Fig. 8. P-M interaction surface defined **in element Type 3**

22.4: Element TYPE 4: Dummy column element

`W (data1)`	**:**	Weight of the element for P-Δ effect	F
`Mi (data2)`	**:**	Mass on the node i	F
`Mj (data3)`	**:**	Mass on the node j	F
`MDOF (data4)`	**:**	Mass DOF	I

Notes

1, The element can be used to model a lean-on column for P-D effect

2, To exclude P-Δ effect , set W = 0

3, Mi and Mj are the lumped mass assigned to nodes i and j.

4, DOF option in global coordinate for the assigned mass (MDOF=1 for X-direction, =2 for Y-direction, =3 for Rotational direction). Current version allows mass in only X-direction

22.5: Element TYPE 5: Rotational spring element with stiffness and strength deterioration capabilities

`Ke (data1)`	:	Initial stiffness	F
`As (data2)`	:	Strain hardening ratio	F
`AsNeg (data3)`	:	Strain hardening ratio in the negative direction	F
`My_pos (data4)`	:	Positive yield moment	F
`My_neg (data5)`	:	Negative yield moment	F
`LamdaS (data6)`	:	Basic strength deterioration parameter	F
`LamdaK (data7)`	:	Unloading stiffness deterioration parameter	F
`LamdaA (data8)`	:	Accelerated reloading stiffness deterioration parameter	F
`LamdaD (data9)`	:	Post-capping strength deterioration parameter	F
`Cs (data10)`	:	Exponent for basic strength deterioration	F
`Ck (data11)`	:	Exponent for unloading stiffness deterioration	F
`Ca (data12)`	:	Exponent for accelerated reloading stiffness deterioration	F
`Cd (data13)`	:	Exponent for post-capping strength deterioration	F
`Thetap_pos (data14)`	:	Plastic rotation capacity for positive loading	F
`Thetap_neg (data15)`	:	Plastic rotation capacity for negative loading	F
`Thetapc_pos (data16)`	:	Post-capping rotation capacity for positive loading	F
`Thetapc_neg (data17)`	:	Post-capping rotation capacity for negative loading	F
`K (data18)`	:	Residual strength ratio	F
`KNeg (data19)`	:	Residual strength ratio for negative loading	F
`Thetau_pos (data20)`	:	Ultimate rotation capacity for positive loading	F
`Thetau_neg (data21)`	:	Ultimate rotation capacity for negative loading	F
`DPlus (data22)`	:	Composite action factor for positive loading	F
`DNeg (data23)`	:	Composite action factor for negative loading	F

Notes

1. The element is a rotational zero length element to simulate strength and stiffness deterioration at the plastic hinge region at the beam end during cyclic rotational response

2. The element parameter can be determined from regression analysis of experimental data and detail information on the hysteresis rule and regression parameters, θ_p, θ_{pc}, Λ in for wide flange beam section, are shown below

For Non Reduced beam section,

$$\theta_p = 0.087\left(\frac{h}{t_w}\right)^{-0.365} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.14} \cdot \left(\frac{L}{d}\right)^{-0.14} \cdot \left(\frac{d}{c_{unit}^1 \cdot 21''}\right)^{-0.721} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.23}$$

$$\theta_{pc} = 5.70\left(\frac{h}{t_w}\right)^{-0.565} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.80} \cdot \left(\frac{d}{c_{unit}^1 \cdot 21''}\right)^{-0.28} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.43}$$

$$\Lambda = 500\left(\frac{h}{t_w}\right)^{-1.34} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.595} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.36}$$

For Reduced beam section,

$$\theta_p = 0.19\left(\frac{h}{t_w}\right)^{-0.314} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.10} \cdot \left(\frac{L_b}{r_y}\right)^{-0.119} \cdot \left(\frac{L}{d}\right)^{0.113} \cdot \left(\frac{d}{c_{unit}^1 \cdot 21''}\right)^{-0.76} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.07}$$

$$\theta_{pc} = 9.62\left(\frac{h}{t_w}\right)^{-0.513} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.863} \cdot \left(\frac{L_b}{r_y}\right)^{-0.108} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.36}$$

$$\Lambda = 500\left(\frac{h}{t_w}\right)^{-1.138} \cdot \left(\frac{b_f}{2t_f}\right)^{-0.632} \cdot \left(\frac{L_b}{r_y}\right)^{-0.205} \cdot \left(\frac{c_{unit}^2 \cdot F_y}{50}\right)^{-0.391}$$

Where, h/t_w is the fillet to fillet web depth over web thickness ratio; L_b/r_y is the ratio between beam unbraced length L_b over radius of gyration about the weak axis of the cross section; $b_f/2t_f$ is the flange width to thickness ; L/d is the shear span to depth ratio of the beam; F_y is the yield strength of the flange of the beam in ksi; c_{unit}^1 and c_{unit}^2 are coefficients for units conversion. They both are 1.0 if inches and ksi are used, and they are 0.0254 and 0.145, respectively if *d* is in meters and *Fy* is in MPa; and Λ is a cumulative plastic rotation parameter.

3. The element deterioration parameters, Λ_S, Λ_K, Λ_A, and Λ_D, are equal to the value of Λ obtained from the regression equation above. The plastic rotation capacity parameters θ_p and θ_{pc} in both positive and negative directions are set equal in the current version of the element. According to Lignos and Krawinkler [7], the value of the post-yielding stiffness ratios, **`As`** and **`AsNeg`**, are determined to have the ratio of capping moment to effective yield moment equal to 1.1. The value of the residual strength ratios, **`K`** and **`KNeg`**, is recommended as 0.4. The value of θ_{pu} in both negative and positive directions is recommended as 0.04 rad. The values of **`Cs, Ck, Ca and Cd,`** are 1.0 by default. The values of **`DPlus, DNeg`** are 1.0 by default.

4. More detail information on this element can be found in Lignos and Krawinkler [7].

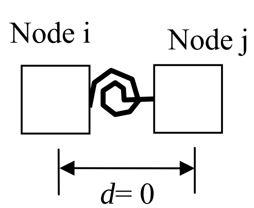


Fig. 9. Schematic view of element Type 5

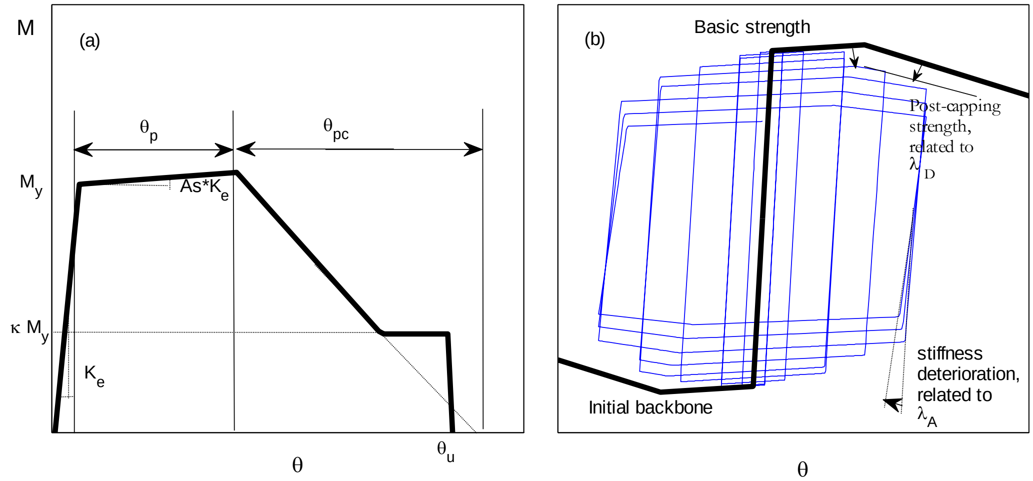


Fig. 10. Element Type 5 rotational behaviour (a) Envelop curve and (b) cyclic behaviour

22.6: Element TYPE 6: Displacement Based Fiber Beam-Column Element

`SECID (data1)`	:	Section ID number	I
`NIP (data2)`	:	Number of integration points along the length of element	F
`Load (data3)`	:	Element gravity load per unit length	F

Notes

1. Integration scheme in the element is based on Gauss-Lobatto quadrature rule and two integration points at the element ends are included as shown in Figure 9

22.7: Element TYPE 7: Force Based Fiber Beam-Column Element

`SECID (data1)`	:	Section ID number	I
`NIP (data2)`	:	Number of integration points along the length of element	F
`Load (data3)`	:	Element gravity load per unit length	F
`ITER (data4)`	:	Maximum number of iteration	F

`TOL (data5)` : Tolerance F

Notes

1. Integration scheme in the element is based on Gauss-Lobatto quadrature rule
2. Maximum iteration and tolerance need to be specified to enable the iterative form of the flexibility formulation and the theoretical development for this element can be found in Spacone, et al. [**8**]

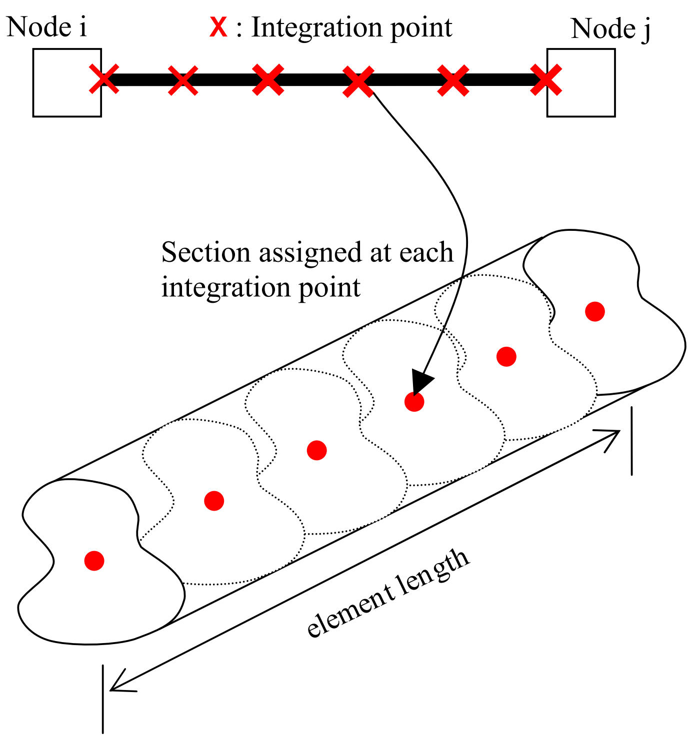


Fig. 11. Schematic view of beam-column element Type 6 and Type 7 [4]

22.8: Element TYPE 8: Zerolength Element

`DOF (data1)` : Direction of element in global coordinate I

`MATID (data2)` : Material ID number F

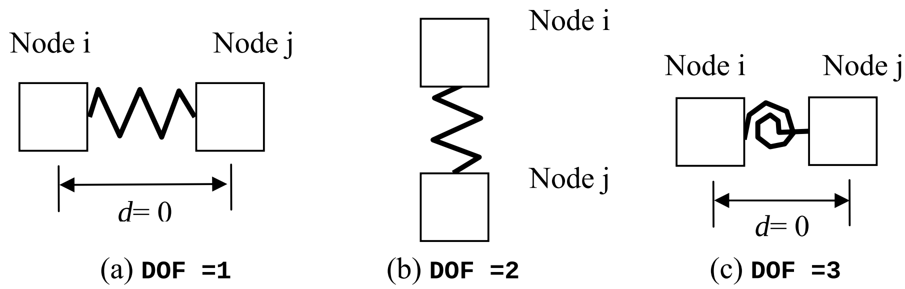


Fig. 12. Schematic view of element Type 8

22.9: Element TYPE 9: 2D Planar Panel Zone Element

`NODEI`	:	Node i	I
`NODEJ`	:	Node j	I
`NODEK`	:	Node k	I
`NODEL`	:	Node l	I
`DAMPK`	:	Stiffness proportional damping	F
`DAMPM`	:	Mass proportional damping	F
`MATID (data1)`	:	Material ID number	I
`ICOL (data2)`	:	Column section moment of inertia	F
`ACOL (data3)`	:	Column section area	F
`BCOL (data4)`	:	Column flange width	F
`tw (data5)`	:	Column web thickness	F
`tf (data6)`	:	Column flange thickness	F
`td (data7)`	:	Double plate thickness	F
`tcnt (data8)`	:	Continuity plate thickness	F
`SHMATID(data9)`	:	Shear material ID number	I
`Mass (data10)`	:	Small mass assigned to the element	F

Notes

1. The element requires four node and the nodes are defined in counter-clockwise rotation.

2. Shear material in the panel zone element is defined with Trilinear material type (material type = 5).

3. The detail information on this element can be found in Seo, et al [9]

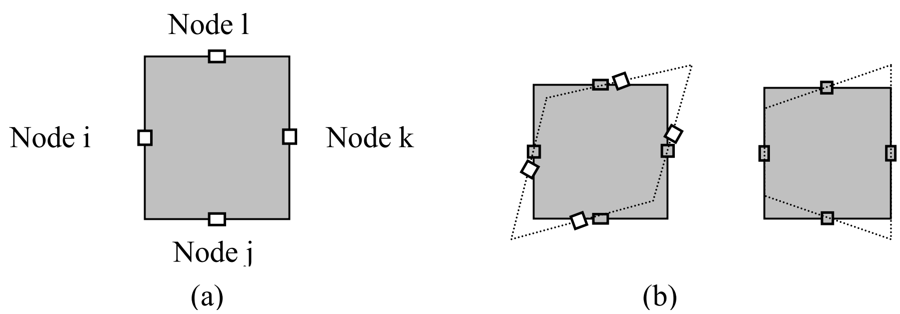


Fig. 13. (a) Schematic view of element Type 9 and (b) Deformation modes considered in the element

23. Comment line

Any alphanumeric characters

24. Comment line

Any alphanumeric characters

25. Gravity Analysis data block

NODE	**DOF**	**GRAVITYLOAD**

NODE	:	Node ID number	I
DOF	:	DOF	I
GRAVITYLOAD	:	gravity load	F

Notes

1. Gravity analysis data block is optional. Gravity analysis is performed followed by dynamic transient analysis, when nonzero value is assigned to **NGN** in the Structure data block.

2. DOF in global coordinate (DOF=1 for X-direction, =2 for Y-direction, =3 for Rotational direction)

26. Comment line

Any alphanumeric characters

27. Comment line

Any alphanumeric characters

28. Analysis data block

```
T1  T2  KSI  SF  DT  INT
```

`T1`	:	First period of vibration	I
`T2`	:	Second period of vibration	F
`KSI`	:	Modal damping ratio	F
`SF`	:	Scale factor of accelerogram	F
`DT`	:	Time step of accelerogram	F
`INT`	:	The number of steps interpolated within **`DT`** for analysis	I

29. Comment line

Any alphanumeric characters

30. Comment line

Any alphanumeric characters

31. Numerical Integration Option block

```
TYPE       data
```

`TYPE`	:	Integration algorithm	I
`data`	:	Integration parameters	F

30.1: TYPE 1: C-R algorithm

`Lambda`	:	1 by default, Integration parameters for CR algorithm	F

Notes

1. The C-R algorithm is an unconditional explicit numerical integration algorithm and detail information can be found in Chen and Ricles [1]

30.2: TYPE 2: Rosenbrock -W algorithm

`Gamma`	:	0.5 by default, Integration parameters for Rosenbrock-W algorithm	F

Notes

1. The algorithm is an unconditional explicit numerical integration algorithm and detail information can be found in Lamarche, et al. [2]

5. Verification examples

5.1: Verification example 1: Elastic two-story frame

1. DATA

1.1 Units

kN-m-s.

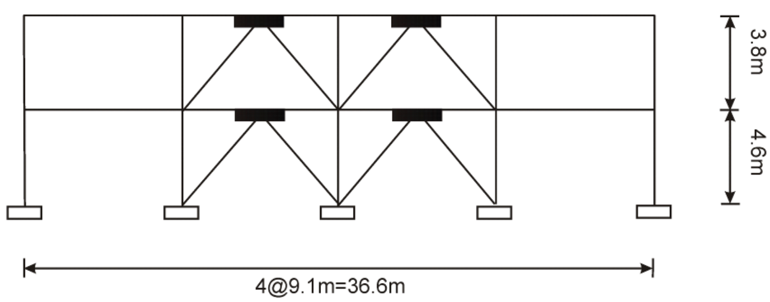
1.2 Elements used

Type1 – Linear Beam Column Element

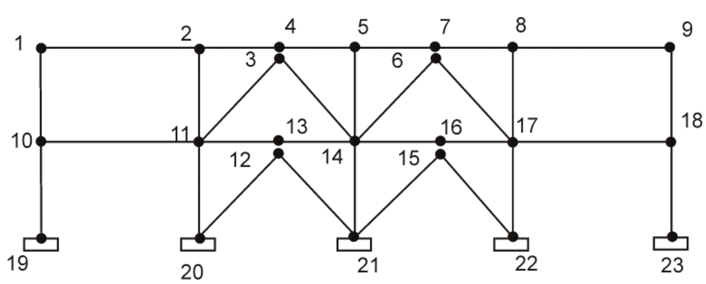
Type2 – Experimental Element

Type4 – Dummy Column Element

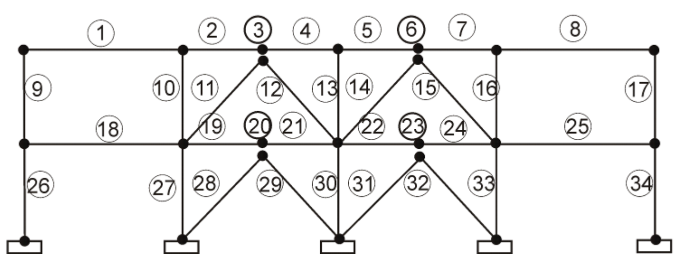
1.3 Geometry of structure – Node numbering – Element numbering



Node Numbering



Element Numbering



1.4 Restrained DOF

19 20 21 22 23: Ux=0 Uy=0 THETAz=0

3 6 12 15: THETAz=0

1.5 Constrained DOF

1 2 4 5 7 8 9: Ux

10 11 13 14 16 17 18: Ux

3 4: Uy

6 7: Uy

12 13: Uy

15 16: Uy

3 6: Ux

12 15: Ux

1.6 Element Section/Properties

All columns: W14x120

Beams 1st floor: W24x55

Beams 2nd floor: W18x40

Braces 1st floor: A=5.881e-3 I=0.0

Braces 2nd floor: A=3.0e-3 I=0.0

Dampers 1st floor: Stiffness=8461

Dampers 2nd floor: Stiffness=5615

1.7 P-Δ and concentrated masses – Two-storey lean on column

Lean-on column section: A=0.0976 I=7.125e-4

Pinned at the base

1st Floor Load: 6618.44

2nd Floor Load: 4633.03

Stiffness matrix (conventional + geometric): $\begin{bmatrix} 7009 & -4630.2 \\ -4630.2 & 1982.34 \end{bmatrix}$

Mass matrix: $\begin{bmatrix} 805 & 0 \\ 0 & 578 \end{bmatrix}$

1.8 Time-history data

See input file below and directory for damping, dt, acceleration record, etc.

1.9 Linear Elastic Material Properties

E=200000000.0

1.10 HybridFem InputFile

Input file below is written for HybridFEM v 4.1

HybridFEM input file (version 4.1)

```
UNITS: KN-m-sec  HybridFEM V4.1 Input
STRUCTURE DATA BLOCK
NN  NE  NM NDIM  NEN  NDN   ND   NSLAVED
23  35   1    2    2    3   19     18
NODAL COORDINATE DATA BLOCK
NODE     X       Y          UX              UY         THETA
 1       0       8.4         1              2             3
 2     9.1       8.4         1              4             5
 3   13.65       8.4         6              7             200
 4   13.65       8.4         1              7             8
 5    18.2       8.4         1              9             10
 6   22.75       8.4         6             11             200
 7   22.75       8.4         1             11             12
 8    27.3       8.4         1             13             14
 9    36.4       8.4         1             15             16
10       0       4.6         17            18             19
11     9.1       4.6         17            20             21
12   13.65       4.6         22            23             200
13   13.65       4.6         17            23             24
14    18.2       4.6         17            25             26
15   22.75       4.6         22            27             200
16   22.75       4.6         17            27             28
17    27.3       4.6         17            29             30
18    36.4       4.6         17            31             32
19       0       0           200           200            200
20     9.1       0           200           200            200
21    18.2       0           200           200            200
22    27.3       0           200           200            200
23    36.4       0           200           200            200
MATERIAL DATA BLOCK
MATERIAL   E
 1          200000000.0
ELEMENT TYPE AND CONNECTIVITY DATA BLOCK
ELEM    TYPE   NODE1   NODE2    data5    data6    data7    data8    data9    data10    data11    data12
data13   data14   data15
 1         1      1       2         1          7.613e-3      2.547e-4       0.586          1         1
 2         1      2       4         1          7.613e-3      2.547e-4       0.586          1         1
 3         2      3       4         5615       1             0              0
 4         1      4       5         1          7.61e-3       2.55e-4        0.586          1         1
 5         1      5       7         1          7.61e-3       2.55e-4        0.586          1         1
 6         2      6       7         5615       1             0              0
 7         1      7       8         1          7.61e-3       2.55e-4        0.586          1         1
 8         1      8       9         1          7.61e-3       2.55e-4        0.586          1         1
 9         1     10       1         1          2.28e-2       5.74e-4        0              1         1
10         1     11       2         1          2.28e-2       5.74e-4        0              1         1
11         1     11       3         1          3.00e-3       0              0.231          1         1
12         1     14       3         1          3.00e-3       0              0.231          1         1
13         1     14       5         1          2.28e-2       5.74e-4        0.0            1         1
14         1     14       6         1          3.00e-3       0              0.231          1         1
15         1     17       6         1          3.00e-3       0              0.231          1         1
16         1     17       8         1          2.28e-2       5.74e-4        0              1         1
17         1     18       9         1          2.28e-3       5.74e-4        0              1         1
18         1     10      11         1          1.05e-2       5.62e-4        0.8085         1         1
19         1     11      13         1          1.05e-2       5.62e-4        0.8085         1         1
20         2     12      13         8461       1             0              0
21         1     13      14         1          1.05e-2       5.62e-4        0.8085         1         1
22         1     14      16         1          1.05e-2       5.62e-4        0.8085         1         1
23         2     15      16         8461       1             0              0
24         1     16      17         1          1.05e-2       5.62e-4        0.8085         1         1
25         1     17      18         1          1.05e-2       5.62e-4        0.8085         1         1
26         1     19      10         1          2.28e-2       5.74e-4        0              1         1
27         1     20      11         1          2.28e-2       5.74e-4        0              1         1
28         1     20      12         1          5.88e-3       0              0.4528         1         1
29         1     21      12         1          5.88e-3       0              0.4528         1         1
30         1     21      14         1          2.28e-2       5.74e-4        0              1         1
31         1     21      15         1          5.88e-3       0              0.4528         1         1
32         1     22      15         1          5.88e-3       0              0.4528         1         1
33         1     22      17         1          2.28e-2       5.74e-4        0              1         1
34         1     23      18         1          2.28e-2       5.74e-4        0              1         1
35         4     18       9         7009       -4630.2       -4630.2        1982.34        805       578
1       1
HYBRID TESTING DATA BLOCK
T1      T2      KSI   SF  DT
1.18   0.38    0.02   1   0.009765625
--------END OF CONFIGURATION------------------

ELEMENT LIBRARY AND ELEMENT DATA
type1: elastic beam column element
type2: experimental link element with initial stiffness
type3: hinge (concentrated plasticity) beam column element (drain2dx type 2)
type4: 2 story gravity column
```

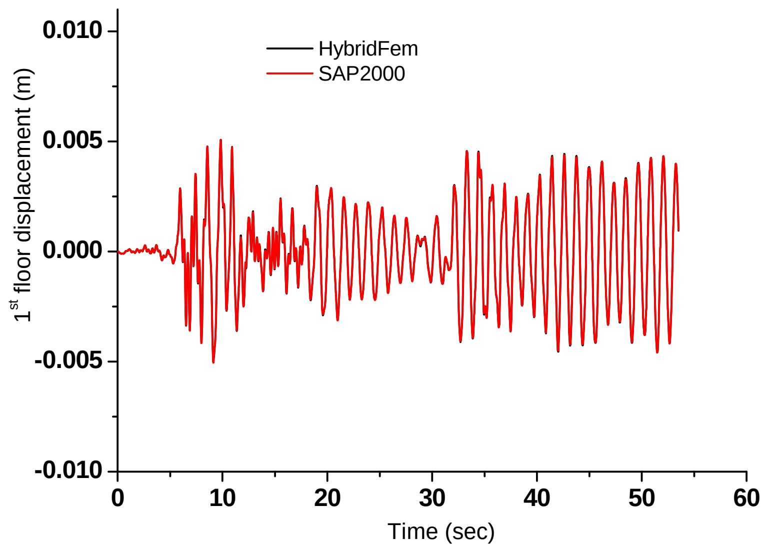
2. RESULTS

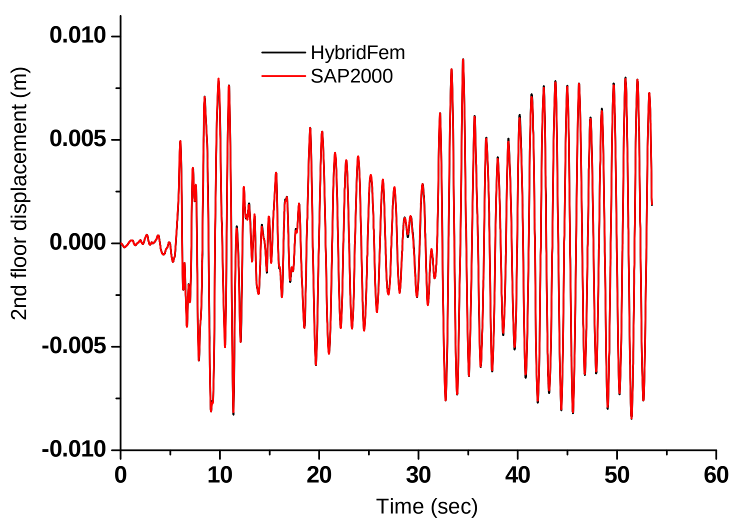
2.1 Modal analysis

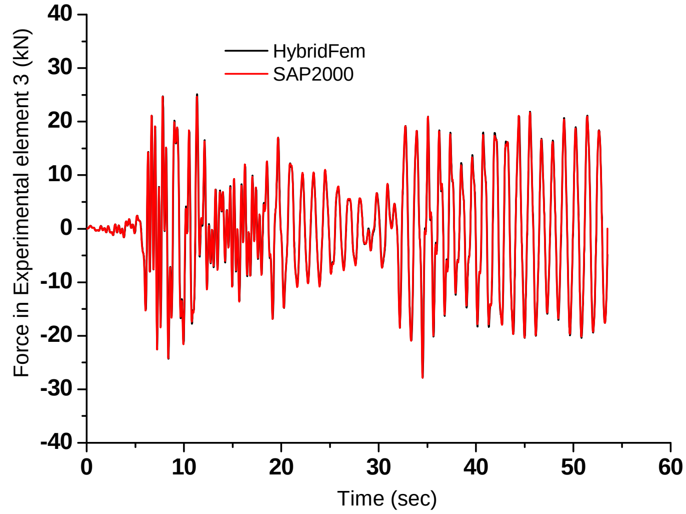
SAP2000: T_1=1.18 T_2=0.38

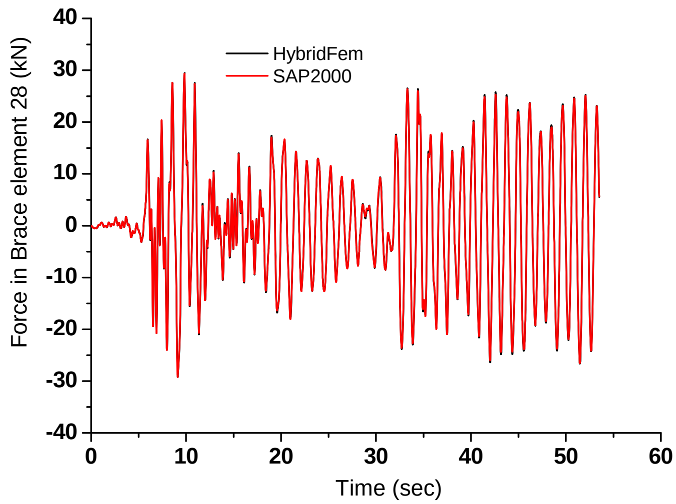
HybridFem: T_1= 1.18 T_2=0.38

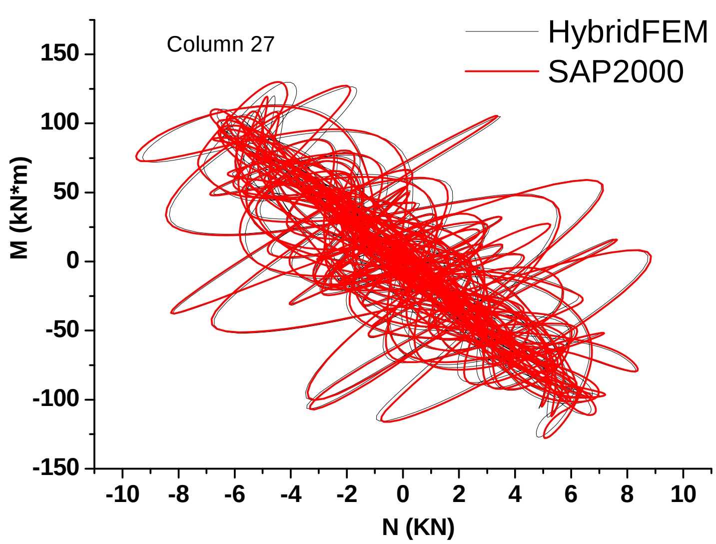
2.2 Time-history analysis

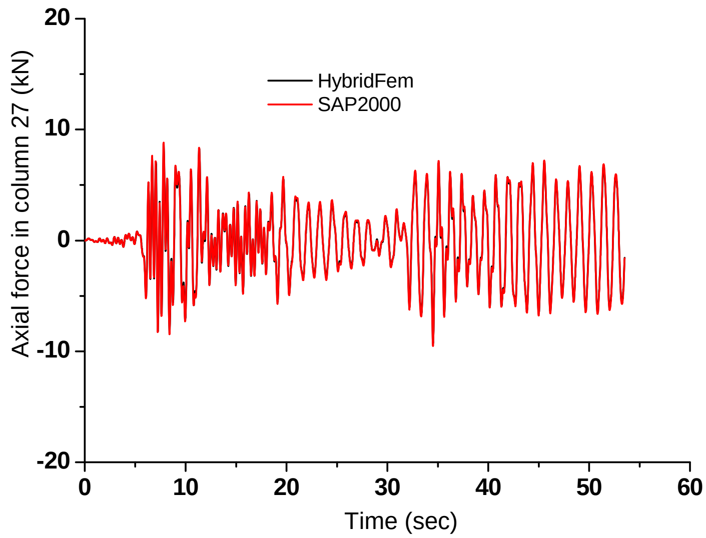


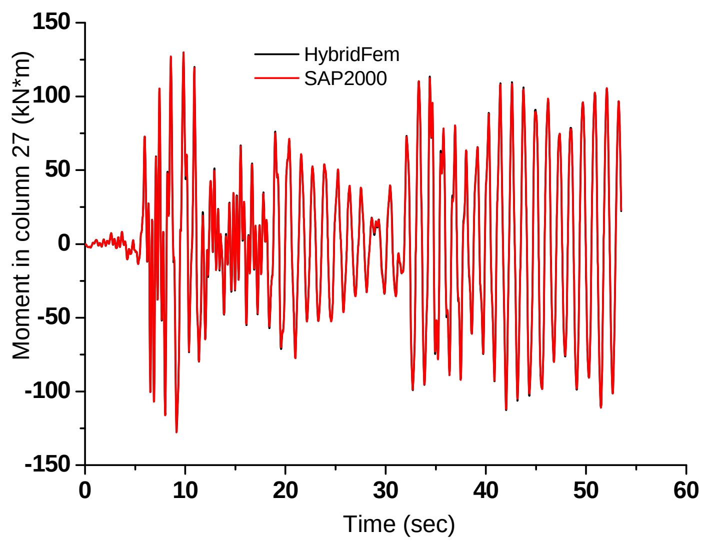



HybridFem
SAP2000
Force in Experimental element 3 (kN)
Time (sec)


HybridFem
SAP2000
Force in Brace element 28 (kN)
Time (sec)


Column 27
HybridFEM
SAP2000
M (kN*m)
N (KN)


20
10
0
-10
-20
0
10
20
30
40
50
60
Axial force in column 27 (kN)
Time (sec)
HybridFem
SAP2000


150
100
50
0
-50
-100
-150
0
10
20
30
40
50
60
Moment in column 27 (kN*m)
Time (sec)
HybridFem
SAP2000

5.2: Verification example 2: Inelastic response of two-storey frame

1. DATA

1.1 Units

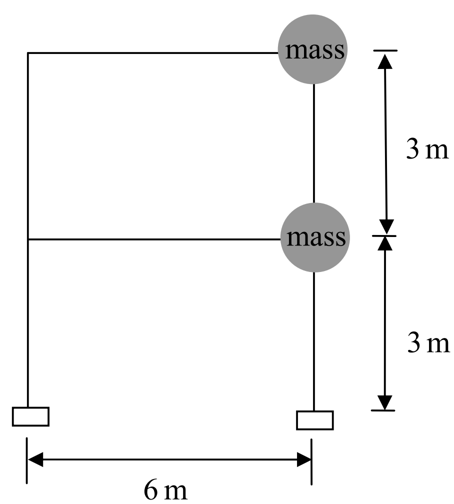
kN-m-s.

1.2 Elements Used

Type3 – Nonlinear Beam Column Element

Type4 – Dummy Element (not available after Version 4.2.3)

1.3 Geometry of Structure



1.4 Element Section/Properties

All columns: W14x120

Beams 1st floor: W24x55

Beams 2nd floor: W24x55

1.5 Concentrated Masses

Mass matrix: $\begin{bmatrix} 612.5 & 0 \\ 0 & 612.5 \end{bmatrix}$

1.6 Time-history Data

See input file for damping, dt, acceleration record, etc.

1.7 Nonlinear Material Properties

E=200000000.0   fy=345000

1.9 Input Files

Input files of HybridFEM v 4.1 and DRAIN-2DX are given in the next pages

HybridFEM input file (version 4.1)

```
UNITS: KN-m-sec
STRUCTURE DATA BLOCK
NN  NE  NM NDIM  NEN  NDN   ND   NSLAVED
6  7  1   2   2   3   6   2
NODAL COORDINATE DATA BLOCK
NODE    X     Y       UX          UY        THETA
 1     0     6       1           2         3
 2     6     6       1           4         5
 3     0     3       6           7         8
 4     6     3       6           9         10
 5     0     0      200         200        200
 6     6     0      200         200        200
MATERIAL DATA BLOCK
MATERIAL   E
 1       200000000.0
ELEMENT TYPE AND CONNECTIVITY DATA BLOCK
ELEM     TYPE   NODE1   NODE2     data5      data6       data7        data8        data9   data10      data11  data12    data13    data14  data15
 1       3     1     2      1       0.0105      5.619e-4      0.8085        0.03   6.007e-3        0.3    757.62    3622.5   1     1
 2       3     3     4      1       0.0105      5.619e-4      0.8085        0.03   6.007e-3        0.3    757.62    3622.5   1     1
 3       3     3     1      1       0.0228      5.744e-4      1.7556        0.03   5.512e-3        0.3    1198.53   7866     1     1
 4       3     4     2      1       0.0228      5.744e-4      1.7556        0.03   5.512e-3        0.3    1198.53   7866     1     1
 5       3     5     3      1       0.0228      5.744e-4      1.7556        0.03   5.512e-3        0.3    1198.53   7866     1     1
 6       3     6     4      1       0.0228      5.744e-4      1.7556        0.03   5.512e-3        0.3    1198.53   7866     1     1
 7       4     4     2      0       0         0            0           612.64  612.64          1      1
HYBRID TESTING DATA BLOCK
T1     T2        KSI   SF  DT
1.4   0.41       0.03    10 0.01
--------END OF CONFIGURATION------------------
ELEMENT LIBRARY AND ELEMENT DATA
type1: elastic beam column element
type2: experimental link element with initial stiffness
type3: hinge (concentrated plasticity) beam column element (drain2dx type 2)
type4: 2 story gravity column
```

DRAIN-2DX input file

```
!This is the DRAIN.INP input file for structure:frame
!erase permanent files
*STARTXX
!name of structure,execute,show progress,consider p-delta effects,perform energy
calculations
     frame         0 1 1 1                                                 frame
!NODE COORDINATES
*NODECOORDS
C        1         0         0
C        2         6         0
C        3         0         3
C        4         6         3
C        5         0         6
C        6         6         6
!RESTRAINTS
!Fixed at the ground level
*RESTRAINTS
S 111         1
S 111         2
!CONSTRAINTS
!All floors assumed to be diaphragms
*SLAVING
S 100         3         4         4
S 100         5         6         6
!NODAL MASSES: computed from a)structure selfweight(autocalculated in program)
!                            b)uniform load on beams
*MASSES
S 110   0.26844         1                                            1       0.2
S 110   0.26844         2                                            1       0.2
S 110   306.595         3                                            1       0.2
S 110   306.595         4                                            1       0.2
S 110   306.326         5                                            1       0.2
S 110   306.326         6                                            1       0.2
*ELEMENTGROUP
   02    1    1          0.003                                     COLUMNS
    2    0    2
    1    2e+008      0.03    0.0228 0.0005744    4    4    2  0.005512  0.30.0001
    2    2e+008      0.03    0.0228 0.0005744    4    4    2  0.005512  0.30.0001
    1    2   1198.53   1198.53      7866      7866  1.0 0.15  1.0 0.15
    2    2   1198.53   1198.53      7866      7866  1.0 0.15  1.0 0.15
    1         1         3              1         1    1
    2         2         4              1         1    1
    3         3         5              2         2    2
    4         4         6              2         2    2
*ELEMENTGROUP
   02    1    1          0.003                                       BEAMS
    2    0    2
    1    2e+008      0.03    0.0105 0.0005619    4    4    2  0.006007  0.30.0001
    2    2e+008      0.03    0.0105 0.0005619    4    4    2  0.006007  0.30.0001
    1    2    757.62    757.62    3622.5    3622.5  1.0 0.15  1.0 0.15
    2    2    757.62    757.62    3622.5    3622.5  1.0 0.15  1.0 0.15
    1         3         4              1         1    1
    2         5         6              2         2    2
*GENDISP
         1    1        -1
         3    1         1
*GENDISP
         3    1        -1
         5    1         1
*GENDISP
         1    1         0
         3    1         1
*GENDISP
         3    1         0
         5    1         1
*RESULTS
NSD    001
E        1
```

DRAIN-2DX input file- Continued

```
*ELEMLOAD
 kata                                                            katakoryfa
G   1    2
    1    0         1        0.0        0.0        0.0        0.0        0.0        0.0
    2    0         1        0.0        0.0        0.0        0.0        0.0        0.0
    1              1         1
    2              1         1
    3              2         1
    4              2         1

G   2    2
    1    0         1        0.0        0.0        0.0        0.0        0.0        0.0
    2    0         1        0.0        0.0        0.0        0.0        0.0        0.0
    1              1         1
    2              2         1

*ACCNREC
 elce           elce             (F15.7)                                     elce
 5479    1    0    2                          0.01      0.01
*PARAMETERS
OD       0         0    0         0    0      0.01    0         0    0     54.79
DC  1           -100
*GRAV                                                           KATAKORYFA FORTIA
E     kata
*ACCN                                                            Time history for:elce
     54.79 5480    1      0.01         3         3
1     elce
*STOP
```

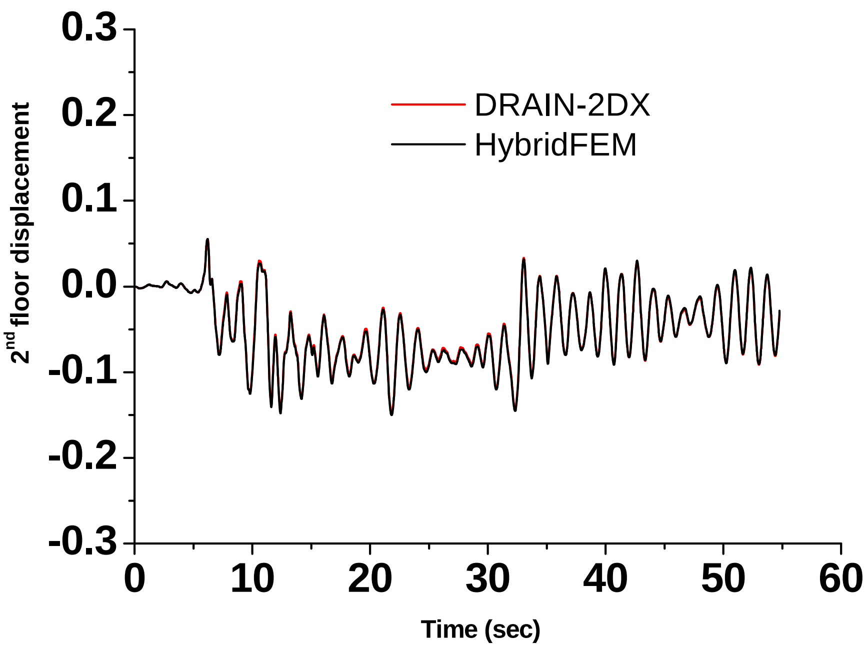
2. RESULTS

2.1 Modal Analysis

HybridFEM: T_1=1.4 T_2=0.4

DRAIN: T_1= 1.4 T_2=0.4

2.2 Time-history Analysis



5.3: Verification example 3: Simulating stiffness and strength deterioration in the cyclic response of steel components using element type 5

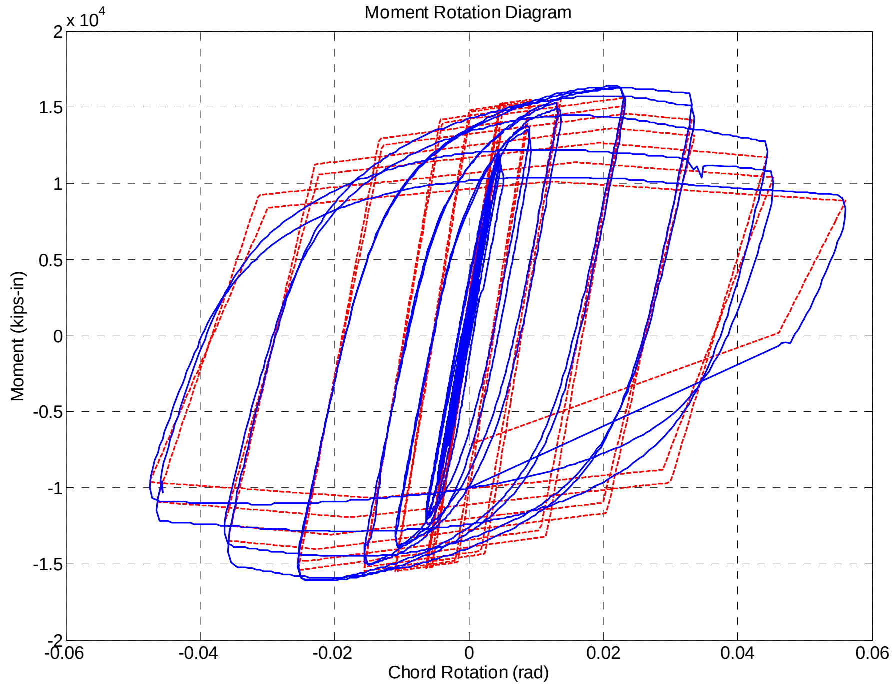
1. Input Parameters for Element Type 5

```
pKe=2790995;
pAs=0.023;
pAsNeg = 0.023;
pMy_pos=15300;
pMy_neg=-15200;
pLamdaS=1.10;
pLamdaK=1.00;
pLamdaA=1.00;
pLamdaD=1.00;
pCs=1.0;
pCk=1.0;
pCa=1.0;
pCd=1.0;
pThetap_pos=0.022;
pThetap_neg=0.022;
pThetapc_pos=0.22;
pThetapc_neg=0.22;
pK=0.40;
pKNeg =0.40;
pThetau_pos=0.40;
pThetau_neg=0.40;
DPlus = 1.0;
DNeg  = 1.0;
```

2. Experimental Results

Refer to Lignos and Krawinkler [7]

3. Results



Moment Rotation Diagram
x 10^4
2
1.5
1
0.5
0
-0.5
-1
-1.5
-2
-0.06
-0.04
-0.02
0
0.02
0.04
0.06
Moment (kips-in)
Chord Rotation (rad)

5.4: Verification example 2: Two-storey frame with P-Δ effect

1. DATA

1.1 Units

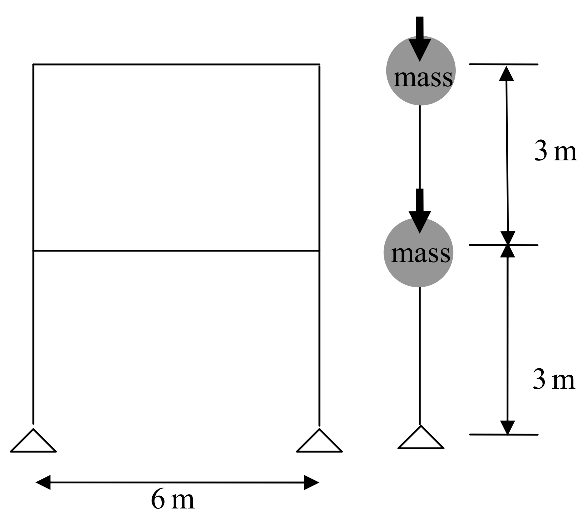
kN-m-s.

1.2 Elements Used

Type1 – Elastic Beam-Column Element

Type4 – Dummy Column Element

1.3 Geometry of Structure



1.4 Element Section/Properties

All columns: W14x120

Beams 1st floor: W24x55

Beams 2nd floor: W24x55

Lean-on column section: A=0.0976   I=7.125e-4

1.5 Concentrated Masses

Mass matrix: $\begin{bmatrix} 612.5 & 0 \\ 0 & 612.5 \end{bmatrix}$

1.6 Seismic Weight

Weight at 2nd story on lean on column : 6010

Weight at 1st story on lean on column : 6010

1.7 Time-history Data

See input file for damping, dt, acceleration record, e.t.c

1.8 Linear Elastic Material Properties

E=200000000.0

1.9 Directory

HybridFEM input script and input motion directory:

…**HybridFEMv423\Systems\Element1Example\\...**

1.10 Input Files

Input files of HybridFEM v 4.2.3 and OpenSEES are given in the next pages

HybridFEM input file (version 4.2.3)

```
UNITS: KN-m-sec
STRUCTURE DATA BLOCK
NN      NE      NM      NSEC    NDIM    NEN     NDN     ND      NSLAVED NGN
9       10      1       0       2       2       3       6       2       0
NODAL COORDINATE DATA BLOCK
NODE    X               Y               Z
1               0               6               0
2               6               6               0
3               0               3               0
4               6               3               0
5               0               0               0
6               6               0               0
7               7               0               0
8               7               3               0
9               7               6               0
BOUNDARY CONDITION
NODE  X Y Z
5 1 1 0
6 1 1 0
7 1 1 0
CONSTRAINT
M S  dof
4 8 1 0 0
2 9 1 0 0
MATERIAL DATA BLOCK
ID  TYPE      E
1               1               200000000.0
ELEMENT TYPE AND CONNECTIVITY DATA BLOCK
ELEM  TYPE  NODE1  NODE2  DampK   DampM    {Element Properties} 3
1       1       1       3       1 1 1 .022774148 .000574399 0.1
2       1       3       5       1 1 1 .022774148 .000574399 0.1
3       1       2       4       1 1 1 .022774148 .000574399 0.1
4       1       4       6       1 1 1 .022774148 .000574399 0.1
5       1       1       2       1 1 1 .010451592 .000561912 0.1
6       1       3       4       1 1 1 .010451592 .000561912 0.1
7       1       7       8       1 1 1  0.0976  7.125e-4  0.1
8       1       8       9       1 1 1  0.0976  7.125e-4  0.1
9       4       7       8       1 1 12020.0   0.0  612.64  1
10      4       8       9       1 1 6010.0   0.0  612.64  1
HYBRID TESTING DATA BLOCK
T1              T2              KSI     SF              DT   Interpolations
1.4     0.41    0.00    1       0.001    5
INTEGRATOR
TYPE   {Parameters}
1  1.0
--------END OF CONFIGURATION------------------

ELEMENT LIBRARY AND ELEMENT DATA
type1: elastic beam column element
type2: experimental link element with initial stiffness
type3: hinge (concentrated plasticity) beam column element (drain2dx type 2)
type4: 2 story gravity column
type5: SNAP (Stanford U) bilinear zero-length element with stiffenss and strength deterioration capabilities

INTEGRATOR LIBRARY
type1: CR Method
type2: Rosenbrock-W Method
```

OpenSEES input file

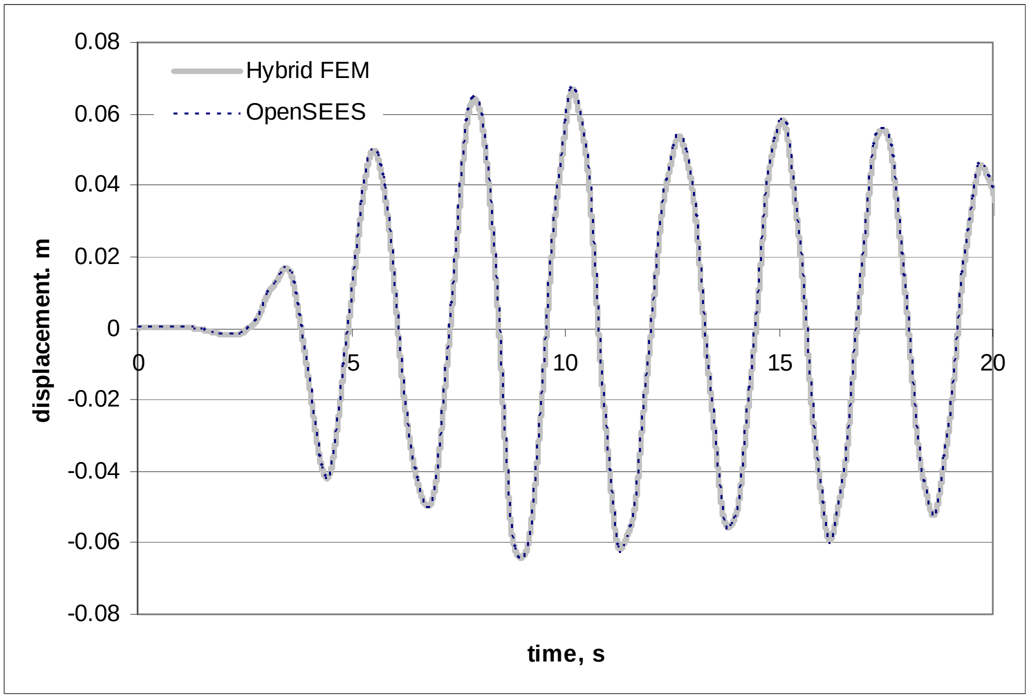
```
# To check element 4 in HybridFEM

    model BasicBuilder -ndm 2 -ndf 3

######################################
#   Define Basic Materials           #
######################################

   set E     200000000.0
   set Fy    345000.
   set b     0.01

################################################
#   Define Nodes and Boundary condition        #
################################################

   node 1 0 6
   node 2 6 6
   node 3 0 3
   node 4 6 3
   node 5 0 0
   node 6 6 0
   node 7 7 0
   node 8 7 3
   node 9 7 6
   set M [expr 612.64];
   set smallM [expr 0.001]
   mass 8 $M $smallM  $smallM
   mass 9 $M $smallM  $smallM
   fix 5 1 1 0
   fix 6 1 1 0
   fix 7 1 1 0
   equalDOF 4 8 1
   equalDOF 2 9 1

################################################
#   Define Elements, gravity load and record   #
################################################
set PTrans 10
geomTransf PDelta $PTrans
set LTrans 11
geomTransf Linear $LTrans

element elasticBeamColumn 1 1 3 0.022774148 200000000. 0.000574399 $LTrans
element elasticBeamColumn 2 3 5 0.022774148 200000000. 0.000574399 $LTrans
element elasticBeamColumn 3 2 4 0.022774148 200000000. 0.000574399 $LTrans
element elasticBeamColumn 4 4 6 0.022774148 200000000. 0.000574399 $LTrans
element elasticBeamColumn 5 1 2 0.010451592 200000000. 0.000561912 $LTrans
element elasticBeamColumn 6 3 4 0.010451592 200000000. 0.000561912 $LTrans
element elasticBeamColumn 7 7 8 0.0976 200000000. 7.125e-4 $PTrans
element elasticBeamColumn 8 8 9 0.0976 200000000. 7.125e-4 $PTrans

pattern Plain 1 Linear {
       load 8 0 -6010 0
       load 9 0 -6010 0
}
recorder Node -file Displ.out -time -node 8 9 -dof 1 disp;

######################################
#   Static Analysis                  #
######################################

system UmfPack
constraints Transformation
test EnergyIncr 1e-10 60 1
numberer Plain
algorithm Linear
integrator LoadControl 1
analysis Static
analyze 1
loadConst -time 0.0
```

OpenSEES input file-Continued

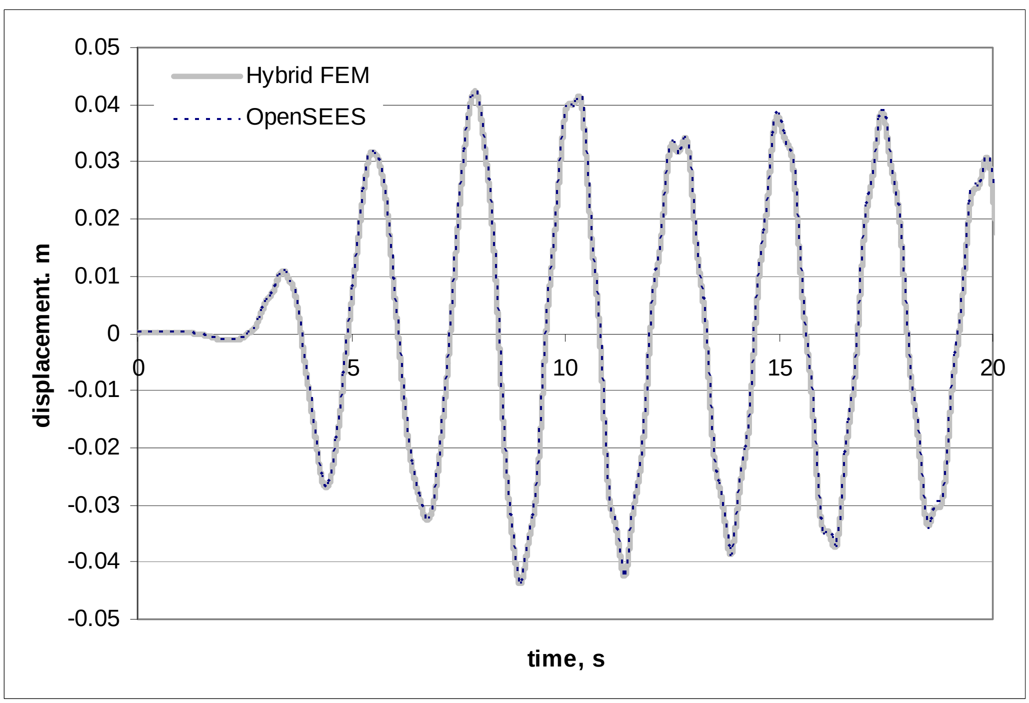
```
 ######################################
#  EQ Analysis                        #
######################################

set outgm "NR0CPC196.OPS"
set dt 0.01
set duration 20.0
set Nsteps [expr int($duration/$dt)];
set Gaccel "Series -dt $dt -filePath $outgm -factor 1.0";

pattern UniformExcitation 2 1 -accel $Gaccel

test EnergyIncr 1.0e-5 10 0
constraints Transformation
numberer Plain
algorithm Newton
integrator Newmark 0.5 0.25
analysis Transient
analyze $Nsteps $dt
wipe
```

2. RESULTS

2^{nd} story floor displacement comparison



1^{st} story floor displacement comparison



5.5: Verification example 2: Two-storey frame using displacement based beam column elements

1. DATA

1.1 Units

kN-m-s.

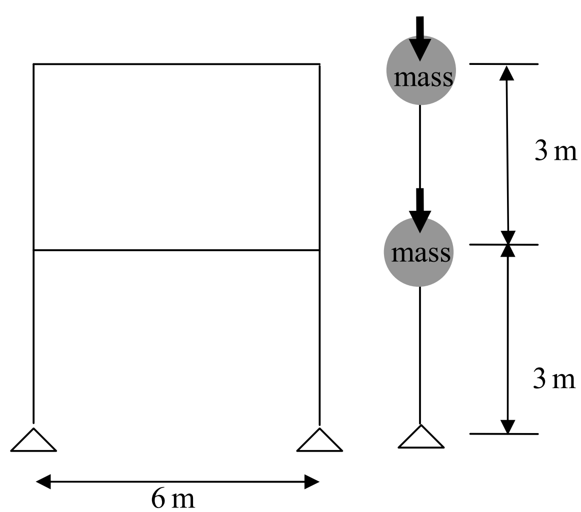
1.2 Elements Used

Type1 – Elastic Beam-Column Element

Type4 – Dummy Column Element

Type6 – Displacement Beam-Colum Element

1.3 Geometry of Structure



1.4 Element Section/Properties

All columns: W14x120

Beams 1st floor: W24x55

Beams 2nd floor: W24x55

Lean-on column section: A=0.0976 I=7.125e-4

1.5 Concentrated Masses

Mass matrix: $\begin{bmatrix} 612.5 & 0 \\ 0 & 612.5 \end{bmatrix}$

1.6 Seismic Weight

Weight at 2nd story on lean on column : 6010

Weight at 1st story on lean on column : 6010

1.7 Time-history Data

See input file for damping, dt, acceleration record, e.t.c

1.8 Nonlinear Material Properties

E=200000000.0   fy=345000

1.9 Directory

HybridFEM input script and input motion directory:

…**HybridFEMv423\Systems\Element6Example\...**

1.10 Input Files

Input files of HybridFEM v 4.2.3 and OpenSEES are given in the next pages

HybridFEM input file (version 4.2.3)

```
UNITS: KN-m-sec
STRUCTURE DATA BLOCK
NN  NE  NM      NSEC    NDIM   NEN     NDN     ND      NSLAVED        NGN
9      10      2     2      2         2       3       6       2       2
NODAL COORDINATE DATA BLOCK
NODE    X              Y              Z
 1            0              6              0
 2            6              6              0
 3            0              3              0
 4            6              3              0
 5            0              0              0
 6            6              0              0
 7            7              0              0
 8            7              3              0
 9            7              6              0
BOUNDARY CONDITION
NODE  X Y Z
5 1 1 0
6 1 1 0
7 1 1 0
CONSTRAINT
M S  dof
4 8 1 0 0
2 9 1 0 0
MATERIAL DATA BLOCK
ID Type E sigmaY E2
1 2 200000000.0 345000. 0.01
2 1 200000000.0
Section Data Block
SectionID SectionType Section data MaterialID {W14x120: h  bf tf tw}
1 1 0.367792 0.372618 0.023876 0.014986 3 10 1
2 1 0.598678 0.177927 0.012827 0.010033 3 10 1
ELEMENT TYPE AND CONNECTIVITY DATA BLOCK
ELEM  TYPE  NODE1  NODE2  DampK   DampM    {Element Properties} 3
1 6 1 2 1  1 2 5 1
2 6 3 4 1  1 2 5 1
3 6 5 3 1  1 1 5 1
4 6 3 1 1  1 1 5 1
5 6 6 4 1  1 1 5 1
6 6 4 2 1  1 1 5 1
7  1 7 8 1 1 2 0.0976 7.125e-4  0.1
8  1 8 9 1 1 2 0.0976 7.125e-4  0.1
9  4 7 8 1 1 12020.0 0.0  612.64  1
10 4 8 9 1 1 6010.0  0.0  612.64  1
GRAVITY LOADING BLOCK
NODE DOFNUM LOAD
8 2 -6010
9 2 -6010
HYBRID TESTING DATA BLOCK
T1             T2             KSI     SF             DT   Interpolations
1.4     0.41    0.00    3       0.001    5
INTEGRATOR
TYPE   {Parameters}
1  1.0
--------END OF CONFIGURATION------------------

ELEMENT LIBRARY AND ELEMENT DATA
type1: elastic beam column element
type2: experimental link element with initial stiffness
type3: hinge (concentrated plasticity) beam column element (drain2dx type 2)
type4: 2 story gravity column
type5: SNAP (Stanford U) bilinear zero-length element with stiffenss and strength
deterioration capabilities

INTEGRATOR LIBRARY
type1: CR Method
type2: Rosenbrock-W Method
```

OpenSEES input file

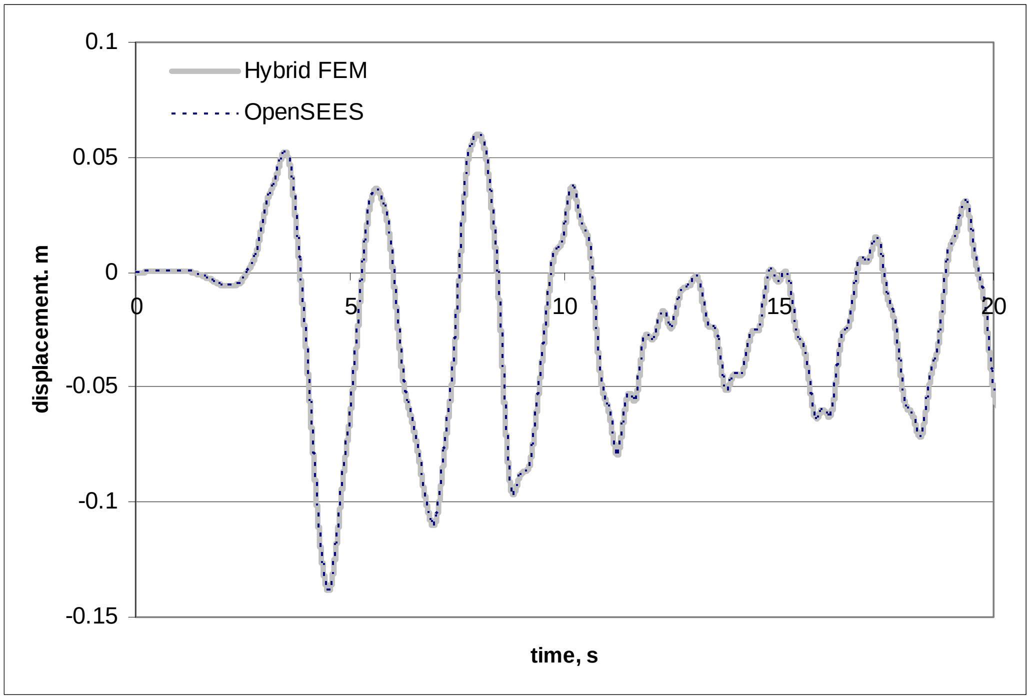
```
# To check element 4 in HybridFEM

    model BasicBuilder -ndm 2 -ndf 3
    source WFsection.tcl
    logFile test.txt

######################################
#   Define Basic Materials           #
######################################

   set E     200000000.0
   set Fy    345000.
   set b     0.01

################################################
#   Define Nodes and Boundary condition        #
################################################

   node 1 0 6
   node 2 6 6
   node 3 0 3
   node 4 6 3
   node 5 0 0
   node 6 6 0
   node 7 7 0
   node 8 7 3
   node 9 7 6

   set M [expr 612.64];
   set smallM [expr 0.001]
   mass 8 $M $smallM  $smallM
   mass 9 $M $smallM  $smallM

   fix 5 1 1 0
   fix 6 1 1 0
   fix 7 1 1 0
   equalDOF 4 8 1
   equalDOF 2 9 1

################################################
#   Define Elements                            #
################################################
   #  Beam integration points
   set nI5 5

  # Define parameters
  set ABm   0.010451592
  set IBm   0.000561912
  set ICl   0.000574399
  set ACl   0.022774148
  set dCl   0.367792
  set twCl  0.014986
  set bfCl  0.372618
  set tfCl  0.023876
  set dBm   0.598678
  set twBm  0.010033
  set bfBm  0.177927
  set tfBm  0.012827
  set tdp   0.0;     # double plate thickness
  set tcnt  0;       # continuity plate thickness

  # Define sections for beams
  uniaxialMaterial Steel01 1000 $Fy $E $b
  WFsection 1001 1000 $dBm $twBm $bfBm $tfBm 10 3

  # Columns section and materials
  WFsection 2001 1000 $dCl $twCl $bfCl $tfCl 10 3
```

OpenSEES input file- Continued

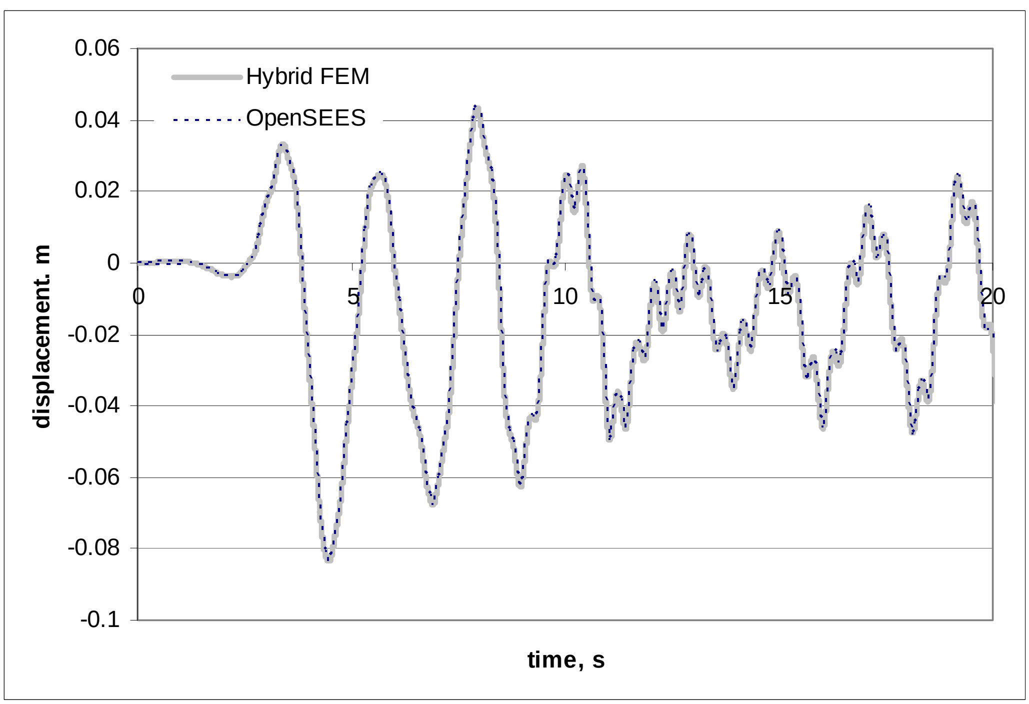
```
################################################
#   Define Elements and gravity load           #
################################################
set PTrans 10
geomTransf PDelta $PTrans
set LTrans 11
geomTransf Linear $LTrans

element dispBeamColumn 1 1 3 $nI5 2001 $LTrans -mass [expr 0.0001/9.81]
element dispBeamColumn 2 3 5 $nI5 2001 $LTrans -mass [expr 0.0001/9.81]
element dispBeamColumn 3 2 4 $nI5 2001 $LTrans -mass [expr 0.0001/9.81]
element dispBeamColumn 4 4 6 $nI5 2001 $LTrans -mass [expr 0.0001/9.81]
element dispBeamColumn 5 1 2 $nI5 1001 $LTrans -mass [expr 0.0001/9.81]
element dispBeamColumn 6 3 4 $nI5 1001 $LTrans -mass [expr 0.0001/9.81]
element elasticBeamColumn 7 7 8 0.0976 200000000. 7.125e-4 $PTrans
element elasticBeamColumn 8 8 9 0.0976 200000000. 7.125e-4 $PTrans

pattern Plain 1 Linear {
        load 8 0 -6010 0
        load 9 0 -6010 0
}

#####################################
# recorder and analysis             #
#####################################

recorder Node -file Displ.out -time -node 8 9 -dof 1 disp;

system UmfPack
constraints Transformation
test EnergyIncr 1e-10 60 1
numberer Plain
algorithm Linear
integrator LoadControl 1
analysis Static
analyze 1
loadConst -time 0.0

set outgm "NR0CPC196.OPS"
set dt 0.01
set duration 20.0
set Nsteps [expr int($duration/$dt)];
set Gaccel "Series -dt $dt -filePath $outgm -factor 3.0";

pattern UniformExcitation 2 1 -accel $Gaccel

test EnergyIncr 1.0e-5 10 0
constraints Transformation
numberer Plain
algorithm Newton
integrator Newmark 0.5 0.25
analysis Transient
analyze $Nsteps $dt
wipe
```

2. RESULTS

2nd story floor displacement comparison



1st story floor displacement comparison



5.6: Verification example 2: Inelastic one-story frame using panel zone elements

1. DATA

1.1 Units

kN-m-s.

1.2 Elements Used

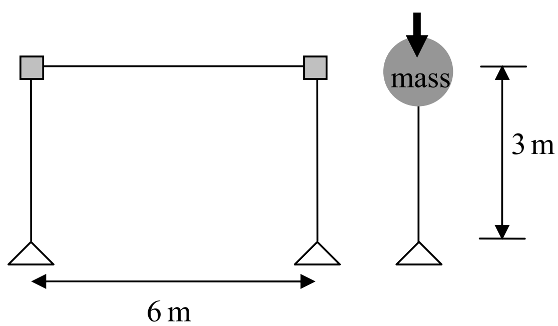
Type1 – Elastic Beam-Column Element

Type4 – Dummy Column Element

Type6 – Displacement Beam-Colum Element

Type9 – Panel Zone Element

1.3 Geometry of Structure



1.4 Element Section/Properties

All columns: W14x120 (Linear elements)

Beams 1st floor: W24x55 (Nonlinear element)

Lean-on column section: A=0.0976   I=7.125e-4

1.5 Concentrated Masses

Mass: 612.64 on the lean-on column node

1.6 Seismic Weight

Seismic weight at 1st story on lean on column : 6010

1.7 Time-history Data

See input file for damping, dt, acceleration record, etc.

1.8 Nonlinear Material Properties

E=200000000.0   fy=345000

1.9 Directory

HybridFEM input script and input motion directory:

…**HybridFEMv423\Systems\Element9Example\...**

1.10 Input Files

Input files of HybridFEM v 4.2.3 and OpenSEES are given in the next pages

HybridFEM input file (version 4.2.3)

```
UNITS: KN-m-sec
STRUCTURE DATA BLOCK
NN NE NM NSEC NDIM NEN NDN ND NSLAVED NGN
12 7 3 2 2 2 3 6 1 1
NODAL COORDINATE DATA BLOCK
NODE X Y Z UX UY THETA
1   0.  0.  0.
2  -0.183896 3. 0.
3   0. 2.700661 0.
4   0.183896 3. 0.
5   0. 3.299339 0.
6   6. 0. 0.
7   5.816104 3. 0.
8   6. 2.700661 0.
9   6.183896 3. 0.
10  6. 3.299339 0.
11  7. 3.       0.
12  7.0 0.0   0.
BOUNDARY CONDITION
NODE  X Y Z
1 1 1 0
6 1 1 0
12 1 1 0
CONSTRAINT
M  S  dof
11 9 1 0 0
MATERIAL DATA BLOCK
ID Type E sigmaY E2
1 2 200000000.0 345000. 0.01
2 5 1097.9 1341.719 0.00155 0.006201
3 1 200000000.0
Section Data Block
SectionID SectionType Section data MaterialID {W14x120: h  bf tf tw}
1 1 0.367792 0.372618 0.023876 0.014986 3 10 1
2 1 0.598678 0.177927 0.012827 0.010033 3 10 1
ELEMENT TYPE AND CONNECTIVITY DATA BLOCK
ELEM TYPE NODE1 NODE2 DampK   DampM {secID nIP DistLoad}
1 6 1 3  1 1 1 5 0.1
2 6 6 8  1 1 2 5 0.1
3 6 4 7  1 1 1 5 0.1
4 9 2 3 4  5 1 1 3 0.000574399 0.022774148 0.372618 0.014986 0.023876 0. 0. 2 0.1
5 9 7 8 9 10 1 1 3 0.000574399 0.022774148 0.372618 0.014986 0.023876 0. 0. 2 0.1
6 1 12 11 1 1 3 0.0976 0.0007125 0.1
7 4 12 11 1 1 6010. 0. 612.64 1
GRAVITY LOADING BLOCK
DOFNUM LOAD
11 2 -6010
HYBRID TESTING DATA BLOCK
T1  T2  SI  SF  DT
0.127443  10.0 0.0  3.0  0.001 2
INTEGRATOR
TYPE   {Parameters}
1 1
```

OpenSEES input file

```
    model BasicBuilder -ndm 2 -ndf 3
    source WFsection.tcl
    logFile test.txt

######################################
#   Define Basic Materials           #
######################################

# Steel Material
set E     200000000.0
set G     [expr $E/(2*(1+0.3))];
set Fy    345000.
set b     0.01

################################################
#   Define Nodes and Boundary Conditions       #
################################################

node 1 0.      0.
node 2 -0.183896 3.
node 3 0. 2.700661
node 4 0.183896 3.
node 5 0. 3.299339
node 6 6. 0.
node 7 5.816104 3.
node 8 6. 2.700661
node 9 6.183896 3.
node 10 6. 3.299339
node 11 7. 3.
node 12 7. 0.

mass 11  612.64 0.0 0.0;
fix 1 1 1 0
fix 6 1 1 0
fix 12 1 1 0
equalDOF 9 11 1

################################################
#   Define Elements                            #
################################################
#  Beam integration points
set nI5 5

# Define parameters
set ABm   0.010451592
set IBm   0.000561912
set ICl   0.000574399
set ACl   0.022774148
set dCl   0.367792
set twCl  0.014986
set bfCl  0.372618
set tfCl  0.023876
set dBm   0.598678
set twBm  0.010033
set bfBm  0.177927
set tfBm  0.012827
set tdp   0.0;     # double plate thickness
set tcnt  0;       # continuity plate thickness

# Define sections for beams
uniaxialMaterial Steel01 1000 $Fy $E $b
WFsection 1001 1000 $dBm $twBm $bfBm $tfBm 10 3

# Columns section and materials
WFsection 2001 1000 $dCl $twCl $bfCl $tfCl 10 3

# define material for panel zone element
set d1   $dCl
set d2   [expr $dBm + 2.0*$tcnt]
```

OpenSEES input file- continued

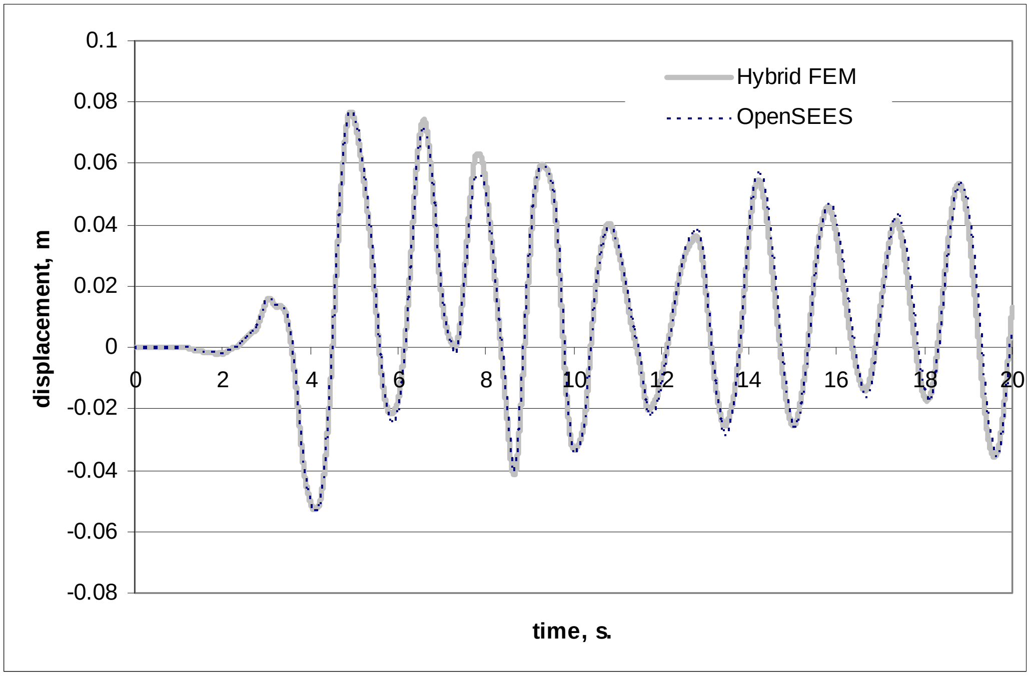
```
#  Axial stiffness
set SecA1 [expr $d2*($twCl+$tdp) + 2.*$tcnt*($bfCl-$twCl) ]
set K1    [expr $E*$SecA1/$d1];
set SecA2 [expr $ACl + ($d1 - 2.*$tfCl)*$tdp]
set K2    [expr $E*$SecA2/$d2];
#  Bending stiffness
set I11   [expr $d2*$d2*$d2*$twCl/12.];
set I12   [expr 2.*$tcnt*$tcnt*$tcnt*($bfCl-$twCl)/12.];
set I13   [expr 2.*($bfCl-$twCl)*$tcnt*($d2-$tcnt)*($d2-$tcnt)/4];
set I14   [expr $d2*$d2*$d2*$tdp/12.];
set K4    [expr $E*($I11+$I12+$I13+$I14)/$d1];
set I21   [expr ($d1-2.*$tfCl)*($d1-2.*$tfCl)*($d1-2.*$tfCl)*$tdp/12.];
set K5    [expr $E*($I21+$ICl)/$d2];
#  Asym Bending stiffness
set K6   [expr 3.*$E*($I11+$I12+$I13+$I14)/$d1];
set K7   [expr 3.*$E*($I21+$ICl)/$d2];
#  Asym Shear Stiffness
set K8   [expr 6.*$E*$G*$SecA1*$d1/(3.*$E*$d2*$d2 +2.*$G*$d1*$d1)];
set K9   [expr 6.*$E*$G*$SecA2*$d2/(3.*$E*$d1*$d1 +2.*$G*$d2*$d2)];

############################################################################
#  Shear stiffness for Panel Zone element
############################################################################
set Vy     [expr $Fy*$dCl*($twCl + $tdp)/1.732];
set Vupar1 [expr $Fy*$dCl*$twCl/1.732];
set Vupar2 [expr 1.0 + 3.45*$bfCl*$tfCl*$tfCl/($dBm*$dCl*$twCl)];
set Vupar3 [expr $Fy*($dCl-$tfCl)*$tdp/1.732];
set Vu     [expr $Vupar1*$Vupar2+ $Vupar3];
# shear strain gamma
set gammay [expr $Fy/($G*1.732)];
set gammau [expr 4.*$gammay];
# deformation due to distortion
set Dy     [expr $gammay *$dBm];
set Du     [expr $gammau *$dBm];
# equivalent spring stiffness
set Kinit  [expr $Vy/$Dy]
set Ky     [expr ($Vu-$Vy)/($Du-$Dy)]
set Ku     [expr 0.*$Kinit]
# decompose spring stiffness
set KEppPZ  [expr $Kinit-$Ky]
set FySt01  [expr $Ky*$Du]
set aSt01   [expr $Ku/$Ky]

uniaxialMaterial Elastic 1001   [expr $K1*100]
uniaxialMaterial Elastic 1002   [expr $K2*100]
uniaxialMaterial Elastic 1004   [expr $K4*3]
uniaxialMaterial Elastic 1005   [expr $K5*3]
uniaxialMaterial Elastic 1006   [expr $K6*100]
uniaxialMaterial Elastic 1007   [expr $K7*100]
uniaxialMaterial Elastic 1008   [expr $K8*100]
uniaxialMaterial Elastic 1009   [expr $K9*100]

# inelastic material for shear mode deformation
uniaxialMaterial ElasticPP 1101 $KEppPZ   $Dy
uniaxialMaterial Steel01   1102 $FySt01   $Ky   $aSt01
uniaxialMaterial Parallel  1003 1101 1102

############################################################################
#  Define elements
############################################################################
set PTrans 11
geomTransf PDelta $PTrans

element dispBeamColumn 1 1 3 $nI5 2001 $Trans -mass [expr 0.01/9.81]
element dispBeamColumn 2 6 8 $nI5 2001 $Trans -mass [expr 0.01/9.81]
element dispBeamColumn 3 4 7 $nI5 1001 $Trans -mass [expr 0.01/9.81]
element LehighJoint 4 2 3 4 5  1001 1002 1003 1004 1005 1006 1007 1008 1009
element LehighJoint 5 7 8 9 10 1001 1002 1003 1004 1005 1006 1007 1008 1009
element elasticBeamColumn 6 12 11 0.0976 200000000. 7.125e-4 $PTrans
```

OpenSEES input file- continued

```
#####################################
# gravity load
####################################

pattern Plain 1 Linear {
        load 11 0 -6010 0
}

#####################################
# recorder
####################################

recorder Node -file Displ.out -time -node 11 -dof 1 disp;

######################################
#   Analysis                         #
######################################

system UmfPack
constraints Transformation
test EnergyIncr 1e-10 60 1
numberer Plain
algorithm Linear

integrator LoadControl 1
analysis Static
analyze 1
loadConst -time 0.0


######################################
# Display Frames                     #
######################################

set lambda  [eigen 1];
set lambda1 [lindex $lambda 0];
set omega1  [expr pow($lambda1,0.5)];
set T1      [expr 2.*3.1415/$omega1];
puts "$T1 "

set outgm "NR0CPC196.OPS"
set dt 0.01
set timeIncr [expr $dt/10];
set endTime 20.0
set Nsteps [expr int($endTime/$timeIncr )];
set Gaccel "Series -dt $dt -filePath $outgm -factor 3.0";

pattern UniformExcitation 2 1 -accel $Gaccel

set currentTime 0.0;
set calDt  $timeIncr;
source SolutionAlgorithm.tcl;
```

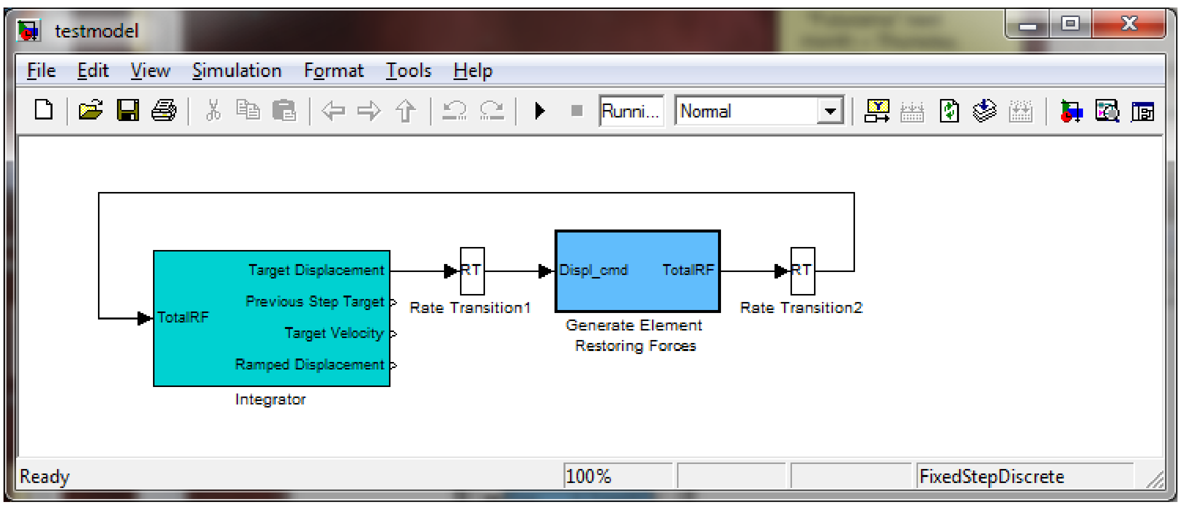
2. RESULTS

Floor displacement comparison



6. REAL-TIME SIMULATION

The HybridFEM program has the capabilities of running models in real-time using SIMULINK, Real Time Workshop and xPC. In the example systems, the file, "RealtimeSetup.m" contains the code to set up the system and the file, "RealtimeModel.mdl" contains the SIMULINK blocks that run the integration algorithm and compute the restoring forces. For new configuration, the user must run "ModelGenerator.m" in order to create a new SIMULINK model. The model is multi-tasking meaning that the integration time step is a multiple of the fundamental sample rate. This allows for actuator ramping at the fundamental sample rate while the complex integration algorithm performs its calculations over the full time step.



In the above system, the light blue block, "Integrator" represents the Integration algorithm. It generates the displacement command and accepts the total restoring force as input. It connects to the "Generate Element Restoring Forces" block which performs the per element restoring force calculations. The "Rate Transition" blocks are necessary to bridge the multi-tasking architecture since the "Integrator" block runs at the fundamental sample rate and the "Generate Element Restoring Forces" block runs at the integration time step.

To interface with this model, the user can connect to either the “Target Displacement” or “Ramped Displacement” output from the “Integrator” block. The “Target Displacement” output is the displacement command generated at every integration time step and the “Ramped Displacement” is the same single run through a linear interpolator at the fundamental sampling rate

7. REFERENCES

[1] Chen ,C. and J.M. Ricles, (2007) Development of Direct Integration Algorithms for Structural Dynamics using Discrete Control Theory, Journal of Engineering Mechanics, ASCE. Vol. 134, No. 8., p. 676-683

[2] Lamarche, C., P., Bonelli, A., Bursi, O., S., and Tremblay, R., (2009) A Rosenbrock-W Method for Real-Time Dynamic Substructuring and Pseudo-Dynamic Testing, Earthquake Engineering & Structural Dynamics, Vol. 38, No. 9., p. 1071–1092

[3] Prakash, V., Powell, G., H., and Campbell, S., (1993) DRAIN-2DX Base Program Description and User Guide, Version 1.10, UCB/SEMM-1993/17, Berkeley: Department of Civil Engineering, University of California.

[4] Mazzoni, S., McKenna, F., Scott, H., M., and Fenves, G., L., (2000) Open System for Earthquake Engineering Simulation User Command-Language Manual, Pacific Earthquake Engineering Research Center, University of California, Berkeley

[5] Wen, Y., K., (1976) Method for Random Vibration of Hysteretic Systems, Journal of Engineering Mechanics. ASCE. Vol. 102, No. 2. p. 249-263

[6] Wu, S., (1995) Seismic Analysis and Retrofit of Non-Ductile Reinforced Concrete Frame Buildings, Ph.D. Dissertation, Department of Civil and Environmental Engineering, Lehigh University.

[7] Lignos DG, Krawinkler H. (2007) A Database in Support of Modeling of Component Deterioration for Collapse Prediction of Steel Frame Structures. ASCE Structures Congress, SEI institute, Long Beach CA.

[8] Spacone, E., Ciampi, V., and Filippou, F., C., (1992) A Beam Element for Seismic Damage Analysis. Report No. UCB/EERC-92/07. Earthquake Engineering Research Center, College of Engineering, University of California, Berkeley

[9] Seo, C-Y, Lin Y-C, Sause R, Ricles J. (2009) Development of Analytical Models for 0.6 Scale Self-Centering MRF with Beam Web Friction Devices. 6th International Conference for Steel Structures in Seismic Area (STESSA), Philadelphia.