NARLabs
國家實驗研究院

Introduction of the PISA3D and its applications

Ming-Chieh Chuang (莊明介), Ph.D.
Assistant Researcher
(mcchuang@ncree.narl.org.tw)
National Center for Research on Earthquake Engineering (NCREE), Taiwan

www.narlabs.org.tw

April 9, 2018

Outline

NARLabs

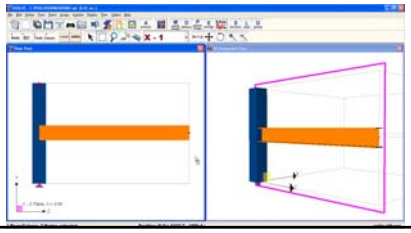
- Introduction of PISA3D
- Demonstration of PISA3D

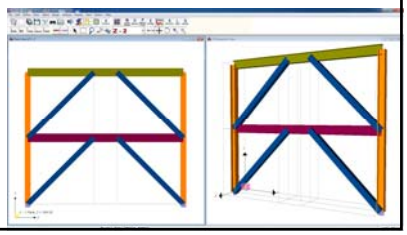
Model 1-1 and Model 1-2

Simulations of the Subassemblage Testing of Beam-to-Column Connection

Model 2

Simulations of the 2-story Eccentrically Braced Frame (EBF)





NARLabs

PISA3D基本功能介紹
Introduction of PISA3D

NARLabs

Finite Element Analysis Tools

- Commercial packages
  - User friendly
  - Less flexibility
  - Examples:
    - ABAQUS / ANSYS
      - General finite element analysis
    - SAP / ETABS
      - For civil / structural engineers
- Academic programs
  - Necessary for research, plentiful libraries
  - Poor in graphical user interface
  - Examples:
    - DRAIN family
      (DRAIN-2D; DRAIN-2DX; DRAIN-3DX, by UC Berkeley)
      (DRAIN2D+ & VIEW2D by NTU)
    - OpenSees by PEER

Features of PISA3D

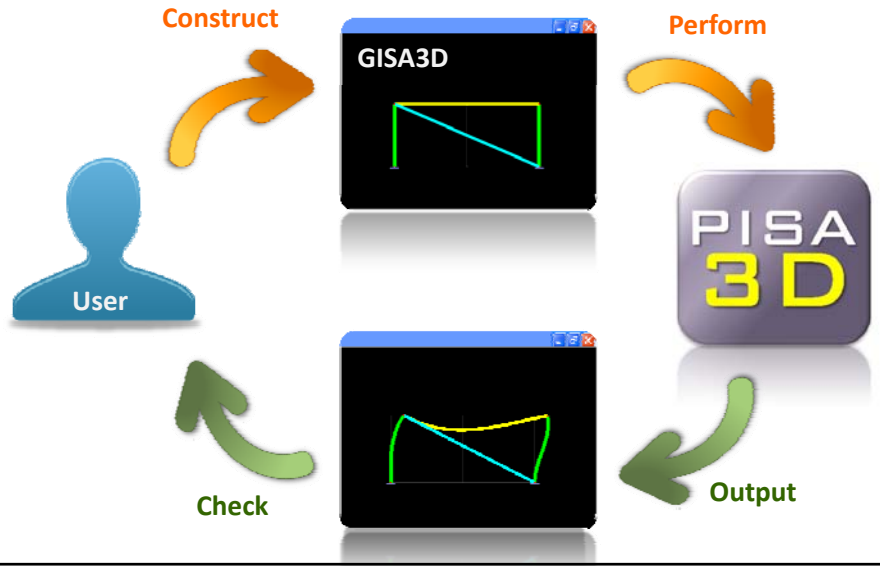
Platform of Inelastic Structural Analysis for 3D Systems

- Static/dynamic analysis
- Linear/nonlinear analysis
- Friendly user interface – GISA3D
- Plentiful material/element models to simulate Steel/RC/Composite structures
- High efficiency and performance of analyses
- Object-oriented concept for engineers and developers

[Lin et al., 2009]

Lin, B. Z., Chuang, M. C., & Tsai, K. C. (2009). Object-oriented development and application of a nonlinear structural analysis framework. *Advances in engineering software*, 40(1), 66-82.

PISA3D with GUI (GISA3D)



The Milestones of PISA3D

NARLabs

PISA3D (VISA3D)

Developed by Prof. Keh-Chyuan Tsai' s research team

GISA3D

Awarded by 國家實驗研究院

2002 *2005* *2006* *2007* *2008* *2009* *2012*

Certified by CUPSEA

Awarded by 國家實驗研究院 & ***E-DEFENSE***

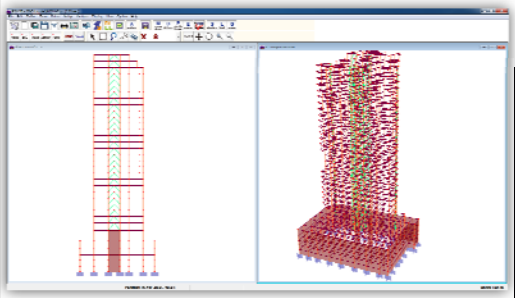
Awarded by 國家實驗研究院 & ***E-DEFENSE***

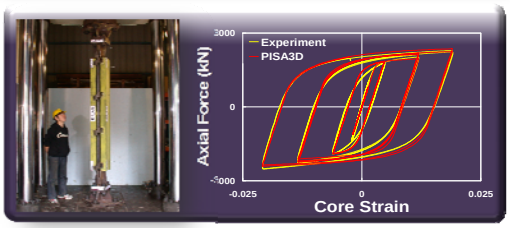
Awarded by ***E-DEFENSE***

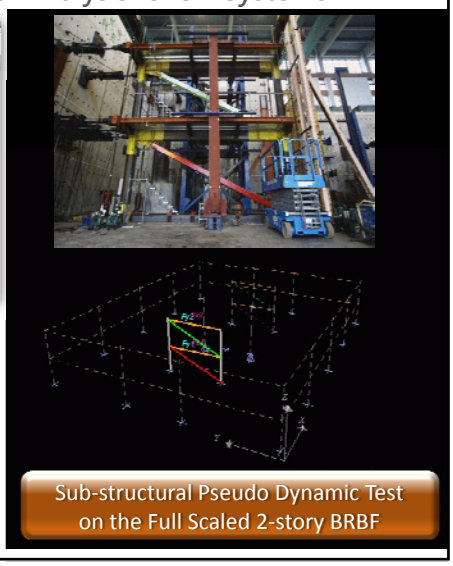
PISA3D

NARLabs

Platform of Inelastic Structural Analysis for 3D Systems

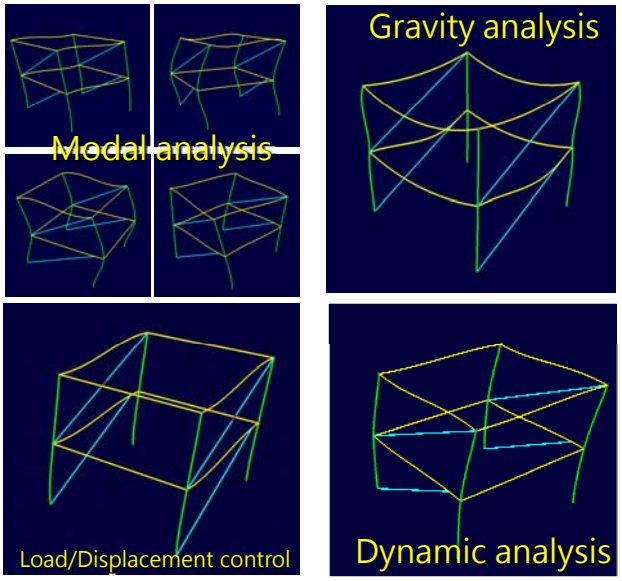






Sub-structural Pseudo Dynamic Test on the Full Scaled 2-story BRBF

PISA3D Analysis Methods



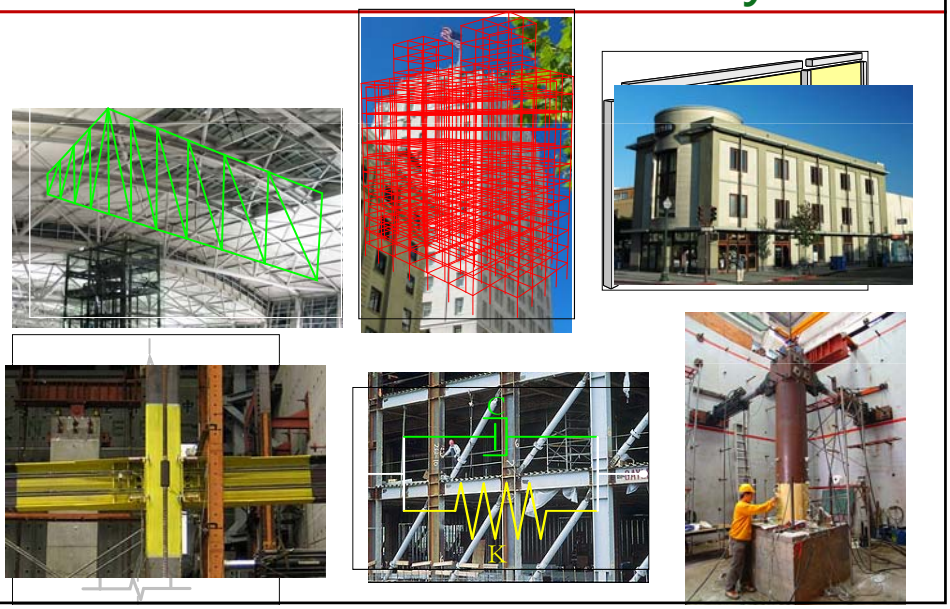
Analysis Method Library of PISA3D

- **Nonlinear Static Analysis**
  One Step, Load Control, Displacement Control
- **Modal Analysis**
  Mode Shapes and Periods
- **Nonlinear Dynamic Analysis**
  Time-History of Ground Acceleration
  (Newmark, Operator-Splitting)
- **Combined Analysis**
  Static, dynamic, eigen analyses can be combined in any preferred order in a single run.

  e.g., ***{ Disp. Ctrl. + Disp. Ctrl. + … }***
  { Dynamic + Dynamic + … + Modal + … }
  { Dynamic + Disp. Ctrl. + … }

NARLabs

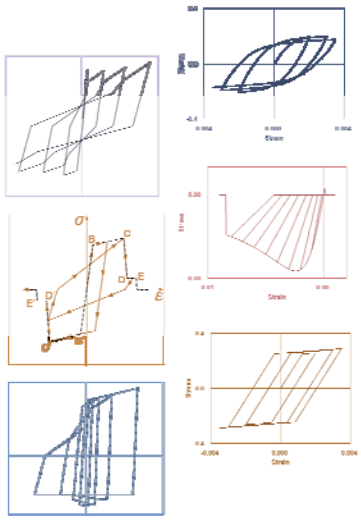
Nonlinear Element Library



NARLabs

Nonlinear Element Library

- PISA3D Element
  - Truss
  - BeamColumn
  - FiberBeamColumn
  - Panel
  - Joint
  - 6DJoint
  - Damper
  - BilinearDamper
- PISA3D Material
  - Elastic
  - BiIlnear
  - Bilinear02
  - BilinearElastic
  - Hardening
  - Degrading
  - Fracture
  - Buckle
  - Concrete
  - TensionOnlyBilinear
  - OpenSees Material
    - Bond_SP01
    - BoucWenMaterial
    - Concrete01
    - Concrete04
    - HysteresticMaterial
    - Steel01



More than 120 different features of elements are available in PISA3D.

Certification

結構應用電子計算機程式
認 證 證 書

證書字號：結軟登字第015-3號
程式名稱：PISA3D (Platform of Inelastic Structu Analysis for 3D Systems)
程式所有人：財團法人國家實驗研究院
代 理 人 ：（空白）

PISA3D (Platform of Inelastic Structural Analysis for 3D Systems) 結構分析程式業經本公會「電子計算機程式認證小組」依據本公會程式認證審核程序之規定審查通過。本公會認可該電腦分析程式除側推分析部分外，自中華民國107年4月12日起至中華民國109年4月11日止適合於中華民國境內使用，惟使用者據以完成之分析及設計結果仍應自行研判其正確性並負設計責任。

中華民國結構工程技師公會全國聯合會
理事長 張 荻 薇

中 華 民 國 107 年 3 月 20 日

PISA3D has been certified by Chinese Union of Professional Structural Engineer Associations.

13

PISA3D軟體操作說明
Demonstration of PISA3D

NARLabs

Something You Have to Know

- URL for Download :

  http://w3.ncree.org/download/PISA3D/PISA3D SE R3.2(2011-09-05).exe

  http://tinyurl.com/83losom

- Download free without registration (Node limit is 200)

- You can find the user manual and examples in

  C:\Program Files\PISA3D\PISA3D SE\PISA3D R3.2

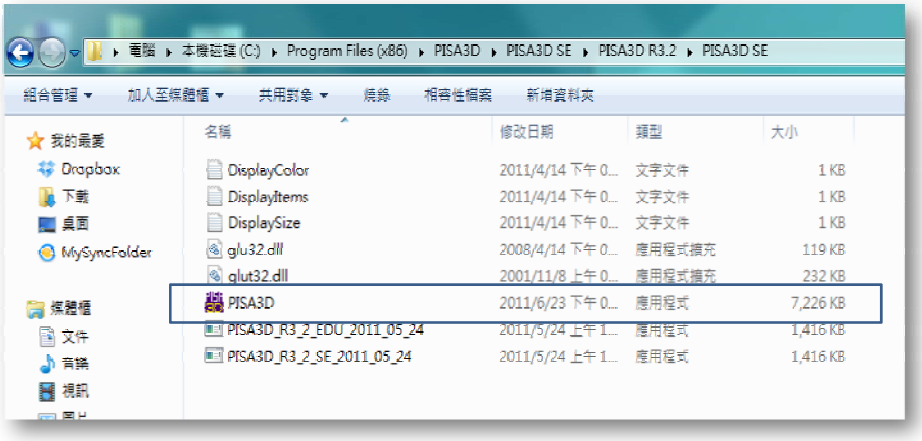
- You must avoid saving the project by using a long path.

- You must define the Tags of materials, sections...etc. without blank space.

- If the program can not work on your PC/Laptop, try to update the drivers of the graphics card.

NARLabs

Update your PISA3D



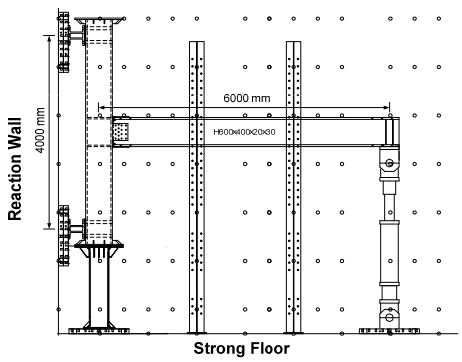
Default path:

C:\Program Files\PISA3D\PISA3D SE\PISA3D R3.2\PISA3D SE

Model 1

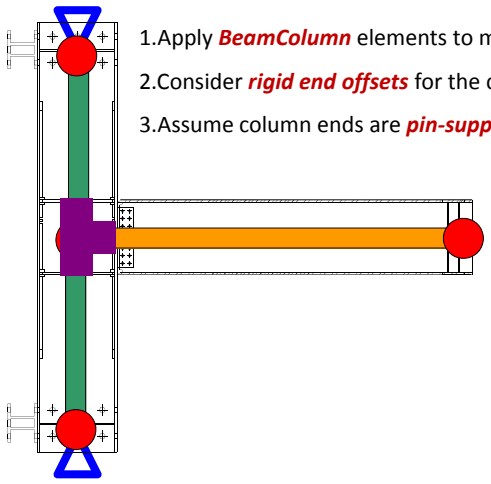
Simulations of the Subassemblage Testing
of Beam-to-Column Connection

The subassemblage Testing of Beam-to-column Connection



Member	Section	Length (mm)	F_y (kN/mm^2)	Self Weight (per unit of length) (kN/mm)
Beam	H600x400x20x30	6,000	0.42	2.67e-3
Column	Box550x550x30	4,000	0.42	4.79e-3

Model 1-1
(W/O P.Z. deformation effect)



1.Apply ***BeamColumn*** elements to model the columns and beams.

2.Consider ***rigid end offsets*** for the column and the beam members.

3.Assume column ends are ***pin-supported***.

Demonstration of Model 1-1

<u>The skills listed below will be illustrated in this example.</u>

1.Begin a Project.

2.Draw BeamColumn Elements.

3.Select Any Objects(Elements/Nodes) of Your Model.

4.Define and Assign Material/Section/Rigid End Offsets.

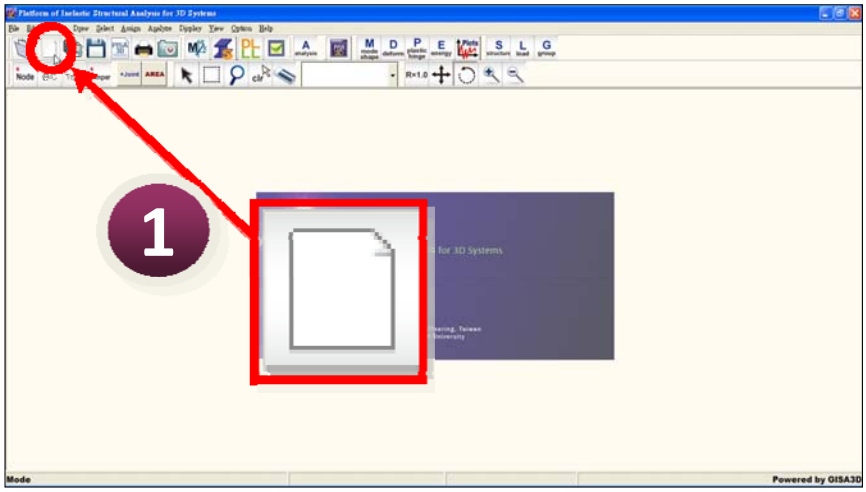
5.Define Static Load Case and Assign Element/Nodal Loads.

6.Perform the Static Analysis and the Disp. Control Analysis.

7.Extract the Results of Analyses.

Begin a model

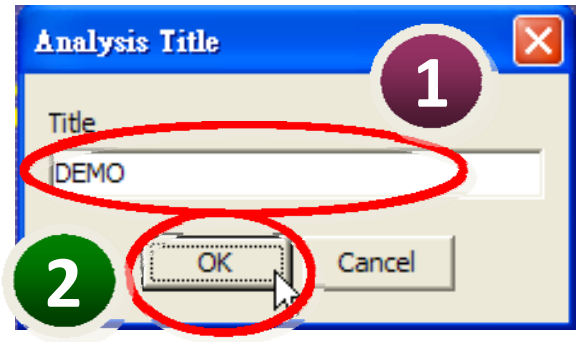
1.Click the button to create a new project.



Input the Analysis Title

1.It is optional to input the analysis title.

2.Click the ***OK*** button.



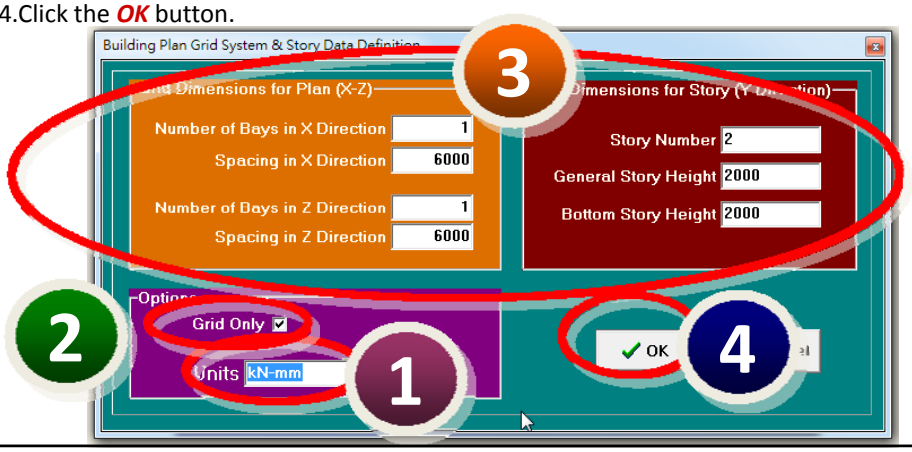
Create the Basic Grid System

1.Specify the units: ***kN-mm***.

2.Select the ***Grid Only*** mode to initialize your model.

3.Input the details about your grid system.

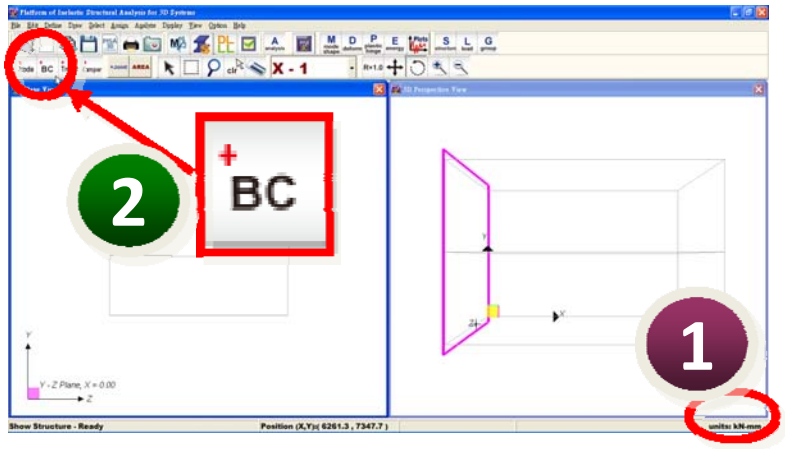
4.Click the ***OK*** button.



Activate Draw Mode for Drawing BeamColumn Element

1.Here you can find and check the units.

2.Click the button to start to draw a BeamColumn element.

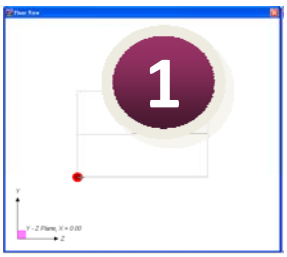


Draw a BeamColumn Element

1.When the mouse position is close to the grid intersection, red point means ***snap to grid function*** is available.

2.Click the left mouse button and drag mouse to define a BeamColumn element.

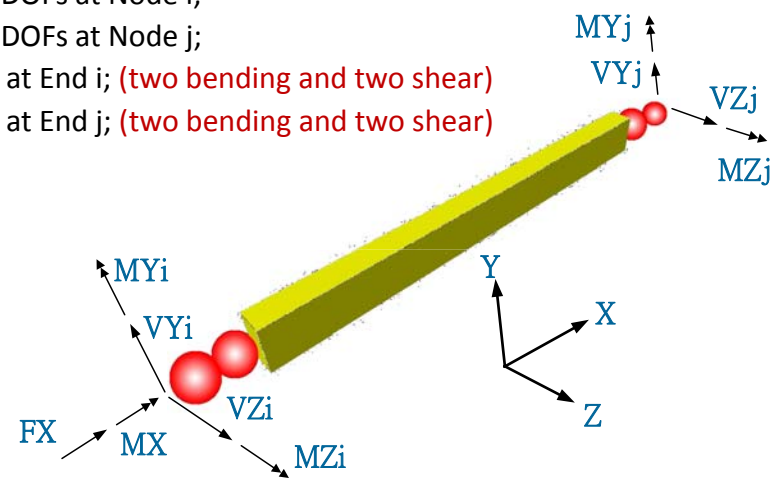
3.Repeat step 2, construct the column and the beam.



BeamColumn Element

- The zero-length plastic hinge (lumped plasticity)

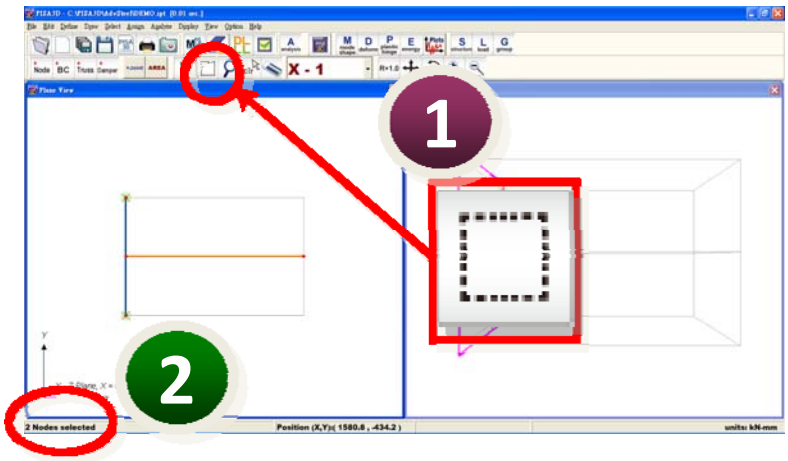
(1) 6 global DOFs at Node i;
(2) 6 global DOFs at Node j;
(3) 4 hinges at End i; (two bending and two shear)
(4) 4 hinges at End j; (two bending and two shear)



Rubber Band Window for Select Nodes/Elements

1.Click the button.

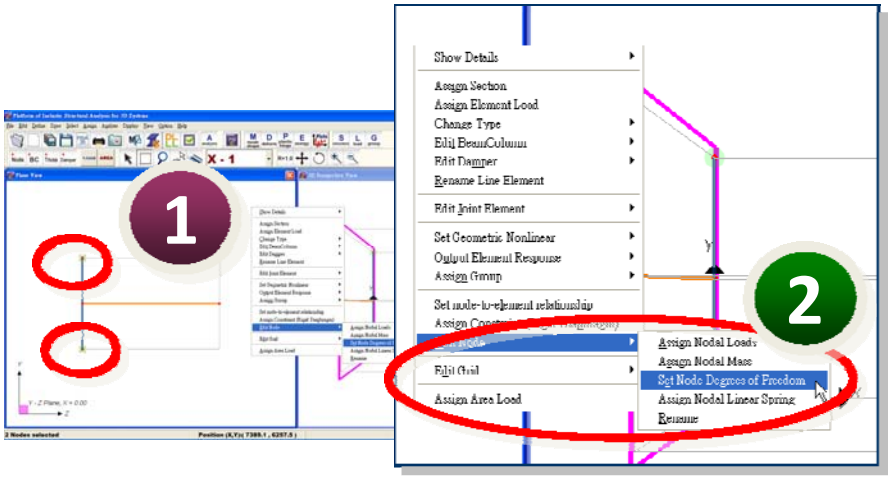
2.Information of selected objects are listed here.



Edit the Node Degrees of Freedom

1.Select the **two nodes**, then click right mouse button to show the popup menu.

2.Select and Click the ***Edit Node>Set Node Degrees of Freedom***.

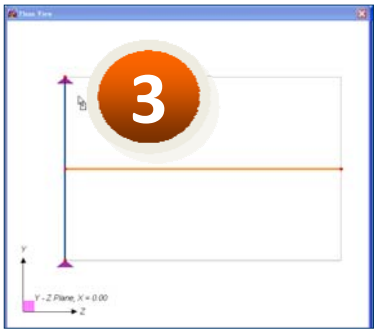


Edit the Node Degrees of Freedom

1.Click ***Pin*** button using the fast set function.

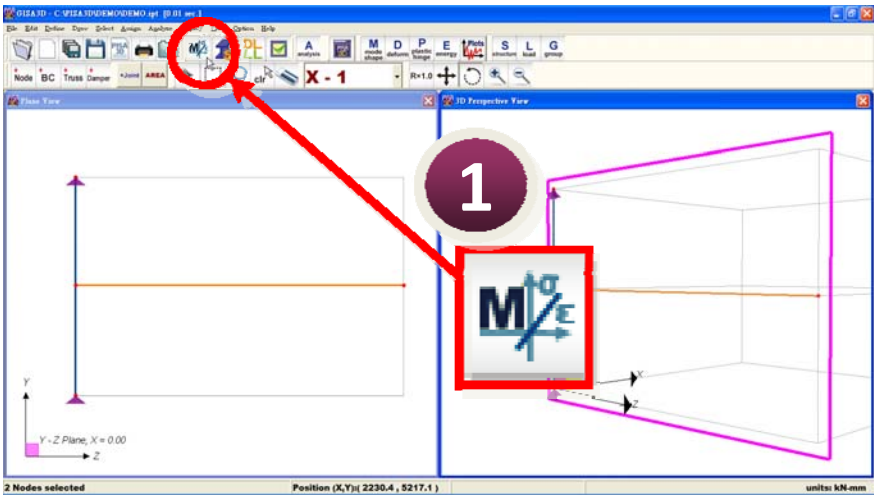
2.Click the ***OK*** button.

3.Immediate representation of the specified DOF/Boundary Condition.



Define the New Material Properties

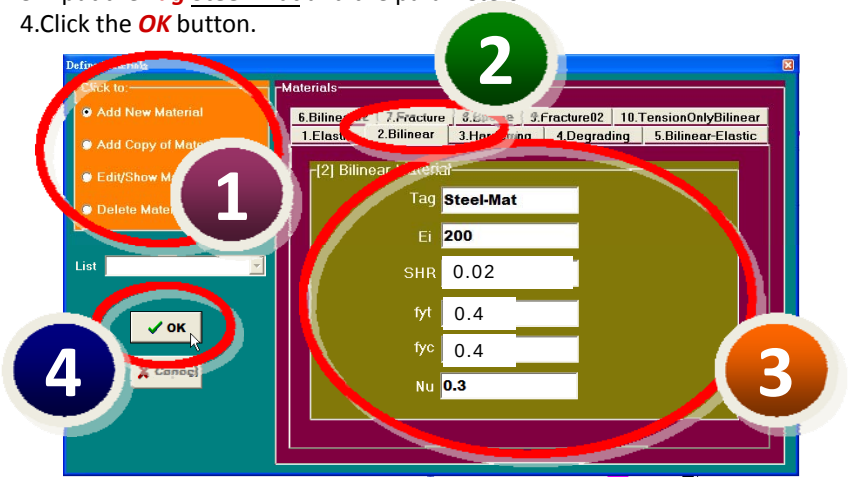
1.Click the button to activate the form of define materials.



NARLabs

Define the New Material Properties

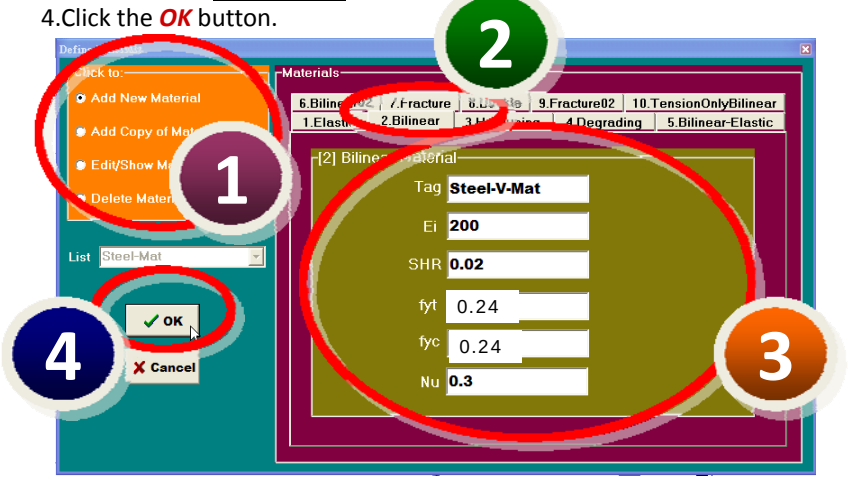
1.Make sure the selected option is ***Add New Material***.
2.Select the ***Bilinear*** page.
3.Input the ***Tag*** **Steel-Mat** and the parameters.
4.Click the ***OK*** button.



NARLabs

Define the New Material Properties

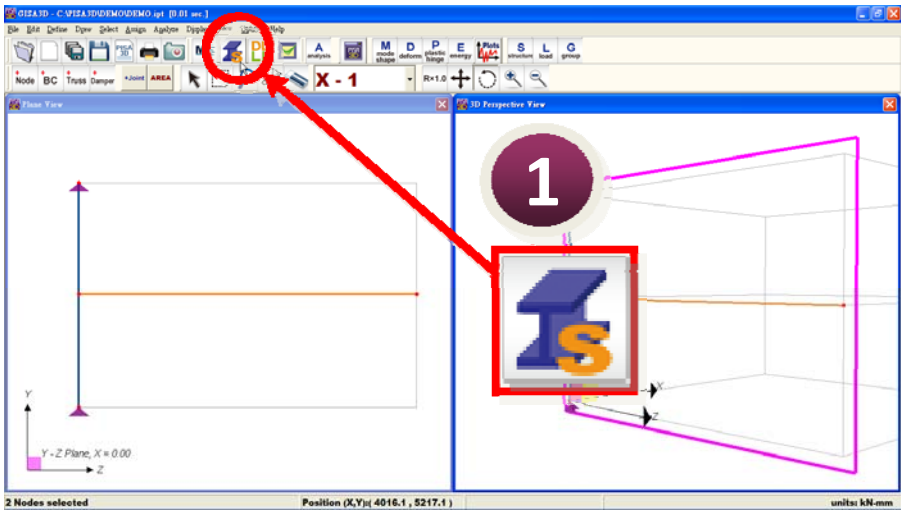
1.Make sure the selected option is ***Add New Material***.
2.Select the ***Bilinear*** page.
3.Input the ***Tag*** **Steel-V-Mat** and the parameters.
4.Click the ***OK*** button.



NARLabs

Define the Sections for BeamColumn Elements

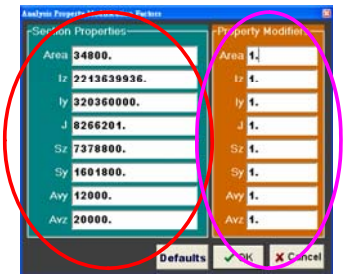
1.Click the button to activate the form for define sections.

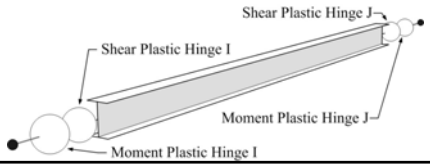


NARLabs

BeamColumn Element

Command: BeamColumn *(Manual: G02)*





$$M_y = S \times f_y$$

$$V_y = A_v \times f_y$$

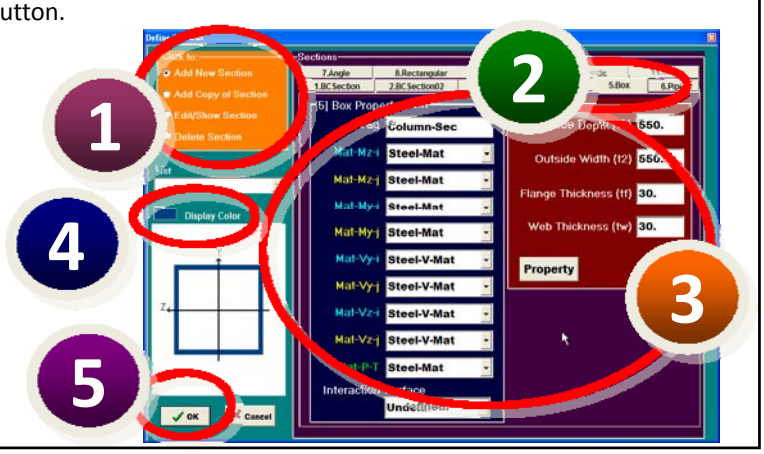
Element *Section* *Material*

(Hinge model)

NARLabs

Define the Sections for BeamColumn Elements

1.Make sure the selected option is ***Add New Section***.
2.Select the ***Box*** page.
3.Input the parameters.
4.Click the panel to specify a color the you want to display the section.
5.Click the ***OK*** button.



NARLabs

Define the Sections for BeamColumn Elements

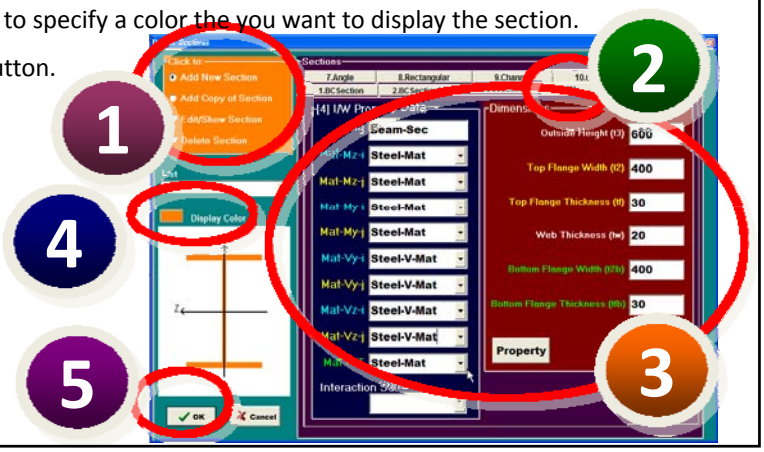
1.Make sure the selected option is ***Add New Section***.

2.Select the ***I/W*** page.

3.Input the parameters.

4.Click the panel to specify a color the you want to display the section.

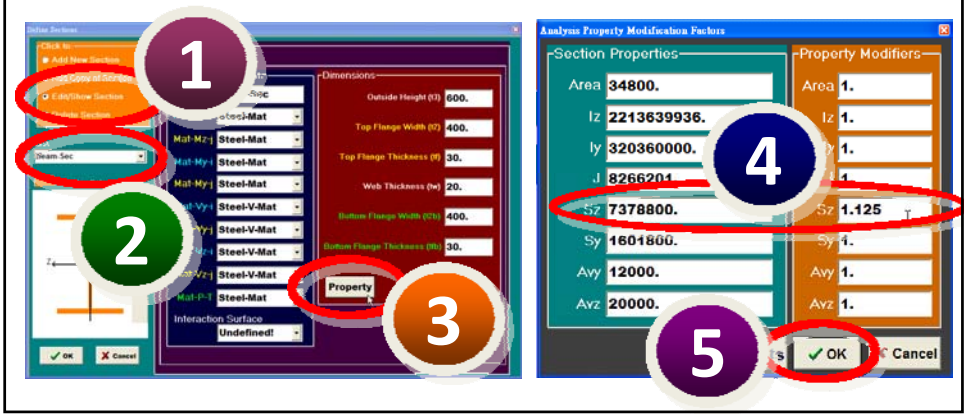
5.Click the ***OK*** button.



NARLabs

Edit the Section Properties

1.Make sure the selected option is ***Edit/Show Section***.
2.Select the Section named Beam-Sec from ***drop-down list***.
3.Click the ***Property*** button.
4.Input the modified parameters (***Optional***).
5.Click the ***OK*** button.



NARLabs

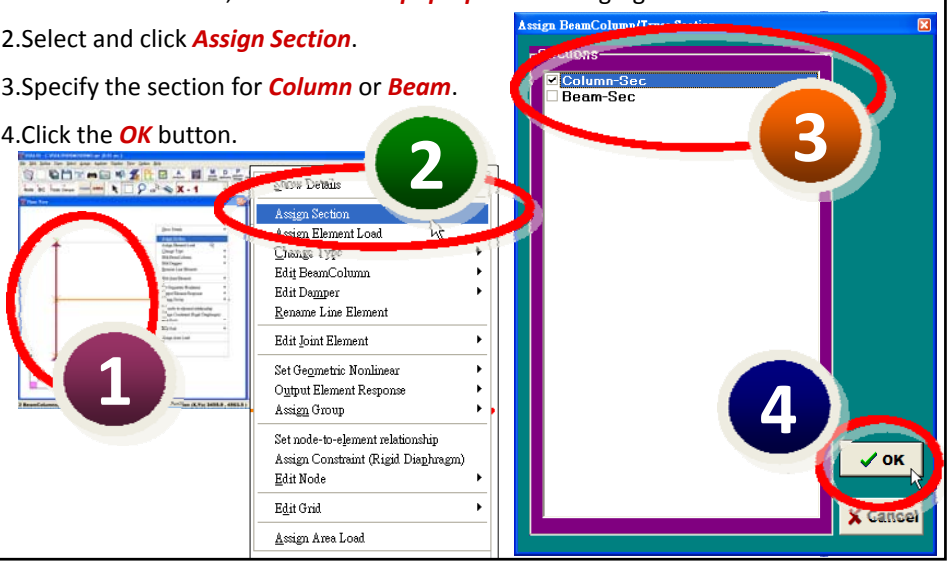
Assign the Section to BeamColumn Element

1.Select the columns, then show the ***pop-up menu*** using right mouse click.

2.Select and click ***Assign Section***.

3.Specify the section for ***Column*** or ***Beam***.

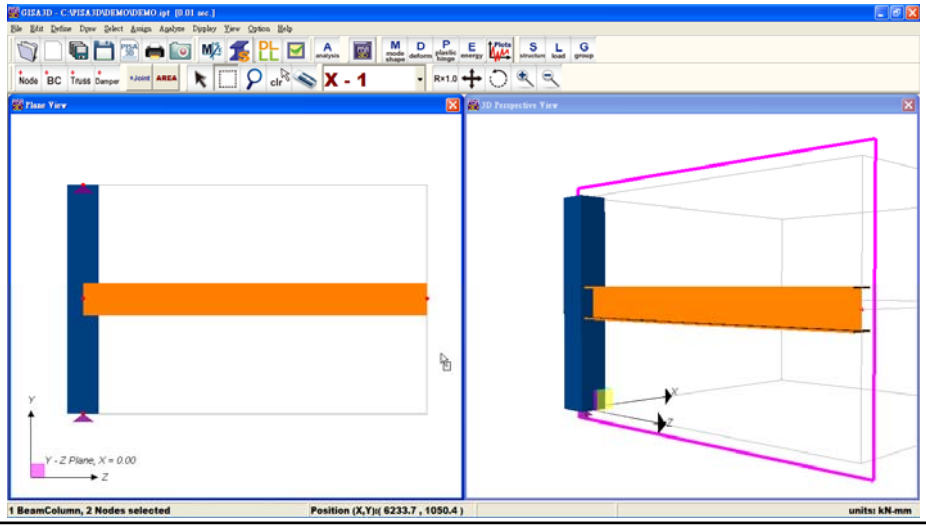
4.Click the ***OK*** button.



NARLabs

Assign the Section to BeamColumn Element

1.Repeat the procedure noted above, but select Beam and assign the section.
2.Extrusions of the sections will be shown on the screen immediately.



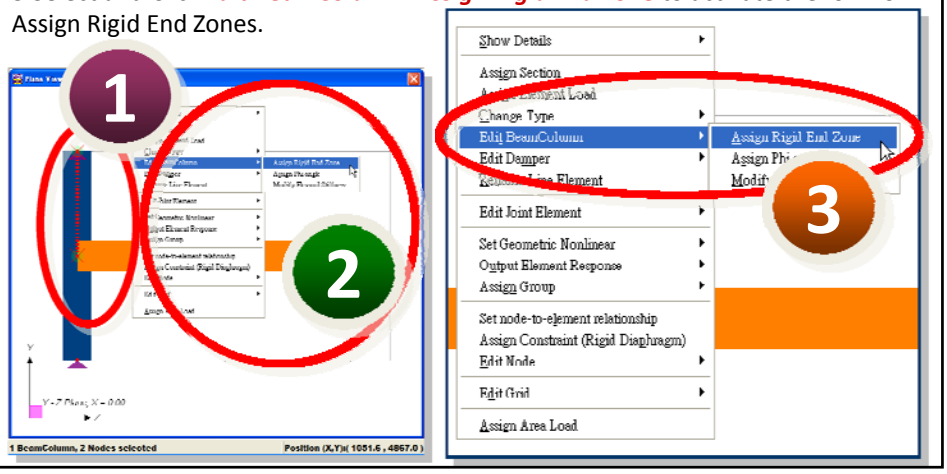
NARLabs

Assign the Rigid End Zone to BeamColumn Element

1.Select Beam or Column.

2.Click right mouse button to activate the ***pop-up menu***.

3.Select and Click ***Edit BeamColumn>Assign Rigid End Zone*** to activate the form of Assign Rigid End Zones.

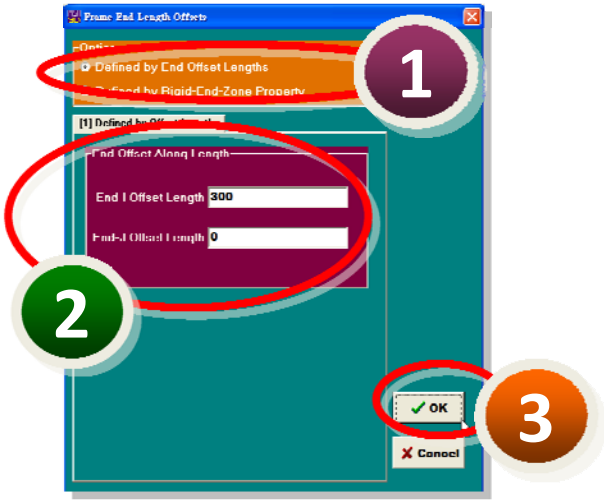


Assign the Rigid End Zone to BeamColumn Element

1.Select the option of ***Defined by End Offset Lengths***.

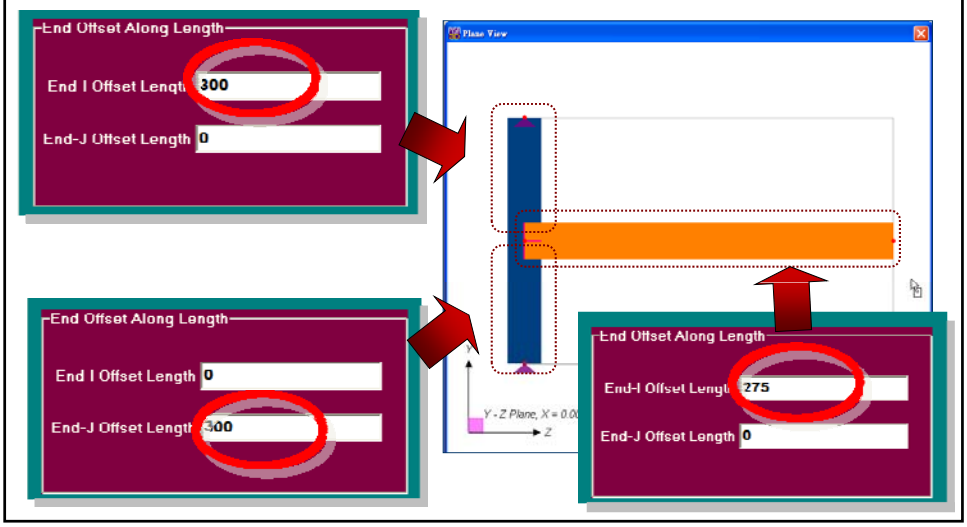
2.Input the End Offset Lengths according to ***BeamColumn Local axis***.

3.Click the ***OK*** button.



Define Offsets of the Rigid End Zone

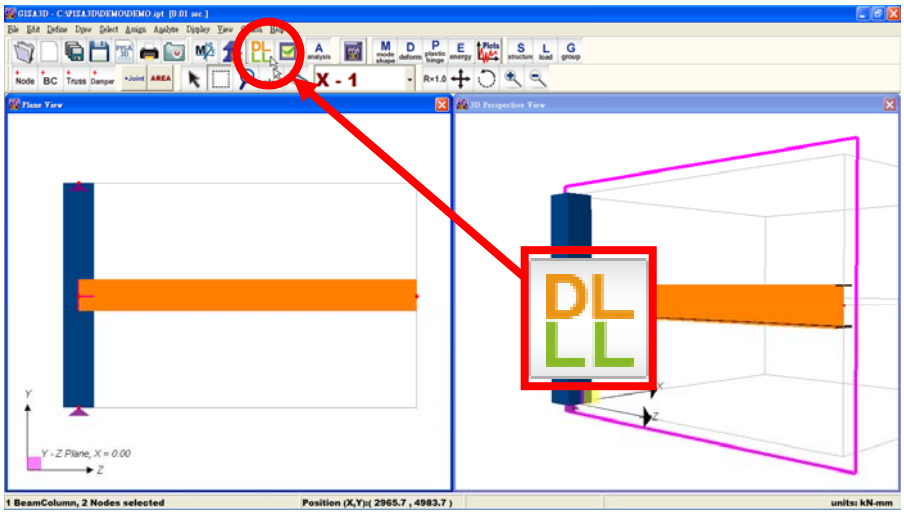
You have to check the ***End I*** and ***End J*** of the BeamColumn Element.



NARLabs

Define the Static Load Cases for Structural Loads

1.Click the button to activate the form of define static load cases.



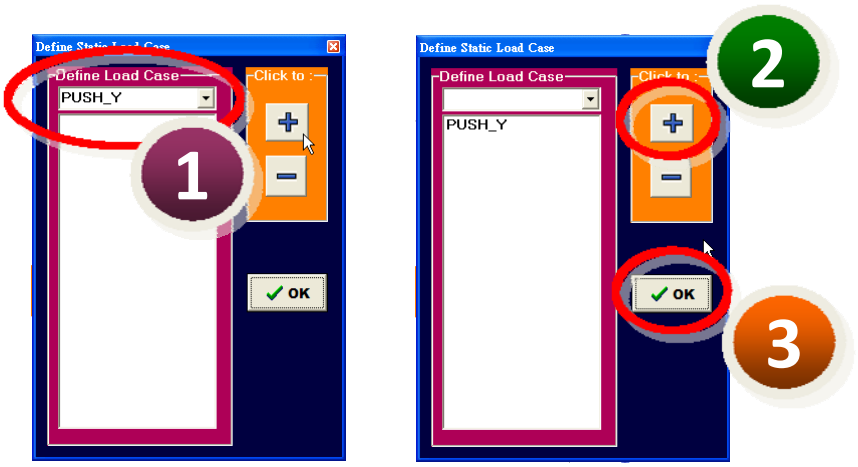
NARLabs

Define the Static Load Cases for Structural Loads

1.Input the tags of the static load cases (***PUSH_Y*** & ***DL***) .

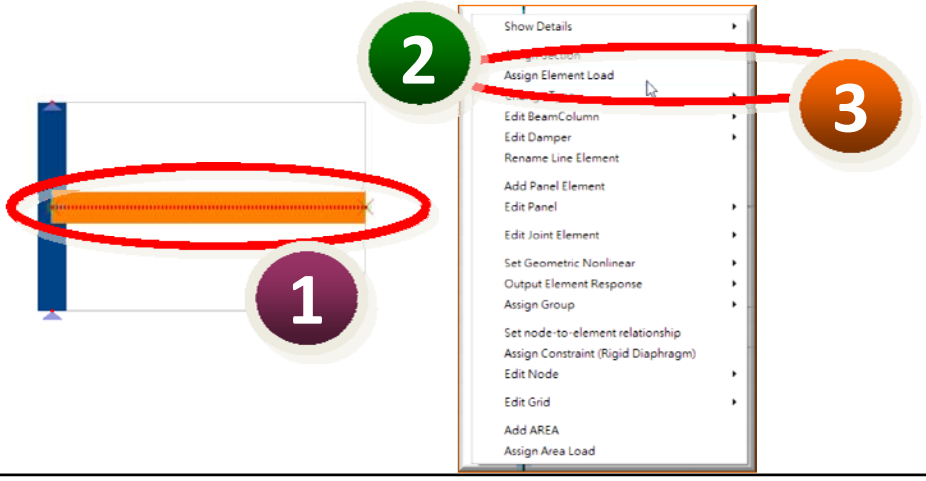
2.Click the ***+*** button.

3.Click the ***OK*** button to exit.



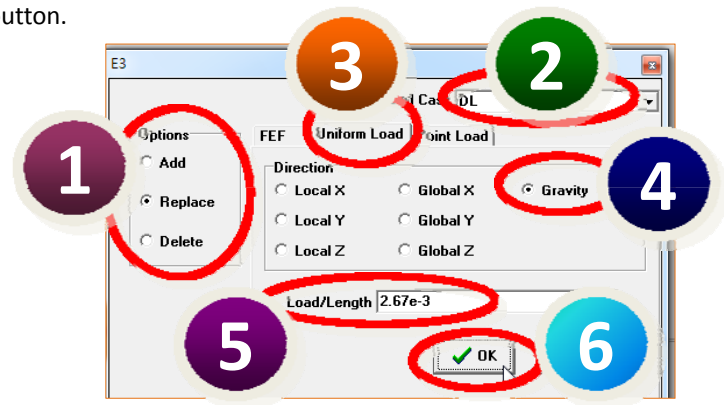
Assign Element Loads

1.Select the ***BeamColumn Element of BEAM***.
2.Click right mouse button to activate the ***pop-up menu***.
3.Select and Click ***Assign Element Load*** to activate the form of Assign Nodal Loads.



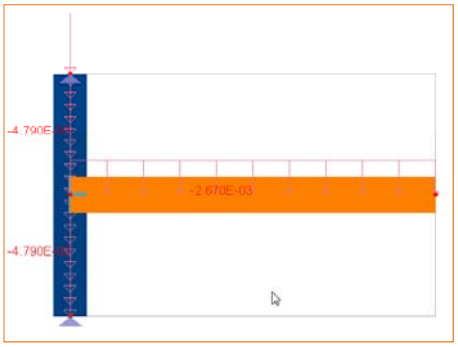
Assign Element Loads

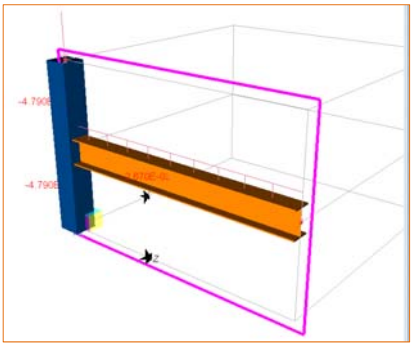
1.Make sure the selected option is ***Replace***.
2.Specify the ***Load Case*** **DL**.
3.Select the ***Uniform Load*** page.
4.Select the Direction ***Gravity***.
5.Input the Load Intensity ***2.67e-3*** kN/mm.
6.Click the ***OK*** button.



Assign Element Loads

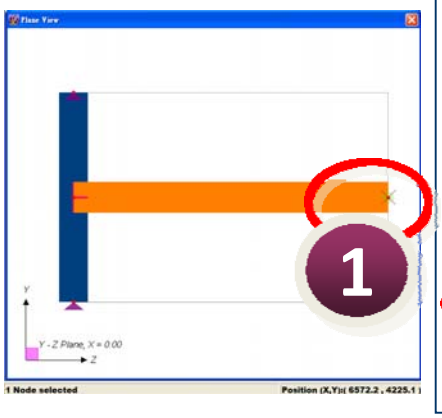
Repeat the operations above, but assign the load intensity ***4.79e-3*** kN/mm to the two BeamColumn elements of column members .

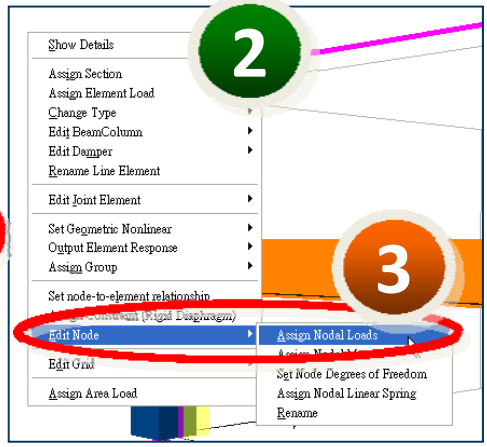




Assign Nodal Loads

1.Select ***Nodes***.
2.Click right mouse button to activate the ***pop-up menu***.
3.Select and Click ***Edit Node>Assign Nodal Loads*** to activate the form of Assign Nodal Loads.





NARLabs

Specify Nodal Loads by using the Static Load Case

1.Select Load Case ***PUSH_Y***.

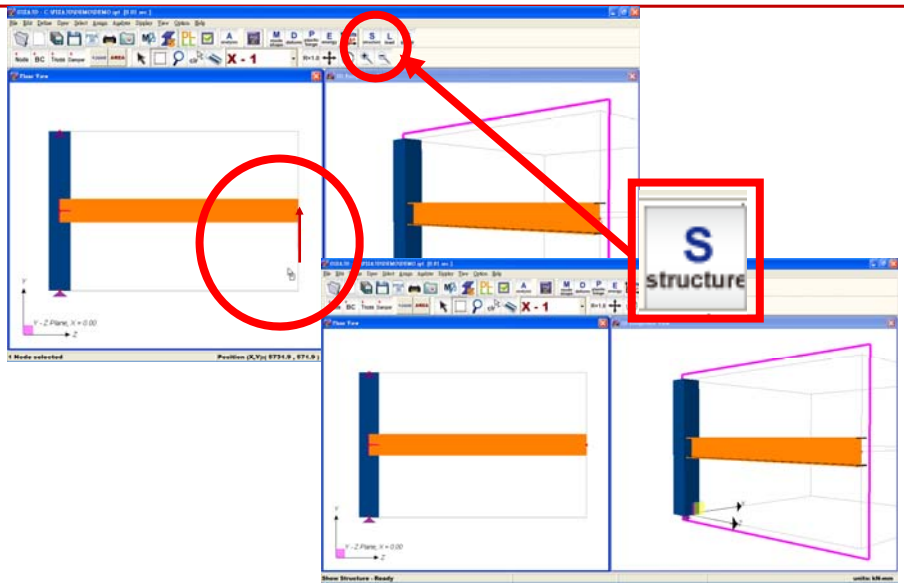
2.Make sure the selected option is ***Replace Existing Loads***.

3.Input the ***Force Load in Y Direction***.

4.Click the ***OK*** button.

NARLabs

Restore the Display of Loads to the Undeformed Geometry

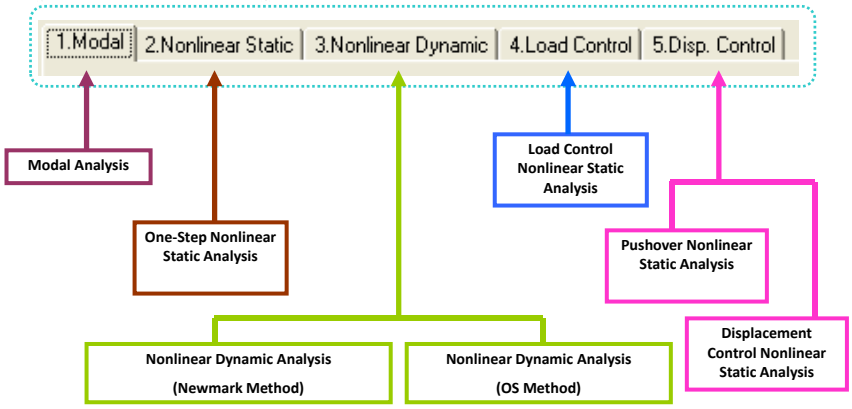


NARLabs

Analysis Setting

***Analyze->Analysis Setting**

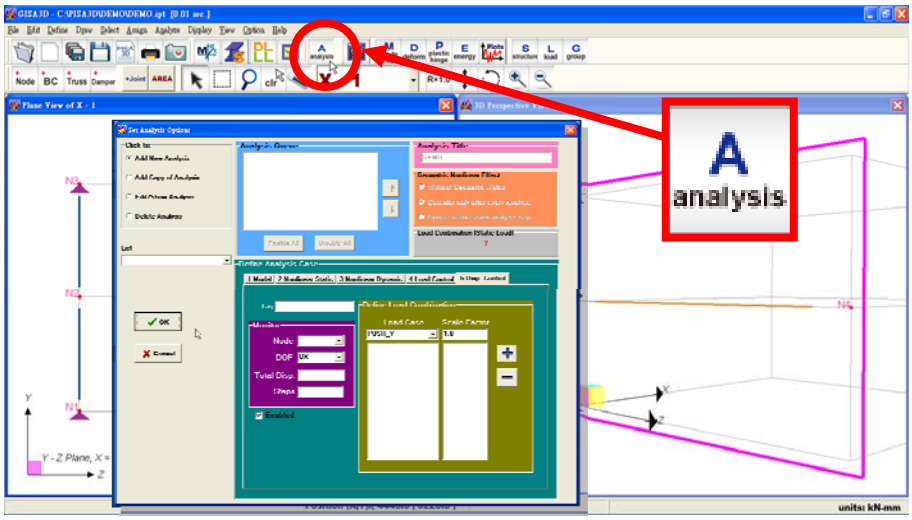
(See PISA3D Manual: PART B.)



NARLabs

Define the Analysis Cases for Analyzing the Model

1.Click the button to activate the form of ***Set Analysis Options***.



NARLabs

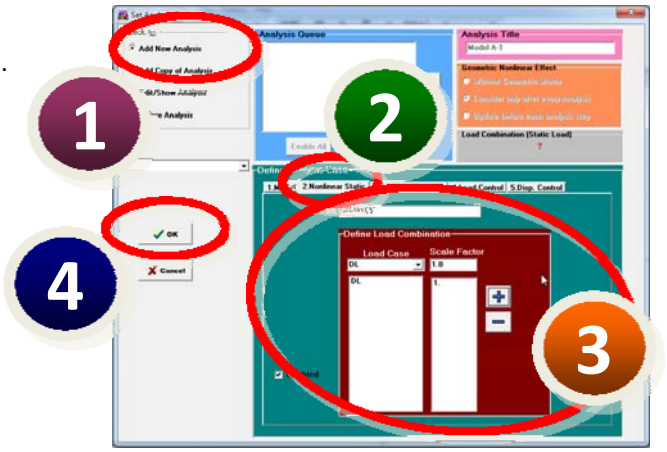
Define the Nonlinear Static Analysis for Gravity Analysis

1.Make sure the selected option is ***Add New Analysis***.

2.Select the ***Nonlinear Static*** page.

3.Input the Tag, the parameters and specify gravity loads using the ***static load case*** **DL**.

4.Click the ***OK*** button.



NARLabs

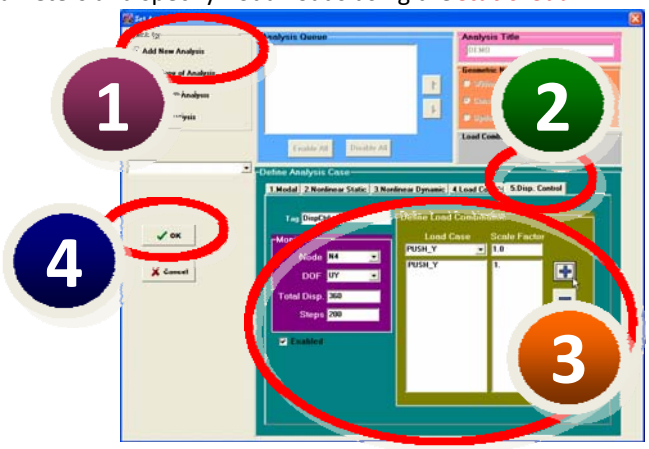
Define the Displacement Control Nonlinear Static Analysis

1.Make sure the selected option is ***Add New Analysis***.

2.Select the ***Disp. Control*** page.

3.Input the Tag, the parameters and specify nodal loads using the ***static load case***.

4.Click the ***OK*** button.



NARLabs

Output Setting

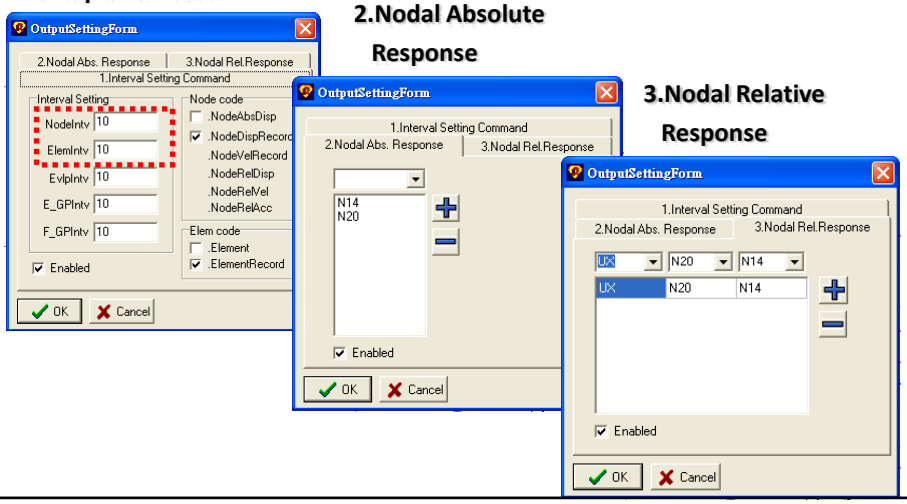
Analyze->Output Setting

(See PISA3D Manual: I01, I02, I03)

1.Output Interval

2.Nodal Absolute Response

3.Nodal Relative Response

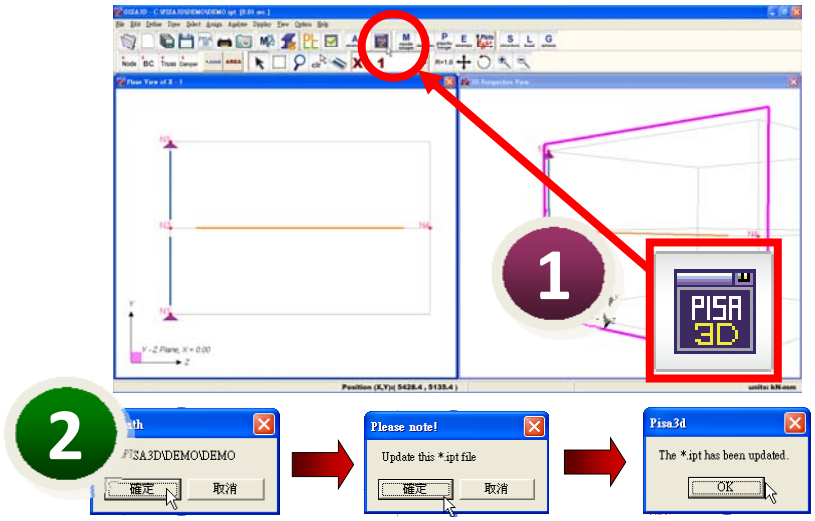


NARLabs

Perform the PISA3D Analyses

1.Click the button to execute the PISA3D.

2.Click on the ***Dialog Box*** to check the ***file path*** or ***updating*** the input file.

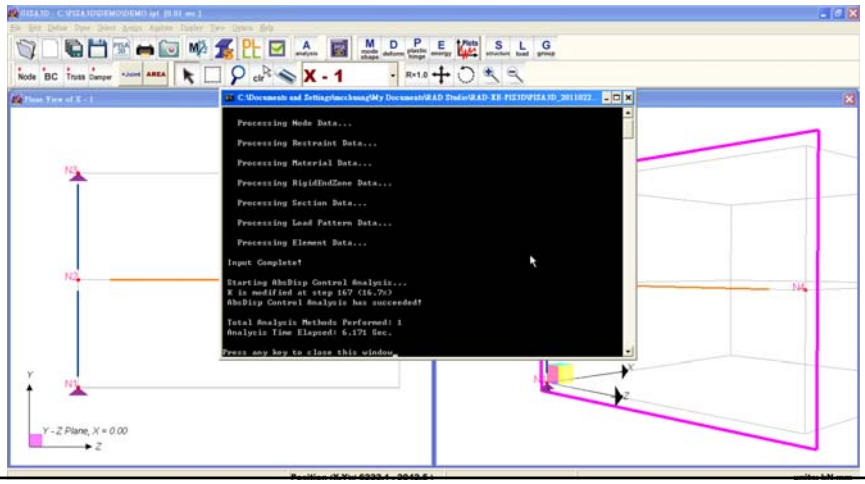


NARLabs

Perform the PISA3D Analyses

1.After all analyses have been performed successfully, Analysis Time Elapsed and the Notification are listed in the bottom of the window.

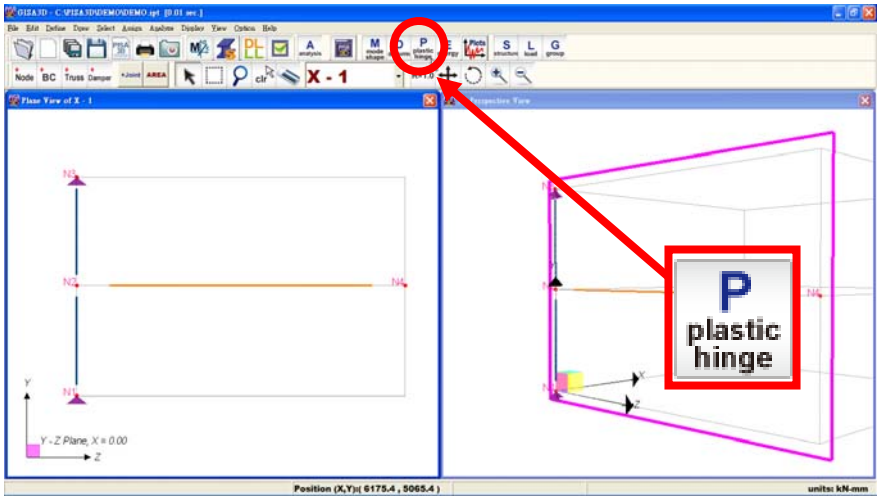
2.***Press any key*** to close this command line window.



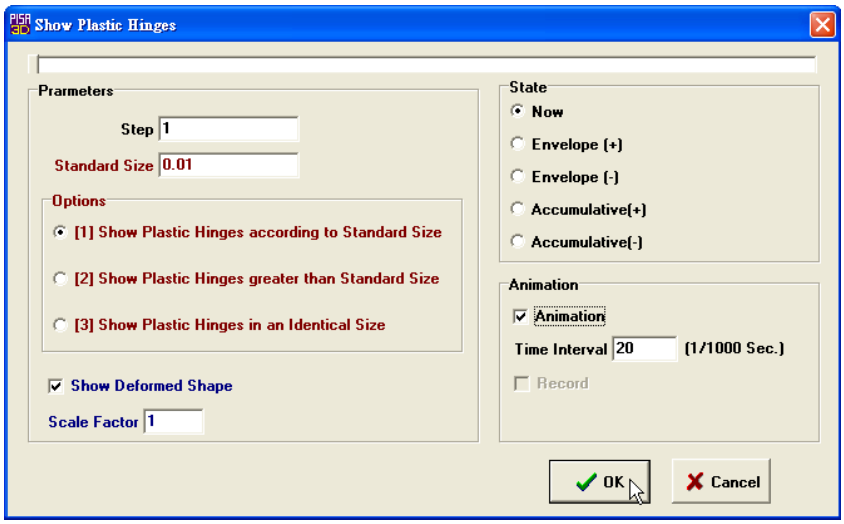
NARLabs

Display the Plastic Hinge Distributions with the Structural Deformation

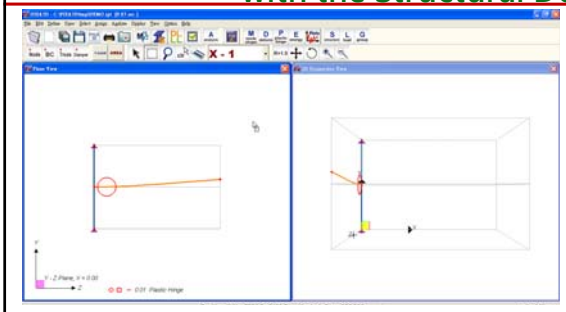
1.Click the button to activate the form of ***Show Plastic Hinges***.



Display the Plastic Hinge Distributions with the Structural Deformation



Display the Plastic Hinge Distributions with the Structural Deformation



About the representation of hinges in BeamColumn elements, circle and square indicate the nonlinear behavior occurred with flexural and shear yielding respectively. The red line shown on middle point of truss element indicates the axial yielding.

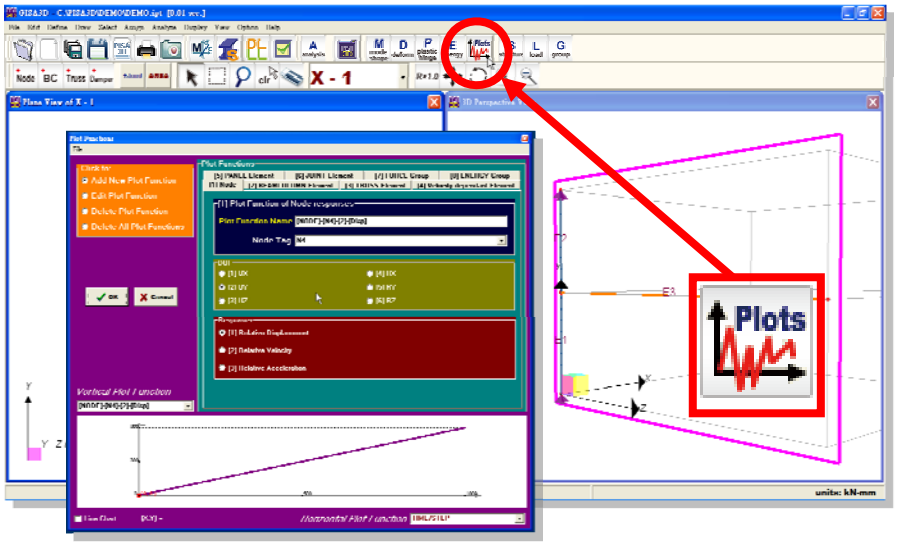
Additionally, the size of the shapes and line mentioned means the magnitudes of the nonlinear behavior, plastic hinge rotation, axial strain level. And the sizes are all normalized with respect to the given or default circle, square and line.

User can find the default standard shapes (0.01 for axial strain and 0.01 radians for rotation) on the bottom of the 2D view windows, and visually make a comparison between the standard one and the hinges showed in the model.

NARLabs

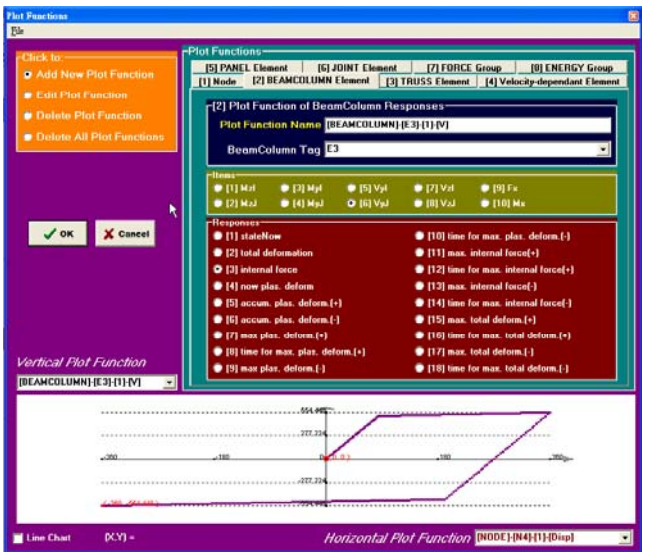
Display the Structural Response by using the Plot Functions

1.Click the button to activate the form of ***Plot Functions***.

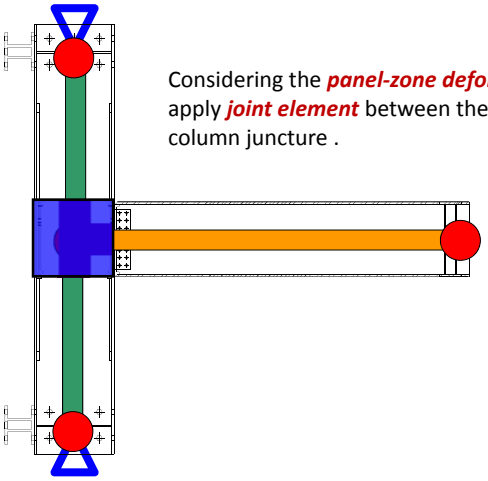


NARLabs

Display the Structural Response using the Plot Functions



Model 1-2
(W/ P.Z. deformation effect)



Considering the ***panel-zone deformation*** to refine the model A-1, apply ***joint element*** between the double nodes at the beam-to-column juncture .

Demonstration of Model 1-2

<u>The skills listed below will be illustrated in this example.</u>

1.Define a New Node Constrained by Other Nodes.

2.Add a Joint Element.

3.Define and Assign Joint Element Properties.

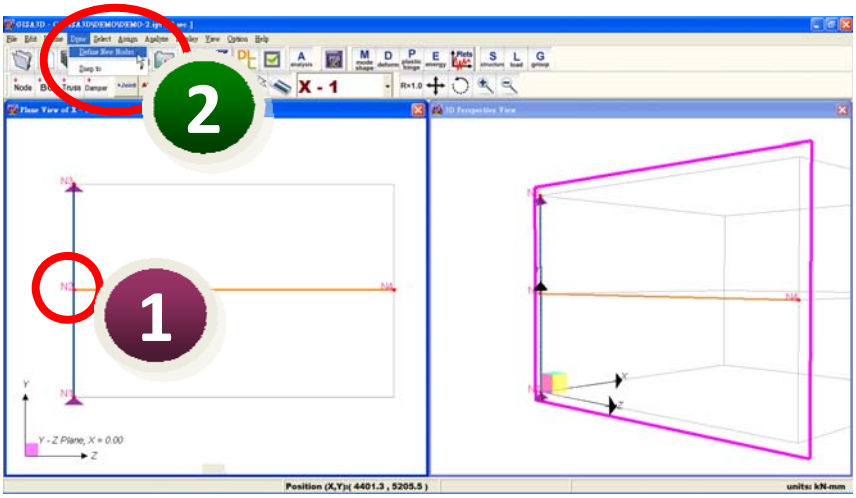
4.Change the Node-to-Element Relationship.

NARLabs

Create the New Node Identical to the Existing Node

1.Find the ***Tag of the existing node***.

2.Select the ***Draw>Define New Nodes*** from the main menu.



NARLabs

Create the New Node Identical to the Existing Node

1.Select ***the Existing Node*** **N2**.

2.Input the Tag of the New Node.

3.Check the Position and Edit ***the Degrees of Freedom.***

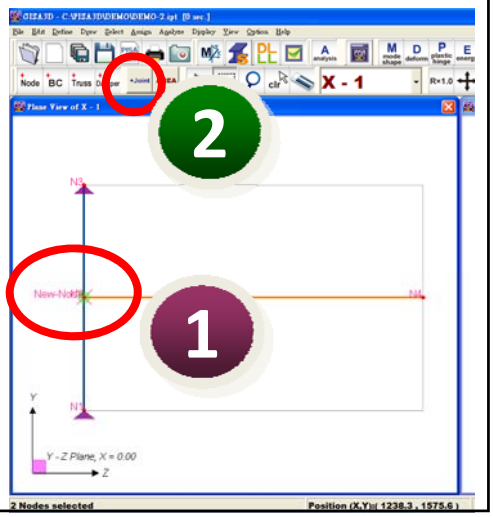
4.Click the ***OK*** button.

5.Click the ***Cancel*** button to close the form.

Create the Joint Element by using Two Nodes Share the Same Location in Space

1.Select ***a pair of Nodes***.

2.Click the ***+Joint*** button.



Panel Zone Model

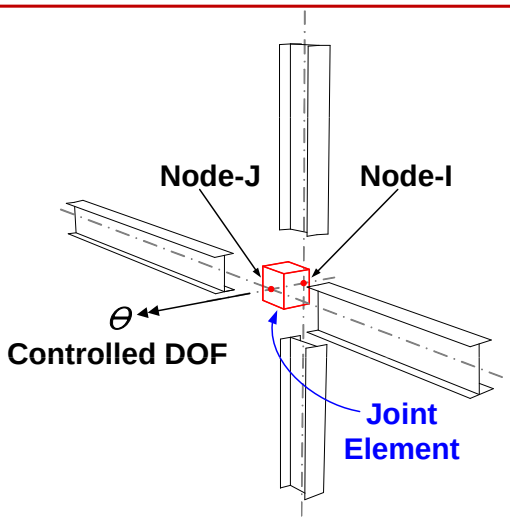
Command: Joint, Material

$$K = \frac{G d_b d_c t_j}{1 - d_b / h}$$

$$M_y = \frac{0.6 F_y d_b d_c t_j}{1 - d_b / h}$$

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

Input:
F_y & E_i in material,
K in Joint element

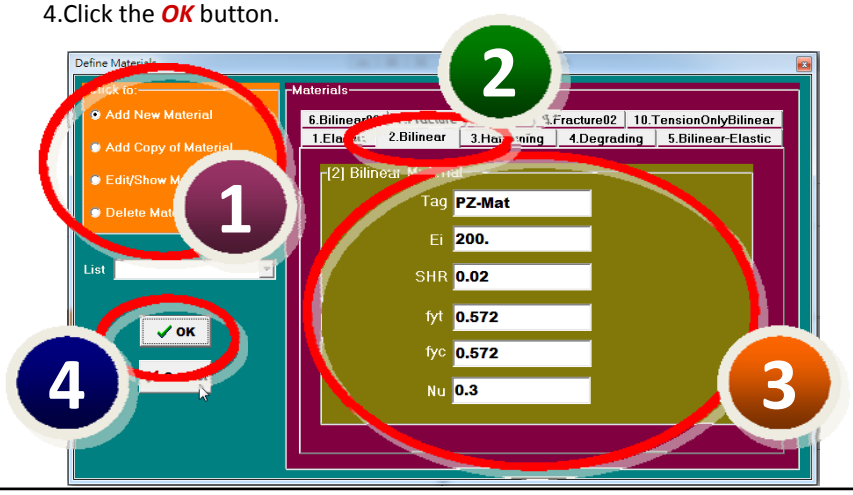


(See manual: G03, Joint Element)

NARLabs

Define the New Material Properties

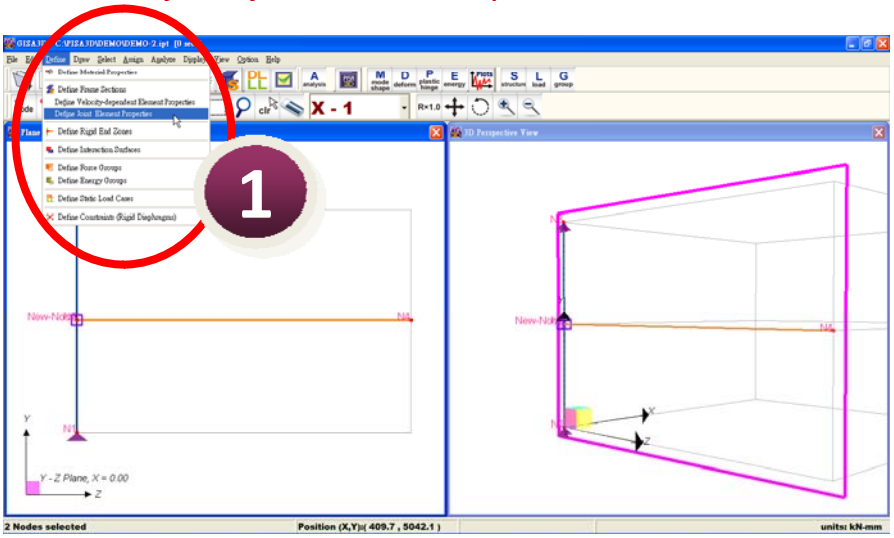
1.Make sure the selected option is ***Add New Material***.
2.Select the ***Bilinear*** page.
3.Input the ***Tag*** **PZ-Mat** and the parameters.
4.Click the ***OK*** button.



NARLabs

Define the Joint Properties

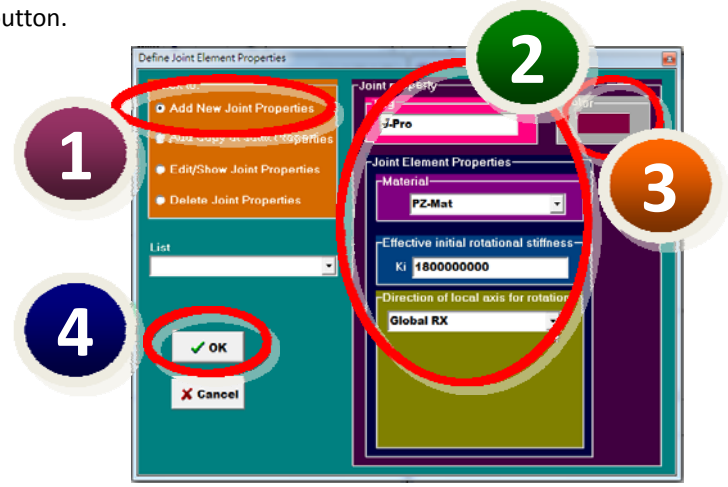
1.Select the ***Define>Define Joint Element Properties***.



NARLabs

Define the Joint Properties

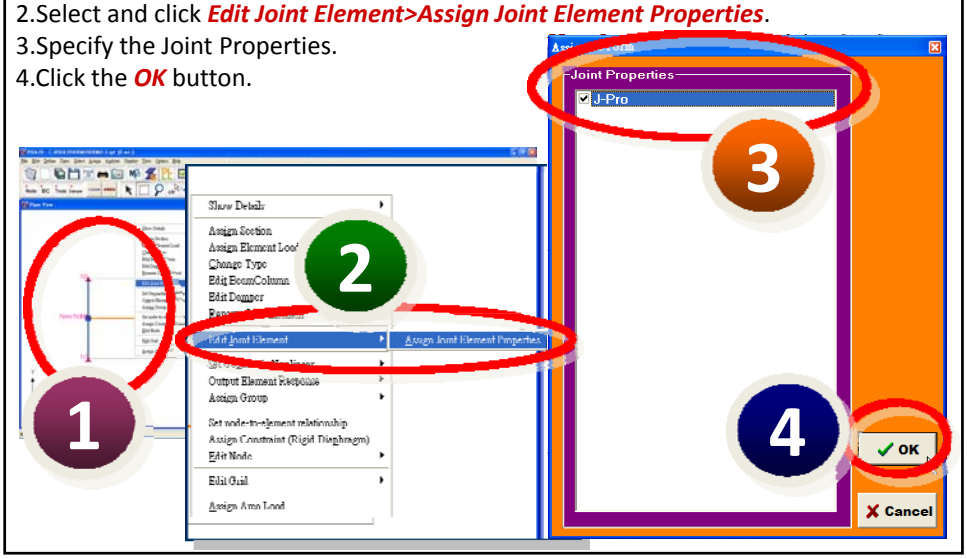
1.Make sure the selected option is ***Add New Joint Properties***.
2.Input the ***Tag*** **J-Pro** and the parameters.
3.Click the panel to specify the color that you want to display the property.
4.Click the ***OK*** button.



NARLabs

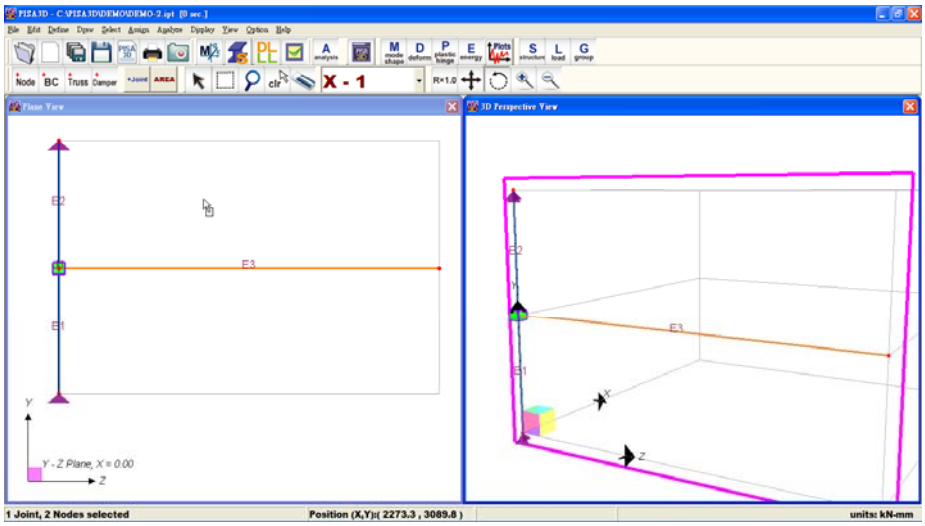
Assign the Joint Properties to the Joint Element

1.Select the ***Joint Element***, then show the pop-up menu using right mouse click.
2.Select and click ***Edit Joint Element>Assign Joint Element Properties***.
3.Specify the Joint Properties.
4.Click the ***OK*** button.



NARLabs

Assign the Joint Properties to the Joint Element

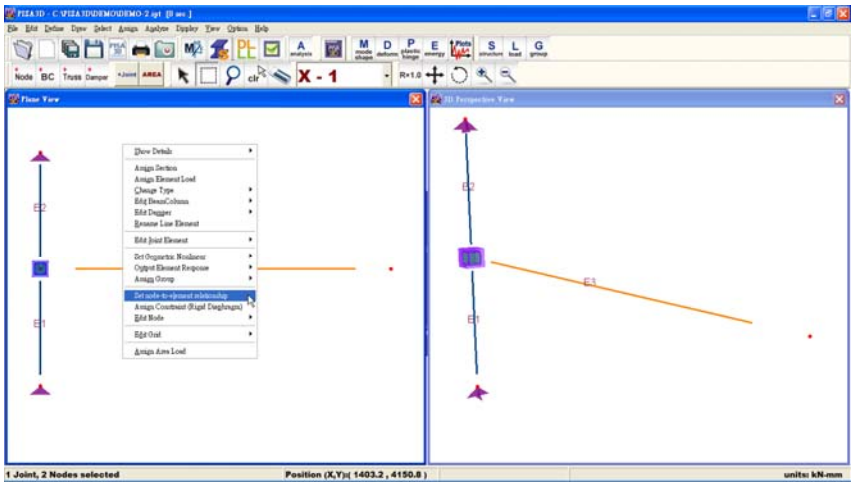


NARLabs

Set the Node-to-Element Relationship

1.Select the ***Pair of Nodes***, then show the pop-up menu using right mouse click.

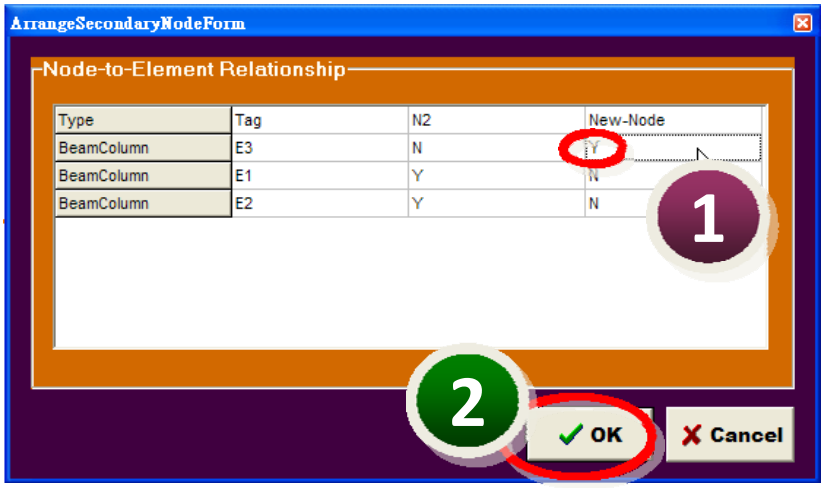
2.Select and click ***Set node-to-element relationship***.



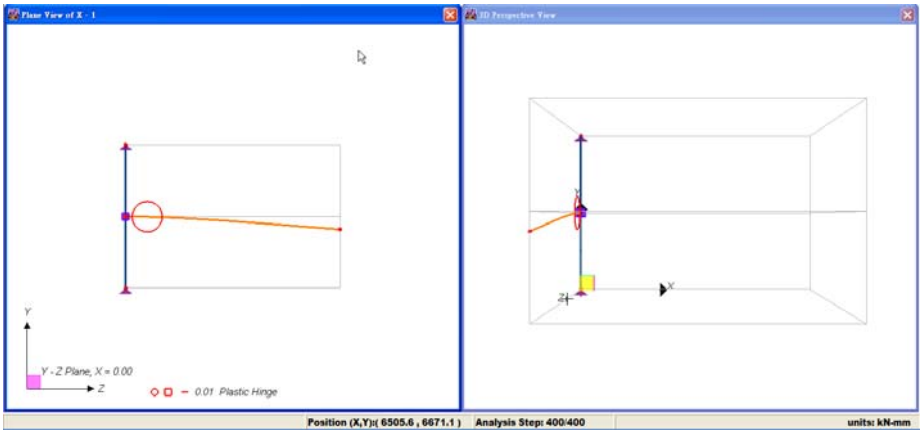
Set the Node-to-Element Relationship

1.Double clicks to change ***the node to element relationship***.

2.Click the ***OK*** button.

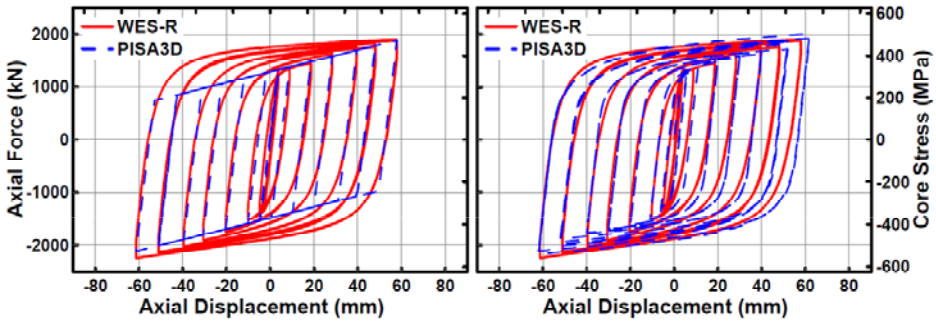


Structural Response with the Deformation of the Panel Zone



Simulation of the Buckling Restrained Brace

Bilinear Hardening (Two-Surface Model) *(Dafalias and Popov, 1975)*

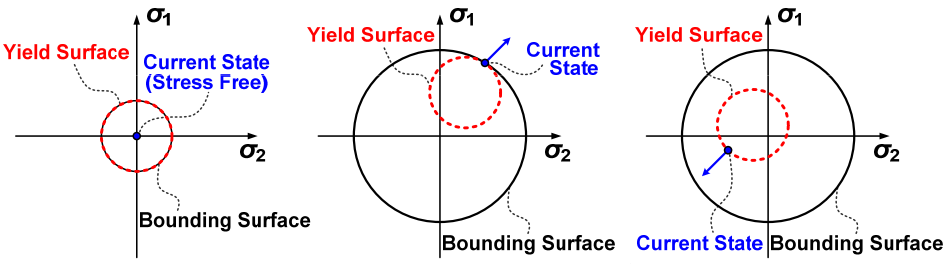


Dafalias, YF, Popov, EP. (1975). "A model of nonlinearly hardening materials for complex loading." Acta Mechanica, 21(3): 173-192.

Hardening Material

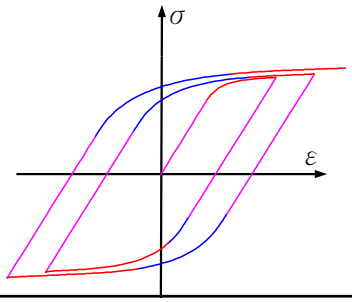
Command: Material, Hardening (Manual: F03)

Two-Surface Model



Bounding Surface
(Isotropic Hardening)

Yield Surface
(Kinematic Hardening)



78

Hardening Material

NARLabs

Command: Material, Hardening (Manual: F03)

			C_1^+	C_2^+	C_1^-	C_2^-	C_3	C_4	
E_i	f_{yp}	f_{yc}	H_{iso1+}	H_{iso2+}	H_{iso1-}	H_{iso2-}	H_{kin1}	H_{kin2}	BS/YS

$$F_T = F_E + F_P$$

F_T: total flexibility
F_E: elastic flexibility
F_P: plastic flexibility

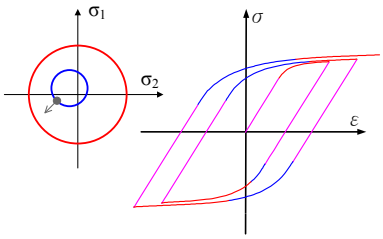
Isotropic hardening:

$$F_{Pi} = \frac{F_E}{\left[C_1 + (1 - C_1)e^{\left(-\frac{BS}{YS}C_2\right)}\right]}$$

BS/YS ↑, $F_{Pi} \rightarrow F_E/C_1$,

$E_T = E_E * C_1/(1+C_1)$,

If C_1 is small, $E_T \approx E_E C_1$



Kinematic hardening:

$$F_{Pk} = \frac{F_{Pi}}{\left[1 + C_3\left(\frac{\delta}{\delta_{ini} - \delta}\right)^{C_4}\right]}$$

When the material begins kinematic hardening:
$\delta = \delta_{ini}$, $F_{Pk} = 0$

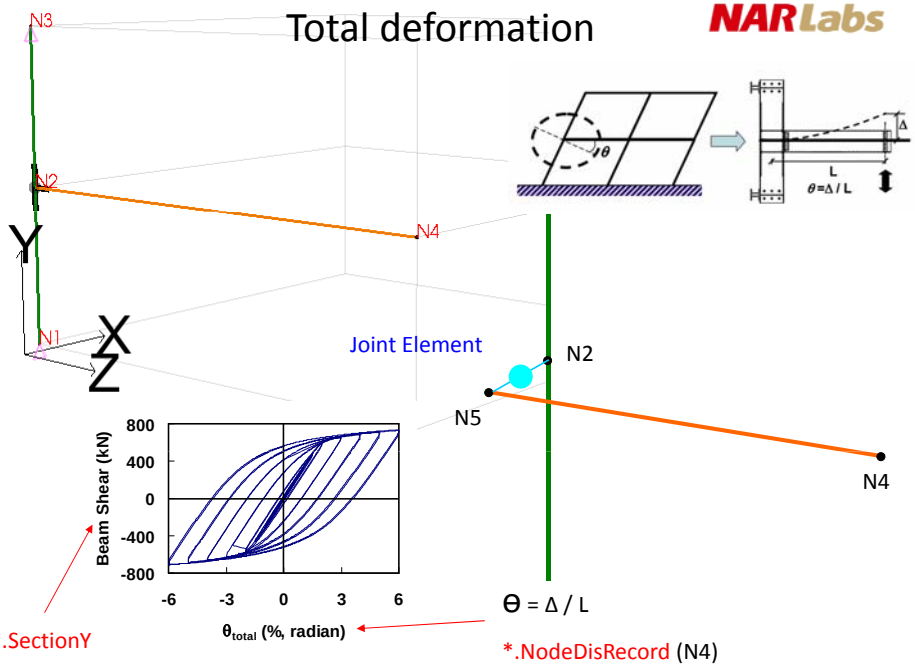
When the material begins isotropic hardening:
$\delta = 0$, $F_{Pk} = F_{Pi}$

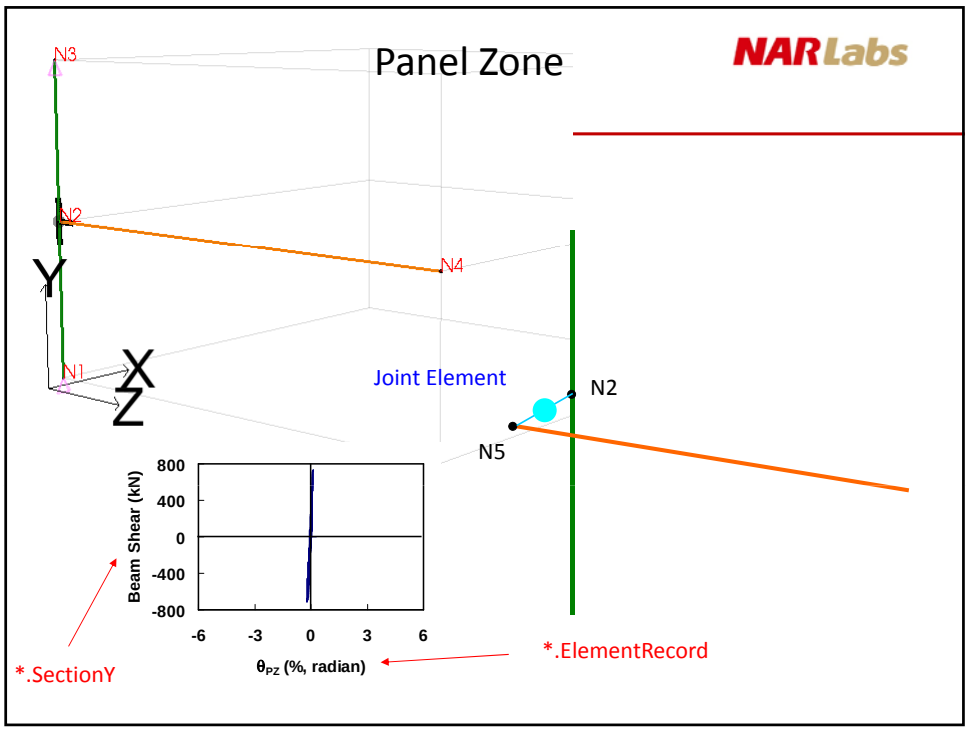
δ: the distance of the stress state under consideration from the corresponding bound

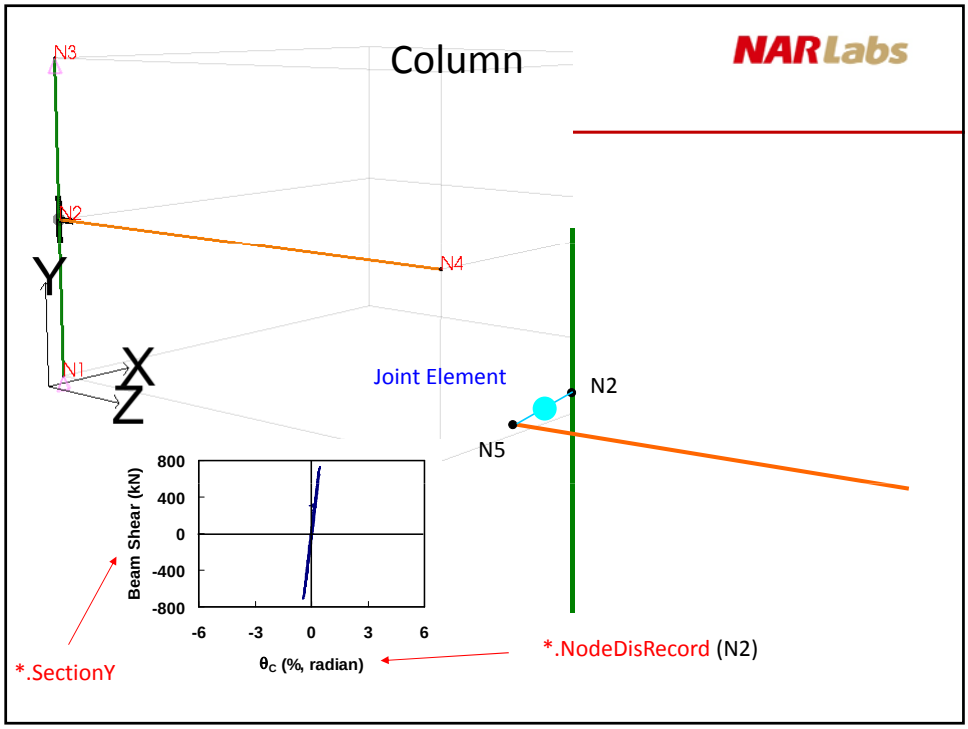
δ_{ini}: the value of δ at the initiation of yielding for each loading process

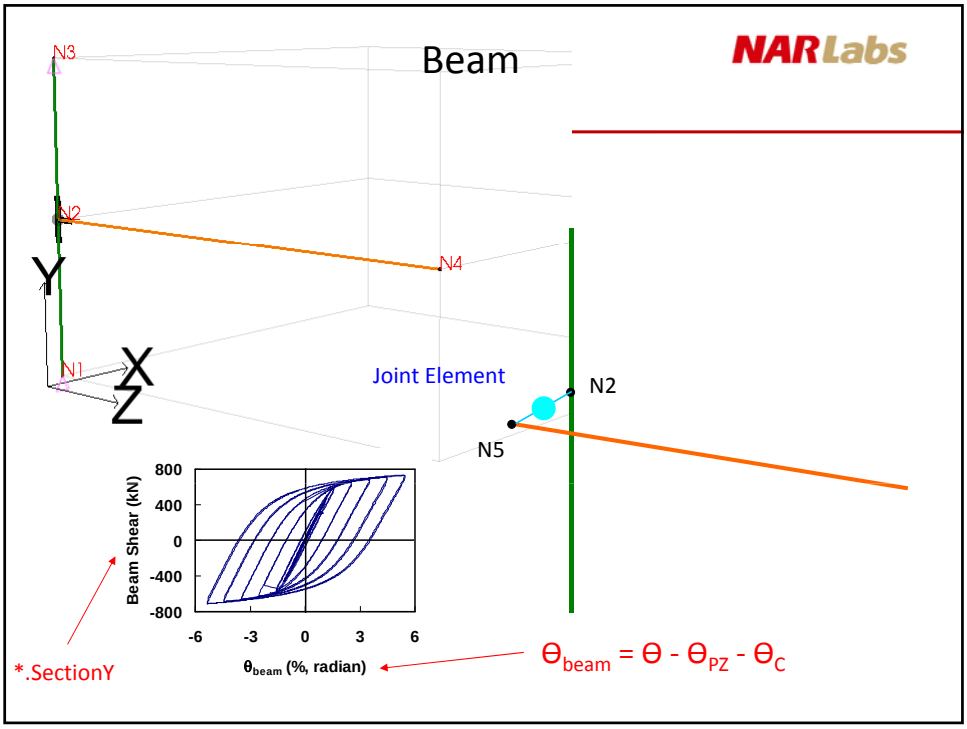
Total deformation

NARLabs




Panel Zone
NARLabs
N3
N2
N4
N1
Y
X
Z
Joint Element
N2
N5
Beam Shear (kN)
800
400
0
-400
-800
-6
-3
0
3
6
θPZ (%, radian)
*.SectionY
*.ElementRecord


Column
NARLabs
N3
N2
N4
N1
Y
X
Z
Joint Element
N2
N5
Beam Shear (kN)
800
400
0
-400
-800
-6
-3
0
3
6
θC (%, radian)
*.SectionY
*.NodeDisRecord (N2)

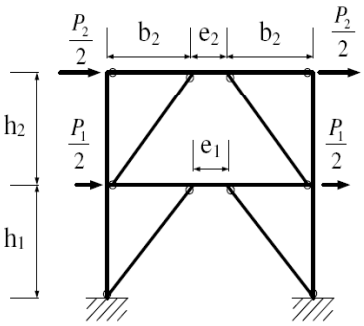


NARLabs

Model 2

Simulations of the 2-story Eccentrically Braced Frame (EBF)

The example case of 2-story EBF



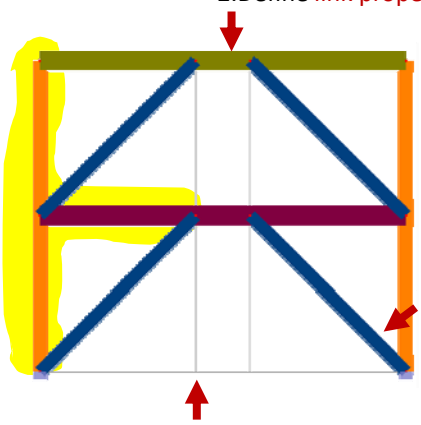
- The link lengths, e1=e2=1.4m.
- The beam segment, b2=4.0m.
- The story height, h1=h2=4.0m.
- The LRFD seismic lateral forces P2=200 tf, P1=100 tf.

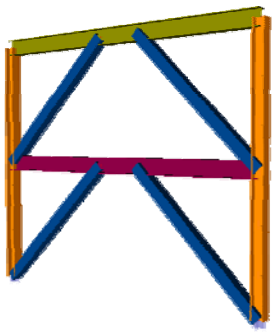
Only for demonstration!

Component	Section	Fy (kN/mm²)
Upper Beam	H500x300x11x25	0.344
Lower Beam	H500x300x16x33	0.344
Column	H350x350x15x32	0.344
Brace	HSS 320x320x10	0.344

What's new in this case

2.Define link properties



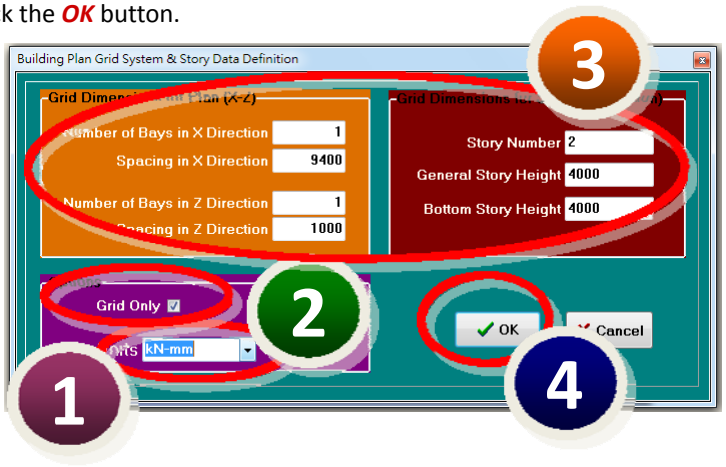


3.Add truss elements to simulate braces

1.Add/Adjust grid data (X-Z Plane)

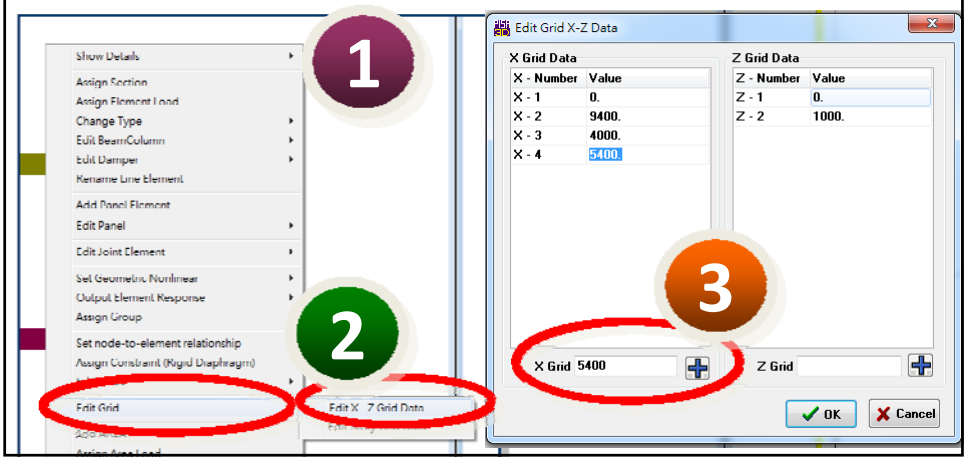
Initialize the EBF Model

1.Specify the units: ***kN-mm***.
2.***Deselect*** the ***Grid Only*** mode to initialize your model.
3.Input the details about your grid system.
4.Click the ***OK*** button.



Edit the Grids

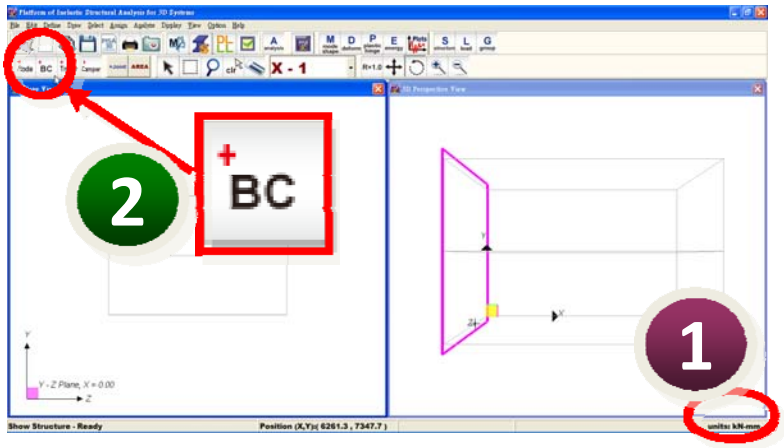
1.Click right mouse button to activate the ***pop-up menu***.
2.Select and Click ***Edit Grid>Edit X-Z Grid Data*** to activate the form for editing the X-Z Grid Data .
3.Input the ***new grid data*** and click ***+*** button to add.



Activate Draw Mode for Drawing BeamColumn Element

1.Here you can find and check the units.

2.Click the button to start to draw a BeamColumn element.

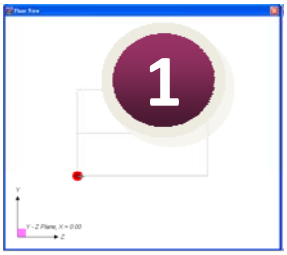


Draw a BeamColumn Element

1.When the mouse position is close to the grid intersection, red point means ***snap to grid function*** is available.

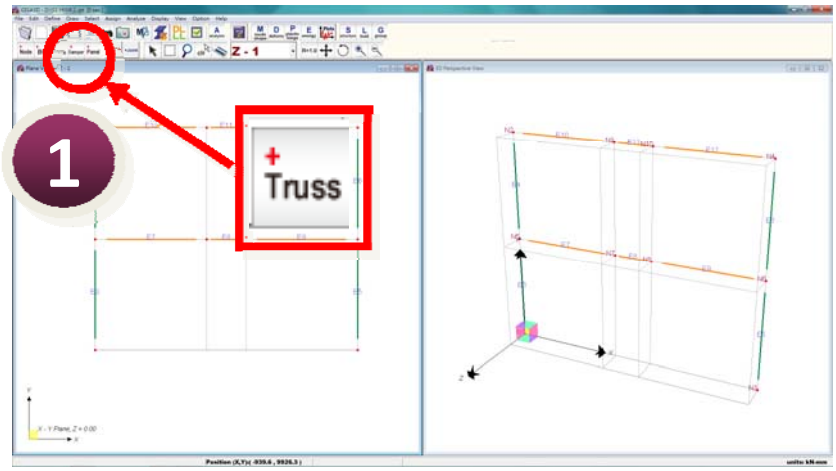
2.Click the left mouse button and drag mouse to define a BeamColumn element.

3.Repeat step 2, construct the column and the beam.



Activate Draw Mode for Drawing Truss Element

1.Click the button to start to draw a Truss element.

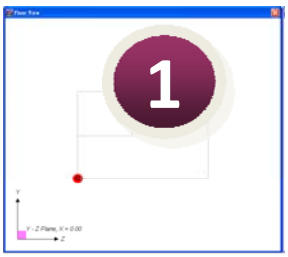


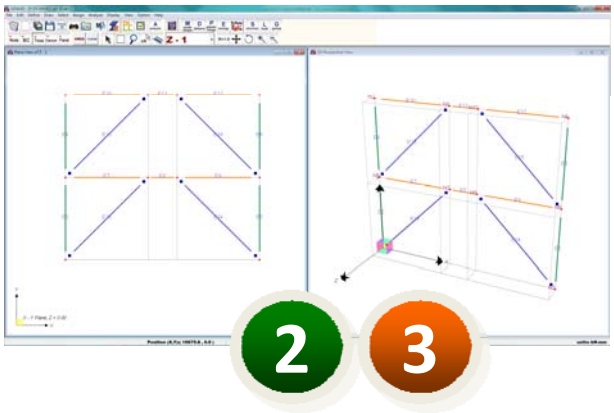
Draw a Truss Element

1.When the mouse position is close to the grid intersection, red point means ***snap to grid function*** is available.

2.Click the left mouse button and drag mouse to define a Truss element.

3.Repeat step 2, construct the braces.

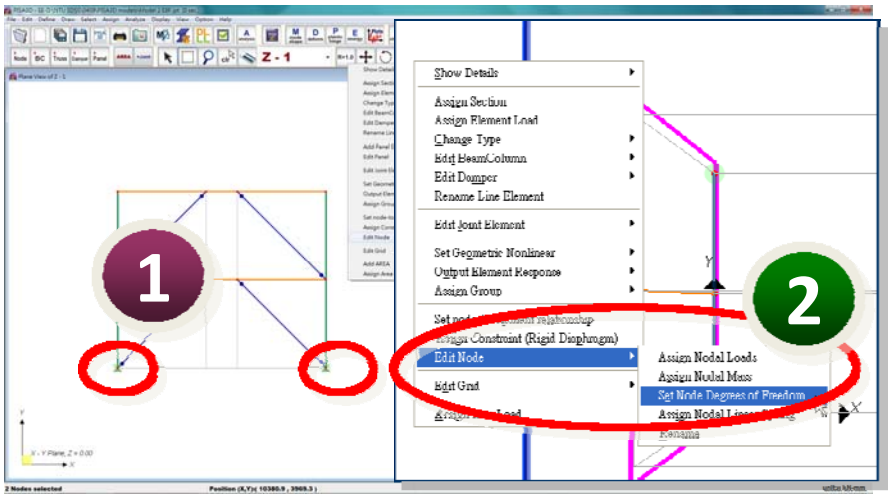




Edit the Node Degrees of Freedom

1.Select the **two nodes**, then click right mouse button to show the popup menu.

2.Select and Click the ***Edit Node>Set Node Degrees of Freedom***.

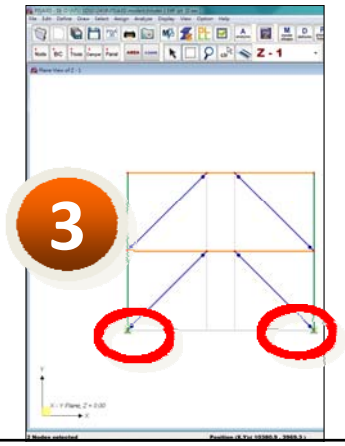


Edit the Node Degrees of Freedom

1.Click ***Pin*** button using the fast set function.

2.Click the ***OK*** button.

3.Immediate representation of the specified DOF/Boundary Condition.



Alternative way to create truss element

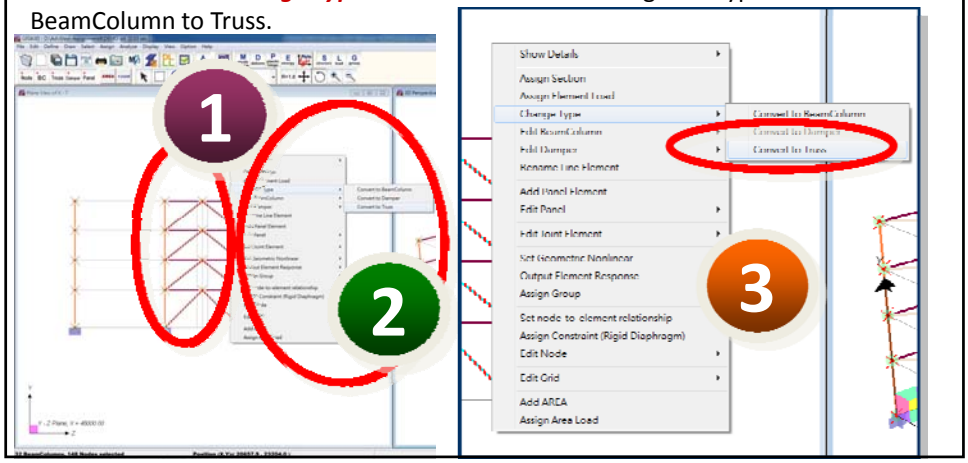
NARLabs

Change the Type of Element

1.Select the BeamColumn ekements

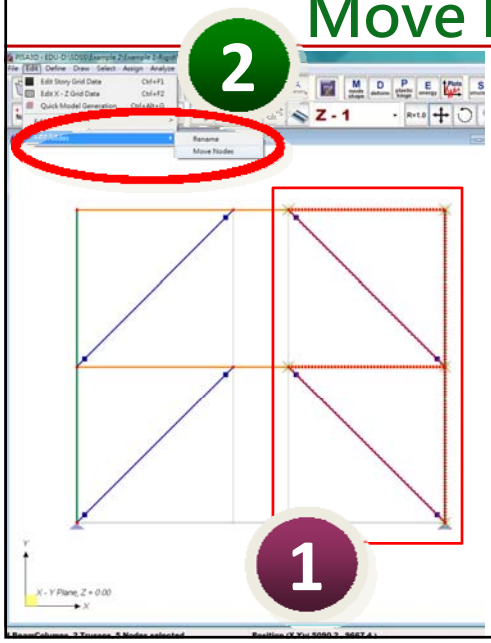
2.Click right mouse button to activate the ***pop-up menu***.

3.Select and Click ***Change Type>Convert to Truss*** to change the type from BeamColumn to Truss.



NARLabs

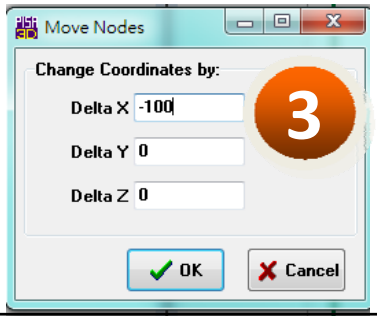
Move Nodes



1.Select the **nodes which should be moved**.

2.Select and Click the ***Edit>Edit Node>Move Nodes***.

3.Input the value of shift.



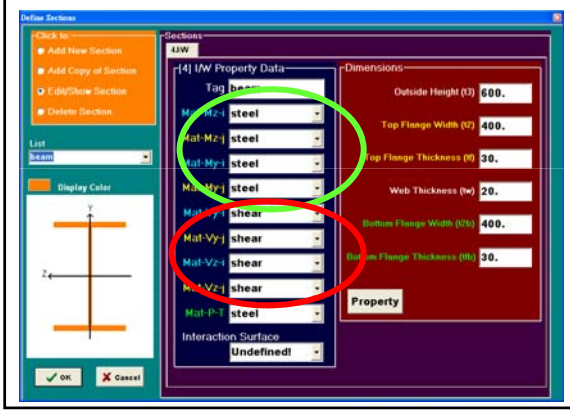
96

EBF-Link Model

Command: BeamColumn, BCSection03, Material

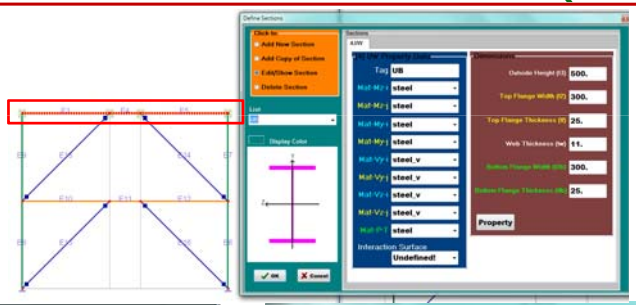
$M_y = S \times F_y$

$V_y = A_v \times \tau_y = A_v \times (0.6F_y)$

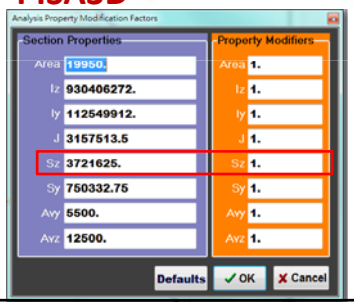


97

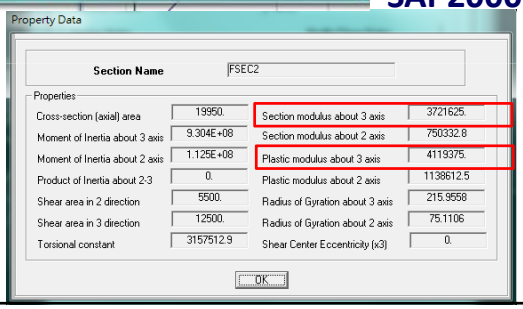
Section of EBF-Link (UB)



PISA3D

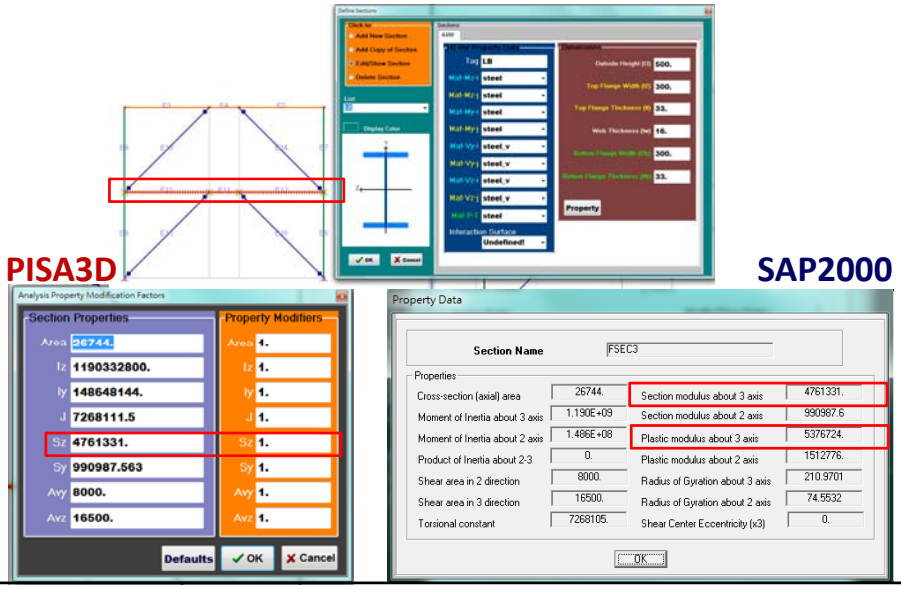


SAP2000



98

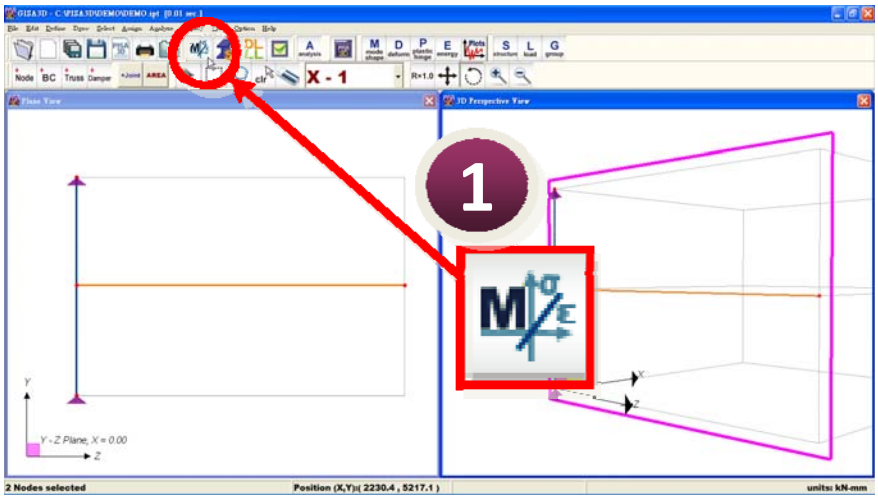
Section of EBF-Link (LB)



99

Define the New Material Properties

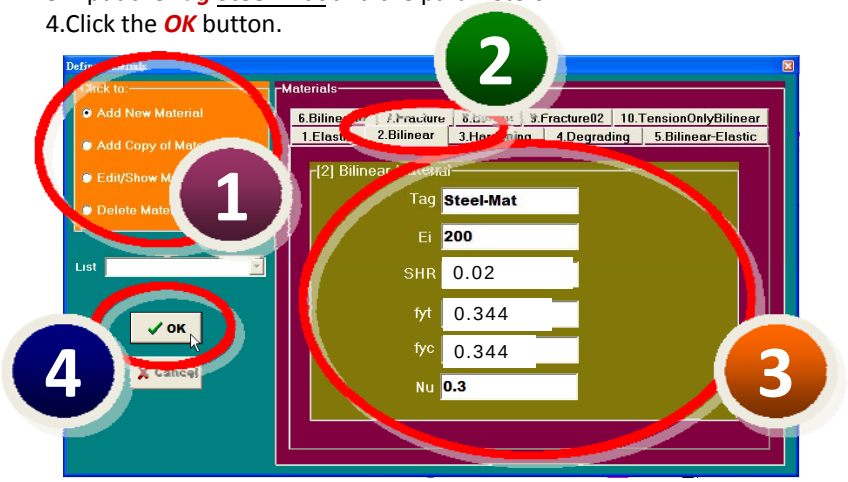
1.Click the button to activate the form of define materials.



NARLabs

Define the New Material Properties

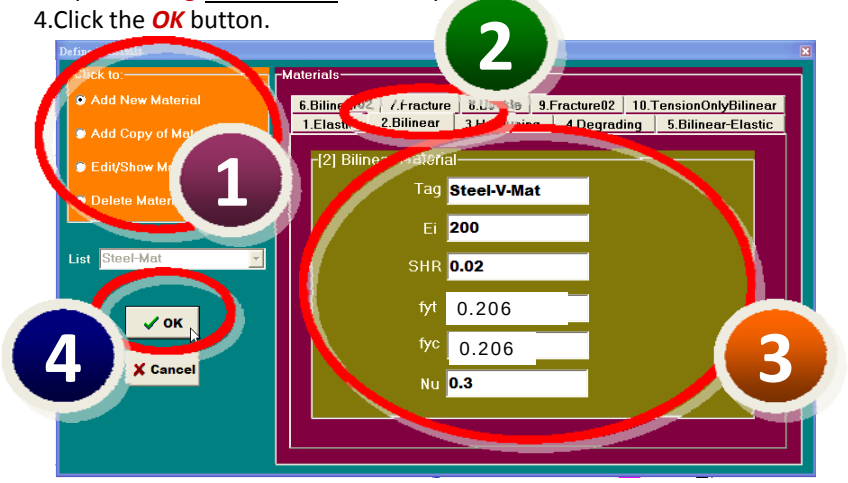
1.Make sure the selected option is ***Add New Material***.
2.Select the ***Bilinear*** page.
3.Input the ***Tag*** **Steel-Mat** and the parameters.
4.Click the ***OK*** button.



NARLabs

Define the New Material Properties

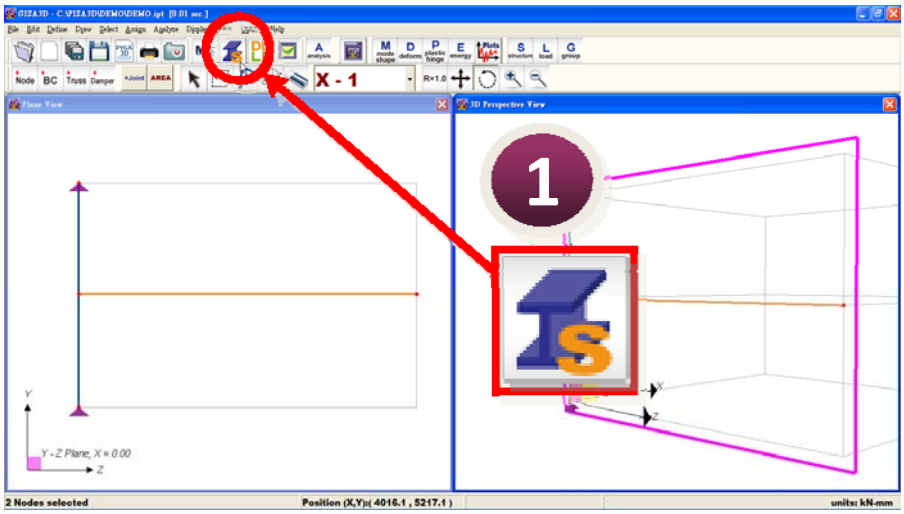
1.Make sure the selected option is ***Add New Material***.
2.Select the ***Bilinear*** page.
3.Input the ***Tag*** **Steel-V-Mat** and the parameters.
4.Click the ***OK*** button.



NARLabs

Define the Sections for BeamColumn Elements

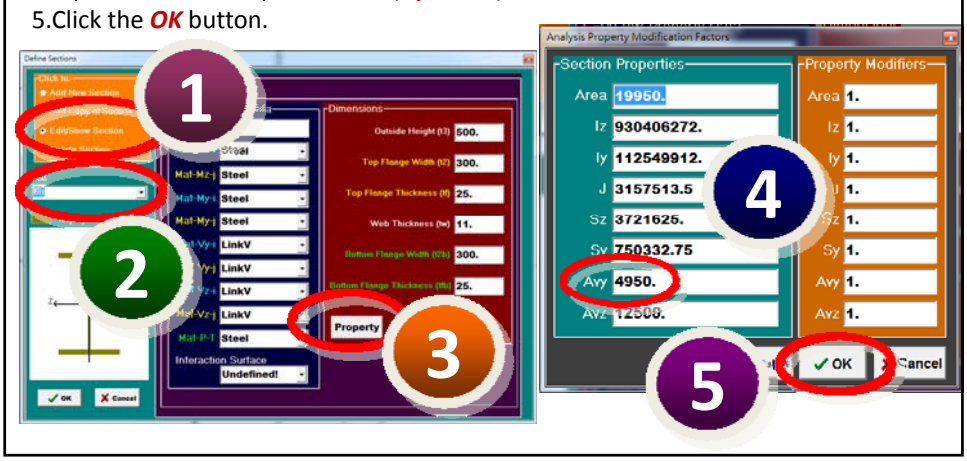
1.Click the button to activate the form for define sections.



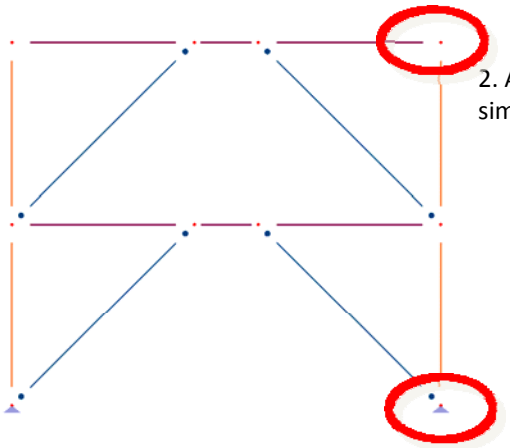
NARLabs

Edit the Section Properties

1.Make sure the selected option is ***Edit/Show Section***.
2.Select the Section named Beam-Sec from ***drop-down list***.
3.Click the ***Property*** button.
4.Input the modified parameters (***Optional***).
5.Click the ***OK*** button.



Change the column base and all the beam-to-column connections from rigid to pin

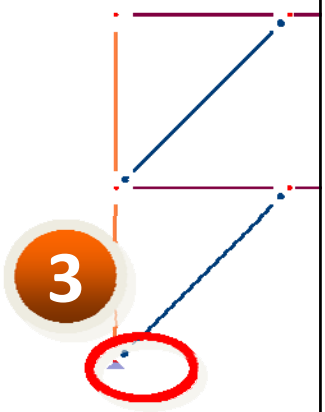


2. Add and constrain a New Node to simulate the pin connection

1. Edit the Node Degrees of Freedom

Edit the Node Degrees of Freedom

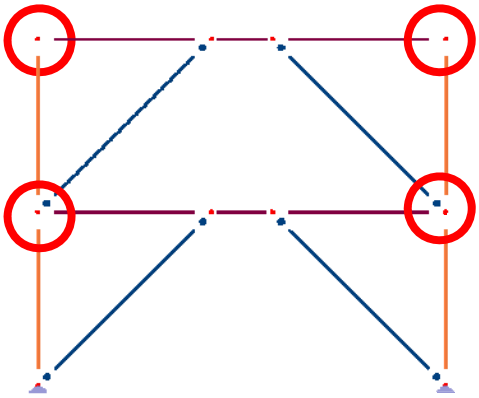
1.Click ***Pin*** button using the fast set function.
2.Click the ***OK*** button.
3.Immediate representation of the specified DOF/Boundary Condition.



Create the New Node Identical to the Existing Node

1.Find the ***Tag of the existing nodes***.

2.Select the ***Draw>Define New Nodes*** from the main menu.



Create the New Node Identical to the Existing Node

1.Select ***the Existing Node*** **N5**.

2.Input the Tag of the New Node.

3.Check the Position and Edit ***the Degrees of Freedom.***

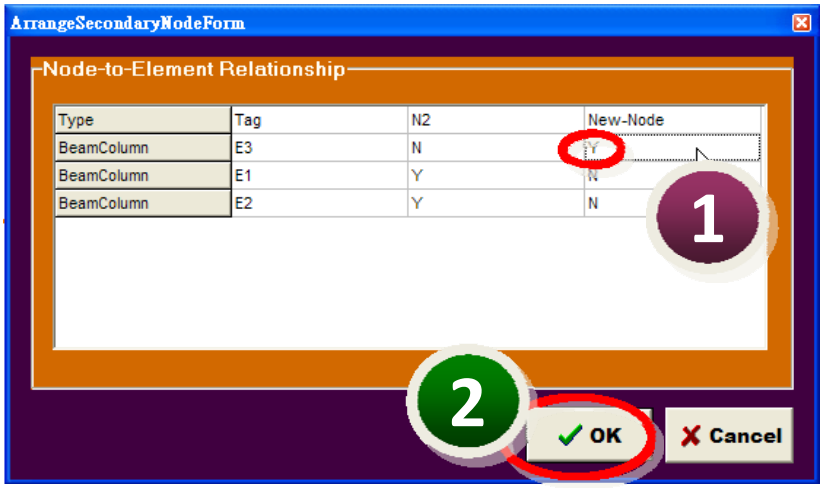
4.Click the ***OK*** button.

5.Click the ***Cancel*** button to close the form.

Set the Node-to-Element Relationship

1.Double clicks to change ***the node to element relationship***.

2.Click the ***OK*** button.



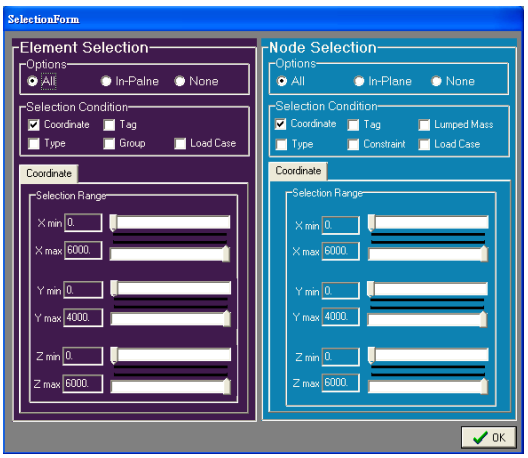
Quick Search

設定Element搜尋範圍

設定Node搜尋範圍

****Select>Search***

Ctrl+Alt+S



可選定項目，以**交集**方式過濾，進行條件式搜尋Element

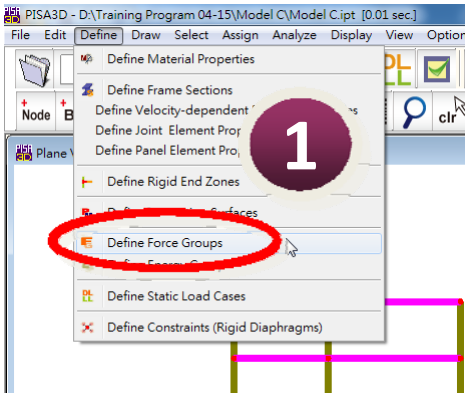
可選定項目，以**交集**方式過濾，進行條件式搜尋Node

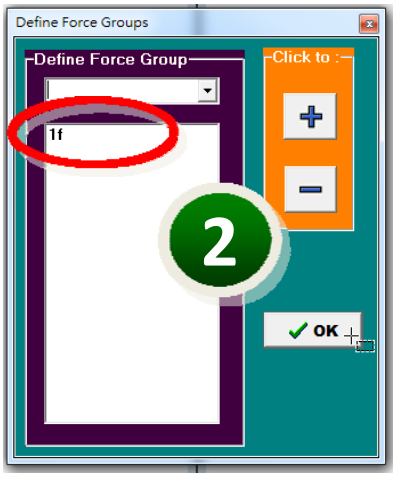
NARLabs

Define the Force Group

1.Select and Click the ***Define>Define Force Group*** from the main menu.

2.Add the new ***Force Group*** **1f**.

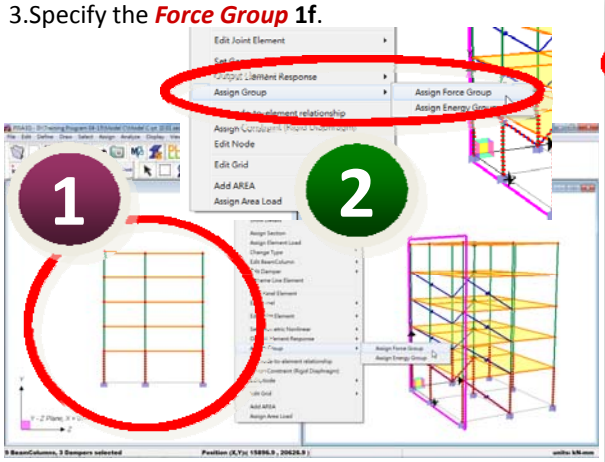


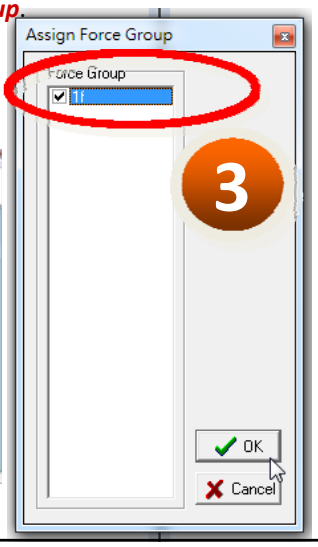


NARLabs

Assign the Force Group

1.Select elements, then click right mouse button to show the popup menu.
2.Select and Click the ***Assign Group>Assign Force Group***.
3.Specify the ***Force Group*** **1f**.



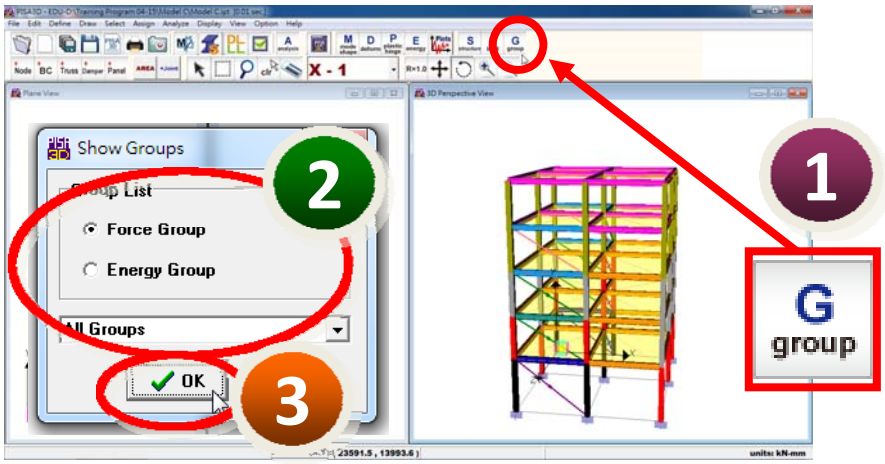


Display the Members of the Force Group

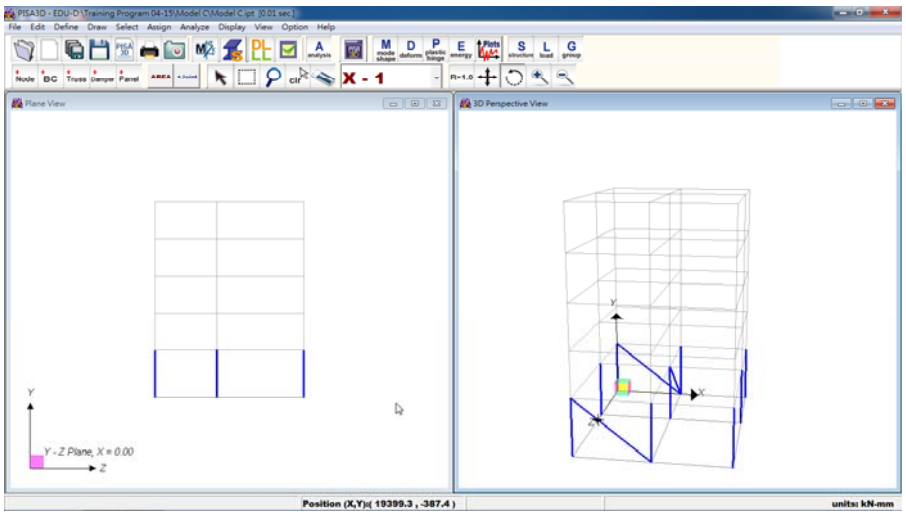
1.Click the button to activate the form of ***Show Groups***.

2.Click to show force group, then select a Group/All Groups to show.

3.Click the ***OK*** button.



Display the Members of the Force Group



The Prefix Code "GUI_"

Element BilinearDamper Tag I J MatTag Area Fr C1 C2 Out Energy_Gp Force_Gp Model_Type

```
Element  BilinearDamper  E106 N39 N33 GUI_dp0xMat0f_of_E106 1  200  3.13  0.21  1 0  0  0
Material Elastic GUI_dp0xMat0f_of_E106  0  0
GUI_Element  VD_Element  E106 N39 N33 BilinearDamper250  1 0  0
```

```
GUI DAPMER PROPERTIES  BilinearDamper250  KELVIN MODEL  DEFINED By STIFFNESS  0  DEFINED By BILINEAR DAMPER  200  3.13  0.21
GUI_VELOCITY_DRP_PROPERTIES_DISPLAY_COLOR BilinearDamper250 1  0.5  0.5
```

Model C - 記事本

檔案(F) 編輯(E) 格式(O) 檢視(V) 說明(H)

```
PISA3D
bstory
KN
KM
ControlData    GeometricNL  0
GUI_Analysis  ModeShape  modal  6  1  2  0.02  0.02  ON
Analysis  ModeShape  6  1  2  0.02  0.02

GUI Analysis  Gravity  gravity  DL  1  ON
Analysis  Gravity  DL  1  GUI_AREA_LOAD_DL  1

GUI_Analysis  Dynamic  Newmark  dynamic  100x.txt  1  1  1  100y.txt  1  1  1  100z.txt  1  -1  1  0.01  4096  0.13963  0.0028636  0  ON
Analysis  Dynamic  Newmark  dynamic XGndMot  1  dynamic YGndMot  1  dynamic ZGndMot  1  0.01  4096  0.13963  0.0028636  0

LoadPattern  GroundAccel  dynamic XGndMot  <D:\Training Program 04-15\Model C\100x.txt>  1  1
LoadPattern  GroundAccel  dynamic_YGndMot  <D:\Training Program 04-15\Model C\100y.txt>  1  1
LoadPattern  GroundAccel  dynamic ZGndMot  <D:\Training Program 04-15\Model C\100z.txt>  1  -1

GUI GRID  XDIR  0  5000  10000
GUI_GRID  YDIR  0  3850  6850  9850  12850  15835
GUI GRID  ZDIR  0  5000  12000
```

Contents of Output Files (1)

1. Echo
   包含分析模型的資料與分析過程的訊息，可供使用者檢核模型正確性，錯誤訊息亦會輸出於此檔。

2. Element
   在每個指定要輸出的時間點上，輸出使用者指定要輸出的所有元素反應。

3. ElemRecord
   印出上述.Element檔案的所有內容，但是會將每個元素的反應依時間順序排列在一起，方便作圖。此檔只有在分析完成後才會產生。

4. ElemEnvelope
   在每個指定要輸出的時間點上，輸出使用者指定要輸出的所有元素，到此時間點為止的最大反應。

5. SectionX, SectionY, SectionZ
   輸出X/Y/Z向的元素群組斷面力。

6. Energy
   印出系統應變能、阻尼能、動能、外力輸入能量、元素群組應變能量，與不平衡能量。

Contents of Output Files (2)

7. NodeAbsDisp
   在每個指定要輸出的時間點上，輸出使用者指定要輸出的所有節點的位移反應。

8. NodeDisRecord
   印出上述.NodeAbsDisp檔案的所有內容，但是會將每個節點的反應依時間順序排列在一起，方便作圖。此檔只有在分析完成後才會產生。

9. NodeVelRecord, NodeAccRecord
   如同.NodeDisRecord，但記載的是速度與相對加速度的反應

10. NodeRelDisp, NodeRelVel, NodeRelAcc
    根據使用者的指定，輸出節點間的相對位移、速度與加速度反應，例如可用來觀測層間變位。

11. NodeEnvelope
    在每個指定要輸出的時間點上，輸出使用者指定要輸出的所有節點，到此時間點為止的最大反應。

12. Eigen
    振態分析結果的輸出檔，記載自然週期、振態，與比例阻尼的α、β值。

Contents of Output Files (3)

【.Energy file】

ADST_1.Energy - 記事本

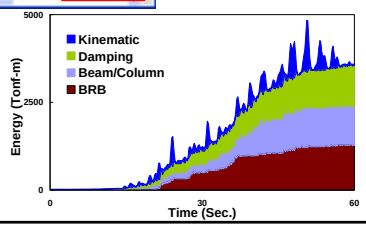
PISA3D Energy Distribution File

Analysis Time/Step	Unbalanced Energy_Ratio	Input Energy	Kinetic Energy	Damping Energy	Strain Energy	Element Group-BC	Element Group-LINK
0.100	0.00%	3.237E+001	3.458E+000	6.459E+000	2.245E+001	2.176E+001	6.141E-001
0.200	-0.00%	1.856E+002	7.191E+001	1.630E+001	9.738E+001	8.779E+001	3.697E+000
0.300	0.00%	2.260E+002	9.238E+001	4.583E+001	8.775E+001	4.578E+001	8.067E-001
0.400	0.00%	2.152E+002	7.704E+001	5.065E+001	8.753E+001	4.581E+001	6.143E-001
0.500	0.00%	2.391E+002	1.437E+002	6.659E+001	2.884E+001	2.268E+001	9.477E-001
0.600	0.00%	2.473E+002	1.240E+002	7.873E+001	4.460E+001	4.107E+001	9.285E-002
0.700	0.00%	2.364E+002	1.124E+002	9.205E+001	3.201E+001	2.101E+001	9.491E-001
0.800	0.00%	2.822E+002	5.714E+001	1.030E+002	1.220E+002	1.019E+002	4.171E+000
0.900	0.00%	4.618E+002	6.164E+001	1.132E+002	2.870E+002	2.358E+002	1.091E+001
1.000	0.00%	5.804E+002	1.385E+002	1.278E+002	3.142E+002	1.892E+002	1.128E+001
1.100	0.00%	5.642E+002	1.253E+002	1.497E+002	2.892E+002	1.324E+002	5.250E+000
1.200	0.00%	8.339E+002	2.991E+002	1.903E+002	3.445E+002	2.630E+002	1.223E+001
1.300	0.00%	1.120E+003	6.396E+002	2.051E+002	2.753E+002	2.406E+002	8.186E+000
1.400	0.00%	1.149E+003	7.764E+002	2.755E+002	9.716E+001	8.054E+001	1.779E-001
1.500	0.00%	1.315E+003	6.668E+002	3.283E+002	3.198E+002	3.137E+002	2.425E+000
1.600	0.00%	1.400E+003	8.057E+002	3.632E+002	2.310E+002	1.667E+002	4.276E+000
1.700	-0.00%	1.510E+003	6.114E+002	4.099E+002	4.889E+002	3.164E+002	1.120E+001
1.800	-0.00%	1.434E+003	1.587E+002	4.325E+002	8.432E+002	6.233E+002	1.638E+001

< 10~20% is better !

Element Group

Energy Distribution



Contents of Output Files (4)

【.Element file】

ADST_1.Element - 記事本

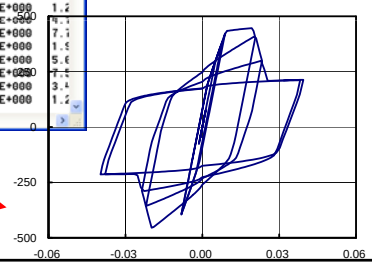
```
Elem.  Yield     Axial        Total        Now_Plas.     Accum._Plastic Extension     Total       Hysteresi
Tag    Code      Force      Extension      Extension      Positive      Negative    StrainEngy    StrainEng
E25      1    -1.234E+003  -2.444E+001   -3.744E+000    5.807E+002   -5.845E+002   1.362E+006   1.585E+00

Joint Element
Elem.  Yield   Internal      Total       Now_Plas.    Accum._Plastic Rotation      Total       Hysteresis
Tag    Code     Moment      Rotation     Rotation      Positive      Negative    StrainEngy    StrainEngy
J2       0    1.159E+006   4.841E-003   3.428E-003    1.057E-002   -7.139E-003   2.221E+004   2.245E+004
J3       0    1.025E+006   4.671E-003   3.441E-003    1.534E-002   -1.190E-002   3.336E+004   3.452E+004
J10      0    1.159E+006   4.839E-003   3.425E-003    1.056E-002   -7.139E-003   2.221E+004   2.244E+004
J11      0    1.025E+006   4.672E-003   3.441E-003    1.534E-002   -1.190E-002   3.336E+004   3.451E+004


Element Response at Time:  30.0000

BeamColumn Element
Elem.  -- -- Plas Hinge Yied Code -- --   BendingMom.  BendingMom.  BendingMom.  BendingMom.  ShearForce
Tag    MzI MzJ MyI MyJ UyI UyJ UzI UzJ     Z_I_Now      Z_J_Now      Y_I_Now      Y_J_Now      Y_I_Now
E13     0   1   0   0   0   0   0   0   -6.887E+005   1.238E+006  -1.404E+000   1.712E+000   7.851E+002

Truss Element
Elem.  Yield     Axial        Total        Now_Plas.     Accum._Plastic Extension     Total       Hysteresi
Tag    Code      Force      Extension      Extension      Positive      Negative    StrainEngy    StrainEng
E25      1    -1.265E+003  -3.039E+001   -9.174E+000    5.807E+002   -5.899E+002   1.370E+006   1.593E+00

Joint Element
Elem.  Yield   Internal      Total       Now_Plas.    Accum._Plastic Rotation      Total       Hysteresis
Tag    Code     Moment      Rotation     Rotation      Positive      Negative    StrainEngy    StrainEngy
J2       0    1.020E+006   4.652E-003   3.428E-003    1.057E-002   -7.139E-003   2.200E+004   2.245E+004
J3       0    6.930E+005   4.216E-003   3.441E-003    1.534E-002   -1.190E-002   3.297E+004   3.452E+004
J10      0    1.020E+006   4.649E-003   3.425E-003    1.056E-002   -7.139E-003   2.200E+004   2.244E+004
J11      0    6.925E+005   4.216E-003   3.441E-003    1.534E-002   -1.190E-002   3.297E+004   3.451E+004
```

Contents of Output Files (5)

【.ElementRecord file】

ADST_1.ElemRecord - 記事本

```
29.400   0   0   0   0   0   0   0   0    1.340E+005  -3.997E+003  -1.650E+000   2.01
29.500   0   0   0   0   0   0   0   0    1.027E+004   3.109E+005  -1.585E+000   1.93
29.600   0   0   0   0   0   0   0   0   -1.679E+005   3.557E+005  -1.531E+000   1.86
29.700   0   0   0   0   0   0   0   0   -3.892E+005   5.149E+005  -1.523E+000   1.85
29.800   0   1   0   0   0   0   0   0   -5.202E+005   9.693E+005  -1.500E+000   1.82
29.900   0   1   0   0   0   0   0   0   -6.352E+005   1.155E+006  -1.450E+000   1.76
30.000   0   1   0   0   0   0   0   0   -6.887E+005   1.238E+006  -1.404E+000   1.71


E25 : Truss Element
Analysis   Yield    Axial        Total        Now_Plas.     Accum._Plastic Extension     T
Time/Step  Code     Force      Extension      Extension      Positive      Negative    Str
   0.100     0    1.635E+000   2.742E-002   0.000E+000    0.000E+000    0.000E+000   2.2
   0.200     0    1.871E+001   3.137E-001   0.000E+000    0.000E+000    0.000E+000   2.5
   0.300     0    4.947E+001   8.296E-001   0.000E+000    0.000E+000    0.000E+000   2.6
   0.400     0    4.939E+001   8.282E-001   0.000E+000    0.000E+000    0.000E+000   2.6
   0.500     0    1.669E+001   2.800E-001   0.000E+000    0.000E+000    0.000E+000   2.3
   0.600     0   -1.223E+001  -2.051E-001   0.000E+000    0.000E+000    0.000E+000   1.2
   0.700     0   -2.386E+001  -4.001E-001   0.000E+000    0.000E+000    0.000E+000   4.1
   0.800     0   -3.044E+001  -5.104E-001   0.000E+000    0.000E+000    0.000E+000   7.7
   0.900     0   -4.874E+001  -8.175E-001   0.000E+000    0.000E+000    0.000E+000   1.9
   1.000     0   -8.219E+001  -1.378E+000   0.000E+000    0.000E+000    0.000E+000   5.6
   1.100     0   -9.489E+001  -1.591E+000   0.000E+000    0.000E+000    0.000E+000   7.5
   1.200     0   -6.375E+001  -1.069E+000   0.000E+000    0.000E+000    0.000E+000   3.4
   1.300     0   -3.808E+001  -6.385E-001   0.000E+000    0.000E+000    0.000E+000   1.2
```



Contents of Output Files (6)

【.ElemEnvelope file】

```
J2      1.455E+006   26.600  -1.275E+006   23.690   7.219E-003   26.600  -2.300E-003
J3      1.545E+006   26.590  -1.326E+006   23.350   9.686E-003   26.590  -3.686E-003
J10     1.455E+006   26.600  -1.275E+006   23.690   7.216E-003   26.600  -2.301E-003
J11     1.545E+006   26.590  -1.326E+006   23.350   9.684E-003   26.590  -3.687E-003

Element Response Envelopes at Time:  26.9000

BeamColumn Element
Elem.    ---- Max. Bending_Moment Z_axis_I_end------    --- Max. Bending_Rotation Z_axis_I_en
Tag       Positive      Time     Negative      Time      Positive      Time     Negative
E13      8.705E+005    15.230  -1.043E+006   25.130   1.865E-003   15.070  -2.423E-003

Truss Element
Elem.    ----------- Maximum Axial Force-----------    ------------ Maximum Extension ------
Tag       Positive      Time     Negative      Time      Positive      Time     Negative
E25      1.330E+003    26.830  -1.364E+003   25.100   7.696E+001   16.050  -1.198E+002

Joint Element
Elem.    --------- Maximum Internal Moment----------    ------------ Maximum Rotation -------
Tag       Positive      Time     Negative      Time      Positive      Time     Negative
J2       1.455E+006   26.600  -1.275E+006   23.690   7.219E-003   26.600  -2.300E-003
J3       1.545E+006   26.590  -1.326E+006   23.350   9.686E-003   26.590  -3.686E-003
J10      1.455E+006   26.600  -1.275E+006   23.690   7.216E-003   26.600  -2.301E-003
J11      1.545E+006   26.590  -1.326E+006   23.350   9.684E-003   26.590  -3.687E-003
```

Contents of Output Files (7)

【.NodeAbsDisp file】

```
PISA3D  Nodal Absolute Displacement File

Nodal Absolute Displacement at Time:  0.1000

Tag        UX          UY          UZ          RX          RY
N14   -1.541E-001   0.000E+000   3.366E-001   0.000E+000  -7.246E-011   0.6
N20   -1.523E-001   0.000E+000   3.700E-001   0.000E+000  -4.887E-011   0.6

Nodal Absolute Displacement at Time:  0.2000

Tag        UX          UY          UZ          RX          RY
N14    1.292E-001   0.000E+000   7.517E-001   0.000E+000   1.403E-009   0.6
N20    1.015E-001   0.000E+000   1.113E+000   0.000E+000   1.738E-010   0.6

Nodal Absolute Displacement at Time:  0.3000

Tag        UX          UY          UZ          RX          RY
N14   -3.352E-001   0.000E+000   1.110E-001   0.000E+000   1.329E-010   0.6
N20   -3.520E-001   0.000E+000   1.054E+000   0.000E+000   2.870E-009   0.6

Nodal Absolute Displacement at Time:  0.4000

Tag        UX          UY          UZ          RX          RY
N14   -3.420E-001   0.000E+000   5.409E-002   0.000E+000   1.126E-008   0.6
N20   -4.911E-001   0.000E+000   9.954E-001   0.000E+000   4.929E-009   0.6

Nodal Absolute Displacement at Time:  0.5000
```

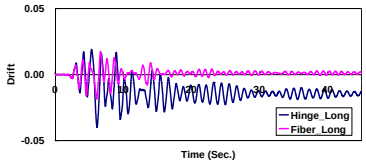
Contents of Output Files (8)

【.NodeDisRecord file】

ADST_1.NodeDisRecord - 記事本

檔案(F) 編輯(E) 格式(O) 檢視(V) 說明(H)

```
N14      29.300    4.516E+000   0.000E+000   7.664E+000   0.000E+000   3.342E
N14      29.400   -8.328E+000   0.000E+000   6.349E-001   0.000E+000   4.035E
N14      29.500   -1.046E+001   0.000E+000  -1.219E+001   0.000E+000   1.187E
N14      29.600   -2.046E+001   0.000E+000  -6.958E+000   0.000E+000   1.035E
N14      29.700   -3.679E+001   0.000E+000  -2.019E+000   0.000E+000   1.018E
N14      29.800   -4.727E+001   0.000E+000  -1.801E+001   0.000E+000   1.608E
N14      29.900   -5.880E+001   0.000E+000  -4.425E+001   0.000E+000   1.915E
N14      30.000   -6.079E+001   0.000E+000  -6.183E+001   0.000E+000   1.870E


Tag   Time/Step       UX           UV           UZ           RX           RY
N20       0.100   -1.523E-001   0.000E+000   3.700E-001   0.000E+000  -4.887E
N20       0.200    1.015E-001   0.000E+000   1.113E+000   0.000E+000   1.738E
N20       0.300   -3.520E-001   0.000E+000   1.054E+000   0.000E+000   2.870E
N20       0.400   -4.911E-001   0.000E+000   9.954E-001   0.000E+000   4.929E
N20       0.500   -5.780E-001   0.000E+000   6.630E-001   0.000E+000   2.330E
N20       0.600   -3.723E-001   0.000E+000  -4.512E-002   0.000E+000   7.420E
N20       0.700   -6.367E-001   0.000E+000  -7.598E-001   0.000E+000   1.398E
N20       0.800   -6.620E-001   0.000E+000  -1.328E+000   0.000E+000   1.871E
N20       0.900   -6.991E-001   0.000E+000  -2.141E+000   0.000E+000   1.988E
N20       1.000   -4.888E-001   0.000E+000  -2.618E+000   0.000E+000   1.697E
N20       1.100   -4.576E-001   0.000E+000  -2.295E+000   0.000E+000   1.184E
N20       1.200   -1.127E+000   0.000E+000  -2.432E+000   0.000E+000   6.295E
N20       1.300   -1.600E+000   0.000E+000  -1.797E+000   0.000E+000   2.544E
N20       1.400   -9.430E-001   0.000E+000  -5.538E-001   0.000E+000   4.732E
N20       1.500    1.768E-001   0.000E+000   7.490E-001   0.000E+000   1.428E
N20       1.600    1.434E+000   0.000E+000   1.723E+000   0.000E+000   2.371E
N20       1.700    3.030E+000   0.000E+000   2.770E+000   0.000E+000   2.071E
N20       1.800    4.242E+000   0.000E+000   3.183E+000   0.000E+000  -5.679E
N20       1.900    4.440E+000   0.000E+000   2.419E+000   0.000E+000  -5.750E
N20       2.000    3.836E+000   0.000E+000   7.770E-001   0.000E+000  -1.196E
N20       2.100    3.302E+000   0.000E+000  -4.415E-001   0.000E+000  -1.639E
N20       2.200    3.361E+000   0.000E+000  -1.155E+000   0.000E+000  -1.676E
```



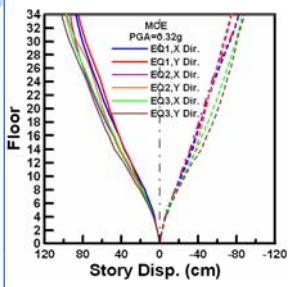
Contents of Output Files (9)

【.NodeEnvelope file】

ADST_1.NodeEnvelope - 記事本

檔案(F) 編輯(E) 格式(O) 檢視(V) 說明(H)

```
Time:  22.6000

Node  ------------- UX Displacement -------------    ------------- UY Displacement ----
Tag    Positive       Time     Negative      Time      Positive       Time     Negative
N14   1.716E+002    22.400   -3.758E+002   19.600    0.000E+000     0.000   0.000E+000
N20   3.682E+002    22.550   -8.097E+002   19.540    0.000E+000     0.000   0.000E+000

Node  ------------- UX Velocity -----------------    ------------- UY Velocity --------
Tag    Positive       Time     Negative      Time      Positive       Time     Negative
N14   3.958E+002    22.170   -3.888E+002   16.930    0.000E+000     0.000   0.000E+000
N20   6.123E+002    22.180   -6.751E+002   18.830    0.000E+000     0.000   0.000E+000

Node  ------------- UX Acceleratoin -------------    ------------- UY Acceleratoin ----
Tag    Positive       Time     Negative      Time      Positive       Time     Negative
N14   3.122E+003    16.440   -3.222E+003   18.380    0.000E+000     0.000   0.000E+000
N20   2.831E+003    15.960   -3.046E+003   22.440    0.000E+000     0.000   0.000E+000


Time:  22.7000

Node  ------------- UX Displacement -------------    ------------- UY Displacement ----
Tag    Positive       Time     Negative      Time      Positive       Time     Negative
N14   1.716E+002    22.400   -3.758E+002   19.600    0.000E+000     0.000   0.000E+000
N20   3.682E+002    22.550   -8.097E+002   19.540    0.000E+000     0.000   0.000E+000

Node  ------------- UX Velocity -----------------    ------------- UY Velocity --------
Tag    Positive       Time     Negative      Time      Positive       Time     Negative
N14   3.958E+002    22.170   -3.888E+002   16.930    0.000E+000     0.000   0.000E+000
N20   6.123E+002    22.180   -6.751E+002   18.830    0.000E+000     0.000   0.000E+000

Node  ------------- UX Acceleratoin -------------    ------------- UY Acceleratoin ----
```



Contents of Output Files (10)

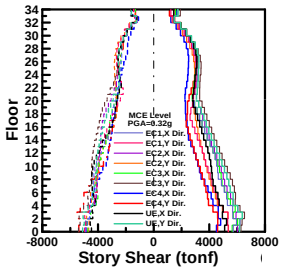
【.SectionX/Y/Z file】

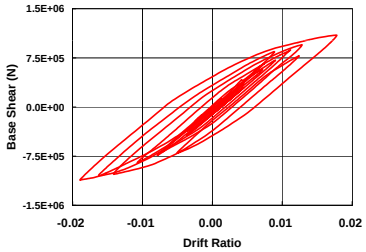
FIBERCONTEST4_SectionX - 記事本

檔案(F) 編輯(E) 格式(O) 檢視(V) 說明(H)

PISA3D X-Direction Group Force File

Time/Step	Group-1F	Group-2F	Group-3F	Group-4F
0	1.017E+001	5.821E-001	7.214E-002	1.445E-002
0.010	2.484E+000	-4.306E-001	7.083E-002	-2.573E-003
0.020	1.704E+002	-3.125E+001	2.319E+000	-4.843E-001
0.030	3.737E+002	-4.611E+001	-2.400E+000	1.491E-001
0.040	3.383E+000	7.925E+001	-2.375E+001	3.105E+000
0.050	-1.039E+003	2.994E+002	-3.674E+001	3.028E+000
0.060	-2.306E+003	4.143E+002	3.061E+000	-6.668E+000
0.070	-3.005E+003	1.867E+002	1.187E+002	-2.396E+001
0.080	-2.649E+003	-4.443E+002	2.689E+002	-3.146E+001
0.090	-1.304E+003	-1.299E+003	3.523E+002	-3.368E+000
0.100	6.849E+002	-2.072E+003	2.579E+002	7.620E+001
0.110	3.173E+003	-2.506E+003	-7.586E+001	1.957E+002
0.120	6.107E+003	-2.412E+003	-6.316E+002	3.117E+002
0.130	9.090E+003	-1.616E+003	-1.323E+003	3.580E+002
0.140	1.144E+004	-8.582E+000	-1.990E+003	2.598E+002
0.150	1.252E+004	2.343E+003	-2.414E+003	-4.443E+001
0.160	1.228E+004	5.103E+003	-2.357E+003	-5.775E+002
0.170	1.131E+004	7.735E+003	-1.634E+003	-1.296E+003
0.180	1.052E+004	9.733E+003	-2.054E+002	-2.074E+003
0.190	1.038E+004	1.086E+004	1.781E+003	-2.711E+003
0.200	1.069E+004	1.125E+004	4.015E+003	-2.970E+003
0.210	1.091E+004	1.121E+004	6.134E+003	-2.642E+003
0.220	1.089E+004	1.098E+004	7.844E+003	-1.610E+003
0.230	1.087E+004	1.067E+004	8.991E+003	1.217E+002
0.240	1.108E+004	1.035E+004	9.568E+003	2.416E+003
0.250	1.152E+004	1.009E+004	9.696E+003	5.032E+003
0.260	1.222E+004	9.910E+003	9.585E+003	7.663E+003
0.270	1.327E+004	9.819E+003	9.470E+003	9.997E+003
0.280	1.429E+004	9.929E+003	9.538E+003	1.177E+004





國家實驗研究院

Thank you.

www.narlabs.org.tw