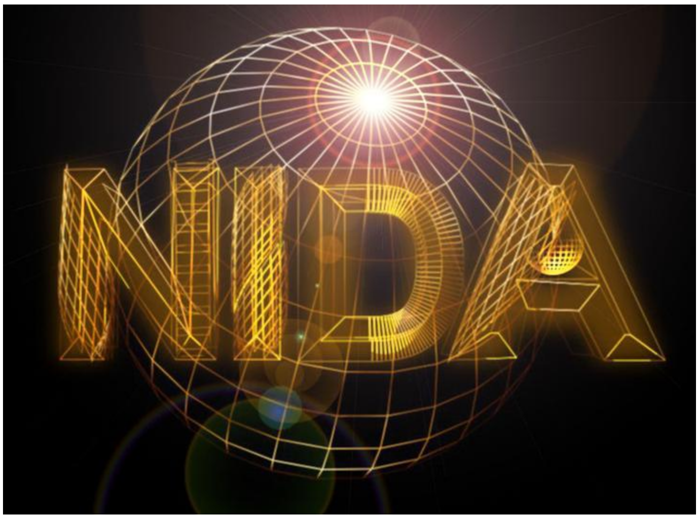
Welcome to NIDA Version 9.0

NIDA - Nonlinear Integrated Design and Analysis

User Manual and Analysis Theory



Since 1996

Second-Order (Direct) Elastic & Plastic Analysis

To

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

Without Assumption of Effective Length

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

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

Table of Contents

1. DESCRIPTION

1.1 Introduction

In current most widely used practice for structural design, an analysis is carried out to determine forces and moments of all members in a frame under external loads. The resistance of the members is then determined using the formulae in design codes and compared with these moments and forces due to factored loads. When the resistance is larger than the applied forces and moments, the structure is deemed to be adequately safe. This process is illustrated in Figure 1.1.

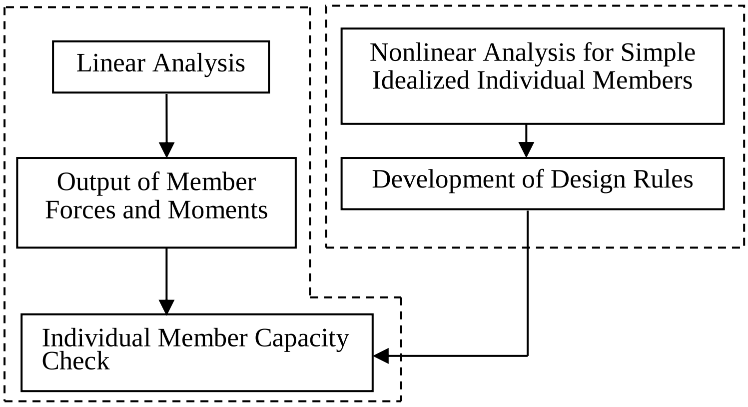


Figure 1.1 Conventional Design Procedure

In this conventional process, the analysis is linear, but the actual behaviour of the structure is non-linear and buckling occurs before reaching the squash or yield load. Many realistic effects such as buckling and material yield are not considered in analysis. The checking of buckling strength of members by design formulae is carried out at a latter and independent stage using the design code formulae. These two steps are separated but, buckling, instability and second-order effects influence the stiffness and thus the force and moment distribution in the whole structural system. The linear analysis program is therefore incorrect in force and moment computation. Furthermore, error due to the effective length assumption used in design codes makes the final design output unreliable and inaccurate.

For slender members and most steel or metal structures, instability or second-order effects will amplify the stress calculated from a linear analysis, so that the conventional method is either uneconomical or exceeding the adequate safety margin. Various design codes (e.g. Eurocode-3 [2005], AISC-LRFD [2010], and CoPHK [2011]) require the designer to include the P-Δ effect or stability check in the analysis and design when the simplified formulae in codes cannot cover the effects.

“Advanced analysis” is a new design method for steel structure in which the first-order and second-order or buckling effects are included, together with the characteristics of realistic structures, so that section capacity check as presented below is adequate for strength design.

$$\frac{P}{p_y A} + \frac{(M_y + P\Delta_y + P\delta_y)}{M_{cy}} + \frac{(M_z + P\Delta_z + P\delta_z)}{M_{cz}} = \phi \le 1 \tag{1.1}$$

where

Δ	Displacement due to sway of the frame measured at nodes
δ	Displacement due to member curvature or bowing, measured along a member
P	Axial force in member
A	Cross sectional area
p_y	Design strength
M_{cy}	Moment capacity about principal y-axis
M_{cz}	Moment capacity about principal z-axis
M_y	External moments about principal y-axis
M_z	External moments about principal z-axis
φ	Section capacity factor. If φ >1, member fails in section capacity check.

The method has been stated in Eurocode-3 (2005), AISC-LRFD (2010) and CoPHK (2011) and well-researched (Chen and Chan 1995). Using this more rigorous approach, we can directly compute the applied sectional moments, forces or their resulting stress due to factored loads and compare against the structural resistance. In doing so, we can then skip the complex and unreliable process of checking individual member buckling strength based on simplified boundary and loading conditions. This better process is summarised in Figure 1.2.

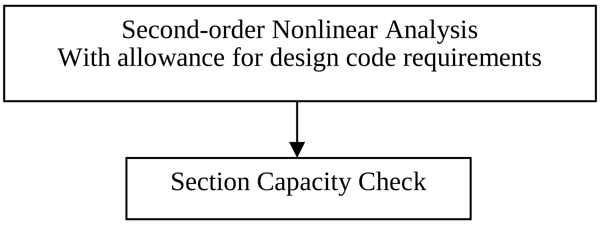


Figure 1.2 Nonlinear Analysis Design Procedure

This program is aimed for this type of second-order non-linear analysis and the design meeting the code requirement for strength and stability design. When using the plastic modulus in input sectional properties, it follows the most widely used "first-plastic-hinge design concept" that the design load is obtained as the load causing the formation of the first plastic hinge. When the elastic modulus is used for input sectional properties, the analysis becomes an elastic analysis and design. The effect of effective length is automatically and accurately considered in the computer program and need not be assumed as the P-δ and the P-Δ moment simulating the effective length is automatically determined from the buckling analysis of the frame. In reality, in many practical situations, the effective length can hardly be determined (For example, the effective length of scaffolding, domes and many types of steel frames). The use of the linear analysis computer programs may be dangerous in actual design.

More details about nonlinear analysis can be found in the papers and book by Chan and Chui (2000), and Chan (2001).

For design purpose, the present program adopts the Eurocode3 (2005) and the CoPHK (2011) design approaches where the initial imperfection is obtained from Table 5.1 in the Eurocode3 (2005) and Table 6.1 in the CoPHK (2011). For design of cold-formed and welded sections, the initial imperfection is amplified to simulate the effects of residual stress.

This program contains conventional beam-column, shell and cable elements.

1.2 Features of NIDA

The present program, **NIDA**, is the ninth commercial version for non-linear analysis and combined design and analysis of frame, cable and shell structures.

At the moment of preparing this manual, NIDA contains the linear analysis, geometric and material non-linear static and dynamic analysis, response spectrum analysis, eigenvalue buckling and vibration analysis of framed and shell structures.

Special features contain second-order P-Δ-δ analysis with considerations both of global and local initial imperfections, semi-rigid connections, load and construction sequences and cable structures.

The theory of the program can be found in Chan and Zhou (1995), Chan and Chui (2000) and Chan (2001). For non-linear solution method, Chan (1988) should be referred to. The program follows the concept associated with the ultimate limit state design code with the "first-plastic-hinge" limit load criterion used in the AS4100 (1990), BS5950 (1990, 2000), Eurocode-3 (2005), AISC-LRFD (2010),and CoPHK (2011).

1.3 Important Disclaimer

Considerable care has been taken to ensure the accuracy of the program. Nevertheless, responsibility for the use of the program rests with the user and the authors will not be responsible for any kind of damage caused by the use or misuse of the computer program.

1.4 Program Limitations

The program checks the lateral-torsional buckling of a beam using the code effective length method in which the effective length for beams can be easily assumed using code formulae whereas the effective length for columns is very much more complicated to assume. Also, the local plate buckling will be checked by the effective strength method. The effects of eccentric connection in unsymmetrical sections like angle and channel are ignored in default and need to be considered manually if these effects are significant.

1.5 Assumptions

The analysis program is based on the second-order non-linear elastic and plastic, displacement based stiffness method of analysis. The right hand rule is assumed as the sign convention for the whole computer program.

1.6 Copyright

The computer program NIDA is proprietary and copyrighted products. Worldwide rights of ownership rest with Professor S.L. Chan. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Professor S.L. Chan is explicitly prohibited.

Several trademarks and registered trademarks have been introduced in this manual as follows:

Windows, Excel, Access : Trademark of Microsoft Corporation

AutoCAD : Registered Trademark of Autodesk, Inc.

SAP2000, ETABS : Registered Trademark of Computer and Structure, Inc.

STAAD : Registered Trademark of Bentley.

1.7 Installation of NIDA

The program can be used in Windows 9x, Windows 2000, Windows XP, Windows Vista and Windows 7 compatible.

The package available to you will contain a manual, a USB key for protection of the version of your own program and a CD.

Insert the compact disc to your computer and automatically the program, NIDA, is installed inside a directory you prefer. Alternatively, you can install the program by activating ***install.exe*** in the CD directory. During the installation process, it requires the directory, user name and company name. Click appropriate buttons until the whole installation process is completed which is indicated by a message on screen.

After installation of NIDA, you need to install the key driver for standalone version and the server of network version. For network version, the client computer need a system variable in "***Environment Variables***" named as **NSP_HOST** with the value of IP address of the server. For more details, the document "NIDA Installation – Step by step" can be referred to.

2. GETTING STARTED

This chapter includes different kinds of NIDA commands, providing a demonstration for the general process of computer model creation by NIDA.

Start a model by clicking the ***File*** *> New* command to access the New Model form.

Add structural objects from one of the built-in NIDA templates by clicking the ***File****>New Model from Templates*. See New Model from Template for detailed information.

Set the units and degrees of freedom of the model by clicking the ***Analysis****>Analysis & Design Parameters Setting*.

Use the commands available in the ***Construct*** group to define Material, Frame Section, Shell Section, Load Case , Combined Load Case, Combined Member and Group and to create Node, Member, Spring element, Floor and Area.

Use the ***Edit*** commands to modify the geometric properties of the model. There are two special commands, Mesh and Split member, providing certain operations. The *Edit > Mesh* command is used to mesh the areas, floors and shells. The *Edit > Split members* command can be used to break the member at intersections with selected members or selected Nodes.

Use the ***Gr-Assign*** commands to assign relative properties to the model, i.e. the properties for the members (e.g. Section, End condition), nodes (e.g. Boundary condition, Nodal spring), shells , floors and areas, and to assign different types of loading (e.g. Member Loads, Nodal Loads, and Floor Pressure).

If you want to read the input file of the model in text format, you can use the ***Tools*** *> Show Data File* command. You can directly edit and then save the data file. Please re-open the data file so that your change can be taken effect.

Use the ***Analysis*** *> Set Analysis Cases…* to define and modify the analysis cases

Use the ***Analysis*** *> Analysis & Design Parameters Setting…* to select the steel design Code, determinate the floor stiffness, and direction of Gravity and so on.

The current model can be run by clicking ***Analysis*** *>Run*. Several individual models can be run simultaneously by clicking ***Analysis*** *>Run a Batch Files*.

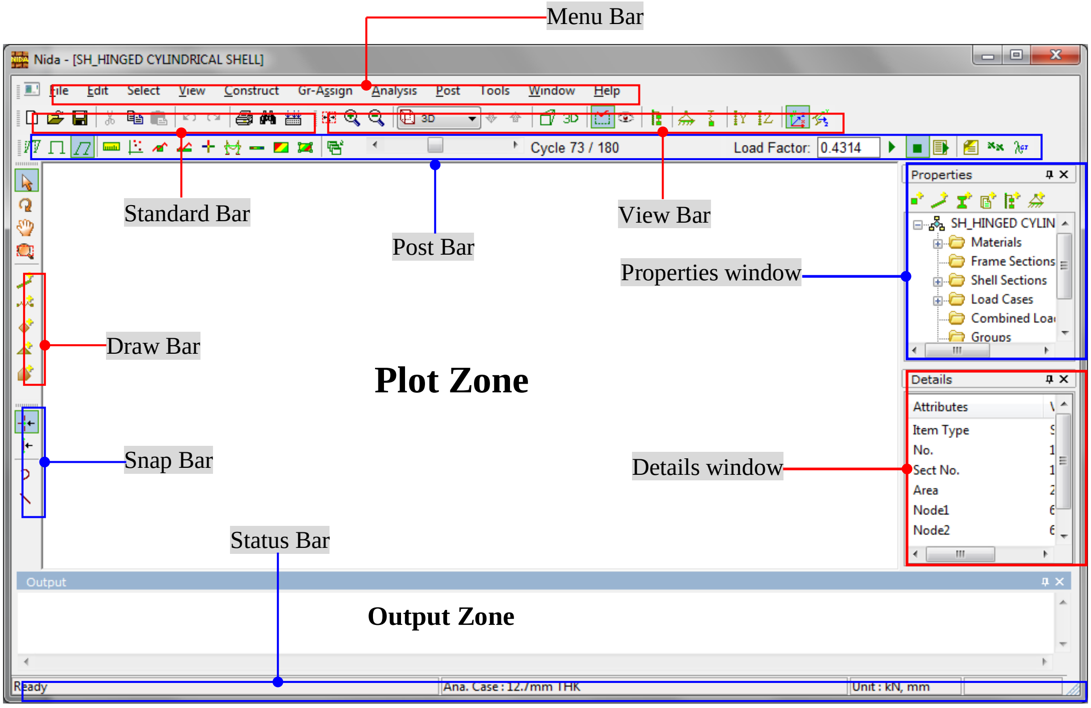
The deformed shape will be displayed automatically after the analysis is completed. The animation of the deformed shape or mode shape can be played by clicking the **Play** button in the Post bar. The cycle and speed of animation can be specified in the animation option bottom.

To view the result, click the ***Post*** commands to display analysis results on your model. The Nodal Reaction, Load Deflection Curve/Reaction, Member Statistics, Bending Moment/Shear Diagram and Shell Result can be generated and displayed on screen by applying corresponding functions.

Alternatively, use the **Post**>*Show Result Files* to show analysis results in text file.

3. GRAPHICAL USER INTERFACE

3.1 Main Screen



3.2 Menu Bar

The Menu Bar provides the most commonly used functions in NIDA. The details of each operation are provided in Menus Chapter.

3.3 Standard Bar

The Standard Bar includes shortcuts of certain functions (e.g. *New Project, Open Project, Save, …*) available in the Menu bar for the ease of structure construction.

3.4 Plot Zone

The drawing area visualizes the overall structure to the user. It mainly shows:

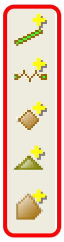
- the position of nodes,
- how objects are connected,
- the direction and magnitude of loadings,
- the direction of member local axis, and
- the presence of the boundary conditions, end releases, etc.

3.5 View Bar

The view bar provides shortcuts to view certain properties of the model easily, including:

- Fit to Screen
- Zoom In
- Zoon Out
- Set 2-D or 3-D View
- Perspective Mode
- Extrude View
- Show Selection Only
- Visible/Invisible
- Set Visibility of Loadings
- Show Boundary Condition
- Show Nodal Spring
- Show Moment Release in y-direction
- Show Moment Release in z-direction
- Show Global Axis
- Show Member Local Axis

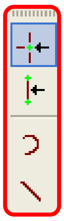
3.6 Draw Bar



The draw bar provides shortcuts to create certain objects easily, including:

- Add Member
- Add Spring Element
- Add Floor Element
- Add Shell Element
- Add Area Object

3.7 Snap Bar

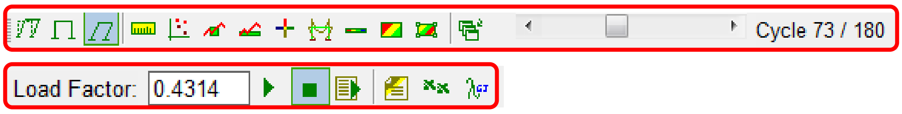


The snap bar provides shortcuts to select certain objects easily, including:

- Snap Node
- Snap Member
- Select by intersecting curve

- Select by intersecting line

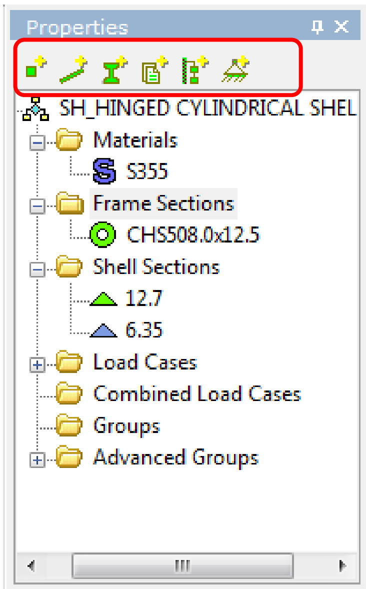
3.8 Post Bar



The post bar provides shortcuts of to show certain analysis results easily, including:

- Show Undeformed and Deformed Shape
- Show Undeformed Shape
- Show Deformed Shape
- Display Scale
- Show Section Capacity Factor Statistics
- Show Load Deflection Curve/Reaction
- Show Member Statistics
- Show Nodal Displacements/Reactions
- Show Bending Moment/Shear Diagram
- Section Capacity Color
- Show Nodal Displacement
- Show Shell Nodal Stress
- Select Analysis Case
- Select Load Cycle
- Play Animation
- Stop Animation
- Animation Options
- To Export Summary of Analysis Results
- To Export Statistics of Analysis Results
- Export Eigen-buckling Load Factors

3.9 Properties Window



In the top of the properties window, there is also a toolbar named “Add Bar”. It helps to add the following objects to the NIDA project.

- Nodes
- Members
- Sections
- Load Cases
- Loadings
- Boundary Conditions

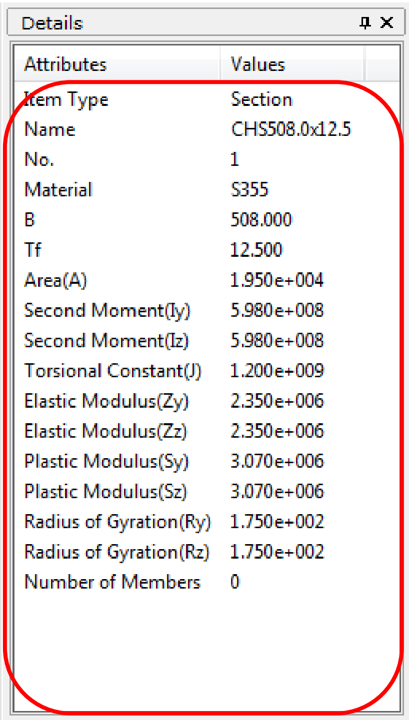
The Properties Window gives a systematic view of the project. It shows the main components of the structure in format of a tree.

The topmost level is the project name.

The components on the second level are :

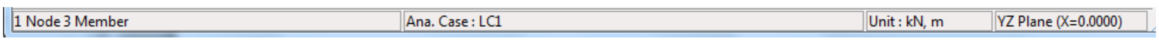
- Materials
- Frame Sections
- Shell Sections
- Load Cases
- Combined Load Cases
- Groups
- Advanced Groups

3.10 Details Window



The Details Window gives further details about the components of the project. When the user clicks on any object of the project, its own attributes will be available in this window immediately, providing a quick view to the user.

3.11 Status Bar



The status bar shows the following items:

- The number of objects selected or the status information about what the program is doing currently
- The activated analysis case
- The current units
- The information of 3D view or 2D view

4. MENUS

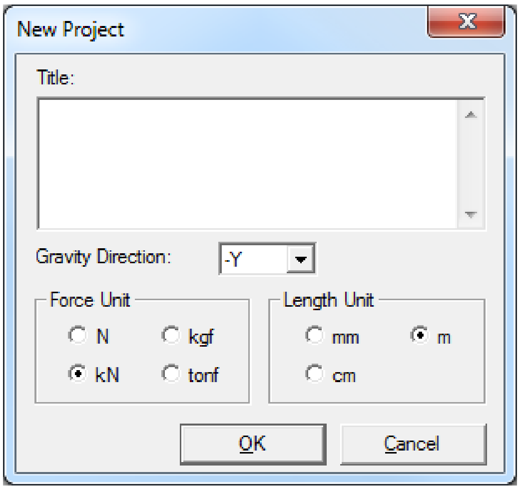
4.1 File

4.1.1 New

Click the ***File*** *> New* command to start a new model. Enter the project title in the window shown below.

Select the force unit, length unit, and direction of gravity.

Shortcut : Toolbar : Keys : Ctrl+N



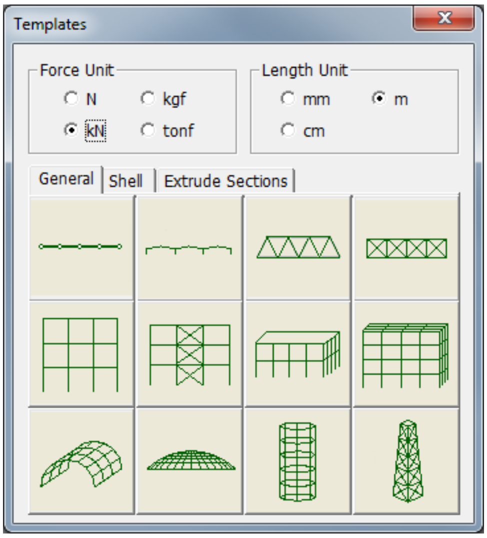
4.1.2 New Model from Templates

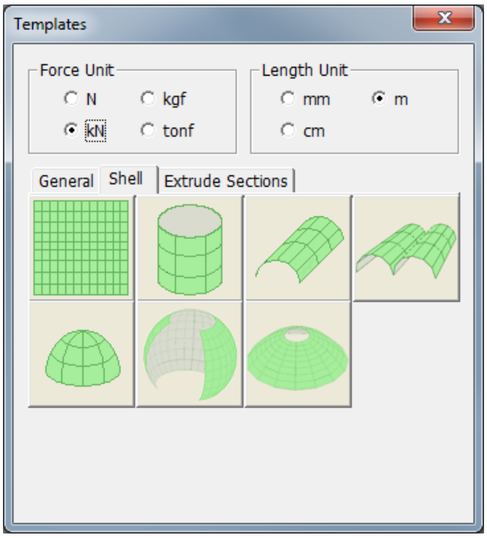
Click the ***File*** *> New Model from Templates* command to start a new model.

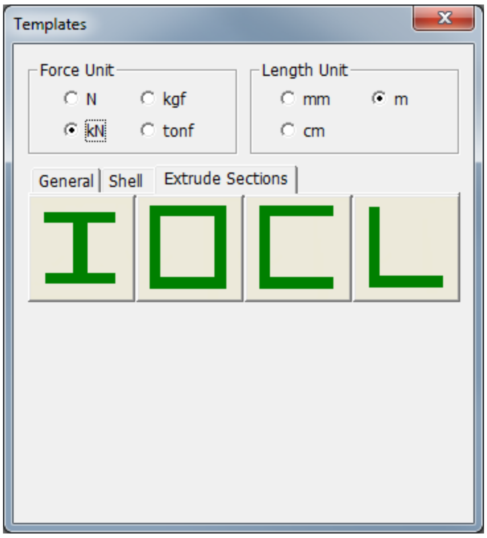
The templates contain a number of common structural forms to assist the user to prepare a data file quickly. The list of the structural forms will be displayed on the next page.

In current version, there are typical frame structural forms, several shell type structures as well as some extrude sections available for selection.

Click the one preferred and input necessary parameters, the structure will then appear on the screen.

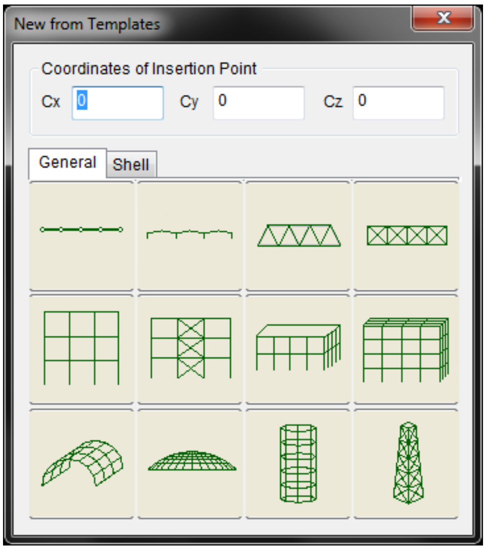






4.1.3 Add from Templates

Click the ***File*** *> Add from Templates* command to add a template model to the model being constructed.



Input coordinates of insertion point, click a template preferred and input necessary parameters, the structure will then appear on screen.

4.1.4 Open

Click the ***File*** > *Open* command to open an existing NIDA data file (*.dat). Select the directory where the NIDA file is located. The project will be loaded and the structure will be shown on the screen.

Shortcut : Toolbar :  Keys : Ctrl+O

4.1.5 Close

Click the ***File*** > *Close* command to close the current project.

4.1.6 Save

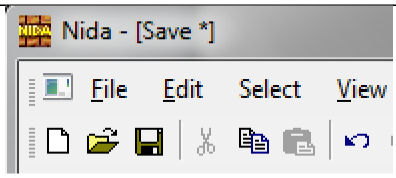
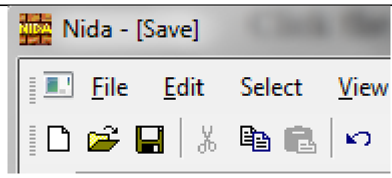
Click the ***File*** > *Save* command to save a NIDA model.

NIDA will provide a file path and name for a new model. If the model has been saved before, NIDA will save the file with the same filename in the same location, and the previous file will be automatically overwritten.

Once there has been a change in data file, there will be an asterisk “*” sign shown in the Window Title before saving.

After saving, the asterisk “*” sign will hide until the project is changed again.

Shortcut : Toolbar :  Keys : Ctrl+S

	
Once there has been a change in	After saving, the asterisk “*”

data file, there will be an asterisk "*" sign shown in the Window Title before saving.	sign will hide until the project is changed again.

Note: The model will be saved automatically when the analysis is running.

4.1.7 Save as

Click the ***File** > Save As* command to save the current project to a new location.

4.1.8 Import DXF

Click the ***File** > Import DXF* command to import an AutoCAD generated file, with a "**.dxf**" extension usually. Only nodal coordinates and connectivity can be imported. For sake of convenience, sections are exported/imported as layers.

Noted that the imported file must be further modified before NIDA can produce an analysis.

4.1.9 Import STAAD

Click the ***File** > Import STAAD* command to import an STAAD generated file, with a "**.std**" extension.

Noted that the imported file must be further modified before NIDA can produce an analysis.

4.1.10 Import ETABS (*.mdb)

Click the ***File** > Import ETABS (*.mdb)* command to import an ETABS generated file, with a "**.mdb**" extension.

Noted that the imported file must be further modified before NIDA can produce an analysis.

4.1.11 Import SAP2000 (*.mdb)

Click the ***File** > Import SAP2000 (*.mdb)* command to import a SAP2000 generated file, with a "**.mdb**" extension.

Noted that the imported file must be further modified before NIDA can produce an analysis.

4.1.12 Export DXF

Click the ***File** > Export DXF* command to export an AutoCAD generated file with a "**.dxf**" extension usually. Only nodal coordinates and connectivity can be exported.

4.1.13 Export BMP

Click the ***File*** *> Export BMP* command to export a Bitmap generated file with a "**.dmp**" extension usually.

4.1.14 Print

Click the ***File*** *> Print* to enter the printer setting page to select the printer, specify the range of pages to be printed, the number of copies, and set the page size, orientation and output format.

Shortcut : Toolbar :  Keys : Ctrl+P

4.1.15 Print Preview

Click the ***File*** *> Print Preview* to show the document which would appear when printed. The print preview window will appear and replace the main window when you select this command. View one or two pages at the same time, zoom in or zoom out the document, and start the print job.

4.1.16 Print Setup

Click the ***File*** *> Print Setup* to set the color, font, size and date information, and to select the information which is required to be included such as the company name, section, and material information.

4.1.17 Exit

Click the ***File*** *> Exit* to quit the program NIDA.

4.2 Edit

4.2.1 Undo

Click the ***Edit** >Undo* command to cancel last operation in model processing.

Shortcut : Toolbar :  Keys : Ctrl+Z

4.2.2 Redo

Click the ***Edit** > Redo* command to restore last action in model processing.

Shortcut : Toolbar :  Keys : Ctrl+Y

4.2.3 Cut

Click the ***Edit** > Cut* command to cut the selected objects from the current model.

Shortcut : Toolbar :  Keys : Ctrl+X

4.2.4 Copy

Click the ***Edit** > Copy* command to copy the selected objects (nodes, members, shells, floors, areas …) to the current model.

Shortcut : Toolbar :  Keys : Ctrl+C

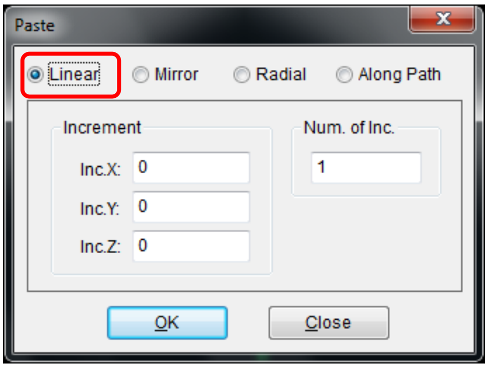
4.2.5 Paste

Click the ***Edit** > Paste* command to paste the selected objects to the current model.

Shortcut : Toolbar :  Keys : Ctrl+V

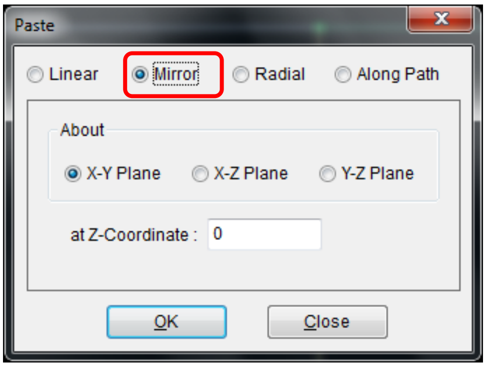
There are 4 methods to paste new objects: *Linear*, *Mirror*, *Radial, and Along Path*.

- Linear



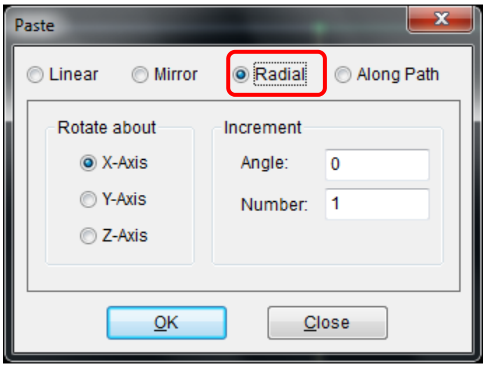
Enter the number of increments, increments along X, Y and Z axes. And then, click OK button, the new structural objects such as nodes, members, and shells will be shown on the screen.

- Mirror



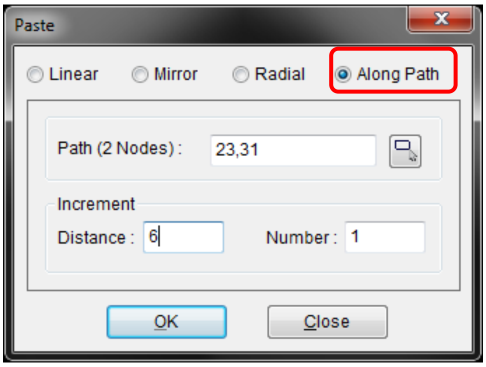
Select the X-Y, X-Z or Y-Z plane, and then input the coordinate of the corresponding axis. Click OK button and the new structural objects such as nodes, members, and shells will be shown on the screen.

- Radial



The increment angle is positive for clockwise. Click OK button and the new structural objects such as nodes, members, and shells will be shown on the screen.

- Along Path



Pick up two nodes as a path, put into the incremental distance, click OK button and the new structural objects such as nodes, members, and shells will be shown on the screen.

4.2.6 Paste Special

Click the ***Edit*** *> Paste special* command to paste the selected nodes, members, floors, shells or all objects from *Text/MS Excel file*, other *NIDA model* to the current NIDA one.

The number of objects pasted would be listed.

4.2.7 Delete

Click the ***Edit*** *> Delete* command to delete the selected objects from the current model.

- For Node, Member, Shell, Floor and Area

Select nodes, members, shells, floors and/or areas on the screen.

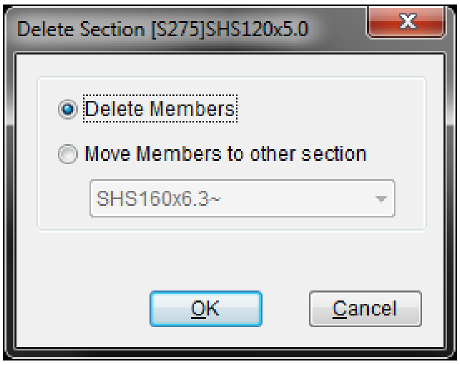
Click the ***Edit*** *> Delete* command or press <**DEL**> key on the keyboard.

- For Material

Select one of the materials from the Properties Window, right click and choose delete or press <**DEL**> key on the keyboard.

- For Section

Select one of the sections from the Properties Window. Press <**DEL**> key on the keyboard then choose "Delete Members" or "Move Members to other Section", see the options as below.

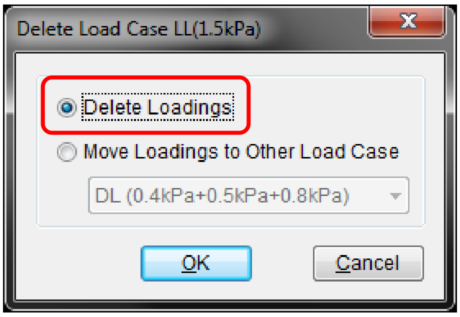


Press OK button.

- For Load Case

Select one of the load cases from the Properties Window.

Press <**DEL**> key on the keyboard and then choose "Delete Loadings" or "Move Loadings to other Load Case", see the options as below



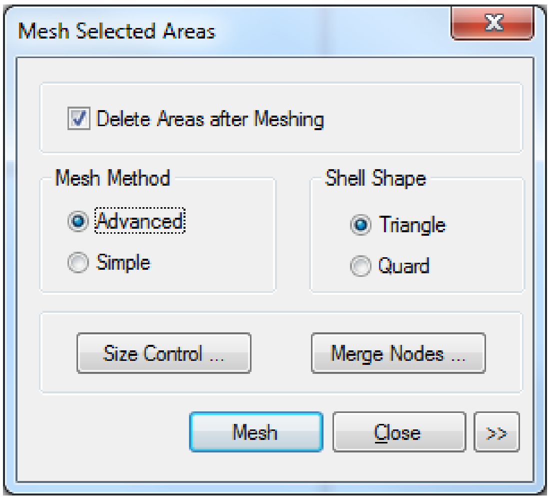
Press OK button.

- For other objects (Group, Combined Load Case, etc.)

Select the objects from the Properties Window or on the screen and press <**DEL**> key on the keyboard.

4.2.8 Mesh Areas

Click the ***Edit*** *> Mesh Areas* command to mesh the selected area objects into shell elements.



Delete Objects after Meshing option. The selected area objects would be deleted after meshing if this option is selected.

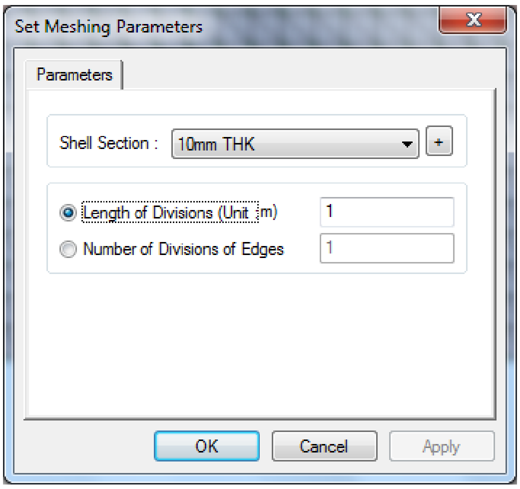
Mesh Method - Advanced option. A group of high quality shell elements would be created by using this option. This function is very useful for meshing the complex area objects.

Mesh Method - Simple option. A group of simple shell elements would be created by using this option. This function is suitable for meshing the regular area objects.

Shell Shape - Triangle option. The shell elements with triangle shape would be created by clicking this option.

Shell Shape - Quard option. The shell elements with quadrilateral shape would be created by clicking this option.

Normally, the objects to be meshed should be assigned the shell section and mesh size. If the user forgets to do the assignment or wants to mesh the objects with new meshing parameters, click **Size Control** button to set meshing parameters as below.

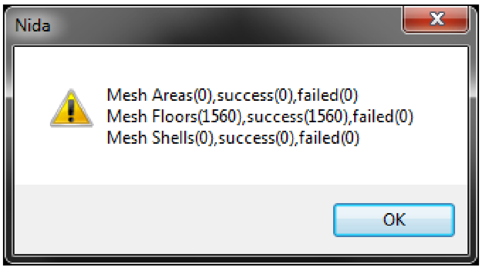


Shell Section option. Choose which kind of shell section to be meshed to.

Length of Divisions option. Each edge of selected objects will be divided into several parts by the assigned division length.

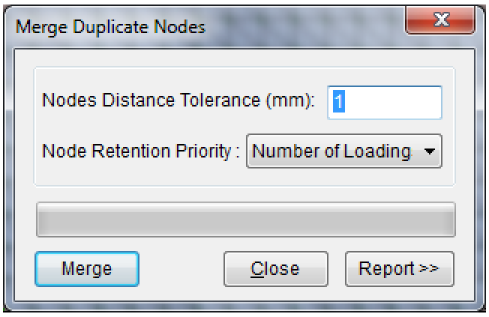
Number of Divisions of Edges option. Each edge of selected objects will be divided into N parts by the assigned number N.

After the appropriate options are selected, the meshing process can be started by clicking Mesh button.



Click OK button to return to the main window of Mesh. The detailed operation can be checked by clicking the >> button.

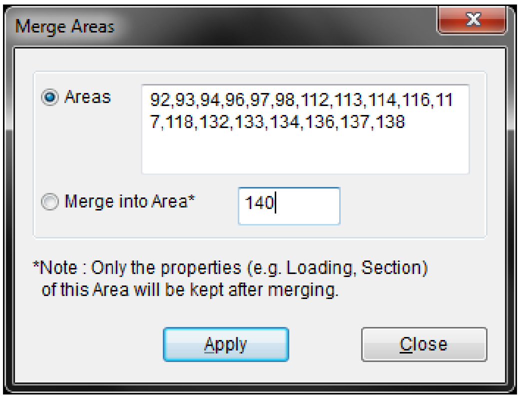
Click Merge button to merge duplicate nodes before meshing if needed.



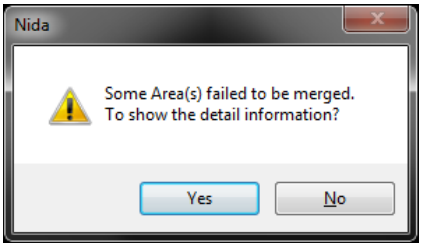
More details can be referred to [Merge Duplicate Nodes](#).

4.2.9 Merge Areas

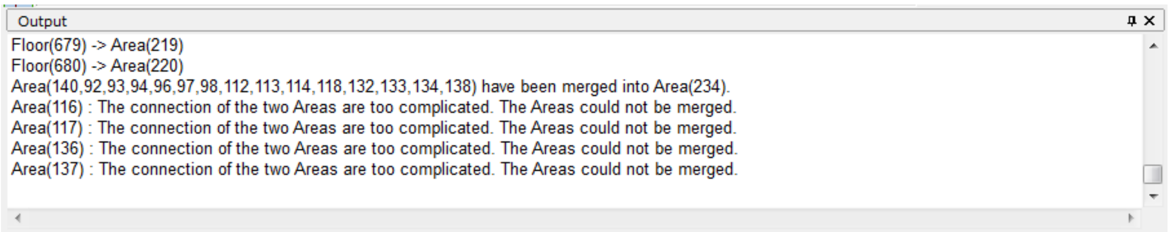
Click the ***Edit** > Merge Areas* command to merge the selected area(s) into one.



To merge Areas into the big Area*, the selected Areas should be adjacent to Area*. You may pick up areas on screen to input Areas or Area*.



Sometimes not all areas input can be successfully merged into one. You may click Yes button of the popup dialog to get the information shown in **Output Zone.**

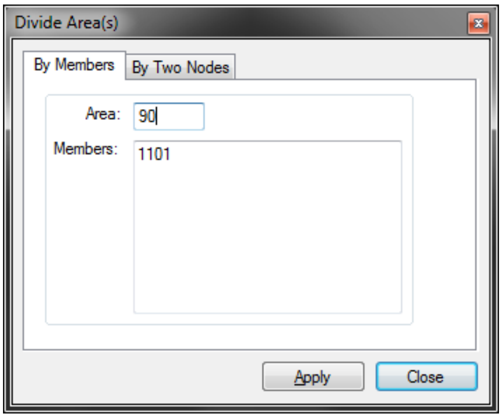


In more complicated case, more trials are required if the first merging operation fails.

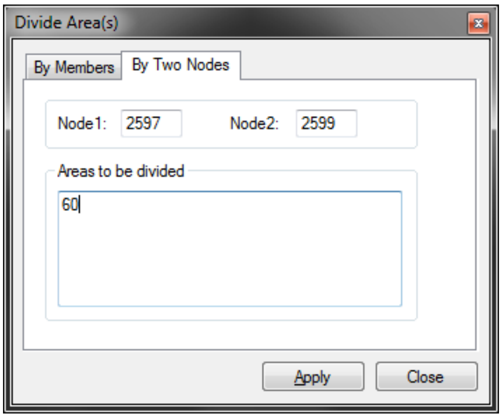
4.2.10 Divide Areas

Click the ***Edit*** *> Divide Areas* command to divide the selected area(s) into two parts by members of two nodes.

When By Member(s) is active, only one selected area can be divided by selected members.

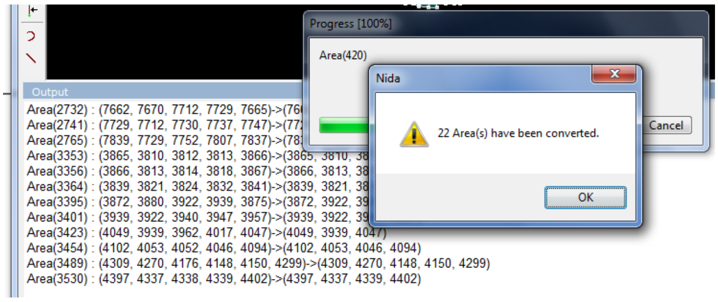


When By Two Nodes is active, several selected areas can be divided by the line linked by two selected nodes.

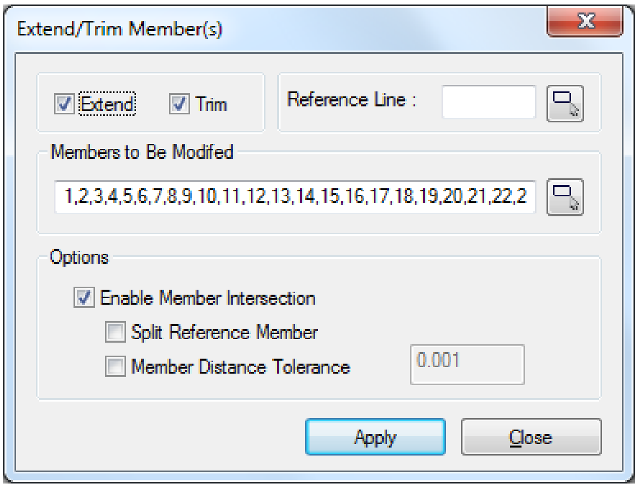


4.2.11 Drop Edge Nodes of Areas

Click the ***Edit*** *> Drop Edge Nodes of Areas* command to drop nodes (if any) of the edges of the selected areas.



4.2.12 Trim/Extend Members



Click the ***Edit*** *> Trim/Extend Members* command to modify members by trimming or extending them with the reference line which they are not parallel to.

User needs to input the reference line by selecting a member on screen. Further, user needs to select members for trimming or extending or both.

Enable Member Intersection

To enable splitting reference member at the intersection points (if any) between the modified members and reference member.

Split Reference Member

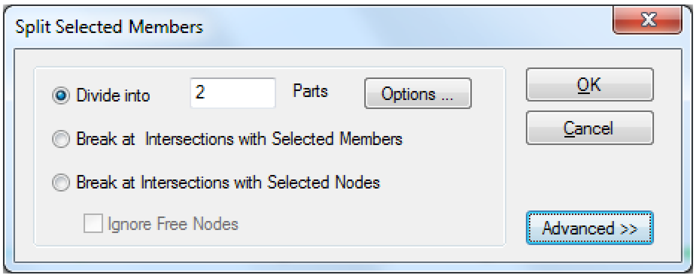
If this option is active, the reference member will be split into two members at the intersection point.

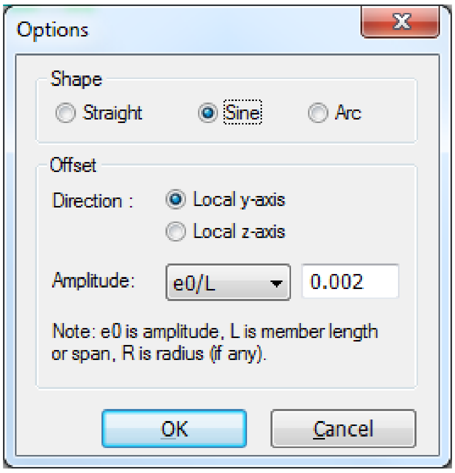
Member Distance Tolerance

This indicates to split the reference member only if the distance of two points perpendicularly between the modified member and reference member is less than the [*member distance tolerance*].

4.2.13 Split Members

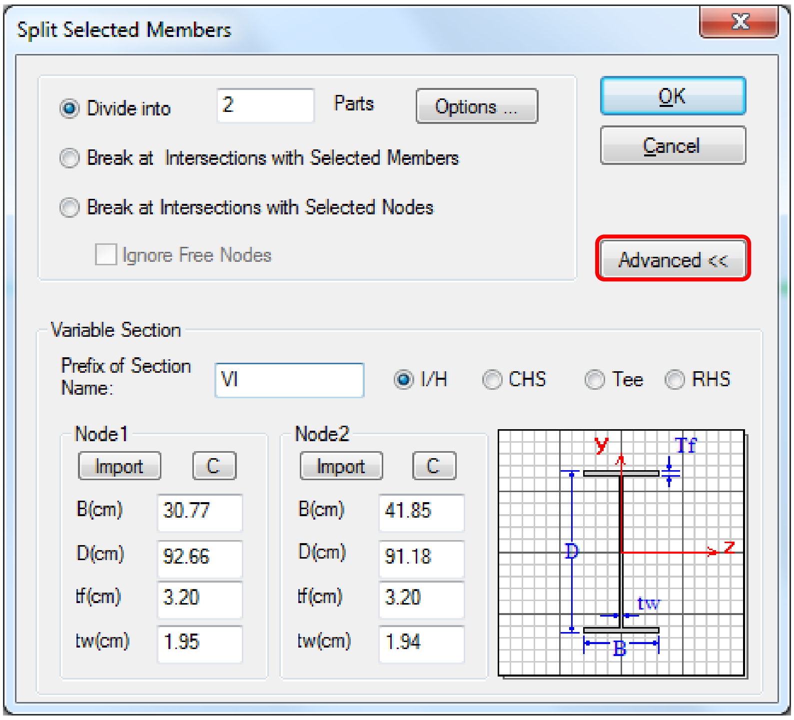
Click the ***Edit*** *> Split Members* command to split the selected members into **n** equal parts. In "Options", you can offset the member with a given value along member local y or z direction to specify the member imperfection manually or to create a curved member by several elements.





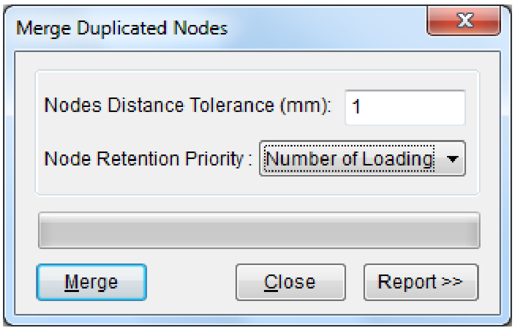
You can break the members at intersections with selected members or selected nodes.

In Advanced setting, the variable I, CHS, Tee and RHS sections can be simulated by a number of members with a series of uniform sections. The parameters of the section at two ends will be input and the section will be changed uniformly from the large section to a small one.



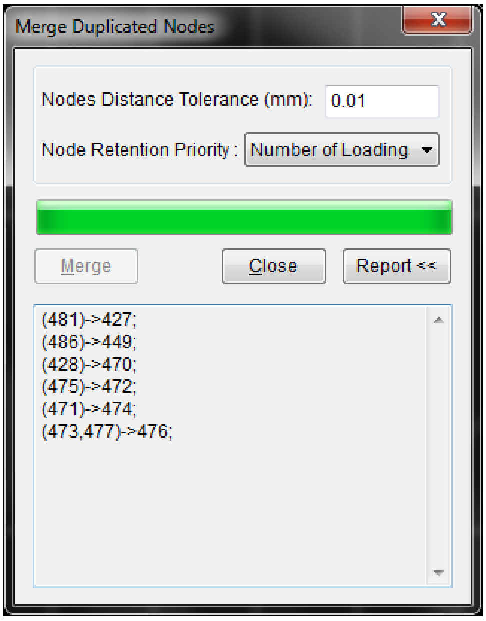
4.2.14 Merge Duplicate Nodes

Click the ***Edit*** *> Merge Duplicate Nodes* command to join those nodes with duplicate properties.



The **Nodes Distance Tolerance** value according to the minimum size and the units of the structure should be input before starting this action. After that, the nodes whose distances are less than [*Nodes Distance Tolerance*] will be merged to one.

The **Node Retention Priority** option will help to select the node to be remained automatically.



Click the Report button to show the details of the nodes merged already.

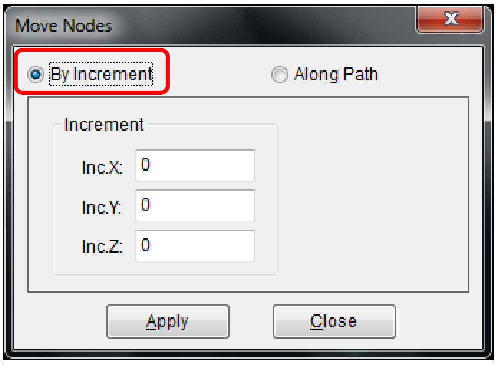
4.2.15 Move Nodes

Click the ***Edit** > Move Nodes* command to move the selected nodes (including the objects connected to the nodes) to the new position.

Shortcut : Toolbar : None Keys : Ctrl+M

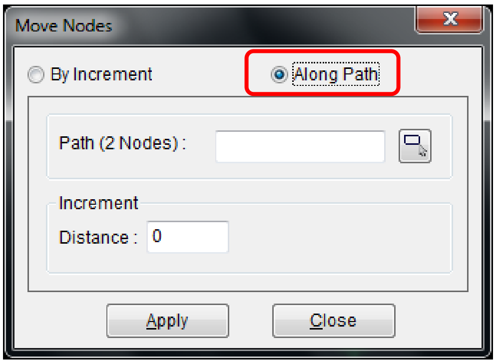
- By Increment

This option is to move the selected nodes to a new position by incremental coordinates in X, Y and Z axes with respective to the original one.



- Along Path

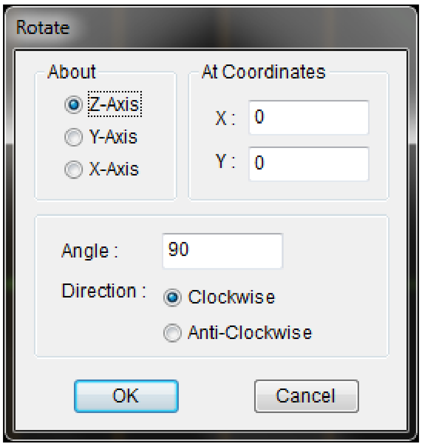
This option is to move the selected nodes along a path defined by 2 nodes.



Note: The connected objects by the nodes will be changed after this operation.

4.2.16 Rotate Nodes

Click the ***Edit*** *> Rotate Nodes* command to rotate the nodes (including the objects connected to the nodes) about X, Y or Z axis.



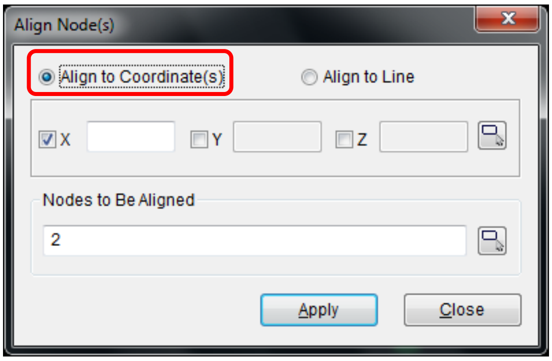
User needs to select an axis and fill in two coordinates to determine the reference axis. And then fill in the angle and select the direction to rotate. After that, the selected objects will be re-generated by rotating the reference axis with the angle.

The original objects will be deleted forever.

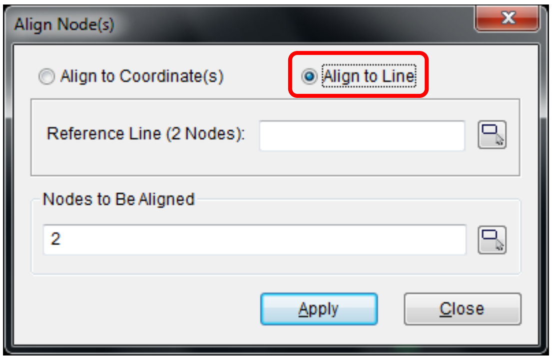
4.2.17 Align Nodes

Click the ***Edit*** *> Align Nodes* command to align selected nodes (including the objects connected to the nodes) to X, Y and/or Z coordinates or lines.

Align to Coordinate(s) option. The command is to align the selected nodes to the given coordinate(s).

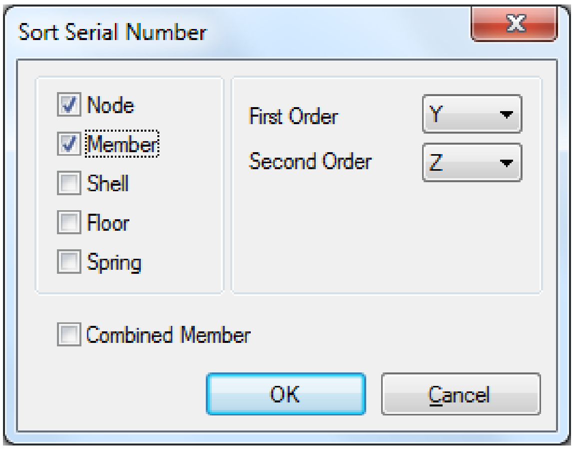


Align to Line option. The command is to align the selected nodes to the reference line.



4.2.18 Sort Serial Numbers

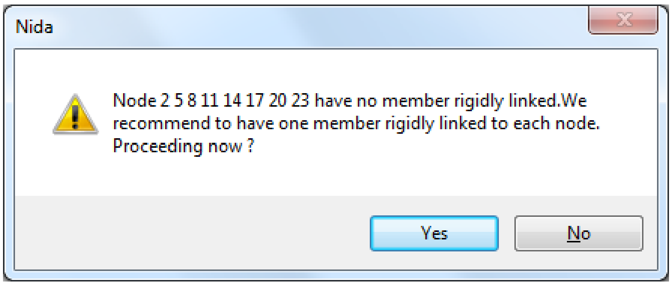
Click the ***Edit*** *> Sort Serial Numbers* command to re-sort the number of node, member, shell, floor and spring elements according to their coordinates.



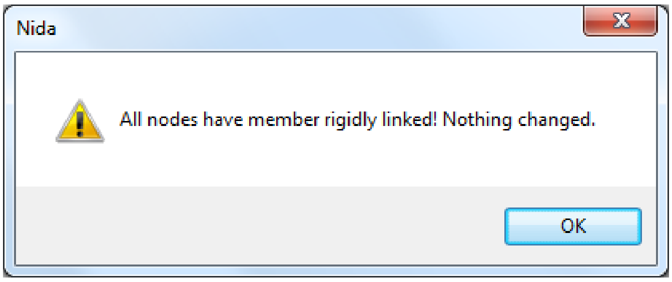
4.2.19 Check Node for Rigid Body Rotation

Click the ***Edit*** *> Check Node for Rigid Body Rotation* command to check if there are any nodes which have not been rigidly linked.

If some of the nodes have not been rigidly linked, they will be shown as below.



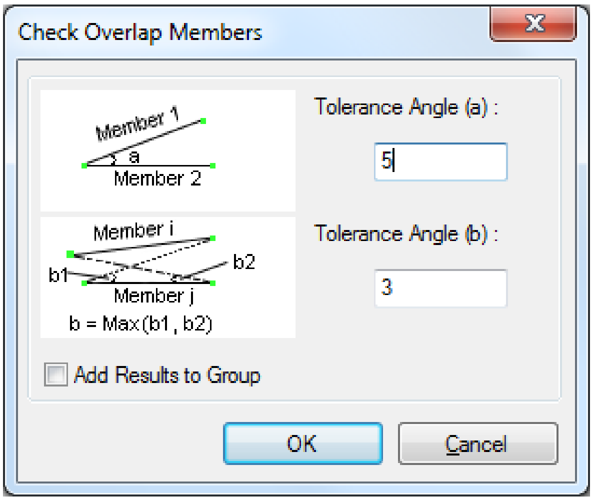
If all nodes have been rigidly linked, then a message dialog will pop up as below.



Important: Once some of the nodes have not been rigidly linked, the stiffness matrix may be singular and the structure will be unstable, and the stability problem may occur in the process of analysis. This checking is very important for nonlinear analysis.

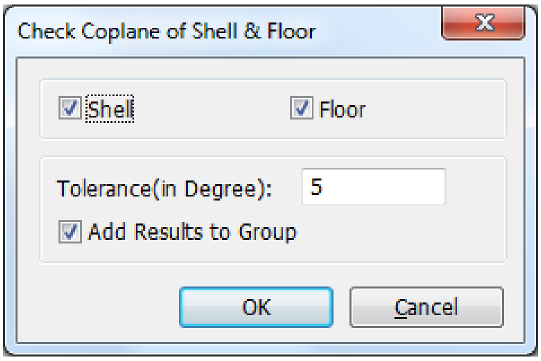
4.2.20 Check Overlap Members

Click the ***Edit*** *> Check Overlap Members* command to check if there are any members overlapping each other. The overlapped members can be stored in group for further operation.



4.2.21 Check Coplane of Shell & Floor

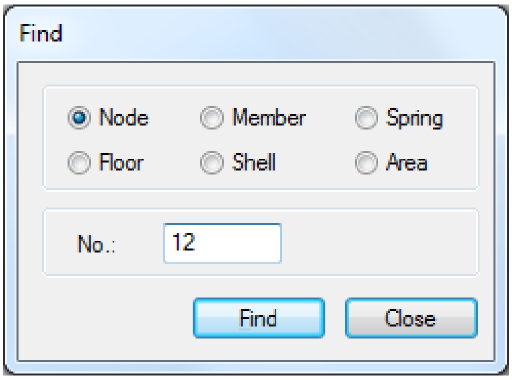
Click the ***Edit*** *> Check Coplane of Shell & Floor* command to check if there are any four nodes of shell and/or floor elements not in the same plane. The shell and/or floor elements with noncoplanar nodes can be stored in group for further operation.



4.2.22 Find

Click the ***Edit** > Find* command to find the node, member, shell, spring, floor and area objects by their unique numbers in the project quickly.

Shortcut : Toolbar : None Keys : Ctrl+F

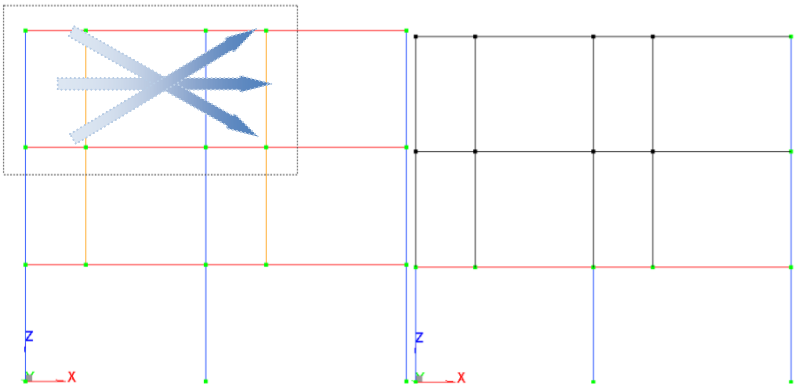


Input the No. of a certain kind of objects (e.g. Node, Member, Shell, Spring, Floor and Area) and click Find button. If the number is valid, the corresponding object will be highlighted on the screen.

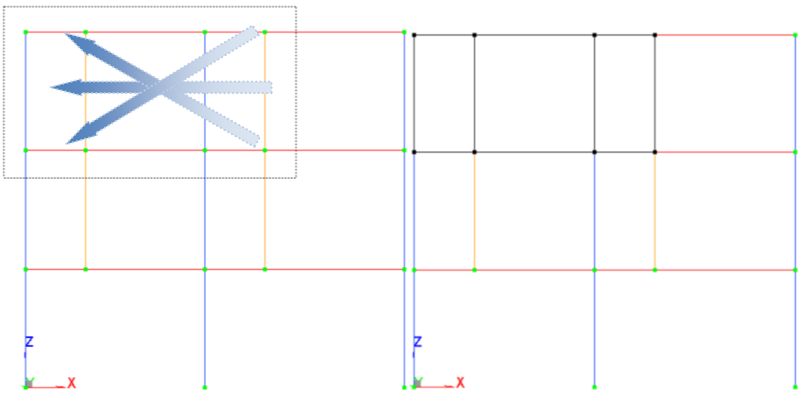
4.3 Select

When the ***Select*** function is on, left click and hold the mouse to enclose all the objects to be selected before releasing. There is a general rule regarding the direction to which the mouse moves.

When the mouse moves rightwards following the paths like the arrows below, objects with any part enclosed by the dotted line will be selected, indicated in black.



When the mouse moves leftwards following the paths like the arrows below, only objects with all nodes completely enclosed by the dotted line will be selected, indicated in black below as well.



To multi-select or add objects in addition to the existing selection, press the ***Ctrl*** key while undergoing the ***Select*** procedure;

Or to unselect any one of a group of objects, press the ***Ctrl*** key and click on the objects to be unselected.

4.3.1 Select All

Click the ***Select*** > *Select All* command to select all objects of the project.

Shortcut : Toolbar : None Keys : Ctrl+A

4.3.2 Deselect

Click the ***Select*** > *Deselect* command to deselect the objects which were selected on the screen.

Shortcut : Toolbar : None Keys : Ctrl+D

4.3.3 Inverse

Click the ***Select*** > *Inverse* command to inverse the selection so that the unselected object can be selected and the selected object becomes unselected.

Shortcut : Toolbar : None Keys : Ctrl +Shift +I

4.3.4 Nodes by Boundary Condition

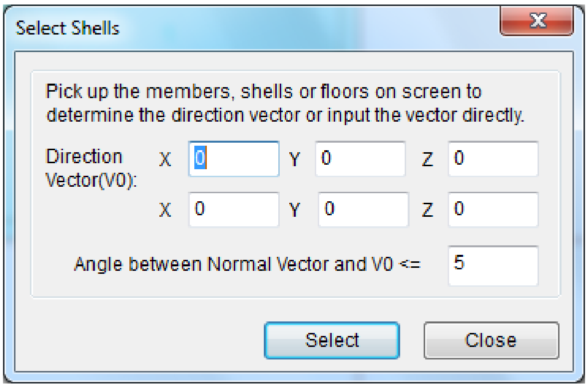
Click the ***Select*** > *Nodes by Boundary Condition* command to select the nodes by typical boundary conditions such as Fixed, Pin, Roller, Fully Free.

4.3.5 Members by Plastic Hinge

Click the ***Select*** > *Members by Plastic Hinge* command to select the members which are allowed for plastic hinges or those are not allowed for plastic hinges by clicking “Allowed” or “Disallowed”.

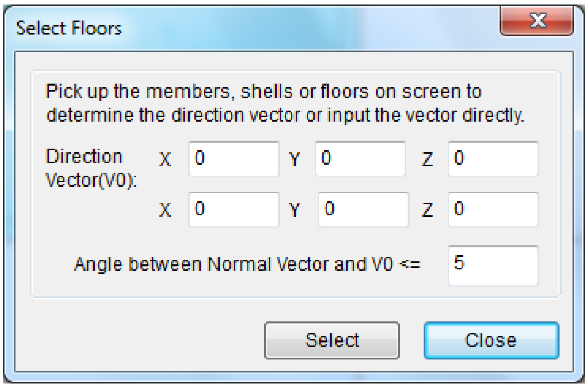
4.3.6 Shells by Normal Direction

Click the ***Select*** > *Shells by Normal Direction* command to select the shells by normal direction. The direction vector can be defined by picking up members, shells or floors on the screen.



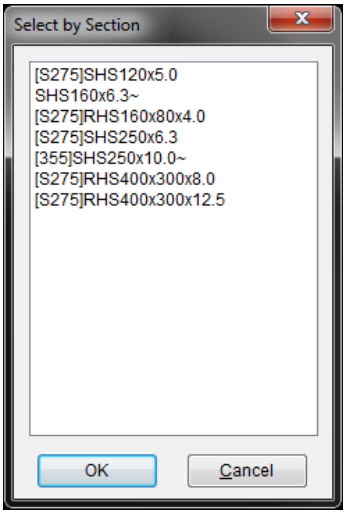
4.3.7 Floors by Normal Direction

Click the ***Select*** > *Floors by Normal Direction* command to select the floors by normal direction. The direction vector can be defined by picking up members, shells or floors on the screen.



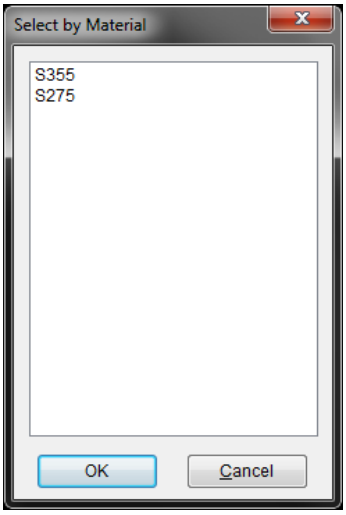
4.3.8 Sections

Click the ***Select*** > *Sections* command to select members by section name.



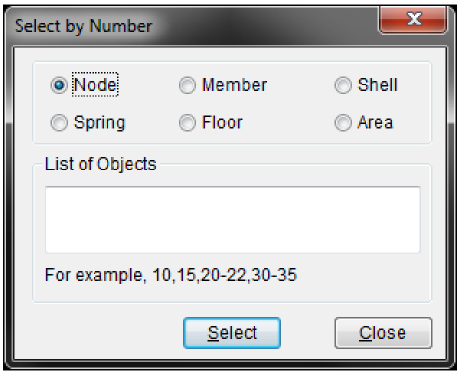
4.3.9 Materials

Click the ***Select*** > *Materials* command to select members by material name.



4.3.10 Multi-select Objects

Click the ***Select*** *> Multi-select Objects* command to select the specified range of objects by number.

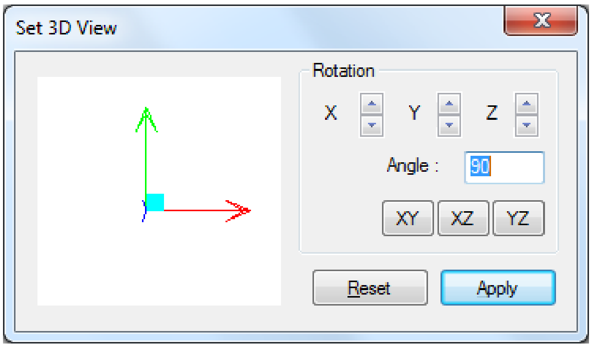


Select the object type and input the number of the objects or pick up them from the screen. Click the Select button and the selected objects will be highlighted on screen.

4.4 View

4.4.1 Set 3-D View

Click the ***View*** > *Set 3-D View* command to adjust the 3D view.



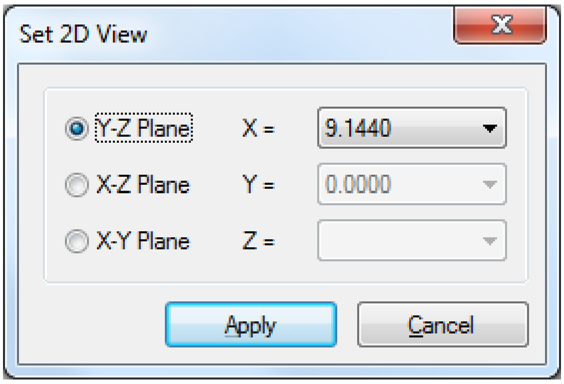
The X, Y or Z axis can be rotated with a specific angle for better view.

The 3-D view could also be set by clicking the  button in view bar.

Relative topic: [Set 2-D View]

4.4.2 Set 2-D View

Click the ***View*** > *Set 2-D View* command to set the 2D View.



The Y-Z, X-Z or X-Y plane could be selected by choosing an appropriate X, Y or Z coordinate.

The 2-D view in Y-Z, X-Z or X-Y plane could also be set by clicking the ,  or button in view bar.

Relative topic: [Set 3-D View]

4.4.3 Zoom

Click the ***View*** > *Zoom* command to enlarge a specific part of the model to view.

Shortcut : Toolbar : Keys : None

4.4.4 Zoom In and Zoom Out

Click the ***View*** > *Zoom In* or ***View*** > *Zoom Out* command to zoom in or zoon out the model on the screen.

Shortcut : Toolbar : Keys : +,- (in Num. keyboard)

4.4.5 Fit to Screen

Click the ***View*** > *Fit to Screen* command to redraw and fit the entire structure on the screen.

Shortcut : Toolbar : Keys : Alt +Z

4.4.6 Pan

Click the ***View*** > *Pan* command to move the entire structure on the screen. The mouse pointer will change to “Hand” sign.

Shortcut : Toolbar : Keys : Arrow (up, down, left, right)

4.4.7 Rotate

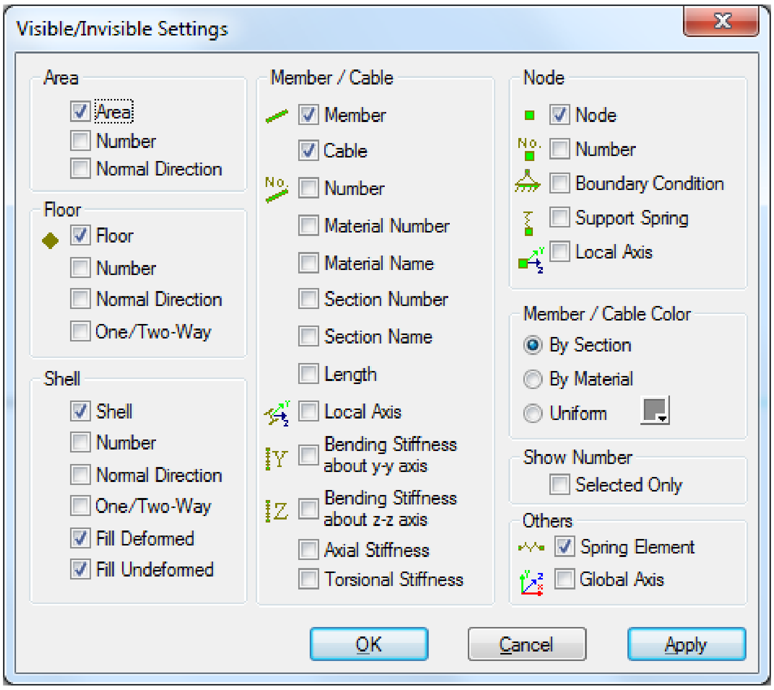
Click the ***View*** > *Rotate* command to rotate the entire structure on the screen. Semi-circular arrow will be shown on the plotted area for rotating the structure by moving the mouse.

Shortcut : Toolbar : Keys : A, S, D, W

4.4.8 Visible/Invisible

Click the ***View*** > *Visible/Invisible* command to select the different objects to be displayed on the screen.

Shortcut : Toolbar : Keys : F4



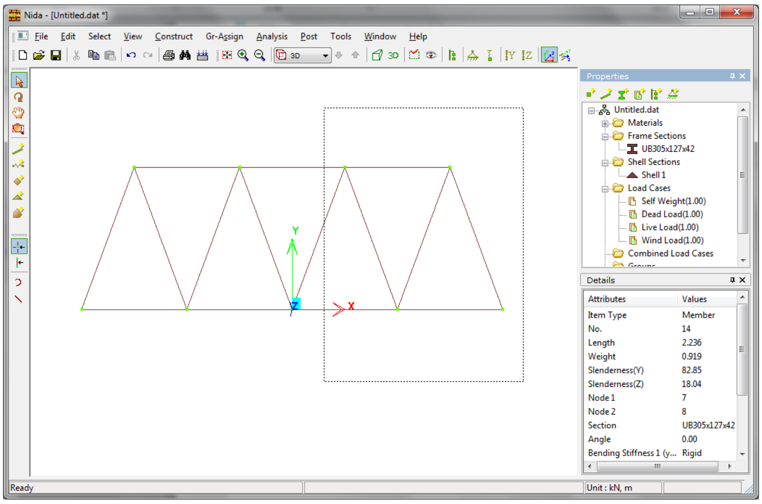
Select the item by a “tick” in the box, and then press OK or Apply button. Your selected items will be displayed on plotted area.

4.4.9 Show Selection Only

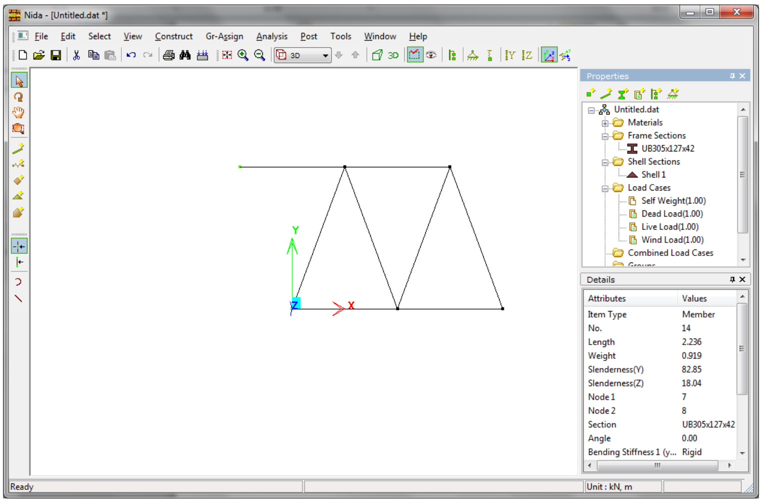
Click the ***View*** > *Show Selection Only* command to view the selected objects only.

Shortcut : Toolbar : Keys : None

- Before operation:



- After operation:



To reshow all objects, un-press  button in the Toolbar.

4.4.10 Remove Selection from View

Click the ***View*** *> Remove Selection from View* command to remove the selected objects from the screen. To show the hide objects again click the ***View*** *> Show Selection Only* command.

Shortcut : Toolbar : None Keys : Alt +R

4.4.11 Extrude View

Click the ***View*** *>Extrude View* command to access the extrude view. The section shape of members can be shown on the screen. Select member(s) and zoom in, then the shape can be viewed more clearly. This function can help to identify the member local axis.

Shortcut : Toolbar :  Keys : None

Un-press  button, then the members will be redrawn by line only.

4.4.12 Perspective Mode

Click the ***View*** *> Perspective Mode* command to view the model in perspective modes.

Shortcut : Toolbar :  Keys : None

4.4.13 Toolbar

Click the ***View*** *> Toolbar* command to select the toolbar items, standard, view, snap, output, post, properties and details window, to be shown in the main window.

To reset the toolbars to default location click the ***View*** *> Toolbar> Reset* command.

Reference: Graphical User Interface

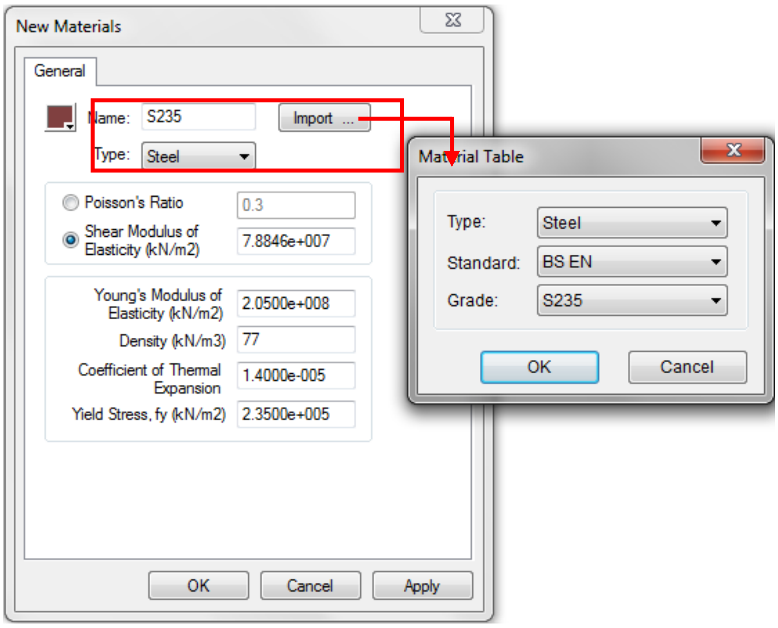
4.4.14 Status Bar

Click the ***View*** > *Status Bar* command to select whether or not to show the status bar at the bottom of the window.

4.5 Construct

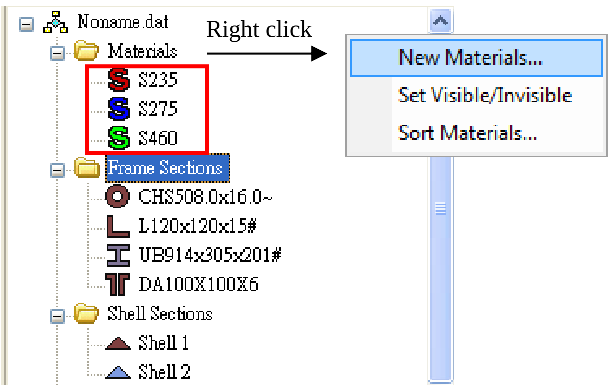
4.5.1 New Material

Click the ***Construct*** > *New Material* command to define a new material.

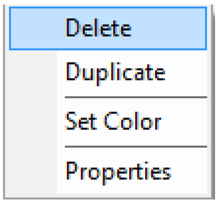


Click the Import button to import a new material according to the available design codes, and the material properties will be filled in the material dialog automatically. The material properties can also be input manually.

After the material is defined, the new material entry will be shown in the Properties Window on the right side of the screen. Right click the head of material, then a pop-up menu will be shown as below.



Delete, **Duplicate** or edit the properties of the selected material, and **Set Color** for the material by right clicking the Material Row of Properties window.



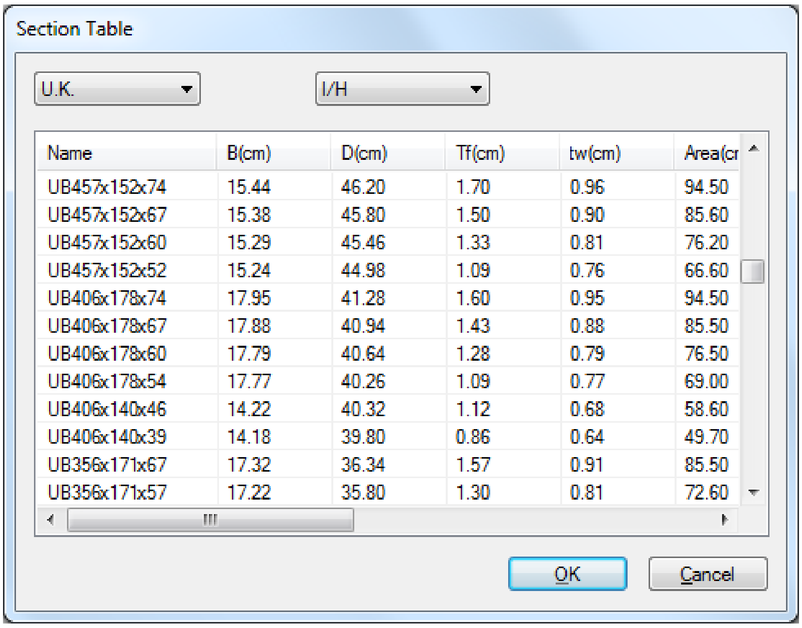
To edit a material, double click the head of material and the Material Dialog will pop up. Change the values and click OK button

To **Delete** a material, highlight the head of material and press <Delete> key. Noted that only the mateiral which has not been assigned to any objects can be deleted directly.

4.5.2 New Frame Section

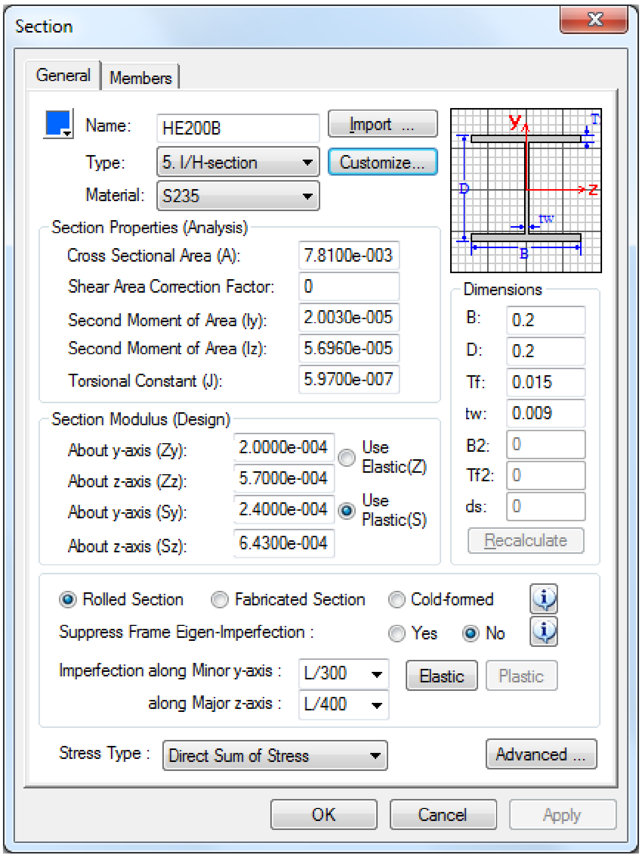
Click the ***Construct*** *> New Frame Section* command to define a new frame section. A group of sections could be added by using ***Construct*** *> New Frame Section>Import Sections* command and the section type be selected from available section table. There are several commonly used section types available according to the five major standard codes such as U.K., China, Europe, U.S. and Japan codes.

Shortcut : Toolbar : Keys : None



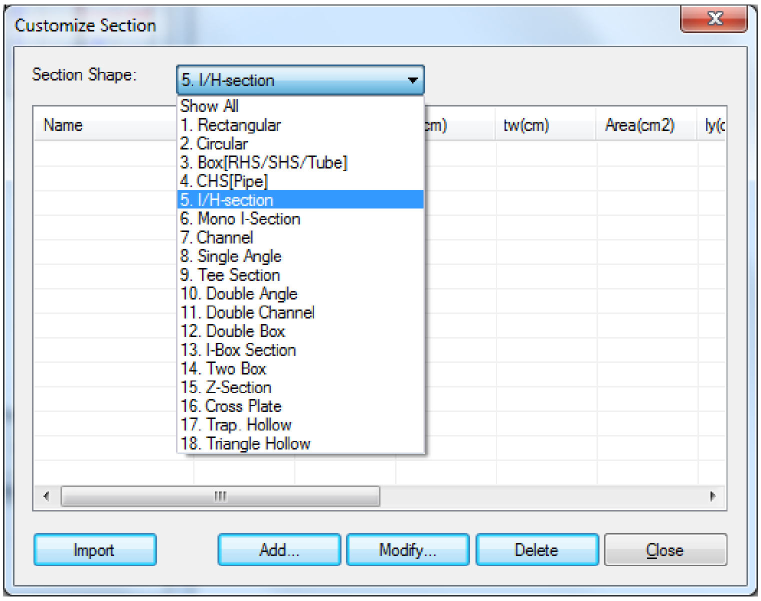
Select the frame section(s) and click OK button.

Sections could also be added one-by-one using ***Construct*** *> New Frame Section> New Sections* command.



Sections could be imported from available section table by clicking Import button in the New Section Window.

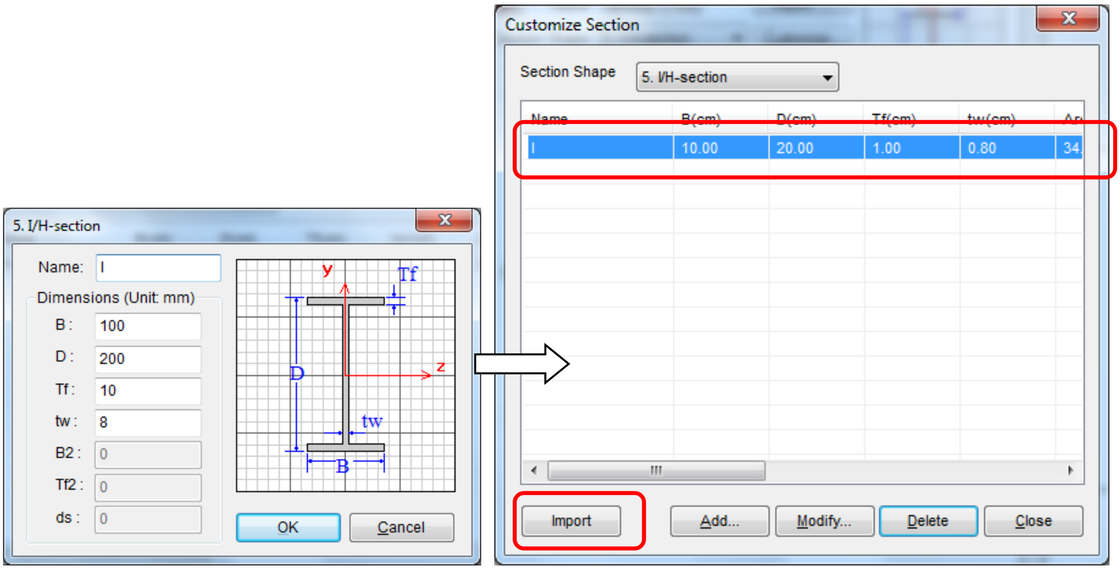
User can also define a section by input the dimensions directly by clicking the Customize button.



There are 18 types of section available for customization, i.e. *Rectangular*, *Circular*, *Box [RHS/SHS]*, *CHS [Tube]*, *I/H-Section*, *Mono I-Section*, *Channel*, *Single Angle*, *Tee Section*, *Double Angle*, *Double Channel*, *Double Box*, *I-Box Section*, *Two Box*, *Z-Section*, *Cross Plate*, *Trap. Hollow and Triangle Hollow*.

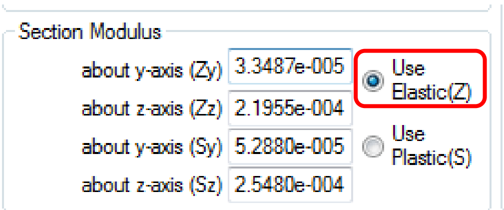
Noted that only the sections with typical shape (e.g. Box [RHS/SHS], CHS [Tube], I/H-Section, Channel) specified in design code can perform member stability check.

Select the section shape, and then click the Add button to input the dimensions of the section. After that, click the Import button to import this section, and the section properties of the customized section will be input in the New Section window.



Either elastic section modulus or plastic section modulus can be selected by clicking Use Elastic (Z) or Use Plastic(S) button. The plastic section modulus is limited to 1.2 times the elastic modulus by default and user can modify it if needed.

Generally, NIDA will check the section classification by design code if the design function is active in analysis. If “b/T” and “D/t” ratios cannot meet the requirements of the code (e.g. Class 3 and Class 4 sections), the elastic section modulus will be automatically used even the option “Use Plastic(S)” is selected.

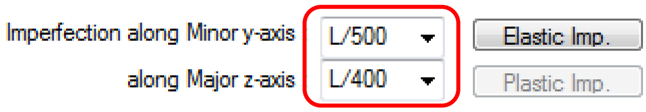


Suppress Frame Eigen-imperfection option enables the ignorance or allowance of the members under this section group in computation of Eigen-imperfection of the frame. This may be required by bracing members to avoid buckling mode of the frame being dominated by the buckling mode of the braces.

Select the section type by clicking the Rolled section, Fabricated section or Cold-formed section button. In some design codes (e.g. CoPHK 2011) the design strength p_y of the compression member using fabricated section should be reduced by 20 N/mm^2. For other cases, user may enlarge the member imperfection by code for fabricated and cold-formed sections.

By default, the member local P-δ imperfection will be set for different section types according to the design Code in use. For Eurocode-3 (2005), there are two types of initial imperfection to select from, i.e. “Elastic Imperfection” and “Plastic Imperfection” respectively.

The member imperfection is valid for nonlinear analysis and time history analysis. For other analysis cases, the member imperfection will be automatically ignored.



.

Generally, the **Stress Type** of circular and circular hollow sections should be “Square-root of Stress” while other type of sections can be “Direct Sum of Stress”

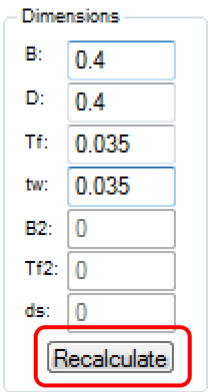
$$\text{Square Root of Stress} = \frac{P}{A} + \frac{\sqrt{M_y^2 + M_z^2}}{M_c}$$

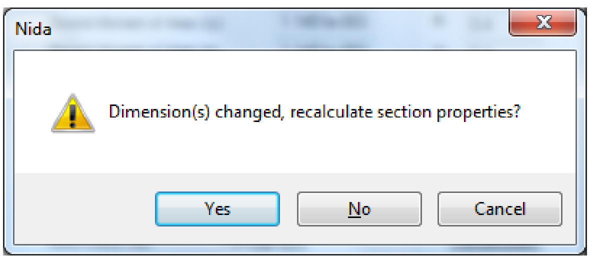
$$\text{Direct Sum of Stress} = \frac{P}{A} + \frac{M_y}{M_{cy}} + \frac{M_z}{M_{cz}}$$

To create a group of tension members, set **Stress Type** as “Tension Only”. This option is usually for cable elements.

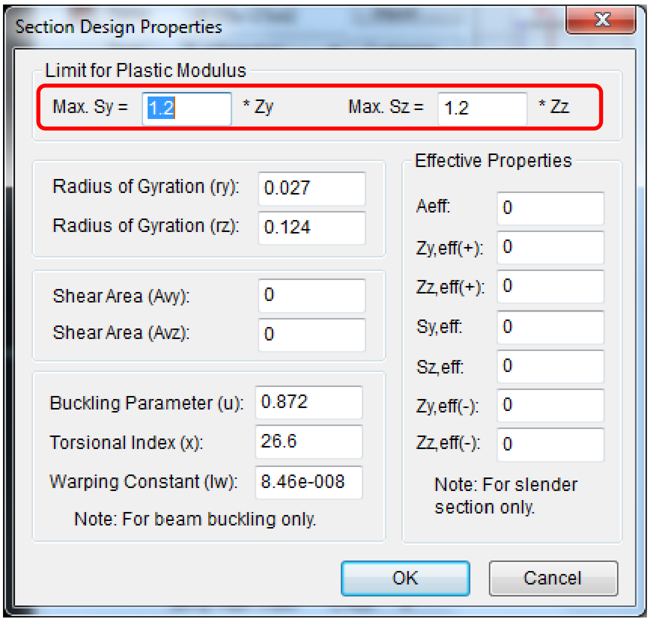
To create a group of compression members, set **Stress Type** as “Compression Only”. This option is usually for soil spring element, infilled wall.

User can modify the section properties by changing the dimensions. After changing the dimensions, user needs to click the **Recalculate** button to confirm and calculate the new values to overwrite the old ones.

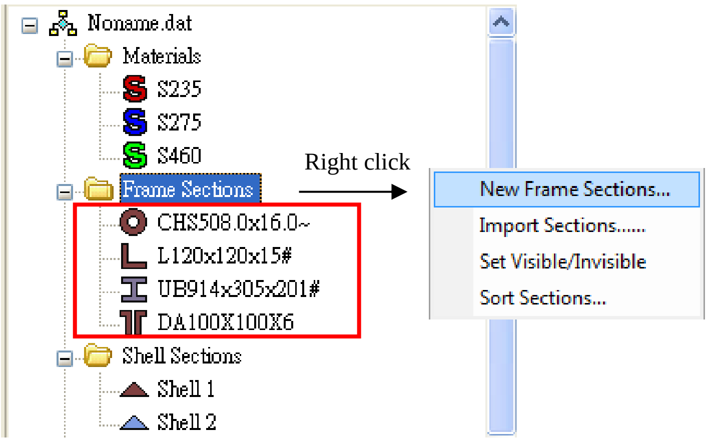




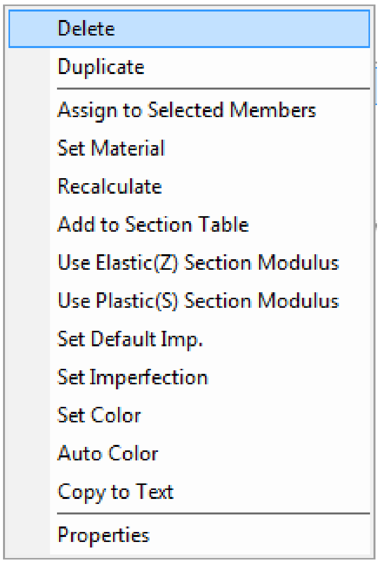
Click the Advanced button to view the advanced section design properties information. If the properties are zero or blank, the program will automatically calculate them by their dimensions if needed. Once the properties are input, the program will adopt the user defined values for design. For slender (Class 4) section, the effective properties should be filled in if the effective width method is used, otherwise, the effective stress method will be used. Also, the plastic limit could be modified in this dialog.



The new frame section entry will be shown in the Properties Window on the right side of the screen. Right click the head of Frame Section, a pop-up menu will be shown.

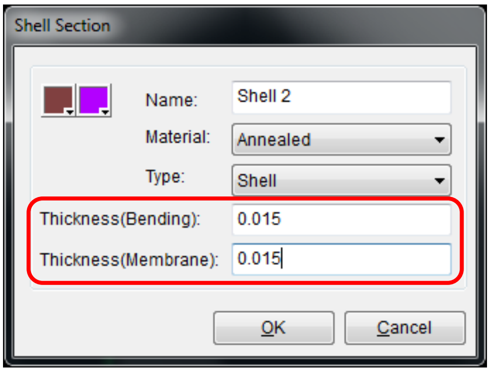


Delete, duplicate and change the settings of the frame section by right clicking the Frame Sections Row of Properties window.



4.5.3 New Shell Section

Click the ***Construct*** > *New Shell Section* command to define a new shell section.

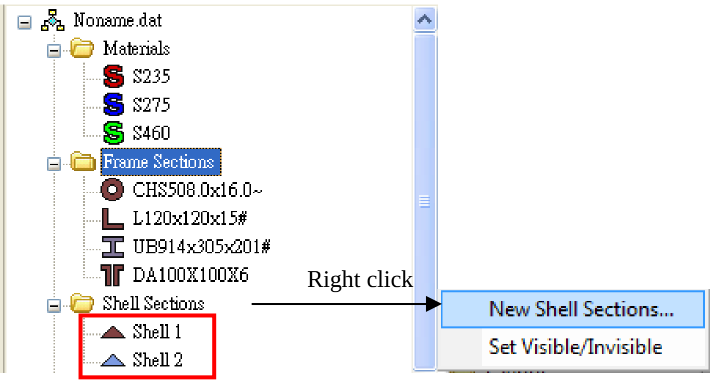


Input the name of the shell, thickness of the shell for bending and membrane, also select the defined material and type for the shell section in the Shell Section window. If the “Membrane Only” or “Plate Only” is selected, more boundary conditions may be needed to restrain the in-plane or out-of-plane degrees of freedom.

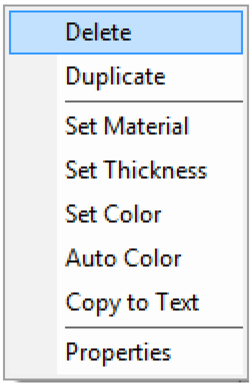
In most cases, the thicknesses for bending action and membrane action should be same. For some structural elements, the stiffness contribution of bending and membrane may depend on the behavior of materials used. For instant, the bending stiffness of concrete slab may be significantly affected by the concrete crack and in such case the bending thickness should be reduced accordingly.

Noted that when the thicknesses are different the self-weight and structural masses will be calculated using the maximum one.

After the shell section is defined, the new shell section entry will be shown on the Properties Window on the right side of the screen.



Right click the head of Shell Sections, and a pop-up menu is shown. Delete, duplicate and change the settings of the shell section by right clicking the Shell Sections Row of Properties Window.

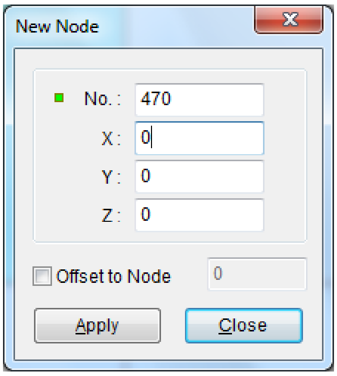


4.5.4 New Nodes

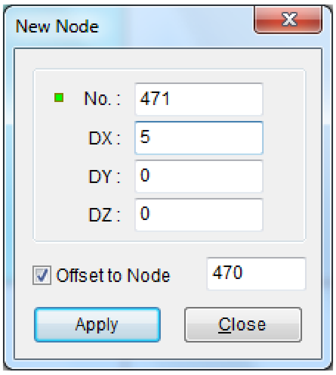
Click the ***Construct*** *> New Nodes* command to define a new node.

Shortcut : Toolbar :  Keys : <u>None</u>

Fill in the nodal coordinates in the dialog, and then a new node will be created and shown on the screen.



Offset mode could also be used to create a new node by filling in incremental coordinates.

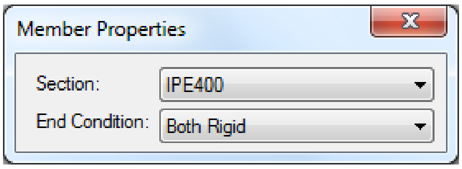


The node properties can also be changed by simply double clicking it on the screen and then a node properties dialog will pop up.

4.5.5 New Members

Click the ***Construct*** > *New Members* command to define a new member.

Shortcut : Toolbar :  Keys : <u>None</u>

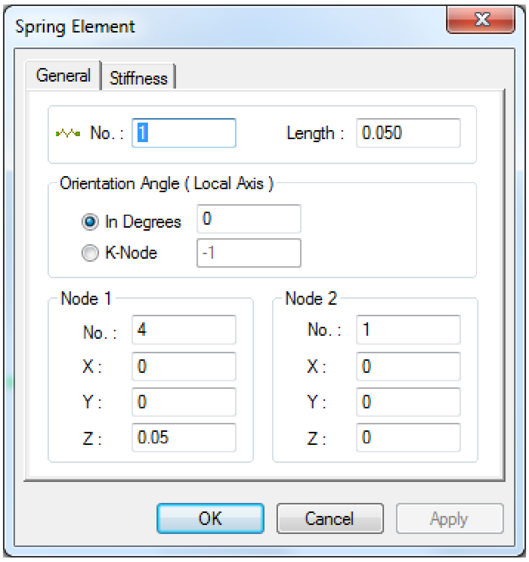


Select the section and end condition of the new member and select the starting and ending node for the new member.

4.5.6 New Spring Elements

Click the ***Construct*** > *New Spring Elements* command to define a new spring element.

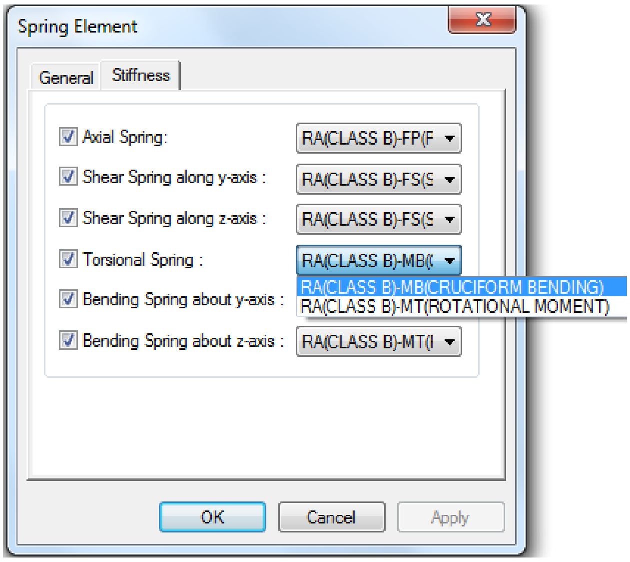
Shortcut : Toolbar :  Keys : <u>None</u>



User can input or select two nodes on the screen to define a spring element. Each node has six degrees of freedom. The definition of the local axis of the spring element is same as general beam-column element. The local axis is important as the element stiffness is formed in this coordinate system first and then transferred to global coordinate system.

The element stiffness matrix is formed by six independent springs, i.e., axial spring, shear springs along y/z-axis, torsional spring, bending springs about y/z-axis. If the contribution of the spring is ignorable the corresponding spring can be left blank.

Noted that the spring model can be defined in "Semi-Rigid/Spring Models". If the nonlinear spring model(s) is used, the spring element may behave nonlinearly.

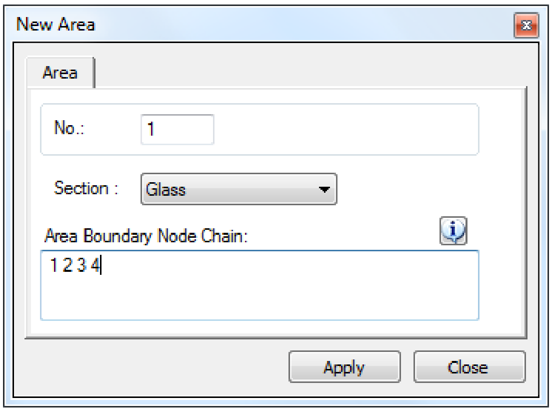


The new spring element would appear on the screen when the OK button is clicked.

4.5.7 New Areas

Click the ***Construct*** > *New Areas* command to define a new area.

Shortcut : Toolbar : Keys : None



In the New Area window, input or pick up the nodes on the screen which are used to form an area. Click the Apply button and a new area will be created.

A new area could also be created by clicking the shortcut and picking up the nodes on the screen.

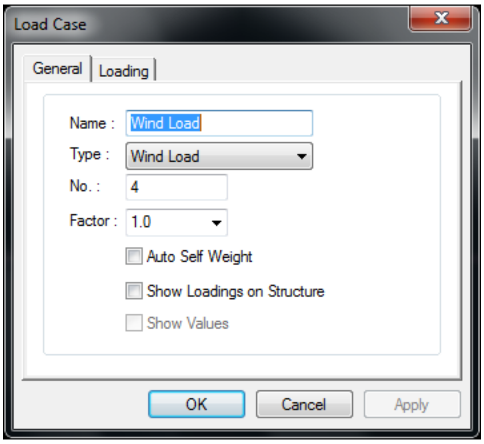
To obtain more information about the construction of an area, click the information button.

Warning: Areas are not structural objects and therefore all their properties such as section assignment and loading assignment are invalid in the analysis before they are converted to floor or shell elements. The area properties will be converted to the corresponding floor/shell properties after they are converted to floor/shell elements. The function of **AREA** is to help to create complex structures with floor/shell elements.

4.5.8 New Load Cases

Click the ***Construct*** > *New Load Cases* command to define a new load case.

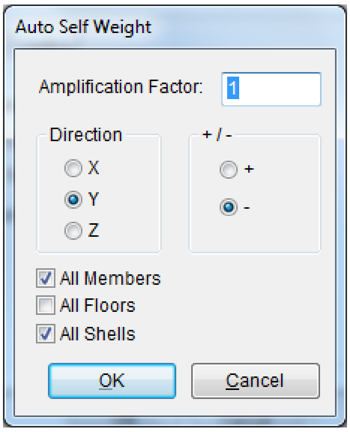
Shortcut : Toolbar : Keys : None



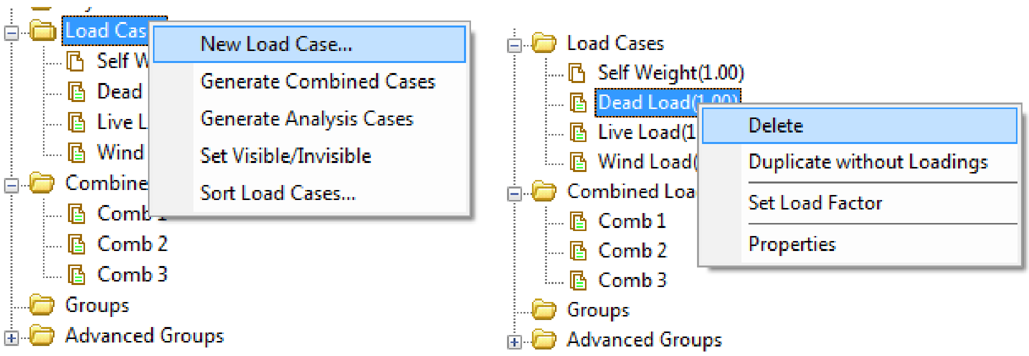
A load case is a container to save loadings, such as dead loads, live loads, wind loads and so on.

Fill in the load case name and factor in the New Load Case window. Click Show Loadings on Structure and Show Values buttons to show all of the loadings as well as the values belong to them on the screen.

If the self-weight are to be added to all members, shells or floors in the load case, click the Auto Self Weight button. Click Settings button and a dialog will pop up as below.



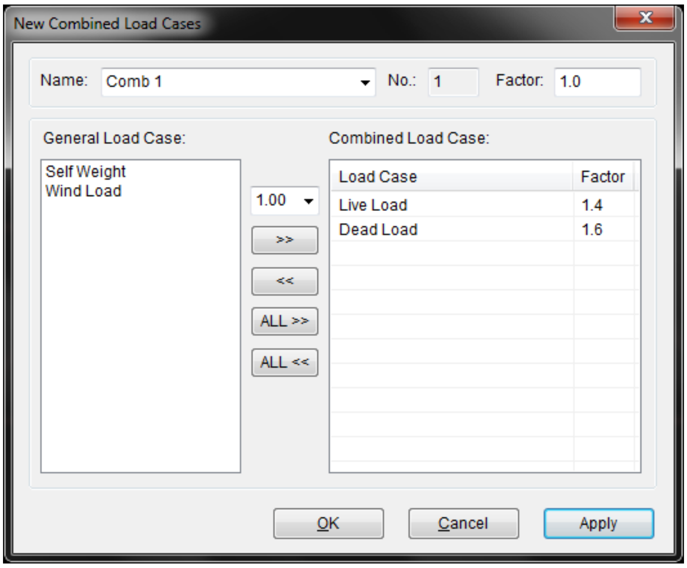
After the load case is defined, the new load case entry will be shown on the Properties Window on the right side of the screen.



Right click the head of Load Cases, a pop-up menu will be shown. Delete, duplicate and change the load factor and properties of the load case by right clicking the Load Cases of Properties window.

4.5.9 New Combined Load Cases

Click the ***Construct*** > *New Combined Load Cases* command to define a new combined load case.



A combined load case is a group of load cases with load factors for ULS (ultimate limit state) check, SLS (serviceability limit state) check or other purposes. A combined load case can be used in a linear, nonlinear, eigenvalue buckling or time history analysis.

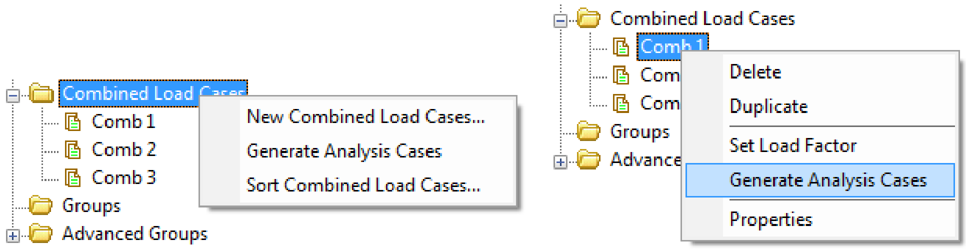
Fill in name and factor for the new combined load case in the Combined Load Case Window.

To **add** load cases into the combined load case, one or more existed load cases should be selected from the left side, provide a load factor for the load cases, and press >> button to add to the new combined load case. All of the load cases could be added together into the combined load cases by pressing ALL >> button.

To **remove** load cases from the combined load case, you have to select one or more existing load cases from right side and press << button to remove from the new combined load case. All of the load cases can also be removed from the combined load cases simultaneously by pressing ALL << button.

After the combined load case is defined, the new combined load case entry will be shown in the Properties Window on the right side of screen. Right click the head of Combined Load Cases, and a pop-up menu will be shown. Delete, duplicate and change the load factor and properties of the load case by right clicking the Combined Load Cases of Properties window.

Analysis cases could also be generated directly with the selected combined load cases.



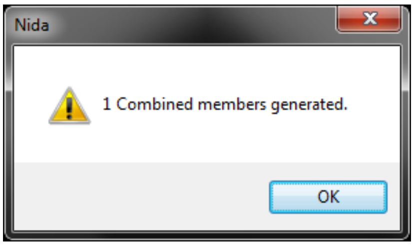
4.5.10 New Combined Members

Click the ***Construct*** > *New Combined Members* command to combine a group of selected elements into a combined member for easy checking of deflection or consider the imperfection as a single member. For the latter use, the combined members will be like super elements.

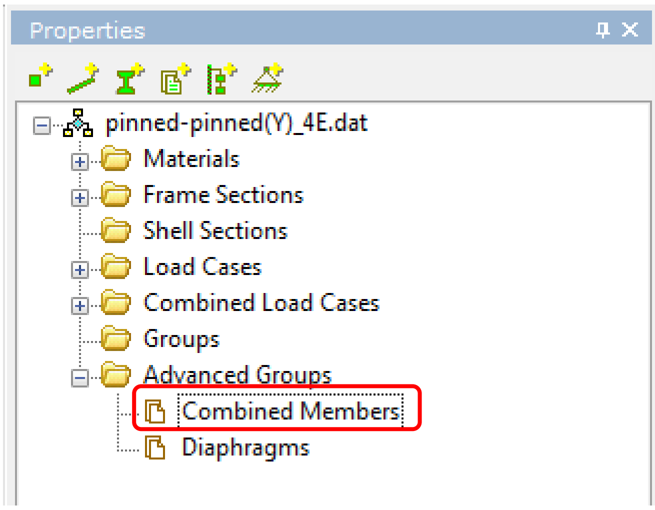
All elements in a combined member should be continuous and parallel to each other. When considering member imperfection, all elements in a combined member should be assigned with same section and the local axes of all elements should be same direction.

Shortcut : Toolbar : None Keys : F7

If several members are successfully combined, a message will show as follows:



To edit, delete the combined members, double click the head of "Combined Members" in Properties Window on the right side of the screen as below.

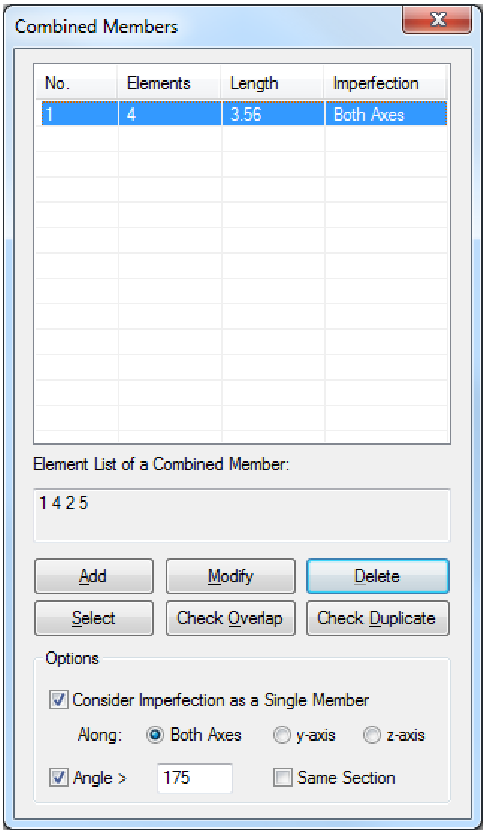


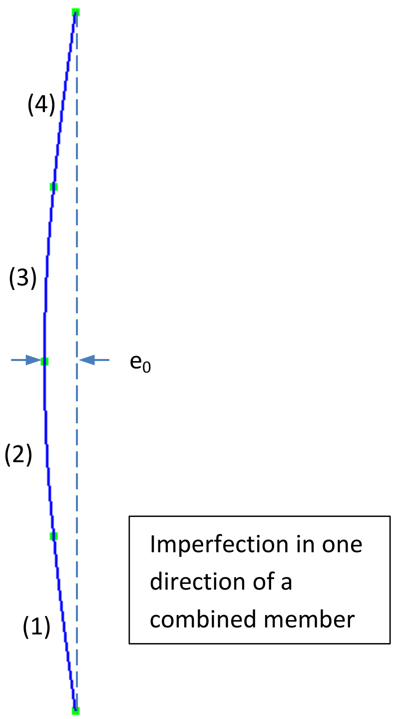
A dialog will pop up as below. This dialog shows the number of elements, total length and setting of imperfection of each combined member.

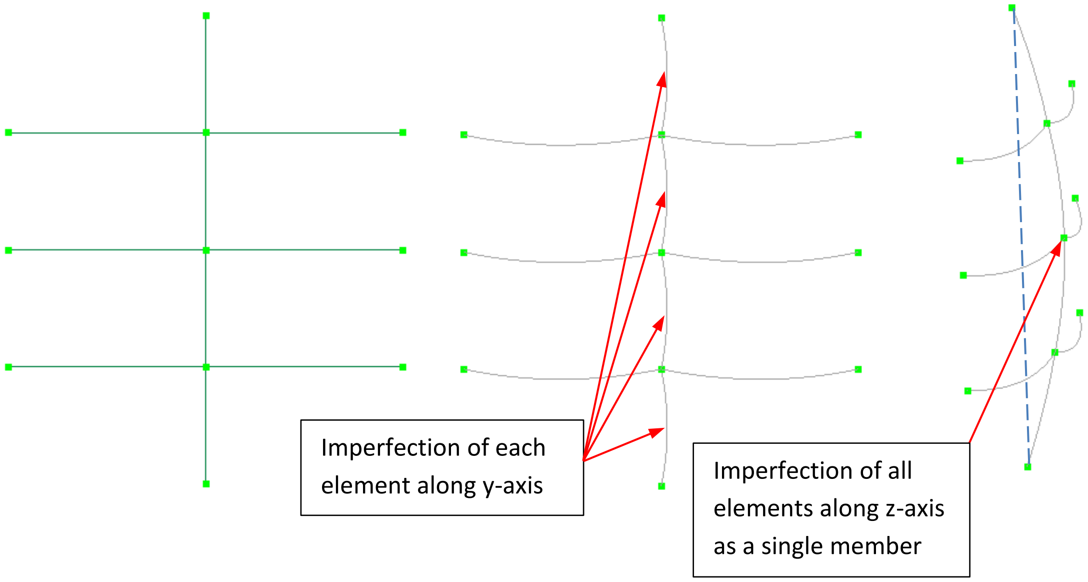
To add a combined member, please select a group of elements on screen and then click "**Add**" button.

To modify the imperfection, select a number of combined members and choose the direction such as "Both Axes", "y-axis" and "z-axis" in "Options" and then click "**Modify**" button.

"Both Axes" means that the imperfections of both major and minor axes will be treated as a single member which follows a sinusoidal curve. "Y-axis" means that the imperfection of the combined member in y axis will be treated as a single member while the imperfection in z axis will follow the individual elements of it.







Imperfection of a combined member in "z-axis"

Generally "one element per member" is adequate to capture the structural behavior in NIDA. The imperfection of combined member is usually for a member divided into several elements or a physical member spanning across multiple members and its buckling lengths in two directions are different.

4.5.11 Diaphragms

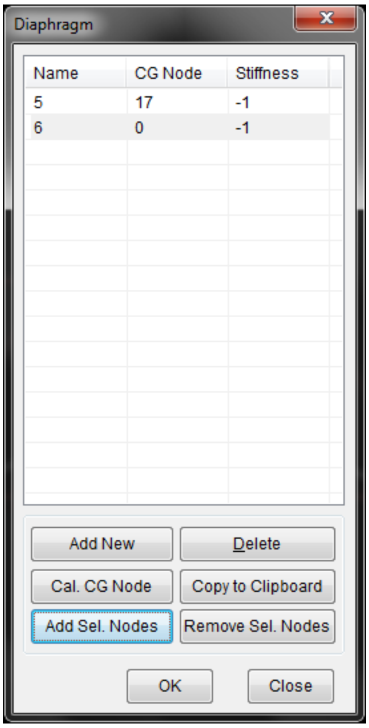
Click the ***Construct*** > *Diaphragms* command to define a new diaphragm. A diaphragm is a group of nodes in the same plane connected to a C.G. Node with internal spring elements with infinite stiffness "-1". Generally, the nodes at the same story can be assigned to a diaphragm.

Click Add New to define a diaphragm. The default stiffness of internal spring elements is set as "-1" which means infinite stiffness. User can change the stiffness if needed. When the stiffness is zero, the diaphragm will be ignored in the analysis. The default C.G. Node is zero which means the C.G. Node should be further determined by user.

Select a number of nodes on the screen and choose Add Sel. Nodes, and then click OK button to add nodes to the selected diaphragm.

Select a number of nodes on the screen and choose Remove Sel. Nodes, and then click OK button to remove nodes from the selected diaphragm.

User can edit the CG Node and Stiffness manually or click Cal. CG Node to determine it automatically.

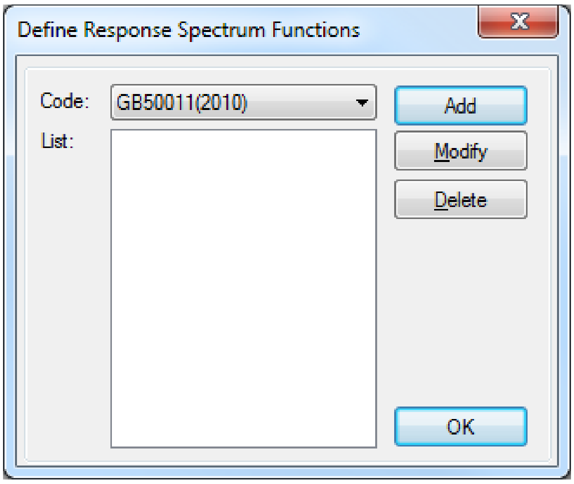


After the diaphragm is defined, the new diaphragm entry will be shown on the Advanced Groups of Properties window on the right side of screen.

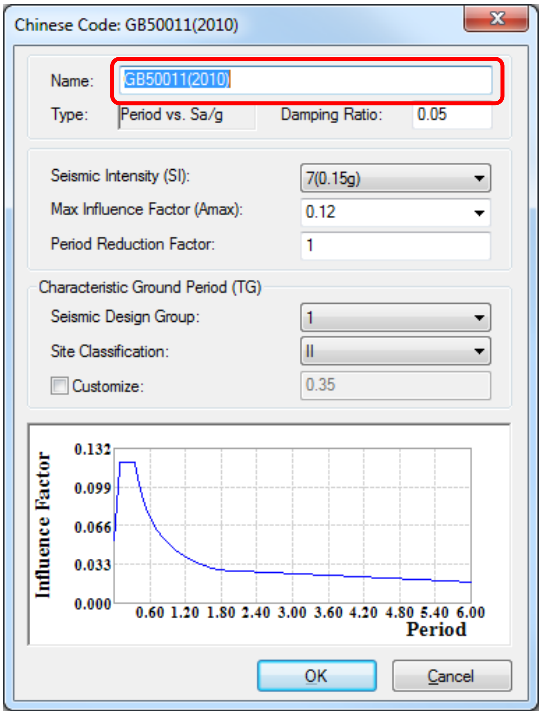
4.5.12 Response Spectrum Functions

Click the ***Construct*** *> Response Spectrum Functions* command to define a new response spectrum function.

Three seismic design codes, i.e. GB50011-2010 (China), Eurocode8 (2004) and IS 1893:2002 (India), and User Defined are available for defining a new response spectrum. After selecting the design code, click the Add button to define a new response spectrum function.



- GB50011 (2010)



Seismic Intensity (SI) level: Six levels are available for selection.
Max Influence Factor (Amax): Two values are available for selection.
Characteristic ground period (TG): This value can be determined by.

(1)Site Classification and Seismic Design Group

Site Classification: Four site types are available for selection.

Seismic Design Group: Three groups are available for selection.

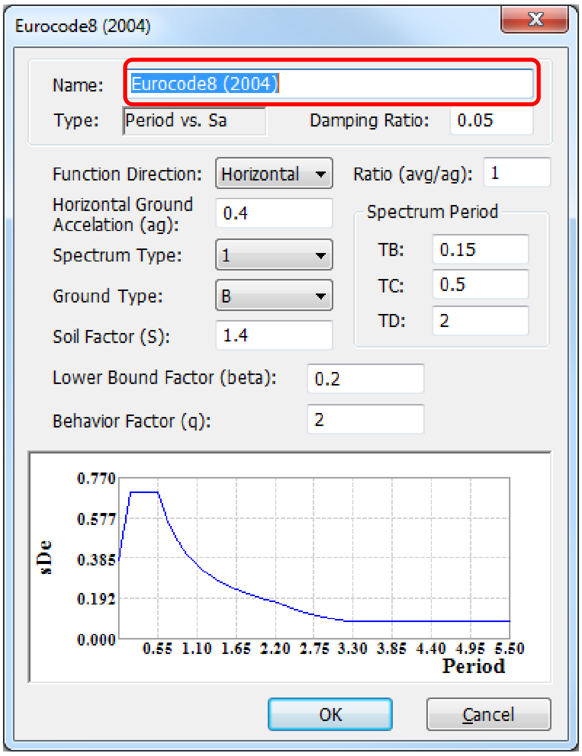
(2)User Defined

Customize: Define the characteristic ground period TG manually.

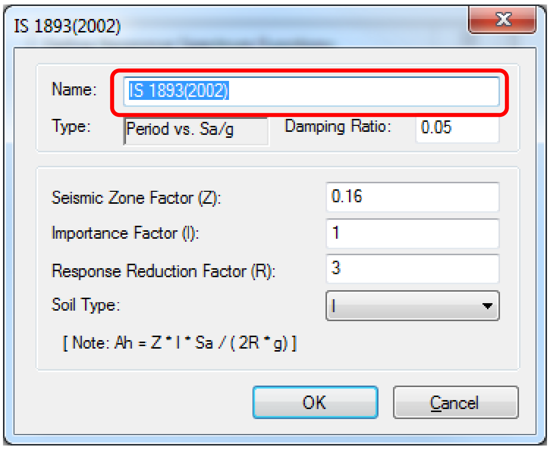
Function Damping Ratio: Enter the function damping ratio.

Period Reduction Factor: Range from 0 to 1.0. This value is to consider the influence of non-structural elements such as infill walls to the periods.

- Eurocode8 (2004)



- IS1893 (2002)



Seismic Zone Factor (Z): Enter the value of seismic zone factor

Important Factor (I): Enter the value of importance factor

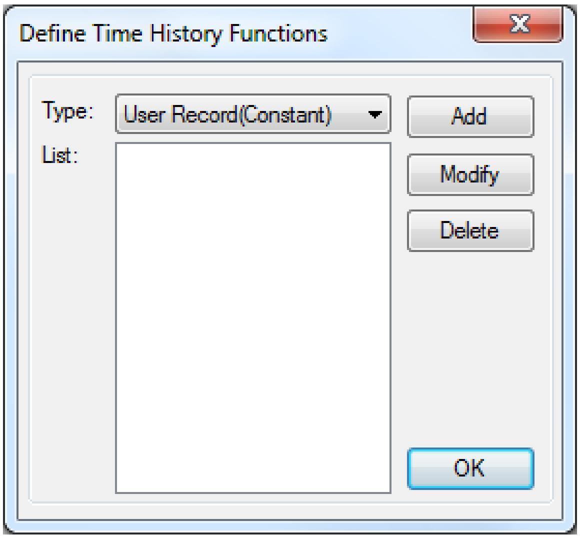
Response Reduction Factor (R’): Enter the response reduction factor

Soil Type: Select the soil type

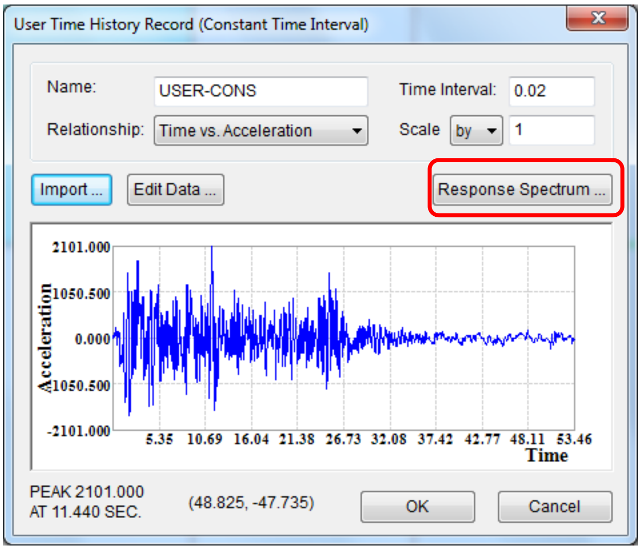
4.5.13 Time History Functions

Click the ***Construct*** *> Define Time History Functions* command to define a new time history function.

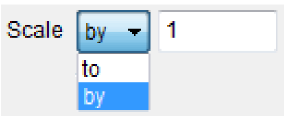
Seven types of functions, i.e. *User Record (Constant)*, *User Record (Variable)*, *Sine Function*, *Cosine Function*, *Ramp Function*, *Sawtooth Function* and *Triangular Function* are available for defining a new time history.



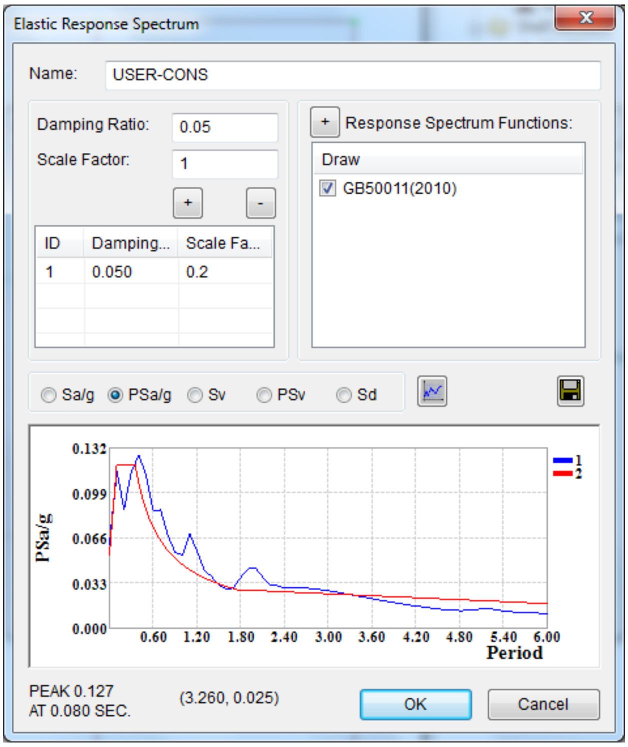
- User Record (Constant)



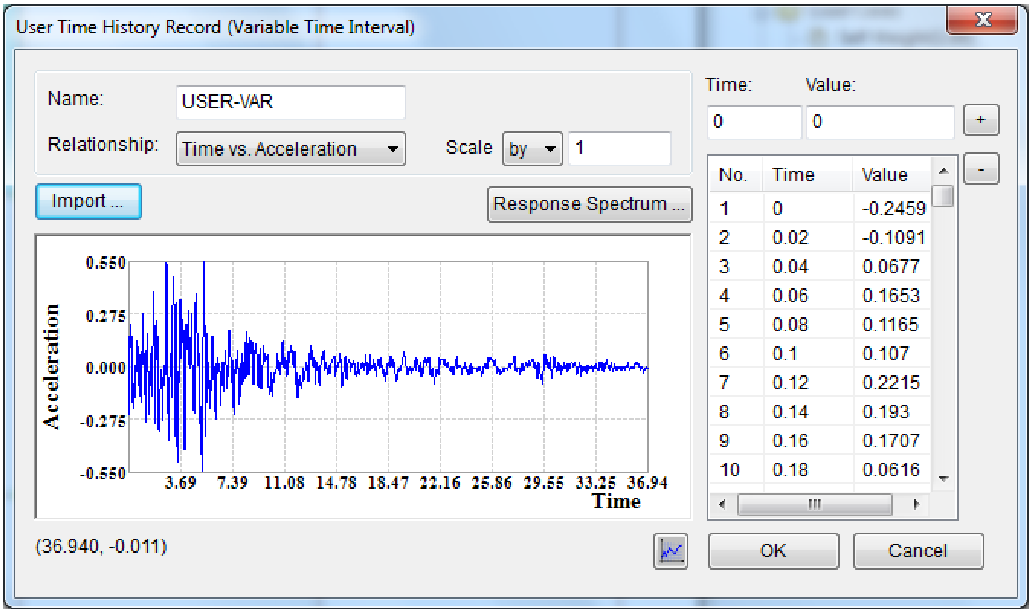
To scale an earthquake or artificial wave, user can select "By" or "To" mode. For "By" option, the wave will be scaled by multiplying the factor. For "To" option, the scale factor is equal to [*given value*] / [*peak value*].



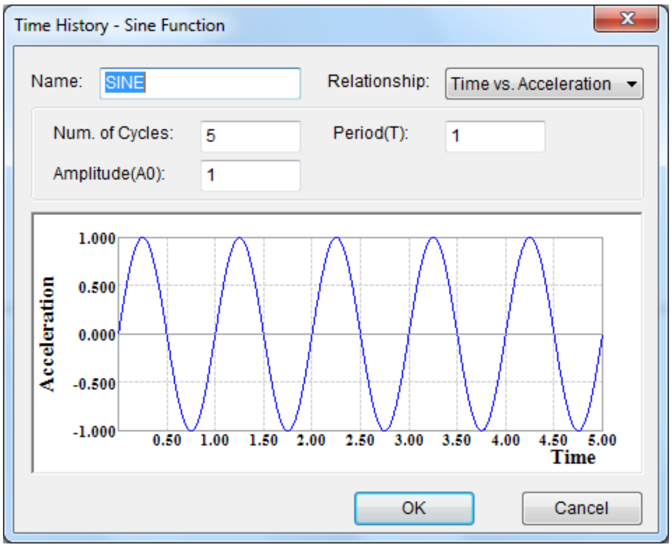
To create elastic response spectrum, click Response Spectrum button. The obtained spectrum can be plotted against those defined in Response Spectrum Functions.



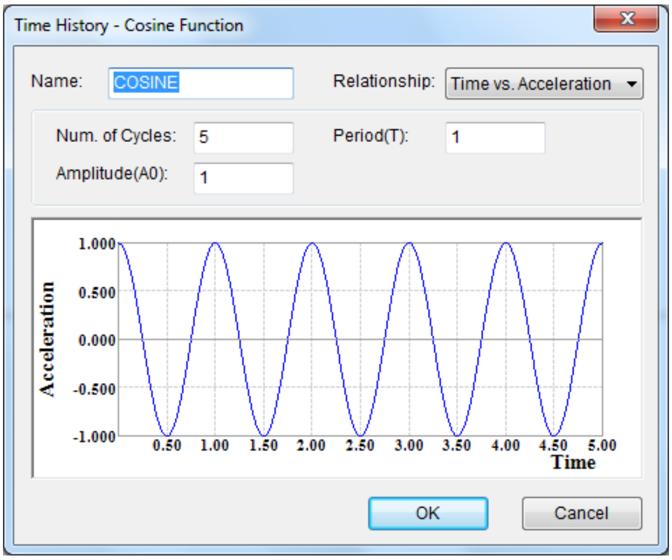
- User Record (Variable)



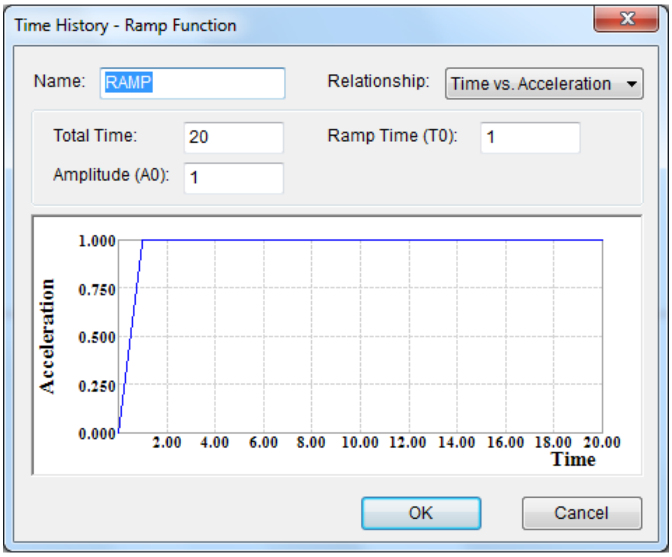
- Sine Function



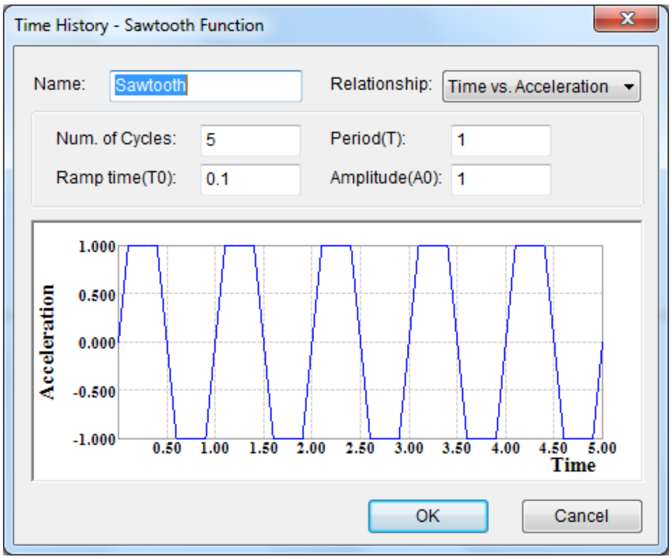
- Cosine Function



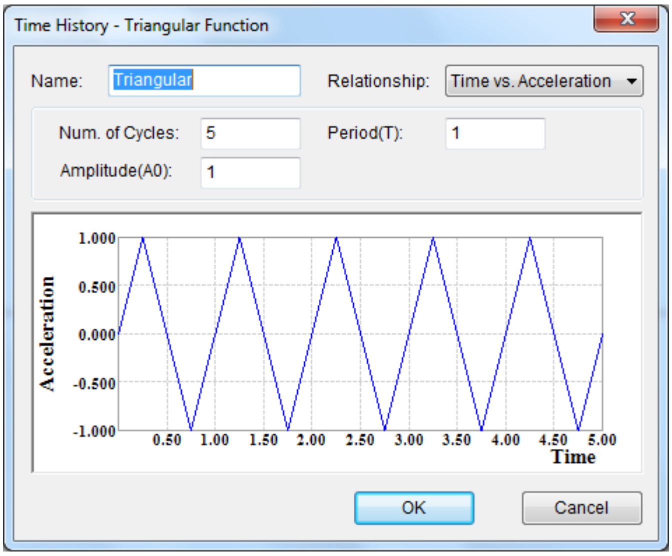
- Ramp Function



- Sawtooth Function

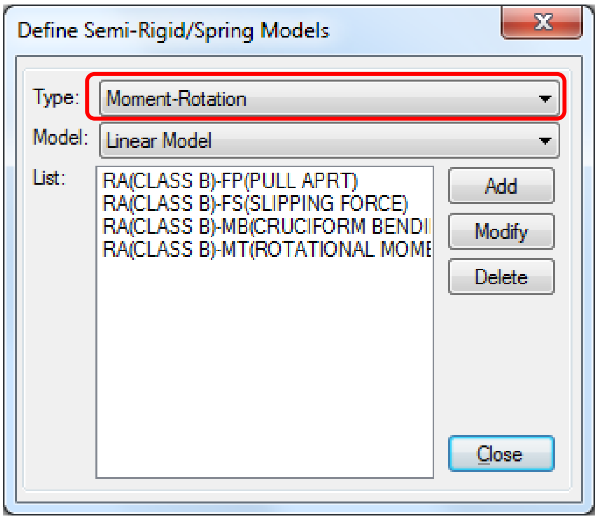


- Triangular Function



4.5.14 Semi-Rigid/Spring Models

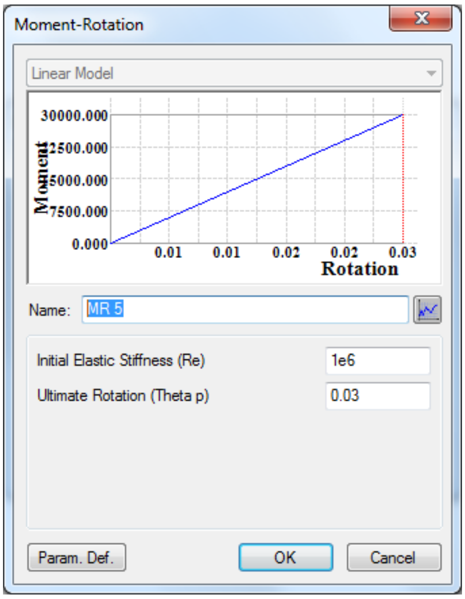
Click the ***Construct*** > *Semi-Rigid/Spring Models* command to define a semi-rigid model for beam-column element or a spring model for spring element and support spring.



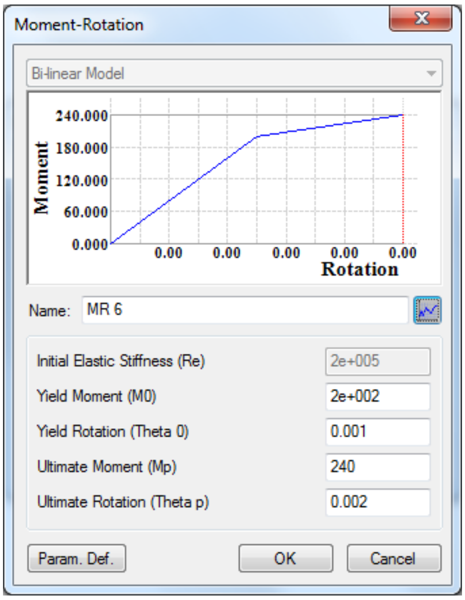
Type: Two kinds of relationships, i.e. Moment -Rotation and Force-Displacement are available.

- Moment-Rotation

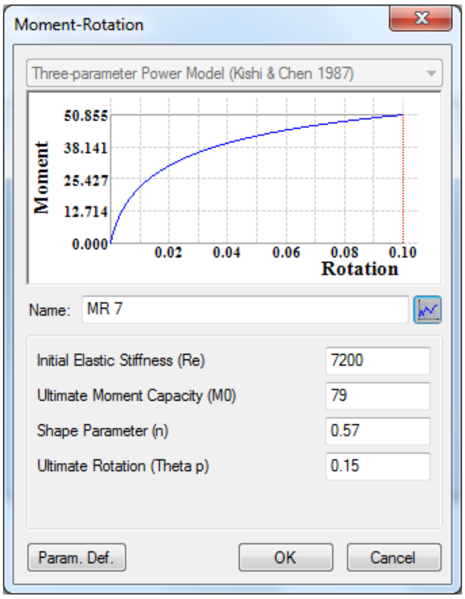
  (1) Linear Model



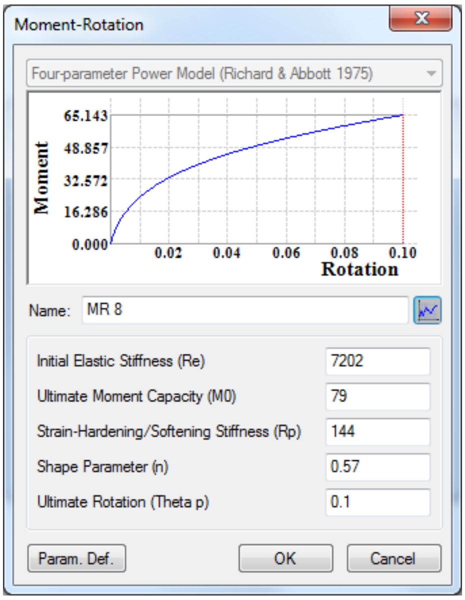
(2) Bi-linear Model



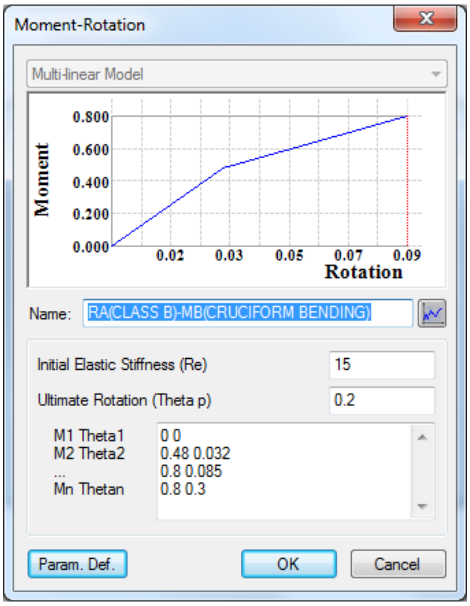
(3) Three-parameter power model



(4) Four-parameter power model

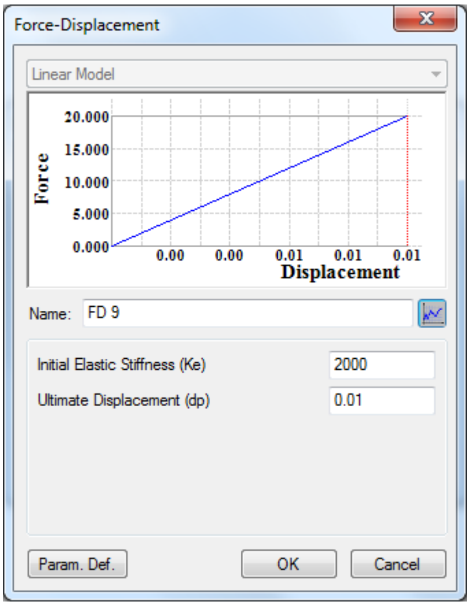


(5) Multi-linear model

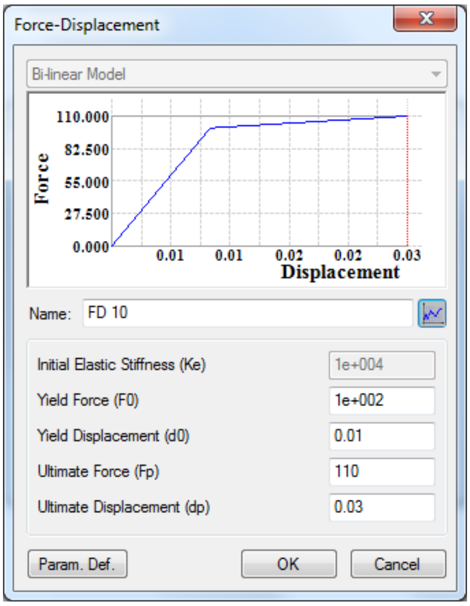


- Force-Displacement

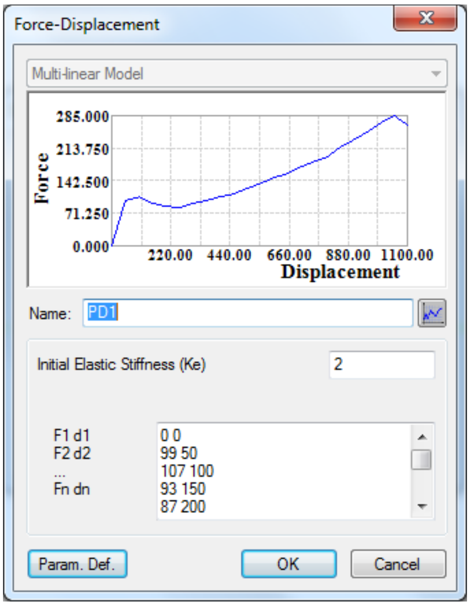
(1) Linear Model



(2) Bi-linear Model



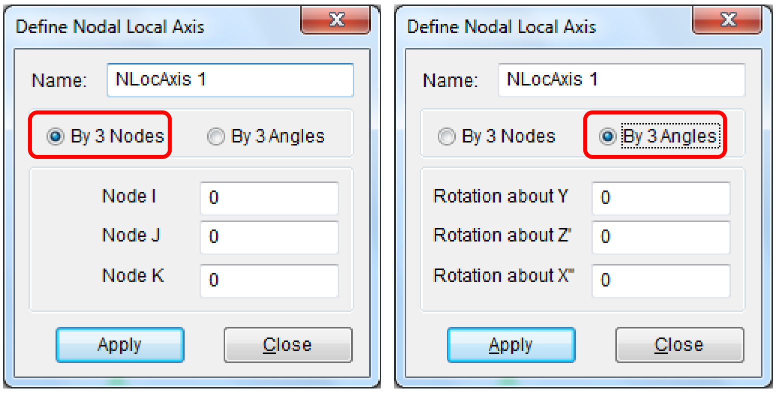
(3) Multi-linear Model



4.5.15 Node Local Axis

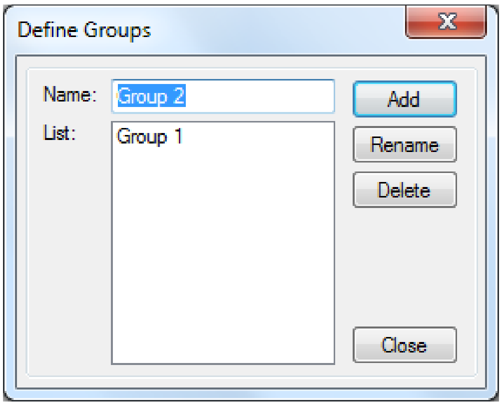
Click the ***Construct*** *> Node Local Axis* command to define the local axis for specific nodes.

There are two methods to create a node local axis, i.e. By 3 Nodes and By 3 Angles.



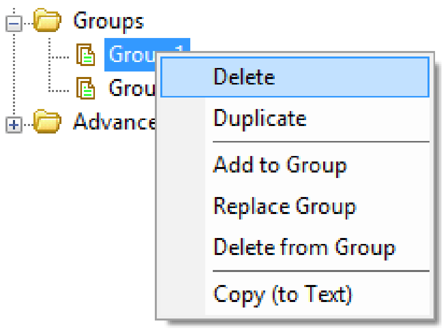
4.5.16 Groups

Click the ***Construct*** > *Groups* command to define a new group.



A group can contain a group of nodes, members, shells, floors, areas and their combination.

You can add, rename and delete the group. After a group is defined, a new group entry will be shown under the Groups of Properties window on the right side of screen.



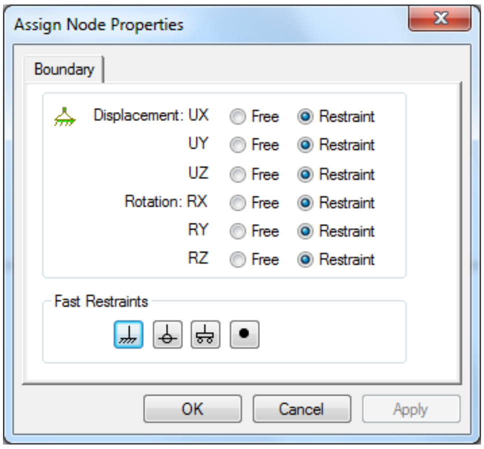
4.6 Gr-Assign

4.6.1 Nodes

4.6.1.1 Boundary Conditions

Click the ***Gr-Assign** >Nodes>Boundary Conditions* command to set the boundary condition of an individual node or a group of nodes.

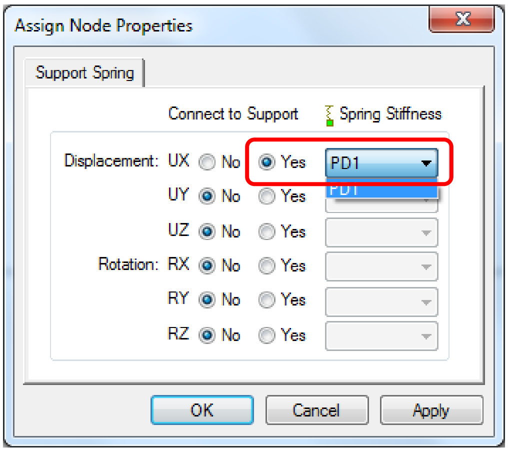
Shortcut : Toolbar : Keys : None



The boundary condition of the node could also be changed by double clicking the node and changing it in Boundary tab window.

4.6.1.2 Node Support Springs

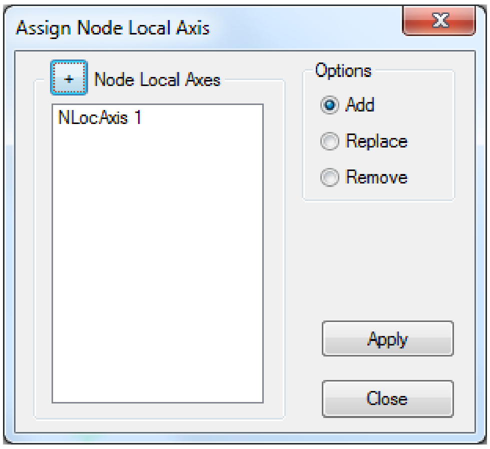
Click the ***Gr-Assign** >Nodes>Node Support Springs…* command to set the node support spring of the selected nodes.



The spring model(s) used should be defined in "Semi-Rigid/Spring Models" first. The support spring could also be changed by double clicking the node in "Spring" tab of node properties window.

4.6.1.3 Node Local Axis

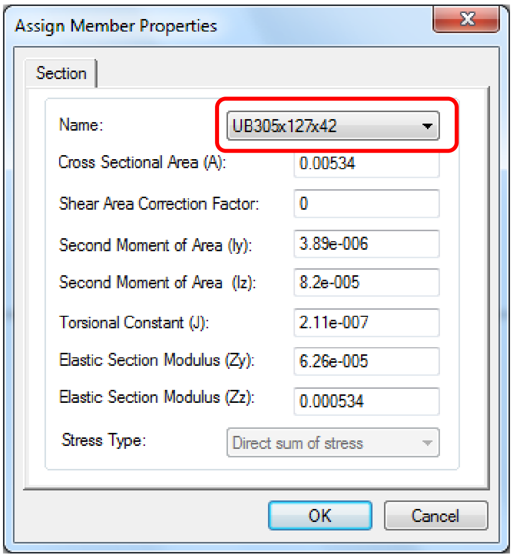
Click the ***Gr-Assign** >Nodes>Node Local Axis* command to set the node local axes of the selected nodes. The node local axis is used to consider inclined support and the joint loads in local axis.



4.6.2 Members

4.6.2.1 Section

Click the ***Gr-Assign** > Members>Section* command to assign a defined section to an individual member or a group of members.

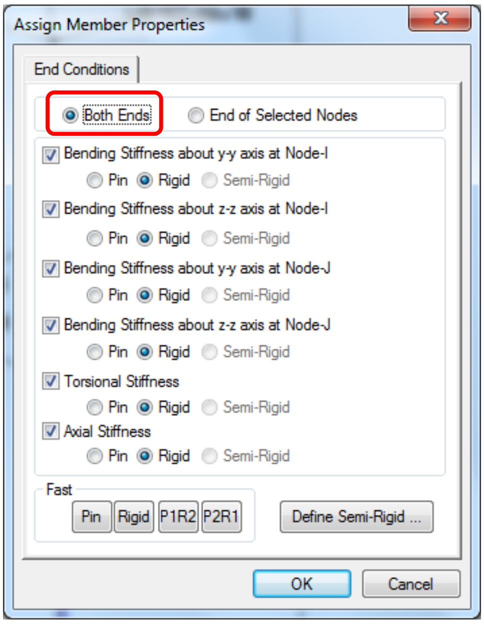


The member section could also be changed by double clicking the member and changing it in member properties window.

The frame section can be defined in New Frame Section.

4.6.2.2 End Conditions

Click the ***Gr-Assign** > Members>End Conditions* command to assign the end condition to both ends of an individual member or a group of members.

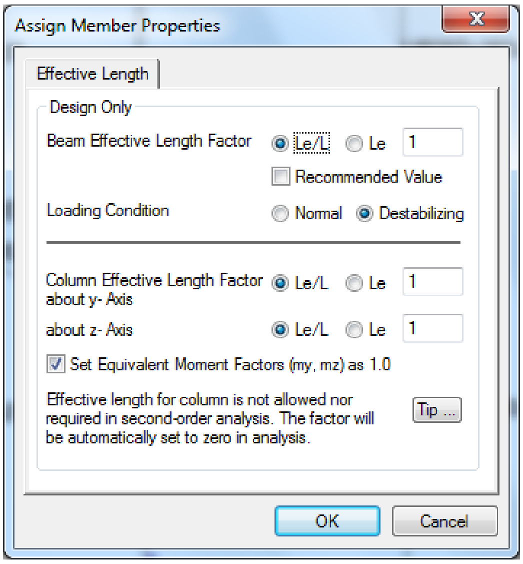


You can also change the connection stiffness of both ends by double clicking the member and change it in “End Conditions” tab of member property window.

Fast function enables a quick select of the end condition. The Pin button means the pin ends for both nodes. The Rigid button means the rigid ends for both nodes. The P1R2 button means pin end for the first node and rigid end for the second node. The P2R1 button means pin end for the second node and rigid end for the first node. Also Define Semi-Rigid button can be used to define semi-rigid connections.

4.6.2.3 Effective Length

Click the ***Gr-Assign*** *> Members>Effective Length* command to assign the effective length or effective length factor to an individual member or a group of members (Beam and Column), and to select the loading condition (Normal or Destabilizing) for beam design.



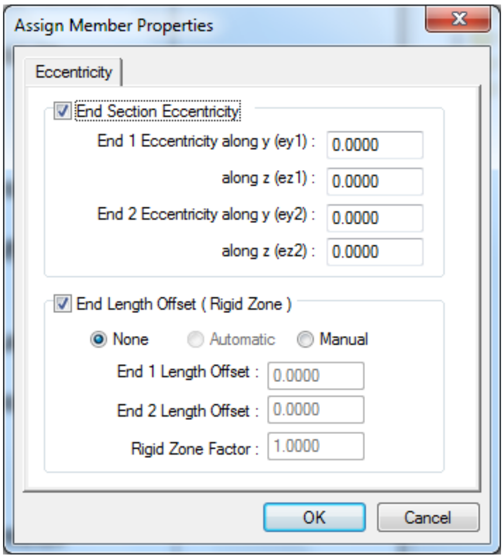
You can also assign the effective length of the member by double clicking the member and assign in Design tab of member properties window.

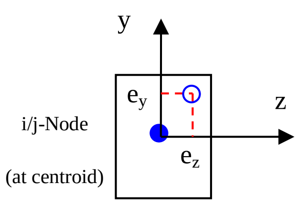
Click the Tip button to view the definition of effective length.

Note: The recommended beam and column effective length factors refer to CoPHK (2011) or BS5950 (2000). The column effective length factors are invalid in nonlinear analysis.

4.6.2.4 Eccentricity

Click the ***Gr-Assign*** *> Members>Eccentricity* command to assign eccentricity to an individual member or a group of members.





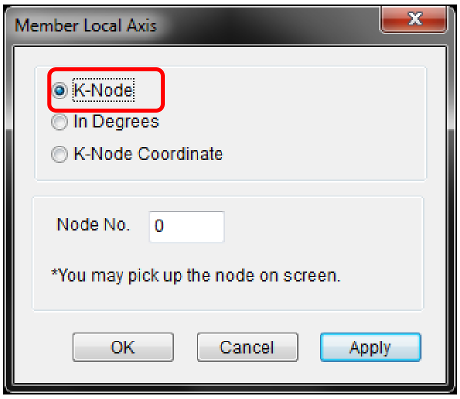
End Section Eccentricity

End Length Offset

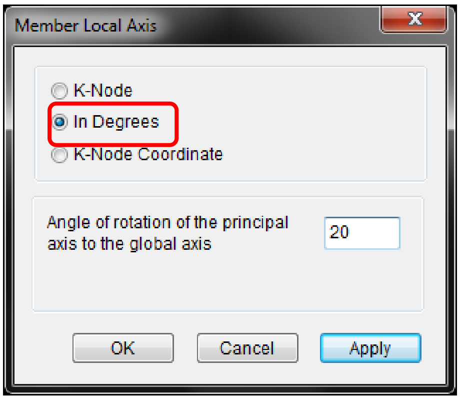
4.6.2.5 Member Local Axis

Click the ***Gr-Assign*** *> Members>Member Local Axis* command to define the member local axis system by K-Node, degrees or K-Node coordinates.

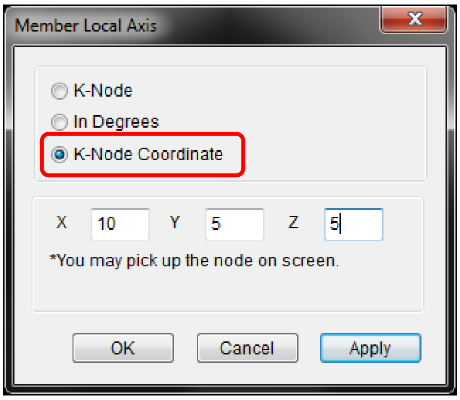
(1) When "**K-Node**" is chosen, a node should be picked up on screen or input directly. Note: the k-node should be defined before clicking Apply and local xy-plane will tilt to contain k-node.



(2) When “**In Degree**” is chosen, an angle is required to be specified.

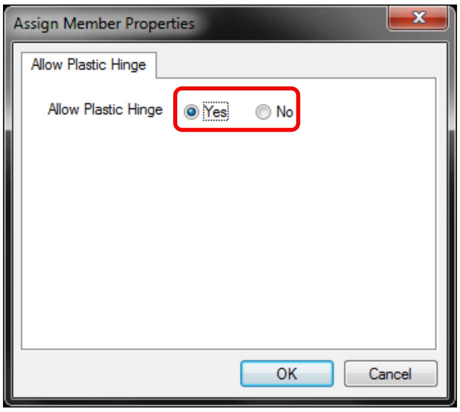


(3) When “**K-Node Coordinates**” is chosen, 3 coordinates should be input in global coordinate system to define a k-node. Note: k-node needs not be defined before using it.



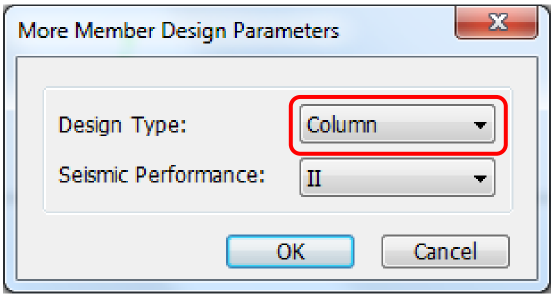
4.6.2.6 Plastic Hinge

Click the ***Gr-Assign*** *> Members>Plastic Hinge* command to choose whether or not to allow for plastic hinge.



4.6.2.7 More Design Parameters

Click the ***Gr-Assign** > Members>More Design Parameters…* command to set the member design type such as column, beam and brace and the seismic performance according to GB 50011 of the selected members.



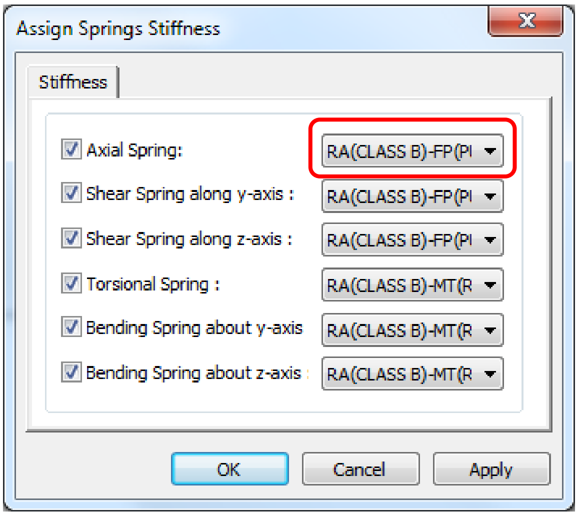
4.6.2.8 Reverse Connectivity

Click the ***Gr-Assign** > Members>Reverse Connectivity* command to reverse the connectivity of the selected members.

4.6.3 Springs

4.6.3.1 Stiffness of Spring

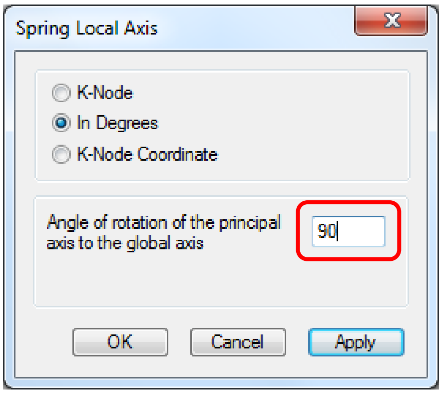
Click the ***Gr-Assign** >Springs> Stiffness* command to assign one or more than one defined spring models to an individual spring element or a group of spring elements.



You can also select and double click a spring element and then change its stiffness in spring properties window.

4.6.3.2 Spring Local Axis

Click the ***Gr-Assign*** *> Springs> Local Axis…* command to define the spring local axis system by K-Node, degrees or K-Node coordinates.



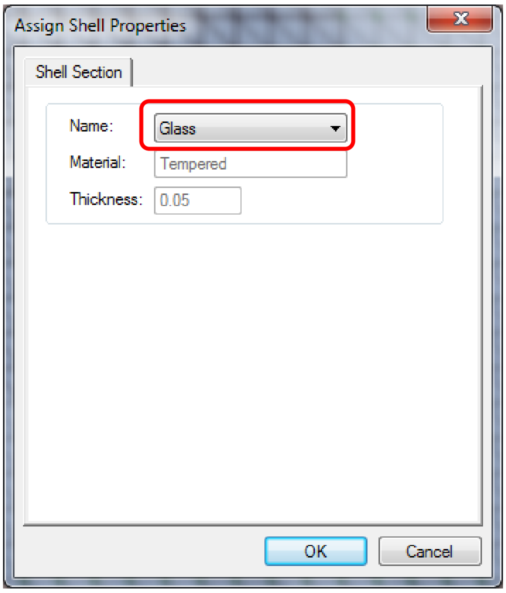
4.6.3.3 Reverse Connectivity

Click the ***Gr-Assign*** *> Springs>Reverse Connectivity* command to reverse the connectivity of the selected spring elements.

4.6.4 Shells

4.6.4.1 Section

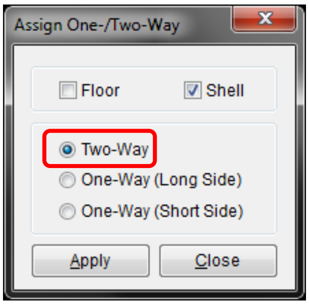
Click the ***Gr-Assign*** *>Shells> Section* command to assign a section to an individual shell or a group of shells.



You can also select and double click a shell and then change its section in shell properties window.

4.6.4.2 One-Way/Two-Way

Click the ***Gr-Assign*** *>Shells>One-Way/Two-Way* command to assign the load distribution mode to an individual shell or a group of shells.



4.6.4.3 Reverse Normal Direction

Click the ***Gr-Assign*** *>Shells>Reverse Normal Direction* command to reverse the normal direction of the selected shells.

4.6.4.4 Convert to 2 Triangular Shells

Click the ***Gr-Assign*** *>Shells>Convert to 2 Triangular Shells* command to convert the selected shells to 2 triangular shells.

4.6.4.5 Convert to 4 Triangular Shells

Click the ***Gr-Assign*** *>Shells>Convert to 4 Triangular Shells* command to convert the selected shells to 4 triangular shells.

4.6.4.6 Convert to Floor

Click the ***Gr-Assign*** *>Shells>Convert to Floor* command to convert the selected shells to floors.

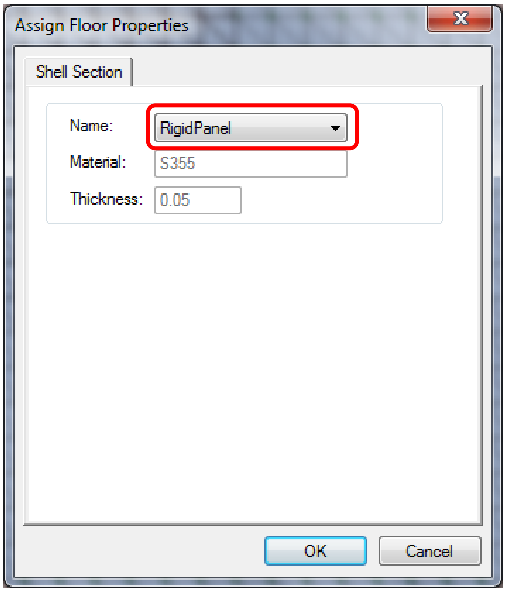
4.6.4.7 Convert to Area

Click the ***Gr-Assign*** *>Shells>Convert to Area* command to convert the selected shells to areas.

4.6.5 Floors

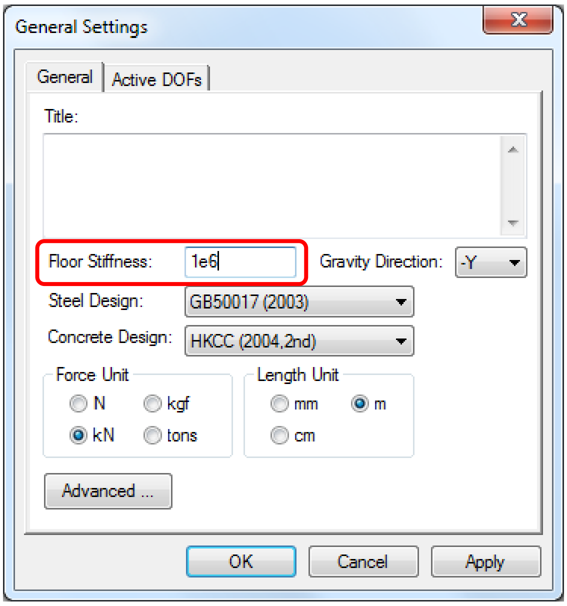
4.6.5.1 Section

Click the ***Gr-Assign*** *>Floors> Section* command to assign a section to an individual floor or a group of floors.



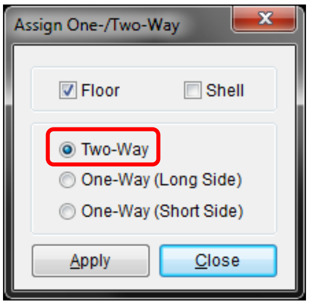
You can also change the section by clicking the floor and changing it in floor properties window.

Noted that the section assigned to floor elements for calculation of self-weight and structural mass and also for conversion of shell elements but not the stiffness. The stiffness of all floor elements are defined in ***Analysis****>Analysis & Design Parameters Setting* as below.



4.6.5.2 One-Way/Two-Way

Click the ***Gr-Assign*** *>Floors>One-Way/Two-Way* command to assign the load distribution mode to an individual floor or a group of floors.



4.6.5.3 Reverse Normal Direction

Click the ***Gr-Assign*** *>Floors>Reverse Normal Direction* command to reverse the normal direction of the selected floors.

4.6.5.4 Convert to 2 Triangular Shell Elements

Click the ***Gr-Assign*** *>Floors>Convert to 2 Triangular Shell Elements* command to convert the selected floors to 2 triangular shell elements.

4.6.5.5 Convert to 4 Triangular Shell Elements

Click the ***Gr-Assign*** *>Floors>Convert to 4 Triangular Shell Elements* command to convert the selected floors to 2 triangular shell elements

4.6.5.6 Convert to Quard Shells

Click the ***Gr-Assign*** *>Floors>Convert to Quard Shells* command to convert the selected floors to shell in a rectangular shape.

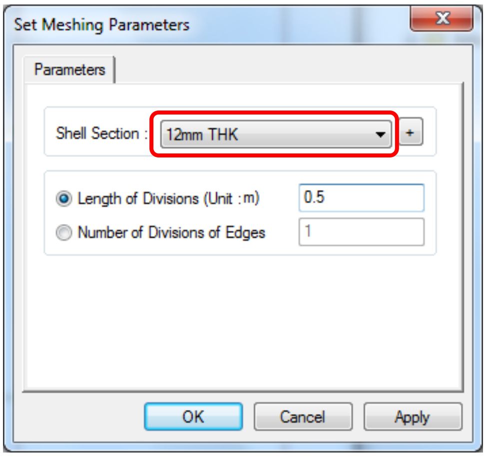
4.6.5.7 Convert to Area

Click the ***Gr-Assign*** *>Floors>Convert to Area* command to convert the selected floors to areas.

4.6.6 Areas

4.6.6.1 Set Meshing Parameters

Click the ***Gr-Assign*** *>Areas>Set Meshing Parameters* command to set up the parameters for meshing operation.



Select the shell section and input the mesh size or the number of the parts for all edges for meshing.

4.6.6.2 Reverse Normal Direction

Click the ***Gr-Assign*** *>Areas>Reverse Normal Direction* command to reverse the normal direction of the selected areas.

4.6.6.3 Convert to Floor Elements

Click the ***Gr-Assign*** *>Area>Convert to Floor Elements* command to convert the selected areas to floor element.

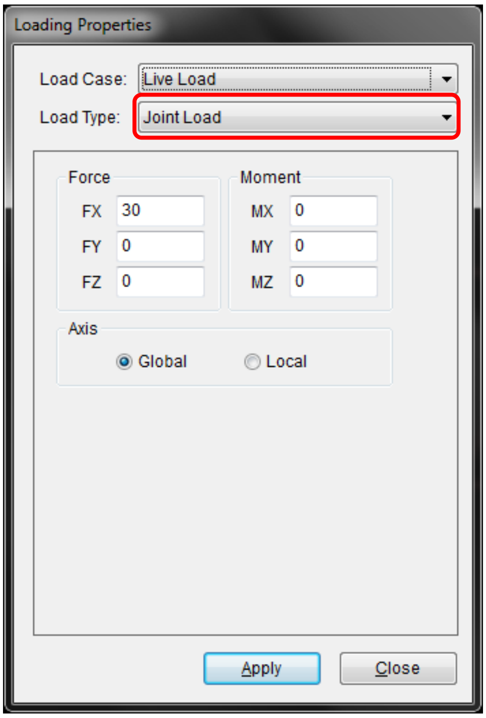
4.6.6.4 Drop Edge Nodes of Areas

Click the ***Gr-Assign*** *>Drop Edge Nodes of Areas* command to drop nodes (if any) at the edges of the selected areas.

4.6.7 Node Loads

4.6.7.1 Joint Loads

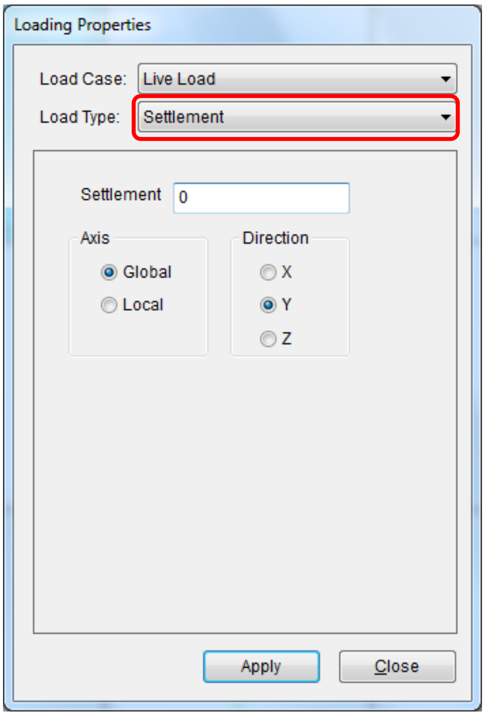
Click the ***Gr-Assign*** *>Nodal Loads>Joint Loads* command to assign joint load to selected nodes.



Fill in the value of forces and moments in X, Y and Z directions.

4.6.7.2 Settlements

Click the ***Gr-Assign*** *>Nodal Loads>Settlements* command to assign support settlement to the selected nodes with boundary conditions.

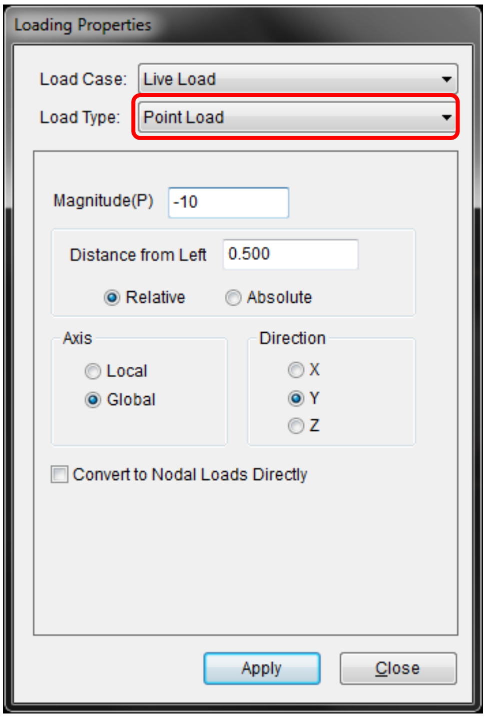


Fill in the value of settlement and select the axis and direction.

4.6.8 Member Loads

4.6.8.1 Point Loads

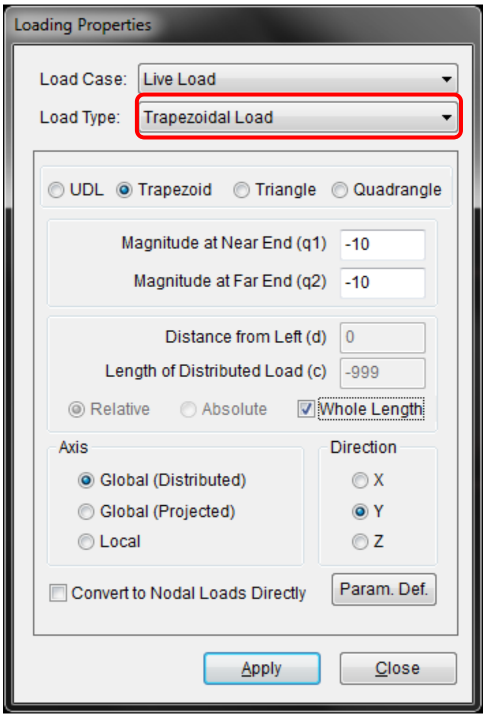
Click the ***Gr-Assign*** *>Member Loads>Point Loads* command to assign the point load to selected members.



Fill in the magnitude, distance from left (i.e. distance from the node with smaller number to one with larger number), load distance type, and axis and direction.

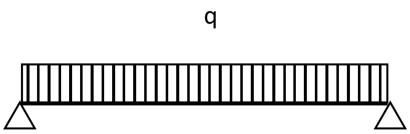
4.6.8.2 Trapezoidal Loads

Click the ***Gr-Assign*** *>Member Loads>Trapezoidal Loads* command to assign the trapezoidal load to selected members.

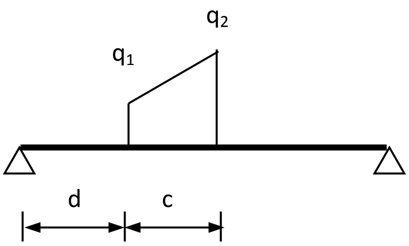


Four types of trapezoidal loads are available, i.e. UDL, Trapezoid, Triangle and Quadrangle.

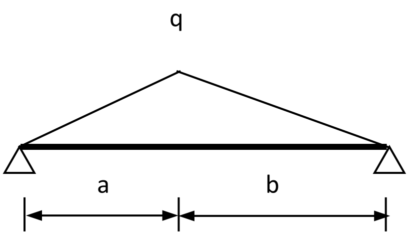
For Uniformly Distributed Load (UDL), you have to fill in magnitude q of the load, element axis and direction.



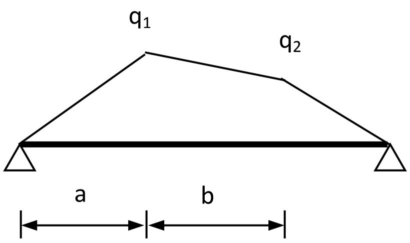
For Trapezoid load, you have to fill in magnitude at near and far end (count from the node with smaller node number), distance from left (i.e. distance from the node with smaller number to one with larger number), load distance type, length of the distributed load, element axis and direction. If the “Whole Length” is selected, that means the load is applied to the whole member and the distance and length will be disabled.



For Triangle load, you have to fill in magnitude of the load at top, distance from left (i.e. distance from the node with smaller number to one with larger number), load distance type, length of the distributed load, element axis and direction.



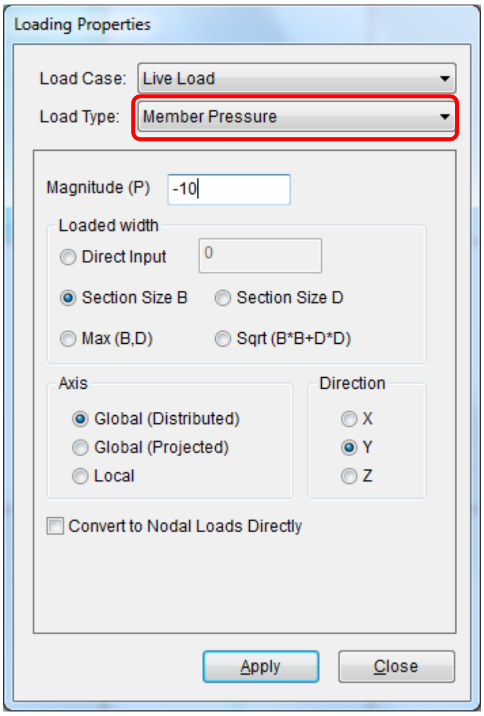
For Quadrangle load, you have to fill in magnitude of load in 1 and 2, distance from left (i.e. distance from the node with smaller number to one with larger number), and distance from 1 to 2, load distance type, element axis and direction.



Click Apply button and a new trapezoidal load entry will be shown on the load case properties and it can be activated by double-clicking the load in the right properties window.

4.6.8.3 Member Pressure

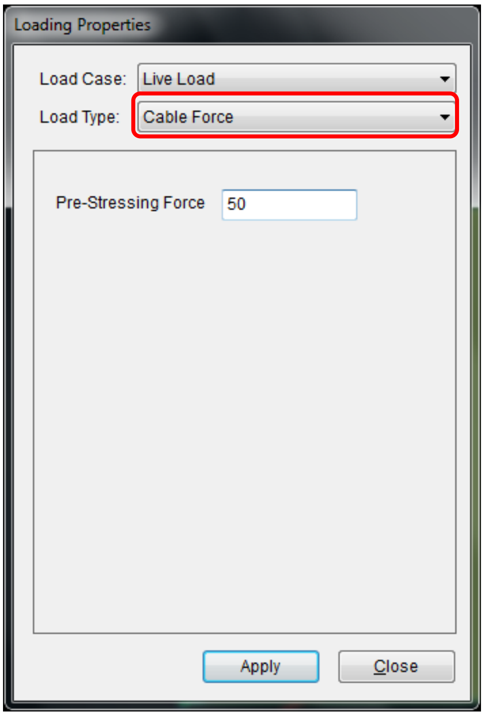
Click the ***Gr-Assign*** *>Member Loads>Member Pressure* command to assign pressure to selected members.



Fill in the magnitude of member pressure.

4.6.8.4 Cable Forces

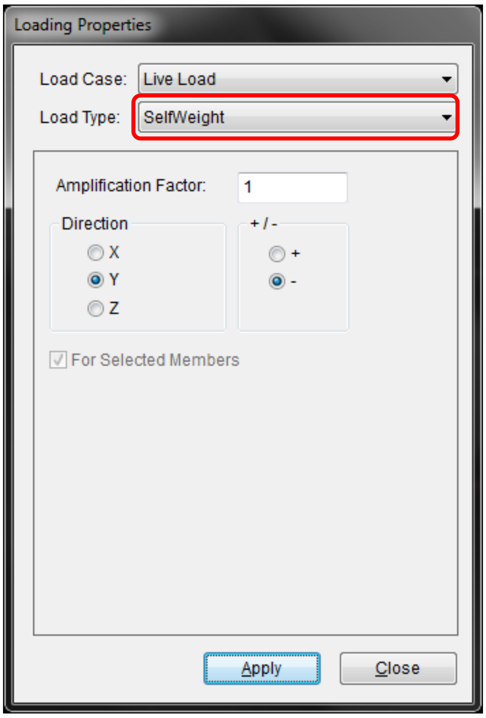
Click the ***Gr-Assign*** *>Member Loads>Cable Forces* command to assign the cable load to selected members.



Fill in the magnitude of pre-stressing force.

4.6.8.5 Self Weight

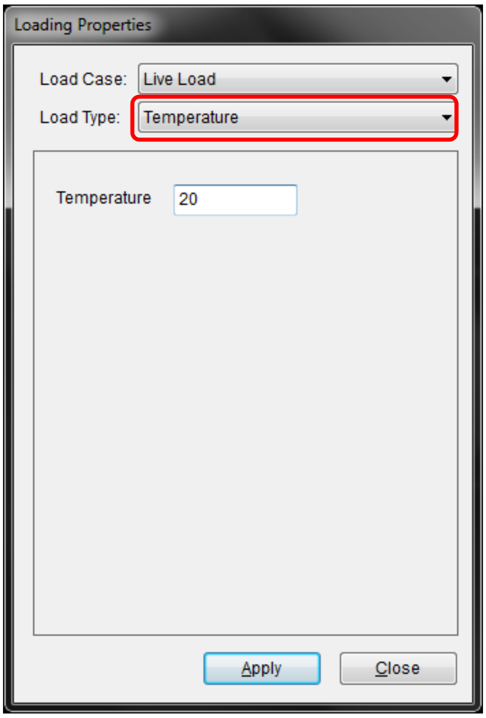
Click the ***Gr-Assign*** *>Member Loads>Self Weights* command to assign the self-weight to selected members.



Fill in the amplification factor, select the sign and direction for selected beam-column elements only. The other structural objects do not allow for self-weight individually.

4.6.8.6 Temperature

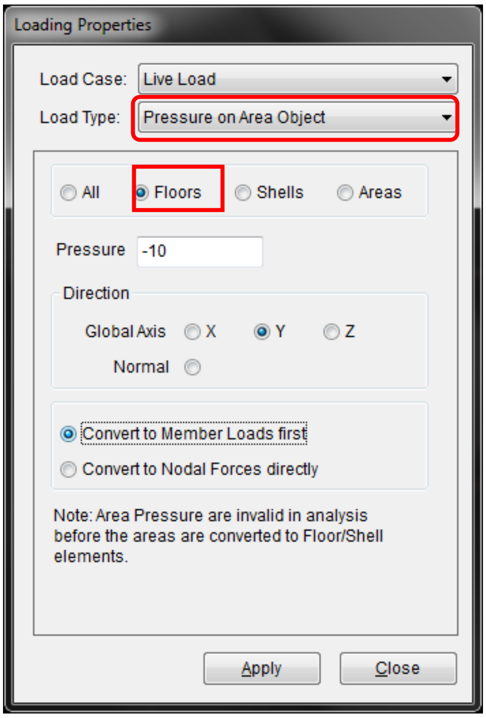
Click the ***Gr-Assign*** *>Member Loads>Temperature* command to assign the temperature load to selected members.



Fill in the magnitude of temperature for selected beam-column elements.

4.6.9 Floor Pressure

Click the ***Gr-Assign*** *>Floor Pressures* command to assign pressure to selected floors.

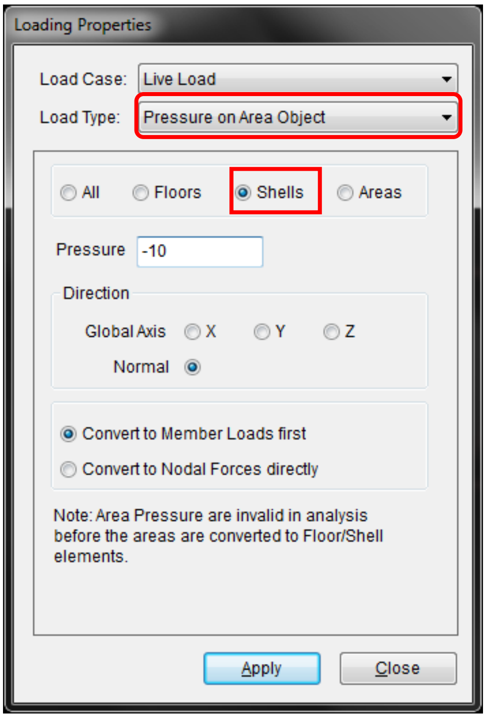


Fill in the magnitude and select the direction of the pressure. The pressure can be converted to either member loads or nodal forces directly.

- Convert to Member Loads First option. This command is to convert the floor pressures to member loads first by one-way or two-way mode if there are some beam-column elements under the floor elements.
- Convert to Nodal Forces Directly option. This command is to convert the floor pressures to joint loads directly.

4.6.10 Shell Pressure

Click the ***Gr-Assign*** *>Shell Pressure* command to assign pressure to selected shells.

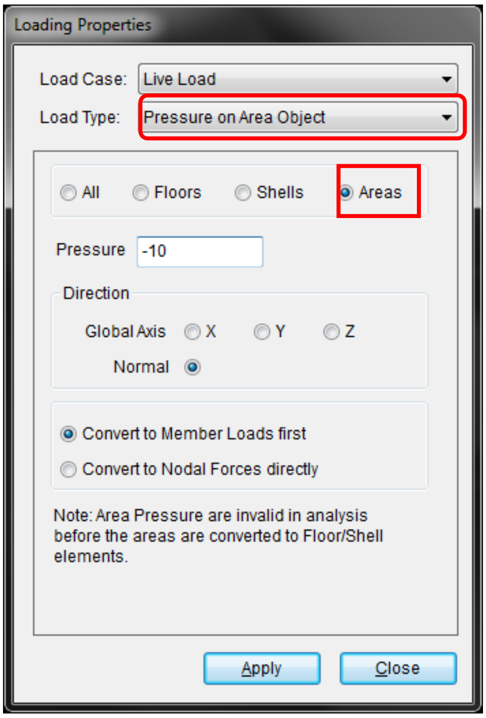


Fill in the magnitude and select the direction of the pressure. The pressure can be converted to either member loads or nodal forces directly.

- Convert to Member Loads First option. This command is to convert the shell pressures to member loads first by one-way or two-way mode if there are some beam-column elements under the shell elements.
- Convert to Nodal Forces Directly option. This command is to convert the shell pressures to joint loads directly.

4.6.11 Area Pressure

Click the ***Gr-Assign*** *>Area Pressure* command to assign pressure to selected areas.



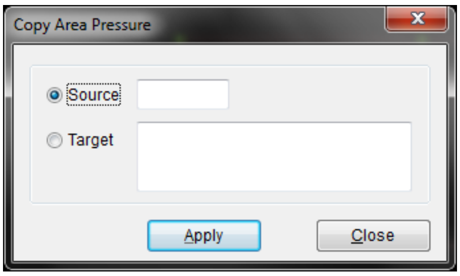
Fill in the magnitude and select the direction of the pressure. The pressure can be converted to either member loads or nodal forces directly.

- Convert to Member Loads First option. This command is to convert the area pressures to member loads first by one-way or two-way mode if there are some beam-column elements under the areas.
- Convert to Nodal Forces Directly option. This command is to convert the area pressures to joint loads directly.

Warning: Area pressures are invalid in the analysis before the area(s) are converted to floor/shell elements.

4.6.12 Copy Area Pressures

Click the ***Gr-Assign*** *>Copy Area Pressures* command to copy the area pressure from one specified area to several areas.



Input the area No. in “Source”. You can also pick up an area on the screen.

Input several areas in “Target”. You can also select the areas from the screen directly.

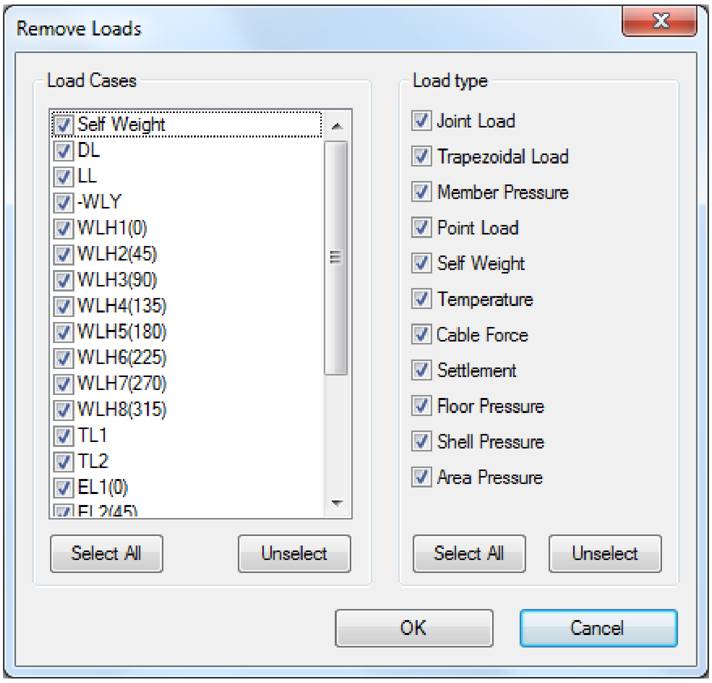
Click Apply button and all pressures in different load cases of the “Source” area can be copied to the “Target” areas.

Click Close button to quit this window.

After this operation, all "Target" areas will be assigned with the same pressures in many load cases of the "Source" area.

4.6.13 Remove Loads

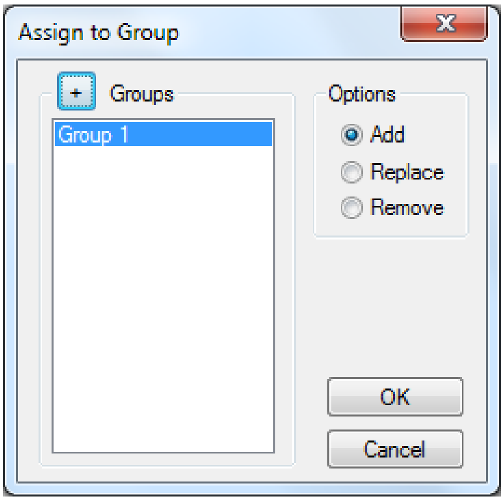
Click the ***Gr-Assign*** *>Remove Loads* command to delete the loadings within the selected load cases in the selected objects.



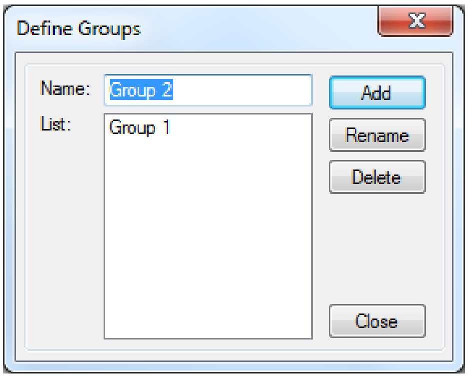
Select the load cases inside the load case and different load type to remove the loading of selected objects.

4.6.14 Assign to Group

Click the ***Gr-Assign*** *>Assign to Group* command to add the selected objects to the specific group.



Choosing a group, the selected objects can be added, replaced or deleted from the group. The group can be defined by clicking the  button on the left side of Groups.



Note: Node, member, shell, floor and area objects can be assigned to one group.

4.7 Analysis

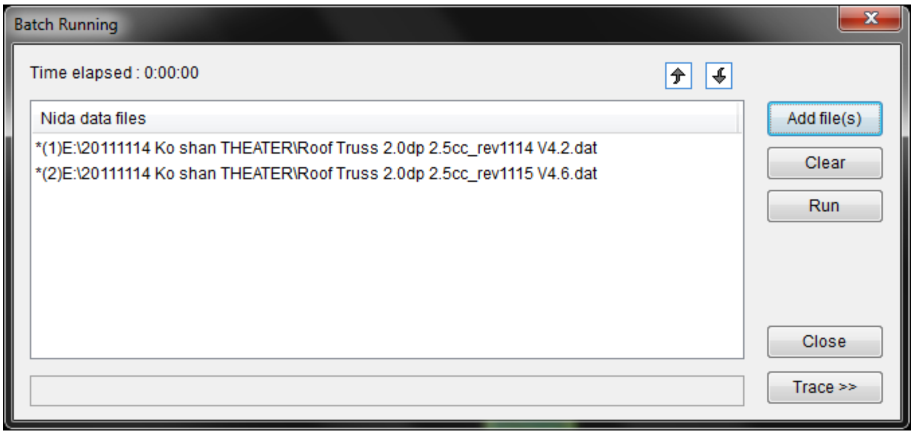
4.7.1 Run

Click the ***Analysis***>*Run* command to run a number of analysis cases which are set up in Set Analysis Case step.

Shortcut : Toolbar : None Keys : F5

4.7.2 Run a Batch of Files

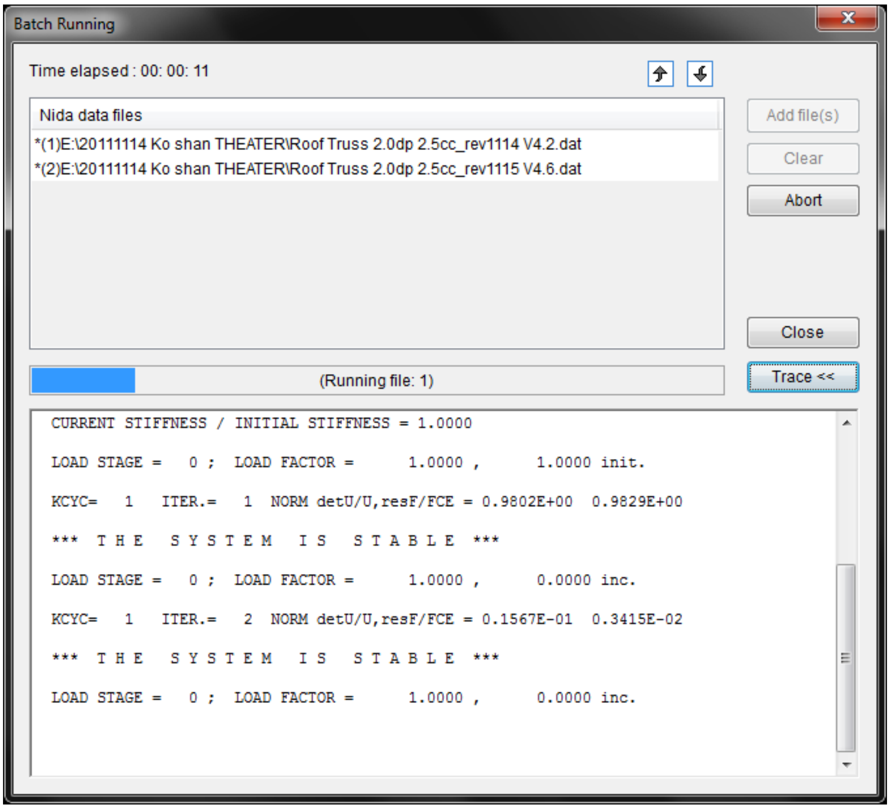
Click the ***Analysis***>*Run a Batch of Files* command to run a batch of NIDA data files.



Click the Add file(s) button to add the NIDA files to run.

Click the Clear button to remove the selected NIDA data files.

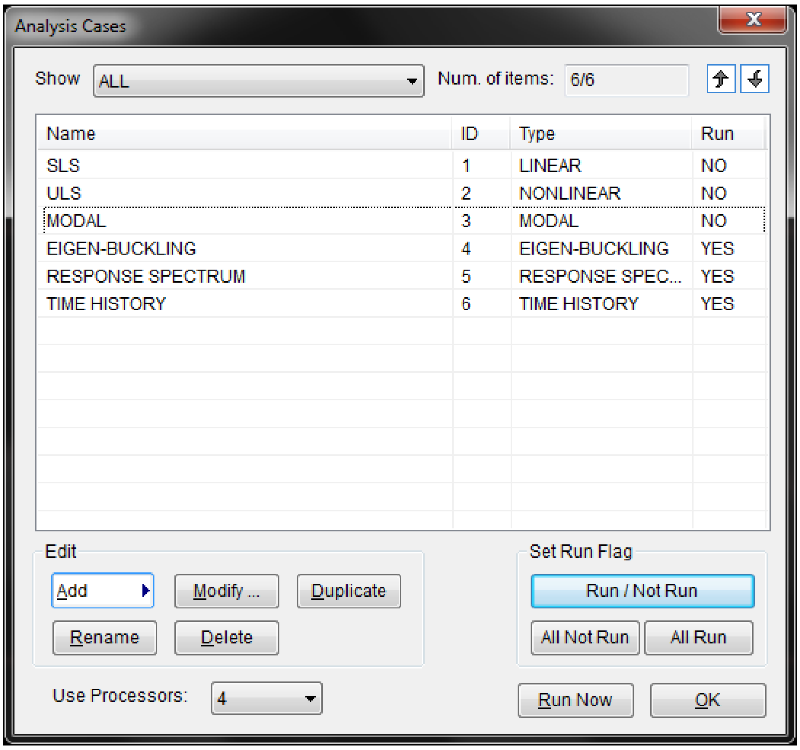
Click the Run button to run the files in sequence.



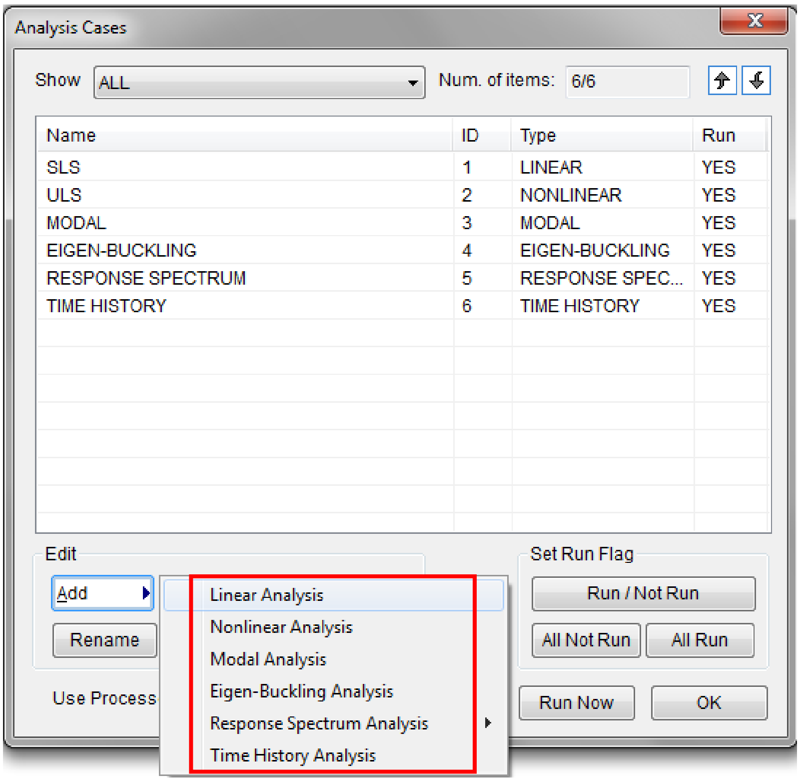
During the analysis process, the message from analysis can be viewed by clicking the Trace button.

4.7.3 Set Analysis Case

Click the ***Analysis***>*Set Analysis Case* command to define analysis cases.



Click Add button to add analysis cases. Six types of analysis (Linear Analysis, Nonlinear Analysis, Modal Analysis, Eigen-buckling Analysis, Response Spectrum Analysis and Time History Analysis) can be selected.



Click Modify button to modify the analysis case.

Click Rename button to change the name of analysis case

Click Delete button to delete the analysis case.

Click Run/Do Not Run button to toggle the selection status of analysis.

Click All Not Run button to set all analysis cases in un-active status.

Click All Run button to set all analysis cases in active status.

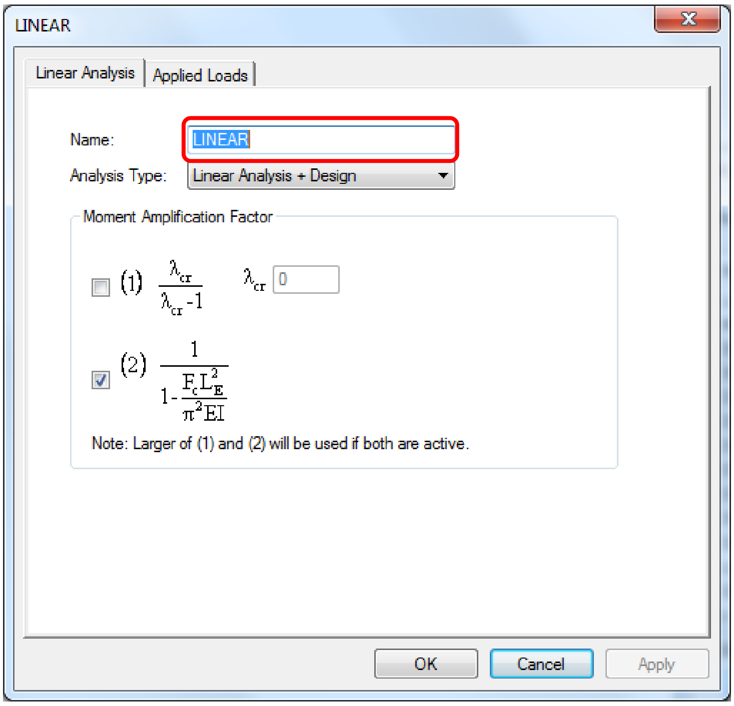
Click Run Now Button to start running the active analysis cases.

4.7.3.1 Linear Analysis

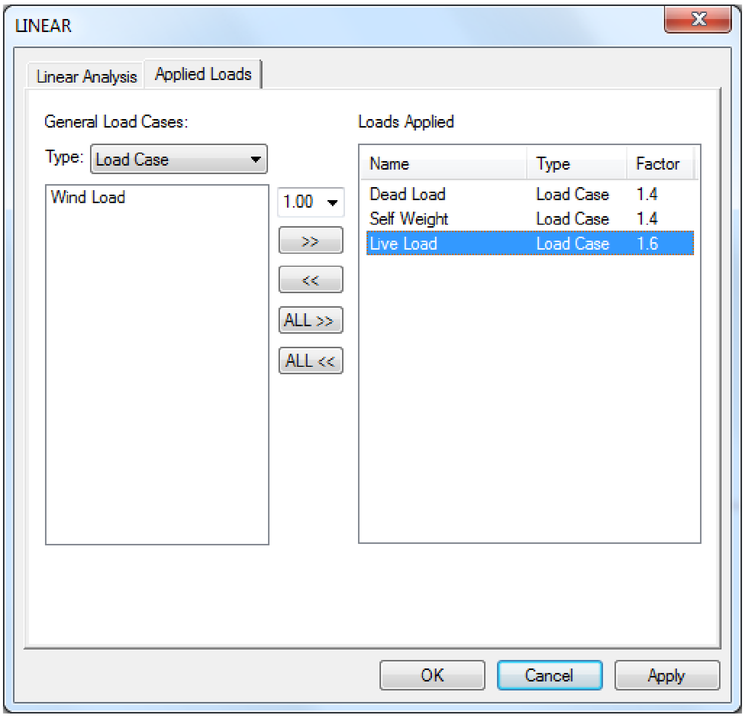
Click ***Linear Analysis*** to define a linear type analysis case. Give a case name and select the linear analysis type.

Noted that when selecting ‘Linear Analysis + Design’ NIDA will check the member adequacy by selected design code, for example, design strength reduced by plate thickness, section classification, lateral-tortional bucking, column buckling. Otherwise, the checking by code will be skipped.

The linear moment can be amplified by filling in the parameters in ‘Moment Amplification Factor’ such as elastic critical load factor and/or the equation from CoPHK (2011) when using CoPHK for steel design.



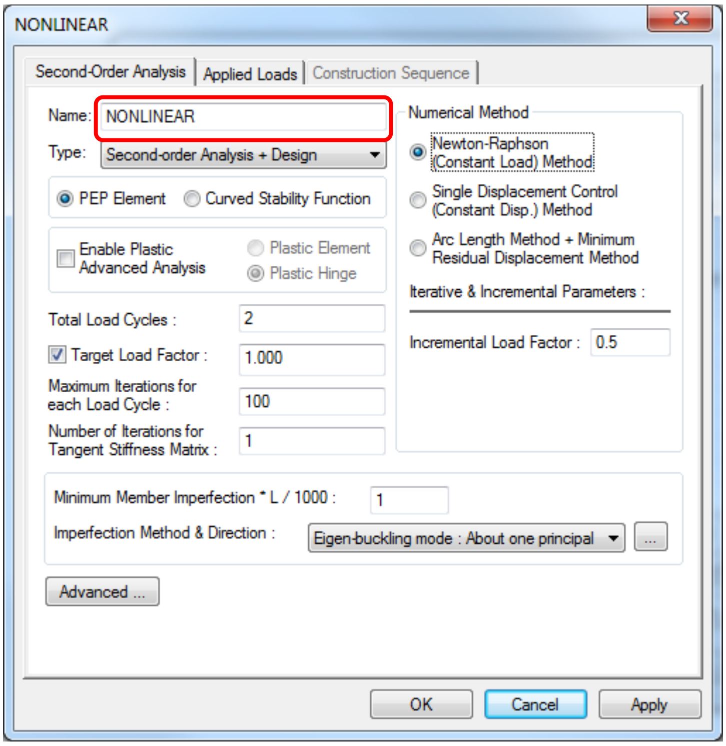
Click the ***Applied Loads*** tab to select the type of general load cases (Load case, Combined and Spectrum Analysis Case) to be added to analysis case. Different factors can be assigned to the analysis case.



Reference: Definition of Linear Analysis

4.7.3.2 Nonlinear Analysis

Click ***Nonlinear Analysis*** to define a second-order elastic or plastic analysis case. Give a case name and select the nonlinear analysis type.



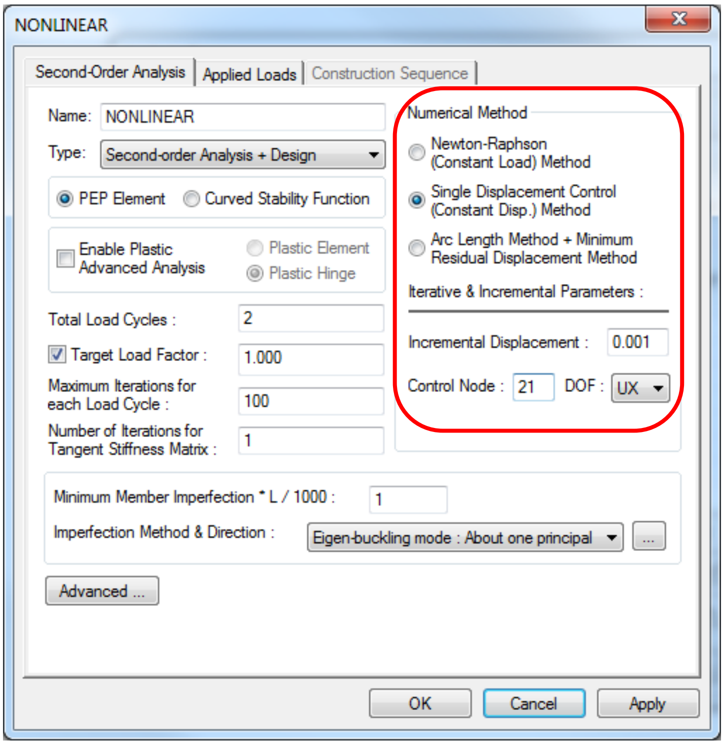
Noted that when selecting 'Second-order Analysis + Design' NIDA will check the member adequacy by selected design code, for example, design strength reduced by plate thickness, section classification, lateral-tortional bucking, column buckling. Otherwise, the checking by code will be skipped.

Either PEP Element (Chan and Zhou 1994) or Curved Stability Function (Chan and Gu 2005) can be used to simulate the beam-column elements with initial member imperfection.

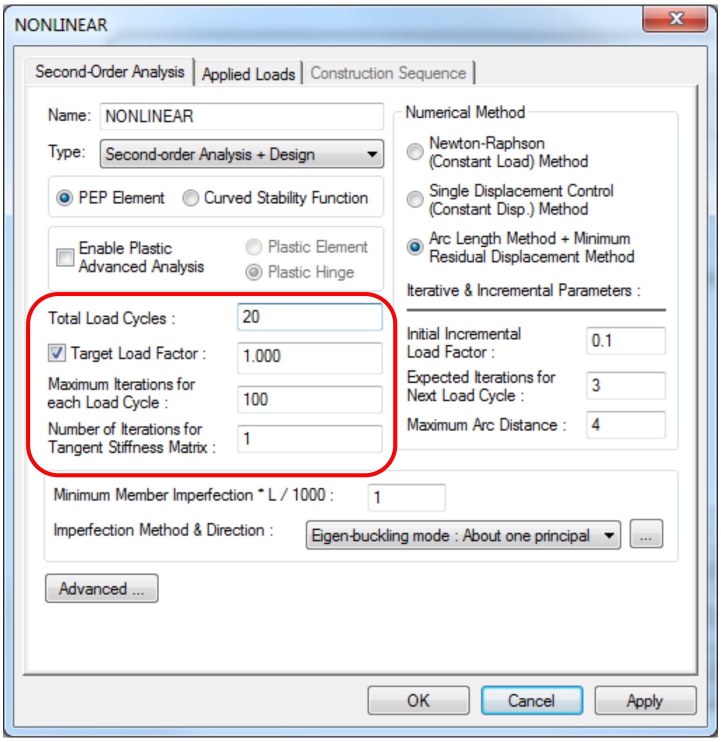
Click the Enable Plastic Advanced Analysis to allow for the material yielding in beam-column elements. Either Plastic Element or Plastic Hinge can be used for plastic analysis method.

The nonlinear solution in NIDA is essentially based on the incremental-iterative procedure. There are three numerical methods for Load Incremental Scheme and Interactive Scheme, i.e. Newton-Raphson (Constant Load) Method, Single Displacement Control (Constant Displacement) Method, and Arc Length Method + Minimum Residual Displacement Method.

In Single Displacement Control method and Arc Length Method + Minimum Residual Displacement Method, user can set the parameters for load incremental scheme and interactive scheme.



It is also required to specify the total load cycles, target load factor, maximum iterations for each load cycle, number of iterations for tangent stiffness. The definition of each parameter is explained in General Nonlinear Parameters for Analysis.



Total Load Cycles

It describes the maximum number of total load cycles for the nonlinear analysis. The analysis process will be terminated once the maximum cycle reaches. The analysis process may be also terminated when input errors or divergence found, or when the target load factor reaches if Target Load Factor function is enable.

Target Load Factor

It describes the expected load factor for the nonlinear analysis. The analysis process will be terminated once the load factor reaches the target one even though the load cycles may be less than the total load cycles. The load factor may be not able to reach the target one due to structural problem or limitation of total load cycles. For the former user need to redesign the structure while for the latter user may increase the total load cycles.

Maximum Iterations for each Load Cycle

It describes the maximum number of iteration for each load cycle in the analysis. If the equilibrium condition is satisfied before this number is reached or the iteration number is equal to this assigned iteration number, another load step will be imposed until the permitted number of load steps is reached. The tolerance for equilibrium check is 0.1 % by default. That is, when the Euclidean norms of the unbalanced displacements and the unbalanced forces are less than respectively 0.1 % of the total applied forces and the total accumulated displacements, the equilibrium condition is assumed to have been satisfied.

Number of Iterations for Tangent Stiffness Matrix

It describes the number of iterations for the tangent stiffness matrix to reform during the iterative process. When this number is specified to be very large or simply equal to the “maximum number of iterations for each load cycle” above, the iterative scheme will then become the modified Newton Raphson method. If the number here is specified as "1”, it becomes the Newton Raphson method. If the number is between these two extremes, the method is a mixed Newton-Raphson method. When compared to modified Newton Raphson method, the Newton Raphson method generally requires less number of iterations for convergence, but longer time for each iteration. It is recommended to use the Newton Raphson method

Incremental Load Factor

This factor will be used as the first load factor used for the analysis and the load factor increment in subsequent analysis. It is different from the design load factor behind “header load” which is multiplied to the input load to obtain the design load vector and will not appear in the plotting of equilibrium or load-deflection curve with its value generally taken as, for example, 1.6 for wind, 1.4 for self-weight etc. The load factor described here is used as the ratio of the current applied load to the input design load. For example, if a structure yields at a load factor of 2.6, it means when the applied load is 2.6 of the design load, the structure yields.

Parameters for Numerical Methods

1. Newton-Raphson (Constant Load) Method
   a. Incremental load factor – constant load increment ratio
2. Single Displacement Control (Constant Displacement) Method
   a. Incremental displacement – constant displacement increment
   b. Control node – monitored node
   c. DOF - degree of freedom of control node
3. Arc Length Method + Minimum Residual Displacement Method
   a. Initial incremental load factor – load increment ratio for first load cycle
   b. Expected iterations for next load cycle – iterations to control the load increment
   c. Maximum arc distance – limit of arc distance increment

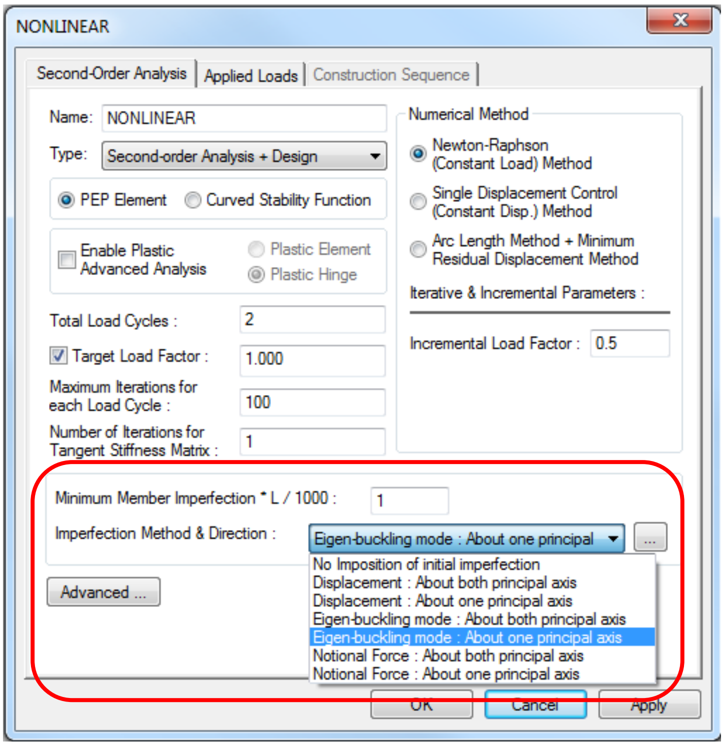
Minimum Member Imperfection

The minimum magnitude of initial imperfection is taken as 1 (*Input value*) /1000 of the member length if the initial imperfections are allowed.

Imperfection Method & Direction

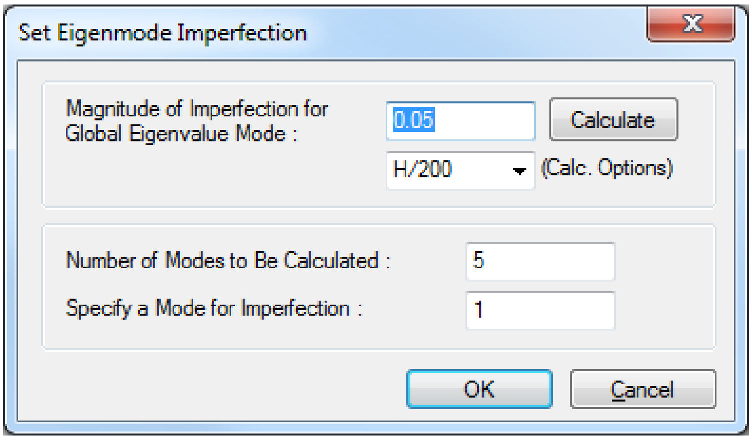
It describes the direction of initial imperfection. It can be no initial imperfection, initial imperfection in one principal plane causing less severe effect than initial imperfection in both the principal planes.

The frame global and member imperfections should be defined in analysis. There are three types of imperfections including 'displacement (DIMP)', 'eigen-buckling mode (EIMP)', and 'notional force (NIMP)'. The imperfection could be added in one or both principal axes of a beam-column element. If no initial imperfection is required, select the 'No Imposition of Initial Imperfection'. Click the Setting… button to set the required parameters for imperfection analysis.



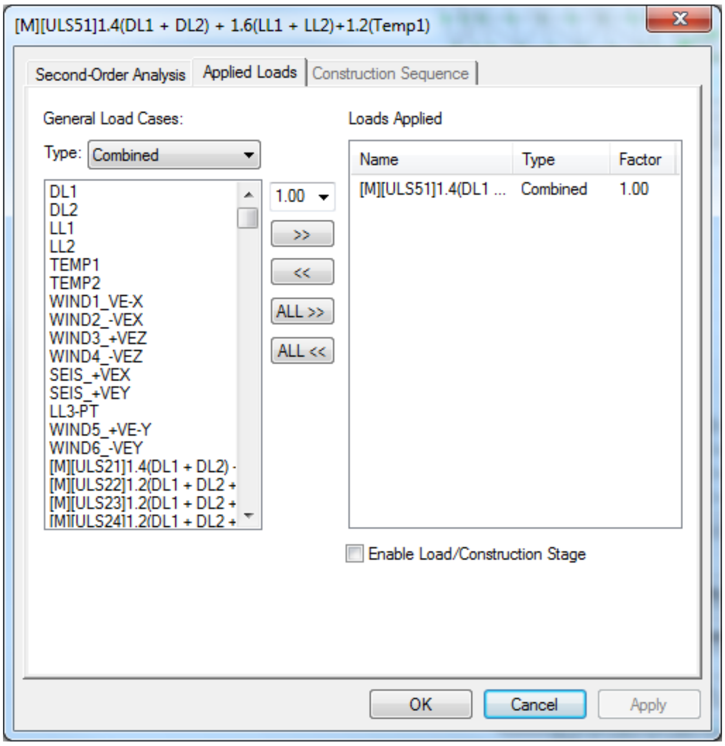
Magnitude of Imperfection for Global Eigenvalue Mode

This value is the magnitude of imperfection when eigenvalue buckling mode is adopted. After the eigenvalue analysis, the eigen-mode is determined and a set of initial imperfection is determined for the structure with this mode shape. This number is for the magnitude (maximum) of the initial deflection for the eigen-mode which is then added to the initial geometry of the structure.

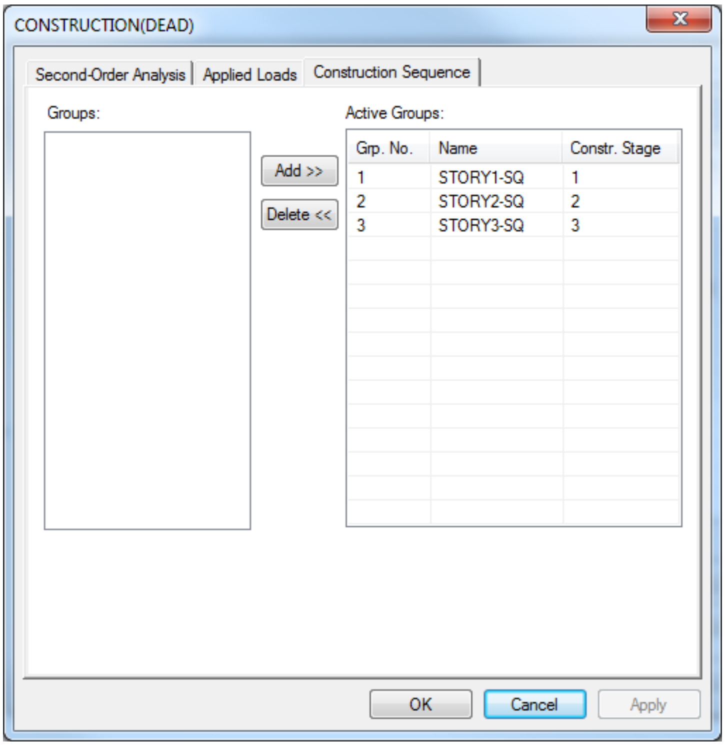


Click the ***Advanced Setting*** button to select the recovery force method.

Click the ***Applied Loads*** tab to add loading for analysis. Appropriate load factor can be input for load combination purpose in the analysis case.



Click the ***Construction Sequence*** tab to add different construction stage.



References:

Definition of Nonlinear Analysis

Numerical Method

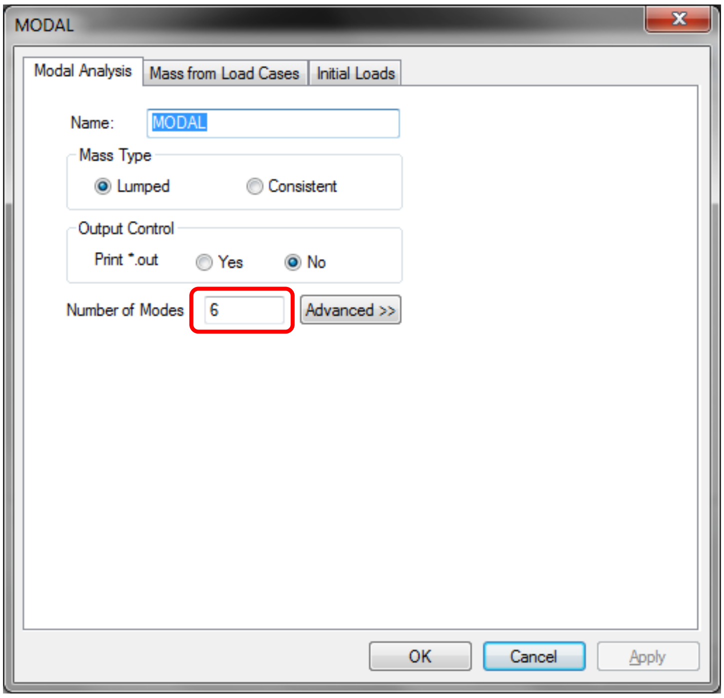
[Load Incremental Scheme]

[Interactive Scheme]

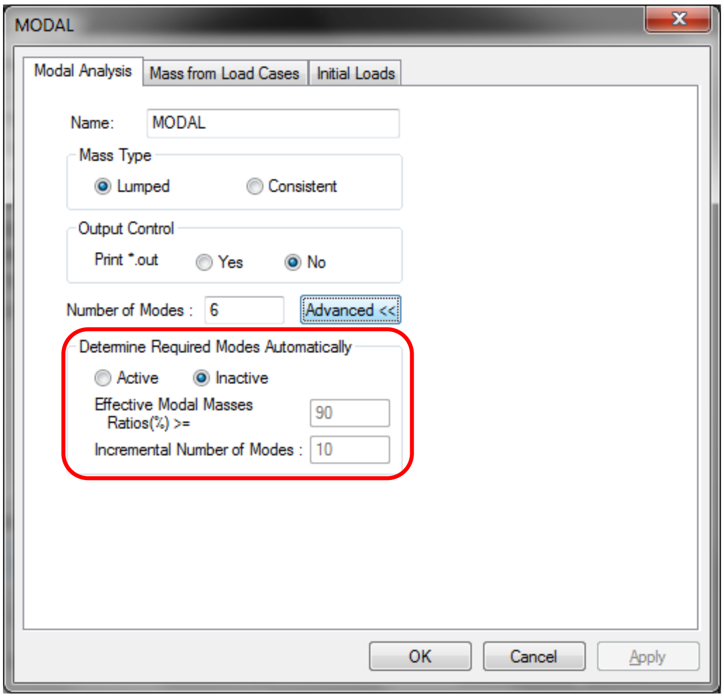
[Imperfection]

4.7.3.3 Modal Analysis

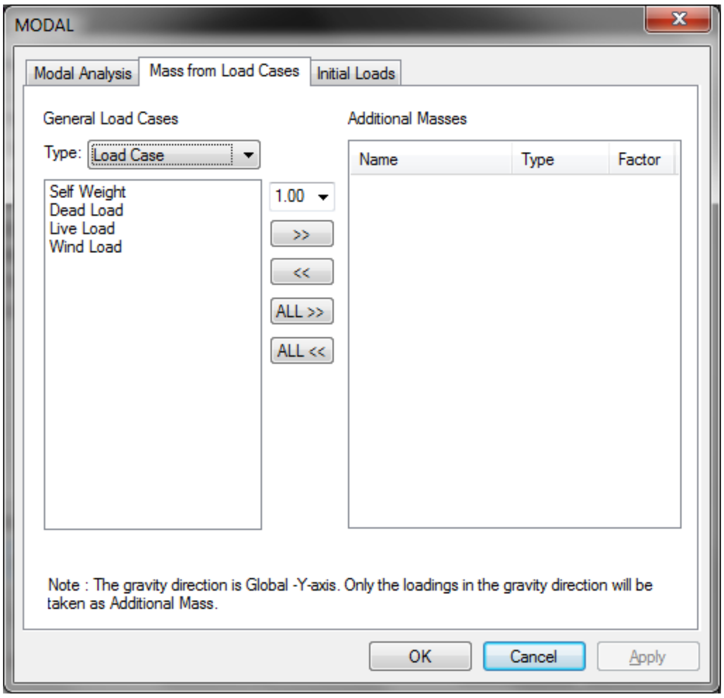
Click the ***Modal Analysis*** to define a modal analysis case. Give a case name, select the mass type (Lumped or Consistent) and determine whether the output results have to be printed in *.out file or not. The number of required modes should be input.



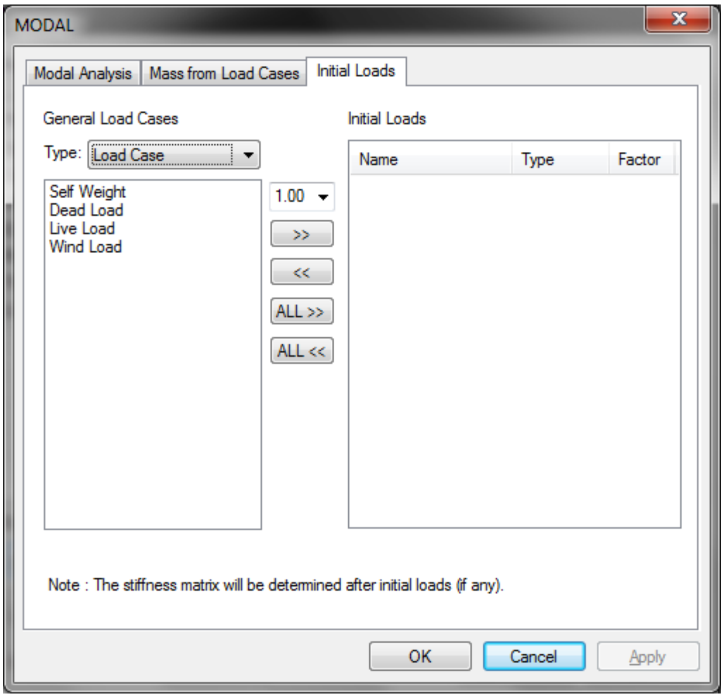
Click the ***Advanced*** button to activate the function for determining the mode numbers automatically by specifying the effective modal masses ratios.



Click the Mass from Load Cases tab to define additional masses from load cases. The self-weight of structural objects will be automatically included in the modal analysis. However, if additional masses due to other dead and live loads have to be considered, you may consider them by adding load cases which include mass components. Noted that only the loadings in gravity direction will be added to the mass matrix.



Click the Initial Loads tab to consider stiffness change in structural system due to loads like temperature and pretension forces. The effect of initial load is not necessary for all kinds of structures.

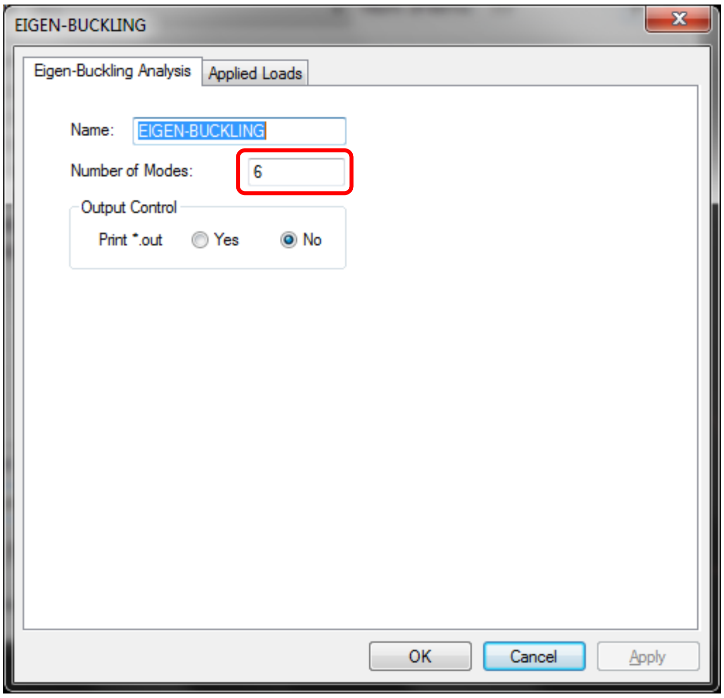


Reference:

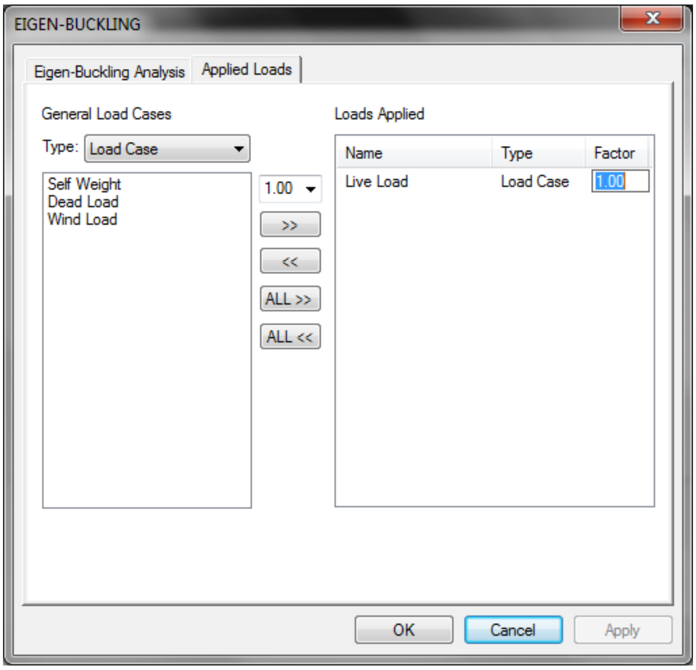
[Definition of Modal Analysis]

4.7.3.4 Eigen-buckling Analysis

Click the ***Eigen-buckling Analysis*** to define an eigen-buckling analysis case. Give a case name and determine whether the output results have to be printed in ***.out** file or not. The number of modes required for observation should be given.



Click the ***Applied Loads*** tab to add the loads for analysis. The default load factor for each load case is 1.0 and it can be changed if needed.

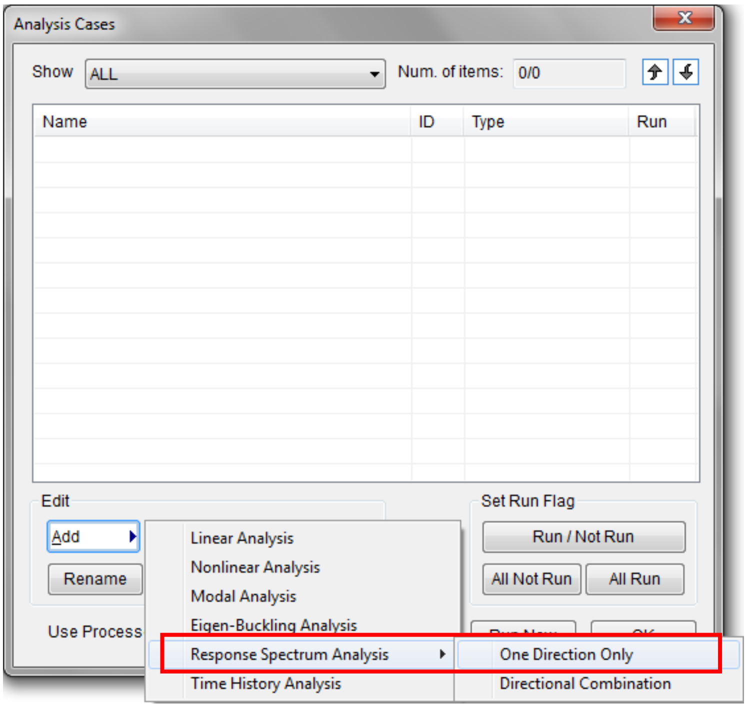


Reference:

[Definition of Eigen-buckling Analysis]

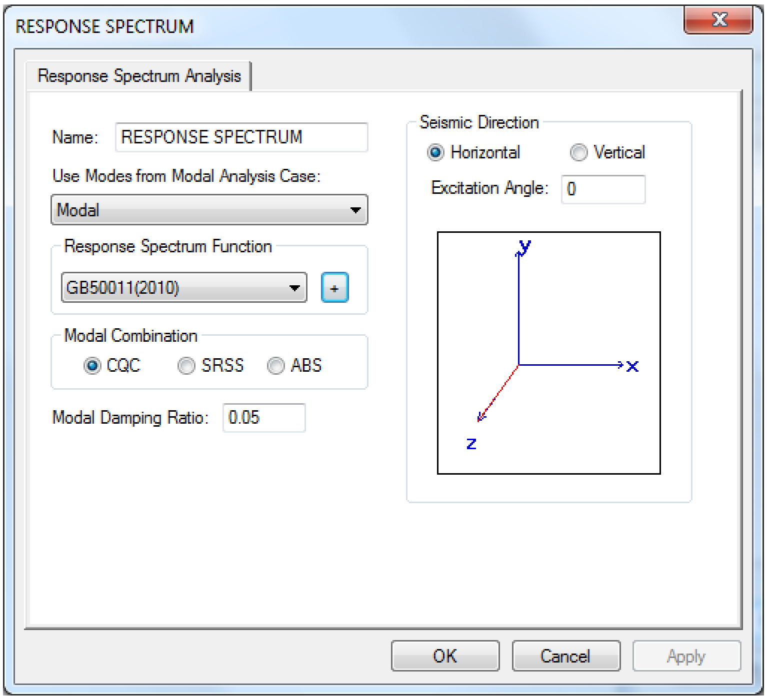
4.7.3.5 Response Spectrum Analysis

Click the ***Response Spectrum Analysis*** and select **One Direction Only** or **Directional Combination** to define a response spectrum analysis case.



- Response Spectrum Analysis - One Direction Only

Define a response spectrum analysis in one direction.



For one direction only, input the name of analysis case and select the Use Modes from Modal Analysis Case.

Select the Response Spectrum Function from the dropdown list if any or click the Define button to create a new spectrum function for selection.

Select the Modal Combination , there are three methods available, i.e. CQC (Complete Quadratic Combination), SRSS (Square Root of the Sum of the Squares) and ABS (Absolute Sum).

<u>CQC (Complete Quadratic Combination) Method</u>

The CQC method takes into account the statistical coupling between closely-spaced modes caused by modal damping. You may specify a CQC damping ratio (damp) measured as a fraction of critical damping: 0 <= damp<1. This should reflect the damping that is present in the structure being modeled. Note that the modal damping ratio is different from the function damping ratio defined in the Response Spectrum Function. The latter is developed independently for an assumed value of structural damping. Normally these two damping values should be the same. If the damping is zero, this method degenerates to the SRSS method.

<u>SRSS (Square Root of the Sum of the Squares) Method</u>

The SRSS method is to combine the modal results by taking the square root of the sum of their squares. This method does not take into account any coupling of Modes as in the CQC method.

ABS (Absolute Sum) Method

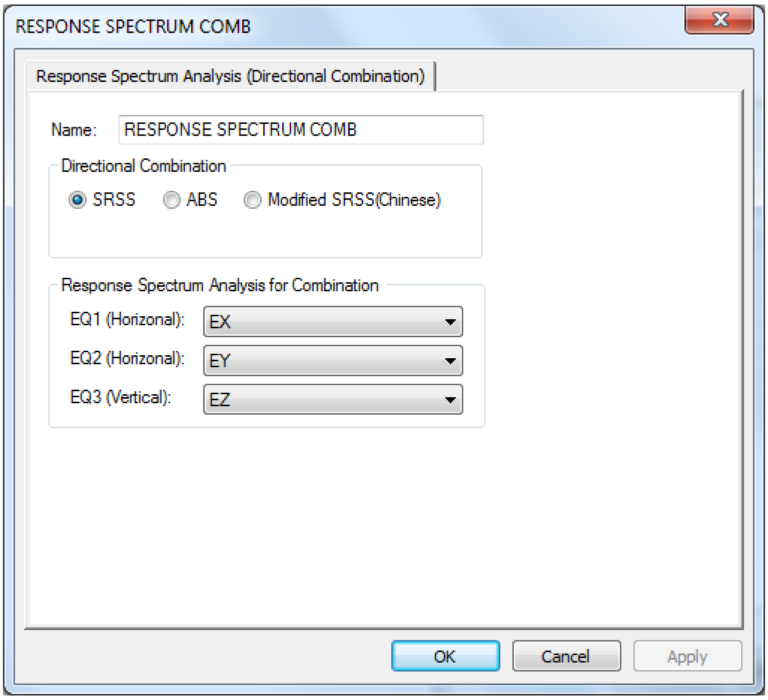
The ABS is to combine the modal results by taking the sum of their absolute values. This method is usually over- conservative.

Modal Damping Ratio: For CQC method only. This ratio is to consider the coupling between closely-spaced modes.

Seismic Direction and Excitation Angle: To define the seismic direction. When the seismic direction is "Horizontal", the "Excitation Angle" should be further defined to indicate the seismic direction. The vertical direction is the same as the gravity direction.

- Response Spectrum Analysis - Directional Combination

Combine the results of two or three "one direction response spectrum analysis".



To select the method of Directional Combination, there are three available – SRSS, ABS and Modified SRSS (Chinese).

SRSS

Combine the directional results by taking the square root of the sum of their squares. This method is in variant with coordinate system, i.e., the results do not depend upon the choice of coordinate system when the given response-spectrum curves are the same. This is recommended for directional combination.

ABS

Combine the directional results by taking the sum of their absolute values. This method is usually over- conservative.

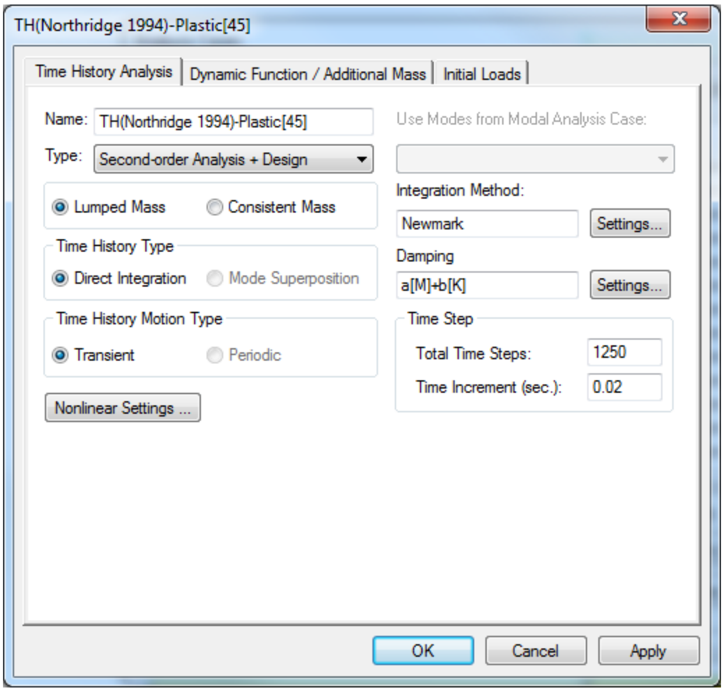
Modified SRSS (Chinese)

Combine the directional results by a modified SRSS method complied with Chinese Seismic Design Code GB50011-2001

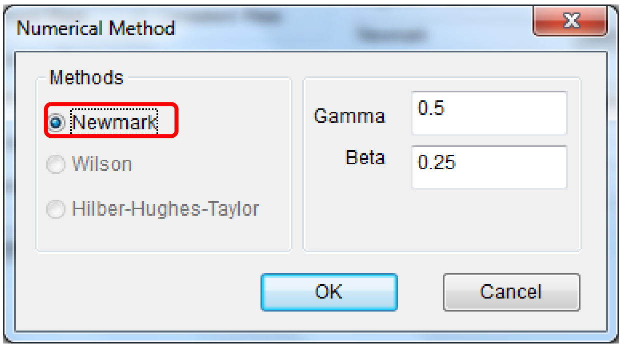
Select the Response Spectrum Analysis cases for combination in EQX, EQY and EQZ directions.

4.7.3.6 Time History Analysis

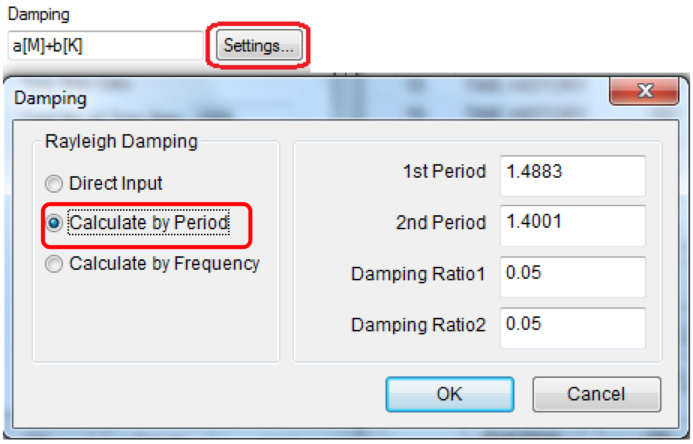
Click the ***Time History Analysis*** to define a time history analysis case. Give a case name and select the mass type, time history type and time history motion type.



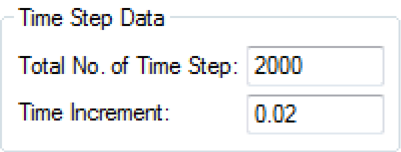
Select an integration method and fill in the parameters as required.



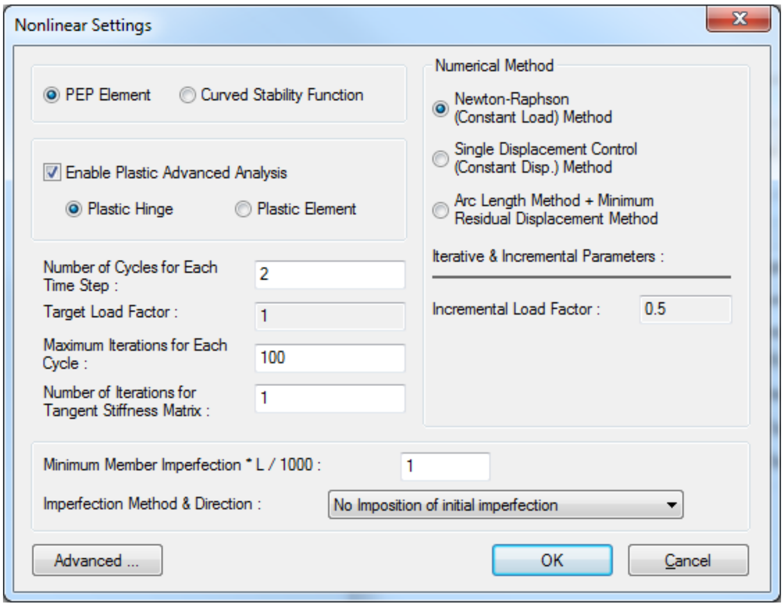
Define the damping as below.



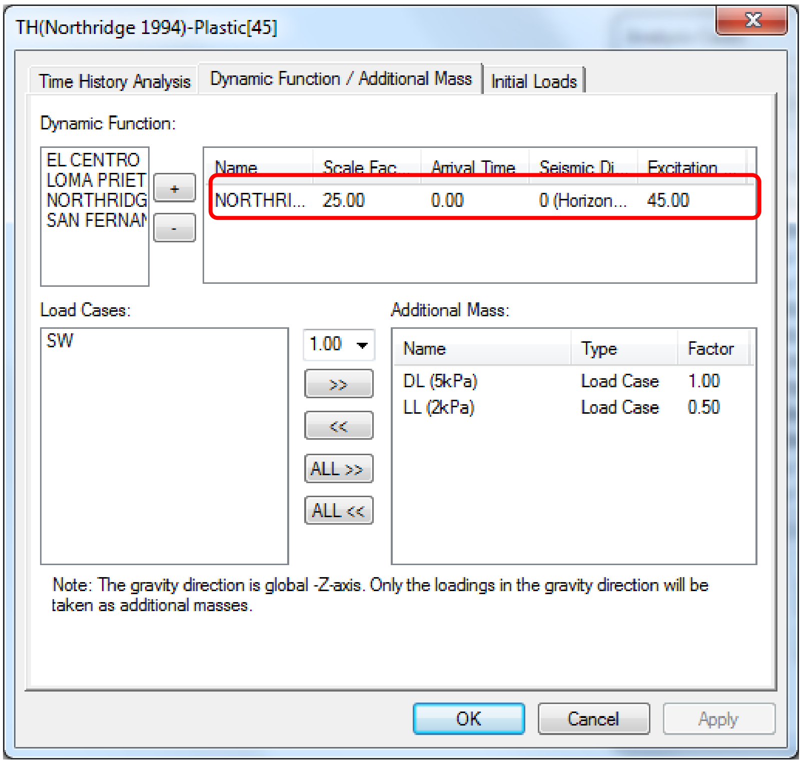
Fill in the total no. of time steps and time increment.



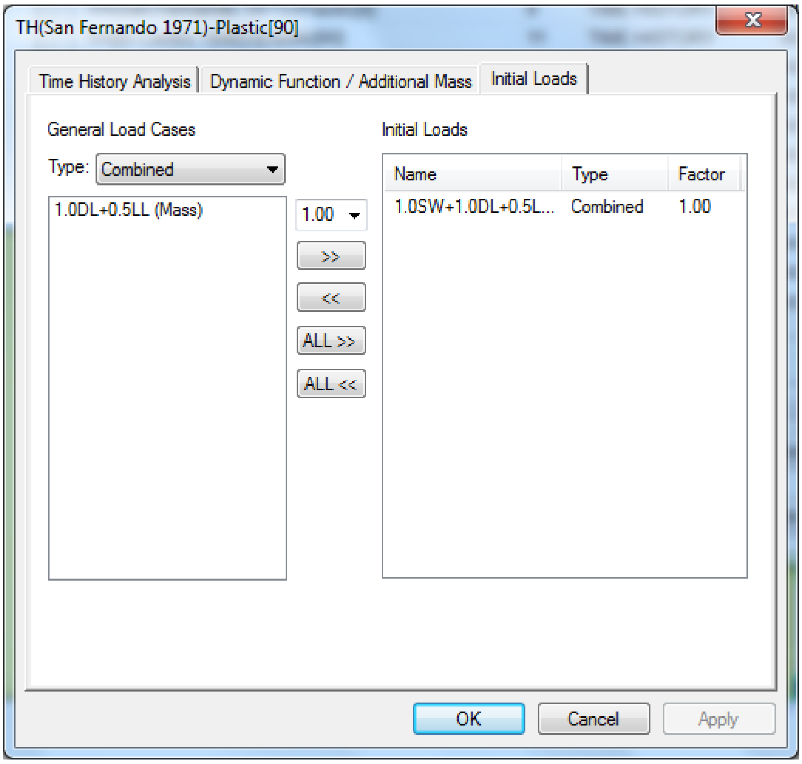
For more advanced settings for nonlinear solution, click Advanced Setting button and a dialog will pop up as below.



Click Dynamic Function / Additional Mass tab to select one or more than one dynamic functions as earthquake input. Also, the additional masses from load cases such as dead load and live load can be input in this tab.

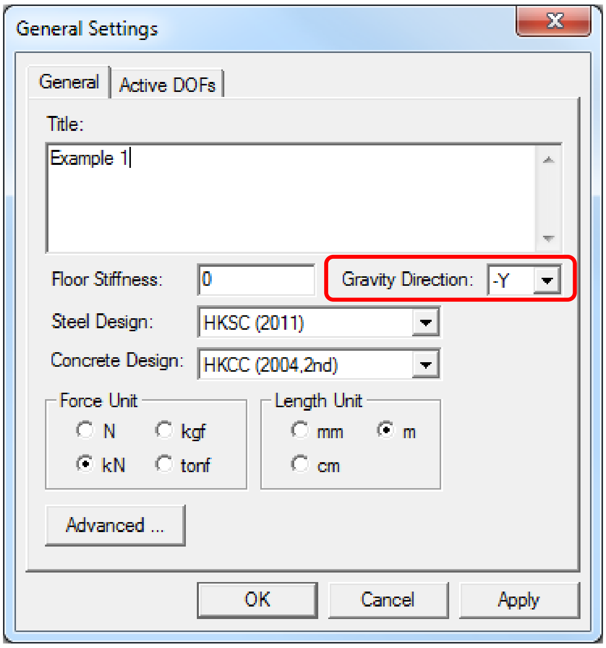


Click Initial Loads tab to consider static loads such as dead load and live load.



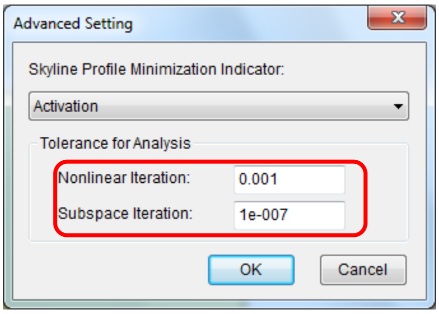
4.7.4 Analysis & Design Parameters Setting

Click the ***Analysis***>*Analysis & Design Parameters Setting* command to change the general settings of the model.

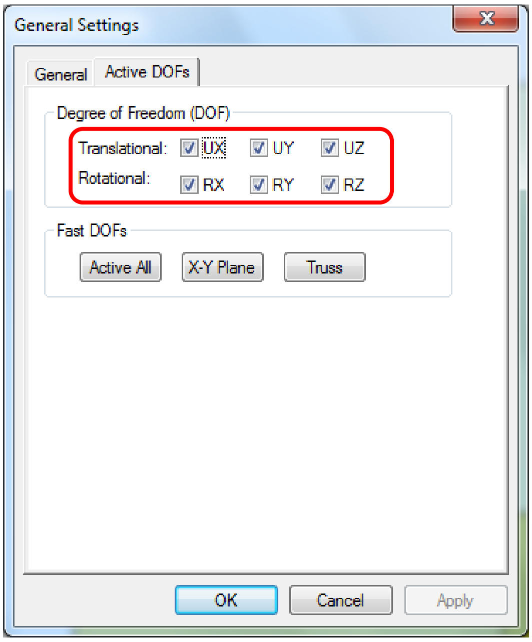


The first tab window is the general information and global control parameters of the project. You can change the project title, define the floor stiffness, set the direction of gravity, select the steel and concrete design code, and change the force unit as well as the length unit.

Click the Advanced button to select the Skyline Profile Minimization Indicator for memory optimization and modify the tolerances for convergence of analysis.



The second tab window is used to set the activation of degrees of freedom in 6 directions (Ux, Uy, Uz, Rx, Ry and Rz). You can use the 'Fast DOFs' function by clicking the Active All, X-Y Plant or Truss button to set up the degrees of freedom in a fast way.



4.8 Post

4.8.1 Show Deformed Shape

Click the ***Post***>*Show Deformed Shape* command to view the deformed shape of the model after finishing the analysis.

Shortcut : Toolbar : Keys : None

4.8.2 Show Undeformed Shape

Click the ***Post***>*Show Undeformed Shape* command to view the undeformed shape of the model.

Shortcut : Toolbar : Keys : None

4.8.3 Show Deformed & Undeformed Shape

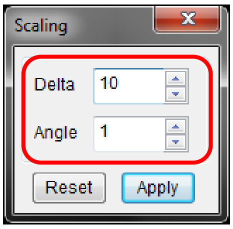
Click the ***Post***>*Show Deformed & Undeformed Shape* command to view the deformed and undeformed shape of the model together at the same window.

Shortcut : Toolbar : Keys : None

4.8.4 Display Scale

Click the ***Post***>*Display Scale* command to magnify displacement of nodes and rotation of the members of deformed structure.

Shortcut : Toolbar : Keys : None

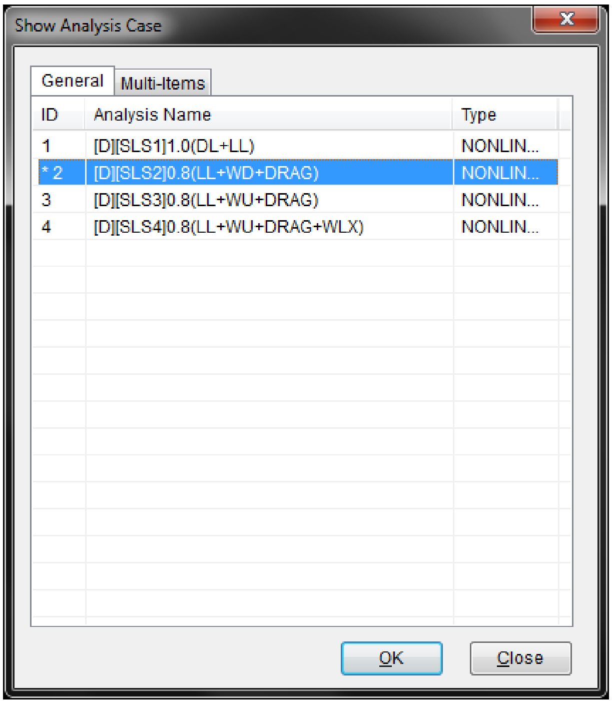


To magnify the deflected shape, change the value of "Delta" or "Angle" or both of them. The displacement or rotation of the nodes will be multiplied by this factor. Click "Reset" button, all values will be reset to 1.

4.8.5 Show Analysis Case

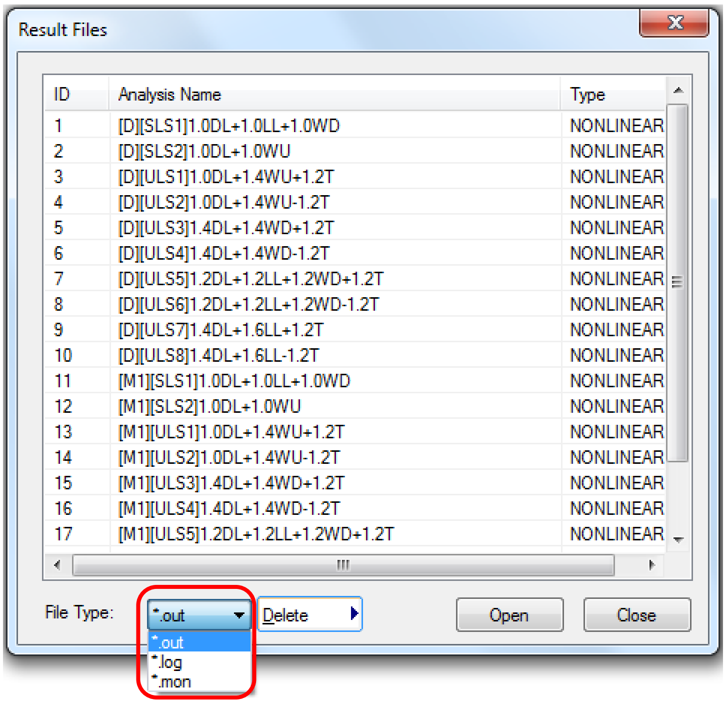
Click the ***Post***>*Show Analysis Case* command to show the results of the selected analysis case on the screen, such as internal forces and moments, nodal displacements and reactions, shell stresses and so on.

Shortcut : Toolbar : Keys : None



4.8.6 Show Result Files

Click the ***Post****>Show Result Files* command to open the results in text file. There are four types of result file associated with different information.



- *.out

It is a text file which includes the detailed input data, analysis results such as nodal displacements, member internal forces and moments, shell stresses and nodal reactions. Each analysis case contains a *.out file with analysis case number.

- *.log

It is a text file which stores important messages during analysis procedure, such as load increment, divergence and singular information. Some information of error, warning and note may be important for correct solutions.

Each analysis case contains a *.log file with analysis case number.

- *.mon

It is a text file which stores some messages during data reading procedure. The information may help correct the errors due to modelling.

4.8.7 Nodal Results

4.8.7.1 Reactions/Displacements

Click the ***Post****>Nodal Results >Reactions/Displacements* command to show the nodal reaction forces and displacements when you click the node of deformed structure.

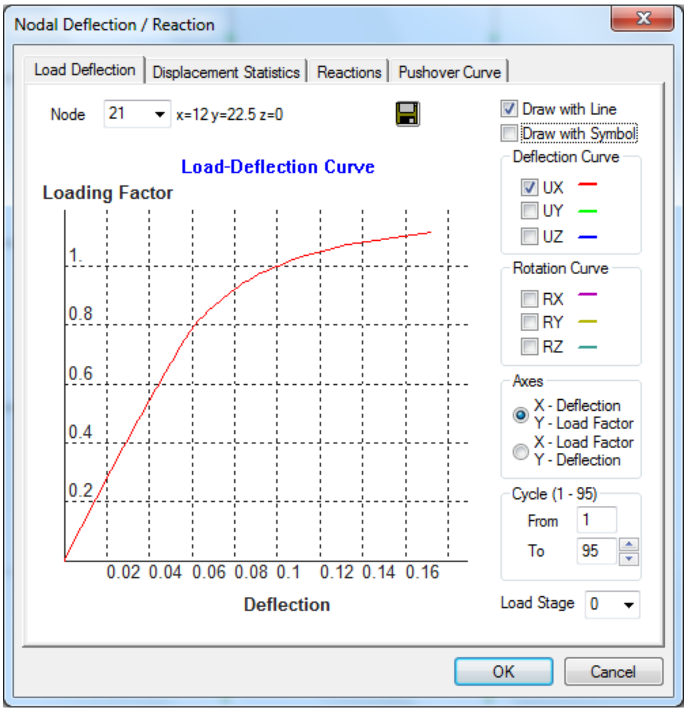
4.8.7.2 Load Deflection Curve/Reactions

Click the ***Post****>Nodal Results>Load Deflection Curve/Reactions* command to show the load-deflection curve, displacement statistics and reaction forces and moments.

Shortcut : Toolbar : Keys : None

- Load-Deflection Curves

When loading is applied to a structure, it will deform according to the load factor. The deflection versus load plot and other information about nodes can be viewed in this Load-Deflection Curve window:

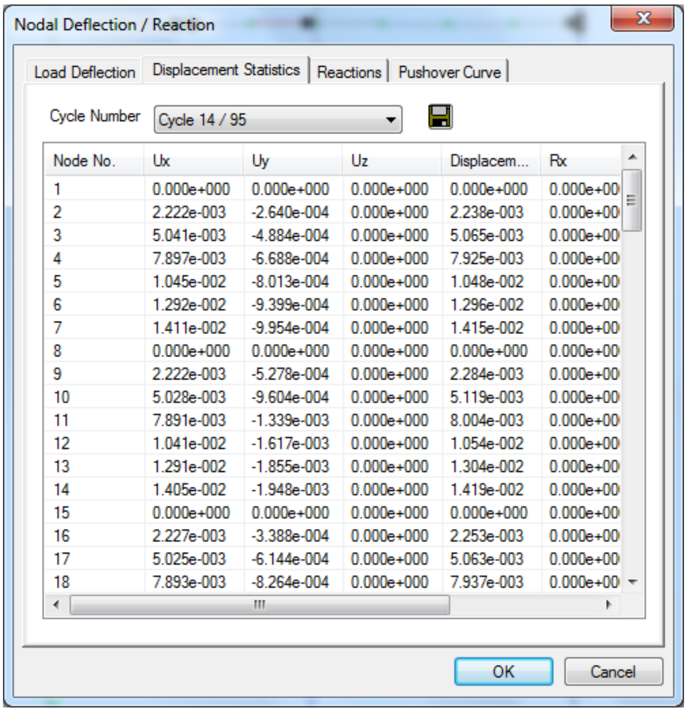


The first tab represents the deflection of selected node against load factor. The two axes represent the change of load factor and deformation of a node. The number of cycle can be set in Cycle box.

Six different curves against load factor can be plotted, i.e. deflections in X, Y, Z, and rotations about X, Y, Z with appropriate selection. Tick the check boxes at right hand side of diagram to show/hide the curves and draw the curve with line or symbol.

To save the results in text file, just click the button  at top right of this window.

- Displacement Statistics



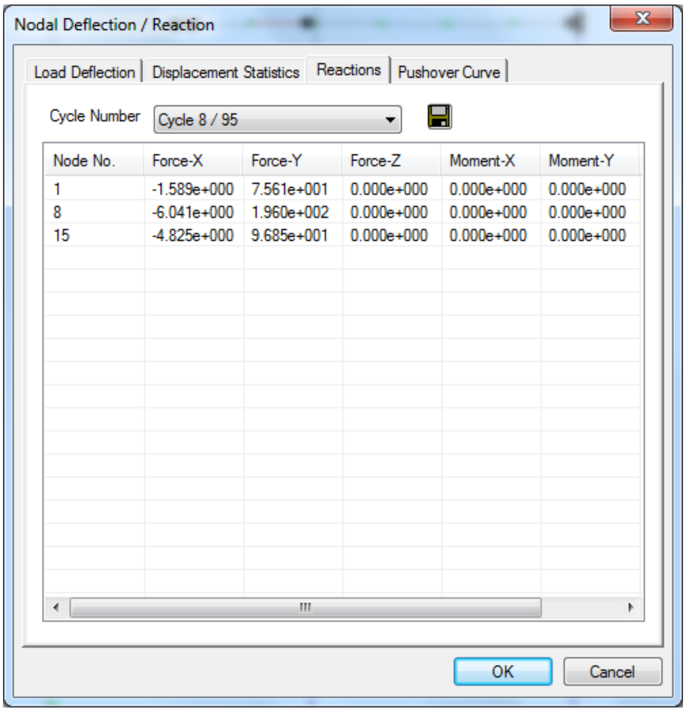
The second tab shows detailed displacements and rotations in specified load cycle and load stage. When cycle number is changed, the results will be changed accordingly.

|Ux|,|Uy| and |Uz| are the displacements in X, Y and Z-axes respectively, Displacement is the total translation between original and new positions, and |Rx|, |Ry| and |Rz| are the rotation about X, Y and Z-axes respectively.

Click the column header (e.g. |Ux|, |Ry|, etc.) to sort that column in ascending or descending order for finding the maximum and minimum values.

To save the results in text file, click the button at top right of this window.

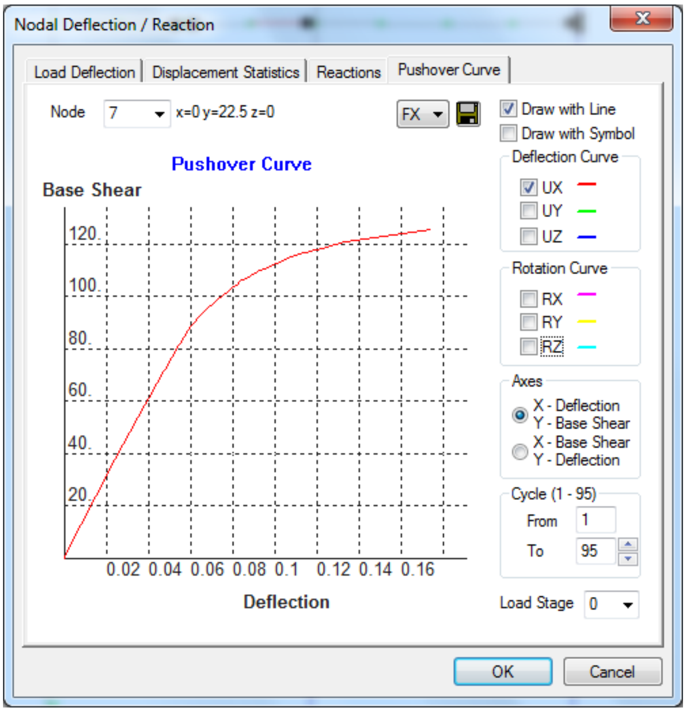
- Reactions



This tab lists the reaction forces and moments of each load cycle.

To save the results in text file, click the button at top right of this window.

- Pushover Curve

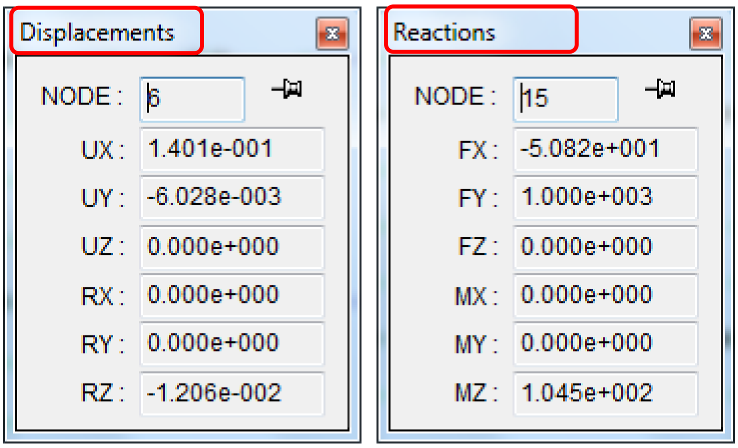


This diagram shows change of base shear against nodal deflection for pushover analysis.

To save the results in text file, click the button at top right of this window.

4.8.7.3 Toggle Displacements/Reactions Dialog

Click the ***Post****>Nodal Results>Toggle Displacements/Reactions Dialog* command to choose whether reactions or displacements to be shown on the popup dialog when double-clicking a node.



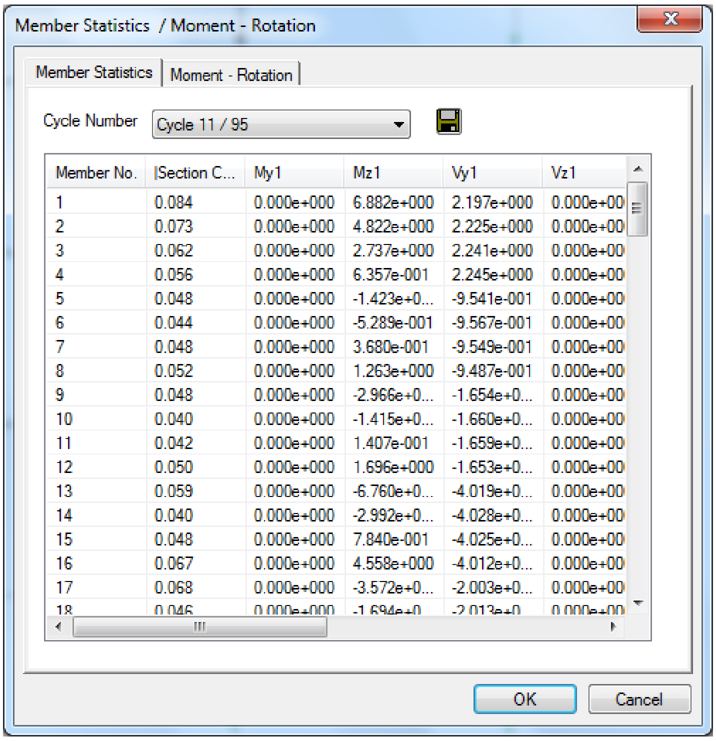
4.8.8 Member Results

4.8.8.1 Member Statistics/Moment-Rotation

Click the ***Post****>Member Results>Member Statistics/Moment-Rotation* command to show the member information such as the section capacity factor, end moment, shear stress, axial force and torsional moment and the Moment-Rotation curve in each load cycle.

Shortcut : Toolbar :  Keys : None

- Member Statistics



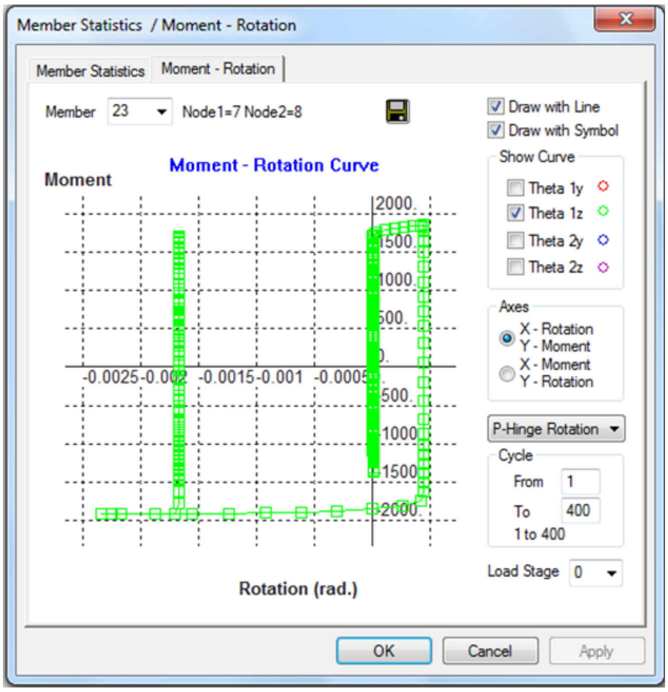
Member No.	Section C...	My1	Mz1	Vy1	Vz1
1	0.084	0.000e+000	6.882e+000	2.197e+000	0.000e+00
2	0.073	0.000e+000	4.822e+000	2.225e+000	0.000e+00
3	0.062	0.000e+000	2.737e+000	2.241e+000	0.000e+00
4	0.056	0.000e+000	6.357e-001	2.245e+000	0.000e+00
5	0.048	0.000e+000	-1.423e+0...	-9.541e-001	0.000e+00
6	0.044	0.000e+000	-5.289e-001	-9.567e-001	0.000e+00
7	0.048	0.000e+000	3.680e-001	-9.549e-001	0.000e+00
8	0.052	0.000e+000	1.263e+000	-9.487e-001	0.000e+00
9	0.048	0.000e+000	-2.966e+0...	-1.654e+0...	0.000e+00
10	0.040	0.000e+000	-1.415e+0...	-1.660e+0...	0.000e+00
11	0.042	0.000e+000	1.407e-001	-1.659e+0...	0.000e+00
12	0.050	0.000e+000	1.696e+000	-1.653e+0...	0.000e+00
13	0.059	0.000e+000	-6.760e+0...	-4.019e+0...	0.000e+00
14	0.040	0.000e+000	-2.992e+0...	-4.028e+0...	0.000e+00
15	0.048	0.000e+000	7.840e-001	-4.025e+0...	0.000e+00
16	0.067	0.000e+000	4.558e+000	-4.012e+0...	0.000e+00
17	0.068	0.000e+000	-3.572e+0...	-2.003e+0...	0.000e+00
18	0.046	0.000e+000	-1.694e+0	-2.013e+0	0.000e+00

When different loading is applied on the structure, the internal forces on the members are changed accordingly. Some useful information about the force on each member: stress, axial force and torsional moment can be found here.

To find the minimum/maximum section capacity factor, axial force or torsional moment among the members in the structure at a certain load factor, select the cycle number and click on column header to sort it in ascending/descending order.

To save the results in text file, click the button at top right of this window.

- Moment-Rotation



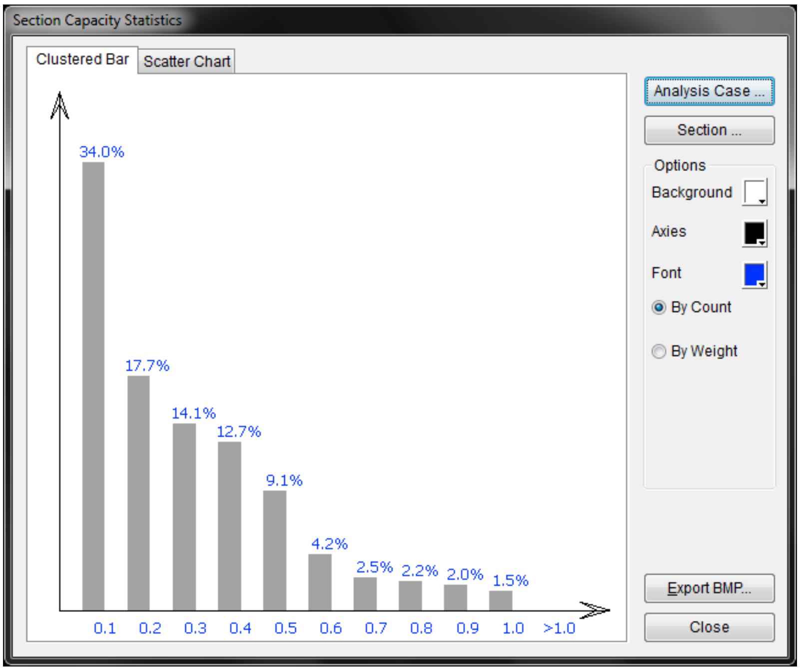
This diagram shows change of moment against rotation for second-order plastic analysis and time history analysis.

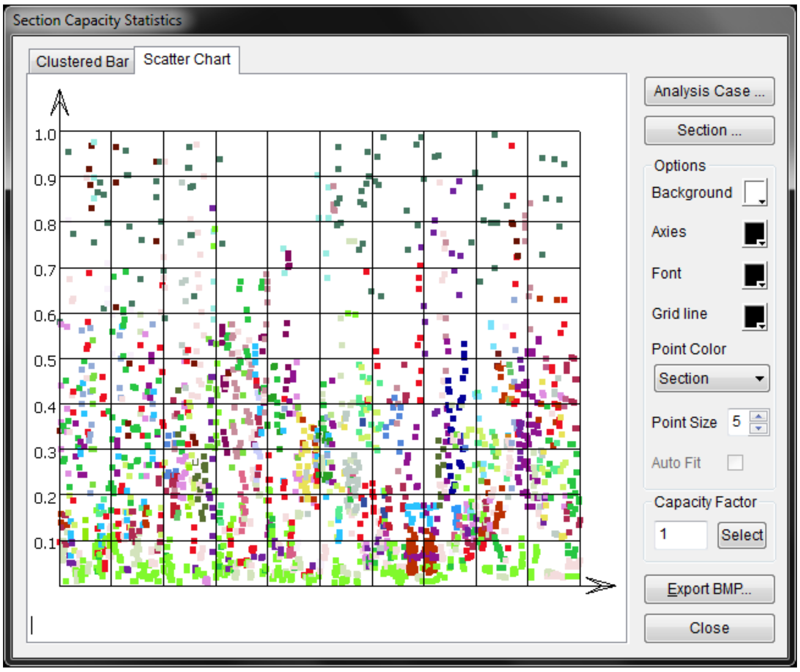
4.8.8.2 Show Section Capacity Factor Statistics

Click the ***Post****>Member Results>Show Section Capacity Factor Statistics* command to show section capacity factor statistics.

Shortcut : Toolbar :  Keys : <u>None</u>

This function is useful to check the utilization of each section group. User can reduce or increase member size according to charts.





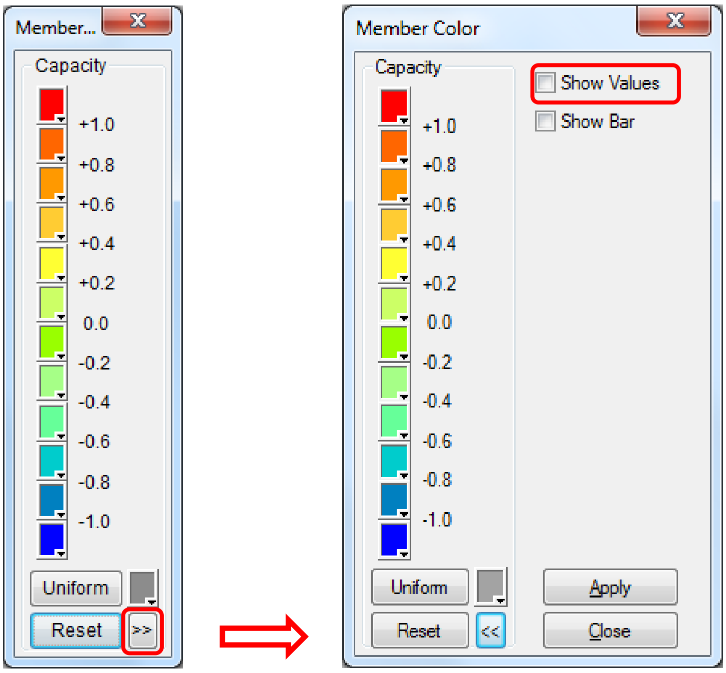
Click the Analysis Case button to select the analysis case to be included.

Click the Section button to select the section to be included.

The color of background, axis, front, grid line and point color can be selected. The chart can be exported to a BMP file.

The color of each member in deformed structure depends on the section capacity factor at each load cycle.

Shortcut : Toolbar : Keys : <u>None</u>



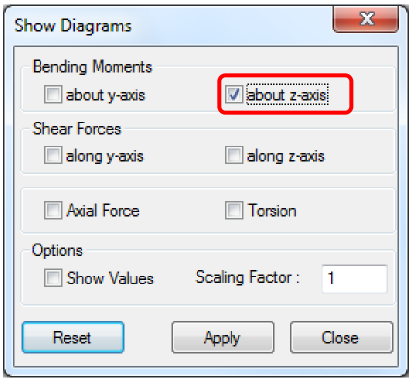
It is divided into 12 different colors by different ranges of section capacity factor. The first one represents the member with stress value larger than or equal to **1.0** (i.e. exceeding the tensile yield stress), whereas the last one represents the stress value is less than **-1.0** (i.e. exceeding the compressive yield stress).

Change these colors by simply clicking the color boxes and select the new one from color palette.

4.8.8.3 Show Bending Moment/Shear Force Diagram

Click the ***Post****>Member Results>Show Bending Moment/Shear Force Diagram* command to show the bending moment and shear force diagram of all the members on the screen in undeformed configuration.

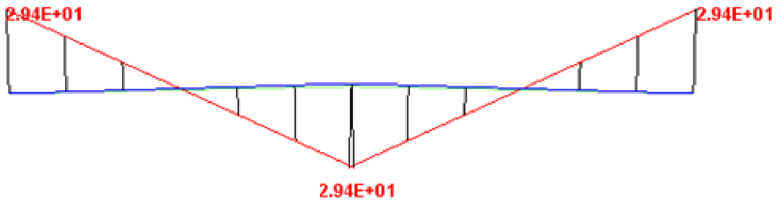
Shortcut : Toolbar : Keys : <u>None</u>



To show bending moment and/or shear force on undeformed structure in main window, select one or more than one items to be shown on the screen.

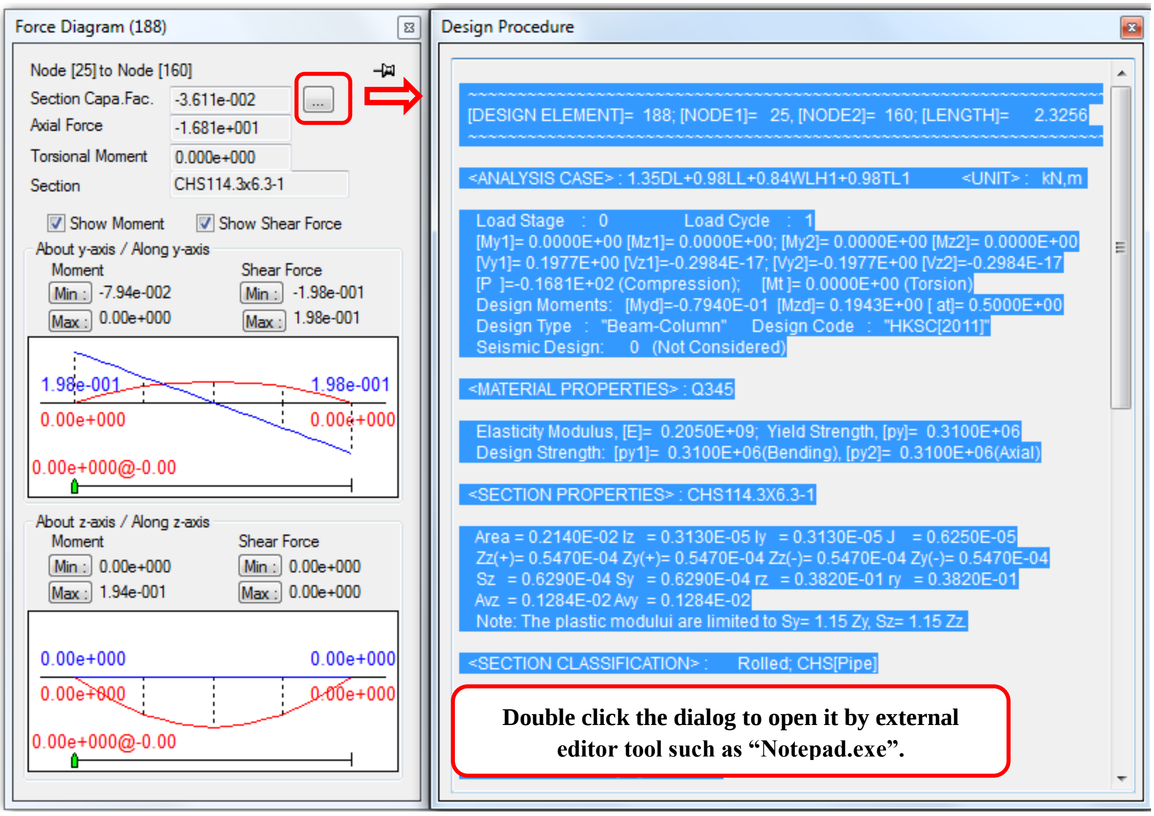
- Bending moment about y-axis
- Bending moment about z-axis
- Shear force along local y-axis (xz plane)
- Shear force along local z-axis (xy plane)
- Axial force
- Value of bending moment, shear force and axial force

Afterwards, tick the members you want to draw curves and press Apply.



Please be reminded that too many curves will reduce the quality of the diagram. It is recommended that only diagrams of selected members would be shown simultaneously on the screen.

The result could also be viewed in a separate window by clicking the specific member, and then a dialog will pop up to show the bending moments and shear forces as below.



The first diagram shows bending moment about y-axis and shear force along y-axis (i.e. xz plane) of the selected member. The second diagram shows bending moment and shear force about z-axis and along z-axis (i.e. xy plane) respectively.

Note: all these diagrams are drawn in member local axis.

4.8.8.4 Show Plastic Hinge

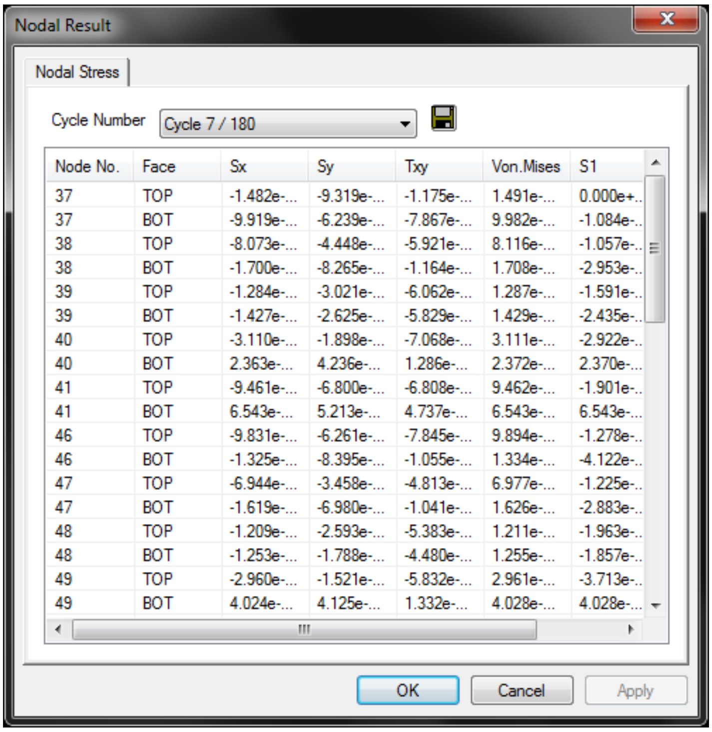
Click the ***Post****>Member Results>Show Plastic Hinge* command to show the plastic hinge if it is a plastic analysis allowing for plastic hinges formation.

The plastic hinge may occur at one or both ends of the member with blue or red circular mark. The blue mark means partially yield plastic hinge while the red one means full yield plastic hinge.

4.8.9 Shell Results

4.8.9.1 List Nodal Stress

Click the ***Post****>Shell Results>List Nodal Stress* command to list the nodal stresses.



Node No.	Face	Sx	Sy	Txy	Von.Mises	S1
37	TOP	-1.482e-...	-9.319e-...	-1.175e-...	1.491e-...	0.000e+..
37	BOT	-9.919e-...	-6.239e-...	-7.867e-...	9.982e-...	-1.084e-..
38	TOP	-8.073e-...	-4.448e-...	-5.921e-...	8.116e-...	-1.057e-..
38	BOT	-1.700e-...	-8.265e-...	-1.164e-...	1.708e-...	-2.953e-..
39	TOP	-1.284e-...	-3.021e-...	-6.062e-...	1.287e-...	-1.591e-..
39	BOT	-1.427e-...	-2.625e-...	-5.829e-...	1.429e-...	-2.435e-..
40	TOP	-3.110e-...	-1.898e-...	-7.068e-...	3.111e-...	-2.922e-..
40	BOT	2.363e-...	4.236e-...	1.286e-...	2.372e-...	2.370e-..
41	TOP	-9.461e-...	-6.800e-...	-6.808e-...	9.462e-...	-1.901e-..
41	BOT	6.543e-...	5.213e-...	4.737e-...	6.543e-...	6.543e-..
46	TOP	-9.831e-...	-6.261e-...	-7.845e-...	9.894e-...	-1.278e-..
46	BOT	-1.325e-...	-8.395e-...	-1.055e-...	1.334e-...	-4.122e-..
47	TOP	-6.944e-...	-3.458e-...	-4.813e-...	6.977e-...	-1.225e-..
47	BOT	-1.619e-...	-6.980e-...	-1.041e-...	1.626e-...	-2.883e-..
48	TOP	-1.209e-...	-2.593e-...	-5.383e-...	1.211e-...	-1.963e-..
48	BOT	-1.253e-...	-1.788e-...	-4.480e-...	1.255e-...	-1.857e-..
49	TOP	-2.960e-...	-1.521e-...	-5.832e-...	2.961e-...	-3.713e-..
49	BOT	4.024e-...	4.125e-...	1.332e-...	4.028e-...	4.028e-..

You may obtain 6 types of information of each node in different load cycles. When cycle number is changed, the load factor will also be varied.

Sx	The nodal stress in x-direction (Global coordinate system)
Sy	The nodal stress in y-direction (Global coordinate system)
Txy	Shear Stress (Global coordinate system)
von.Mises	von Mises Stress
S_1	The First Principle Stress
S_2	The Second Principle Stress

Note: Principle stresses and von Mises stress are calculated as below.

$$S_1 = \frac{S_X + S_Y}{2} + \sqrt{\left(\frac{S_X - S_Y}{2}\right)^2 + T_{XY}^2}$$

$$S_2 = \frac{S_X + S_Y}{2} - \sqrt{\left(\frac{S_X - S_Y}{2}\right)^2 + T_{XY}^2}$$

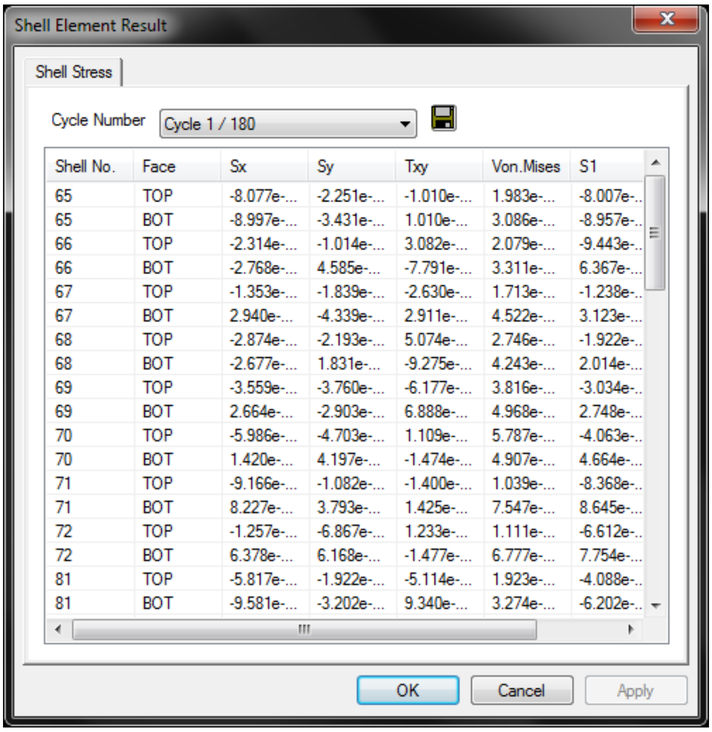
$$S_{VMS} = \sqrt{S_X^2 + S_Y^2 - S_X S_Y + 3T_{XY}^2}$$

Click the column header (e.g. Sx, Von Mises, S_1, etc.) to sort that column in ascending or descending order in order to obtain the maximum and minimum values.

To save the results in text file, click the button at top right of this window.

4.8.9.2 List Shell Stress

Click the ***Post****>Shell Results>List Shell Stress* command to list the shell element stresses.



Shell No.	Face	Sx	Sy	Txy	Von.Mises	S1
65	TOP	-8.077e-...	-2.251e-...	-1.010e-...	1.983e-...	-8.007e-...
65	BOT	-8.997e-...	-3.431e-...	1.010e-...	3.086e-...	-8.957e-...
66	TOP	-2.314e-...	-1.014e-...	3.082e-...	2.079e-...	-9.443e-...
66	BOT	-2.768e-...	4.585e-...	-7.791e-...	3.311e-...	6.367e-...
67	TOP	-1.353e-...	-1.839e-...	-2.630e-...	1.713e-...	-1.238e-...
67	BOT	2.940e-...	-4.339e-...	2.911e-...	4.522e-...	3.123e-...
68	TOP	-2.874e-...	-2.193e-...	5.074e-...	2.746e-...	-1.922e-...
68	BOT	-2.677e-...	1.831e-...	-9.275e-...	4.243e-...	2.014e-...
69	TOP	-3.559e-...	-3.760e-...	-6.177e-...	3.816e-...	-3.034e-...
69	BOT	2.664e-...	-2.903e-...	6.888e-...	4.968e-...	2.748e-...
70	TOP	-5.986e-...	-4.703e-...	1.109e-...	5.787e-...	-4.063e-...
70	BOT	1.420e-...	4.197e-...	-1.474e-...	4.907e-...	4.664e-...
71	TOP	-9.166e-...	-1.082e-...	-1.400e-...	1.039e-...	-8.368e-...
71	BOT	8.227e-...	3.793e-...	1.425e-...	7.547e-...	8.645e-...
72	TOP	-1.257e-...	-6.867e-...	1.233e-...	1.111e-...	-6.612e-...
72	BOT	6.378e-...	6.168e-...	-1.477e-...	6.777e-...	7.754e-...
81	TOP	-5.817e-...	-1.922e-...	-5.114e-...	1.923e-...	-4.088e-...
81	BOT	-9.581e-...	-3.202e-...	9.340e-...	3.274e-...	-6.202e-...

You may obtain 6 types of information of each shell element in different load cycles. When cycle number is changed, the load factor will be changed accordingly.

Sx	The element stress in x-direction (Local coordinate system)
Sy	The element stress in y-direction (Local coordinate system)
Txy	Shear Stress (Local coordinate system)
von.Mises	von Mises Stress
S_1	The First Principle Stress
S_2	The Second Principle Stress

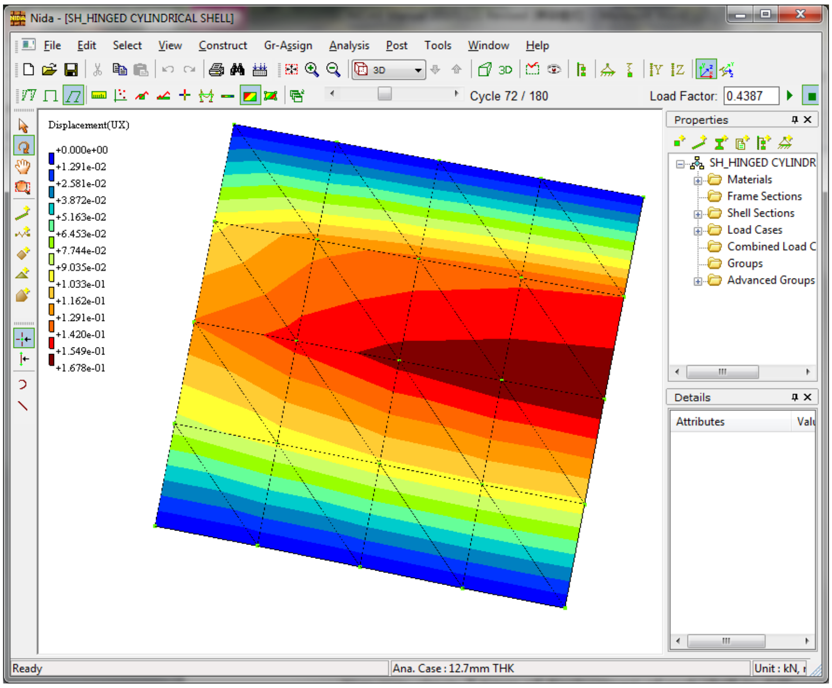
Click the column header (e.g. Sx, Von Mises, S1, etc.) to sort that column in ascending or descending order in order to obtain the maximum and minimum values.

To save the results in text file, click the button at top right of this window.

4.8.9.3 Show Displacements

Click the ***Post****>Shell Results>Show Displacements* command to show the displacement contour.

Shortcut : Toolbar : Keys : None



You may obtain 7 types of displacement of each shell in different load cycles. When cycle number is changed, the load factor will be changed accordingly.

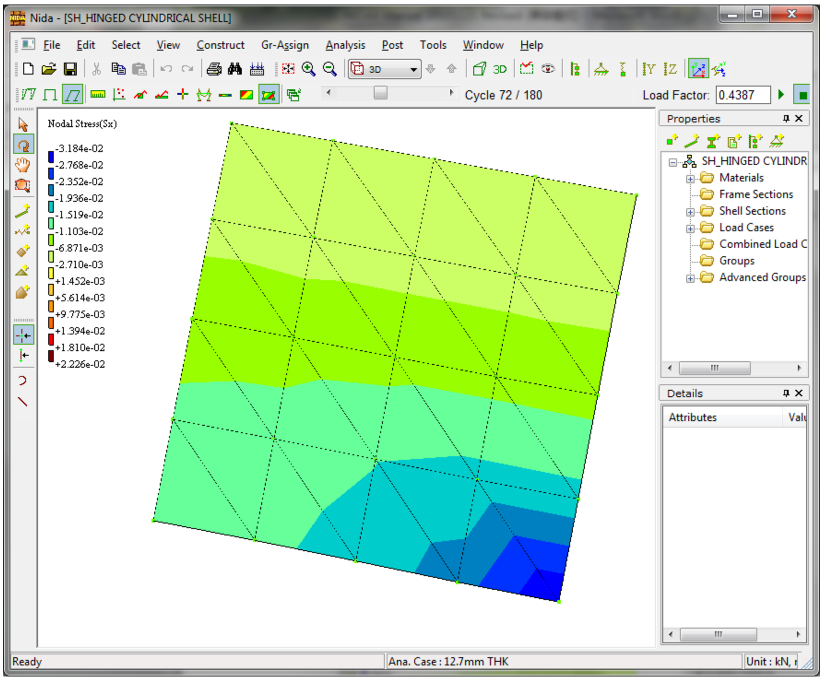
Ux	Displacement in X-axis
Uy	Displacement in Y-axis
Uz	Displacement in Z-axis
Disp	Total displacement, $Disp = \sqrt{U_X^2 + U_Y^2 + U_Z^2}$
Rx	Rotation about X-axis
Ry	Rotation about Y-axis
Rz	Rotation about Z-axis

The different colors in the shell element indicate the different levels of deformation.

4.8.9.4 Show Nodal Stress (Surface)

Click the ***Post****>Shell Results>Show Nodal Stress (Surface)* command to show the nodal surface stress contour.

Shortcut : Toolbar : Keys : None



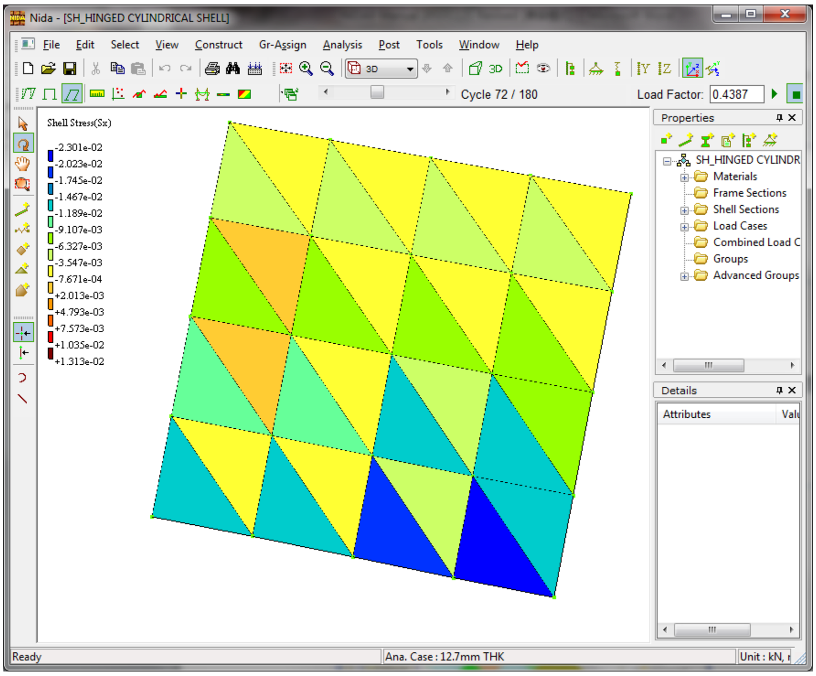
You may obtain 6 types of stress of each node in different load cycles. When cycle number is changed, the load factor will be changed accordingly.

Sx	The nodal stress in x-direction (Global coordinate system)
Sy	The nodal stress in y-direction (Global coordinate system)
Txy	Shear Stress (Global coordinate system)
Von.Mises	von Mises Stress
S1	The First Principle Stress
S2	The Second Principle Stress

The different colors in the shell element mean the different level of stress.

4.8.9.5 Show Shell Element Stresses (Surface)

Click the ***Post****>Shell Results>Show Shell Elem Stress (Surface)* command to show the shell element surface stress contour.



You may obtain 6 types of stress of each node in different load cycles. When cycle number is changed, the load factor will be changed accordingly.

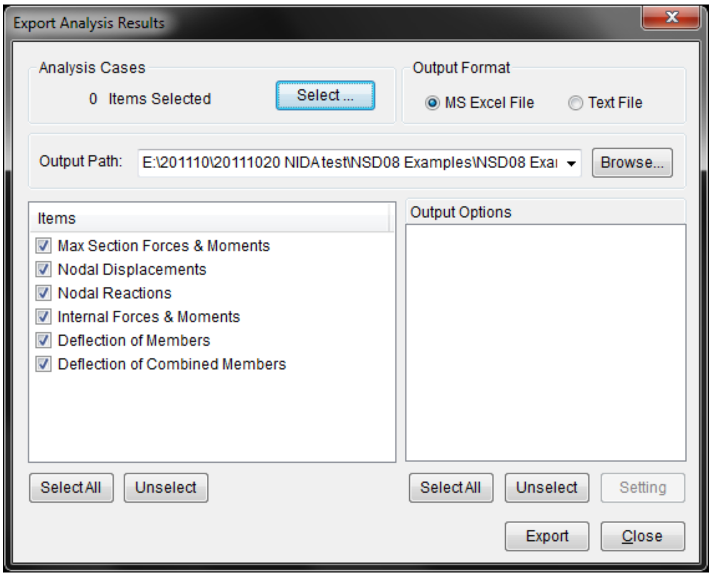
Sx	The element stress in x-direction (Local coordinate system)
Sy	The element stress in y-direction (Local coordinate system)
Txy	Shear Stress (Local coordinate system)
Von.Mises	von Mises Stress
S1	The First Principle Stress
S2	The Second Principle Stress

The different colors in the shell element indicate the different levels of stress.

4.8.10 Export Summary of Analysis Results

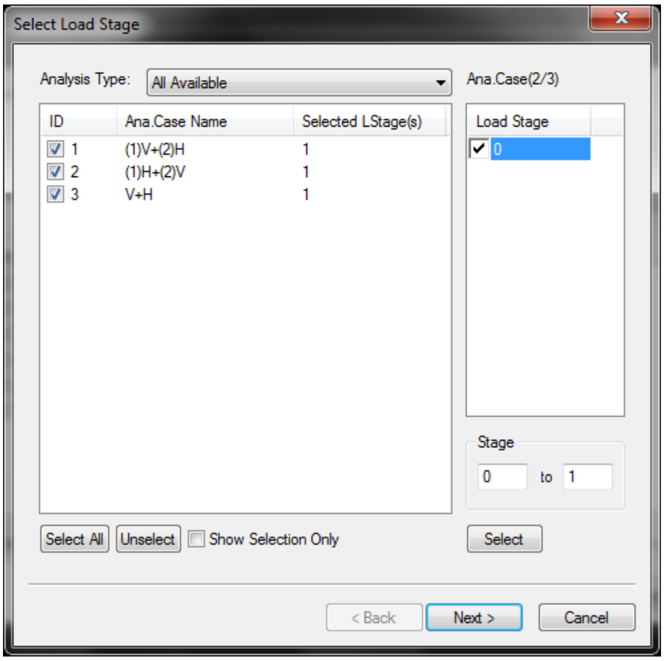
Click the ***Post****>Export Summary of Analysis Results* command to export the analysis results to MS Excel file or Text file.

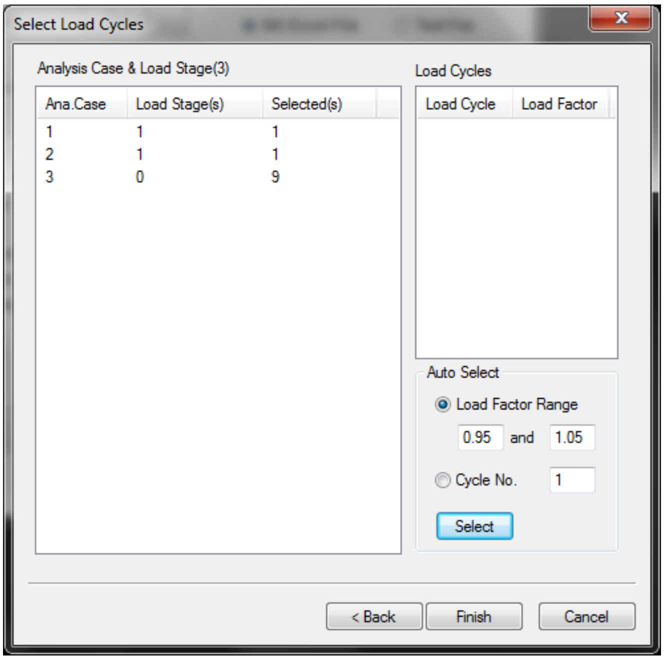
Shortcut : Toolbar : Keys : None



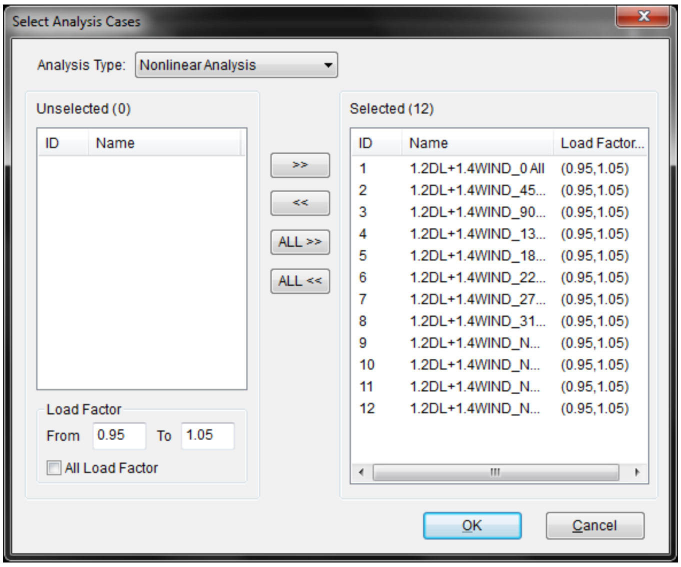
Step 1: Click Select … button to select analysis case, load cycle and/or load stages.

<u>Analysis case(s) with load stages:</u>



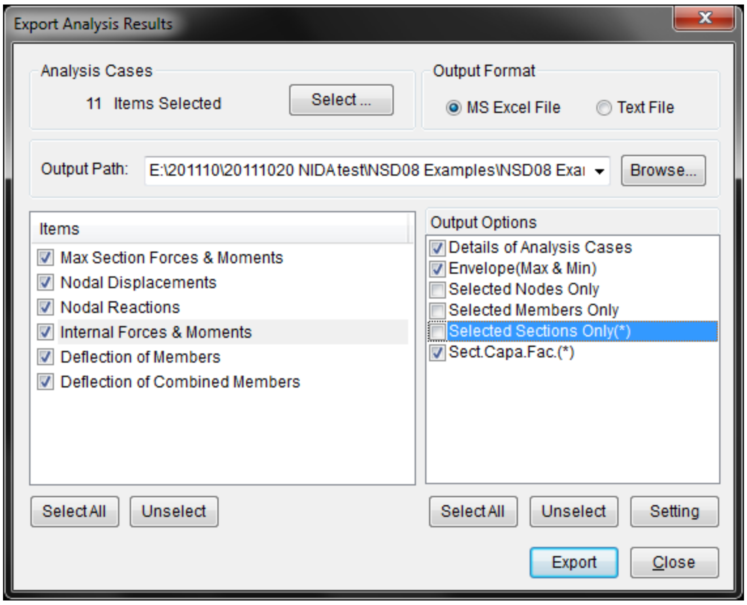


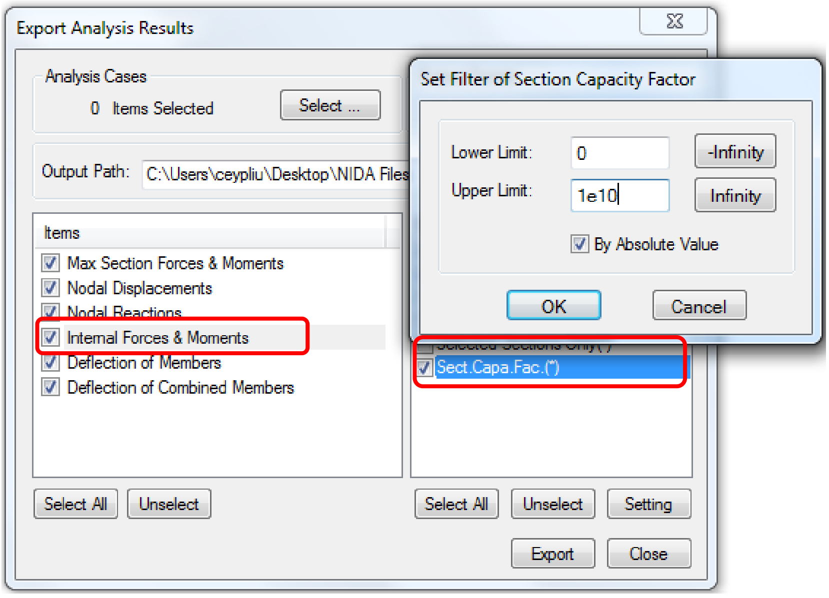
<u>Analysis case(s) without load stages:</u>



Step 2: Click the Browse button to select the output path to save results.

Step 3: Choose the item(s) to be included in output. Items(result type) can be specified (Max Section Forces & Moments, Nodal Displacements, Nodal Reactions, Internal Forces & Moments, Deflection of Members and Deflection of Combined Members); and options are provided for each item. Star (*) in option means further setting is provided.





Click Export button and an MS Excel file or Text File will be shown as below.

- MS Excel File

Ancs No.	Ancs Name
1	1.2DL+1.4WIND_0 All

Member No.	Node No.	Ancs No.	Stage	Cycle	Load Factor	P	My	Mz	T	Vy	Vz	SCFactor
						kN	kN*m	kN*m	kN*m	kN	kN	1
1077	381	1	0	2	1.00	-8.364E+02	-4.112E+00	-6.924E+00	-6.814E-04	-7.451E+00	5.229E+00	-8.392E-01
1077	397	1	0	2	1.00	-8.364E+02	-4.117E+00	-4.812E+00	-6.814E-04	7.417E+00	-5.195E+00	-8.392E-01
1077	381	Max				-8.364E+02	-4.112E+00	-6.924E+00	-6.814E-04	-7.451E+00	5.229E+00	-8.392E-01
1077	381	Min				-8.364E+02	-4.112E+00	-6.924E+00	-6.814E-04	-7.451E+00	5.229E+00	-8.392E-01
1077	397	Max				-8.364E+02	-4.117E+00	-4.812E+00	-6.814E-04	7.417E+00	-5.195E+00	-8.392E-01
1077	397	Min				-8.364E+02	-4.117E+00	-4.812E+00	-6.814E-04	7.417E+00	-5.195E+00	-8.392E-01
1078	383	1	0	2	1.00	-6.850E+02	-6.159E+00	-2.843E+00	6.540E-04	-3.910E+00	6.521E+00	-6.863E-01
1078	398	1	0	2	1.00	-6.850E+02	-4.109E+00	-3.303E+00	6.540E-04	3.876E+00	-6.487E+00	-6.863E-01
1078	383	Max				-6.850E+02	-6.159E+00	-2.843E+00	6.540E-04	-3.910E+00	6.521E+00	-6.863E-01
1078	383	Min				-6.850E+02	-6.159E+00	-2.843E+00	6.540E-04	-3.910E+00	6.521E+00	-6.863E-01
1078	398	Max				-6.850E+02	-4.109E+00	-3.303E+00	6.540E-04	3.876E+00	-6.487E+00	-6.863E-01
1078	398	Min				-6.850E+02	-4.109E+00	-3.303E+00	6.540E-04	3.876E+00	-6.487E+00	-6.863E-01
1125	397	1	0	2	1.00	-8.338E+02	4.117E+00	4.811E+00	-1.381E-05	4.810E+00	-5.318E+00	-7.826E-01
1125	413	1	0	2	1.00	-8.338E+02	4.306E+00	2.810E+00	-1.381E-05	-4.844E+00	5.352E+00	-7.826E-01
1125	397	Max				-8.338E+02	4.117E+00	4.811E+00	-1.381E-05	4.810E+00	-5.318E+00	-7.826E-01
1125	397	Min				-8.338E+02	4.117E+00	4.811E+00	-1.381E-05	4.810E+00	-5.318E+00	-7.826E-01
1125	413	Max				-8.338E+02	4.306E+00	2.810E+00	-1.381E-05	-4.844E+00	5.352E+00	-7.826E-01
1125	413	Min				-8.338E+02	4.306E+00	2.810E+00	-1.381E-05	-4.844E+00	5.352E+00	-7.826E-01
1126	398	1	0	2	1.00	-6.828E+02	4.108E+00	3.303E+00	-4.384E-04	4.815E+00	-3.583E+00	-6.416E-01
1126	414	1	0	2	1.00	-6.828E+02	1.575E+00	4.325E+00	-4.384E-04	-4.849E+00	3.617E+00	-6.416E-01
1126	398	Max				-6.828E+02	4.108E+00	3.303E+00	-4.384E-04	4.815E+00	-3.583E+00	-6.416E-01

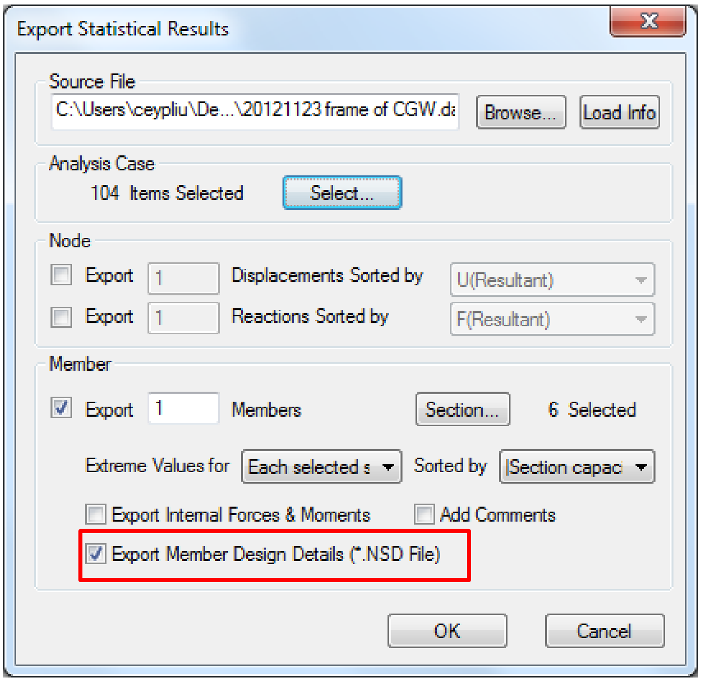
- Text File

```
Unit : kN, m

---------------------------------------------------- Member = 1077 ----------------------------------------------------
NodeI = 381
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -8.364e+02   -4.112e+00   -6.924e+00   -6.814e-04   -7.451e+00   5.229e+00    -8.392e-01
Max                           -8.364e+02   -4.112e+00   -6.924e+00   -6.814e-04   -7.451e+00   5.229e+00    -8.392e-01
Min                           -8.364e+02   -4.112e+00   -6.924e+00   -6.814e-04   -7.451e+00   5.229e+00    -8.392e-01
NodeJ = 397
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -8.364e+02   -4.117e+00   -4.812e+00   -6.814e-04   7.417e+00    -5.195e+00   -8.392e-01
Max                           -8.364e+02   -4.117e+00   -4.812e+00   -6.814e-04   7.417e+00    -5.195e+00   -8.392e-01
Min                           -8.364e+02   -4.117e+00   -4.812e+00   -6.814e-04   7.417e+00    -5.195e+00   -8.392e-01
---------------------------------------------------- Member = 1078 ----------------------------------------------------
NodeI = 383
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -6.850e+02   -6.159e+00   -2.843e+00   6.540e-04    -3.910e+00   6.521e+00    -6.863e-01
Max                           -6.850e+02   -6.159e+00   -2.843e+00   6.540e-04    -3.910e+00   6.521e+00    -6.863e-01
Min                           -6.850e+02   -6.159e+00   -2.843e+00   6.540e-04    -3.910e+00   6.521e+00    -6.863e-01
NodeJ = 398
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -6.850e+02   -4.109e+00   -3.303e+00   6.540e-04    3.876e+00    -6.487e+00   -6.863e-01
Max                           -6.850e+02   -4.109e+00   -3.303e+00   6.540e-04    3.876e+00    -6.487e+00   -6.863e-01
Min                           -6.850e+02   -4.109e+00   -3.303e+00   6.540e-04    3.876e+00    -6.487e+00   -6.863e-01
---------------------------------------------------- Member = 1125 ----------------------------------------------------
NodeI = 397
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -8.338e+02   4.117e+00    4.811e+00    -1.381e-05   4.810e+00    -5.318e+00   -7.826e-01
Max                           -8.338e+02   4.117e+00    4.811e+00    -1.381e-05   4.810e+00    -5.318e+00   -7.826e-01
Min                           -8.338e+02   4.117e+00    4.811e+00    -1.381e-05   4.810e+00    -5.318e+00   -7.826e-01
NodeJ = 413
AncsNo.    LoadStage   Cycle       P            My           Mz           T            Vy           Vz           Capacity
  1            0       2      -8.338e+02   4.306e+00    2.810e+00    -1.381e-05   -4.844e+00   5.352e+00    -7.826e-01
Max                           -8.338e+02   4.306e+00    2.810e+00    -1.381e-05   -4.844e+00   5.352e+00    -7.826e-01
Min                           -8.338e+02   4.306e+00    2.810e+00    -1.381e-05   -4.844e+00   5.352e+00    -7.826e-01
```

4.8.11 Export Statistics of Analysis Results

Click the ***Post****> Export Statistics of Analysis Results* command to export the statistical results in an MS Excel file.

Shortcut : Toolbar : Keys : None



Analysis Cases Selection: refer to Step (1) of **4.18.10 Export Summary of Analysis Results**.

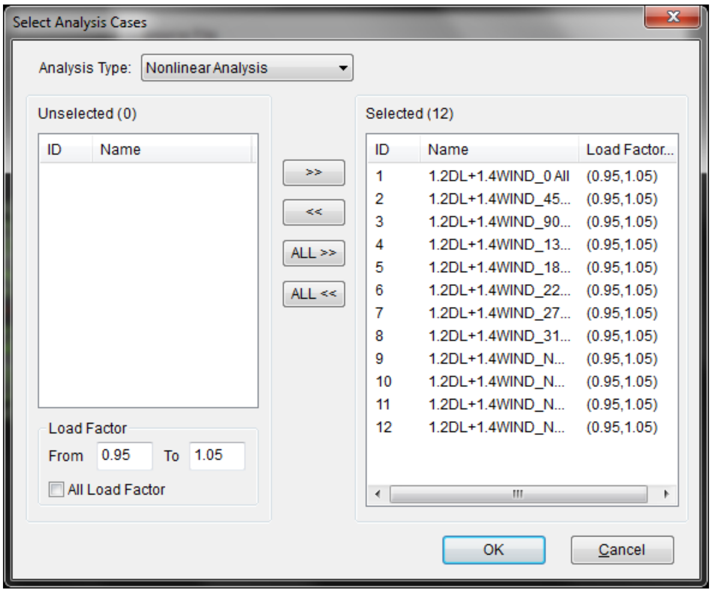
Node result output: Most top (max) **N** result items sorted by **Option** will be exported if the checkbox enables. The item Count 'N' can be specified and Option(Resultant/Ux/Uy/Uz/Rx/Ry/Rz) is provided for displacement while Option(Resultant/Fx/Fy/Fz/Mx/My/Mz) is provided for reaction.

Member result output:

"For" Option: items are categorized by **Each selected section/Each member** or not categorized, i.e. **All**.

"By" Option: items are sorted by **|Section capacity factor|** (Absolute value) /Axial force/Bending moment/Shear force/Torsional moment;

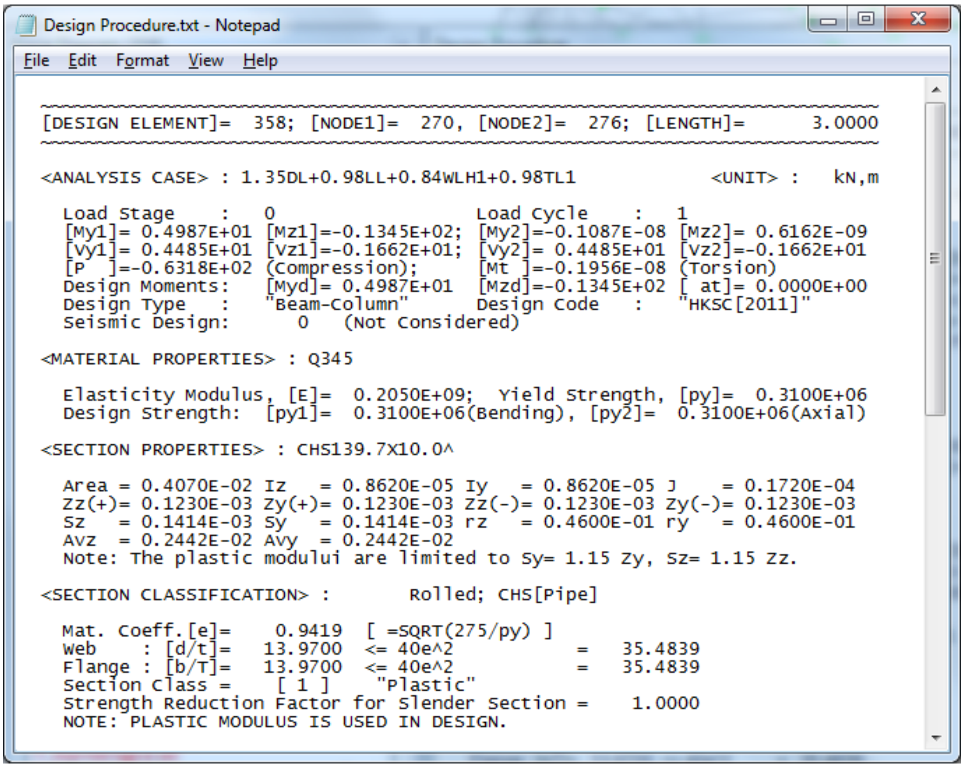
Section filter:



An exported MS Excel file will be shown as below.

ANCS No.	ANCS Name	Load Stage	Cycle	Cycle Fac.	Sect	Sect No.	Member No.	Sect Capa. Fac.
1	1.2DL+1.4WIND_0 All	0	2	1	copy of A150X17	1	1077	-0.839
4	1.2DL+1.4WIND_135 All	0	2	1	copy of A200X25	2	1003	-0.903
8	1.2DL+1.4WIND_315 All	0	2	1	A100X8	3	930	0.139
4	1.2DL+1.4WIND_135 All	0	2	1	copy of A200X30	4	639	-0.966
4	1.2DL+1.4WIND_135 All	0	2	1	A120X8	5	1430	0.241
1	1.2DL+1.4WIND_0 All	0	2	1	A130X10	6	494	-0.730
8	1.2DL+1.4WIND_315 All	0	2	1	A130X12	7	1332	0.415
1	1.2DL+1.4WIND_0 All	0	2	1	A130X9	8	1186	-0.543
1	1.2DL+1.4WIND_0 All	0	2	1	A150X11	9	948	0.485
4	1.2DL+1.4WIND_135 All	0	2	1	A150X12	10	1147	-0.842
8	1.2DL+1.4WIND_315 All	0	2	1	A150X15	11	1273	-0.533
6	1.2DL+1.4WIND_225 All	0	2	1	A200X15	12	1008	-0.809
1	1.2DL+1.4WIND_0 All	0	2	1	A200X20	13	637	-0.965
1	1.2DL+1.4WIND_0 All	0	2	1	A200X25	14	1888	-0.992
8	1.2DL+1.4WIND_315 All	0	2	1	A45X5	15	319	-0.598

"Export Member Design Details (*.NSD file)" option can output the details of design procedure of critical members for calculation report.

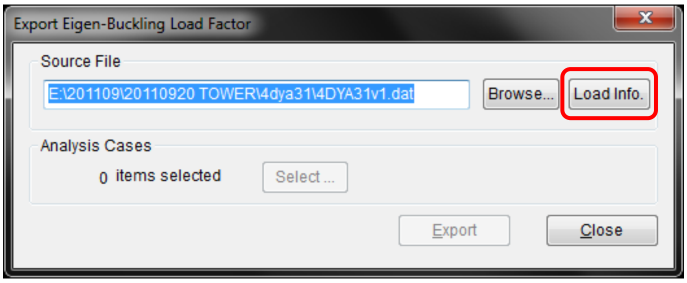


```
Design Procedure.txt - Notepad
File  Edit  Format  View  Help
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
[DESIGN ELEMENT]=  358; [NODE1]=  270, [NODE2]=  276; [LENGTH]=      3.0000
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
<ANALYSIS CASE> : 1.35DL+0.98LL+0.84WLH1+0.98TL1            <UNIT> :   kN,m

  Load Stage     :   0                 Load Cycle     :   1
  [My1]= 0.4987E+01 [Mz1]=-0.1345E+02; [My2]=-0.1087E-08 [Mz2]= 0.6162E-09
  [Vy1]= 0.4485E+01 [Vz1]=-0.1662E+01; [Vy2]= 0.4485E+01 [Vz2]=-0.1662E+01
  [P  ]=-0.6318E+02 (Compression);     [Mt ]=-0.1956E-08 (Torsion)
  Design Moments:   [Myd]= 0.4987E+01  [Mzd]=-0.1345E+02 [ at]= 0.0000E+00
  Design Type    :  "Beam-Column"      Design Code    :  "HKSC[2011]"
  Seismic Design:      0    (Not Considered)

<MATERIAL PROPERTIES> : Q345

  Elasticity Modulus, [E]=  0.2050E+09;  Yield Strength, [py]=  0.3100E+06
  Design Strength:  [py1]=  0.3100E+06(Bending), [py2]=  0.3100E+06(Axial)

<SECTION PROPERTIES> : CHS139.7X10.0^

  Area = 0.4070E-02 Iz   = 0.8620E-05 Iy   = 0.8620E-05 J    = 0.1720E-04
  Zz(+)= 0.1230E-03 Zy(+)= 0.1230E-03 Zz(-)= 0.1230E-03 Zy(-)= 0.1230E-03
  Sz   = 0.1414E-03 Sy   = 0.1414E-03 rz   = 0.4600E-01 ry   = 0.4600E-01
  Avz  = 0.2442E-02 Avy  = 0.2442E-02
  Note: The plastic modului are limited to Sy= 1.15 Zy, Sz= 1.15 Zz.

<SECTION CLASSIFICATION> :       Rolled; CHS[Pipe]

  Mat. Coeff.[e]=    0.9419  [ =SQRT(275/py) ]
  Web     : [d/t]=   13.9700  <= 40e^2           =    35.4839
  Flange  : [b/T]=   13.9700  <= 40e^2           =    35.4839
  Section Class =     [ 1 ]    "Plastic"
  Strength Reduction Factor for Slender Section =     1.0000
  NOTE: PLASTIC MODULUS IS USED IN DESIGN.
```

4.8.12 Export Eigen-buckling Load Factor(s)

Click the ***Post****>Export Eigen-buckling Load Factor(s)* command to export the eigen-buckling load factor to an MS Excel file.

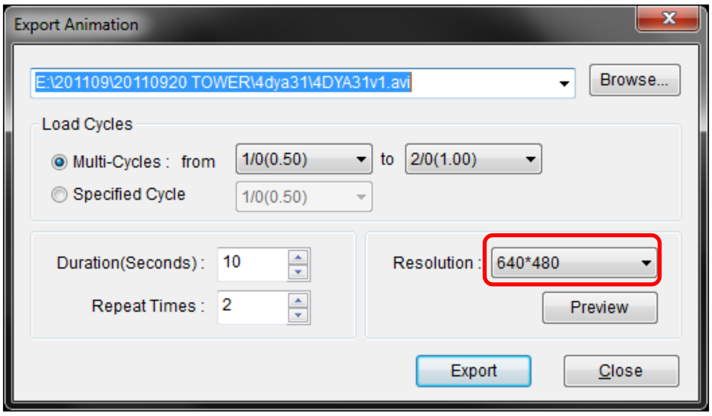
Shortcut : Toolbar : Keys : None



Analysis Cases Selection: refer to Step (1) of **4.18.10 Export Summary of Analysis Results**.

4.8.13 Export Animation AVI

Click the ***Post****>Export Animation AVI* command to export an animation file with AVI format.



Multi-Cycles option provides a video which plays the deformation of the structure under loading from one load cycle to another.

Specified Cycle option provides a video which plays vibration between the undeformed and deformed shape of the specified load cycle.

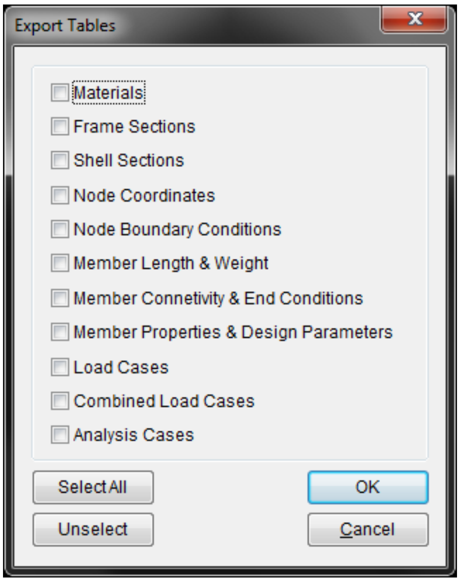
4.9 Tools

4.9.1 Show Data File

Click the ***Tools****>Show Data File* command to open the data file in text format, which contains all information of the model. User can modify the model by editing the file directly and reload it to confirm the change.

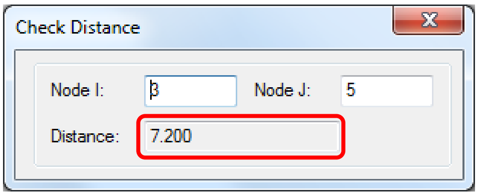
4.9.2 Export Tables

Click the ***Tools****>Export Tables* command to export the definition of the model such as node coordinates, beam and shell connectivity, materials, sections, load cases and load combinations.



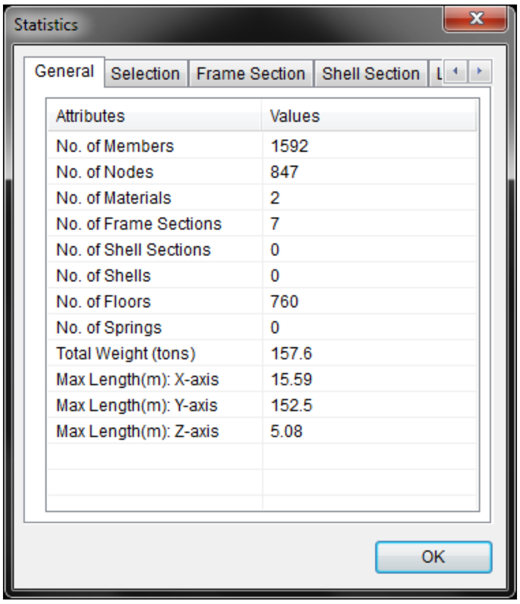
4.9.3 Check Nodal Distance

Click the ***Tools****>Check Nodal Distance* command to calculate the distance between two selected nodes.



4.9.4 Statistics

Click the ***Tools****>Statistics* command to show the information of the model.



The General tab lists the number of member, nodes, materials, sections, shells, floors, the total weight of the model, and the maximum length in X, Y and Z axes.

The Selection tab shows the number of member, node, shell, floor and the total weight of the selected objects.

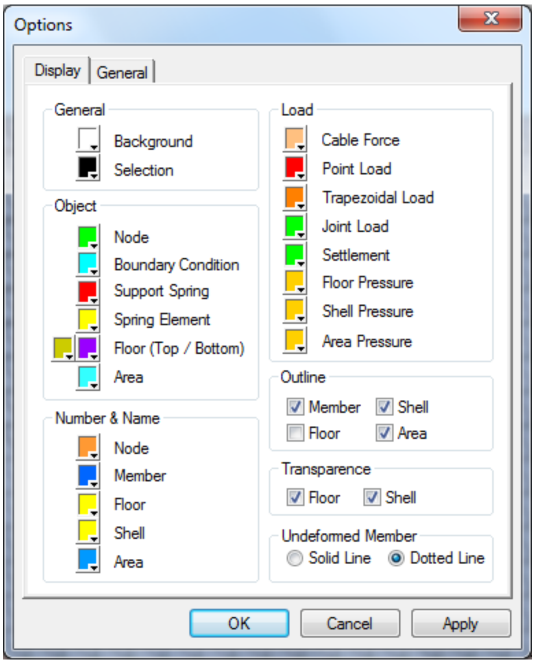
The Frame Section tab shows the number of the members and the total weight of the members in the frame sections.

The Shell Section tab shows the number of the shells and the total weight of the shells in the shell sections.

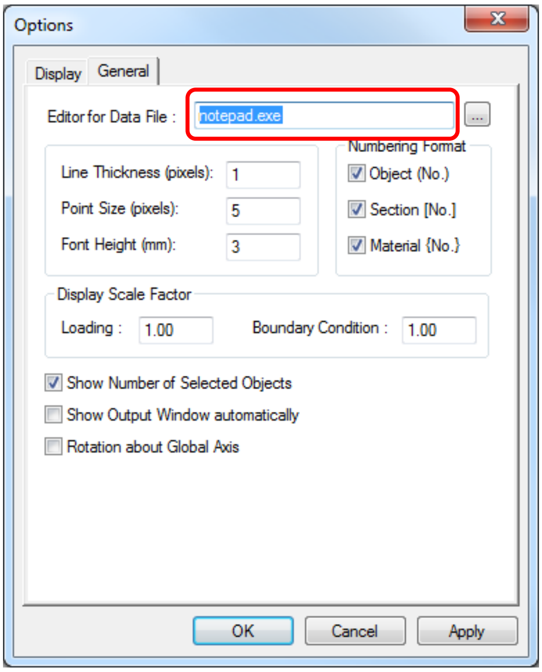
The Load Case tab shows the name, load factor and number of loadings in the corresponding load case.

4.9.5 Options

Click the ***Tools****>Options* command to set the color of each object as well as the display options.



Select an object on the list . The button beside the object will show the current color of that object. Click it to change the color scheme.



Change the thickness of line, choose the tool for opening the data file and so on.

4.10 Window

4.10.1 New Window

Click the ***Window****>New Window* command to open a new window.

4.10.2 Tile Vertically

Click the ***Window****>Tile Vertically* command to show the windows vertically.

4.10.3 Tile Horizontally

Click the ***Window****>Tile Horizontally* command to show the windows horizontally.

4.11 Help

4.11.1 Tutorials

Click the ***Help***>*Tutorials* command to view the step-by-step tutorials.

4.11.2 User Manual

Click the ***Help***>*User Manual* command to view the manual document.

4.11.3 About Nida

Click the ***Help***>*About Nida* command to show the name and version of the program, and the license information.

5. ANALYSIS THEORY

5.1 General Concept for Ultimate Behaviour of Structures

The strut under compression in Figure 5.1 represents the typical behaviour of a member under compression.

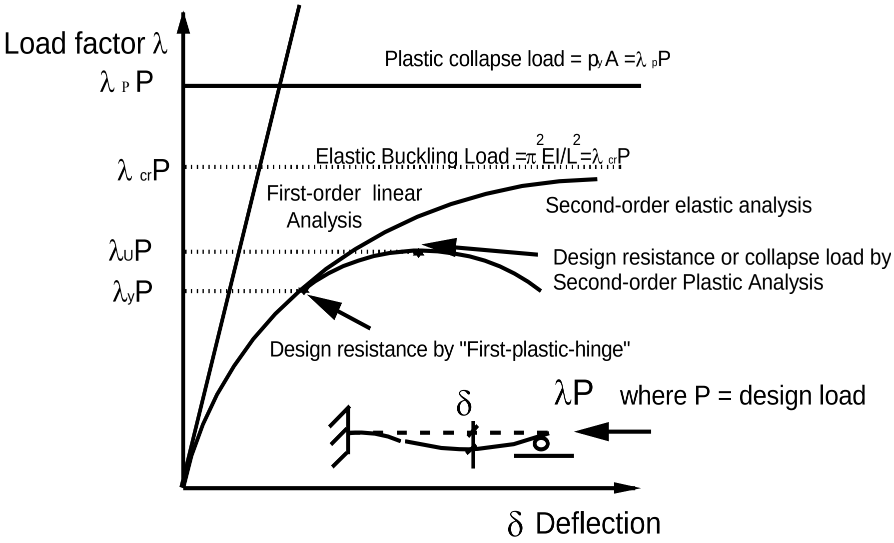


Figure 5.1 Typical Behaviour of An Imperfect Strut

Note: (1) λ is a load factor multiplied to the factored design load P.

(2) λ should not be less than 1 to ensure resistance is not less than design load.

5.1.1 Elastic Critical Load Factor λcr

Elastic Critical Load Factor (λ_{cr}) is an indicator for buckling sensitivity of a structure. The larger the value is, the less sensitive to sway buckling effect the structure is. It can also be used to compute the amplification moment (M) from the moment obtained by a linear analysis ($\overline{M}$) as,

$$M = \frac{\lambda_{cr}}{\lambda_{cr} - 1}\overline{M} \qquad (5.1.1)$$

By definition, λ_{cr} is a factor multiplied to the design load to cause the structure to buckle elastically and drastically, such that the structure does not exhibit pre-buckling deflection. This condition is impossible to attain in practice since structures deform in all directions, no matter how small, once external loads are applied. As the large deflection and material yielding effects are not considered here, the factor is an upper bound solution that cannot be used directly in design. However, λ_{cr} is useful in assessing the stability condition.

The following requirements are imposed in the CoPHK (2011).

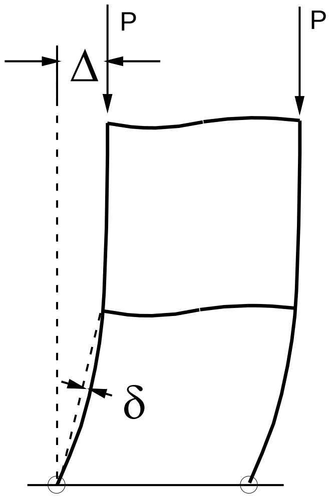
(1) When λ_{cr} < 5, a structure must be designed by a second-order analysis discussed below (i.e. the P-Δ moment must be considered by a second-order analysis and the first-order linear analysis with or without moment amplification cannot be used)

(2) When 5≤ λ_{cr} < 10, the structure is sway-sensitive and moment must be amplified for the sway effect (i.e. the P-Δ moment must be considered and the first-order linear analysis with moment amplification method can be used) and

(3) When λ_{cr} ≥10, the frame is sway insensitive that the sway effect can be ignored (i.e. the P-Δ moment can be ignored)

λ_{cr} can be determined either by the deflection method in the CoPHK (2011) or in computer. In all cases, the P-δ moment must be considered in using imperfect member in analysis or the buckling curves in design code.

Second-order analysis can be used in all cases above.



P-Δ for nodal displacement / sway
P-δ for member curvature / bowing

Figure 5.2 The P-Δ and P-δ Moments

5.1.2 P-Δ-Only Analysis

P-Δ-only analysis is the simplest method in the family of non-linear analyses. In this method, the only considered non-linear effect is the change of structural geometry due to nodal displacements. The process is to simply add displacements to coordinates of all nodes in a structure during an analysis. As the member curvature or the P-δ effect indicated in Figure 5.2 is ignored and the member is assumed to remain perfectly straight in an analysis, design formulae in a design code must be applied in this method for checking the buckling resistance of a member with its length either of an effective member length or multiplied by a rationally determined effective length factor (L_e/L).

5.1.3 P-Δ-δ Elastic Analysis

P-Δ-δ elastic analysis allows for the non-linear effects due to nodal displacement and member bow. The consideration of these two effects is equivalent to reducing the member resistance by the corrected value effective length (unfortunately this is unknown for most except the simplest structures). The resistance of a member in a frame is adequate when the following equation for section capacity check is satisfied.

$$\frac{P}{p_y A} + \frac{(M_y + P\Delta_y + P\delta_y)}{M_{cy}} + \frac{(M_z + P\Delta_z + P\delta_z)}{M_{cz}} = \phi \le 1 \tag{5.1.2}$$

where

Δ	Displacement due to sway of the frame measured at nodes
δ	Displacement due to member curvature or bowing, measured along a member
P	Axial force in member
A	Cross sectional area
p_y	Design strength
M_{cy}	Moment capacity about principal y-axis
M_{cz}	Moment capacity about principal z-axis
M_y	External moments about principal y-axis
M_z	External moments about principal z-axis
φ	Section capacity factor. If φ >1, member fails in section capacity check.

The term "elastic" here indicates that the method disallows stress or moment re-distribution. However, a small degree of material yield for compact or plastic section based on the "first-plastic–hinge" concept can be used in Equation (5.2) by replacing the elastic moduli (Z_Y and Z_Z) with the plastic moduli (S_Y and S_Z). Even with this use of plastic modulus and plastic moment as moment capacity in this "First-plastic-hinge" design approach, stress or moment re-distribution is not allowed.

5.1.4 P-Δ-δ Plastic Analysis or Advanced Analysis

The P-Δ-δ plastic analysis is not the same as the conventional rigid-plastic design based on equalization of external work done and internal strain energy, which ignores the effect of buckling or large deflection. P-Δ-δ plastic analysis allows for buckling and material yielding and it extends the method of elastic analysis to plastic range utilizing the mechanism of moment or stress re-distribution after the first plastic hinge formation. This is in line with the ultimate limit state design which requires the structural resistance allowing for the effects of buckling and plastic yielding larger than the factored ultimate load. In the computer analysis, a plastic hinge is inserted to the end of a member when a section along the member reaches the limit of section capacity in Equation (5.1.2). The process is continued until a plastic collapse mechanism allowing for P-Δ-δ buckling effects is formed at which the load is taken as the collapse load of the frame.

5.1.5 Initial Imperfections

All structures contain imperfections due to member out-of-straightness and frame out-of-plumbness and therefore they MUST be included either in analysis or in design. As the presence of imperfections reduces the load resistance of a practical structure, their ignorance is on the unsafe side in design. The geometrical member out-of-straightness is normally not greater than 0.1% of its length but a larger value is commonly used in order to include the effect of residual stress by an enlarged equivalent geometrical member imperfection. The CoPHK (2011) and the Eurocode 3 (2005) are possibly the only two codes giving explicit imperfections for members of different cross sectional shapes while nearly all design codes allow the use of second-order analysis without giving explicit guides. To include the effect, the imperfections are assumed to have initial values before load applications. Thus,

$$\delta = \delta_0 + \delta_P \quad \text{and} \quad \Delta = \Delta_0 + \Delta_P \tag{5.1.3}$$

in which δ, δ_0 and δ_p are member total, initial and load-induced member bowing displacements and Δ, Δ_0 and Δ_P are frame total, initial and load-induced out-of-plumbness displacements.

5.1.5.1 Frame Imperfection

Frames can hardly be built perfectly vertical and out-of-plumbness should be allowed. In the steel design codes, an inclination of 1/500 or 0.2% is commonly adopted whereas temporary or other structures require a larger value of out-of-plumbness. Although the effect of applying a notional force of 0.5% of the total vertical loads is not exactly the same as the use of slightly inclined structural geometry in an analysis, the codes allow the interchanged use of the two approaches. However, for irregular frames like domes, both methods are difficult to apply, because of the irregularity of the structural geometry. The CoPHK (2011) allows the use of the buckling mode as the imperfection mode. The magnitude of imperfection should be taken to be not less than the construction tolerance in analysis.

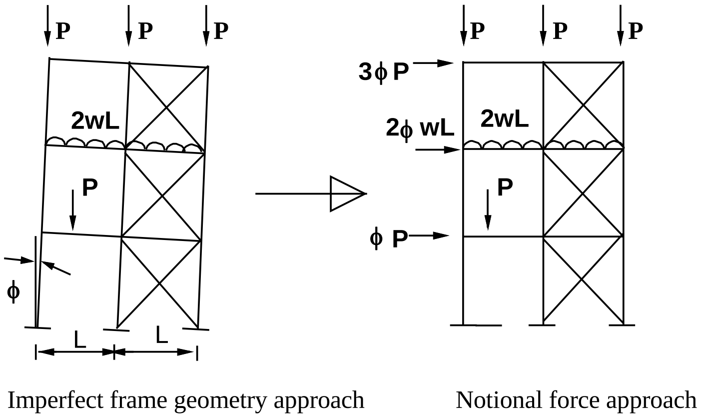


Figure 5.3 Transformation of out-of-plumbness to Horizontal Notional Force

5.1.5.2 Member Imperfection

Similar to frames which cannot be perfectly vertical, members can hardly be perfectly straight. The initial imperfection or crookedness is not totally due to geometrical defect, it also allows for residual stress in different sections. The procedure of curve-fitting to obtain a lower-bounded match of imperfection against the buckling curve in the code was discussed by Cho and Chan (2005) whose proposed imperfections are the basis of the subsequently drafted the CoPHK (2011) and it is similar to the Eurocode 3 (2005).

5.1.6 Elastic vs. Plastic Analysis

Elastic or first-plastic-hinge design has been used for century and plastic analysis is mainly limited to portal frame design. While steel accepted for use in building structures is reasonably ductile and has a minimum elongation at fracture of 15%, the ignorance of favorable effect in redundant structures by using elastic analysis is unjustifiable and un-sustainable for several reasons as follows.

- The elastic analysis discourages engineers to design robust and redundant structure against local failure since the design stops at the first plastic hinge. The approach does not allow engineers to consider in an analysis the strength reserve after first yield of structures while redundant and robust structures do not fail at the first plastic hinge.
- Engineers quite often like to make use of ductility of steel in their design and, for extreme loads during rare events, elastic design is uneconomical and puts a consultant using the elastic design into a non-competitive position.
- From past record, one can hardly find frame failure initiated by formation of plastic hinges in beams. Inspection of steel structures after earthquake showed that member buckling and cracking at connections were more common but plastic hinge in beams was unusual. This indicates buckling and connection are two important aspects in structural steel design against collapse and design allowing for plastic behaviour and stress re-distribution is both a safe and a sensible direction for design.
- For design of structures under static loads, the authors suggest the use of P-Δ-δ plastic analysis in ultimate load design and P-Δ-δ elastic analysis for design under working loads, with both analyses allowing for mandatory frame and member imperfections. This ensures that the structure will not collapse under ultimate loads or yield to store energy under working load condition.

5.1.7 Design Hierarchy

With the introduction of the new design method, it will be useful to summarize the relationship between various design methods for appreciation by engineers in choosing a suitable design method for certain structure. The flow-chart below indicates their relationship.

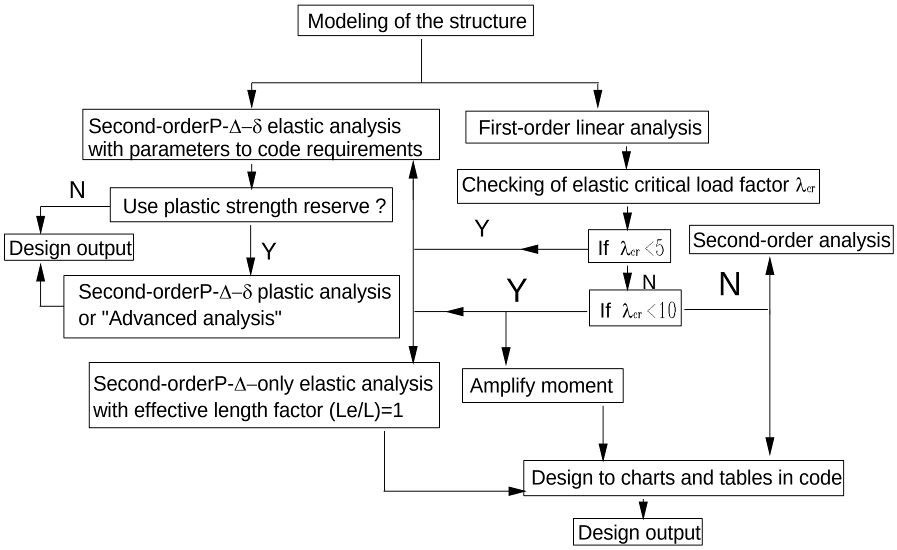


Figure 5.4 Flow-chart of Structural Design under the Contemporary Codes

5.2 Analysis Types

5.2.1 Introduction

Second-order analysis is considered in many design codes as a more reliable and accurate design and analysis tool for slender frames. For example, CoPHK (2011) disallows the use of the first-order linear analysis when the elastic critical load factor (λ_{cr}) is less than 5 while similar requirement also applies for other codes like AS4100 (1998), BS5950 (2000) and the Eurocode 3 (2005). However, unlike the first-order linear analysis which has only one version of assuming the external force proportional to displacement and stress, there are several versions of second-order nonlinear analysis since many aspects of structural behaviour can be considered as non-linear or assumed as linear. Being a powerful tool with saving in material weight, improvement in safety margin and reduction in design effort and time, the second-order nonlinear analysis can be dangerous if not used properly.

5.2.2 Background

In structural analysis and design, a suitable model to represent and simulate the true behaviour of a structure is necessary for obtaining an accurate output. The first-order linear analysis is based on the assumption of elastic material behaviour and undeformed structural geometry for equilibrium check. The elastic analysis does not fully comply with the requirements of the limit state design (LSD) philosophy and additional member check is needed to ensure nonlinear effects due to buckling and plastic failure do not occur under the design loads. Also, the use of plastic moment capacity in design and the adoption of an elastic analysis are inconsistent. More critically, buckling is system behaviour while the design check is member-based and they are not compatible with great uncertainty in effective length factor (Le/L).

A complete non-linear analysis traces the structural response until the limit state is reached with allowance for non-linear geometrical and material behaviour and imperfections. As such, the limit load from the analysis can be directly compared with the factored design load and individual member design becomes unnecessary. Because of this characteristic, the method is sometimes called the “Direct analysis” in the U.S.A. However, in order to simplify the complexity of an analysis, different versions of non-linear analysis are available. The engineer must therefore fully realize the limitation of any one of these versions otherwise disaster due to over-estimation of load resistance may occur. The P-Δ-only version of second-order analysis requires additional checks using the tables in the code and the software is not fully “automatic”. All software for second-order non-linear analysis, like the first-order linear analysis, requires careful input of data file and interpretation of data output. This section introduces the design concept for a second-order elastic and plastic analysis of steel frames used in many real projects. Below is a summary of these methods which are essential in understanding the new design theory.

5.2.3 First-order Linear Analysis

As other structural analysis software, NIDA plots the bending moment and shear force diagrams under the assumption that change in geometry under load does not affect the structural stiffness.

5.2.4 Second-order Analysis

In this second-order analysis, NIDA performs the following functions.

- Calculate the displacements and rotations at all the nodes or junctions of elements or members, allowing for the change of structural geometry (P-δ and P-Δ effects) upon loading.
- Calculate the bending moments about the element cross sectional axes, torsional moment about the longitudinal axis and axial force in the member, allowing for the second-order non-linear effects due to axial force. Shear will also be determined in the computer output.
- Design a structure by section capacity check that effective length is not required to be assumed.
- Check the instability of members as well as global structure.
- Design the structure by a system approach, in contrast to the traditional "member-based" design method.

5.2.4.1 Second-order Elastic Analysis

In the design of steel structures by second-order non-linear analysis, the program increments the load in a step-by-step and incremental-iterative manner. Thus, a small increment of, say, from 5 to 25%, of expected design load is applied to the structure and the displacements are then computed and used to calculate the resistance. Iteration for equilibrium is carried out if they do not balance and convergence is assumed when the error of the norm of the unbalanced forces is smaller than 0.1% of the applied force. After convergence, the section capacity is checked for each member using Equation (5.1.2). When any one of the members fails to meet the section capacity check, it is considered to have failed and it is then indicated in red and shown in post-viewer. After this step, a new load increment is applied and the same iterative procedure is exercised. This incremental-iterative procedure is activated until the specified number of load increments has been applied and the analysis is completed or when divergence occurs. In the whole design and analysis process, no assumption of effective length is needed since the P-δ and P-Δ effects have been considered in Equation (5.1.2).

5.2.4.2 Second-order Plastic Analysis or Advanced Analysis

This type of second-order plastic or advanced analysis is similar to the above elastic analysis except it needs not stop at the first plastic hinge as its design resistant load. When a member fails, a hinge is inserted to the member end close to the hinge position and analysis continues until the collapse load is reached. The collapse load is taken as the load level which does not allow further load increase indicated as a curve reaches plateau, descends or stagnates in the load vs. deflection plot. In design, this collapse load should be greater than or equal to the factored design load in all load cases.

In Plastic Advanced "plastic element" analysis, when a member reaches its design resistance, the axial force and moments of the member are kept constant and not allowed to change with the increasing load. This implies additional loads will be re-distributed to other members.

In plastic advanced "plastic hinge" analysis, when a member reaches its design resistance, a plastic hinge will be inserted to the node close to the location of plastic moment.

Note that, in second-order elastic-plastic analysis, the load increment should be smaller and generally should be less than 1% of the expected design load. Also, the arc length plus minimum residual displacement method should be used with control parameters sufficiently smaller, normally between 2 to 3.

5.2.5 Vibration and Buckling Analysis

NIDA determines the natural frequency by lump mass or consistent mass assumptions and the eigen-value buckling load factor as follows.

- For natural frequency analysis

$$\left|K_L + \omega^2 M\right| = 0 \quad \text{and period} \quad T = 2\pi / \omega \tag{5.2.1}$$

- For eigenvalue buckling analysis

$$\left|K_L + \lambda K_G\right| = 0 \tag{5.2.2}$$

5.3 General Nonlinear Parameters for Analysis

5.3.1 Total Load Cycles

It describes the number of total load cycles for the nonlinear analysis. The number is equal to the total number of load cycles required in the Analysis.

5.3.2 Maximum Iterations for each Load Cycle

It describes the maximum number of iteration for each load cycle in the analysis. If the equilibrium condition is satisfied before this number is reached or the iteration number is equal to this assigned iteration number, another load step will be imposed until the permitted number of load steps is reached. The tolerance for equilibrium check is 0.1 % by default. That is, when the Euclidean norms of the unbalanced displacements and the unbalanced forces are less than respectively 0.1 % of the total applied forces and the total accumulated displacements, the equilibrium condition is assumed to have been satisfied.

5.3.3 Number of Iterations for Tangent Stiffness Matrix

It describes the number of iterations for the tangent stiffness matrix to reform during the iterative process. When this number is specified to be very large or simply equal to the “maximum number of iterations for each load cycle” above, the iterative scheme will then become the modified Newton Raphson method. If the number here is specified as "1”, it becomes the Newton Raphson method. If the number is between these two extremes, the method is a mixed Newton-Raphson method. When compared to modified Newton Raphson method, the Newton Raphson method generally requires less number of iterations for convergence, but longer time for each iteration. It is recommended to use the Newton Raphson method

5.3.4 Incremental Load Factor

This factor will be used as the first load factor used for the analysis and the load factor increment in subsequent analysis. It is different from the design load factor behind “header load” which is multiplied to the input load to obtain the design load vector and will not appear in the plotting of equilibrium or load-deflection curve with its value generally taken as, for example, 1.6 for wind, 1.4 for self-weight etc. The load factor described here is used as the ratio of the current applied load to the input design load. For example, if a structure yields at a load factor of 2.6, it means when the applied load is 2.6 of the design load, the structure yields.

5.3.5 Imperfection Method & Direction

It describes the direction of initial imperfection with different methods. It can be no initial imperfection, initial imperfection in one principal plane causing less severe effect than initial imperfection in both the principal planes. The minimum magnitude of initial imperfection is taken as 1/1000 of the member length if the initial imperfections are allowed. For some sections such as cold-formed sections, this value may not be adequate. For global imperfections of a structure, the notional force can be used in place of member imperfections.

5.3.6 Magnitude of Imperfection for Global Eigenvalue Mode

This value is the magnitude of imperfection when eigenvalue buckling mode is adopted. After the eigenvalue analysis, the eigen-mode is determined and a set of initial imperfection is determined for the structure with this mode shape. This number is for the magnitude (maximum) of the initial deflection for the eigen-mode which is then added to the initial geometry of the structure.

5.4 Numerical Methods for Nonlinear Analysis

5.4.1 Load Incremental Schemes

It describes the solution method for control of load increment size. There are five methods to be selected.

Newton-Raphson (Constant Load) Method
Single Displacement Control Method
Minimum Residual Displacement Method
Arc Length Method
Constant Work Method

5.4.2 Iterative Schemes

It describes the solution method for control of iterative scheme. Same as above are five methods to be selected.

Newton-Raphson (Constant Load) Method
Single Displacement Control Method
Minimum Residual Displacement Method
Arc Length Method
Constant Work Method

5.4.3 Incremental-Iterative Schemes

Generally the numerical methods for incremental-iterative procedure can be any methods mentioned above. For easy use, three common combinations are provided.

Newton-Raphson (Constant Load) Method

For constant load increment method, the Newton-Raphson method will be used for load incremental scheme and iterative scheme.

Single Displacement Control (Constant Displacement) Method

For constant displacement increment method, single displacement control method will be used for load incremental scheme and iterative scheme.

Arc Length Method + Minimum Residual Displacement Method

Arclength method will be used for load incremental scheme and minimum residual displacement method will be used for iterative scheme.

5.4.4 Solution Methods

5.4.4.1 Newton-Raphson (Constant Load) Method

This is the only method providing the response of a structure at the input load in terms of buckling strength and therefore it should be used check the adequacy of a structure under a set of factored design loads. In the Newton-Raphson method, iteration is activated to obtain the equilibrium condition between the applied forces and the internal structural resistance within a load step. Unlike the pure incremental method in which no equilibrium check is performed, the unbalanced force is dissipated via the iterative procedure and can therefore be eliminated by this method. Being free from the drift-off error, the solution is more accurate and the computational time is reduced when compared to the pure incremental method. The general procedure can be summarized by the following recurrent equations. The quantities in the following can be referred to Figure 5.5 and Figure 5.6. The load increment at the **k**-th cycle and the **i**-th iteration, $\{\Delta F_i^k\}$, can be obtained from the external load vector $\{\hat{F}\}$ as,

$$\{\Delta F_i^k\} = \Delta \lambda^k \{\hat{F}\} \tag{5.4.1}$$

in which $\Delta \lambda^k$ is an incremental load factor at the **k**-th load cycle.

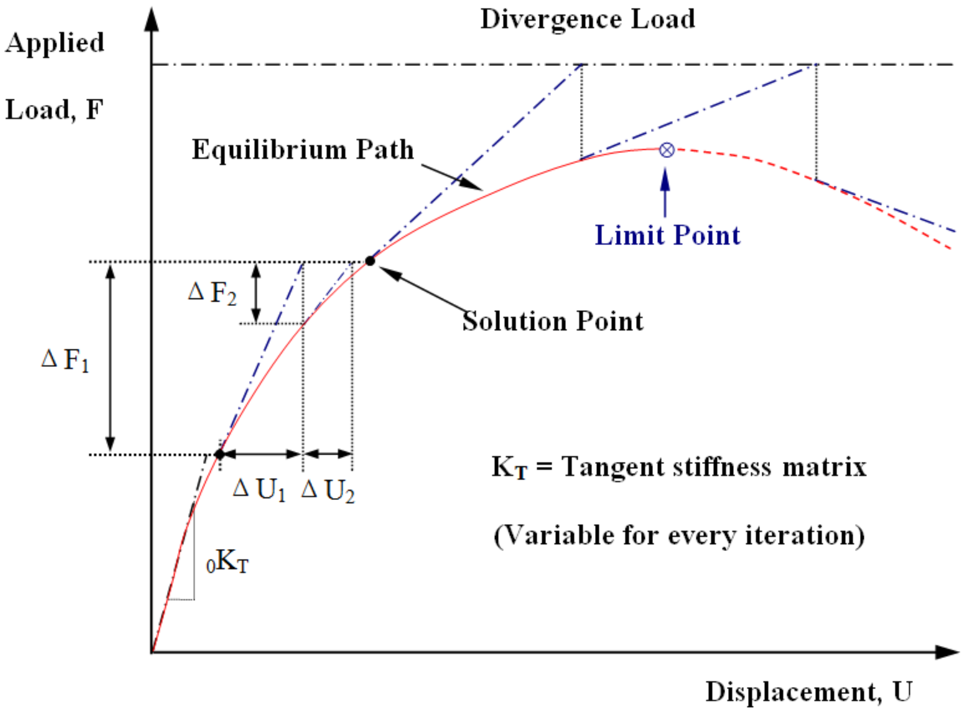


Figure 5.5 Conventional Newton-Raphson Method

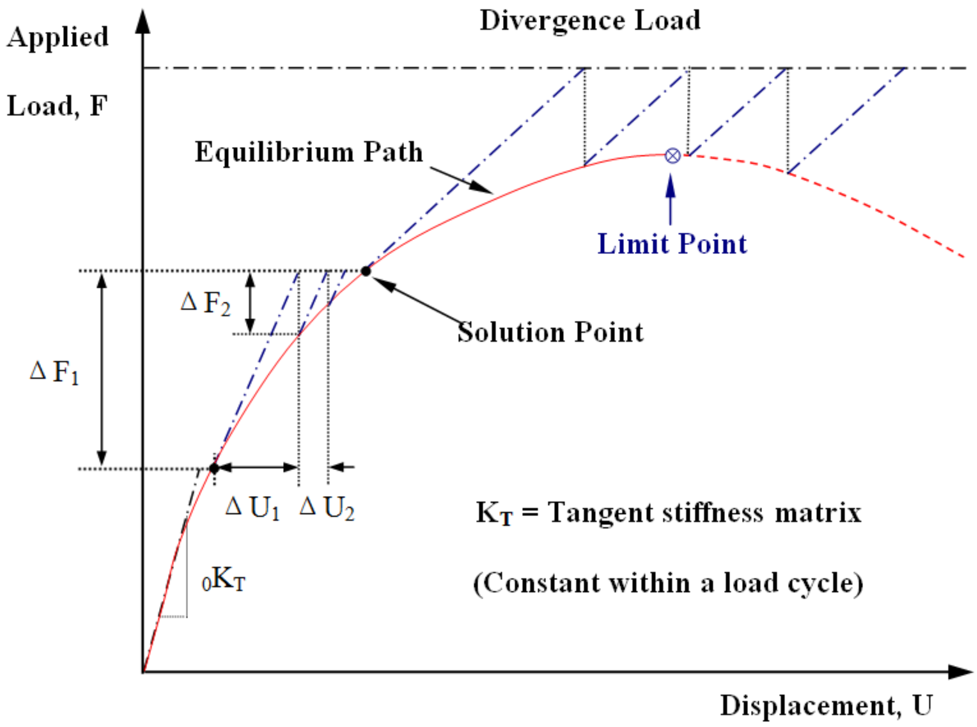


Figure 5.6 Modified Newton-Raphson Method

For the **i**-th iteration within this load step, the displacement increment $\{\Delta r_i^k\}$ can be determined as,

$$\{\Delta r_i^k\} = [K_i^k]_T^{-1}\{\Delta F_i^k\} \tag{5.4.2}$$

in which $[K_i^k]_T$ is the instantaneous tangent stiffness matrix; the superscript **i** is for the **i**-th iteration. Thus, the updated total displacement $\{r_{i+1}^k\}$ can be added by the last displacement $\{r_i^k\}$ to this displacement increment as,

$$\{r_{i+1}^k\} = \{r_i^k\} + \{\Delta r_i^k\} \tag{5.4.3}$$

The resistance of the structure $\{R_{i+1}^k\}$ and the unbalanced force $\{\Delta F_{i+1}^k\}$ can be calculated as,

$$\begin{cases} \{R_{i+1}^k\} = \sum[\,{}^eK_{i+1}^k]\{r_{i+1}^k\} \\ \lambda^k = \lambda^{k-1} + \Delta\lambda^k \\ \{\Delta F_{i+1}^k\} = \lambda^k\{\hat{F}\} - \{R_{i+1}^k\} \end{cases} \tag{5.4.4}$$

in which $[\,{}^eK_{i+1}^k]$ is the element stiffness at local coordinate and $\{r_{i+1}^k\}$ is the element nodal displacement extracted from the global displacement vector and transformed to the element local axis. The procedure from Equations (5.4.2) to (5.4.4) is repeated until convergence is reached.

For the conventional Newton-Raphson method, the tangent stiffness matrix of the structure $[K_i^k]_T$ will be updated at every iteration while for the modified Newton-Raphson method, it is reformed only in the first iteration and is kept unchanged within the load cycle.

The conventional and the modified Newton-Raphson methods usually provide a rapid rate of convergence in the stable equilibrium range. However, when approaching the limit point of the load-deflection curve, a large number of iterations will be required even for a small load increment. In the Newton-Raphson method, the solution point at the specified applied load level is sought and therefore any unloading path cannot be traced. Consequently, the solution scheme diverges near the critical point due to the ill-conditioning of the tangent stiffness matrix or simply due to the size of the load increment being greater than the limit load as shown in the Figure 5.5 and Figure 5.6.

5.4.4.2 Single Displacement Control Method

Unlike the load control methods previously described, a constraint equation for displacement is imposed in this approach. The displacement control method was originally proposed by Argyris (1965). In his study, however, the symmetrical nature of the tangent stiffness matrix is destroyed by adding the displacement constraint equation. In order to retain the symmetrical property of the tangent stiffness matrix, Batoz and Dhatt (1979) imposed the constraint for displacement via iteration. According to their procedure, a single displacement degree of freedom is chosen to be constrained. The accumulated value of this displacement within a load cycle is given by,

$$ {}^{j}r_i^k = {}^{j}r_{i-1}^k + (\Delta \lambda_i^k \; {}^{j}\hat{r} + \Delta \; {}^{j}r_i^k) \tag{5.4.5} $$

in which ${}^{j}r_i^k$ is the accumulated displacement for the **j**-th degree of freedom, at **k**-th load cycle and in the **i**-th iteration; $\Delta \lambda_i^k$ is the incremental load factor; and ${}^{j}\hat{r}$ and $\Delta \; {}^{j}r_i^k$ are respectively the displacements for the unbalanced force and for the reference load vector at the **j**-th degree of freedom.

In the first iteration, we have ${}^{j}r_0^k = \Delta \; {}^{j}r_1^k = 0$ and hence Equation (5.4.5) can be simplified as,

$$ \Delta \lambda_1^k = - \frac{{}^{j}\tilde{r}}{{}^{j}\hat{r}} \tag{5.4.6} $$

in which ${}^{j}\tilde{r}$ is the specified displacement increment at the **j**-th degree of freedom. After the first load increment, the control displacement is kept constant so that,

$$ {}^{j}r_i^k = {}^{j}r_{i-1}^k \tag{5.4.7} $$

Substituting Equation (5.4.7) into (5.4.5), the adjustment on the load factor at **i**-th iteration, $\Delta \lambda_i^k$, can be obtained as,

$$ \Delta \lambda_i^k = - \frac{\Delta \; {}^{j}r_i^k}{{}^{j}\hat{r}} \tag{5.4.8} $$

The diagrammatic presentation of the above procedures is depicted in Figure 5.7.

The constant displacement method does not exhibit any difficulty in passing the snap-through limit point but fails to converge in snap-back problems. Thus, it is usually used in conjunction with other solution schemes in order to solve general nonlinear problems. For example, Sabir and Lock (1972) used the constant load Newton-Raphson method, which can handle the snap-back but not the snap-through problem, together with the constant displacement method in their nonlinear analysis of shell structures exhibiting snap-through and snap-back behaviour.

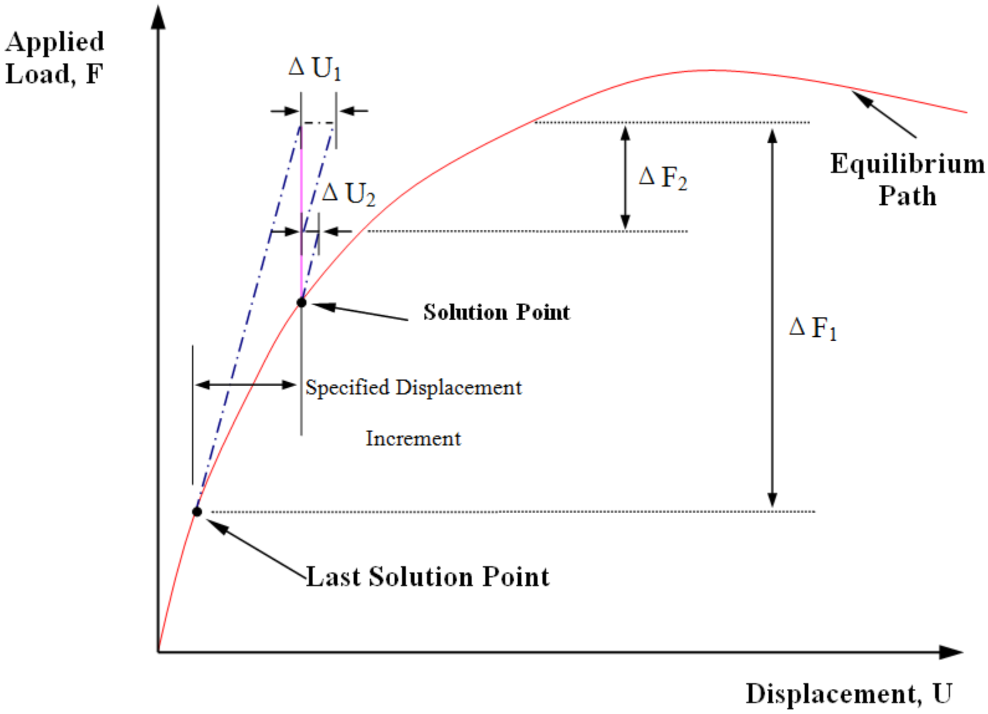


Figure 5.7 Displacement Control Method

5.4.4.3 Constant Work Method

Similar to the constant displacement method, the basic idea of the constant work method is to impose a constraint equation to guide the incremental load. In this case, the work done by the external loading is kept constant within a load increment. The use of this concept on nonlinear analysis has been reported by many researchers including Honecher (1980), Powell and Simons (1981), Karamanlidis et al. (1981), Bathe and Dvorkin (1983) and Yang (1984).

Before introducing the constant work constraint, Equation (5.4.5) is first rewritten in terms of the complete displacement vector instead of a single degree of freedom as,

$$\{r\}_i^k = \{r\}_{i-1}^k + \left(\Delta \lambda_i^k \{\hat{r}\} + \{\Delta r\}_i^k\right) \tag{5.4.9}$$

Assume δW_o to be the specified work increment in a load step. For the first iteration, $\{r\}_{i-1}^k$ and $\{\Delta r\}_i^k$ are zeros. Hence, we have,

$$\delta W_o = \{F\}_1^k \{r\}_1^k = (\Delta \lambda_1^k)^2 \{\hat{F}\}^T \{\hat{r}\} \tag{5.4.10}$$

and,

$$\Delta \lambda_1^k = \sqrt{\left(\frac{\delta W_o}{\{\hat{F}\}^T \{\hat{r}\}} \right)} \qquad (5.4.11)$$

in which δW_o can be calculated according to the work done by the specified load in the first cycle; and $\{\hat{r}\}$ is the displacement increment due to the reference external load as,

$$\{\hat{r}\} = [K_i^k]_T^{-1} \{\hat{F}\} \qquad (5.4.12)$$

For the second and subsequent iterations, the external work done is kept constant and thus there should be no work done due to the change in displacement. This condition can be expressed as,

$$\lambda_i^k \{\hat{F}\}^T \left(\Delta \lambda_i^k \{\hat{r}\} + \{\Delta r\}_i^k \right) = 0 \qquad (5.4.13)$$

and the load factor, $\Delta \lambda_i^k$, can be expressed as,

$$\Delta \lambda_i^k = - \frac{\{\hat{F}\}^T \{\Delta r\}_i^k}{\{\hat{F}\}^T \{\hat{r}\}} \qquad (5.4.14)$$

Graphical illustration of the procedure for the constant work method is shown in Figure 5.8. It can be seen from the figure that the load increment is directed towards the load-deformation curve so that there would not be any difficulty in handling both general snap-through and snap-back problems. Exception of this will be the case when the loaded degrees of freedoms exhibit snap-back behaviour. This characteristic has been discussed by Chan and Ho (1990). However, Bathe and Dvorkin (1983) showed that its rate of convergence is slow when compared to the arc-length method which is discussed in the following sub-section.

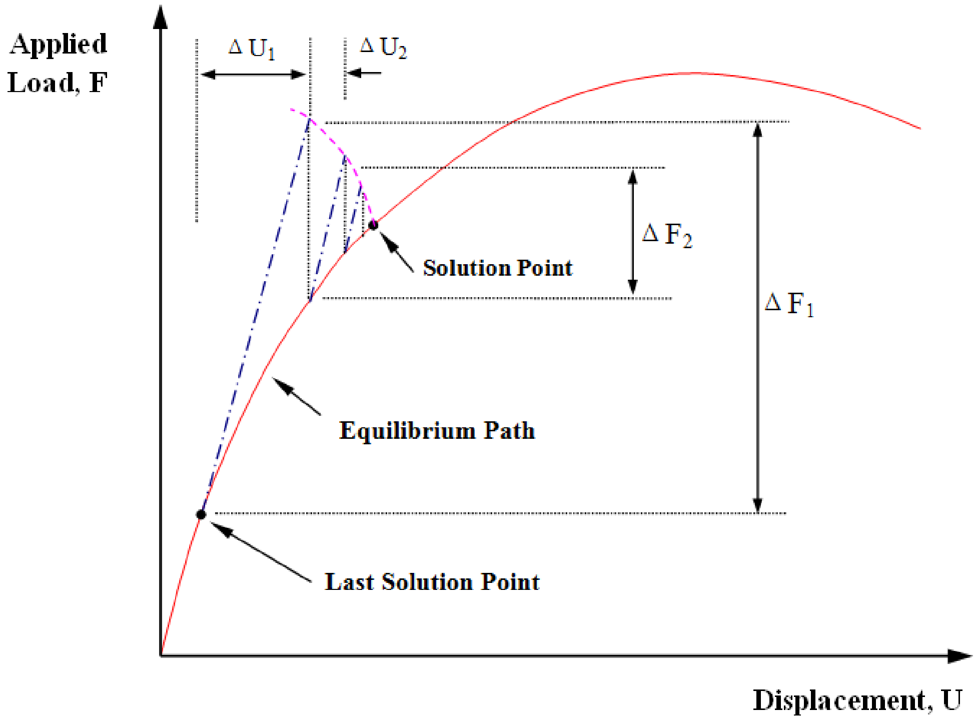


Figure 5.8 Constant Work Method

5.4.4.4 Arc Length Method

Different forms of the arc-length method have been proposed by Wempner (1971), Riks (1979) and Ramm (1980; 1981) for nonlinear analysis. In NIDA, the spherical constant arc-length method suggested by Crisfield (1981) is adopted as it shows to be more reliable than others.

The basic concept of the spherical arc-length method is to constrain the load increment so that the dot product of displacement along the iteration path remains constantly in the 2-dimensional plane of load versus deformation. This means that the constraint condition is given by,

$$\left[\{r\}_{i-1}^{k} + \Delta\lambda_i^k\{\hat{r}\} + \{\Delta r\}_i^k\right]^T \left[\{r\}^{k} + \Delta\lambda_i^k\{\hat{r}\} + \{\Delta r\}_i^k\right] = \hat{S}^2 \qquad (5.4.\ 15)$$

in which $\hat{S}$ is the specified constant arc-length.

For the first iteration, $\{r\}_0^k = \{\Delta r\}^k = 0$. Thus, the first incremental load factor is given by,

$$\Delta\lambda_1^k = \frac{\hat{S}}{\pm\sqrt{\{\hat{r}\}^T\{\hat{r}\}}} \qquad (5.4.\ 16)$$

It is noted that the sign for the load increment will be the same as that for the determinant of the updated tangent stiffness matrix. In other words, a positive determinant will lead to an increase in loading while a negative determinant will result in a decreasing load. This concept of choosing the sign was first suggested by Bergan and Soreide (1978) in their studies using the method of current stiffness parameter.

The load increment for the second and subsequent iterations can be calculated by expanding Equation (5.4.15), which yields a quadratic equation in $\Delta\lambda_i^j$ as,

$$b_2\,(\Delta\lambda_i^k)^2 + b_1\,(\Delta\lambda_i^k) + b_0 = 0 \qquad (5.4.\ 17)$$

where,

$$\begin{cases} b_2 = \{\hat{r}\}^T \bullet \{\hat{r}\} \\ b_1 = 2\left(\{r\}_{i-1}^k + \{\Delta r\}_i^k\right)^T \bullet \{\hat{r}\} \\ b_0 = \left(\{r\}_{i-1}^k + \{\Delta r\}_i^k\right)^T \bullet \left(\{r\}_{i-1}^k + \{\Delta r\}_i^k\right) - \hat{S}^2 \end{cases} \qquad (5.4.\ 18)$$

In general, two different roots for $\Delta\lambda_i^k$ can be obtained from Equation (5.4.17). The proper solution to be used would be the one which satisfies the following condition as,

$$\Delta\lambda_i^k\ \{\hat{F}\}\ \{r\}_i^k\ >\ 0 \qquad (5.4.\ 19)$$

In case when both roots satisfy Equation (5.4.19), the one close to the linear solution given below should be chosen as,

$$\Delta\lambda_i^k = -\frac{b_0}{b_1} \qquad (5.4.\ 20)$$

If the solution point is approaching the limit point, a smaller arc-length should be used to prevent divergence. The specified arc distance at a particular **i**-th iteration is given by,

$$\hat{S}_k = \hat{S}_{k-1} \left(\frac{I_d}{I_{k-1}} \right)^{\frac{1}{2}} \tag{5.4. 21}$$

in which $\hat{S}_k$ and $\hat{S}_{k-1}$ are the arc-lengths for the **k**-th and **(k-1)**-th load cycle; I_d is the desired number of iterations; and I_{k-1} is the number of iterations used in the last or **(k-1)**-th cycle.

The procedure of the spherical arc-length method is illustrated in Figure 5.9. Owing to its accuracy, reliability and satisfactory rate of convergence, it is probably the most popular method for nonlinear analysis and it was noted to be robust and stable for pre- and post-buckling analysis.

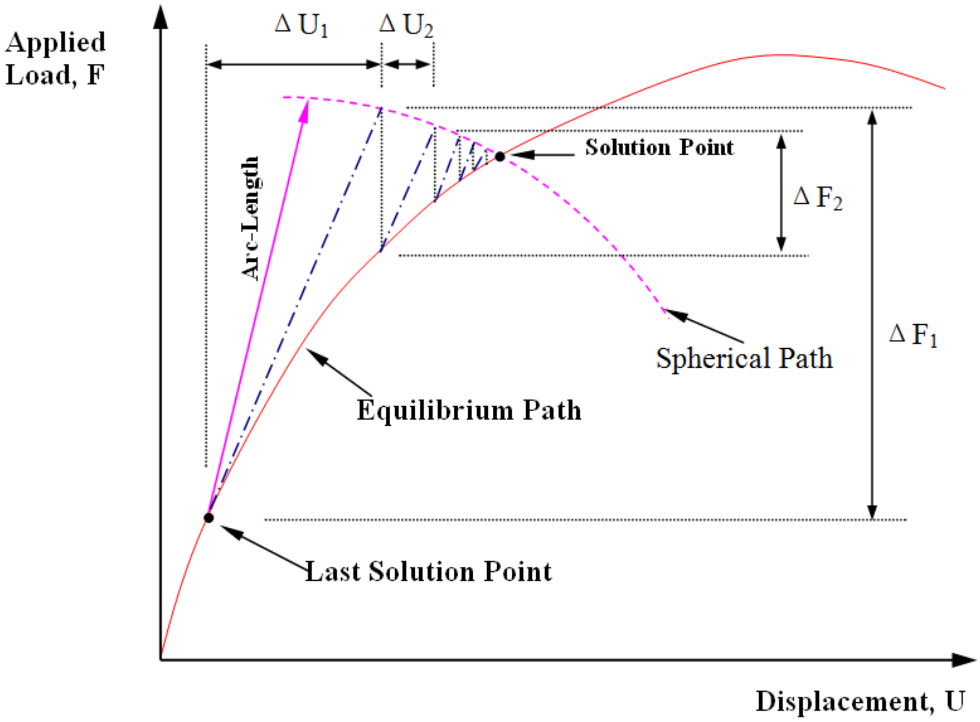


Figure 5.9 Arc-length Method

5.4.4.5 Minimum Residual Displacement Method

The basic idea of this method originally proposed by Chan (1988) is to minimize the norm of residual displacement in each iteration. The constraint equation can be written as,

$$\frac{d}{d\lambda_i^k}\left[\left(\{\Delta r\}_i^k + \Delta\lambda_i^k\{\hat{r}\}\right)^T\left(\{\Delta r\}_i^k + \Delta\lambda_i^k\{\hat{r}\}\right)\right] = 0 \tag{5.4. 22}$$

Re-arrange the above equation, there obtained,

$$\Delta\lambda_i^k = -\frac{\{\hat{r}\}^T \bullet \{\Delta r\}_i^k}{\{\hat{r}\}^T \bullet \{\hat{r}\}} \tag{5.4. 23}$$

in which $\Delta\lambda_i^k$ is the load increment factor to be determined for all iterations except the first one. For the first iteration, the load increment factor is chosen to be,

$$\Delta\lambda_1^k = \Delta\lambda_1^1 \left[\frac{\{\hat{F}\}^T \bullet \{\hat{r}\}_1^1}{\{\hat{F}\}^T \bullet \{\hat{r}\}_1^k} \right]^\gamma \qquad (5.4.\ 24)$$

in which γ is a parameter defined by the analyst for the control of load size and the sign of the load factor is taken to be the same as that of the determinant of the updated tangent stiffness matrix. The choice of $\Delta\lambda_1^k$ is indeed arbitrary. It is later found that the procedure of spherical arc-length method in determining the load factor for the first iteration is more appropriate.

The graphical representation of the procedure is demonstrated in Figure 5.10. From the figure, it can be seen that this constraint condition enforces the iteration path to follow a path normal to the load-deformation curve. It adopts the shortest path to arrive at the solution path by error minimization and thus is considered to be an optimum solution. In addition, the procedure from Equations (5.4.22) to (5.4.24) is much simpler to use than the arc-length method. Generally speaking, owing to its efficiency and effectiveness in tracing the equilibrium path, the minimum residual displacement method is usually chosen to perform the iterative procedure and combined with the part for load size determination in the first iteration by the arc-length method. Unless otherwise specified, the combination for the nonlinear solution strategy is used in NIDA for static analysis.

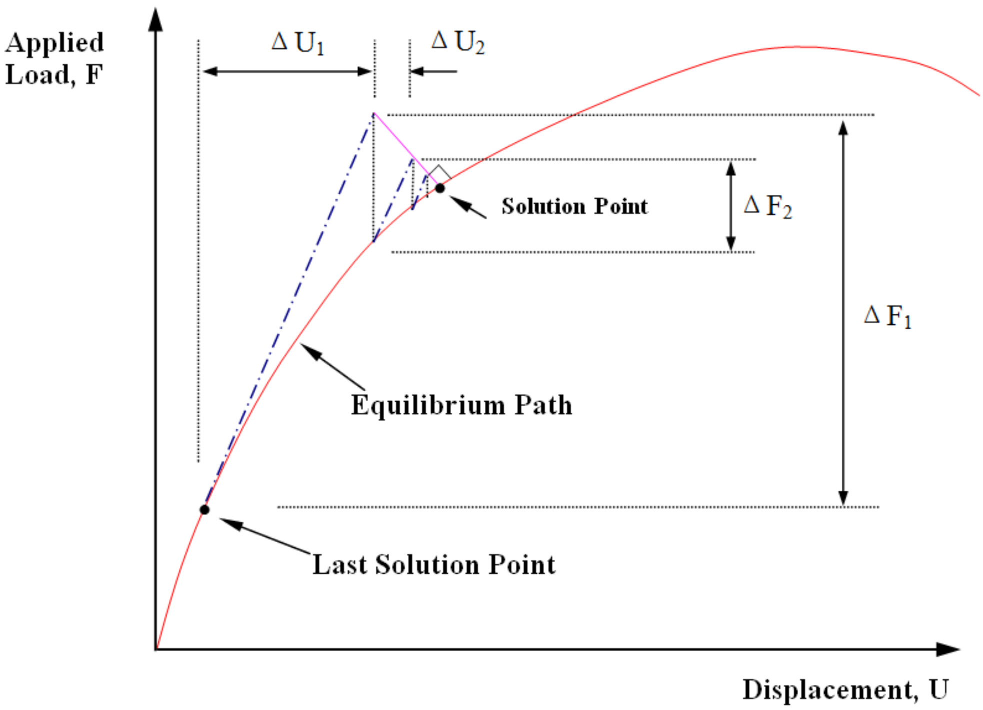


Figure 5.10 Minimum Residual Displacement Method

5.5 Beam-Column Element

5.5.1 Coordinate Systems

All structural analyses require a set of consistent coordinate systems to define the vector quantities. A properly defined coordinate system is essential for the development of reliable, convenient and user-friendly analysis computer program. In some cases the loads are referred to in a global coordinate system such as the structural dead weight and the gravitational loads. In other cases, the loads are referenced by the local coordinate system which belongs to the element itself. This loading type includes the pressure load perpendicular to the element axis. Wind load can act on the projected area about either the global or the local axes, depending on the load case being considered.

About the output of NIDA, nodal displacements are always defined in global coordinates. However, internal forces and moments are output in local coordinate systems.

In NIDA, two coordinate systems are adopted to describe analytical model, i.e., global coordinate system describing the structural model and local coordinate system describing the finite element model. To illustrate the relationship between global system and local system, a coordinate system of reference, named reference coordinate system, i.e., default local coordinate system, is introduced to help transform arbitrary local system to global system.

All coordinate systems are right-handed, 3D Cartesian systems.

5.5.1.1 Global Coordinate System

The global coordinate system OXYZ is a set of coordinate axes for the complete structure. The stiffness and applied forces for all the elements must be transformed to this coordinate system for global superposition.

The location and orientation of global coordinate system is arbitrary. In default the Y -axis is upward, but could be altered according to required conditions.

5.5.1.2 Reference Coordinate System

For sake of convenience, global coordinate system *OXYZ* and local coordinate system $oxyz$ can be linked by reference coordinate system $\overline{o}\overline{x}\overline{y}\overline{z}$, which could be defined as below:

1. $\overline{z}$ -axis always parallels to global *XZ* plane;

2. Angle α between local $\overline{x}-\overline{y}$ plane and global *X-Y* plane should satisfy $0 \le \alpha \le 90$;

3. When element or member is perpendicular to global *XZ* plane, i.e., $\alpha = 90$, $\overline{z}$ -axis and global *Z*-axis are in the same direction.

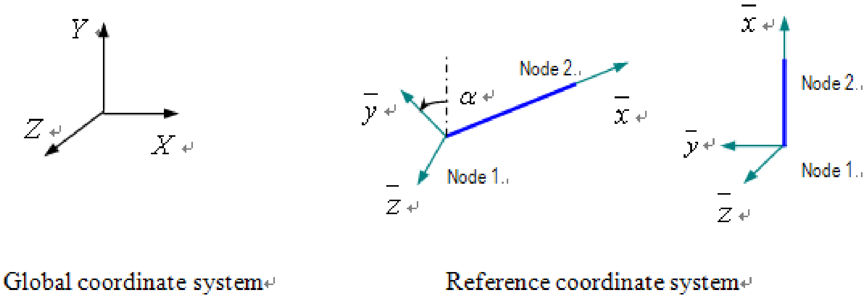


Figure 5.11 Coordinate Systems of Beam-Column Element

Reference coordinate system is the *default local coordinate system* in NIDA.

5.5.1.3 Local Coordinate System

In default, local coordinate system $oxyz$ is the same as reference coordinate system $\overline{oxyz}$.

However, the default local coordinate system may be oriented differently from the considered member section. Generally, an angle, named β (see Figure 5.12), lies between the real local coordinate system and reference coordinate system.

Once the orientation of local coordinate system is different from that of reference coordinate system, i.e. $\beta \neq 0$, the angle β should be specified in NIDA or edited in the input file manually. Alternatively, K-node (see Figure 5.13) could be specified to determinate the local xy plane expediently.

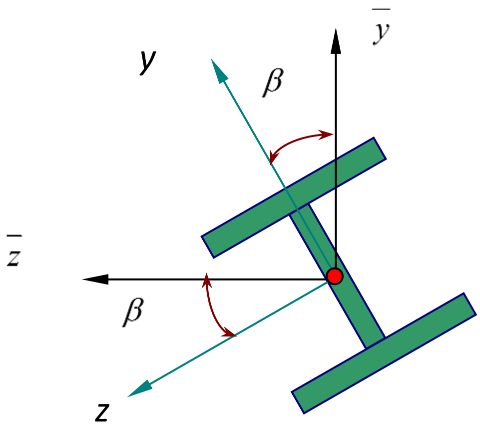


Figure 5.12 Orientation Angle β

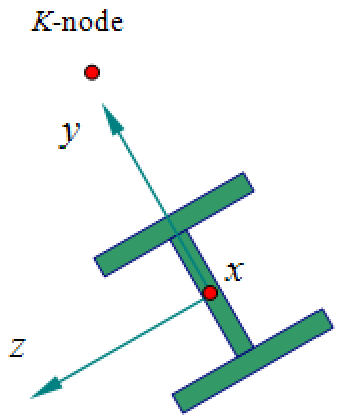


Figure 5.13 K-node for Local Axis

5.5.2 PEP Element

5.5.2.1 Introduction

Pointwise Equilibrating Polynomial (PEP) element based on a fifth-order polynomial shape function was developed to enable the second-order analysis by using one single element per member. With the inclusion of initial imperfection, the improved PEP element shows good performance in predicting the buckling behaviour of slender structure. The behaviour of high slenderness ratio steel member can be precisely captured by using PEP element. On the contrary, the conventional cubic Hermite element can only reflect the behaviour of the low slenderness ratio member. With the invention of PEP element, practical and reliable second-order analysis and design become possible.

5.5.2.2 Assumptions

The PEP element was based on the following assumptions.

- The theory is based on the underlying assumption of the beam-column theory by Timoshenko and Gere.
- The element is prismatic and elastic.
- The applied loads are conservative and nodal.
- Small strain but arbitrarily large deflection is considered.
- Warping is ignored.
- The member but not the frame is assumed to be prevented against any out-of-plane deformation due to instability caused by moments.
- Rotations between the tangent at member ends and the chord joining the two ends are small.

5.5.2.3 Formulations

The shape function of the lateral displacement along one principal plane of the PEP element as shown in Figure 5.14 is assumed to be a fifth-order polynomial which is given by

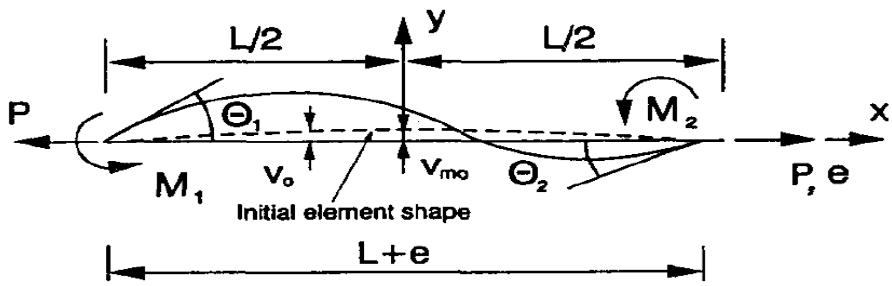


Figure 5.14 Basic Forces vs. Displacements Relations in PEP Element (with Initial Imperfection)

$$v = a_0 + a_1 x + a_2 x^2 + a_3 x^3 + a_4 x^4 + a_5 x^5 \tag{5.5. 1}$$

The initial curvature as shown in Figure 5.14 is assumed to be a quadratic function which is given by

$$v_0 = \delta_0 \left(1 - \frac{4x^2}{L^2}\right) \tag{5.5. 2}$$

In order to solve the six unknown coefficients of Equation (5.5.1), six boundary conditions are required. The differential equilibrium equation which takes the effect of axial force into consideration can be written as follows:

$$EIv'' = P(v + v_0) + \frac{M_1 + M_2}{L}\left(x + \frac{L}{2}\right) - M_1 \tag{5.5. 3}$$

Differentiating both sides, obtained

$$EIv''' = P\left(v' + v_0{}'\right) + \frac{M_2 - M_1}{L} \tag{5.5. 4}$$

For equilibrium state, the boundary conditions are given by

$$\text{At } x = 0, \qquad \begin{aligned} EIv'' &= P(v + \delta_0) - \frac{M_1 - M_2}{2} \\ EIv''' &= Pv' + \frac{M_1 + M_2}{L} \end{aligned} \tag{5.5. 5}$$

The four boundary conditions for geometrical compatibility are:

$$\text{At } x = -\frac{L}{2}, \qquad v = 0 \text{ and } v' = \theta_1 \tag{5.5. 6}$$

$$\text{At } x = \frac{L}{2}, \qquad v = 0 \text{ and } v' = \theta_2$$

Applying boundary conditions from Equations (5.5.5) to (5.5.6), the coefficients of Equations (5.5.1).can be solved. Unlike the stability function, the shape function derived of a PEP element is valid for positive, zero and negative values of P.

For axial deformation and torsion, their respective shape functions can be assumed to be linear as:

$$u = b_0 + b_1 x \tag{5.5. 7}$$

$$\theta = c_0 + c_1 x$$

At $x = 0$, $\quad u = u_1$ and $\theta_x = \theta_{x1}$

At $x = L$, $\quad u = u_2$ and $\theta_x = \theta_{x2}$

With the above boundary conditions, the shape functions of axial deformation and twist can be obtained:

$$u = \left(1 - \frac{x}{L}\right) u_1 + \frac{x}{L} u_2$$
$$\theta_x = \left(1 - \frac{x}{L}\right) \theta_{x1} + \frac{x}{L} \theta_{x2} \tag{5.5. 8}$$

The total potential energy function, Π, is given by:

$$\Pi = U - V \tag{5.5. 9}$$

in which U is the strain energy and V is the external work done. The strain energy, U, can be written as:

$$U = \frac{1}{2}\int_L EAu'^2 dx + \frac{1}{2}\int_L EIv'' dx + \frac{1}{2}\int_L P\left(v'^2 + 2v'v_0{}'\right) dx \tag{5.5. 10}$$

The secant stiffness matrix can be found from the first variation of the strain energy function.

The tangent stiffness matrix can be found from the second variation of the total potential energy function.

5.5.2.4 Transformation Matrices

Before carrying a second-order analysis, the element stiffness matrix must be transformed from the local system to the global system. The stiffness matrices above are derived basically for an element has 6 degrees of freedom, namely, $[\delta_x \ \theta_{y1} \ \theta_{z1} \ \theta_x \ \theta_{y2} \ \theta_{z2}]$. These 6 independent forces and moments are required to be expressed in terms of 12 degrees of freedom as:

$$[F] = [T][P] \tag{5.5. 11}$$

in which [F] is the 12×1 local element force vector $[F_{x1} \ F_{y1} \ F_{z1} \ M_{x1} \ M_{y1} \ M_{z1} \ F_{x2} \ F_{y2} \ F_{z2} \ M_{x2} \ M_{y2} \ M_{z2}]^{\mathrm{T}}$, $[P]$ is the 6×1 elemental force vector $[F_x \ M_{y1} \ M_{z1} \ M_x \ M_{y2} \ M_{z2}]^{\mathrm{T}}$ and $[T]$ is the transformation matrix as

$$[T]=\begin{bmatrix} -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1/L & 0 & 0 & 1/L \\ 0 & -1/L & 0 & 0 & -1/L & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1/L & 0 & 0 & -1/L \\ 0 & 1/L & 0 & 0 & 1/L & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

With the transformation matrix $[T]$, 6×6 the element stiffness matrix $[k_e]$ in the local coordinate system having the dimension of 12×12 can be obtained as

$$[{}_e k_T]=[T][k_e][T]^T \tag{5.5. 12}$$

Rotational transformation matrix for principal axes

For asymmetric members such as angles, the principle axes are not parallel to the global axes. In order to relate the sectional properties of the member in the principal axes to those parallel to the global axes, a rotational transformation matrix is required as shown as follows.

$$[L_2]'=\begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\phi & \sin\phi \\ 0 & -\sin\phi & \cos\phi \end{bmatrix} \tag{5.5. 13}$$

And

$$[L_2]=\begin{bmatrix} [L_2]' & 0 & 0 & 0 \\ 0 & [L_2]' & 0 & 0 \\ 0 & 0 & [L_2]' & 0 \\ 0 & 0 & 0 & [L_2]' \end{bmatrix} \tag{5.5. 14}$$

Local to global transformation matrix

In order to transform the stiffness matrix for the elements from the local coordinate system to the global coordinate system, the following transformation matrix is required.

$$[L_1]' = \begin{bmatrix} c_x & c_y & c_z \\ -\frac{c_x c_y}{\sqrt{c_x^2 + c_z^2}} & \sqrt{c_x^2 + c_z^2} & -\frac{c_y c_z}{\sqrt{c_x^2 + c_z^2}} \\ -\frac{c_z}{\sqrt{c_x^2 + c_z^2}} & 0 & \frac{c_x}{\sqrt{c_x^2 + c_z^2}} \end{bmatrix} \tag{5.5. 15}$$

And

$$[L_1] = \begin{bmatrix} [L_1]' & & & \\ & [L_1]' & & \\ & & [L_1]' & \\ & & & [L_1]' \end{bmatrix} \tag{5.5. 16}$$

in which c_x, c_y and c_z are the direction cosines of x, y and z axes.

However, when the member is vertical, i.e. $|c_y| = 1$, $c_x = c_z = 0$, the following matrix for $[L_1]'$ should be used.

$$[L_1]' = \begin{bmatrix} 0 & -c_y & 0 \\ c_y & 0 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{5.5. 17}$$

The overall transformation matrix $[L]$ is the product of the rotational and the local to global transformation matrices. i.e.

$$[L] = [L_1][L_2] \tag{5.5. 18}$$

The global tangent stiffness matrix of an element is given by

$$[k_T] = [L]^T [{}_e k_T][L] \tag{5.5. 19}$$

5.5.3 Curved Stability Function

Real structural members contain imperfection and so design codes require implicit (via use of different buckling curves) or explicit (via element formulation) consideration of initial imperfection in element geometry. The conventional stability function with high accuracy in beam-column analysis is improved to consider member initial bowing.

Imperfections are first assumed to be in a half sine curve with an assigned amplitude at mid-span as follows (see also Figure 5.15).

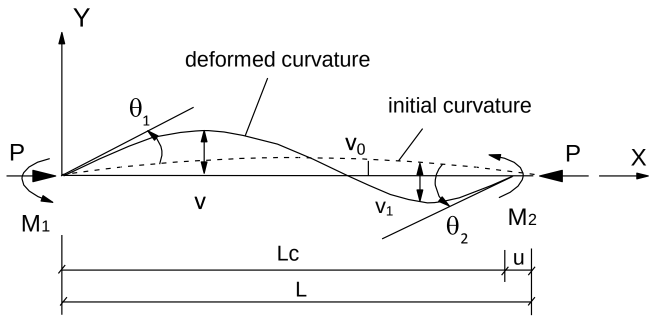


Figure 5.15 Stability Function Element with Initial Crookedness

$$v_0 = v_{mo} \sin \frac{\pi x}{L} \tag{5.5. 20}$$

in which v0 is the lateral deflection, v_{mo} is the magnitude of imperfection at the mid-span, x is the distance along the element longitudinal axis and L is the element length (Figure 5.15).

The equilibrium equation along the element length can be expressed as,

$$EI\frac{d^2 v_1}{dx^2} = -P(v_0 + v_1) + \frac{M_1 + M_2}{L}x - M_1 \tag{5.5. 21}$$

in which EI is the flexural rigidity, M_1 and M_2 are the nodal moments and v_1 is the lateral displacement induced by loads.

Making use of the boundary conditions that when x=0 and x=L, v_1=0, we have,

$$v_1 = \frac{M_1}{P}\left[\frac{\sin(\phi - kx)}{\sin\phi} - \frac{L - x}{L}\right] - \frac{M_2}{P}\left[\frac{\sin kx}{\sin\phi} - \frac{x}{L}\right] + \frac{q}{1-q} v_{mo} \sin\frac{\pi x}{L} \tag{5.5. 22}$$

Superimposing the deflection to the initial imperfection, we have the final offset of the element centroidal axis from the axis joining the two ends of the element as,

$$v = v_1 + v_0 = \frac{M_1}{P}\left[\frac{\sin(\emptyset - kx)}{\sin\emptyset} - \frac{L-x}{L}\right] - \frac{M_2}{P}\left[\frac{\sin kx}{\sin\emptyset} - \frac{x}{L}\right] + \frac{1}{1-q} v_{mo} \sin\frac{\pi x}{L} \tag{5.5. 23}$$

in which,

$$k = \sqrt{\frac{P}{EI}}, \phi = kL, q = \frac{P}{P_{cr}} \tag{5.5. 24}$$

And P_{cr} is the buckling axial force parameter given by $P_{cr} = \frac{\pi^2 EI}{L^2}$.

Differentiating Equation (5.5.22) with respect to x, and expressing the rotations at two ends in terms of the nodal rotations as, $\frac{dv_1}{dx}|_{x=0} = \theta_1, \frac{dv_1}{dx}|_{x=L} = \theta_2$, we have,

$$M_1 = \frac{EI}{L}\left[c_1\theta_1 + c_2\theta_2 + c_0\left(\frac{v_{mo}}{L}\right)\right] \tag{5.5. 25}$$

$$M_2 = \frac{EI}{L}\left[c_1\theta_1 + c_2\theta_2 - c_0\left(\frac{v_{mo}}{L}\right)\right] \tag{5.5. 26}$$

in which c_1, c_2 and c_0 are stability functions given in appendix. Axial strain can be expressed in terms of the nodal shortening, the bowing due to initial imperfection and deflection as,

$$\varepsilon = \frac{\delta_0}{L} - \frac{\delta_c}{L} + \frac{u}{L} \tag{5.5. 27}$$

in which u is the nodal shortening, δ_0 and δ_c are the shortening due to bowing of initial imperfection and deflection given by,

$$\delta_o = \frac{1}{2}\int_L \left[\frac{dv_o}{dx}\right]^2 dx \ ; \delta_c = \frac{1}{2}\int_L \left[\frac{dv}{dx}\right]^2 dx \tag{5.5. 28}$$

Substituting Equations (1) and (4) into Equations (10) and (11), we have,

$$P = EA\,\varepsilon = EA\left[\frac{u}{L} - b_1(\theta_1+\theta_2)^2 - b_2(\theta_1-\theta_2)^2 - b_{vs}\frac{v_{mo}}{L}(\theta_1-\theta_2) - b_{vv}\left(\frac{v_{mo}}{L}\right)^2\right] \tag{5.5. 29}$$

in which b_1, b_2, b_{vs} and b_{vv} are curvature functions and A is the cross sectional area. These functions need to be repeatedly derived for the cases of positive, zero and negative values of axial force parameter, q. Expressions for b_1 and b_2 are given in appendix for completeness and these coefficients are originally given by Oran who, however, considered only a straight element and b_{vs} and b_{vv} are listed in appendix.

Defining [F] and [u] as the basic nodal variables at two ends of an element as,

$$[F] = [\,M_{1z}\; M_{2z}\; M_{1y}\; M_{2y}\; M_t\; P\,]^T \tag{5.5. 30}$$

$$[u] = [\theta_{1z}\; \theta_{2z}\; \theta_{1y}\; \theta_{2y}\; \theta_t\; u\,]^T \tag{5.5. 31}$$

the tangent stiffness equation for the incremental forces and displacements can then be written as,

$$[\Delta F] = [L]^T(\{T\}^T[k_e][T] + [N])[L][\Delta u] \tag{5.5. 32}$$

in which [L] is the global to local transformation matrix, [T] is the transformation matrix relating the 6 local forces and moments to the 12 nodal forces and moments in local element coordinate system, [N] is the geometric matrix accounting for nodal translations and $[k_e]$ is the element tangent stiffness matrix in the local co-ordinate system given by,

$$[k_e] = \frac{EI}{L}\begin{bmatrix} \eta_z c_{1z} + \frac{G_{1z}^2}{\pi^2 H} & \eta_z c_{2z} + \frac{G_{1z}G_{2z}}{\pi^2 H} & \frac{G_{1z}G_{1y}}{\pi^2 H} & \frac{G_{1z}G_{2y}}{\pi^2 H} & 0 & \frac{G_{1z}}{LH} \\ \eta_z c_{2z} + \frac{G_{1z}G_{2z}}{\pi^2 H} & \eta_z c_{1z} + \frac{G_{2z}^2}{\pi^2 H} & \frac{G_{2z}G_{1y}}{\pi^2 H} & \frac{G_{2z}G_{2y}}{\pi^2 H} & 0 & \frac{G_{2z}}{LH} \\ \frac{G_{1y}G_{1z}}{\pi^2 H} & \frac{G_{1y}G_{2z}}{\pi^2 H} & \eta_y c_{1y} + \frac{G_{1y}^2}{\pi^2 H} & \eta_y c_{2y} + \frac{G_{1y}G_{2y}}{\pi^2 H} & 0 & \frac{G_{1y}}{LH} \\ \frac{G_{2y}G_{1z}}{\pi^2 H} & \frac{G_{2y}G_{2z}}{\pi^2 H} & \eta_y c_{2y} + \frac{G_{1y}G_{2y}}{\pi^2 H} & \eta_y c_{1y} + \frac{G_{2y}^2}{\pi^2 H} & 0 & \frac{G_{2y}}{LH} \\ 0 & 0 & 0 & 0 & \frac{GJ + Pr_0^2}{EI} & 0 \\ \frac{G_{1z}}{LH} & \frac{G_{2z}}{LH} & \frac{G_{1y}}{LH} & \frac{G_{2y}}{LH} & 0 & \frac{\pi^2}{L^2 H} \end{bmatrix} \tag{5.5. 33}$$

5.6 Shell Element

5.6.1 Triangular Shell

In NIDA, the triangular shell element is a flat type element which superimposes plate bending and membrane element stiffness, in conjunction with the Kirchhoff thin shell theory.

Membrane Stiffness

In NIDA, the constant strain triangular (CST) with drilling degree of freedom developed by Allman (1984) is used. The strain vector can be expressed in terms of the displacement in the element and then to the nodal displacements as,

$$[\varepsilon]=\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_{xy}\end{bmatrix}=\begin{bmatrix}\dfrac{\partial u}{\partial x}\\ \dfrac{\partial v}{\partial y}\\ \dfrac{\partial u}{\partial y}+\dfrac{\partial v}{\partial x}\end{bmatrix}=[B_m][U_m] \tag{5.6. 1}$$

in which $[u_m]=[u_1\ v_1\ \theta_{z1}\ u_2\ v_2\ \theta_{z2}\ u_3\ v_3\ \theta_{z3}]^T$ is the nine element nodal displacements and rotations (see Figure 5.16) and $[B_m]$ is the strain vs. displacement matrix given by,

$$[B_m]=\frac{1}{2A}\times$$

$$\begin{bmatrix} -y_3 & 0 & \dfrac{yy_3}{2} & y_3 & 0 & -\dfrac{yy_3}{2} & 0 & 0 & 0\\ 0 & x_3-x_2 & \dfrac{x(x_3-x_2)}{2} & 0 & -x_3 & \dfrac{(x_2-x)x_3}{2} & 0 & x_2 & -\dfrac{x_2(x_3-x)}{2}\\ x_3-x_2 & -y_3 & \dfrac{-xy_3-(x_3-x_2)y}{2} & -x_3 & y_3 & \dfrac{(x-x_2)y_3+x_3y}{2} & x_2 & 0 & \dfrac{x_2(y_3-y)}{2}\end{bmatrix} \tag{5.6. 2}$$

in which x_i and y_i are the coordinates of vertex i in local coordinate with x-axis passing through node 1 and 2.

The membrane element stiffness can then be obtained by a standard procedure as,

$$[K_m]=\int_A [B_m]^T[D_m]\ [B_m]\ tdA \tag{5.6. 3}$$

in which $[D_m]$ is the elasticity matrix, t is the element thickness and A is the element area. The elasticity matrix is defined as follows,

$$[D_m]=\frac{E}{1-\nu^2}\begin{bmatrix}1 & \nu & 0\\ \nu & 1 & 0\\ 0 & 0 & \dfrac{1-\nu}{2}\end{bmatrix} \tag{5.6. 4}$$

where ν is the Poisson's ratio and E is the Young's modulus of elasticity.

Bending Stiffness

Discrete Kirchhoff triangular (DKT) element proposed by Batoz et al. (1980) is used in NIDA. In the element, the Kirchhoff hypothesis is imposed on the element at its corners and mid-side nodes only. The bending stiffness of the element is based on the assumption of zero shear strain energy. The curvature $[k]$ of the plate can be derived in terms of the nodal displacement $[u_b]$ as,

$$[k]=\begin{bmatrix} \dfrac{\partial \beta_x}{\partial x} \\ \dfrac{\partial \beta_y}{\partial y} \\ \dfrac{\partial \beta_x}{\partial y}+\dfrac{\partial \beta_y}{\partial x} \end{bmatrix}=[B_b][u_b] \tag{5.6. 5}$$

in which $[u_b]=[w_1\ \theta_{x1}\ \theta_{y1}\ w_2\ \theta_{x2}\ \theta_{y2}\ w_3\ \theta_{x3}\ \theta_{y3}]$ is the nodal displacement degrees of freedom, as shown in Figure 5.16, β_x and β_y are the rotations of the normal to the undeformed middle surface in the x-z and y-z planes and $[B_b]$ is the strain vs. displacement matrix given by,

$$[B_b]=\frac{1}{2A}\begin{bmatrix} y_3[H_{x,\xi}] \\ -x_3[H_{y,\xi}]+x_2[H_{y,\eta}] \\ -x_3[H_{x,\xi}]+x_2[H_{x,\eta}]+y_3[H_{y,\xi}] \end{bmatrix} \tag{5.6. 6}$$

where ξ and η are the area coordinates; $[H_{x,\xi}]$ and $[H_{x,\eta}]$ are the derivatives of the interpolation function $[H_{x(\xi,\eta)}]$ with respect to ξ and η while $[H_{y,\xi}]$ and $[H_{y,\eta}]$ are the derivatives of the interpolation function $[H_{y(\xi,\eta)}]$ with respect to ξ and η .

Using the principle of total potential energy, the bending stiffness of the element can be derived as,

$$[K_b]=2A\int_0^1 [B_b]^T [D_b] \ [B_b]d\xi d\eta \tag{5.6. 7}$$

in which $[D_b]$ is elasticity matrix for plate bending and it can be expressed as follows.

$$[D_b]=\frac{Et^3}{12(1-\nu^2)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix} \tag{5.6. 8}$$

Geometric Stiffness Matrix

The geometric stiffness matrix is formed by the product of the initial stress and the quadratic strain tensor energy due to the motion of an element. The initial stresses are taken as the average of their respective element stresses calculated from the membrane stiffness. Mathematically the geometric stiffness matrix can be written as Zienkiewicz and Taylor (1989)

$$[K_G]=\int_v [G]^T \begin{bmatrix} T_x & T_{xy} \\ T_{xy} & T_y \end{bmatrix} [G]dv \tag{5.6. 9}$$

in which T_x , T_y and T_{xy} are the product of the average membrane stresses and the plate thickness.

The matrix $[G]$ is the derivative of the shape functions given by

$$[G]=\begin{bmatrix} \left(\dfrac{\partial H_x(\xi,\eta)}{\partial x}\right)^T \\ \left(\dfrac{\partial H_y(\xi,\eta)}{\partial y}\right)^T \end{bmatrix} \tag{5.6. 10}$$

Element Stiffness Matrix

The flat shell element is superposed by CST and DKT and the complete element stiffness can be obtained from Eqns (5.6.3) and (5.6.7) as,

$$[K_L]=[K_m]+[K_b] \tag{5.6. 11}$$

in which $[K_L]$ is the shell element composed of membrane and bending stiffness. The incremental tangent matrix can be written as below.

$$[\Delta K_e]=[\Delta K_L]+[\Delta K_G] \tag{5.6. 12}$$

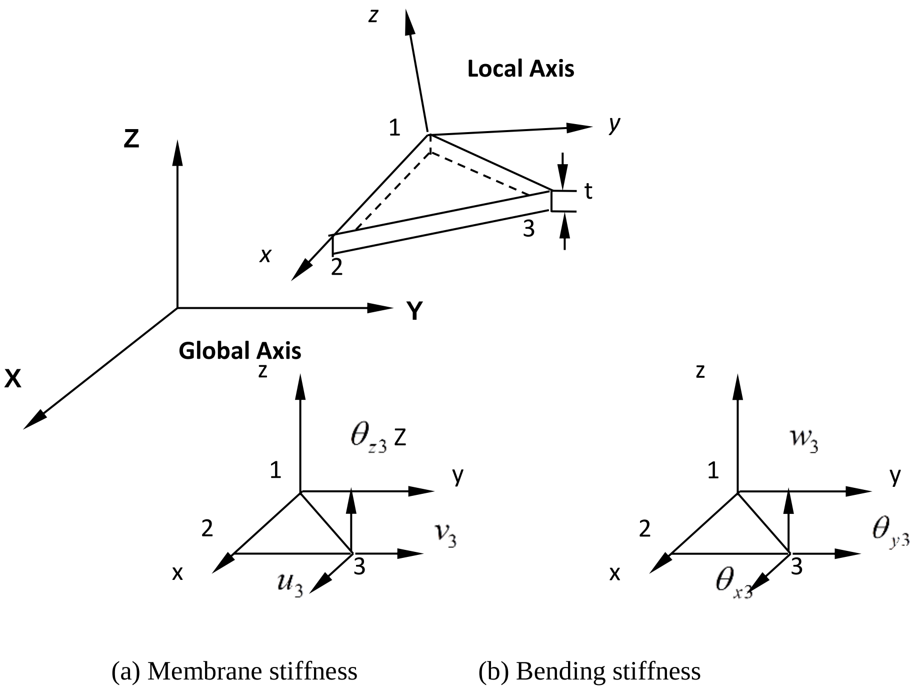


Figure 5.16 Shell Element (CST + DKT)

5.7 Response Spectrum Analysis

5.7.1 Response Spectra of SDOF Systems

Although a real structure may be very complicated and possess many degrees of freedom, the design spectra are essentially obtained from a single-degree-of-freedom (SDOF) system as shown in Figure 5.17. For this typical SDOF system, the equation of motion can be formulated by expressing the equilibrium of all forces acting on the system.

$$F(t)-F_I(t)-F_D(t)-F_S(t)=0 \tag{5.7. 1}$$

in which, $F(t)$ is time-varying force, $F_I(t)$ is inertial force, $F_D(t)$ is damping force, $F_S(t)$ is elastic restoring force. The governing equation of motion can be rewritten as

$$m\ddot{u}(t)+c\dot{u}(t)+ku(t)=-m\ddot{u}_g(t) \tag{5.7. 2}$$

where m is mass of the SDOF system produced inertial force, c is damping constant presented energy dissipation mechanism, k is linear stiffness of the spring provided elastic resistance.

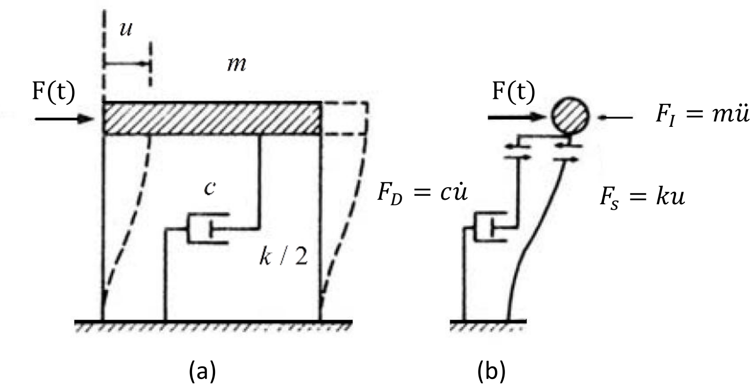


Figure 5.17 SDOF System under Horizontal Force

By defining the natural frequency and damping ratio as

$$\omega=\sqrt{\frac{k}{m}} \tag{5.7. 3}$$

$$\zeta=\frac{c}{2\sqrt{km}} \tag{5.7. 4}$$

The governing equation of a SDOF system subjected to ground acceleration $\ddot{u}_g(t)$ becomes

$$\ddot{u}(t)+2\omega\zeta\dot{u}(t)+\omega^2u(t)=-\ddot{u}_g(t) \tag{5.7. 5}$$

The Eq. (5) clearly shows that for a given $\ddot{u}_g(t)$ the displacement response $u(t)$ of the system depends only on its natural frequency ω (or period T) and damping ratio ζ. That is, the same response $u(t)$ will be produced for any two systems having the same values T and ζ under a specified earthquake. Thus, the response spectrum determined from a SDOF system can be applied to any structures though their stiffness and mass may be very different.

The evaluation of the dynamic response such as displacement, velocity, acceleration, internal force and stress at every time instant during an earthquake is usually conducted by numerical integration (e.g. Newmark's method, central difference method). For engineering application purpose, only the maximum absolute values of displacement, velocity and acceleration responses experienced by a structure are of interest as below:

$$\begin{cases} S_d = |u(t)|_{\max} = S_d(T,\zeta) \\ S_v = |\dot{u}(t)|_{\max} = S_v(T,\zeta) \\ S_a = |\ddot{u}(t)|_{\max} = S_a(T,\zeta) \end{cases} \tag{5.7. 6}$$

in which S_d is spectral displacement, S_v is spectral velocity, S_a is spectral acceleration.

In engineering practice, the following approximations are generally employed:

$$\begin{cases} S_{pv} = \omega S_d \cong S_v \\ S_{pa} = \omega^2 S_d \cong S_a \end{cases} \tag{5.7. 7}$$

Where S_{pv} is pseudo-spectral velocity, S_{pa} is pseudo-spectral acceleration. The prefix 'pseudo' indicates that such values do not correspond to the actual peak spectral velocity and acceleration. The above spectral relationships significantly expedite the construction of earthquake response spectra because only the spectral displacement S_d needs to be determined by numerical integration.

A plot of the peak value of a spectral ordinate (i.e. S_d, S_v or S_{pv}, S_a or S_{pa}) against the natural period T is called the response spectrum for that spectral ordinate. Each such plot is for SDOF systems having a fixed damping ratio ζ and therefore a family of plots will be produced to cover the range of damping values encountered in real structures.

5.7.2 Design Spectra

The response spectrum mentioned above is obtained from a specified past earthquake record. The shape of the response spectrum is normally very irregular for a given earthquake. However, certain similarities exist among the earthquake ground motions recorded under similar conditions. Thus, the design spectra can be derived from statistical analyses based on the response spectra obtained from earthquakes with common characteristics. The design spectra in seismic codes are usually presented as smooth curves and/or straight lines.

Acceleration response spectra are commonly implemented in seismic design codes such as GB50011, Eurocode 8 and UBC97 because they are related directly to the base shear used in the seismic design. For long period structures, the displacement response spectra become more and more important for seismic design.

It is convenient that the pseudo-spectral acceleration S_{pa} is normalized by gravity acceleration g (GB50011, UBC97) or design acceleration a_g (Eurocode 8) to a dimensionless quantity. In GB50011, this dimensionless quantity is called seismic influence coefficient defined as

$$\alpha = S_{pa} / g \tag{5.7. 8}$$

5.7.3 Modal Response Spectrum Analysis

Modal response spectrum analysis (MRSA) is used to find the maximum responses rather than the full responses during a likely earthquake. This method is performed using mode superposition.

For a multi-degree-of-freedom structure, the governing equation of motion of the structure undergone an earthquake is given as

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = -[M]\{\ddot{u}_g(t)\} \tag{5.7. 9}$$

where $[M]$ is mass matrix, $[C]$ is damping matrix, $[K]$ is stiffness matrix, and $\{\ddot{u}_g(t)\}$ is ground acceleration vector.

Making use of the modal decomposition method,

$$\{u(t)\} = [\Phi]\{q\} = \sum_{j=1}^{n} \{\phi\}_j q_j(t) \tag{5.7. 10}$$

the following modal equation can be obtained

$$\ddot{q}_j(t) + 2\omega_j \zeta_j \dot{q}_j(t) + \omega_j^2 q_j(t) = -\gamma_j \ddot{u}_g(t) \tag{5.7. 11}$$

in which ω_j is natural frequency, ζ_j is damping ratio, and

$$\gamma_j = \frac{\{\phi\}^T{}_j [m]\{1\}}{M_j} = \frac{\{\phi\}^T{}_j [m]\{1\}}{\{\phi\}^T{}_j [m]\{\phi\}_j} = \frac{\sum_{i=1}^{n} m_i \phi_{ji}}{\sum_{i=1}^{n} m_i \phi_{ji}^2} \tag{5.7. 12}$$

in which $\{\phi\}_j$ is the modal shape for jth mode which can be determined by modal analysis, and γ_j is the earthquake participation factor for jth mode.

The modal response $q_j(t)$ for jth mode can be obtained by the Duhamel integral expression

$$q_j(t) = -\frac{\gamma_j}{\omega_j} \int_0^t e^{-\zeta_j \omega_j (t-\tau)} \ddot{u}_g(\tau) \sin \omega_j (t-\tau) d\tau = \gamma_j \Delta_j(t) \tag{5.7. 13}$$

where

$$\Delta_j(t) = -\frac{1}{\omega_j} \int_0^t e^{-\zeta_j \omega_j (t-\tau)} \ddot{u}_g(\tau) \sin \omega_j (t-\tau) d\tau \tag{5.7. 14}$$

Once the modal response $q_j(t)$ is obtained, the displacement and acceleration responses for ith DOF in physical coordinates can be determined by

$$\begin{cases} u_i(t) = \sum_{j=1}^{n} q_j(t)\phi_{ji} = \sum_{j=1}^{n} \gamma_j \Delta_j(t)\phi_{ji} \\ \ddot{u}_i(t) = \sum_{j=1}^{n} \ddot{q}_j(t)\phi_{ji} = \sum_{j=1}^{n} \gamma_j \ddot{\Delta}_j(t)\phi_{ji} \end{cases} \tag{5.7. 15}$$

The inertia force on ith DOF is expressed as

$$F_i(t) = m_i[\ddot{u}_i(t) + \ddot{u}_g(t)] \tag{5.7. 16}$$

Using the relationship $\sum_{j=1}^{n} \gamma_j \phi_{ji} = 1$, we have

$$\ddot{u}_g(t) = \sum_{j=1}^{n} \gamma_j \phi_{ji} \ddot{u}_g(t) \tag{5.7. 17}$$

Substituting Eqs. (5.7.15) and (5.7.17) into Eq. (5.7.16), the inertia force on ith DOF for jth mode can be written as

$$F_{ji}(t) = m_i \gamma_j \phi_{ji}[\ddot{\Delta}_j(t) + \ddot{u}_g(t)] \tag{5.7. 18}$$

For engineering applications, we only need to know the maximum inertia force.

$$F_{ji} = \alpha_j \gamma_j \phi_{ji} G_i \tag{5.7. 19}$$

where

$$\alpha_j = |\ddot{\Delta}_j(t) + \ddot{u}_g(t)|_{\max} \tag{5.7. 20}$$

$$G_i = m_i g \tag{5.7. 21}$$

The displacement response of ith DOF for jth mode can be calculated as

$$u_{ji} = \frac{1}{\omega_j^2} \alpha_j \gamma_j \phi_{ji} \tag{5.7. 22}$$

Thus, the other responses such as internal force and stress can be further determined from displacement response.

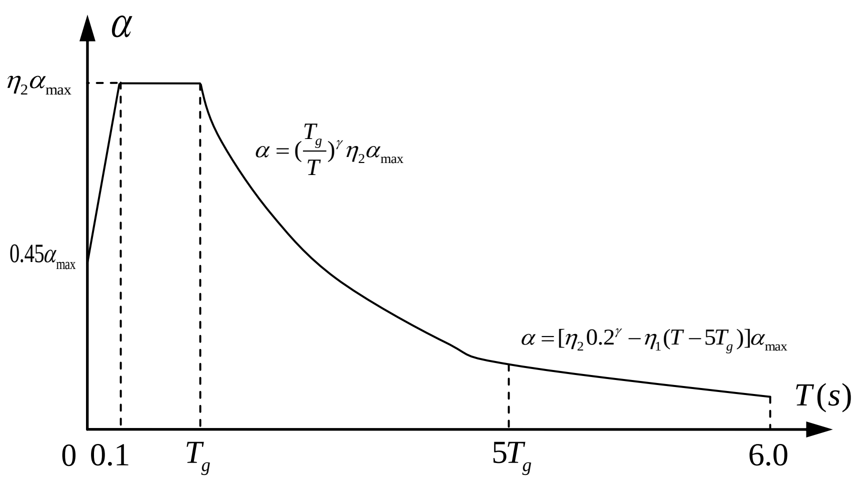


Figure 5.18 Seismic Influence Coefficient (GB50011)

Although there are many seismic design codes in the world, the difference between them when using MRSA mainly lies in the difference of determination of the coefficient α_j. Eq. (5.7.8) shows the definition of the coefficient α_j. Figure 5.18 shows the curve for determination of α_j specified in GB50011.

5.7.4 Participating Mass Ratio

From above, it can be seen that the response spectrum analysis is based on the modal analysis which is used to find the natural frequencies and the vibration modes of the structures. When performing a modal analysis for a structure with large degrees of freedom, it only need to consider a relatively small number of modes p in the response calculations, such that p << n (n is the number of all modes). Otherwise, it needs huge computer time to calculate all vibration modes. Thus, the displacement response in Eq. (5.7.15) can be approximated as

$$\hat{u}_i(t) = \sum_{j=1}^{p} q_j(t)\phi_{ji} = \sum_{j=1}^{p} \gamma_j \Delta_j(t)\phi_{ji} \quad (p \ll n) \tag{5.7. 23}$$

in which $\hat{u}_i(t)$ represents the truncated response for p < n.

However, the number p cannot be too small though small value of p means time saving. Otherwise, some important higher-mode effects will be ignored and, consequently the output may be not accurate enough. For this, the effective mass concept is widely used to determine the number of modes p to be input in the modal analysis. The effective mass for jth mode, M_{ej}, is defined as

$$M_{ej} = \{\phi\}_j^T [m]\{I\}\gamma_j = \frac{\left(\sum_{i=1}^{n} m_i \phi_{ji}\right)^2}{\sum_{i=1}^{n} m_i \phi_{ji}^2} \tag{5.7. 24}$$

The sum of the effective masses for all modes is equal to the total mass M of the structure, i.e.

$$M = \sum_{j=1}^{n} M_{ej} \tag{5.7. 25}$$

This leads to a means for determining the number of truncated modes necessary to accurately represent the structure response. If the structural response is calculated from the truncated modes, the ratio calculated by the sum of the effective masses from these modes over the total mass M should be not less than a predefined percentage. This ratio is called participating mass ratio which is given by

$$P_M = \sum_{j=1}^{p} M_{ej} / M \tag{5.7. 26}$$

Many seismic design codes specify that at least 90% of the participating mass of the structure must be included in the response spectrum analysis.

5.7.5 Combination of Modal Responses

Noted that the maximum responses for different modes do not occur simultaneously and therefore the maximum structural response cannot be obtained by taking the sum of the maximum modal responses. Two methods have been widely used for modal combination, i.e. the square-root-of-the-sum-of-the-squares (SRSS) method, and the complete-quadratic-combination (CQC) method.

(1) SRSS method

The SRSS method is usually applied for calculating the maximum response for two-dimensional systems exhibiting well-separated modes. In this method,

$$S_{EK} = \sqrt{\sum_{j=1}^{p} S_j^2} \tag{5.7. 27}$$

in which, S_{EK} is the structural response such as internal force and displacement considering all selected modes, and S_j is the structural response for jth mode.

(2) CQC method

The CQC method is usually applied for calculating the maximum response for three-dimensional systems and/or systems with closely spaced modes. In this method,

$$S_{EK} = \sqrt{\sum_{j=1}^{p}\sum_{k=1}^{p} \rho_{jk} S_j S_k} \tag{5.7. 28}$$

in which, ρ_{jk} is the correlation coefficient for jth mode and kth mode. When using constant damping ratio ζ , this coefficient is calculated as

$$\rho_{jk} = \frac{8\zeta^2(1+r)r^{3/2}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2} \tag{5.7. 29}$$

where $r = {}^{\omega_j}/_{\omega_k}$ and must be not greater than 1.0.

5.7.6 Combination of the Effects of the Components of the Seismic Action

In real world, an earthquake may come from any directions such that a designed structure should be capable of resisting earthquake shaking from all possible directions. Generally, two horizontal components and one vertical component of seismic action should be considered to act simultaneously on a spatial structure. There are denoted here as S_{EX} and S_{EY} for the structural responses of the two horizontal components and S_{EZ} for the vertical. Since the peak value of the seismic action effects do not occur simultaneously, a combination rule is required to produce reasonable results. Two approaches have been widely used for directional combination, i.e. the square-root-of-the-sum-of-the-squares (SRSS) method, and the absolute-sum (ABS) method.

(1) SRSS method

The SRSS method for directional combination assumes the components are independent of each other. In this method,

$$S_{EK} = \sqrt{S_{EX}^2 + S_{EY}^2 + S_{EZ}^2} \tag{5.7. 30}$$

(2) ABS method

The ABS method assumes that when the maximum response from one component occurs, the responses from the other two components are taken part of their maximum. In this method,

$$S_{EK} = \max(\bar{S}_1, \bar{S}_2, \bar{S}_3) \tag{5.7. 31}$$

where

$$\begin{cases} \bar{S}_1 = S_{EX} + \lambda(S_{EY} + S_{EZ}) \\ \bar{S}_2 = S_{EY} + \lambda(S_{EZ} + S_{EX}) \\ \bar{S}_3 = S_{EZ} + \lambda(S_{EX} + S_{EY}) \end{cases} \tag{5.7. 32}$$

The value λ in Eq. (5.7.32) is usually taken as 0.3 in seismic design codes.

5.8 Time History Analysis

5.8.1 Direct Integration for Equation of Motion

The incremental form of the equation of motion Eq. (5.7.8) can be written as.

$$[M]\{\Delta\ddot{u}\}+[C]\{\Delta\dot{u}\}+[K]\{\Delta u\}=\{\Delta F\} \tag{5.8. 1}$$

in which $\{\Delta F\}$ is equal to $-[M]\{\Delta\ddot{u}_g\}$. For simplicity, the "(t)" in acceleration $\ddot{u}(\mathrm{t})$, velocity $\dot{u}(\mathrm{t})$ and displacement $u(\mathrm{t})$ will be omitted hereafter.

Noted that the damping matrix $[C]$ is usually employed the Rayleigh damping model which is given as

$$[C]=a[M]+b[K] \tag{5.8. 2}$$

in which a is mass proportional coefficient, and b is stiffness proportional coefficient. The two coefficients can be calculated by

$$\begin{cases} a=\dfrac{4\pi(\zeta_1 T_1-\zeta_2 T_2)}{(T_1^2-T_2^2)} \\ b=\dfrac{T_1T_2(\zeta_2 T_1-\zeta_1 T_2)}{\pi(T_1^2-T_2^2)} \end{cases} \tag{5.8. 3}$$

in which T_1 and T_2 are the first and second natural periods of the structure respectively, and ζ_1 and ζ_2 are the damping ratios corresponding to T_1 and T_2 respectively.

MRSA solves the dynamic equilibrium equation by mode superposition approach while THA widely adopts numerical integration method. In NIDA [1], the Newmark method is utilized for step-by-step solution of Eq. (5.8.2).

Newmark truncated the Taylor's series for displacement $\{u\}$and velocity $\{\dot{u}\}$ and finally expressed them as,

$$\{^{t+\Delta t}\dot{u}\}=\{^{t}\dot{u}\}+(1-\gamma)\Delta t\{^{t}\ddot{u}\}+\gamma\Delta t\{^{t+\Delta t}\ddot{u}\} \tag{5.8. 4}$$

$$\{^{t+\Delta t}u\}=\{^{t}u\}+\Delta t\{^{t}\dot{u}\}+(0.5-\beta)(\Delta t)^2\{^{t}\ddot{u}\}+\beta(\Delta t)^2\{^{t+\Delta t}\ddot{u}\} \tag{5.8. 5}$$

where $\{^{t}u\}$, $\{^{t}\dot{u}\}$ and $\{^{t}\ddot{u}\}$ are the total displacement, velocity and acceleration vectors at time t, and Δt is time increment. The parameters γ and β define the variation of acceleration over a time step and determine the stability and accuracy characteristics of the method. Typically, $\gamma=0.5$ and $1/6\le\beta\le 1/4$ can provide stable results.

By using Eqs. (5.8.4) and (5.8.5), the equation of motion Eq. (5.8.1) can be finally written as,

$$[K_{eff}]\{^{t}\Delta u\}=[\Delta F_{eff}] \tag{5.8. 6}$$

in which

$$[K_{eff}]=c_1[M]+c_4[C]+[K] \tag{5.8. 7}$$

$$[F_{eff}]=\{^t\Delta F\}-(c_2[M]+c_5[C])\{^t\dot{u}\}-(c_3[M]+c_6[C])\{^t\ddot{u}\} \tag{5.8. 8}$$

with

$$\begin{cases} c_1=\dfrac{1}{\beta(\Delta t)^2};\ c_2=-\dfrac{1}{\beta\Delta t};\ c_3=-\dfrac{1}{2\beta} \\ c_4=\dfrac{\gamma}{\beta\Delta t};\quad c_5=-\dfrac{\gamma}{\beta};\quad c_6=-(\dfrac{\gamma}{2\beta}-1)\Delta t \end{cases} \tag{5.8. 9}$$

After obtaining $\{^t\Delta u\}$ from Eq. (5.8.6), the incremental velocity $\{^t\Delta\dot{u}\}$ and acceleration $\{^t\Delta\ddot{u}\}$ can be calculated by

$$\{^t\Delta\dot{u}\}=c_4\{^t\Delta u\}+c_5\{^t\dot{u}\}+c_6\{^t\ddot{u}\} \tag{5.8. 10}$$

$$\{^t\Delta\ddot{u}\}=c_1\{^t\Delta u\}+c_2\{^t\dot{u}\}+c_3\{^t\ddot{u}\} \tag{5.8. 11}$$

Further, the total vectors for next time step are updated as

$$\begin{cases} \{^{t+\Delta t}u\}=\{^t u\}+\{^t\Delta u\} \\ \{^{t+\Delta t}\dot{u}\}=\{^t\dot{u}\}+\{^t\Delta\dot{u}\} \\ \{^{t+\Delta t}\ddot{u}\}=\{^t\ddot{u}\}+\{^t\Delta\ddot{u}\} \\ \{^{t+\Delta t}F\}=\{^t F\}+\{^t\Delta F\} \end{cases} \tag{5.8. 12}$$

For nonlinear dynamic analysis, iterations for solving Eq. (5.8.6) are needed for correction of equilibrium error. To check the equilibrium, both the displacement and force norms are recommended, i.e.

$$\frac{\{^t\Delta u\}_i^T\{^t\Delta u\}_i}{\{^{t+\Delta t}u\}_i^T\{^{t+\Delta t}u\}_i}<\text{TOLERANCE} \tag{5.8. 13}$$

$$\frac{\{^t\Delta F^*\}_i^T\{^t\Delta F^*\}_i}{\{^{t+\Delta t}F\}_i^T\{^{t+\Delta t}F\}_i}<\text{TOLERANCE} \tag{5.8. 14}$$

in which the subscript "i" is the number of iterations within a time step, and $\{^t\Delta F^*\}$ is the unbalanced residual force increment vector determined by

$$\{^t\Delta F^*\}=\{^{t+\Delta t}F\}-([M]\{^{t+\Delta t}\ddot{u}\}+[C]\{^{t+\Delta t}\dot{u}\}+\{^{t+\Delta t}R\}) \tag{5.8. 15}$$

where $\{^{t+\Delta t}R\}$ is the resisting force of the complete structure.

Once the conditions given in Eqs. (5.8.13) and (5.8.14) are satisfied, the procedure presented in Eqs. (5.8.6-5.8.14) is repeated for next time step until the target time steps reach or the structure is collapsed.

5.8.2 Selection of Earthquake Wave

It should be pointed out that the artificial/recorded/simulated waves of ground motion selected for a time history analysis may significantly affect the outcome. Therefore, seismic design codes explicitly or implicitly specify some requirements for selecting earthquake waves when performing a nonlinear dynamic analysis.

The theoretical background for selection of earthquake wave is generally based on the three characteristics of ground motion, i.e. peak ground motion, time duration and frequency content. Peak ground motion, primarily peak ground acceleration (PGA), influences the vibration amplitude and has been commonly employed to scale earthquake design spectra and acceleration time histories. Time duration of ground motion affects the severity of ground shaking. For example, an earthquake with a high PGA poses a high hazard potential, but if it is sustained for only a short period of time it is unlikely to inflict significant damage to many types of structures. On the contrary, an earthquake with a moderate PGA and a long duration can build up damaging motions in certain types of structures. When the frequency content of the ground motion is close to the natural frequencies of the structure, the resonant phenomenon, in which the vibration amplitude of the structure grows indefinitely in theory, will occur.

From above, the general rules for selection of earthquake waves in GB50011 are listed as below.

(1) Minimum Time Duration

The duration of the input wave should be long enough, which is generally taken as not less than 5 to 10 times of the fundamental period of the structure.

(2) Minimum Number of Waves

GB50011 specifies that at least 2 sets of recorded strong earthquake waves and 1 set of artificial wave, based on the seismic intensity, design seismic group and site classification, should be employed.

(3) Minimum Base Shear

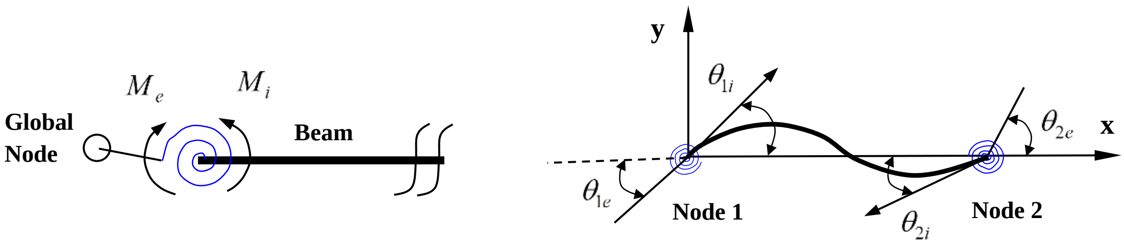
The seismic action represented by the input waves should conform, on average, to the 5% damping elastic response spectrum so that the waves used may have the statistical meaning to some extent. GB50011 states that when performing an elastic time history analysis, the base shear obtained from each wave shall not be less than 65% of that from the response spectrum method, and the average value from all waves shall not be less than 80% of that from the response spectrum method.

5.9 Semi-Rigid Connection

A semi-rigid connection must have the required strength, stiffness and ductility. Some typical moment versus rotation (M-θ) curves and various analytical and mathematical models representing the M-θ relationships can be found in previous chapters. Conveniently, a joint can be considered in an analysis as dimensionless with location at the intersection of the element centre lines. Further, a rotational spring element satisfying the M-θ relationship is inserted into each end of beam element to model the connection behaviour. The joint equilibrium condition can be expressed as,

$$M_e + M_i = 0 \tag{5.9. 1}$$

in which M_e and M_i are the moments at the two ends of a connection (see Figure 5.19a). The corresponding external node is connected to the global node and the internal node is joined to the beam element.



(a) Equilibrium at a joint (b) External and internal rotations

Figure 5.19 Modeling of Semi-Rigid Jointed Member

The stiffness of the connection, S, can be related to relative rotations at the two ends of the connection spring as,

$$S = \frac{M_e}{(\theta_e - \theta_i)} = \frac{M_i}{(\theta_i - \theta_e)} \tag{5.9. 2}$$

where θ_e and θ_i are the conjugate rotations for the moments M_e and M_i (see Figure 5.19b). Rewriting Equation (2) in matrix form, the stiffness matrix of a connection spring can be written as,

$$\begin{bmatrix} S & -S \\ -S & S \end{bmatrix} \begin{bmatrix} \theta_e \\ \theta_i \end{bmatrix} = \begin{bmatrix} M_e \\ M_i \end{bmatrix} \tag{5.9. 3}$$

A typical beam element bending stiffness matrix can be expressed as,

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{bmatrix} \theta_{1i} \\ \theta_{2i} \end{bmatrix} = \begin{bmatrix} M_{1i} \\ M_{2i} \end{bmatrix} \tag{5.9. 4}$$

in which k_{ij} is the stiffness coefficients of a prismatic beam. Here, the imperfect PEP element proposed by Chan and Zhou (1995) is adopted and more details can be referred to the original reference. Therefore, a hybrid element can be obtained by directly adding the two ends connection stiffness to the PEP element bending stiffness matrix as,

$$\begin{bmatrix} S_1 & -S_1 & 0 & 0 \\ -S_1 & k_{11}+S_1 & k_{12} & 0 \\ 0 & k_{21} & k_{22}+S_2 & -S_2 \\ 0 & 0 & -S_2 & S_2 \end{bmatrix} \begin{bmatrix} \theta_{1e} \\ \theta_{1i} \\ \theta_{2i} \\ \theta_{2e} \end{bmatrix} = \begin{bmatrix} M_{1e} \\ M_{1i} \\ M_{2i} \\ M_{2e} \end{bmatrix} \tag{5.9. 5}$$

in which the first subscript refers to node 1 or node 2. The internal degrees of freedom of the stiffness expression can be eliminated by a standard static condensation procedure. The stiffness expression of a beam element with both ends connected to a pair of springs can be finally written as,

$$[k_e][\theta_e] = [M_e] \tag{5.9. 6}$$

in which,

$$[\theta_e] = [\theta_{1e} \quad \theta_{2e}]^{\mathrm{T}}, \quad [M_e] = [M_{1e} \quad M_{2e}]^{\mathrm{T}} \tag{5.9. 7}$$

$$[k_e] = \frac{1}{\beta} \begin{bmatrix} S_1\beta - S_1^2(S_2+k_{22}) & k_{12}S_1S_2 \\ k_{21}S_1S_2 & S_2\beta - S_2^2(S_1+k_{11}) \end{bmatrix} \tag{5.9. 8}$$

and β is given by,

$$\beta = \det \begin{vmatrix} k_{11}+S_1 & k_{12} \\ k_{21} & k_{22}+S_2 \end{vmatrix} \tag{5.9. 9}$$

where S_1 and S_2 are the connection stiffness at nodes 1 and 2, respectively. When the spring stiffness S_i is zero, it means that the corresponding end is pinned end; when the spring stiffness S_i is infinite, it means that the corresponding end is rigid end. For semi-rigid case, the spring stiffness S_i can be determined by the given M-θ function.

5.10 Plastic Hinge Method

In the process of time history analysis by NIDA, a simple, accurate and efficient method for determining the plastic hinge(s) is used to capture the progressive strength and stiffness degradation of the structure under an earthquake attack.

The basis of the plastic hinge method is cross-section plastification. Material yielding is accounted for by zero-length plastic hinges at one or both ends of each element. Plasticity is assumed to be lumped only at the ends of an element, while the portion within the element is assumed to remain elastic throughout the analysis.

In NIDA, two predefined section springs (see Figure 5.20), which are used to simulate plastic hinge, will be set at the two ends of each beam-column element. The end section springs will be finally formulated into the element stiffness matrix of the curved stability function beam-column element (Chan and Gu 2000) which has been widely used for second-order P-Δ-δ analysis. More details about the plastic hinge method can be referred to Chan and Chui (2000).

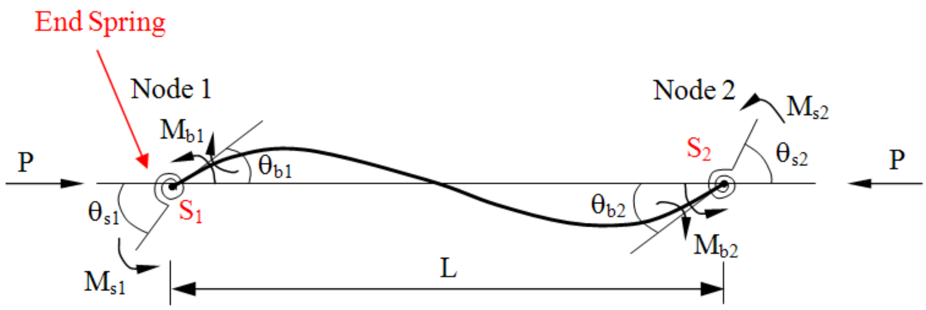


Figure 5.20 Internal Forces of the Curved Element with End Springs

The hysteresis model for steel material used in NIDA is shown in Figure 5.21. As illustrated in Figure 5.21, initial yielding occurs at point A when the first yield moment capacity Mei is attained. On the curve AB, the gradual yielding occurs and the plastic moment capacity Mp is reached point B. When unloading takes place at point B, gradual yielding characteristics disappears and the path follows the line BDC in which the moment at point C is less than the initial yield moment Mei at point D. On reloading, the path moves along the line CD under the perfectly elastic state and then follows the curve DE under the partial yielding state. Similarly, under unloading conditions at point E, the path moves along EFG'H.

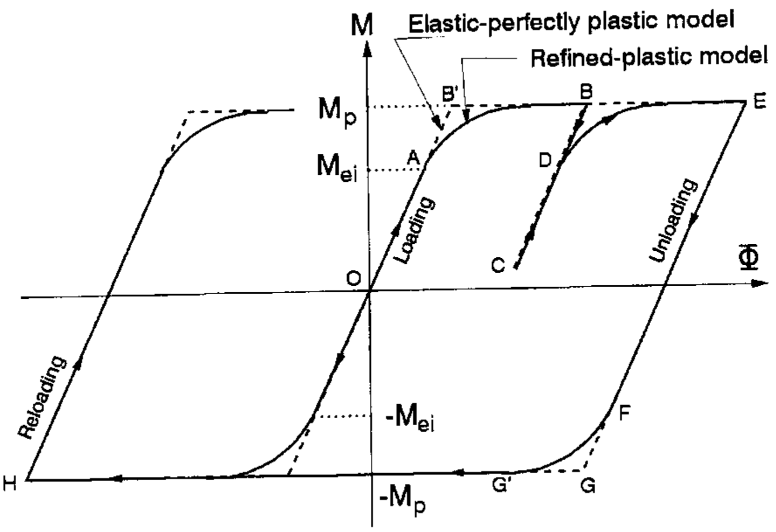


Figure 5.21 Elastic-Perfectly Plastic & Refined-Plastic Models

5.11 Load and Construction Sequences

Sequential loads and segmental structural components are considered by assigning design load cases and structural elements to different load stages and they are applied by the load stage number sequentially, see Figure 5.22. Applying a set of loads in the first load stage and keeping them constant in the second load stage with a set of additional loads allow us to simulate load sequence.

The strength, stability and deflection checks in the conventional design are essentially based on the complete structure without consideration of construction process. In fact, the behaviour of the components or units in the erection process may be different from the ideal case because instability and excessive deflection may occur in the construction stage with limited propping. Furthermore, shortening and undesirable deformations of the incomplete structure under self-weight and construction loads are inevitable. Generally speaking, the structural self-weight, external loads, boundary conditions and materials are depended on stages during the construction process and their variations are easily overlooked in conventional design.

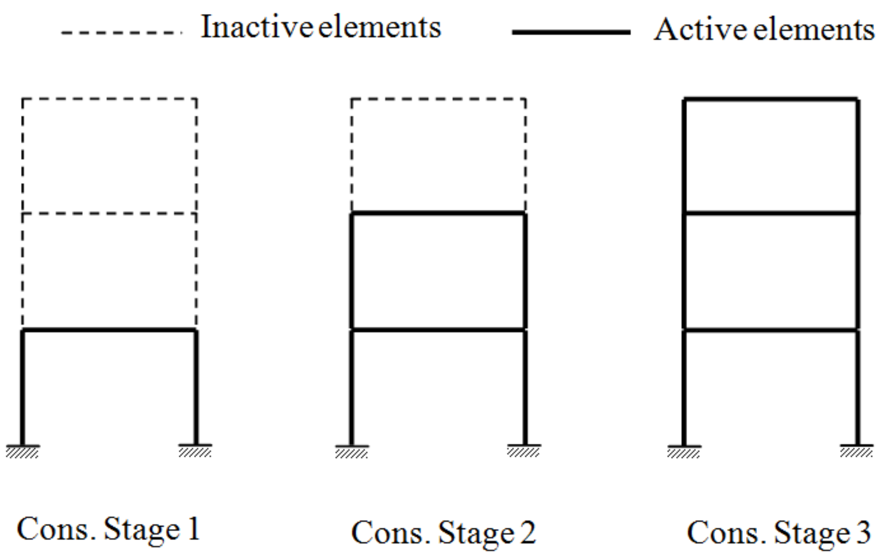


Figure 5.22 Construction Sequence Analysis

The deformation and stress level during construction are key considerations by structural engineers.

Unlike previous work using the first-order linear analysis which treats the limit states design and the construction analysis as two separate procedures with analysis by computers and design by charts, the proposed method integrates analysis to design and therefore the design and analysis models are consistent. Moreover, the P-Δ and P-δ effects, initial imperfections, pre-deformation and material yielding are all taken into account. The flow chart of the proposed procedure is shown in Figure 5.23.

The steps in the flowchart in Figure 5.23 are explained as follows.

(1) Define Construction Sequence

Engineers group structural elements in the design model into a number of construction units according to the construction scheme and assign each of them a unique "Construction ID". Only the elements in the active groups are considered in analysis at a particular loading stage with other non-active structural elements ignored. New elements are added to the structure at initial stage.

(2) Apply Dead Loads (DL) and Construction Loads (CL)

Dead loads are mainly from the structural self-weight and a load factor of 1.2 is suggested. Construction loads include the live loads, self-weight of equipments and others.

(3) Pre-set Deformation

Excessive deflections can be reduced by pre-cambering through pre-set displacements.

(4) Construction Analysis

Stiffness matrix of the members in active groups is solved with the nonlinear incremental-iterative procedure adopted to improve convergence.

(5) Strength and Instability Check

With the consideration of initial imperfections and P-Δ and P-δ effects, a fast and accurate instability and strength checks can be conducted by the section capacity check in Eq. (9). Modification is needed when some members fail.

(6) Deflection Check

As checking for serviceability limit state, this step ensures deflection limit is not violated otherwise member re-sizing or pre-cambering is activated.

(7) Permanent Load Design Check

The conventional ultimate and serviceability limit state design load checks are carried out as the final safety and serviceability checks. The consideration of load sequence may affect the final force and moment distribution as some construction loads are present earlier than some members which do not share these construction loads.

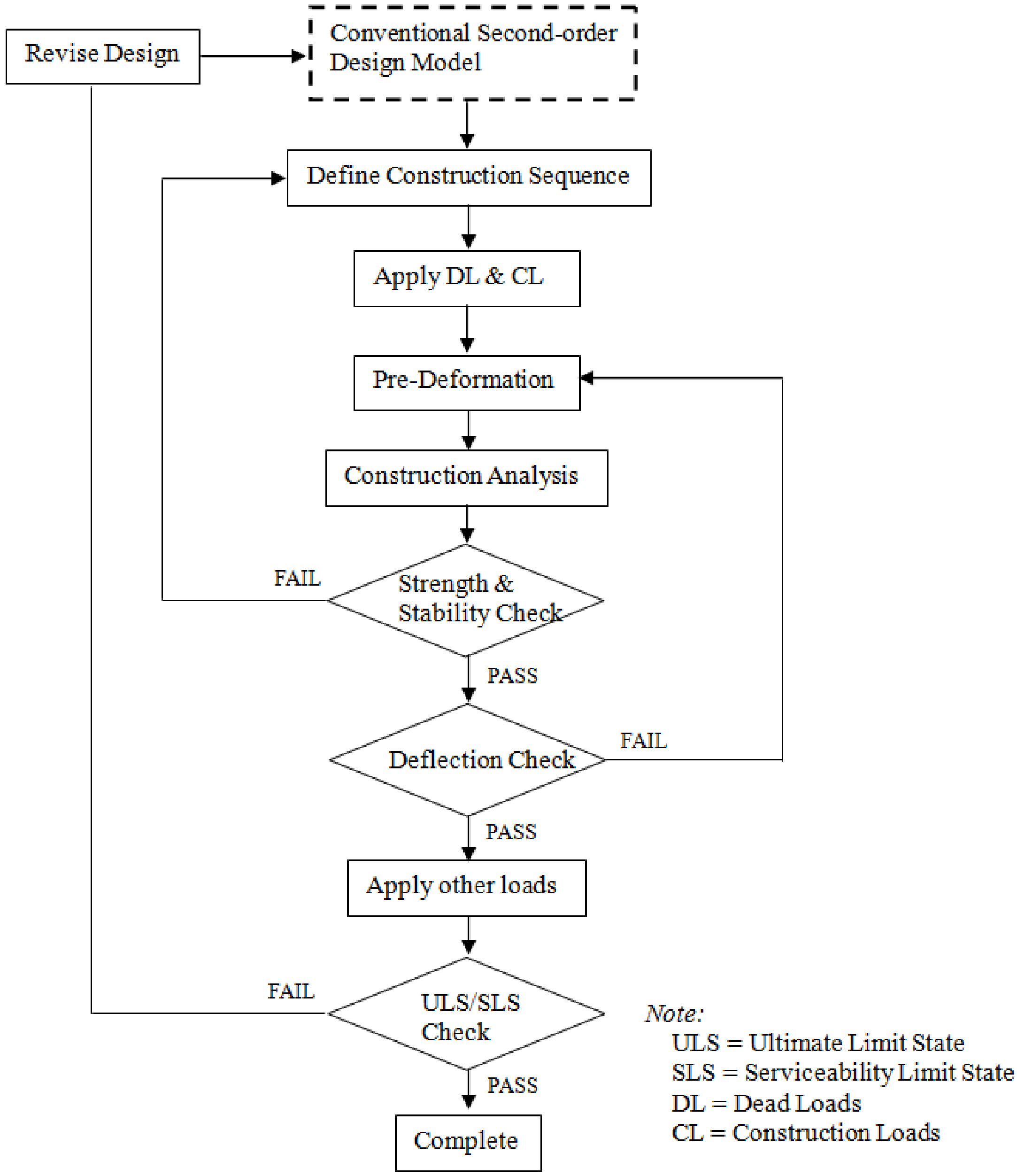


Figure 5.23 Flow Chart of the Integrated Design and Construction Procedure

6. EXPLANATION OF INPUT FILE

6.1 Comment and General Rules

- Using "!" for comment which will be ignored when reading the file.
- All names such as material, section, load case should be put into "< >".

6.2 Project Title and Information

To record the project title and other project information and will be output in *.out file.

***TITLE**

...

Noted that more than one lines is allowed, but should not include the KEYWORDs.

6.3 Version

To verify whether the data file is supported by the current version of NIDA or not.

***VERSION** [Number]

6.4 Active Degree of Freedom

To active some of the degrees of freedom for analysis. User can set 2D problem such as XY plan here

***ACTIVEDOF** [Number]

[Number]

(1)0 or 1 only;

(2)0 = inactive; 1= active

(3)Combine with maximum 6 digitals, for example, 111111

6.5 Unit System

To define the units used in the project.

***UNIT** [Force Unit] [Length Unit]

Item	Unit
Force	N, kN, kgf, tonf

Length	mm, cm, m
Rotation	DEGREE, RAD
Temperature	^{o}C
Time	s (second)

6.6 Numbering Optimization

Active or inactive the nodal numbering optimization for reducing memory used in analysis.

***KMIN** [Number]

[Number]

= 0 Inactive nodal numbering optimization for reducing memory used;

= 1 Auto active this function (default);

= 2 Always active this function.

6.7 Tolerance

Set the tolerances to change the accuracy of analysis results.

***TOLERANCE** [Number1] [Number2]

[Number1] : For nonlinear iteration, Default Value = 1e-3

[Number2] : For subspace iteration, Default Value = 1e-7

6.8 Design Code Used

To define the design codes for design purpose.

***DESIGN** [Steel Code] {[Concrete Code] [Composite Code]}

[Steel Code] : Steel design code used

Number	Design Code
1~10	HKSC (2011)
11~20	BS5950 (2000)
21~30	Eurocode3 (2005)
31~40	GB50017 (2003)
41~50	AISC-LRFD (2010)
> 50	Undefined

[Concrete Code] : Concrete design code used

Number	Design Code

1~10	HKCC (2004, 2nd)
11~20	BS8110 (1997)
21~30	Eurocode2 (2004)
31~40	GB 50010 (2002)
> 40	Undefined

[Composite Code] : Composite design code used

Number	Design Code
1~10	HKSC (2005)
11~20	BS5950 (2000)
21~30	Eurocode4 (2004)
31~40	GBJ138 (2001)
> 40	Undefined

6.9 Gravity Direction

To define the direction of gravity. It is important to define the gravity direction as many parameters are related to it. For modal analysis or time history analysis, user may need to add additional mass from load cases (in the gravity direction) into the structural system.

***GRAVITY** [Number]

[Number] : Direction of gravity

= +/-1, Global axis +/-X;

= +/-2, Global axis +/-Y;

= +/-3, Global axis +/-Z;

6.10 Analysis Case

To define analysis case and analysis parameters.

***ANTY**

SOLU [NO.][TYPE][IF DESIGN /dummy] [IF RUN:1=RUN,0=NOT RUN] <NAME>

[TYPE]

Number	Analysis Case Type
1~10	Linear Analysis
11~20	Nonlinear analysis
21~30	Modal analysis
31~40	Eigen-buckling analysis
41~50	Response spectrum analysis
51~60	Time history analysis
>60	Undefined

6.10.1 Linear Analysis

SOLU [NO.][TYPE=1~5][IF DESIGN][IF RUN:1=RUN,0=NOT RUN] <NAME>

LINP [Method_AmpMoment=0][Elastic Buckling Factor = 0]

USE [1=LOADCASE] [NO.] [FACTOR]

USE [2=COMBLOAD] [NO.] [FACTOR]

Note: Method_AmpMoment : Method for Moment Amplification (Linear Design only)

(1) HKSC (2005/2011)

= 0 , No moment amplification;

= 1 , $\dfrac{\lambda_{cr}}{\lambda_{cr}-1}$;

= 2 , $\dfrac{1}{1-\dfrac{F_c L_E^2}{\pi^2 EI}}$; (Default)

= 3 , Larger of 1 and 2;

(2) GB50017 (2003)

= 0 , No moment amplification;

= 1 , N.A.;

= 2 , $\dfrac{1}{1-0.8\dfrac{N}{N'_E}}$, where $N'_E = \pi^2 EA/(1.1\lambda^2)$; (Default)

= 3 , N.A.;

6.10.2 Nonlinear Analysis

SOLU [NO.][TYPE=11~20,11/13=PEP,12/14=STABILITY FUNCTION, 13~14 BEAM BUCKING] [IF DESIGN] [IF RUN:1=RUN,0=NOT RUN] <NAME>

NONP […]

Note: All data should be in one line.

No.	Parameter	Description
1	MYIELD	0=Elastic; 1=Plastic Element; 2=Plastic Hinge
2	Force Recovery Scheme	1= Total Secant Stiffness Method; 2= Incremental Tangent Stiffness Method
3	CYCLES	Total Load Cycles, 1
4	ITERATIONS	Number of Iterations for Each Load Cycle, 1
5	INC LOADFACTOR	Incremental Load Factor, 1.0
6	NUMERICAL METHOD	Numerical Method: 11=NR, 22=Single Disp., 43=ArcLen+MRD

7	ITER-PARAMETER	Parameter for Numerical Method, 4
8	INC-PARAMETER	Parameter for Numerical Method, 5
9	TANGENTS	Number of Iterations for Tangent Stiffness Matrix, 1
10	IMPERFECTION DIRECTION	+/-1, DIMP; +/-2, EIMP; +/-3, NIMP 0, No Imperfection
11	MIN. MEMBER IMPERFECTION	Minimum Member Imperfection, 1/1000.
12	Stiffness Reduction Factor	0.8~1.0 (AISC-LRFD 2010)
13	Target Load Factor	Target Load Factor, 1.0 (-1=Disable this function)

*IMPERFECTION DIRECTION: To determine **GLOBAL** and **LOCAL** imperfection*

(a) IMPERFECTION DIRECTION=+/-1: "Displacement", using deformed shape of applied forces as global imperfection;

DIMP [MAGNITUDE OF GLOBAL IMPERFECTION, L]

(b) IMPERFECTION DIRECTION=+/-2: "Eigen-buckling mode", using specified buckling mode as global imperfection, need the following parameters:

EIMP [MAGNITUDE OF GLOBAL IMPERFECTION, L][>0, Directly Calc. MODES,5] [iTH MODE USED,1]

(c) IMPERFECTION DIRECTION=+/-3: "Notional Forces", using notional horizontal forces as global imperfection, need the following parameters:

NIMP [Angle about vertical axis (HORIZONTAL Direction), 0, DEGREE][% of Gravity Load]

Note: For all methods, "+ ve" considers member imperfections in both principal axis while "-ve" considers only one principal axis which is adequate for most structures.

NUMERICAL METHOD (3 given schemes)

[NUMERICAL METHOD]

No.	Description
1	NEWTON-RAPHSON
2	DISPLACEMENT CONTROL
3	MINIMUM RESIDUAL DISPLACEMENT
4	ARC-LENGTH
5	CONSTANT WORK

(a) *Constant Load Increment* [11]

Iterative Parameter : None

Incremental Parameter : None

(b) *Constant Displacement Increment* [22]

Iterative Parameter : Control Node + DOF (NODE.DOF)

Incremental Parameter : Incremental Displacement (L)

(3) *Advanced Technique* [43]

Iterative Parameter : Load Size of First Step

Incremental Parameter : Maximum Arc Distance

AGRP [Construction Stage ID] [Active Group ID]

USE [1=LOADCASE] [NO.] [FACTOR] [Load/Construction stage]

USE [2=COMBLOAD] [NO.] [FACTOR] [Load/Construction stage]

[LOAD/Construction STAGE]

= -2 : For construction stage

= -1 : For initial imperfections

= 0: Reserved, for traditional NO load stage

> 0: Defined by User

Note: If no “Load/Construction Stage” defined by user, this value should be left as **BLANK**.

6.10.3 Modal Analysis

SOLU [NO.][TYPE=21~30,21=LUMPED MASS,22=CONSISTENT MASS][dummy][IF RUN:1=RUN,0=NOT RUN] <NAME>

FREQ [EIGENVALUES,6] [IF PRINT: 1=PRINT, 0=NOT PRINT] [IF ACTIVE AUTO FUNCTION: 1=ACTIVE, 0=INACTIVE] [Modal Participating Mass Ratios,90%] [INC modes,10]

USE [*-1=LOADCASE*] [NO.] [FACTOR] !for additional mass

USE [1=LOADCASE] [NO.] [FACTOR] !for determination of initial stiffness

USE [2=COMBLOAD] [NO.] [FACTOR]

6.10.4 Eigen-Buckling Analysis

SOLU [NO.][TYPE=31~40][dummy][IF RUN:1=RUN,0=NOT RUN] <NAME>

BUCK [EIGENVALUES, 6] [IF PRINT]

USE [1=LOADCASE] [NO.] [FACTOR]

USE [2=COMBLOAD] [NO.] [FACTOR]

6.10.5 Response Spectrum Analysis

RESPONSE SPECTRUM ANALYSIS

SOLU [NO.][TYPE=41~45][dummy][IF RUN:1=RUN,0=NOT RUN] <NAME>

SPEC [MODAL ANALYSIS CASE NO.]

[MODAL COMBINATION:**1=CQC**,2=SRSS,3=ABS]

[SEISMIC DIRECTION: **0: HORIZONTAL**, 1: VERTICAL] [EXCITATION ANGLE, **0**]

[Modal Damping Ratio, **0.05**]

USE [4=SEISMIC SPECTRUM FUNCTION] [NO.] [dummy] !Response spectrum function

RESPONSE SPECTRUM ANALYSIS(DIRECTIONAL COMBINATION)

SOLU [NO.][TYPE=46~50][dummy][IF RUN:1=RUN,0=NOT RUN] <NAME>

DCMB [DIRECTIONAL COMBINATION:**1=SRSS**,2=ABS,3=Modified SRSS(Chinese)]

[Horizontal Spectrum Ana.: EQ1] [Horizontal Spectrum Ana.: EQ2] [Vertical Spectrum Ana.: EQ3] {Scale factor for ABS, 0.3}

6.10.6 Time History Analysis

SOLU [NO.][TYPE=51~60][IF DESIGN][IF RUN:1=RUN,0=NOT RUN] <NAME>

[TYPE]

Number	Analysis Case Type	Mass Type
52	Nonlinear analysis (PEP)	Lumped
53	Nonlinear analysis (STABILITY FUNCTION)	Lumped
55	Nonlinear analysis (PEP)	Consistent
56	Nonlinear analysis (STABILITY FUNCTION)	Consistent

TIHI […]

No.	Parameter	Description
1	TIME HISTORY TYPE	1= Direct integration; 2= Mode superposition
2	TIME HISTORY MOTION TYPE	1= Transient; 2= Periodic
3	TOTAL NO. OF TIME STEP	<=0 depend on the time history function defined; > 0 stop when the total time reached
4	TIME INCREMENT	0.02 second
5	MODAL ANALYSIS CASE No.	MODAL ANALYSIS CASE USED for mode superposition (ignorable)

NONP […]

Note: All data should be in one line.

No.	Parameter	Description
1	MYIELD	0=Elastic; 1=Plastic Element; 2=Plastic Hinge
2	Force Recovery Scheme	1= Total Secant Stiffness Method; 2= Incremental Tangent Stiffness Method
3	CYCLES	Total Load Cycles, 1
4	ITERATIONS	Number of Iterations for Each Load Cycle, 1
5	INC LOADFACTOR	Incremental Load Factor, 1.0
6	NUMERICAL METHOD	Numerical Method: 11=NR, 22=Single Disp., 43=ArcLen+MRD
7	ITER-PARAMETER	Parameter for Numerical Method, 4
8	INC-PARAMETER	Parameter for Numerical Method, 5
9	TANGENTS	Number of Iterations for Tangent Stiffness Matrix, 1
10	IMPERFECTION DIRECTION	+/-1, DIMP; +/-2, EIMP; +/-3, NIMP 0, No Imperfection
11	MIN. MEMBER IMPERFECTION	Minimum Member Imperfection, 1/1000.

12	Stiffness Reduction Factor	0.8~1.0 (AISC-LRFD 2010)
13	Target Load Factor	Target Load Factor, 1.0 (-1=Disable this function)

*<u>IMPERFECTION DIRECTION: To determine **GLOBAL** and **LOCAL** imperfection</u>*

(a) *IMPERFECTION DIRECTION=+/-1*: “Displacement”, using deformed shape of applied forces as global imperfection;

<u>DIMP</u> [MAGNITUDE OF GLOBAL IMPERFECTION, L]

(b) *IMPERFECTION DIRECTION=+/-2*: “Eigen-buckling mode”, using specified buckling mode as global imperfection, need the following parameters:

<u>EIMP</u> [MAGNITUDE OF GLOBAL IMPERFECTION, L][>0, Directly Calc. MODES,5] [iTH MODE USED,1]

(c) *IMPERFECTION DIRECTION=+/-3*: “Notional Forces”, using notional horizontal forces as global imperfection, need the following parameters:

<u>NIMP</u> [Angle about vertical axis (HORIZONTAL Direction), 0, DEGREE][% of Gravity Load]

Note: For all methods, “+ ve” considers member imperfections in both principal axis while “-ve” considers only one principal axis which is adequate for most structures.

<u>DINT</u> [NUMERICAL METHOD FOR DIRECT INTEGRATION, 1=Newmark] [Parameters …]

(1) Newmark: [Gamma, 0.5] [Beta, 0.25]

<u>DAMP</u> [TYPE] [PARAMETERS …] – **Rayleigh Damping**

(1) [Type=1, Direct input, {C}=a*{M}+b*{K}] [Alpha, Mass Proportional coefficient] [Beta, Stiffness Proportional coefficient]

(2) [Type=2, Calculate by Period] [T1, 1st period] [T2, 2nd period] [Damping Ratio1 ξ_1] [Damping Ratio2 ξ_2]

(3) [Type=3, Calculate by Frequency] [f1, 1st frequency] [f2, 2nd frequency] [Damping Ratio1 ξ_1] [Damping Ratio2 ξ_2]

Note:

$$\alpha = \frac{4\pi(\xi_1 T_1 - \xi_2 T_2)}{(T_1^2 - T_2^2)}$$

$$\beta = \frac{T_1 T_2(\xi_2 T_1 - \xi_1 T_2)}{\pi(T_1^2 - T_2^2)}$$

$$(\alpha + \beta\omega_i^2 = 2\xi_i\omega_i\,,\quad f = \frac{1}{T})$$

<u>USE</u> [*-1=LOADCASE*] [NO.] [FACTOR] !for additional mass

<u>USE</u> [1=LOADCASE] [NO.] [FACTOR] !for static loads

<u>USE</u> [2=COMBLOAD] [NO.] [FACTOR] !for static loads

USE [5=DYNAMIC FUNCTION] [NO.] [FACTOR] [Arrival Time, 0, s]

[SEISMIC DIRECTION: **0: HORIZONTAL**, 1: VERTICAL]

[EXCITATION ANGLE, **0**, DEGREE] !for time history function (Earthquake)

6.11 Material

To define material properties.

***MATP**

MAT [NO.][TYPE] <NAME> COLOR

Number	Material Type
0	OTHER
1	STEEL
2	CONCRETE
3	ALUMINIUM
4	GLASS
5	REBAR

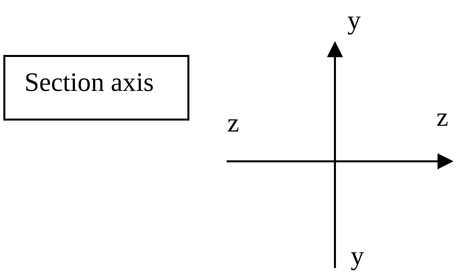
[E (short term for concrete), F/L^2] [-1~0 =Poisson's ratio; >0 = G] [Density, F/L^3] [Thermal coefficient][Yield strength, F/L^2] {[Yield tensile strength, F/L^2] [Long term for concrete: E, F/L^2]}

6.12 Frame Section

To define frame section properties.

***SECT**

Note: The section ID for all sections should be unique.



6.12.1 General and Steel Section

SSTL [NO.][TYPE][SHEAR FACTOR] [SHAPE] <NAME> COLOR

LINE1: [MAT] [IMP-Y] [IMP-Z] [Limit SY, 1.2] [Limit SZ, 1.2]

[B (B1)][D][Tf (Tf1)][tw]{[B2][Tf2][ds]}

LINE2: [AREA, L^2] [IY, L^4] [IZ] [J,L^4] [ZY(+), L^3] [ZZ(+)] [SY, L^3] [SZ]

{[Iw, L^6] [ZY(-)] [ZZ(-)]}

LINE3: {[Avy, L^2] [Avz] [ry, L] [rz] [u] [x]}

LINE4: {[A_{eff}] [$Z_{y,eff}$(+)] [$Z_{z,eff}$(+)] [$S_{y,eff}$] [$S_{z,eff}$] [$Z_{y,eff}$(-)] [$Z_{z,eff}$(-)]}

Note: (1) Line 2&3 can be ignored when the dimensions are defined.

(2) Line 2&3 cannot be ignored when LINE4 exists.

[SHAPE] – 5 DIGITAL AS BELOW: {N_1}$N_2N_3N_4N_5$

N1 – Indicator of considering eigen-imperfection or not (0,1)

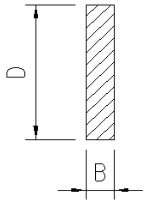
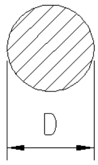
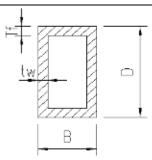
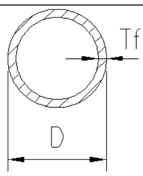
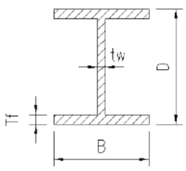
N2 – Indicator of using elastic or plastic modulus

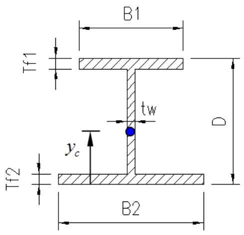
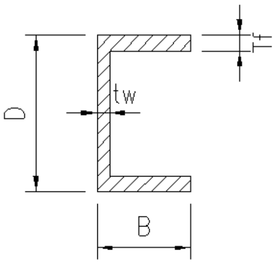
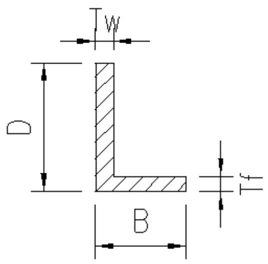
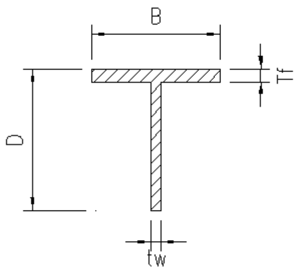
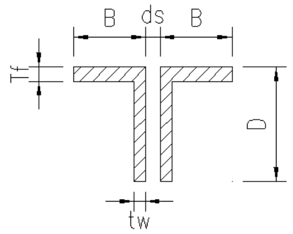
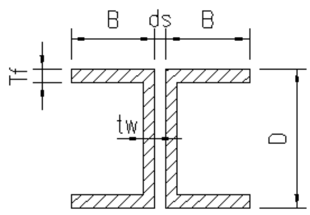
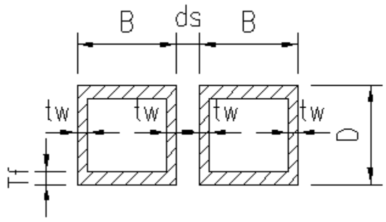
1~2: Effective stress method (1=Elastic, 2=Plastic)

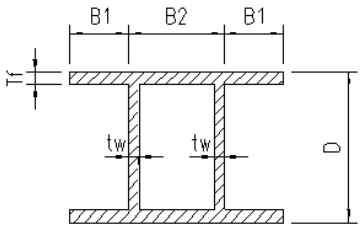
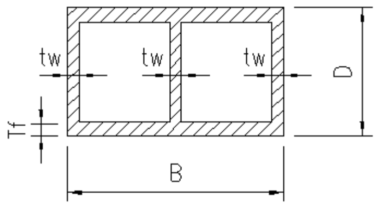
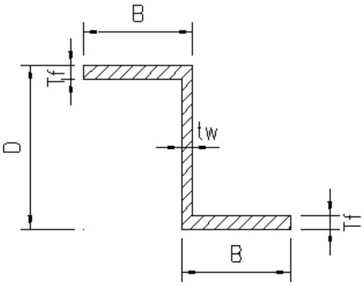
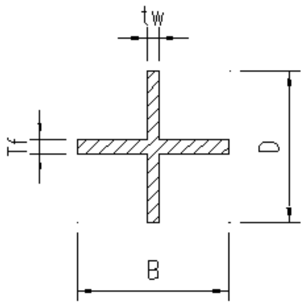
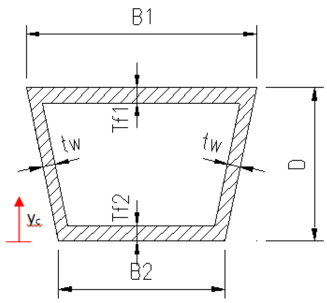
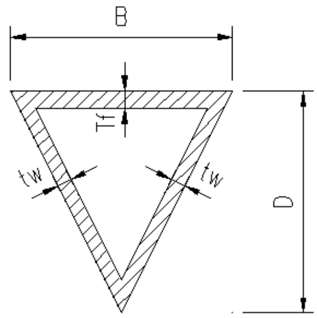
N3 – Formed type of section (1= rolled, 2=fabricated, 3= cold-formed)

N4 – Section Axis (0 = yz axis; 1 = uv axis)

N5 – Stress Type for Stress Computation (1=Direct sum of stress, 2=Square-root of stress, 3= Tension Only, 4=Compression Only)

Type	Name	Dimensions	Figure
0	Other	N.A.	N.A.
1	Rectangular	B,D	
2	Circular	D	
3	Box	B,D,Tf,tw	
4	Pipe	D,Tf	
5	I/H section(double symmetric)	B,D,Tf,tw	

6	Monosymmetric I-section	B (B1),D,Tf1,tw,B2,Tf2	
7	Channel	B,D,Tf,tw	
8	Single angle	B,D,Tf,tw	
9	Tee section	B,D,Tf,tw	
10	Double angle	B,D,Tf,tw,ds	
11	Double channel	B,D,Tf,tw,ds	
12	Double box	B,D,Tf,tw,ds	

13	I-Box section	B (B1),,D,Tf,tw,B2	
14	Two boxes	B,D,Tf,tw	
15	Z-section	B,D,Tf,tw	
16	Cross plate	B,D,Tf,tw	
17	Trap-hollow section	B,D,Tf,tw,B2,Tf2	
18	Triangular-hollow section	B,D,Tf,tw	

6.13 Area Section

To define area type section properties.

<u>*ASECT</u>

Note: The section ID for all sections should be unique.

6.13.1 Shell Section

<u>SSHL</u> [NO.] <NAME> COLOR

[MAT] [Type] [THICKNESS-BENDING] [THICKNESS-MEMBRANE] {[IsThickPlate]}

<u>[TYPE]</u>

= 1 : Shell (Membrane+ Plate)

= 2 : Membrane only

= 3 : Plate only

<u>[IsThickPlate]</u>

= 0 : Thin Plate (Default)

= 1 : Thick Plate

6.14 Node

<u>*NODE</u>

[NO.] [X, L] [Y, L] [Z, L]

6.15 Semi-rigid Connection and Spring Models

<u>*SEMIRIGID</u>

<u>SRModel</u> [NO.>0] [Relationship] [TYPE] [Initial Stiffness] [Ultimate Rot./Disp.] [SIGN] <NAME>

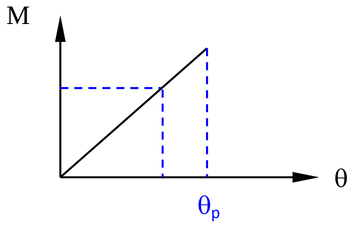
Note: [Ultimate Rot./Disp.] =0 means no limitation.

[Relationship=1]: Moment -Rotation

[TYPE=1] : Linear Model

SRModel [NO.] [1] [1] [R_e Initial stiffness, F*L/RAD] [θ_p Ultimate Rotation, RAD] [SIGN] <NAME>

$$M = R_e \theta$$

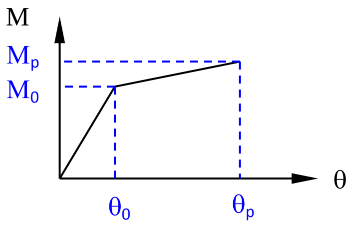


[TYPE=2] : Bi-linear Model

SRModel [NO.] [1] [2] [$R_e = M_0/\theta_0$ or dummy] [θ_p Ultimate Rotation, RAD] [SIGN] <NAME>

[M_0 Yield moment, F*L], [θ_0 Yield Rotation, RAD] , [M_p Ultimate moment, F*L]

$$M = \begin{cases} R_e\theta, \quad R_e = M_0/\theta_0 & (\theta \le \theta_0) \\ M_0 + \dfrac{M_p - M_0}{\theta_p - \theta_0}(\theta - \theta_0) & (\theta_0 < \theta \le \theta_p) \\ 0 \qquad (\theta > \theta_p) & \end{cases}$$

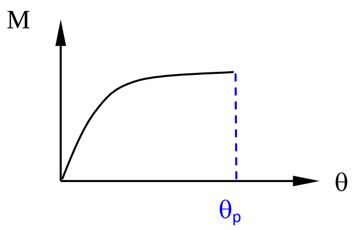


[TYPE=3] : Three-parameter power model (Kishi and Chen 1987)

SRModel [NO.] [1] [3] [R_e Initial stiffness, F*L/RAD] [θ_p Ultimate Rotation, RAD] [SIGN] <NAME>

[M_0 Reference moment, F*L], [n Shape parameter, NO UNIT]

$$\begin{cases} M = \dfrac{R_e\theta}{[1+(\theta/\theta_0)^n]^{1/n}} \\ R = \dfrac{dM}{d\theta} = \dfrac{R_e}{[1+(\theta/\theta_0)^n]^{(n+1)/n}} \\ \theta = \dfrac{M}{R_e[1-(M/M_0)^n]^{1/n}} \end{cases} \quad \text{where } \theta_0 = M_0/R_e$$



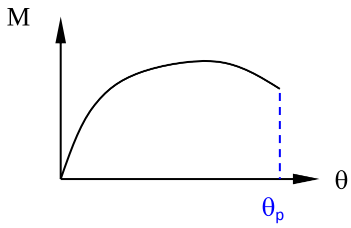
[TYPE=4] : Four-parameter power model (Richard and Abbott 1975)

SRModel [NO.] [1] [4] [R_e Initial stiffness, F*L/RAD] [θ_p Ultimate Rotation, RAD] [SIGN] <NAME>

[M_0 Reference moment, F*L], [R_p Strain-hardening/softening stiffness, F*L/RAD],

[n Shape parameter, NO UNIT]

$$\begin{cases} M = \dfrac{(R_e - R_p)\theta}{\{1+[(R_e - R_p)\theta / M_0]^n\}^{1/n}} + R_p\theta \\ R = \dfrac{dM}{d\theta} = \dfrac{R_e - R_p}{\{1+[(R_e - R_p)\theta / M_0]^n\}^{(n+1)/n}} + R_p \end{cases}$$



$$\theta_0 = \frac{M_0[-1+(1-R_e/R_p)^{n/(1+n)}]^{1/n}}{R_e - R_p} \quad (R = \frac{dM}{d\theta} = 0)$$

[TYPE=5] : Multi-linear Model

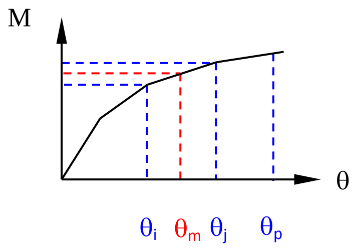
SRModel [NO.] [1] [5] [R_e Initial stiffness, F*L/RAD] [θ_p Ultimate Rotation, RAD] [SIGN] <NAME>

[M_1, F*L], [θ_1, RAD]

[M_2], [θ_2]

…

[M_n], [θ_n]



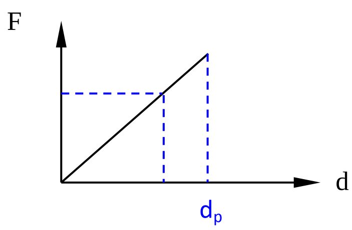
$$\begin{cases} \Delta\theta = \theta_j - \theta_i \\ \Delta M = M_j - M_i \\ R_t = \dfrac{\Delta M}{\Delta\theta} \\ M_m = M_i + R_t\theta_m \end{cases} \qquad \text{j=i+1}, \ \theta_i < \theta_m < \theta_j$$

[Relationship=2]: Force-Displacement

[TYPE=1] : Linear Model

SRModel [NO.] [2] [1] [k_e Initial stiffness, F/L] [d_p Ultimate Displacement, L] [SIGN] <NAME>

$$F = k_e d$$

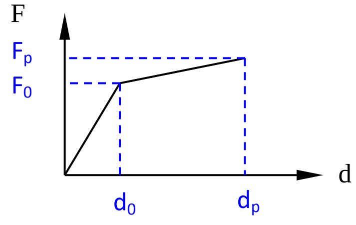


[TYPE=2] : Bi-linear Model

SRModel [NO.] [2] [2] [$k_e=F_0/d_0$ or dummy] [d_p Ultimate Displacement, L] [SIGN] <NAME>

[F_0 Yield force, F], [d_0 Yield Displacement, L] , [F_p Ultimate force, F]

$$F = \begin{cases} k_e d, \quad k_e = F_0 / d_0 & (d \le d_0) \\ F_0 + \dfrac{F_p - F_0}{d_p - d_0}(d - d_0) & (d_0 < d \le d_p) \\ 0 \quad (d > d_p) \end{cases}$$



[TYPE=3] : Multi-linear Model

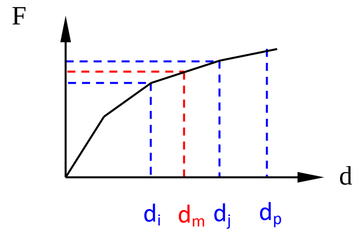
<u>SRModel</u> [NO.] [2] [3] [k_e Initial stiffness, F/L] [d_p Ultimate Displacement, L] [SIGN] <NAME>

[F_1, F], [d_1, L]

[F_2], [d_2]

…

[F_n], [d_n]



$$\begin{cases} \Delta d = d_j - d_i \\ \Delta F = F_j - F_i \\ k_t = \dfrac{\Delta F}{\Delta d} \\ F_m = F_i + k_t d_m \end{cases}$$

j=i+1, $d_i < d_m < d_j$

6.16 Beam-Column Element

To define beam-column elements.

<u>*BEAM</u>

[Mem NO.][SECT] [JOT1][JOT2] *[Design Type:0=By Program (beam-column), 1= column, 2= beam, 3=brace][If Igored plastic hinge,-1=ignore,0=Default hinge model, >0 Plastic hinge model][Seismic Design=0 No Seismic (Default),4,3,2,1]*

6.17 End Conditions

To define the end connection of beam-column element.

<u>*RELEASE</u>

[Mem NO.] [SPRING 1~6: IIY, IIZ, JJY, JJZ, *IJT*, *IJX*]

IIY: connection stiffness at i-node and about y-y axis [-1=Rigid, 0=Pin, >0 Semirigid Model]

IIZ: connection stiffness at i-node and about z-z axis [-1=Rigid, 0=Pin, >0 Semirigid Model]

JJY: connection stiffness at j-node and about y-y axis [-1=Rigid, 0=Pin, >0 Semirigid Model]

JJZ: connection stiffness at j-node and about z-z axis [-1=Rigid, 0=Pin, >0 Semirigid Model]

IJT: torsional spring [-1=Rigid, 0=Pin]

IJX: axial spring [-1=Rigid, 0=Pin, >0 Semirigid Model]

6.18 Effective Length

<u>*EFFLEN</u>

[Mem NO.] [BEAM LE] [LOADING CONDITION] *[COLUMN LE-Y, L for <0] [COLUMN LE-Z, L for <0][Equivalent Moment Factors (my, mz): 1=Set as 1.0 (Default); 0=By program]*

Parameters	Steel Design	Other Cases
[BEAM LE]	Beam effective length	0 (N.A.)
[LOADING CONDITION]	0,1	0 (N.A.)

6.19 Combined Member

To define a complete beam-column element with a number of segments for deflection checking.

<u>*SEGMENT</u>

<u>MEMB</u> [No.] [Start Node] [End node] {[Type]} <Name>

[Beam-column No. List …]

6.20 Eccentricity

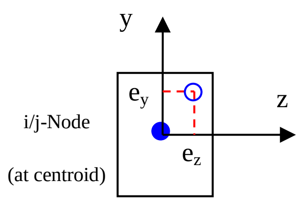
To define the eccentricity (offsets) of beam-column end.

<u>*BMECCE</u>

[Mem NO.] [Type=1, End Section Eccentricity] [ey_i, L] [ez_i, L] [ey_j, L] [ez_j, L]

[Mem NO.] [Type=2, End Length Offset (for Rigid Zone)] [0=Direct input, 1=By Program][e_i>0, L] [e_j>0, L] [Rigid Zone Factor= 0~1(Default), L1=L-f*(e_i +e_j)>=0]

Note: The end eccentricities and length offsets are relative to the centroid point of cross section in member local axis.



End Section Eccentricity

End Length Offset

6.21 Shell Element

To define shell elements.

<u>*SHELL</u>

[NO.] [SECT] [TYPE,1] [JOT1] [JOT2] [JOT3]

6.22 Floor Element

To define floor elements.

<u>*FLOOR [STIFFNESS, F/L]</u>

[NO.][Shell Section] [JOT1] [JOT2] [JOT3] {[JOT4]}

[Shell Section] : 0 No shell section assignment

>0 Shell section ID

6.23 Beam-Column/Spring Element Local Axis

To define the local axis of beam-column or spring element.

<u>*ANGL</u>

[MEMBER NO.] [1=ANGLE; 2=KP] [VALUE, DEGREE for ANGLE]

[SPRING NO.] [11=ANGLE; 12=KP] [VALUE, DEGREE for ANGLE]

6.24 Node Local Axis

To define the node local coordinate system.

<u>*NSYS</u>

<u>NSYS</u> [NO.] [TYPE] <NAME>

<u>TYPE=1(By 3 Nodes)</u>

[Node i] [Node j] [Node k] <u>(Right-hand rule)</u>

$V_1 = \vec{ij} = \{X_j - X_i\}$ (Local x-axis)

$V_2^{'} = \overrightarrow{jk} = \{X_k - X_j\}$ (Reference axis)

$V_3 = V_1 \times V_2^{'}$ (Local z-axis)

$V_2 = V_3 \times V_1$ (Local y-axis)

TYPE=2 (By 3 Angles)

[ROTATION,ABOUT Y θ_G, DEGREE] [ROTATION,ABOUT Z' θ_e, DEGREE]

[ROTATION,ABOUT X'' β, DEGREE]

6.25 Assignment of Node Local Axis

To assign the node local coordinate system to nodes.

***NLAXES**

NLAXES [NSYS NO.]

[Node No. List …]

6.26 Spring

To define spring element and node support spring.

***SPRI**

SUPP …
ADJA…

6.26.1 Support Spring

SUPP [NODE] [DOF] [SRMODEL NO.]

6.26.2 Spring Element

ADJA [No.] [Type=2] [iNode] [jNode] [Ka] [Kby] [Kbz] [Kt] [Kvy] [Kvz]

Ka : Local axial element stiffness (along x-axis)

Kby : Local bending element stiffness about y-axis

Kbz : Local bending element stiffness about z-axis

Kt : Local tortion element stiffness (about x-axis)

Kvy : Local shear element stiffness along y-axis

Kvz : Local shear element stiffness along z-axis

Note: (1) Ka, Kvy, Kvz refer to *SEMIRIGID (SRMODEL NO.) – Force vs Disp;

(2) Kby, Kbz, Kt refer to *SEMIRIGID (SRMODEL NO.) – Moment vs Rotation;

(3) Ka, Kby, Kbz, Kt, Kvy or Kvz =0 is for ignorable spring.

6.27 Boundary Condition

To define the boundary conditions for nodes.

<u>*BOUN</u>

[NODE] [VALUE]

[VALUE]: Max. 6 number together, 0=Free, 1=Restraint, e.g. 110101

6.28 Group

<u>*GROUP</u> [NO.] <NAME>

@NODE

… [Node No. List, Max. Num per line = 10 (if any)]

@BEAM

… [Member No.List, Max. Num per line = 10 (if any)]

@SHELL

… [Shell No.List, Max. Num per line = 10 (if any)]

@FLOOR

… [Floor No.List, Max. Num per line = 10 (if any)]

@SPRING

… [Spring No.List, Max. Num per line = 10 (if any)]

@AREA

… [Area No.List, Max. Num per line = 10 (if any)]

6.29 One/Two-Way

To define one-way direction for floor/shell pressure assignment.

<u>*ONEWAY</u>

[Obj. Type: 1=Floor, 2=Shell] [Obj. No.] [Direction: 1=Side 1-3, 2=Side 2-4]

6.30 Load Case

<u>*LOADCASE</u> [NO.] [FACTOR] [TYPE]<NAME>

Number	Load Case Type
0	OTHER
1	DEAD LOAD
2	LIVE LOAD
3	WIND LOAD
4	TEMPERATURE
5	SEISMIC
6	EARTH
7	WATER

6.30.1 Self Weight

SELF [-1][SYS][+/-][DIR][Amplification Factor] !FOR ALL MEMBERS

SELF [-2][SYS][+/-][DIR][Amplification Factor] !FOR ALL SHELLS

SELF [-3][SYS][+/-][DIR][Amplification Factor] !FOR ALL FLOORS

[SYS]

= GL : Global axis (Not allowed local axis)

[DIR]

= X : X-axis; = Y : Y-axis; = Z : Z-axis

6.30.2 Pretension (Cable) Force

PRES [MEMBER][PRE-TENSION FORCE, F]

6.30.3 Joint Load

JNTL [NODE] [FX, F][FY, F][FZ, F][MX, F*L][MY, F*L][MZ, F*L] {[SYS]}

[SYS]

= LO : Nodal Local axis

= GL : Global axis (Default)

6.30.4 Member Point Load

PNTL [MEMBER][VALUE, F][DIST FROM LEFT, L][SYS][DIR] {[IsNodalLoad]}

[SYS]

= LO : Local axis

= GL : Global axis

[DIR]

= X : X-axis; = Y : Y-axis; = Z : Z-axis

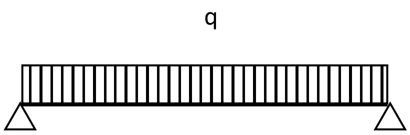
[IsNodalLoad]

= 0 : Consider as Member Load (Default)

= 1 : Convert to Nodal Loads Only

6.30.5 Member UDL

UDL [MEMBER] [q, F/L] [SYS] [DIR] {[IsNodalLoad]}



[SYS]

= LO : Local axis

= GL : Global axis

= TG : Global axis (Distributed along Member)

[DIR]

= X : X-axis; = Y : Y-axis; = Z : Z-axis

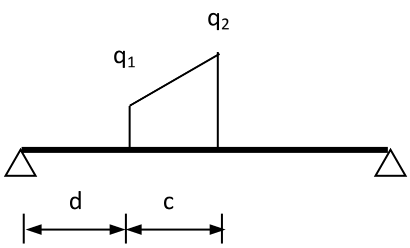
[IsNodalLoad]

= 0 : Consider as Member Load (Default)

= 1 : Convert to Nodal Loads Only

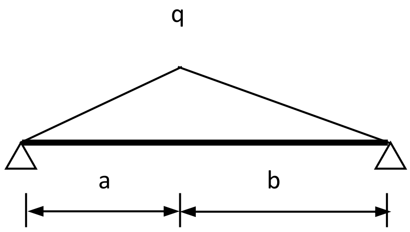
6.30.6 Member TRAP

TRAP [MEMBER] [q_1, F/L] [q_2, F/L] [d, L] [c, L] [SYS] [DIR] {[IsNodalLoad]}



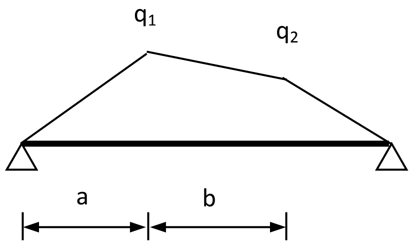
6.30.7 Member TRAPT

TRAPT [MEMBER] [q, F/L] [a, L] [SYS] [DIR] {[IsNodalLoad]}



6.30.8 Member TRAPQ

<u>TRAPQ</u> [MEMBER] [q_1, F/L] [q_2, F/L] [a, L] [b, L] [SYS] [DIR] {[IsNodalLoad]}



6.30.9 Settlement

<u>LODP</u> [NODE][DOF=1,2,3][DISPLACEMENT, L] {[SYS]}

<u>[SYS]</u>

= LO : Nodal Local axis

= GL : Global axis (Default)

6.30.10Temperature

<u>TEMP</u> [MEMBER][VALUE, ^{o}C]

6.30.11Floor Pressure

<u>FPRE</u> [FLOOR] [TYPE] [DIR=N,X,Y,Z] [P1, F/L^2] {[P2~ P4]}

<u>[TYPE]</u>

= 0 : CONVERT TO MEMBER LOADS AND THEN TO NODES

= 1 : CONVERT TO NODES DIRECTLY

6.30.12Shell Pressure

<u>SPRE</u> [SHELL] [TYPE] [DIR=N,X,Y,Z] [P1, F/L^2] {[P2~ P4]}

[TYPE]

= 0 : CONVERT TO MEMBER LOADS AND THEN TO NODES

= 1 : CONVERT TO NODES DIRECTLY

6.30.13 Area Pressure

APRE [Area No] [TYPE] [DIR=N,X,Y,Z] [P (Uniform) , F/L^2]

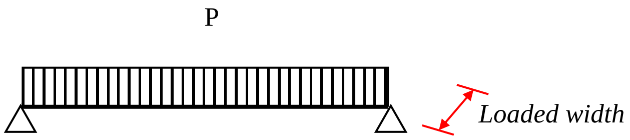
[TYPE]

= 0 : CONVERT TO MEMBER LOADS AND THEN TO NODES

= 1 : CONVERT TO NODES DIRECTLY

6.30.14 Member Pressure

MPRE [MEMBER] [P, F/L^2] [SYS] [DIR] [Loaded width]{[IsNodalLoad]}



Note: Similar to UDL.

[SYS]

= LO : Local axis

= GL : Global axis

= TG : Global axis (Distributed along Member)

[DIR]

= X : X-axis; = Y : Y-axis; = Z : Z-axis

[Loaded width]

< 0 : width input by user, L

= 1 : Section “B”

= 2 : Section “D”, (Default)

= 3 : Max (B, D)

= 4 : SQRT(B*B+D*D)

[IsNodalLoad]

= 0 : Consider as Member Load (Default)

= 1 : Convert to Nodal Loads Only

6.31 Load Combination

***COMBLOAD**

COMB [NO.] [FACTOR] <NAME>

[LOADCASE NO.][FACTOR]

...

COMB [NO.] [FACTOR] <NAME>

[LOADCASE NO.][FACTOR]

...

6.32 Response Spectrum Function

***SEISFUNC**

SFUNC [NO.] [TYPE] [Relationship] [Function Damping Ratio, 0.05] [Direction]<NAME>

[Relationship]

=1 Period [L/s^2] vs. Sa/g

=2 Period [L/s^2] vs. Sa [L/s^2]

[Direction]

=0 Horizontal

=1 Vertical

[TYPE] = 0 User defined

[T1, s] [Sa(1)/g]

[T2, s] [Sa(2)/g]

…

[Ti, s] [Sa(i)/g]

[TYPE] = 1 GB50011 (2010)

[Max Influence Factor: Amax=0.12] [Seismic Intensity: SI=7.5] [Characteristic Ground Period: TG=0.35] [PERIOD REDUCTION FACTOR, 1.0]

[TYPE] = 2 Eurocode8 (2004)

Note: All data should be in one line.

No.	Parameter	Description
1	ag	Horizontal Ground Acceleration
2	Spectrum Type	Spectrum Type (1~2), 1
3	Ground Type	Ground Type (1~5 = A,B,C,D,E), 2
4	Soil Factor	Soil Factor(S), 2
5	Accel. Ratio	Acceleration Ratio(avg/ag) , -1 (<0 not applicable)
6	TB	Spectrum Period(TB) , 0.15
7	TC	Spectrum Period(TC) , 0.5
8	TD	Spectrum Period(TD) , 2.0
9	Beta	Lower Bound Factor(Beta) , 0.2
10	q	Behavior Factor (q) , 2

[TYPE] = 7 IS 1893 (2002)

[Seismic Zone Factor: Z=0.16] [Importance Factor: I=1.0] [Response Reduction Factor: R=3] [Soil Type I~III = 1]

6.33 Dynamic (Time History) Function

***DYNAFUNC**

DFUNC [NO.] [TYPE] [Relationship] [Time interval, s] [Scale Factor] <NAME>

[Relationship]

=1 Acceleration [L/s^2] vs. Time

=2 Displacement [L]vs. Time

=3 Force [F] vs. Time

[Scale Factor]

>0 Scale by this factor

<0 Scale to this value (Scale factor = This value / Peak value)

[TYPE]

=1 User time history record (Constant time interval)

[value1] [value2] … [value i] …

=2 User time history record (Variable time interval)

[Time0] [value] [Time1] [value] … [Timei] [value] …

=3 Time History Sine Function:

[Total No. of cycles] [Period: T] [Amplitude: A0]

$$A = A_0 \sin(\frac{2\pi}{T}\ t)$$

=4 Time History Cosine Function:

[Total No. of cycles] [Period: T] [Amplitude: A0]

$$A = A_0 \cos(\frac{2\pi}{T}\ t)$$

=5 Time History Ramp Function:

[Total Time] [Ramp time: t0] [Amplitude: A0]

$$A = \begin{cases} k_0 t, & (t \le t_0) \quad \text{where } k_0 = A_0 / t_0 \\ A_0, & (t > t_0) \end{cases}$$

=6 Time History Sawtooth Function

[Total No. of cycles] [Period: T] [Ramp time: t0] [Amplitude: A0]

$$A = \begin{cases} k_0 t, & (t \le t_0) \quad \text{where } k_0 = A_0 / t_0 \\ A_0, & (t_0 < t \le 0.5T - t_0) \\ A_0 - k_0(t - 0.5T + t_0), & (0.5T - t_0 < t \le 0.5T + t_0) \\ -A_0, & (0.5T + t_0 < t \le T - t_0) \\ -A_0 + k_0(t - T + t_0), & (T - t_0 < t \le T) \end{cases}$$

=7 Time History Triangular Function

[Total No. of cycles] [Period: T] [Amplitude: A0]

$$A = \begin{cases} k_0 t, & (t \le 0.25T) \quad \text{where } k_0 = 4A_0 / T \\ A_0 - k_0(t - 0.25T), & (0.25T < t \le 0.75T) \\ -A_0 + k_0(t - 0.75T), & (0.75T < t \le T) \end{cases}$$

6.34 Area

Areas are not structural objects and therefore all their properties such as section assignment and loading assignment are invalid in the analysis before they are converted to floor or shell elements. The area properties will be converted to the corresponding floor/shell properties after they are converted to floor/shell elements. The function of AREA is to help to create complex structures with floor/shell elements.

***AREA**

AREA [AREA NO.] [NODE NO. LIST]

AMESH [SHELL SECTION NO.] [AREA NO. LIST]

ALSIZE [Area No.] [$M_1(S_1)$~ $M_n(S_n)$, M_i<0 for parts , S_i>0 for size, M_i/S_i =0 for at least one]

ASHELL [AREA NO.] [MESHED SHELL NO. LIST]

AFLOOR [AREA NO.] [MESHED FLOOR NO. LIST]

6.35 Diaphragm

To group a number of nodes to consider the diaphragm action.

***DIAPHRAGM**

DIAPH [No.] [STIFFNESS] [CG Node] <Name> COLOR

{NODE List}

[Stiffness] =-1 : Rigid, the stiffness is determined by Program;

= 0: No stiffness, the diaphragm will be ignored in Program;

> 0: Semi-rigid

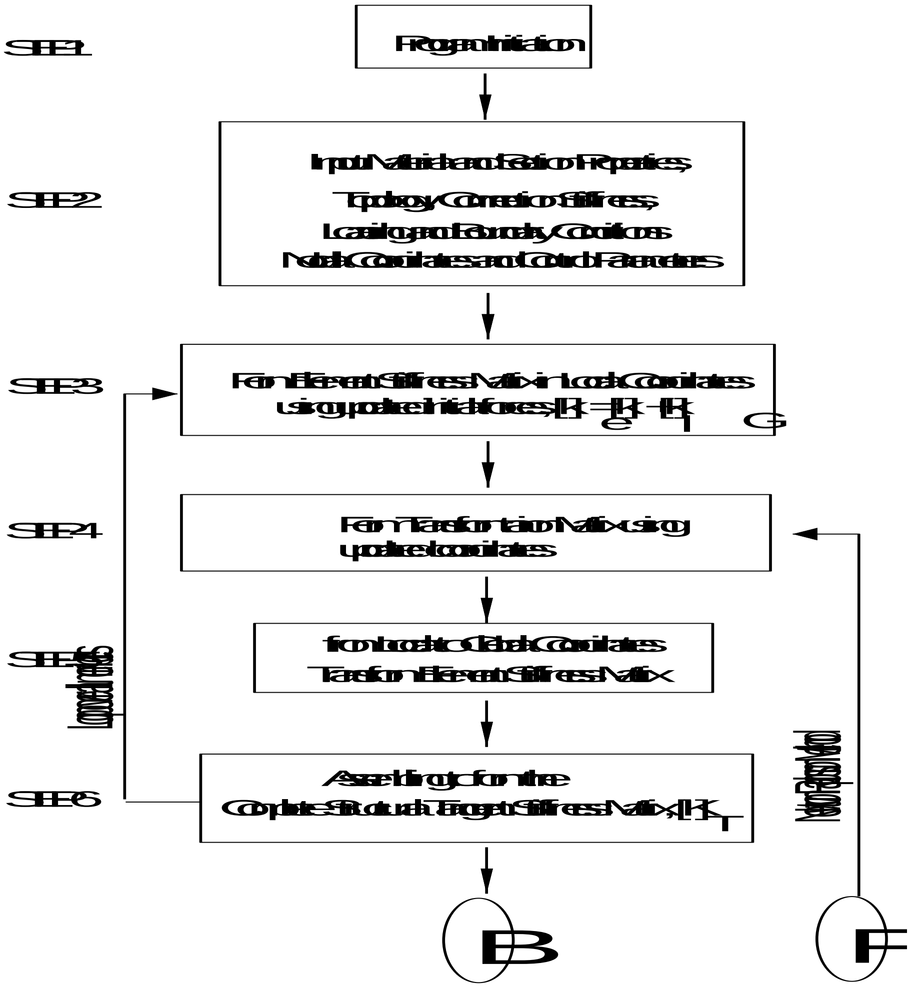
[CG Node] = Node of diaphragm at Geometrical centroid

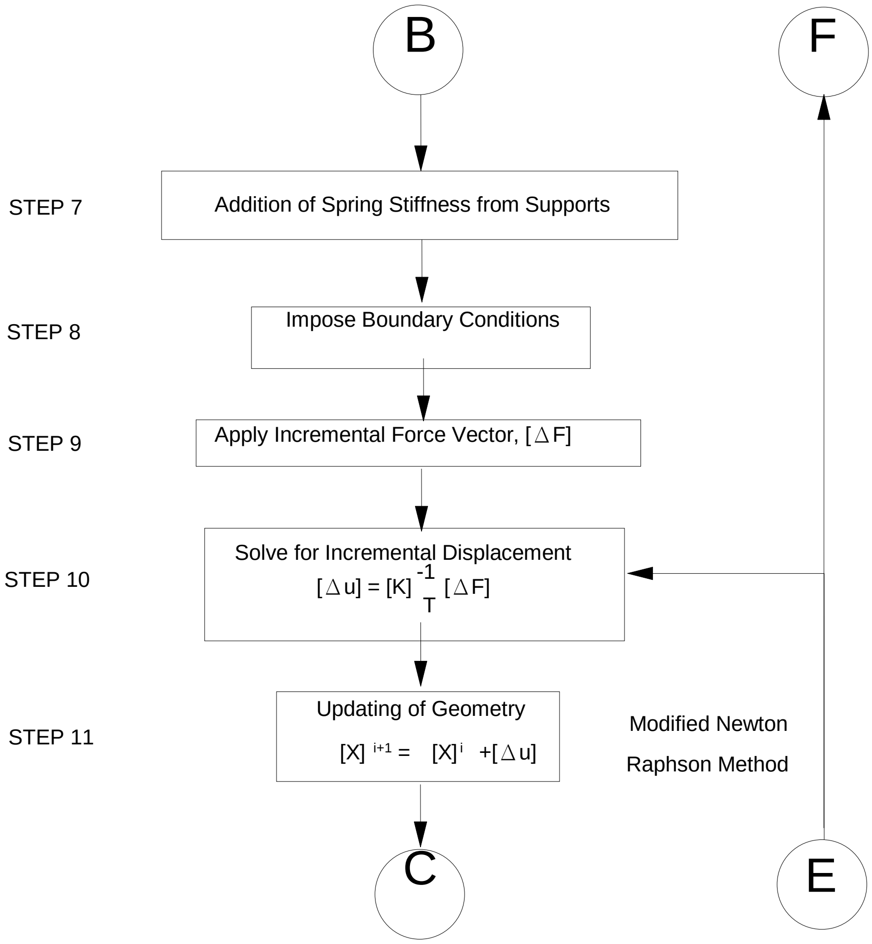
6.36 End of File

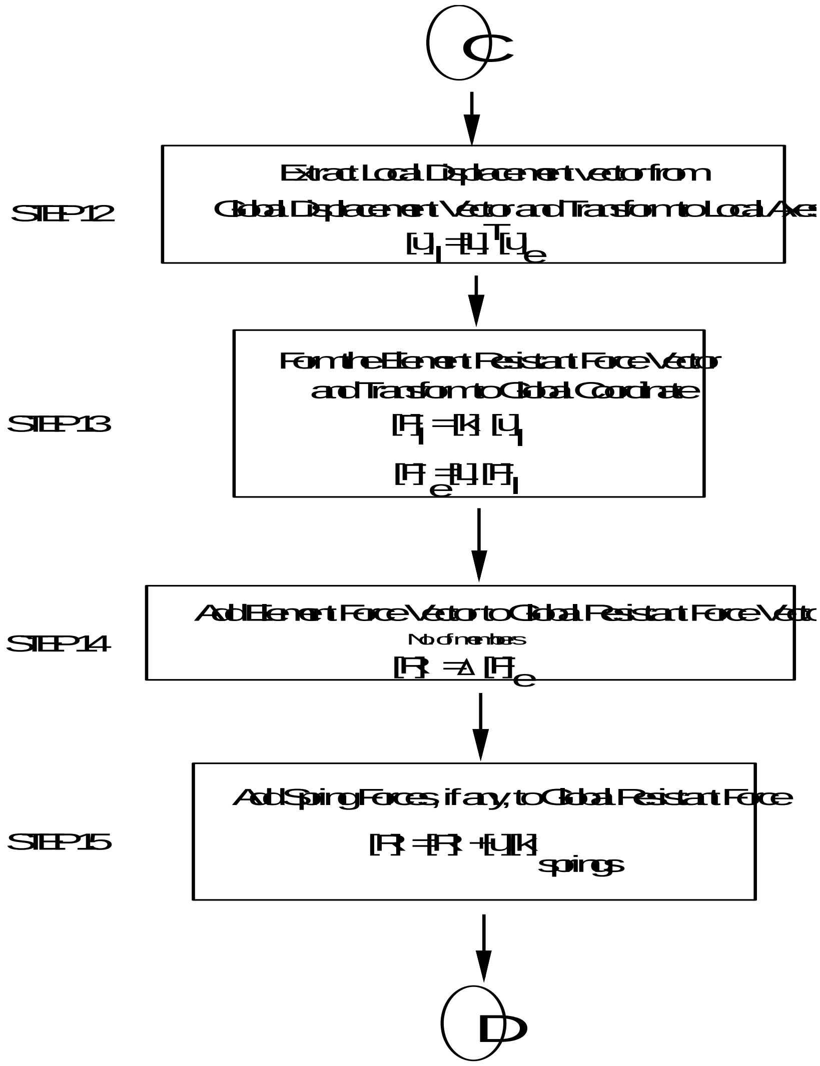
To indicate the end of the input file.

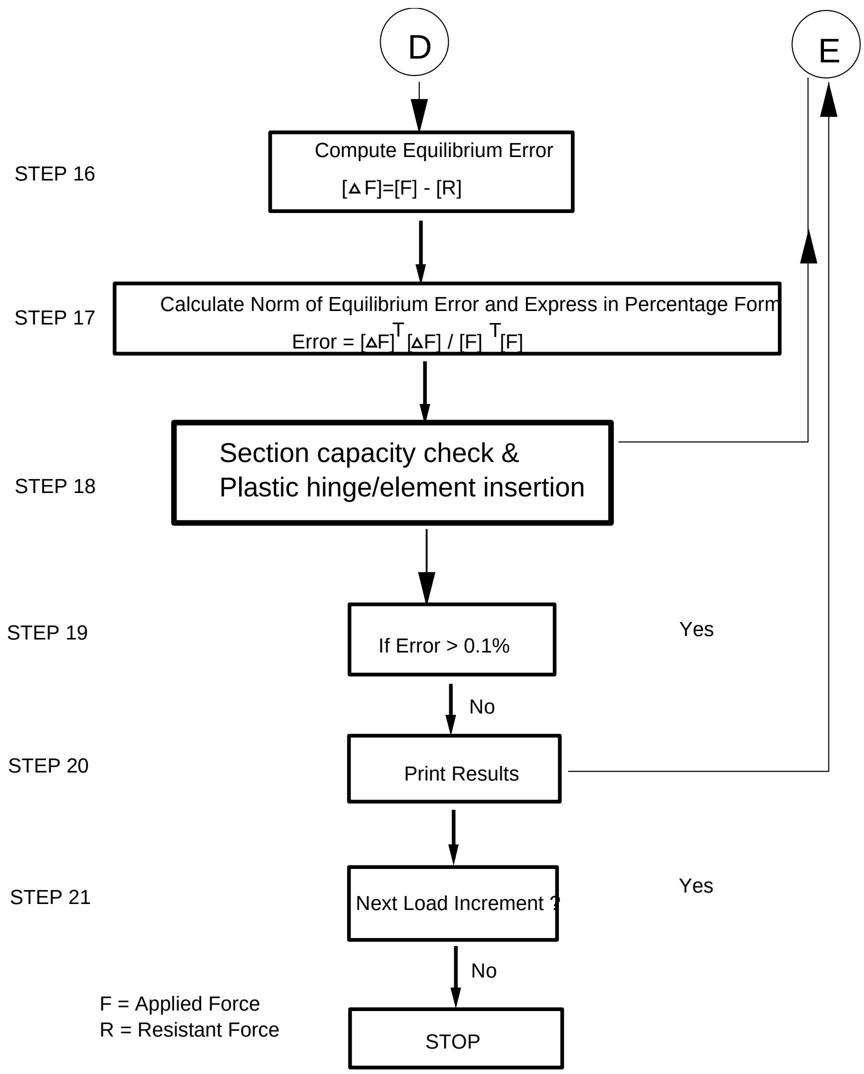
***END**

7. APPENDIX – FLOW CHART FOR ANALYSIS




B
F
STEP 7
Addition of Spring Stiffness from Supports
STEP 8
Impose Boundary Conditions
STEP 9
Apply Incremental Force Vector, [∆F]
STEP 10
Solve for Incremental Displacement
[∆u] = [K]T -1 [∆F]
STEP 11
Updating of Geometry
[X] i+1 = [X] i +[∆u]
Modified Newton
Raphson Method
C
E


C
STEP 12
Extract Local Displacement vector from
Global Displacement Vector and Transform to Local Axes
[u]l = [L]T [u]e
STEP 13
Form the Element Resistant Force Vector
and Transform to Global Coordinate
[F]l = [k] [u]l
[F]e = [L] [F]l
STEP 14
Add Element Force Vector to Global Resistant Force Vector
No. of members
[R] = Σ [F]e
STEP 15
Add Spring Forces, if any, to Global Resistant Force
[R] = [R] + [u][k]
springs
D


D
E
STEP 16
Compute Equilibrium Error
[ΔF]=[F] - [R]
STEP 17
Calculate Norm of Equilibrium Error and Express in Percentage Form
Error = [ΔF]T [ΔF] / [F] T[F]
STEP 18
Section capacity check &
Plastic hinge/element insertion
STEP 19
If Error > 0.1%
Yes
No
STEP 20
Print Results
STEP 21
Next Load Increment ?
Yes
No
F = Applied Force
R = Resistant Force
STOP

8. REFERENCES

Argyris, J.H. (1965), "Continua and Discontinua", Proceedings of 1st Conference on Matrix Methods in Structural Mechanics, Wright-Patterson AFB, Ohio, pp. 11-189.

Argyris, J., Haase, M., Mlejnek, H.P. and Schmolz, P.K. (1986), "Trunc for Shells - An Element Possibly to The Taste of Bruce irons", 1(22), pp.93-115.

Australian Standards AS4100, Steel Structures, Standards Association of Australia, Sydney, Australia 1998.

Bathe, K.J. and Dvorkin, E.N. (1983), "On the Automatic Solution of Nonlinear Finite Element Equations", Computers and Structures, 17(5-6), pp. 871-879.

Batoz, J.L. and Dhatt, G. (1979), "Incremental Displacement Algorithms for Nonlinear Problems", International Journal of Numerical Methods in Engineering, 14, pp. 1262-1267.

Bergan, P.G. and Soreide, T.H. (1978), "Solution of Large Displacement and Instability Problems using the Current Stiffness Parameter", In : Bergan, P.G., Finite Elements in Nonlinear Mechanics, Tapir, Trondheim, pp. 647-669.

BSI (1990). Structural use of steelwork in building – Part 1: Code of practice for design – Rolled and welded sections, BS5950. London; BSI

BSI (2000). Structural use of steelwork in building – Part 1: Code of practice for design – Rolled and welded sections, BS5950. London; BSI

Carpenter, N., Stolarski, H. and Belyschko, T., Improvements in 3-node triangular shell elements. Internal Journal for Numerical Methods in Engineering, Vol.23, No.9, pp.1643-1667, 1986

CEN (2005). Eurocode 3 Design of steel structures – Part 1-1: General rules and rules for building, BS EN 1993-1-1. London: CEN BSI

Chan, S.L. (1988), "Geometric and Material Non-linear Analysis of Beam-Columns and Frames using the Minimum Residual Displacement Method", International Journal of Numerical Methods in Engineering, 26, pp. 2657-2699.

Chan, S.L. and Ho. G.W.M. (1990), "A Comparative Study on the Nonlinear Numerical Algorithms", Proceeding of the Third International Conferences in Numerical

Chan, S.L. and Zhou, Z.H., "Second-order elastic analysis of frames using single imperfect element per member", Journal of Structural Engineering, American Society of Civil Engineers, vol. 121, no.6, June, 1995, pp.939-945.

Chan, S.L., “Non-Linear behaviour and design of steel structures”, invited review paper, Journal of Construction Steel Research, vol. 57, no.12, December, 2001, pp.1217-1232.

Chan, S.L. and Chan, S.T.P., “Proper second-order and advanced analysis of steel frames”, proceedings for annual seminar 2005, Joint Structural Division, 7 June 2005, Hong Kong Convention & Exhibition Centre, pp.53-66.

Chan, S.L. and Chui, P.P.T., “Non-linear static and cyclic analysis of steel frames with semi-rigid connections”, Elsevier, 2000, pp.336.

Chen, W.F. and Chan, S.L., Second Order Inelastic Analysis of Steel Frames using Element with Mid-span and End Springs, March, Vol.121, No.3, Journal of Structural Engineering, ASCE, 1995, pp. 530-541.

Cho, S.H. and Chan, S.L. (2005) . Practical second-order analysis and design of single angle trusses by an equivalent imperfection approach. Steel and Composite Structures (6)5: 443-458.

Code of Practice for Structural Use of Steel 2005. Buildings Department. 2005, Hong Kong SAR Government.

Crisfield, M.A. (1981), "A Faster Incremental / Iterative Solution Procedure that Handles Snap-Through", Computers and Structures, 13, pp. 55-62.

Engineering and Applications, University of Swansea, U.K., Published by Elsevier Applied Science, pp. 552-565.

Honecker, A. (1980), "Entwicklung Implementierung and Austestung von Losungsalgorithmen fur Gleichungssysteme mit Singular Werdender Funktionalmatrix", Diploma Thesis, Berlin.

Karamanlidis, D., Honecher, A. and Knothe, K. (1981), "Large Deflection Finite Element Analysis of Pre- and Post- Critical Response of Thin Elastic Frames", in Nonlinear Finite Element Analysis in Structural Mechanics, Edited by Wunderlich, W., Stein, E. and Bathe, K.J., Springer-Verlag, Berlin Heidelberg, N.Y.

Lay K.S., Shell finite element formulated on shell middle surface, March 1, 1994, Journal of Engineering Mechanics, Vol.119, No.10, 1993, pp. 1973-1992.

Neal, B.G., “The plastic methods of structural analysis”, Chapman and Hall, 1977.

Powell, G. and Simons, J. (1981), "Improved Iteration Strategy for Nonlinear Structures", International Journal of Numerical Methods in Engineering, 17, pp. 1455-1467.

Ramm, E. (1980), "Strategies for Tracing the Nonlinear Response Near Limit Points", Nonlinear Finite Element Analysis in Structural Mechanics, Proceedings of the Europe-U.S. Workshop, pp.63-89, Bochum, Germany, July 28-31.

Ramm, E. (1981), "Strategies for Tracing the Nonlinear Response Near Limit Points", Nonlinear Finite Element Analysis in Structural Mechanics, Edited by Wunderlich, W., Stein, E. and Bathe, K.J., Springer-Verlag, Berlin, pp. 63-89.

Riks, E. (1979), "An Incremental Approach to the Solution of Snapping and Buckling Problems", International Journal of Solids and Structures, 15, pp. 529-551.

Sabir, A.B. and Lock, A.C. (1972), "The Application of Finite Elements to the Large Deflection Geometrically Nonlinear Behaviour of Cylindrical Shells", Variational Methods in Engineering, Edited by Brebbia, C.A. and Tottenham, H., Southampton University Press, pp. 7/66-7/75.

Surana, K.S., Geometrically Nonlinear Formulation for Curved Shell Elements, International Journal for Numerical Methods in Engineering, Vol.19, No.4, pp.581-615, 1983.

Timoshenko, K.S., and Gere, J.M., Mechanics of Materials 3rd Edition, PWS-KWNT Publishing Company, Boston, 1990

Yang, Y.B. (1984), "Linear and Nonlinear Analysis of Space Frames with Non-Uniform Torsion Using Interactive Computer Graphics", Ph.D Thesis, Department of Structural Engineering, Cornell University.

Vogel, U., Calibrating frames. Stahlbau, 54, October, 1985, pp.295-311.

Wempner, G.A. (1971), "Discrete Approximations Related to Nonlinear Theories of Solids", International Journal of Solids and Structures, 7, pp. 1581-1599.

Williams, F.W., "An approach to the non-linear behaviour of the members of a rigid jointed place framework with finite deflection", Quarterly Journal of Mechanics and Applied Mathematics, vol. 17, 1964, pp.451-469.