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

Copyright © 2013 Professor S.L. Chan. All Rights Reserved.

No portion of this document may be reproduced – mechanically, electronically, or by any other means, including photo copying – without written permission of Prof. SL Chan.

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. TU	TORIAL 1 - SIMPLE BENCHMARK EXAMPLE FOR TESTING OF SOFTWAR	E - 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	
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. TI	TORIAL 2 – A BRACED AND AN UNBRACED PORTAL – SECOND ORDER A	NALYSIS
AND EI	LASTIC CRITICAL LOAD FACTOR ANALYSIS	
2.1		22
2.1	STEP 2 CREATE A NEW PROJECT FUE	
2.2	STEP 2 - CREATE A NEW PROJECT FILE	25 25
2.5	STEP 4 ADD NODES	
2.4	STEP 5 ADD SECTIONS	
2.5	STEP 5 - ADD SECTIONS	27 22
2.0	STEP $7 - ADD$ MEMBERS	
2.7	STED 8 - SET VICIDI E/INVICIDI E	
2.0	STEP $Q = \Delta DD I OAD CASES$	
2.9	STEP 10 – ADD LOADINGS	38
2.10	STEP 11 – SET VISIBILITY OF LOADINGS	39
2.11	STEP 12 – SET AND RUN ANALYSIS CASES	40
2.12	STEP 12 – VIEW RESULT: EIGEN-BUCKLING MODE	
2.14	STEP 14 – VIEW RESULT: NONLINEAR ANALYSIS	
2.15	STEP 15 – VIEW RESULT: SHOW ANALYSIS PROCEDURE OF NONLINEAR ANALYSIS	
2.16	STEP 16 – DELETE MEMBERS FOR UNBRACED CASE	
2.17	STEP 17 – UNBRACED PORTAL FRAME	
3. TU	TORIAL 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	5 STEP 6 – COPY AND PASTE NODES	64
3.7	7 STEP 7 – ADD SECTIONS	67
3.8	STEP 8 – ADD MEMBERS	71
3.9	STEP 9 – ADD BOUNDARY CONDITIONS	75
3.1	0 STEP 10 – SET VISIBLE/INVISIBLE	77
3.1	1 STEP 11 – ROTATE MEMBER LOCAL AXIS	78
3.1	2 STEP 12 – ADD LOAD CASES	83
3.1	3 STEP 13 – ADD LOADINGS	85
3.1	4 STEP 14 – SET VISIBILITY OF LOADINGS	91
3.1	5 STEP 15 – ADD LOAD COMBINATIONS	92
3.1	6 STEP 16 – SET AND RUN ANALYSIS CASES	95
3.1	17 STEP 17 – VIEW RESULT: EIGEN-BUCKLING MODE	104
3.1	8 STEP 18 – VIEW RESULT: VIEW ANOTHER ANALYSIS CASE	108
3.1	9 STEP 19 – VIEW RESULT: CHECK THE STRUCTURAL ADEQUACY BY SECOND-ORDER ANALYSIS (I)	
Ex	PORT STATISTICS OF ANALYSIS RESULTS	110
3.2	20 STEP 20 – VIEW RESULT: CHECK THE STRUCTURAL ADEQUACY BY SECOND-ORDER ANALYSIS (II)	
Ex	PORT SUMMARY OF ANALYSIS RESULTS	114
3.2	STEP 21 – VIEW RESULT: CHECK THE STRUCTURAL ADEQUACY BY SECOND-ORDER ANALYSIS (III) S	SHOW
SE	CTION CAPACITY FACTOR STATISTICS	116
4.	TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS	124
4. 4.1	TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS Step 1 – Build the structural model	 124 124
4. 4.1 4.2	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. 	124 124 124
4. 4.1 4.2 4.3	TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS STEP 1 – BUILD THE STRUCTURAL MODEL STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION STEP 3 – DEFINE A MODAL ANALYSIS	124 124 124 126
4. 4.1 4.2 4.3 4.4	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS	124 124 124 126 128
4.1 4.2 4.3 4.4 4.5	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES 	124 124 124 126 128 129
4.1 4.2 4.3 4.4 4.5 4.6	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER 	124 124 124 126 128 129
4.1 4.2 4.3 4.4 4.5 4.6 AC	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS 	124 124 124 126 128 129 131
4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES. STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS. TOTONS 	124 124 126 128 129 131 134
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5.	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES. STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS. TUTORIAL 5 – TIME HISTORY ANALYSIS	124 124 124 126 128 129 131 134 136
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5.	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS. TUTORIAL 5 – TIME HISTORY ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. 	124 124 124 126 128 129 131 134 134 137
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5.	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES. STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS. TUTORIAL 5 – TIME HISTORY ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE ONE OR MORE THAN ONE TIME HISTORY FUNCTIONS. 	124 124 124 126 128 129 131 134 136 137 137
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5. 5.1 5.2 5.3	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. STEP 3 – DEFINE A MODAL ANALYSIS. STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER TIONS. STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS. STEP 1 – BUILD THE STRUCTURAL MODEL. STEP 2 – DEFINE ONE OR MORE THAN ONE TIME HISTORY FUNCTIONS. STEP 3 – DEFINE A TIME HISTORY ANALYSIS CASE 	124 124 124 126 128 129 131 134 134 137 137 138
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5. 5.1 5.2 5.3 5.4	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS	124 124 124 126 128 129 131 134 134 137 137 138 142
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5. 5.1 5.2 5.3 5.4 5.5	TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS STEP 1 – BUILD THE STRUCTURAL MODEL 2 STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION. 3 STEP 3 – DEFINE A MODAL ANALYSIS 4 STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. 5 STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES 5 STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER 7 STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS 7 STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS 8 STEP 1 – BUILD THE STRUCTURAL MODEL 9 STEP 2 – DEFINE ONE OR MORE THAN ONE TIME HISTORY FUNCTIONS. 9 STEP 3 – DEFINE A TIME HISTORY ANALYSIS CASE 9 STEP 3 – DEFINE A TIME HISTORY ANALYSIS CASE 9 STEP 4 – RUN TIME HISTORY ANALYSIS CASE(S). 5 STEP 5 – VIEW RESULT: NODAL DISPLACEMENTS AND BASE SHEAR.	124 124 124 126 128 129 131 131 134 137 137 138 142 142
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5. 5.1 5.2 5.3 5.4 5.5 5.6	TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS STEP 1 – BUILD THE STRUCTURAL MODEL 2 STEP 2 – DEFINE A RESPONSE SPECTRUM FUNCTION 3 STEP 3 – DEFINE A MODAL ANALYSIS 4 STEP 4 – CREATE "SINGLE DIRECTION" MRSA CASES. 5 STEP 5 – CREATE "DIRECTIONAL COMBINATION" MRSA CASES 5 STEP 6 – CREATE "LOAD COMBINATION" CONSIDERING BOTH EARTHQUAKE ACTION AND OTHER 7 STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS 7 STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS 7 STEP 7 – RUN ALL ANALYSIS CASES AND VIEW RESULTS 7 STEP 7 – DEFINE ONE OR MORE THAN ONE TIME HISTORY FUNCTIONS. 8 STEP 2 – DEFINE ONE OR MORE THAN ONE TIME HISTORY FUNCTIONS. 9 STEP 3 – DEFINE A TIME HISTORY ANALYSIS CASE 4 STEP 4 – RUN TIME HISTORY ANALYSIS CASE(S). 5 STEP 5 – VIEW RESULT: NODAL DISPLACEMENTS AND BASE SHEAR. 5 STEP 6 – VIEW RESULT: PLASTIC HINGE ROTATION OF ALL TIME STEPS	124 124 124 126 128 129 131 134 134 137 137 138 142 142 144
4. 4.1 4.2 4.3 4.4 4.5 4.6 AC 4.7 5. 5.1 5.2 5.3 5.4 5.5 5.6 5.7	 TUTORIAL 4 – MODAL RESPONSE SPECTRUM ANALYSIS	124 124 124 126 128 129 131 134 137 137 137 138 142 144 145

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'.

New Projec	t	$\overline{\mathbf{X}}$
Title:		
Tutorial 1		
		<u> </u>
Gravity Dire	ction: -Y	~
-Force Uni	t	Length Unit
ΟN	🔿 kgf	⊖mm ⊙m
⊙ kN	◯ tons	⊖ cm
	ОК	Cancel

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);

General Settings			
General Active DC)Fs		
Title:			
Tutorial 1			*
			-
Floor Stiffness:	0	Gravity Direction:	-Y •
Steel Design:	HKSC (2011)	-	
Concrete Design:	HKSC (2011) BS5950 (2000)		
Force Unit	Eurocode3 (20 GB50017 (200 AISC-LRFD (20	05) 3))10)	
⊚ kN ⊚ to	ns	cm	
Advanced			
	ОК	Cancel	Apply

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

General Settings
General Active DOFs
Degree of Freedom (DOF)
Translational: 🗹 UX 🗹 UY 🔽 UZ
Rotational: RX RY RZ
Fast DOFs
Active All X-Y Plane Truss
OK Cancel Apply

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

1.3 Step 3 – Create material

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

<u>V</u> iew	Construct	Gr-Assign	<u>A</u> nalysis	Post	Тос
NO.	New N	<u>M</u> aterial			ſ
	New Frame Section				•
	New S	Shell Sections.			

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;

Material	
General	
Name: S275	Import
Type: Steel	•
Poisson's Ratio:	0.3
 Shear Modulus of Elasticity (kN/m2): 	7.8846e+007
Young's Modulus of Elasticity (kN/m2):	2.0500e+008
Density (kN/m3):	77
Coefficient of Thermal Expansion:	1.4000e-005
Yield Stress, fy (kN/m2):	2.7500e+005
	OK Cancel

Click OK to continue.

1.4 Step 4 – Add nodes

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

<u>V</u> iew	<u>C</u> on	struct	Gr-Assign	<u>A</u> nalysis	<u>P</u> ost	Тос
0		New	<u>M</u> aterial			•
		New	Frame Section	I		•
	New		Shell Sections.			
		New	Nodes			
		New	M <u>e</u> mbers			

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

Properties	ųΧ
🔹 🖈 🖻 🖬 🦀	
Add Node Jaterials	
1	

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

New Node	New Node
No.: 1 X: 0 Y: 0 Z: 0	 No.: 2 X: 5 Y: 0 Z: 0
Offset to Node <u>Apply</u> <u>C</u> lose	Offset to Node 1 Apply Close

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;

	Details	ά×
	Attributes	Values
	Item Type	Node
	No.	2
	X	5.000
	Y	0.000
	Z	0.000
	Boundary-UX	Free
Υ.	Boundary-UY	Free
- V	 Boundary-UZ 	Free
<u>≮</u> →^	Boundary-RX	Free
	Boundary-RY	Free
	Boundary-RZ	Free
	Loadings	0
	Num. of Members	0
	Num. of Springs	0
	Num. of Floors	0
	Num. of K_Nodes	0
	Num. of Shells	0
	<	>

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;

Node	<u> </u>
Coordinates	Boundary Support Spring Loading
	No.: 2 X: 5
	Y: 0
	2.0
	OK Cancel Apply

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>;

Properties 🛛 📮 🗙	
+ 🔎 🐒 🖻 🖹 🎢	
🖃 🖧 Untitled	
🕀 🗁 Materials	
🖨 🗁 Frame Sections	
New Frame Sections	
Import Sections	

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

New Sections		x
General		
Name: Sect 2	Import	<u>У</u> ,
Type: 4. CHS[Pipe]	▼ Customize	
Material: S355	-	, , , , , , , , , , , , , , , , , , ,
- Section Properties (Analysis)		
Cross Sectional Area (A):	0.0000e+000	
Shear Area Correction Fact	or: 0	Dimensions
Second Moment of Area (ly): 0.0000e+000	B: 0
Second Moment of Area (Iz): 0.0000e+000	D: 0
Torsional Constant (J):	0.0000e+000	Tf: 0
Section Modulus (Design)		tw: 0
About y-axis (Zy): 0.0	0000e+000 Use	B2: 0
About z-axis (Zz): 0.0	Tf2: 0	
About y-axis (Sy): 0.0	ds: 0	
About z-axis (Sz): 0.0	0000e+000	Recalculate
Rolled Section Fab	pricated Section Cold fo	med 🚺
Suppress Frame Ligen-Imper	rection :	No
Imperfection along Minor y-a	kis : L/500 🔻 Elastic	c Plastic
along Major z-ax	kis: L/500 👻	
Stress Type : Square-root of	of Stress	Advanced
	OK Cance	el Apply

Select cross sections under different categories, country or shape;

U.K. 🗸	CHS/Cir. 🗸
U.K.	Box/Rect.
Japan	CHS/Cir.
Europe	VH
U.S.	Channel
China	Angle

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

J.K. 🗸]	CHS/Cir.	~			
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	1
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	'n
CHS76.1x2.9 [^]	7.61	N/A	0.29	N/A	6.67	1
CHS60.3x5.0~	6.03	N/A	0.50	N/A	8.69	L
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	
د]					>	1

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;

 $P - \delta$ 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.

Section					×
Gene	ral Membe	ers			
	Name:	O88.9x3.2∼	Import	1 2	y ,
	Type:	4. CHS[Pipe]	▼ Customize		
	Material:	S275	-	- ((<mark>}</mark> z
Sec	tion Proper	ties (Analysis)			
0	ross Sectio	nal Area (A):	8.6200e-004		
S	ihear Area (Correction Factor:	0	Dimen	isions
S	econd Mon	nent of Area (ly):	7.9200e-007	B:	0.0889
S	econd Mon	nent of Area (Iz):	7.9200e-007	D:	0
1	orsional Co	nstant (J):	1.5800e-006	Tf:	0.0032
Sec	tion Modulu	us (Design)		tw:	0
(A	bout y-axis	(Zy): 1.7800e	-005 Use	B2:	0
A	bout z-axis	(Zz): 1.7800e	+-005 Elastic(Z)	Tf2:	0
A N	bout y-axis	(Sy): 2.1360e	-005 O Use Plastic(S)	ds:	0
	bout z-axis	(Sz): 2.1360e	-005	Re	calculate
	Rolled Sec	tion 🔘 Fabricate	ed Section 🔘 Cold+	formed	
Su	ippress Fran	ne Eigen-Imperfectio	n: 🔘 Yes	No	(1)
Im	perfection a	long Minory-axis :	L/500 - Flas	tic	lastic
	a	long Major z-axis :	L/500 -		100110
St	ress Type :	Square-root of Stre	ess 🔻	A	dvanced
			OK Can	cel	Apply

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;

Member Proper	ties	X
Section:	CHS88.9x3.2~	~
End Condition:	Both Rigid	~
	Both Rigid Both Pin	
	One Pin one Rigid(P1R2) One Rigid one Pin(P2R1)	

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.

Member Proper	ties	X
Section:	CHS88.9x3.2~	~
End Condition: Both Rigid		~

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;

	Node
	Displacement: UX
	UY 🔘 Free 💿 Restraint
•	UZ 🔿 Free 💿 Restraint
_	Rotation: RX 💿 Free 🔿 Restraint
4	RY 💿 Free 🔿 Restraint
	RZ 💿 Free 🔿 Restraint
	Fast Restraints
	OK Cancel Apply

Click *Apply* and *OK* to continue;

or select the node to be restrained;

__1____2

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

Properties	ά×
- > 🖍 🖻 😭	*
🖃 🖧 Untitled	
🕀 🗁 Materials	Boundary Conditions

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

Node			X
General Boundary Suppo	rt Spring	Loading	
🝌 Displacement: UX	◯ Free	 Restraint 	
UY	◯ Free	Restraint	
UZ	◯ Free	 Restraint 	
Rotation: RX	◯ Free	 Restraint 	
RY	◯ Free	 Restraint 	
RZ	◯ Free	 Restraint 	
Fast Restraints	•		
ОК		Cancel Apply	

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;

Visible/Invisible Settings		
Area	Member / Cable	Node
Area	🛹 🗹 Member	Node
Number	Cable	No. 🔽 Number
	🎐 🗹 Number	Boundary Condition
Floor	Material Number	Support Spring
Number	Material Name	🕰 🗌 Local Axis
Normal Direction	Section Number	-Member / Cable Color-
One/Two-Way	Section Name	By Section
_Shell	Length	O By Material
Shell	🙀 🗌 Local Axis	🔾 Uniform
Number	Bending Stiffness	Show Number
Normal Direction	- about y-y axis	Selected Only
One/Two-Way	about z-z axis	Others
	Axial Stiffness	Me Spring Element
	Torsional Stiffness	Global Axis
	<u>o</u> k	<u>Cancel</u> <u>Apply</u>

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;

Properties		
* 2 2	📑 🖹 🔏	
😑 🖧 Untit	led Ma Add Load Case	

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

New Load Cas	ies 🔀
Name : Type : No. : Factor :	Axial Load Live Load 5 1.0 Auto Self Weight
	Show Loadings on Structure Show Values OK Cancel Apply

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;

Loading Prope	erties		
Load Case: Load Type:	Axial Load		~
Force FX 10 FY 0		Moment MX 0 MY 0	
Axis	Global	O Local	
		<u>A</u> pply	<u>C</u> lose

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;

Show Loading	
Axial Load	Load Type Joint Load Settlement Trapezoidal Load Member Pressure Point Load Cable Force Pressure on Arrow
Select All ✓ Show Values	Apply Close

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;

Analysis Cases					
Show	ALL	~	Num. of	items: 0/0	\$
Nam	e		ID	Туре	Run
Edit Add Re Use	name	Linear Analysis Nonlinear Analysis Modal Analysis Eigen-Buckling Analysis Response Spectrum Analysis Time History Analysis		Set Run Flag Run / Not Ru Ali Not Run Al Run Now	in I Run OK

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;

lonlinear		×
Second-Order Analysis Ap	plied Loads Construction	on Sequence
Name: Nonlinear Type: Second-order And	alysis + Design 🛛 🔻	Numerical Method Newton-Raphson (Constant Load) Method
PEP Element	urved Stability Function	Single Displacement Control (Constant Disp.) Method
Enable Plastic Advanced Analysis	 Plastic Element Plastic Hinge 	Arc Length Method + Minimum Residual Displacement Method Iterative & Incremental Parameters :
Total Load Cycles :	120	
Target Load Factor : Maximum Iterations for each Load Cycle : Number of Iterations for Tangent Stiffness Matrix :	100	
Minimum Member Imperfe Imperfection Method & Di Advanced	rection * L / 1000 : 1 rection : Displacem	ent : About both principal axis 🔹
		OK Cancel Apply

Apply 'Load Case – Axial Load' to this analysis case;

econd-Order Analysis Applie	ed Loads Cons	truction Sequence		
General Load Cases		Loads Applied		
Type: Load Case	*	Name	Туре	Factor
	1.00 ▼	Axial Load	Load Case	1.00
		Enable Load/Co	nstruction Stage	

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

nalysis Cases			
Show ALL 🔽 N	lum. of	items: 1/1	∱ 4
Name	ID	Туре	Run
NONLINEAR	1	NONLINEAR	YES
Edit		Set Run Flag	
Add Modify Duplicate		Run / Not Ru	un 🔵
Rename Delete		All Not Run A	ll Run
Use Processors: 1		Run Now	<u>о</u> к

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;

Analysis complete!
Start Time : 2011-11-22 15:38:22 Time Elapsed : 00:00:13 Process 1
KCYC= 120 ITER.= 2 NORM detU/U,resF/FCE = 0.6383E-02 0.6047E-04
*** THE SYSTEM IS STABLE ***
LOAD STAGE = 0 ; LOAD FACTOR = 1.2000 , 0.0000 inc.
KCYC= 120 ITER.= 3 NORM detU/U,resF/FCE = 0.2526E-03 0.2310E-05
[1] Beam-column elements have been completed
Date : 20111122 Time : 15:38:34 Zone : +0800
Show Trace OK Cancel

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

📅 Nida - [Untitled]			
Eile Edit Select View Construct Gr-A	<u>s</u> sign <u>A</u> nalysis <u>P</u> ost Tools <u>W</u> indow <u>H</u> elp)	
	👯 🔍 Q 📴 30 🔍 🚸 🔶 rfl 31		
//] [] /7 📟 🖄 🛩 🕂 🙀 🗕 🗖	- 🖻 ≤ 🚺 🔹 DIMP	Load Factor: 0.0050	▶ ■ ● ≤ ×× %

1.13 Step 13 – View result: structural deformation

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

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp	
ର୍ ପ୍ 🛛		Show De	formed Shap	be	7
Show Undeformed Shape				nape	
	× .	Show De	formed & Ur	ndeformed	-

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

Show Undeformed and Deformed Shape

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



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

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

/sis E	<u>P</u> ost Tools <u>W</u> indow <u>H</u> elp	
2	Show Deformed Shape Show Undeformed Shape Show Deformed & Undeformed Display Scale Show Analysis Case Show Result Files	 ▶ ▲ IY IZ 2 3 Load Factor: 1.0000 ▶ ■ ●
	Nodal Results	Reactions/Displacements
	Member Results	Load Deflection Curve/Reactions
	Shell Results	Show Seleted Displacements/Reactions

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.





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
 - $P-\Delta$ 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'.

New Projec	t		
Title:			
Tutorial 2 I	Braced		
Gravity Dire	ction: -Y	~	
-Force Uni	t	CLength Unit	t
ΟN	🔿 kgf	◯mm	⊙ m
⊙ kN	◯ tons	<mark>⊖ cm</mark>	
	ОК		Cancel

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);

General Settings		
General Active DO	DFs	
Title:		
Tutorial 2 Braced	~ ~	
Floor Stiffness:	0 Gravity Direction: -Y 🔻	
Steel Design:	HKSC (2011) -	
Concrete Design:	HKSC (2011) BS5950 (2000) Eurocode3 (2005) GB50017 (2003) PAISC-LRFD (2010)	
Advanced	ns Cm	
	OK Cancel Apply	

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

General Settings	X
General Active DOFs	
Degree of Freedom (DOF)	
Translational: 🗹 UX 🔽 UY 🔲 UZ	
Rotational: RX RY RZ	
Fast DOFs	
Active All X-Y Plane Truss	
OK Cancel Apply	

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

2.3 Step 3 – Create material

Click the *Construct > New Material* command,

<u>V</u> iew	Construct	Gr-A <u>s</u> sign	<u>A</u> nalysis	<u>P</u> ost	Тос
0	New <u>M</u> aterial				ſ
	New I	New Frame Section			
	New 9	New Shell Sections			

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;

New Materials	
General	
Name: S275	Import
 Poisson's Ratio: Shear Modulus of Elasticity (kN/m2): 	0.3 7.8846e+007
Young's Modulus of Elasticity (kN/m2):	2.0500e+008
Density (kN/m3): Coefficient of Thermal Expansion:	77 1.4000e-005
Yield Stress, fy (kN/m2):	2.7500e+005
ОК	Cancel Apply

Click OK to continue.

2.4 Step 4 – Add nodes

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

<u>V</u> iew	Construct	Gr-Assign	<u>A</u> nalysis	<u>P</u> ost	Too
ß	New	Material			Ē
-	New	Frame Section	I		• =
	New	Shell Sections.			
	New	Nodes			
	New	Members			

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

Properties	$^{\circ}$ ×	
💽 🖍 😰 😰 🖌		
Add Node laterials		



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

New Node	New Node 🛛 🔀
• No.: 1	No.: 5
X: 0	X: 15
Y: 0	Y: 5
Z: 0	Z: 0
Offset to Node	Offset to Node
Apply <u>C</u> lose	Apply <u>C</u> lose

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;

Node 🛛 🔀
General Boundary Support Spring Loading
No.: 5
X: 15
Y: 5
Z: 0
OK Cancel Apply

Click OK to continue.

2.5 Step 5 – Add sections

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

<u>V</u> iew	<u>C</u> on	struct	Gr-A <u>s</u> sign	<u>A</u> nalysis	Post	Т	ools	<u>W</u> indow	<u>H</u> elp	
(m)		New I	<u>M</u> aterial			_	~	& & f	7 3D	(9)
_		New F	Frame Section			۲		Import Sec	tions	1
		New S	Shell Sections.					New Section	ons	

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>;

Properties 🛛 📮 🗙		
+ 🔎 🐒 🖻 🖹 🎢		
🖃 🖧 Untitled		
🕀 🗁 Materials		
🚊 🛅 Frame Sections		
New Frame Sections		
Import Sections		

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

New Sections			×	
General				
Name:	Sect 2	Import	<u>У</u> д	
Type:	4. CHS[Pipe]	▼ C <u>u</u> stomize		
Material:	S355	•		
-Section Propert	ies (Analysis)			
Cross Section	nal Area (A):	0.0000e+000		
Shear Area C	Correction Factor:	0	Dimensions	
Second Morr	ent of Area (ly):	0.0000e+000	B: 0	
Second Morr	ent of Area (Iz):	0.0000e+000	D: 0	
Torsional Cor	nstant (J):	0.0000e+000	Tf: 0	
Section Modulu	ıs (Design)		tw: 0	
About y-axis	(Zy): 0.0000e	+000 Use	B2: 0	
About z-axis	(Zz): 0.0000e	+000 Elastic(Z)	Tf2: 0	
About y-axis	(Sy): 0.0000e	+000 Ose Plastic(S)	ds: 0	
About z-axis	(Sz): 0.0000e	+000	Recalculate	
Rolled Sec	tion 🔘 Fabricate	ed Section 🔘 Cold-f	ormed 🕕	
Suppress Fram	ne Eigen-Imperfectio	n : 💿 Yes	No	
Imperfection a	long Minory-axis :	L/500 - Elasi	tic Plastic	
along Major z-axis : L/500 -				
Stress Type :	Square-root of Stre	ess 🔻	Advanced	
		OK Can	cel Apply	

Select cross sections under different categories, country or shape;

U.K. 🗸	VH 🗸
U.K.	Box/Rect.
Japan	CHS/Cir.
Europe	VH
U.S.	Channel
China	Angle

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

J.K. 💌		VH	~		
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
<]	1111				>

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;

Section	×
General Members	
Name: UC356x368x153# Import Type: 5. I/H-section ▼ Customize	*
Material: S275	₽ >Z
Cross Sectional Area (A): 1.9500e-002	
Shear Area Correction Factor: 0	Dimensions
Second Moment of Area (ly): 1.7600e-004	B: 0.3705
Second Moment of Area (Iz): 4.8500e-004 Torsional Constant (I): 2.5100e-006	D: 0.362
Section Modulus (Design)	tw: 0.0123
About y-axis (Zy): 9.4800e-004 Use	B2: 0
About z-axis (Zz): 2.6800e-003 Use	lf2: 0 ds: 0
About y-axis (Sy). Plastic(S) About z-axis (Sz): 2.9700e-003	Recalculate
 Rolled Section ○ Fabricated Section ○ Cold Suppress Frame Eigen-Imperfection : ○ Yes Imperfection along Minor y-axis : along Major z-axis : L/300 ▼ 	formed i
Stress Type : Direct Sum of Stress	Advanced
ОК Саг	Apply

 $P - \delta$ 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;

J.K. 💌	1	Channel	~		
Name	B(cm)	D(cm)	Tf(cm)	tw(cm)	Area(
[430x100x64	10.00	43.00	1.90	1.10	83.40
[380x100x54	10.00	38.00	1.75	0.95	70.10
[300x100x46	10.00	30.00	1.65	0.90	58.90
[300x90x41	9.00	30.00	1.55	0.90	53.30
[260x90x35	9.00	26.00	1.40	0.80	45.40
[260x75x28	7.50	26.00	1.20	0.70	35.90
[230x90x32	9.00	23.00	1.40	0.75	41.60
[230x75x26	7.50	23.00	1.25	0.65	33.20
[200x90x30	9.00	20.00	1.40	0.70	37.90
[200x75x23	7.50	20.00	1.25	0.60	30.40
[180x90x26	9.00	18.00	1.25	0.65	34.10
[180x75x20	7.50	18.00	1.05	0.60	26.60
[150x90x24	9.00	15.00	1.20	0.65	30.40
[150x75x18	7.50	15.00	1.00	0.55	22.80
[125x65x15#	6.50	12.50	0.95	0.55	19.00
<					>
-					

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

General Memb	bers			
Name:	[230x90x32	Import	Y _A T	
Type:	7. Channel	▼ Customize		
Material:	S275	S275 🔻		
Section Prope	rties (Analysis)			
Cross Secti	onal Area (A):	4.1000e-003		
Shear Area	Correction Factor:	0	Dimensions	
Second Mo	ment of Area (ly):	3.3400e-006	B: 0.09	
Second Mo	ment of Area (Iz):	3.5180e-005	D: 0.23	
Torsional C	onstant (J):	1.9300e-007	Tf: 0.014	
Section Modu	lus (Design)		tw: 0.0075	
About y-axis	s (Zy): 5.5000	De-005 👝 Use	B2: 0	
About z-axis	s (Zz): 3.0600	De-004 Elastic(Z)	Tf2: 0	
About y-axis	s (Sy): 9.8900	De-005 Our Use	ds: 0	
About z-axis	s (Sz): 3.5500	De-004	Recalculate	
Rolled Se	ection 🔘 Fabrica	ated Section 🔘 Cold	l-formed	
Suppress Fra	me Eigen-Imperfect	tion : 💿 Yes	💿 No 🚺	
Imperfection	along Minory-axis :	L/300 V	Plastic	
	along Major z-axis :	L/300 -		
Stress Type	Direct Sum of St	ress 🔻	Advanced	
	[OK Car	Apply	

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;

Member Properties				
Section:	UC356x368x153#	~		
End Condition:	Both Rigid	~		
	Both Rigid Both Pin One Pin one Rigid(P1R2) One Rigid one Pin(P2R1)			

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,

Member Properties			
Section:	[230x90x32	~	
End Condition:	Both 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;

General boundary Support Spring Loading
🝌 Displacement: UX 🔿 Free 💿 Restrain
UY 🔿 Free 💿 Restrai
UZ 🔿 Free 💿 Restrai
Rotation: RX 💿 Free 🔿 Restraiv
RY
RZ 💿 Free 🔘 Restrai

Click *Apply* or *OK* to continue;

or select the node to be restrained;

7

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

Properties	Ф × Ф
- 2 🖬 🖻 🖻	*
🕞 🖧 Untitled	
🗄 🗁 Materials	Boundary Conditions

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

Node				
General Boundary Supp	oort Spring	Loading		
🝌 Displacement: UX	Free	Restraint		
UY	∕ ○ Free	Restraint		
UZ	C Free	 Restraint 		
Rotation: RX	• Free	Restraint		
RY	· • Free	◯ Restraint		
RZ	• Free	Restraint		
Fast Restraints				
OK Cancel Apply				

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;

Visible/Invisible Settings				
Area	_Member / Cable	Node		
Area	🛹 🗹 Member	Node		
Number	Cable	No. Vumber		
Normal Direction	Number	📥 🗹 Boundary Condition		
Floor	Material Number	👔 🔲 Support Spring		
IFI00F	Material Name	🕰 🗌 Local Axis		
Normal Direction	Section Number			
One/Two-Way	Section Name	Member / Cable Color		
Shall		By Section		
Shell	🛁 Local Axis			
Number	Y ■ Bending Stiffness about v-v axis	Show Number		
	Bending Stiffness	Selected Only		
Fill Deformed	about z-z axis	Others		
Fill Undeformed	Axial Stiffness	Spring Element		
	<u>o</u> k	Cancel Apply		

Tick 'Bending Stiffness about y-y/z-z axes' to view the end releases of member 4 and

5; or press the buttons IY IZ 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;

New Load Ca	ses	<
General		
Name :	Vertical Load	
Type :	Other 🖌	
No. :	5	
Factor :	1.0	
	Auto Self Weight	
	Show Loadings on Structure	
	Show Values	
	OK Cancel Apply	5

Click *Apply* to continue adding more load cases;

New Load Cas	ses 🛛 🔀
General	
Name :	Horizontal Load
Type :	Other 🗸
No. :	6
Factor :	1.0
	Auto Self Weight
	Show Loadings on Structure
	Show Values
	OK Cancel Apply

Click **OK** to finish this step.

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

Properties 🏾 🗜	x	
🛃 🥕 🛣 🖻 🖬 🎢		
🗄 🗁 Materials	^	
🚊 🗁 Frame Sections		
UC356x368x153#		
[230x90x32		
🗄 🗁 Shell Sections		
🚊 🖓 🧰 Load Cases	≡	
🖸 Self Weight(1.00)		
🕒 Dead Load(1.00)		
📴 Live Load(1.00)		
📴 Wind Load(1.00)		

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;

Properties	д×		
🔹 🥕 🐒 🖬			
🖃 🖧 Untitled			
Haterials Add Load			
E Frame Sections			

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

Loading Properties	
Load Case: Vertical load Load Type: Joint Load	
Force FX 0 FY -1000 FZ 0 Axis ③ Global	Moment MX 0 MY 0 MZ 0
	Apply Close

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

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

Loading Properties	
Load Case: Horizontal load	
Force Moment FX 60 FY 0 FZ 0 Axis Image: Organization of the second se	
<u>Apply</u> <u>Close</u>	

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;

Show Loading	
Load Case ✓ Vertical load ✓ Horizontal load	Load Type Joint Load Settlement Trapezoidal Load Member Pressure Point Load Cable Force Pressure on Arrow
Select All Show Values	Apply Close

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;

Analysis Cases					
Show	ALL	Num. of	items: 4/4	∱ €	
Nam	e	ID	Туре	Run	
Edit Add Re Use	Linear Analysis Nonlinear Analysis name Eigen-Buckling Analysis Process Response Spectrum Analysis Time History Analysis		Set Run Flag Run / Not Run All Not Run All Run Now	Run	

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

EIGEN-BUCKLING	X
Eigen-Buckling Analysis Applied Loads	
Name: Braced EIGEN-BUCKLING	
Number of Modes: 6	
Output Control Print *.out O Yes O No	
OK Cancel Apply	

igen-Buckling Analysis Applied I General Load Cases	Loads	Loads Applied		
Type: Load Case	1.00 🗸	Name Vertical load Horizontal load	Load Case Load Case	Eactor

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;

Analysis	Cases			
Show	ALL	Num. of i	tems: 1/1	À 4
Nam	e d EIGEN-BUCKLING	ID 1	Type EIGEN-BUCKLING	YES
Edit Add Re Use	 Linear Analysis Nonlinear Analysis Modal Analysis Eigen-Buckling Analysis Process Response Spectrum Analysis Time History Analysis 		Set Run Flag Run / Not Ru All Not Run All Run Now	n Run OK

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;

	ied Loads Constructio	n Sequence
Name: Braced NONLINEA	R	Numerical Method
Type: Second-order Analy	vsis + Design 🔹 🔻	Newton-Raphson (Constant Load) Method
PEP Element O Curv	red Stability Function	Single Displacement Control (Constant Disp.) Method
Enable Plastic	Plastic Element	Arc Length Method + Minimum Residual Displacement Method
Advanced Analysis	(@) Plastic Hinge	Iterative & Incremental Parameters :
Total Load Cycles :	1000	
Target Load Factor :		Incremental Load Factor : 0.001
Maximum Iterations for each Load Cycle :	100	
Number of Iterations for Tangent Stiffness Matrix :	1	
Minimum Member Imperfect Imperfection Method & Dire	ion * L / 1000 : 2 ction : Eigen-buck	ling mode : About both principal 🔻 🛄
Minimum Member Imperfect Imperfection Method & Dire Advanced	ion * L / 1000 : 2 ction : Eigen-buck : Eigenmode Imperfe	ling mode : About both principal
Minimum Member Imperfect Imperfection Method & Dire Advanced	ion * L / 1000 : 2 ction : Eigen-buck : Eigenmode Imperfe Magnitude of Imperfe Global Eigenvalue M	ling mode : About both principal ▼ ection ction for bde : H/200 ▼ (Calc. Options)

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;

General Load Cases	truction Sequence		
Type: Load Case	Name Vertical load Horizontal load	Type Load Case Load Case	Factor 1.00 1.00
	Enable Load/Const	truction Stage	

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

Analysis	Cases			
Show	ALL	Num. of	items: 2/2	\$
Nam Brace Brace	e d EIGEN-BUCKLING d NONLINEAR	1 2	Type EIGEN-BUCKLING NONLINEAR	Run YES YES
Edit	▶ Modify Duplicate		Set Run Flag Run / Not Ru	
Use	name Delete		All Not Run Al	I Run

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;

Analysis complete!	
Start Time : 2011-11-23 11:21:52 Time Elapsed : 00:01:27 Process 1 🗸	
BEGIN SUBSPACE TO FIND EIGENVALUES & EIGENMODES	
MODE NO. LOAD FACTOR	
1 14.6793 2 35.4036 3 62.7160 4 353.7503 5 446.7995	
Date : 20111123 Time : 11:23:18 Zone : +0800	
Show Trace OK Cancel	

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

📅 Nida - [Untitled]		
Eile Edit Select View Construct Gr-Assign Analysis Post	Tools <u>W</u> indow <u>H</u> elp	
🗅 🚅 🔒 👗 🖻 💼 🗠 🖙 🎒 🖀 🛗 🔢 🍳 🔍 🗎 30	💌 🚸 🖆 3D 🞽 💿 👔 📥 👔	IY IZ 🔀 🛠
17 П 🕖 📟 🗄 🛷 🛥 🕂 🙀 🗕 🗖 🐹 😤 🛛 💷	> Mode 1 / 6	Buckling LF: 14.6793 🕨 🔳 📑 😤 🛰 🌾

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,

<u>A</u> nalysis	Post	Tools <u>W</u> indow <u>H</u> elp			
ର୍ ପ୍ 🏼		Show Deformed Shape	ļ	, '	ł
<	\checkmark	Show Undeformed Shape			
		Show Deformed & Undeformed	ł		_
		Display Scale			
		Show <u>A</u> nalysis Case			
		Show <u>R</u> esult Files			
		Nodal Results	Þ		
		Member Results	•		
		Shell Results	•		
		Export Summary of Analysis Results			
		Export Statistics of Analysis Results			
		Export Eigen-Buckling Load Factors			
		Export Animation AVI			

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

Buckling LF: 14.6793 🕨 🔳 📑 🕌 🛰	79 7	
	Exp	ort Eigen-Buckling Load Factors

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

Export Eigen-Buckling Load Factor	X
Source File	
C:\Documents and Settings\xu\Desktop\Untit	ed.dat Browse Load Info.
Analysis Cases	
0 items selected Select	
	Export Close
1 items selected Select	
	<u>Export</u>

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

Select Analysis Case	$\overline{\mathbf{X}}$
Analysis Type: EIGEN-BUCKLIN	IG 💌
Load Factor From 0.8	To 1.2
Unselected	Selected
ID Name	ID Name I Braced EIGEN-BUC ALL >> I
	<u>O</u> K <u>C</u> ancel

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;

		- (" - 🕞	Untitled_lo	adfactor -	Microsoft Exc	el _	= X
	Home	Insert Page Layout	Formulas	Data Revie	w View Acr	obat 🞯 –	∎ x
Pa	aste ✓ ✓	imes New R($*$ 11 $*$ B $\mathbf{Z} \underline{\mathbf{U}} \mathbf{A}^* \mathbf{A}^*$ $\mathbf{B} \mathbf{A}^* \mathbf{A}^*$ Font		■ III N ■ III N N N	% umber Styles	Cells ✓	· ᢓ7· · ∄∄· ·
	G8	- (0	f _x				*
	А	В		С	D	E	
1	Ancs No.	Ancs Nan	ne	Cycle No	Load Facto	r	
2	1						
	1	Braced EIGEN-BU	ICKLING	1	14.67	79	
3	1	Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING	1	14.67 35.40	79)4	
3 4	1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING	1 2 3	14.67 35.40 62.71	79)4 .6	
3 4 5	1 1 1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING CKLING	1 2 3 4	14.67 35.40 62.71 353.75	79 04 66 50	
3 4 5 6	1 1 1 1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING CKLING CKLING	1 2 3 4 5	14.67 35.40 62.71 353.75 446.79	79 04 06 00 09	
3 4 5 6 7	1 1 1 1 1 1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING CKLING CKLING CKLING	1 2 3 4 5 6	14.67 35.40 62.71 353.75 446.79 5369.69	79 14 6 50 99 95	
3 4 5 6 7 8	1 1 1 1 1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING CKLING CKLING CKLING	1 2 3 4 5 6	14.67 35.40 62.71 353.75 446.79 5369.69	79 14 50 99 95	
3 4 5 6 7 8	1 1 1 1 1 1	Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU Braced EIGEN-BU	CKLING CKLING CKLING CKLING CKLING CKLING		14.67 35.40 62.71 353.75 446.79 5369.69	79 14 66 09 99	

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

<u>A</u> nalysis	Post Tools Window Help	
ର୍ ପ୍ 🏼	Show Deformed Shape	*
24	Show Undeformed Shape	
	Show Deformed & Undeformed	

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

 Image: Image

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

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp	
ତ୍ତ୍ର 🛛		Show <u>D</u> ef	ormed Shap	e	*
<		Show <u>U</u> nd	leformed Sh	ape	
	×	Show Def	ormed & Un	deformed	-
		Display So	:ale		

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

77 Π /7		<u>ii</u> 🔊	~	÷	₩		2	5
	Adj	ust Scalir	ng Fa	ctor	1			

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

Scaling	
Delta	20
Angle	20
Rese	t Apply

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;

Z			Mode 1 / 6	Buckling LF: 14.6793
	Se	lect Analysis Case		

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

Show A	nalysis Case		×
Gene	ral Multi-Items		
ID	Analysis Name	Type	
*1	Braced EIGEN-BUCKLING	FIGEN-B	
2	Braced NONLINEAR	 NONLIN	
_			
		Close	
		Linse	

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

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp	
ର୍ ପ୍ 🛽		Show <u>D</u> ef	formed Shap	be in the second se	*
2		Show Un	deformed Sł	nape	
	\sim	Show Def	formed & Ur	ndeformed	-
					 1

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

17	٦	Π	hut	15	~	~	÷	М	
⊾ Sh	ow	Unde	forme	d an	d De	forme	ed Sh	nape	1

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

<u>A</u> nalysis	Post	t Tools <u>W</u> indow <u>H</u> elp	
ର୍ ବ୍ 🏼		Show Deformed Shape	►
<		Show <u>U</u> ndeformed Shape	
	~	Show Deformed & Undeformed	_
		Display Scale	

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

17 N /7		🚊 🛷 🛥 🕂 🚧 🗕 🗖 🕱 🖷	î
	Adj	just Scaling Factor	

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;

177	臣 🖌	~	* *]	2	5		
k				Mem	ber S	Sect. (Capa. F	actor

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

<u>A</u> nalysis	Post	Tools <u>W</u> indow <u>H</u> elp	_
29		Show Deformed Shape	→ 3
<		Show <u>U</u> ndeformed Shape	
	×.	Show Deformed & Undeformed	_
		Display Scale	
		Show <u>A</u> nalysis Case	
		Show <u>R</u> esult Files	

Click the *Post > Show Result Files* command;

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

ID	Analysis Name	Туре
1	Braced EIGEN-BUCKLING	EIGEN-BUCK.
2	Braced NONLINEAR	NONLINEAR
31- Tu		

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;

```
- 0 ×
tut2(2).log - Notepad
File Edit Format View Help
                                                                                 ٠
  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)
 *** THE
             SYSTEM
                           IS STABLE ***
 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 ...
                               _____
 *** THE SYSTEM IS STABLE ***
 CURRENT STIFFNESS / INITIAL STIFFNESS = 1.0000
 LOAD STAGE = 0 ; LOAD FACTOR =
                                       0.0010 ,
                                                     0.0010 init.
                     1 NORM detU/U,resF/FCE = 0.1000E+01 0.2635E-02
 KCYC=
         1
             ITER.=
```

2.16 Step 16 – Delete members for unbraced case

Click the *Post > Show Undeformed Shape* command;

<u>A</u> nalysis	Post	Tools	<u>W</u> indow	<u>H</u> elp	
a Q [[Show <u>D</u> ef	formed Shap	be	₩.
<		Show Un	deformed Sh	nape	
	1	Show Det	formed & Un	deformed	

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



The structure returns back to its undeformed shape;





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

Select by Section	\mathbf{X}
UC356x368x153#	
[230x90x32	
ОК	Cancel

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,

<u>F</u> ile	<u>V</u> iew	Tools	<u>H</u> elp	
D	<u>N</u> ew			Ctrl+N
	New Mo	odel from	Template	s
2	Open			Ctrl+O

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'.

New Project			×
Title:			
Tutorial 3			~
	_		Ŧ
Gravity Direct	ion: -Z	-	
- Force Unit -		- Length Unit	
© N	🔘 kgf	🔘 mm	🔘 m
kN	tons	🔘 cm	
	ОК		Cancel

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);

General Settings
General Active DOFs
Title:
Tutorial 3
Floor Stiffness: 0 Gravity Direction: -Z -
Steel Design: HKSC (2011)
Concrete Design: HKCC (2004,2nd)
Force Unit
©N ©kgf ©mm ⊚m
Advanced
Auvanceu

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

General Settings
General Active DOFs
Degree of Freedom (DOF)
Translational: VX VV VZ
Rotational: RX RY RZ
Fast DOFs
Active All X-Y Plane Truss
OK Cancel Apply

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>;

Properties		
* 2 2	🖻 🖬 🍰	
😑 🖧 Untit	led.dat	
÷ 🔁	Materials	
	New Materia	ls
- 6	Set Visible/Ir	nvisible
<u>.</u>	Sort Materia	ls

Default steel grade set up in the program is S275;

Properties	ųх
🔹 🥕 🐒 📑 🔛 🦽	
🖃 🖧 Untitled.dat	
🖨 🗁 Materials	
S275	

Double click <\$275> to view and change the material properties in the <Materials> window if required, such as changing the material color only for this tutorial;

Name: S275	Import
Type: Steel	•
Poisson's Ratio:	0.3
Shear Modulus of Elasticity (kN/m2):	8.0769e+007
Young's Modulus of Elasticity (kN/m2):	2.1000e+008
Density (kN/m3):	77
Coefficient of Thermal Expansion:	1.4000e-005
Yield Stress, fy (kN/m2):	2.7500e+005

Click OK to continue.

3.4 Step 4 – Set gravity direction

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

Select	View	Construct	Gr-A <u>s</u> sign	<u>A</u> nalysis
B. 6	:	. Q		

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

Set 3D View	×	
	Rotation	
∧	X 🔄 Y 📄 Z 🚔	Z
	Angle : 90	
▶ →	XY XZ YZ	X
	Reset Apply	

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

Set 3D View	x	
	Rotation X Y Z Z Angle : 90 XY XZ YZ	→ <mark>7</mark>
	Reset Apply	

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;

General Settings				
General Active DO	Fs			
Title:				
Tutorial 3	*			
Floor Stiffness:	0 Gravity Direction: -Z -			
Steel Design:	HKSC (2011) -			
Concrete Design:	HKCC (2004,2nd)			
Force Unit	Length Unit			
🔘 N 🔘 kg	ff ⊚mm ⊚m			
🔍 kN 🔘 to	ns 🔘 cm			
Advanced				
	OK Cancel Apply			

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;

Node 🛛 🗙
General Boundary Support Spring Loading
No.: 2
X · 3
Y · 3
7. 0
Ζ. υ
OK Cancel Apply

Click OK to continue.

3.6 Step 6 – Copy and paste nodes

Click the *Select > Select All* command,

<u>E</u> dit	Select	<u>V</u> iew	<u>C</u> onstruct	Gr-A <u>s</u> sign	<u>A</u> nalysis
3 %	Se	lect All		Ctrl+A	
	De	select		Ctrl+D	

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



Click the *Edit* > *Copy* command;

<u>F</u> ile	<u>E</u> dit	Select	<u>V</u> iew	<u>C</u> onstruct	Gr-A <u>s</u> sign	<u>A</u> nalysis
🚔 (ю	<u>U</u> ndo			Ctrl+Z	
	Сн	<u>R</u> edo			Ctrl+Y	
	ж	Cu <u>t</u>			Ctrl+X	
		<u>С</u> ору			Ctrl+C	
	B	<u>P</u> aste			Ctrl+V	

or click the <Copy> shortcut in the toolbar;



Click the *Edit* > *Paste* command;

<u>F</u> ile	<u>E</u> dit	Select	<u>V</u> iew	<u>C</u> onstruct	Gr-A <u>s</u> sign	<u>A</u> nalysi
🗃 (ю	<u>U</u> ndo			Ctrl+Z	20
	CH	<u>R</u> edo			Ctrl+Y	
	Ж	Cu <u>t</u>			Ctrl+X	
	8	<u>С</u> ору			Ctrl+C	
	B	Paste			Ctrl+V	

or click the <Paste> shortcut in the toolbar;



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

Paste			
⊙ Linear		○ Radial	O Along Path
Increm	ent		um. of Inc.
Inc.X	0		2
Inc.Y	0		
Inc.Z	3		
	<u>о</u> к		ose

Click OK	, then a message	box will pop u	p recommending	merging du	plicate nodes:
011011 0 11	, mon a mossinge	oon nin pop .			

Nida	$\overline{\mathbf{X}}$
⚠	We strongly recommend you to merge duplicate nodes. Do you want to proceed?
	Yes No

Click Yes to proceed and the message box will close;

Merge Duplicate N	lodes 🛛 🔀
Nodes Distance 1 Node Retention P	Folerance (mm):
Merge	<u>C</u> lose Report >>

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

Nida	
⚠	None of selected nodes is merged.
	ОК

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

Merge Duplicate Nodes
Nodes Distance Tolerance (mm): 1
Node Retention Priority : Number of Loading 🗸
Merge Close Report >>

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;

New Sections				×	
General					
Name:	Sect 2	Import	y 2	• • • • • • • • • • • • • • • • • • •	
Type:	4. CHS[Pipe]				
Material:	S355	S355 👻			
-Section Proper	ties (Analysis)				
Cross Sectio	nal Area (A):	0.0000e+000			
Shear Area (Correction Factor:	0	Dimensio	ns	
Second Mon	nent of Area (ly):	0.0000e+000	B: 0		
Second Mon	nent of Area (Iz):	0.0000e+000	D: 0		
Torsional Co	nstant (J):	0.0000e+000	Tf: 0		
Section Module	Section Modulus (Design)				
About y-axis	About y-axis (Zy): 0.0000e+000 Use				
About z-axis	About z-axis (Zz): 0.0000e+000 Elastic(Z)				
About y-axis (Sy): 0.0000e+000 Use			ds: 0		
About z-axis (Sz): 0.0000e+000 Recalculate					
Rolled Section Fabricated Section Cold-formed					
Suppress Frame Eigen-Imperfection : O Yes O No					
Imperfection along Minor y-axis : L/500 - Elastic Plastic					
along Major z-axis : L/500 🗸					
Stress Type : Square-root of Stress					
		OK Can	cel	Apply	

Select cross sections under different categories, country or shape;

U.K. 🗸	VH 💌
U.K.	Box/Rect.
Japan	CHS/Cir.
Europe	VH
U.S.	Channel
China	Angle

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

J.K. 💌		VH	~		
Name	B(cm)	D(cm)	Tf(cm)	tw(cm)	Area(
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
UBP305x305x186#	32.09	32.83	2.56	2.55	237.00
UBP305x305x149#	31.60	31.85	2.07	2.06	190.00
(Ш				>

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

Section			×		
General Membe	ers				
Name:	UB152x89x16	<u>y,</u>			
Type:	5. I/H-section				
Material:	S275	 S275 ▼			
- Section Proper	ties (Analysis)		tw		
Cross Sectio	nal Area (A):	1.9000e-003			
Shear Area (Correction Factor:	0	Dimensions		
Second Mon	nent of Area (ly):	5.2600e-007	B: 0.0887		
Second Mon	nent of Area (Iz):	7.2640e-006	D: 0.1524		
Torsional Co	nstant (J):	3.2400e-008	Tf: 0.0077		
Section Modulu	Section Modulus (Design)				
About y-axis	(Zy): 2.0200e	-005 👝 Use	B2: 0		
About z-axis	(Zz): 1.0900e	-004 Elastic(Z)	Tf2: 0		
About y-axis	About y-axis (Sy): 3.1200e-005 Use				
About z-axis (Sz): 1.2300e-004			Recalculate		
 Rolled Section Fabricated Section Cold-formed Suppress Frame Eigen-Imperfection : Yes No 					
Imperfection along Minor y-axis : L/500 👻 Elastic Plastic					
along Major z-axis : L/400 👻					
Stress Type : Direct Sum of Stress					
		OK Can	cel Apply		
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;

 $P - \delta$ 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 \langle Section Table \rangle window with similar section properties to those required for the columns;

J.K. 💌		VH	~		
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
<]	11				
<	1	00.10	0.04	0.00	>

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

Section	×			
General Members				
Name: UB203x102x23	y, r			
Type: 5.1/H-section ▼ Customize				
Material: S275				
Section Properties (Analysis)	e tw			
Cross Sectional Area (A): 3.2330e-003				
Shear Area Correction Factor: 0	Dimensions			
Second Moment of Area (ly): 1.5000e-006	B: 0.1018			
Second Moment of Area (Iz): 2.2354e-005	D: 0.2032			
Torsional Constant (J): 1.0000e-007	Tf: 0.0093			
Section Modulus (Design)	tw: 0.0054			
About y-axis (Zy): 3.2200e-005 Use	B2: 0			
About z-axis (Zz): 2.0700e-004	Tf2: 0			
About y-axis (Sy): 4.9800e-005 Use	ds: 0			
About z-axis (Sz): 2.3400e-004	<u>R</u> ecalculate			
Rolled Section Fabricated Section Cold-	formed 🚺			
Suppress Frame Eigen-Imperfection : O Yes	No			
Imperfection along Minor v-axis : 1/500				
along Major z-axis : L/400 V	T lastic			
Stress Type : Direct Sum of Stress	Advanced			
OK Can	cel Apply			

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;

Member Properties		×
Section:	UB203x102x23	~
End Condition:	Both Rigid	~

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,

Member Properties		×
Section:	UB152x89x16	~
End Condition:	Both 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;

Select by Section	
UB152x89x16	
002034102423	
<u>0</u> K	Cancel

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;

Paste			
⊙ Linear		ORadial	O Along Path
Increm	ent		um. of Inc.
Inc.X	0		1
Inc.Y	0		
Inc.Z	3		
	<u>о</u> к		ose

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

Nida	
⚠	We strongly recommend you to merge duplicate nodes. Do you want to proceed?
	Yes No

Click Yes to proceed and the message box will close;

Merge Duplicate N	lodes 🛛 🔀
Nodes Distance	Tolerance (mm): 1
Node Retention F	Priority : Number of Loading
]
Merge	Close Report >>

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

Nida	
⚠	4 nodes have been merged.
	ОК

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';

Select by Number		
 Node 	OMember	◯ Shell
◯ Spring	○ Floor	○ Area
List of Object	ts	
1-4		
For example,	10,15,20-22,30-35	
	Select	Close

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;

Construct	Gr-A <u>s</u> sign <u>A</u> nalysis	<u>P</u> ost Tools <u>W</u> indow	Help	
> 🚑 🖊	Nodes	•	Boundary Conditions	τ
	<u>M</u> embers	• 1	Nodal Springs	
	Shells	•	Nodal Local Axis	
	-1	N		

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

Assign Node Properties		
Boundary		
Displacement: UX	O Free	Restraint
UY	O Free	 Restraint
UZ	◯ Free	 Restraint
Rotation: RX	◯ Free	 Restraint
RY	◯ Free	 Restraint
RZ	◯ Free	 Restraint
- Fast Restraints		
<u>₩</u> 4 ÷	•	
		OK Cancel

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;

Visible/Invisible Settings				
Area	Member / Cable	Node		
Area Area Number Normal Direction Floor Floor Floor Number Normal Direction One/Two-Way Shell	Member Member Cable No Number Material Number Material Name Section Number Length	Node Node Node Number Boundary Condition Support Spring Coal Axis Member / Cable Color By Section By Material		
Shell Number Normal Direction One/Two-Way Fill Deformed Fill Undeformed	 Local Axis Y Bending Stiffness about y-y axis Z Bending Stiffness about z-z axis Axial Stiffness Torsional Stiffness 	O Uniform Show Number ☐ Selected Only Others → ☐ Spring Element ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓		
	<u>Q</u> K	<u>Cancel</u> <u>Apply</u>		

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,

onstruct	Gr-A <u>s</u> sign <u>A</u> nalysis <u>P</u> ost 1	Tools <u>W</u> indow <u>H</u> elp
i 🚑 🖊	Nodes) 🔄 🗇 3D 🖄 🐵 📘 📥
	<u>M</u> embers	<u>S</u> ection
	Shells	End Conditions
	Floors	Effective Length
	Areas	 Eccentricity
	Nodal Loads	Member Local <u>A</u> xis

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;

Load Case	Load Case		
General Load	ling		
Name :	Self Weight		
Type :	Dead Load 🗸		
No. :	1		
Factor :	1.0		
	Auto Self Weight Settings		
	Show Loadings on Structure		
	Show Values		
	OK Cancel Apply		

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

Auto Self Weight		
Amplification Factor:	1	
Direction	1	
OX	O+	
ΟY	<u>o</u> -	
⊙Z		
All Members		
All Floors		
All Shells		
<u>0</u> K	Cancel	

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;

onstruct	Gr-/	A <u>s</u> sign	<u>A</u> nalysis	<u>P</u> ost	Tools	V	Vindo
- 🚑 M		<u>N</u> odes	5			۲	슙
		<u>M</u> emb	ers			۲	_
		Shells				۲	
		Floors				۲	
		Areas	l.			۲	
		Nodal	Loads			۲	
		Memb	er Loads			۲	
		Floor	Pressure				
		Shell F	Pressure				

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

Loading Properties
Load Case: Dead Load 🗸
Load Type: Pressure on Area Object 💌
O All ⊙ Floors O Shells O Areas Pressure -5 Direction
Global Axis ○ X ○ Y ⊙ Z Normal ○
 Convert to Member Loads first Convert to Nodal Forces directly
Note: Area Pressure are invalid in analysis before the areas are converted to Floor/Shell elements.
<u>Apply</u> <u>C</u> lose

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

Loading Properties
Load Case: Live Load
Load Type: Pressure on Area Object 💌
○ All ⊙ Floors ○ Shells ○ Areas
Pressure -2
Direction
Global Axis OX OY OZ
⊙ Convert to Member Loads first
O Convert to Nodal Forces directly
Note: Area Pressure are invalid in analysis before the areas are converted to Floor/Shell elements.
<u>Apply</u> <u>Close</u>

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*;

Select by Number		
 Node 	OMember	O Shell
◯ Spring	○ Floor	○ Area
List of Object	S	
6,7		
For example,	10,15,20-22,30-35	
	Select	Close

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

Loading Properties	
Load Case: Wind Load	~
Load Type: Joint Load	~
Force	Moment
FX 0	MX 0
FY -11.25	MY 0
FZ 0	MZ 0
Axis	
⊙ Global	OLocal
	<u>Apply</u> <u>C</u> lose

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;

Select by Number		×
⊙ Node	OMember	O Shell
◯ Spring	○ Floor	○ Area
List of Object	S	
10,11		
For example,	10,15,20-22,30-35	
	Select	Close

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

Loading Properties	
Load Case: Wind Lo Load Type: Joint Loa	ad 💌
Force FX 0 FY -5.625 FZ 0 Axis ③ Global	Moment MX 0 MY 0 MZ 0 Local
	<u>Apply</u>

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;

Show Loading	$\overline{\mathbf{X}}$
Load Case ✓ Self Weight ✓ Dead Load ✓ Live Load ✓ Wind Load	Load Type ✓ Joint Load Settlement Trapezoidal Load Member Pressure Point Load Cable Force ✓ Pressure on Arrow ✓ Temperature
Select All Show Values	Apply Close

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';

General Load Case:			
Windland		Combined Load Case:	
	1.60 ▼ >> << ALL >> ALL <<	Load Case Self Weight Dead Load Live Load	Fac 1.4 1.4 1.6

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

New Combined Load Case	s
Name: LC2	No.: 2 Factor: 1.0
General Load Case:	Combined Load Case:
Live Load	1.40 Self Weight 1.4 >> Self Weight 1.4 Dead Load 1.4 Wind Load 1.4 ALL >> ALL <
	QK <u>C</u> ancel <u>Apply</u>

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

w Combined Load Cases	No.: 3 Factor: 1.0
General Load Case:	Combined Load Case:
L	<u>O</u> K <u>C</u> ancel <u>Apply</u>

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;

Analysis	Cases				×
Show	ALL	~	Num. of	items: 4/4	\$
Nam	e		ID	Туре	Run
-Edit-				Set Run Flag	
Add Re	ename	Linear Analysis Nonlinear Analysis Modal Analysis Eigen-Buckling Analysis		Run / Not Ru	n Run
Use	Process	Response Spectrum Analysis Time History Analysis		Run Now	ОК

Give the name 'LC1' to the 1st load combination case;

EIGEN-BUCKLING	X
Eigen-Buckling Analysis Applied Loads	
Name: LC1	
Number of Modes: 6	
Output Control	
Print *.out 🔿 Yes 💿 No	
	OK Cancel Apply

Apply the load combination LC1;

igen-Buckling Analysis Applied Loads	5		
General Load Cases	Loads Applied		
Type: Combined ▼ LC2 LC3 	Name LC1	Combined	Factor
ALL			

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;

Analysis	Cases					
Show	ALL		v 1	Num. of	items: 1/1	\$
Nam LC1	e			ID 1 - - - - - - - - - - - - -	Type EIGEN-BUCKLING	YES
Edit- Add Re Use	name Process	Linear Analysis Nonlinear Analysis Modal Analysis Eigen-Buckling Analysis Response Spectrum Analysis Time History Analysis			Set Run Flag Run / Not Ru All Not Run Al Run Now	I Run

Give the name 'LC2' to the 2nd load combination case;

GEN-BUCKLING			Đ
Eigen-Buckling Analysis App	ied Loads	 	
Name: LC2			
Number of Modes:	6		
- Output Control			
Print .out OYes	⊙ No		

Apply the load combination LC2;

Type: Combined	1.00 🗸	Name LC2	Type Combined	Factor
	<pre>>> ALL >></pre>			
	ALL <<			

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;

Analysis	s Cases					
Show	ALL		V N	um. of i	tems: 2/2	\$
Nam LC1 LC2	e			ID 1 2	Type EIGEN-BUCKLING EIGEN-BUCKLING	Run YES YES
Edit					Set Run Flag	
Add Re Use	ename Process	Linear Analysis Nonlinear Analysis Modal Analysis Eigen-Buckling Analysis Response Spectrum Analysis Time History Analysis			Run / Not Ru	II Run

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

IGEN-BUCKLING		
Eigen-Buckling Analysis Applied Loads		
Name: IC2		
Number of Modes: 6		
Output Control		
Print .out Tes INO		

Apply the load combination LC3;

IGEN-BUCKLING Eigen-Buckling Analysis Applied Loads			
General Load Cases	Loads Applied		
Type: Combined	Name LC3	Type Combined	Factor
	ОК	Cancel	Apply

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

Analysis Cases					
Show	ALL	Num. of	items: 3/3	\$	
Nam LC1 LC2 LC3	e	ID 1 2 3	Type EIGEN-BUCKLING EIGEN-BUCKLING EIGEN-BUCKLING	Run YES YES	
Edit Add Re Use	Modify Duplicate		Set Run Flag Run / Not Ru All Not Run Al Run Now	IRun OK	

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>;

Properties 🛛 🕹 🗙			
📲 🥕 🛣 🖻 🖬 🎢			
🖃 🖧 tut3.dat			
🕀 🗁 Materials			
🚊 🗁 Frame Sections			
UB203x102x23			
🗄 🗁 Load Cases			
🚊 🛅 Combined Load Cases			
New Combined Load Cases	s		
Generate Analysis Cases			
Sort Combined Load Cases	s		

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

Set Analysis Case	X
Analysis Type: Nonlinear Analysis Case Name Prefix 2nd-order + Combined Name + Serial No. Suffix	
	< Back Next > Cancel

Click *Next* to continue;

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

Name:	Numerical Method				
Type: Second-order Analysis + Design 🗸	 Newton-Raphson (Constant Load) Method 				
PEP Element Curved Stability Function	Single Displacement Control (Constant Disp.) Method				
Enable Plastic Advanced Analysis Plastic Element Plastic Hinge	Arc Length Method + Minimum Residual Displacement Method				
Total Load Cycles : 1 Number of Iterations for each Load Cycles : 100					
Number of Iterations for Tangent Stiffness Matrix : Incremental Load Factor :					
Minimum Member Imperfection * L / 1000 1 Imperfection Method & Direction : Displacement : About one principal axis Setting					
Advanced					

Click *Finish* to complete adding analysis cases;

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



Analysis Cases 🛛 🔀					
Show ALL	Num. of	items: 6/6	7 4		
Name	ID	Туре	Run		
LC1	1	EIGEN-BUCKLING	YES		
LC2	2	EIGEN-BUCKLING	YES		
LC3	3	EIGEN-BUCKLING	YES		
2nd order LC1	4	NONLINEAR	YES		
2nd order LC2	5	NONLINEAR	YES		
2nd order LC3	6	NONLINEAR	YES		
Edit		Set Run Flag			
Add Modify Duplicate Run / Not Run Rename Delete All Not Run All Ru			in I Run		
Use Processors: 1		Run Now	<u>о</u> к		

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;

Analysis complete!		
Start Time : 2011-11-30 1	1:39:00 Time Elapsed : 00:00:03 Pro	cess 1 🗸
BEGIN SUBSPACE TO FIND	EIGENVALUES & EIGENMODES	~
MODE NO. LOAD FACTOR		
1 8.1501		
2 11.5863		
3 15.5827		
4 20.0029		
5 28.9532		
 Date : 20111130	Time : 11:39:02 Zone : +080	
Show Trace	ок	Cancel

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

📅 Nida - [Untitled]			
Eile Edit Select View Construct Gr-Assign Analysis Post	Tools <u>W</u> indow <u>H</u> elp		
D 😅 🔲 % 🖻 💼 🗠 🖂 🖨 👭 🛗 🐯 🍳 Q 📴 30	💌 🚸 💧 🗇 3D 🗹 👁 📔 📥 🕻	IY IZ 🔀 🛠	
77 口 // 罒 Ё 〃 ┵ + ☆ ー z ヌ 客 📧	Mode 1 / 6	Buckling LF: 14.6793	• 🔳 📑 🐔 🌬

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,

<u>A</u> nalysis	Post	Tools <u>W</u> indow <u>H</u> elp				
ର୍ ପ୍ 🏼		Show Deformed Shape		÷ ₹		
<	×	Show Undeformed Shape				
Show Deformed & Undeformed		Show Deformed & Undeformed				
		Display Scale				
		Show <u>A</u> nalysis Case				
		Show <u>R</u> esult Files				
Nodal Results		Nodal Results	×			
		Member Results	►			
		Shell Results	۲			
		Export Summary of Analysis Results				
		Export Statistics of Analysis Results				
		Export Eigen-Buckling Load Factors				
		Export Animation AVI				

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

Buckling LF: 6.6649	省 🔹	Her.	
		Exp	ort Eigen-Buckling Load Factors

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

Export Eigen-Buckling Load Factor	×
Source File	1
C:\Documents and Settings\xu\Desktop\step by step\tut3\Ur Browse Load Info.	J
Analysis Cases	ĥ
0 items selected Select	
	ו
<u>Expon</u>	J
·	
3 items selected Select	
<u>Export</u>	

Click *Select* and choose the analysis case in the <Select Analysis Case> window;
Select Analysis Case		
Analysis Type: EIGEN-BUCKLIN	G 🗸	
Load Factor From 0.8	To 1.2	All Load Factor
Unselected	\$	Selected
ID Name	>> << ALL >> ALL <<	ID Name 1 LC1 2 LC2 3 LC3
		<u>O</u> K <u>C</u> ancel

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;

	💽 🕼 🖤 🛛 🔍 🖓 🗘 🕫 Untitled_loadfactor - Microsoft Ex 💶 📼 🗙						
	Home	Insert Page Lay	Formula D	ata Review Vi	ew Acrobat	🕑 – 🗖 X	
$\begin{array}{c c} \hline \\ \hline \\ \hline \\ \hline \\ Paste \\ \hline \\ Clipboa \\ \hline $				Styles •	$\Sigma \stackrel{\sim}{\sim} \stackrel{\sim}{2} \stackrel{\sim}{\sim} $		
	A1	- (0	f_x	Ancs No.		*	
	А	В	С	D	E	F 두	
1	Ancs No.	Ancs Name	Cycle No	Load Factor			
2	3	LC3	1	7.131			
3	3	LC3	2	9.790			
4	3	LC3	3	12.521			
5	3	LC3	4	17.654			
6	3	LC3	5	21.816			
7	3	LC3	6	31.601			
8							
9	2	LC2	1	7.567			
10	2	LC2	2	11.066			
11	2	LC2	3	13.283			
12	2	LC2	4	19.588			
13	2	LC2	5	21.473			
14	2	LC2	6	29.820			
15		1.01					
16	1	LCI	1	0.005			
1/	1	LCI	2	10.589			
18	1	LCI	3	12.326			
19	1	LCI	4	15.128			
20	1	LCI	2	42.225			
21	1	LUI	0	42.9//			
II I	► ► Loa	ad factor of cy	cles 🦯 🔁				
Rea	dy		E	III I00%	0	ii. 🕀 💎	

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

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp				
ର୍ ପ୍ 🛛		Show Deformed Shape						
2		Show Undeformed Shape						
	× .	Show Deformed & Undeformed						

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;

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp				
ତ୍ତ୍ର 🛛		Show Def	formed Shap	be and the second se	₩			
<		Show Undeformed Shape						
	\checkmark	Show Deformed & Undeformed						
		Display Si	cale					

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

MΠ Π		<u>11</u> 🛷	~	÷	М		;	5
	Adju	ust Scali	ng Fa	ctor	1			

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

Scaling		
Delta	5	
Angle	5	*
Reset		Apply

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;



Sh	now Ai	nalysis Case		
	Gene	ral Multi-Items		
	ID	Analysis Name		Туре
	* 1	LC1		EIGEN-B
	2	LC2		EIGEN-B
-	3	LC3		EIGEN-B
-				
-				
-				
-				
-				
-				
			<u>0</u> K	Close

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

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

<u>A</u> nalysis	Post	Tools	<u>W</u> indow	<u>H</u> elp				
ର୍ ପ୍ 🛛		Show <u>D</u> eformed Shape						
34		Show Undeformed Shape						
	× .	Show Deformed & Undeformed						

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;

<u>A</u> nalysis	<u>P</u> ost	Tools	<u>W</u> indow	<u>H</u> elp				
ର୍ ବ୍ 🏼		Show <u>D</u> ef	formed Shap	e		*		
<		Show Undeformed Shape Show Deformed & Undeformed						
		Display So	cale					

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

νν Π ΖΖ		<u>:</u> 🛷	~	÷	₩		;	r
	Adju	ust Scalir	ng Fac	tor	1			

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

Scaling		
Delta	4	
Angle	4	*
Reset		Apply

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



With its elastic critical load factor $\lambda_{cr} = 7.5670$;

Mode 1 / 6 Buckling LF: 7.5670

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



With its elastic critical load factor $\lambda_{cr} = 7.1314$. Mode 1 / 6
Buckling LF: 7.1314

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

<u>A</u> nalysis	<u>P</u> ost	Tools <u>W</u> indow <u>H</u> elp		
ର୍ ପ୍ 🏼	\checkmark	Show Deformed Shape	÷	ξ
<		Show Undeformed Shape Show Deformed & Undeformed		
		Display Scale		
		Show <u>A</u> nalysis Case		
		Show <u>R</u> esult Files		
		Nodal Results		
		Member Results		
		Shell Results		
		Export Summary of Analysis Results		
		Export Statistics of Analysis Results		
		Export Eigen-Buckling Load Factors		

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

Load Factor:	1.0000		1	Ner Yer
				Export Statistics of Analysis Results

port Statistical Results	×							
Source File								
C:\Users\hknida\Desktop\NIDA Files\Tutorials\tut3.dat Browse Load Info								
Analysis Case								
0 Items Selected Select								
Node								
Export 1 Displacements Sorted by	U(Resultant) 🔹							
Export 1 Reactions Sorted by	F(Resultant)							
Member								
Export 1 Members	Section 0 Selected							
Extreme Values for Each selected sect 💌	Sorted by Section capacity f v							
Export Internal Forces & Moments	Add Comments							
Export Member Design Details (*.NSD File)								
	OK Creat							
	Cancel							

'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;

Select Analysis Cases	×
Analysis Type: Nonlinear Analysis	▼
Unselected (0)	Selected (3)
ID Name AL AL AL AL AL AL AL AL AL	ID Name Load Facto 4 2nd order LC1 (0.95,1.05) 5 2nd order LC2 (0.95,1.05) 6 2nd order LC3 (0.95,1.05) <
	<u>O</u> K <u>Cancel</u>

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';

Export Statistical Results
Source File C:\Users\hknida\Desktop\NIDA Files\T\tut3.dat Browse Load Info
Analysis Case 3 Items Selected Select
Node
Export 1 Displacements Sorted by U(Resultant)
Export 1 Reactions Sorted by F(Resultant)
Member
Export 1 Members Section 2 Selected
Extreme Values for Each selected sect Sorted by ISection capacity f
Export Internal Forces & Moments
Export Member Design Details (*.NSD File)
OK Cancel

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;

Select Sections	X
Available Section	Selected Section
ID Name	ID Name 2 UB152x89x16 3 UB203x102x23 <
	<u>Q</u> K <u>C</u> ancel

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;

C) 🖬 🤊	- (- 1) -			tut3 - Micr	osoft Excel				- =	x
	Home	Insert Pag	ge Layout F	ormulas	Data	Review Vie	w Acrob	at	6) - 🗖	×
Pa	iste	Times New R(\cdot 12 B I \underline{U} \cdot A^{\cdot} \vdots \cdot \bigcirc \bullet \bullet \bullet \bullet		≡ ≣ ≫∕*	← General	6 , Styles	Har Insert ▼ Market ▼ Delete ▼ Format ▼ Cells	Σ · A · Z · Sort & F · Filter · S	ind & elect *		
Cemp	A1	+ (o	<i>f</i> ∗ AN	CS No.			cens	Cutting			¥
	А	В	С	D	E	F	G	Н		I	T
1	ANCS N	o. ANCS Name	Load Stage	Cycle	Cycle Fac.	Sect	Sect No.	Member No.	Sect Ca	pa. Fac	
2		5 2nd order LC2	0	1	1	UB152x89x16	2	11		-1.299)
3		5 2nd order LC2	0	1	1	UB203x102x23	3 3	1		-0.929)
4											-
-14-4	If I Member sort by Sect.Cap.Fac.								1		
Rea	dy						- E I	100% 😑 –			

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:
                      (Not Considered)
                   0
<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)]
          [d/t] = 34.1852 \le 80e/(1+r1)
  Web
                                               =
                                                   67.7695
                   5.4731 <= 9e
 Flange : [b/T]=
                                                    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;

<u>A</u> nalysis	<u>P</u> ost	Tools <u>W</u> indow <u>H</u> elp	_	
€ Q [<	*	Show <u>D</u> eformed Shape Show <u>U</u> ndeformed Shape Show Deformed & Undeformed Display Scale Show <u>A</u> nalysis Case	*	•
		Show <u>R</u> esult Files		
		Nodal Results		
		Member Results		
		Shell Results		
		Export Summary of Analysis Results		
		Export Statistics of Analysis Results		

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

Load Factor: 1.0000	۲		🚰 🛰 λοτ
			Export Summary of Analysis Results

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

Export Analysis	Results	$\overline{\mathbf{X}}$
Analysis Case 0 Item	ns Selected Select	Output Format
Output Path:	C:\Documents and Settings\xu\Deskt	pp/step by step/tut3 V Browse
Items Max Sectio Nodal Disp Nodal Rea Internal Fo Deflection Deflection	n Forces & Moments blacements ctions rces & Moments of Members of Combined Members	Output Options
Select All	Unselect	Select All Unselect Setting Export Close

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

Select Analysis Cases	×
Analysis Type: Nonlinear Analysis]
Unselected (0)	Selected (3)
ID Name >> ALL <	ID Name Load Facto 4 2nd order LC1 (0.95,1.05) 5 2nd order LC2 (0.95,1.05) 6 2nd order LC3 (0.95,1.05)
Load Factor From 0.95 To 1.05 All Load Factor	<
	<u>Q</u> K <u>C</u> ancel

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

Export Analysis Results	$\overline{\mathbf{X}}$
Analysis Cases 3 Items Selected Select	Output Format
Items Max Section Forces & Moments Modal Displacements Nodal Reactions Internal Forces & Moments Deflection of Members Deflection of Combined Members	Output Options Output Options
Select All Unselect	Select All Unselect Setting Export Close

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;

<u>A</u> nalysis P	ost Tools <u>W</u> indow <u>H</u> elp	
€, C, [Show <u>D</u> eformed Shape Show <u>U</u> ndeformed Shape Show Deformed & Undeformed Display Scale Show <u>A</u> nalysis Case Show <u>R</u> esult Files Nodal Results	 ↓ IY IZ 2 4 ± Load Factor: 1.0000 ▶ ■ ■
	Member Results	Member Statistics
	Shell Results	Show Section Capacity Factor Statistics
	Export Summary of Analysis Results Export Statistics of Analysis Results	Show Bending Moment/Shear Force Diagram Show Plasic Hinge

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

🖺 🖉 🛥 🕂 🚧 🗕 🗖 🕱 🗟
Show Section Capacity Factor Statistics

Section Capacity Statistics	
Clustered Bar Scatter Chart	
	Analysis Case Section Options Background , Axes
	Font
	• By Count
	O By Weight
↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	Export BMP Close

The <Section Capacity Statistics> window will appear;

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

Details	
Analysis Case Type: All Available	
ID Name Load Stage Load Cycle ✓ 4 2nd or 0 ✓ 5 2nd or 0 ✓ 6 2nd or 0	Select a Load Stage Select a Load Cycle Select Items* • Load Factor Range 0.95 and 1.05 • Cycle No. 1 Select *Note:For Nonlinear Analysis Cases or Time History
Select All Unselect Show Selection Only	<u>O</u> K <u>C</u> ancel

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;

Details	$\overline{\mathbf{X}}$
Analysis Case Type: All Available	
ID Name Load Stage Load Cycle ✓ 4 2nd or 0 1(1.00) ✓ 5 2nd or 0 1(1.00) ✓ 6 2nd or 0 1(1.00)	Select a Load Stage Select a Load Cycle
	Select Items* Load Factor Range 0.95 and 1.05 Cycle No. 1 Select *Note:For Nonlinear Analysis Cases or Time History
Select All Unselect Show Selection Only	<u>O</u> K <u>C</u> ancel

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;

Select Sections	$\overline{\mathbf{X}}$
Available Section	Selected Section
ID Name	ID Name 2 UB152x89x16 3 UB203x102x23 <<
	<u>O</u> K <u>C</u> ancel

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 $5 - 2^{nd}$ order LC2;

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

8	<	DIMP
Sele	ect Analysis Case	

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

Show A	nalysis Case		
Gene	eral Multi-Items		
ID	Analysis Name		Туре
1	LC1		FIGEN-B
2	LC2		EIGEN-B
3	LC3		EIGEN-B
4	2nd order LC1		NONLIN
* 5	2nd order LC2		NONLIN
6	2nd order LC3		NONLIN
_			
	I		
		<u>о</u> к	<u>C</u> lose

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 2^{nd} 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.



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,

Define Response Spectrum Functions					
	[
Code:	GB50011(2010) 🔹	Add			
List:		Modify			
		<u>D</u> elete			
		ОК			

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.

Chinese Co	ode: GB50011(2010)
Name: Type:	GB50011(2010) Period vs. Sa/g Damping Ratio: 0.05
Seismi Max In Period	c Intensity (SI): 7(0.15g) fluence Factor (Amax): 0.12 Reduction Factor: 1
Characte Seismi Site Cl	eristic Ground Period (TG) c Design Group: 1 assification: II stomize: 0.35
1.0 0.0 0.0 0.0 0.0 0.0	132 1999 166 133 100 0.60 1.20 1.80 2.40 3.00 3.60 4.20 4.80 5.40 6.00 Period
	OK <u>C</u> ancel

Click OK to continue.

4.3 Step 3 – Define a modal analysis

Analysis Cases			X
Show: ALL	Num. of	f Items: 0/0	∱
Name	ID	Туре	Run
Edit Add Linear Analysis Rename Use Processors Eigen-Buckling Analysis Response Spectrum Analysis		Set Run Flag Run / Not All Not Run Run Now	Run All Run OK
Time History Analysis		-	×
Modal Analysis Mass from Load Cases Initial Loads]		
Mass Type			
Output Control Print *.out Yes			
Number of Modes : 15 Advanced <<			
Determine Required Modes automatically C Active Intervet Adda Masses Ratios(%) >= Incremental Number of Modes : 10			
	ОК	Cancel	Apply

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.

MODAL				-X	
Modal Analysis Mass from Load	Cases Initia	al Loads			
General Load Cases: Additional Masses					
Type: Load Case 🔹		Name	Туре	Factor	
DL	1.00 👻	SDL	Load Case	1.00	
WL .	>>	LL	Load Case	0.0	
	<				
	ALL >>				
	ALL <<				
-					
Note : The gravity direction is global Factor-axis. Only the loadings in the gravity direction will be taken as additional masses.					
		ОК	Cancel	Apply	

Click *OK* to continue.

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

Ana	lysis comple	ete!								×
St	tart Time	: 2012-	-12	-09 21:	02:24 T	ime Elapse	ed : 00	:00:05 (Process 1	•
	мора	L PA	R	тісі	PATI	NG МА	SS	RATI	05:	^
	MODE NUMBER	PERIOD (Sec.)	I	<1> INE UX	UVIDUAL N UY	10DE (%) UZ	<2> CUM SUM-UX	ULATIVE SUM-UY	SUM (%) SUM-UZ	
	1	2.5341	T	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	1	0.00	62.93	0.00	69.62	63.20	0.00	
	4	1.4555	1	0.00	0.03	0.00	69.62	63.24	0.00	
	5	1.0749	1	18.60	0.00	0.00	88.22	63.24	0.00	
	6	1.0041	1	0.00	0.58	0.00	88.22	63.81	0.00	
	7	0.8699	1	0.00	13.39	0.00	88.22	77.21	0.00	
	8	0.7186	1	0.00	6.71	0.00	88.22	83.91	0.00	
	9	0.6149	1	0.00	2.99	0.00	88.22	86.90	0.00	
	10	0.5364	1	4.09	0.00	0.00	92.31	86.90	0.00	
	11	0.5257	1	0.00	0.00	0.00	92.31	86.90	0.00	
	12	0.3752	1	4.65	0.00	0.00 _	96 97	86 90	0 00	
	13	0.3715	Т	0.00	0.00	0.00	96.97	86.91	0.00	-
C	Show Trac	ze						<u>0</u> K	Ca	ncel

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

Show: ALL	•	Num. o	f Items: 0/0	∱ €
Name		ID	Туре	Run
Edit			Set Bun Flag	
Add	Linear Analysis		Run / Not F	lun
<u>R</u> ename	Nonlinear Analysis Modal Analysis		All Not Run	All Run

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

Create a response spectrum analysis case in global x direction;

RS-U1-CQC	
Response Spectrum Analysis	1
Name: FS-U1-CQC Use Modes from Modal Analysis Case: MODAL Response Spectrum Function GB50011(2010) • • Modal Combination © CQC © SRSS © ABS Modal Damping Ratio: 0.05	Seismic Direction Horizontal Vertical Excitation Angle: 0
[OK Cancel Apply

(Create a resp	onse spectrum	analysis	case in	global	v direction:
	create a resp	onse speenan	i anai joio	ease m	Broom	j aneenon,

RS-U2-CQC	×
Response Spectrum Analysis	1
Name: RS-U2:CQC Use Modes from Modal Analysis Case: MODAL Response Spectrum Function (GB50011(2010) Modal Combination CQC SRSS ABS Modal Damping Ratio: 0.05	Seismic Direction Horizontal Vertical Excitation Angle: 90 y a x
	OK Cancel Apply

Create a response spectrum analysis case in global z direction;

Name: RS-U3-CQC Use Modes from Modal Analysis Case: MODAL Response Spectrum Function (GB50011(2010)	Seismic Direction Horizontal Excitation Angle: 0
Modal Combination © CQC © SRSS © ABS Modal Damping Ratio: 0.05	

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.

nalysis Cases				×
Show: ALL	•	Num. of	f Items: 0/0	\$
Name		ID	Туре	Run
Edit			Set Run Flag	
Add	Linear Analysis Nonlinear Analysis Modal Analysis		Run / Not Ru	n II Run
Use Processors	Eigen-Buckling Analysis Response Spectrum Analysis	×	Run Now One Direction On	ок ly
	Time History Analysis		Directional Comb	ination

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

RS-U1U2-CQC-SRSS
Response Spectrum Analysis (Directional Combination)
Name: RS-U1U2(0.85)-CQC-SRSS
Directional Combination
SRSS ABS Modified SRSS(Chinese)
Response Spectrum Analysis for Combination
EQ1 (Horizonal): RS-U1-CQC
EQ2 (Horizonal): RS-U2-CQC -
OK Cancel Apply

RS-U1U2-CQC-SRSS	J
Response Spectrum Analysis (Directional Combination)	
Name: RS-U1(0.85)U2-CQC-SRSS Directional Combination SRSS ABS Modified SRSS(Chinese)	
Response Spectrum Analysis for Combination EQ1 (Horizonal): EQ2 (Horizonal): RS-U1-CQC	
OK Cancel Apply	

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":

1.2DL+0.28WL+1.3EQX		x
Linear Analysis Applied Loads		
Name: Analysis Type: Linear Analysis + Design Moment Amplification Factor (1) $\frac{\lambda_{cr}}{\lambda_{cr}-1}$ λ_{cr} 0 (2) $\frac{1}{1-\frac{F_{c}L^{2}_{E}}{\pi^{2}EI}}$ Note: Larger of (1) and (2) will be used	if both are active.	
1.2DL+0.28WL+1.3EQX	OK Cancel Apply	×
General Load Cases:	Loads Applied	
Type: Combined Load Case Combined Spectrum Analysis Case >>	Name Type Factor DL Load Case 1.20 SDL Load Case 1.20 WL Load Case 0.28 PS II1 COC Space Space	
ALL >> ALL <<	RS-0 FCac Spectrum 1.30	

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

1.2DL+0.28WL+1.3EQY			×
Linear Analysis Applied Loads			
Name: 1.2DL+0.28WL+1.3EQ Analysis Type: Linear Analysis + Desig Moment Amplification Factor (1) $\frac{\lambda_{cr}}{\lambda_{cr}-1}$ λ_{cr} (1) (2) $\frac{1}{1-\frac{F_{c}L^{2}_{E}}{\pi^{2}EI}}$ Note: Larger of (1) and (2) will be used	ĭ n ▼ if both are active.		
	ОК	Cancel	Apply
1.2DL+0.28WL+1.3EQY			×
Linear Analysis Applied Loads			
General Load Cases:	Loads Applied		
Type Combined Load Case Combined 1.00	Name DL SDI	Type Load Case	Factor
Subscription Analysis Case Image: Subscription Analysis Case <tr< td=""><td>WL RS-U2-CQC</td><td>Load Case Load Case Spectrum</td><td>1.20 1.20 0.28 1.30</td></tr<>	WL RS-U2-CQC	Load Case Load Case Spectrum	1.20 1.20 0.28 1.30

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

1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ	_	-	x
Linear Analysis Applied Loads			
Name: $\begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} \begin{array}{c} $	COX+EQY)+0.5EQZ Design ▼		
	ОК	Cancel	Apply
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ			
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ			
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases:	Loads Applied		
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined	Loads Applied	Type F	actor
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined	Loads Applied Name RS-U1-CQC RS-U1-CQC	Type F Spectrum 1	Eactor 1.30
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Combined Spectrum Analysis Case >>	✓ Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U2-CQC RS-U3-CQC	Type F Spectrum T Spectrum Spectrum (=actor 1.30 1.50
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case >> <<	Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL	Type F Spectrum 1 Spectrum 1 Spectrum 1 Load Case 1 Load Case 1	Eactor 1.30 1.20 1.20
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Combined Sectrum Analysis Case >> Combined Sectrum Analysis Case >> Combined Sectrum Analysis Case >>	Coads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL	Type F Spectrum Spectrum Spectrum Load Case T Load Case 1 Load Case 1	Eactor 1.30 1.30 0.50 1.20 1.20 1.20 1.22 1.22
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case	 ✓ Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL 	Type F Spectrum 1 Spectrum 1 Spectrum 1 Load Case 1 Load Case 1 Load Case 1	Eactor 1.30 1.20 1.20 1.28
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case ALL 2 ALL 2	 Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL 	Type F Spectrum Spectrum Spectrum Case Load Case Case Load Case Case	Eactor 1.30 1.30 1.20 1.20 1.20 1.20 1.22
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case <<< ALL ALL	 Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL 	Type F Spectrum Spectrum Spectrum C Load Case Load Case Load Case C	Eactor 1.30 1.30 1.20 1.20 1.20 1.20 1.22
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case	 ✓ RS-U1-CQC RS-U2-CQC RS-U2-CQC DL SDL WL 	Type F Spectrum Spectrum Spectrum Load Case T Load Case (Eactor 1.30 1.30 1.20 1.20 1.20 1.22 1.20 1.28
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case ALL a	 Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL 	Type F Spectrum S Spectrum (Load Case Load Case (Factor 1.30 1.20 1.20 1.20 1.22
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Combined Spectrum Analysis Case (< ALL 2 ALL 2	 ✓ RS-U1-CQC RS-U2-CQC RS-U2-CQC DL SDL WL 	Type F Spectrum Spectrum Load Case Load Case Coad Case Coa	Eactor 1.30 1.30 1.20 1.20 1.20 1.20 1.20 1.20 1.20 1.20 1.20 1.20
1.2DL+0.28WL+1.3(EQX+EQY)+0.5EQZ Linear Analysis Applied Loads General Load Cases: Type: Combined Load Case Combined Spectrum Analysis Case (< ALL ALL	 Loads Applied Name RS-U1-CQC RS-U2-CQC RS-U3-CQC DL SDL WL 	Type F Spectrum Spectrum Spectrum Load Case T Load Case C	Eactor 1.30 1.30 1.20 1.20 1.20 1.22 1.28

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 $E=2.034 \times 105$ 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.

88.96 kN	22	W24X110		23	W24X110		24		Roof
66.72 kN	W14X176	28) W24X110	W14X211	1 4 20	35 w24x110	W14X176	QD 21	3.9624m	Level 7
55.60 kN	W14X176	2)) W24X110	W14X211	13 17	34) W24X110	W14X176	20 18	3.9624m	Level 6
44.48 kN	M14X211	26) W24X130	W14X246	12 14	33 W24X130	W14X211	19 15	3.9624m	Level 5
33.36 kN	M14X211	25) W24X130	W14X246	1) 11	32 W24X130	W14X211	(18) 12	3.9624m	Level 4
22.24 kN	- W14X246	24)) W24X160	W14X287	198	3D W24X160	W14X246	17 9	3.9624m	Level 3
11.12 kN	W14X246	23) W24X160	W14X287	9 5	30 W24X160	W14X246	16 6	4.1148m	Level 2
	W14X246	22) 9.144m	J W14X287	8 2 77	29 9.144m	W14X246	15 3	4.1148m	Ground
	\vdash								

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

Analysis Cases				×
Show: ALL	•	Num. of	Items: 0/0	\$
Name		ID	Туре	Run
Edit			Set Run Flag	
Add Rename	Linear Analysis Nonlinear Analysis Modal Analysis		Run / Not Ru	un VI Run
Use Processors	Eigen-Buckling Analysis Response Spectrum Analysis Time History Analysis	ŀ	Run Now	ОК

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

nine nia		Jynamic Function / Addition	iai Mass Initiai Loads	
Name:	Nonlineartime	history	Use Modes from Modal Ana	alysis Case:
Type:	Second-order	Analysis + Design 🔹 🔻		-
٥L	umped Mass	Consistent Mass	Integration Method:	
	1 k T	0	Newmark	Settings
lime	History Type		Damping	
O Di	rect Integration	Mode Superposition	a[M]+b[K]	Settings
Time	History Motion	Гуре	Time Step	
Tr	ansient	Periodic	Total Time Steps:	400
_		ה	Time Increment (sec.):	0.02
Nonli	inear Settings	J		

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:

Numerical Method		×
Methods Newmark Wilson Hilber-Hughes-Taylor	Gamma: Beta:	0.5 0.25
Hilber-Hughes-Taylor	ОК	Cancel

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

Damping		×
Rayleigh Damping		
🔘 Direct Input	1st Period	1.2732
Calculate by Period	2nd Period	0.4313
Calculate by Frequency	Damping Ratio1	0.05
	Damping Ratio2	0.05
	ОК	Cancel

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

Nonlinear Settings			×
PEP Element O Curved	Stability Function	Numerical Method Newton-Raphson (Constant Load) Method	
 Enable Plastic Advanced Ar Plastic Hinge 	nalysis) Plastic Element	Single Displacement Cont (Constant Disp.) Method Arc Length Method + Mini Residual Displacement Method	rol imum ethod
Number of Cycles for Each Time Step :	50	Iterative & Incremental Param	ieters :
Target Load Factor :	1	Incremental Load Factor :	0.02
Maximum Iterations for Each Cycle :	100		
Number of Iterations for Tangent Stiffness Matrix :	1		
Minimum Member Imperfection	L / 1000 : 0		
Imperfection Method & Direction	No Imposition	n of initial imperfection	•
Advanced		ОК	<u>C</u> ancel

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.

Time History Analysis	Dynamic Fur	ction / Additiona	al Mass Initial	Loads	
Dynamic Function:					
EL-Centro +	Name EL-Centro	Scale Fac 20.00	Arrival Time 0.00	Seismic Di 0 (Horizon	Excitation 0.00
Load Cases:		Add	itional Mass:		
		1.00 ▼ Na	ame intMass	Type Load Ca	Factor ase 1.00
		ALL >> ALL <<	Select load additional LL	case(s) to mass such	consider as SDL,
Note: The gravity di taken as additional r	rection is globa nasses.	I -Z-axis. Only th	e loadings in th	e gravity directi	ion will be

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:
Time History Analysis Dyn	amic Function / Additi	onal Mass Initial	Loads	
Dynamic Function: D	ouble click the	name		
EL-Centro + EL-	ne Scale Fac. Centro 20.00	Anival Time 0.00	Seismic Di 0 (Horizon	Excitation 0.00
	ynamic Function			
Load Cases:	Name:	EL-Centro		
	Scale Factor:	20		Factor
	Arrival Time:	0	C	ase 1.00
	Seismic Direction			
	Horizontal	Vertical		
	Excitation Angle	: 0		
Note: The gravity dire taken as additional ma	z		d	tion will be
	×	y		Apply
	L			

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* ...;

	-			_						
Analysis ⊇ ∢	Post Tools Window Internal Use Help Image: Show Deformed Shape Show Undeformed Shape Show Deformed & Undeformed Image: Show Deformed & Undeformed Display Scale Show Analysis Case Show Result Files Show Result Files		Help	IY IZ 12 43 Load Factor: 1.0000 ► ■ ■ 12 ×× %						
		Nodal Res Member I Spring Re Shell Resu Export Su Export Sta Export Eig Export An	sults Results sults ilts mmary of A itistics of Ar gen-Buckling imation AV	nalysis Results . nalysis Results g Load Factors .	• • •	List Noc List Noc Show N Toggle Load De Show Se	dal Displaceme dal Reactions lodal Displacer <u>Reactions/Dis</u> eflection Curve eleted Displace	ents ment Conto <u>placements</u> e/Reactions ements/Rea	ur <u>Dialog</u> ctions	

or press the button of post toolbar as below;

177 L 🔽 🚥 🗒 🗖	- ∽[⊦ ₩	Z 🔀	탑	•
	Show L	oad-Deflec	tion Curv	/e/React	tion

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...*;

<u>A</u> nalysis <u>P</u> o	st Tools <u>W</u> indow Internal Use <u>H</u> elp				
Q 🖸 🔽	Show <u>D</u> eformed Shape	IY IZ 🖄 🙀			
•	Show <u>U</u> ndeformed Shape	Load Factor: 1.0000 🕨 🔳 💽 🍋 🛰 🐙			
	Show Deformed & Undeformed				
	Display Scale				
	Show <u>A</u> nalysis Case				
	Show <u>R</u> esult Files				
	Nodal Results				
	Member Results	List Member Internal Forces & Moments			
	Spring Results	Member Load Deflection Curve			
	Shell Results	Show Section Capacity Factor Statistics			
	Export Summary of Analysis Results	Show Bending Moment/Shear Force Diagram			
	Export Statistics of Analysis Results	✓ Toggle Show Plasic Hinge			
	Export Eigen-Buckling Load Factors				
	Export Animation AVI				

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.