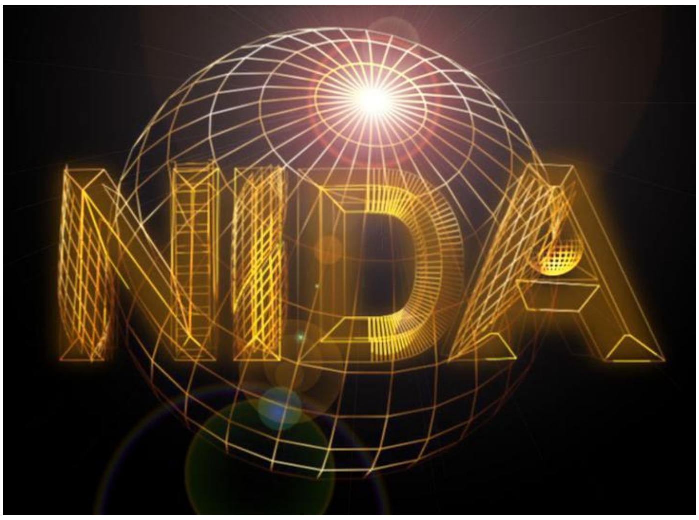
Welcome to NIDA Version 9.0

NIDA - Nonlinear Integrated Design and Analysis

NIDA Tutorials

Step-by-Step



Since 1996

Second-Order (Direct) Elastic & Plastic Analysis

To

Eurocode 3 (2005), AISC-LRFD (2010) and CoPHK (2011)

Without Assumption of Effective Length

Add: The Hong Kong Polytechnic University,
Hung Hom, Kowloon, Hong Kong, P.R. China

Tel: (852) 2766 6047
Fax: (852) 2334 6389
Email: ceslchan@polyu.edu.hk
Web: http://www.nidacse.com/

Table of Contents

1. TUTORIAL 1 - SIMPLE BENCHMARK EXAMPLE FOR TESTING OF SOFTWARE - A STRUT UNDER AXIAL FORCE 1

1.1 STEP 1 – START NIDA 1
1.2 STEP 2 – CREATE A NEW PROJECT FILE 1
1.3 STEP 3 – CREATE MATERIAL 4
1.4 STEP 4 – ADD NODES 5
1.5 STEP 5 – ADD SECTIONS 6
1.6 STEP 6 – ADD MEMBERS 9
1.7 STEP 7 – ADD BOUNDARY CONDITIONS 11
1.8 STEP 8 – SET VISIBLE/INVISIBLE 12
1.9 STEP 9 – ADD LOAD CASES 13
1.10 STEP 10 – ADD LOADINGS 14
1.11 STEP 11 – SET VISIBILITY OF LOADINGS 15
1.12 STEP 12 – SET AND RUN ANALYSIS CASES 16
1.13 STEP 13 – VIEW RESULT: STRUCTURAL DEFORMATION 19
1.14 STEP 14 – VIEW RESULT: SHOW LOAD-DEFLECTION CURVE 20

2. TUTORIAL 2 – A BRACED AND AN UNBRACED PORTAL – SECOND ORDER ANALYSIS AND ELASTIC CRITICAL LOAD FACTOR ANALYSIS 22

2.1 STEP 1 – START NIDA 22
2.2 STEP 2 – CREATE A NEW PROJECT FILE 23
2.3 STEP 3 – CREATE MATERIAL 25
2.4 STEP 4 – ADD NODES 26
2.5 STEP 5 – ADD SECTIONS 27
2.6 STEP 6 – ADD MEMBERS 32
2.7 STEP 7 – ADD BOUNDARY CONDITIONS 34
2.8 STEP 8 – SET VISIBLE/INVISIBLE 35
2.9 STEP 9 – ADD LOAD CASES 36
2.10 STEP 10 – ADD LOADINGS 38
2.11 STEP 11 – SET VISIBILITY OF LOADINGS 39
2.12 STEP 12 – SET AND RUN ANALYSIS CASES 40
2.13 STEP 13 – VIEW RESULT: EIGEN-BUCKLING MODE 45
2.14 STEP 14 – VIEW RESULT: NONLINEAR ANALYSIS 48
2.15 STEP 15 – VIEW RESULT: SHOW ANALYSIS PROCEDURE OF NONLINEAR ANALYSIS 50
2.16 STEP 16 – DELETE MEMBERS FOR UNBRACED CASE 52
2.17 STEP 17 – UNBRACED PORTAL FRAME 55

3. TUTORIAL 3 – SPACE FRAME 56

3.1 STEP 1 – START NIDA 56
3.2 STEP 2 – CREATE A NEW PROJECT FILE 57
3.3 STEP 3 – CREATE MATERIAL 58
3.4 STEP 4 – SET GRAVITY DIRECTION 60
3.5 STEP 5 – ADD NODES 61

3.6 Step 6 – Copy and paste nodes 64
3.7 Step 7 – Add sections 67
3.8 Step 8 – Add members 71
3.9 Step 9 – Add boundary conditions 75
3.10 Step 10 – Set visible/invisible 77
3.11 Step 11 – Rotate member local axis 78
3.12 Step 12 – Add load cases 83
3.13 Step 13 – Add loadings 85
3.14 Step 14 – Set visibility of loadings 91
3.15 Step 15 – Add load combinations 92
3.16 Step 16 – Set and run analysis cases 95
3.17 Step 17 – View result: eigen-buckling mode 104
3.18 Step 18 – View result: View another analysis case 108
3.19 Step 19 – View result: check the structural adequacy by second-order analysis (I) Export Statistics of Analysis Results 110
3.20 Step 20 – View result: check the structural adequacy by second-order analysis (II) Export Summary of Analysis Results 114
3.21 Step 21 – View result: check the structural adequacy by second-order analysis (III) Show Section Capacity Factor Statistics 116

4. TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS 124

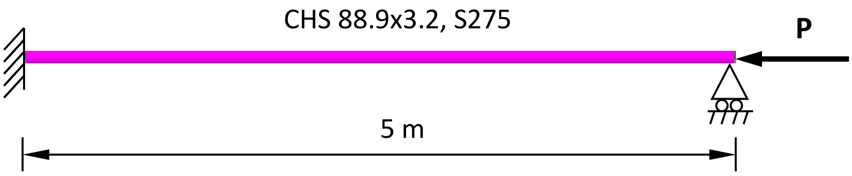
4.1 Step 1 – Build the structural model 124
4.2 Step 2 – Define a response spectrum function 124
4.3 Step 3 – Define a modal analysis 126
4.4 Step 4 – Create “single direction” MRSA cases 128
4.5 Step 5 – Create “directional combination” MRSA cases 129
4.6 Step 6 – Create “load combination” considering both earthquake action and other actions 131
4.7 Step 7 – Run all analysis cases and view results 134

5. TUTORIAL 5 – TIME HISTORY ANALYSIS 136

5.1 Step 1 – Build the structural model 137
5.2 Step 2 – Define one or more than one time history functions 137
5.3 Step 3 – Define a time history analysis case 138
5.4 Step 4 – Run time history analysis case(s) 142
5.5 Step 5 – View result: nodal displacements and base shear 142
5.6 Step 6 – View result: plastic hinge rotation of all time steps 144
5.7 Step 7 – View result: plastic hinge location 145
5.8 Step 8 – View and export results 146

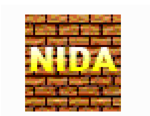
1. TUTORIAL 1 - Simple Benchmark Example for Testing of Software - A Strut under Axial Force

The column of CHS 88.9x3.2, grade S275 steel and length 5m has a boundary condition as one end pin and one end fixed. Determine the axial load resistance and buckling load of the column by second order analysis. Do not assume effective length for the column as it is unknown for most compression members in real frames.



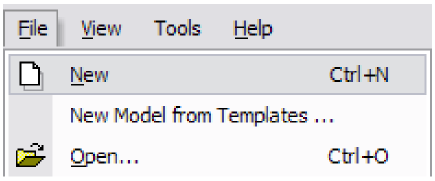
1.1 Step 1 – Start NIDA

Double click the "NIDA" icon on the desktop or in the corresponding folder upon successful installation of NIDA.



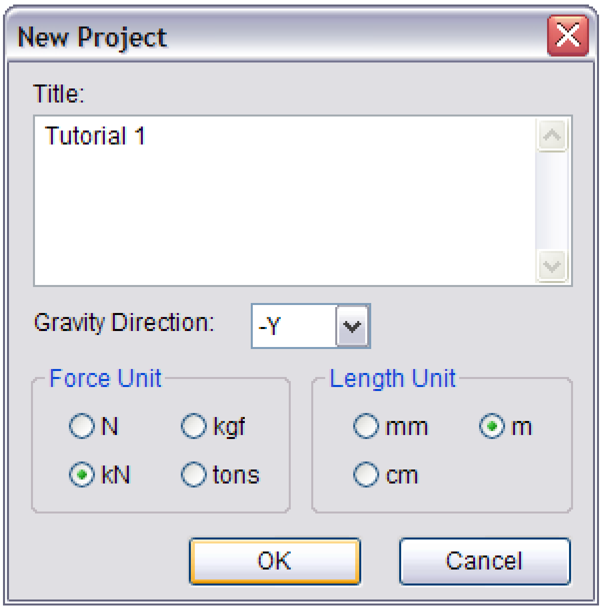
1.2 Step 2 – Create a new project file

Click the ***File > New*** command,



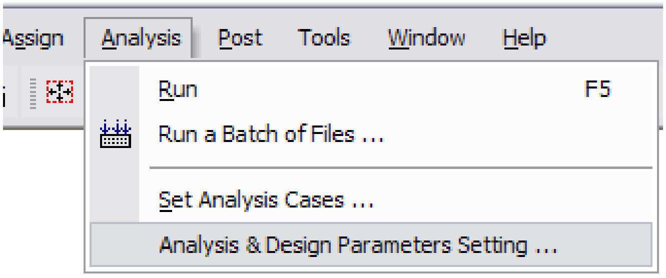
or find the <New> shortcut button in the toolbar;

Enter the new project <Title> 'Tutorial 1' in the popped window and select the <Gravity Direction> to be '-Y', <Force Unit> 'kN' and <Length Unit> 'm'.



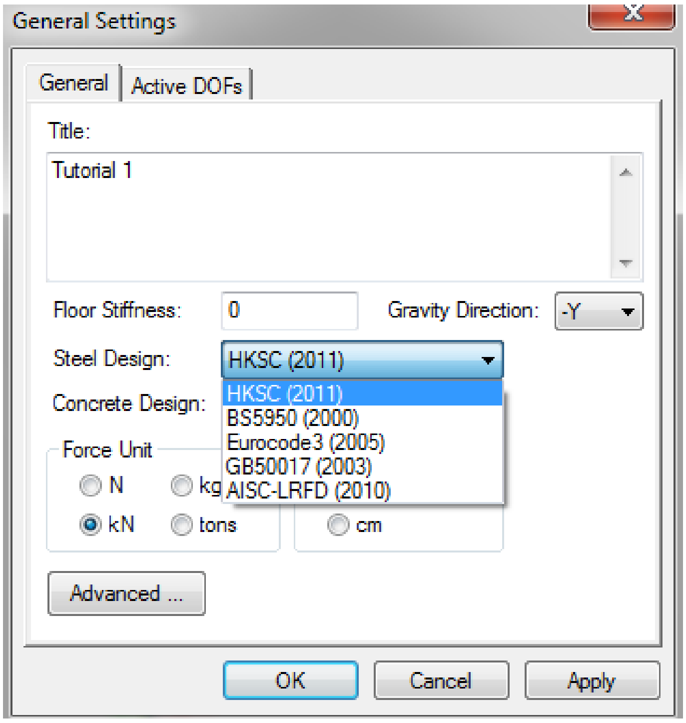
Click ***OK*** to continue.

Click the ***Analysis*** > ***Analysis & Design Parameters setting*** command,

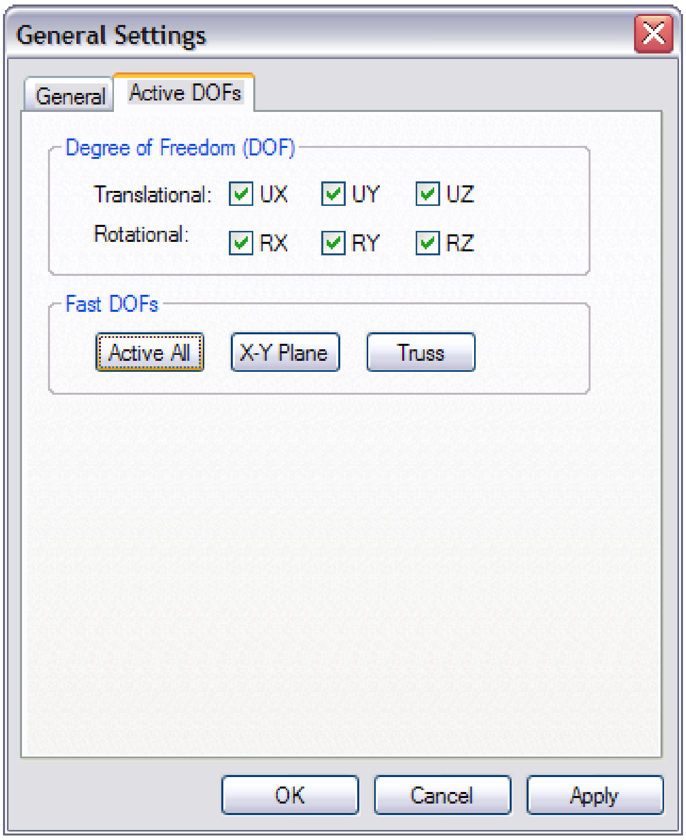


The parameters already set could also be viewed or changed in the <General Settings> window;

Select the steel design code HKSC (2011);



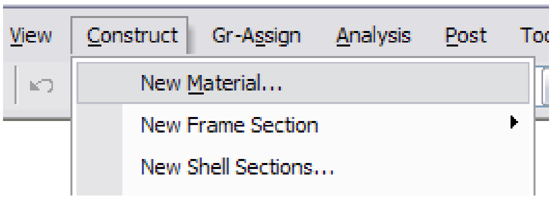
Activate all the degrees of freedom under 'Active DOFs';



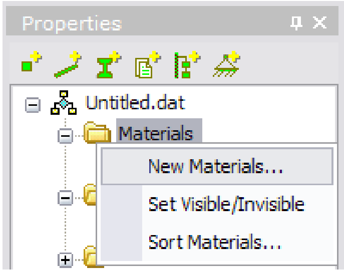
Click ***Apply*** and then ***OK*** to continue.

1.3 Step 3 – Create material

Click the ***Construct* > *New Material*** command,

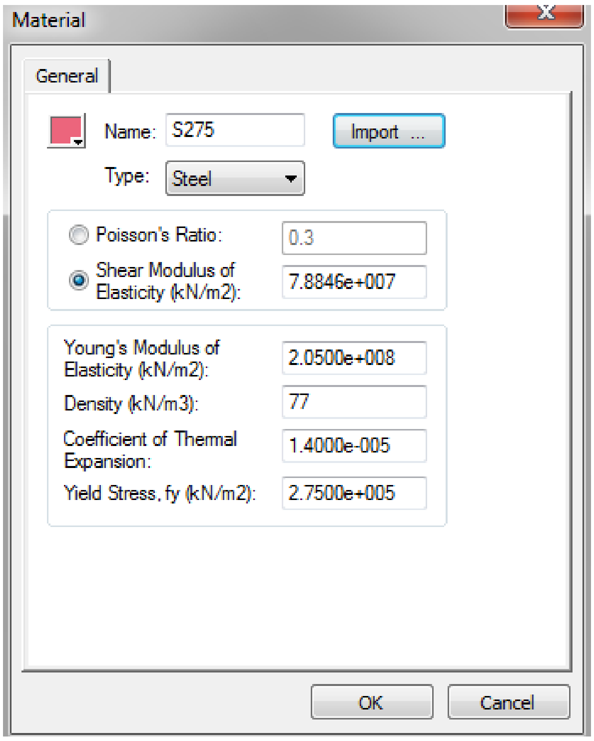


or right click <Materials> in the ***Properties*** window and then click <New Materials>;



Default steel grade set up in the program is S275;

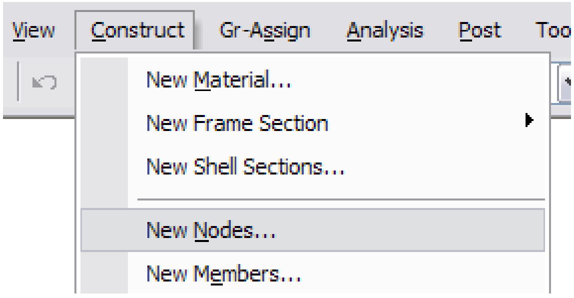
Double click <S275> to view and change the material properties in the <Materials> window if required, such as changing the material colour only for this tutorial;



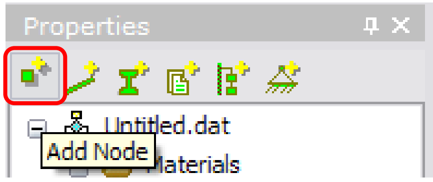
Click ***OK*** to continue.

1.4 Step 4 – Add nodes

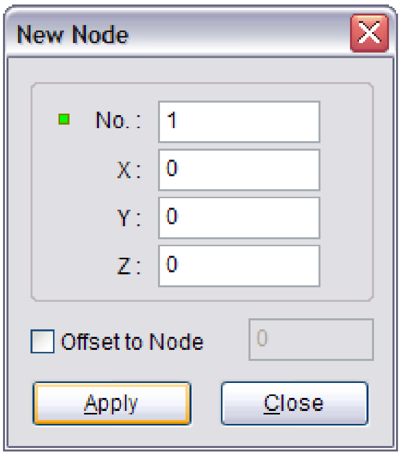
Click the ***Construct*** > ***New Nodes*** command,

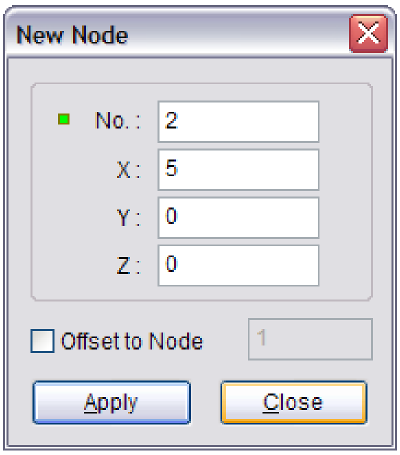


or click the <Add Node> shortcut in the ***Properties*** window;



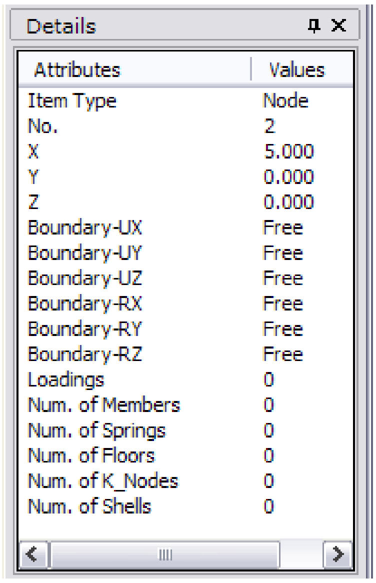
Enter the x, y and z coordinates for node No. 1 and click ***Apply*** to continue adding nodes;



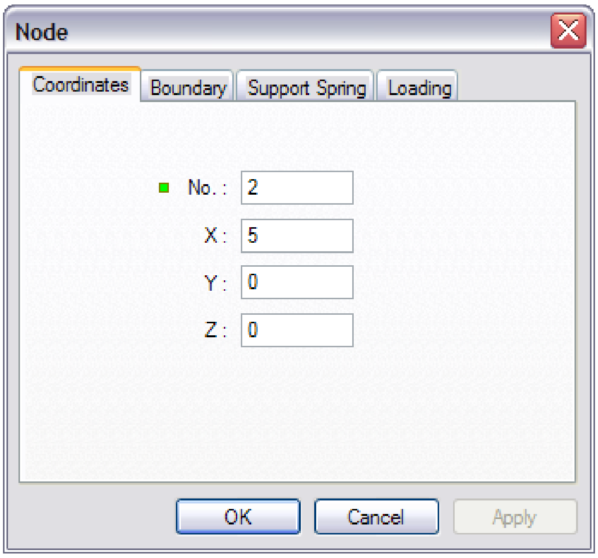


On completion of the last node construction, click ***Apply*** and then ***Close*** to finish this step.

To view the details of a particular node, click the node, then all the details including coordinates, boundary conditions, loadings, etc, are displayed in the ***Details*** window;



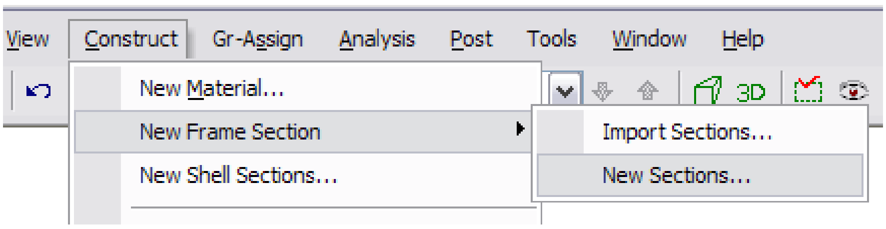
or double click the node to view the node details under ‘Coordinates’, ‘Boundary’, ‘Support Spring’ and ‘Loading’ menus, make changes in the ***Node*** window if needed;



Click ***OK*** to continue.

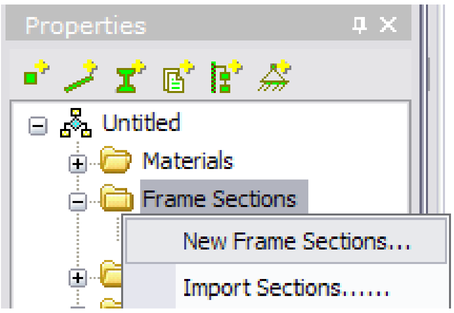
1.5 Step 5 – Add sections

Click the ***Construct*** > ***New Frame Section*** > ***New Sections*** command,

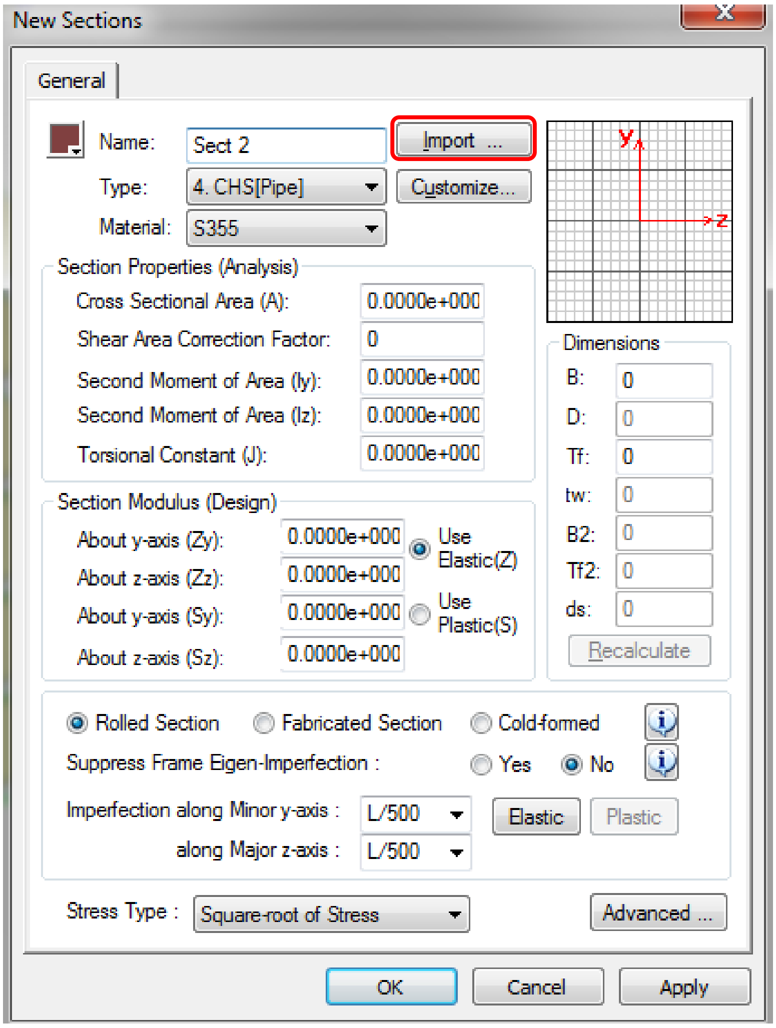


or click the <Add Section> shortcut in the ***Properties*** window;

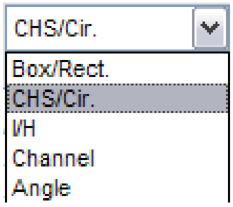
or right click <Frame Sections> in the ***Properties*** window and then click <New Frame Sections>;



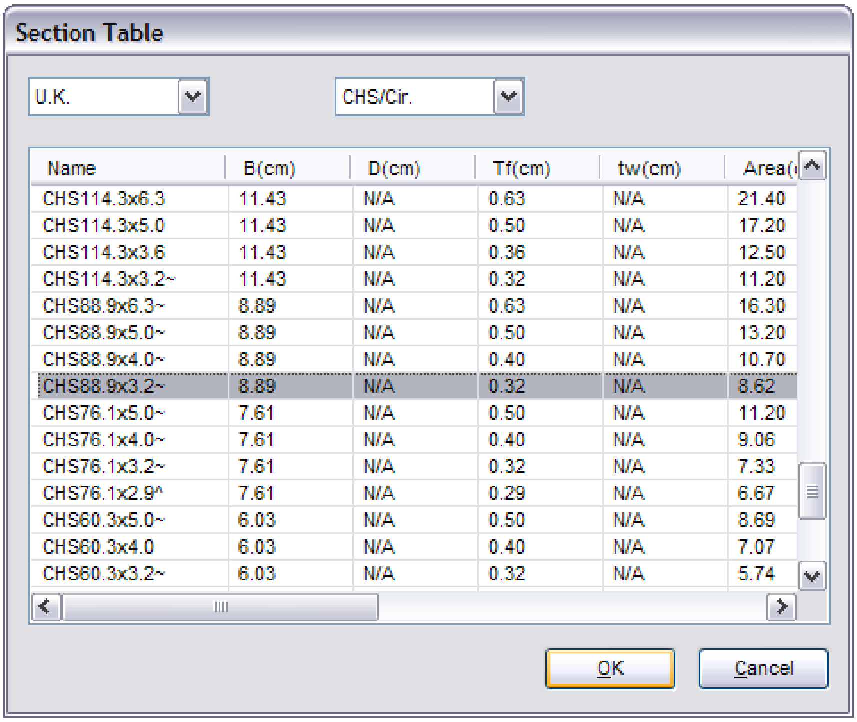
Click <Import> button in the <New Sections> window to import a new cross section;



Select cross sections under different categories, country or shape;



Select ‘CHS 88.9x3.2~’ in the <Section Table>;



Name	B(cm)	D(cm)	Tf(cm)	tw(cm)	Area(
CHS114.3x6.3	11.43	N/A	0.63	N/A	21.40
CHS114.3x5.0	11.43	N/A	0.50	N/A	17.20
CHS114.3x3.6	11.43	N/A	0.36	N/A	12.50
CHS114.3x3.2~	11.43	N/A	0.32	N/A	11.20
CHS88.9x6.3~	8.89	N/A	0.63	N/A	16.30
CHS88.9x5.0~	8.89	N/A	0.50	N/A	13.20
CHS88.9x4.0~	8.89	N/A	0.40	N/A	10.70
CHS88.9x3.2~	8.89	N/A	0.32	N/A	8.62
CHS76.1x5.0~	7.61	N/A	0.50	N/A	11.20
CHS76.1x4.0~	7.61	N/A	0.40	N/A	9.06
CHS76.1x3.2~	7.61	N/A	0.32	N/A	7.33
CHS76.1x2.9^	7.61	N/A	0.29	N/A	6.67
CHS60.3x5.0~	6.03	N/A	0.50	N/A	8.69
CHS60.3x4.0	6.03	N/A	0.40	N/A	7.07
CHS60.3x3.2~	6.03	N/A	0.32	N/A	5.74

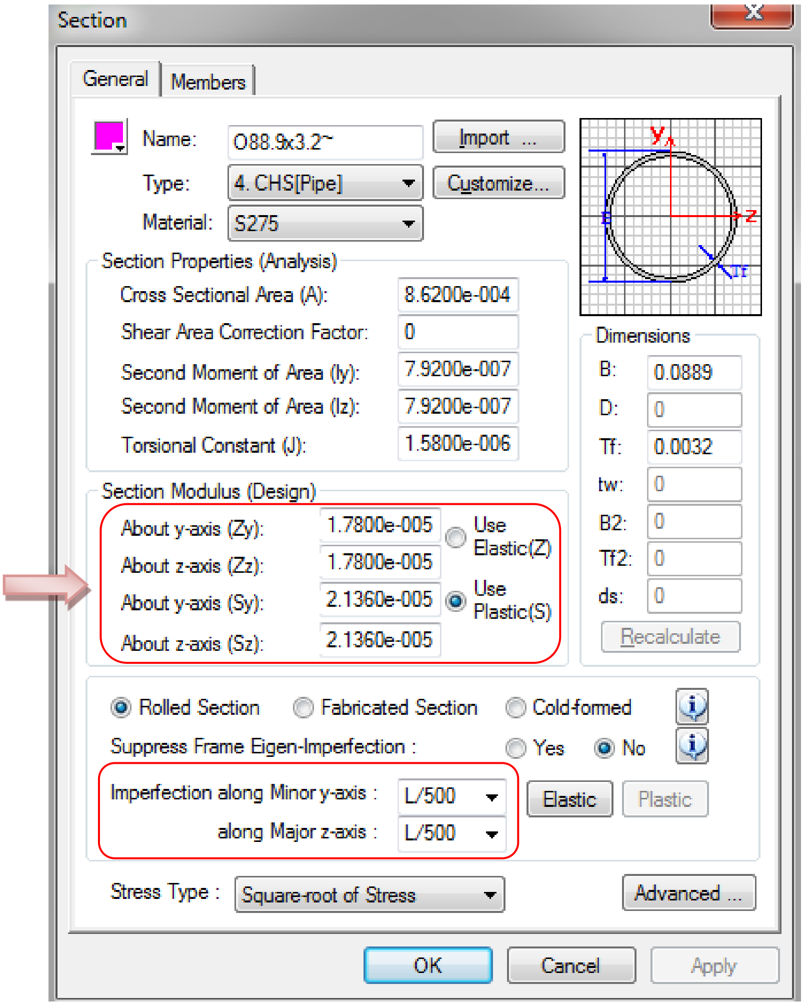
Click ***OK***, then all the section properties are displayed in the <New Sections> window;

Select a different colour for each section, check all the section properties and make changes if required;

Change the plastic modulus S about both axes from 2.3500e-005 to 2.1360e-005;

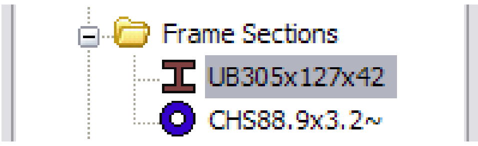
Select ‘Use Plastic (S)’ for second order analysis;

P – δ imperfections of the member in this section are generated automatically by the software according to the design code in use, but could be manually adjusted as well to fulfil special requirement.



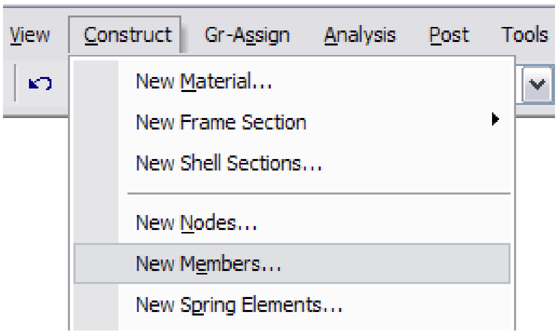
Click ***Apply*** to continue importing different sections and ***OK*** to finish adding sections.

There is a default frame section 'UB 305 x 127 x 42' generated by the software. Delete it or change the properties if needed.



1.6 Step 6 – Add members

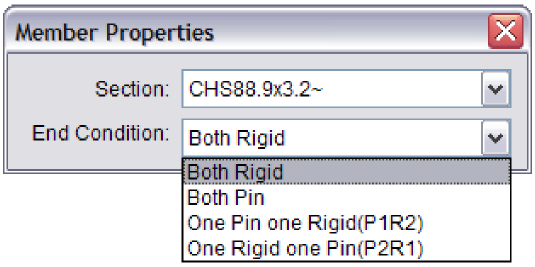
Click the ***Construct*** > ***New Members*** command,



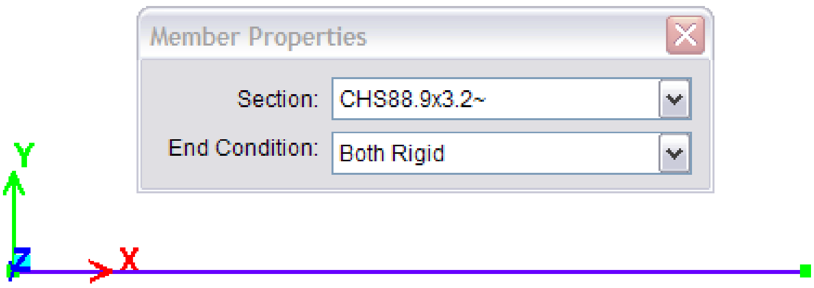
or click the <Add Member> shortcut in the ***Properties*** window;

or click the <Add Member> shortcut in the toolbar;

Select the ‘Section’ and ‘End Condition’ in the < Member Properties > window;



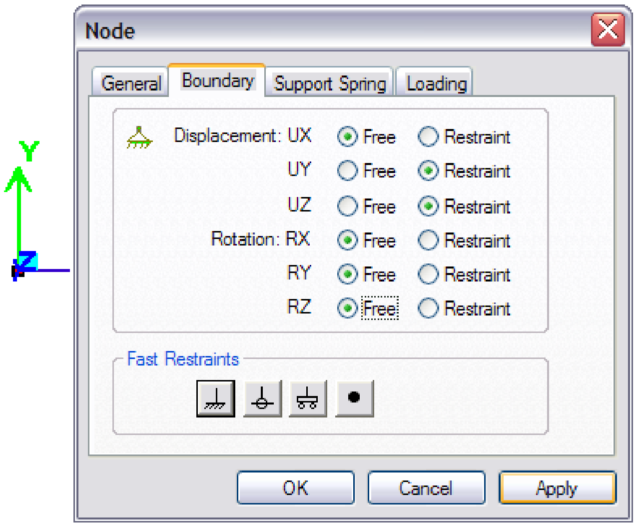
Click the start node and then the end node; if this end node is connected to other members, continue to click the next end node and right click to stop adding members.



Click ***Close*** in dialog or press ***Esc*** in keyboard to finish this step.

1.7 Step 7 – Add boundary conditions

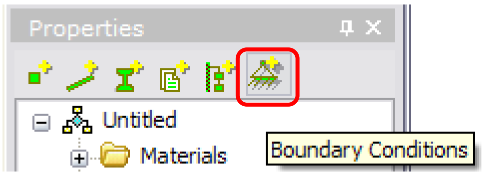
Double click the boundary node, set the restraints of displacement in Y and Z axes manually in the ***Node*** window for this pin end node;



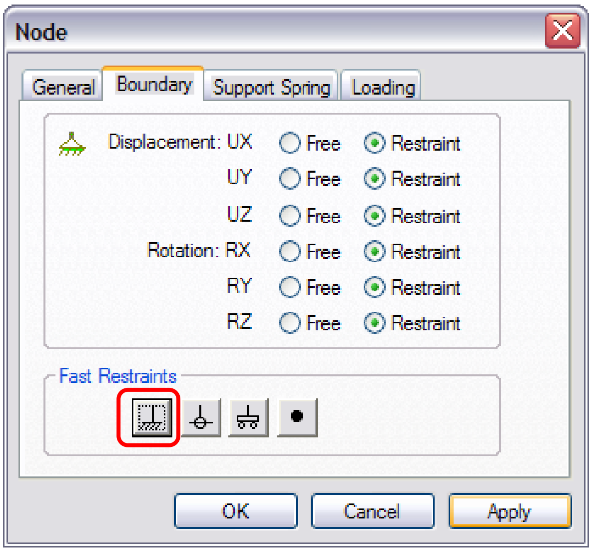
Click ***Apply*** and ***OK*** to continue;

or select the node to be restrained;

and click the <Boundary Conditions> shortcut in the ***Properties*** window;



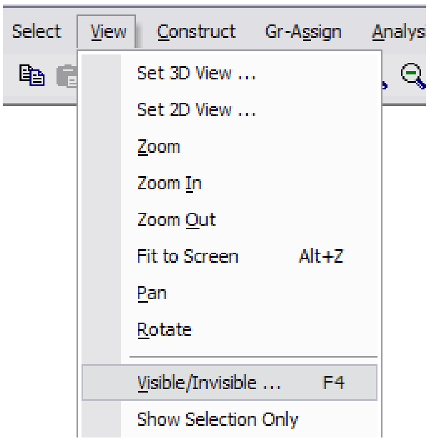
Set the other end node to be fixed by using ‘Fast Restraints’;



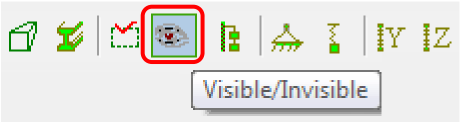
Click ***Apply*** and ***OK*** to finish this step.

1.8 Step 8 – Set visible/invisible

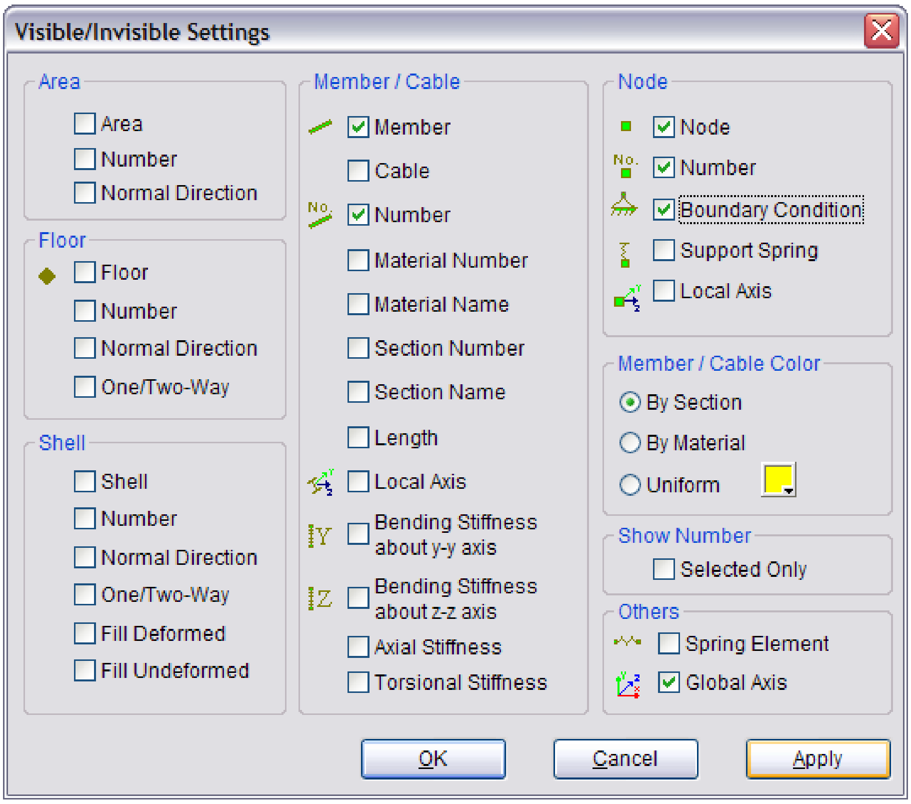
Click the ***View*** > ***Visible/Invisible*** command,



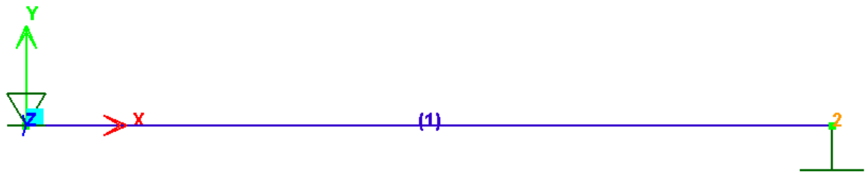
or click the <Visible/Invisible> shortcut in the toolbar;



Tick 'Member', 'Member-Number', 'Node', 'Node-Number', 'Node-Boundary Condition', 'Global Axis' and select 'By Section' in the <Visible/Invisible Settings> window;

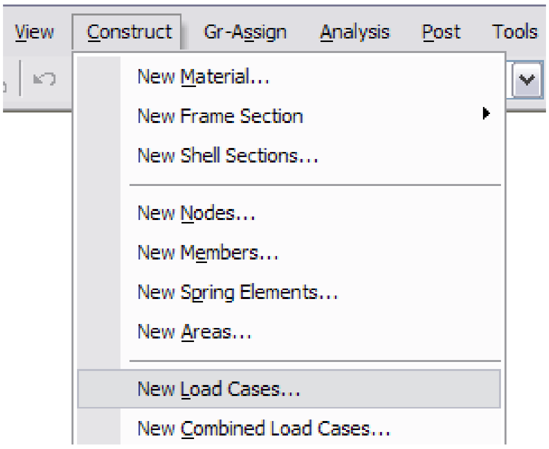


Click ***Apply*** and ***OK*** to view the structure with properties constructed.

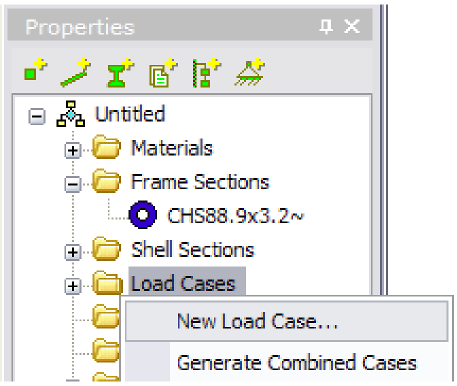


1.9 Step 9 – Add load cases

Click the ***Construct*** > ***New Load Cases*** command,

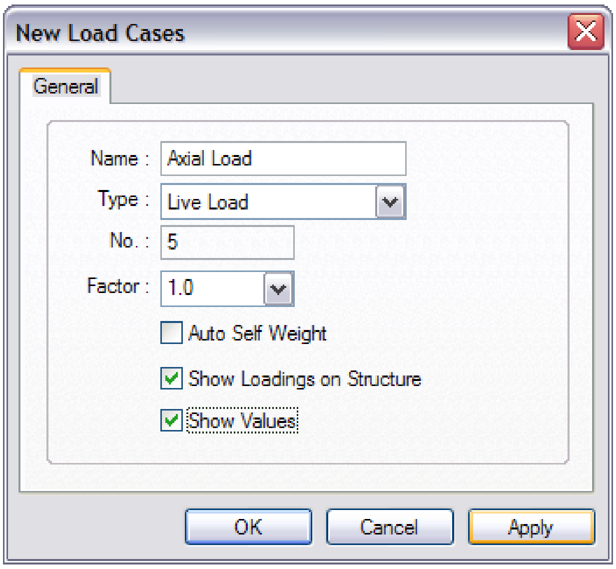


or right click <Load Cases> in the ***Properties*** window and then click <New Load Case>;



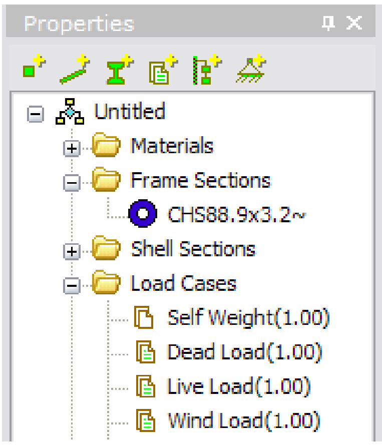
or click the <Add Load Case> shortcut in the ***Properties*** window;

Enter the 'Name' of new load case, select the load 'Type' and 'Factor' in the <New Load Cases> window;



Click ***Apply*** to continue adding new load cases and ***OK*** to finish this step.

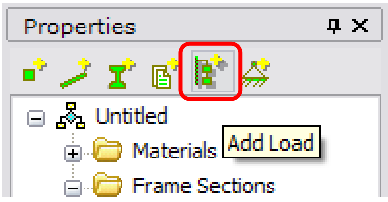
There are four load cases set as default in the software;



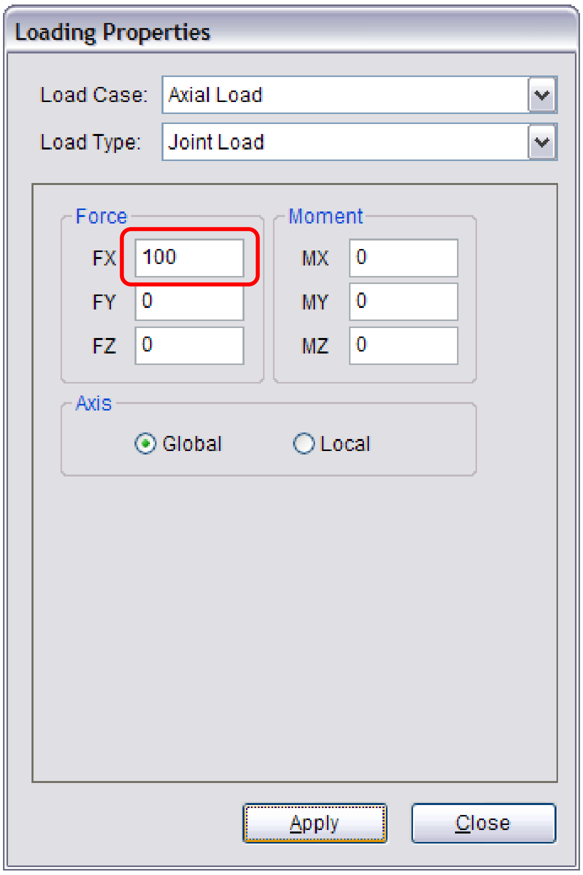
Double click any load case to view or change the settings of this load case or right click and select 'delete' as requirement.

1.10 Step 10 – Add loadings

Select node No. 1 and click the <Add Load> shortcut in the ***Properties*** window;



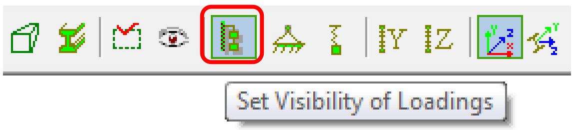
Select 'Load Case', 'Load Type', 'Axis' and enter 100 kN 'Force' in positive X direction in the <Loading Properties> window;



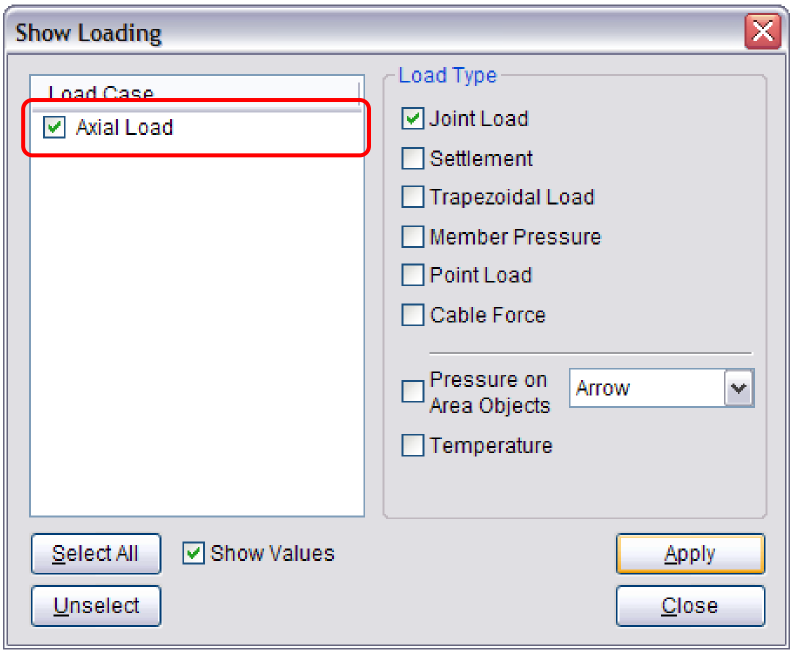
Click ***Apply*** to continue.

1.11 Step 11 – Set visibility of loadings

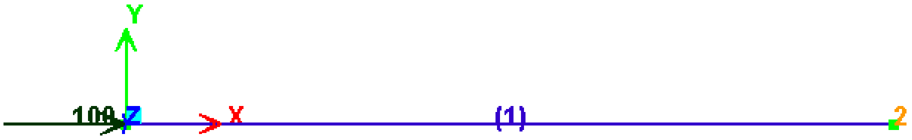
Click the <Set Visibility of Loadings> shortcut in the toolbar;



Select the 'Load Case', 'Load Type' and tick 'Show Values' in the <Show Loading> window;

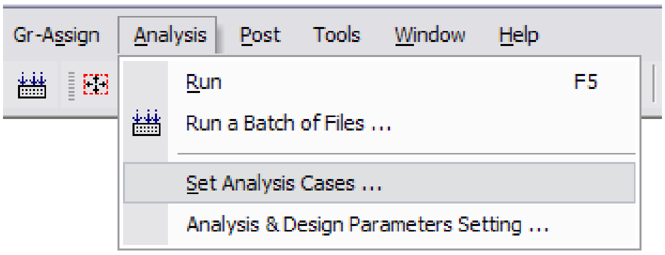


Click ***Apply*** and ***Close*** to finish setting. Loading is then visible.

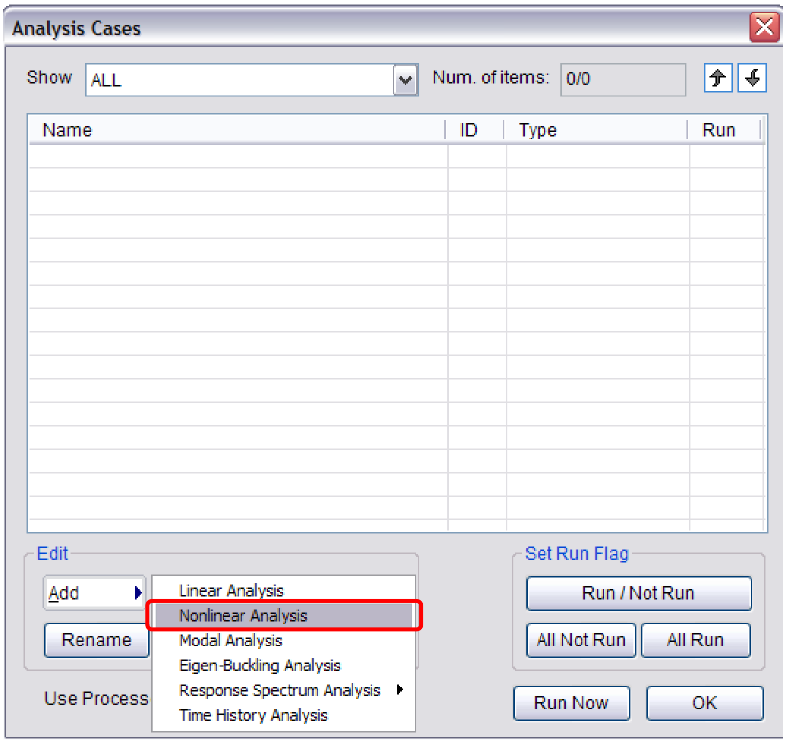


1.12 Step 12 – Set and run analysis cases

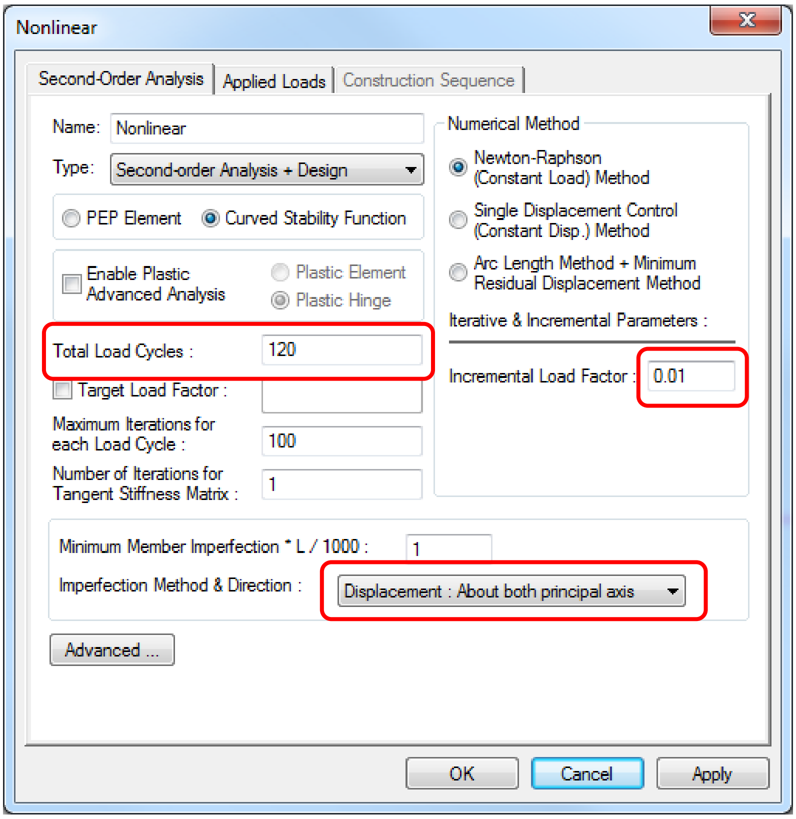
Click the ***Analysis*** > ***Set Analysis Cases*** command;



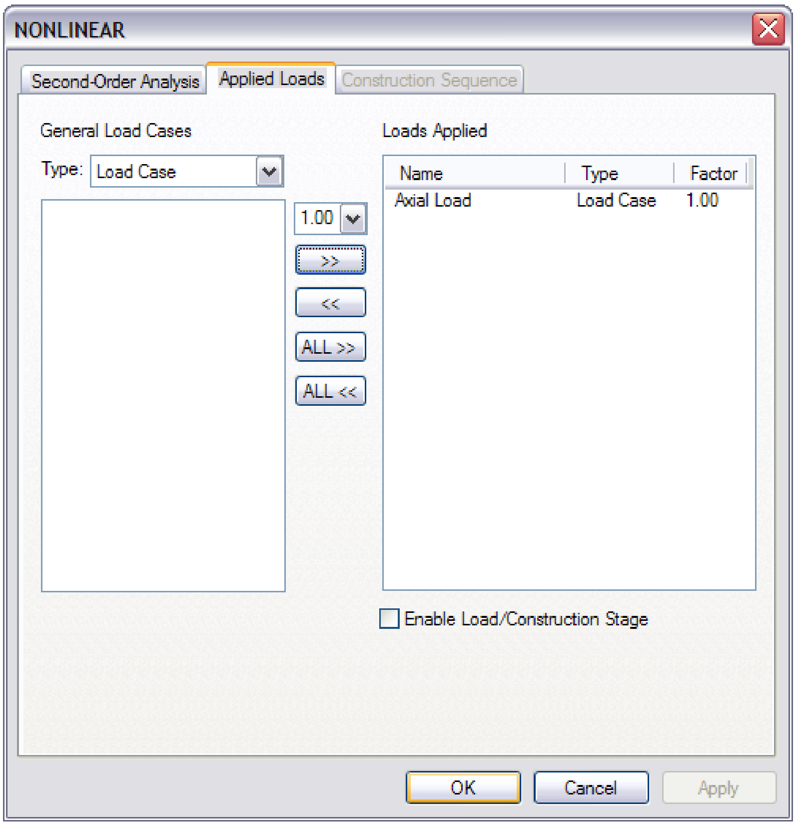
Add 'Nonlinear Analysis' to determine axial load resistance;



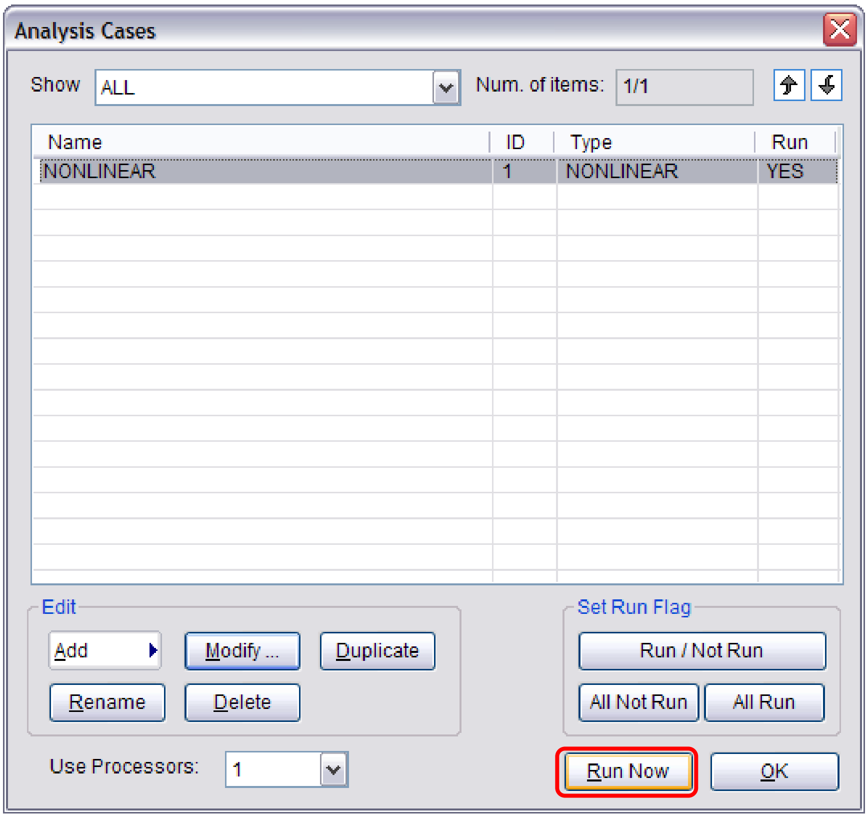
Select ‘Nonlinear Analysis + Design’, enter the ‘Total Load Cycles’ to be 120, ‘Incremental Load Factor’ to be 0.01 and select ‘Imperfection Method & Direction’ in the <NONLINEAR> window;



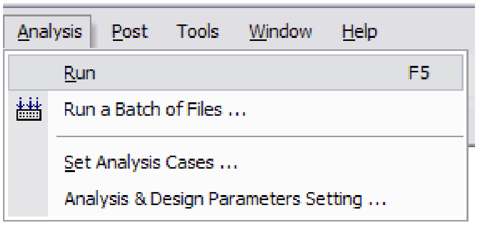
Apply ‘Load Case – Axial Load’ to this analysis case;



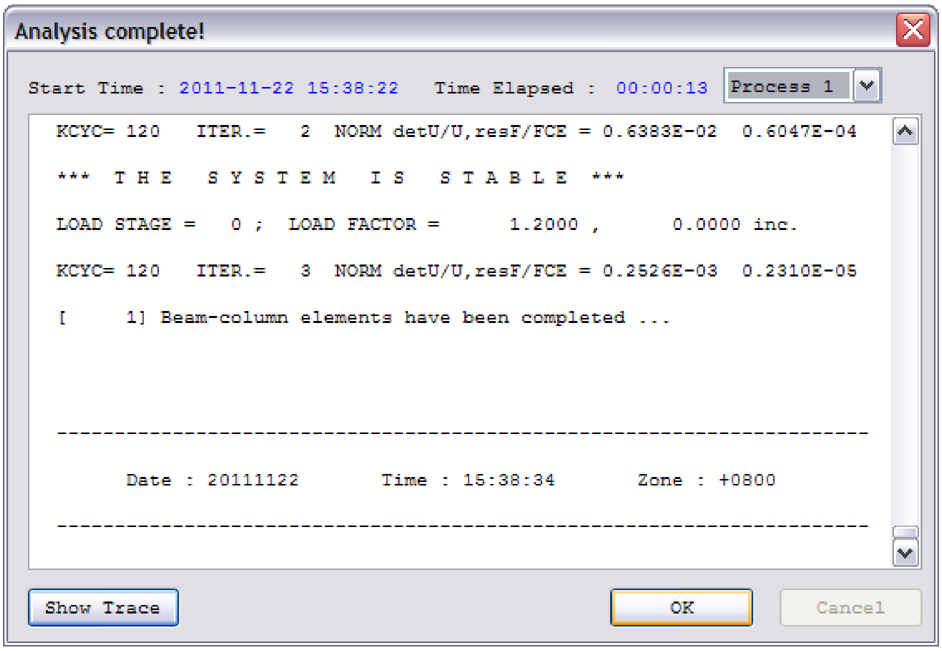
Click ***OK*** to go back to the <Analysis Cases> window;



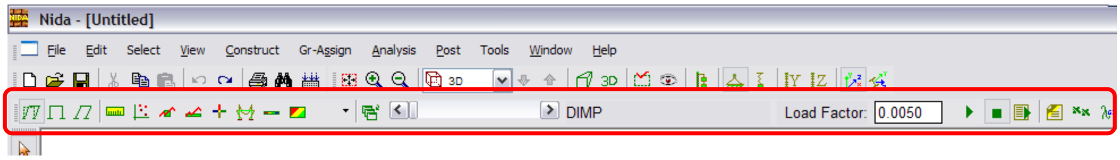
Click ***Run Now*** to run the analysis case or click ***OK*** to close this window and click the ***Analysis > Run*** command.



When the analysis is completed, click OK to close the <Analysis complete!> window;

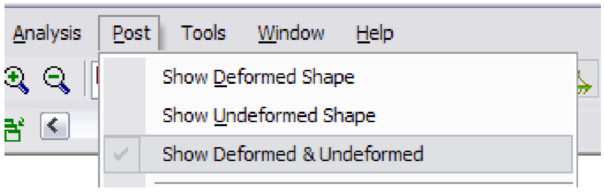


and the post analysis toolbar will appear under the function toolbar;

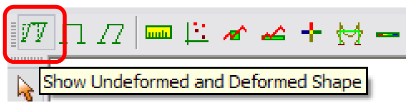


1.13 Step 13 – View result: structural deformation

Click the ***Post > Show Deformed & Undeformed*** command;

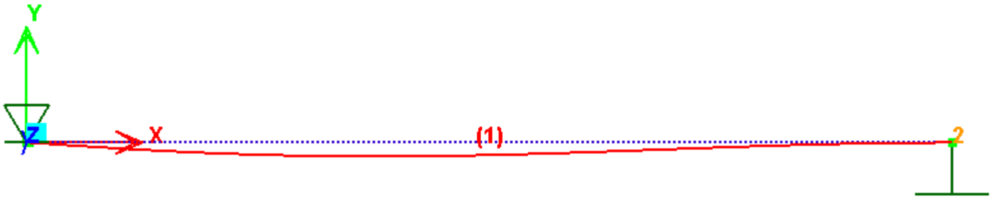


or click the <Show Undeformed and Deformed Shape> shortcut in the toolbar;



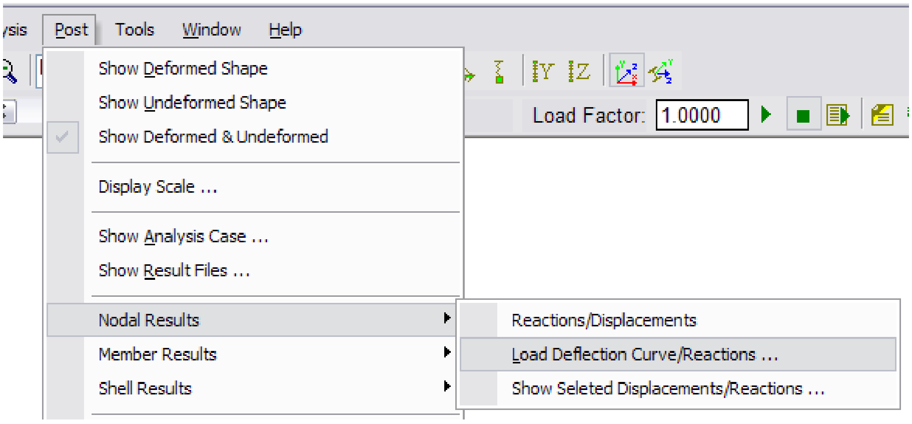
Change the load factor in the sliding bar to view the deformed shape under different load factors;

The deformed shape of the structure under 100kN * (LF=1.2) is shown as below;

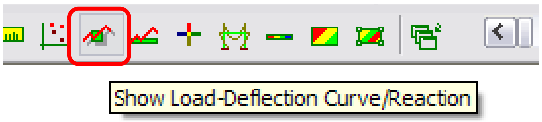


1.14 Step 14 – View result: show load-deflection curve

Click the ***Post*** > ***Show Nodal Results*** > ***Load Deflection Curve/Reaction*** command;

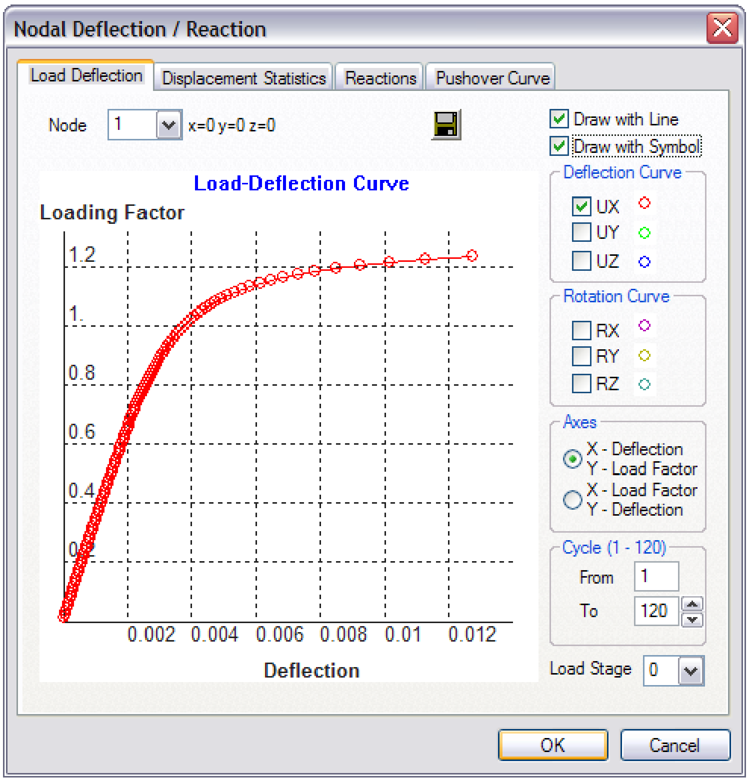


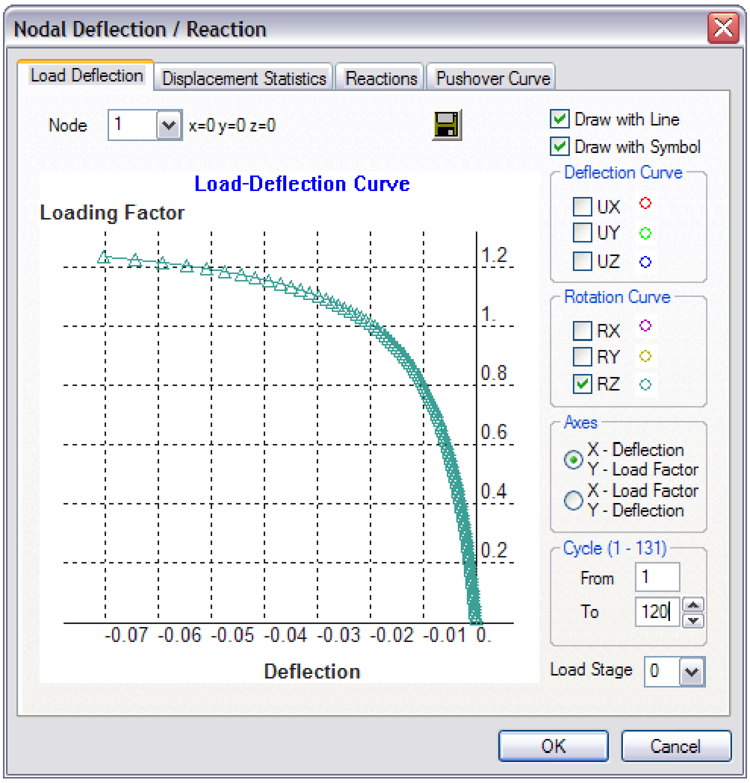
or click the <Show Load-Deflection Curve/Reaction> shortcut in the toolbar;



Select the node for which the Load (y) – Deflection (x) curve is determined, tick ‘Draw with Line’ and ‘Draw with Symbol’;

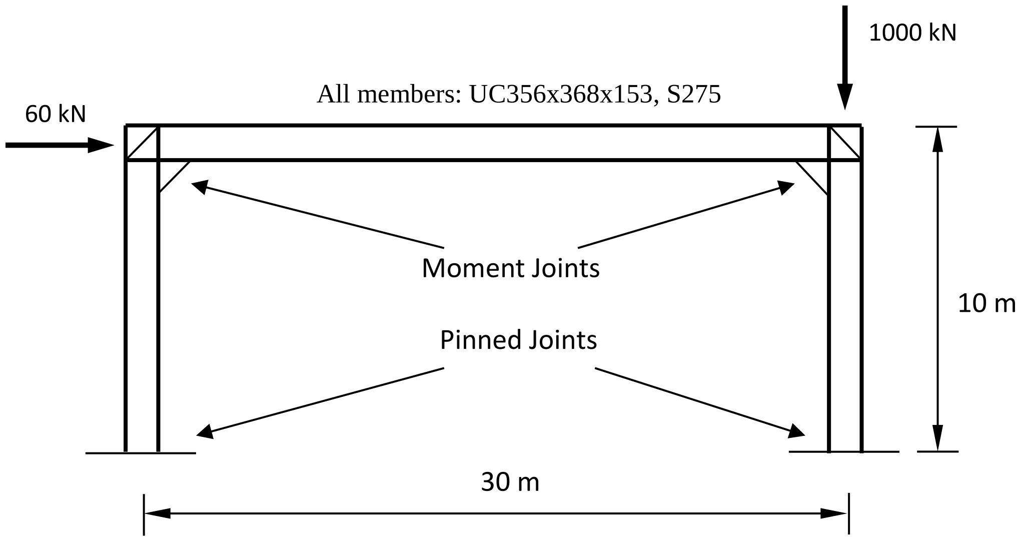
For this tutorial, the Load – Deflection curve for horizontal deflection along X axis of node 1 and rotation about Z axis of node 1 are both obtained in the <Nodal Deflection / Reaction> windows as below.


Nodal Deflection / Reaction
Load Deflection
Displacement Statistics
Reactions
Pushover Curve
Node
1
x=0 y=0 z=0
Draw with Line
Draw with Symbol
Load-Deflection Curve
Loading Factor
Deflection
Deflection Curve
UX
UY
UZ
Rotation Curve
RX
RY
RZ
Axes
X - Deflection
Y - Load Factor
X - Load Factor
Y - Deflection
Cycle (1 - 120)
From
To
120
Load Stage
0
OK
Cancel


Nodal Deflection / Reaction
Load Deflection
Displacement Statistics
Reactions
Pushover Curve
Node
1
x=0 y=0 z=0
Draw with Line
Draw with Symbol
Load-Deflection Curve
Loading Factor
Deflection
Deflection Curve
UX
UY
UZ
Rotation Curve
RX
RY
RZ
Axes
X - Deflection
Y - Load Factor
X - Load Factor
Y - Deflection
Cycle (1 - 131)
From
To
120
Load Stage
0
OK
Cancel

2. TUTORIAL 2 – A Braced and an Unbraced Portal – Second Order Analysis and Elastic Critical Load Factor Analysis

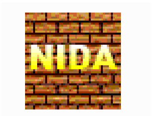
The section used for both columns and beam is UC356x368x153 and grade S275 steel. Assume the portal is braced and unbraced as two cases. Cross brace of channel 230 x 90 x 32 of same steel grade is used for the braced case.



1. Determine Elastic Critical Load Factor λ_{cr} for both cases.
2. Carry out P-Δ-δ analysis and perform member check as per clause 8.9.1 of the CoP for the Structural Use of Steel 2011 for both cases
   - P-δ effects are considered by incorporating Table 6.1 during analysis
   - P-Δ effects are considered by iterative procedure of adding deflections to the original structural geometry
   - Most unfavorable direction such as through the use of the elastic buckling mode for the initial bow be established when using Table 6.1

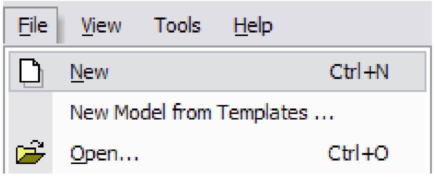
2.1 Step 1 – Start NIDA

Double click the "NIDA" icon on the desktop or in the corresponding folder upon successful installation of NIDA.



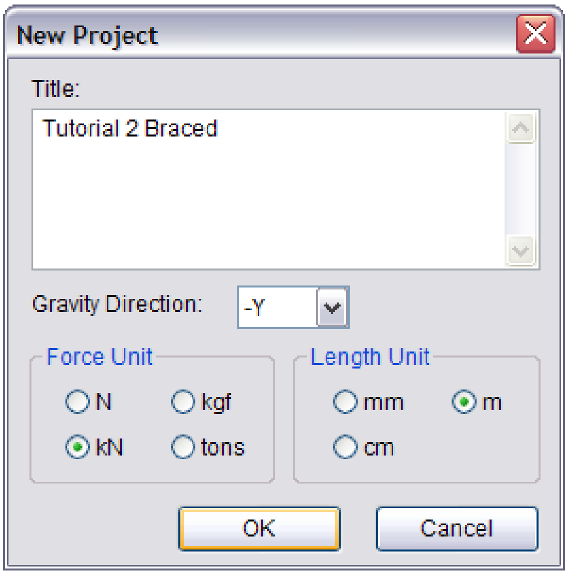
2.2 Step 2 – Create a new project file

Click the ***File* > *New*** command,



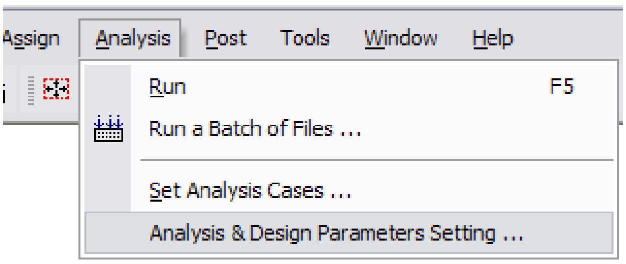
or find the <New> shortcut button in the toolbar;

Enter the new project <Title> 'Tutorial 2 Braced' in the popped window and select the <Gravity Direction> to be '-Y', <Force Unit> 'kN' and <Length Unit> 'm'.



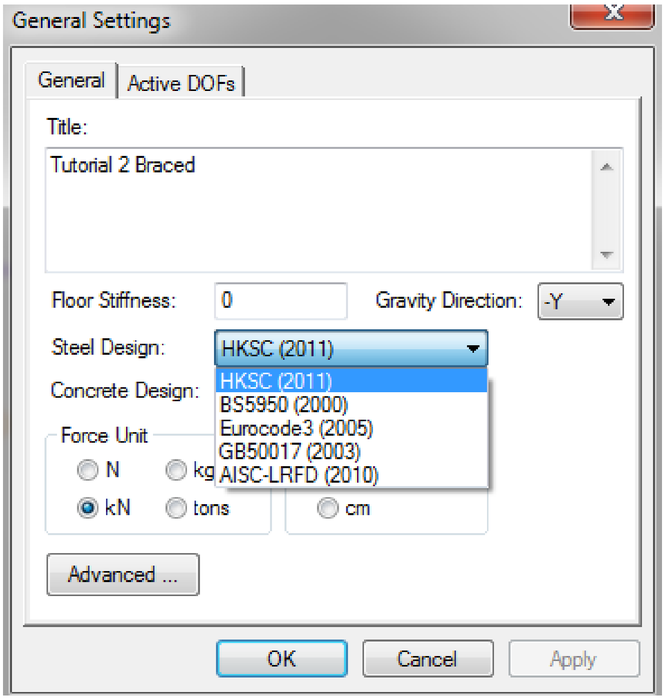
Click ***OK*** to continue.

Click the ***Analysis* > *Analysis & Design Parameters setting*** command,

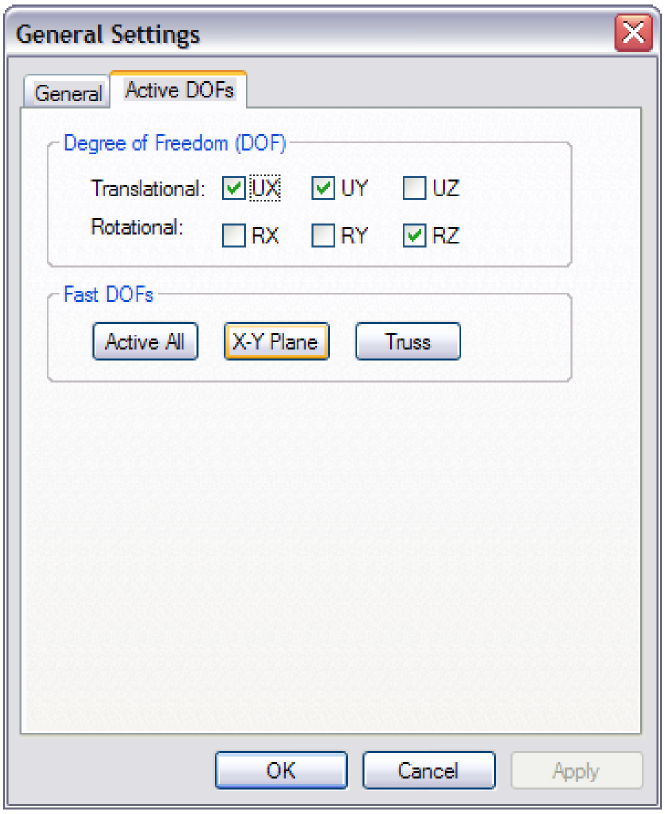


The parameters already set could also be viewed or changed in the <General Settings> window;

Select the steel design code HKSC (2011);



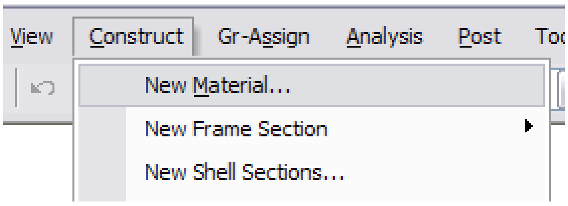
Select 'Fast DOFs – X-Y Plane' under 'Active DOFs' for a plane portal frame;



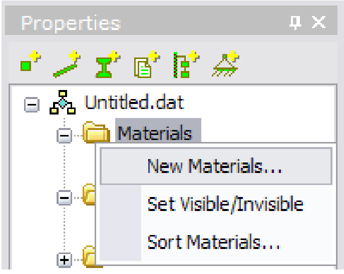
Click ***Apply*** and then ***OK*** to continue.

2.3 Step 3 – Create material

Click the ***Construct*** **>** ***New Material*** command,

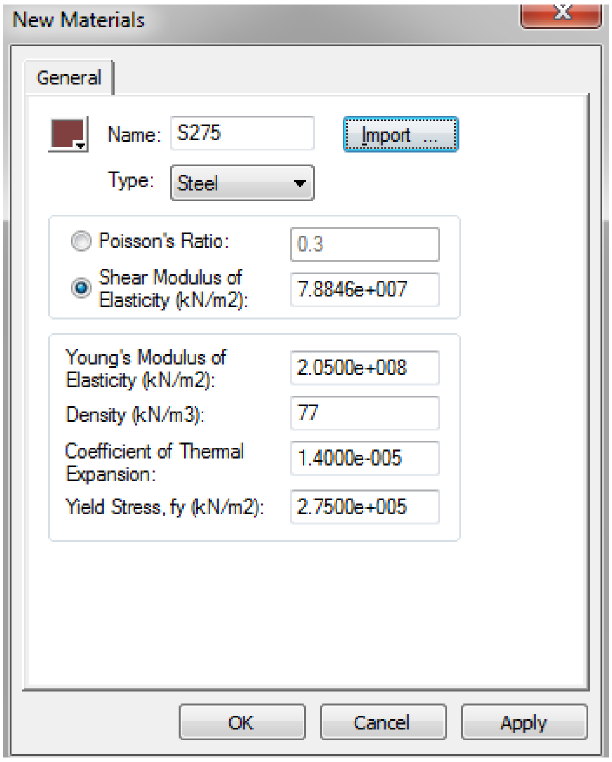


or right click <Materials> in the ***Properties*** window and then click <New Materials>;



Default steel grade set up in the program is S275;

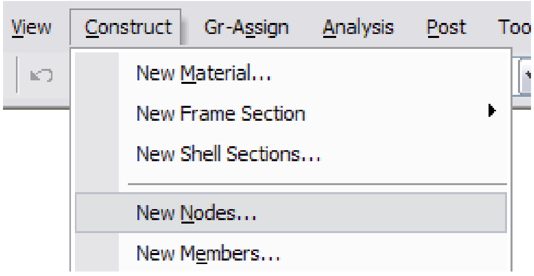
Double click <S275> to view and change the material properties in the <Materials> window if required, such as changing the material colour only for this tutorial;



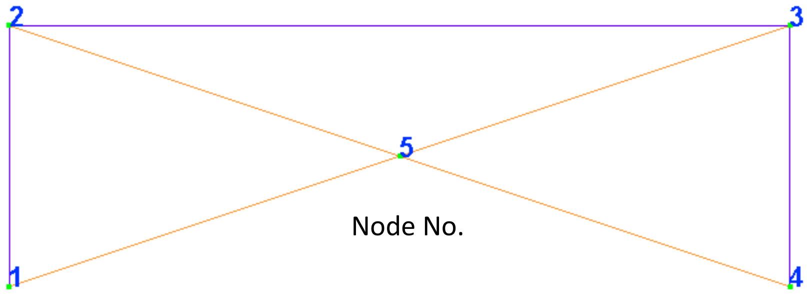
Click ***OK*** to continue.

2.4 Step 4 – Add nodes

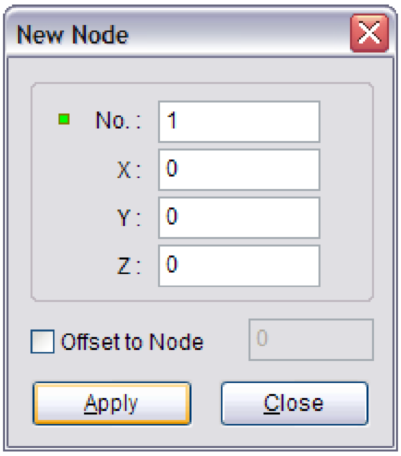
Click the ***Construct*** **>** ***New Nodes*** command,

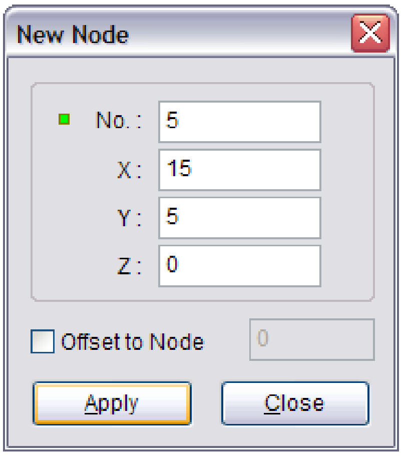


or click the <Add Node> shortcut in the ***Properties*** window;



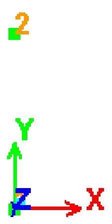
Enter the x, y and z coordinates for node No. 1 and click ***Apply*** to continue adding nodes;

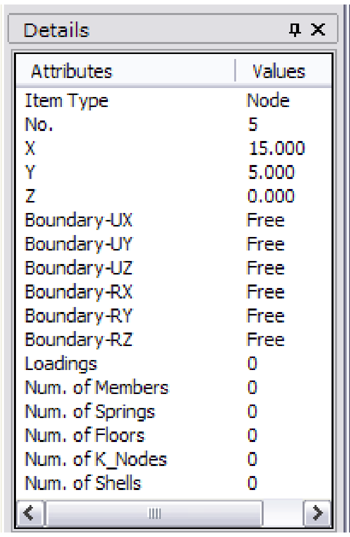




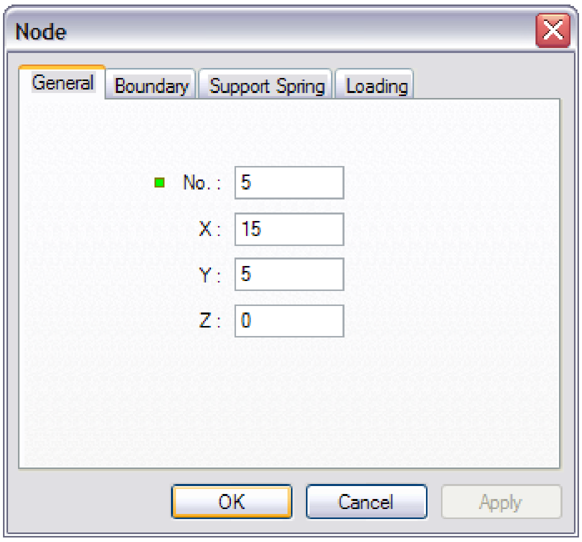
On completion of the last node construction, click ***Apply*** and then ***Close*** to finish this step.

To view the details of a particular node, click the node, then all the details including coordinates, boundary conditions, loadings, etc, are displayed in the ***Details*** window;





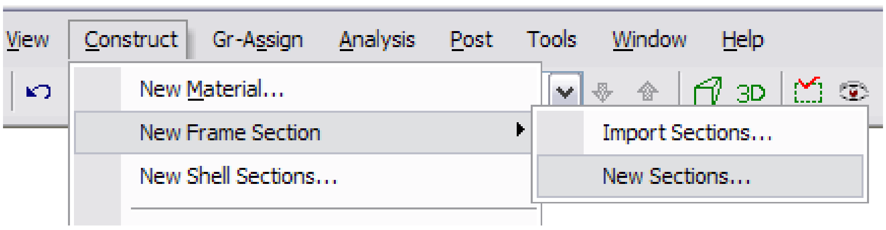
or double click the node to view the node details under ‘Coordinates’, ‘Boundary’, ‘Support Spring’ and ‘Loading’ menus, make changes in the ***Node*** window if needed;



Click ***OK*** to continue.

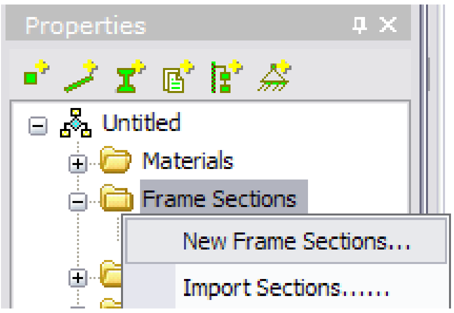
2.5 Step 5 – Add sections

Click the ***Construct*** > ***New Frame Section*** > ***New Sections*** command,

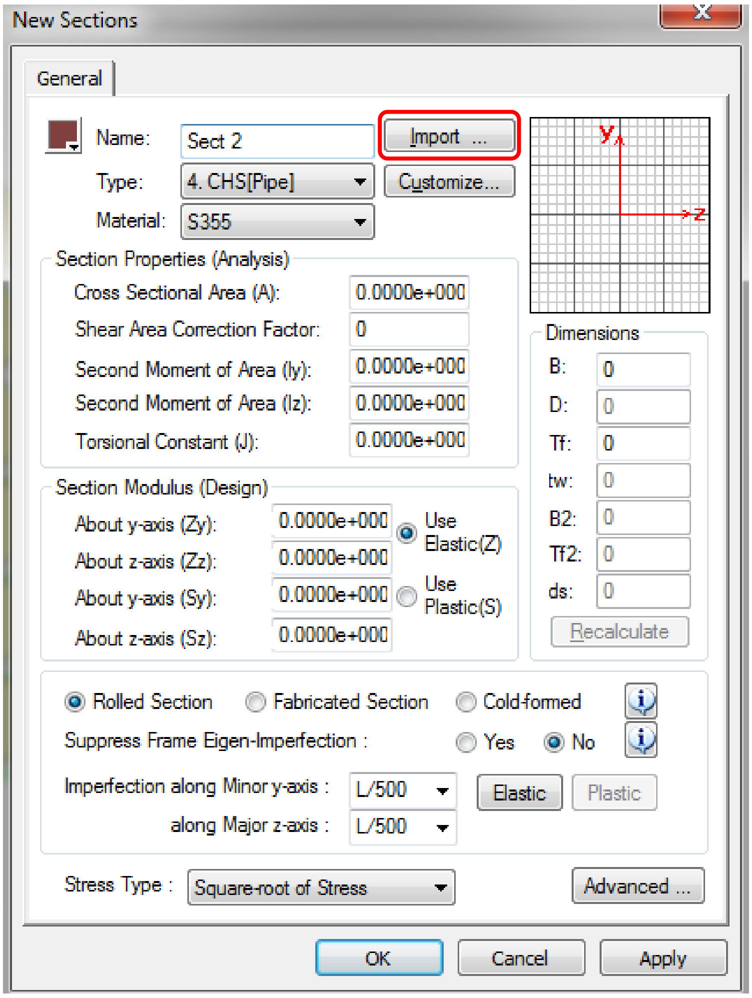


or click the <Add Section> shortcut in the ***Properties*** window;

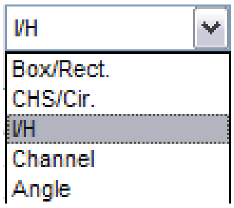
or right click <Frame Sections> in the ***Properties*** window and then click <New Frame Sections>;



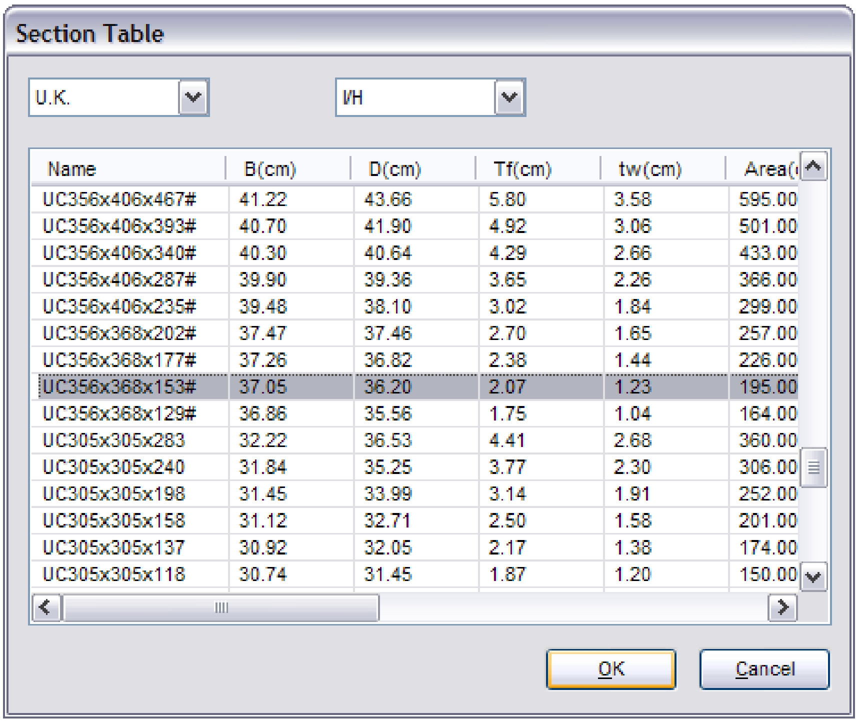
Click <Import> button in the <New Sections> window to import a new cross section;



Select cross sections under different categories, country or shape;



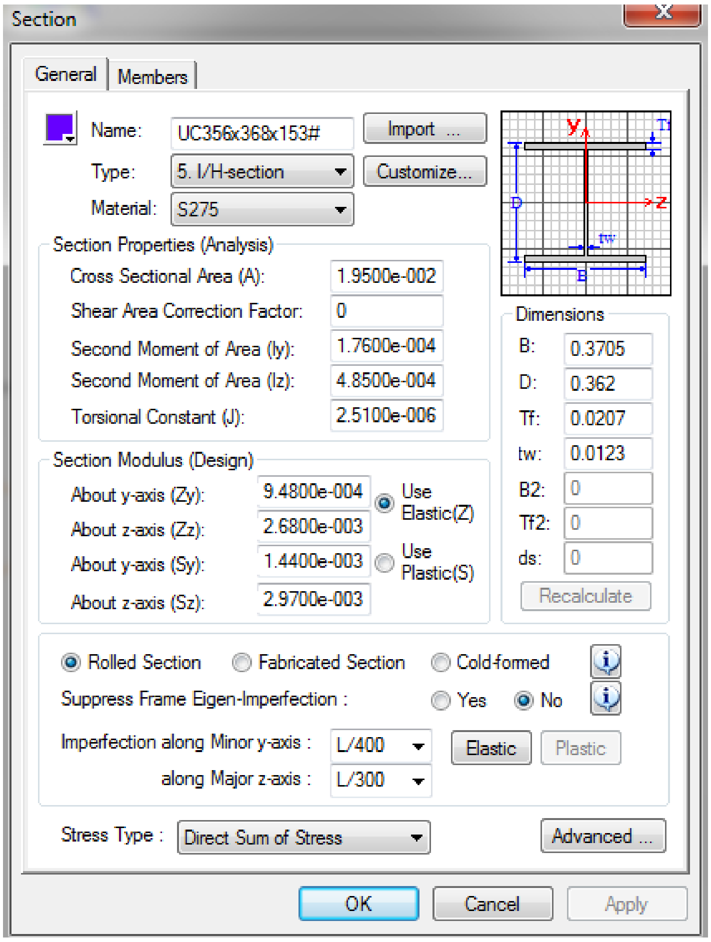
Select ‘UC 356 x 368 x 153#’ in the <Section Table>;



Name	B(cm)	D(cm)	Tf(cm)	tw(cm)	Area((
UC356x406x467#	41.22	43.66	5.80	3.58	595.00
UC356x406x393#	40.70	41.90	4.92	3.06	501.00
UC356x406x340#	40.30	40.64	4.29	2.66	433.00
UC356x406x287#	39.90	39.36	3.65	2.26	366.00
UC356x406x235#	39.48	38.10	3.02	1.84	299.00
UC356x368x202#	37.47	37.46	2.70	1.65	257.00
UC356x368x177#	37.26	36.82	2.38	1.44	226.00
UC356x368x153#	37.05	36.20	2.07	1.23	195.00
UC356x368x129#	36.86	35.56	1.75	1.04	164.00
UC305x305x283	32.22	36.53	4.41	2.68	360.00
UC305x305x240	31.84	35.25	3.77	2.30	306.00
UC305x305x198	31.45	33.99	3.14	1.91	252.00
UC305x305x158	31.12	32.71	2.50	1.58	201.00
UC305x305x137	30.92	32.05	2.17	1.38	174.00
UC305x305x118	30.74	31.45	1.87	1.20	150.00

Click ***OK***, then all the section properties are displayed in the <New Sections> window;

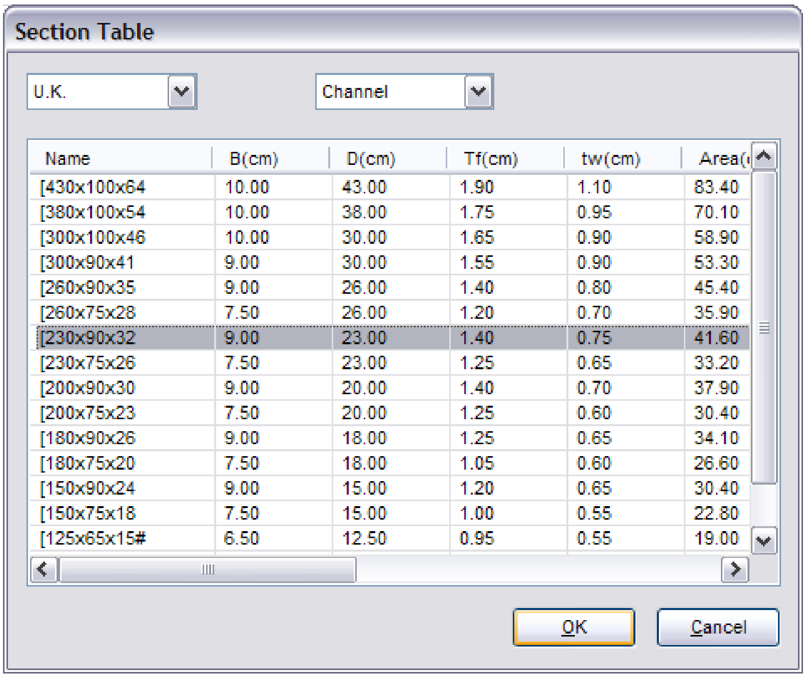
Select a different colour for this UC section, check all the section properties and make changes if required;



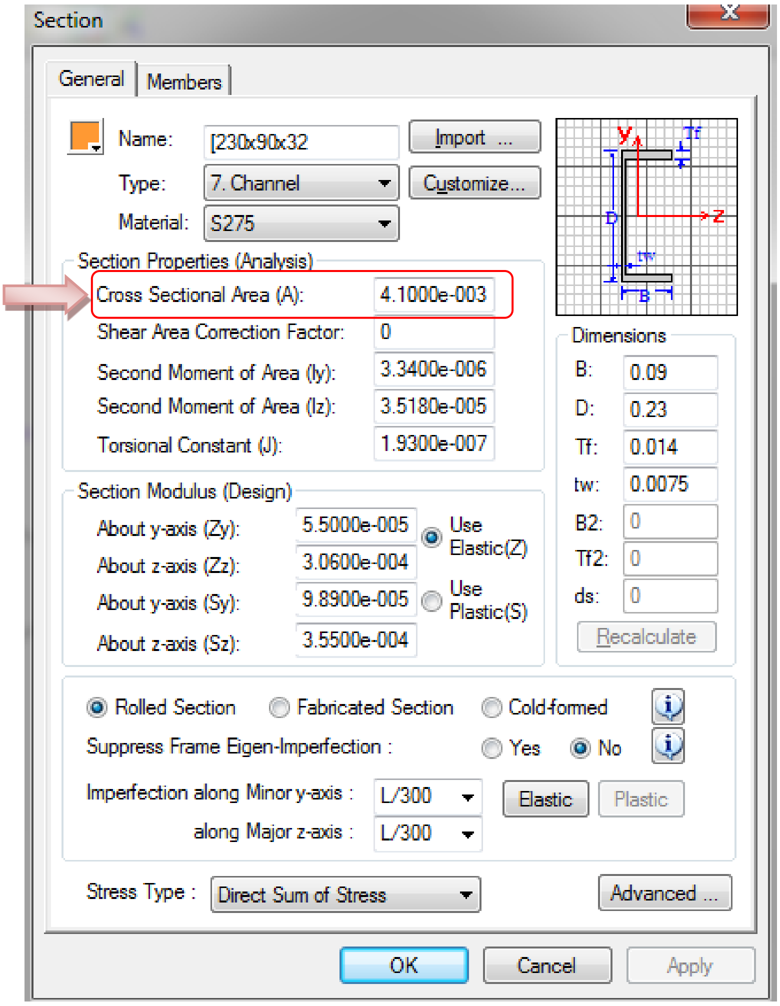
P – δ imperfections of the member in this section are generated automatically by the software according to the design code in use, but could be manually adjusted as well to fulfil special requirement.

Click ***Apply*** and ***Import*** to continue to add a new section Channel 230 x 90 x 32;

Find the Channel 230 x 90 x 32 in the <Section Table> window under the ‘UK-Channel’ category;



Click ***OK***, then all the section properties are displayed in the <New Sections> window;

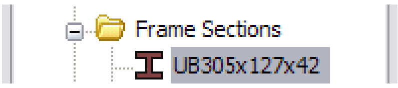


Select a different colour for this channel section, check all the section properties;

Change the cross sectional area from 4.1600e-003 to 4.1000e-003;

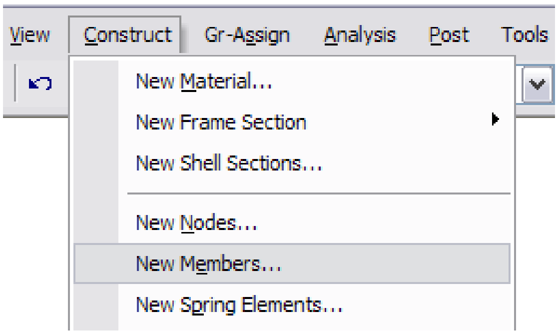
Click ***OK*** to finish adding sections.

There is a default frame section ‘UB 305 x 127 x 42’ generated by the software. Delete it or change the properties if needed.

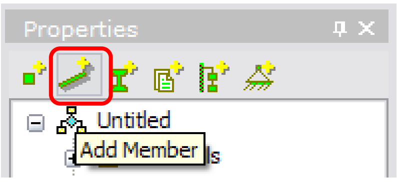


2.6 Step 6 – Add members

Click the ***Construct*** **>** ***New Members*** command,

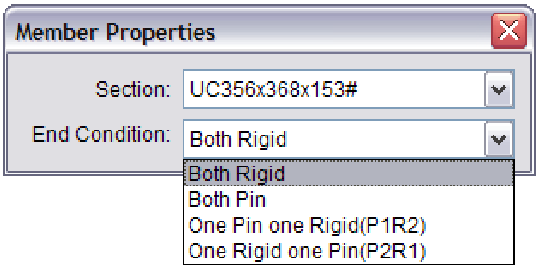


or click the <Add Member> shortcut in the ***Properties*** window;



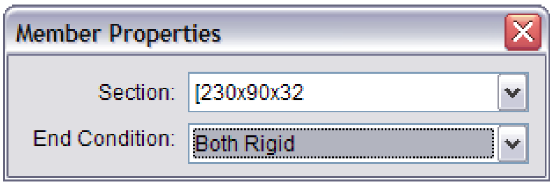
or click the <Add Member> shortcut in the toolbar;

Select the ‘Section’ and ‘End Condition’ in the <Properties of Member> window;

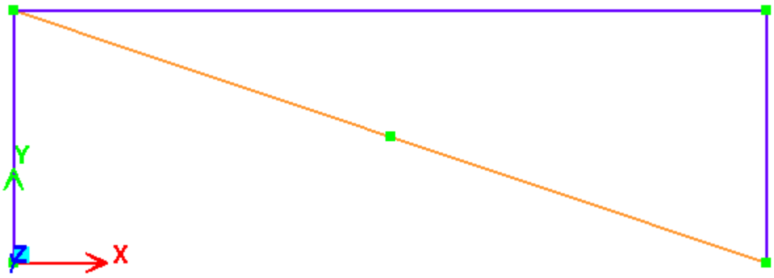


Click the start node and then the end node; if this end node is connected to other members, continue to click the next end node and right click to stop adding members.

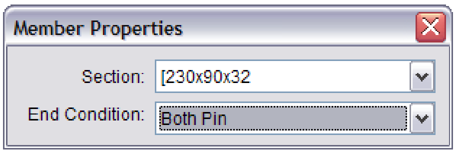
Change the member to channel section with both ends rigid,

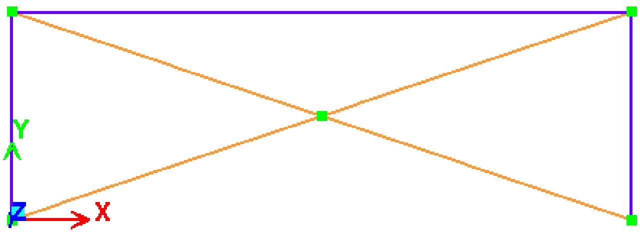


Click the left top node, the centre point and the right bottom node to add two rigidly connected bracing members;



Add the other two bracing members with both ends pin connected;

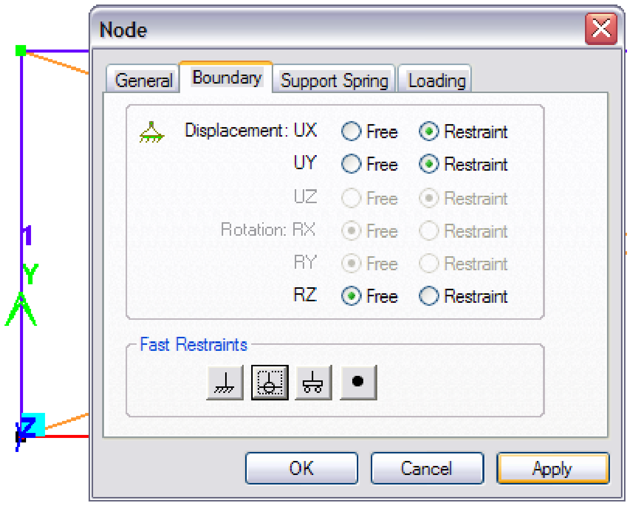




Click ***Close*** in dialog or press ***Esc*** in keyboard to finish this step.

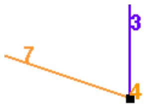
2.7 Step 7 – Add boundary conditions

Double click the left boundary node, use the ‘Fast Restraints’ option to automatically set this pin node in the ***Node*** window;

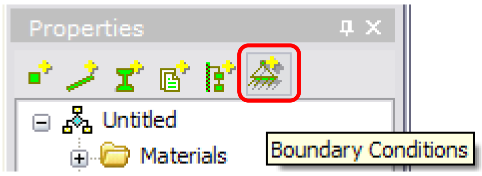


Click ***Apply*** or ***OK*** to continue;

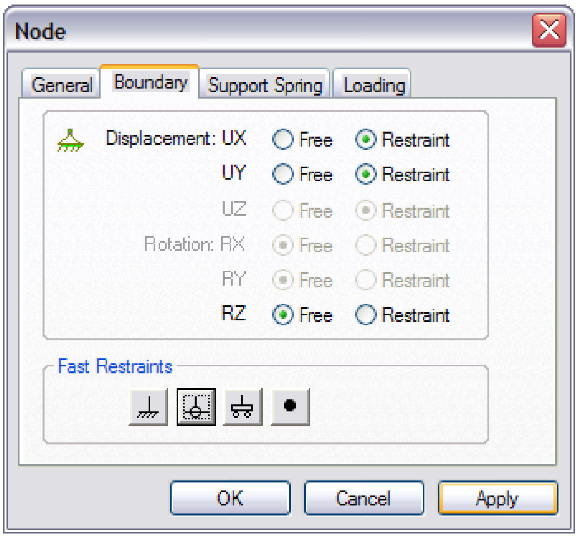
or select the node to be restrained;



and click the <Boundary Conditions> shortcut in the ***Properties*** window;



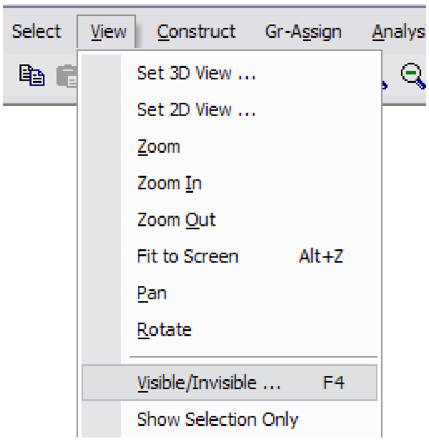
Set the this node to be pin end using ‘Fast Restraints’;



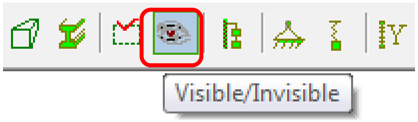
Click ***Apply*** or ***OK*** to finish this step.

2.8 Step 8 – Set visible/invisible

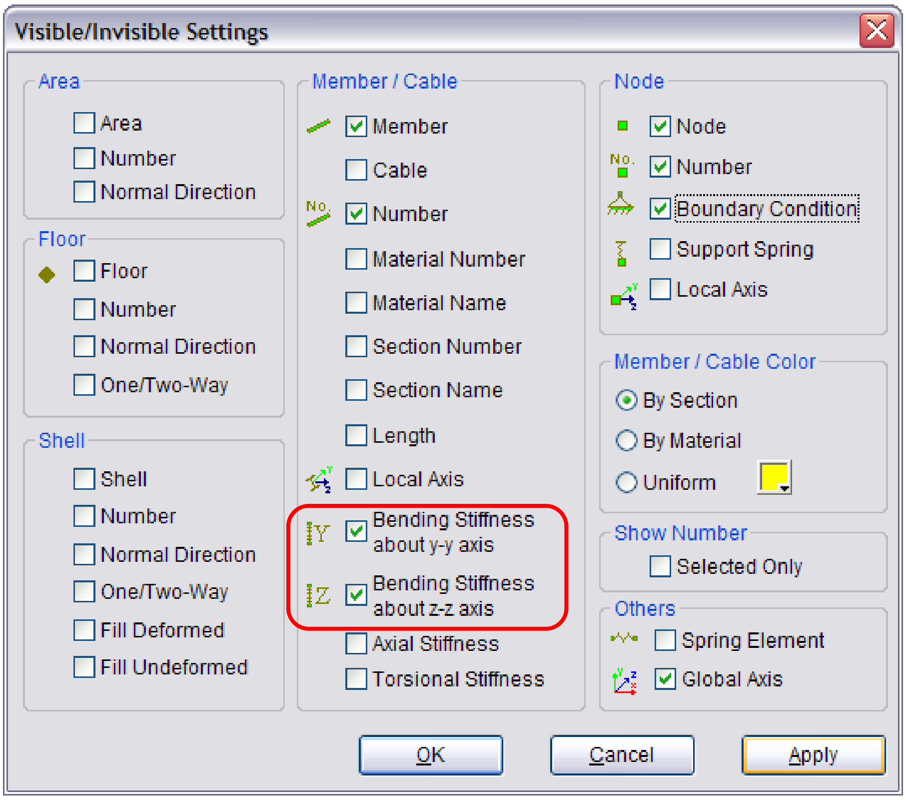
Click the ***View* > *Visible/Invisible*** command,

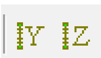


or click the <Visible/Invisible> shortcut in the toolbar;

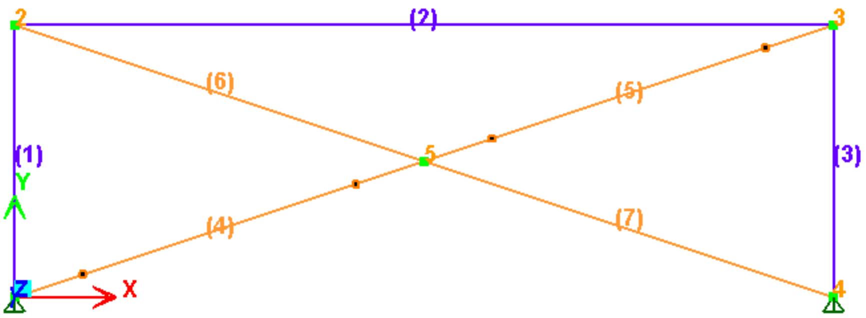


Tick 'Member', 'Member-Number', 'Node', 'Node-Number', 'Node-Boundary Condition', 'Global Axis' and select 'By Section' in the <Visible/Invisible Settings> window;



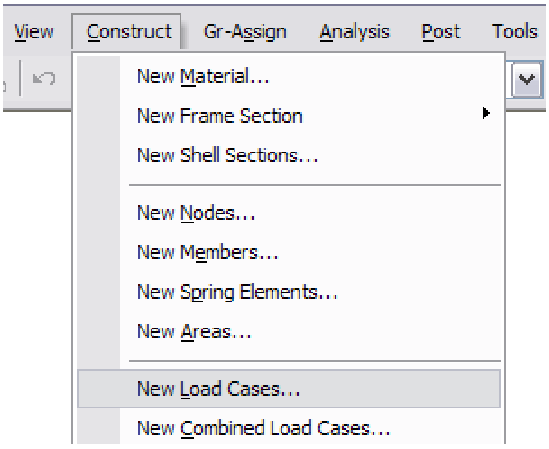
Tick 'Bending Stiffness about y-y/z-z axes' to view the end releases of member 4 and 5; or press the buttons  in toolbar.

Click ***Apply*** or ***OK*** to view the structure with properties constructed.

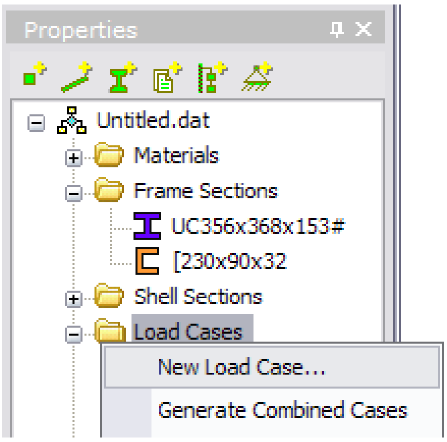


2.9 Step 9 – Add load cases

Click the ***Construct*** **>** ***New Load Cases*** command,

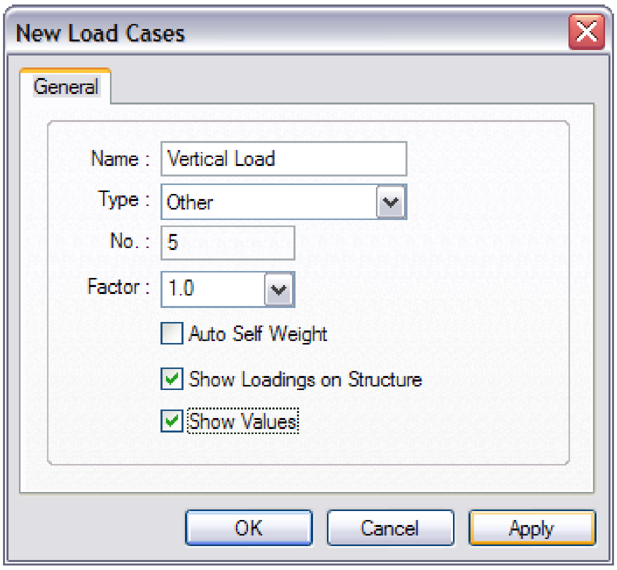


or right click <Load Cases> in the ***Properties*** window and then click <New Load Case>;

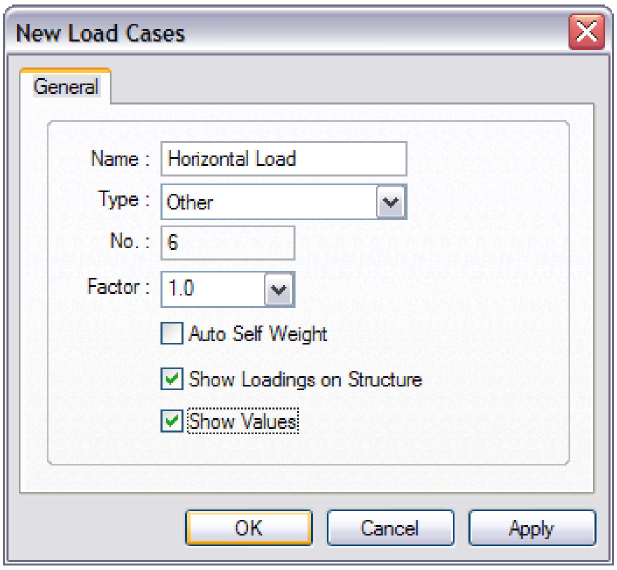


or click the <Add Load Case> shortcut in the ***Properties*** window;

Enter the 'Name' of new load case, select the load 'Type' and 'Factor' in the <New Load Cases> window;

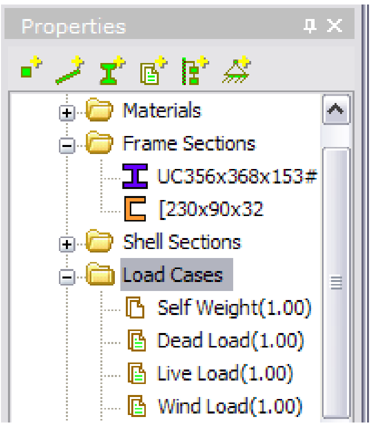


Click ***Apply*** to continue adding more load cases;



Click ***OK*** to finish this step.

There are four load cases set as default in the software;

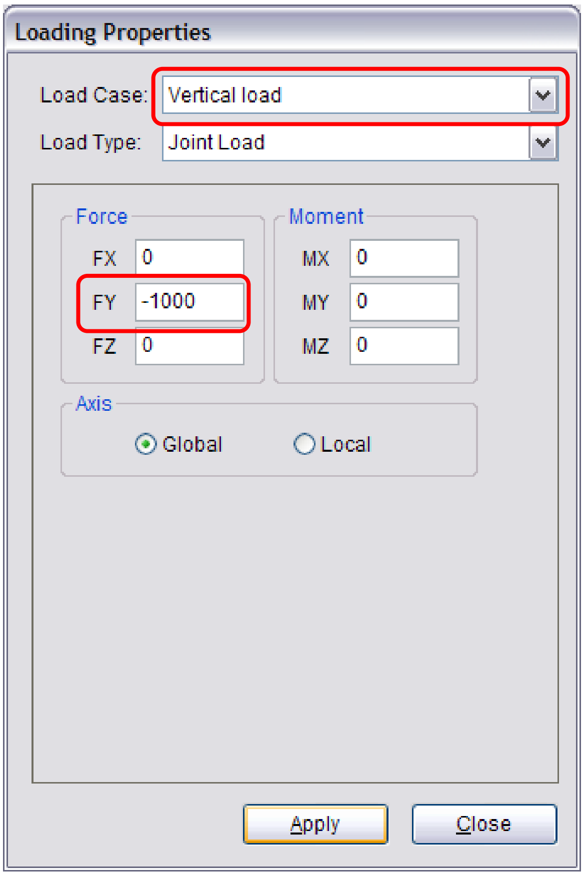


Double click any load case to view or change the settings of this load case or right click and select ‘delete’ as requirement.

2.10 Step 10 – Add loadings

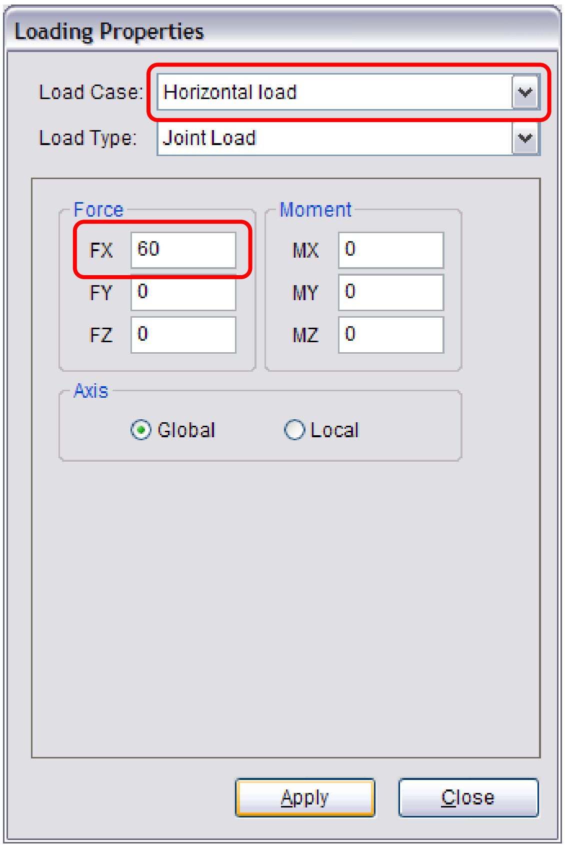
Select node No. 3 and click the <Add Load> shortcut in the ***Properties*** window;

Select ‘Load Case’, ‘Load Type’, ‘Axis’ and enter 1000 kN ‘Force’ in negative Y direction in the <Loading Properties> window;



Click ***Apply*** and ***Close*** to continue adding loads;

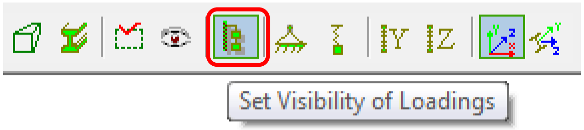
Select node 2 and apply the same procedure as for node 3;



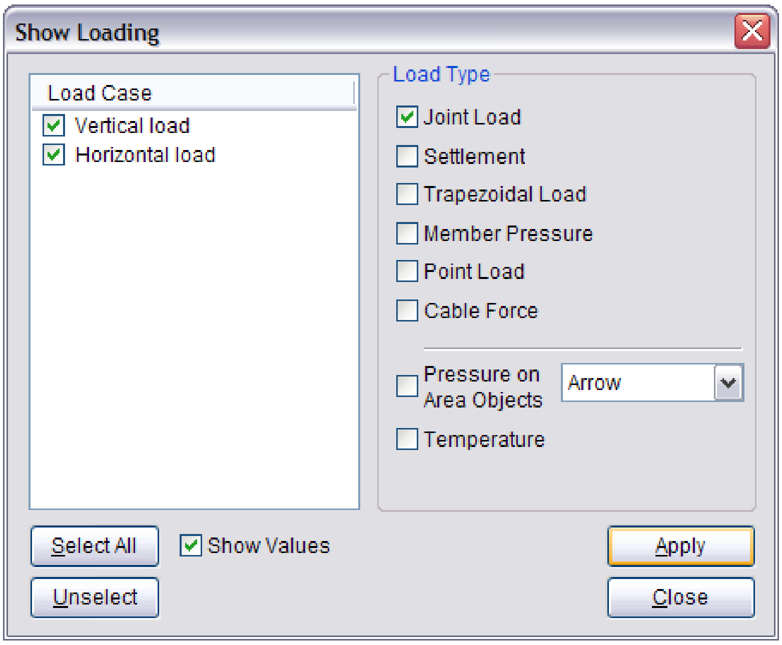
Click ***Apply*** and ***Close*** to finish this step.

2.11 Step 11 – Set visibility of loadings

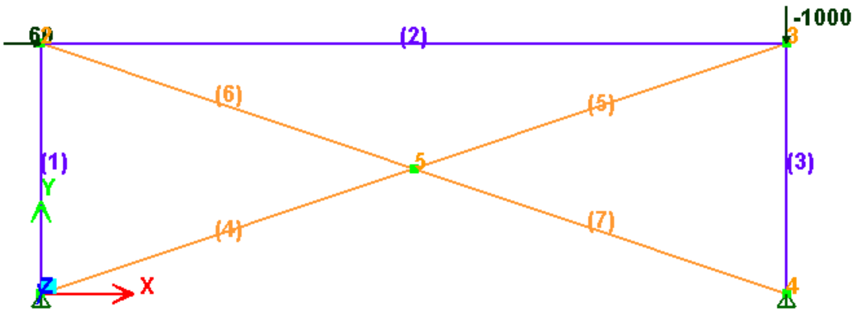
Click the <Set Visibility of Loadings> shortcut in the toolbar;



Select the ‘Load Case’, ‘Load Type’ and tick ‘Show Values’ in the <Show Loading> window;

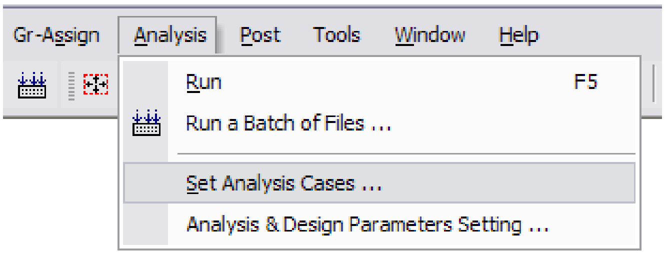


Click ***Apply*** and ***Close*** to finish setting. Loading is then visible.

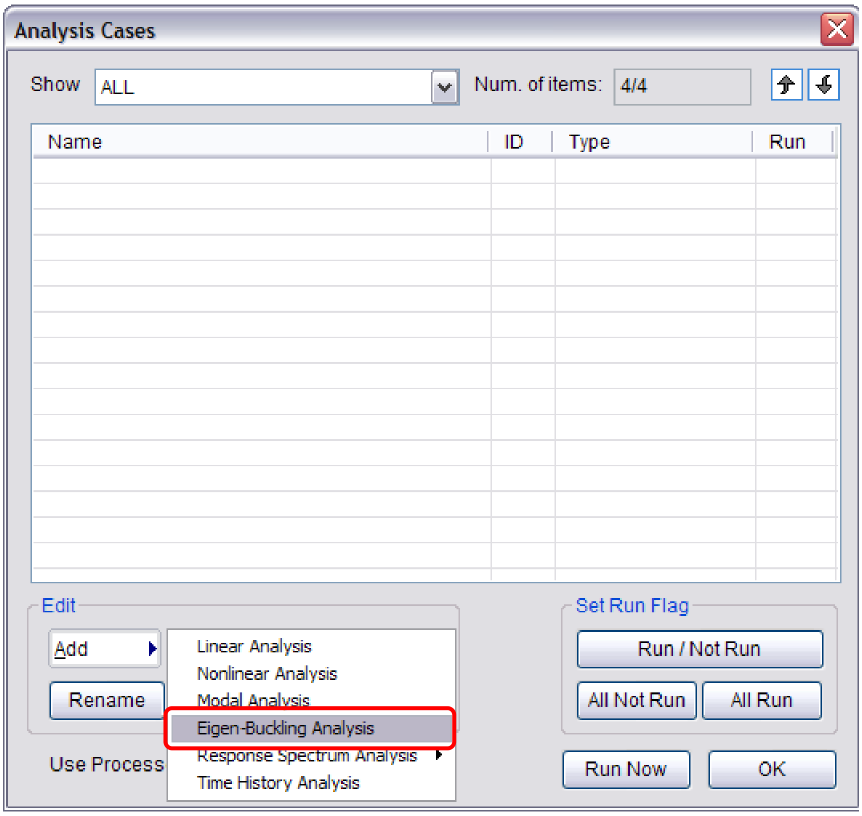


2.12 Step 12 – Set and run analysis cases

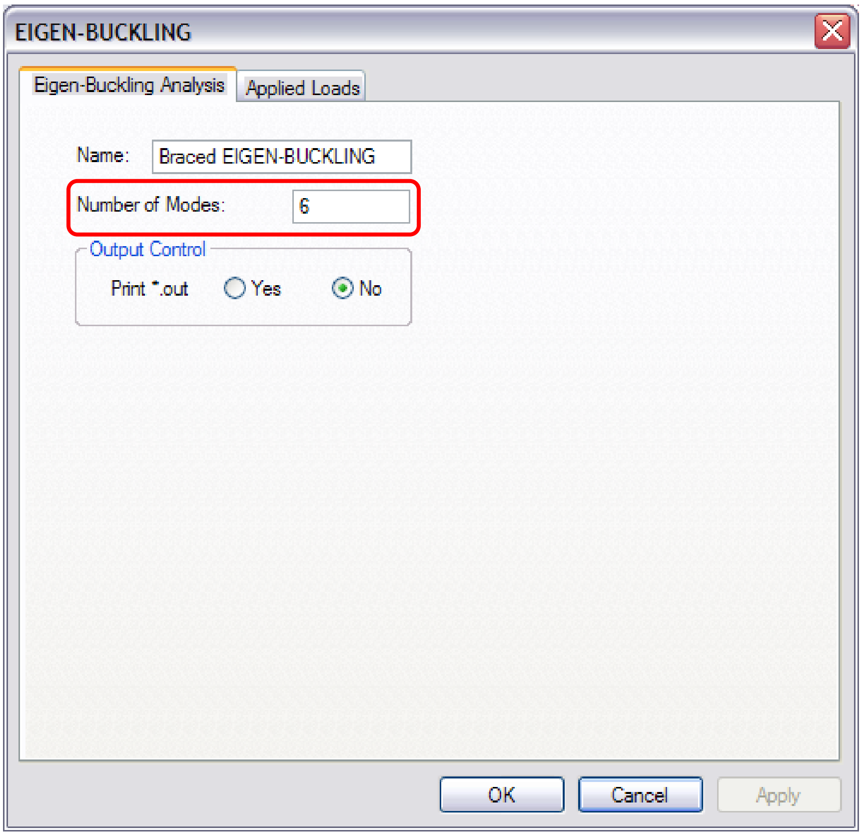
Click the ***Analysis > Set Analysis Cases*** command;



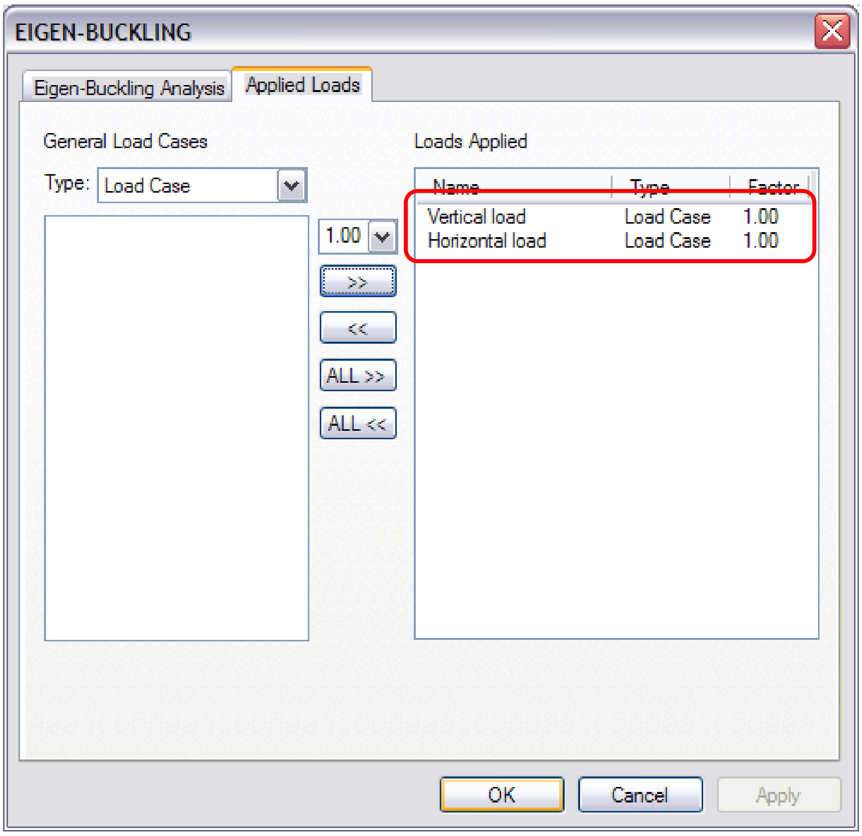
Add 'Eigen-Buckling Analysis' to determine elastic critical load factor λ_{cr} for the braced frame;



Enter the name of analysis to be 'Braced EIGEN-BUCKLING' in the <EIGEN-BUCKLING> window;

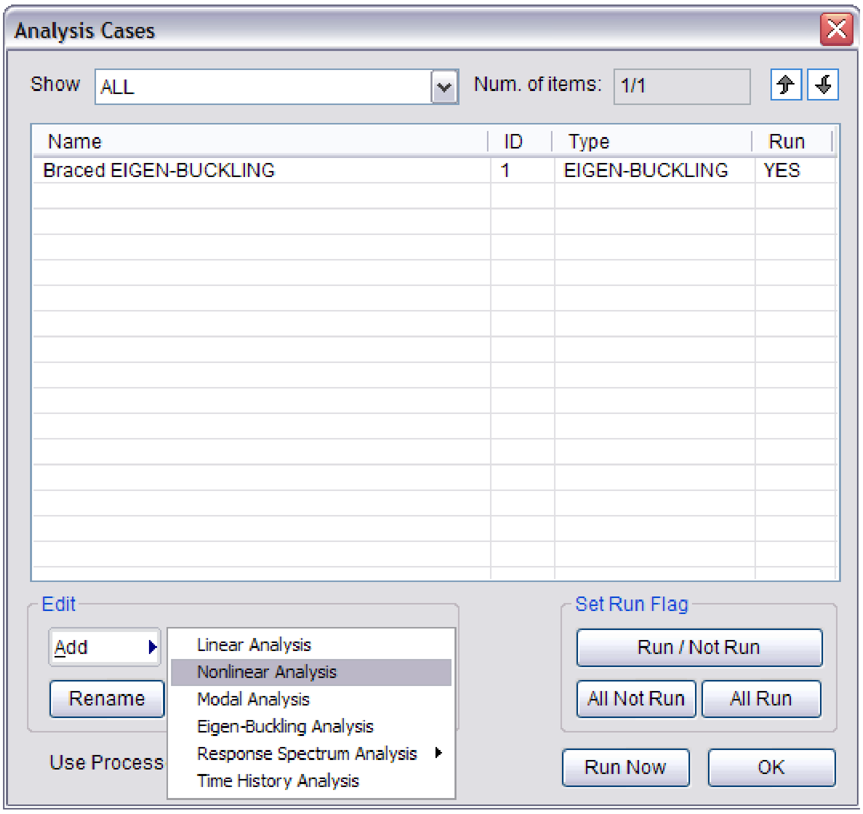


Apply ‘Load Case – Vertical/Horizontal Load’ to this analysis case;

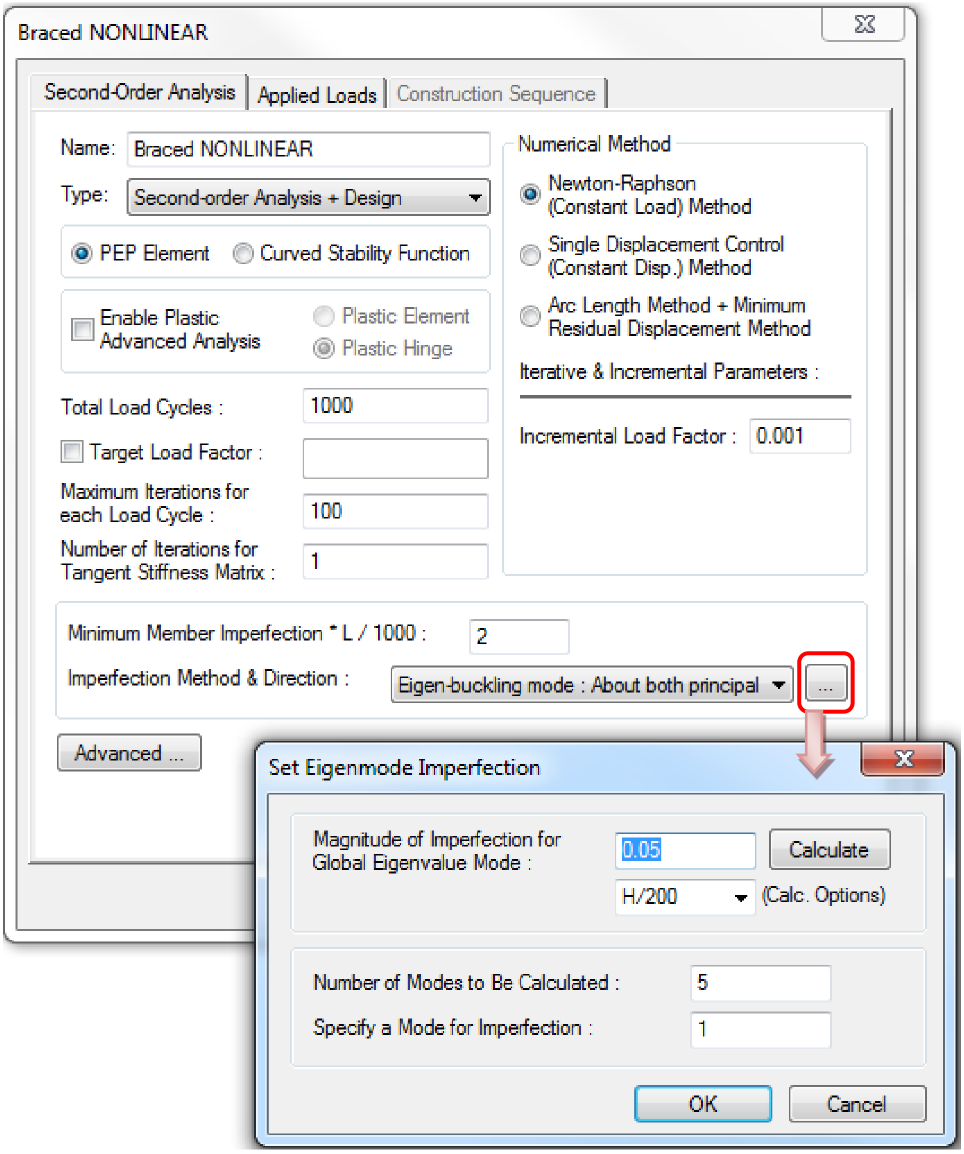


Click ***OK*** to go back to the <Analysis Cases> window;

Add ‘Nonlinear Analysis’ to carry out P-Δ-δ analysis and perform member check;

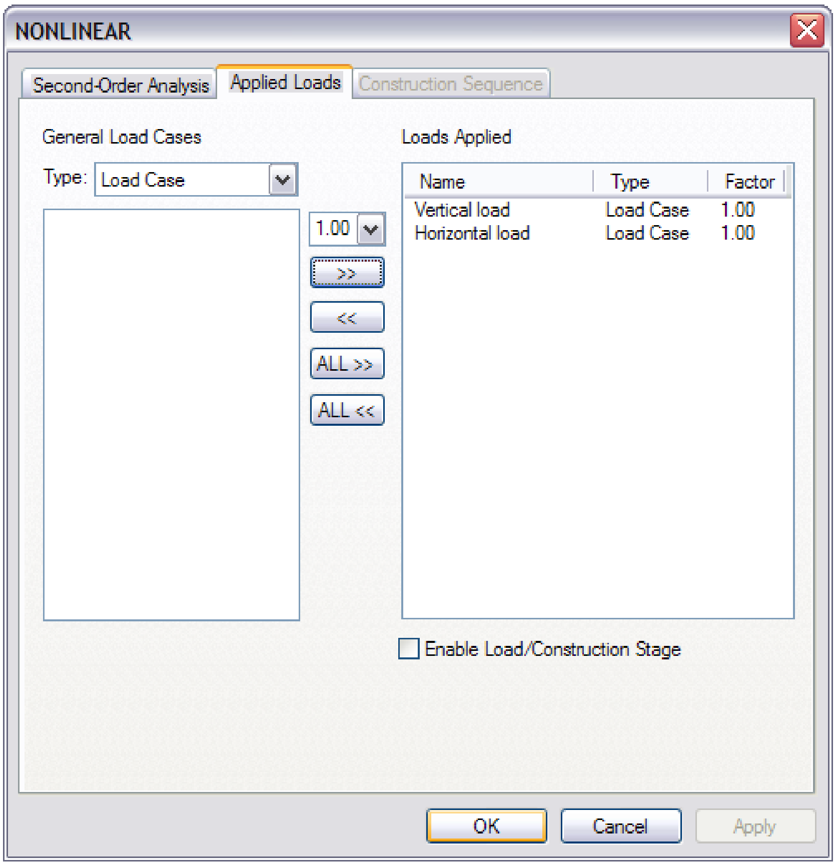


Select ‘Nonlinear Analysis + Design’, enter the ‘Name’ to be ‘Braced NONLINEAR’, the ‘Total Load Cycles’ to be 1000, ‘Incremental Load Factor’ to be 0.001 and select ‘Imperfection Method & Direction’ in the <NONLINEAR> window;

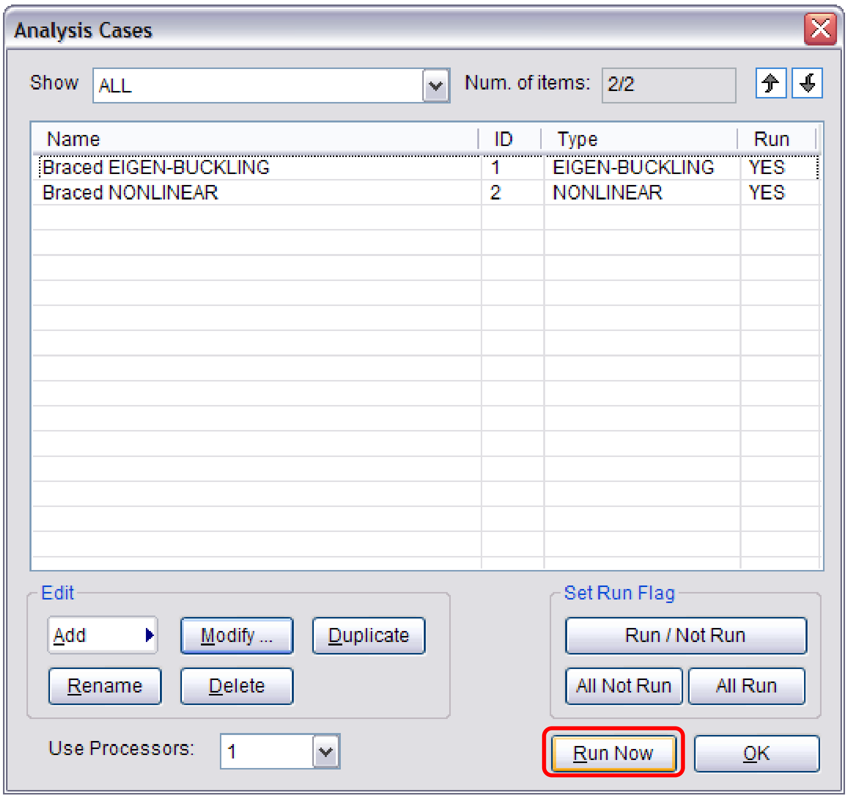


To fulfil the requirement of implementing P-Δ imperfection, click ***Setting***, enter the ‘Magnitude of Imperfection for Global Eigenvalue Mode’ in the <Set Eigenmode Imperfection> window or select an ‘Auto Cal. Option’ according to the design code applied, which is H/200 for this tutorial and click ***Auto Cal*** to obtain the value of ‘Magnitude’;

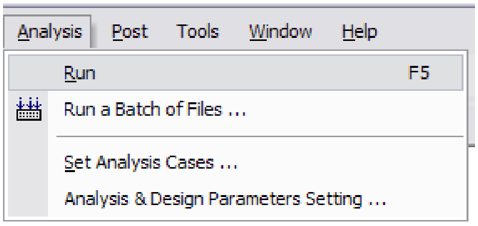
Apply ‘Load Case – Vertical/Horizontal Load’ to this analysis case;



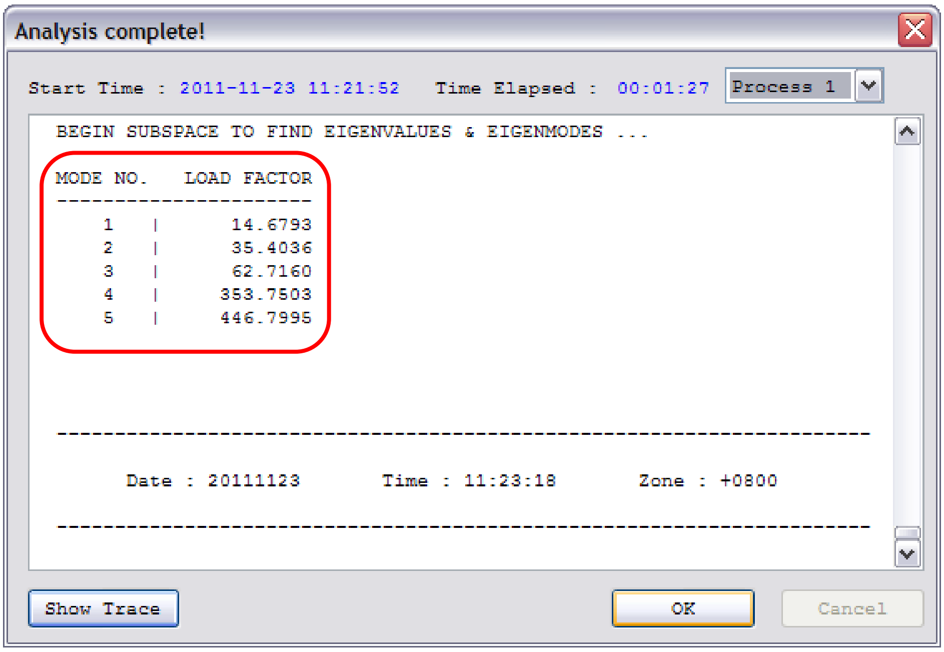
Click ***OK*** to go back to the <Analysis Cases> window;



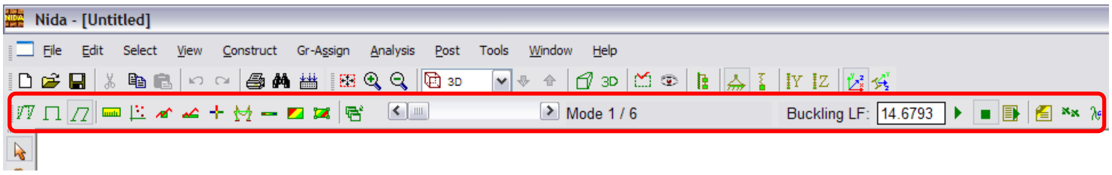
Click ***Run Now*** to run both analysis cases or click ***OK*** to close this window and click the ***Analysis > Run*** command.



When the analysis is completed, click OK to close the <Analysis complete!> window;



and the post analysis tool bar will appear under the function tool bar;



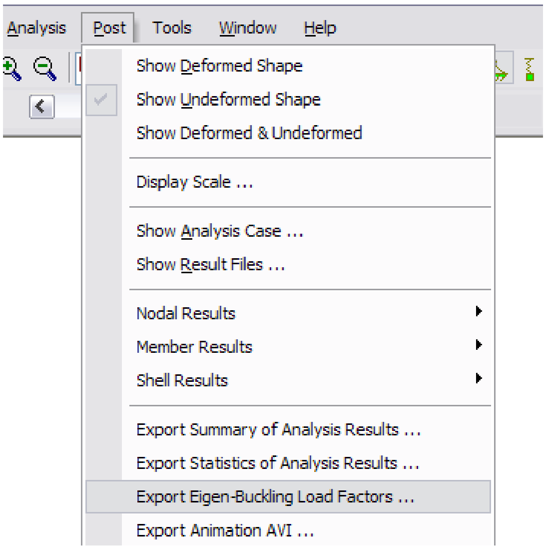
2.13 Step 13 – View result: eigen-buckling mode

For more than one analysis case, the first analysis case will display automatically upon completion of program running. For this tutorial, the Eigen-Buckling Analysis result displays the 6 eigen-modes firstly;

The mode could be selected through the sliding bar and the corresponding ‘Buckling LF’ could be viewed in the tool bar, so the elastic critical load factor for the braced case, which is also the first mode buckling load factor, is

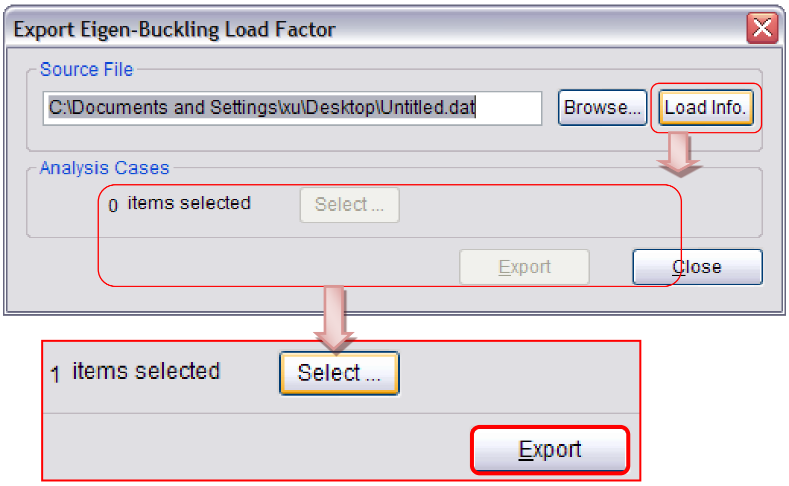
$\lambda_{cr} = 14.6793$;

Click the ***Post > Export Eigen-Buckling Load Factors*** command,

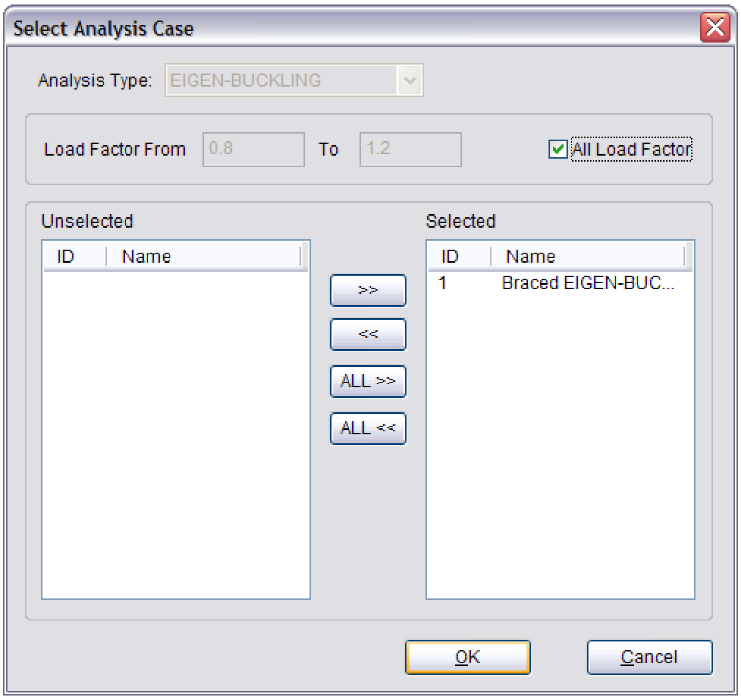


or click <Export Eigen-Buckling Load Factors> shortcut in the post analysis toolbar;

Select the 'Source File' direction, click ***Load Info*** in the <Export Eigen-Buckling Load Factor> window to select the analysis case to be exported;

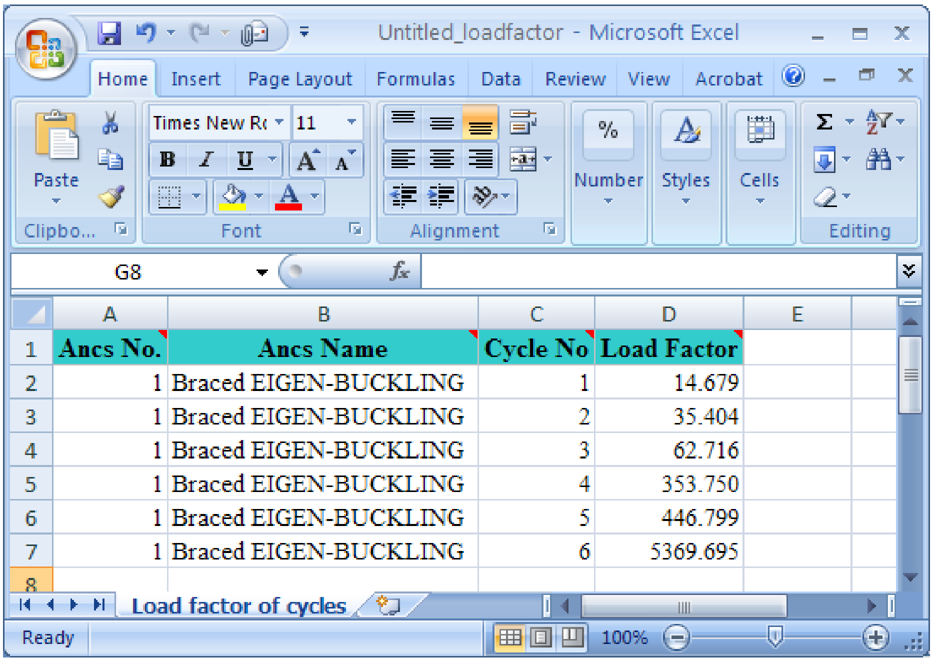


Click ***Select*** and choose the analysis case in the <Select Analysis Case> window;

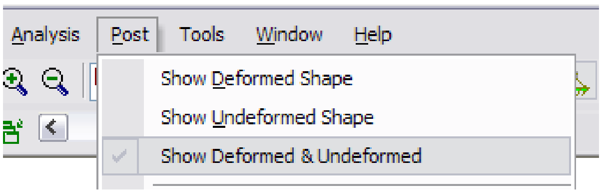


Click ***OK*** to go back to the <Export Eigen-Buckling Load Factor> window;

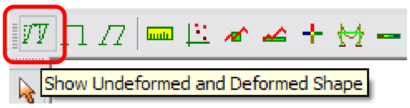
Click ***Export*** , an excel spreadsheet titled '***_loadfactor' will pop up including load factors for all the eigen-buckling modes;



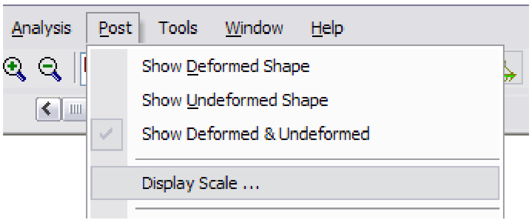
Click the ***Post*** > ***Show Deformed & Undeformed*** command;



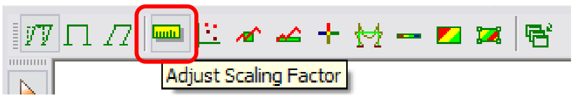
or click the <Show Undeformed and Deformed Shape> shortcut in the toolbar;



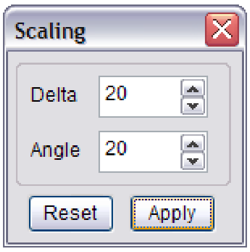
To view the first eigen-buckling mode deformation in a larger scale, click the ***Post*** > ***Display*** command;



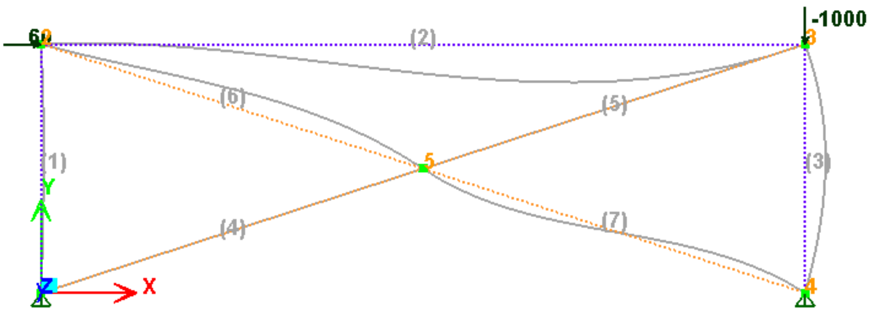
or click the <Adjust Scaling Factor> shortcut in the toolbar;



Enter scaling factors of ‘Delta’ and ‘Angle’ in the <Scaling> window;



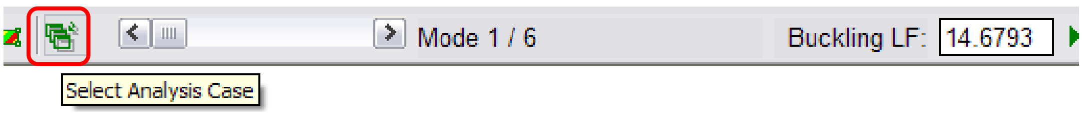
The first eigen-buckling mode deformation of the structure is shown as below;



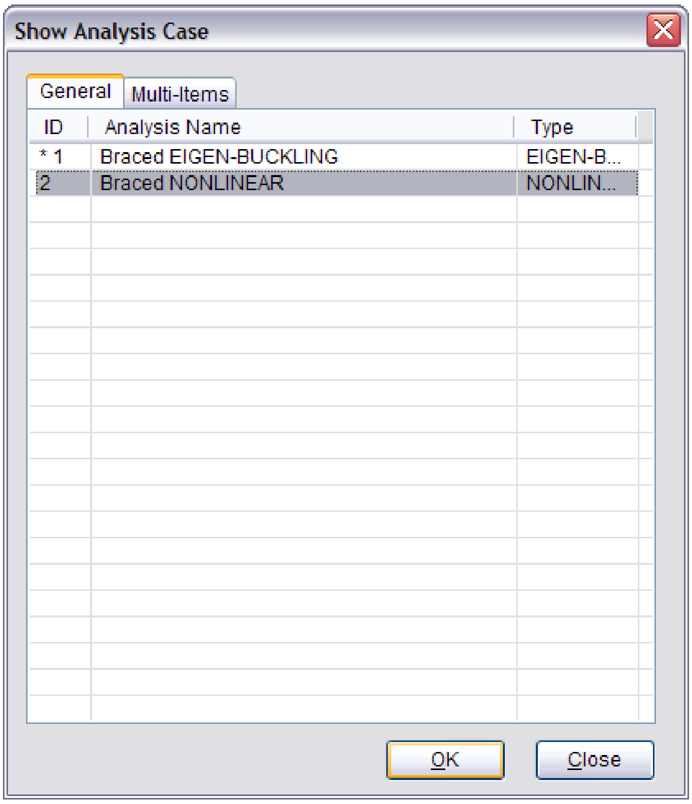
Change the load factor in the sliding bar to view the deformed shapes of higher buckling modes.

2.14 Step 14 – View result: nonlinear analysis

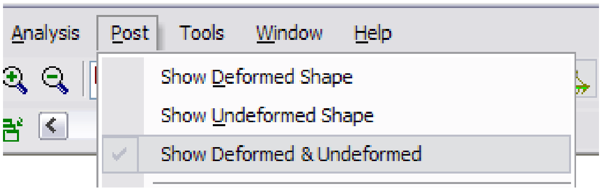
To view the other analysis result, click <Select Analysis Case> shortcut near the sliding bar in the post analysis toolbar;



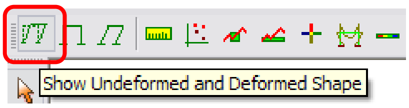
Select the 'Braced NONLINEAR' case from the <Show Analysis Case> window;



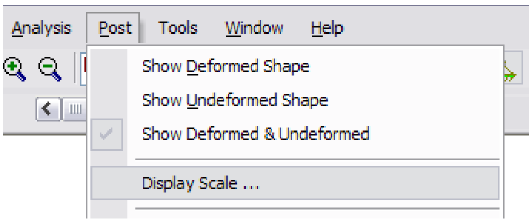
Click the ***Post*** **>** ***Show Deformed & Undeformed*** command;



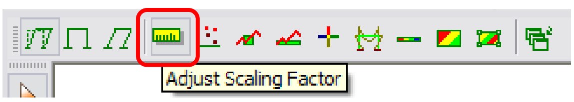
or click the <Show Undeformed and Deformed Shape> shortcut in the toolbar;



To view the nonlinear deformation in a larger scale, click the ***Post*** **>** ***Display Scale*** command;

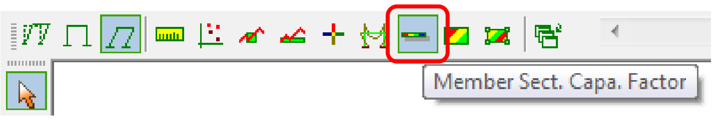


or click the <Adjust Scaling Factor> shortcut in the toolbar;

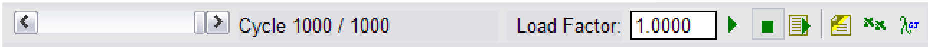


Enter scaling factors of ‘Delta’ and ‘Angle’ in the <Scaling> window;

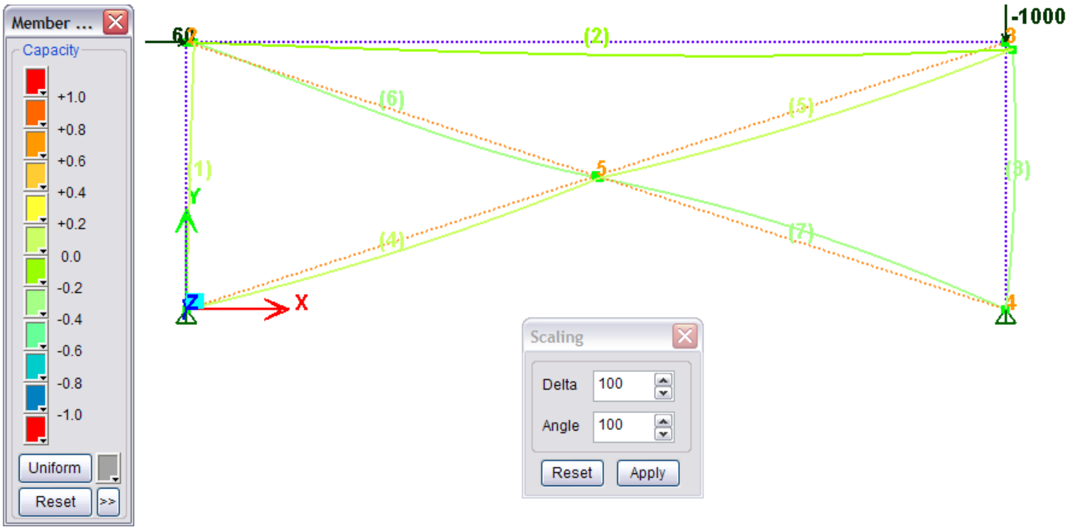
Click the <Member Sect. Capa. Factor> shortcut in the post analysis toolbar, the range of section capacity from +1.0 to -1.0 will be displayed in the <Member> toolbar;



Color of each member in the deformed structure indicates its range of section capacity at different load factor;

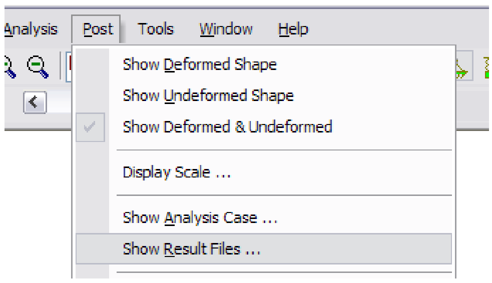


At load factor = 1.000, the deformed shape of structure and the section capacities of deformed members is shown as below;

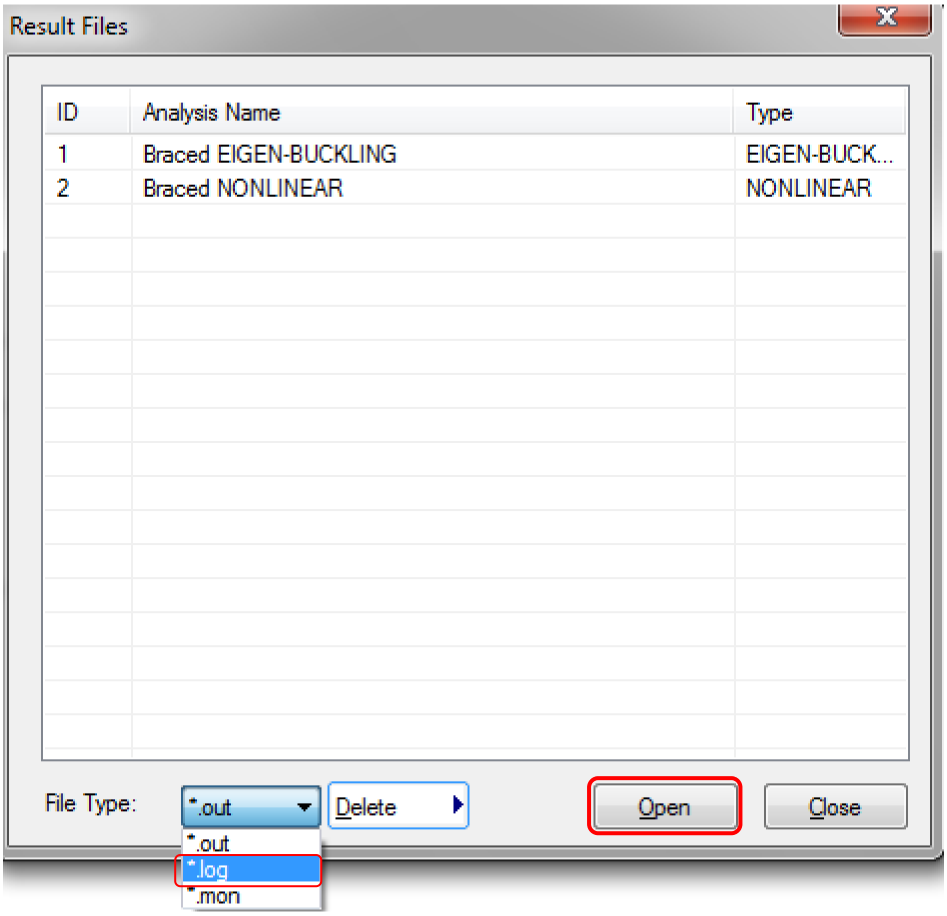


2.15 Step 15 – View result: Show analysis procedure of nonlinear analysis

Click the ***Post*** > ***Show Result Files*** command;

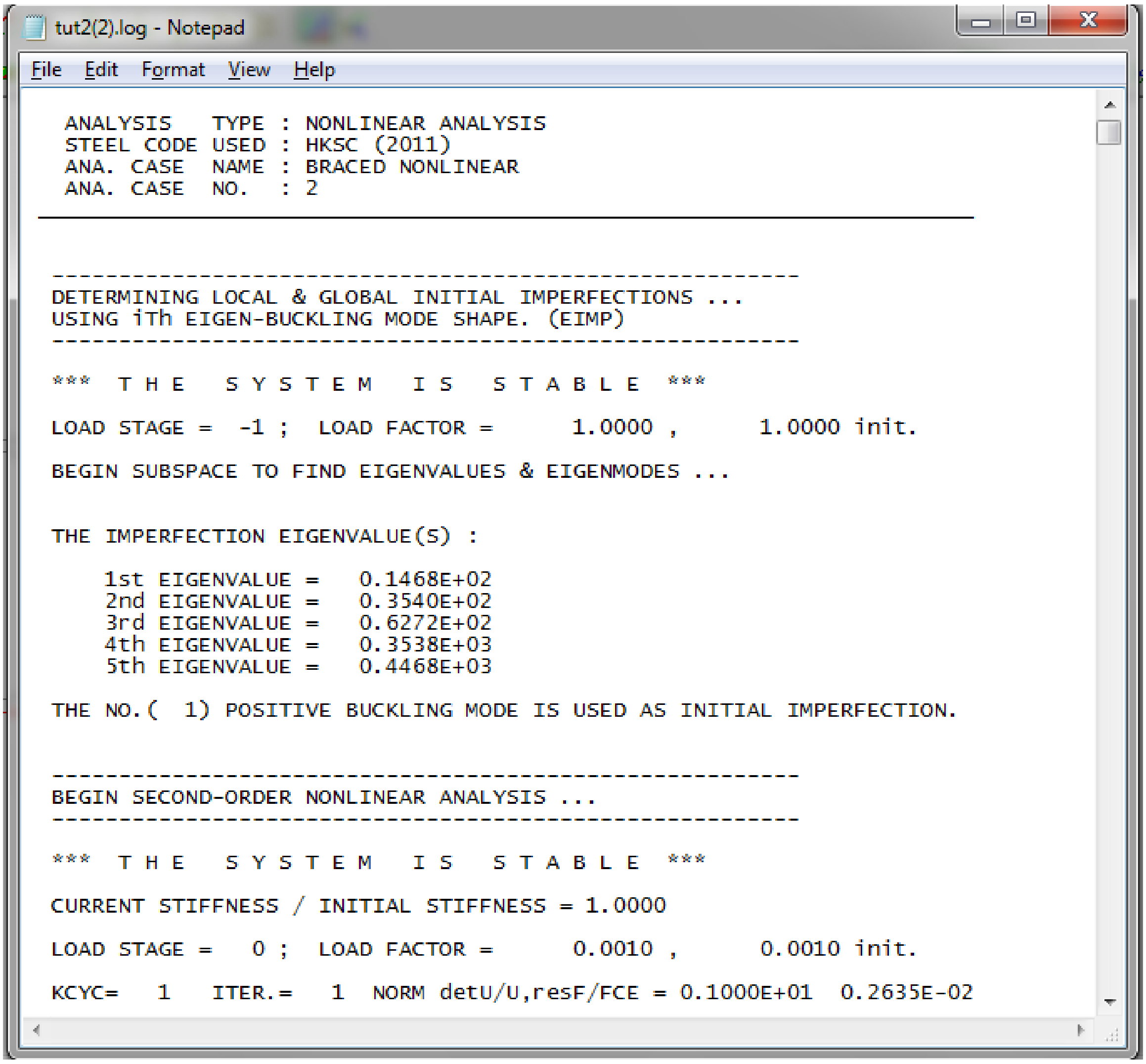


Select the <*.log> file for the ‘Braced NONLINEAR’ analysis case from the <Result Files> window;



Click ***Open***, a text document showing load stage, load cycle, load increment and convergence and so on. The possible warning and error messages will be also printed here for checking model;

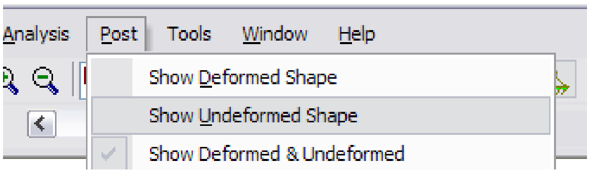
Part of messages has been extracted as below;



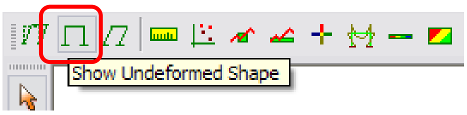
```
tut2(2).log - Notepad
File  Edit  Format  View  Help

 ANALYSIS   TYPE : NONLINEAR ANALYSIS
 STEEL CODE USED : HKSC (2011)
 ANA. CASE  NAME : BRACED NONLINEAR
 ANA. CASE  NO.  : 2

------------------------------------------------------------
DETERMINING LOCAL & GLOBAL INITIAL IMPERFECTIONS ...
USING ith EIGEN-BUCKLING MODE SHAPE. (EIMP)
------------------------------------------------------------

***  T H E   S Y S T E M   I S   S T A B L E  ***

LOAD STAGE =  -1 ;  LOAD FACTOR =      1.0000 ,      1.0000 init.

BEGIN SUBSPACE TO FIND EIGENVALUES & EIGENMODES ...

THE IMPERFECTION EIGENVALUE(S) :

    1st EIGENVALUE =   0.1468E+02
    2nd EIGENVALUE =   0.3540E+02
    3rd EIGENVALUE =   0.6272E+02
    4th EIGENVALUE =   0.3538E+03
    5th EIGENVALUE =   0.4468E+03

THE NO.(  1) POSITIVE BUCKLING MODE IS USED AS INITIAL IMPERFECTION.

------------------------------------------------------------
BEGIN SECOND-ORDER NONLINEAR ANALYSIS ...
------------------------------------------------------------

***  T H E   S Y S T E M   I S   S T A B L E  ***

CURRENT STIFFNESS / INITIAL STIFFNESS = 1.0000

LOAD STAGE =   0 ;  LOAD FACTOR =      0.0010 ,      0.0010 init.

KCYC=   1   ITER.=   1   NORM detU/U,resF/FCE = 0.1000E+01   0.2635E-02
```

2.16 Step 16 – Delete members for unbraced case

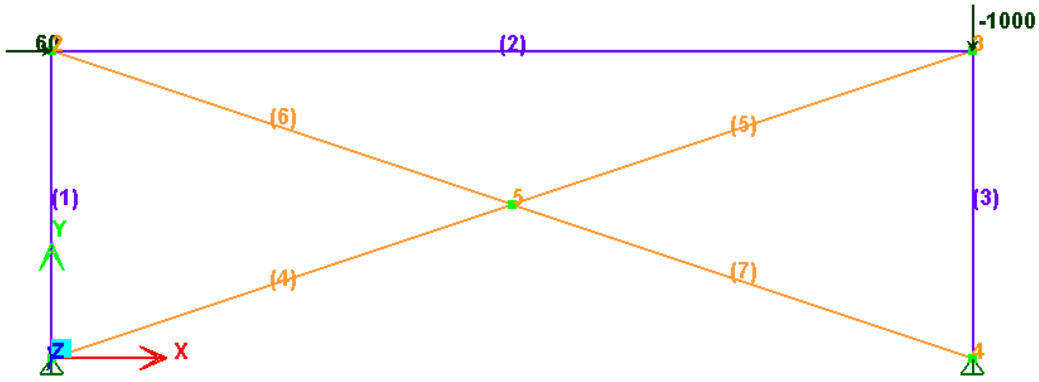
Click the ***Post*** **>** ***Show Undeformed Shape*** command;



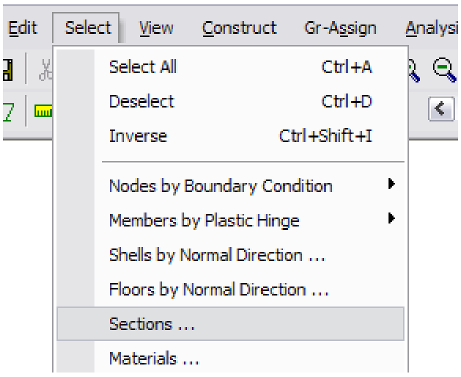
or click the <Show Undeformed Shape> shortcut in the post analysis toolbar;



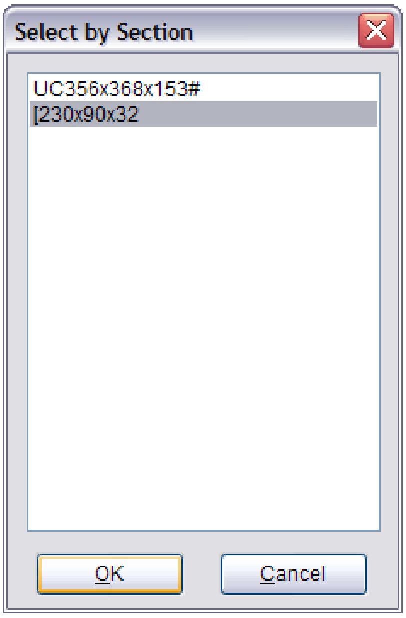
The structure returns back to its undeformed shape;



Click the ***Select*** **>** ***Sections*** command;

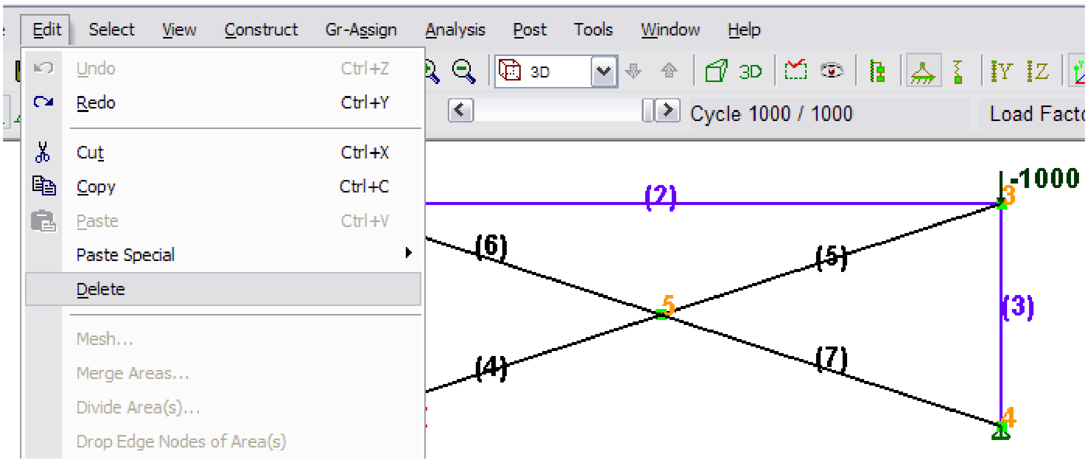


Choose the channel section 230 x 90 x 32 in the <Select by Section> window;



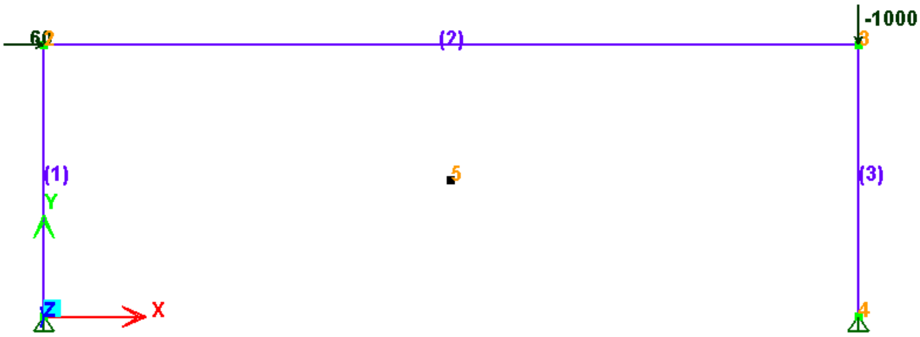
Click ***OK*** to continue, four bracing members (4) – (7) are then selected, or simply select the members on the screen;

Click the ***Edit*** **>** ***Delete*** command;

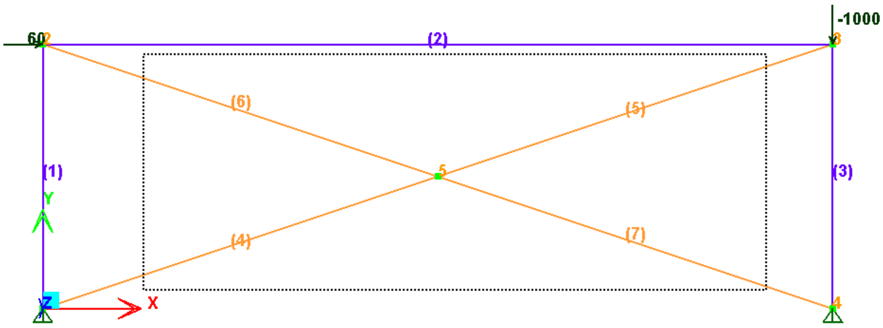


The four bracing members disappear leaving only the node 5;

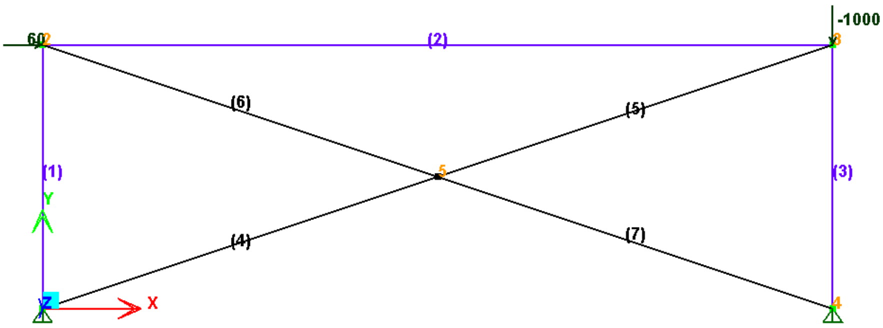
Click on the node 5 and repeat the above deleting procedure or press the 'Delete' button on the keyboard directly;



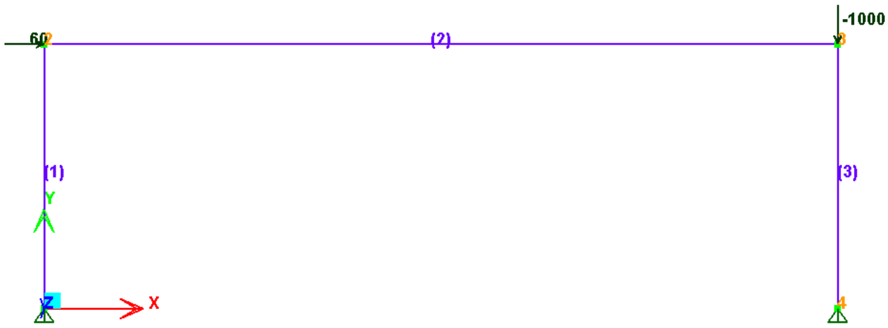
or left click and hold the mouse to select all the members and nodes to be deleted;



Release the mouse, then members (4) – (7) and node 5 are selected;



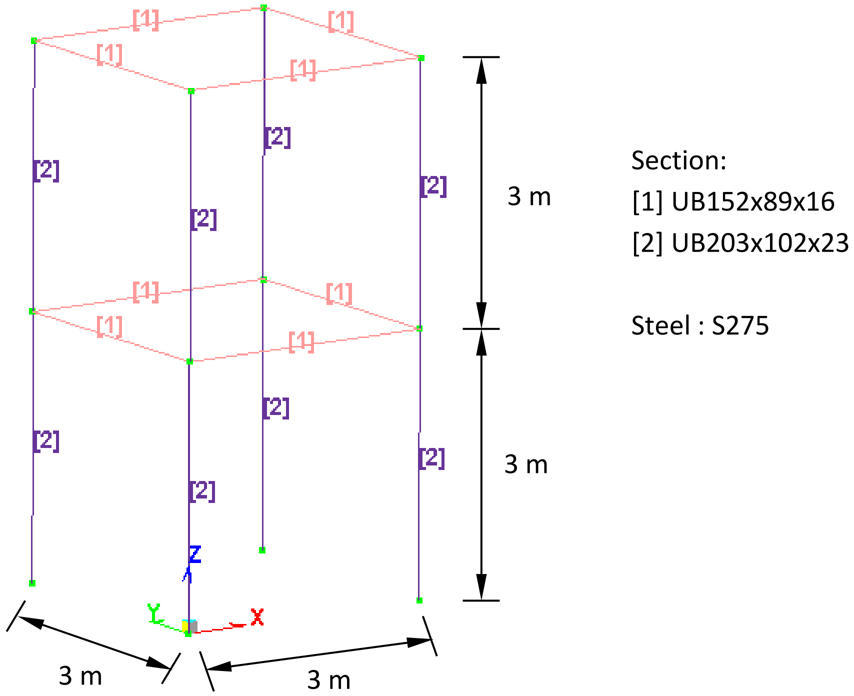
Repeat the deleting procedure above, the unbraced frame is then constructed;



2.17 Step 17 – Unbraced portal frame

All the other properties and load cases of the unbraced frame remain the same as the braced one, perform the Eigen-Buckling Analysis and the Nonlinear Analysis again, the analysis result is then obtained.

3. TUTORIAL 3 – Space Frame



For vertical members, local y-axis in global Y-direction;

For horizontal members, local xy plane parallel to global z axis.

Determine the elastic critical load factor λ_{cr} and check the structural adequacy by second-order analysis under the following load combinations:

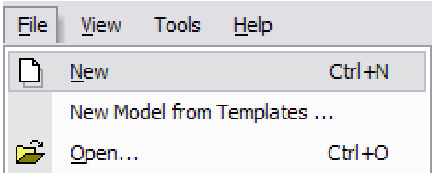
- LC1: 1.4 Dead Load + 1.6 Live Load
- LC2: 1.4 Dead Load + 1.4 Wind Load
- LC3: 1.2 Dead Load + 1.2 Live Load + 1.2 Wind Load

3.1 Step 1 – Start NIDA

Double click the “NIDA” icon on the desktop or in the corresponding folder upon successful installation of NIDA.

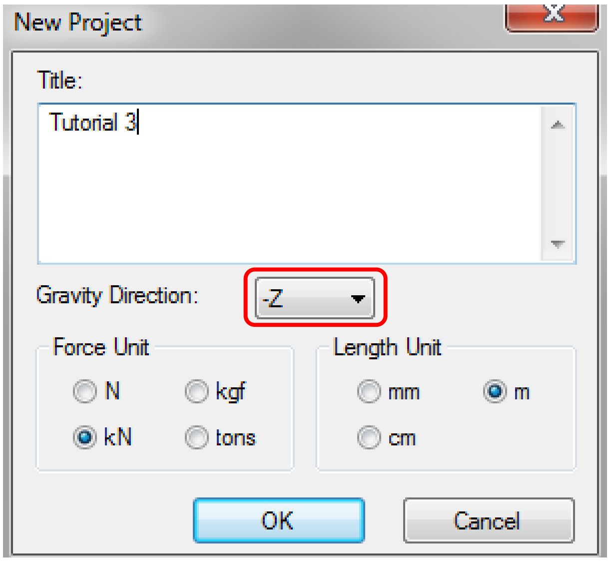
3.2 Step 2 – Create a new project file

Click the ***File* > *New*** command,



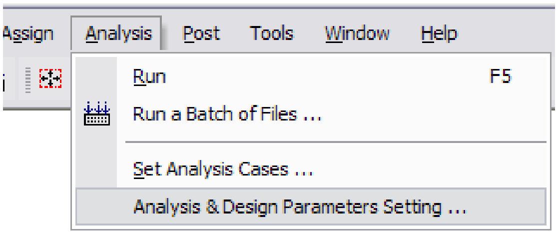
or find the <New> shortcut button in the toolbar;

Enter the new project <Title> ‘Tutorial 2 Braced’ in the popped window and select the <Gravity Direction> to be ‘-Y’, <Force Unit> ‘kN’ and <Length Unit> ‘m’.



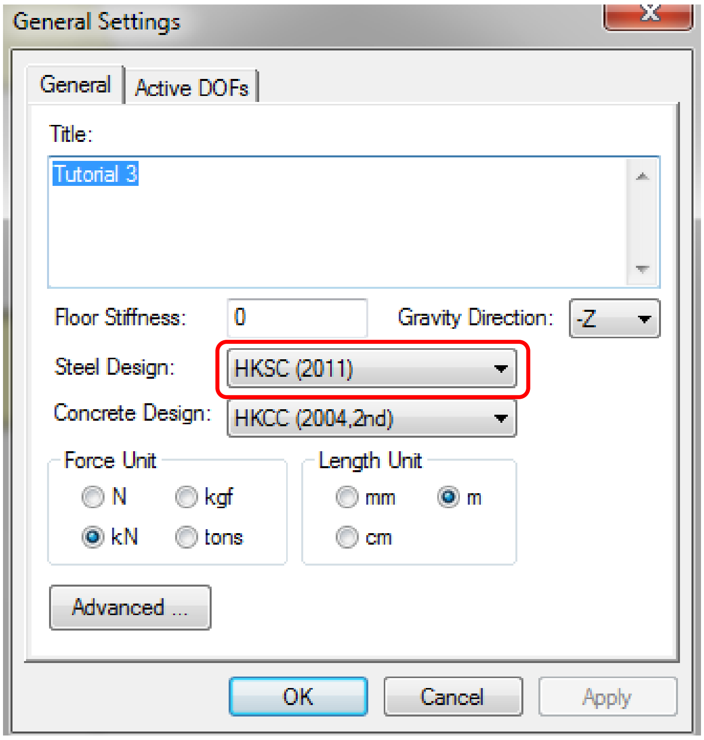
Click ***OK*** to continue.

Click the ***Analysis* > *Analysis & Design Parameters Setting*** command,

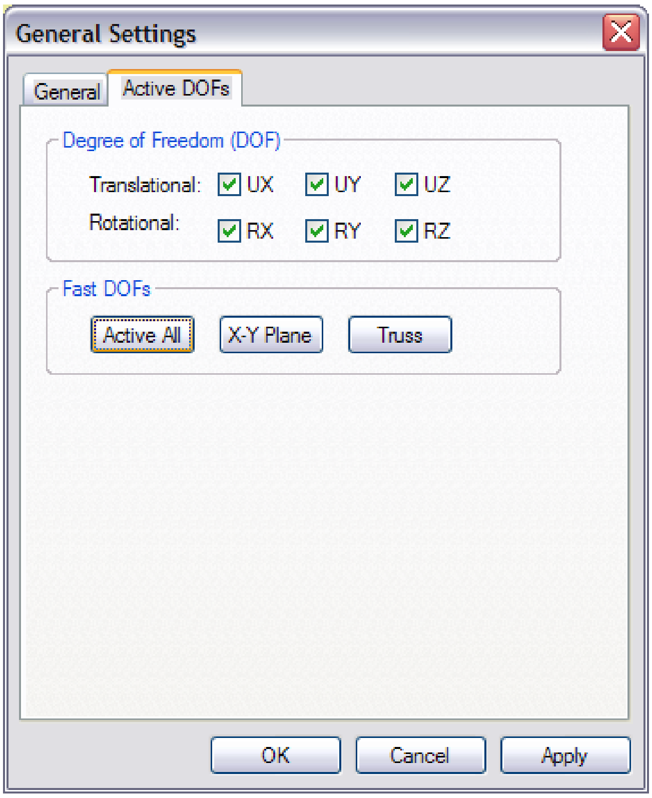


The parameters already set could also be viewed or changed in the <General Settings> window;

Select the steel design code HKSC (2011);



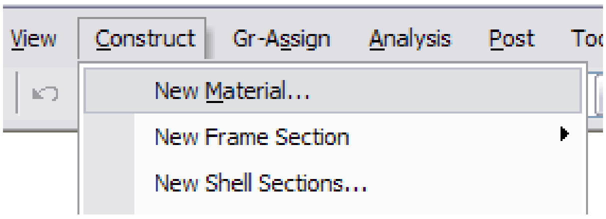
Select 'Fast DOFs – Active All' under 'Active DOFs' for a plane portal frame;



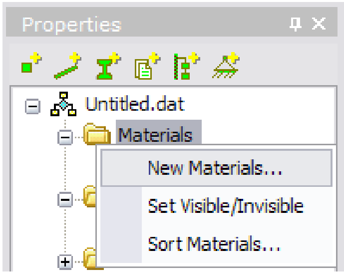
Click ***Apply*** and then ***OK*** to continue.

3.3 Step 3 – Create material

Click the ***Construct*** > ***New Material*** command,

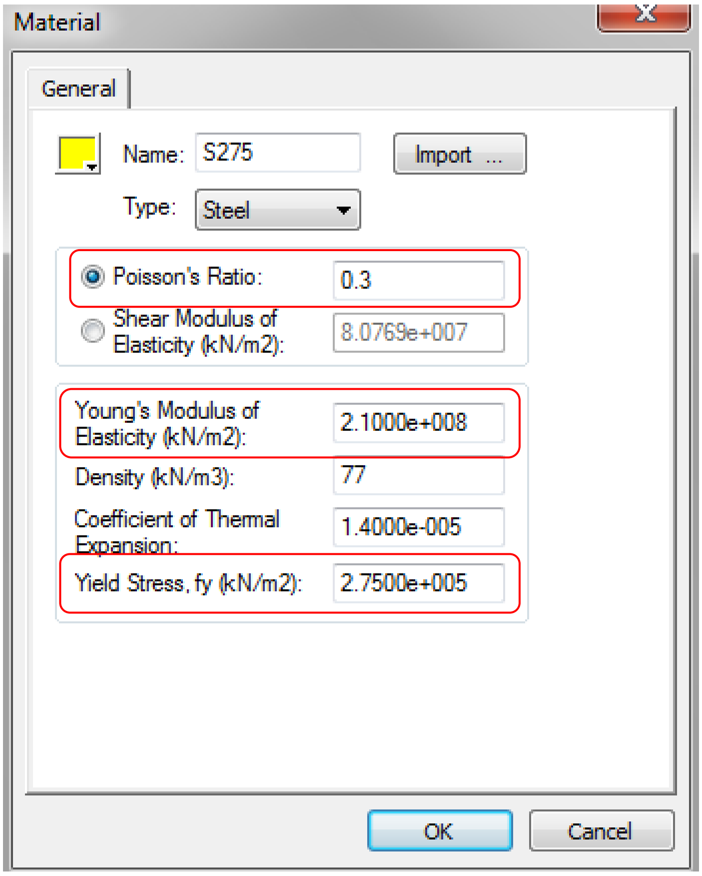


or right click <Materials> in the ***Properties*** window and then click <New Materials>;



Default steel grade set up in the program is S275;

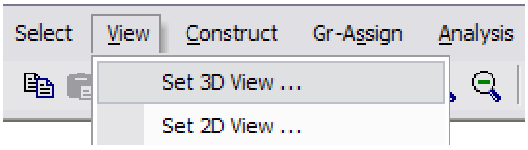
Double click <S275> to view and change the material properties in the <Materials> window if required, such as changing the material color only for this tutorial;



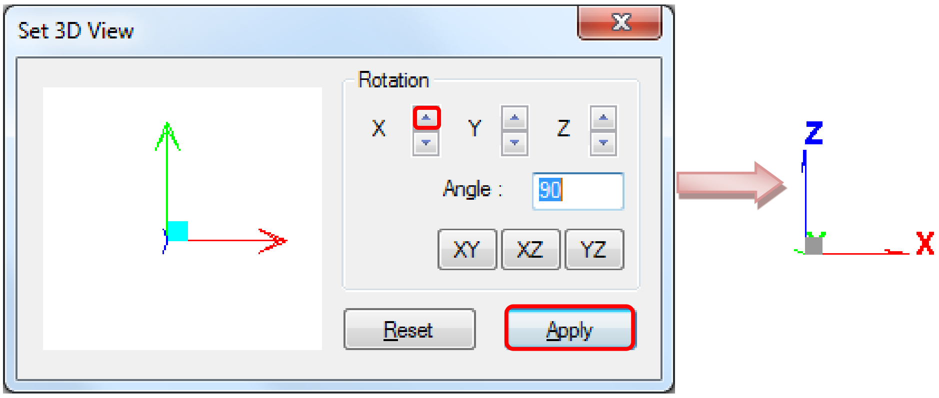
Click ***OK*** to continue.

3.4 Step 4 – Set gravity direction

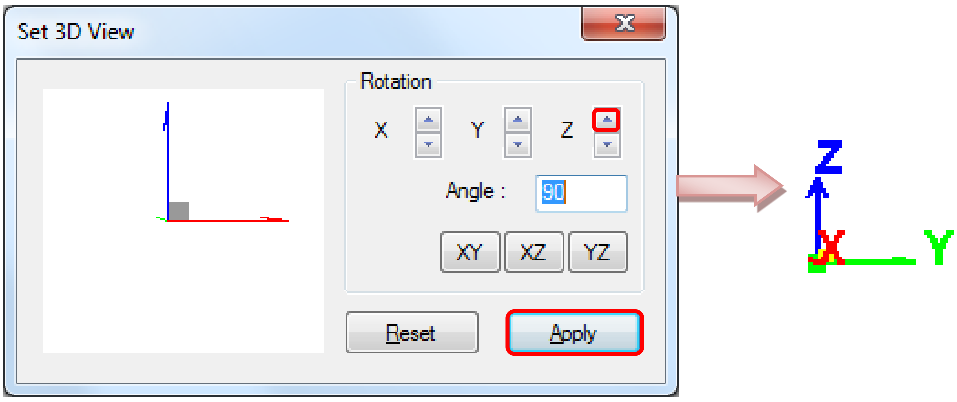
Click the ***View*** > ***Set 3D View*** command,



Enter 90 for ‘Angle’, click the up arrow of X and ***Apply*** to rotate the existing coordination 90 degrees about X axis;

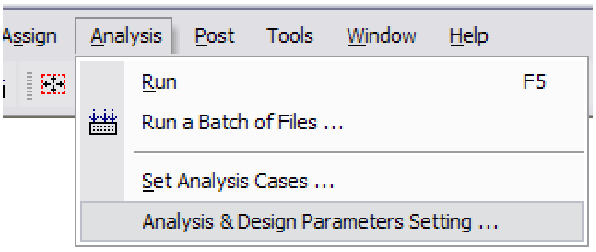


Click the up arrow of Z and ***Apply*** to further rotate the existing coordination 90 degrees about Z axis;

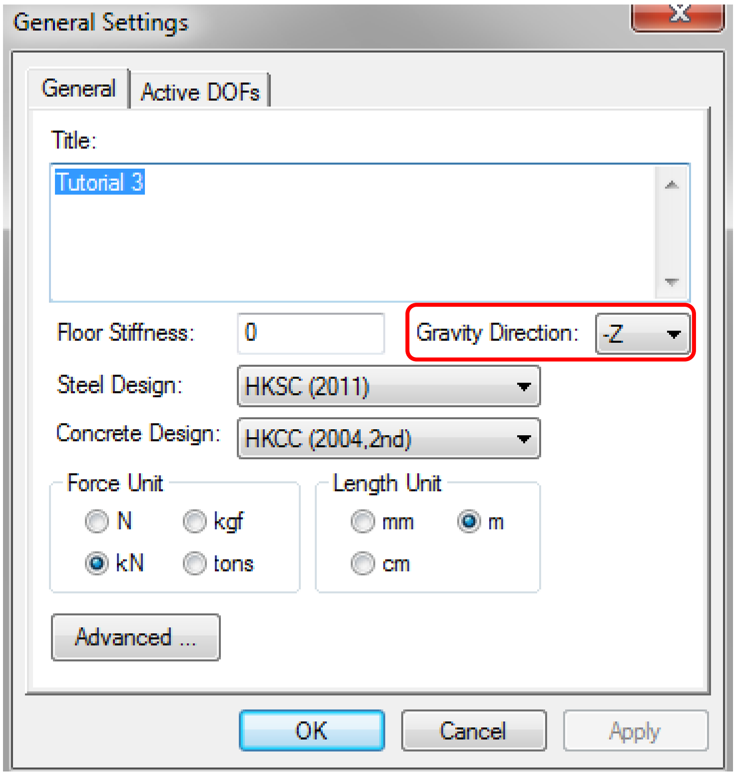


Click ***Close*** in dialog or ***Cancel*** button to finish rotation;

Click the ***Analysis*** > ***Analysis & Design Parameters Setting*** command,



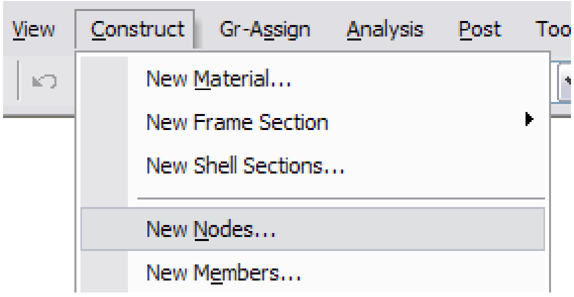
Change the ‘Gravity Direction’ to be ‘-Z’ in the <General Settings> window;



Click ***OK*** to finish this step.

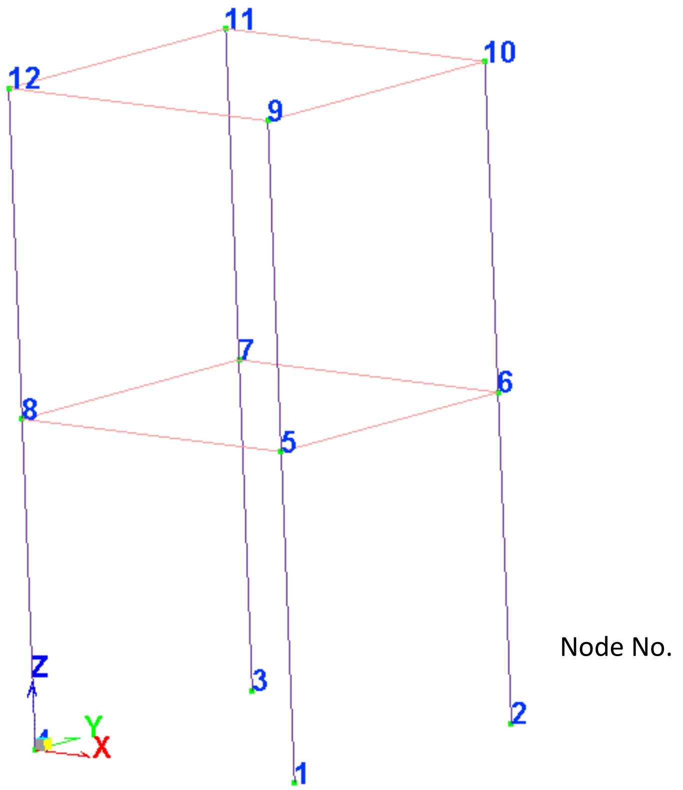
3.5 Step 5 – Add nodes

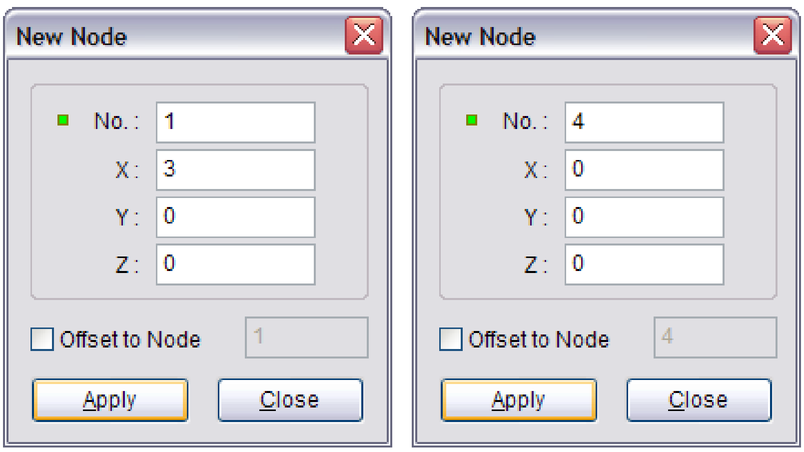
Click the ***Construct*** > ***New Nodes*** command,



or click the <Add Node> shortcut in the ***Properties*** window;

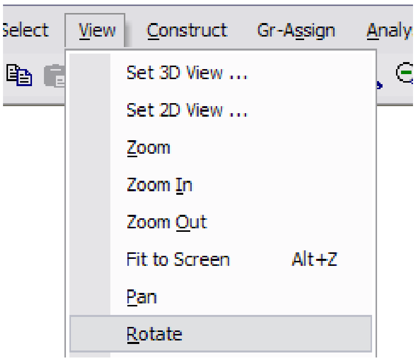
Enter the x, y and z coordinates for node No. 1 and click ***Apply*** to continue adding nodes on the xy plane with z = 0;



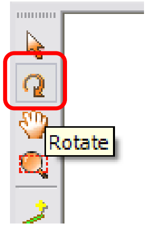


On completion of the last node within this plane, click ***Apply*** and then ***Close*** the <New Node> window;

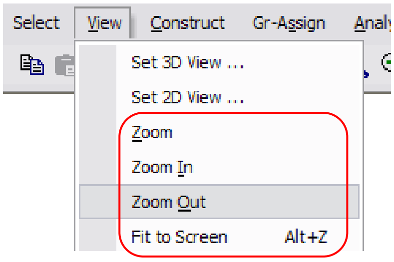
Click the ***View*** > ***Rotate*** command to view the overlapping nodes in a different angle of view,



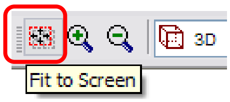
or click the <Rotate> shortcut in the toolbar;

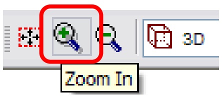


Click the ***View*** > ***Zoom In / Zoom Out / Fit to Screen*** adjust the size of the structure on the screen;



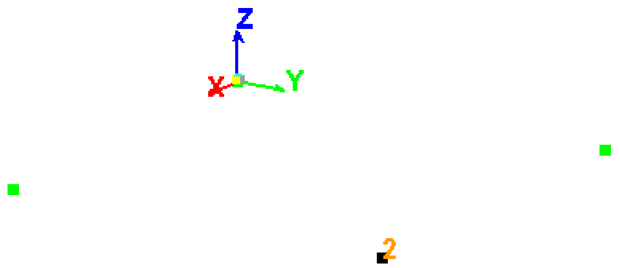
or click the <Fit to Screen / Zoom In / Zoom Out> shortcut in the toolbar;

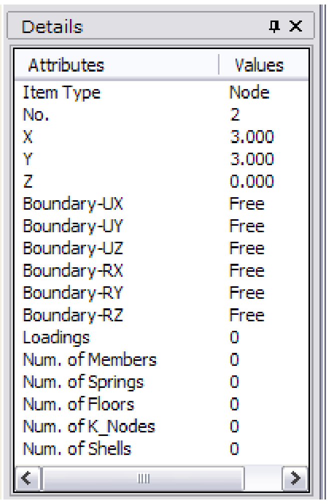




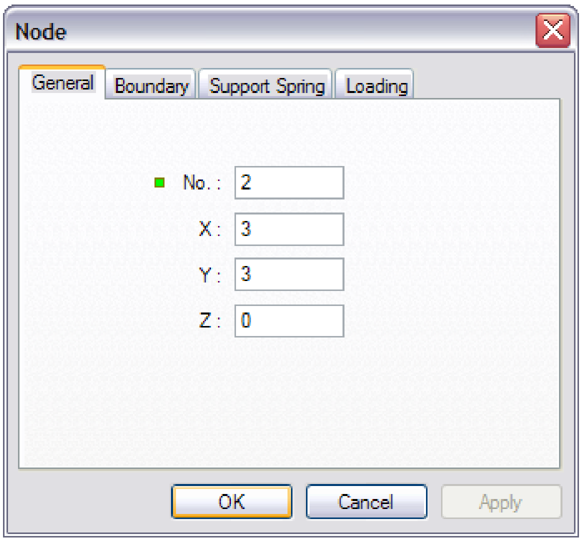
or roll mouse wheel up and down to zoom in and out to obtain the best fit view;

To view the details of a particular node, click the node, then all the details including coordinates, boundary conditions, loadings, etc, are displayed in the ***Details*** window;





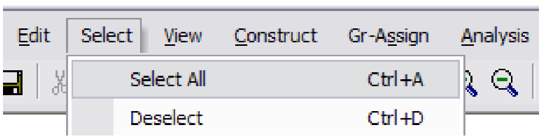
or double click the node to view the node details under ‘Coordinates’, ‘Boundary’, ‘Support Spring’ and ‘Loading’ menus, make changes in the ***Node*** window if needed;



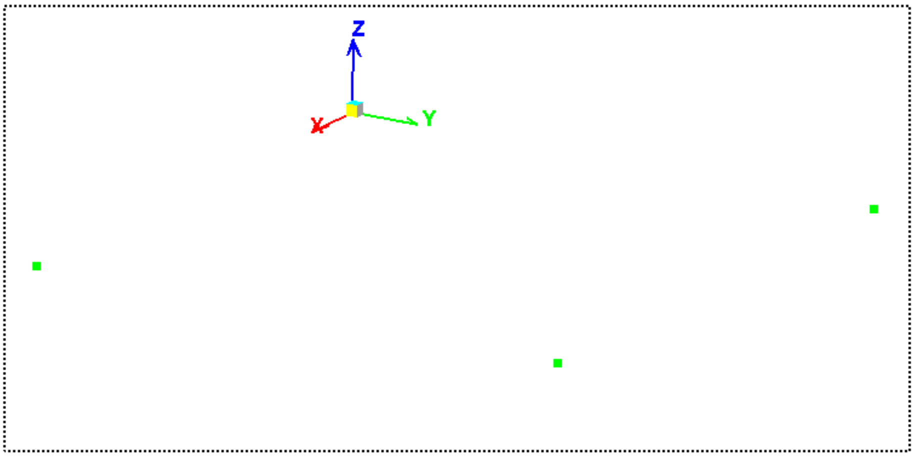
Click ***OK*** to continue.

3.6 Step 6 – Copy and paste nodes

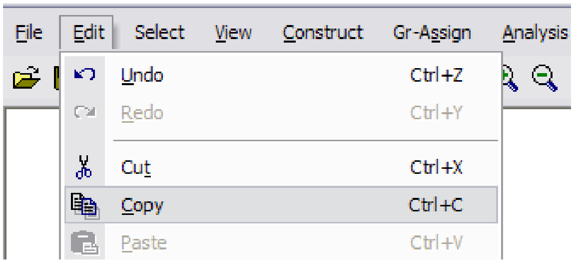
Click the ***Select > Select All*** command,



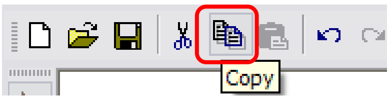
or left click and hold the mouse to frame all the nodes in,



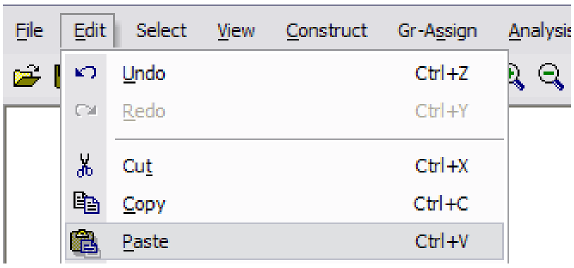
Click the ***Edit* > *Copy*** command;



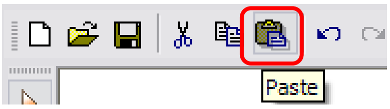
or click the <Copy> shortcut in the toolbar;



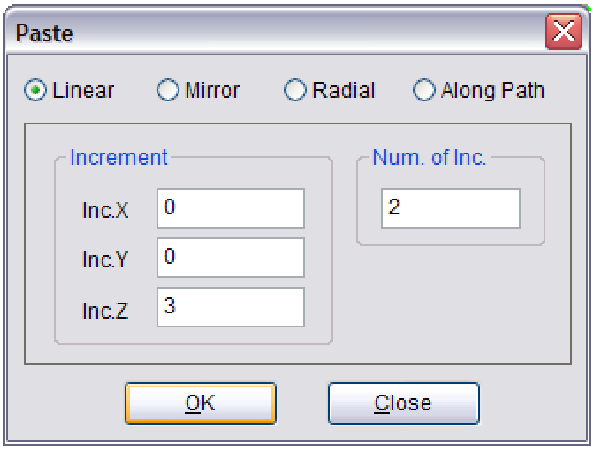
Click the ***Edit* > *Paste*** command;



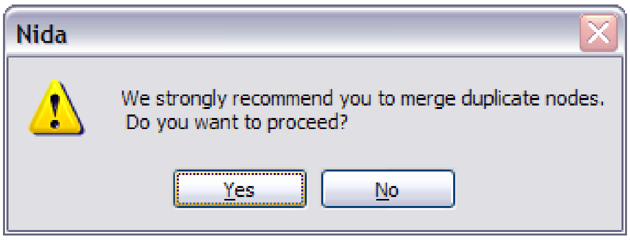
or click the <Paste> shortcut in the toolbar;



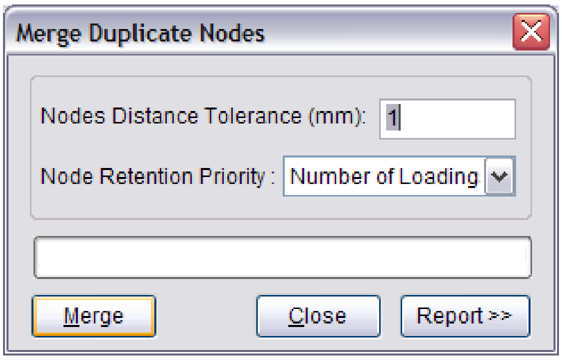
Select 'Linear', enter the 'Increment' along Z axis and 'Num of Inc.' in the <Paste> window;



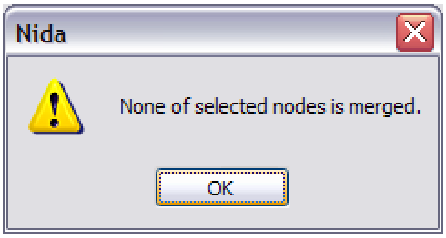
Click ***OK***, then a message box will pop up recommending merging duplicate nodes;



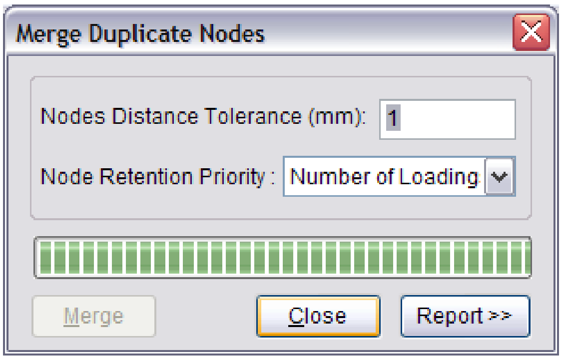
Click ***Yes*** to proceed and the message box will close;



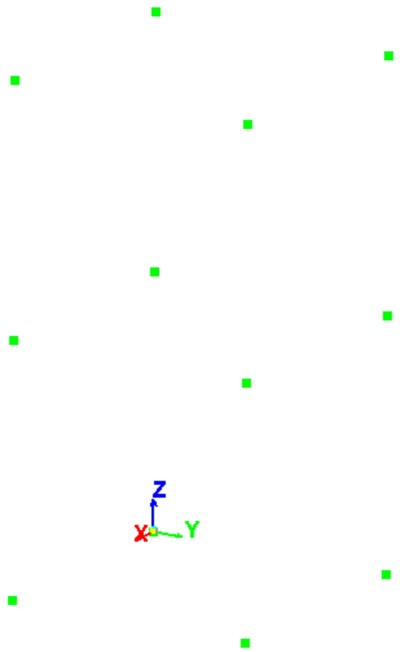
Click ***Merge***, a new message box will pop up to show the number of nodes merged;



Click ***OK*** , and ***Close*** the <Merge Duplicate Nodes> window to finish this step;

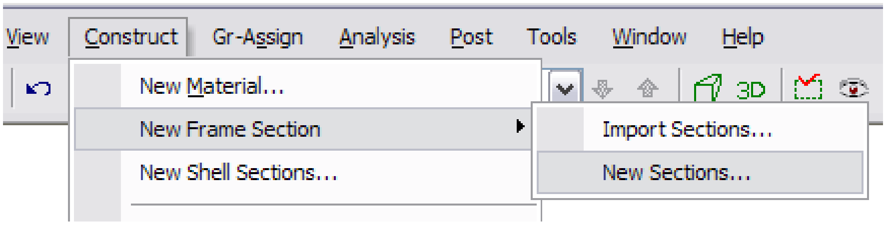


Then all the nodes are constructed as shown below.



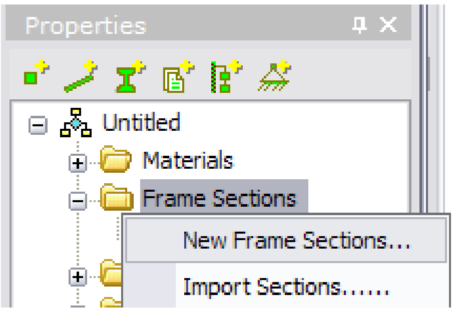
3.7 Step 7 – Add sections

Click the ***Construct*** > ***New Frame Section*** > ***New Sections*** command,

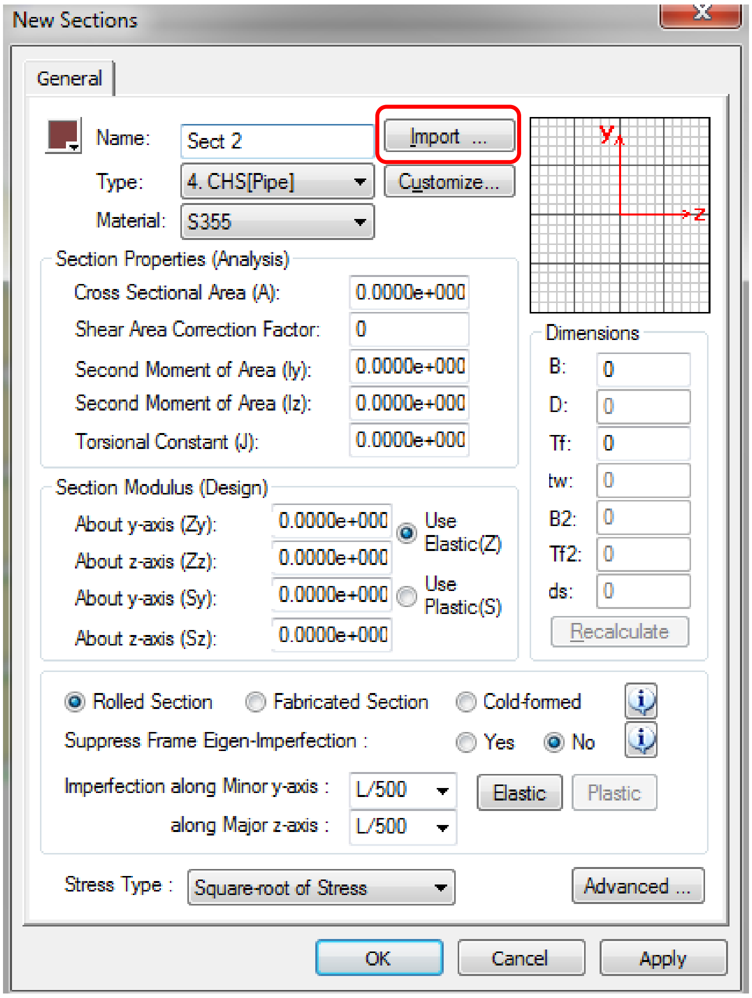


or click the <Add Section> shortcut in the ***Properties*** window;

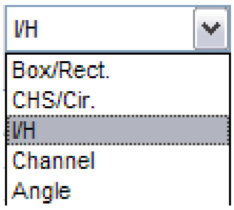
or right click <Frame Sections> in the ***Properties*** window and then click <New Frame Sections>;



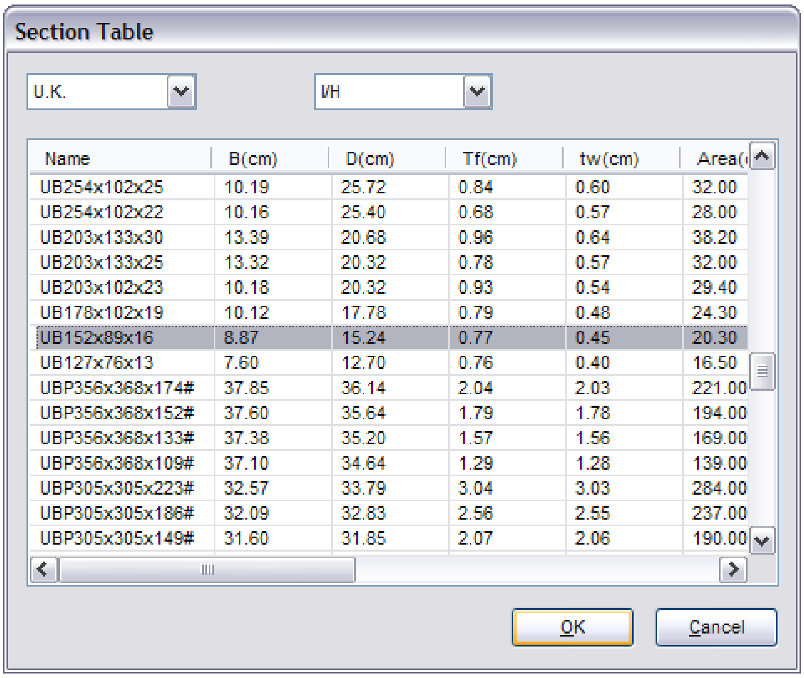
Click <Import> button in the <New Sections> window to import cross sections;



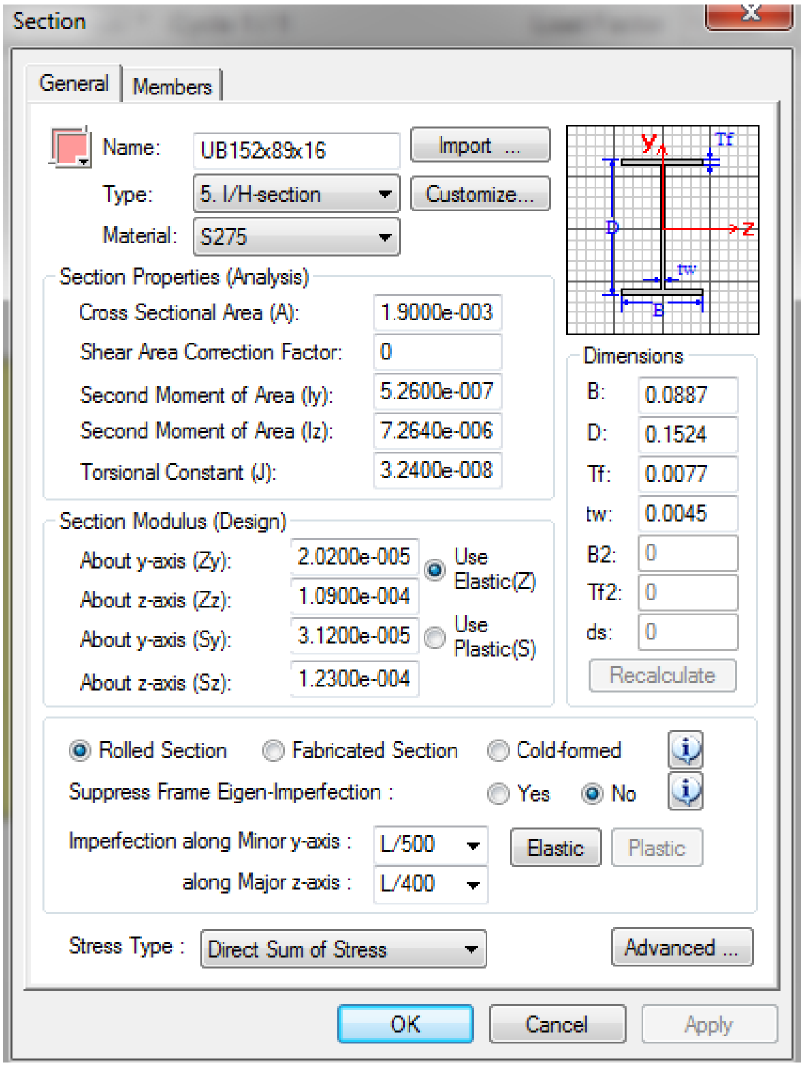
Select cross sections under different categories, country or shape;



Choose a UB section UB 152 x 90 x 16 from the <Section Table> window;



Click ***OK*** to import UB 152 x 89 x 16 and then all the section properties are displayed in the <New Sections> window;

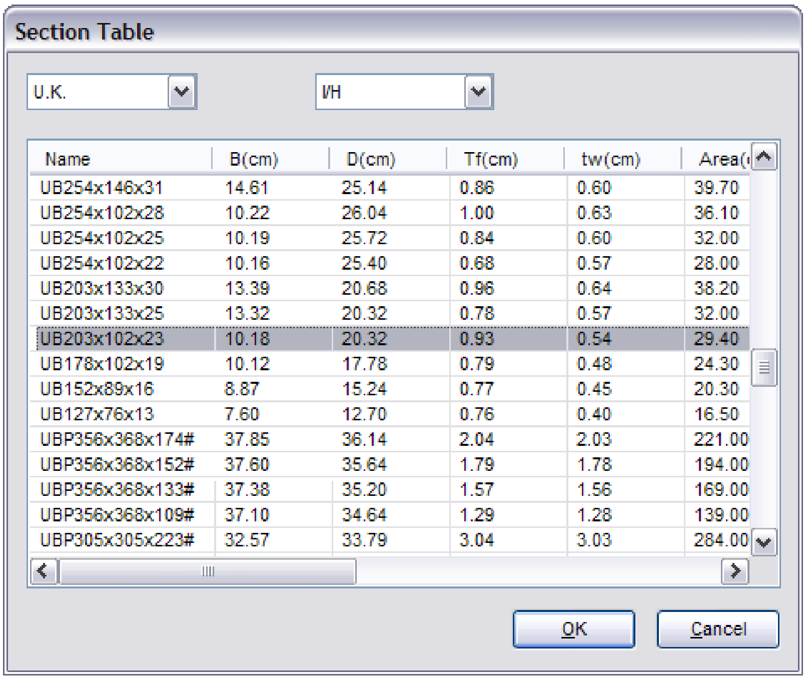


Select a different colour for this UB section, check all the section properties and make changes to the area, second moment of area and torsional constant as requirement;

P – δ imperfections of the member in this section are generated automatically by the software according to the design code in use, but could be manually adjusted as well to fulfil special requirement.

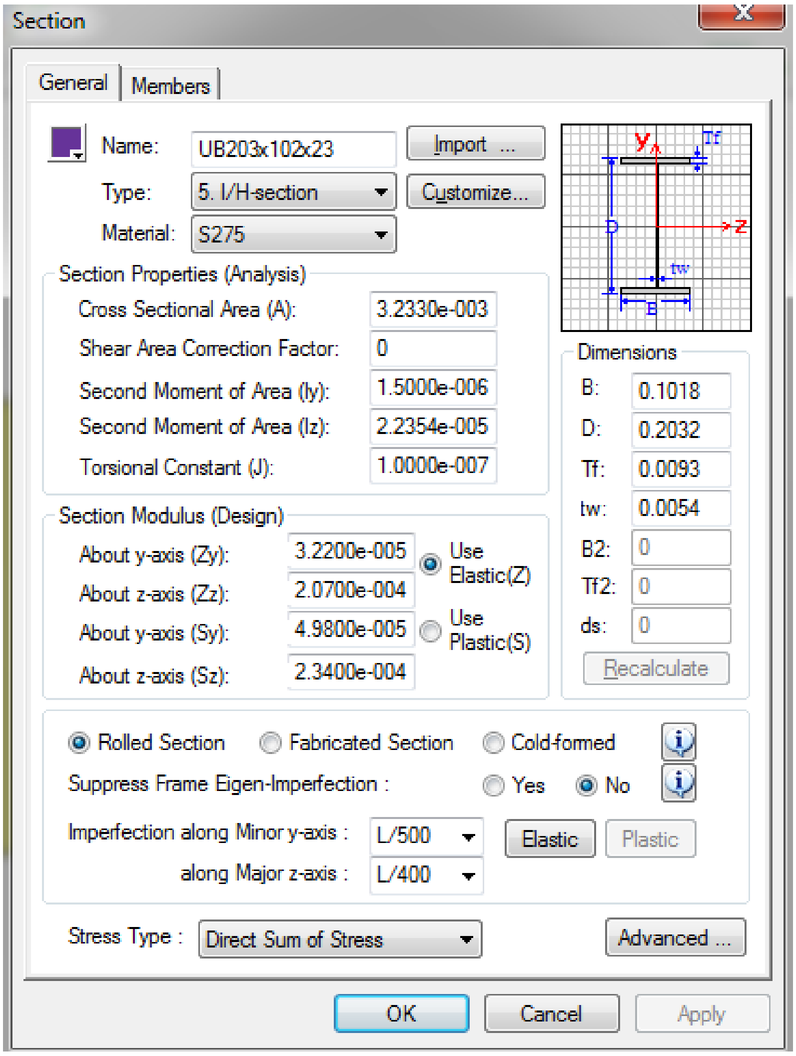
Click ***Apply*** and ***Import*** to continue to add a new section for the column;

Choose a UB section UB 203 x 102 x 23 from the <Section Table> window with similar section properties to those required for the columns;



Name	B(cm)	D(cm)	Tf(cm)	tw(cm)	Area(
UB254x146x31	14.61	25.14	0.86	0.60	39.70
UB254x102x28	10.22	26.04	1.00	0.63	36.10
UB254x102x25	10.19	25.72	0.84	0.60	32.00
UB254x102x22	10.16	25.40	0.68	0.57	28.00
UB203x133x30	13.39	20.68	0.96	0.64	38.20
UB203x133x25	13.32	20.32	0.78	0.57	32.00
UB203x102x23	10.18	20.32	0.93	0.54	29.40
UB178x102x19	10.12	17.78	0.79	0.48	24.30
UB152x89x16	8.87	15.24	0.77	0.45	20.30
UB127x76x13	7.60	12.70	0.76	0.40	16.50
UBP356x368x174#	37.85	36.14	2.04	2.03	221.00
UBP356x368x152#	37.60	35.64	1.79	1.78	194.00
UBP356x368x133#	37.38	35.20	1.57	1.56	169.00
UBP356x368x109#	37.10	34.64	1.29	1.28	139.00
UBP305x305x223#	32.57	33.79	3.04	3.03	284.00

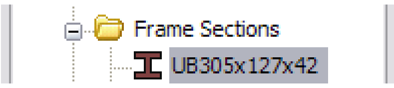
Click ***OK*** to import UB 203 x 102 x 23, then all the section properties are displayed in the <New Sections> window;



Select a different color for this section for columns and check all the section properties;

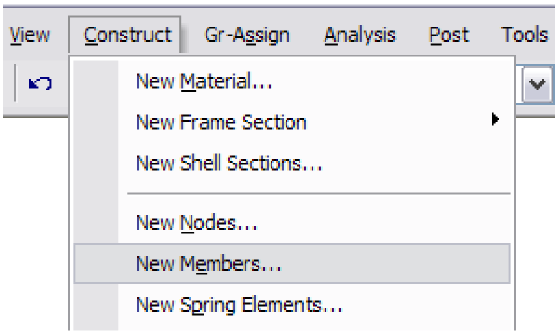
Click ***OK*** to finish adding sections.

There is a default frame section 'UB 305 x 127 x 42' generated by the software. Delete it or change the properties if needed.



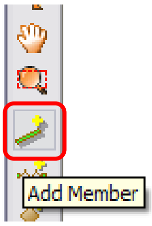
3.8 Step 8 – Add members

Click the ***Construct*** > ***New Members*** command,

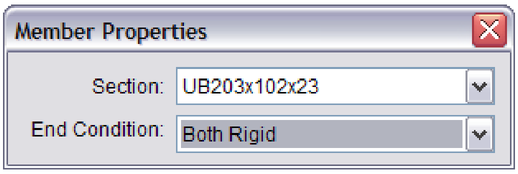


or click the <Add Member> shortcut in the ***Properties*** window;

or click the <Add Member> shortcut in the toolbar;

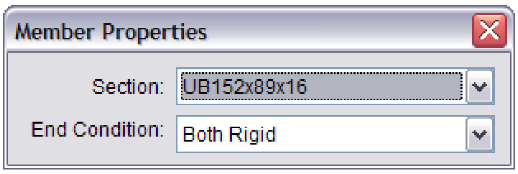


Select the 'Section' and 'End Condition' for the column members in the <Properties of Member> window;



Click the start node and then the end node; if this end node is connected to other members, continue to click the next end node and right click to stop adding members.

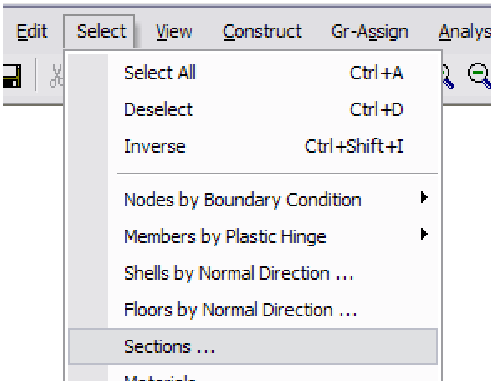
Change the member to beam section with both ends rigid,



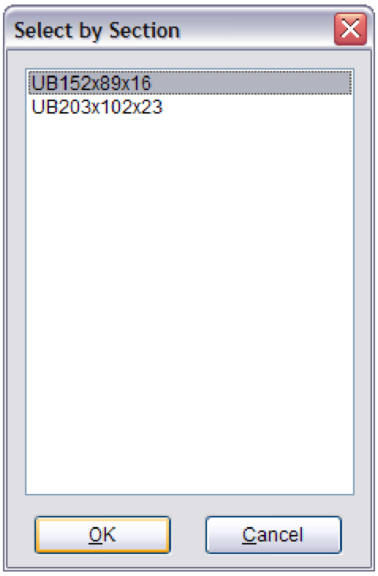
Add members to form the lower level beams first and then click ***Close*** in dialog or press ***Esc*** in keyboard;

Apply copy and paste procedure to the beams for the upper level ones;

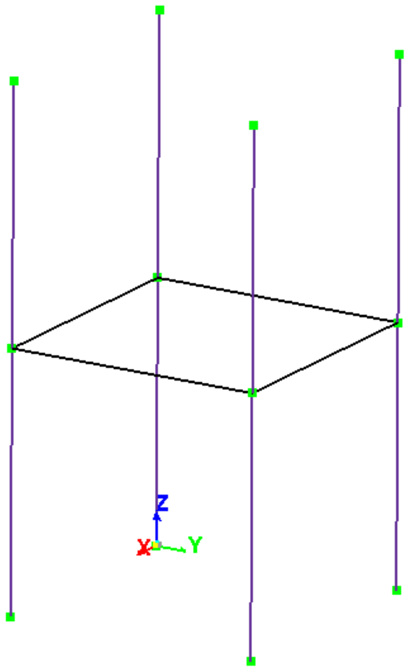
Click the ***Select > Sections*** command;



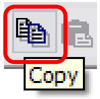
Select beam section UB 152 x 89 x 16 from the <Select by Section> window;



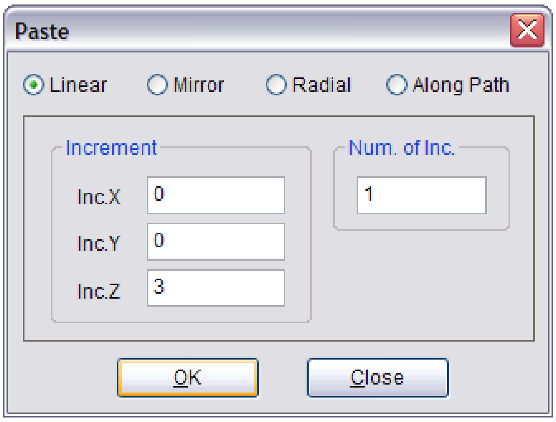
Click ***OK*** to close the window then the beam members are selected, or simply select the beam members on the screen;



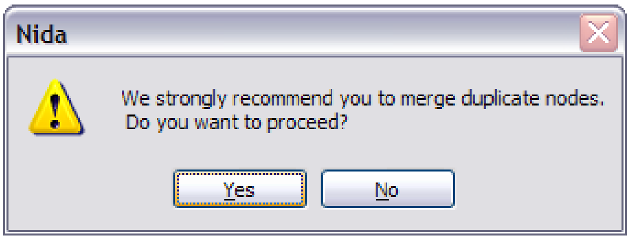
Click the <Copy> shortcut and the <Paste> shortcut in the toolbar;



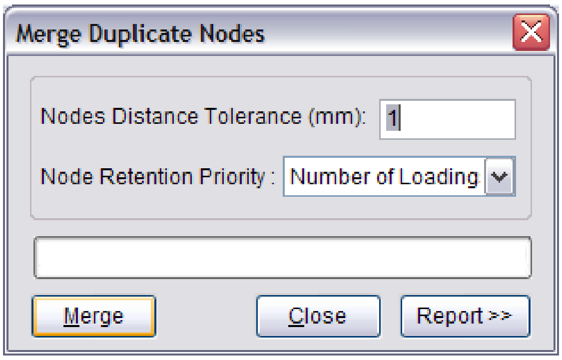
Select ‘Linear’, enter the ‘Increment’ along Z axis and ‘Num of Inc.’ in the <Paste> window;



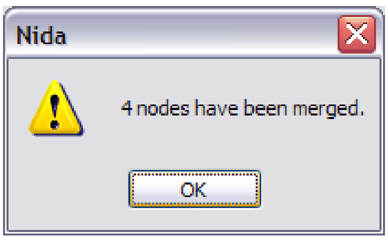
Click ***OK***, then a message box will pop up recommending merging duplicate nodes;



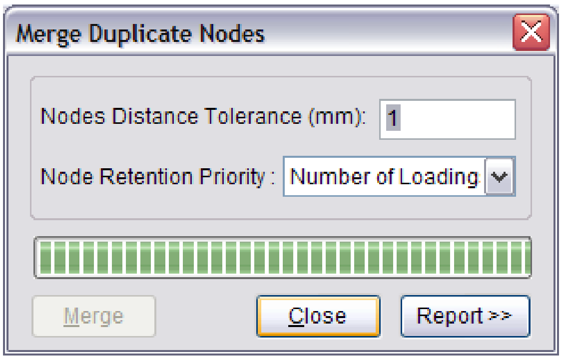
Click ***Yes*** to proceed and the message box will close;



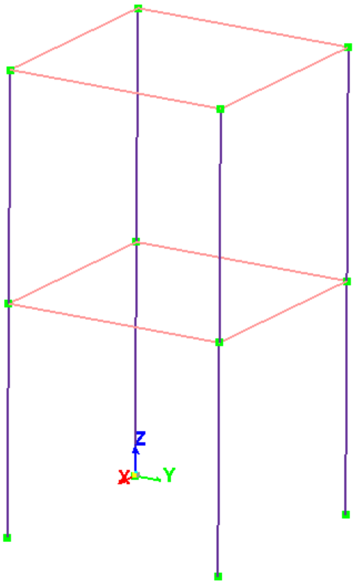
Click ***Merge***, a new message box will pop up to show the number of nodes merged;



Click ***OK*** , and ***Close*** the <Merge Duplicate Nodes> window to finish this step;



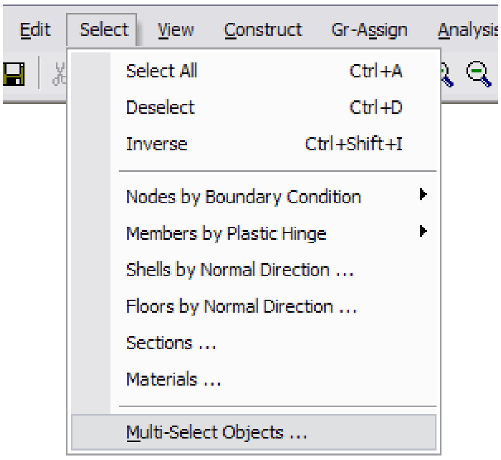
Then all the nodes are constructed as shown below.



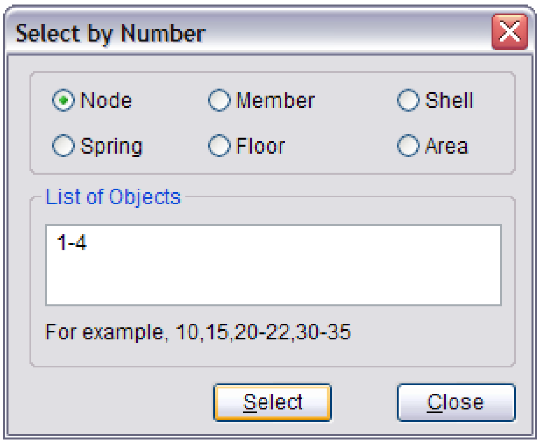
Check the numberings of this structure and change them according to a successive sequence if needed.

3.9 Step 9 – Add boundary conditions

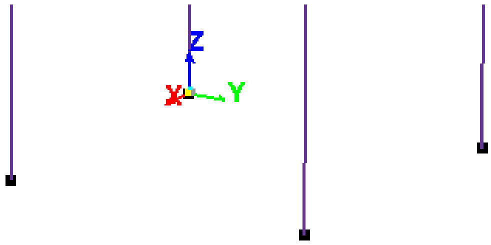
Click the ***Select > Multi-Select Objects*** command,



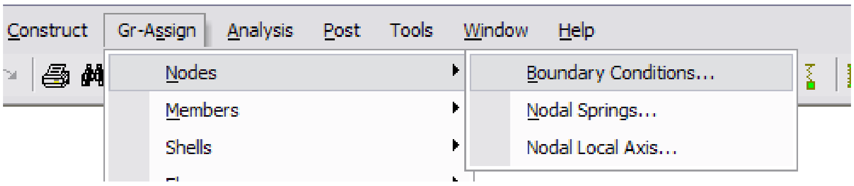
Select ‘Node’ and enter ‘1-4’ to the ‘List of Objects’;



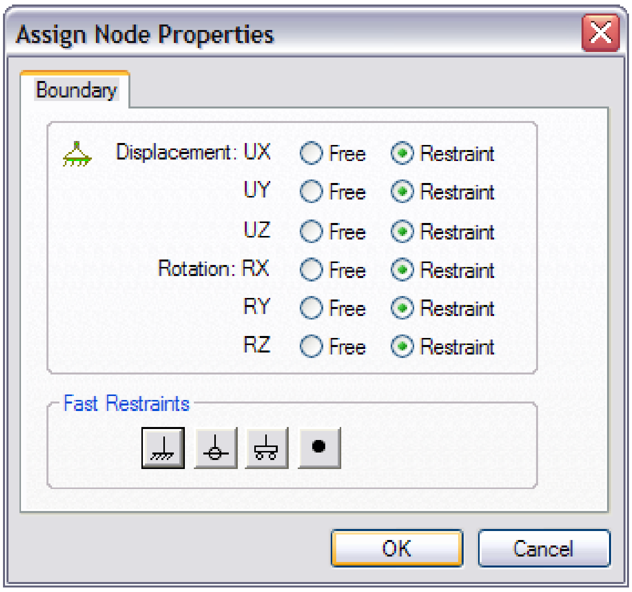
Click ***Select*** then the selected nodes are shown as below;



Click the ***Gr-Assign > Nodes > Boundary Conditions*** command to assign boundary conditions to the nodes in a group;



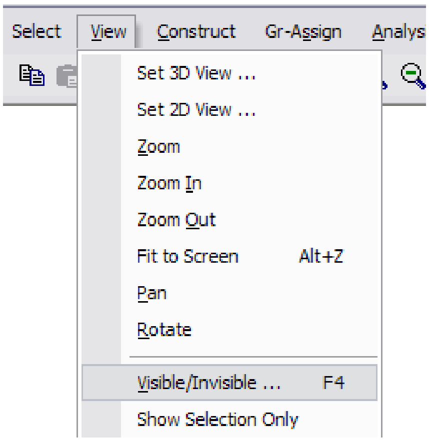
Click the ‘Fast Restraints – Fixed’ to restrain all the degrees of freedom for these nodes in the <Assign Node Properties> window;



Click ***OK*** to finish this step.

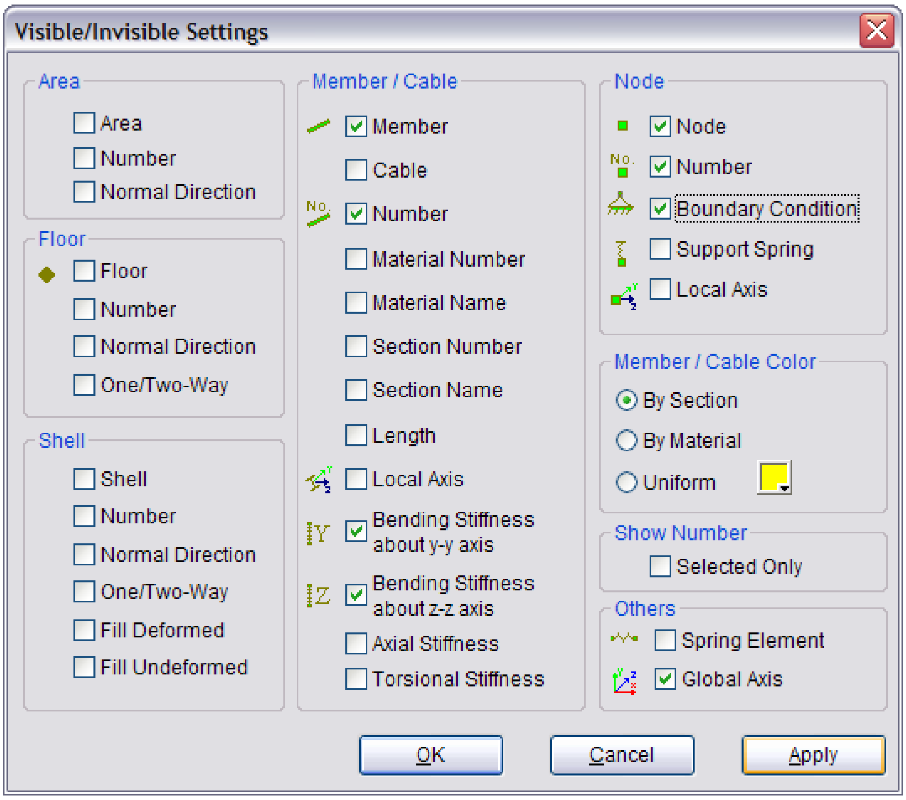
3.10 Step 10 – Set visible/invisible

Click the ***View*** > ***Visible/Invisible*** command,

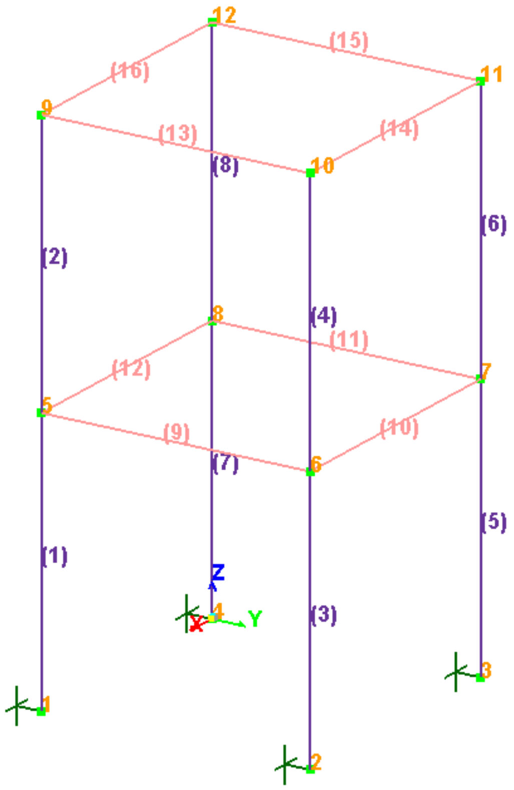


or click the <Visible/Invisible> shortcut in the toolbar;

Tick 'Member', 'Member-Number', 'Node', 'Node-Number', 'Node-Boundary Condition', 'Global Axis' and select 'By Section' in the <Visible/Invisible Settings> window;

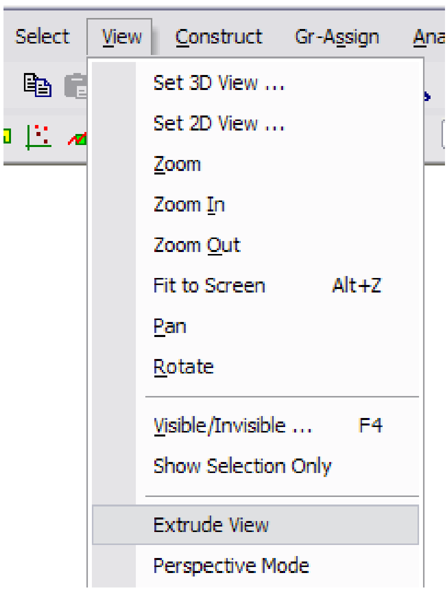


Click ***Apply*** and ***OK*** to view the structure with properties constructed.

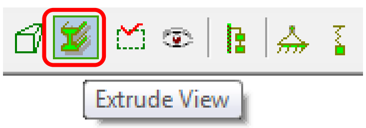


3.11 Step 11 – Rotate member local axis

Click the ***View*** **>** ***Extrude View*** command,



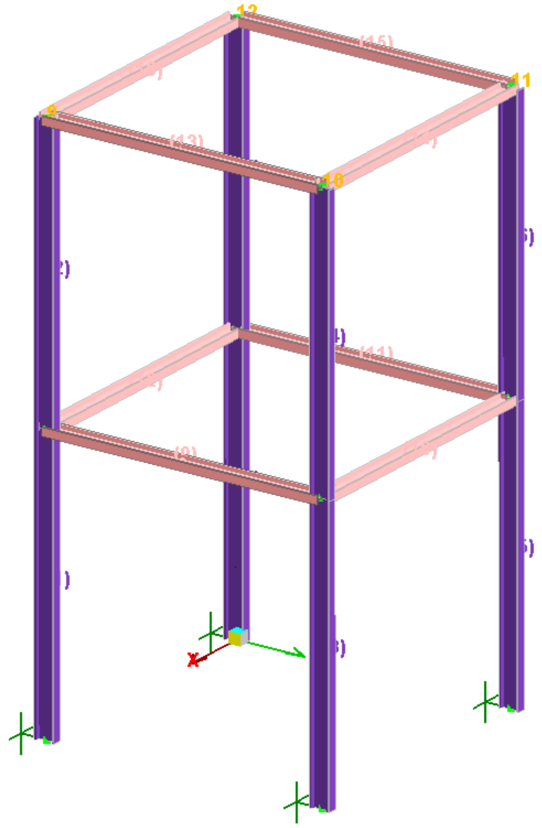
or click the <Extrude View> shortcut in the toolbar;



For the vertical members, local y-axis should lie in global Y-direction;

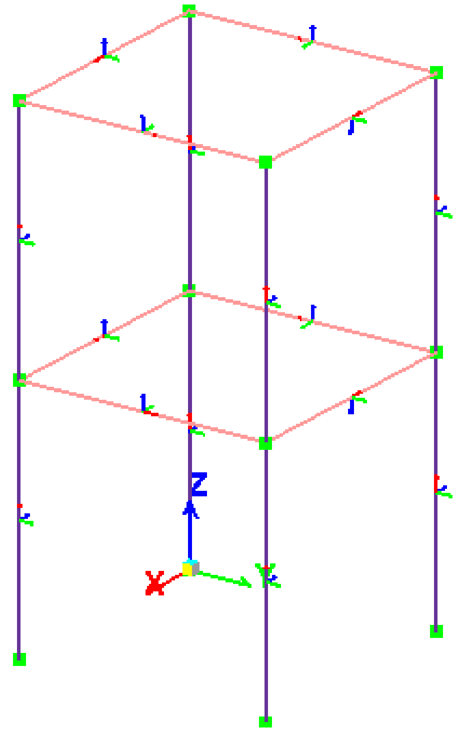
For the horizontal members, local xy plane should parallel to global z axis;

As shown in the following extrude view of the structure, the beams cannot meet the requirement;

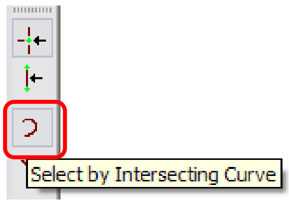


Click the <Show Member Local Axis> shortcut in the toolbar to view the local axis of each member;

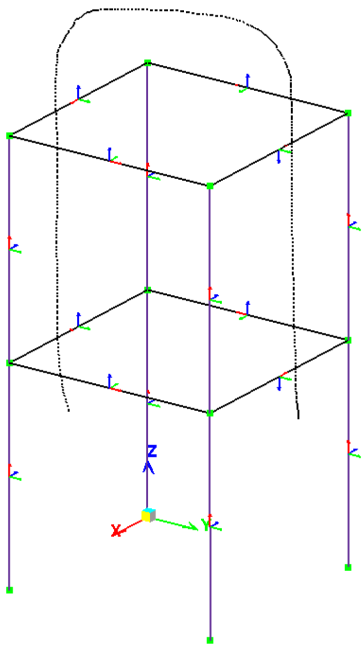
Red arrow indicates local x axis, green arrow indicates local y axis and blue arrow indicates local z axis;



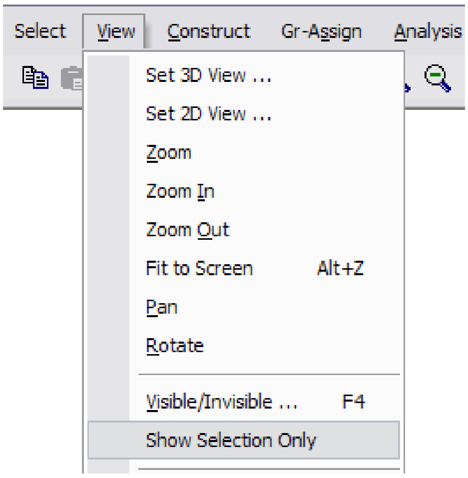
Click the <Select by Intersection Curve> shortcut in the toolbar;



Left click and hold the mouse to draw a line intersecting all the members to be selected;

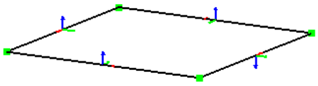


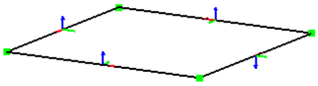
Click the ***View*** > ***Show Selection Only*** command to show the selected members only on the screen;



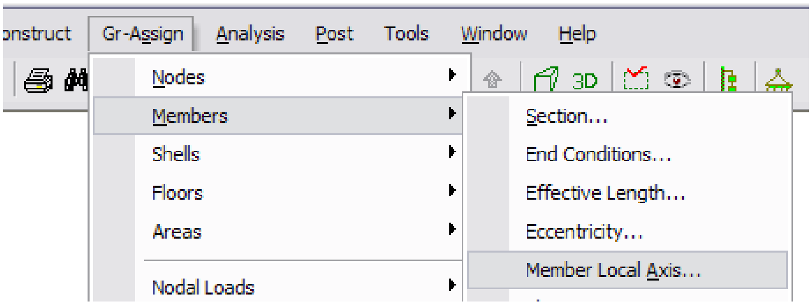
or click the <Show Selection Only> shortcut in the toolbar;

Only the selected beam members are displayed;

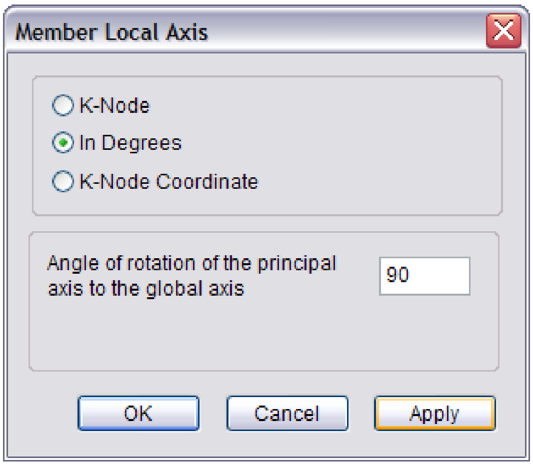




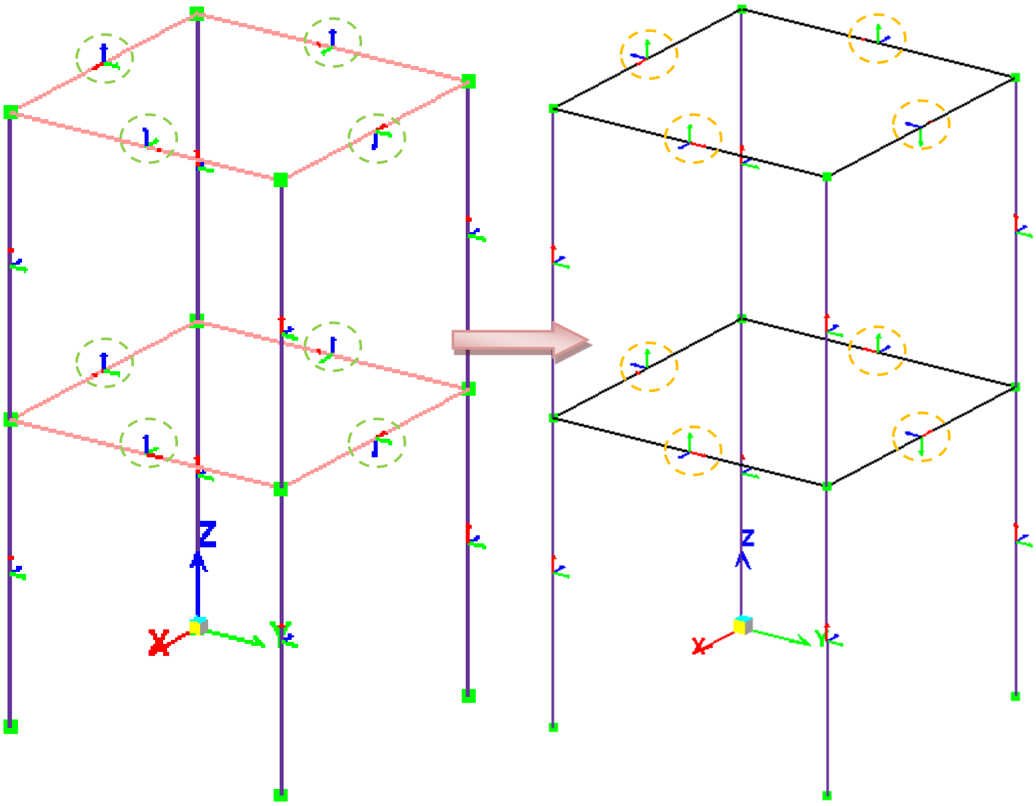
Click the ***Gr-Assign*** > ***Members*** > ***Member Local Axis*** command,



Select the local axis to be set 'In Degrees' in the <Member Local Axis> window, enter the 'Angle of rotation of the principal axis to the global axis' to be 90;

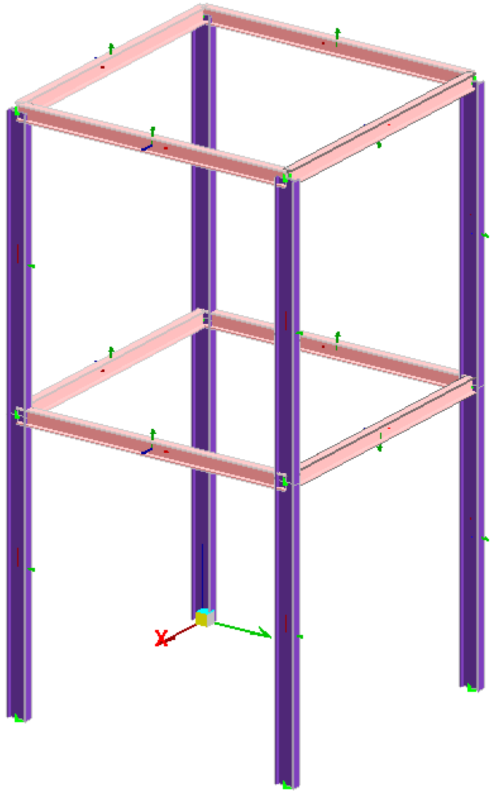


Click ***Apply*** and ***OK*** to close the <Member Local Axis> window and view the rotation of local axis of the selected member;



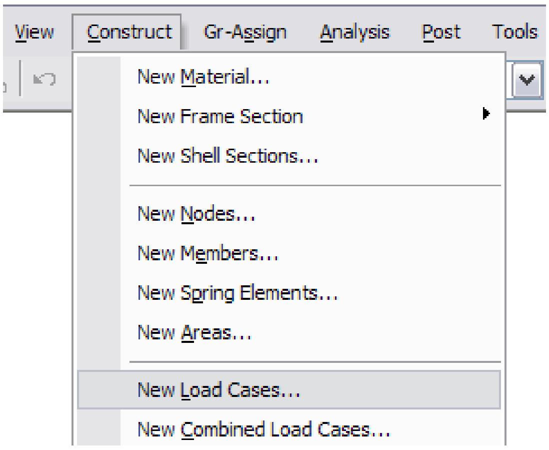
The local xy plane of each beam member is turned to parallel to global z axis;

Click the <Extrude View> shortcut in the toolbar to view the 3D orientation of each member.

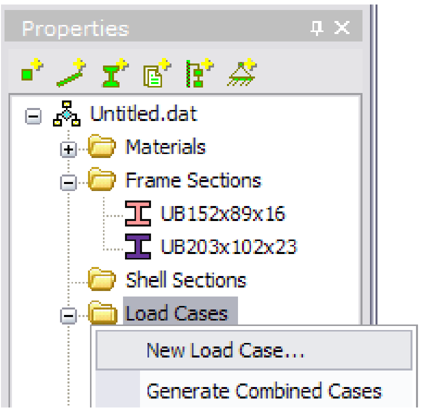


3.12 Step 12 – Add load cases

Click the ***Construct > New Load Cases*** command,

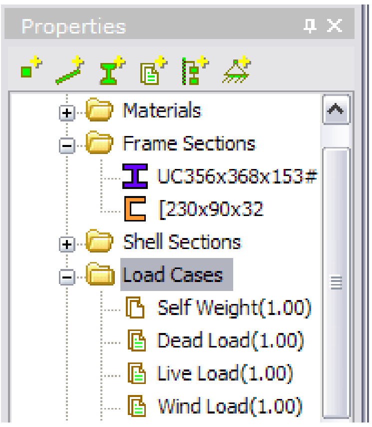


or right click <Load Cases> in the ***Properties*** window and then click <New Load Case>;

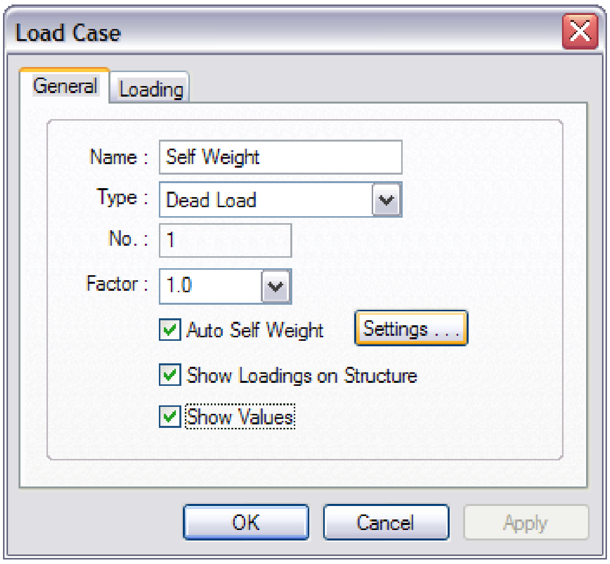


or click the <Add Load Case> shortcut in the ***Properties*** window;

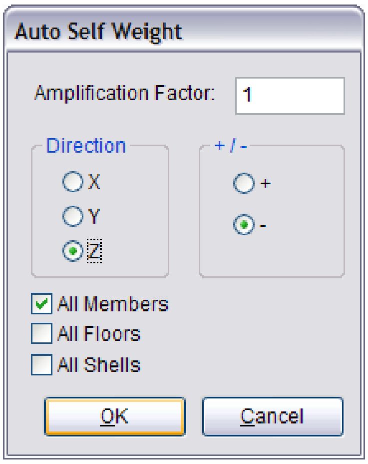
There are four load cases set as default in the software, just the four load cases required for this tutorial;



Double click 'Load Case – Self Weight' to view and change the settings;



Tick 'Auto Self Weight' in the <Load Case – General> window and click 'Settings';



Check the settings of auto self weight in the popped window and select ‘All Members’ only, set the direction of self weight to be in negative Z direction;

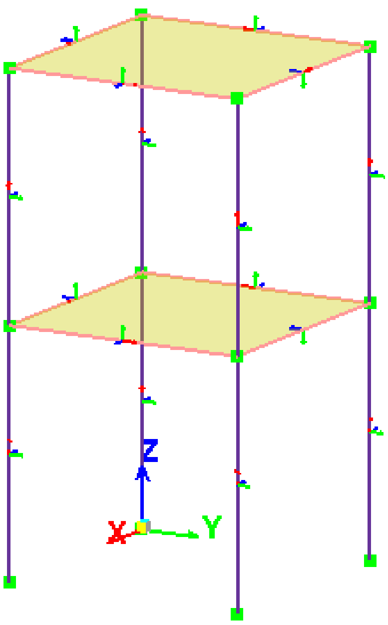
Click ***OK*** to close the <Auto Self Weight> window and click ***OK*** to close the <Load Case> window;

For all the other three load cases, keep the default settings.

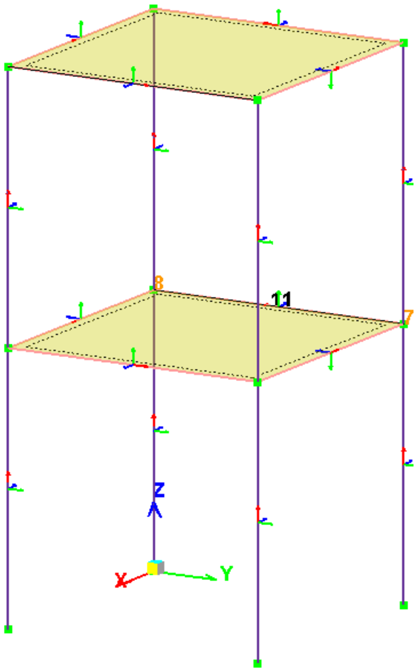
3.13 Step 13 – Add loadings

Click the <Add Floor> shortcut in the toolbar;

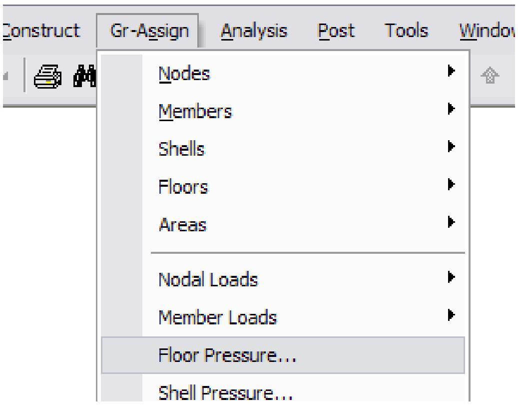
Move the mouse and click on four nodes that can form a floor to apply pressure on;



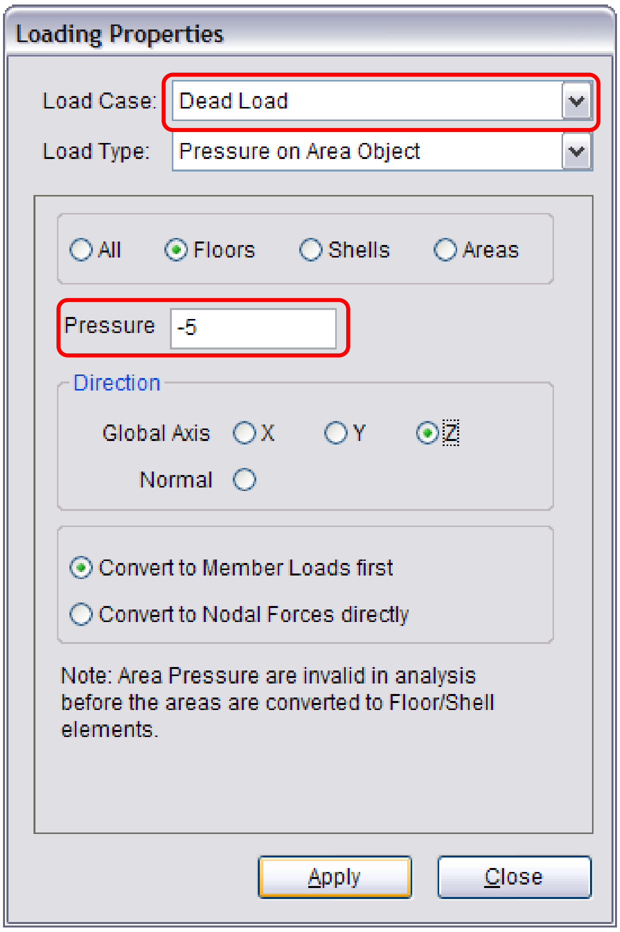
Select the two floors to be loaded;



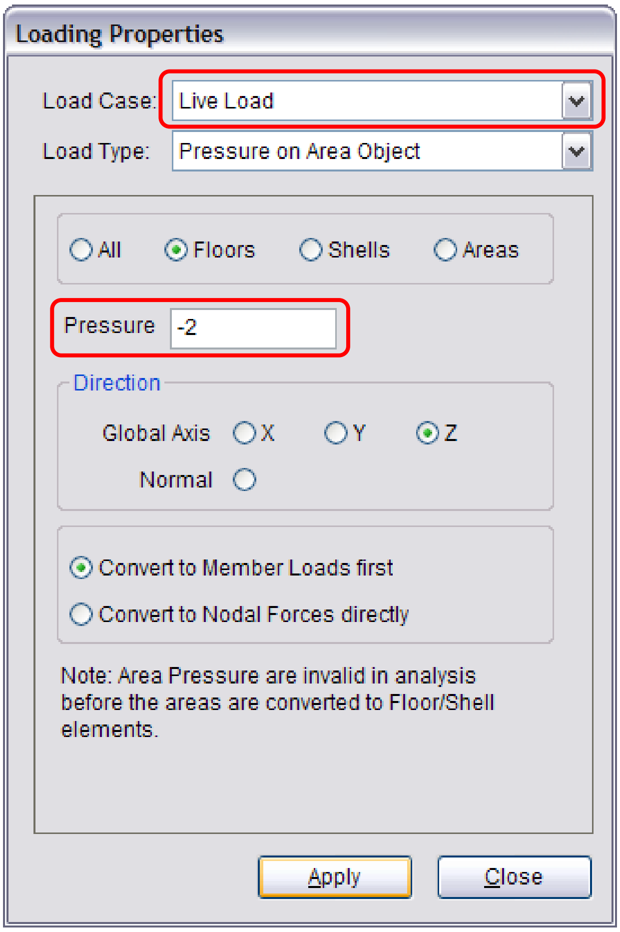
Click the ***Gr-Assign*** > ***Floor Pressure*** command;



Apply 5kN/m^2 dead load pressure to both floors in negative Z direction and select 'Convert to Member Loads first';

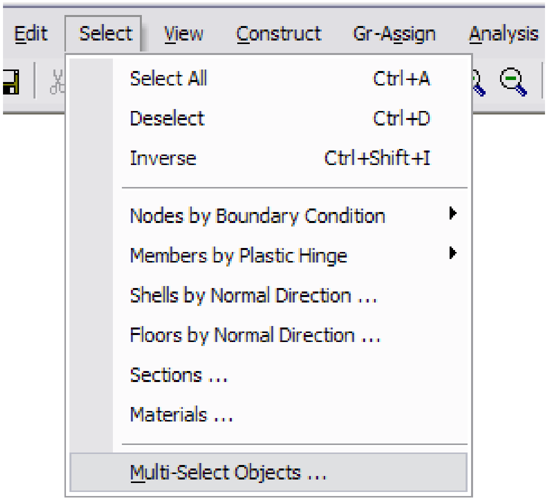


Click ***Apply*** to continue adding 2kN/m^2 live load pressure to both floors in negative Z direction;

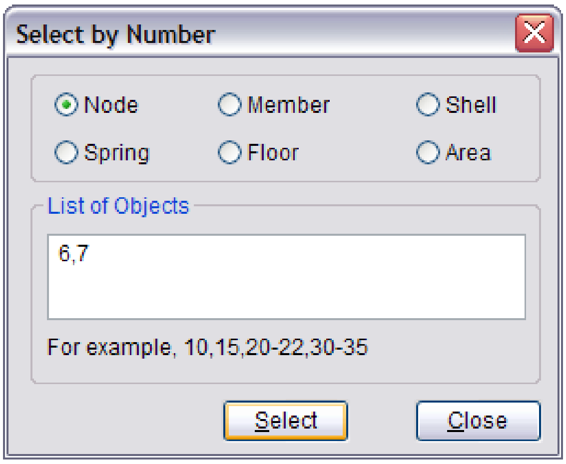


Click ***Apply*** and ***Close*** to continue adding other loads;

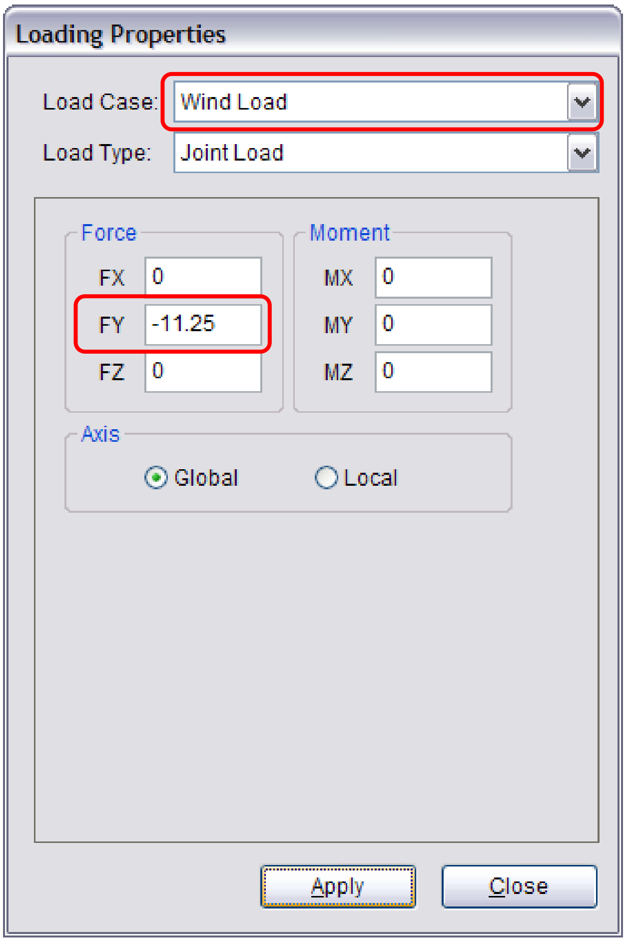
Click the ***Select > Multi-Select Objects*** command;



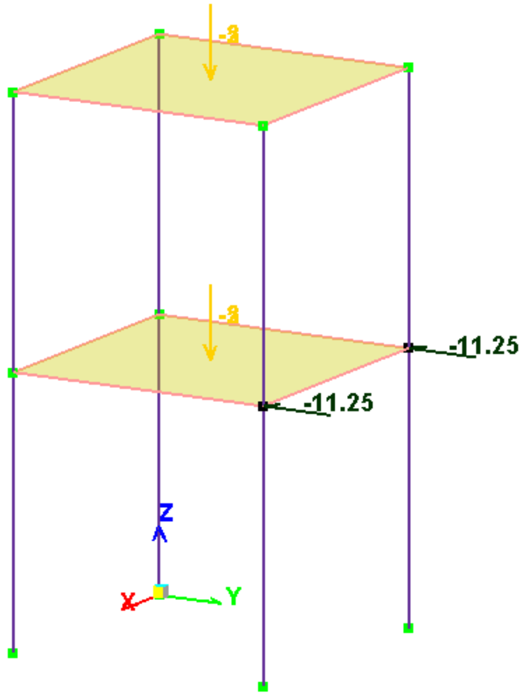
Enter Node '6, 7' of the lower layer of the frame in the <Select by Number> window and click ***Select***;



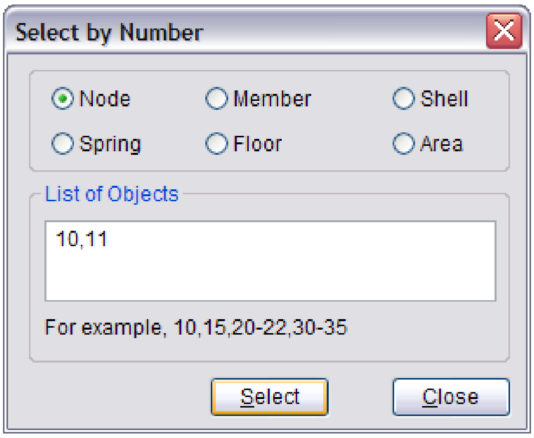
Choose the ‘Load Case – Wind Load’ and ‘Joint Load’, enter the value of force in global negative Y direction;



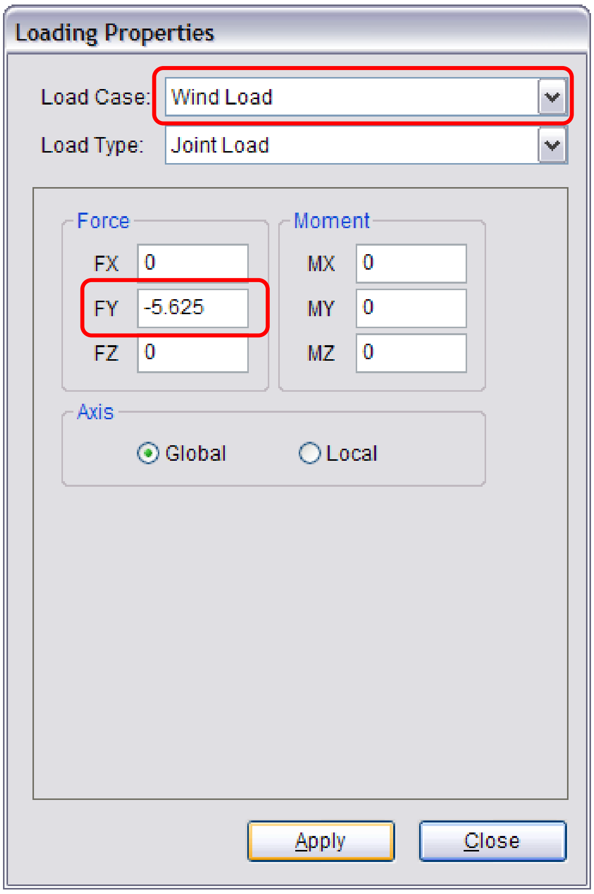
Click ***Apply*** and the wind load will be displayed as below;



Close the <Loading Properties> window and select the other two nodes on the upper layer of frame;



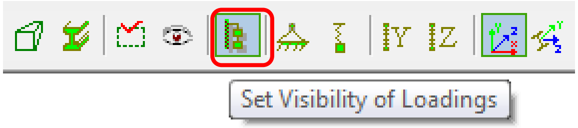
Change the value of wind load to be 5.625 in global negative Y direction;



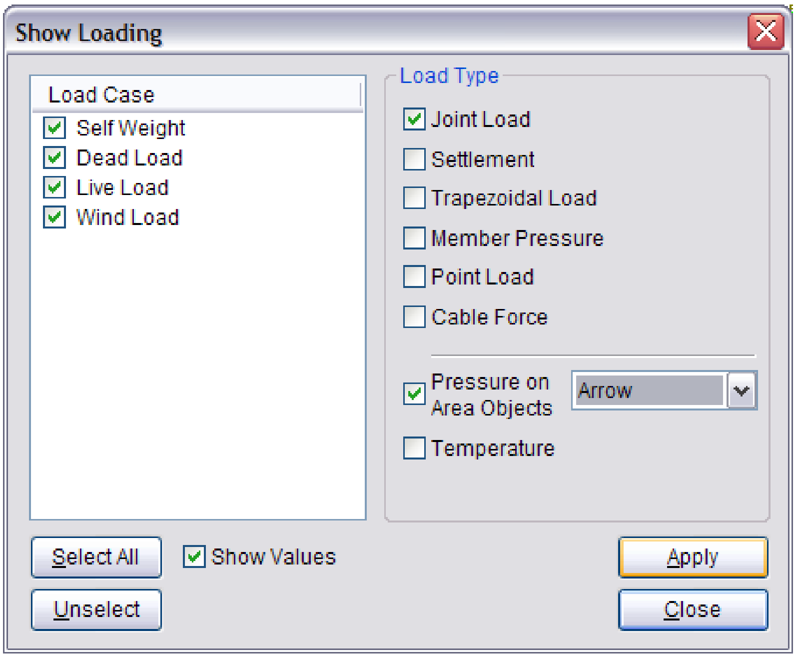
Click Apply and Close the <Loading Properties> window to finish this step;

3.14 Step 14 – Set visibility of loadings

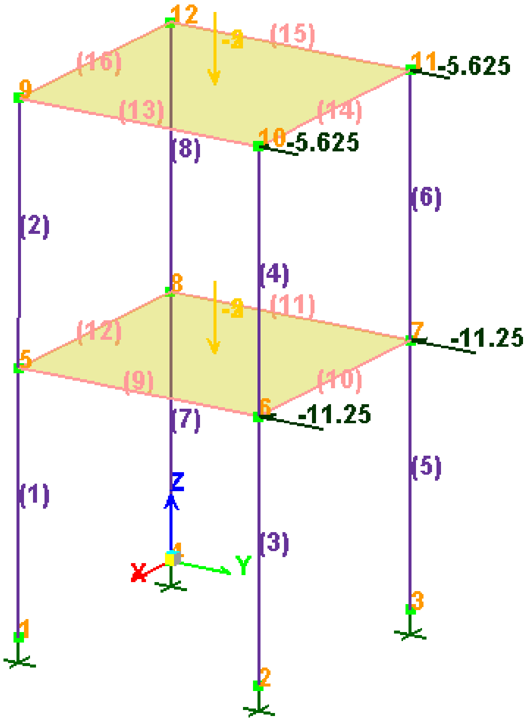
Click the <Set Visibility of Loadings> shortcut in the toolbar;



Select the ‘Load Case’, ‘Load Type’ and tick ‘Show Values’ in the <Show Loading> window;

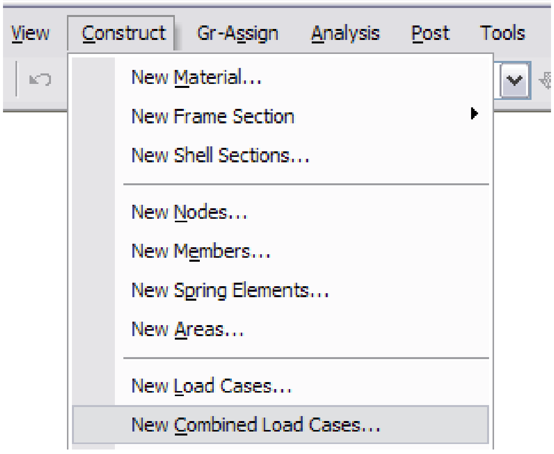


Click ***Apply*** and ***Close*** to finish setting, the whole structure constructed with applied loading is visible as below;

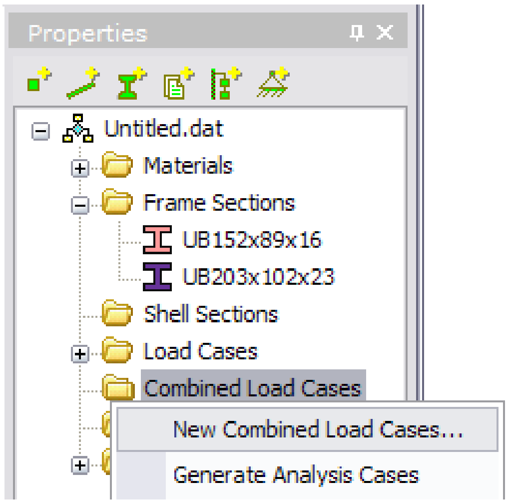


3.15 Step 15 – Add load combinations

Click the ***Construct*** > ***New Combined Load Cases*** command;

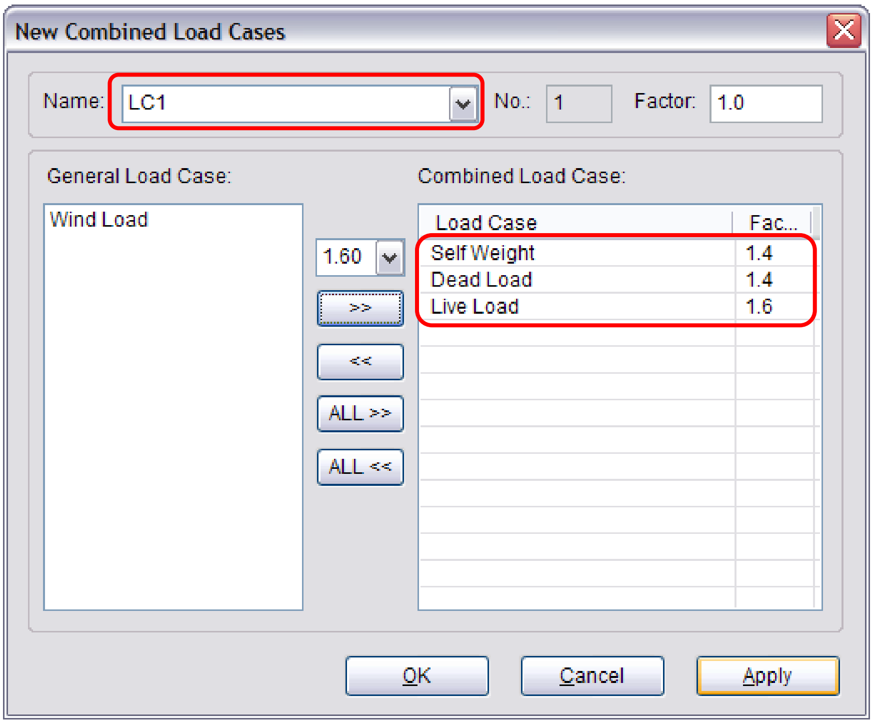


or right click <Combined Load Cases> in the ***Properties*** window and then click <New Combined Load Cases>;

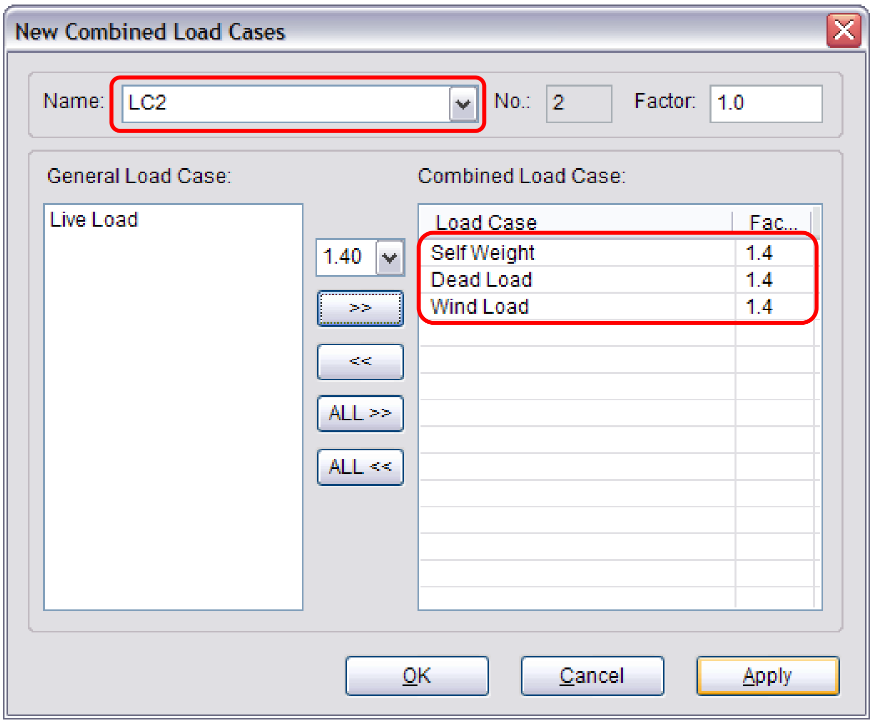


Enter the ‘Name’ of the load case and select load cases from the ‘General Load Case’ menu to the ‘Combined Load Case’ menu with corresponding load factors in the <New Combined Load Cases> window;

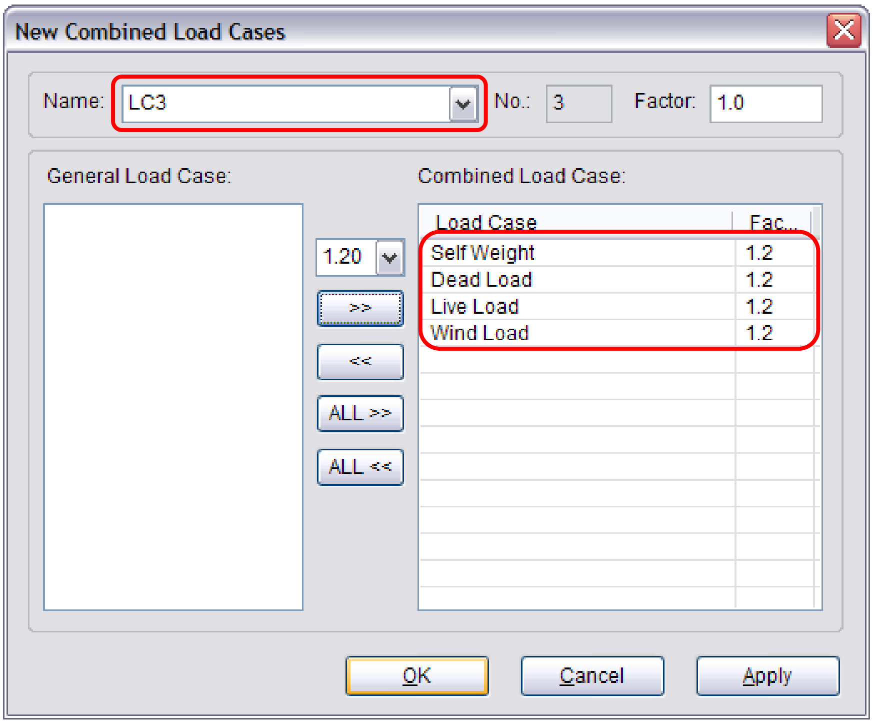
Load type of ‘Self Weight’ is the same as ‘Dead Load’, so apply the same load factor to ‘Self Weight’ as for ‘Dead Load’;



Click ***Apply*** to continue adding new combined load case following the same procedure;



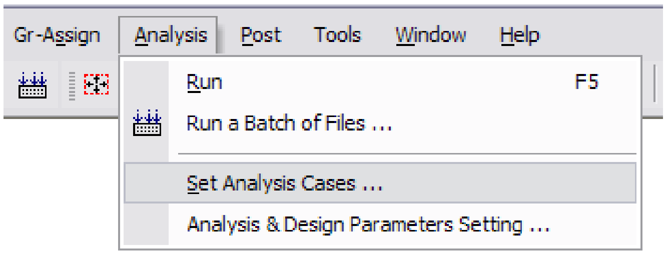
Click ***Apply*** to continue to add the last load case for this tutorial;



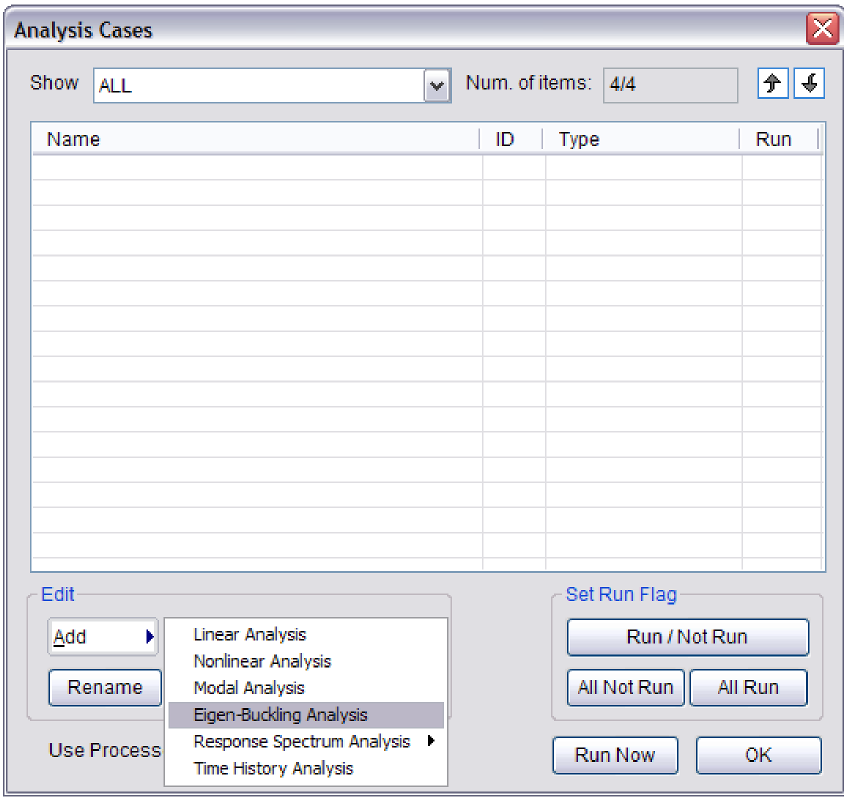
Click ***OK*** to finish this step.

3.16 Step 16 – Set and run analysis cases

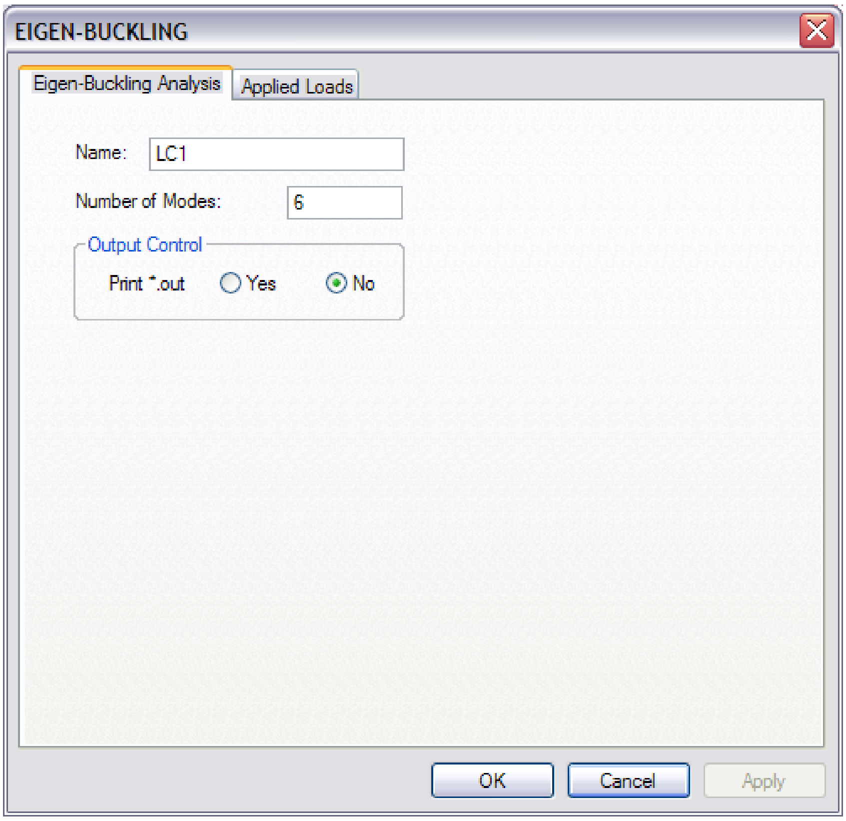
Click the ***Analysis*** > ***Set Analysis Cases*** command;



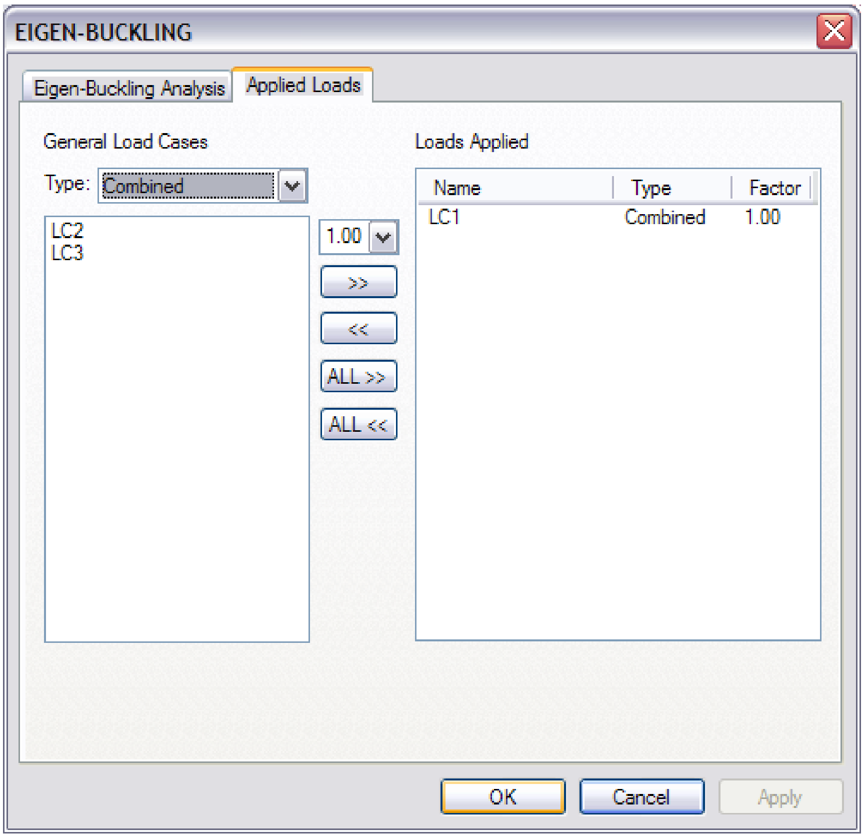
Add 'Eigen-Buckling Analysis' to determine elastic critical load factor λ_{cr} for the frame;



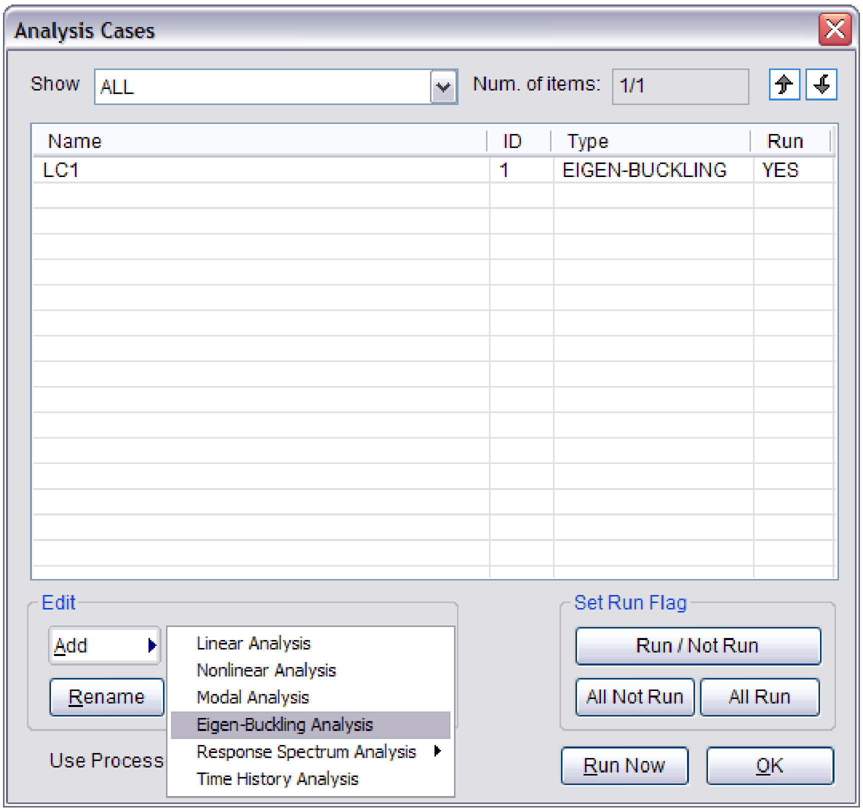
Give the name ‘LC1’ to the 1st load combination case;



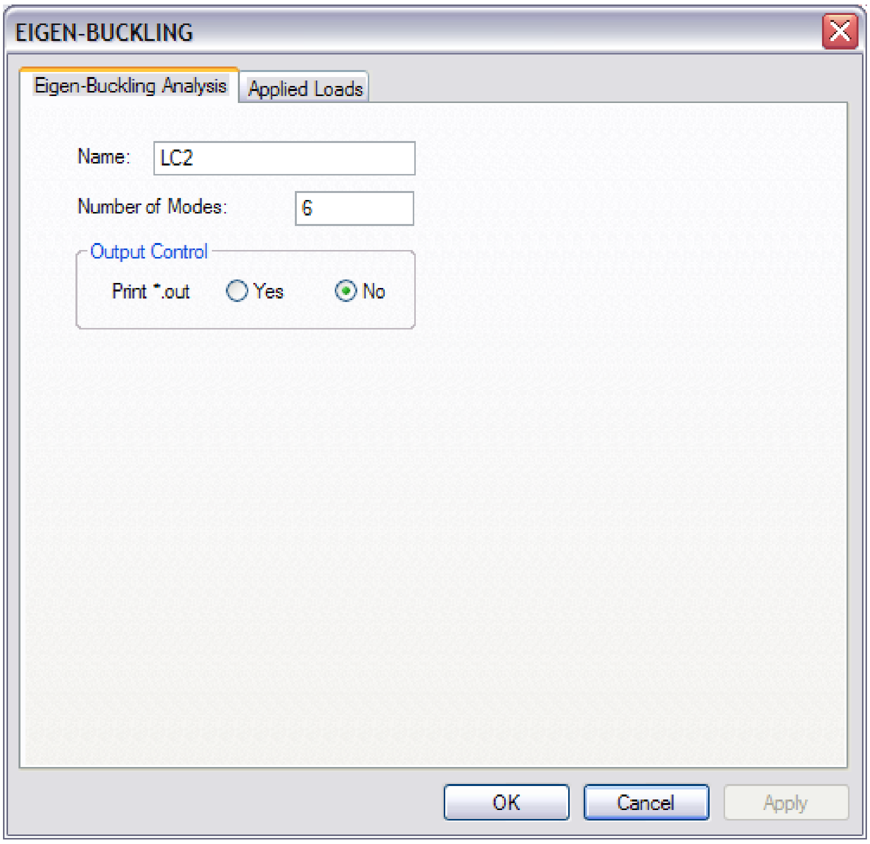
Apply the load combination LC1;



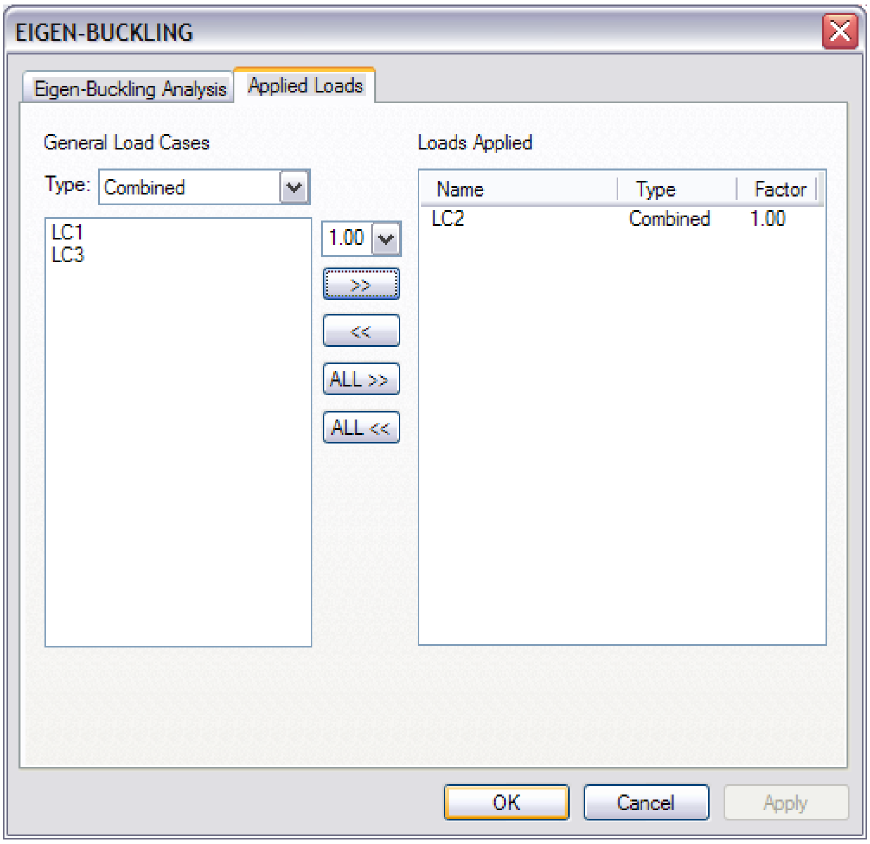
Click OK to close the <EIGEN-BUCKLING> window and add another ‘Eigen-Buckling Analysis’ to determine elastic critical load factor λ_{cr} for load combination 2;



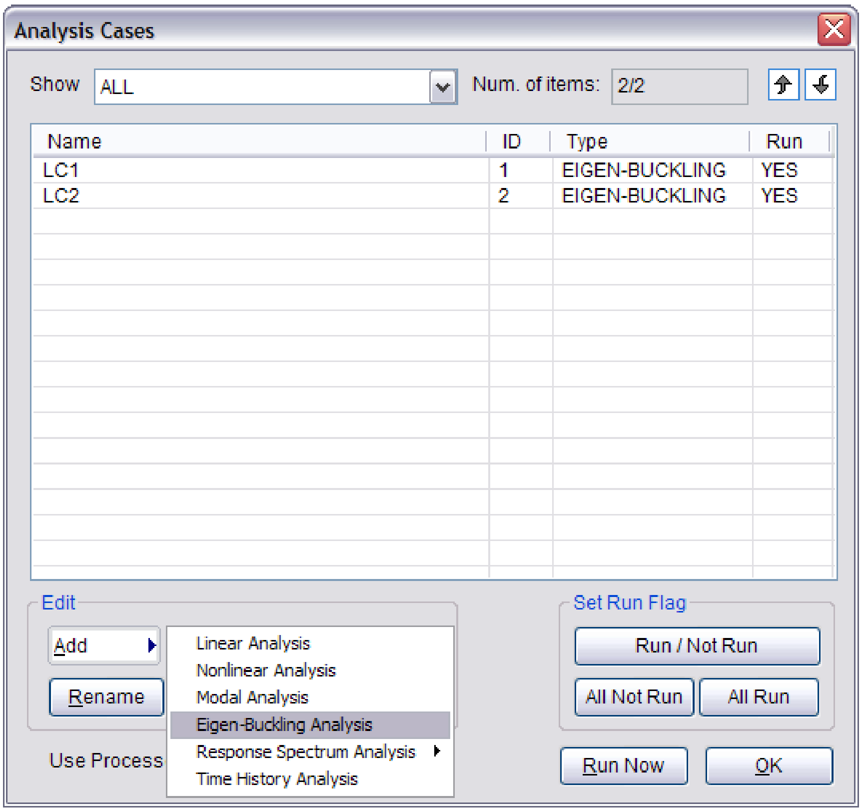
Give the name ‘LC2’ to the 2nd load combination case;



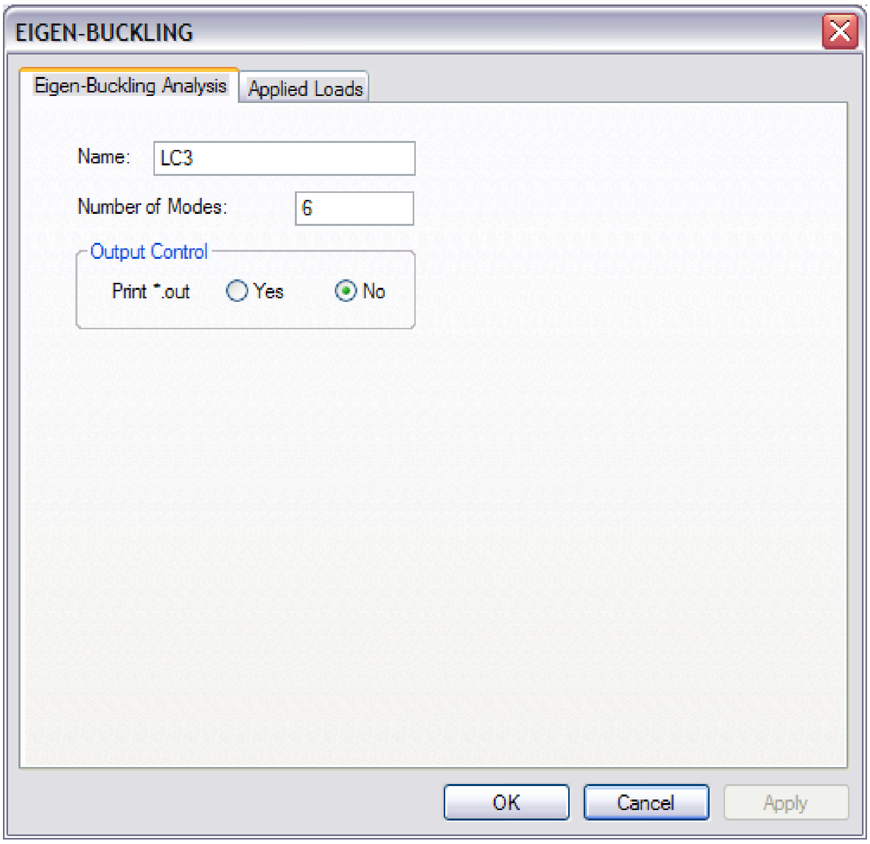
Apply the load combination LC2;



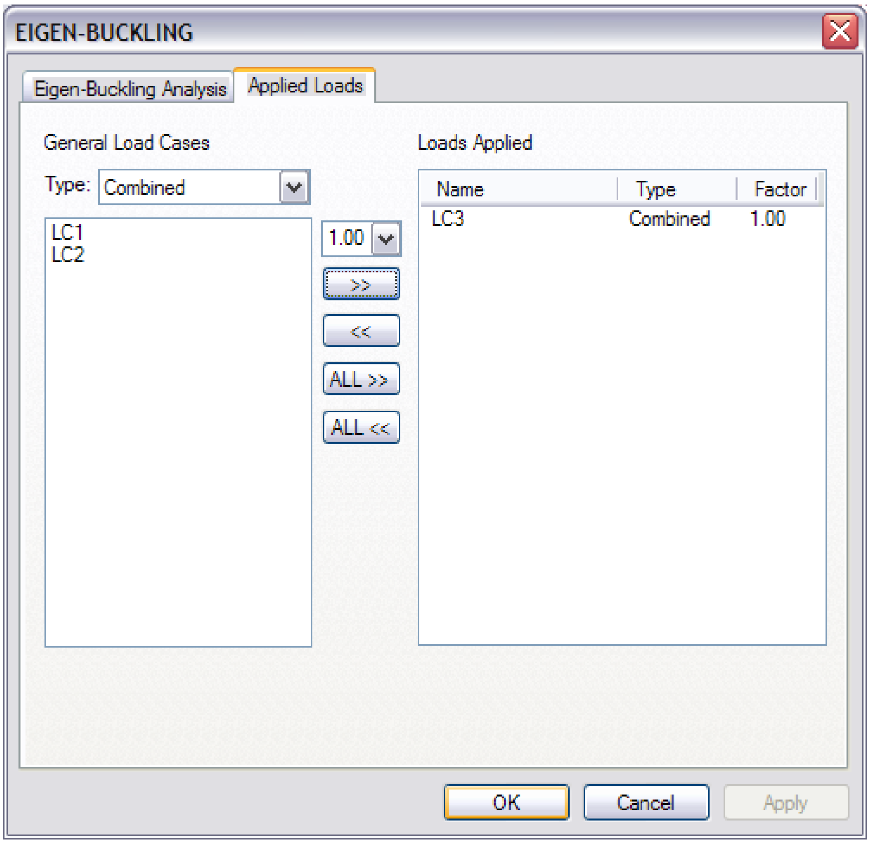
Click OK to close the <EIGEN-BUCKLING> window and add another 'Eigen-Buckling Analysis' to determine elastic critical load factor λ_{cr} for load combination 3;



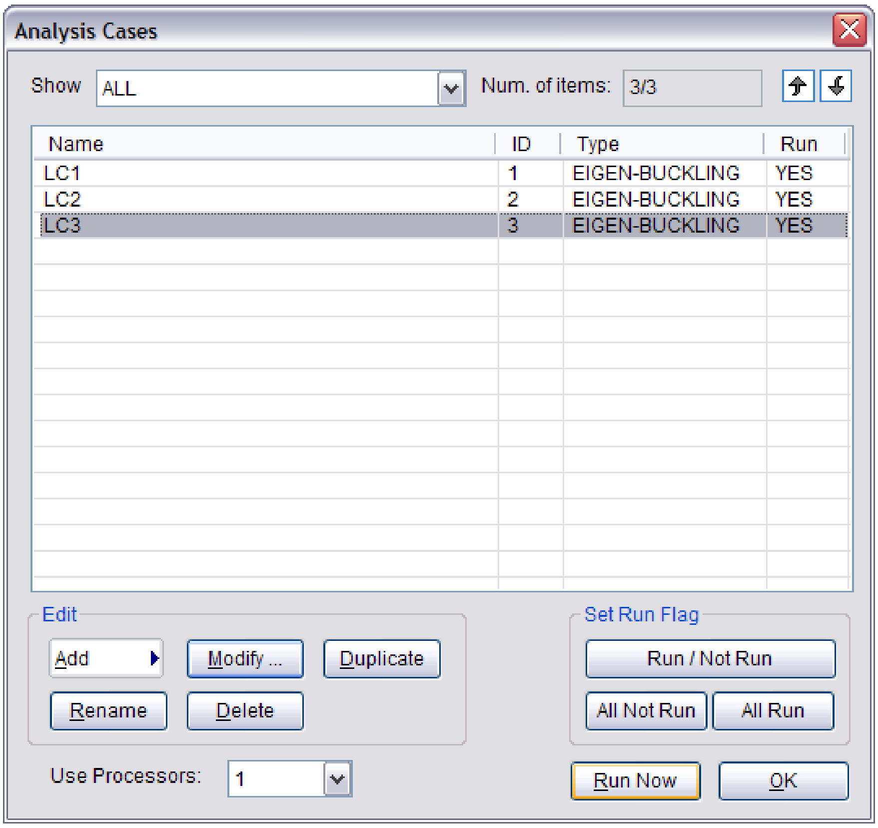
Give the name 'LC3' to the 3rd load combination case;



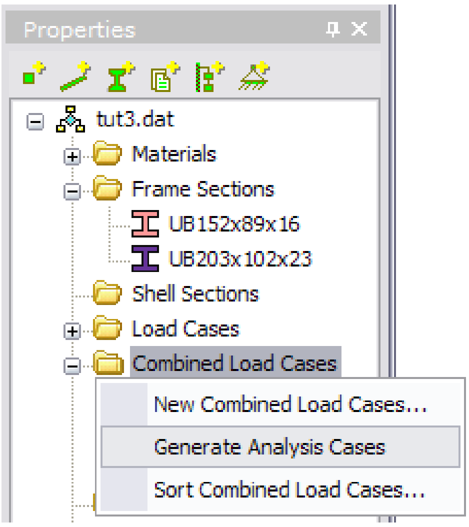
Apply the load combination LC3;



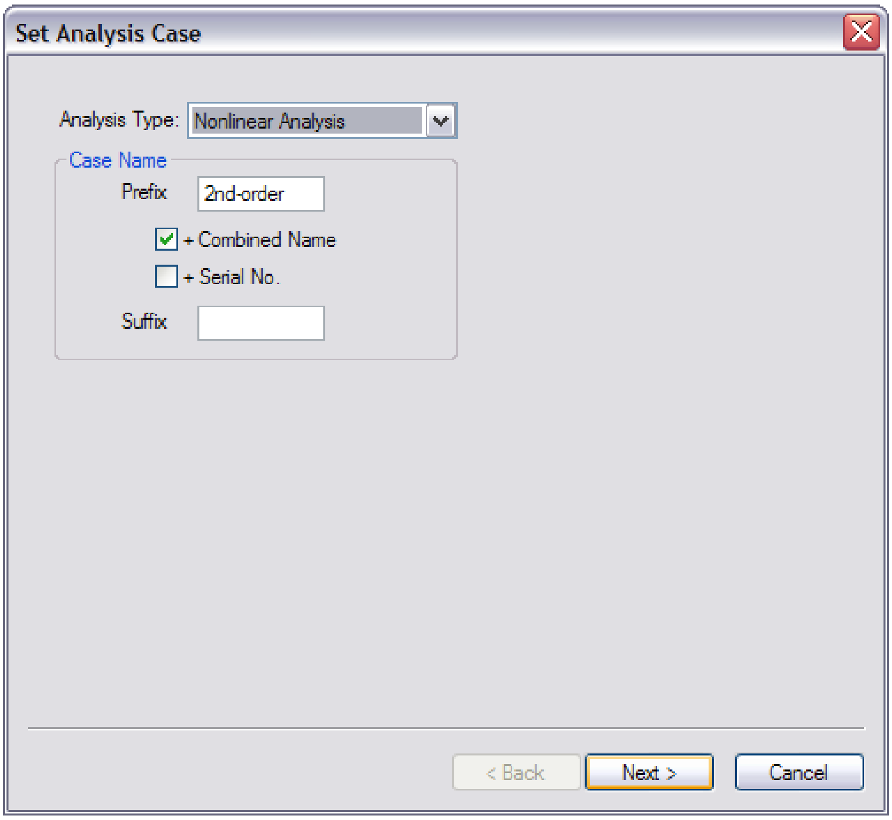
Click OK to close the <EIGEN-BUCKLING> window and go back to the <Analysis Cases> window;



To further add the second order analysis cases with the same series of load combinations, right click <Combined Load Cases> in the ***Properties*** window and click <Generate Analysis Cases>;

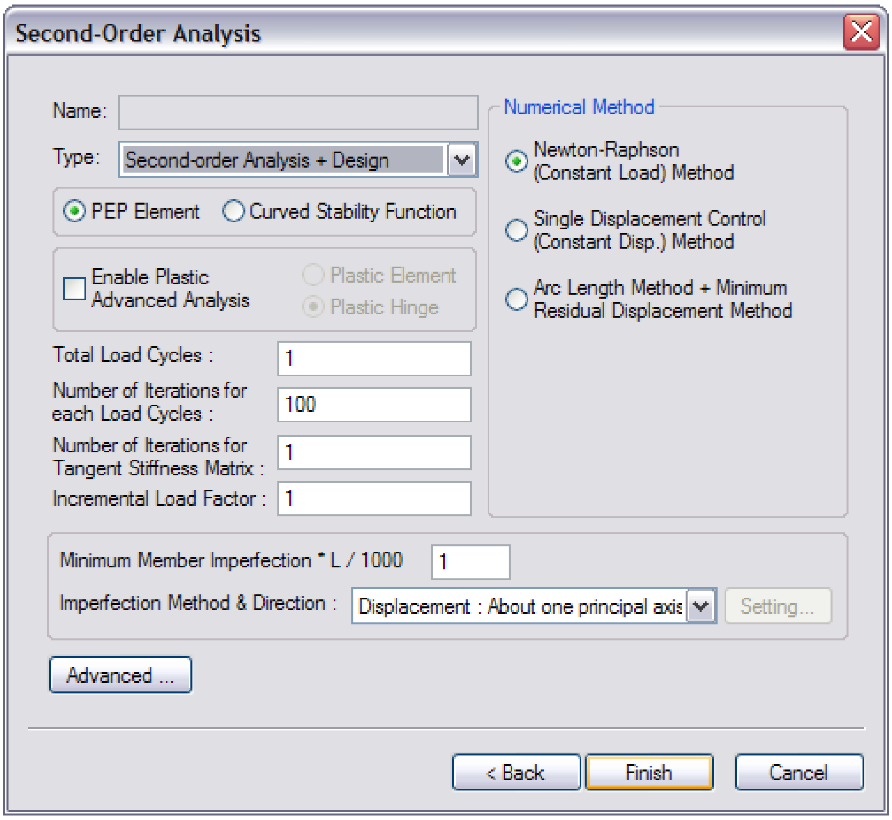


Select 'Nonlinear Analysis' type , enter a 'Prefix – 2nd-order' to 'Combined Name' in the <Set Analysis Case> window;



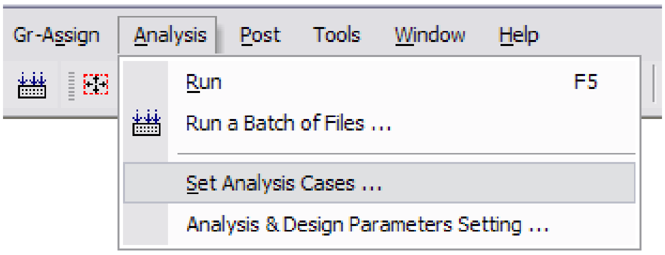
Click ***Next*** to continue;

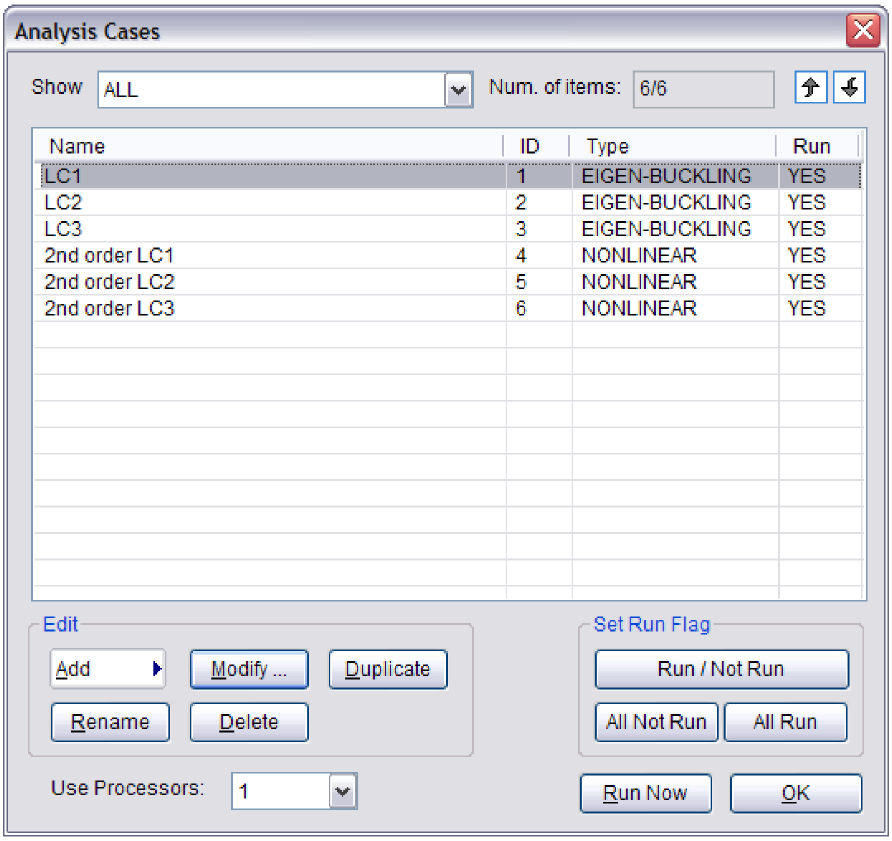
Remain the default settings of 'Second-order Analysis + Design' in the <Second-Order Analysis> window;



Click ***Finish*** to complete adding analysis cases;

Click the ***Analysis*** > ***Set Analysis Cases*** command to view the analysis cases constructed;

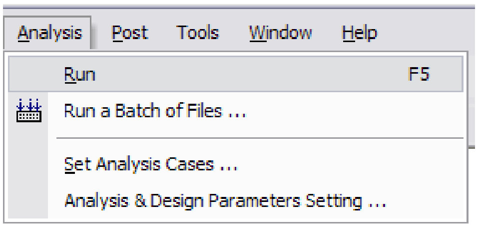




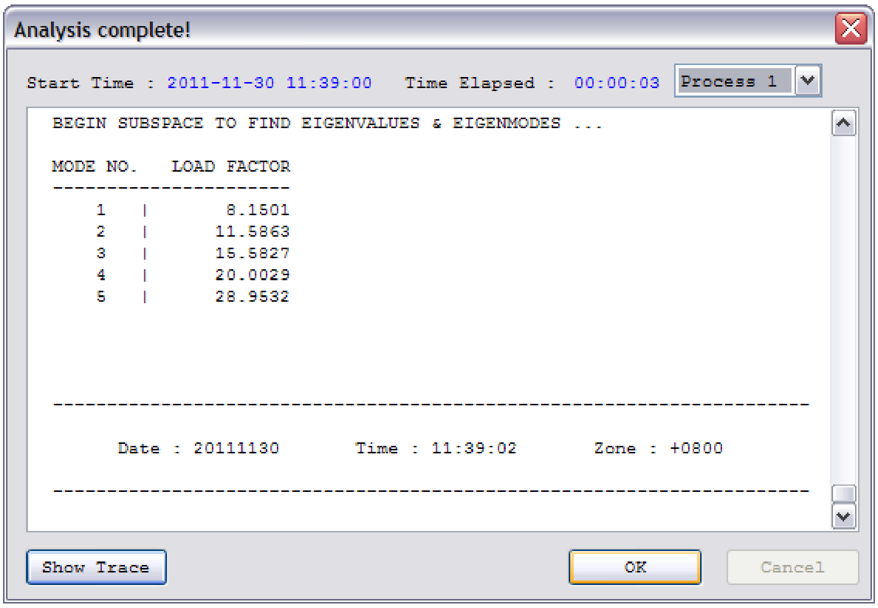
Click ***Run Now*** to run all the analysis cases;

or directly click the ***Analysis*** > ***Run*** command;

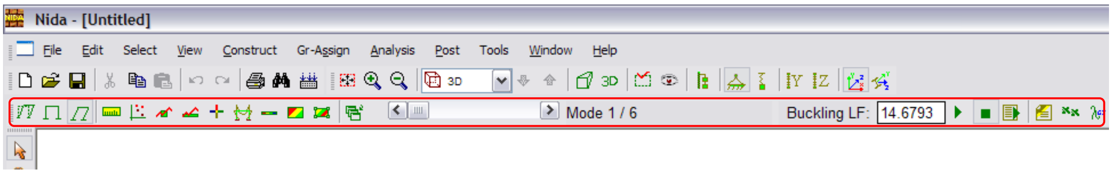
or press **F5** key;



When the analysis is completed, click OK to close the <Analysis complete!> window;



and the post analysis toolbar will appear under the function toolbar;



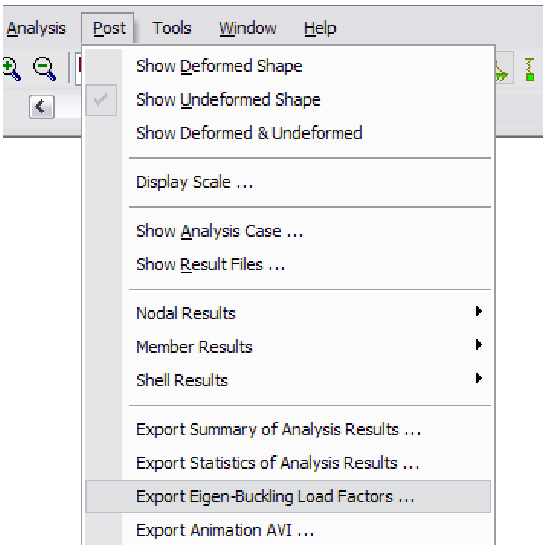
3.17 Step 17 – View result: eigen-buckling mode

For more than one analysis case, the first analysis case will display automatically upon completion of program running. For this tutorial, the Eigen-Buckling Analysis result displays the 6 eigen-modes firstly;

The mode could be selected through the sliding bar and the corresponding ‘Buckling LF’ could be viewed in the tool bar, so the elastic critical load factor for the braced case, which is also the first mode buckling load factor, is

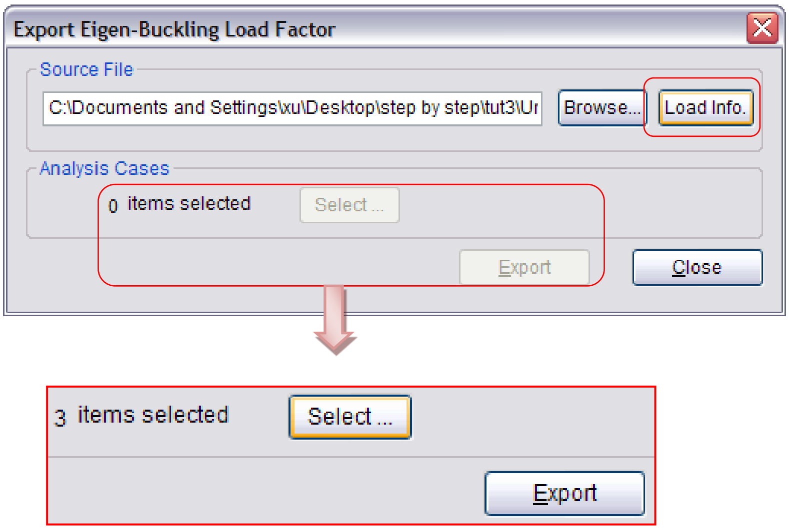
$\lambda_{cr} = 6.6649$;

Click the ***Post*** > ***Export Eigen-Buckling Load Factors*** command,

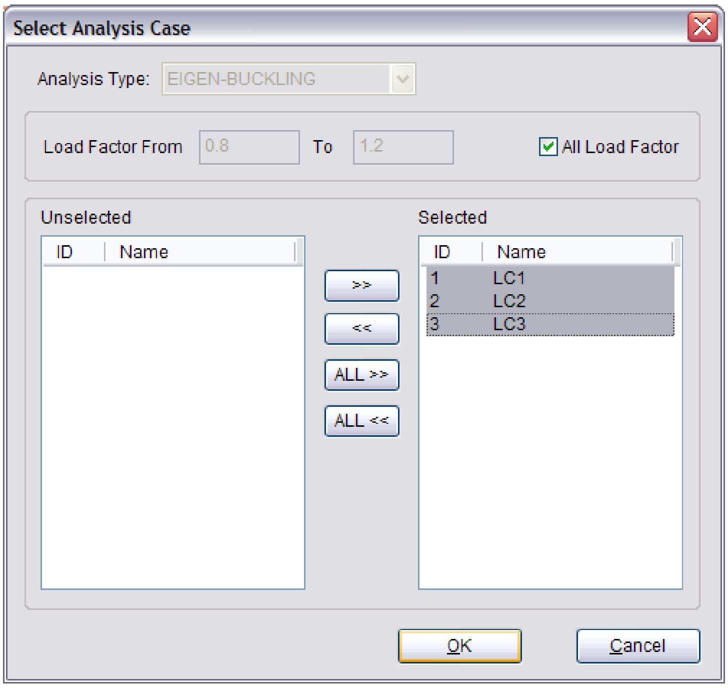


or click <Export Eigen-Buckling Load Factors> short cut in the post analysis toolbar;

Select the 'Source File', click ***Load Info*** in the <Export Eigen-Buckling Load Factor> window to import the analysis case to be exported;

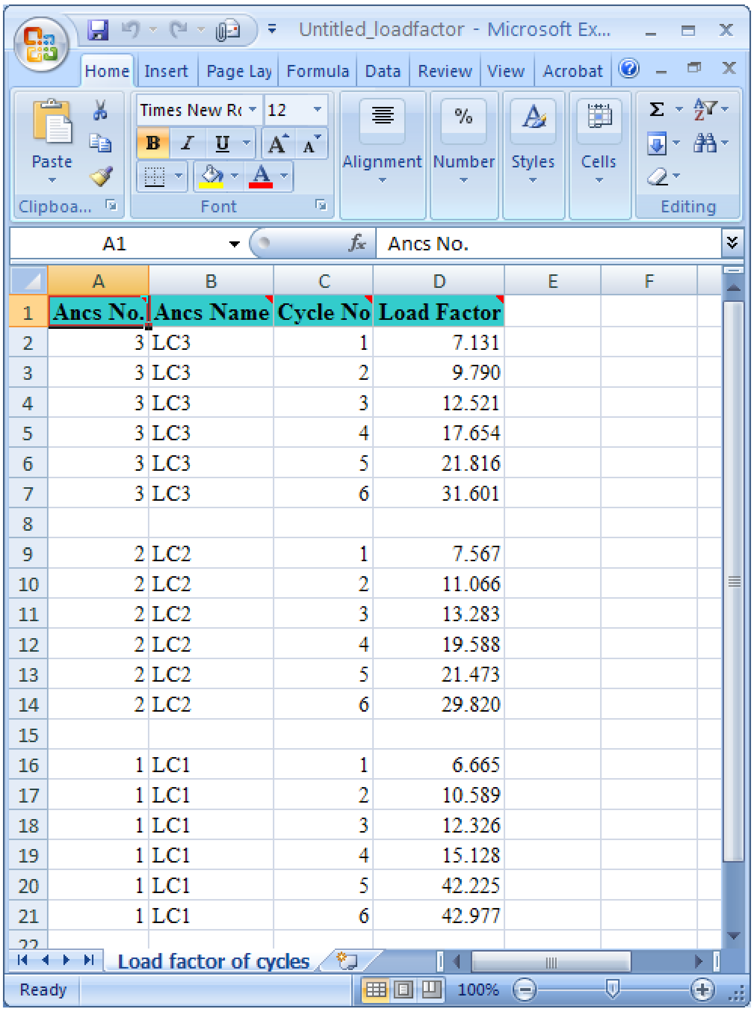


Click ***Select*** and choose the analysis case in the <Select Analysis Case> window;



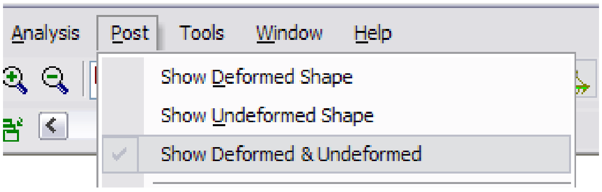
Click ***OK*** to go back to the <Export Eigen-Buckling Load Factor> window;

Click ***Export*** , an excel spreadsheet titled '***_loadfactor' will pop up including load factors for all the eigen-buckling modes;

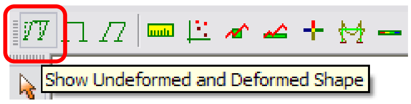


Ancs No.	Ancs Name	Cycle No	Load Factor
3	LC3	1	7.131
3	LC3	2	9.790
3	LC3	3	12.521
3	LC3	4	17.654
3	LC3	5	21.816
3	LC3	6	31.601
2	LC2	1	7.567
2	LC2	2	11.066
2	LC2	3	13.283
2	LC2	4	19.588
2	LC2	5	21.473
2	LC2	6	29.820
1	LC1	1	6.665
1	LC1	2	10.589
1	LC1	3	12.326
1	LC1	4	15.128
1	LC1	5	42.225
1	LC1	6	42.977

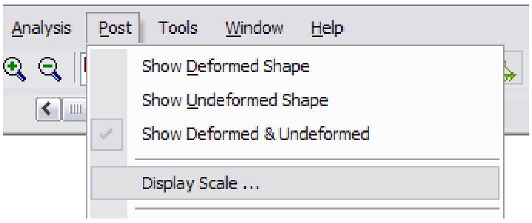
Click the ***Post*** > ***Show Deformed & Undeformed*** command;



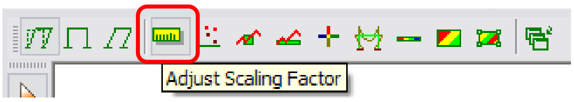
or click the <Show Undeformed and Deformed Shape> shortcut in the toolbar;



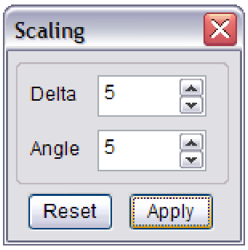
To view the first eigen-buckling mode deformation in a larger scale, click the ***Post*** > ***Display*** command;



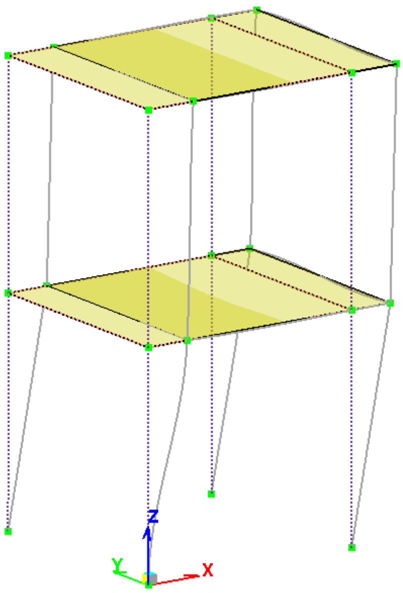
or click the <Adjust Scaling Factor> shortcut in the toolbar;



Enter scaling factors of ‘Delta’ and ‘Angle’ in the <Scaling> window;



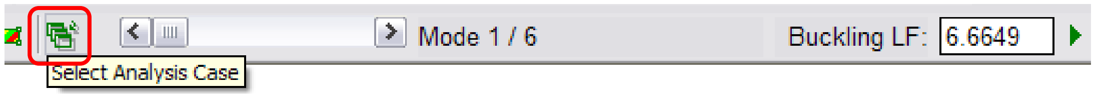
The first eigen-buckling mode deformation of the structure is shown as below;



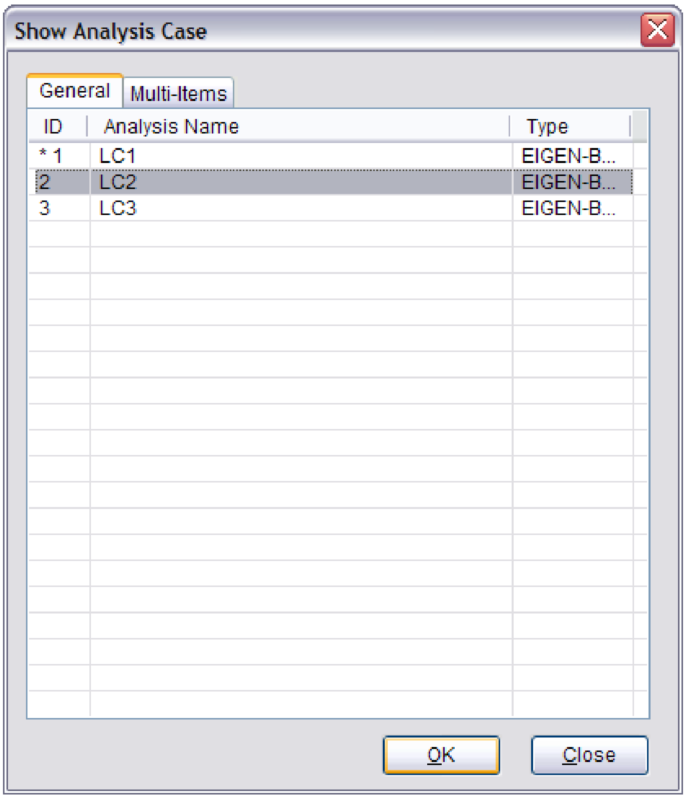
Change the load factor in the sliding bar to view the deformed shapes of higher buckling modes.

3.18 Step 18 – View result: View another analysis case

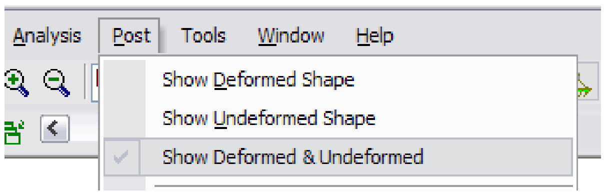
To view another analysis result, click <Select Analysis Case> shortcut near the sliding bar in the post analysis toolbar;



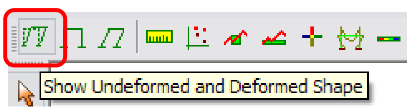
Select the 'LC2' case from the <Show Analysis Case> window;



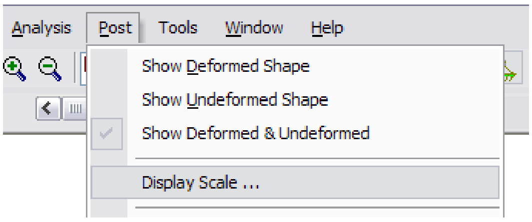
Click the ***Post*** > ***Show Deformed & Undeformed*** command;



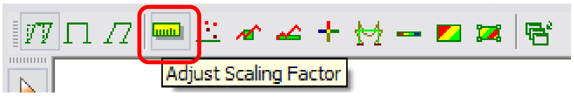
or click the <Show Undeformed and Deformed Shape> shortcut in the toolbar;



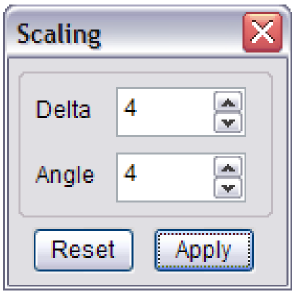
To view the first eigen-buckling mode deformation in a larger scale, click the ***Post*** > ***Display*** command;



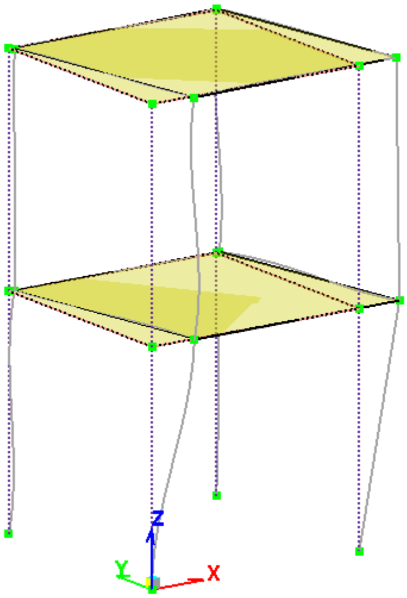
or click the <Adjust Scaling Factor> shortcut in the toolbar;



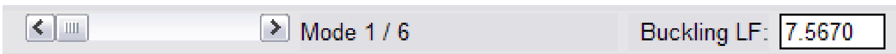
Enter scaling factors of ‘Delta’ and ‘Angle’ in the <Scaling> window;



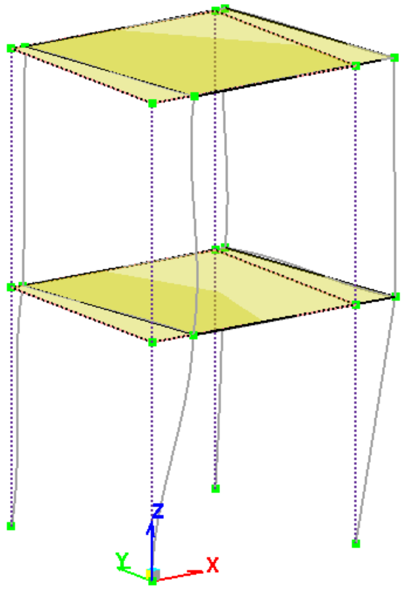
The deformation of the frame under LC2 is show as below;



With its elastic critical load factor λ_{cr} = 7.5670;



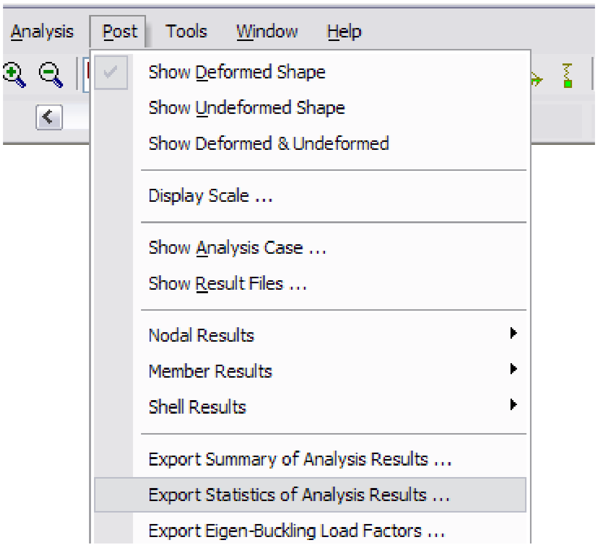
The same procedure applies to the 3rd load combination LC3, the deformation therefore obtained as below;



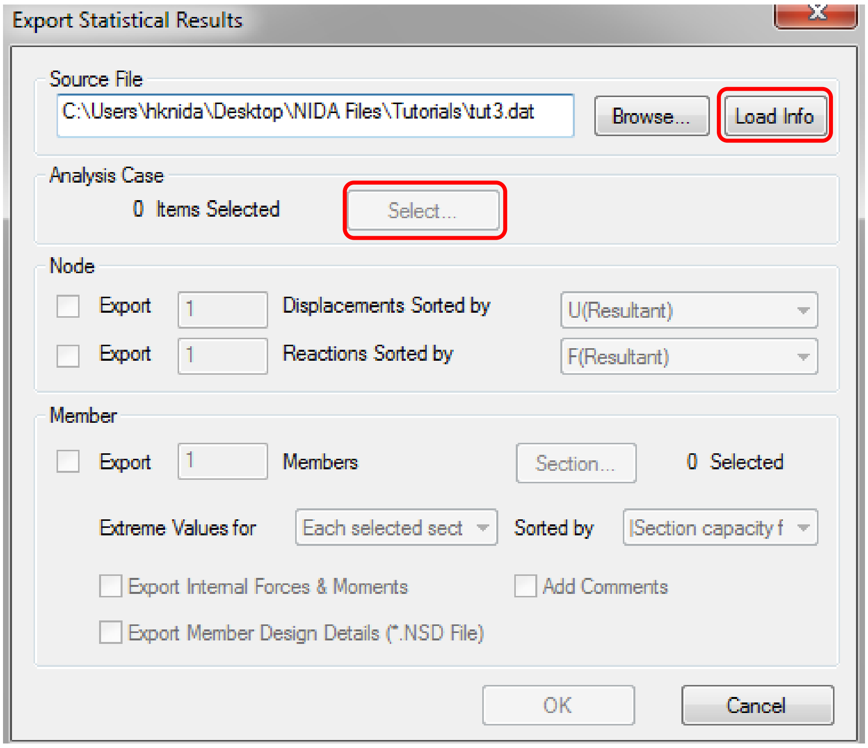
With its elastic critical load factor λ_{cr} = 7.1314.

3.19 Step 19 – View result: check the structural adequacy by second-order analysis (i) Export Statistics of Analysis Results

Click the ***Post*** **>** ***Export Statistics of Analysis Results*** command;

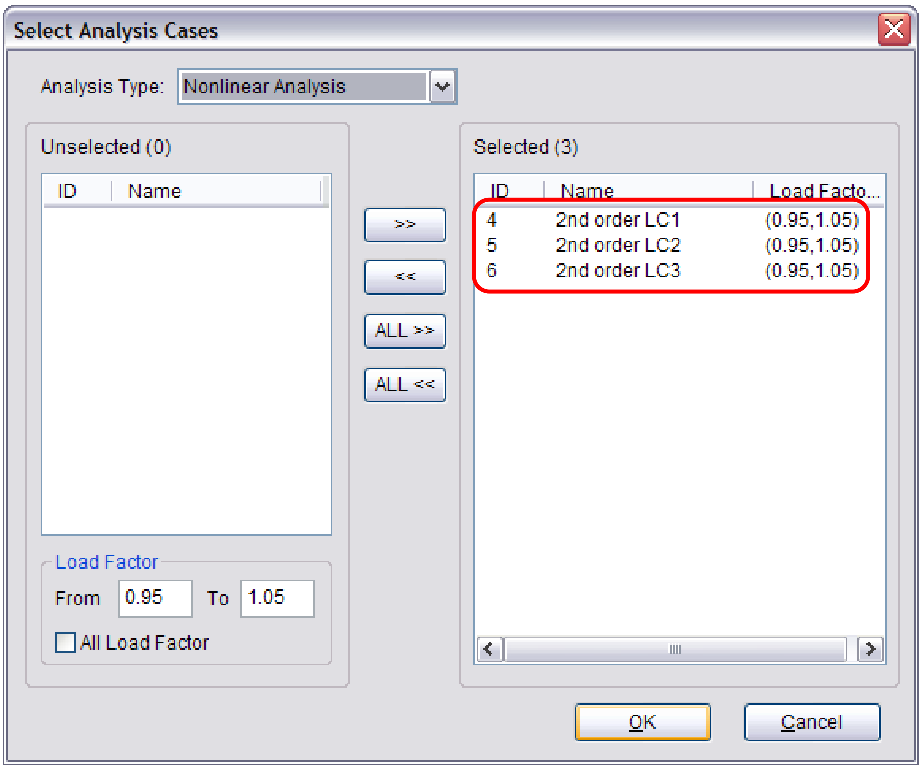


or click the <Export Statistics of Analysis Results> shortcut in the toolbar;



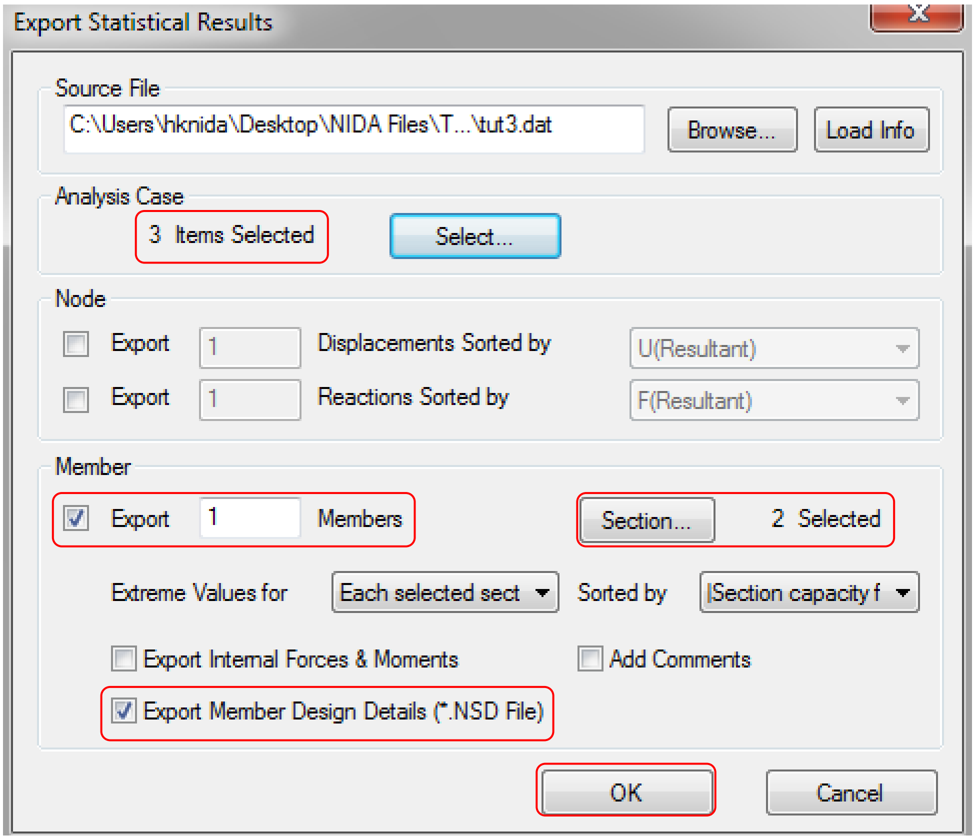
'Browse' the 'Source File' direction in the <Export Statistical Results> window and click the 'Load Info' button, then the 'Select' of 'Analysis Case' option becomes enable;

Click 'Select' for the 'Analysis Case' in the <Export Statistical Results> window, the <Select Analysis Cases> window will pop up;



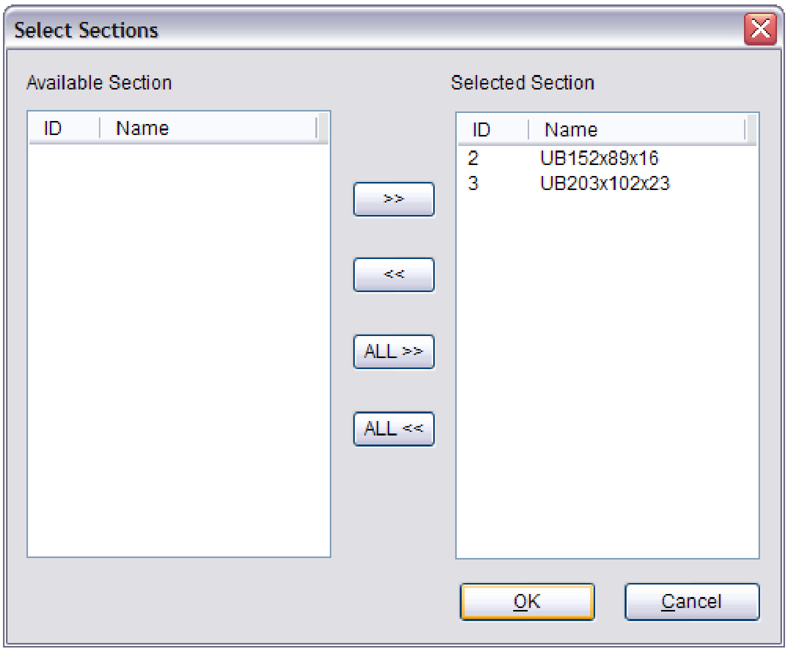
Click ***OK*** to export the analysis cases considered and go back to the <Export Statistical Results> window;

The number of items selected will be shown under ‘Analysis Case’;

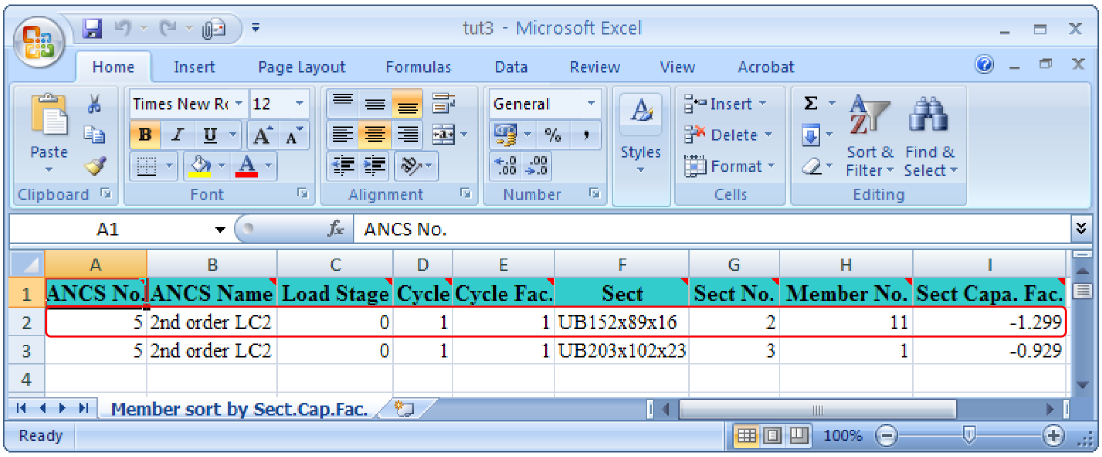


Choose to export ‘1 member extreme value for each selected section by section capacity factor’;

Following a similar process, select the ‘Member – Sections’ by clicking the ‘Section’ button and choose the sections to be exported in the <Select Sections> window;



Click ***OK*** to go back to the <Export Statistical Results> window and click ***OK***, an excel spreadsheet will be generated displaying the extreme section capacity factor for each section used in this tutorial;



ANCS No.	ANCS Name	Load Stage	Cycle	Cycle Fac.	Sect	Sect No.	Member No.	Sect Capa. Fac.
5	2nd order LC2	0	1	1	UB152x89x16	2	11	-1.299
5	2nd order LC2	0	1	1	UB203x102x23	3	1	-0.929

From the above spreadsheet, UB152x89x16 is observed to have a section capacity factor exceeding 1, which would not be acceptable for design.

The member design details based on the selected design code for each critical member can be seen in the ***.nsd file which is located in the directory of data file when ‘Export member design details (*.NSD file)’ option is selected.

Part of the design details is shown as below.

~~~~~~

[DESIGN ELEMENT]= 1; [NODE1]= 1, [NODE2]= 5; [LENGTH]= 3.0000

~~~~~~

<ANALYSIS CASE> : LC2 <UNIT> : kN,m

Load Stage : 0 Load Cycle : 1

[My1]=-0.5001E+00 [Mz1]=-0.2565E+02; [My2]= 0.9307E+00 [Mz2]= 0.1229E+02

[Vy1]= 0.1265E+02 [Vz1]= 0.4770E+00; [Vy2]= 0.1265E+02 [Vz2]= 0.4770E+00

[P]=-0.4947E+02 (Compression); [Mt]= 0.1344E-03 (Torsion)

Design Moments: [Myd]=-0.5001E+00 [Mzd]=-0.2565E+02 [at]= 0.0000E+00

Design Shears : [Vyd]= 0.1265E+02 [Vzd]= 0.4770E+00 [at]= 0.0000E+00

Design Type : "Beam-Column" Design Code : "HKSC[2011]"

Seismic Design: 0 (Not Considered)

<MATERIAL PROPERTIES> : S275

Elasticity Modulus, [E]= 0.2100E+09; Yield Strength, [py]= 0.2750E+06

Design Strength: [py1]= 0.2750E+06(Bending), [py2]= 0.2750E+06(Axial)

<SECTION PROPERTIES> : UB203X102X23

Area = 0.3233E-02 Iz = 0.2235E-04 Iy = 0.1500E-05 J = 0.1000E-06

Zz(+)= 0.2070E-03 Zy(+)= 0.3220E-04 Zz(-)= 0.2070E-03 Zy(-)= 0.3220E-04

Sz = 0.2340E-03 Sy = 0.3864E-04 rz = 0.8460E-01 ry = 0.2360E-01

Avz = 0.1893E-02 Avy = 0.1097E-02

Note: The plastic modului are limited to Sy= 1.20 Zy, Sz= 1.20 Zz.

<SECTION CLASSIFICATION> : Rolled; I/H-Section

Mat. Coeff.[e]= 1.0000 [=SQRT(275/py)]

Web : [d/t]= 34.1852 <= 80e/(1+r1) = 67.7695

Flange : [b/T]= 5.4731 <= 9e = 9.0000

Section Class = [1] "Plastic"

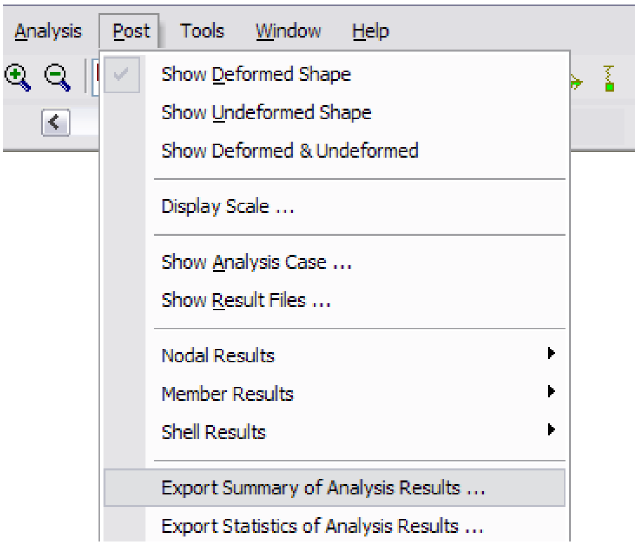
Strength Reduction Factor for Slender Section = 1.0000

NOTE: ELASTIC MODULUS IS USED IN DESIGN.

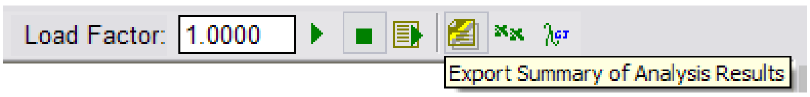
…

3.20 Step 20 – View result: check the structural adequacy by second-order analysis (ii) Export Summary of Analysis Results

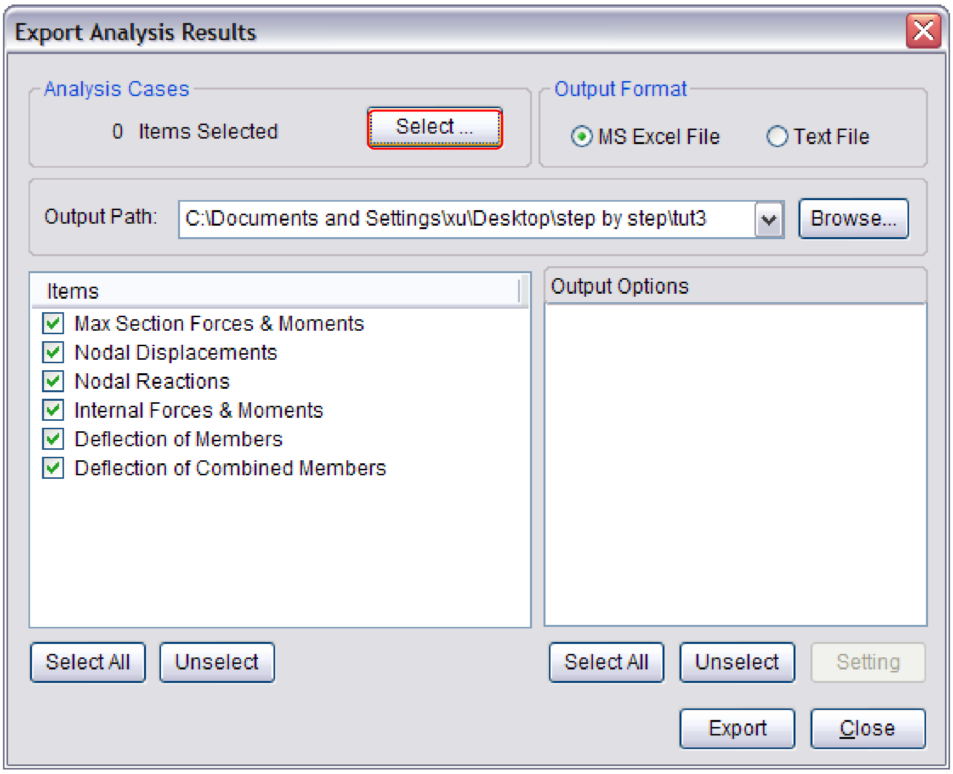
To further examine detailed second-order analysis results, click the ***Post*** **>** ***Export Summary of Analysis Results*** command;



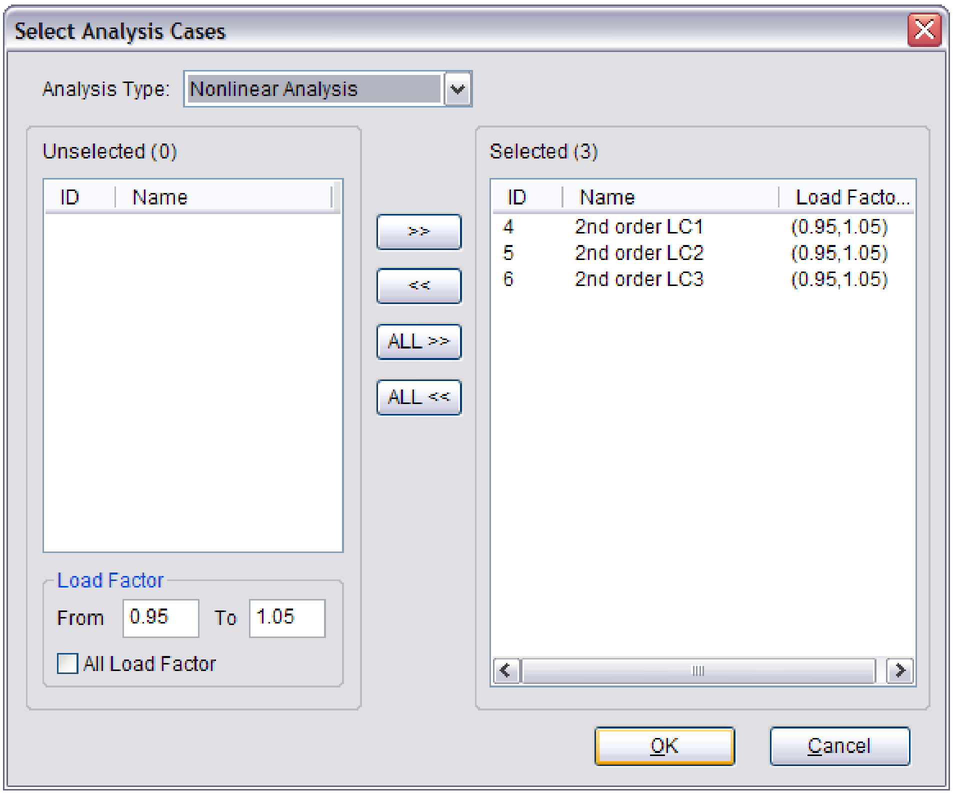
or click the <Export Summary of Analysis Results> shortcut in the toolbar;



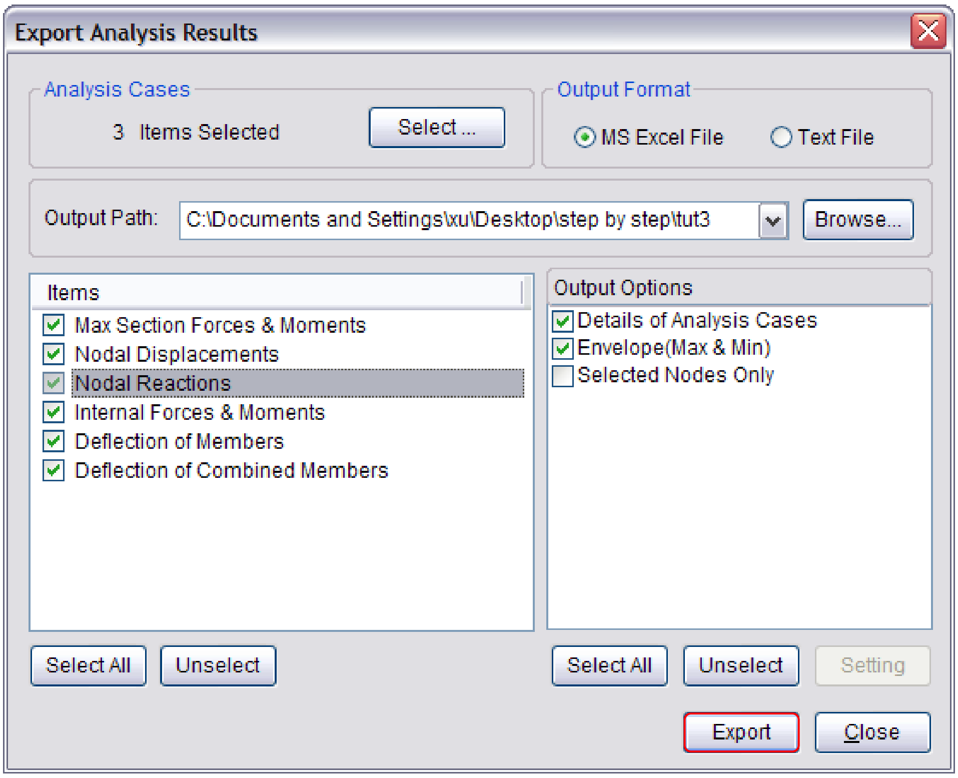
The <Export Analysis Results> window will pop up including the selection of analysis cases, items and output options;



Click ‘Select’, choose the analysis cases to be considered from the <Select Analysis Cases> window;



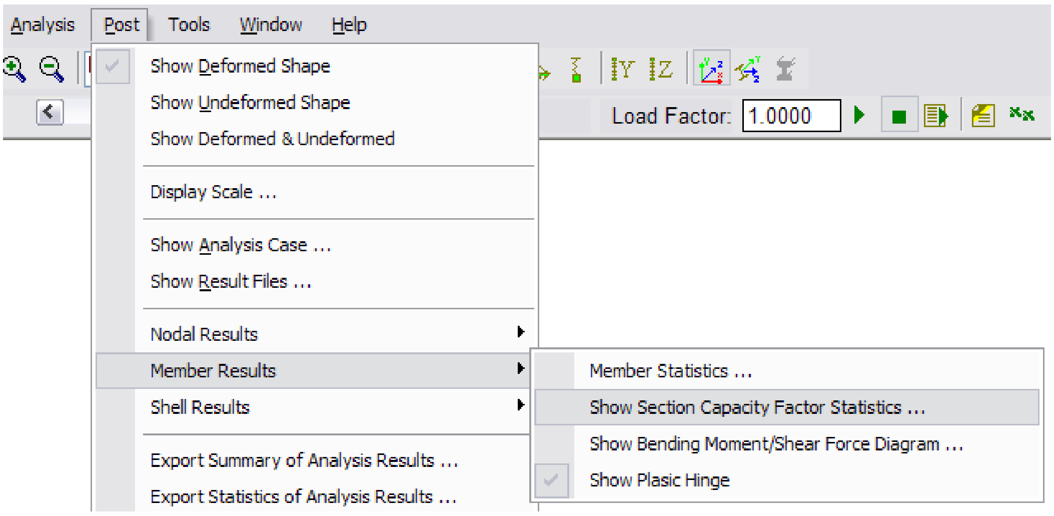
Click ***OK*** to go back to the <Export Analysis Results> window;



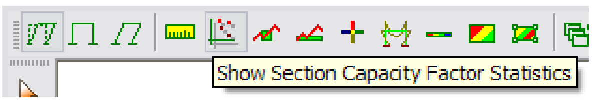
Click ***Export***, all the outputs will be generated by excel spreadsheets.

3.21 Step 21 – View result: check the structural adequacy by second-order analysis (iii) Show Section Capacity Factor Statistics

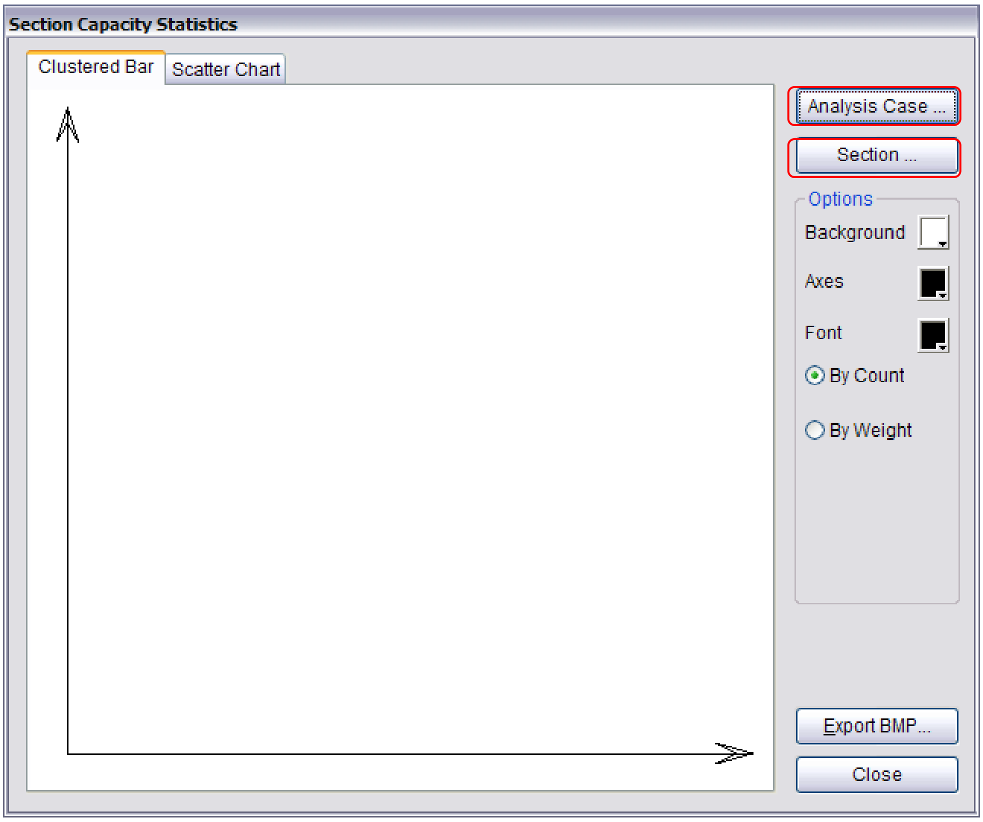
Click the ***Post*** > ***Member Results*** > ***Show Section Capacity Factor Statistics*** command;



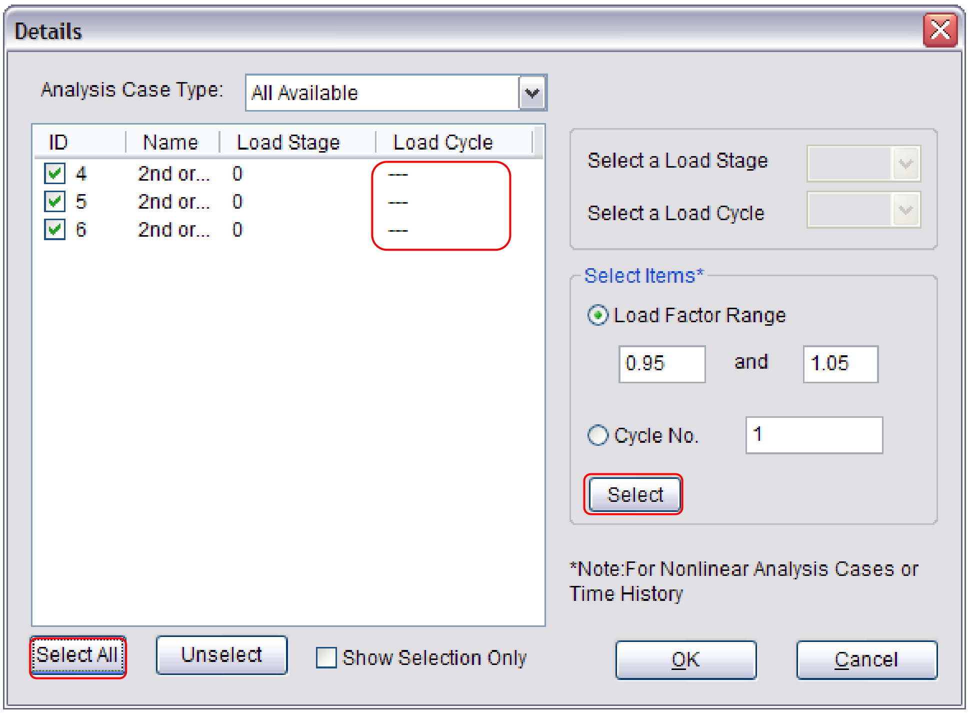
or click the <Show Section Capacity Factor Statistics> shortcut in the toolbar;



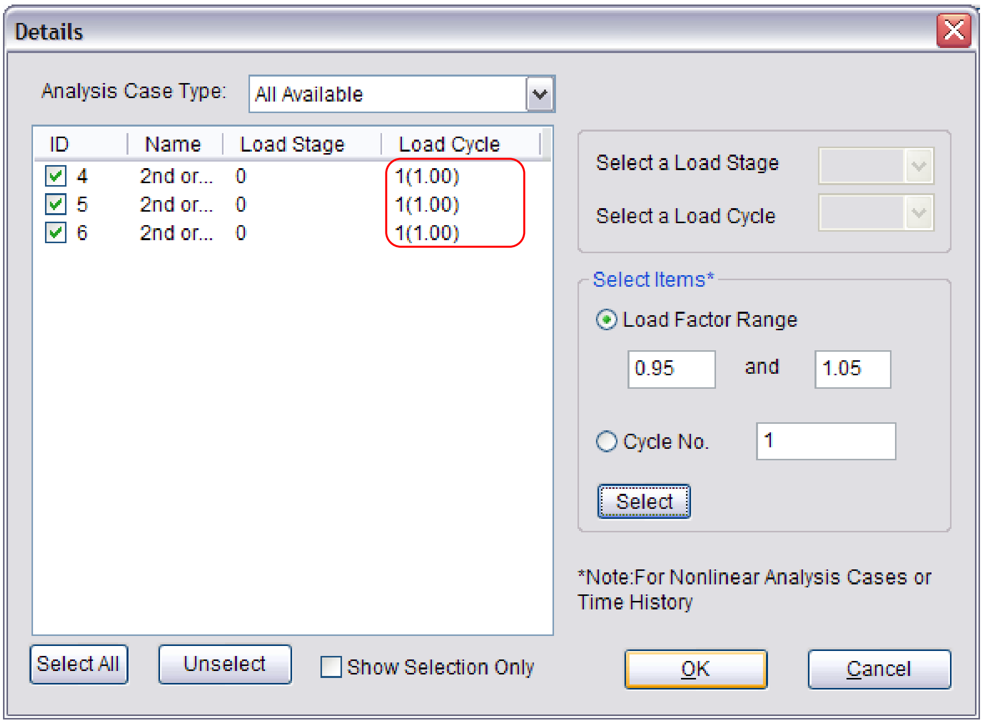
The <Section Capacity Statistics> window will appear;



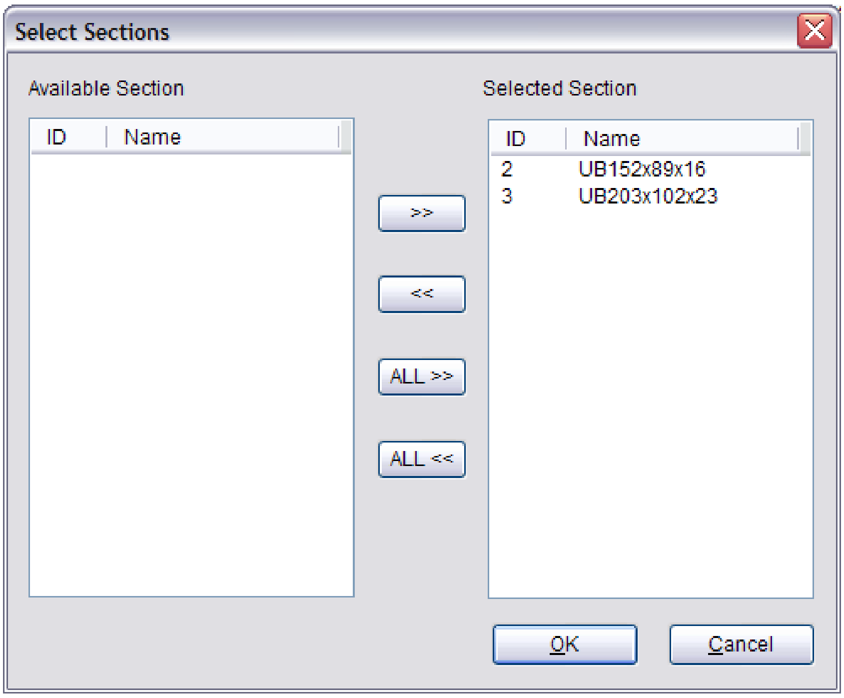
Click ‘Analysis Case’ in the <Section Capacity Statistics> window, the <Details> window will pop up;



Click ‘Select All’ and then ‘Select’ items with ‘Load Factor Range 0.95 – 1.05’, then the ‘Load Cycle’ will be displayed after each analysis case;

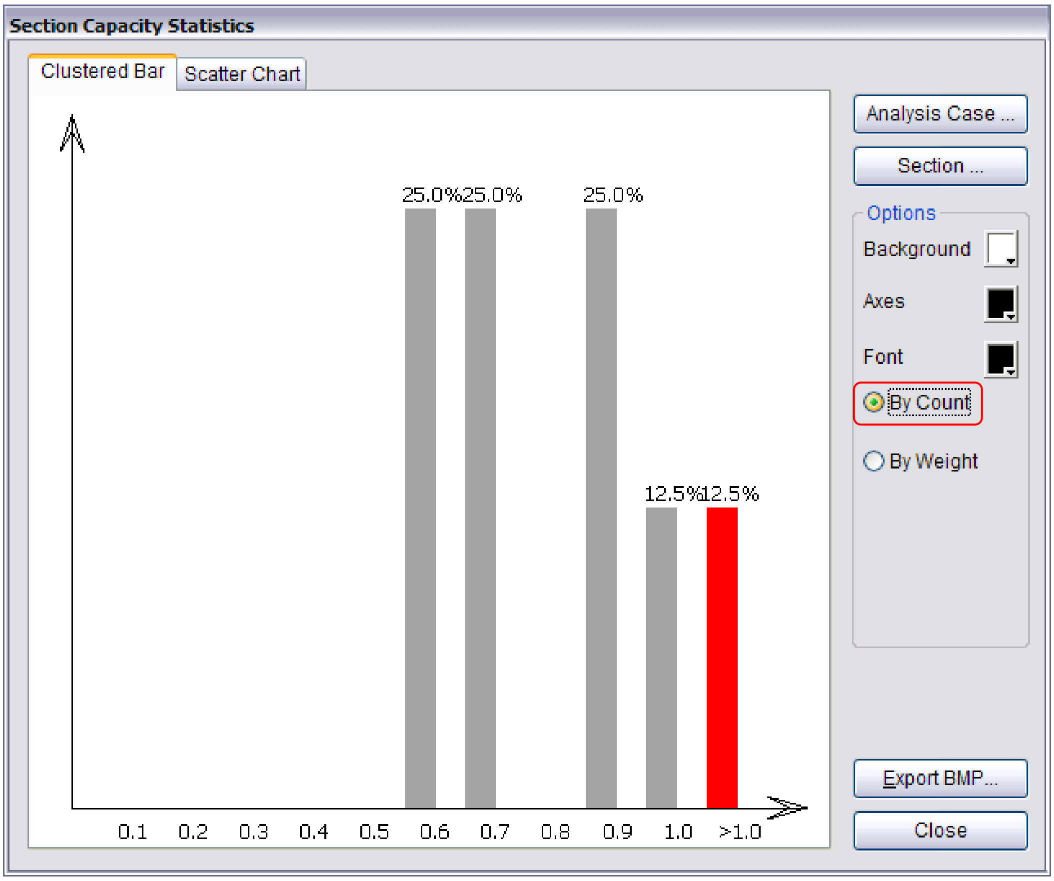


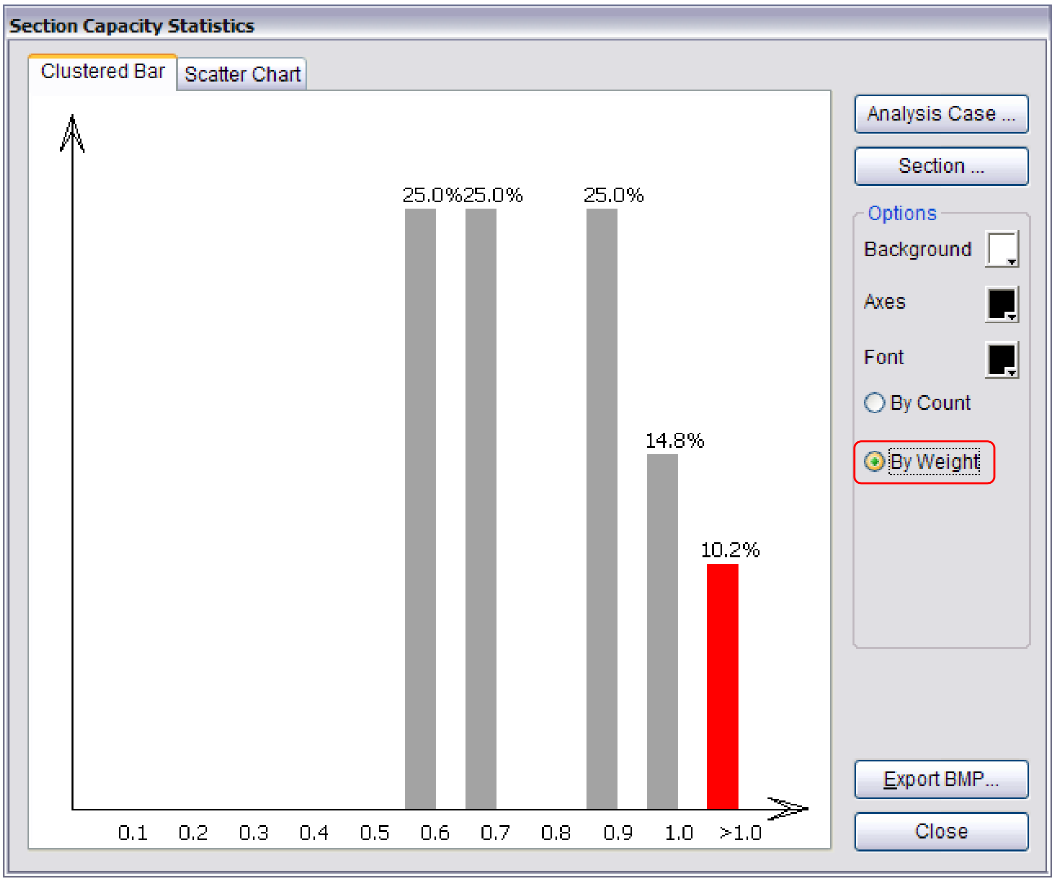
Click ***OK*** to go back to the <Section Capacity Statistics> window and then select sections by clicking ‘Section’ in the <Section Capacity Statistics> window, the <Select Sections> window will then pop up;



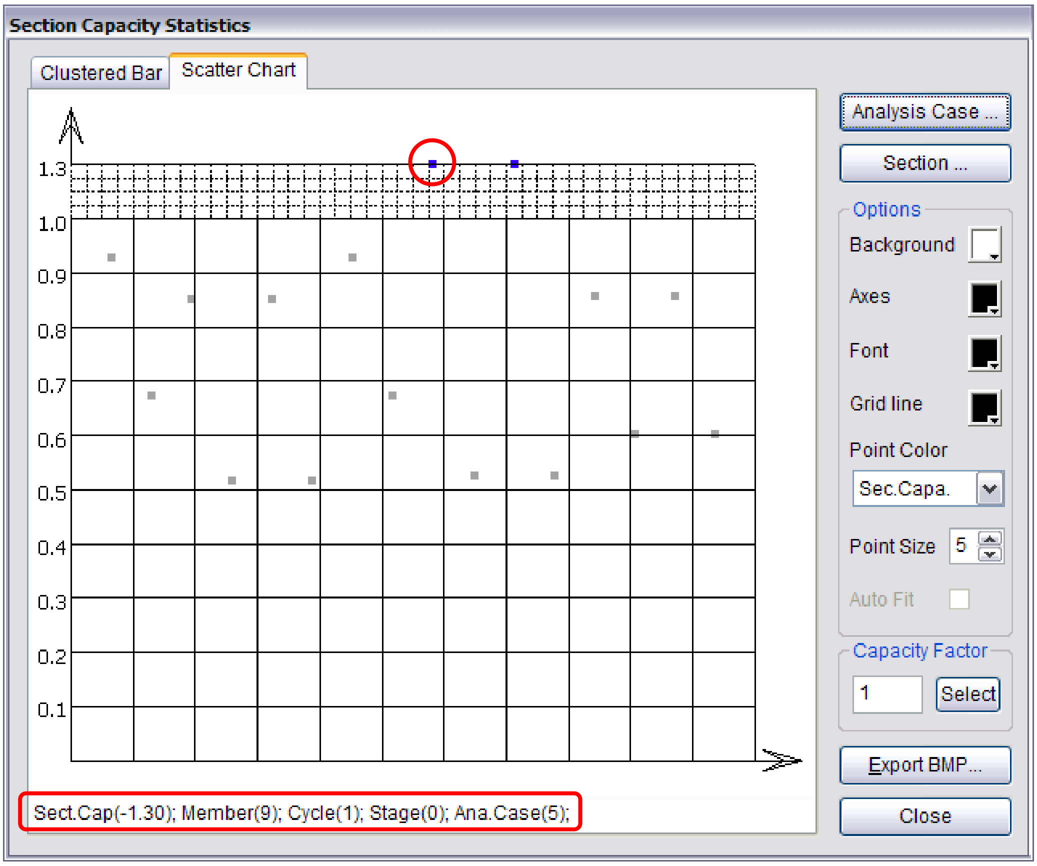
Import the sections to be considered and click ***OK*** to go back to the <Section Capacity Statistics> window;

The bar charts showing the section capacity distributions could be sorted either by counting the number of members –‘By Count’ or by summarizing the weight of members –‘By Weight’ within each percentage range as shown below;

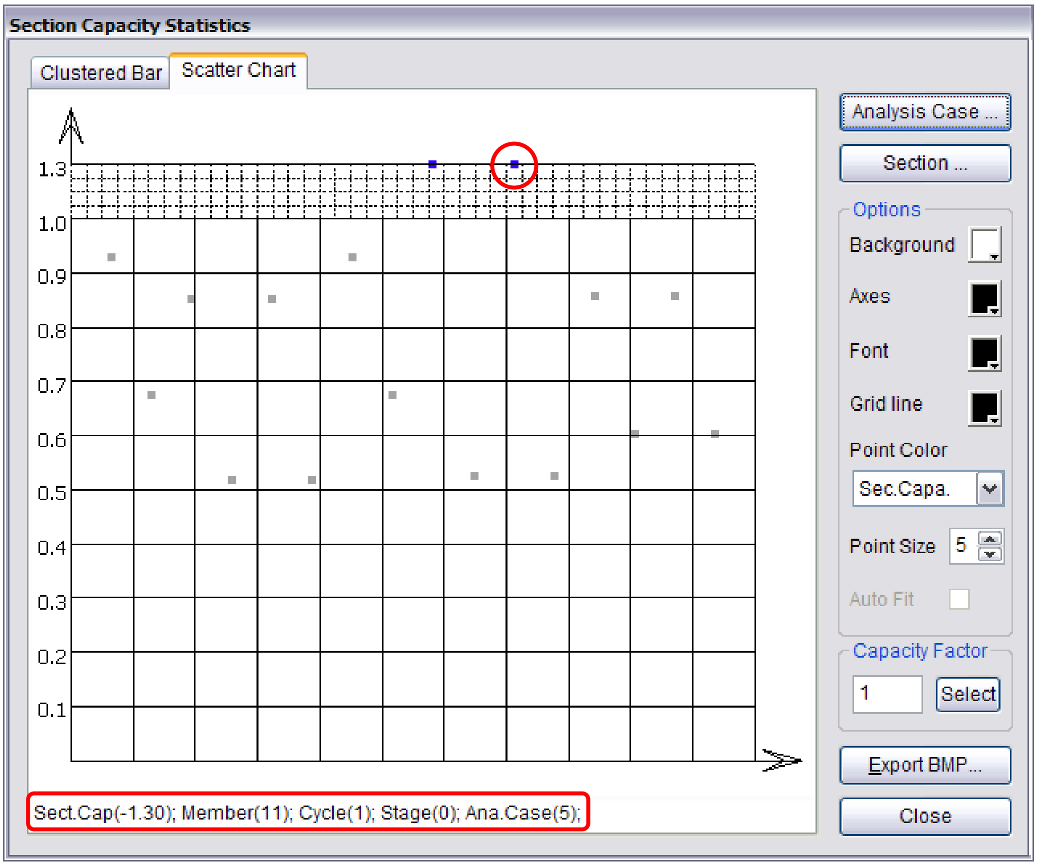




To have an overview of the section capacities of every member, a scatter chart could also be produced;

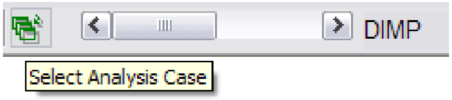


Move the mouse to one of the points with section capacity factor exceeding 1.0, the value of section capacity and member/cycle/stage/analysis case numbers will be displayed below the chart;

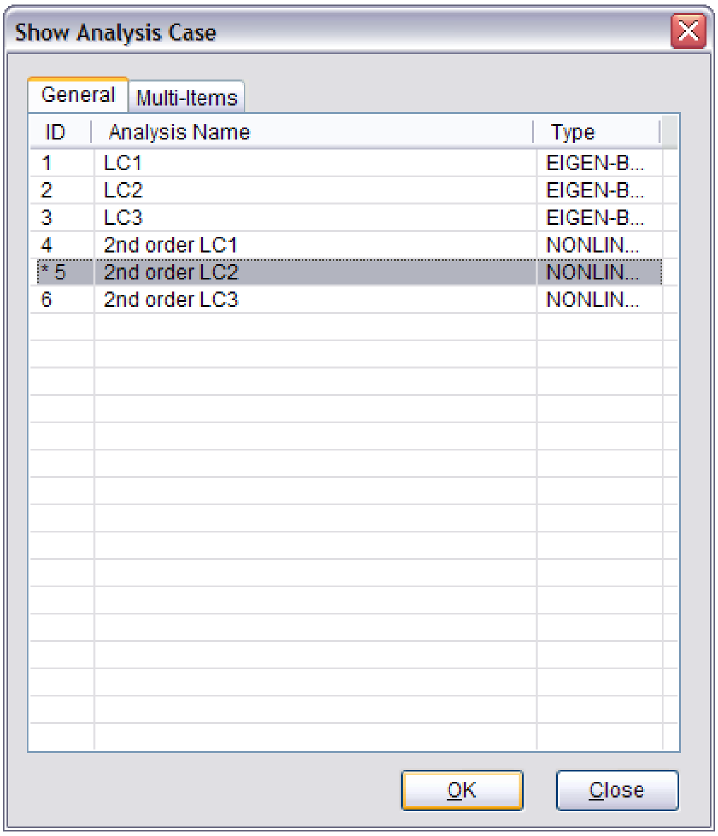


Both of the two members (9) and (11) with section capacity factors -1.3 belong to analysis case 5 – 2nd order LC2;

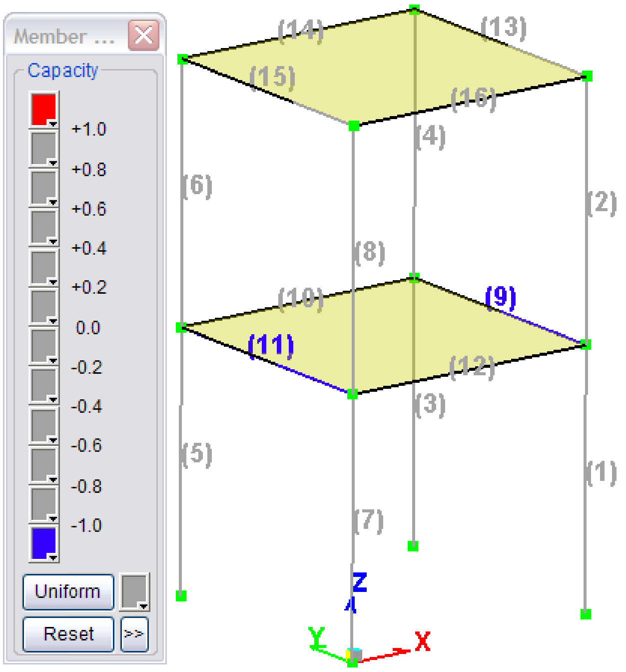
Click the <Select Analysis Case> shortcut in the post analysis toolbar;



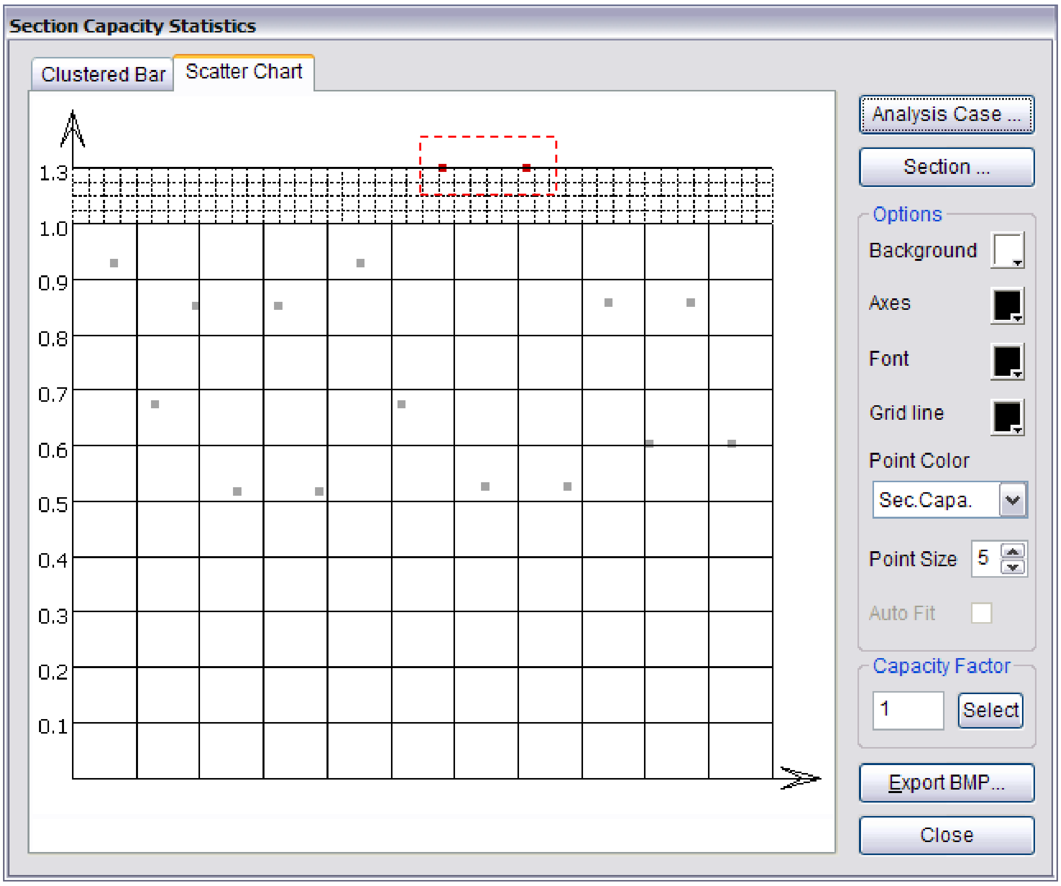
Select analysis case ‘5 – 2nd order LC2’ from the <Show Analysis Case> window;



Click ***OK*** to close the <Show Analysis Case> window, the deformed configuration is then displayed on the screen as below;

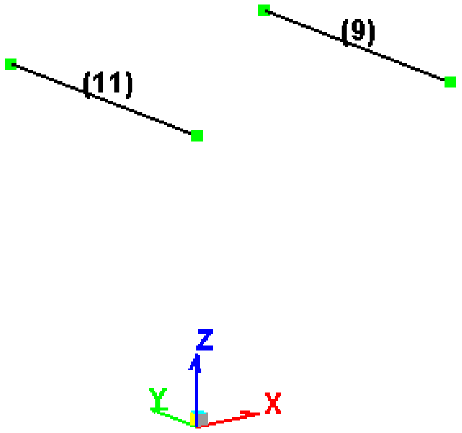


Select both of the two points described above,



Click the <Show Selection Only> shortcut in the toolbar;

The two members that have failed the 2nd order analysis are then displayed as below.



4. TUTORIAL 4 – Modal Response Spectrum Analysis

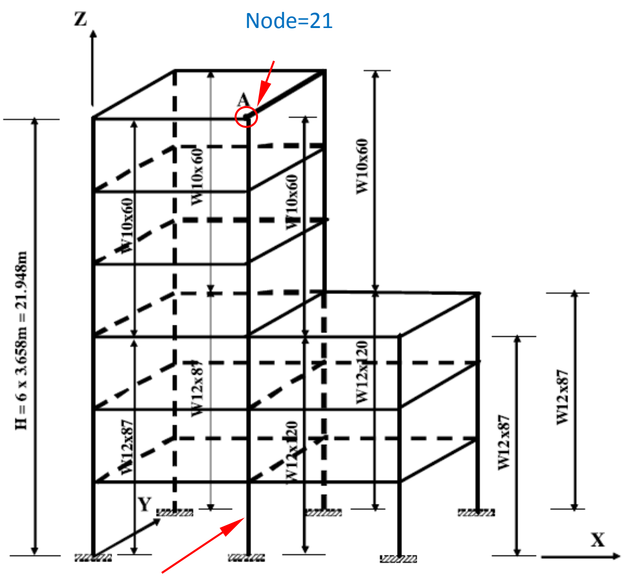
A six-story rigid steel space frame originally studied by Orbison et al. is adopted here for demonstration of the modal response spectrum analysis (MRSA) by NIDA. The details of the space frame are given as below:

(a) Geometrical dimensions and section sizes: shown in Figure as below;

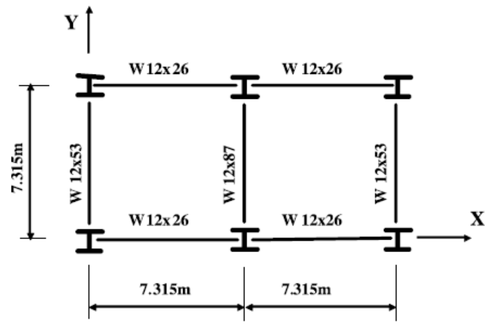
(b) The material properties for all members: Young's modulus E=206850 MPa, shear modulus G=79293 MPa, yield strength py=250 MPa;

(c) Applied loads: uniform floor pressure=9.6 kN/m2, wind loads simulated by point loads of 53.376 kN in Y-direction at every beam-column joints;

(d) Boundary conditions: all columns are fixed to foundation.



(a) 3D view



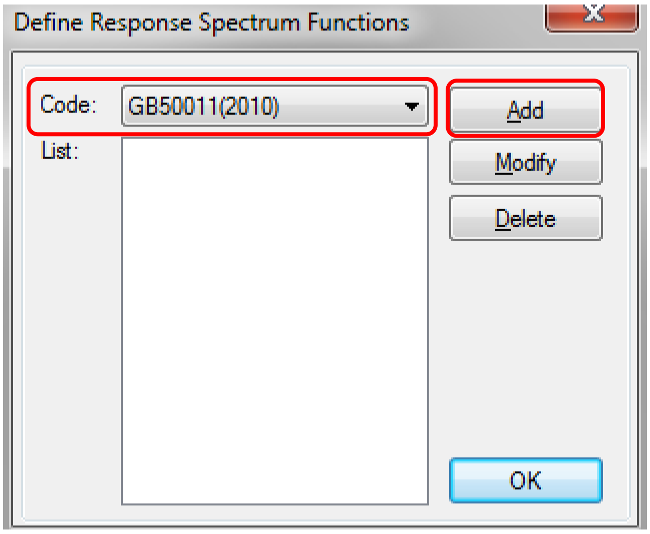
(b) Plan view

4.1 Step 1 – Build the structural model

Input the nodal coordinates, material properties, section properties, applied loads, boundary conditions, etc., more details can be referred to tutorial 1 to 3.

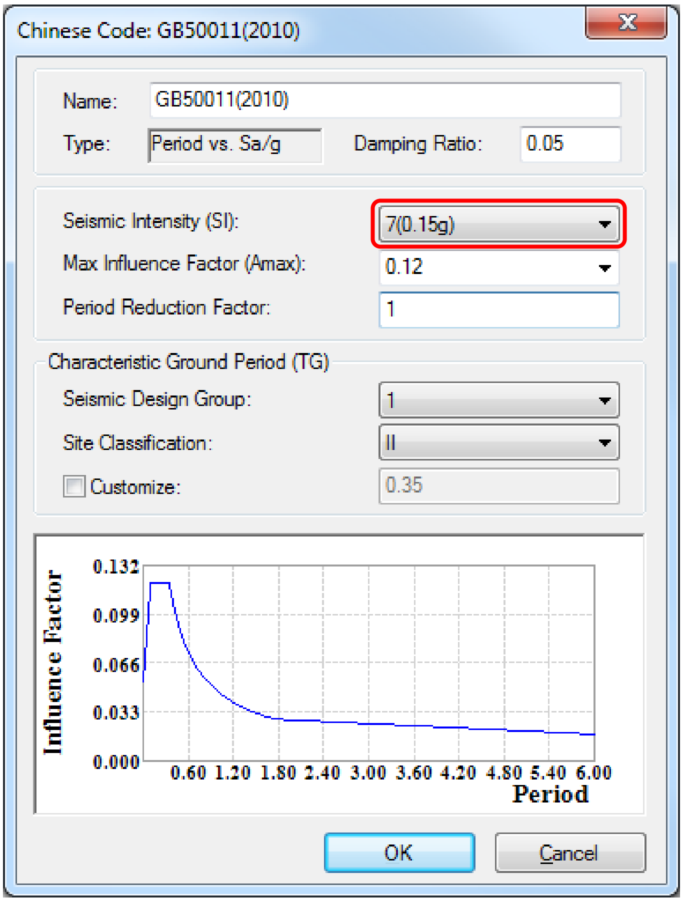
4.2 Step 2 – Define a response spectrum function

Click the ***Construct*** > ***Response Spectrum Functions*** command,



Select the seismic design code ‘GB50011 (2011)’ and click ***Add*** to create a response spectrum function;

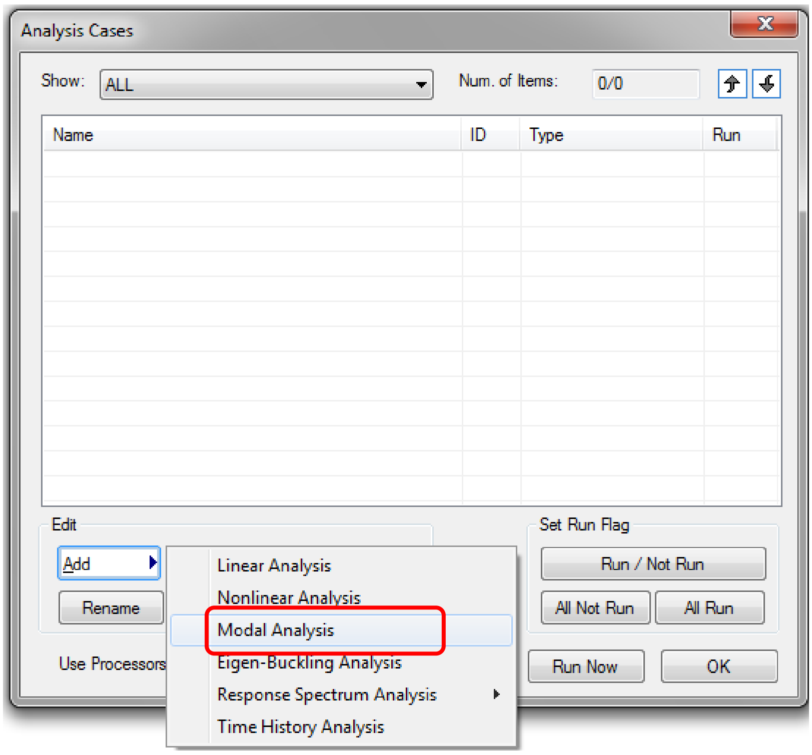
According to GB50011, the design seismic intensity of Hong Kong is 7 (0.15g), and the seismic design group is 1. The site classification should be given by official department or site investigation, it is specified as 2 here for demonstration.

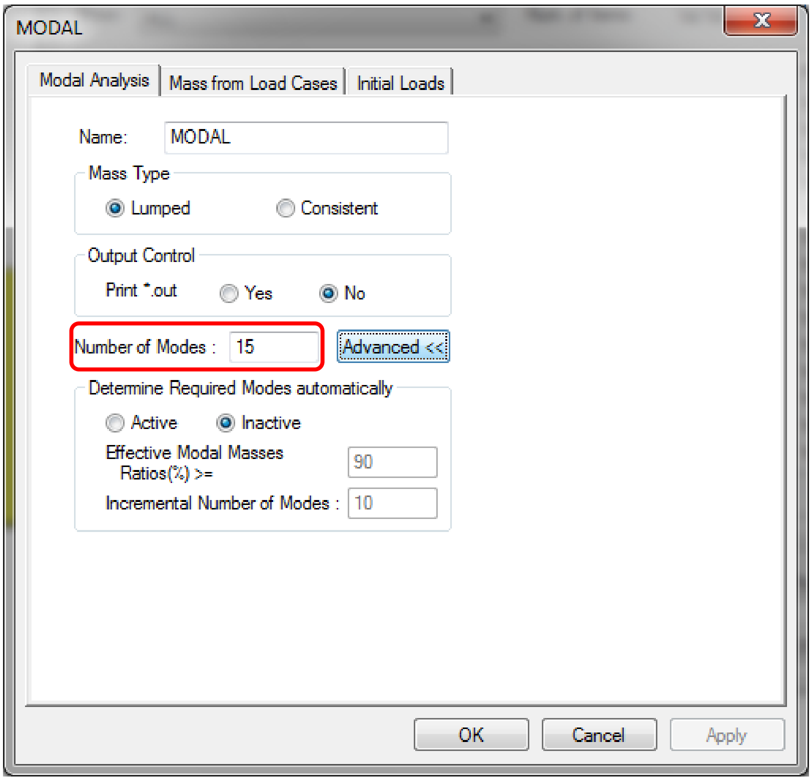


Click ***OK*** to continue.

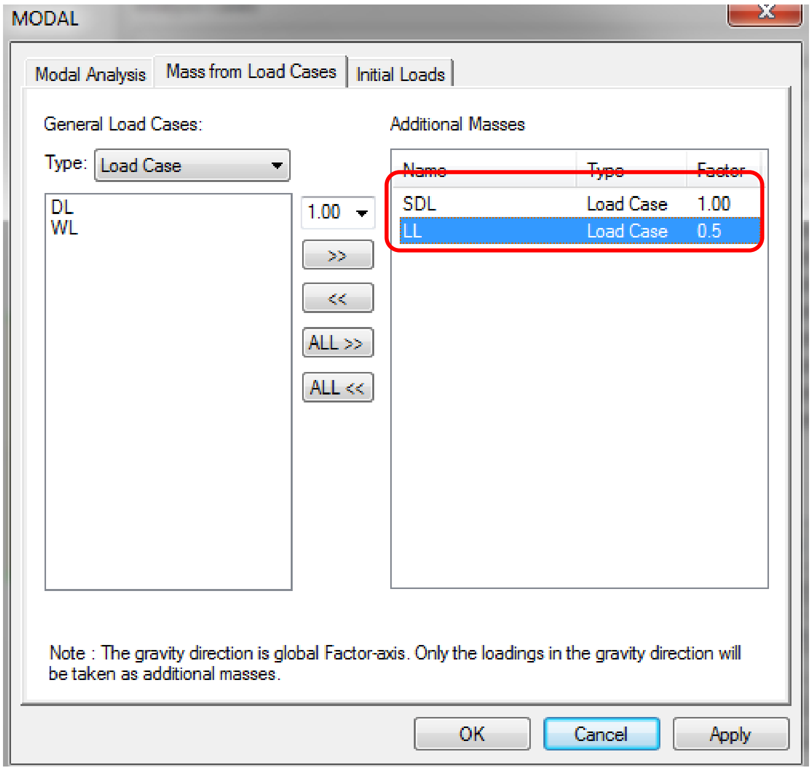
4.3 Step 3 – Define a modal analysis

Click the ***Analysis*** > ***Set Analysis Cases…*** command,



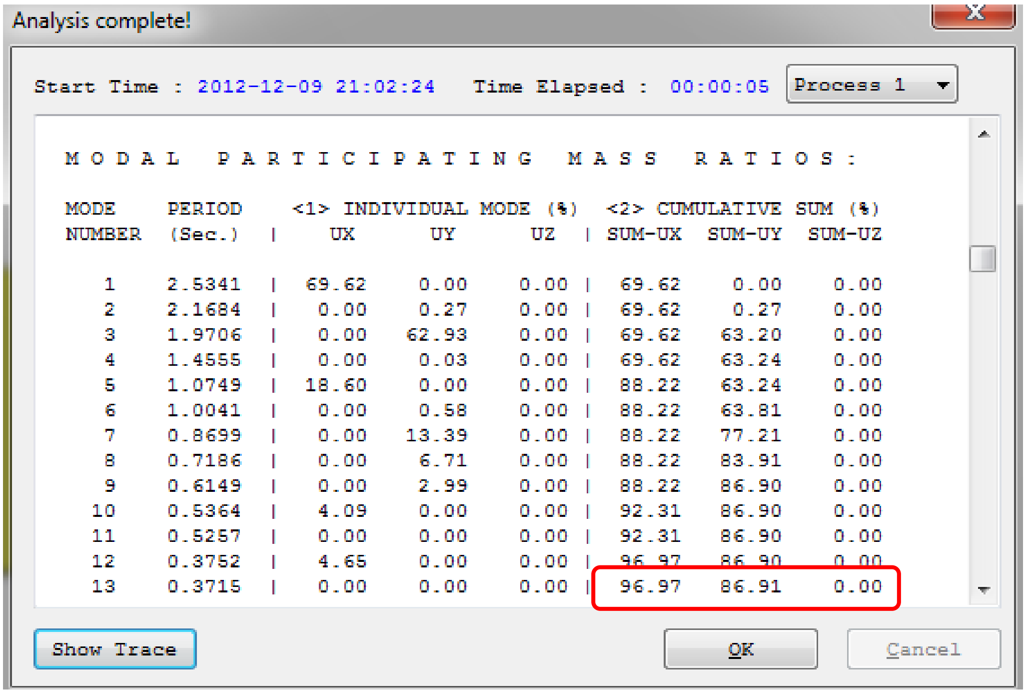


It should carefully consider the structural masses to get the natural periods and vibration modes. Besides structural self-weight, the other dead loads and part of live loads should be taken as structural masses manually, seen figure as below. Noted that the structural self-weight is automatically included and therefore user need not add them as additional masses once more.



Click ***OK*** to continue.

After performing the modal analysis, the modal participating mass ratios will be shown in the trace window as below.


Analysis complete!

Start Time : 2012-12-09 21:02:24 Time Elapsed : 00:00:05 Process 1

M O D A L P A R T I C I P A T I N G M A S S R A T I O S :

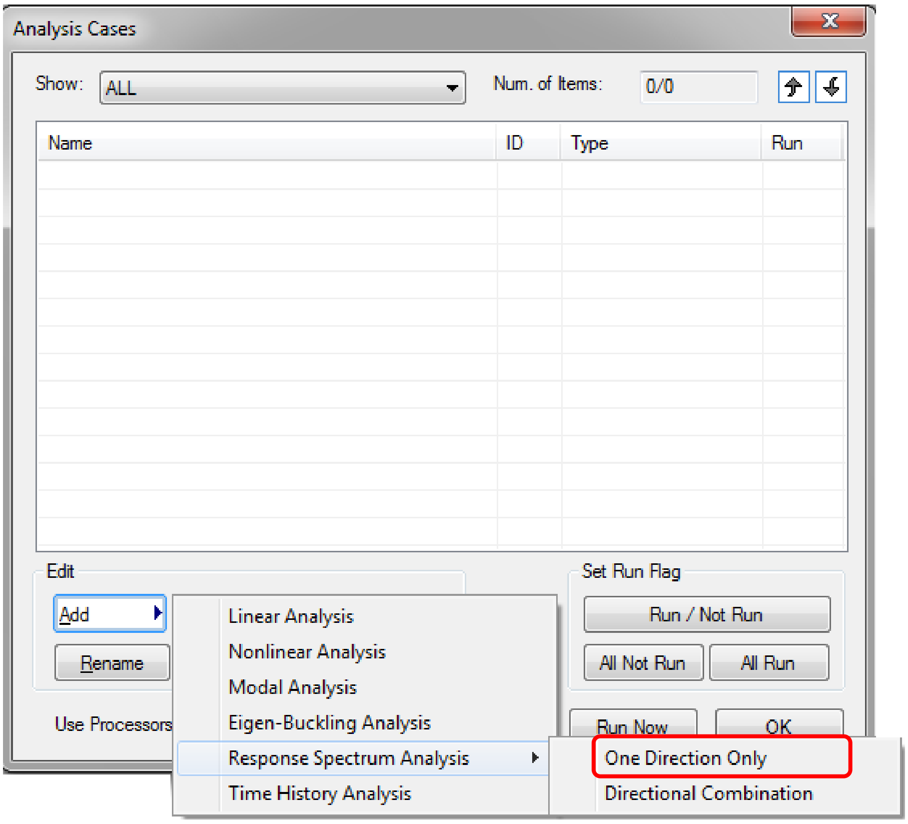
MODE NUMBER	PERIOD (Sec.)	<1> INDIVIDUAL MODE (%) UX	UY	UZ	<2> CUMULATIVE SUM (%) SUM-UX	SUM-UY	SUM-UZ
1	2.5341	69.62	0.00	0.00	69.62	0.00	0.00
2	2.1684	0.00	0.27	0.00	69.62	0.27	0.00
3	1.9706	0.00	62.93	0.00	69.62	63.20	0.00
4	1.4555	0.00	0.03	0.00	69.62	63.24	0.00
5	1.0749	18.60	0.00	0.00	88.22	63.24	0.00
6	1.0041	0.00	0.58	0.00	88.22	63.81	0.00
7	0.8699	0.00	13.39	0.00	88.22	77.21	0.00
8	0.7186	0.00	6.71	0.00	88.22	83.91	0.00
9	0.6149	0.00	2.99	0.00	88.22	86.90	0.00
10	0.5364	4.09	0.00	0.00	92.31	86.90	0.00
11	0.5257	0.00	0.00	0.00	92.31	86.90	0.00
12	0.3752	4.65	0.00	0.00	96.97	86.90	0.00
13	0.3715	0.00	0.00	0.00	96.97	86.91	0.00

Show Trace OK Cancel

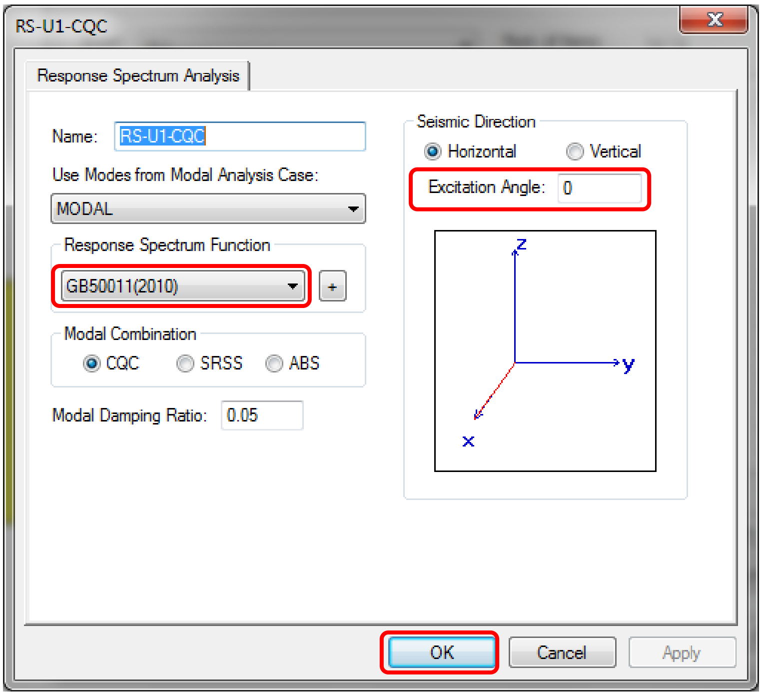
Noted that the number of modes should meet the requirement of “at least 90% of the participating mass” in codes.

4.4 Step 4 – Create "single direction" MRSA cases

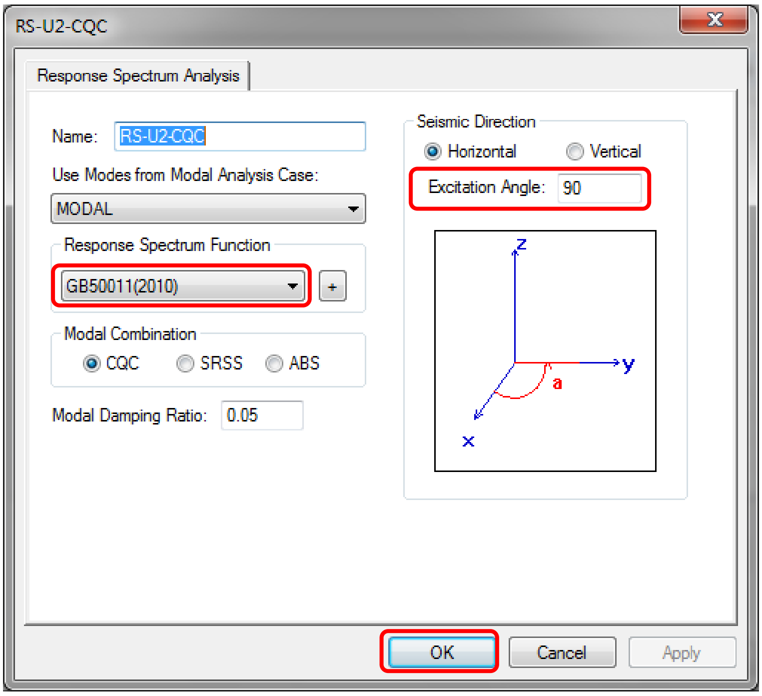
Click the ***Analysis*** > ***Set Analysis Cases…*** command,



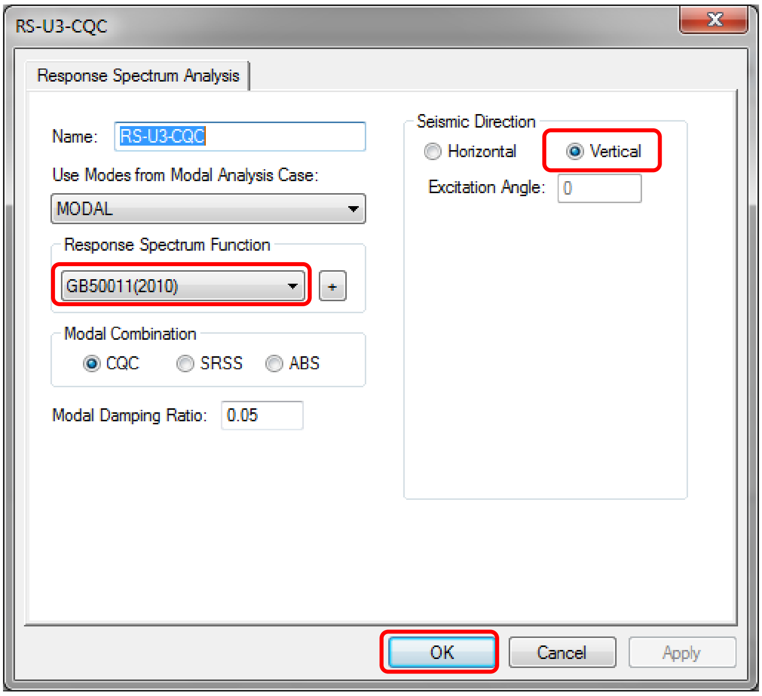
Create a response spectrum analysis case in global x direction;



Create a response spectrum analysis case in global y direction;



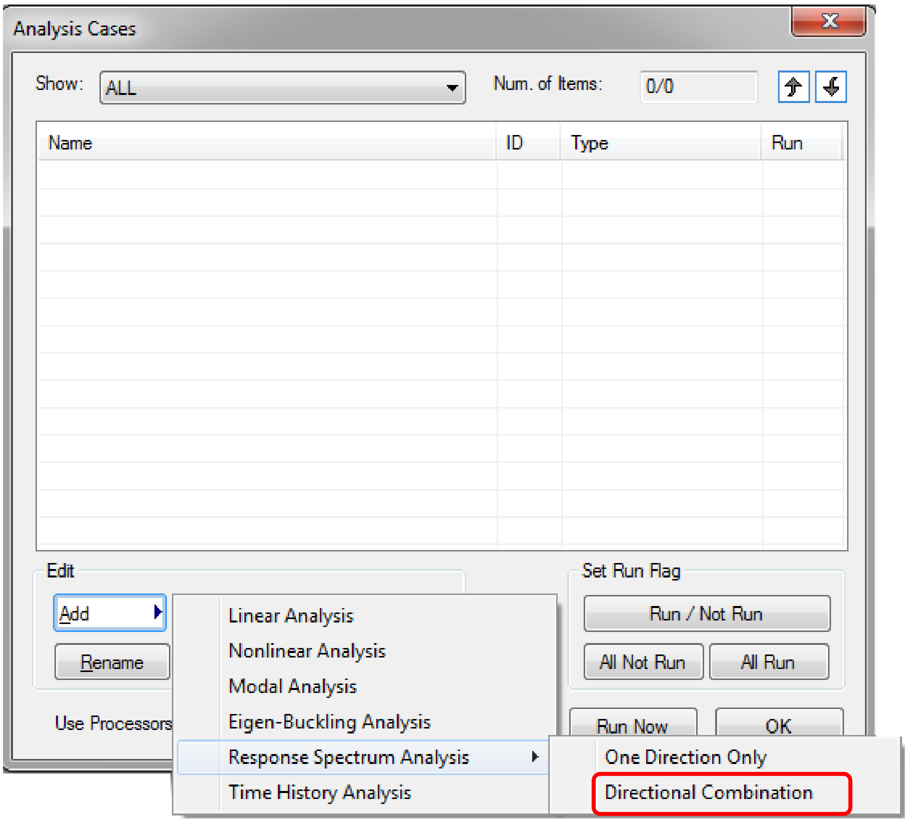
Create a response spectrum analysis case in global z direction;

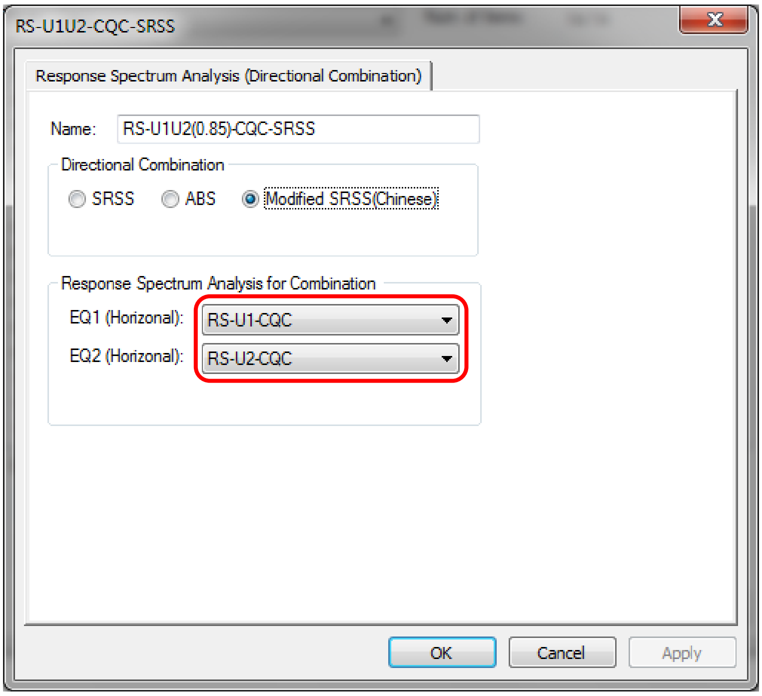


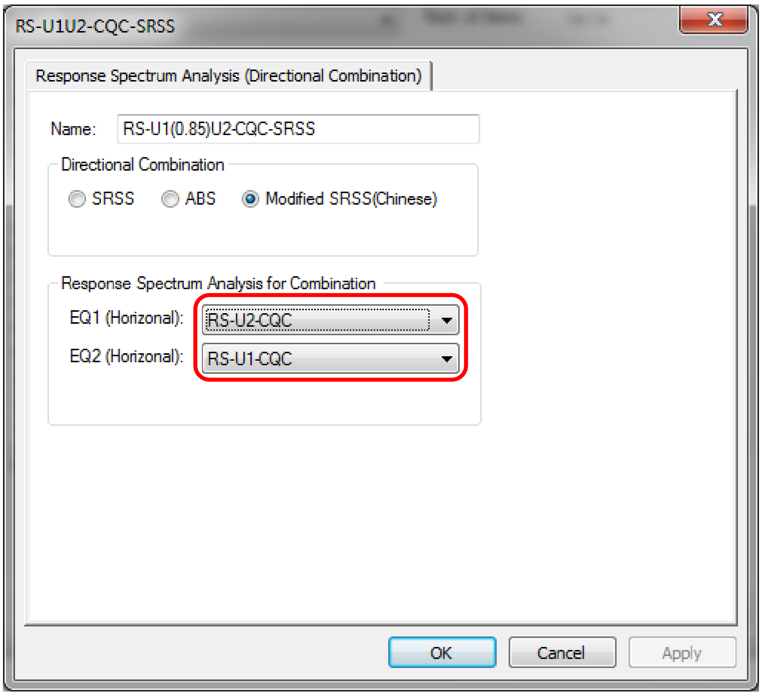
4.5 Step 5 – Create "directional combination" MRSA cases

If the two horizontal components (vertical component may be also needed) of the seismic action need to be considered simultaneously, user need to conduct directional combination of several MRSA cases.

Click the ***Analysis*** > ***Set Analysis Cases…*** command,







4.6 Step 6 – Create "load combination" considering both earthquake action and other actions

Click the ***Analysis*** > ***Set Analysis Cases…*** command;

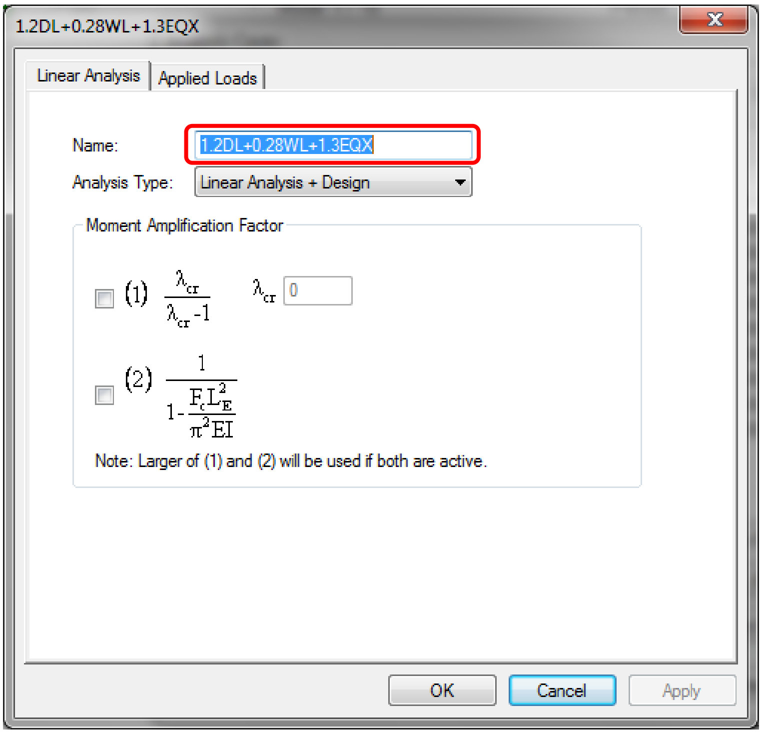
According to GB50011, the following load combinations are recommended in Hong Kong:

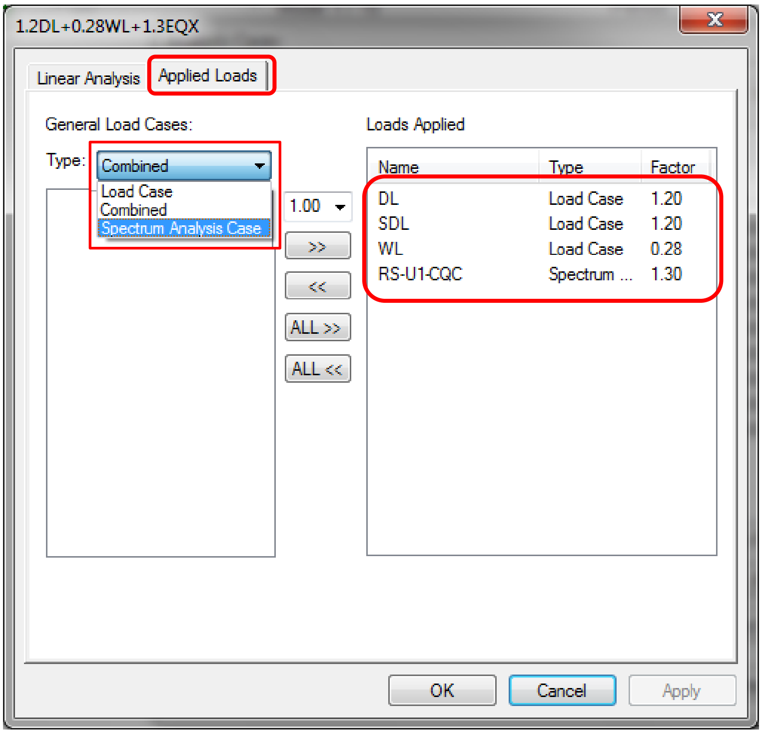
(a) 1.2DL+0.28WL+1.3EQX

(b) 1.2DL+0.28WL+1.3EQY

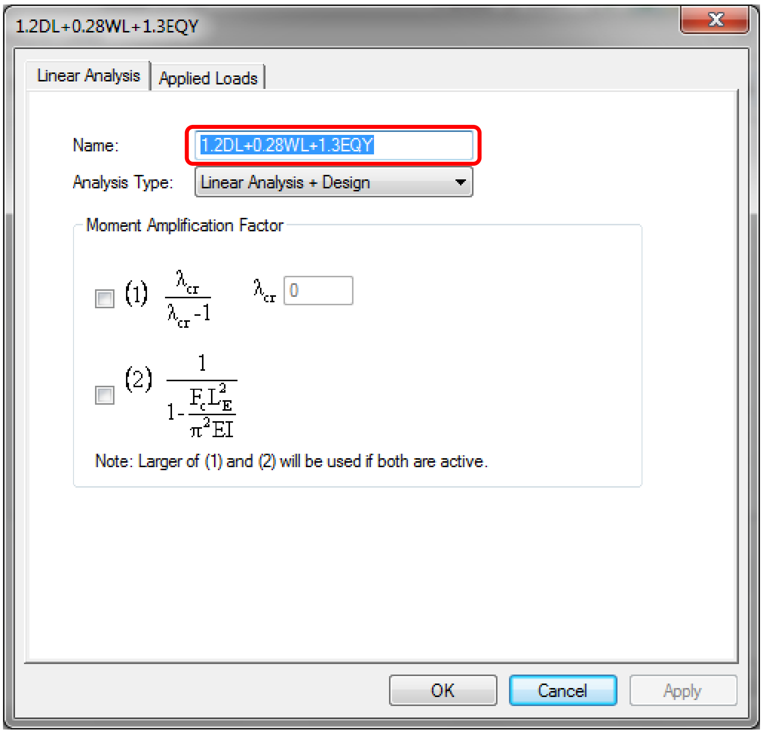
(c) 1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ

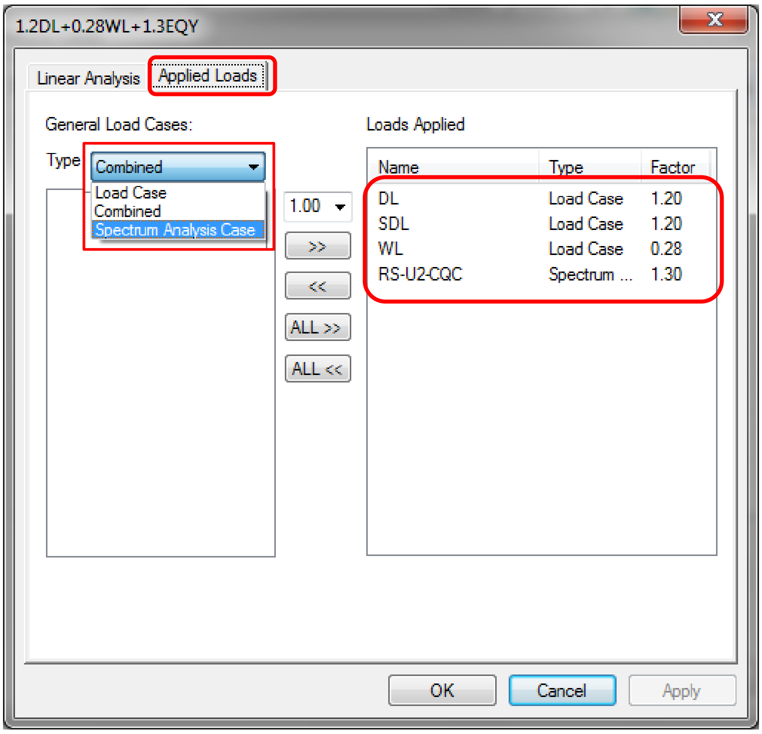
Create a linear analysis case "1.2DL+0.28WL+1.3EQX":



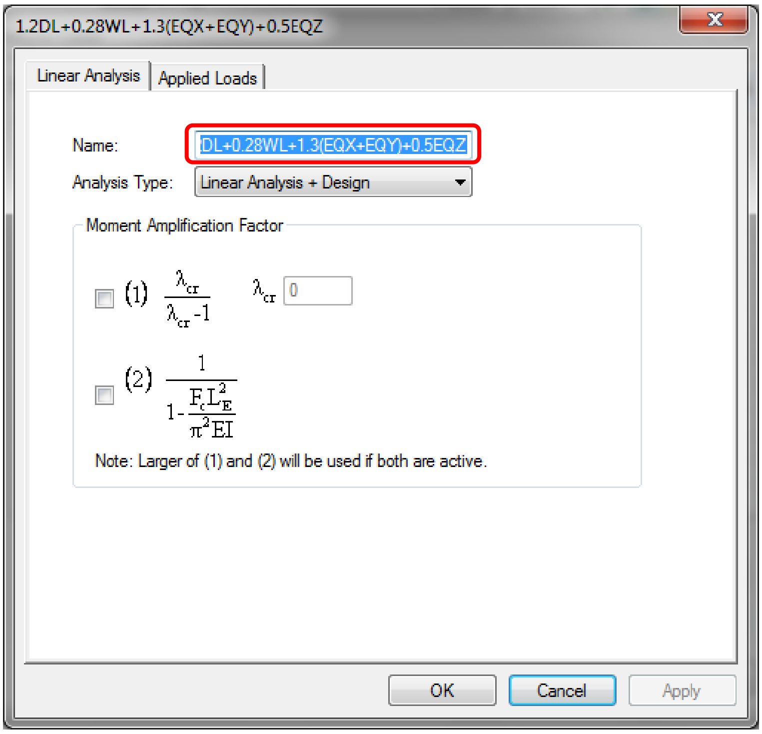


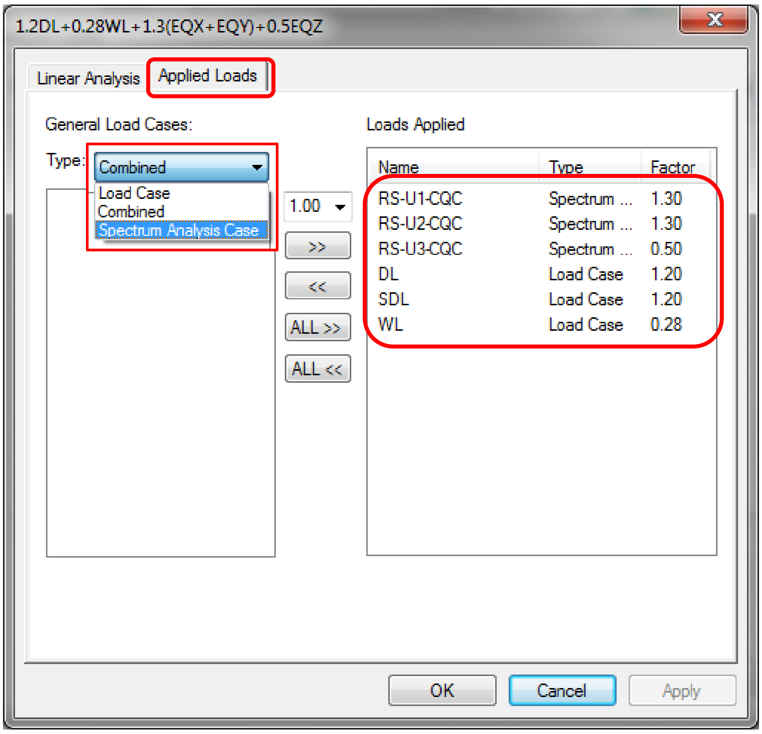
Create a linear analysis case “1.2DL+0.28WL+1.3EQY”:





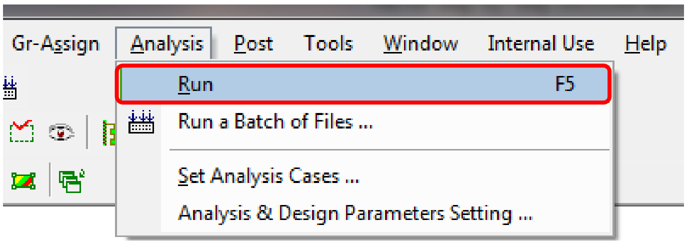
Create a linear analysis case “1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ”:





4.7 Step 7 – Run all analysis cases and view results

Click the ***Analysis*** > ***Run*** or press **F5** key to run all analysis cases;



The procedures for viewing and exporting analysis results can be referred to tutorial 1 to 3.

5. TUTORIAL 5 – Time History Analysis

A seven-story 2D steel frame is used here for demonstration of time history analysis by NIDA. The details of the 2D frame are given below:

(a) Geometrical dimensions and section sizes: shown in Figure as below;

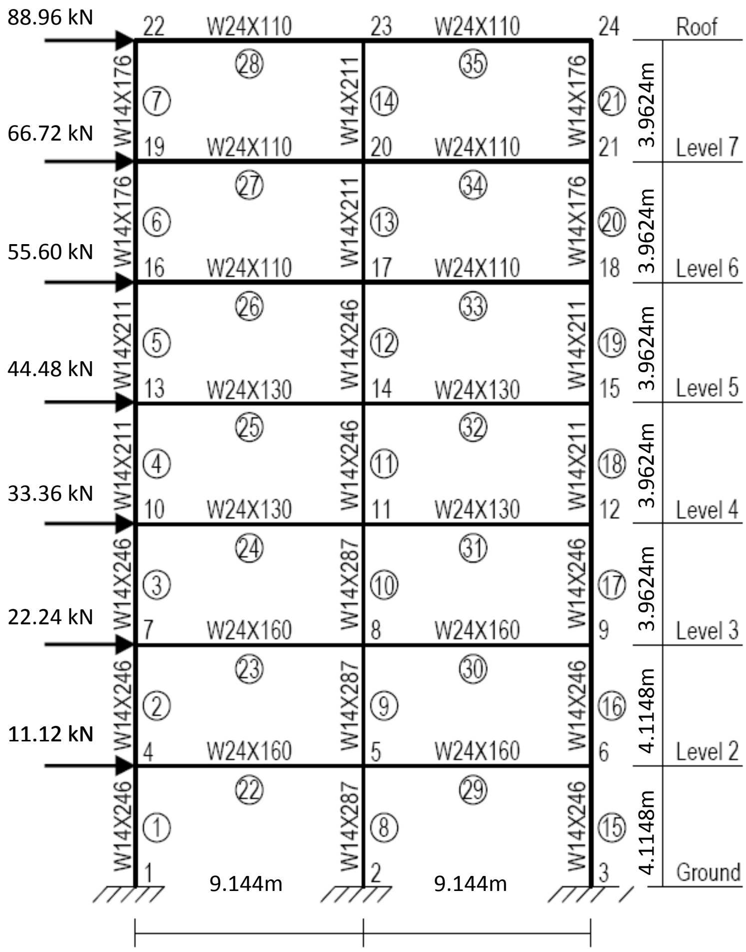
(b) The material properties for all members: Young's modulus E=2.034x105 MPa, Poisson's ratio v=0.3, yield strength py=250 MPa;

(c) Applied static loads: shown in Figure 10;

(d) Boundary conditions: all columns are fixed to foundation and member connections are rigid;

(e) Mass: 85 812.16 kg at each story (node 5, 8, 11, 14, 17, 20 and 23);

(f) Earthquake wave: the N-S component of the El Centro 1940.



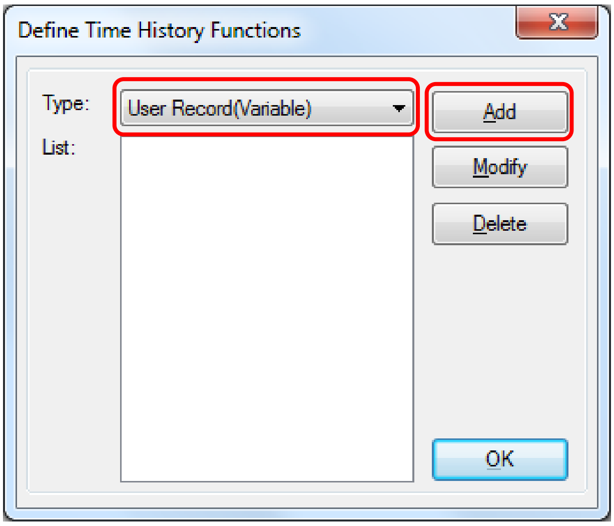
The procedure for performing time history analysis in NIDA (2010) is detailed as below.

5.1 Step 1 – Build the structural model

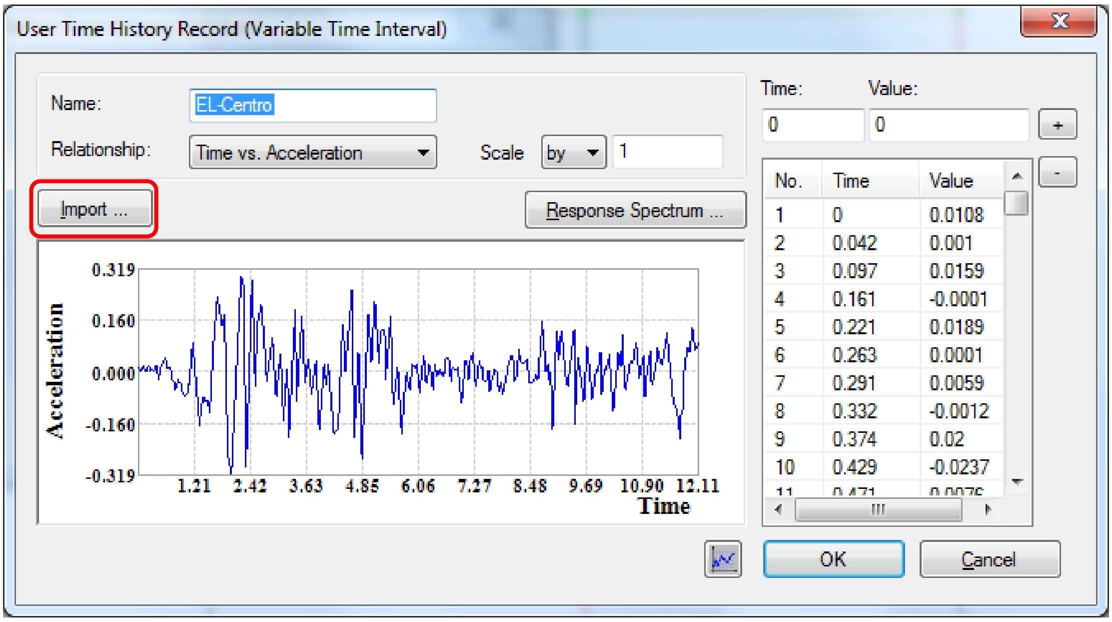
Input the nodal coordinates, material properties, section properties, applied loads, boundary conditions, etc., more details can be referred to tutorial 1 to 3.

5.2 Step 2 – Define one or more than one time history functions

Click the ***Construct > Time History Functions*** command,



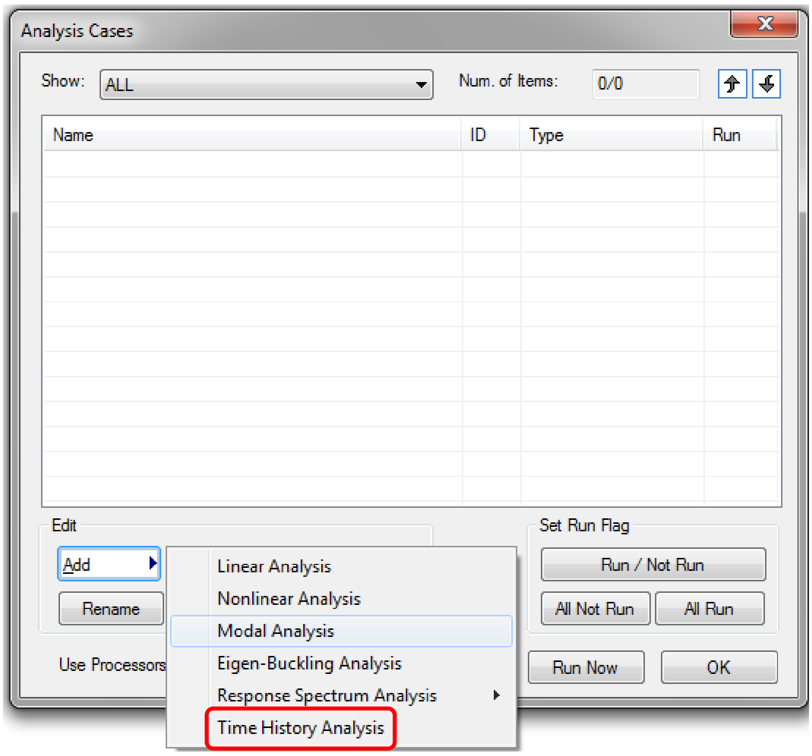
Select the type 'User Record (Variable)' and click ***Add*** to create a time history function;

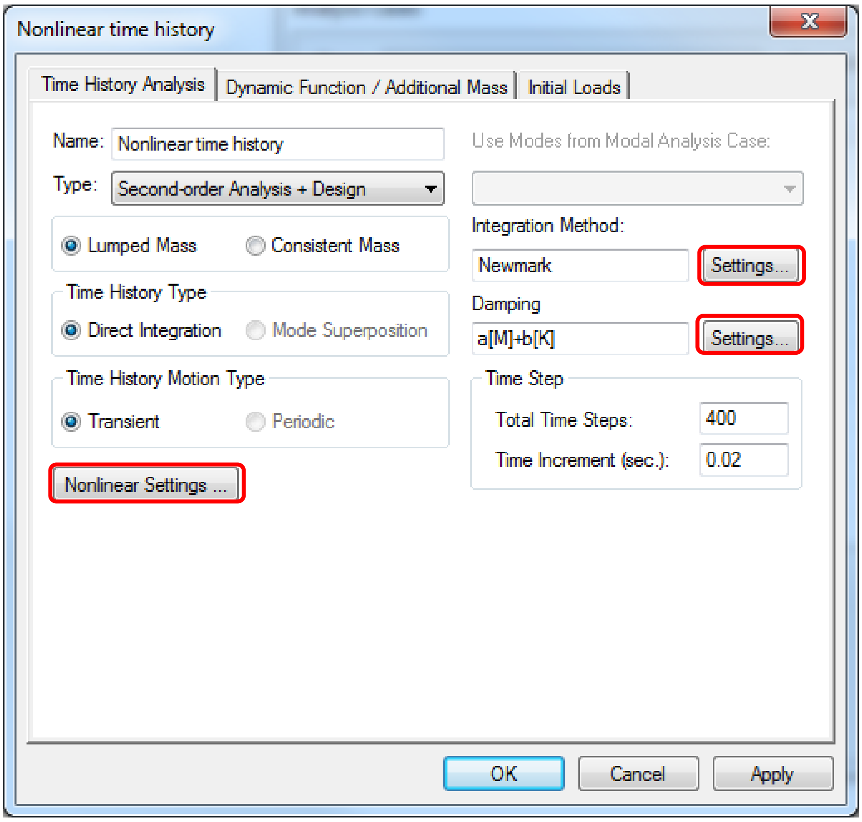


Click ***OK*** to continue.

5.3 Step 3 – Define a time history analysis case

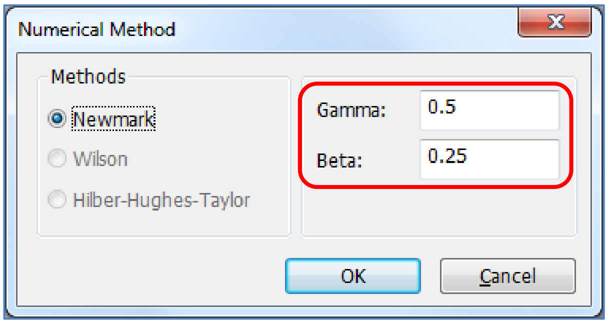
Click the ***Analysis*** > ***Set Analysis Cases…*** command,



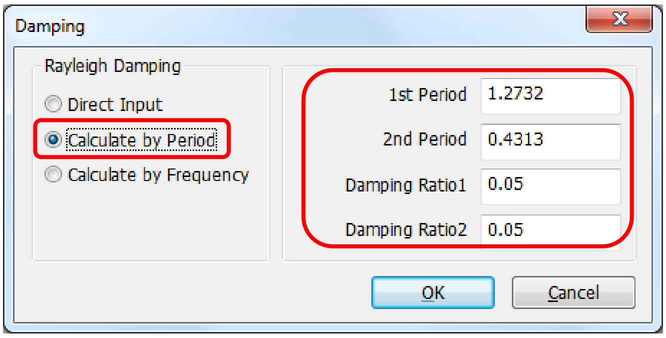


Input the case name, select 'Lumped Mass', 'Direct Integration', 'Transient'; give the 'Total Num. of Time Step' as 400 and 'Time Increment' as 0.02 second.

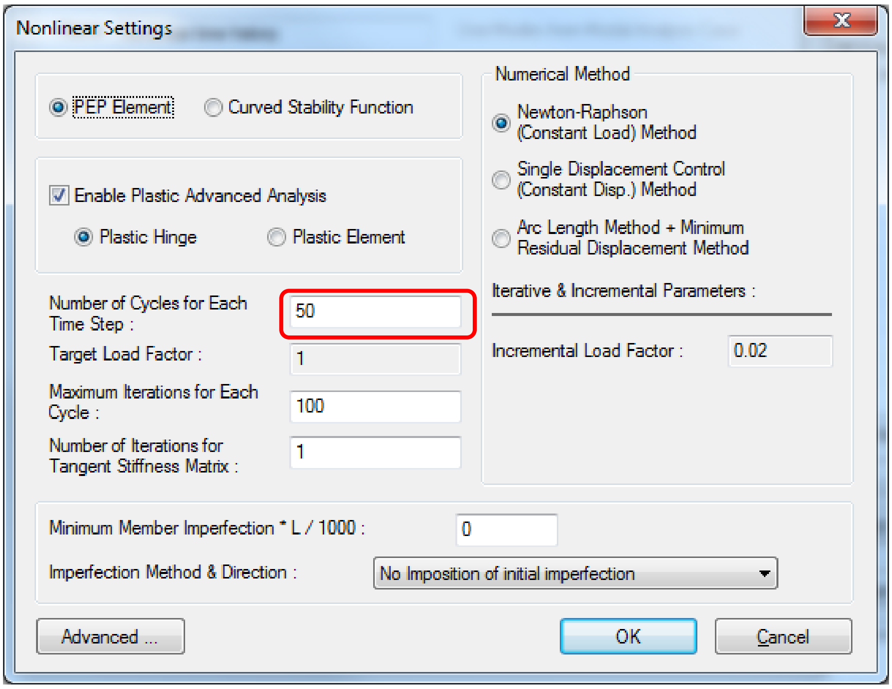
Click 'Settings' of integration method to give the parameters of Newmark method as below:



Click 'Settings' of damping to give the parameters of 'Rayleigh Damping' as below:



Click ‘Advanced Settings’ to set the parameters for nonlinear solution as below:

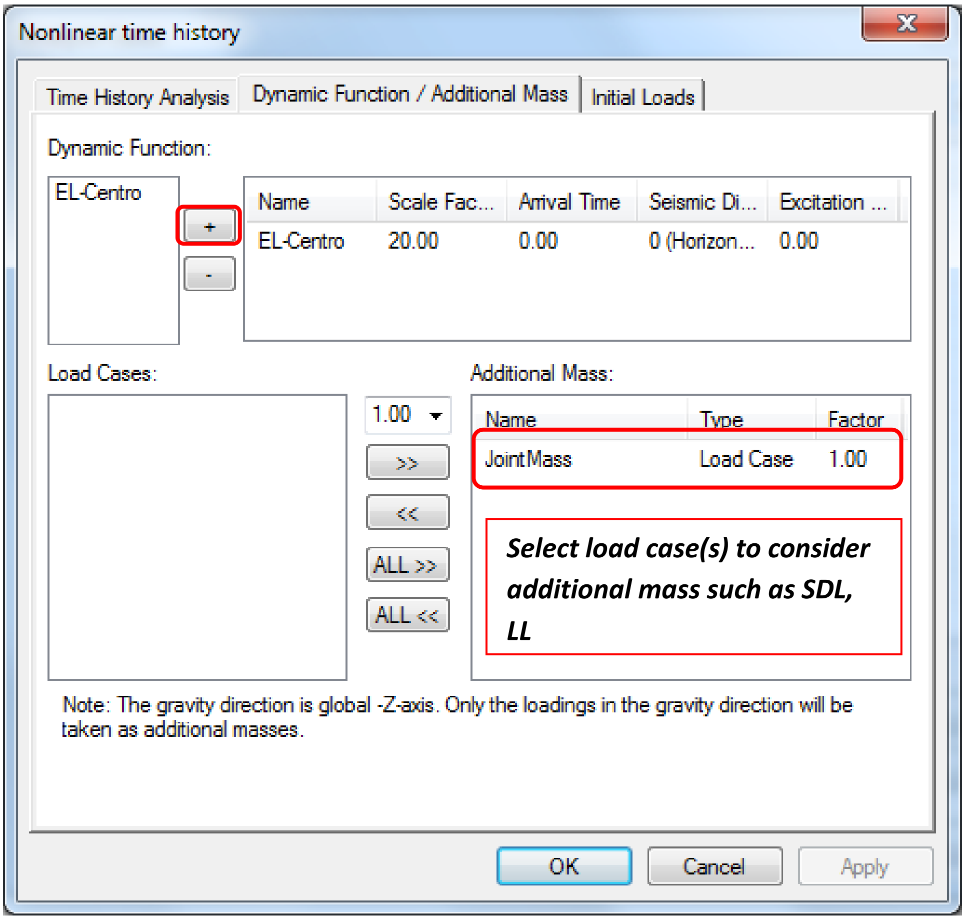


Noted that the Newton-Raphson method is used for the nonlinear incremental-iterative solution when performing a time history analysis. In some cases the structural behaviour may be highly nonlinear and therefore several cycles in each time step are needed.

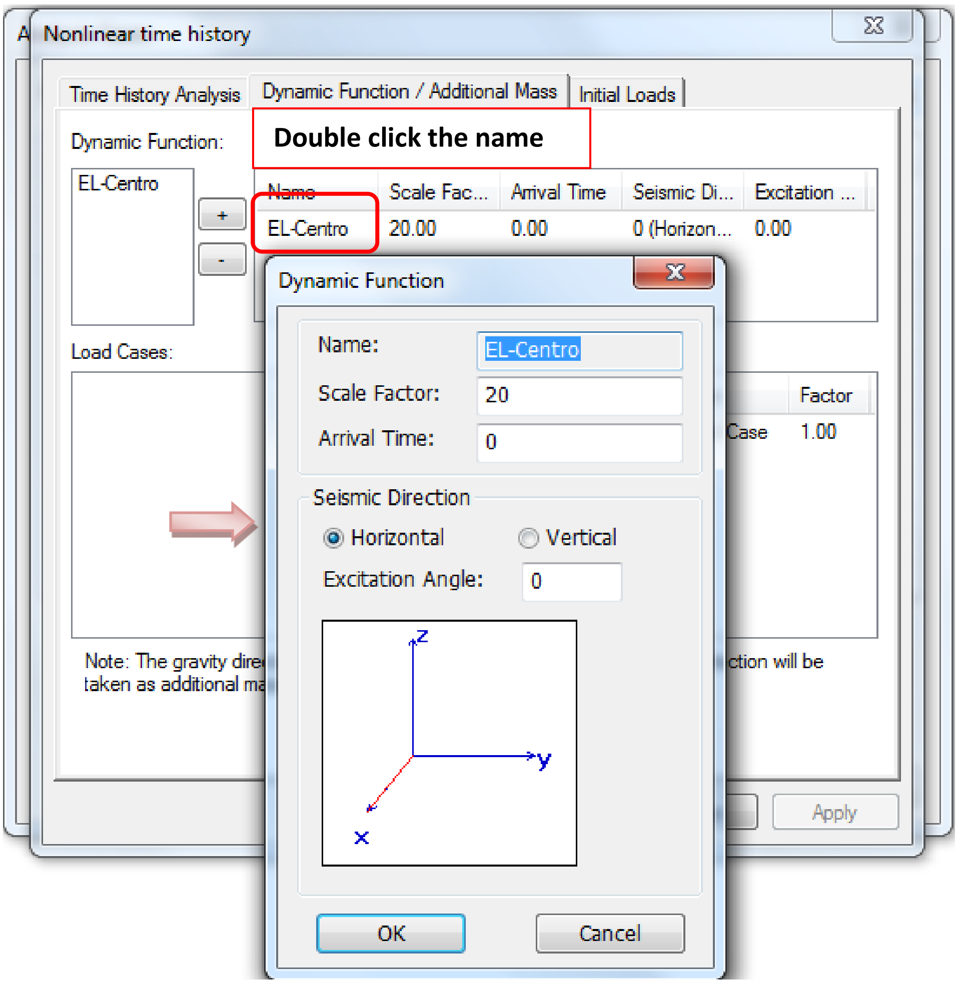
Click ‘Dynamic Function/Additional Mass’ tab to specify one or more than earthquake records as seismic input and the additional mass may need to be considered here.

Select one of dynamic functions and click ‘>>’ to add it as seismic input for this analysis case;

Select load case(s) and click ‘>>’ to add it as additional mass for this analysis case. Noted that the structural self-weight has been automatically included.



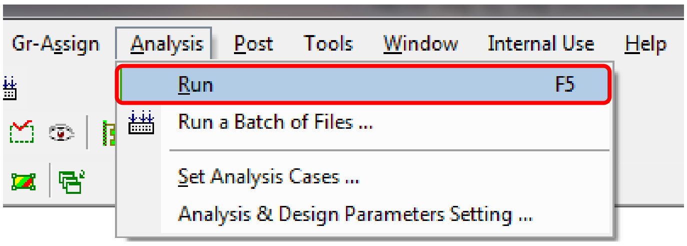
Double click the name of dynamic function and a dialog will pop up to set the 'Scale Factor', 'Arrival Time' and the seismic direction as below:



Click ‘Initial Loads’ tab to consider static loads such as dead loads before earthquake attack.

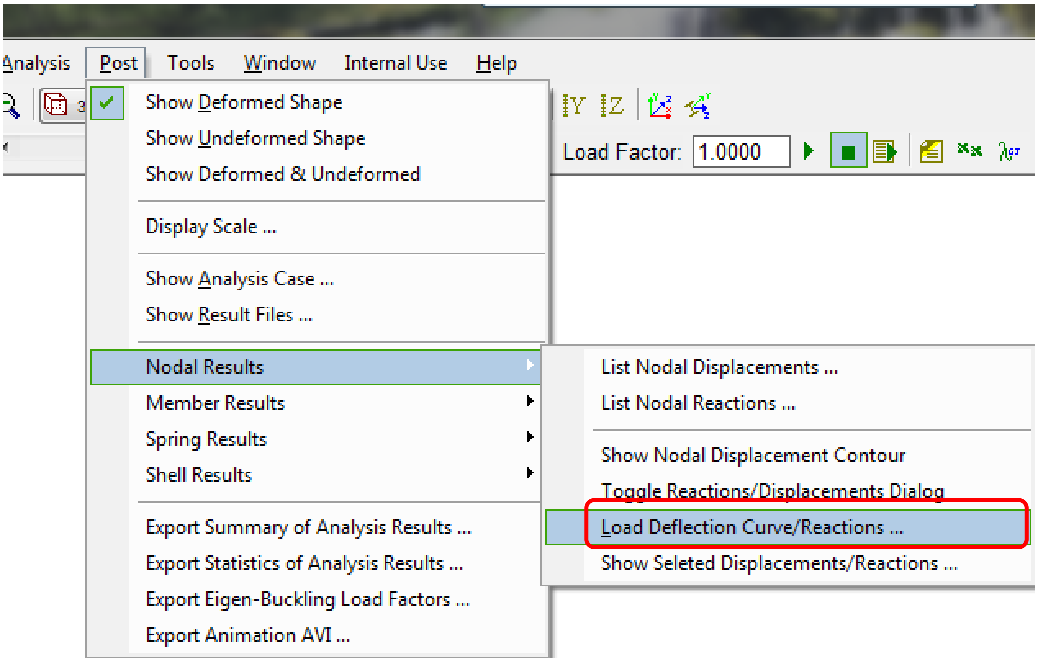
5.4 Step 4 – Run time history analysis case(s)

Click the ***Analysis*** > ***Run*** or press **F5** key to run all analysis cases;

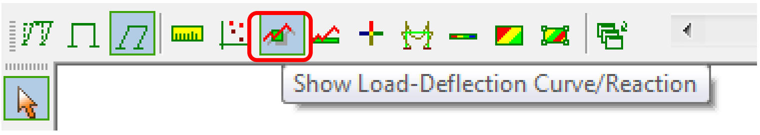


5.5 Step 5 – View result: nodal displacements and base shear

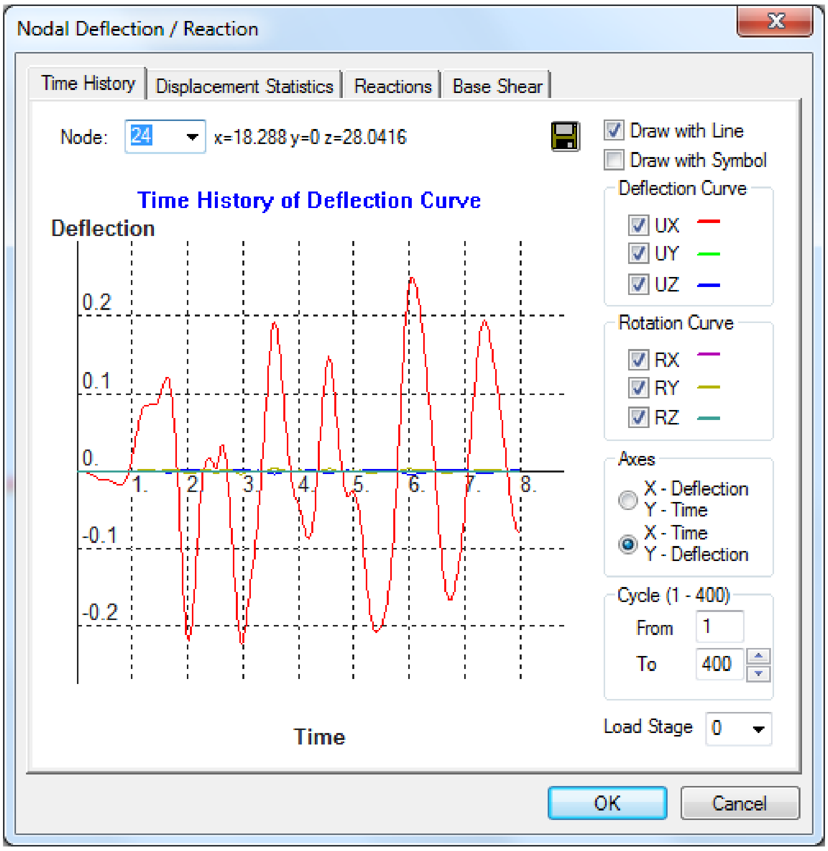
Click the ***Post*** > ***Nodal Results***> ***Load Deflection Curve/Reactions …*** ;



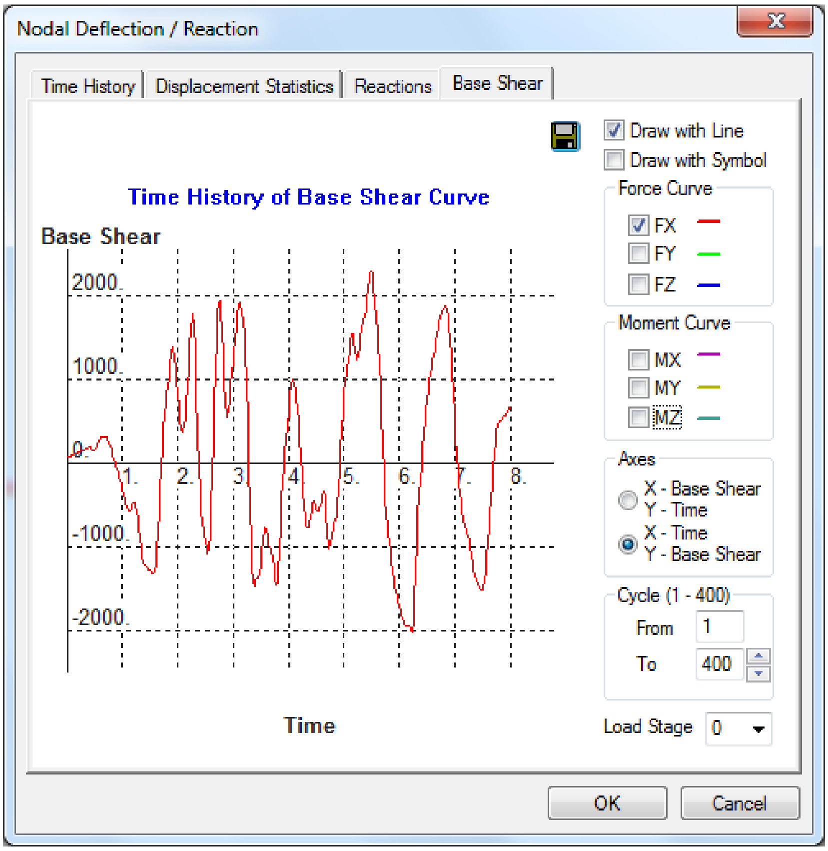
or press the button of post toolbar as below;



Select node to observe the time history of nodal displacement(s);

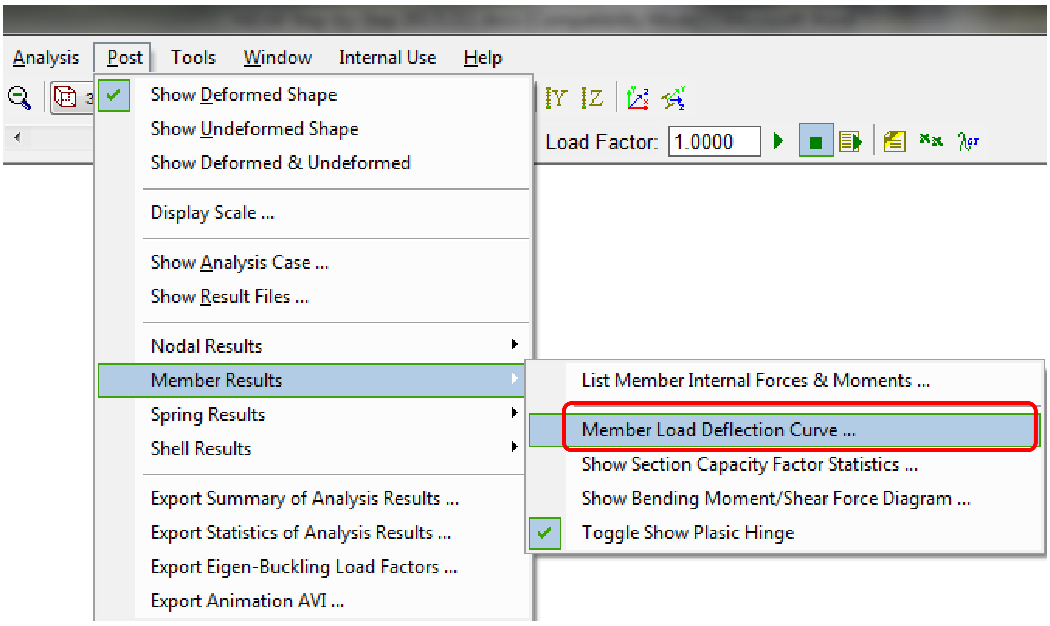


Click 'Base Shear' tab to observe the time history of base shear in X, Y or Z direction;



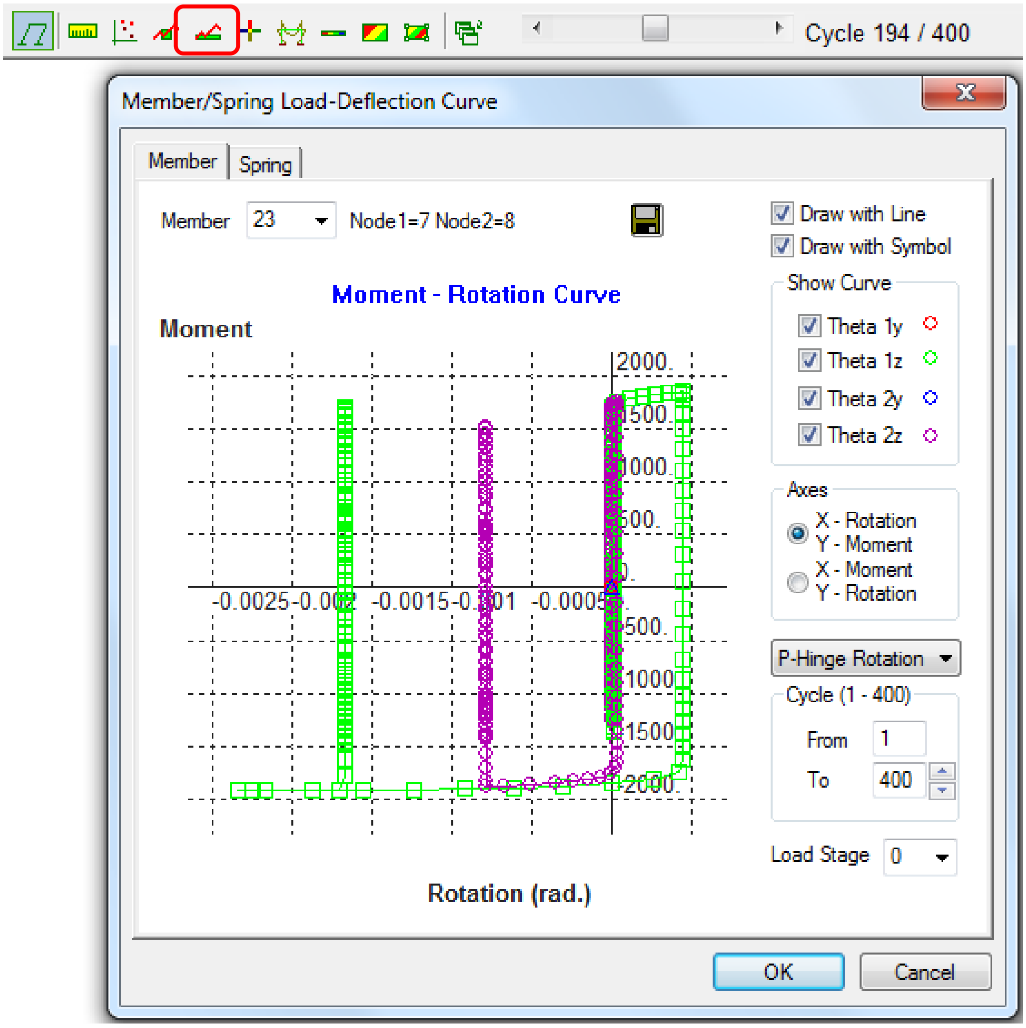
5.6 Step 6 – View result: plastic hinge rotation of all time steps

Click the ***Post*** > ***Member Results***> ***Member Load Deflection Curve…*** ;



or press the button of post toolbar as below;

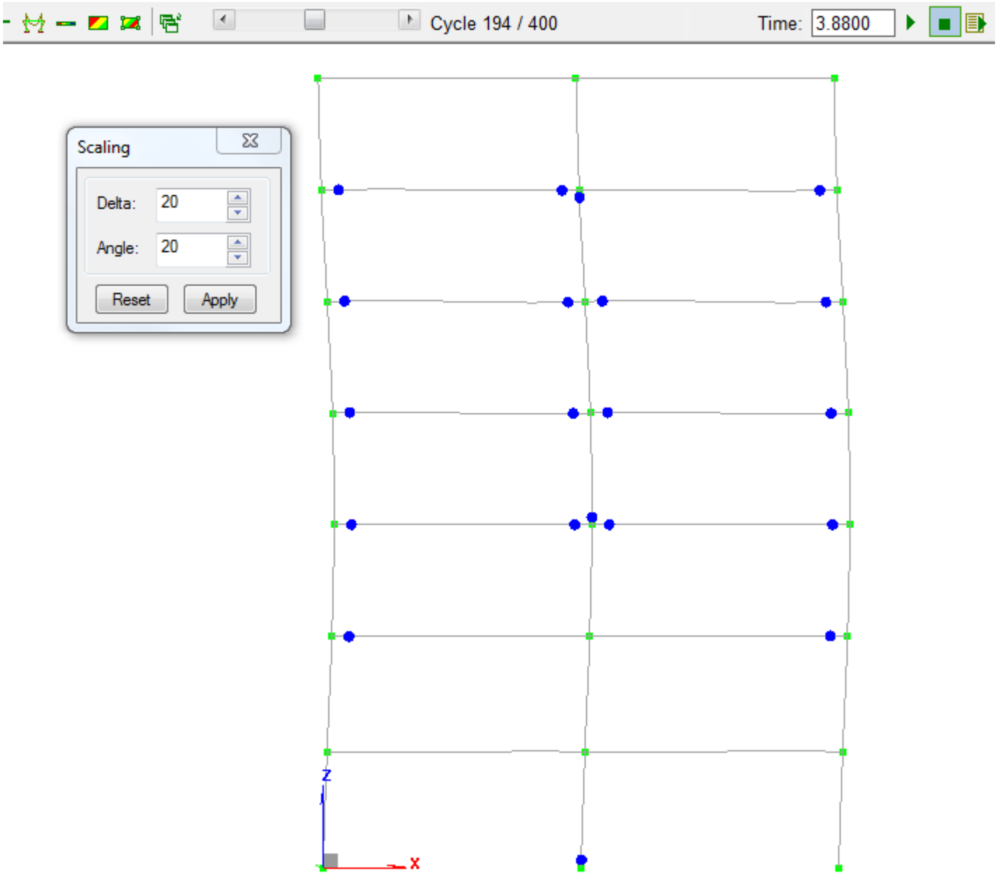
Select member to observe the time history of plastic hinge rotation;



5.7 Step 7 – View result: plastic hinge location

Plastic hinge (s) may be formed in different time step (load cycle);

Plastic hinge (s) will be plotted on screen with dot marker as below:



To toggle show plastic hinge (s) on screen click the ***Post* > *Member Results*> *Toggle Show Plastic Hinge***

5.8 Step 8 – View and export results

The procedures for viewing and exporting analysis results can be referred to tutorial 1 to 3.