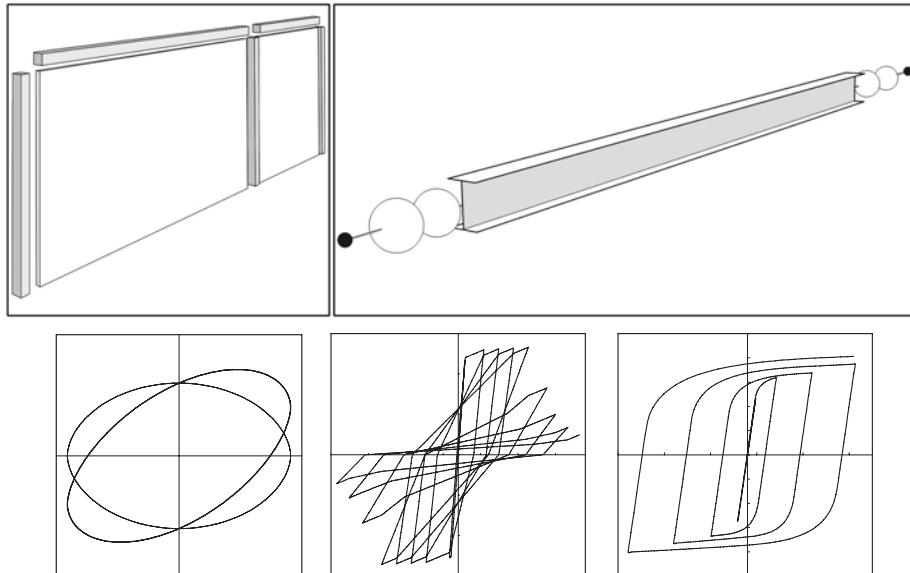


---

*Platform of Inelastic Structural Analysis  
for 3D Systems*

---

**PISA3D**  
**Standard Edition R3.2**  
**User's Manual**



**Bo-Zhou Lin**

**Yi-Jer Yu**

**Ming-Chieh Chuang**

**Keh-Chyuan Tsai**

---

*National Center for Research on Earthquake Engineering, Taiwan*

*Department of Civil Engineering, National Taiwan University*

All rights reserved, March, 2011

## Table of Contents

<b>Introduction</b>	<b>I</b>
<b>Notes on Running Analysis</b>	<b>II</b>
<b>Contents of Output Files</b>	<b>IV</b>
<b>PART A. Initial Setting Command</b>	<b>A-1</b>
A01. Analysis Platform Line	A-1
A02. Analysis Title Line	A-1
A03. Force Unit Line	A-2
A04. Length Unit Line	A-2
<b>PART B. Analysis Method Command</b>	<b>B-1</b>
B01. One-Step Nonlinear Static Analysis	B-1
B02. Load Control Nonlinear Static Analysis	B-2
B03. Displacement Control Nonlinear Static Analysis	B-3
B04. Modal Analysis	B-5
B05. Nonlinear Dynamic Analysis (Newmark Method)	B-6
B06. Nonlinear Dynamic Analysis (OS Method)	B-8
<b>PART C. Analysis Control Setting Command</b>	<b>C-1</b>
C01. Geometric Nonlinear Effect	C-1
<b>PART D. Node Data Command</b>	<b>D-1</b>
D01. Node Generation	D-1
D02. Nodal Degree of Freedom Assign	D-2
D03. Nodal Lump Mass Assign	D-4
D04. Nodal Linear Spring	D-5
D05. Constraint Definition (Rigid Diaphragm)	D-6
<b>PART E. Load Pattern Command</b>	<b>E-1</b>
E01. Nodal Load	E-1
E02. Ground Acceleration Record	E-2
<b>PART F. Material Definition Command</b>	<b>F-1</b>
F01. Elastic Material	F-1
F02. Bilinear Material	F-3
F03. Hardening Material	F-5

F04. Degrading Material	F-8
F05. Bilinear-Elastic Material	F-12
F06. Bilinear02 Material	F-14
F07. Fracture Material	F-16
F08. Buckle Material	F-21
F09. TensionOnlyBilinear Material	F-28

---

<b>PART G. Element Definition Command</b>	<b>G-1</b>
---	------------

---

G01. Truss Element	G-1
G02. BeamColumn Element	G-5
G03. Joint Element	G-12
G04. Panel Element	G-16
G05. Damper Element	G-22
G06. BilinearDamper Element	G-27

---

<b>PART H. Other Correlative Element Command</b>	<b>H-1</b>
--	------------

---

H01. Frame Section Definition - BCSection Command	H-1
H02. Rigid End Zone Definition	H-3
H03. Interaction Surface Definition (Wide-Flange Steel Type)	H-5
H04. Interaction Surface Definition (RC Column Type)	H-9
H05. Element Load (Fixed End Force)	H-11
H06. Element Load Case	H-13
H07. Element Section Definition - BCSection02 Command	H-14
H08. Element Section Definition - BCSection03 Command	H-16

---

<b>PART I. Output Setting Command</b>	<b>I-1</b>
---------------------------------------	------------

---

I01. Output Interval Setting Command	I-1
I02. Output Nodal Absolute Responses	I-3
I03. Output Nodal Relative Responses	I-4
I04. Output element responses in a specified file	I-5
I05. Output nodal responses in a specified file	I-6

---

<b>PART J. End Input Command</b>	<b>J-1</b>
----------------------------------	------------

---

J01. Termination of Input File	J-1
--------------------------------	-----

## References

## Introduction

Structural analysis is a very important procedure in modern research and practice of earthquake engineering. The demands on nonlinear dynamic structural analysis are increasing and changing rapidly. This program adopts the Unified Process and C++, a combined programming language with object-oriented mechanisms, to construct a new object-oriented structural analysis computational platform entitled “Platform of Inelastic Structural Analysis for 3D Systems”, PISA3D.

PISA3D provides a user-friendly input format for commands with full free format. Engineers and researchers can apply PISA3D to simulate the responses of structural systems under static loads or displacements, cyclic loads or displacements, earthquake ground accelerations, and earthquake aftershocks by combining various analysis methods. In addition, the PISA3D is well extensible and easy to maintain due to object-oriented nature of the framework. Users can make replacements, derivation, or combination of the object libraries in PISA3D to solve different types of problems in the structural simulation.

Users can conveniently build nonlinear numerical models for 3D structures using the material and element libraries provided in PISA3D. Currently, there are 8 types of yielding rules in the material library, including linearly-elastic, bilinear, two-surface plastic hardening, 3 parameters degrading, Bilinear-Elastic, Bilinear02, Fracture and Buckle Material. The element library of PISA3D currently consists of 5 types of nonlinear elements, including truss element, beamcolumn element with the hinge model, joint element, structural panel element, and velocity-dependent damper element with the Kelvin model. All these elements can use the objects in the material library, therefore, a total of 40 elements with different characteristics is available for the simulation of the structural responses.

This manual describes the commands of nonlinear elements and materials in detail, and available analysis commands are also explained. A graphic user interface GISA3D is also available to allow the user to graphically inspect the accuracy of the numerical model, statically or dynamically display the mode shapes, deformed shapes, extent and locations of plastic hinge formations.

## Notes on Running Analysis

1. At the beginning of any analysis, the existing output files would be deleted and replaced by new output files.  
So be sure that the access authority of these files is not held by other applications.
2. Any single command must occupy one line in the input file.  
The beginning of any command in the input file must precisely match the Control Words described in this manual.
3. In any input file, one can make notes or comments by starting a line with any symbols except the Control Words.  
That is, any line does not start with Control Words will be treated as comments or notes.
4. Any characters can be entered in either capital or lowercase letters, and all letters are taken into capital cases by PISA3D.
5. Some text editors use tab characters to replace successive blanks. Be sure that no tabs exist while saving input files.
6. In the command with itself tag, like node, element, or material, the tag must be unique in the same type of command. But the same tags over different types of commands (ex: element and node) are available.
7. Some parameters in commands have default values, so the user can leave these parameters blanks to use the default values.  
If the user wants to input some of these parameters oneself, the default values before the objective parameters must be input.
8. The global coordinate system is always set up as:  
Positive X axis is in the orientation towards right side of the screen.  
Positive Y axis is in the orientation towards upper side of the screen,  
and positive Z axis is in the orientation towards out of the screen.
9. For large 3D structural models and analysis methods with thousands of analysis

steps, the size of output files would be massive while the output intervals are too small. Increasing the output intervals can save the analysis time and the disk storage space significantly. Additionally, only specify interested elements and nodes to output will also save much analysis time and space.

10. The input file can be put in any folder, but the file name (including the folder path) should be all in English characters and rightful regulation of MS Windows.

## Contents of Output Files

After executing PISA3D, a number of permanent files are created. All files have names of the form XXX.Ext, where XXX is the input file name and Ext is the extension name. The extension names have the following meanings.

1. Echo

A text file containing an echo of the input data plus an analysis log. The user can check the correctness of the model by the echo data. Additionally, any user's input errors would be recorded in this file.

2. Element

A text file containing responses of the output elements at analysis steps specified to record.

This file is created while command Output->OutFlag->ElemCode = 0 or 1.

3. ElemRecord

A text file containing the contents of XXX.Element, but rearrange each output element's own responses together by analysis time sequence.

This file is created while command Output->OutFlag->ElemCode = 1 or 2.

4. ElemEnvelope

A text file containing response envelopes of the output elements at analysis steps specified to record.

5. SectionX

A text file containing X direction sectional forces of element groups at analysis steps specified to record.

6. SectionY

A text file containing Y direction sectional forces of element groups at analysis steps specified to record.

7. SectionZ

A text file containing Z direction sectional forces of element groups at analysis steps specified to record.

8. Energy  
A text file containing system energy distribution and strain energy of element groups at analysis steps specified to record.
9. NodeAbsDisp  
A text file containing absolute displacements of the output nodes at analysis steps specified to record.  
This file is created while command Output->OutFlag->NodeCode = 0 or 1.
10. NodeDisRecord  
A text file containing the contents of XXX.NodeAbsDisp, but rearrange each output node's own displacements together by analysis time sequence.  
This file is created while command Output->OutFlag->NodeCode = 1 or 2.
11. NodeVelRecord  
A text file containing the absolute velocities to the ground of output nodes at analysis steps specified to record.  
This file is created while any dynamic analysis method exists and command Output->OutFlag->NodeCode = 1 or 2.
12. NodeAccRecord  
A text file containing the absolute acceleration to the ground of output nodes at analysis steps specified to record.  
This file is created while any dynamic analysis method exists and command Output->OutFlag->NodeCode = 1 or 2.
13. NodeRelDisp  
A text file containing relative displacements of specified nodes at analysis steps specified to record.  
This file is created while command Output->NodeRel exists.
14. NodeRelVel  
A text file containing relative velocities of specified nodes at analysis steps specified to record.  
This file is created while any dynamic analysis method exists and command Output->NodeRel exists.



15. NodeRelAcc  
A text file containing relative acceleration of specified nodes at analysis steps specified to record.  
This file is created while any dynamic analysis method exists and command Output->NodeRel exists.
16. NodeEnvelope  
A text file containing response envelopes of the output nodes at analysis steps specified to record.
17. Eigen  
A text file containing modal analysis results, This file is created while any modal analysis method exists.
18. VISA3D  
A binary file containing analysis data for post processing program VISA3D.
19. showElem  
A binary file containing responses of elements for VISA3D.
20. showNode  
A binary file containing responses of nodes for VISA3D.
21. showMode  
A binary file containing modal analysis results for VISA3D.
22. showEngy  
A binary file containing energy distribution data for VISA3D.

## PART A. Initial Setting Command

The following 4 commands initialize the analysis problem, including analysis platform indication “PISA3D”, problem title, force unit, and length unit. These 4 commands must be put at the beginning of the input file.

### A01. Analysis Platform Line

#### Command:

PISA3D

PARAMETER	DESCRIPTION
PISA3D	This control word "PISA3D" is necessary to start executing PISA3D, and it must be put on the first line of the input file.

### A02. Analysis Title Line

#### Command:

Any strings

PARAMETER	DESCRIPTION
Any strings	One can put any strings on this line to describe the project analyzed.  This line would not affect the analysis process, but at least one character or numeral is necessary.

### **A03. Force Unit Line**

#### **Command:**

Force Unit

PARAMETER	DESCRIPTION
Force Unit	Put the force unit on this line, and at least one character is necessary.

### **A04. Length Unit Line**

#### **Command:**

Length Unit

PARAMETER	DESCRIPTION
Length Unit	Put the length unit on this line, and at least one character is necessary.

#### **EXAMPLE:**

PISA3D

A 6 Stories 3D Frame Earthquake Simulation

kN

mm

## PART B. Analysis Method Command

These commands identify the analysis methods executed on the model. In a single analysis, PISA3D can perform plurality of analysis methods with the same/different types in any preferred order. The number of the analysis methods in one analysis procedure is unlimited, but PISA3D always performs analysis as the order of these analysis commands.

### B01. One-Step Nonlinear Static Analysis

This analysis method will apply the static load patterns specified by user in one step.

#### Command:

```
Analysis Gravity LoadPtn_1? LoadFac_1? ... LoadPtn_N? LoadFac_N?
```

PARAMETER	DESCRIPTION
Analysis	Control Word.
Gravity	Perform one-step static analysis.
LoadPtn_1	The tag of the 1st applied load pattern.
LoadFac_1	The load factor of the 1st load pattern.
LoadPtn_N	The tag of the Nth applied load pattern.
LoadFac_N	The load factor of the Nth load pattern.

#### EXAMPLE:

```
Analysis gravity DL 1.4 LL 1.7
```

//This command specifies a one step static analysis method with load pattern 1.4DL+1.7LL.

## B02. Load Control Nonlinear Static Analysis

In this analysis method, the load patterns will be applied in a number of steps with increments.

### Command:

```
Analysis LoadControl LoadPtn_1? LoadFac_1? ... LoadPtn_N? LoadFac_N?  
LoadSteps?
```

PARAMETER	DESCRIPTION
Analysis	Control Word.
LoadControl	Perform iterative-incremental static analysis with the load control method.
LoadPtn_1	The tag of the 1st applied load pattern.
LoadFac_1	The load factor of the 1st load pattern.
LoadPtn_N	The tag of the Nth applied load pattern.
LoadFac_N	The load factor of the Nth load pattern.
LoadSteps	Number of analysis steps to reach the values of the defined load patterns during the analysis process, must > 0.

### EXAMPLE:

```
Analysis LoadControl DL 1.4 LL 1.7 EQ 0.5 100
```

//This command specifies a load control analysis with load pattern  $1.4 \times DL + 1.7 \times LL + 0.5 \times EQ$  in 100 steps.

### B03. Displacement Control Nonlinear Static Analysis

In this method, the displacement will be applied in a number of steps with increments. The load patterns in displacement control method are just distribution reference of the external force, and the analysis completes when the total displacement is applied.

#### Command:

```
Analysis AbsDispControl LdPtn_1? LdFac_1? ... LdPtn_N? LdFac_N? totalDisp?  
Steps? NodeTag? DOF?
```

PARAMETER	DESCRIPTION
Analysis	Control Word.
AbsDispControl	Perform iterative-incremental static analysis with the displacement control method.
LoadPtn_1	The tag of the 1st applied load pattern.
LoadFac_1	The load factor of the 1st load pattern.
LoadPtn_N	The tag of the Nth applied load pattern.
LoadFac_N	The load factor of the Nth load pattern.
totalDisp	Total displacement during this analysis procedure.
Steps	Number of analysis steps to apply the total displacement during this analysis procedure, must > 0.
NodeTag	The tag of the node to be controlled.
DOF	The controlled DOF of the node. UX = Global X direction displacement. UY = Global Y direction displacement. UZ = Global Z direction displacement. RX = Global X axis rotation. RY = Global Y axis rotation. RZ = Global Z axis rotation.

**EXAMPLE:**

Analysis AbsDispControl DL 1.2 LL 1.6 10 5 n15 UX

Analysis AbsDispControl DL 1.2 LL 1.6 -20 10 n15 UX

Analysis AbsDispControl DL 1.2 LL 1.6 30 15 n15 UX

Analysis AbsDispControl DL 1.2 LL 1.6 -40 20 n15 UX

//These commands specify a sequence of displacement control analyses with load

//pattern  $1.2 \times DL + 1.6 \times LL$ , and control DOF UX of node n15 to displacement 10, -10,

//20, and -20.

## B04. Modal Analysis

Modal analysis is to calculate the mode shapes and the natural periods. This analysis method can be executed for any state of the model, and proceeding/following analysis methods would not be affected. In modal analysis, user can get the coefficients ( $\alpha$  and  $\beta$ ) of linear damping ( $[C] = \alpha [M] + \beta [K]$ ) by specifying two modes and the corresponding damping ratios.

### Command:

```
Analysis ModeShape reqdModes? ID_M1? ID_M2? C1? C2?
```

PARAMETER	DESCRIPTION
Analysis	Control Word.
ModeShape	Perform modal analysis.
reqdModes	Number of modes required to be calculated, must > 0 If "reqdModes" is greater than the number of free DOFs, only the number of free DOFs would be calculated.
ID_M1	The ID number of the 1st mode specified to get $\alpha$ and $\beta$ . (Must > 0 and <= reqdModes).
ID_M2	The ID number of the 2nd mode specified to get $\alpha$ and $\beta$ . (Must > 0 and <= reqdModes).
C1	The damping ratio of the 1st mode specified to get $\alpha$ and $\beta$ .
C2	The damping ratio of the 2nd mode specified to get $\alpha$ and $\beta$ .

### NOTE:

1. The specification of the two modes must be different in order to calculate  $\alpha$  and  $\beta$ .

### EXAMPLE:

```
Analysis ModeShape 6 1 2 0.02 0.02
```

//This command specifies a mode shape analysis to calculate 6 modes, and use mode 1 and mode 2, 0.02 as damping ratio, to get  $\alpha$  and  $\beta$ .



## B05. Nonlinear Dynamic Analysis (Newmark Method)

Dynamic analysis can get the structural responses under ground acceleration. At most 3 directions of the acceleration records can be input.

### Command:

Analysis Dynamic Newmark XGndMotion? XFac? YGndMot? YFac?  
ZGndMot? ZFac? StpLength? AnaSteps?  $\alpha$ ?  $\beta$ ?  $\beta_0$ ?

PARAMETER	DESCRIPTION
Analysis	Control Word.
Dynamic	Perform dynamic analysis.
Newmark	With Newmark- $\beta$ integration algorithm.
XGndMotion	Tag of X-direction(horizontal) ground motion load pattern.
XFac	Magnitude factor of X-direction(horizontal) ground motion.
YGndMot	Tag of Y-direction(vertical) ground motion load pattern.
YFac	Magnitude factor of Y-direction(vertical) ground motion.
ZGndMot	Tag of Z-direction(horizontal) ground motion load pattern.
ZFac	Magnitude factor of Z-direction(horizontal) ground motion.
StpLength	Time length of each integration step, must $> 0$ .
AnaSteps	Total integration steps, must $> 0$ .
$\alpha$	Mass proportional damping factor.
$\beta$	Tangent stiffness proportional damping factor.
$\beta_0$	Initial stiffness proportional damping factor. (Default Value = 0)

### NOTE:

1.  $\text{StpLength} \times \text{AnaSteps} = \text{Total analysis time}$ . The user can change either StpLength or AnaSteps as requirements.
2. One can check the correctness and precision of dynamic analysis by the **Unbalanced Energy Ratio** in the file XXX.Energy. Lower Unbalanced Energy Ratio during analysis reflects higher analysis correctness and precision. Shorten

StpLength can usually raise the precision, but it will also cost more analysis duration.

3. If only one or two direction(s) have ground excitation, the ground motion load pattern of rest direction(s) should input as “None”.
4. The linear structural damping can be modeled as  $[C] = \alpha [M] + \beta [K_t]$ , or  $[C] = \alpha [M] + \beta_0 [K_0]$ . Where  $[K_t]$  is the structural tangent stiffness and  $[K_0]$  is the structural initial stiffness. The user should choose one of  $\beta$  and  $\beta_0$  to input, or the damping effects of the structure will be taken into account improperly as  $\beta$  and  $\beta_0$  are input simultaneously.

### **EXAMPLE:**

```
Analysis Dynamic Newmark EQEW 0.33 none 0 EQSN 0.33 0.04 1000  
1.00221388E-001 3.86725431E-003 0
```

//This command defines a two way (X-dir. and Z-dir.) earthquake analysis.

//Note: This command should occupy one line of the input file.

## B06. Nonlinear Dynamic Analysis (OS Method)

This analysis method can get the structural responses under three-dimensional ground acceleration. The Operator-Splitting (OS) integration algorithm uses techniques of merging predictor-corrector / implicit-explicit in nonlinear finite element analysis. It is confirmed that OS method can save much time and get well analysis accuracy in dynamic analysis. The associated theories can be detailed in [2] and [3]. This method of PISA3D is implemented by Yi-Jer Yu ([r93521237@ntu.edu.tw](mailto:r93521237@ntu.edu.tw)).

### Command:

Analysis Dynamic OS XGndMotion? XFac? YGndMot? YFac? ZGndMot?  
ZFac? StpLength? AnaSteps?  $\alpha$ ?  $\beta$ ?  $\beta_0$ ?

PARAMETER	DESCRIPTION
Analysis	Control Word.
Dynamic	Perform dynamic analysis.
OS	With <i>Operator-Splitting</i> integration algorithm.
XGndMotion	Tag of X-direction(horizontal) ground motion load pattern.
XFac	Magnitude factor of X-direction(horizontal) ground motion.
YGndMot	Tag of Y-direction(vertical) ground motion load pattern.
YFac	Magnitude factor of Y-direction(vertical) ground motion.
ZGndMot	Tag of Z-direction(horizontal) ground motion load pattern.
ZFac	Magnitude factor of Z-direction(horizontal) ground motion.
StpLength	Time length of each integration step, must > 0.
AnaSteps	Total integration steps, must > 0.
$\alpha$	Mass proportional damping factor.
$\beta$	Tangent stiffness proportional damping factor.
$\beta_0$	Initial stiffness proportional damping factor. (Default Value = 0)

**NOTE:**

1. Currently, the OS integration algorithm can **not** be applied to “**Nonlinear Damper Element**”. ( $F_D = CV^n$ ,  $\eta$  is not equal to 1. See Part G05) Therefore, the analyzed model must not be composed of “**Nonlinear Damper Element**” or the analysis result will be incorrect.
2. The OS method provides higher efficiency than the Newmark method does, especially in the circumstance that the nonlinear behavior of the analysis model is apparent. Using OS method can save analysis time.
3. One can also check the correctness and precision of dynamic analysis by the **Unbalanced Energy Ratio** in the file XXX.Energy. Lower Unbalanced Energy Ratio during analysis reflects higher analysis correctness and precision. Shorten StpLength can usually raise the precision, but it will also cost more analysis duration.
4. If only one or two direction(s) have ground excitation, the ground motion load pattern of rest direction(s) should input as “None”.
5. The linear structural damping can be modeled as  $[C] = \alpha [M] + \beta [K_t]$ , or  $[C] = \alpha [M] + \beta_0 [K_0]$ . Where  $[K_t]$  is the structural tangent stiffness and  $[K_0]$  is the structural initial stiffness. The user should choose one of  $\beta$  and  $\beta_0$  to input, or the damping effects of the structure will be taken into account improperly as  $\beta$  and  $\beta_0$  are input simultaneously.

**EXAMPLE:**

Analysis Dynamic OS EQEW 0.33 none 0 EQSN 0.33 0.04 1000 1.00221388E-001  
3.86725431E-003 0

//This command defines a two-way (X-dir. and Z-dir.) earthquake analysis with OS integration algorithm.

## PART C. Analysis Control Setting Command

These types of commands control some settings during the analysis procedure.

### C01. Geometric Nonlinear Effect

This command sets the geometric nonlinear consideration during the analysis procedure. See Reference [1].

#### Command:

ControlData GeometricNL Code?

PARAMETER	DESCRIPTION
ControlData	Control Word.
GeometricNL	Setting geometric nonlinear effect by geometric stiffness matrix.
Code	0 = Do not consider geometric stiffness matrix (Default Value). 1 = Consider geometric stiffness matrix only after every analysis method. 2 = Update geometric stiffness matrix before each analysis step in analysis methods

#### EXAMPLE:

ControlData GeometricNL 1

## PART D. Node Data Command

The following commands are about the nodal data descriptions. Note that all the nodes must have their unique tags.

### D01. Node Generation

This command is to define the nodal coordinates.

#### Command:

Node Tag? Coord-X? Coord-Y? Coord-Z?

PARAMETER	DESCRIPTION
Node	Control Word.
Tag	Unique tag of this node.
Coord-X	Coordinate of X direction.
Coord-Y	Coordinate of Y direction.
Coord-Z	Coordinate of Z direction. (Default Value = 0)

#### EXAMPLE:

Node n1 6928.2 4000.0 1000

## D02. Nodal Degree of Freedom Assign

This command defines the restraint of the nodal degree of freedom.

### Command:

DOF Node? UX? UY? UZ? RX? RY? RZ?

PARAMETER	DESCRIPTION
DOF	Control Word.
Node	Tag of the specified node.
UX	Fixity of X-dir. displacement. -1 = fixed. 0 = free. other node's tag = identical to the node.
UY	Fixity of Y-dir. displacement. -1 = fixed. 0 = free. other node's tag = identical to the node.
UZ	Fixity of Z-dir. displacement. -1 = fixed. 0 = free. other node's tag = identical to the node.
RX	Fixity of rotation about the global X-axis. -1 = fixed. 0 = free. other node's tag = identical to the node.
RY	Fixity of rotation about the global Y-axis. -1 = fixed. 0 = free. other node's tag = identical to the node.

RZ	Fixity of rotation about the global Z-axis. -1 = fixed. 0 = free. other node's tag = identical to the node.
----	--

**NOTE:**

1. If there are no DOFs assigned for one node, the default DOFs of this node are set to be all free.
2. If the technique of the same DOFs is used, the identical fixity of the identical node should be assigned to be free in front.

**EXAMPLE:**

DOF    n1    0    0    -1    -1    -1    -1

//This command set dof. UZ, RX, RY, RZ of node n1 to be fixed.

DOF    74    0    0    15    0    -1    0

//This command specifies that dof. UZ of node 74 is equal to dof. UZ of node 15, and dof. RY is fixed.



### D03. Nodal Lump Mass Assign

User can specify the lump mass of a node by this command.

#### Command:

Mass Node? UX? UY? UZ? RX? RY? RZ?

PARAMETER	DESCRIPTION
Mass	Control Word.
Node	Tag of the specified node.
UX	Translational mass associated with X-dir. Displacement, must $\geq 0$ . (Default Value = 0)
UY	Translational mass associated with Y-dir. Displacement, must $\geq 0$ . (Default Value = 0)
UZ	Translational mass associated with Z-dir. Displacement, must $\geq 0$ . (Default Value = 0)
RX	Rotational moment of inertia about the global X-axis, must $\geq 0$ . (Default Value = 0)
RY	Rotational moment of inertia about the global Y-axis, must $\geq 0$ . (Default Value = 0)
RZ	Rotational moment of inertia about the global Z-axis, must $\geq 0$ . (Default Value = 0)

#### NOTE:

1. If there are no lump mass assigned for one node, the default lump mass of this node is set to be all zero.

#### EXAMPLE:

Mass 2 10 10 10 0 0 0

## D04. Nodal Linear Spring

This command allows the user to define the fully-coupled 6-by-6 stiffness spring of any node.

### Command:

Spring NodeTag? Kux? Kuy? Kuz? Krx? Kry? Krz? ... <KMN? ValueKMN?>

PARAMETER	DESCRIPTION
Spring	Control Word.
NodeTag	Tag of the specified node.
Kux	Spring stiffness in global X axis. (Default = 0)
Kuy	Spring stiffness in global Y axis. (Default = 0)
Kuz	Spring stiffness in global Z axis. (Default = 0)
Krx	Spring rotational stiffness about global X axis. (Default = 0)
Kry	Spring rotational stiffness about global Y axis. (Default = 0)
Krz	Spring rotational stiffness about global Z axis. (Default = 0)
KMN	Identify the DOF for off-diagonal spring.  M can be ux, uy, uz, rx, ry, rz.  N can be ux, uy, uz, rx, ry, rz.  Ex: Kuzry means the next spring stiffness value is for UZRY.
ValueKMN	Spring stiffness value of above-mentioned DOF KMN.

### NOTE:

1. <KMN? ValueKMN?> is optional, at most the 15 off-diagonal spring stiffness can be all defined, and the default values are all 0.

### EXAMPLE:

```
Spring N5 0 0 1E5 0 1E3 1000 Kuzrz 300
```

//This command specifies node N5 with spring value:  $K_{UZ}=100000$ ,  $K_{RY}=K_{RZ}=1000$ ,  $K_{UZRZ}=300$  and others=0

## D05. Constraint Definition (Rigid Diaphragm)

This command is to specify the rigid floors of a 3D-building. See Fig. D01. Each floor should be defined as a unique rigid diaphragm, and the center of floor's mass is the master node.

### Command:

Constraint Diaphragm Tag? MasterNd? SlaveNd1? ... SlaveNdN? ...<S NF? NL?>

PARAMETER	DESCRIPTION
Constraint	Control Word.
Diaphragm	Rigid diaphragm for building system.
Tag	Unique tag of this diaphragm.
MasterNd	Tag of the master node.
SlaveNd1	Tag of the slaved node 1.
SlaveNdN	Tag of the slaved node N.
S	To indicate sequential generation of slaved nodes.
NF	Numeral tag of the first slaved node.
NL	Numeral tag of the last slaved node.

### NOTE:

1. The DOF UX/UZ/RX of the master node and the slaved nodes must be free, and the master node can not itself be slaved.  
(i.e. More than one diaphragm can exist at the same level, but only single slaving is permitted for each slaved node.)
2. <S NF? NL?> is optional for the identification of slaved nodes.  
If "S" is input in this command, the 1st and 2nd following nodal tags must be numerals, and the slaved nodes would be generated node by node automatically.

### EXAMPLE:

```
Constraint Diaphragm 1F n7 n1 n2 n3 n4 n5 n6 n8 n9
```

//This command defines a diaphragm 1F. n7 is the master node, and n1~n6, n8 and n9 are slaved.

```
Constraint Diaphragm 2F 15 1 3 4 5 6 7 8 N10 13 17 18 19 20 25 26
```

```
Constraint Diaphragm 2F 15 1 S 3 8 N10 13 S 17 20 25 26
```

//The two commands are of the same meaning. They define master node 15, and slaved nodes 1, 3~8, N10, 13, 17~20, 25, 26.

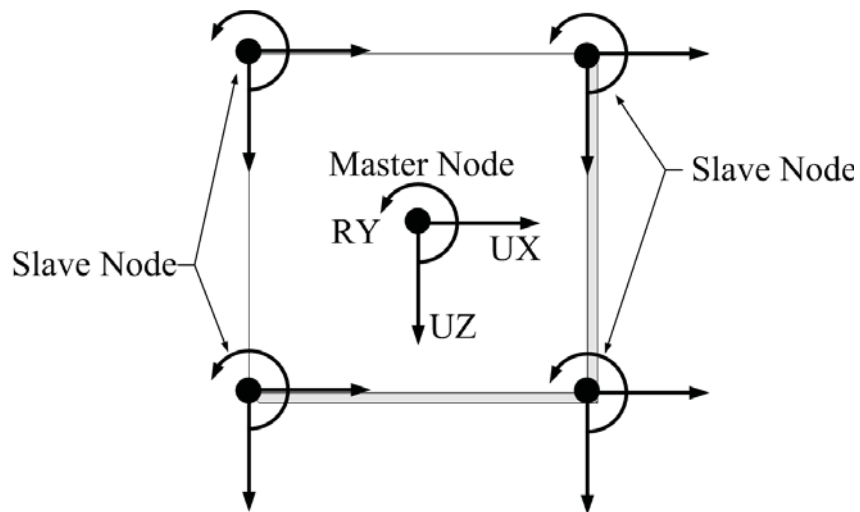


Fig. D01 3D Rigid Diaphragm

## PART E. Load Pattern Command

### E01. Nodal Load

This command defines static nodal loads. Various nodal load commands can have the same tag. These commands would be combined as a single load pattern automatically. If commands have different tags, they would be taken as different patterns.

#### Command:

```
LoadPattern NodalLoad Tag? Node? FX? FY? FZ? MX? MY? MZ?
```

PARAMETER	DESCRIPTION
LoadPattern	Control Word.
NodalLoad	Nodal external load.
Tag	Tag of this load pattern.
Node	Tag of the target node.
FX	External force on the X direction.
FY	External force on the Y direction.
FZ	External force on the Z direction.
MX	External moment about the global X axis. (Counterclockwise as positive by normal view to the X axis.)
MY	External moment about the global Y axis. (Counterclockwise as positive by normal view to the Y axis.)
MZ	External moment about the global Z axis. (Counterclockwise as positive by normal view to the Z axis.)

#### EXAMPLE:

```
LoadPattern NodalLoad DL n1 383 321.4 0 0 0 0
```

```
LoadPattern NodalLoad DL n2 -15 0 130 0 200 0
```

```
//Using two commands to define a load pattern named DL.
```

## E02. Ground Acceleration Record

In this command, the corresponding acceleration record file should be prepared. The dynamic analysis would use the acceleration record as the ground excitation. Different records should have different tags, or the previous one will be replaced.

### Command:

```
LoadPattern GroundAccel Tag? FileName? TimeFactor? MagnFactor?
```

PARAMETER	DESCRIPTION
LoadPattern	Control Word.
GroundAccel	Ground motion as time-history of acceleration.
Tag	Tag of this load pattern.
FileName	File name of the time-history record file, which is in relative path to the input file.
TimeFactor	Time scale factor, must $> 0$ . (Default Value = 1.0)
MagnFactor	Magnitude scale factor. (Default Value = 1.0)

### NOTE:

1. The acceleration record file must be in two columns text format. The first column is time and the second column is the corresponding acceleration value.

### EXAMPLE:

```
LoadPattern GroundAccel EQX CHY076EW.txt 1 9.81
```

//This command defines an acceleration time history named EQX, and the record file is put in the same folder as the input file.

## PART F. Material Definition Command

All the different material objects should have their own tags, even if the types of materials are different. For many elements use an identical material, only one material command for this material is needed. The program will generate the material copies for the elements. This concept is similar to many analysis programs, like SAP and ETABS.

### F01. Elastic Material

This material would always keep linear elastic. See Fig. F01 for the elastic material.

#### Command:

Material Elastic Tag? E? Nu?

PARAMETER	DESCRIPTION
Material	Control Word.
Elastic	Elastic material without yielding state.
Tag	Unique tag of this material.
E	Young's modulus.
Nu	Poisson Ratio, must between 0 and 0.5. (Default Value = 0)

#### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Linear	This material always keeps linear elastic state.

**EXAMPLE:**

Material Elastic mat01 200 0.3

//This Command defines a material named “mat01”, with E=200 and Nu=0.3.

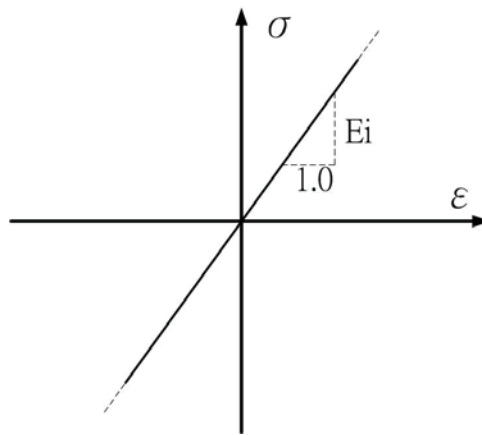


Fig. F01 Elastic Material



## F02. Bilinear Material

This command defines a bilinear material. This material has linear post-yielding tangent to model the strain hardening behavior. See Fig. F02 for the bilinear material.

### Command:

Material Bilinear Tag? Ei? SHR? fyt? fyc? Nu?

PARAMETER	DESCRIPTION
Material	Control Word.
Bilinear	Bilinear yielding rule.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
SHR	Strain hardening ratio, as a proportion of Ei.
fy <sub>t</sub>	Yielding stress in tension.
fy <sub>c</sub>	Yielding stress in compression. (Default Value = -fy <sub>t</sub> )
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)

### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic	The stress has not reached the yielding stress yet.
1	Yielding	The stress reaches the yielding stress.

### EXAMPLE:

```
Material bilinear mat1 200 0.05 0.24 -.22
```

//This command defines a bilinear material with Ei=200, SHR=0.05, fy<sub>t</sub>=0.24, fy<sub>c</sub>=-0.22, and Nu=0.0

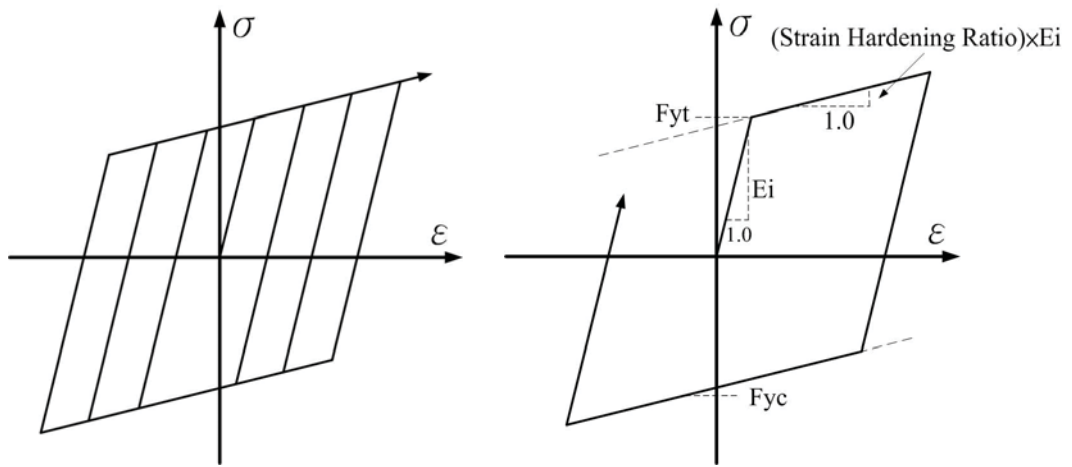


Fig. F02 Bilinear Material

### F03. Hardening Material

This material adopts two-surface plastic hardening rule. See Fig. F03 and Fig. F04 for the hardening material. This material is usually used to model the metal material precisely. Take reference to [1].

#### Command:

Material Hardening Tag? Ei? fyt? fyc? Hiso1+? Hiso2+? Hiso1-? Hiso2-?  
Hkin1? Hkin2? B/Y? Nu? subStep?

PARAMETER	DESCRIPTION
Material	Control Word.
Hardening	Two surface hardening rule.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
fy <sub>t</sub>	Yielding stress in tension.
fy <sub>c</sub>	Yielding stress in compression.
Hiso1+	Positive isotropic hardening parameter C1.
Hiso2+	Positive isotropic hardening parameter C2.
Hiso1-	Negative isotropic hardening parameter C1. (Default Value = Hiso1+)
Hiso2-	Negative isotropic hardening parameter C2. (Default Value = Hiso2+)
Hkin1	Kinematic hardening parameter C3. (Default Value = 1.0)
Hkin2	Kinematic hardening parameter C4. (Default Value = 10.0)
B/Y	Initial ratio of the two surfaces (Boundary/Yielding, i.e. BS/YS). (Default Value = 1.0)
Nu	Poisson Ratio, must between 0 and 0.5. (Default Value = 0)

subStep	Steps of strain incremental segment in yielding state. (Default Value = 10)
---------	--

### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic	The stress has not reached the yielding stress yet.
1	Kinematic Hardening	The stress reaches the yielding surface but below the boundary surface, and keeps kinematic hardening.
2	Isotropic Hardening	The stress reaches the boundary surface, and keeps both isotropic hardening and kinematic hardening.

### NOTE:

1. More “subStep” will waste more analysis time, but it will also improve the analysis correctness and reduce the possibility of converge failure.

### EXAMPLE:

Material Hardening 2 200 0.24 -0.24 0.0045 2.8 .009 2.8 1.0 24 1.3 0.3 100

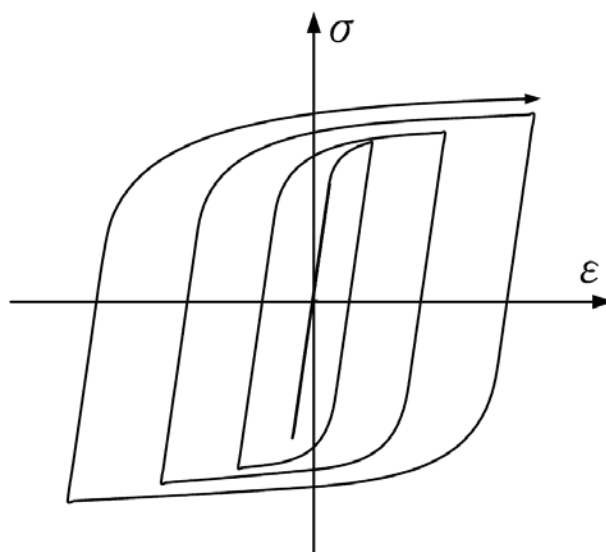
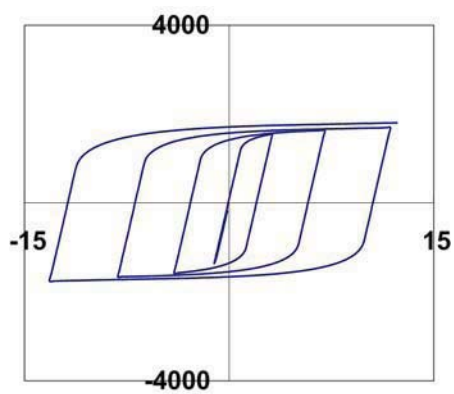
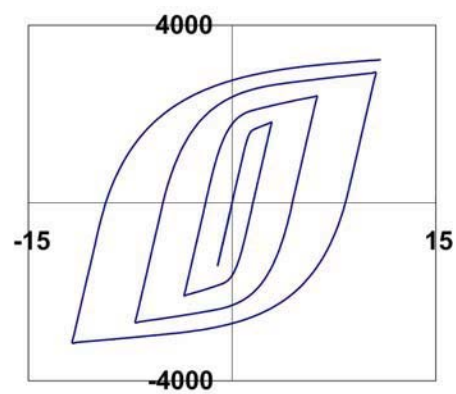


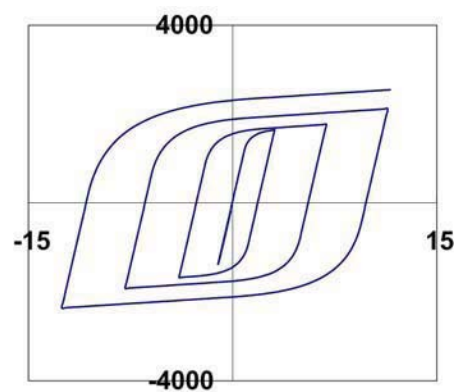
Fig. F03 Hardening Material



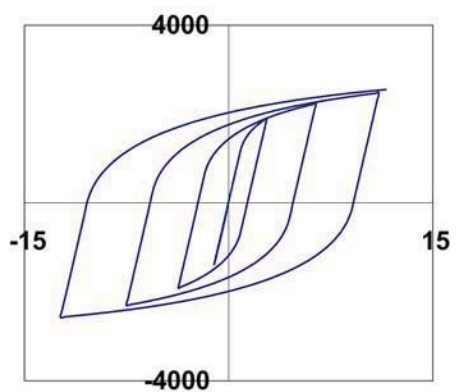
(a)



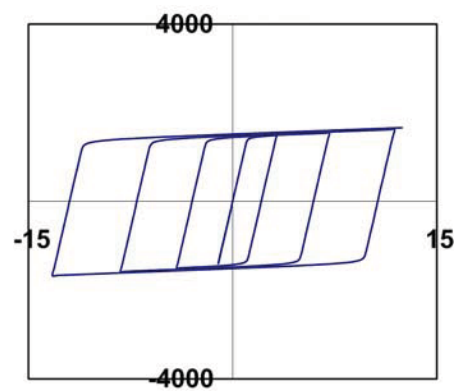
(b)



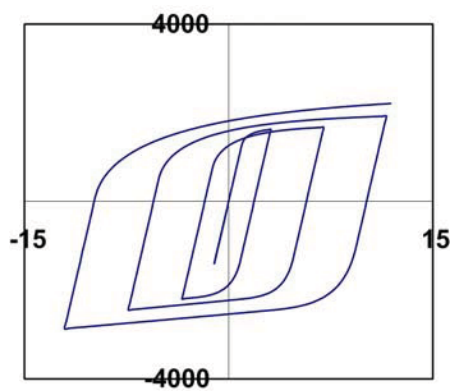
(c)



(d)



(e)



(f)

Fig. F04 Different Parameters of Hardening Material

Case	Ei	fyp	fyc	Hiso1+	Hiso2+	Hiso1-	Hiso2-	Hkin1	Hkin2	BS/YS
(a)	1320	1250	-1250	0.005	5.0	0.005	5.0	1.0	24	1.3
(b)	1320	1250	-1250	0.005	1.7	0.005	1.7	1.0	24	1.3
(c)	1320	1250	-1250	0.015	5.0	0.015	5.0	1.0	24	1.3
(d)	1320	1250	-1250	0.005	5.0	0.005	5.0	1.0	24	2.3
(e)	1320	1250	-1250	0.005	5.0	0.005	5.0	5.0	24	1.3
(f)	1320	1250	-1660	0.005	5.0	0.020	5.0	1.0	24	1.3

## F04. Degrading Material

This material is to model the reinforced concrete component. See Fig. F05 ~ Fig. F09 and Reference [1] for the degrading material. This material adopts 3 parameters (FStiff, FStren and FPinch, or call  $S_1$ ,  $S_2$  and  $S_3$ ) to model the degrading behavior. One can also specify the second set of 3 parameters to model the behavior precisely while the component's ductility ran out.

### Command:

Material Degrading Tag? Ei? fyt? fyc? SHR+? SHR-? FStiff1? FStren1? FPinch1?  
BV? FStiff2? FStren2? FPinch2? Nu?

PARAMETER	DESCRIPTION
Material	Control Word.
Degrading	Three parameters degrading rule.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
fy <sub>t</sub>	Yielding stress in tension.
fy <sub>c</sub>	Yielding stress in compression.
SHR+	Positive strain hardening ratio, as a proportion of Ei.
SHR-	Negative strain hardening ratio, as a proportion of -Ei.
FStiff1	$S_1$ , Stiffness degrading factor of part 1. $S_1$ must be between 0.0 and infinity, generally 1~20.
FStren1	$S_2$ , Strength deterioration factor of part 1, $S_2$ must be between 0.0 and 1.0, generally 0.7~1.0.
FPinch1	$S_3$ , Pinching factor of part 1, $S_3$ must be between 0.0 and 1.0, generally 0.4~1.0.
BV	Strain boundary value of part 1, as a proportion of initial yielding strain. (Default Value = 0) The second set of 3 parameters works up when the strain reaches BV*(initial yielding strain).

FStiff2	$S_1'$ , Stiffness degrading factor of part 2. (Default Value = FStiff1)
FStren2	$S_2'$ , Strength deterioration factor of part 2. (Default Value = FStren1)
FPinch2	$S_3'$ , Pinching factor of part 2. (Default Value = FPinch1)
Nu	Poisson Ratio, must between 0 and 0.5. (Default Value = 0)

### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic Loading	The stress has not reached the yielding stress yet.
1	Unloading State 1	Material unloads before reaching the yielding stress and will keep initial stiffness.
2	Pinching State 1	Material loads but has not reached the pinching point.
3	Pinching State 2	Material loads after the pinching point.
4	Yielding State	The stress reaches the yielding stress.
5	Unloading State 2	Material unloads after reaching the yielding stress and stiffness will degrade as FStiff ( $S_1$ ) sets.

### EXAMPLE:

Material Degrading 1 100 0.2 -0.2 0.05 -0.05 10 0.98 1.0 2.5  
1.0 0.5 0.4 0.2

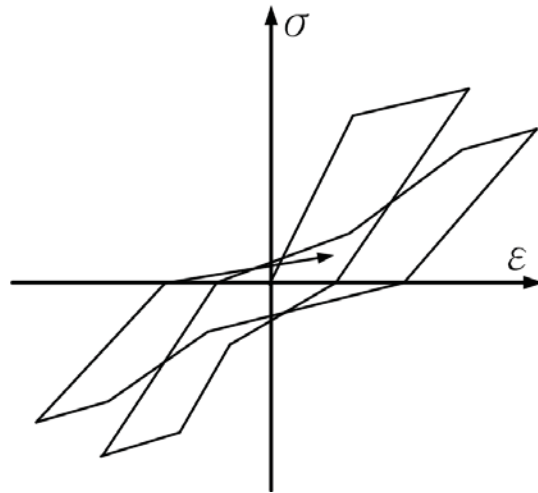


Fig. F05a Degrading Material

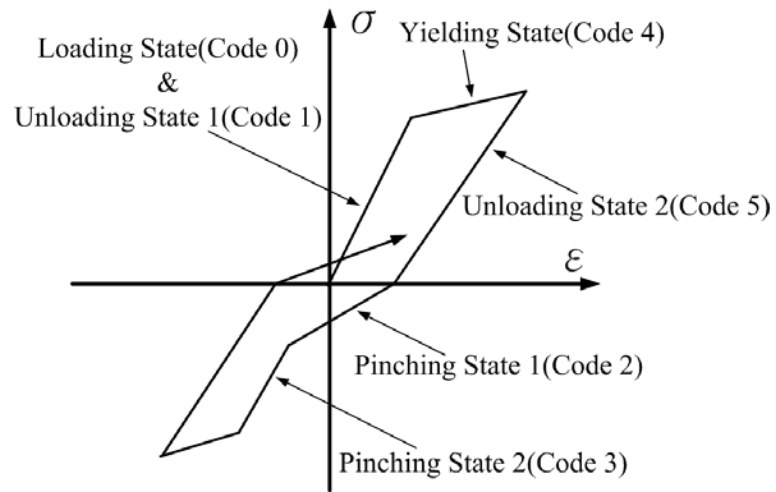


Fig. F05b States of Degrading Material

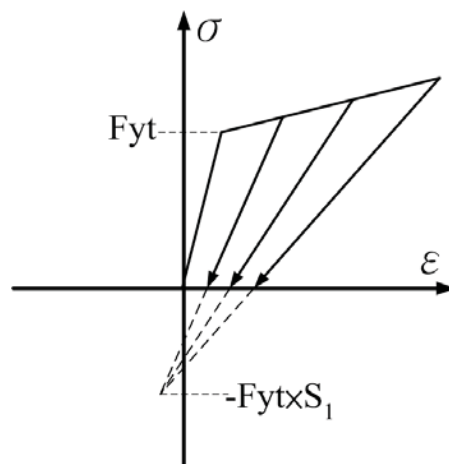


Fig. F06 Stiffness Degrading Factor



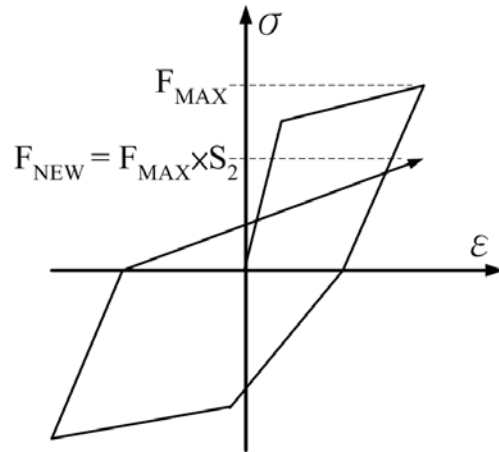


Fig. F07 Strength Deterioration Factor

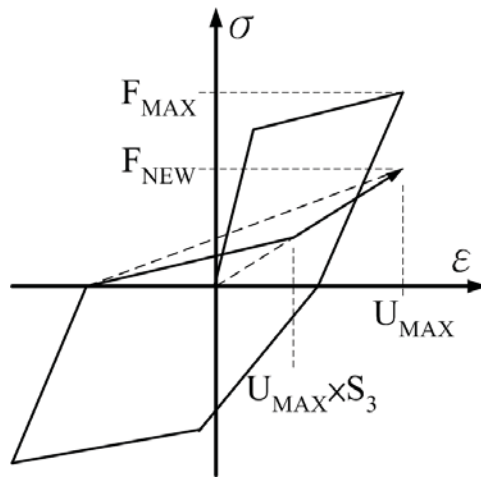


Fig. F08 Pinching Factor

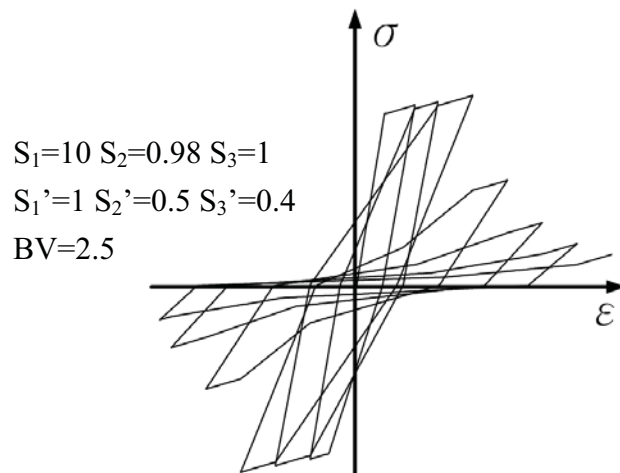


Fig. F09 Using Two Sets of 3 Parameters

## F05. Bilinear-Elastic Material

This material performs nonlinear stress-strain path, but always keeps in elastic. The loading and unloading path would be the same. See Fig. F12 and Fig. F13 for the bilinear-elastic material.

### Command:

```
Material BilinearElastic Tag? Ei+? SHR+? fyt? Nu? Ei-? SHR-? fyc?
```

PARAMETER	DESCRIPTION
Material	Control Word.
BilinearElastic	Elastic material with bilinear strain hardening state.
Tag	Unique tag of this material.
Ei+	Tensile initial Young's modulus.
SHR+	Tensile strain hardening ratio, as a proportion of Ei+.
fy <sub>t</sub>	Yielding stress in tension.
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)
Ei-	Compressive initial Young's modulus. (Default Value = Ei+)
SHR-	Compressive strain hardening ratio, as a proportion of Ei-. (Default Value = SHR+)
fy <sub>c</sub>	Yielding stress in compression. (Default Value = -fy <sub>t</sub> )

### EXAMPLE:

```
Material Bilinearelastic mat1 206 0.03 0.2 0.3 206 0.05 -0.18
```

//This command defines a bilinear-elastic material with Ei+=206, SHR+=0.03, fyt=0.2, Ei-=206, SHR-=0.05, fyc=-0.18, and Nu=0.3

## States of the Material:

One can check the material state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Initial Loading	Initial loading in tension, and the stress is under strain-hardening stress.
1	Initial Loading	Initial loading in compression, and the stress is under strain-hardening stress.
2	Strain Hardening	Strain hardening in tension.
3	Strain Hardening	Strain hardening in compression.

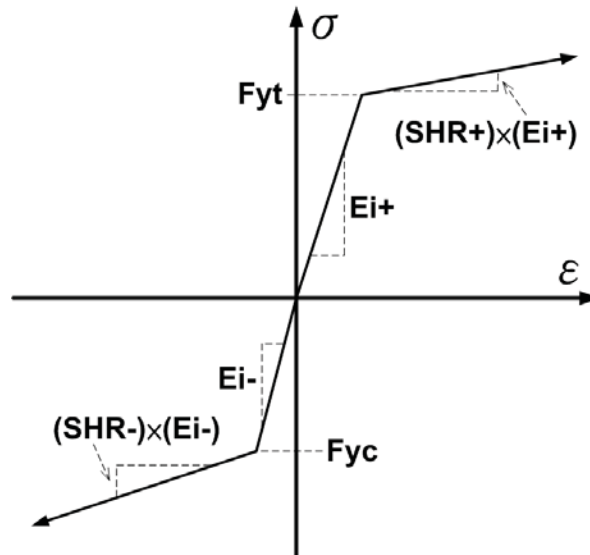


Fig. F12 Bilinear-Elastic Material

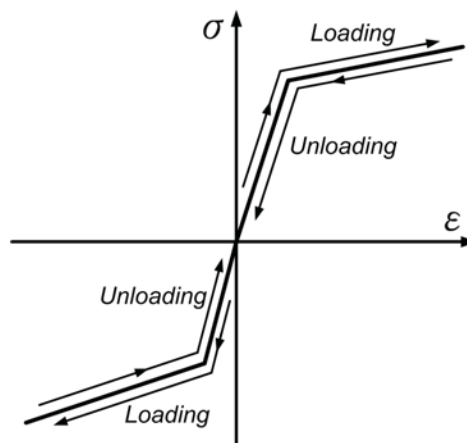


Fig. F13 Stress-Strain Equivalent Path

## F06. Bilinear02 Material

Bilinear02 material is very similar to the Bilinear material, but this material supports the behavior of elastic buckling in compression. See Fig. F14 and Fig. F15 for details.

### Command:

Material Bilinear02 Tag? Ei? SHR? fyt? fyc? Nu? Buckle?

PARAMETER	DESCRIPTION
Material	Control Word.
Bilinear02	Bilinear02 yielding rule.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
SHR	Strain hardening ratio, as a proportion of Ei.
fy <sub>t</sub>	Yielding stress in tension.
fy <sub>c</sub>	Yielding stress in compression. (Default Value = -fy <sub>t</sub> )
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)
Buckle	Code for compression behavior. 0 = Yield in compression (no buckling). 1 = Keep nonlinear elastic in compression (with buckling). (Default Value = 0)

### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic	The stress has not reached the yielding stress yet.
1	Yielding	The stress reaches the yielding stress.

### EXAMPLE:

Material bilinear02 1 200 0.03 0.24 -.22 0.3 1

//This command defines a bilinear02 material with  $E_i=200$ ,  $SHR=0.03$ ,  $f_{yt}=0.24$ ,  $f_{yc}=-0.22$ ,  $Nu=0.3$ , and buckles elastically in compression.

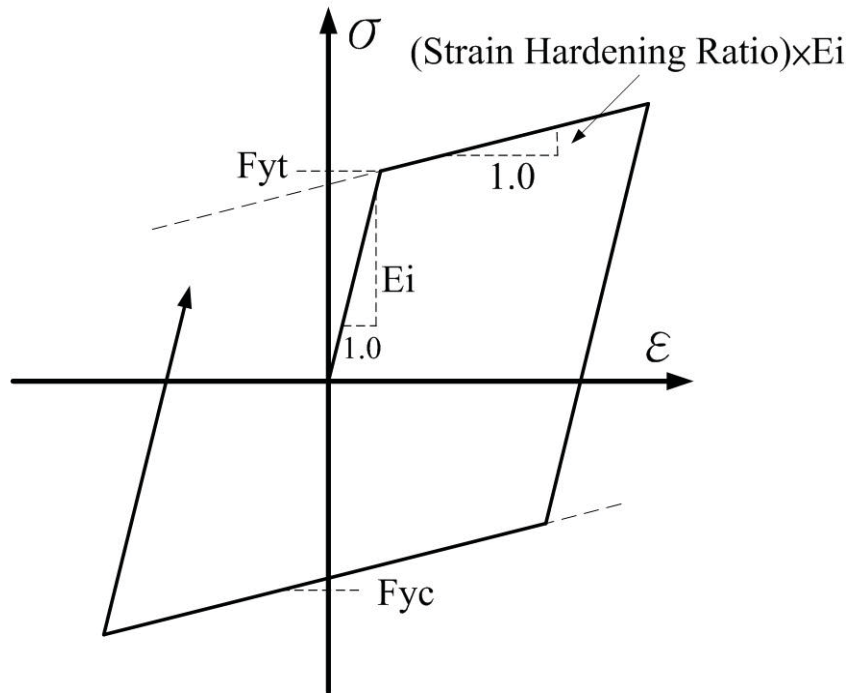


Fig. F14 Parameters of Bilinear02 Material (Buckle = 0)

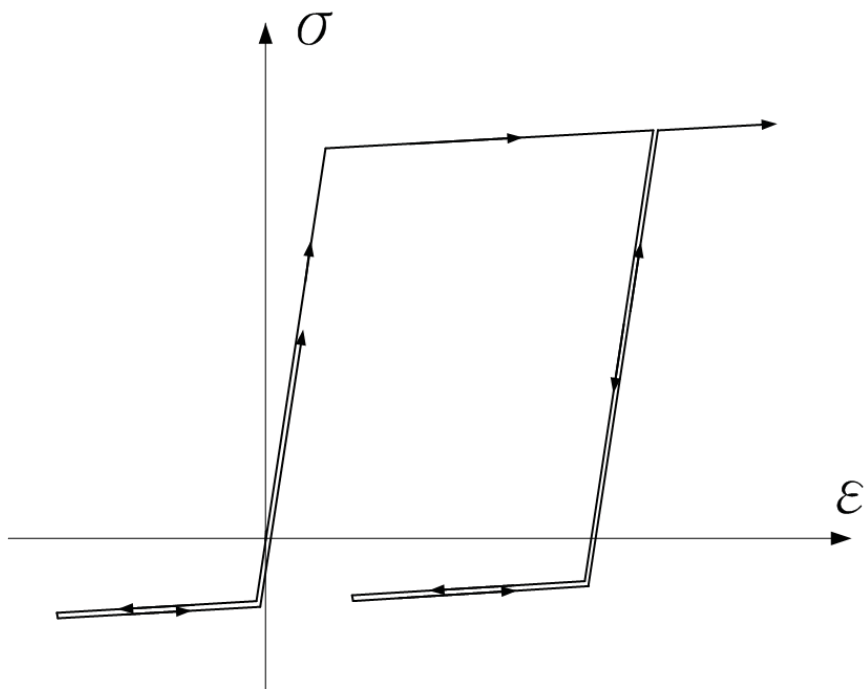


Fig. F15 Stress-Strain Path of Bilinear02 Material (Buckle = 1)

## F07. Fracture Material

This material can be used to model structures having limited ductility. User can specify the loading path by setting the stress and strain coordinates B/C/D/E. After point E, the resisting stress is always equal to zero. The unloading factor of stiffness degrading effect is supported after fracturing point C. The strength deterioration effect is also considered. This material can be used to compose elements in one-way pushover analysis.

### Command:

Material Fracture Tag? Ei? fy? Nu? Ufac? Bx? By? Cx? Cy?  
Dx? Dy? Ex? Ey? B'x? B'y? C'x? C'y? D'x? D'y? E'x? E'y?

PARAMETER	DESCRIPTION
Material	Control Word.
Fracture	Fracture material.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
fy	Yielding stress.
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)
Ufac	Unloading factor of stiffness degrading effect after point C. This parameter must be between 0 and 1. (Default Value = 1) “0” defines a completely degrading behavior to zero strain and stress. “1” specifies unloading tangent that is the same as the initial tangent.
Bx	Strain (X coordinate) of point B, as a scale factor of yielding strain $f_y/E_i$ . $B_x > 0$ .
By	Stress (Y coordinate) of point B, as a scale factor of yielding stress $f_y$ . $B_y > 0$ .

C <sub>x</sub>	Strain (X coordinate) of point C, as a scale factor of yielding strain $f_y/E_i$ . $C_x > 0$ .
C <sub>y</sub>	Stress (Y coordinate) of point C, as a scale factor of yielding stress $f_y$ . $C_y > 0$ .
D <sub>x</sub>	Strain (X coordinate) of point D, as a scale factor of yielding strain $f_y/E_i$ . $D_x > 0$ .
D <sub>y</sub>	Stress (Y coordinate) of point D, as a scale factor of yielding stress $f_y$ . $D_y > 0$ .
E <sub>x</sub>	Strain (X coordinate) of point E, as a scale factor of yielding strain $f_y/E_i$ . $E_x > 0$ .
E <sub>y</sub>	Stress (Y coordinate) of point E, as a scale factor of yielding stress $f_y$ . $E_y > 0$ .
B' <sub>x</sub>	Strain (X coordinate) of point B', as a scale factor of yielding strain $f_y/E_i$ . $B'_x < 0$ , default value = -B <sub>x</sub> .
B' <sub>y</sub>	Stress (Y coordinate) of point B', as a scale factor of yielding stress $f_y$ . $B'_y < 0$ , default value = -B <sub>y</sub> .
C' <sub>x</sub>	Strain (X coordinate) of point C', as a scale factor of yielding strain $f_y/E_i$ . $C'_x < 0$ , default value = -C <sub>x</sub> .
C' <sub>y</sub>	Stress (Y coordinate) of point C', as a scale factor of yielding stress $f_y$ . $C'_y < 0$ , default value = -C <sub>y</sub> .
D' <sub>x</sub>	Strain (X coordinate) of point D', as a scale factor of yielding strain $f_y/E_i$ . $D'_x < 0$ , default value = -D <sub>x</sub> .
D' <sub>y</sub>	Stress (Y coordinate) of point D', as a scale factor of yielding stress $f_y$ . $D'_y < 0$ , default value = -D <sub>y</sub> .
E' <sub>x</sub>	Strain (X coordinate) of point E', as a scale factor of yielding strain $f_y/E_i$ . $E'_x < 0$ , default value = -E <sub>x</sub> .
E' <sub>y</sub>	Stress (Y coordinate) of point E', as a scale factor of yielding stress $f_y$ . $E'_y < 0$ , default value = -E <sub>y</sub> .

## States of the Material:

One can check the material states in the element output which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic loading/unloading	The stress has not reached the yielding stress yet.
1	Yielding	The stress reaches the yielding stress.
2	Fracturing state	The strain exceeds the first fracturing point C, but is under point D.
3	Fracturing state	The strain exceeds the second fracturing point D, but is under point E.
4	Fracturing state	The strain exceeds the third fracturing point F, but the stress has not dropped to zero.
5	Fracturing state	Completely fracture state, zero strength.
6	Unloading/reloading after fracturing	Unloading and reloading after point C, with unloading factor Ufac.
7	Loading state	Loading after unloading from State 6 to zero stress.

### NOTE:

1. The horizontal coordinate of point D must be no less than that of point C. And a too steep behavior between C and D sometimes would cause converging failure in BeamColumn. Modifying the coordinates slightly can improve this situation. For example, modify C(6,1.25)/D(6,0.2) to C(6,1.25)/D(6.1,0.2).

### EXAMPLE:

//See Fig. F16

```
Material Fracture mat1 200 0.24 0.3 1.0 1 1 6 1.25 6 0.2 8 0.2
```

//See Fig. F17

```
Material Fracture mat2 200 0.24 0.3 0.5 1 1 6 1.25 7 0.25 10 0.25
```

//See Fig. F18 and Fig. F19

```
Material Fracture mat3 200 0.24 0.3 .85 1 1 6 1.15 6.5 0.25 8 0.25
```

//See Fig. F20

```
Material Fracture 4 200 0.24 0.3 0.85 1 1 6 1.25 6 0.2 8 0.2 -1 -1 -1 -1 -1 -0.2 -4 -0.2
```

//See Fig. F21

```
Material Fracture 5 200 0.24 0.3 1.0 1 1 4 1.15 6 1.2 8 1.2 -1 -1 -4 -1.15 -6 -1.2 -8 -0.2
```



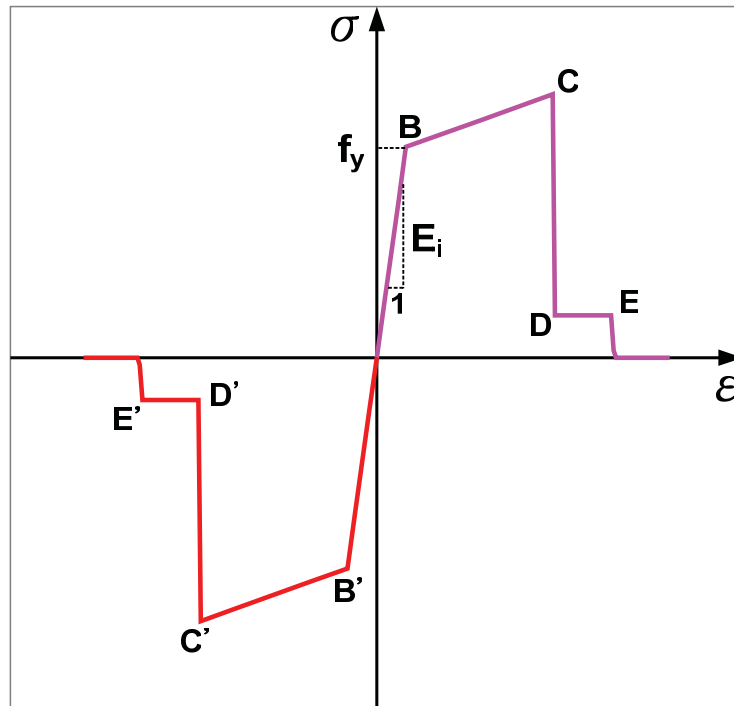


Fig. F16 Path setting of Fracture Material

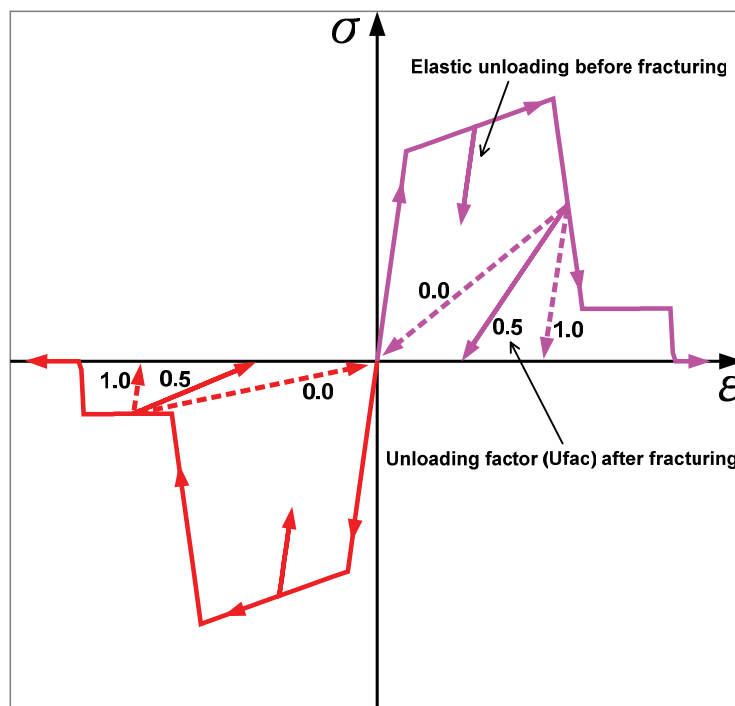


Fig. F17 Unloading factor of Fracture Material

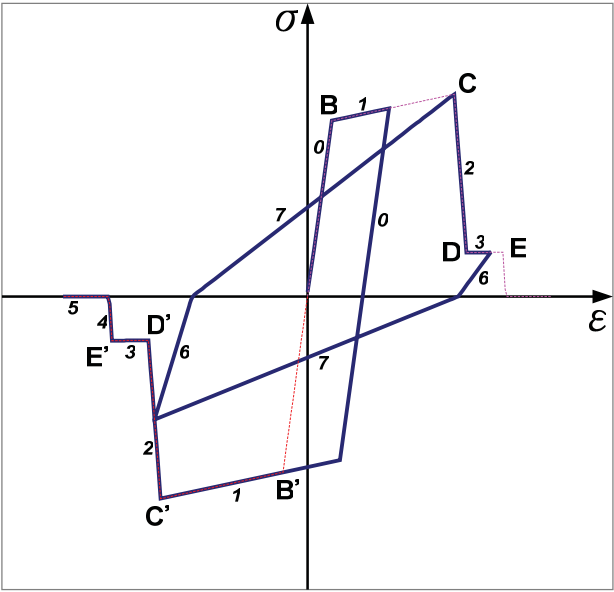


Fig. F18 State codes

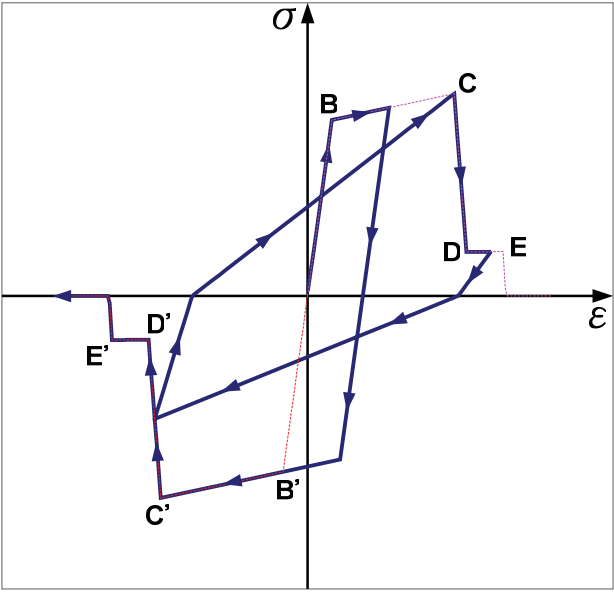


Fig. F19 Cyclic behavior

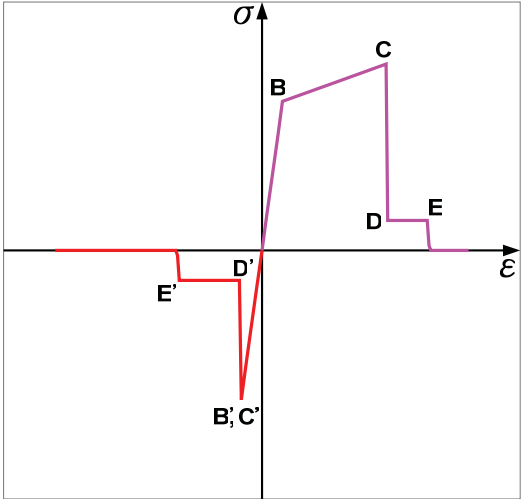


Fig. F20 Example 4

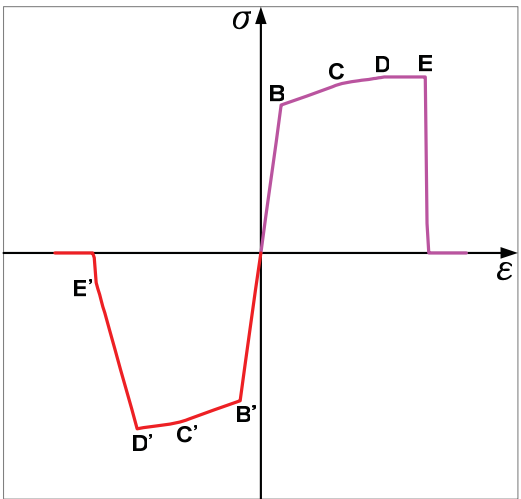


Fig. F21 Example 5

## **F08. Buckle Material**

This material is used to model the brace buckling behavior of brace frames and it's based on phenomenological brace models theory which predefine the shape of axial force – axial displacement response. The brace which this material is specified to is assumed to be pin-ended (see Fig. F22), this implies flexural stiffness of brace is negligible and rotation at the structural joints where the element is connected have no influence on the axial force – axial displacement hysteresis rule. References are given in [Maison and Popov, 1980].

The axial force – axial displacement algorithm simulates the buckling behavior with nine piecewise lines. Users could specify the values of control points and control slope to simulate the hysteretic loop (see Fig. F23, Fig. F24). Three additional rules regarding modifications to the hysteretic behavior due to cyclic loading are:

- (a) The strain control points are shifted by  $\delta y$  on unloading from state 2 or 9 (see Fig. F25). This rule produces a hysteretic loop translation.
- (b) A reduction in the compressive strength dependent on inelastic cycling. In the brace lengthening from state 4, the compressive strength for the next cycle will be the state 4 compressive stress at which the brace lengthening occurred. On entering state 5 the compressive strength will be reduced in future cycle to final compressive strength PCRf. Users could select PCRf to limit the amount of strength reduction (see Fig. F26).
- (c) An inelastic lengthening of the brace at tensile loads less than the uniaxial yield force after inelastic compressive cycling (column growth). The algorithm will translate the control strain U29 a ratio of the amount of inelastic shortening in the previous cycle (see Fig. F26). Users could select GF (growth factor) to simulate column growth phenomenon.

See Fig. F27 for Buckle Material parameters.

**Command:**

Material Buckle Tag? Einit? PCR? fy? SHR? U34? U45? P45? E5?  
 U6? P6? U7? P7? U8? P8? E8? PCRF? GF? Nu?

PARAMETER	DESCRIPTION
Material	Control Word.
Buckle	Buckle Material
Tag	Unique tag of this material.
Einit	Initial Young's modulus.
PCR	Buckling strength.
fy	Yielding stress in tension.
SHR	Strain hardening ratio, as a proportion of Einit.
U34	Strain at intersection of state 3 and 4, as a scale factor of yielding strain fy/Einit.
U45	Strain at intersection of state 4 and 5, as a scale factor of yielding strain fy/Einit.
P45	Stress at intersection of state 4 and 5, as a scale factor of yielding stress fy.
E5	Slope of state 5, as a scale factor of Einit.
U6	Strain of control point for state 6, as a scale factor of yielding strain fy/Einit.
P6	Stress of control point for state 6, as a scale factor of yielding stress fy.
U7	Strain of control point for state 7, as a scale factor of yielding strain fy/Einit.
P7	Stress of control point for state 7, as a scale factor of yielding stress fy..
U8	Strain of control point for state 8, as a scale factor of yielding strain fy/Einit.
P8	Stress of control point for state 8, as a scale factor of yielding

	stress $f_y$ .
E8	Slope of state 8, as a scale factor of $E_{init}$ .
PCRF	Final buckling strength, as a scale factor of yielding stress $f_y$ .
GF	Growth factor.
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)

### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
1	Elastic Loading/Unloading	The stress has not reached the yielding stress or buckling stress yet. State 1 may exit to state 2 or 3.
2	Hardening	The stress reaches the yielding stress. State 2 may exit to state 1.
3	Buckle	The compression stress reaches the buckling strength. State 3 may exit to state 4 or 6.
4	Loading in Compression	Continue loading in compression, and compressive resistance drops. State 4 may exit to state 5 or 6.
5	Loading in Compression	Continue loading in compression, and compressive resistance drops State 5 may exit to state 6.
6	Unloading after buckling	Unloading from state 3, 4, or 5 to zero stress. State 6 may exit to state 5,7,9,1,3,4.
7	Loading in Tension	Loading in tension. State 7 may exit to state 6,8,9.

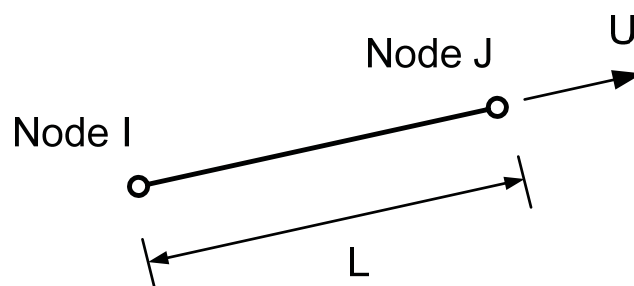
8	Loading in Tenson	Loading in tension. State 8 may exit to state 7 or 9.
9	Loading in Tenson	Loading in tension. State 9 may exit to state 1 or 2.

**EXAMPLE:**

Material	Buckle	B	29000	-47.778	60.7	0.01	-1.1	-3.79	-0.31
-0.013	0.69	1	0.69	0.7	-0.52	0.6	0.08	-0.55	0.3

**Reference:**

Bruce F. Maison and Egor P. Popov. (1980). "Cyclic Response Prediction for Braced Steel Frames. " Journal of the Structural Division, Vol. 106, No. 7, p.1401–p.1416.



Element possesses one local DOF  
U: the axial deformation

Fig. F22 Buckle Material Assumption

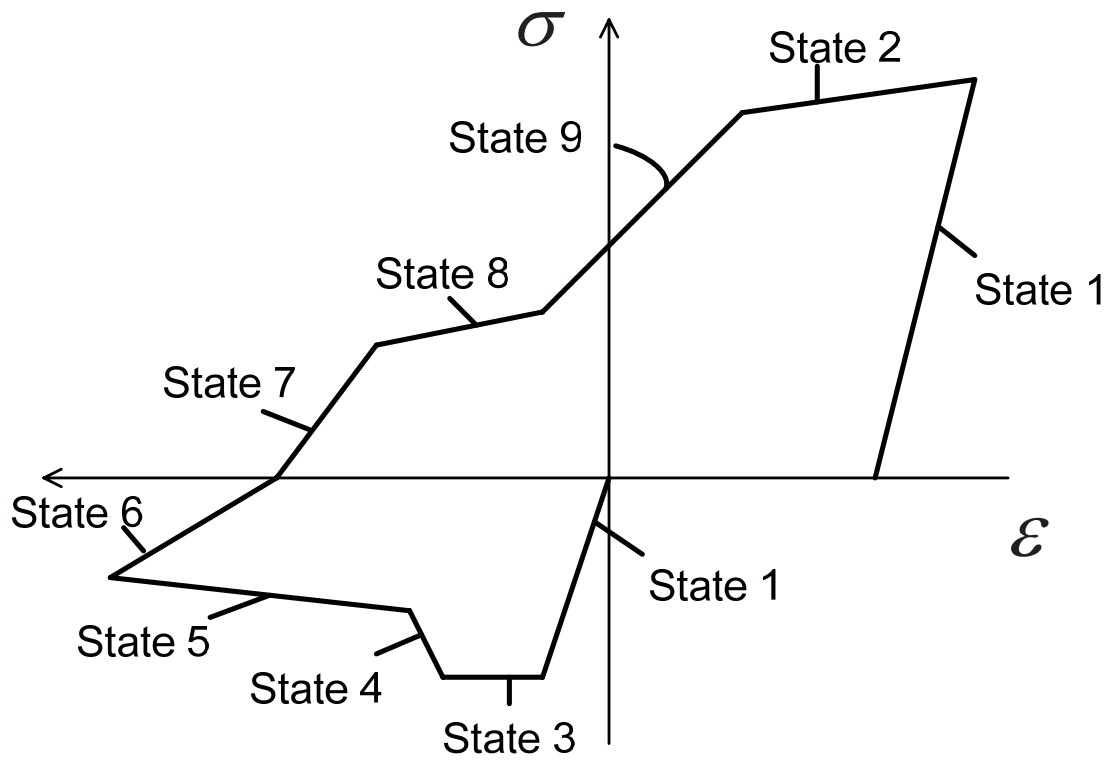


Fig. F23 States of Buckle Material

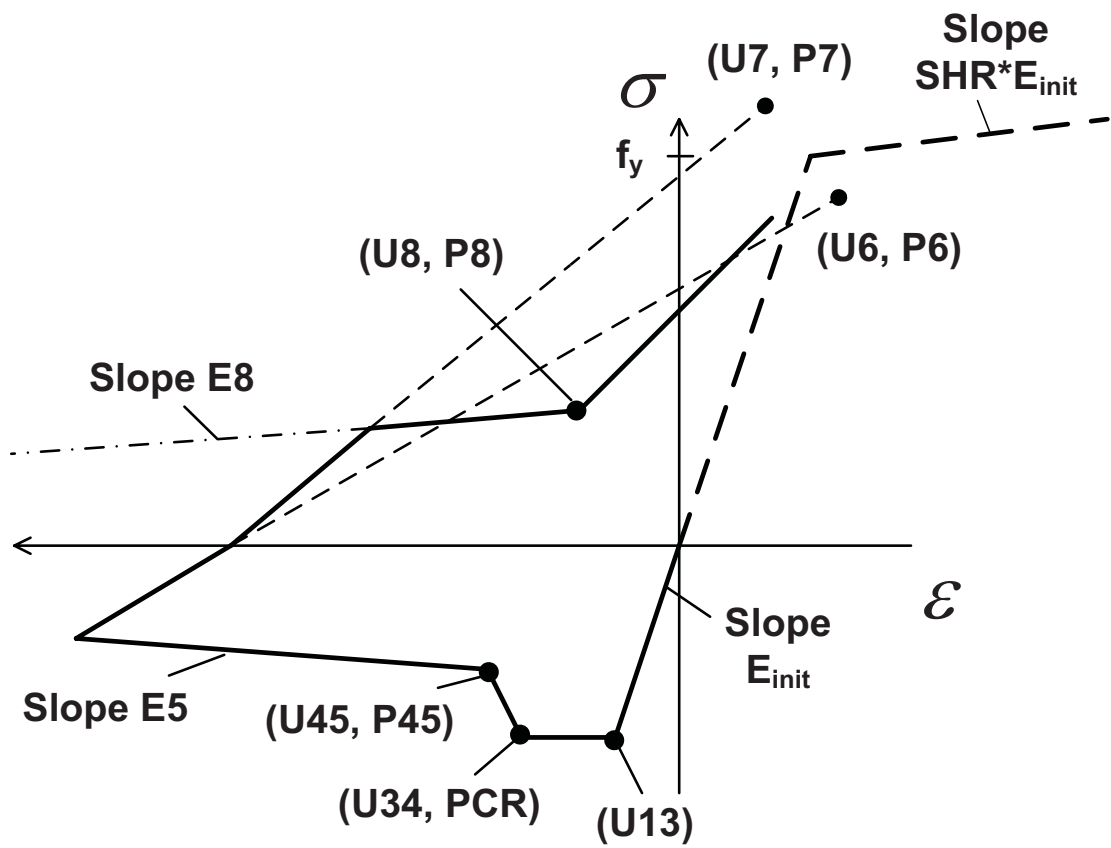


Fig. F24 Buckle Material

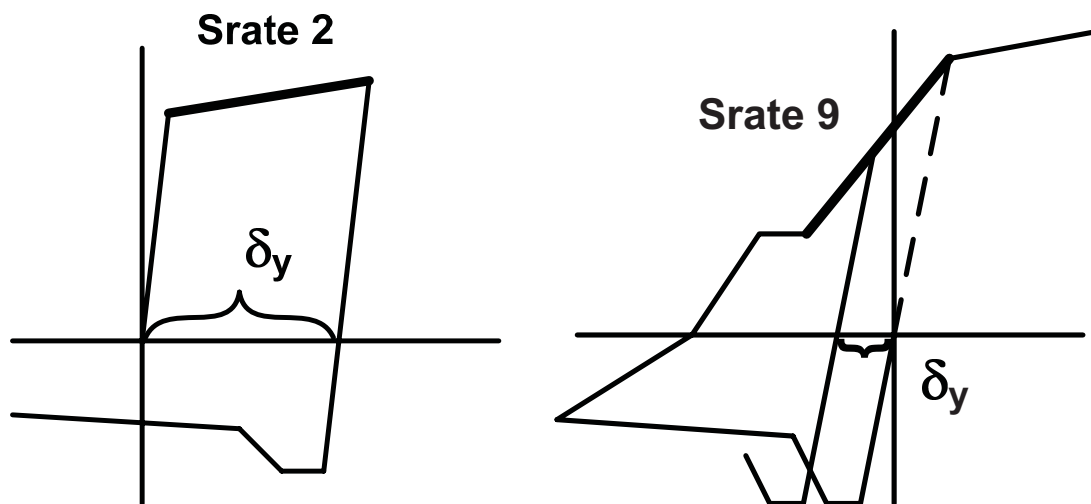


Fig. F25 Control points are shifted by  $\delta_y$

PCRF: Final Buckling  
Strength  
PCR\*: Buckling Strength  
PCR: Initial Buckling Strength

Note:  $PCR \leq PCRF \leq P45$

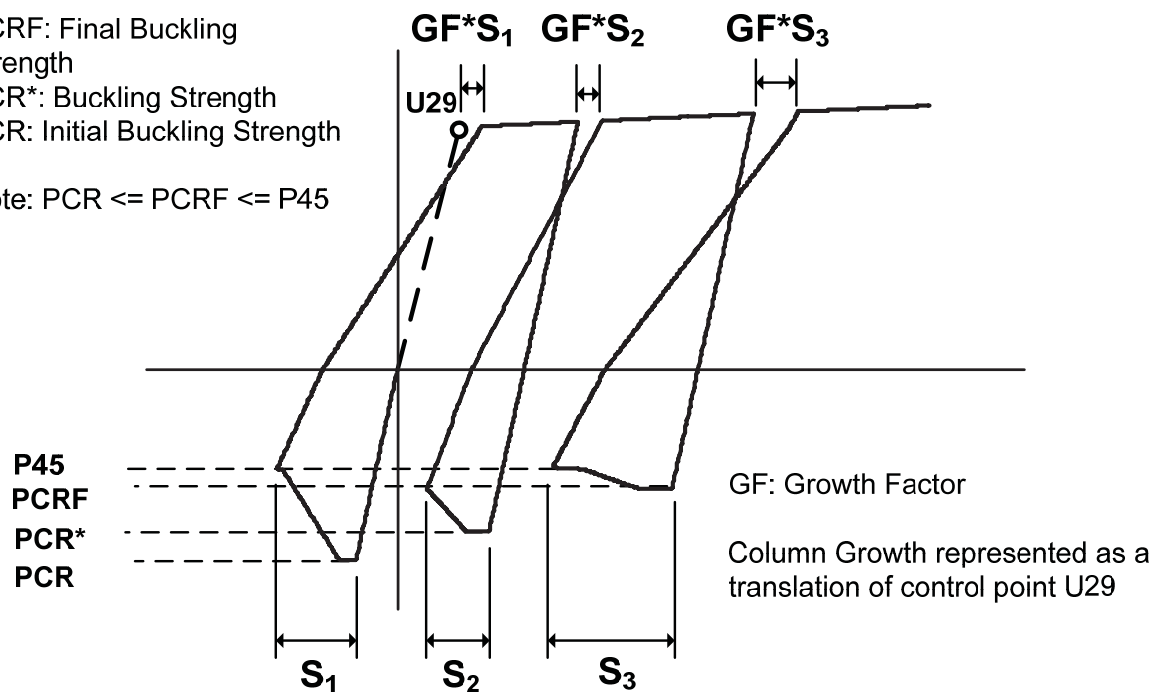


Fig. F26 Strength Reduction and Column Growth



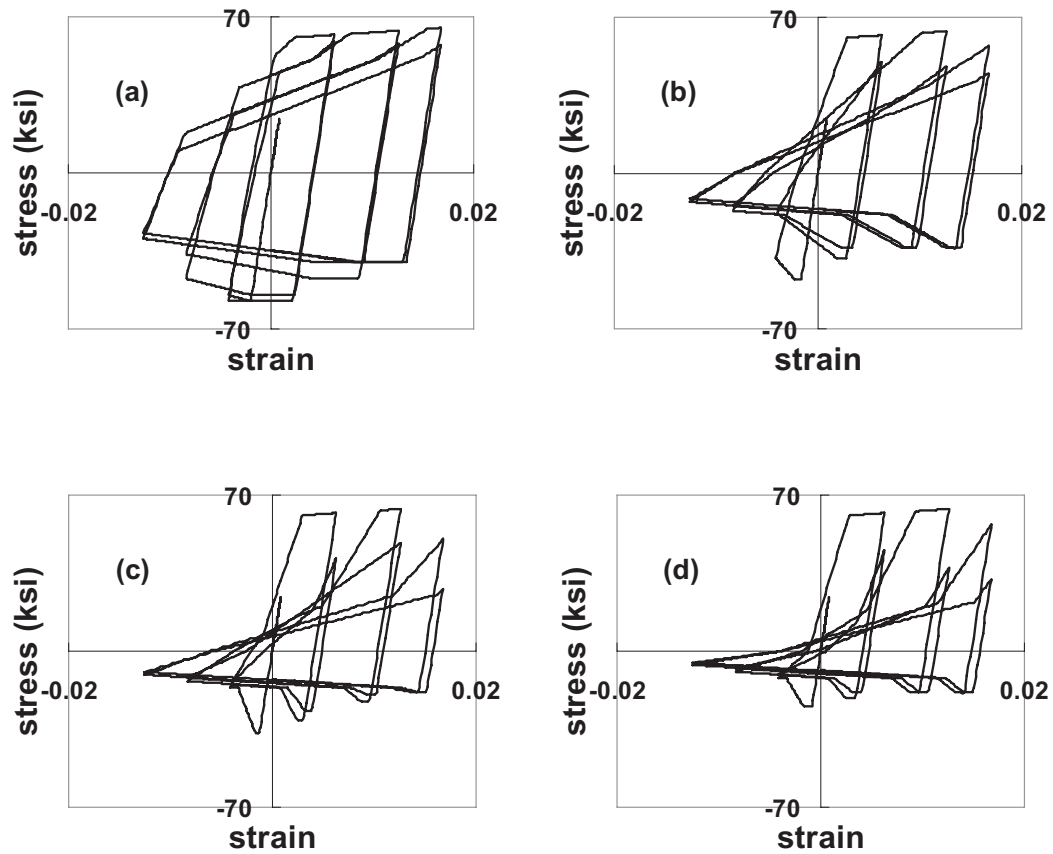


Fig. F27 Parameters for Buckle Material

Case	Einit	PCR	fy	SHR	U34	U45	P45	E5	U6
(a)	29000	-57.42	60.7	0.01	-2.97	-16.5	-0.37	-0.00001	6.76
	P6	U7	P7	U8	P8	E8	PCRF	GF	Nu
	7.03	6.76	5.99	0	0.84	0.06	-0.659	0.13	0.3
Case	Einit	PCR	fy	SHR	U34	U45	P45	E5	U6
(b)	29000	-47.778	60.7	0.01	-1.1	-3.79	-0.31	-0.013	0.69
	P6	U7	P7	U8	P8	E8	PCRF	GF	Nu
	1	0.69	0.7	-0.52	0.6	0.08	-0.55	0.3	0.3
Case	Einit	PCR	fy	SHR	U34	U45	P45	E5	U6
(c)	29000	-36.84	60.7	0.01	-0.8	-1.6	-0.27	-0.01	0.86
	P6	U7	P7	U8	P8	E8	PCRF	GF	Nu
	0.5	0.86	0.43	0	0.33	0.06	-0.3	0.3	0.3
Case	Einit	PCR	fy	SHR	U34	U45	P45	E5	U6
(d)	29000	-24.985	60.7	0.01	-0.8	-1.6	-0.2	-0.01	0.86
	P6	U7	P7	U8	P8	E8	PCRF	GF	Nu
	0.2	0.86	0.39	0	0.33	0.06	-0.3	0.2	0.3

### F09. TensionOnlyBilinear Material

This material is for the strip component modeling of steel panel shear wall (SPSW). See Fig. F28 and F29 for the Tension-Only Bilinear material. In this material, only tensile stress can be taken. This material cannot take compression stress.

#### Command:

Material TensionOnlyBilinear Tag? Ei? SHR? fy? Nu?

PARAMETER	DESCRIPTION
Material	Control Word.
TensionOnlyBilinear	Tension-Only Bilinear yielding and buckling rule.
Tag	Unique tag of this material.
Ei	Initial Young's modulus.
SHR	Strain hardening ratio, as a proportion of Ei.
fy	Yielding stress in tension.
Nu	Poisson ratio, must be between 0 and 0.5. (Default Value = 0)

#### States of the Material:

One can check the material yielding state in the output of the element which uses this material.

CODE	STATE	DESCRIPTION
0	Elastic Loading &Unloading	The stress has not reached the yielding stress yet. Material unloads before reaching the zero stress and will keep initial stiffness.
1	Yielding State	When material loading in tension and reach the yield pt.
2	Buckling State	When material loading/unloading in compression and reach the zero stress.

**EXAMPLE:**

Material    TensionOnlyBilinear    mat    200    0.05    0.131    0.3

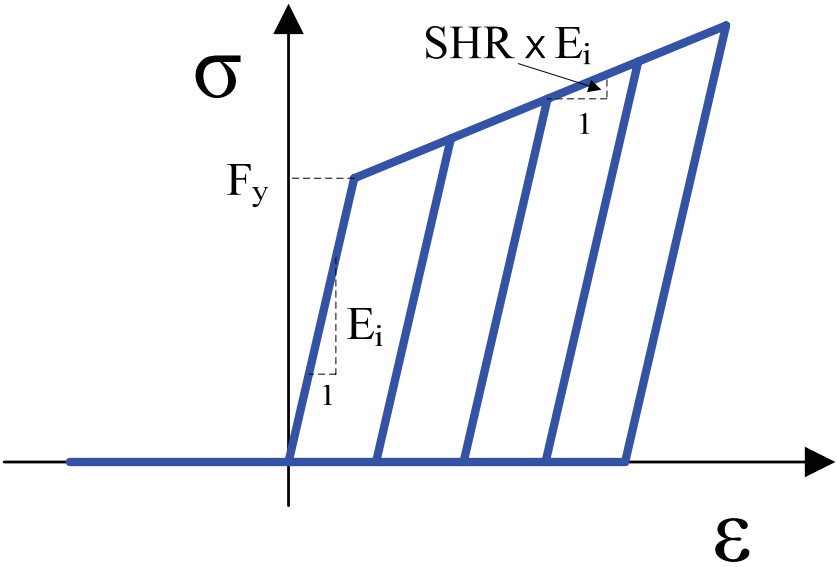


Fig. F28    TensionOnly Material

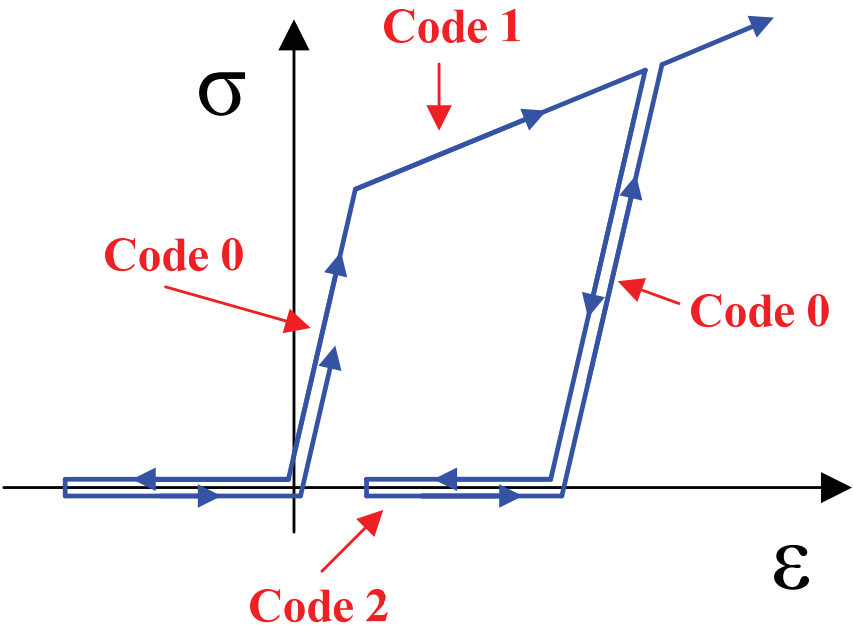


Fig. F29    States of TensionOnlyBilinear Material

## PART G. Element Definition Command

The element can use any material in the material library according to one's modeling requirement. Element's responses are output while the user specifies in the element command. Some elements can take geometric nonlinear effects and element loads into account, see PART H. Additionally, PISA3D can calculate/group strain energy and section force automatically. The user can specify the element's energy/force to required group. All the element objects must have their own tags, even if the types are different between elements.

### G01. Truss Element

This is a general two force member. See Fig. G01, Fig. G02 for the truss element.

#### Command:

Element Truss Tag? I? J? MatTag? Area? GeoK? Out? FEFCase? Fac?  
Energy\_Gp? Force\_Gp?

PARAMETER	DESCRIPTION
Element	Control Word.
Truss	Truss Element.
Tag	Unique tag of this element.
I	Tag of the end node I.
J	Tag of the end node J.
MatTag	Tag of the element's material.
Area	Average cross sectional area of this element.
GeoK	Code for including geometric nonlinear. 0 = Geometric nonlinear effect would not be considered. 1 = Consider geometric nonlinear for this element. (Default Value = 0)

Out	Code for outputting responses. 0 = Responses of this element would not be output. 1 = Output responses of this element. (Default Value = 1)
FEFCase	Tag of the load case for the element load. (Default Value = 0, No load cases.)
Fac	Load factor of the load case for the element load. (Default Value = 0.0)
Energy_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)
Force_Gp	Tag of the section force group. (Default Value = 0, The internal force of this element is not summed up to any section force group.)

## OUTPUT DATA:

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Analysis

Time/Step

Time step in the current analysis method.

Yield

Code

Current state of its composing material, refer to the material's yielding code.

Axial

Force

Current axial force.

Total

Extension

Current total axial deformation of the element, the sum of the elastic and plastic values.

Now\_Plas.

Extension

Current plastic axial deformation.

Accum.\_Plastic Extension

Positive      Negative

Current positive and negative accumulative plastic deformations.

Total

StrainEngy

Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis

StrainEngy

Strain energy absorbed by this element while in material nonlinear state.

Damping

Energy

Total damping energy absorbed by this element.

## 2. The responses below are recorded in the file XXX.ElemEnvelope.

----- Maximum Axial Force-----

Positive      Time      Negative      Time

Envelopes of the axial forces till now and the occurring time, including the positive and negative values.

----- Maximum Extension -----

Positive      Time      Negative      Time

Envelopes of the axial extensions till now and the occurring time, including the positive and negative values.

----- Maximum Plastic Extension-----

Positive      Time      Negative      Time

Envelopes of the axial plastic extensions till now and the occurring time, including the positive and negative values.

### EXAMPLE:

Element Truss e1 N1 N2 1 20000 1 1 0 0 1 1

//This command specifies a truss element with area=20000 and no element loads.

Element Truss brace 15 22 A36 1800 1 1 0 0 B2 2F

//This command specifies a truss named “brace”. Its energy is grouped to group “B2”, and section force is grouped to “2F”.

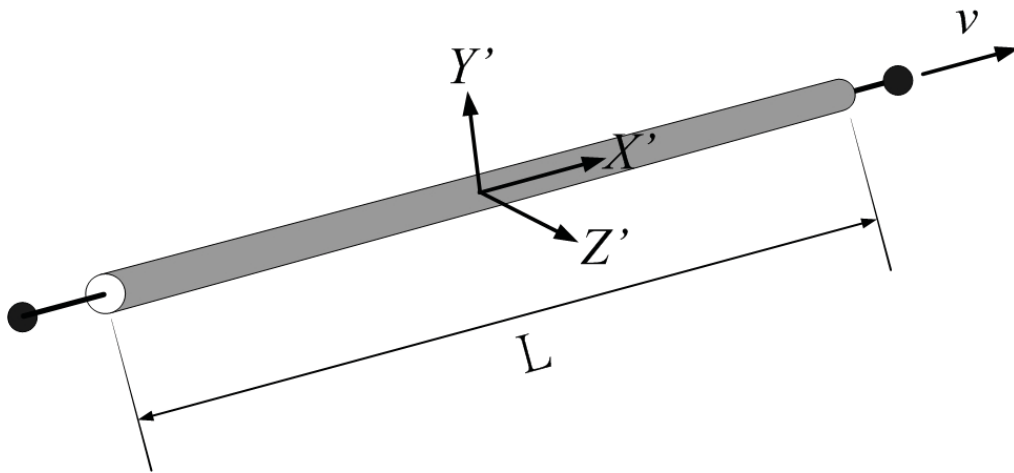


Fig. G01 Local DOF of Truss Element

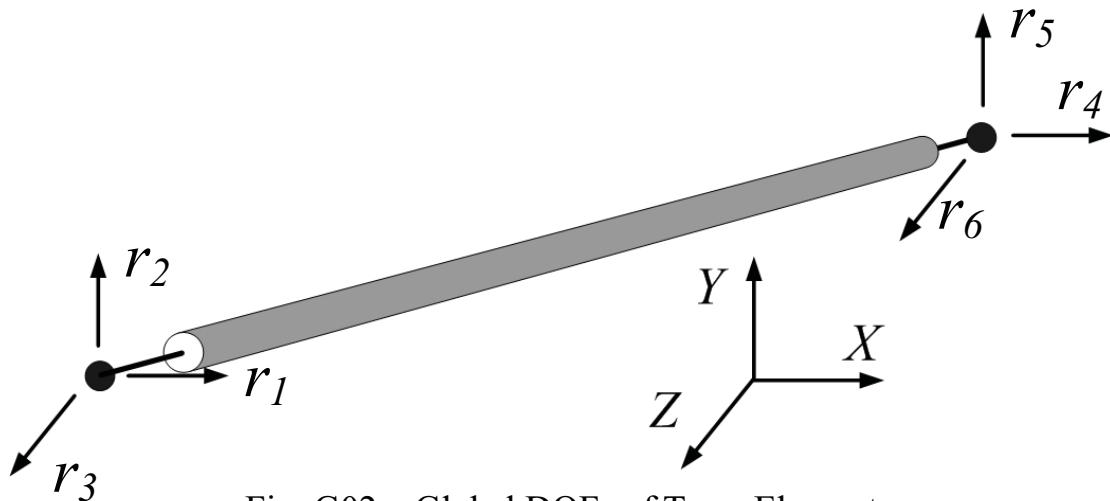


Fig. G02 Global DOFs of Truss Element

## G02. BeamColumn Element

This beamcolumn element adopts the hinge model. This model assumes that internal moments are linear distribution over the element, and the internal shears keep constant over the entire element. In this element, only constant section is available. The user should define a section and its material in the Section Command (see PART H.), and then specifies this section for the beamcolumn. Additionally, rigid end zones for the element and force interaction surface are available for modeling. See Fig. G03 ~ Fig. G07 and Reference [1] for beamcolumn element.

### Command:

Element BeamColumn Tag? I? J? SecTag? REZTag? GeoK? Out? FEFCase? Fac?  
E\_Gp? F\_Gp? PhiY? KZii? KZjj? KZij? KYii? KYjj? KYij?

PARAMETER	DESCRIPTION
Element	Control Word.
BeamColumn	BeamColumn element.
Tag	Unique tag of this element.
I	Tag of the end node I.
J	Tag of the end node J.
SecTag	Tag of the element's section.
REZTag	Tag of the element's rigid end zone. (Default Value = 0, No rigid end zone exists on this element.)
GeoK	Code for including geometric nonlinear. 0 = Geometric nonlinear effect would not be considered. 1 = Consider geometric nonlinear for this element. (Default Value = 0)
Out	Code for outputting responses. 0 = Responses of this element would not be output. 1 = Output responses of this element. (Default Value = 1)



FEFCase	Tag of the load case for the element load. (Default Value = 0, No load cases.)
Fac	Load factor of the load case for the element load. (Default Value = 0.0)
E_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)
F_Gp	Tag of the section force group. (Default Value = 0, The internal force of this element is not summed up to any section force group.)
PhiY	Angle of local Y axis rotates around local X axis, in degrees, and counterclockwise as positive by normal view to local X axis, local X axis is in the direction from nodeI to nodeJ. (Default Value = 0)
KZii	Flexural stiffness factor Kii of local Z axis. (Default Value = 4.0)
KZjj	Flexural stiffness factor Kjj of local Z axis. (Default Value = 4.0)
KZij	Flexural stiffness factor Kij of local Z axis. (Default Value = 2.0)
KYii	Flexural stiffness factor Kii of local Y axis. (Default Value = 4.0)
KYjj	Flexural stiffness factor Kjj of local Y axis. (Default Value = 4.0)
KYij	Flexural stiffness factor Kij of local Y axis. (Default Value = 2.0)

**NOTE:**

1. In 3D analysis, especially for modeling columns or inclined elements, the specification of PhiY must be correct to reflect the strong axis and the weak axis.
2. The flexural stiffness factors can model the different end-boundary conditions. For example, 3, 0, 0 for KZii, KZjj and KZij specifies the element is pin-connection at local Z-axis of J-end. The user can define different values according to proper modeling requirements.

## OUTPUT DATA:

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Analysis

Time/Step

Time step in the current analysis method

-- -- Plas Hinge Yied Code -- --

MzI MzJ MyI MyJ VyI VyJ VzI VzJ

Current state of its composing section, refer to the material's yielding code.

Including the states of the moments (M) and the shears (V), in the local z-axis and y-axis, and at the I-end and the J-end.

BendingMom. (Z\_I\_Now, Z\_J\_Now, Y\_I\_Now, Y\_J\_Now)

End bending moments in the corresponding directions.

ShearForce (Y\_I\_Now, Y\_J\_Now, Z\_I\_Now, Z\_J\_Now)

Element's internal shear forces in the corresponding directions.

Axial            Axial  
Exten.\_Now    Force\_Now

Current axial force and extension.

Torsional      Torsional  
Twist\_Now      Force\_Now

Current torsional force and twist.

Plastic Bending Rotations  
Z\_I\_Now    Z\_I\_Acc.(+)    Z\_I\_Acc.(-)

Current internal plastic bending rotation, and current internal positive and negative accumulative plastic bending rotations, at the I-end of the local Z-axis.

Plastic Shear Rotations  
Y\_I\_Now    Y\_I\_Acc.(+)    Y\_I\_Acc.(-)

Current internal plastic shear deformation, and current internal positive and negative accumulative plastic shear deformation, at the I-end of the local Y-axis.

Total  
StrainEngy  
Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis  
StrainEngy  
Strain energy absorbed by this element while in material nonlinear state.

Damping  
Energy  
Total damping energy absorbed by this element.

## 2. The responses below are recorded in the file XXX.ElemEnvelope.

```
---- Max. Bending_Moment Z_axis_I_end-----
Positive      Time      Negative      Time
```

Envelopes of the bending moments (Z-axis, I-end) till now and the occurring time, including the positive and negative values.

```
--- Max. Bending_Rotation Z_axis_I_end-----
Positive      Time      Negative      Time
```

Envelopes of the bending rotations (Z-axis, I-end) till now and the occurring time, including the positive and negative values.

```
-- Max.Plas. Bending_Rotation Z_axis_I_end
Positive      Time      Negative      Time
```

Envelopes of the plastic bending rotations (Z-axis, I-end) till now and the occurring time, including the positive and negative values.

```
----- Max. Shear_Force Y_axis_I_end-----
Positive      Time      Negative      Time
```

Envelopes of the internal shear forces (Y-axis, I-end) till now and the occurring time, including the positive and negative values.

```
---- Max. Shear_Rotation Y_axis_I_end-----
Positive      Time      Negative      Time
```

Envelopes of the internal shear deformations (Y-axis, I-end) till now and the occurring time, including the positive and negative values.

```
-- Max.Plas. Shear_Rotation Y_axis_I_end --
Positive      Time      Negative      Time
```

Envelopes of the internal plastic shear deformations (Y-axis, I-end) till now and the occurring time, including the positive and negative values.

```
----- Maximum Axial Force-----
Positive      Time      Negative      Time
```

Envelopes of the axial forces till now and the occurring time, including the positive and negative values.

```
----- Maximum Extension -----
Positive      Time      Negative      Time
```

Envelopes of the axial extensions till now and the occurring time, including the positive and negative values.

```
----- Maximum Torsional Force-----
Positive      Time      Negative      Time
```

Envelopes of the torsional forces till now and the occurring time, including the positive and negative values.

```
----- Maximum Twist -----
Positive      Time      Negative      Time
```

Envelopes of the torsional twists till now and the occurring time, including the positive and negative values.

## EXAMPLE:

```
Element BeamColumn 4 3 1 2 5 0 1
```

//This command defines a beamcolumn element named “4”, with end nodes 3 and 1, section 2, rigid end zone 5, no geometric nonlinear effect, and output its responses.

```
Element BeamColumn C1 N1 N2 FSEC1 0 1 1 L1 1.0
```

```
1F 1F 45 4 4 2 0 3 0
```

//This commands defines a beamcolumn C1, end nodes N1 and N2, section FSEC1, no rigid end zone, considering geometric nonlinear effect, output responses, element load case 1.0×L1, grouping energy and force to 1F, local Y-axis rotates 45°, release Y-dir. moment of I end.

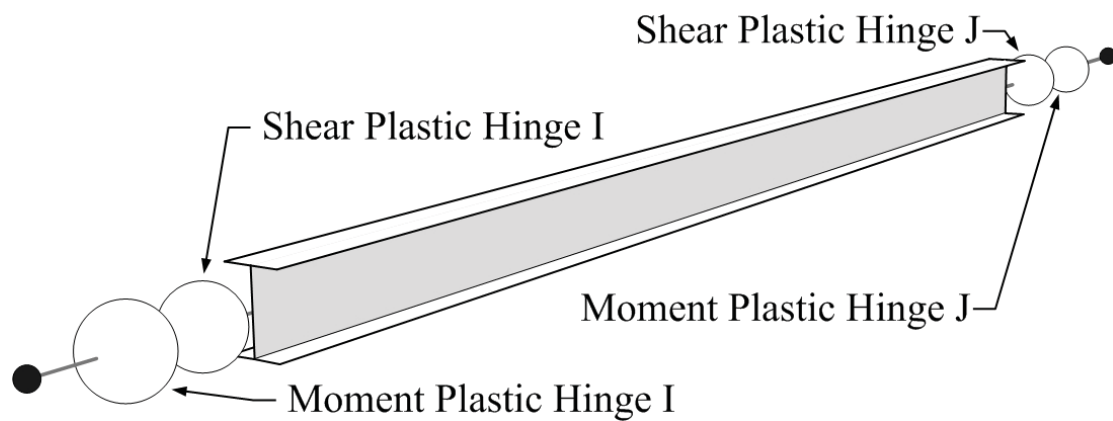


Fig. G03 BeamColumn Element

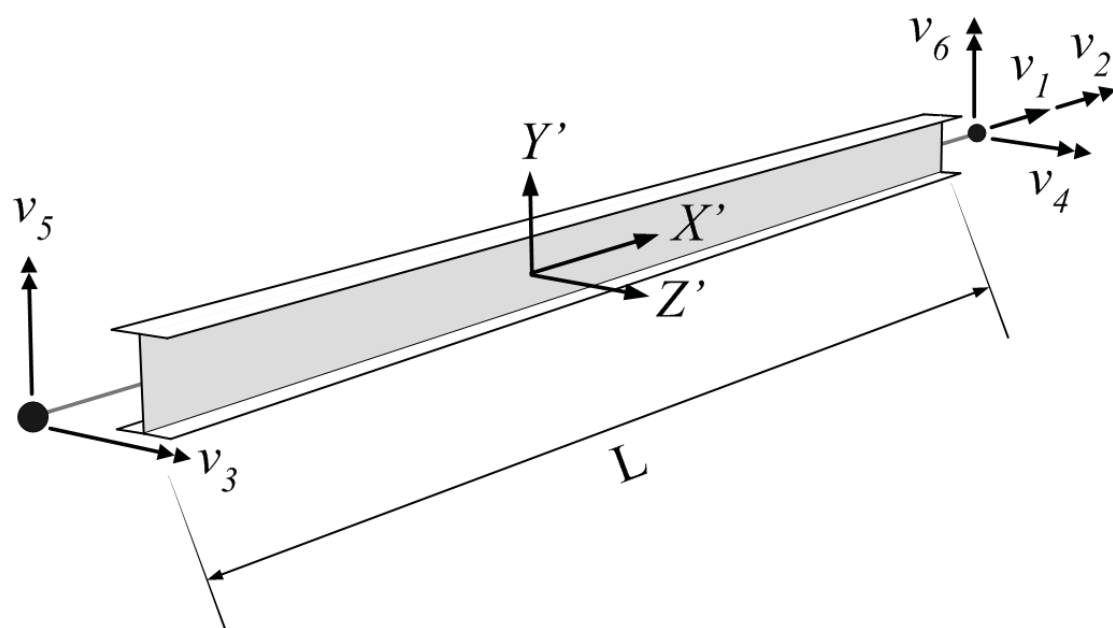


Fig. G04 Local DOFs of BeamColumn

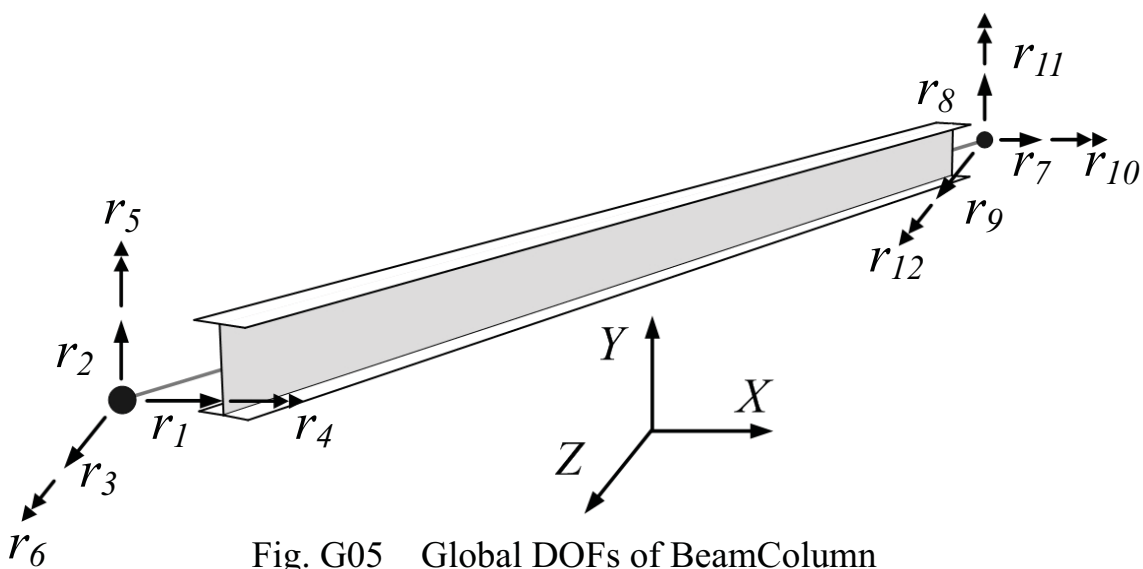


Fig. G05 Global DOFs of BeamColumn

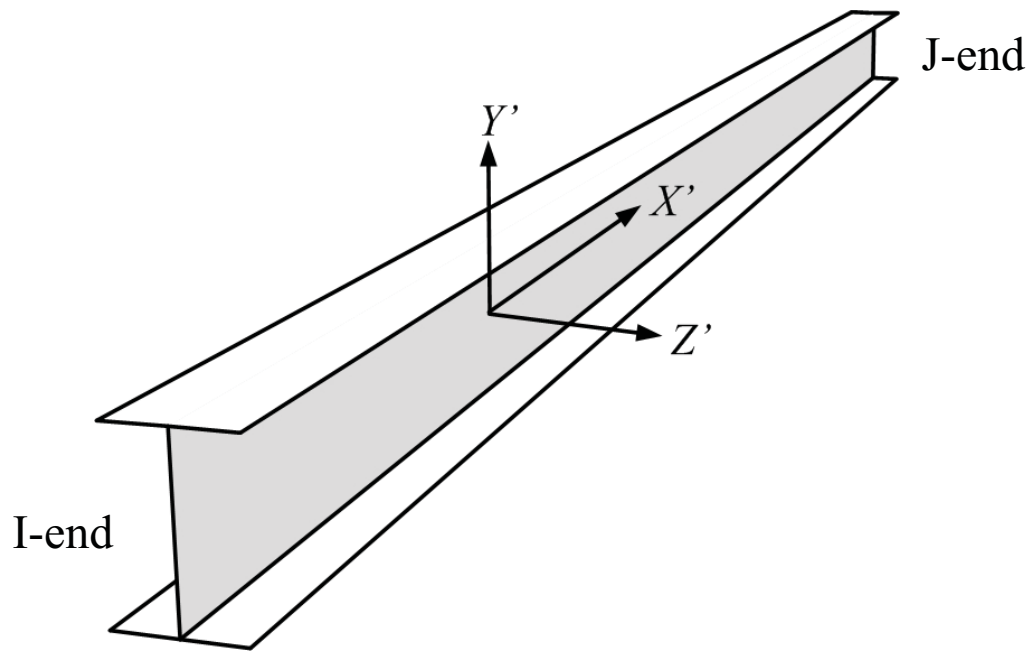


Fig. G06 Default of Local Axes

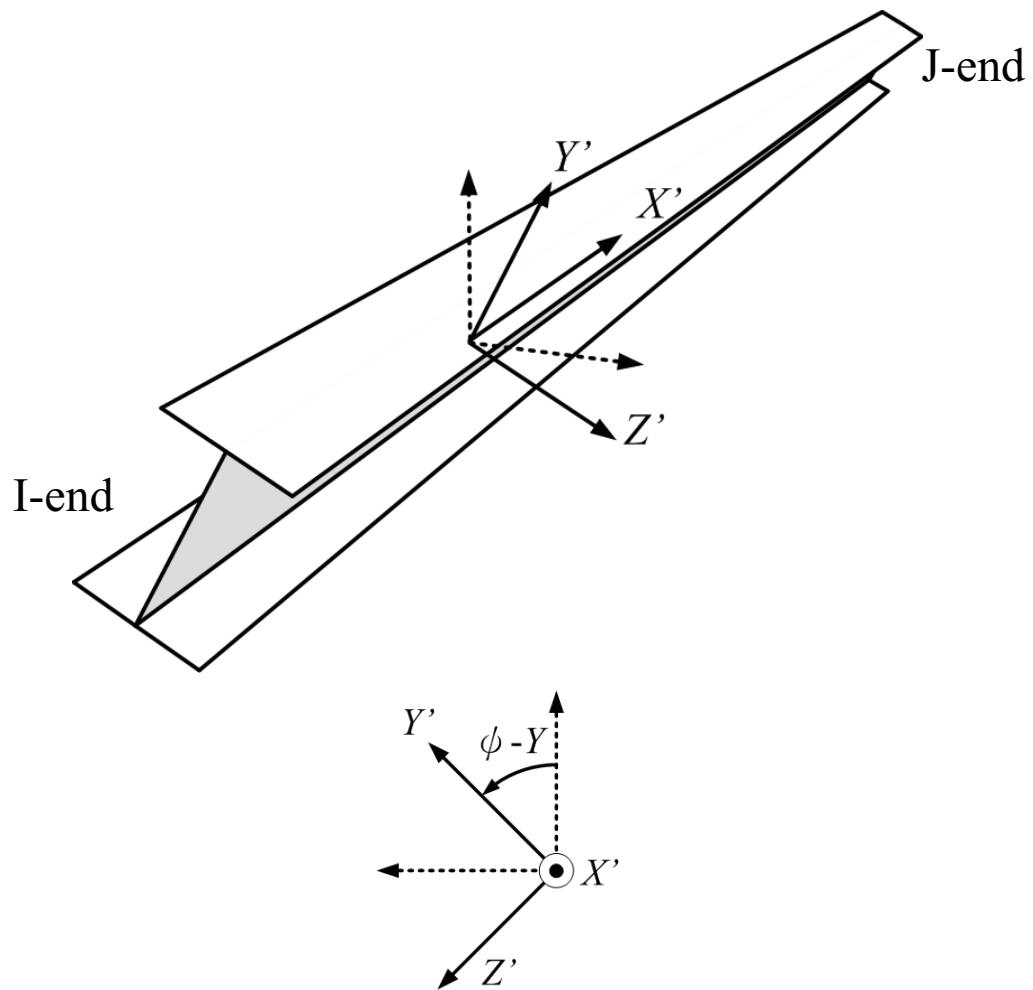


Fig. G07 Rotation of Local Axes

### G03. Joint Element

Joint element is to model the panel zone effect of frames. This is a zero-length element. User must define two nodes at the same position for the joint element. The controlled DOFs (RX or RY or RZ, or the affected rotational DOFs) of the two nodes must be set to be free, and the other DOFs (UX, UY, UZ, and the other rotational DOFs) of these two nodes are identical (see PART D02). Then connect beams to one node, and connect columns to the other node. See Fig. G08 ~ Fig. G10 and Reference [1] for the joint element.

#### Command:

Element Joint Tag? I? J? MatTag? Ki? Dir? Out? Energy\_Gp? NKx? NKy? NKz?

PARAMETER	DESCRIPTION
Element	Control Word.
Joint	Joint element.
Tag	Unique tag of this element.
I	Tag of the end node I.
J	Tag of the end node J.
MatTag	Tag of the element's material.
Ki	Effective initial rotational stiffness. (moment per radian.)
Dir	Direction of local axis for rotation (controlled DOF). RX = Local RX axis is equal to global RX axis. RY = Local RY axis is equal to global RY axis. RZ = Local RZ axis is equal to global RZ axis. (Input 0 if local axis is defined by node K in back.)
Out	Code for outputting responses. 0 = Responses of this element would not be output. 1 = Output responses of this element. (Default Value = 1)

Energy_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)
NKx	X coordinate of “node K” to define local axis.
NKy	Y coordinate of “node K” to define local axis.
NKz	Z coordinate of “node K” to define local axis.

### NOTE:

1. If  $M_y$  of joint element is decided, the  $F_y$  of its material must be input as:

$$F_y = \frac{M_y}{(K_i / E_i)}$$

where  $K_i$  is specified in this element, and  $E_i$  is defined in the material. This means that the yielding moment of this element is calculated as

$$M_y = \frac{K_i \times F_y}{E_i}$$

2. While element's local axis is not identical to global RX/RZ axis, the node K can help user to define element's local axis. If one has defined Dir as RX, RY, or RZ, the specification of node K would be ignored. If Dir is defined as “0”, node K is necessary.

### OUTPUT DATA:

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Analysis

Time/Step

Time step in the current analysis method.

Yield Code

Current state of its composing material, refer to the material's yielding code.

Internal

Moment

Element's internal rotational moment.



Total

Rotation

Current total rotation of the element, the sum of the elastic and plastic values.

Now\_Plas.

Rotation

Current plastic rotation.

Accum.\_Plastic Rotation

Positive      Negative

Current positive and negative accumulative plastic rotations.

Total

StrainEngy

Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis

StrainEngy

Strain energy absorbed by this element while in material nonlinear state.

Damping

Energy

Total damping energy absorbed by this element.

## 2. The responses below are recorded in the file XXX.ElemEnvelope.

----- Maximum Internal Moment-----

Positive      Time      Negative      Time

Envelopes of the internal rotational moments till now and the occurring time, including the positive and negative values.

----- Maximum Rotation -----

Positive      Time      Negative      Time

Envelopes of the total rotations till now and the occurring time, including the positive and negative values.

----- Maximum Plastic Rotation-----

Positive      Time      Negative      Time

Envelopes of the plastic rotations till now and the occurring time, including the positive and negative values.

### EXAMPLE:

```
Element Joint 1 N2 N3 mat2 2e4 RZ 1 1
```

//This command defines a joint element with  $K_i=20000$  and the local axis is the same as global RZ axis. Note that DOF RZ of N2 and N3 must be free, and other 5 DOFs (UX, UY, UZ, RX, RY) of N3 must be identical to N2.

```
Element Joint 2 12 12' 2 2e4 0 1 1 1000 1500 1300
```

//This joint element uses node K(1000, 1500, 1300) to define its local axis, and the local axis is in the direction from node 12 to node K.

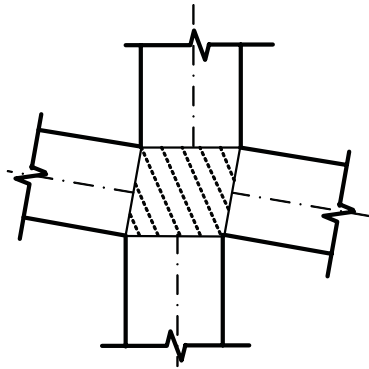


Fig. G08 Panel Zone

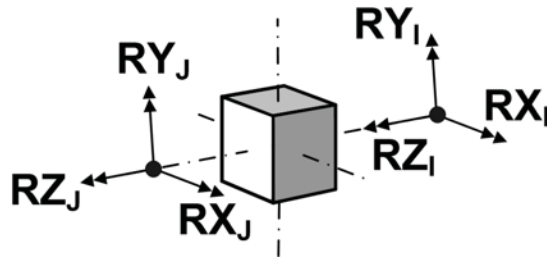


Fig. G09 Global DOFs of Joint Element

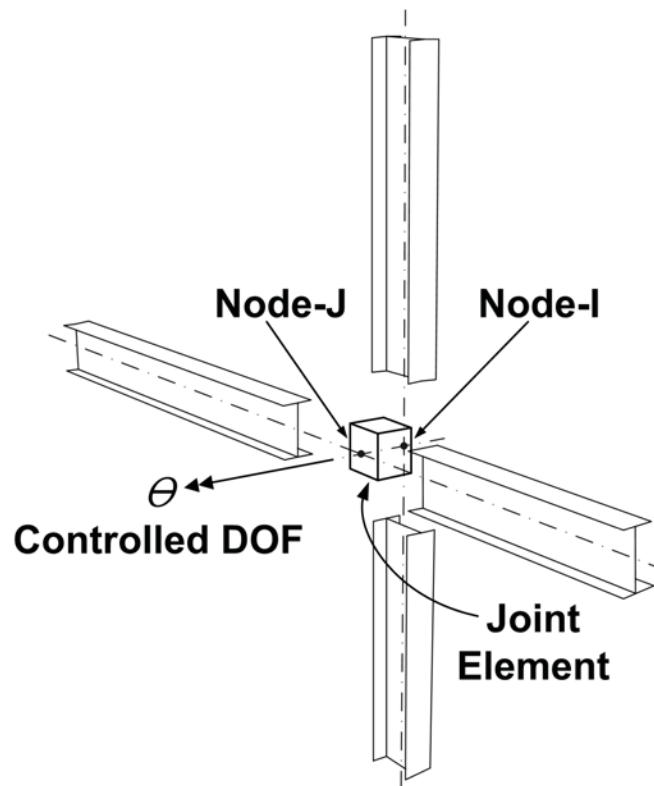


Fig. G10 Usage of Joint Element  
(Node I and node J are of the same coordinate.)

## G04. Panel Element

Panel element is to model the structural wall. Panel is a plane stress element, and the connected 4 nodes must be coplanar. This element has 5 deformation modes, including vertical extension, vertical deflection, horizontal extension, horizontal deflection and horizontal shear deformation. Only the horizontal shear will go into the nonlinear state. See Fig. G11 ~ Fig. G14 for the panel element.

### Command:

Element Panel Tag? I? J? K? L? MatTag? T? Out? Energy\_Gp? Force\_Gp?

PARAMETER	DESCRIPTION
Element	Control Word.
Panel	Panel Element.
Tag	Unique tag of this element.
I	Tag of the end node I. (bottom left)
J	Tag of the end node J. (bottom right)
K	Tag of the end node K. (top right)
L	Tag of the end node L (top left).
MatTag	Tag of the element's material.
T	Average thickness of this element.
Out	Code for outputting responses. 0 = Responses of this element would not be output. 1 = Output responses of this element. (Default Value = 1)
Energy_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)
Force_Gp	Tag of the section force group. (Default Value = 0, The internal force of this element is not summed up to any section force group.)

## NOTE:

1. See Fig. G13, 5 deformation modes exist in the panel element totally, but only the shear deformation ( $q_5$ ) has nonlinear state while analyzers use an inelastic material in this element. Other 4 deformation modes would keep in the elastic states regardless of the element's material.
2. The yielding shear force of this element is calculated by

$$V_y = F_y \times W \times T$$

where  $T$  is defined in the element,  $W$  is the average width of the element, and  $F_y$  is defined in the material of this element.

3. Panel is a plane stress element, so it only provides in-plane stiffness and resisting forces for its four nodes. If there are no boundary elements for the four nodes, it would cause unstable situation to the analysis model.

## OUTPUT DATA:

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Shear

Yield.Code

Current shear stiffness yielding state of its composing material, refer to the material's yielding code.

Shear

Deformation

Current total horizontal shear deformation ( $q_5$ ) of the element, the sum of the elastic and plastic values.

Shear

Force

Current horizontal shear force.

Plastic Shear Deformation

Current      Accum. (+)      Accum. (-)

Current plastic shear deformation,  
current positive and negative accumulative plastic shear deformation.

Vertical

Exten.

Current vertical deformation ( $q_1$ ).

Vertical

Force

Current vertical force.

Vert.\_Bend.

Deform.

Current flexural deformation with the local horizontal axis ( $q_2$ ).

Vert.\_Bend.

Moment

Current bending moment with the local horizontal axis.

Horizontal

Exten.

Current horizontal deformation ( $q_3$ ).

Horizontal

Force

Current horizontal force.

Hori.\_Bend.

Deform.

Current flexural deformation with the local vertical axis ( $q_4$ ).

Hori.\_Bend.

Moment

Current bending moment with the local vertical axis.

Total

StrainEngy

Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis

StrainEngy

Strain energy absorbed by this element while in material nonlinear state.

Damping

Energy

Total damping energy absorbed by this element.

2. The responses below are recorded in the file XXX.ElemEnvelope.

----- Max. Shear\_Force -----

Positive      Time      Negative      Time

Envelopes of the horizontal shear forces till now and the occurring time, including the positive and negative values.

----- Max. Shear\_Deform. -----

Positive      Time      Negative      Time

Envelopes of the horizontal shear deformations till now and the occurring time, including the positive and negative values.

----- Max. Plastic\_Shear\_Deform. -----

Positive      Time      Negative      Time

Envelopes of the plastic shear deformations till now and the occurring time, including the positive and negative values.

----- Max. Vertical\_Force -----

Positive      Time      Negative      Time

Envelopes of the vertical forces till now and the occurring time, including the positive and negative values.

----- Max. Vertical\_Exten. -----

Positive      Time      Negative      Time

Envelopes of the vertical deformations till now and the occurring time, including the positive and negative values.

----- Max. Vertical\_Bending\_Moment -----

Positive      Time      Negative      Time

Envelopes of the vertical bending moments till now and the occurring time, including the positive and negative values.

----- Max. Vertical\_Bending\_Deform. -----  
Positive      Time      Negative      Time

Envelopes of the vertical bending deformations till now and the occurring time, including the positive and negative values.

----- Max. Horizontal\_Force -----  
Positive      Time      Negative      Time

Envelopes of the horizontal forces till now and the occurring time, including the positive and negative values.

----- Max. Horizontal\_Exten. -----  
Positive      Time      Negative      Time

Envelopes of the horizontal deformations till now and the occurring time, including the positive and negative values.

----- Max. Horizontal\_Bending\_Moment -----  
Positive      Time      Negative      Time

Envelopes of the horizontal bending moments till now and the occurring time, including the positive and negative values.

----- Max. Horizontal\_Bending\_Deform. -----  
Positive      Time      Negative      Time

Envelopes of the vertical bending deformations till now and the occurring time, including the positive and negative values.

**EXAMPLE:**

Element    Panel    W1    5    6    4    3    Steel    110    1    1    1

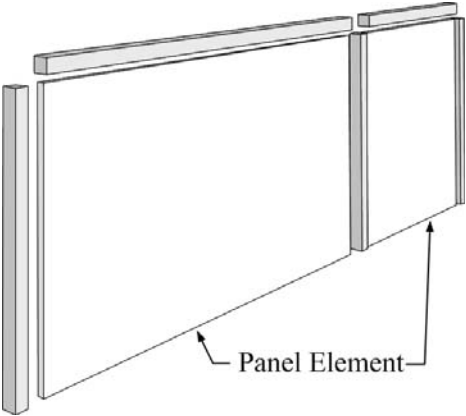


Fig. G11    Panel Element

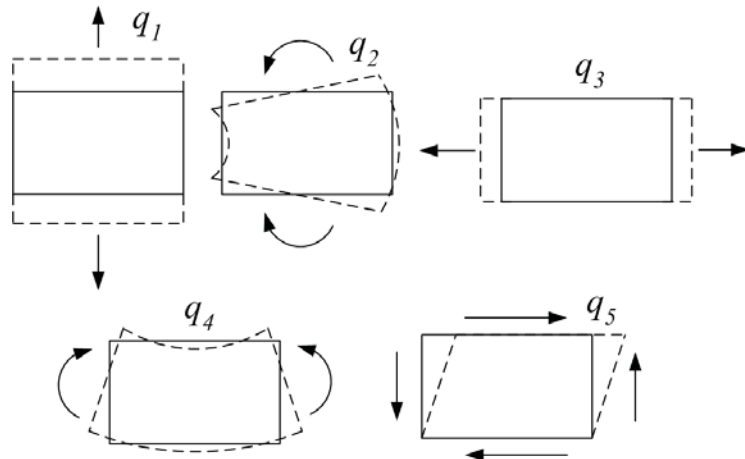


Fig. G12 Deformation Modes

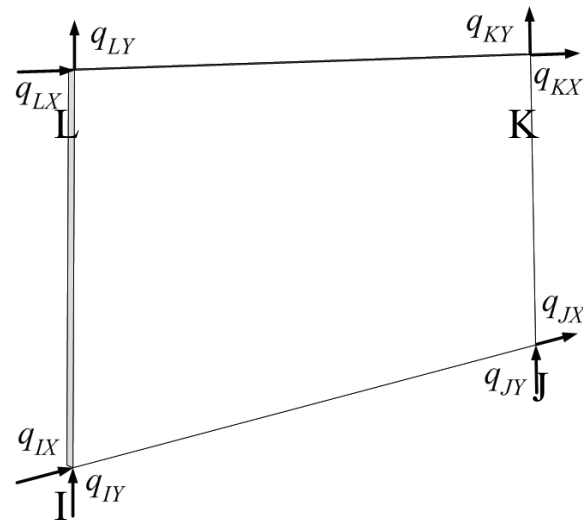


Fig. G13 Local Nodal DOFs of Panel Element

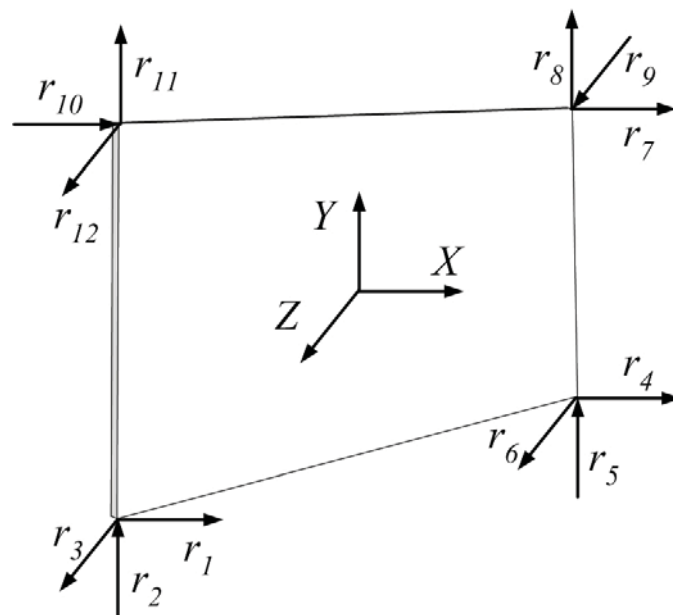


Fig. G14 Global DOFs of Panel Element



## G05. Damper Element

There are two damper types in this element. Users could adopt Kelvin model and Maxwell model by specifying Model\_type = 0 and 1, respectively. The element adopting Kelvin model is a parallel connection of a spring component and a damper component (Fig. G15a). The element adopting Maxwell model is a series connection of a spring component and a damper component (Fig. G15b). Figure G16 shows the damping force of the damper element.

### Command:

Element Damper Tag? I? J? MatTag? Area? C?  $\eta$  ? Out? Energy\_Gp? Force\_Gp?  
Model\_type?

PARAMETER	DESCRIPTION
Element	Control Word.
Damper	Damper Element.
Tag	Unique tag of this element.
I	Tag of the end node I.
J	Tag of the end node J.
MatTag	Tag of the element's material.
Area	Average cross sectional area of this element.
C	Damping coefficient.
$\eta$	Damping exponent. (Default Value = 1.0)
Out	Code for outputting responses. 0 = Responses of this element would not be output. 1 = Output responses of this element. (Default Value = 1)
Energy_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)

Force_Gp	Tag of the section force group. (Default Value = 0, The internal force of this element is not summed up to any section force group.)
Model_Type	The type of the damper 0 = Kelvin model 1 = Maxwell model (Default Value = 0)

### NOTE:

1. The stiffness of this element is defined as  $K = EA/L$ ; where  $E$  is specified in the element's material, and  $L$  represents the length of the element. (length between nodes I and J)
2. The damping force of this element:  $F_D = C\dot{X}V^n$

### OUTPUT DATA:

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Analysis

Time/Step

Time step in the current analysis method.

Total

Force

Current total force.

Total

Extension

Current total extension of the element, the sum of the elastic and plastic values.

Damping

Force

Current damping force.

Stiffness

Force

Current stiffness force.

relative

Velocity

Current velocity of the element. (relative velocity between nodes I and J)

Stiff.

Yie.Code

Current stiffness yielding state of its composing material, refer to the material's yielding code.

Now\_Plas.

Extension

Current plastic extension.

Accum.\_Plastic Extension

Positive      Negative

Current internal positive and negative accumulative plastic extension.

Total

StrainEngy

Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis

StrainEngy

Strain energy absorbed by this element while in material nonlinear state.

Damping

Energy

Total damping energy absorbed by this element.

2. The responses below are recorded in the file XXX.ElemEnvelope.

----- Maximum Total Force-----

Positive      Time      Negative      Time

Envelopes of the internal force till now and the occurring time, including the positive and negative values.

----- Maximum Extension -----

Positive      Time      Negative      Time

Envelopes of the total extension till now and the occurring time, including the positive

and negative values.

----- Maximum Damping Force-----  
Positive      Time      Negative      Time

Envelopes of the damping force till now and the occurring time, including the positive and negative values.

----- Maximum Stiffness Force-----  
Positive      Time      Negative      Time

Envelopes of the stiffness force till now and the occurring time, including the positive and negative values.

----- Maximum Plastic Extension-----  
Positive      Time      Negative      Time

Envelopes of the plastic extension till now and the occurring time, including the positive and negative values.

**EXAMPLE:**

```
Element  Damper  D1  1  6  1  10  30  0.3  1  2  0  0
//This command defines a nonlinear damper with  $\eta=0.3$  (Kelvin model)
Element  Damper  D1  1  6  1  10  30  0.3  1  2  0  1
//This command defines a nonlinear damper with  $\eta=0.3$  (Maxwell model)
```

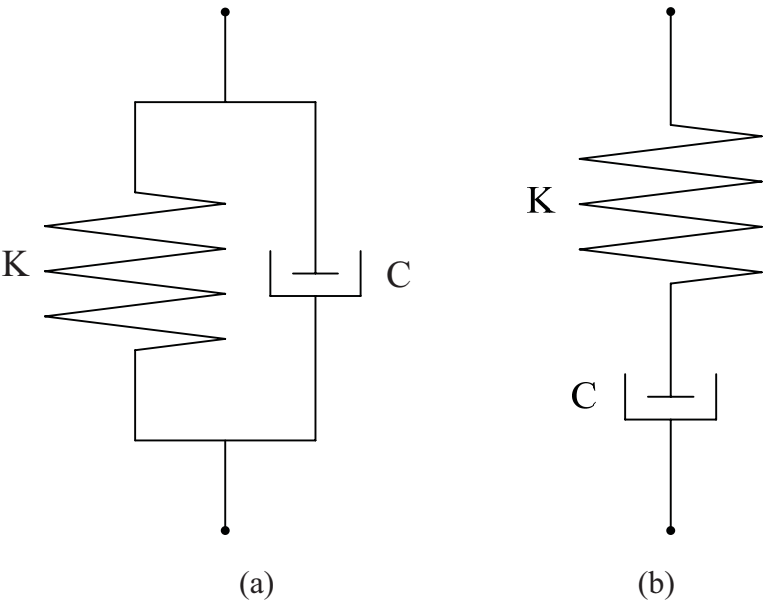


Fig. G15 (a) Kelvin model and (b) Maxwell model of Damper element

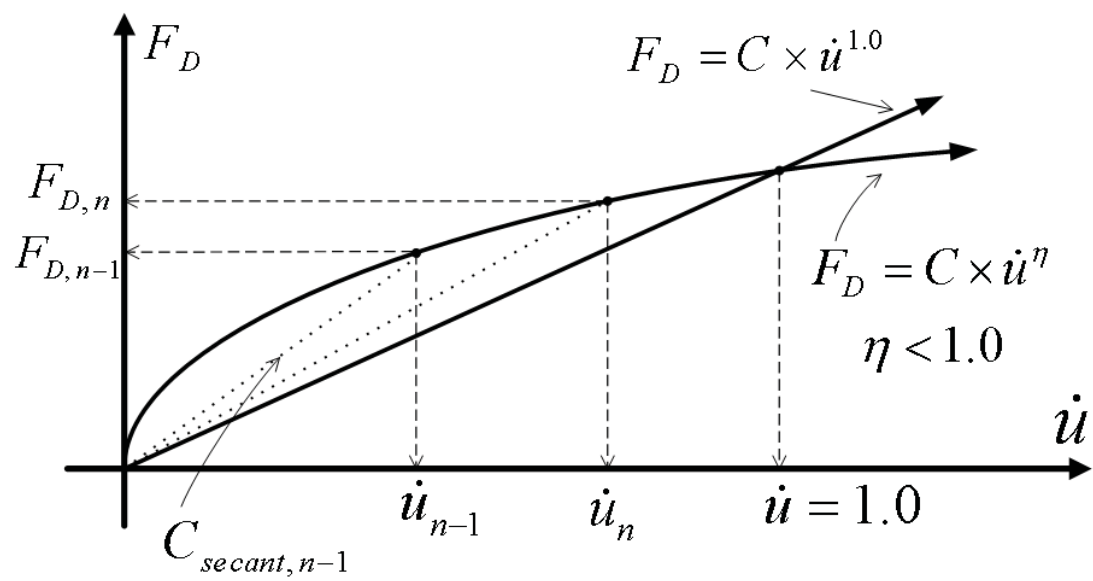


Fig. G16 Damping Force with Various  $\eta$

## G06. BilinearDamper Element

The BilinearDamper element is used to model the behavior of damper which possesses bilinear velocity-damping force path (in Fig. G17). There are two damper types in this element. Users could adopt Kelvin model and Maxwell model by specifying Model\_type = 0 and 1, respectively. The element adopting Kelvin model is a parallel connection of a spring component and a damper component (Fig. G18a). The element adopting Maxwell model is a series connection of a spring component and a damper component (Fig. G18b).

### Command:

Element BilinearDamper Tag? I? J? MatTag? Area? Fr? C1? C2?  
Out? Energy\_Gp? Force\_Gp? Model\_Type?

PARAMETER	DESCRIPTION
Element	Control Word.
BilinearDamper	Bilinear Damper Element.
Tag	Unique tag of this element.
I	Tag of the end node I.
J	Tag of the end node J.
MatTag	Tag of the element's material.
Area	Average cross sectional area of this element.
Fr	Damping force of point A (as shown in Figure 1)
C1	Slope of segment 1 (as shown in Figure 1)
C2	Slope of segment 2. (as shown in Figure 1)
Out	Code for outputting responses.  0 = Responses of this element would not be output.  1 = Output responses of this element.  (Default Value = 1)

Energy_Gp	Tag of the strain energy group. (Default Value = 0, The strain energy of this element is not summed up to any energy group.)
Force_Gp	Tag of the section force group. (Default Value = 0, The internal force of this element is not summed up to any section force group.)
Model_Type	The type of the damper 0 = Kelvin model 1 = Maxwell model (Default Value = 0)

**NOTE:**

1. The stiffness of this element is defined as  $K = EA/L$ ; where  $E$  is specified in the element's material, and  $L$  represents the length of the element. (length between nodes I and J)

**OUTPUT DATA:**

1. The responses below are recorded in files XXX.element and XXX.ElemRecord.

Analysis

Time/Step

Time step in the current analysis method.

Total

Force

Current total force.

Total

Extension

Current total extension of the element, the sum of the elastic and plastic values.

Damping

Force

Current damping force.

Stiffness

Force

Current stiffness force.

relative

Velocity

Current velocity of the element. (relative velocity between nodes I and J)

Stiff.

Yie.Code

Current stiffness yielding state of its composing material, refer to the material's yielding code.

Now\_Plas.

Extension

Current plastic extension.

Accum.\_Plastic Extension

Positive          Negative

Current internal positive and negative accumulative plastic extension.

Total

StrainEngy

Total strain energy absorbed by this element, including linear part and nonlinear part.

Hysteresis

StrainEngy

Strain energy absorbed by this element while in material nonlinear state.

Damping

Energy

Total damping energy absorbed by this element.

## 2. The responses below are recorded in the file XXX.ElemEnvelope.

----- Maximum Total Force-----

Positive          Time          Negative          Time

Envelopes of the internal force till now and the occurring time, including the positive and negative values.



----- Maximum Extension -----

Positive      Time      Negative      Time

Envelopes of the total extension till now and the occurring time, including the positive and negative values.

----- Maximum Damping Force-----

Positive      Time      Negative      Time

Envelopes of the damping force till now and the occurring time, including the positive and negative values.

----- Maximum Stiffness Force-----

Positive      Time      Negative      Time

Envelopes of the stiffness force till now and the occurring time, including the positive and negative values.

----- Maximum Plastic Extension-----

Positive      Time      Negative      Time

Envelopes of the plastic extension till now and the occurring time, including the positive and negative values.

### EXAMPLE:

```
Element BilinearDamper E5 N5 N3 E 0 79.8 483.6 60 1 E_ED
F_ED 0
```

//This command defines a bilinear damper (Kelvin model)

```
Element BilinearDamper E5 N5 N3 E 20 79.8 483.6 60 1 E_ED
F_ED 1
```

//This command defines a bilinear damper (Maxwell model)

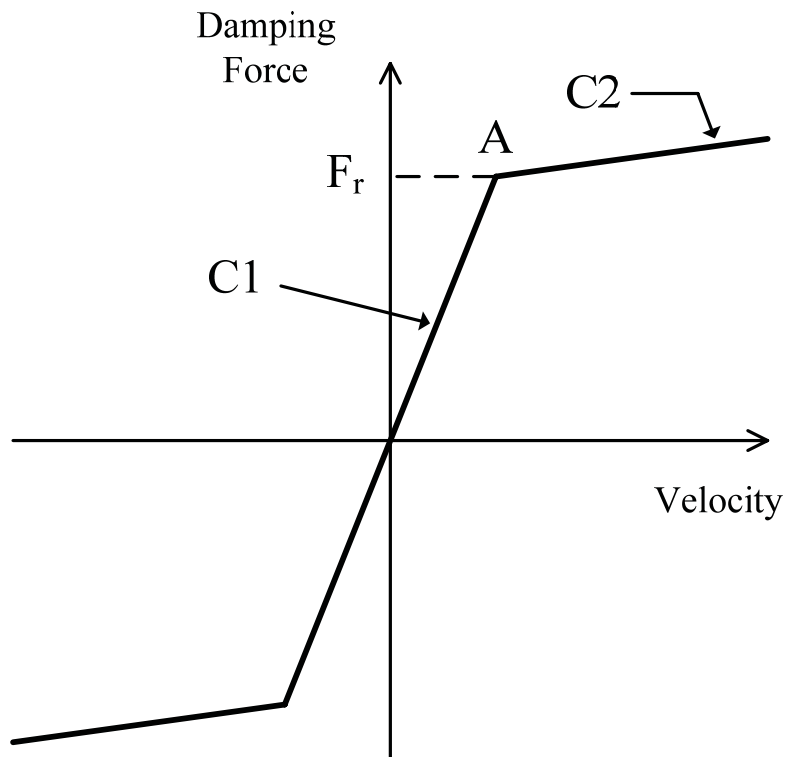


Fig. G17 BilinearDamper Element

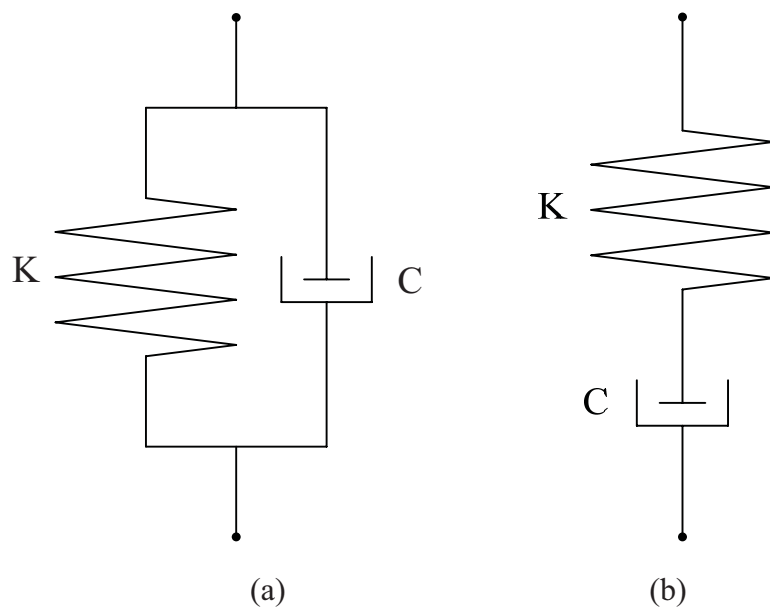


Fig. G18 (a) Kelvin model and (b) Maxwell model

## PART H. Other Associated Element Commands

This part introduces some developed associated element commands.

### H01. Element Section Definition – BCSection Command

This section command is for composing the BeamColumn element (G02). The interaction surface of internal forces is also available (optional) for the section command.

#### Command:

Section BCSection Tag? MatTag? Inter\_Surf? Area? Iz? Iy? J? Sz? Sy? Avy? Avz?

PARAMETER	DESCRIPTION
Section	Control Word.
BCSection	BeamColumn Section.
Tag	Unique tag of this section.
MatTag	Tag of the section's material.
Inter_Surf	Tag of the section's interaction surface. (Input 0 if no interaction surface is considered for this section.)
Area	<b>Effective</b> average cross sectional area.
Iz	<b>Effective</b> moment of Inertia about local Z axis.
Iy	<b>Effective</b> moment of Inertia about local Y axis.
J	<b>Effective</b> torsional constant.
Sz	<b>Effective</b> section modulus about local Z axis.
Sy	<b>Effective</b> section modulus about local Y axis.
Avy	<b>Effective</b> shear area in local Y axis.
Avz	<b>Effective</b> shear area in local Z axis.

**NOTE:**

1. This section uses effective section moduli,  $S_z$  and  $S_y$ , to calculate the yielding moments, i.e.  $M_{Z\_yield} = S_z \times F_y$ ;  $M_{Y\_yield} = S_y \times F_y$ , where  $F_y$  is specified in the material. One can vary  $S_z$ ,  $S_y$ , or  $F_y$  to obtain required yielding moments.
2. This section uses shear areas,  $Av_y$  and  $Av_z$ , to consider shear deformations of the element ends, and to calculate the yielding shear forces.

In this section,  $V_{Y\_yield} = Av_y \times F_y$ ;  $V_{Z\_yield} = Av_z \times F_y$ . One can vary  $Av_y$ ,  $Av_z$ , or  $F_y$  to obtain required yielding shears.

**EXAMPLE:**

```
Section  BCSection  FSEC1    2  YS1   80000   1.07E9   2.67e8  7.324e8
5.333E6  5.333E6  66666.67  66666.67
```

//This command defines a section named “FSEC1” with the interaction surface YS1.

## H02. Rigid End Zone Definition

This command is to define 100% rigid end zones of beamcolumn elements. 3D rigid end zone is available in this command. See Fig. H01 and Fig. H02 for the rigid end zone.

### Command:

RigidEndZone Tag? Xi? Xj? Yi? Yj? Zi? Zj?

PARAMETER	DESCRIPTION
RigidEndZone	Control Word.
Tag	Unique tag of this rigid end zone.
Xi	Projection of eccentricity on global X axis at I end.
Xj	Projection of eccentricity on global X axis at J end.
Yi	Projection of eccentricity on global Y axis at I end.
Yj	Projection of eccentricity on global Y axis at J end.
Zi	Projection of eccentricity on global Z axis at I end.
Zj	Projection of eccentricity on global Z axis at J end.

### EXAMPLE:

RigidEndZone REZ01 25 25 0 0 35 35

RigidEndZone REZ02 10 10 15 15 10 15

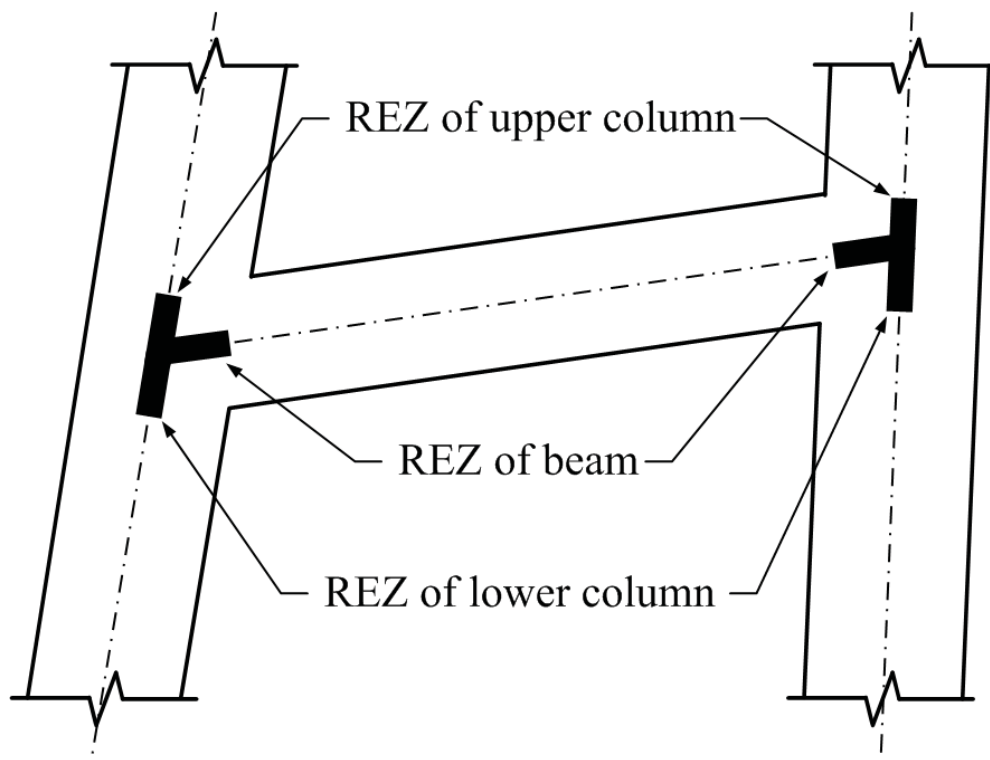


Fig. H01 Rigid End Zone

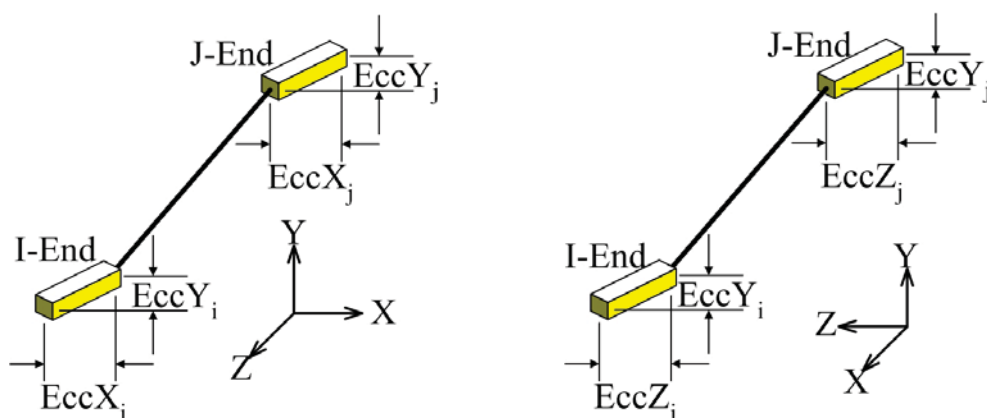


Fig. H02 Definition of Rigid End Zones

### H03. Interaction Surface Definition (Wide-Flange Steel Type)

This is an optional command for the beamcolumn element to consider the interaction effects between the axial force and moments. In this command, the user should specify normalized  $P$ - $M_z$  and  $P$ - $M_y$  curve, and the exponent of  $M_z$ - $M_y$  contour to construct the 3D surface. See Fig. H03(a), Fig. H04 ~ Fig. H07 for the interaction surface.

#### Command:

InteractionSurface Steel Tag? pMA? pPA? pMB? pPB? pME? pPE? pMF? pPF?  
 $\alpha$  1?  $\alpha$  2?

PARAMETER	DESCRIPTION
InteractionSurface	Control Word.
Steel	Wide-flange steel type.
Tag	Unique tag of this interaction surface.
pMA	Normalized M-coordinate of balance point A, must be positive.
pPA	Normalized P-coordinate of balance point A, must be positive.
pMB	Normalized M-coordinate of balance point B, must be positive.
pPB	Normalized P-coordinate of balance point B, must be positive.
pME	Normalized M-coordinate of balance point E, must be positive. (Default Value = pMA)
pPE	Normalized P-coordinate of balance point E, must be positive. (Default Value = pPA)
pMF	Normalized M-coordinate of balance point F, must be positive. (Default Value = pMB)
pPF	Normalized P-coordinate of balance point F, must be positive. (Default Value = pPB)

$\alpha 1$	Z axis exponent for the moment contour of the interaction surface. (Default Value = 1.0)
$\alpha 2$	Y axis exponent for the moment contour of the interaction surface. (Default Value = 1.0)

**NOTE:**

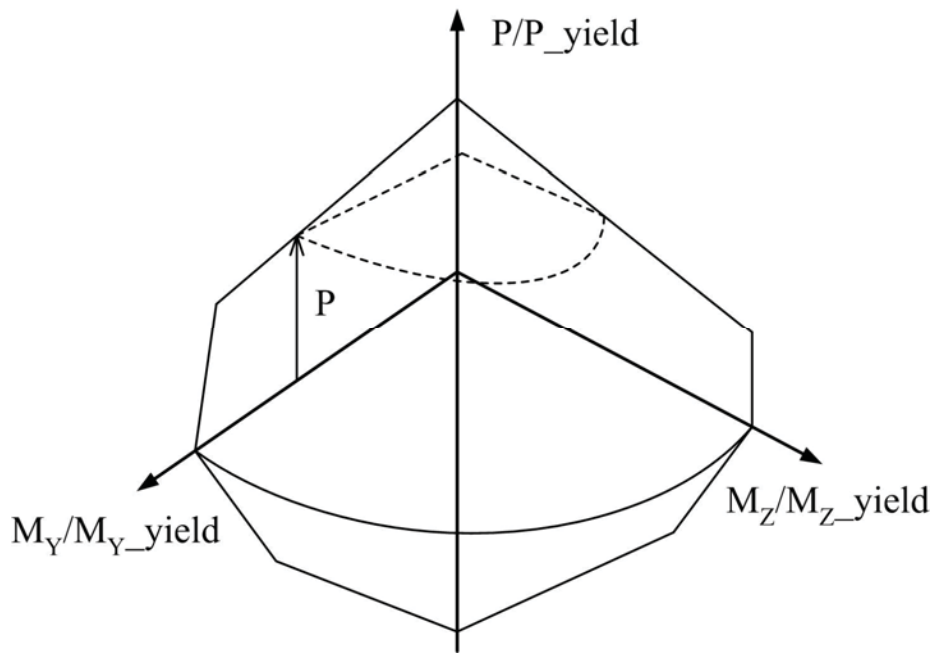
1. All the interaction surface objects should have their own tags, even if the types/features are different between interaction surfaces.

**EXAMPLE:**

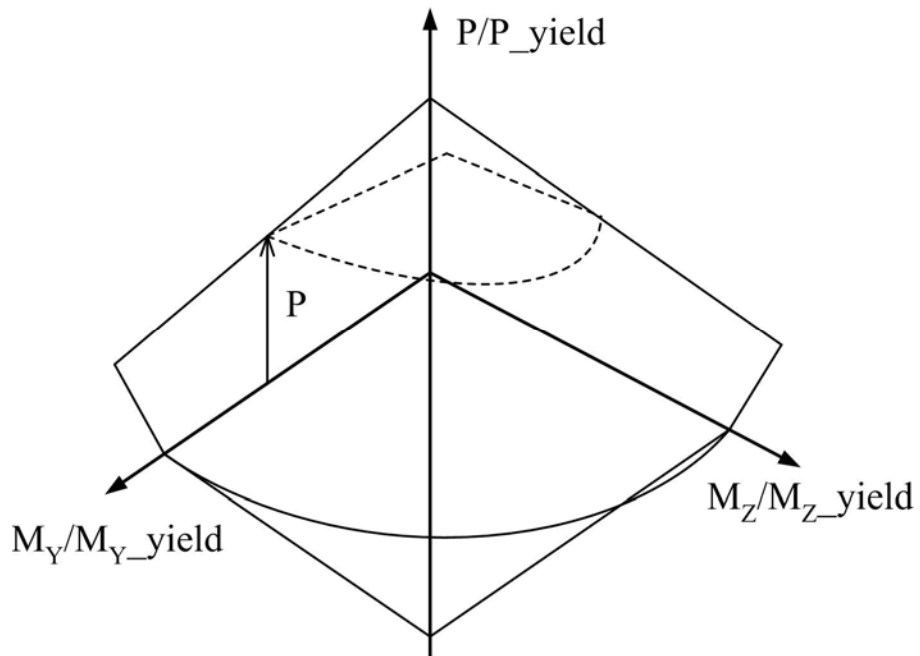
InteractionSurface Steel YS1 1.0 0.15 1.0 0.15 1.0 0.4 1.0 0.4 2 2

//This command defines a general interaction surface of an I-shape steel column.





(a) Wide-Flange Steel Column Type



(b) RC Column Type

Fig. H03 Interaction Surface

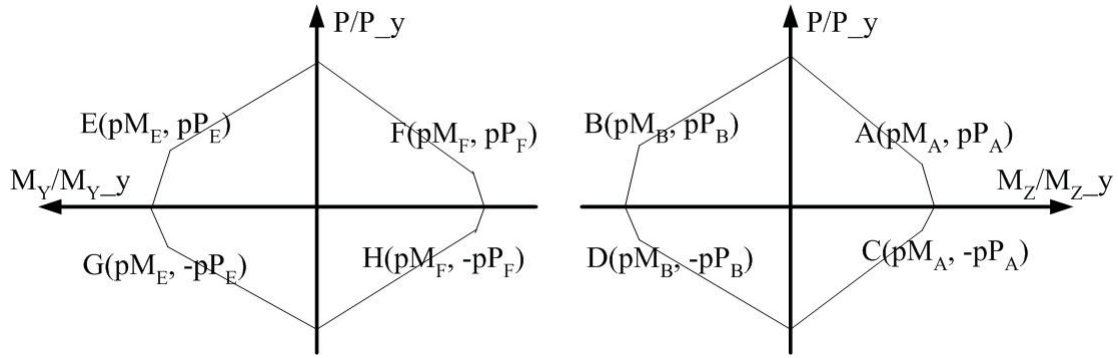


Fig. H04 Wide-Flange Steel Type

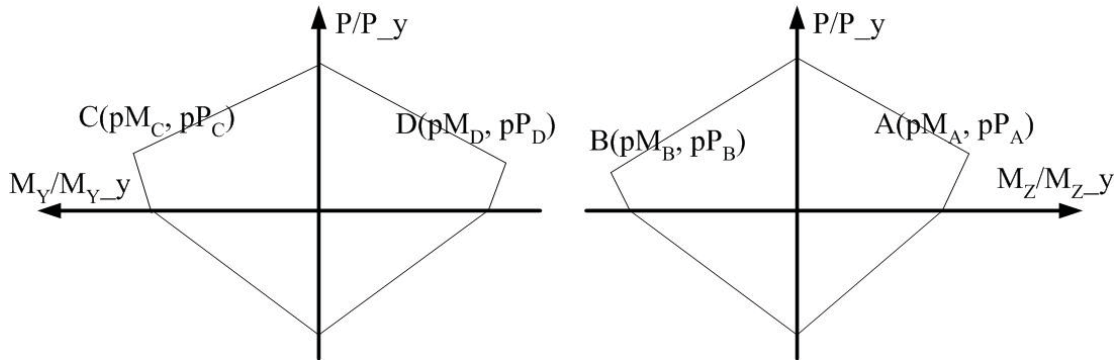


Fig. H05 RC Type

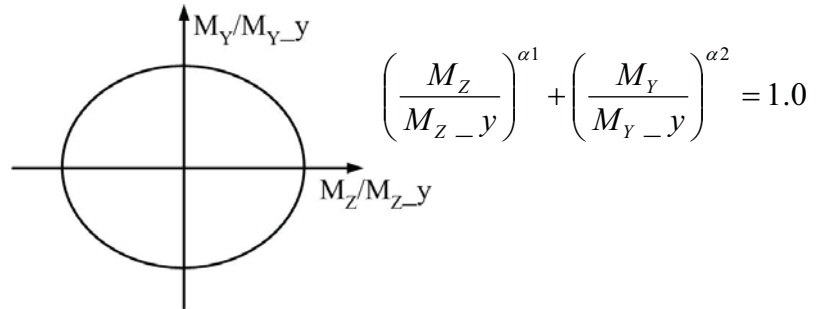


Fig. H06 Moment Contour of The Interaction Surface

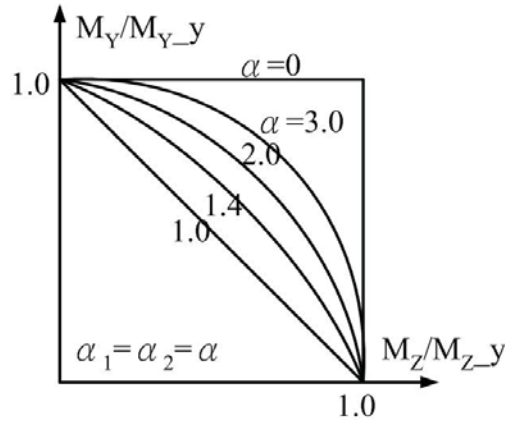


Fig. H07 Moment Contour of The Interaction Surface

(With  $\alpha_1 = \alpha_2$ )

## H04. Interaction Surface Definition (RC Column Type)

See Fig. H03(b), Fig. H04 ~ Fig. H07 for the interaction surface.

### Command:

InteractionSurface RC Tag? pMA? pPA? pMB? pPB? pMC? pPC? pMD? pPD?  
 $\alpha 1?$   $\alpha 2?$

PARAMETER	DESCRIPTION
InteractionSurface	Control Word.
RC	Reinforced concrete column type.
Tag	Unique tag of this interaction surface.
pMA	Normalized M-coordinate of balance point A, must be positive.
pPA	Normalized P-coordinate of balance point A, must be positive.
pMB	Normalized M-coordinate of balance point B, must be positive.
pPB	Normalized P-coordinate of balance point B, must be positive.
pMC	Normalized M-coordinate of balance point C, must be positive. (Default Value = pMA)
pPC	Normalized P-coordinate of balance point C, must be positive. (Default Value = pPA)
pMD	Normalized M-coordinate of balance point D, must be positive. (Default Value = pMB)
pPD	Normalized P-coordinate of balance point D, must be positive. (Default Value = pPB)
$\alpha 1$	Z axis exponent for the moment contour of the interaction surface. (Default Value = 1.0)

$\alpha 2$	Y axis exponent for the moment contour of the interaction surface. (Default Value = 1.0)
------------	---

**EXAMPLE:**

```
InteractionSurface RC 1 1.2 0.15 1.2 0.15 1.1 0.2 1.1 0.2 1.15 1.15
```

//This command defines a RC column interaction surface, and the exponents for the moment contour are set to be more conservative.

## H05. Element Load (Fixed End Force)

This is an optional command for element loads. First of all, the user should transform loads to end loads and then use this command. In analysis, some element load commands will form an element load case (refer to PART H06. Element Load Case), so the user finally specifies this load case in the element command. See Fig. H08 for fixed end forces.

### Command:

ElementLoad FEF Tag? Dir? PXi? PYi? PZi? MXi? MYi? MZi? PXj? PYj? PZj?  
MXj? MYj? MZj?

PARAMETER	DESCRIPTION
ElementLoad	Control Word.
FEF	Fixed end force of the element.
Tag	Unique tag of this element load.
Dir	Direction code.  0 = Forces are in the local coordinate system. 1 = Forces are in the global coordinate system.
PXi	Clamping force in the X axis at I end.
PYi	Clamping force in the Y axis at I end.
PZi	Clamping force in the Z axis at I end.
MXi	Clamping moment about the X axis at I end. (counterclockwise as positive by normal view to the X axis)
MYi	Clamping moment about the Y axis at I end. (counterclockwise as positive by normal view to the Y axis)
MZi	Clamping moment about the Z axis at I end. (counterclockwise as positive by normal view to the Z axis)
PXj	Clamping force in the X axis at J end.
PYj	Clamping force in the Y axis at J end.
PZj	Clamping force in the Z axis at J end.

MXj	Clamping moment about the X axis at J end. (counterclockwise as positive by normal view to the X axis)
MYj	Clamping moment about the Y axis at J end. (counterclockwise as positive by normal view to the Y axis)
MZj	Clamping moment about the Z axis at J end. (counterclockwise as positive by normal view to the Z axis)

**NOTE:**

1. The fixed end forces are to resist external element loads, so the signs of the FEF values are opposite to the signs of the external element loads.

**EXAMPLE:**

ElementLoad FEF EL1 0 0 0 1500 0 36000 0 0 0 1500 0 -36000 0

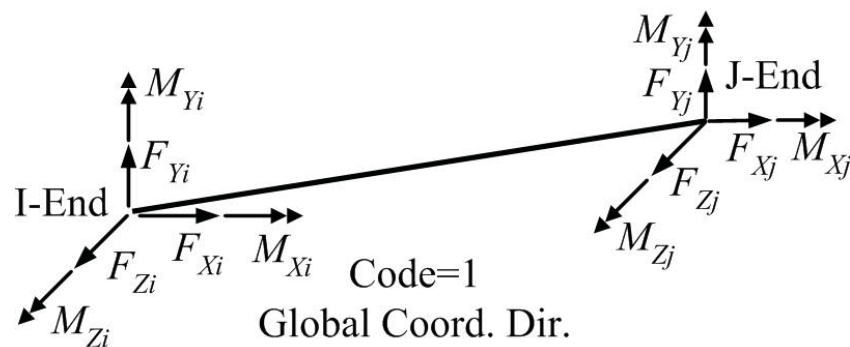
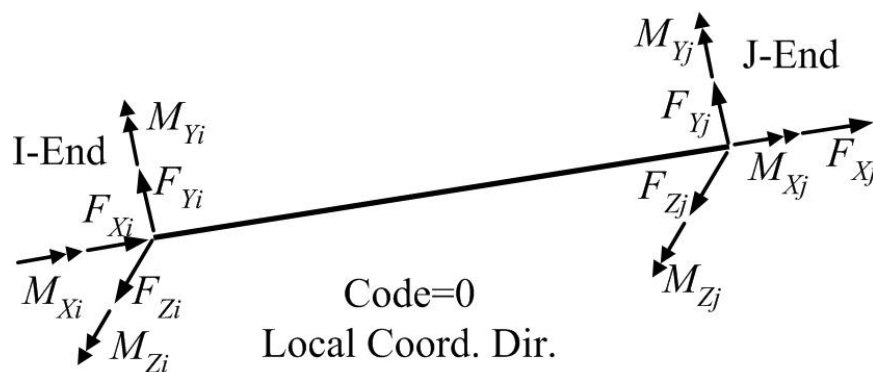


Fig. H08 Fixed End Force

(The arrows indicate the direction of positive values.)

## H06. Element Load Case

One can specify unlimited numbers of element loads and their corresponding load factors in an element load case command.

### Command:

```
ElemLoadCase CaseTag? LpTag1? LpFac1? ... LpTagN? LpFacN?
```

PARAMETER	DESCRIPTION
ElemLoadCase	Control Word.
CaseTag	Unique tag of this element load case.
LpTag1	Tag of the 1st element load.
LpFac1	Load Factor of the 1st element load.
LpTagN	Tag of the Nth element load.
LpFacN	Load Factor of the Nth element load.

### EXAMPLE:

```
ElemLoadCase C1 Ld1 1.4 Ld2 1.4 Ld3 1.7
```

```
//This load case C1 is  $1.4Ld_1 + 1.4Ld_2 + 1.7Ld_3$ .
```

## H07. Element Section Definition – BCSection02 Command

This section command is for composing the BeamColumn element (G02). The main difference between BCSection02 and BCSection (H01) is that user can specify various materials for different degrees of freedom on one BCSection02 so that the BeamColumn can perform various nonlinearities for different degrees of freedom. And the BCSection can only be specified identical material for all the degrees of freedom.

### Command:

Section BCSection02 Tag? Mat\_Mz? Mat\_My? Mat\_Vy? Mat\_Vz? Mat\_P\_T?  
Inter\_Surf? Area? Iz? Iy? J? Sz? Sy? Avy? Avz?

PARAMETER	DESCRIPTION
Section	Control Word.
BCSection02	Define BCSection02.
Tag	Unique tag of this section.
Mat_Mz	Tag of the material to control <b>moment about local Z axis</b> .
Mat_My	Tag of the material to control <b>moment about local Y axis</b> .
Mat_Vy	Tag of the material to control <b>shear in local Y axis</b> .
Mat_Vz	Tag of the material to control <b>shear in local Z axis</b> .
Mat_P_T	Tag of the material to control <b>torsion and axial force</b> .  The torsion and axial force of BCSection02 will keep in linear elastic. Only the Young's modulus and the Poisson ratio are adopted even if this material is nonlinear.
Inter_Surf	Tag of the section's interaction surface.  (Input 0 if no interaction surface is considered for this section.)
Area	<b>Effective</b> average cross sectional area.
Iz	<b>Effective</b> moment of Inertia about local Z axis.
Iy	<b>Effective</b> moment of Inertia about local Y axis.
J	<b>Effective</b> torsional constant.



Sz	<b>Effective</b> section modulus about local Z axis.
Sy	<b>Effective</b> section modulus about local Y axis.
Avy	<b>Effective</b> shear area in local Y axis.
Avz	<b>Effective</b> shear area in local Z axis.

### NOTE:

1. This section uses effective section moduli,  $S_z$  and  $S_y$ , and yielding stresses of the corresponding materials,  $F_{y\_Mat\_Mz}$ ,  $F_{y\_Mat\_My}$  to calculate the yielding moments, i.e.  $M_{Z\_yield} = S_z \times F_{y\_Mat\_Mz}$ ;  $M_{Y\_yield} = S_y \times F_{y\_Mat\_My}$ , where  $F_y$  is specified in the material. One can vary  $S_z$ ,  $S_y$ , or  $F_y$  to obtain required yielding moments.
2. This section uses shear areas,  $Av_y$  and  $Av_z$ , and yielding stresses of the corresponding materials,  $F_{y\_Mat\_Vy}$ ,  $F_{y\_Mat\_Vz}$ , to consider shear deformations of the element, and to calculate the yielding shear forces.

In this section,  $V_{Y\_yield} = Av_y \times F_{y\_Mat\_Vy}$ ;  $V_{Z\_yield} = Av_z \times F_{y\_Mat\_Vz}$ . One can vary  $Av_y$ ,  $Av_z$ , or  $F_y$  to obtain required yielding shears.

### EXAMPLE:

```
Section BCSection02 BC 1 1 2 2 1 0 9784 4E8 1.8E7 4.6E5
1.6E6 1.8E5 4464 4643
```

//This command defines a section named "BC" which uses material "1" to control moments/axial force/torsion and uses material "2" to control shears.

```
Section BCSection02 FSEC1 m1 m1 m1 m1 m1 0 8e4 1.1E9 2.7e8
7.324e8 5.333E6 5.333E6 66666.67 66666.67
```

//This command defines a section named "FSEC1" with identical material "for all degrees of freedom.

//Define materials

```
Material Hardening 1 200 0.24 -.24 0.05 5 0.05 5 5.024 1.0
```

```
Material Degrading 2 200 0.24 -.24 0.001 0.007 1 1 1 2.0 1 1 1 0.3
```

```
Material Fracture m1 200 0.24 0.3 0.8 1 1 7.8 1.25 7.9 0.2 10.4 0.2
```

## H08. Element Section Definition – BCSection03 Command

This section command is for composing the BeamColumn element (G02). Users can specify various materials for different degrees of freedom on both nodes of the element. Therefore, the BeamColumn element can perform various nonlinearities for different degrees of freedom. For example, this section could be used to model the nonlinear responses of T beam.

### Command:

Section BCSection03 Tag? Mat\_Mzi? Mat\_Mzj? Mat\_Myi? Mat\_Myj?  
Mat\_Vyi? Mat\_Vyj? Mat\_Vzi? Mat\_Vzj? Mat\_P\_T? Inter\_Surf? Area?  
Iz? Iy? J? Sz? Sy? Avy? Avz?

PARAMETER	DESCRIPTION
Section	Control Word.
BCSection03	Define BCSection03.
Tag	Unique tag of this section.
Mat_Mzi	Tag of the material to control <b>moment about local Z axis corresponding to node i.</b>
Mat_Mzj	Tag of the material to control <b>moment about local Z axis corresponding to node j.</b>
Mat_Myi	Tag of the material to control <b>moment about local Y axis corresponding to node i.</b>
Mat_Myj	Tag of the material to control <b>moment about local Y axis corresponding to node j.</b>
Mat_Vyi	Tag of the material to control <b>shear in local Y axis corresponding to node i.</b>
Mat_Vyj	Tag of the material to control <b>shear in local Y axis corresponding to node j.</b>
Mat_Vzi	Tag of the material to control <b>shear in local Z axis corresponding to node i.</b>

Mat_Vzj	Tag of the material to control <b>shear in local Z axis corresponding to node j.</b>
Mat_P_T	Tag of the material to control <b>torsion and axial force.</b> The torsion and axial force of BCSection03 will keep in linear elastic. Only the Young's modulus and the Poisson ratio are adopted even if this material is nonlinear.
Inter_Surf	Tag of the section's interaction surface. (Input 0 if no interaction surface is considered for this section.)
Area	<b>Effective</b> average cross sectional area.
Iz	<b>Effective</b> moment of Inertia about local Z axis.
Iy	<b>Effective</b> moment of Inertia about local Y axis.
J	<b>Effective</b> torsional constant.
Sz	<b>Effective</b> section modulus about local Z axis.
Sy	<b>Effective</b> section modulus about local Y axis.
Avy	<b>Effective</b> shear area in local Y axis.
Avz	<b>Effective</b> shear area in local Z axis.

**NOTE:**

$$\begin{aligned}
M_{Zi\_yield} &= S_z \times F_{y\_Mat\_Mzi}, & M_{Zj\_yield} &= S_z \times F_{y\_Mat\_Mzj} \\
M_{Yi\_yield} &= S_y \times F_{y\_Mat\_Myi}, & M_{Yj\_yield} &= S_y \times F_{y\_Mat\_Myj} \\
V_{Yi\_yield} &= A_{vy} \times F_{y\_Mat\_Vyi}, & V_{Yj\_yield} &= A_{vy} \times F_{y\_Mat\_Vyj} \\
V_{Zi\_yield} &= A_{vz} \times F_{y\_Mat\_Vzi}, & V_{Zj\_yield} &= A_{vz} \times F_{y\_Mat\_Vzj}
\end{aligned}$$

where  $F_y$  is specified in the material.

**EXAMPLE:**

//Define section:

Section BCSection03 ss F1 F2 F1 F2 S S S S E 0 8192 2.296e+08 1.73493e+07  
343907 1.14824e+06 173493 3200 4333

//Define materials:

Material Bilinear F1 200 0.01 0.25 -0.13 0.3

Material Bilinear F2 200 0.01 0.13 -0.25 0.3

Material Bilinear S 200 0.01 0.07 -0.07 0.3

Material Elastic E 200 0.3

## PART I. Output Setting Command

This part is to control the required output data. All of these commands are optional and have default values.

### I01. Output Interval Setting Command

One can use this command to set the responses output interval between steps. Additionally, this command can set the output formats for responses of nodes/elements.

#### Command:

Output OutFlag NodeIntv? ElemIntv? EvlpIntv? NodeCode? ElemCode?  
EnergyIntv? F\_GpIntv?

PARAMETER	DESCRIPTION
Output	Control Word.
OutFlag	Output intervals setting during analysis process.
NodeIntv	Interval between steps for output the responses of nodes, expressed as a multiple of a single step, must $> 0$ . (ex: "10" means to output data every 10 steps, Default Value = 1)
ElemIntv	Interval between steps for output the responses of elements, expressed as a multiple of a single step, must $> 0$ . (Default Value = 1)
EvlpIntv	Interval between steps for output response envelopes of nodes and elements expressed as a multiple of a single step, must $> 0$ . (Default Value = 0, only output after the final step of each analysis method.)

NodeCode	<p>Code for output response files of nodes.</p> <p>0 = only output file .NodeAbsDisp</p> <p>2 = only output file .NodeDisRecord/.NodeVelRecord/.NodeAccRecord .NodeRelDisp/.NodeRelVel/.NodeRelAcc.</p> <p>1 = 0 + 2</p>
ElemCode	<p>Code for output response files of elements.</p> <p>0 = only output file .Element</p> <p>2 = only output file .ElemRecord</p> <p>1 = 0 + 2</p>
EnergyIntv	<p>Interval between steps for output the energy distribution, expressed as a multiple of a single step, must &gt; 0. (Default Value = ElemIntv)</p>
F_GpIntv	<p>Interval between steps for output the group section force, expressed as a multiple of a single step, must &gt; 0. (Default Value = ElemIntv)</p>

### NOTE:

1. For large model and analysis methods with thousands of analysis steps, the size of output files would be massive while the output intervals are too small. Increasing the output intervals can save the analysis time and the disk storage space significantly.

### EXAMPLE:

Output Outflag 1 5 0 2 2 5 5

//This command sets to output for nodes every step, output for element every 5 steps, output for envelopes at the last step, output for energy and force group every 5 steps.

Output Outflag 200 10 10 1 1 200 10

//This command sets output for nodes every 200 steps, output for element every 10 steps, output for envelopes every 10 steps, output for energy group every 200 steps, and output for force group every 10 steps.

## I02. Output Nodal Absolute Responses

This command permits users to specify the nodes to output their absolute responses.

### Command:

```
Output  NodeAbs  NodeTag1?  NodeTag2? ... NodeTagN?
```

PARAMETER	DESCRIPTION
Output	Control Word.
NodeAbs	Setting absolute response output for nodes.
NodeTag1	The tag of the 1st output node.
NodeTag2	The tag of the 2nd output node.
NodeTagN	The tag of the Nth output node.

### NOTE:

1. If no output nodes are specified or this command does not exist, responses of all nodes would be output. So only specify required nodes to output will save the analysis time and the disk storage space.

### EXAMPLE:

```
Output  NodeAbs      N2  N15
```

//This command sets only responses of N2 and N15 to output.

### I03. Output Nodal Relative Responses

One can use this command to output the required relative responses between two specified nodes at the specified DOF.

#### Command:

```
Output NodeRel DOF? NodeTag1_1? NodeTag1_2? ... NodeTagN_1? NodeTagN_2?
```

PARAMETER	DESCRIPTION
Output	Control Word.
NodeRel	Setting relative response output for nodes.
DOF	The DOF to be output relative responses (in global coordinate).
NodeTag1_1	The first nodal tag of relative response 1.
NodeTag1_2	The second nodal tag of relative response 1.
NodeTagN_1	The first nodal tag of relative response N.
NodeTagN_2	The second nodal tag of relative response N.

#### NOTE:

1. The response value output is (NodeN\_1-NodeN\_2).
2. In a single command, only one specified DOF is available. The user can specify this command many times if needed.

#### EXAMPLE:

```
output NodeRel UX N1 N2 7 15
```

//This command outputs UX dir. relative responses of N1-N2 and 7-15.

```
output NodeRel RZ 7 15 9 23
```

//This command outputs RZ dir. relative responses of N1-N2, 7-15 and 9-23.



## I04. Output element responses in a specified file

One can use this command to output responses of the required elements in a specified text file. This command can be used many times to create many output files as required. In each single command, the number of output elements is unlimited.

### Command:

Output ElementFile FileName? ElemTag1? ElemTag2? ..... ElemTagN?

PARAMETER	DESCRIPTION
Output	Control Word.
ElementFile	Setting element output in a specified file.
FileName	File name of the specified output file.
ElemTag1	The tag of the 1st output element in the specified file.
ElemTag2	The tag of the 2nd output element in the specified file.
ElemTagN	The tag of the Nth output element in the specified file.

### NOTE:

1. For all the specified elements in this output command, the output codes in the elements' commands must be turned on. See PART G for element definition command.
2. During analysis, if there is any file with identical file name to that in this command, the original file will be deleted and replaced by the new output file.

### EXAMPLE:

Output ElementFile File1.txt e1 e2 d2

Output ElementFile elemOut 2Fcol2

Output ElementFile elemOut 1Fcol3 3Fcol2 4Fcol1

//In the file "File1.txt", the output elements are e1, e2 and d2.

//In the file "elemOut", the output elements are 2Fcol2, 1Fcol3, 3Fcol2 and 4Fcol1.

## I05. Output nodal responses in a specified file

One can use this command to output responses of the required nodes in a specified text file. This command can be used many times to create many output files as required. In each single command, the number of output nodes is unlimited.

### Command:

Output NodeFile FileName? NodeTag1? NodeTag2? ..... NodeTagN?

PARAMETER	DESCRIPTION
Output	Control Word.
NodeFile	Setting nodal output in a specified file.
FileName	File name of the specified output file.
NodeTag1	The tag of the 1st output node in the specified file.
NodeTag2	The tag of the 2nd output node in the specified file.
NodeTagN	The tag of the Nth output node in the specified file.

### NOTE:

1. During analysis, if there is any file with identical file name to that in this command, the original file will be deleted and replaced by the new output file.

### EXAMPLE:

Output NodeFile Nfile.txt N1 N2 N6

//In the file “Nfile.txt”, the output nodes are N1, N2, N6.

## **PART J. End Input Command**

This command stops processing data from the input file. Any commands behind this command would not be handled.

### **J01. Termination of Input File**

#### **Command:**

Stop

PARAMETER	DESCRIPTION
Stop	Control Word.

#### **NOTE:**

1. This command is necessary at the end of the input file.

#### **EXAMPLE:**

STOP

## References

1. Tsai, K.C., Lin, B.Z., “Development of an Object-Oriented Nonlinear Static and Dynamic 3D Structural Analysis Program”, Center for Earthquake Engineering Research, National Taiwan University, Report No. CEER/R92-04, 2003.
2. Thomas J.R. Hughes, Karl S. Pister and Robert L. Taylor, “Implicit-Explicit Finite Elements in Nonlinear Transient Analysis”, *Computer Methods in Applied Mechanics and Engineering*, 159~182, 1979.
3. Masayoshi Nakashima, Takashi Kaminosono, Masatoshi Ishida, and Kazuhiro Ando, “Integration Techniques for Substructure Pseudo Dynamic Test”, Fourth U.S. National Conference on Earthquake Engineering, Vol.2, May, 1990.
4. Yang, Y.B. and Kuo, S.R. (1994). *Theory & Analysis of Nonlinear Framed Structures*, Prentice Hall.
5. Lin, B.Z., Chuang, M.C. and Tsai, K.C., “Object-oriented Development and Application of a Nonlinear Structural Analysis Framework.” *Advances in Engineering Software*, 40, pp. 66-82, 2009.