FrameCE

User Manual

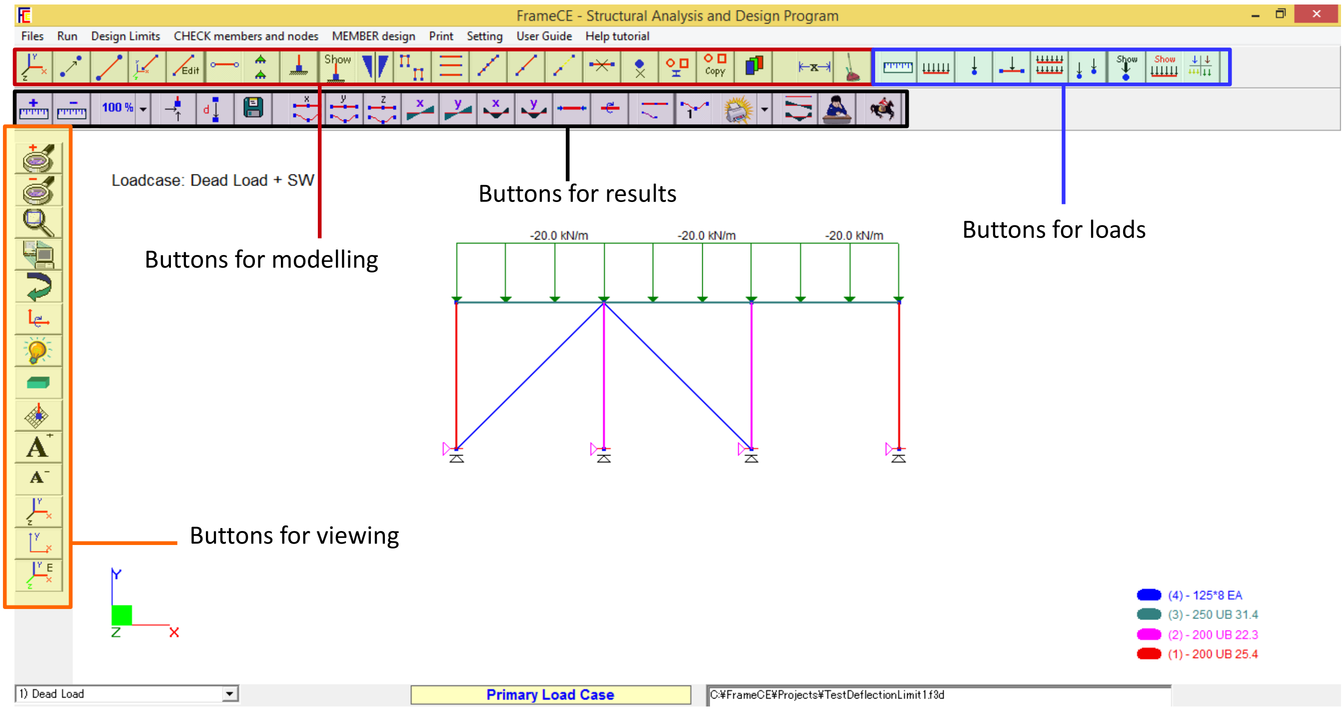
updated 18/06/2015

Table of Contents

FRAMECE MAIN MENU

FrameCE graphical user interface

Buttons in FrameCE are grouped together according to the type of command the each button is used for. The grouping of buttons as shown below.



Creating a project:

There are two ways of creating a project (see Figure 1.) These are **1) Creating a new project using Frame Wizard** and **2) Creating a free form 2D frame model**.

a) **Creating a new project using Frame Wizard –** This option allows the user to use ready made templates of frame configuration. However, since framing systems and configurations in most cases are unique, the templates are there to start the frame modelling but the user may have to edit and modify the models based on the actual structural configuration.

b) **Creating a free form 2D frame model** – If this option of creating a new project is selected, the user can create a 2D frame model by mouse-clicking anywhere inside the project window.

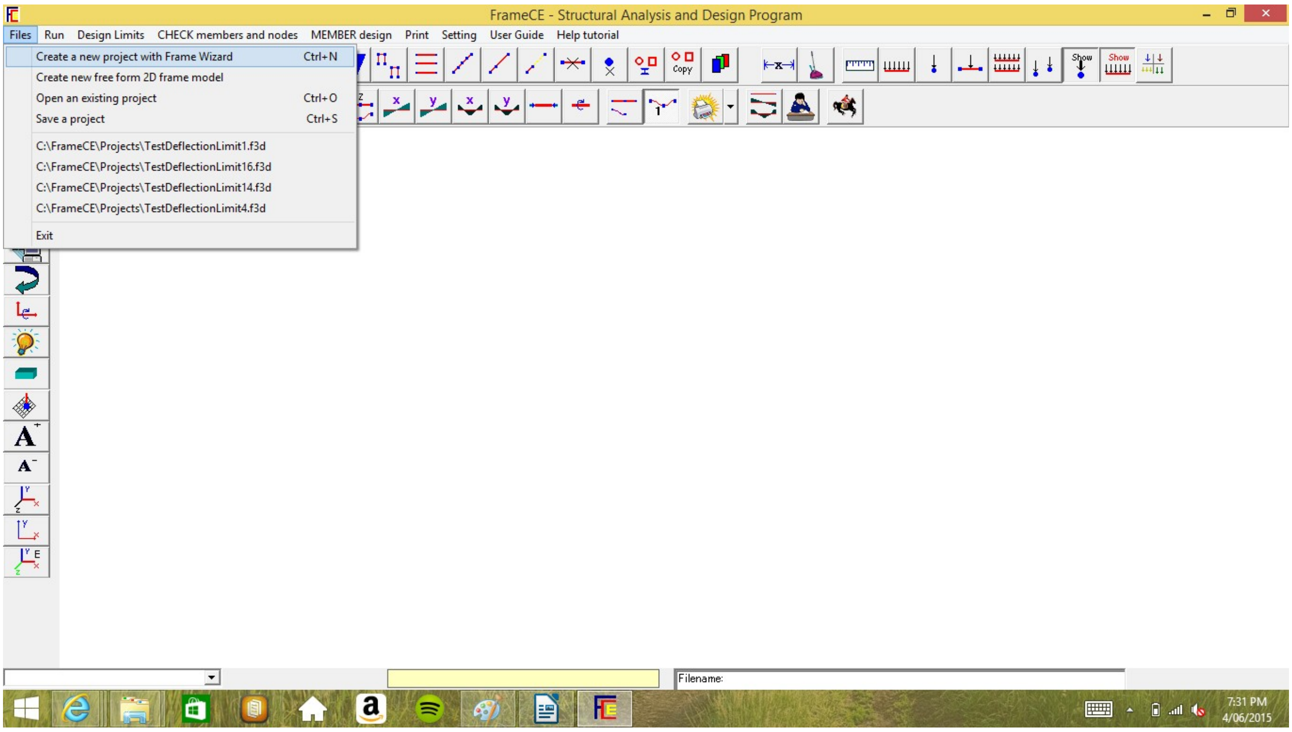


Figure 1: Creating a new project

Design Limits

FrameCE introduces a 'Design Limits' menu (Figure 2) wherein the user can specify the limiting values of deflection and displacements# (see Figure 3) to be used when FrameCE does the automated design calculation (i.e., section sizing).

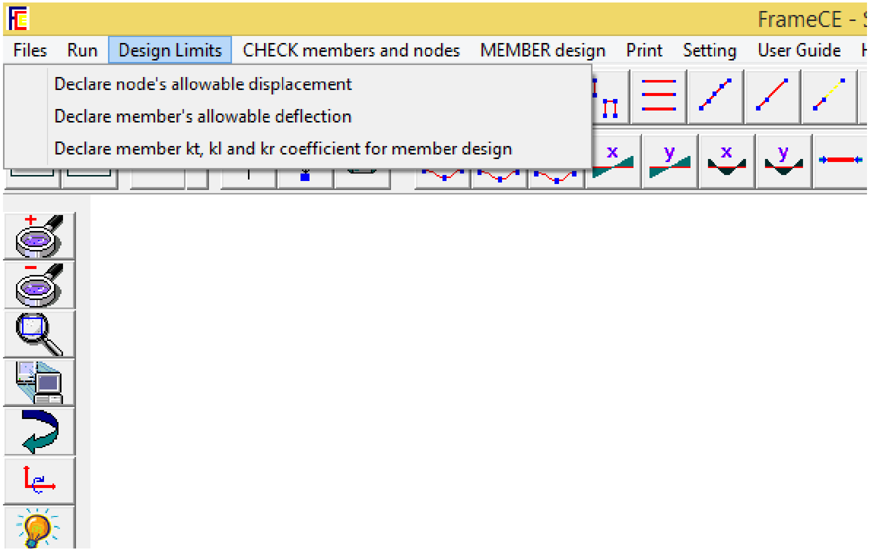


Figure 2: Design limits menu

Displacement and deflection have different meanings in FrameCE. Displacement refers to the change in coordinates of a node before and after the application of loads, whereas deflection refers to the difference in displacements between an element's end nodes and internal points.

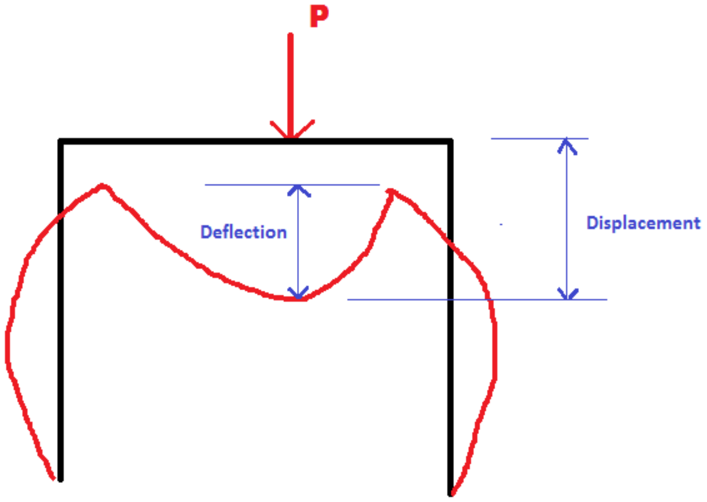


Figure 3: FrameCE definition of 'displacement' and 'deflection'

In Design Limit menu, there are three sub-menus. Each sub-menu is explained below:

a) **Declare node's allowable displacements** – in this sub-menu the maximum nodal displacement is declared by the user. Only nodes with declared allowable displacement will be checked for displacements. This declaration will be used by the program to alert the user which nodes exceeds the allowable displacement values but *is not used in the design of members*.

b) **Declare member's allowable deflection**- in this sub-menu the user can declare the maximum deflection for each member. (*Note how FrameCE defines deflection*). Allowable deflection value for each member is used in the design of member sections.

c) **Declare kt, kl and kr coefficient for member design –** in this sub-menu the coefficients are used by the software to calculate the member effective length.

Checking members and nodes

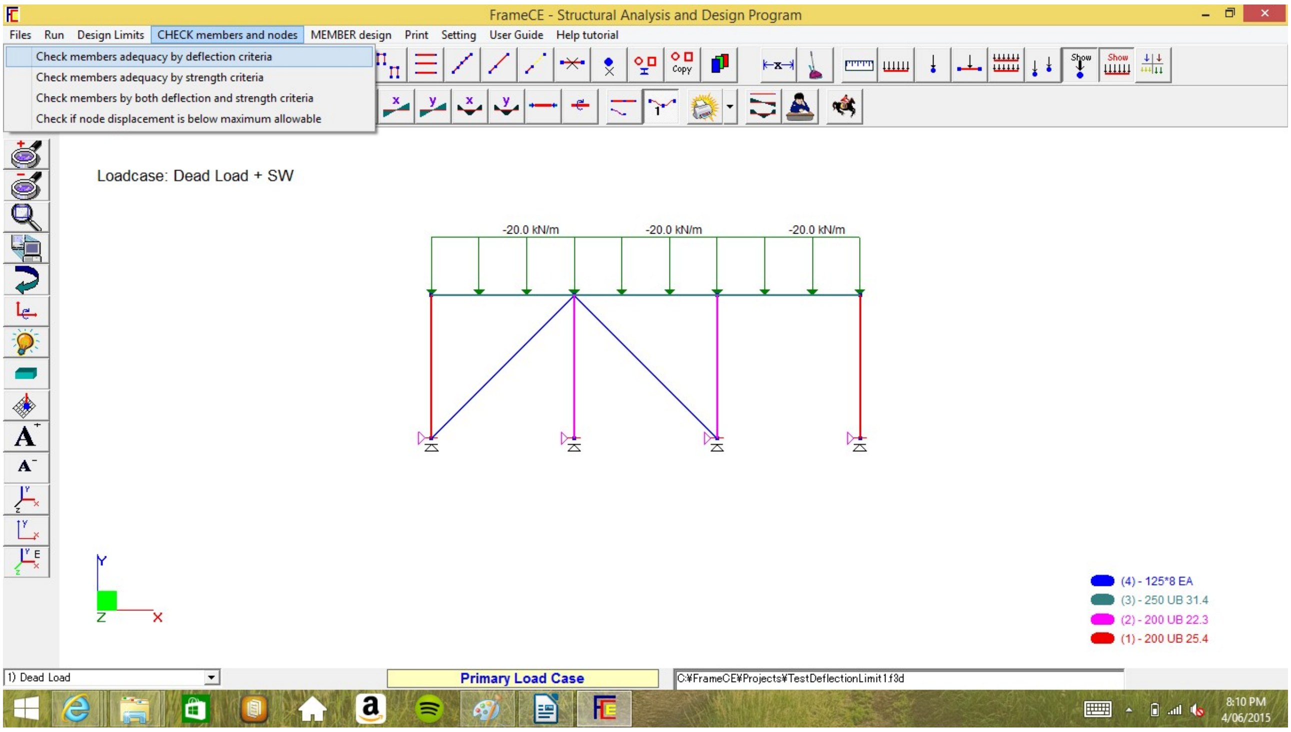


Figure 4: Checking members and nodes menu

This menu investigates the members and/or nodes based on the criteria given below:

a) **Check members adequacy by deflection criteria** – the software will indicate which members fail and which members are safe based on deflection criteria declared in *Design Limits menu.*

b) Check members adequacy by strength criteria – the software will indicate which members fail and which members are safe based on strength criteria.

c) Check members by both deflection and strength criteria - the software will indicate which members fail and which members are safe based on deflection and strength criteria.

d) C**heck if node displacement is below maximum allowable** - the software will indicate which nodes fail and which nodes are safe based on the maximum allowable displacement criteria declared in *Design Limits menu.*

Member section design

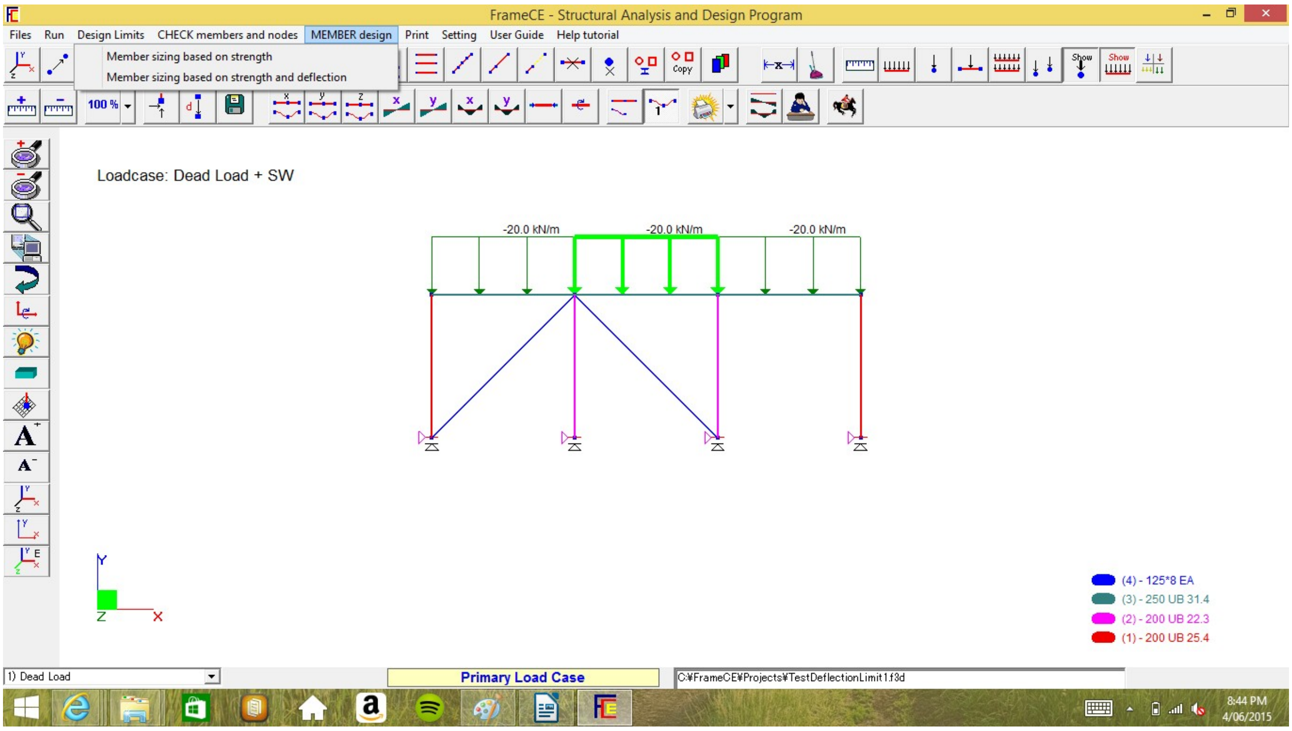


Figure 5: Member design menu

In this menu the frame members can be designed based on AS4100 (Australian standard). There are two sub-menus namely a) *Member sizing based on strength* and b) *member sizing based on strength and deflection*. When designing using b), the allowable deflection for each member is the value the user declared in menu "**Design Limits**". Note that the deflection referred to is the member deflection (see Figure 3) and not the global node displacement. Thus the user must check separately if the global node displacement is within allowable values using the menu "**Check member and nodes**".

Printing

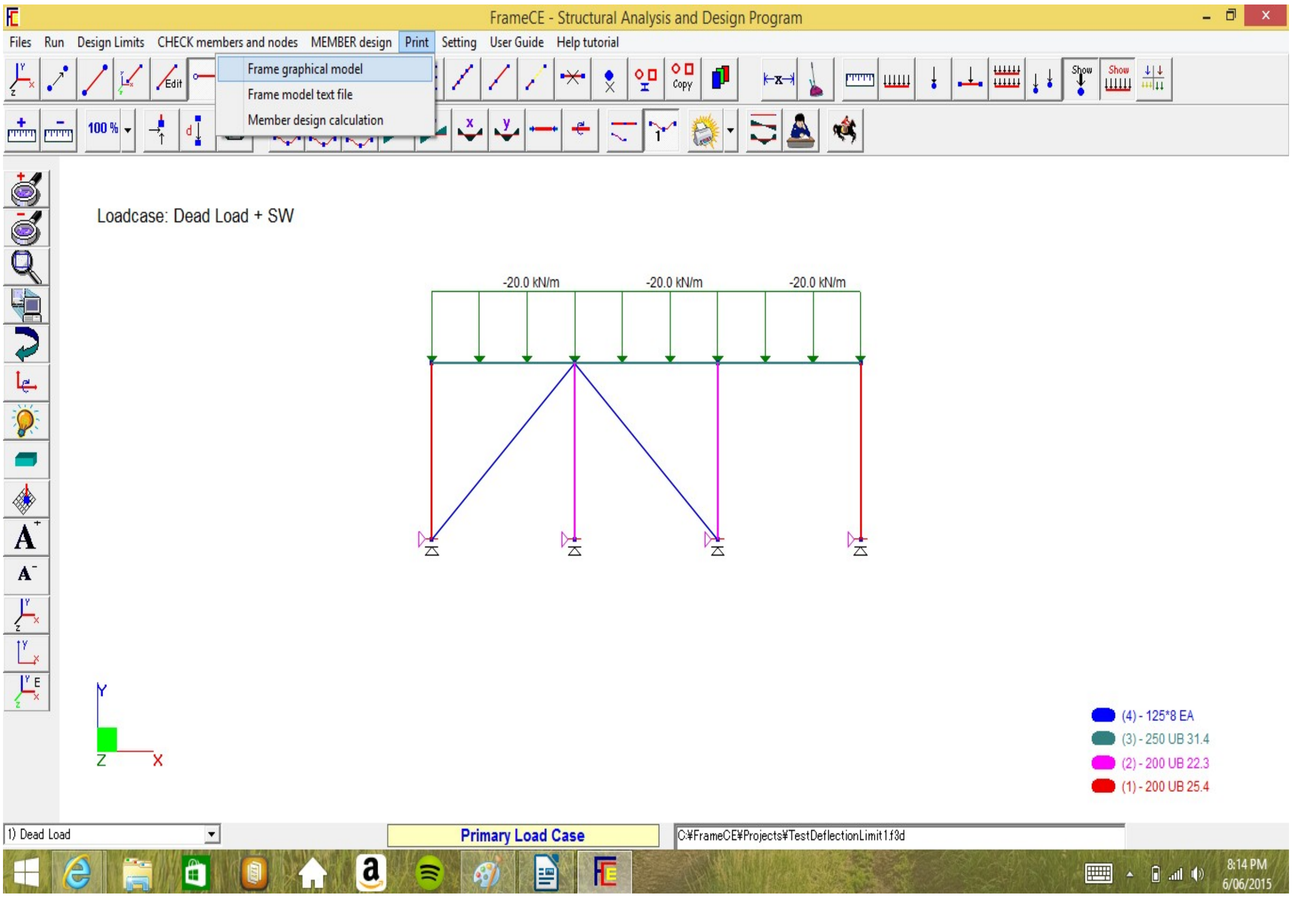


Figure 6: Printing menu

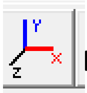
There are three sub-menus for printing, these are

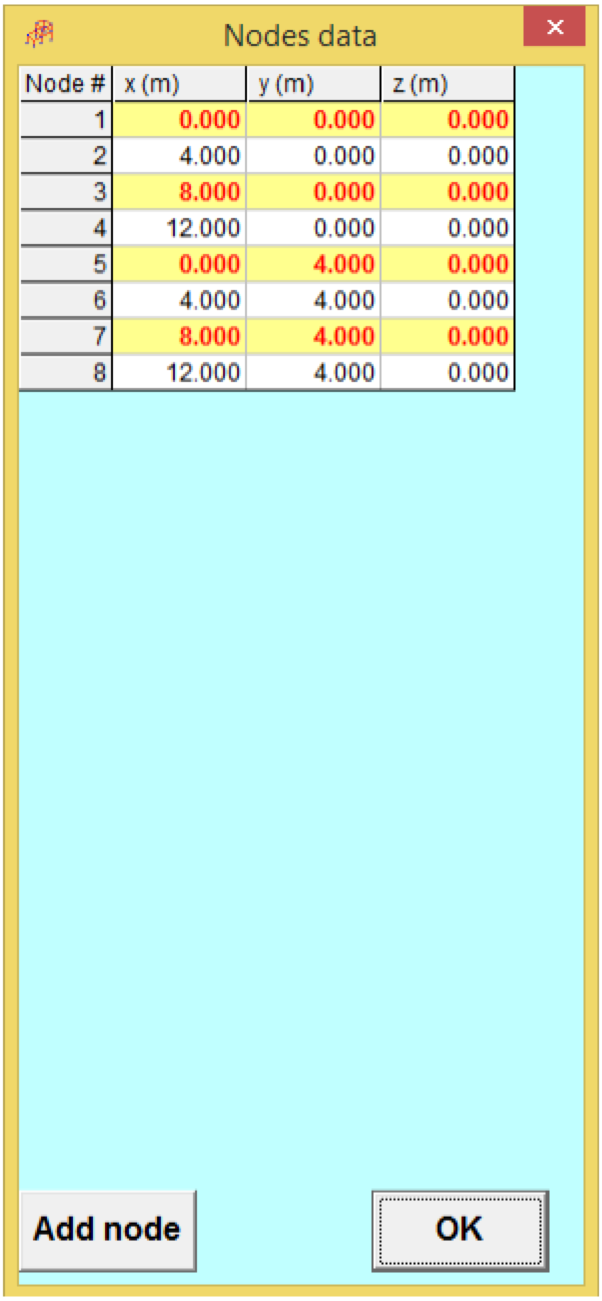
1. **Frame graphical model** - prints the stick model of the frame
2. **Frame model text file** – prints the numerical model of the frame
3. **Member design calculation** – prints the design calculation for each member or members designated for printing. When printing design calculation, the program will prompt the user to indicate which member calculations are to be printed. The calculation is presented for each load case the user identified as the criteria for design. Therefore, for each member, may have multiple pages of design calculation but the section adapted in each member is consistent for all load cases.
4. I*n many instances the user has preferences for member sections to be used in actual construction. In this case the user must edit the section property of these frame members before printing. Before the program prints the calculation, it will investigate each member*

and will alert the user if the preferred sections fail in the design criteria. Otherwise, the program will proceed in printing the calculations.

CREATING THE MODEL

Creating nodes by declaring its global coordinates

Step 1: Click button.  Dialog box shown in Figure 7 will appear.

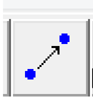


Node #	x (m)	y (m)	z (m)
1	0.000	0.000	0.000
2	4.000	0.000	0.000
3	8.000	0.000	0.000
4	12.000	0.000	0.000
5	0.000	4.000	0.000
6	4.000	4.000	0.000
7	8.000	4.000	0.000
8	12.000	4.000	0.000

Figure 7

Step 2: Click 'Add node' button. Declare each node coordinates in the gird boxes provide. Click 'OK' to finish.

Creating nodes by declaring its coordinates relative to an existing node.

Step 1: Click button.  Dialog box shown in Figure 8 will appear.

Step 2: Click the reference node. The coordinate of the reference node will appear as shown in Figure 9. Declare the distance of the node to be created in the input boxes dx,dy, and dz. Click 'Apply' to create the node. Repeat **Step 2** to create more nodes, otherwise Click 'Exit' button.

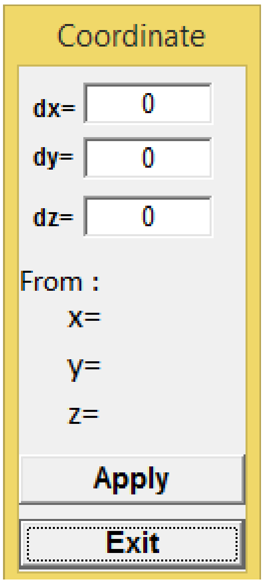


Figure 8

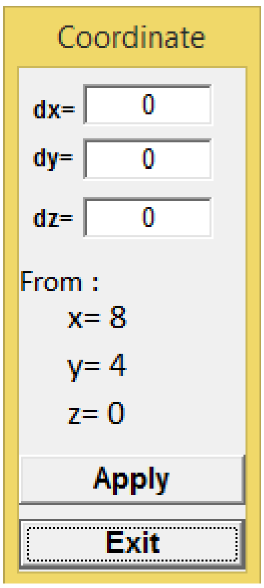
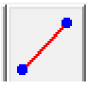


Figure 9

Creating frame elements with existing nodes

Step 1: Click button.  Figure 10 will appear.

Step 2: Click the two nodes that defines the endpoints of the element. Select 'Member Type', section and declare releases (if applicable).

Step 3: Right mouse click to stop the element creation.

Step 4: Click 'Exit' button.

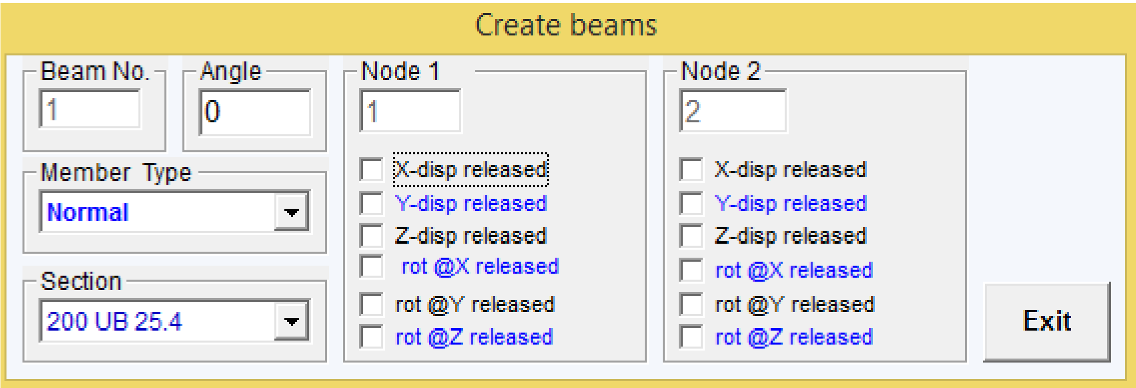
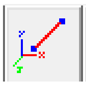


Figure 10

Creating frame elements with one endpoint from existing node

Step 1: Click button.  Figure 11 will appear.

Step 2: Click an existing node and declare distances dx, dy and dz. Select member section from drop-down menu.

Step 3: Click 'Apply' button. Repeat Step 2 to create more elements. Otherwise Click 'Exit' to finish.

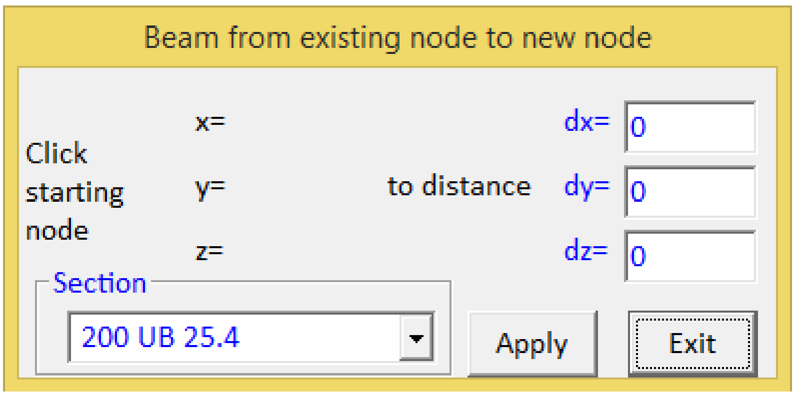


Figure 11

Edit an existing frame elements

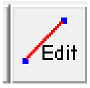
Step 1: Click button. 

Figure 12 will appear.

Step 2: Select (i.e.,. mouse click) one or more elements to be revised.

Step 3: Select Member Type, section and releases (if applicable).

Step 4: Click 'Apply' to finalise the revision. Repeat Step 2 to edit more elements. Otherwise, click 'Exit'.

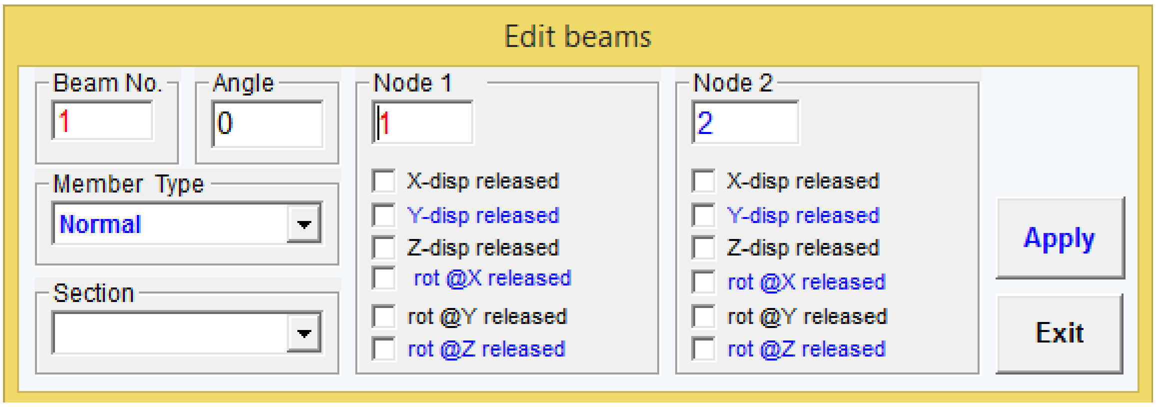


Figure 12

Editing existing frame elements using the ***right mouse click***

Step 1: Click the element or elements you want to edit

Step 2: Click the right button of the mouse. Figure 13 will then pop-up.

Step 3: Select from the menu options the procedure you want to perform.

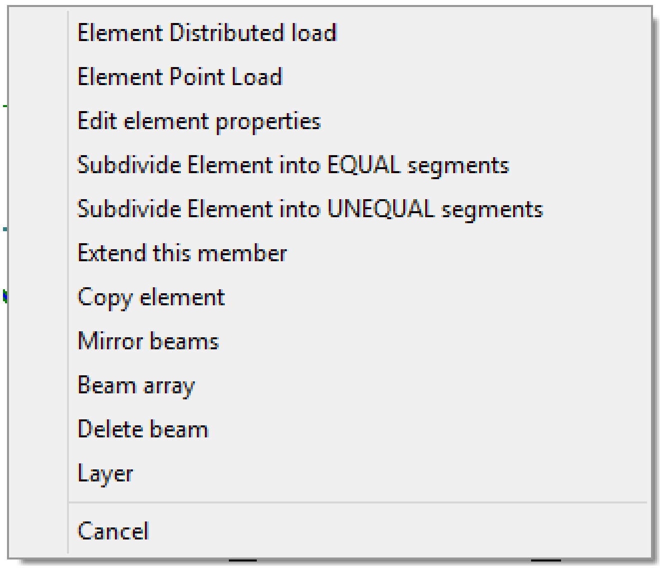
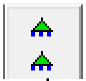


Figure 13

Copying node properties

Node properties are the fixity condition (i.e., *free, fixed or flexible (spring)*).

Step 1: Click button.  Figure 14 will appear.

Step 2: Click the node to be copied (i.e., source node). (After clicking the source node Figure 14 will change to Figure 15). The change from Figure 14 to Figure 15 indicates that you are now in the 'clicking destination node' mode as shown in the clicked 'Clicking destination nodes' button.

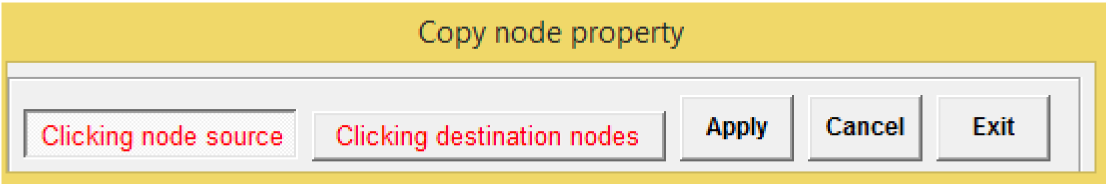


Figure 14

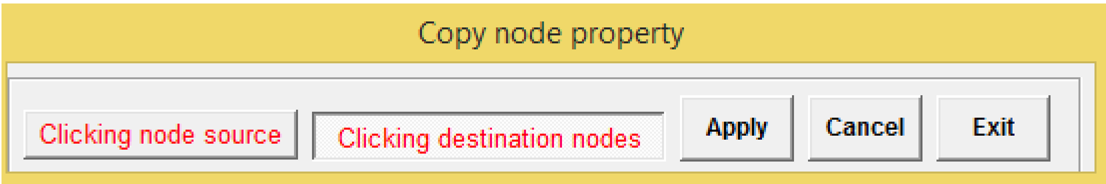
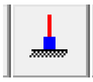


Figure 15

Step 3: Click the destination nodes. (Destination nodes are the nodes that receive the property of the 'sour node').

Step 4: Click 'Apply' button.

Edit or Declare node properties

Step 1: Click button.  Figure 16 will appear.

Step 2: Select one or more nodes. Declare the node properties by completing the input boxes.

Step 3: Click 'Apply' button to finalise. Repeat step 2 to edit other nodes, otherwise click 'Exit' button.

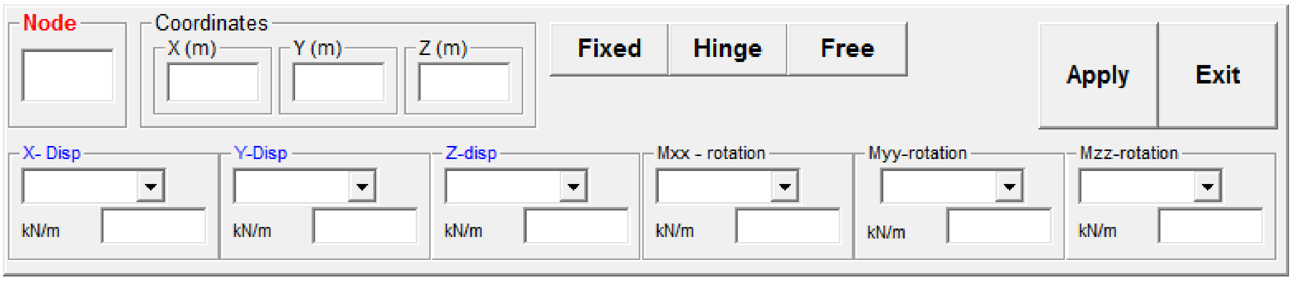


Figure 16

Edit existing nodes using the ***right mouse click***

Step 1: Click the node or nodes you want to edit

Step 2: Click the right button of the mouse. Figure 17 will then pop-up.

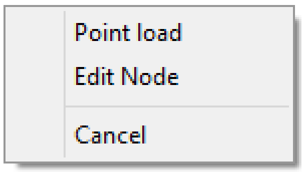


Figure 17

Step 3: Select from the menu options the procedure you want to perform.

Edit an existing frame element by <u>double *clicking*</u>

When an element is double-clicked, Figure 18 will then pop-up and you can edit that element

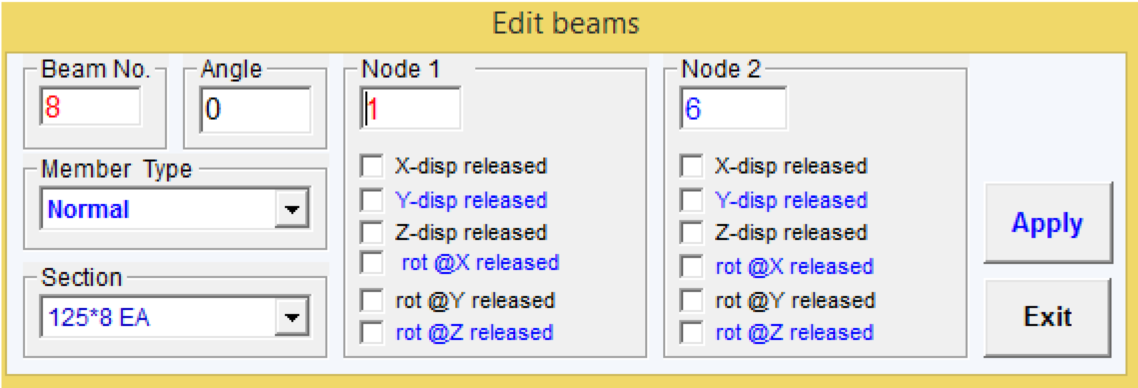


Figure 18

Edit a node by <u>double *clicking*</u>

When a node is double-clicked, Figure 19 will then pop-up and you can edit the that node.

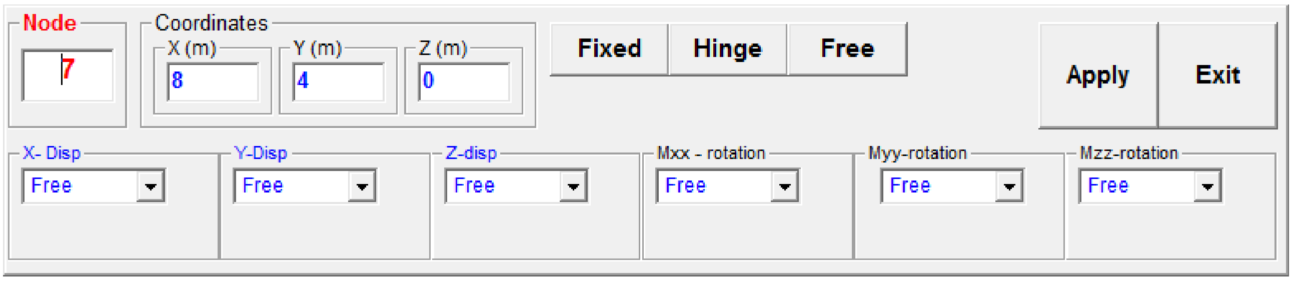
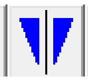


Figure 19

Create elements using coordinate symmetry.

This command will create frame elements symmetrical to existing elements.

Step 1: Click button  Figure 20 will appear. *(Note that the button 'Element to reproduce by symmetry' is in clicked mode. This means that this is the current command.)*

Step 2: Click the elements to be copied (i.e., to be reproduced by symmetry)

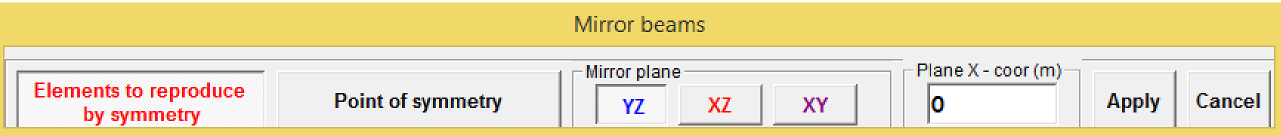


Figure 20

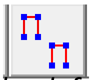
Step 3: Select the mirror plain. (*YZ plane is the default*).

Step 4: Click 'Point of Symmetry' button

Step 5: Click a node. *(This node is the point of symmetry where the mirror plane passes through).*

Step 6: Click 'Apply' button

Copy / Clone elements

Step 1: Click button  Figure 21 will appear. (*Note that button 'Elements to be copied' is in clicked mode by default. This means that the current command is to select the elements to be copied).*

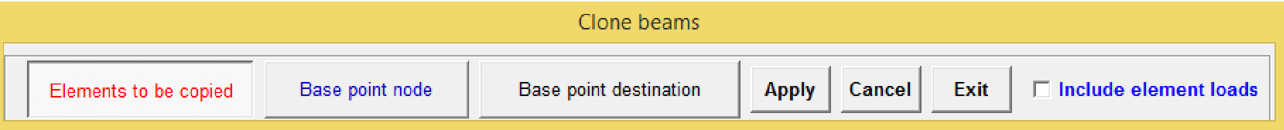


Figure 21

Step 2*:* Click the frame elements to be copied.

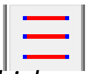
Step 3*:* Click the button 'Basepoint node'.

Step 4*:* Click the *base node*. *(The base node is the reference node of all the elements to be copied. In simple terms, this is the point where you hold all the elements to be copied)*

Step 5*:* Click the button 'Basepoint node destination'.

Step 6. *Click the 'Apply' button.*

Create multiple copies of an element

Step 1: Click button  Figure 22 will appear. (*Note that the button along 'x' axis is clicked by default which means the array is along the x-axis*).

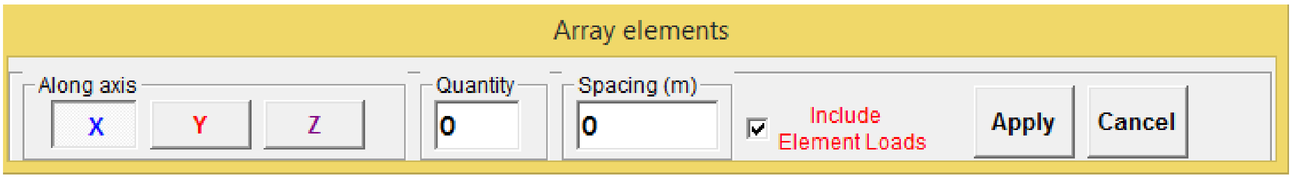


Figure 22

Step 2*:* Select the direction of the array (i.e., x, y or z axis).

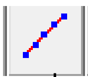
Step 3*:* Declare the quantity (i.e., array size).

Step 4*:* Declare the array spacing.

Step 5: Indicate if element loads are to be copied as well (check = Yes, uncheck=No)

Step 6: Click 'Apply' button.

Subdivide element(s) into equal segments

Step 1: Click button  Figure 23 will appear.

Step 2: Click the frame elements to be subdivided.

Step 3: Declare the number of segments in the input box 'n='

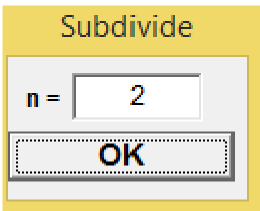
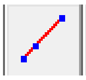


Figure 23

Step 3: Click 'OK' button

Subdivide element(s) into two un-equal segments

Step 1: Click button  Figure 24 will appear.

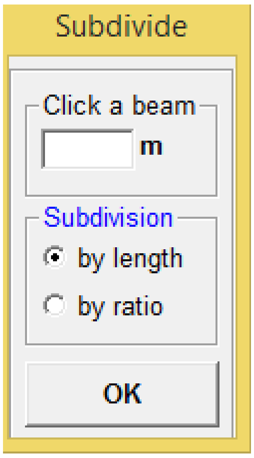


Figure 24

Step 2: Click the frame elements to be subdivided.

Step 3: Declare subdivision type (*by length or by ratio*)

Step 4a: If by length, declare the length of the segment from the first node (*node number is indicated)*

Step 4b: If *by ratio*, declare the percentage of one segment from the first node (*node number is indicated*) as shown in Figure 25

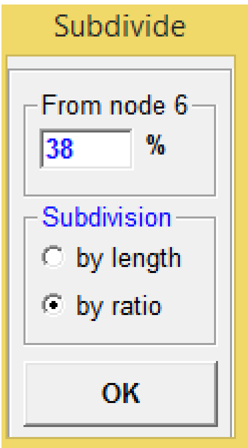
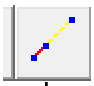


Figure 25

Step 3: Click 'OK' button

Extend an existing element

Step 1: Select the element to be extended

Step 2: Click the button 

Step 3: Click the end node where the element is to be extended. Figure 26 will appear.

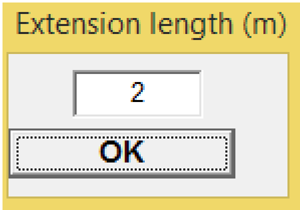


Figure 26

Step 4: Declare extension length in the box provided.

Step 5: Click 'OK' button

Delete an element(s)

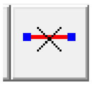
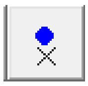
Step 1: Click button  Figure 27 will appear.

Figure 27

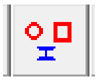
Step 2: Click the elements to be deleted.

Step 3: Click 'Yes' button.

Delete unused nodes

Although the presence of unused nodes does not affect the analysis results, it may slow down the calculation. The user can choose to delete this unused nodes by clicking the button 

Add material sections library in the current project

Step 1: Click button  (Figure 30 will then appear.)

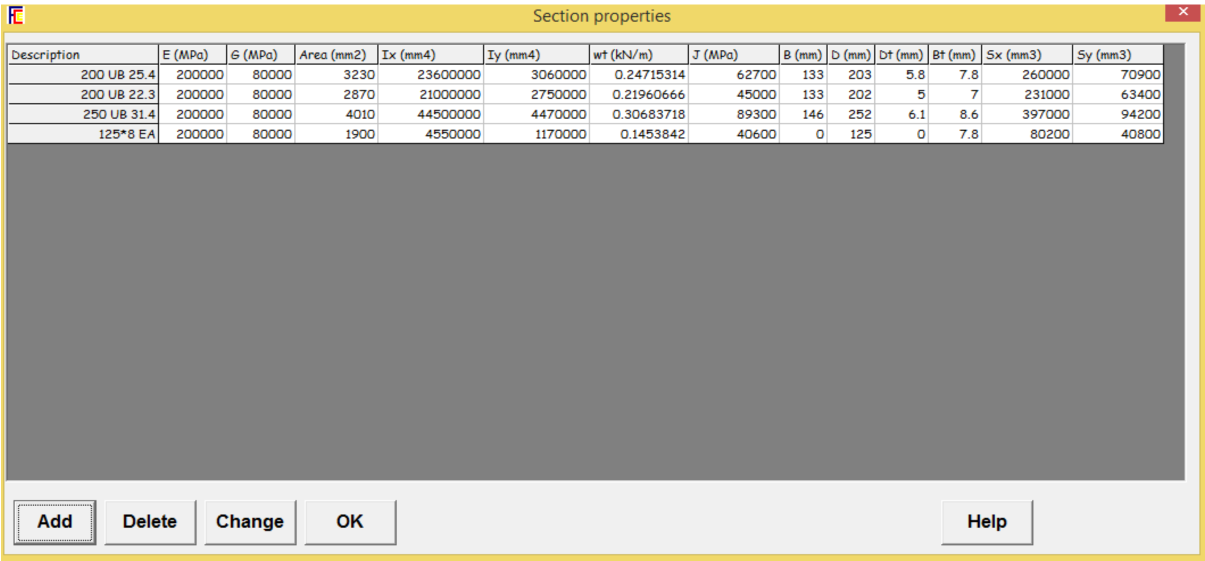


Figure 28

Step 2: Click 'Add' button. (Figure 29 will then appear.)

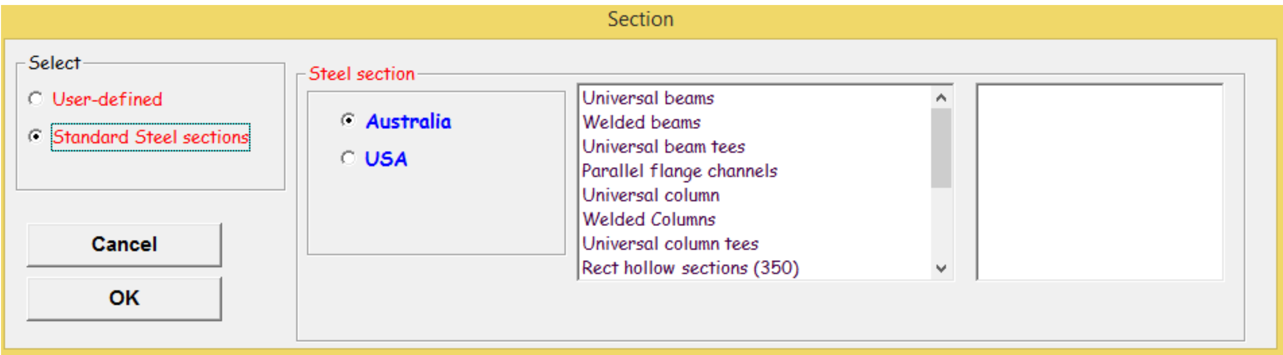
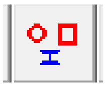


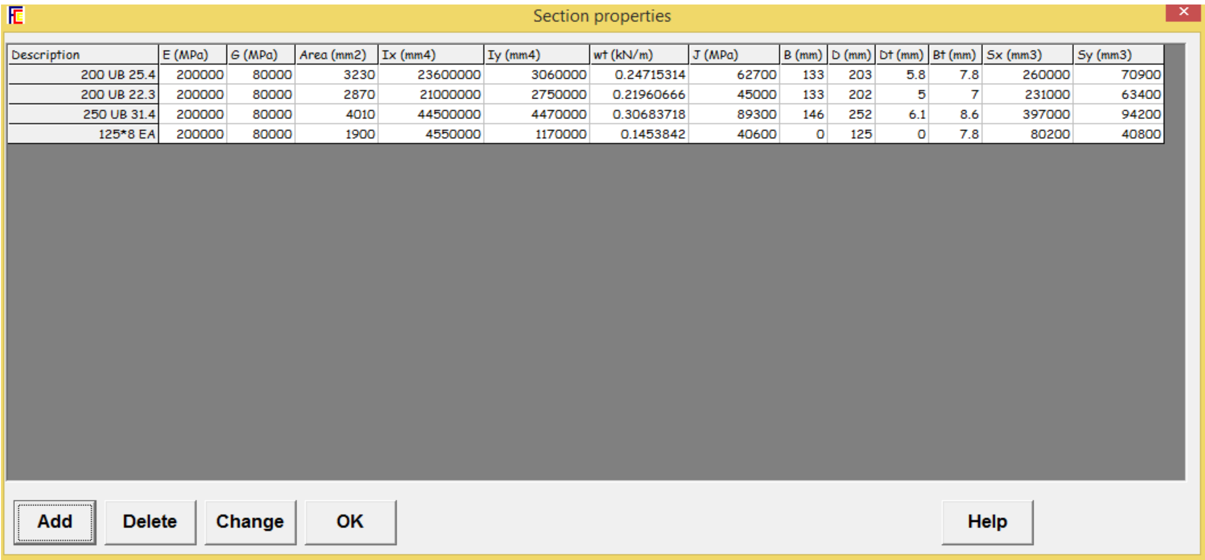
Figure 29

Step 3: Select the appropriate options indicated in Figure 29

Step 4: Click 'OK' button

Delete material sections in the current project

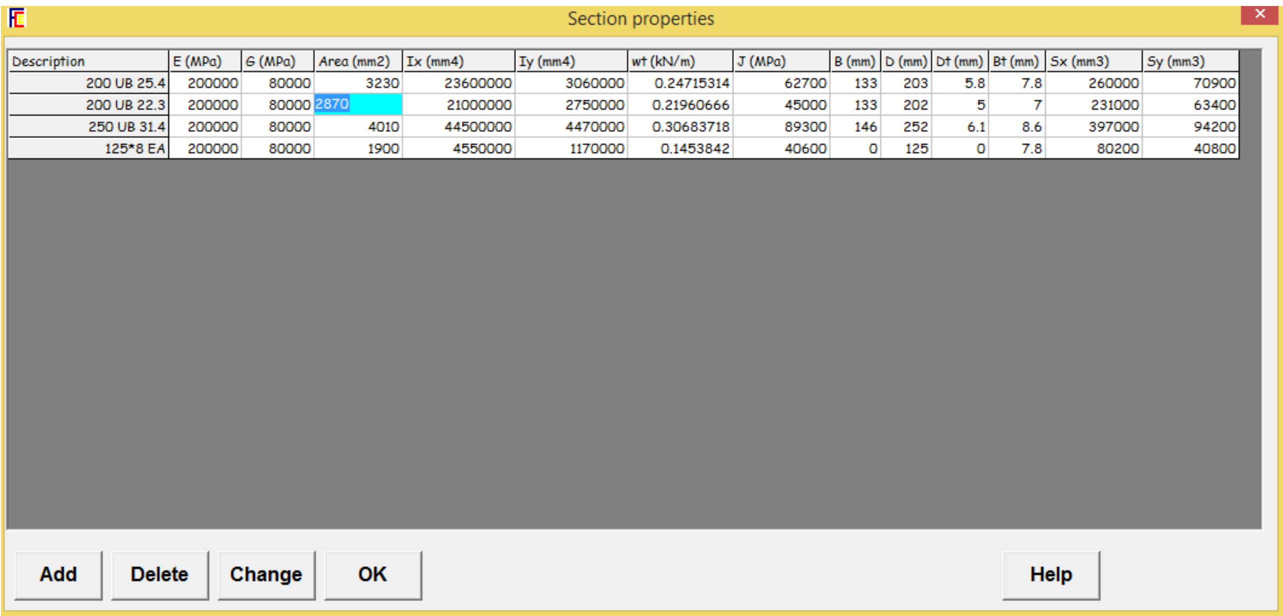
Step 1: Click button  Figure 27 will appear.



Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
200 UB 22.3	200000	80000	2870	21000000	2750000	0.21960666	45000	133	202	5	7	231000	63400
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Figure 30

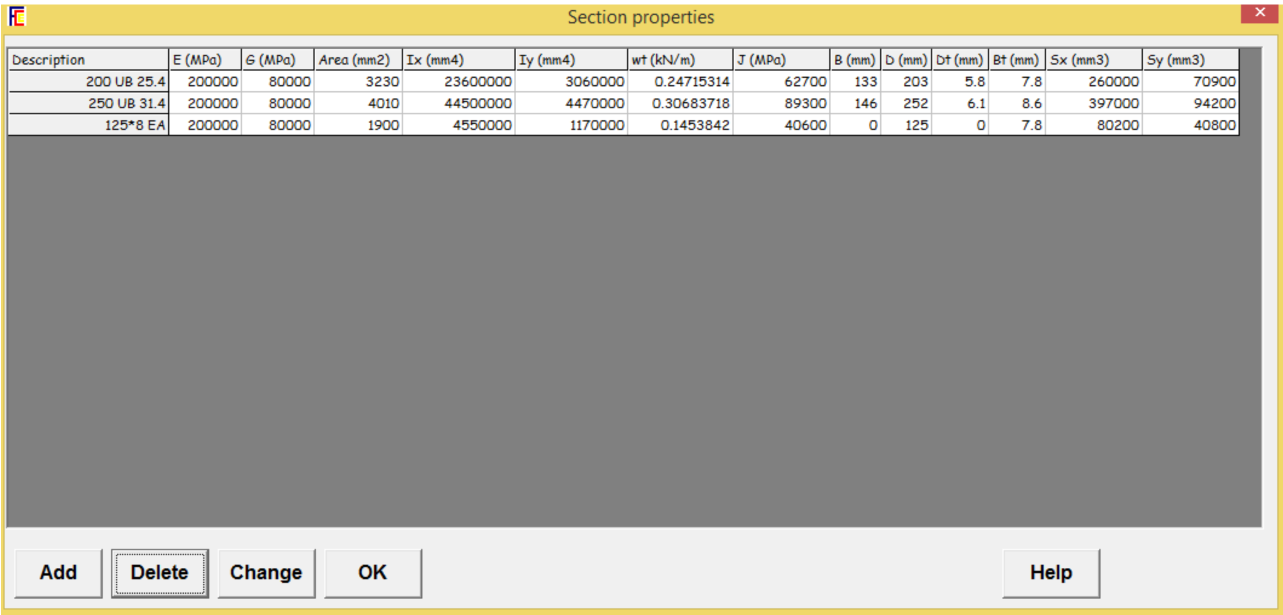
Step 2: Click one cell of the material section to be deleted. (In this example, the 200UB23 is to be deleted. One of the cells (i.e., highlighted by light blue colour) is clicked as shown in Figure 31.



Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
200 UB 22.3	200000	80000	2870	21000000	2750000	0.21960666	45000	133	202	5	7	231000	63400
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Figure 31

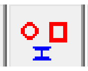
Step 3: Click 'Delete button'. The new material section library will now be without the deleted section as shown in Figure 32

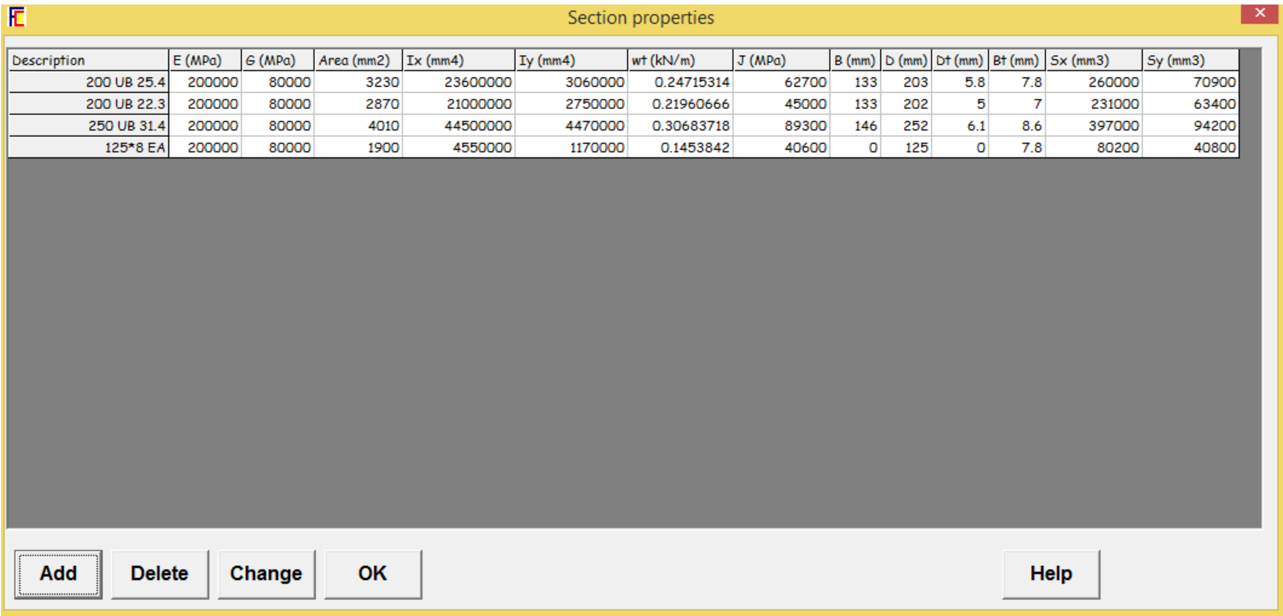


Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Figure 32

Change the material sections in the current project

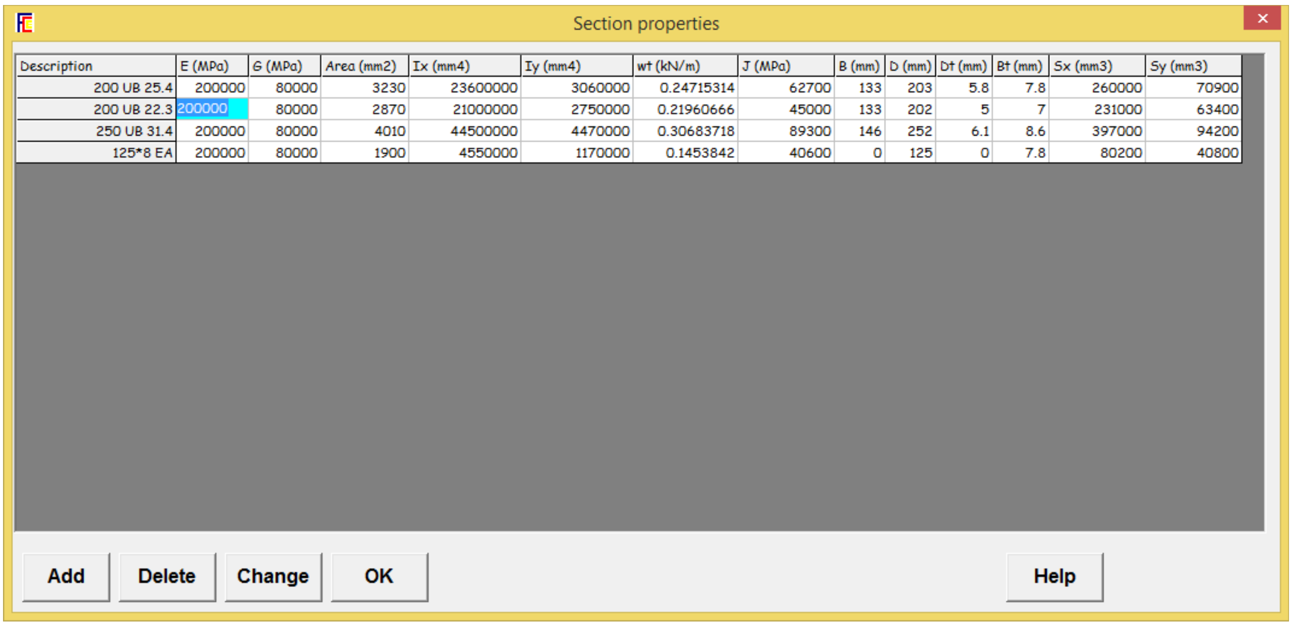
Step 1: Click button  (Figure 33 will then appear.)



Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
200 UB 22.3	200000	80000	2870	21000000	2750000	0.21960666	45000	133	202	5	7	231000	63400
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Figure 33

Step 2: Click one cell of the material section to be changed. (In this example, the 200UB23 is to be changed to 150PFC. One of the cells (i.e., highlighted by light blue colour) is clicked as shown in Figure 34



Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
200 UB 22.3	200000	80000	2870	21000000	2750000	0.21960666	45000	133	202	5	7	231000	63400
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Figure 34

Step 3: Click 'Change' button. (Figure 35 Will then appear)

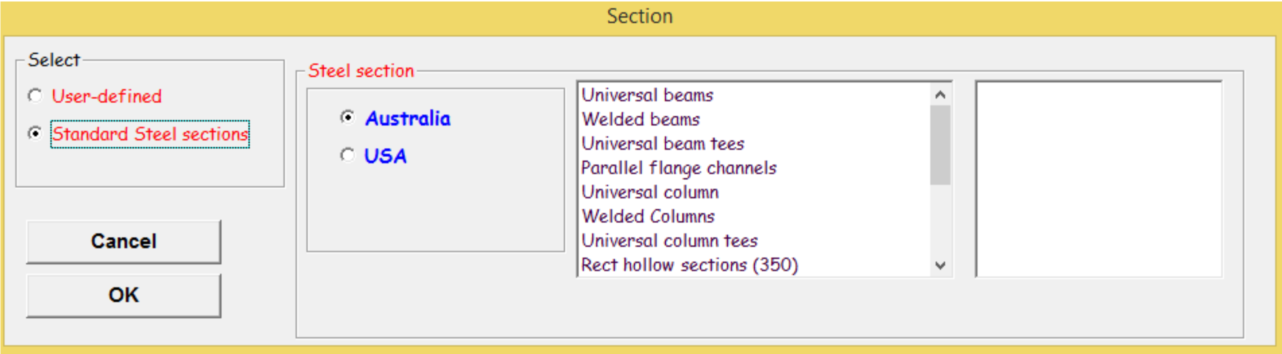


Figure 35

Step 3: Select 'Parallel flange channels' then '150PFC'.

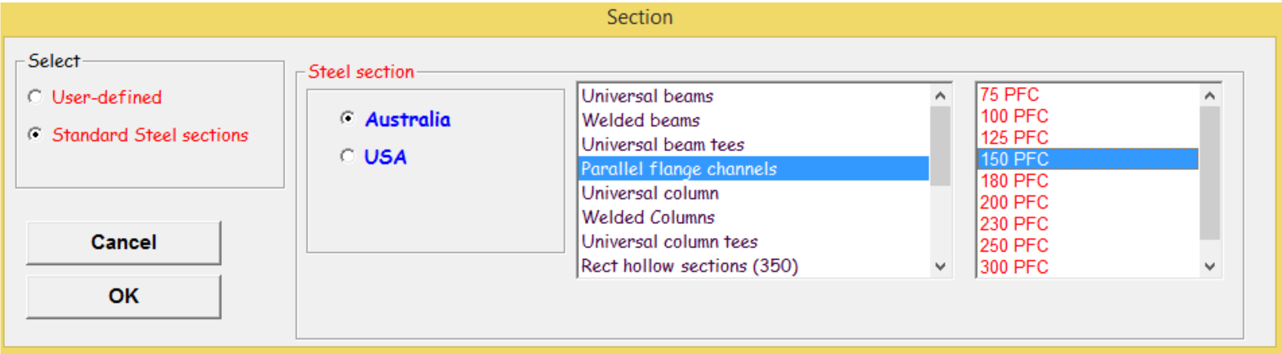
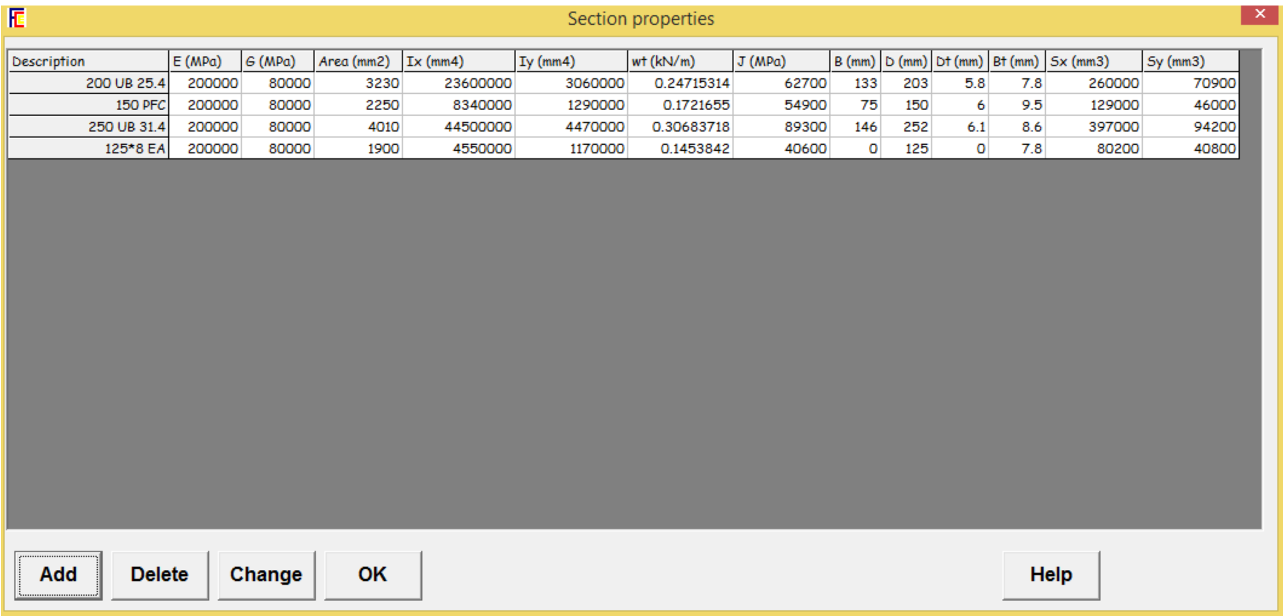


Figure 36

Step 4: Click 'OK' button. Figure 37 will then appear with the '200UB23' replaced with '150PFC'



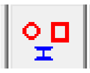
Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
150 PFC	200000	80000	2250	8340000	1290000	0.1721655	54900	75	150	6	9.5	129000	46000
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

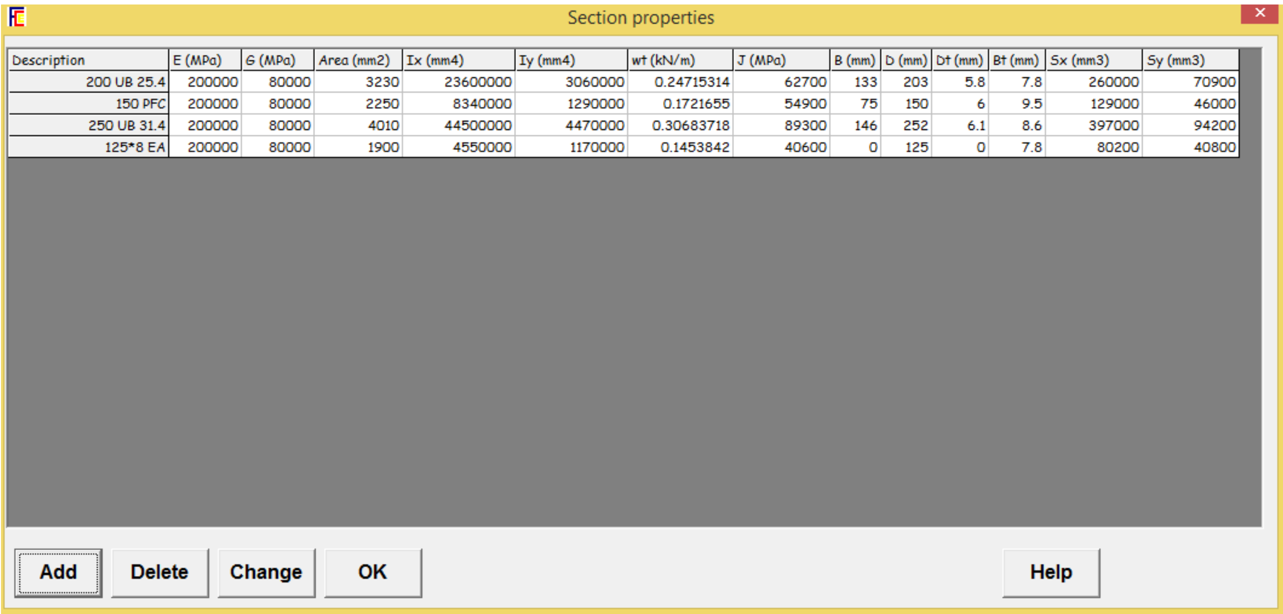
Figure 37

Step 4: Click 'OK' button to exit.

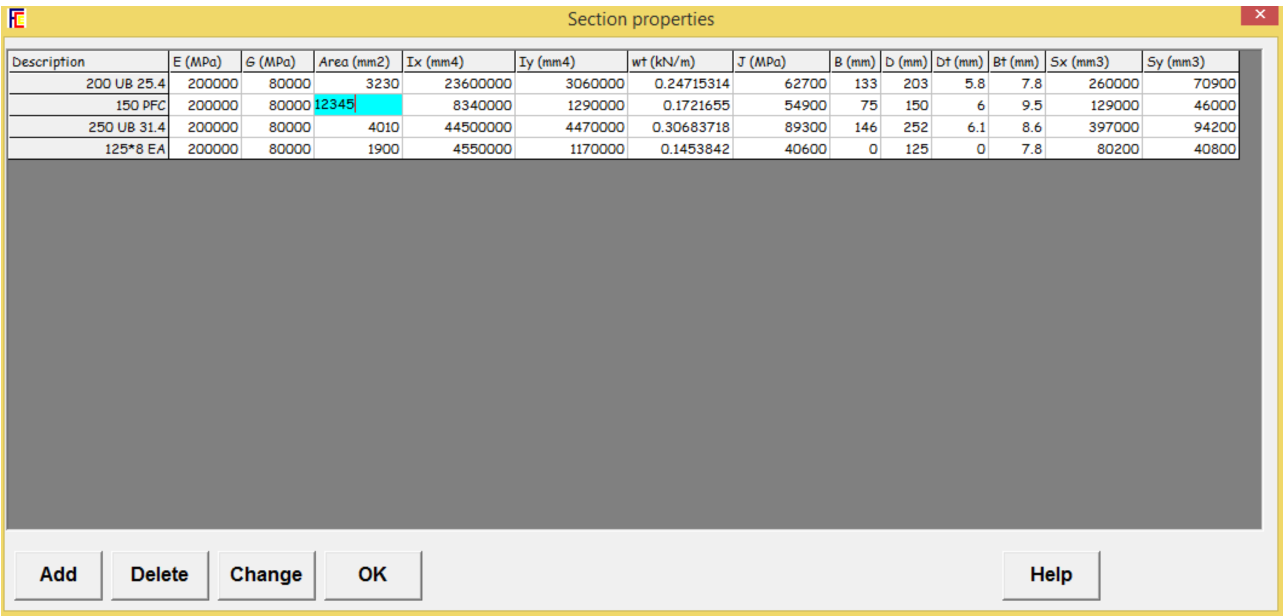
Edit values in the material sections of the current project

In some cases, especially in the user-defined sections, values to section properties needs some revision. The procedure is outlined below.

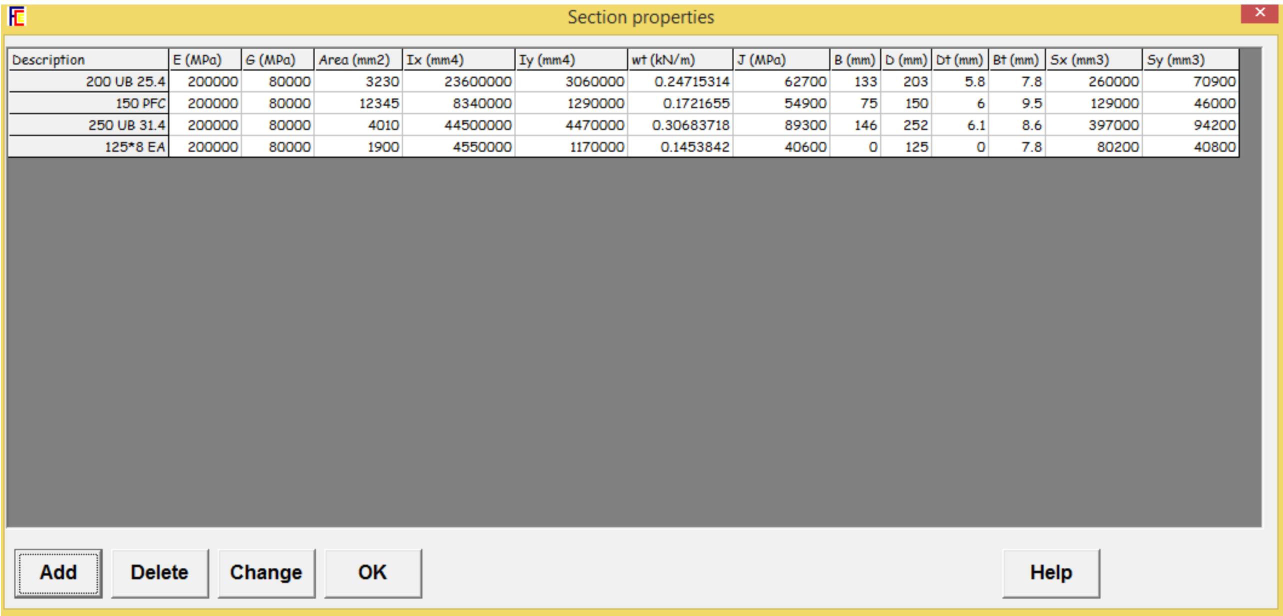
Step 1: Click button  (Figure 38 will then appear.)



Step 2: Select any cell and edit the value. (Example the area of 150PHC is changed to 12345 as shown in Figure 39

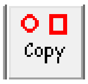


Step 3: Click 'OK' button. The new section properties will appear as shown in Figure 40


Section properties

Description	E (MPa)	G (MPa)	Area (mm2)	Ix (mm4)	Iy (mm4)	wt (kN/m)	J (MPa)	B (mm)	D (mm)	Dt (mm)	Bt (mm)	Sx (mm3)	Sy (mm3)
200 UB 25.4	200000	80000	3230	23600000	3060000	0.24715314	62700	133	203	5.8	7.8	260000	70900
150 PFC	200000	80000	12345	8340000	1290000	0.1721655	54900	75	150	6	9.5	129000	46000
250 UB 31.4	200000	80000	4010	44500000	4470000	0.30683718	89300	146	252	6.1	8.6	397000	94200
125*8 EA	200000	80000	1900	4550000	1170000	0.1453842	40600	0	125	0	7.8	80200	40800

Copy the section property of one element to another elements

Step 1: Click 

button. Figure 41 will then appear. (*Note the button 'Clicking beam source' is currently in clicked mode which means that you are prompted to click the element to be copied*).

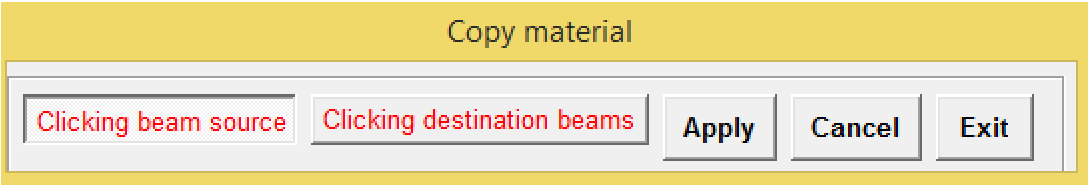


Figure 41

Step 2: Click the element with the section properties to be copied. After clicking the source element, Figure 41 will turn into Figure 42. *(Note the now the button 'Clicking destination beams' is now in clicked mode, which means that the user is prompted to click the elements will will received the material to be copied)*

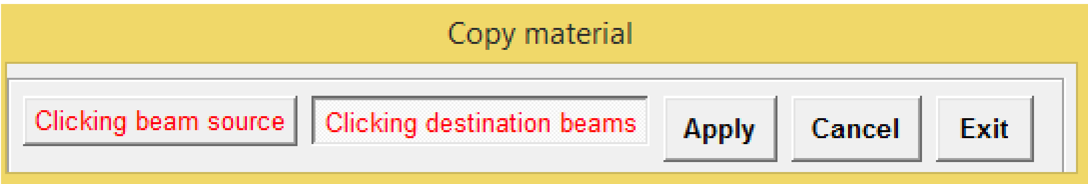


Figure 42

Step 3: Click the elements to receive the section properties.

Step 4: Click 'Apply' button.

Change the element(s) section properties

Step 1: From the list of section properties (see Figure 43) , click one in the list *(An elastic line will then appear between the section name and the mouse cursor.)*

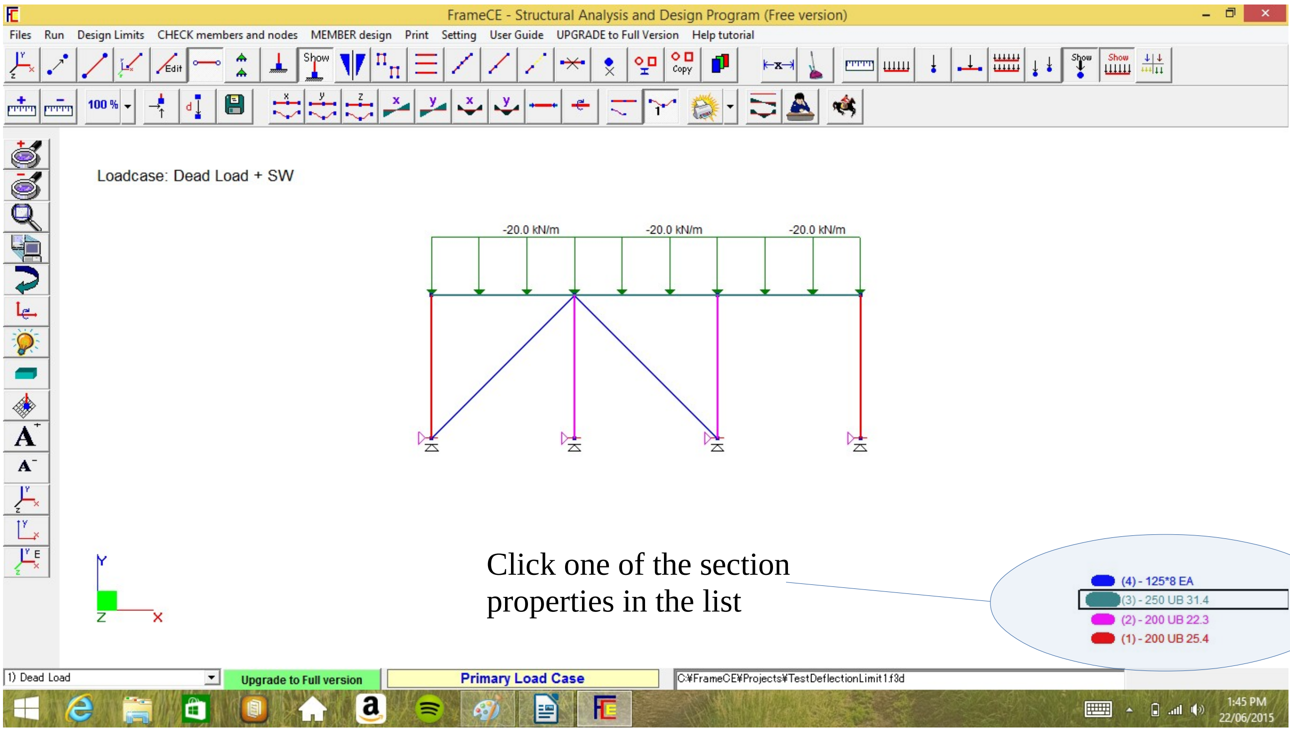


Figure 43

Step 2: Click the frame members that you want to receive the properties of the selected section as shown in Figure 44

Step 3: Right mouse click to stop.

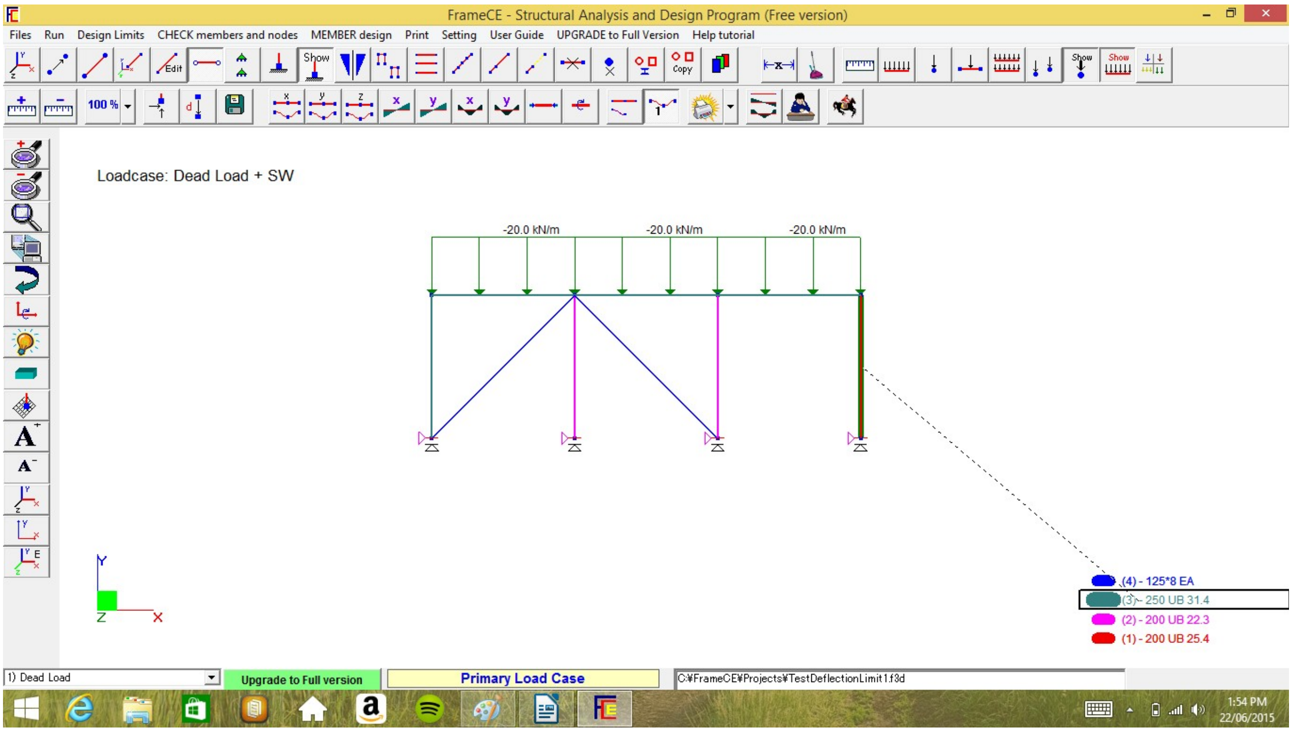
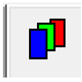

FrameCE - Structural Analysis and Design Program (Free version)
Files Run Design Limits CHECK members and nodes MEMBER design Print Setting User Guide UPGRADE to Full Version Help tutorial
Loadcase: Dead Load + SW
-20.0 kN/m
-20.0 kN/m
-20.0 kN/m
(4) - 125*8 EA
(3) - 250 UB 31.4
(2) - 200 UB 22.3
(1) - 200 UB 25.4
1) Dead Load
Upgrade to Full version
Primary Load Case
C:¥FrameCE¥Projects¥TestDeflectionLimit1.f3d

Figure 44

Create different layers of elements for viewing

When working large projects, the user may need to hide some members. This is particularly necessary when the model has many elements where adding more elements is easier when some elements are hidden. Similarly, when viewing analysis results, some elements are better off hidden due to overlapping graphics of stress results. In cases like this, '*creating layers*' can help the user. The procedure is illustrated below.

Step 1: Click button.  Figure 45 will then appear.

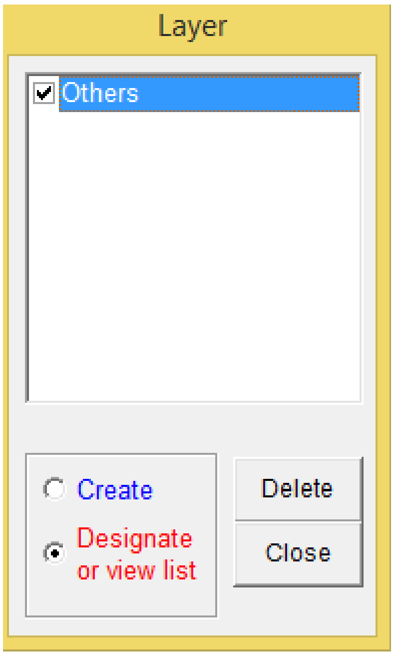


Figure 45

Step 2: Click option 'Create'. (Figure 45 will change to Figure 46)

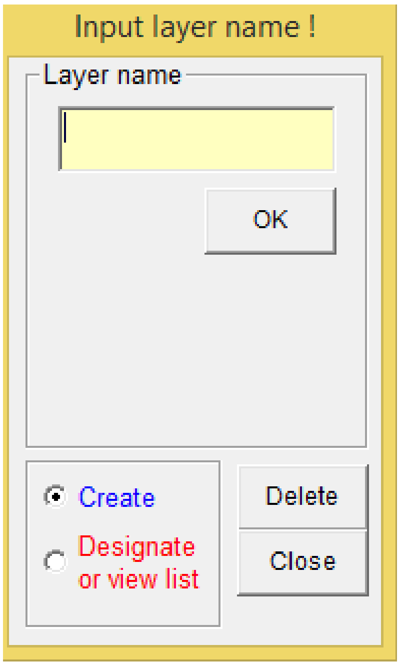


Figure 46

Step 3: Type the layer name. (In this example, let us say '*Test layer 1*'), then click button 'OK'. Figure 46 will then change to Figure 48. (Note that there are now two layers named *Others* and *Test layer 1. Test layer 1* is being highlighted which means that this is the current or default layer.)

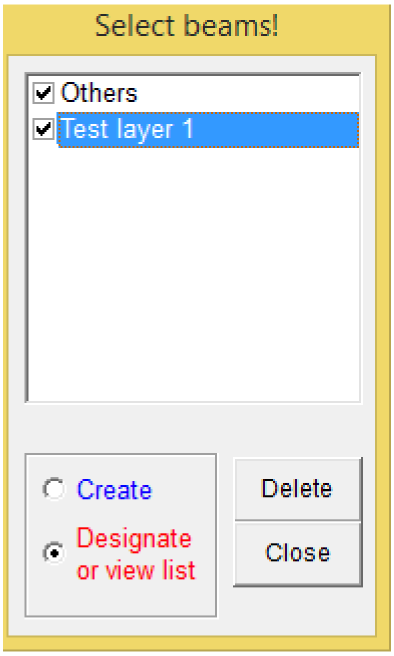


Figure 47

Step 4: Repeat **Step 3** to create more layers. In this example I created three layers, namely, 'Test Layer 1', 'Bracing' and 'Columns'. Also note that all four layers are in 'checked' mode which means that they are visible. The layer name 'Column' is highlighted which indicates that it is the default or current layer. The option button 'Designate or view list' is being selected which means that you can now designate which frame member belongs to current layer 'Column'.

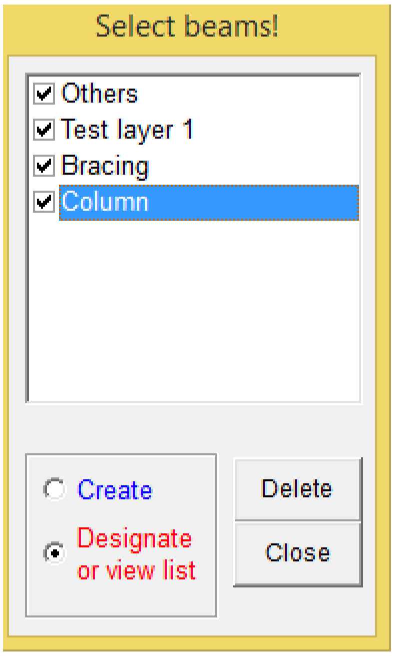


Figure 48

Step 5: Click the frame elements that you want to be included in the highlighted layer. (i.e., *the current layer which is 'Column'*). In this example, I clicked members 1, 2, 3, and 4 as shown in Figure 49. Members 1,2,3 and 4 now belong to layer 'Column'.

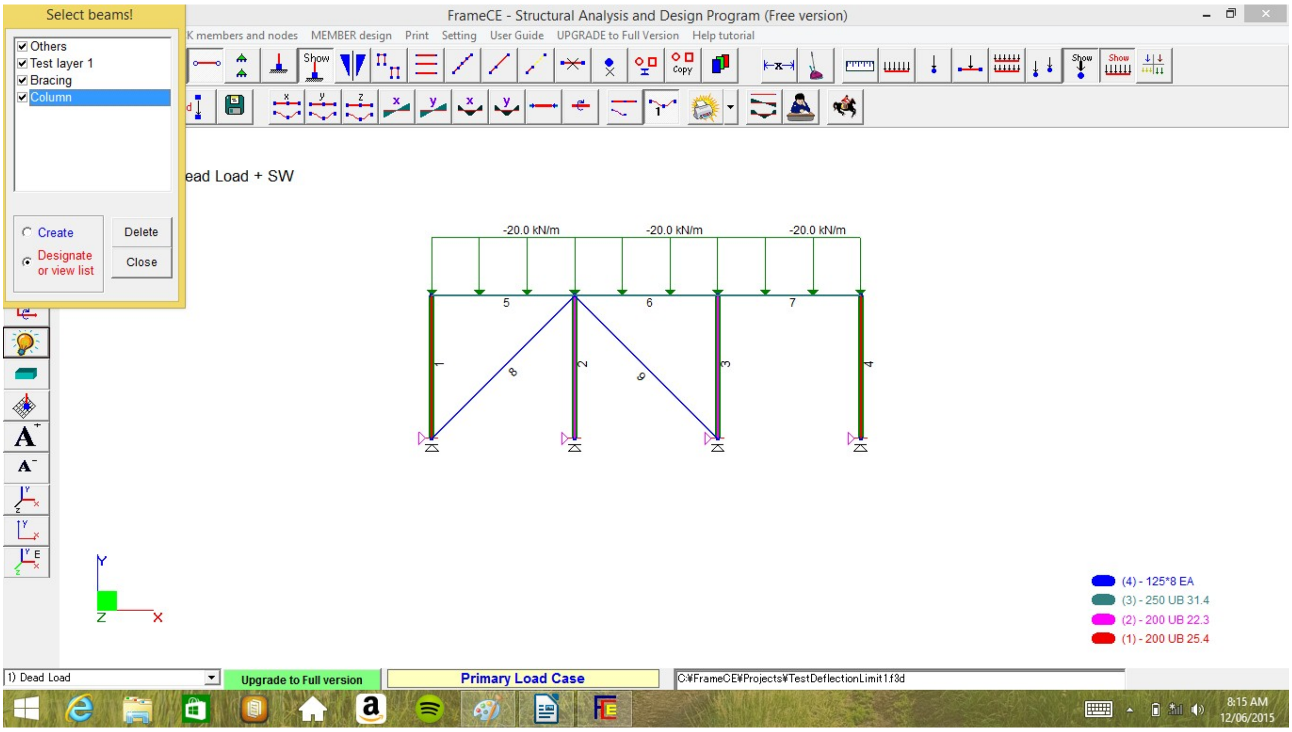
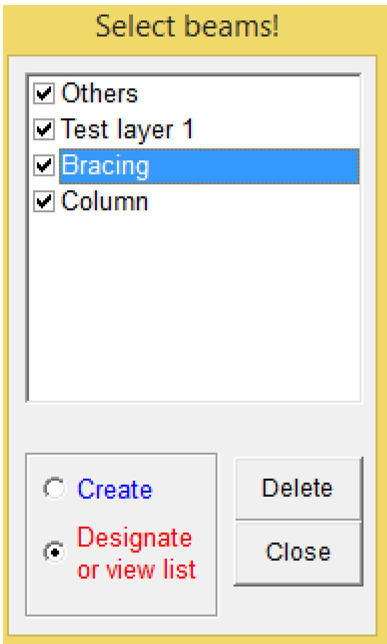


Figure 49

Step 6: Change the current layer by clicking another layer name. *(In this example, I clicked layer 'Bracing' to make this the current layer.)* Note that layer 'Bracing' is now highlighted which indicates that it is now the current layer as shown in Figure 49



Figure

Step 7: Click the frame elements that you want to be included in the highlighted layer. (i.e., *the current layer which is 'Bracing'*). In this example I selected members 8 and 9. Repeat Step 6 to designated members that belong to layer 'Column'. *(In this example I selected members 1,2,3, and 4 as members of layer 'Column' but is not shown here because you already know the step)*. *<u>The rest of the frame members not assigned to any layer name automatically belong to layer 'Others'.</u>*

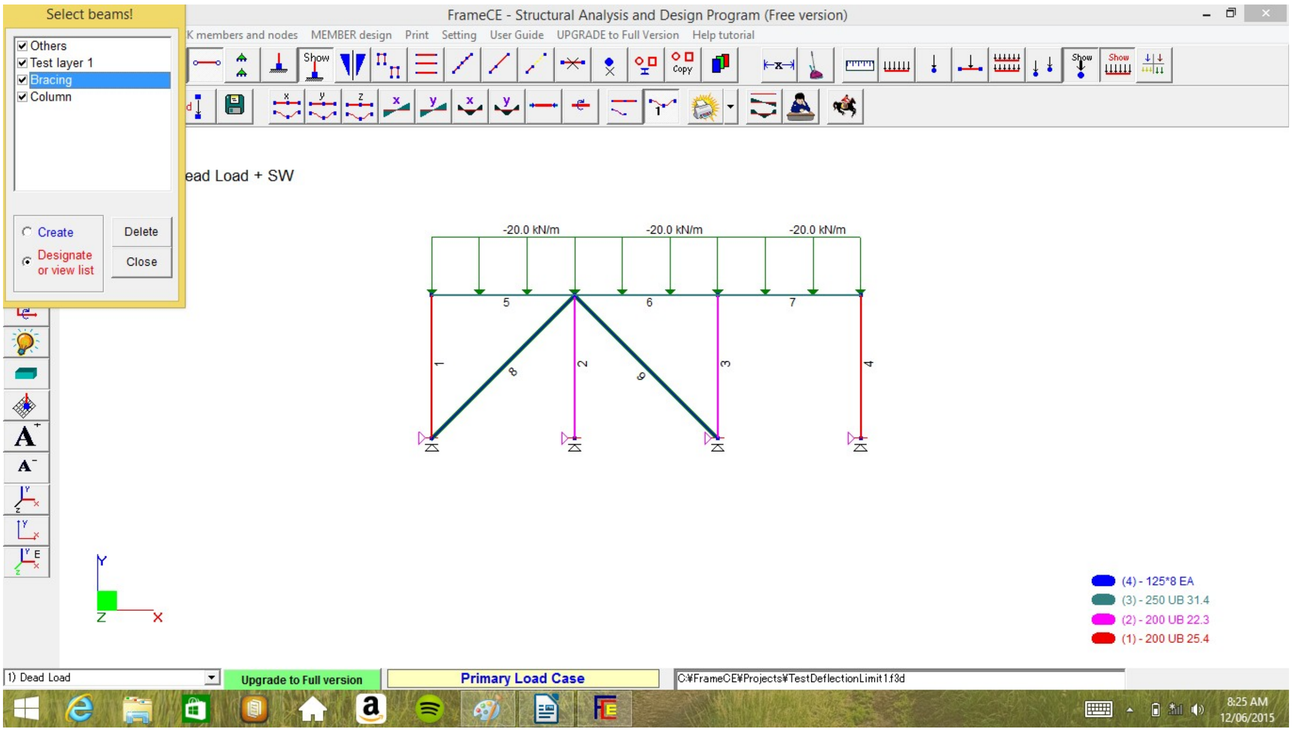


Figure 50

Step 8: Select which layer to show and which select to hide. *Checked* layers will be visible whereas *unchecked* layers are hidden. *__Checking and unchecking layers can be done by clicking the box to the left of the layer name. See Figure 51 and Figure 52.__*

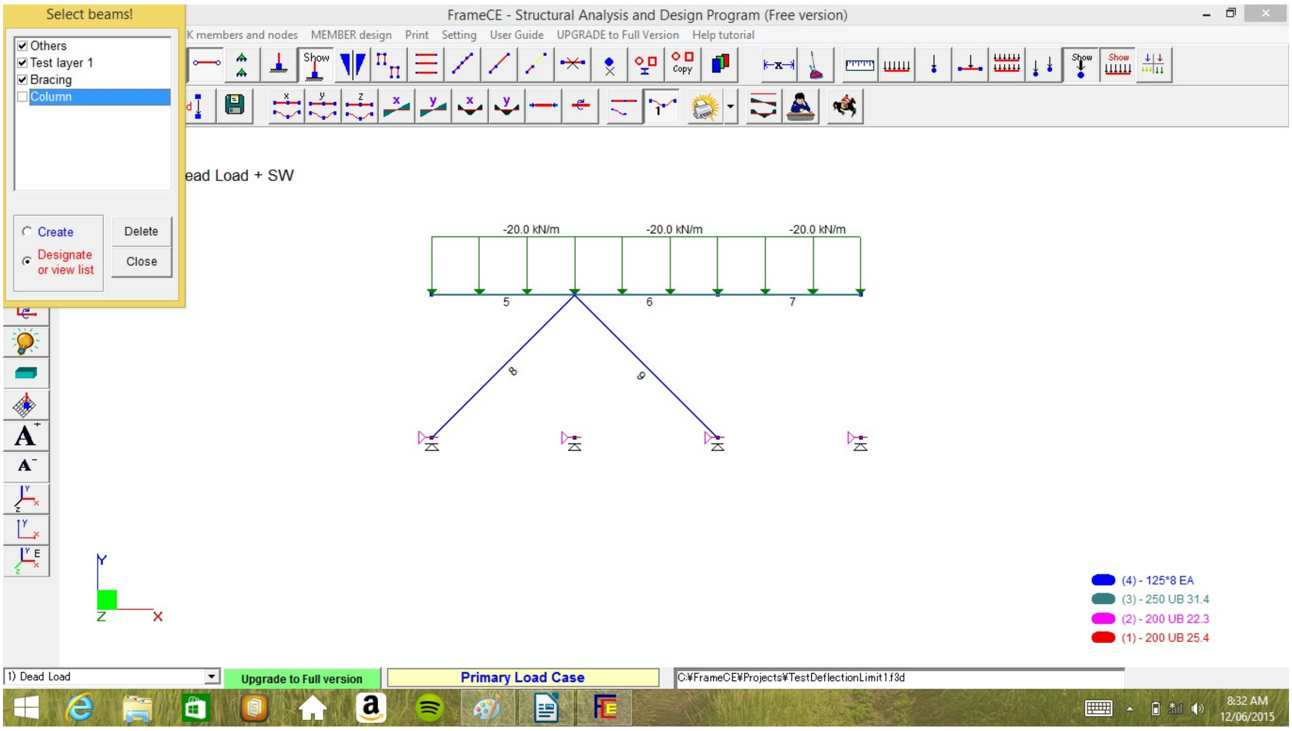


Figure 51

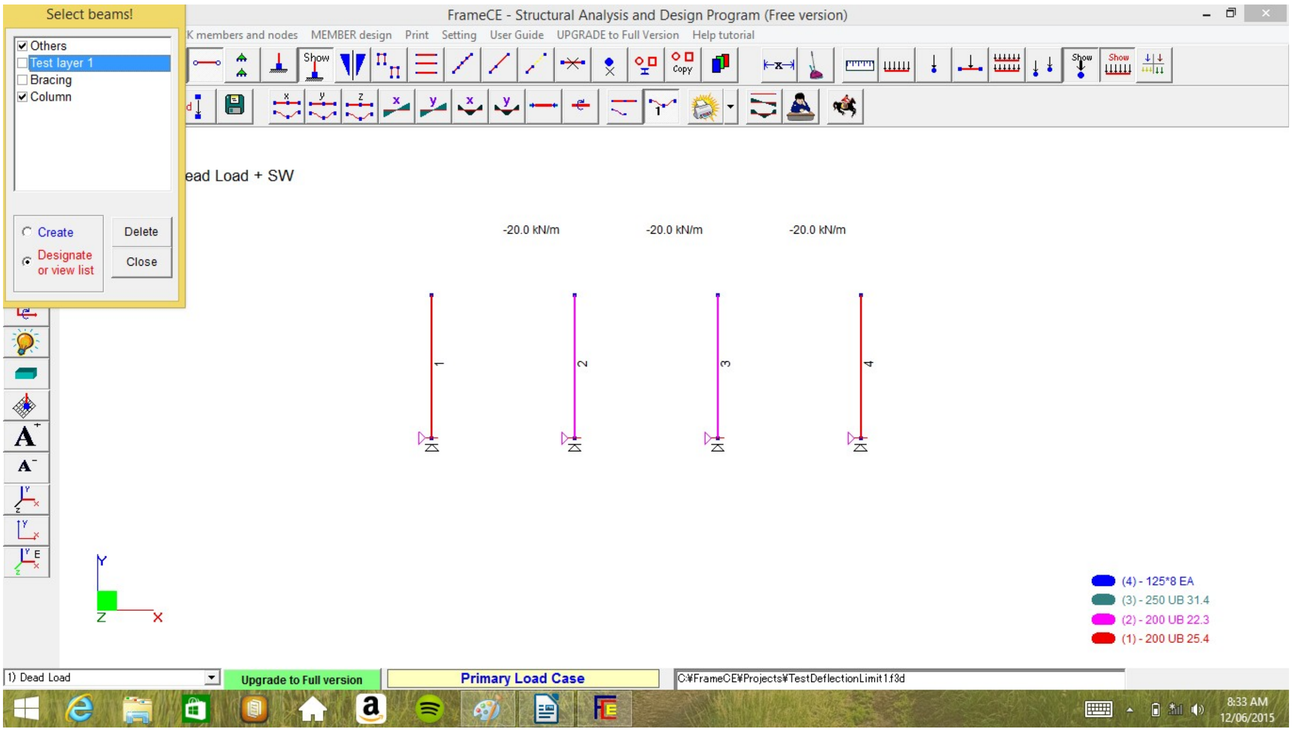
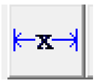


Figure 52

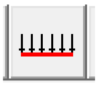
Get the distance between two nodes

Step 1: Click button 

Step 2: (FrameCE will prompt you to click two nodes)

APPLYING LOADS TO THE MODEL

Applying distributed loads to an element

Step 1: Click button  Figure 53 will then appear.

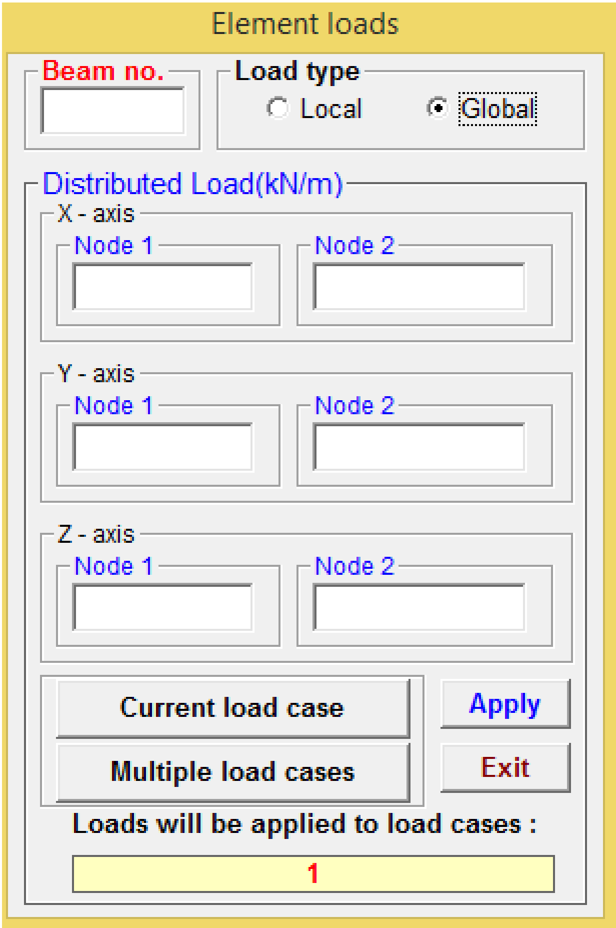


Figure 53

Step 2: Input the load values. *(You don't have to complete all the input boxes provided. Input boxes left blank are assumed to have values of zero)*. Loads can also be applied to multiple load cases by clicking the button 'Multiple load cases'. When this button is clicked, you will then be prompted to select which of the 'primary load cases' the load will be applied.

Step 3: Select the frame members to receive the loads. (In this example, I selected members 5 and 7 to receive the loads of -20 kN/m as shown in Figure 54).

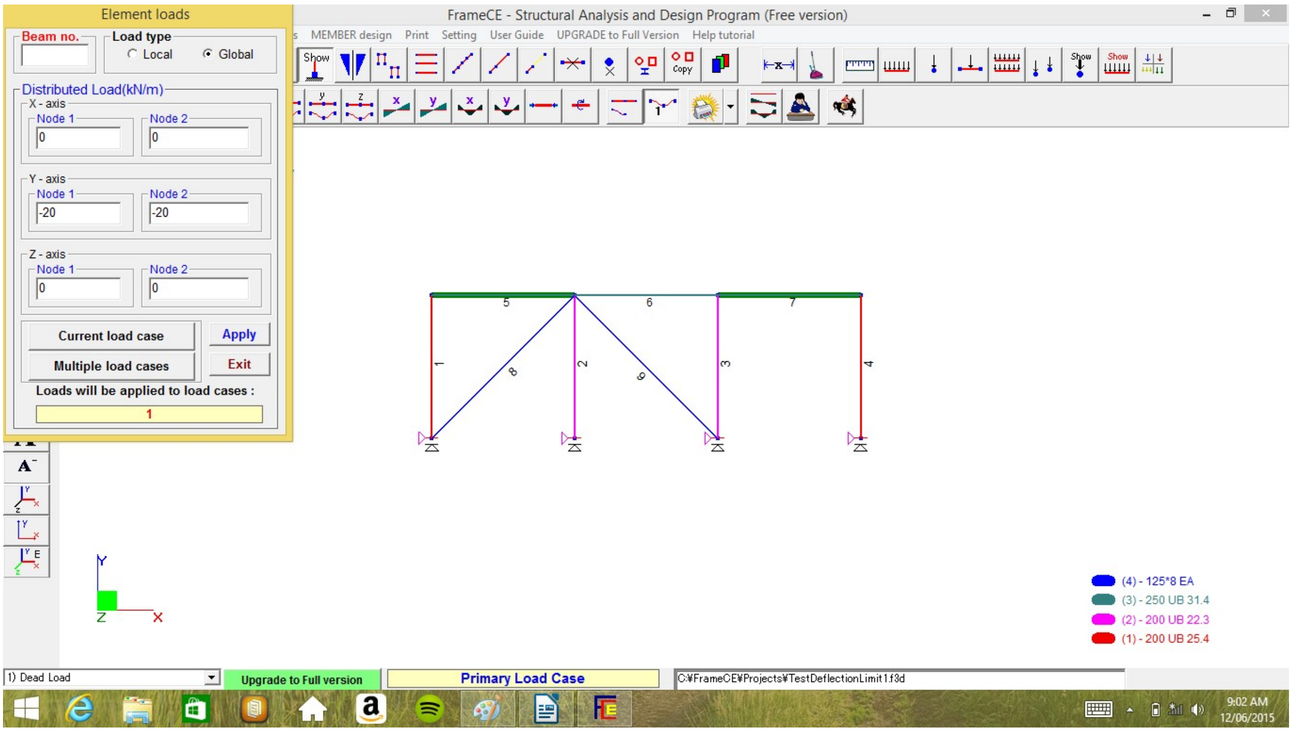
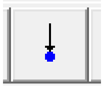


Figure 54

Step 4: Click 'Apply' button.

Step 5: Repeat step 2 to apply element loads to other members, otherwise, click 'Exit' button.

Apply point loads to a node

Step 1: Click button  Figure 55 will then appear.

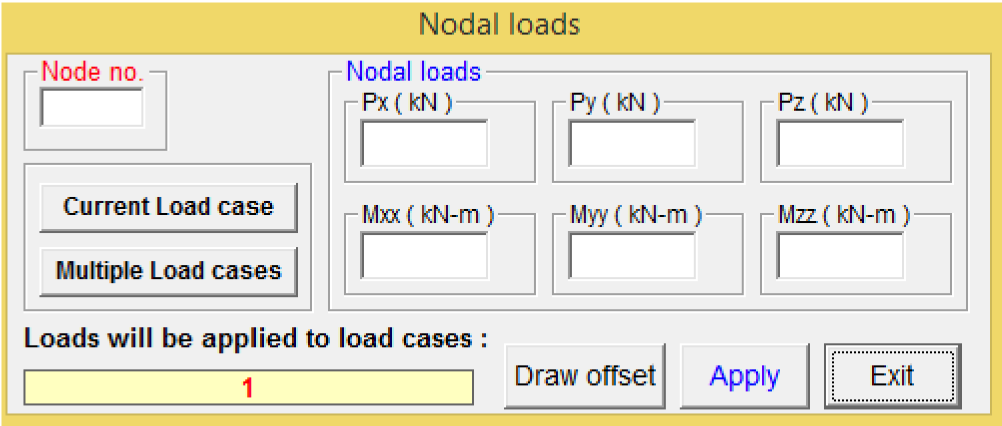


Figure 55

Step 2: Input load values in the boxes provided. *(Boxes left blank are assumed to have values of zero).*

Step 3: Click the nodes to receive the loads

Step 4: Click 'Apply' button

Step 5: Repeat Step 2 to apply point loads to other nodes. Otherwise, click 'Exit' button.

Notes: In Figure 55, if the button 'Draw offset' is clicked, Figure 57 will appear. The X, Y, and Z location offset values are eccentricities used to plot the point loads in the model. This eccentricities for plotting the load in the model are useful it is hard to see the point loads when obstructed by the frame elements. Point loads plotted at a distance from the node helps in viewing the loads. These X, Y, and Z location offset values do not affect the model or the analysis.

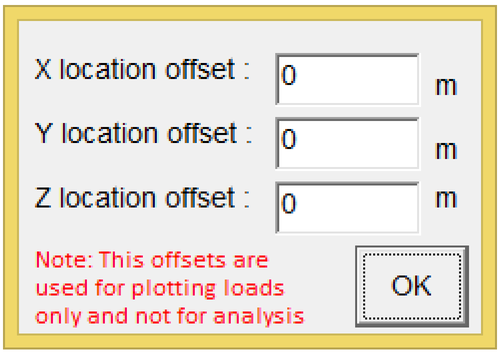
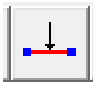


Figure 56

Applying point loads within an element

Step 1: Click button  Figure 57 will then appear.

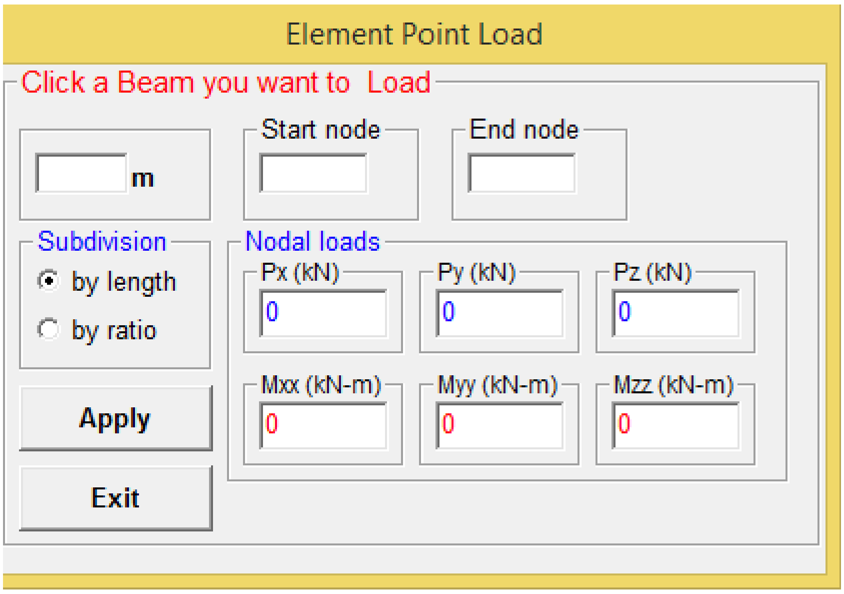


Figure 57

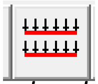
Step 2: Input the load values. *(You don't to complete all the input boxes provided. Input boxes left blank are assumed to have values of zero)*.

Step 3: Select 'Subdivision' option. If option 'by length' is selected, input the distance of the load from the first node of the element. If option 'by ratio' is selected, input the ratio of the distance of the left distance to the total length of the element.

Step 4: Select the frame members to receive the loads.

Step 5: 'Apply button'.

Copy element load from one element to another elements

Step 1: Click button  Figure 58 will then appear. *(Note that button 'Clicking beam source' is in clicked mode which means that you are currently in this mode)*

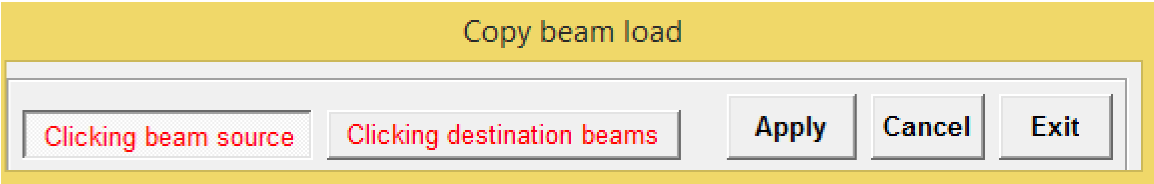


Figure 58

Step 2: Click the element which has the element load to be copied. (After clicking the source element, Figure 58 will change to Figure 59. Note that the button 'Clicking destination beams' are now in clicked mode which indicates that you are now in this mode).

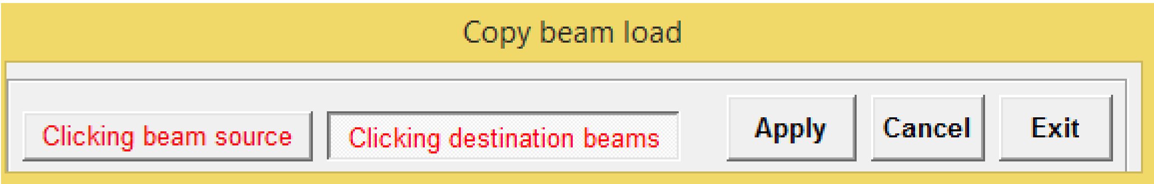
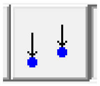


Figure 59

Step 3: Click the elements to receive the loads.

Step 4: Click 'Apply' button.

Copying point load from one node to another nodes

Step 1: Click button  Figure 60 will then appear.

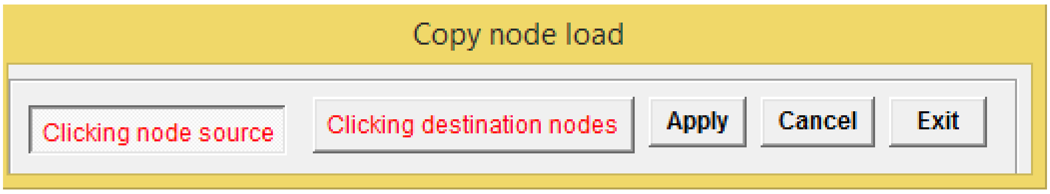


Figure 60

Step 2: Click the node which has the point load to be copied. (After clicking the source node, Figure 60 will change to Figure 61. Note that the button 'Clicking destination nodes' are now in clicked mode which indicates that you are now in this mode).

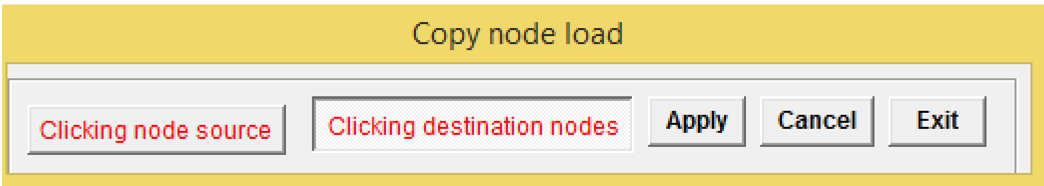
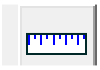


Figure 61

Step 3: Click the nodes to receive the point loads.

Step 4: Click 'Apply' button.

Changing the load graphic scale

Step 1: Click button  Figure 62 will then appear.

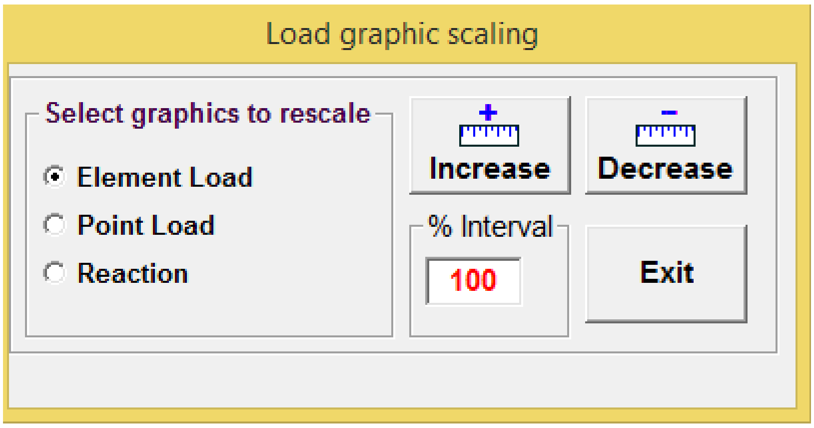


Figure 62

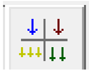
Step 2: Select which load graphics to rescale (i.e., Element load or Point load or Reactions)

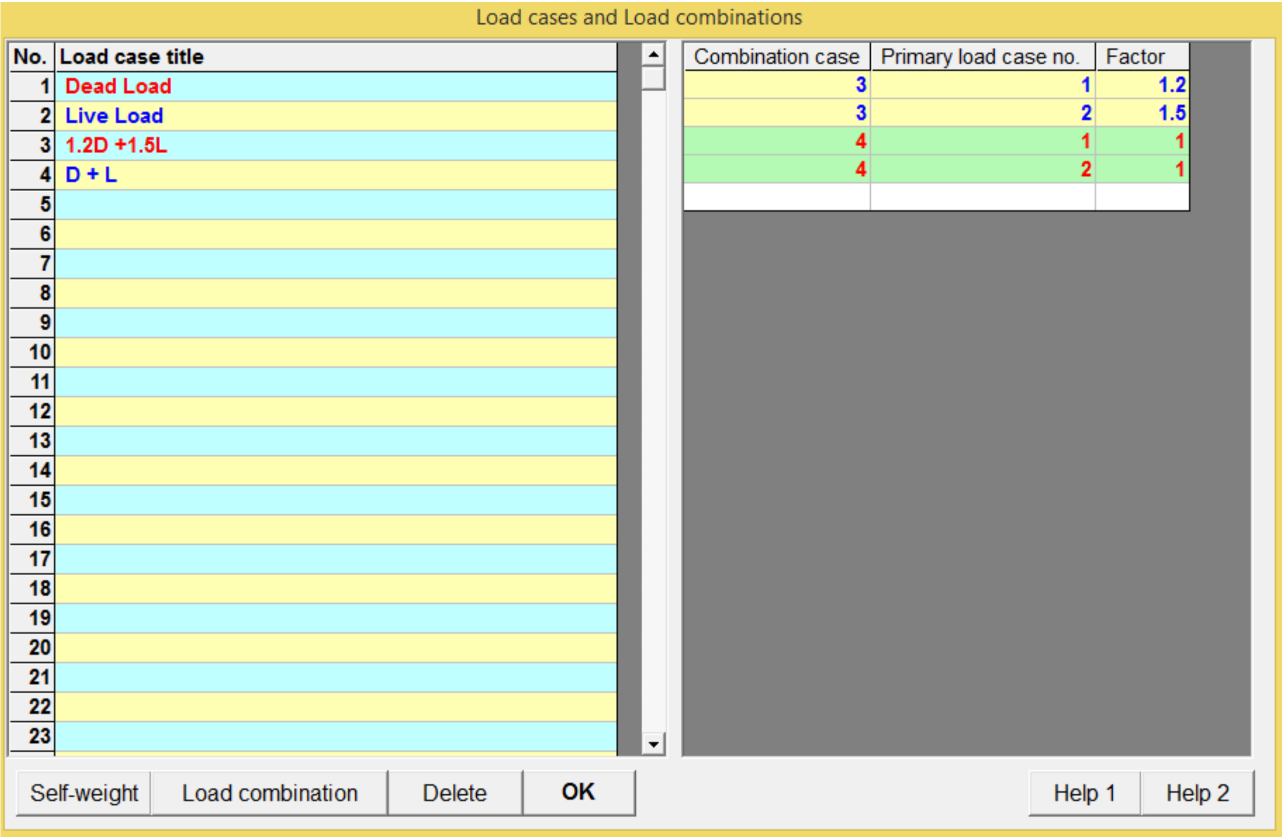
Step 3: Declare the % interval of change. *(The bigger the number the bigger the change)*

Step 4: Click button 'Increase' or 'Decrease'. (Every time you click one of these buttons, you will see changes in the scale)

Step 5: Click 'Exit' to finish.

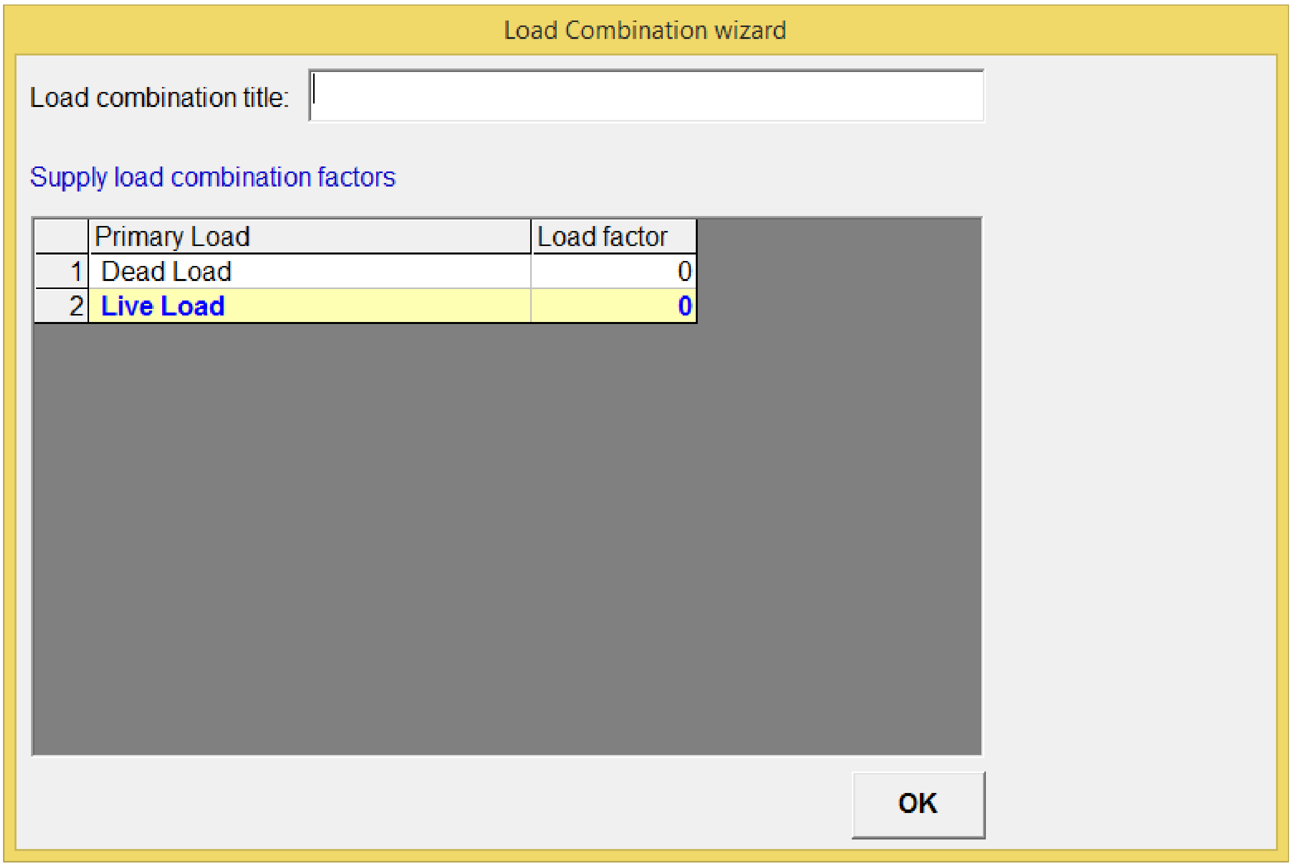
Declare load cases

Step 1: Click button  Figure 63 will then appear.



Step 2: Input manually the load cases titles both for the primary and load combination cases.

Step 3: Input manually the load factors for primary load combinations. This can also be done by clicking the button 'Load combination' and declaring the load factors in the box provided as shown in Figure 64.



Step 4: Declare which load includes the self-weight by clicking the button 'Self-weight'. When the button is clicked, Figure 65 will appear. (*Only the primary load cases will appear*)

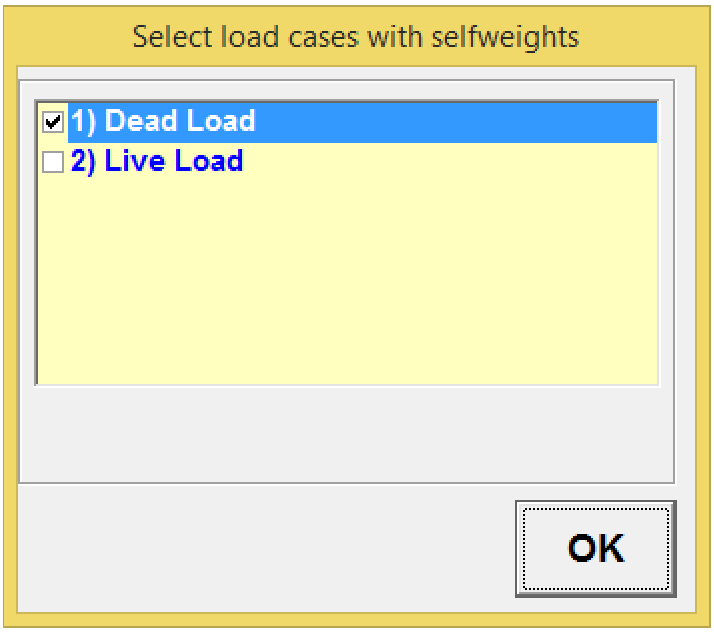


Figure 65

Step 4: Tick the load cases which will include self-weights then click 'OK'. Figure 63 will then re-appear.

Step 5: Click 'OK' button to finish.

Edit the load values

Step1: Click the loads to be edited.

Step2: Click the right button of the mouse (i.e., right mouse click). Figure 66 Will pop-up in case of element loads or Figure 67 in case of point loads.

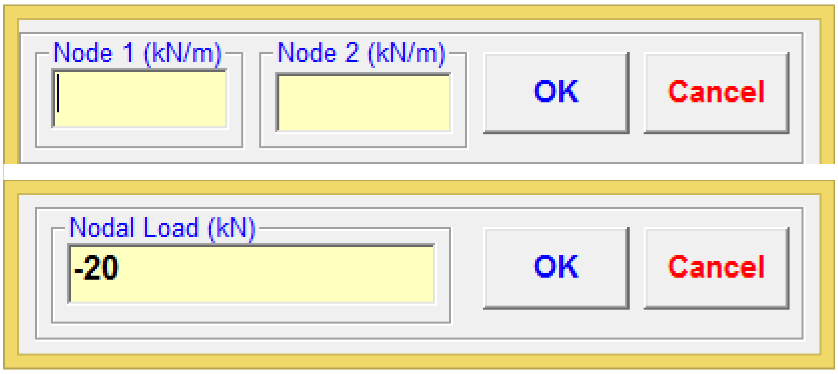


Figure 67

Step 3: Declare load values

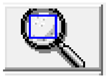
Step 4: Click 'OK' button

Edit the load value by double clicking

Step1: When a load graphics is double-clicked Figure 66 or Figure 67 will pop-up and you can edit its value.

VIEWING THE MODEL

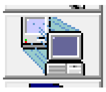
Zoom-in

Step 1: Click button  or the button  when specifying the zooming window using mouse

Zoom-out

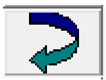
Step 1: Click button 

Pan view

Step 1: Click button 

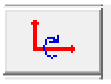
Step 2: Press the left mouse button and drag it while pressing it down. *(You will see that the model moves with the direction of the mouse movement)*

Rotate the model

Step 1: Click button 

Step 2: Press the left mouse button and drag it while pressing it down. *(You will see that the model rotates with the direction of the mouse movement)*

Specify the rotation axis of the model

Step 1: Click button  Figure 68 will then appear.

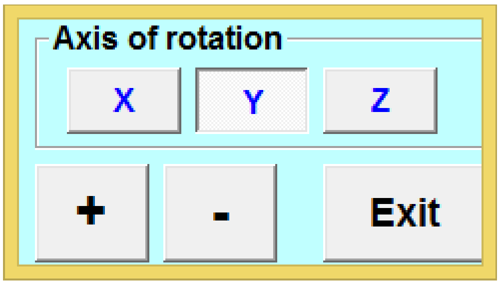
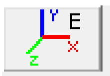


Figure 68

Step 2: Click one of the buttons X or Y or Z then click button '+' or '-'. *(You will see the model rotate about the specifies axis.)*

Specify the point of rotation of the model

Step 1: Click button  Figure 69 will then appear.

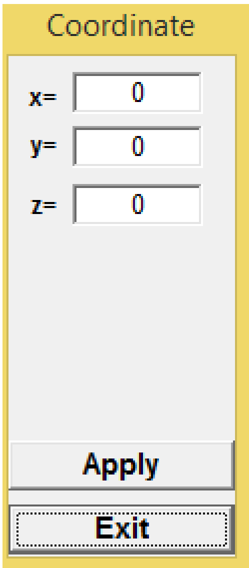


Figure 69

Step 2: Specify the x,y,z coordinate of the point of rotation

Step 3: Click 'Apply' button. *(After you did these steps, when button*  *is clicked, the model will rotate from the x,y,z coordinates you declared in Step 2).*

Show numbering of frame elements and nodes

Step 1: Click button  Figure 70will then appear.

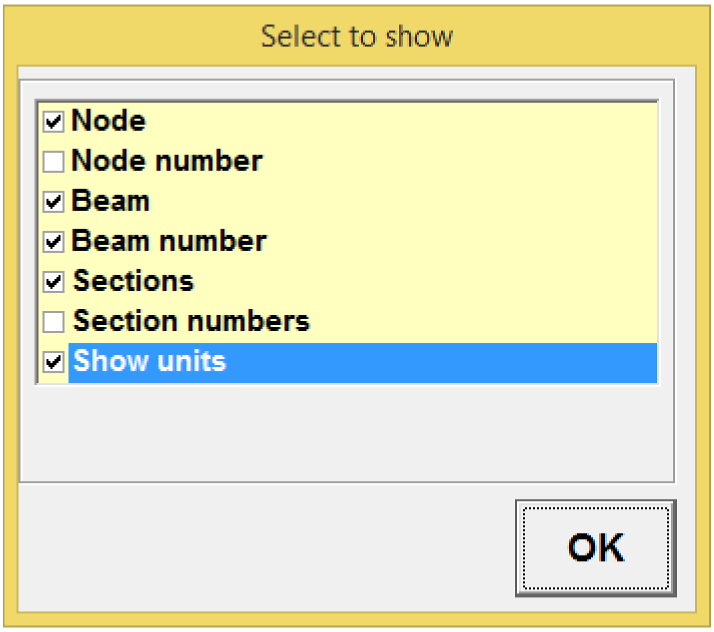
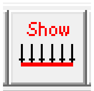


Figure 70

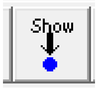
Step 2: Tick the numbering to show.

Step 3: Click 'OK' button to finish.

Show or hide elements loads

Step 1: Click button 

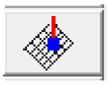
Show or hide point loads

Step 1: Click button 

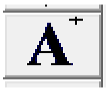
Show frame elements in 3D

Step 1: Click button  (Clicking this button will toggle between 3D and stick element models)

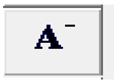
Show or hide base grid

Step 1: Click button 

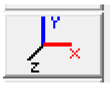
Increase font size

Click button 

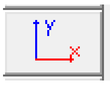
Decrease font size

Click button 

Reset view in 3D axis

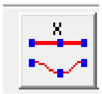
Click button 

Reset view in 2D axis

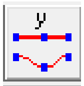
Click button 

VIEWING THE ANALYSIS RESULTS

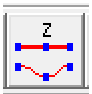
Show x-displacement

Step 1: Click button 

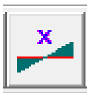
Show y-displacement

Step 1: Click button 

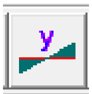
Show z-displacement

Step 1: Click button 

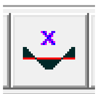
Show shear diagram about the major axis

Step 1: Click button 

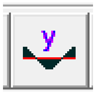
Show shear diagram about the minor axis

Step 1: Click button 

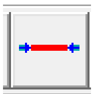
Show moment diagram about the major axis

Step 1: Click button 

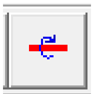
Show moment diagram about the minor axis

Step 1: Click button 

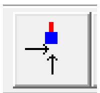
Show axial stresses

Step 1: Click button 

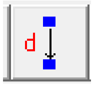
Show torsional stress

Step 1: Click button 

Show reactions

Step 1: Click button 

View the x,y, z displacements at a single node

FrameCE shows the displacements along one axis one at a time. However, the x, y and z displacements at a single node can be viewed by clicking the button.  Figure 71 will then appear. When a node is clicked, the displacements at this node will be indicated.

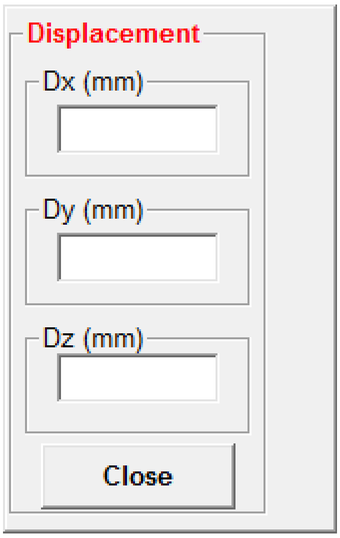


Figure 71

In this example, I clicked node 7 and the displacements at this node is given as shown in Figure 72.

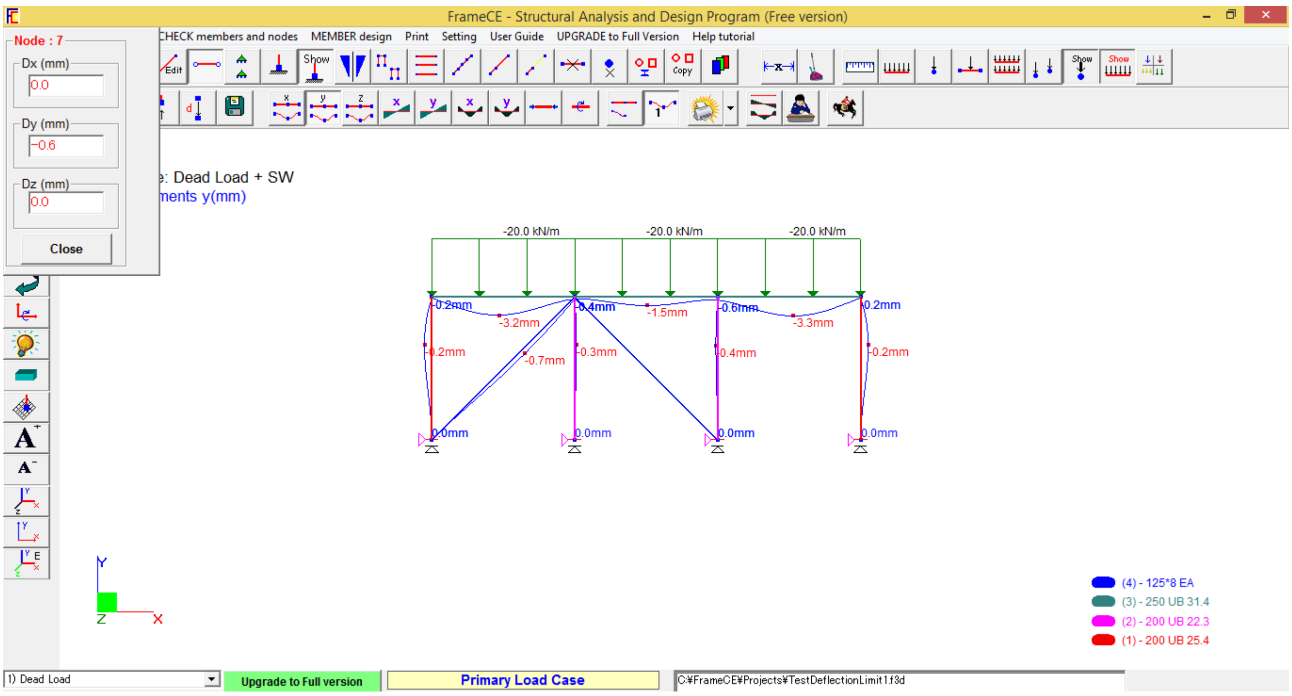
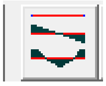


Figure 72

View the analysis results of a single element

Step 1: Click button  Figure 73 will then appear.

Step 2: Click an element. *(The analysis results of that element will then be displayed as shown in Figure 74). In this example I clicked element no. 6.*

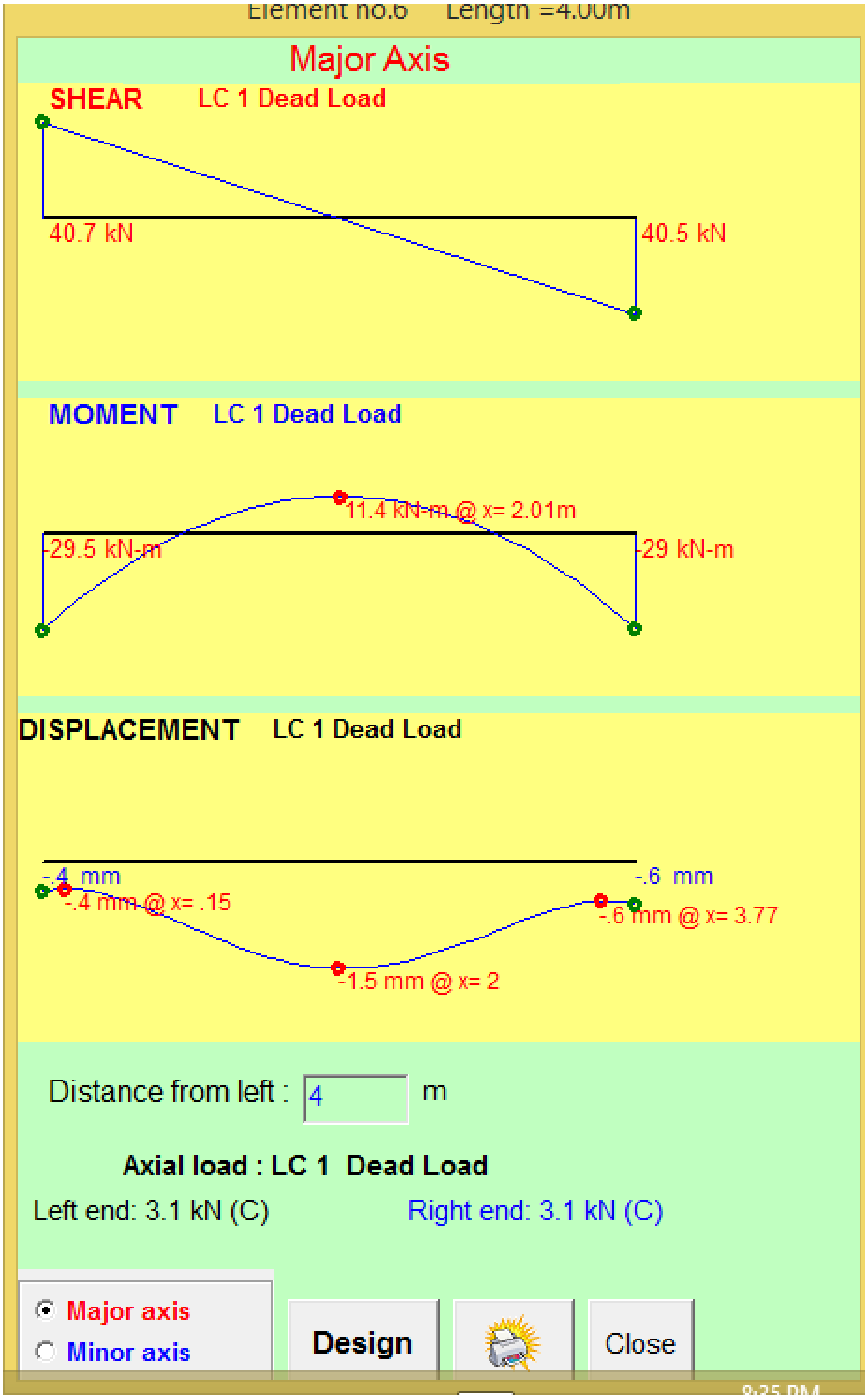


Figure 73

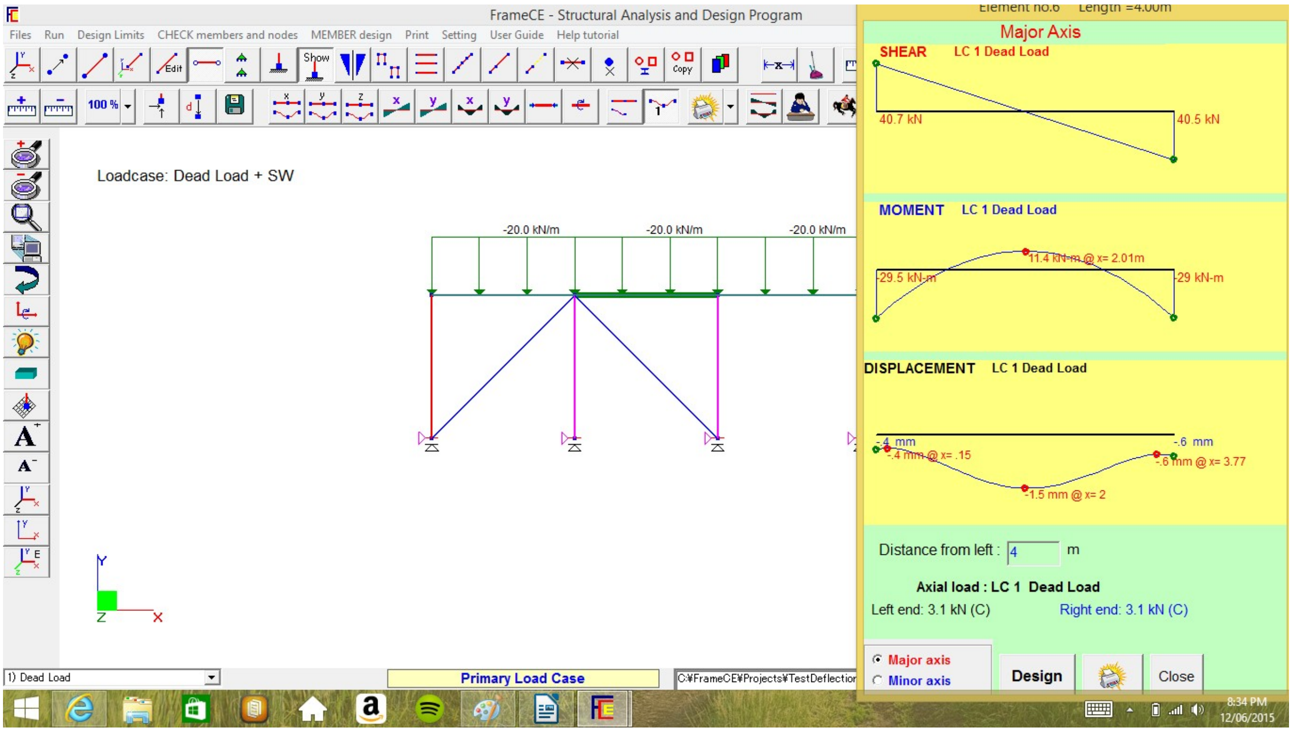
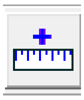
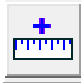

FrameCE - Structural Analysis and Design Program
Files Run Design Limits CHECK members and nodes MEMBER design Print Setting User Guide Help tutorial
Loadcase: Dead Load + SW
-20.0 kN/m
-20.0 kN/m
-20.0 kN/m
Major Axis
SHEAR LC 1 Dead Load
40.7 kN
40.5 kN
MOMENT LC 1 Dead Load
11.4 kN-m @ x= 2.01m
29.5 kN-m
29 kN-m
DISPLACEMENT LC 1 Dead Load
-.4 mm
-.6 mm
-.4 mm @ x= .15
-.6 mm @ x= 3.77
-1.5 mm @ x= 2
Distance from left : 4 m
Axial load : LC 1 Dead Load
Left end: 3.1 kN (C)
Right end: 3.1 kN (C)
Major axis
Minor axis
Design
Close
1) Dead Load
Primary Load Case

Figure 74

Increase the scale of analysis results graphics

Step 1: Click button  repeatedly to get to the desired scale. For example, Figure 75 shows moment diagram. After clicking the button  twice, the moment diagram graphics is now shown in Figure 76. *(Observe the change in scale)*

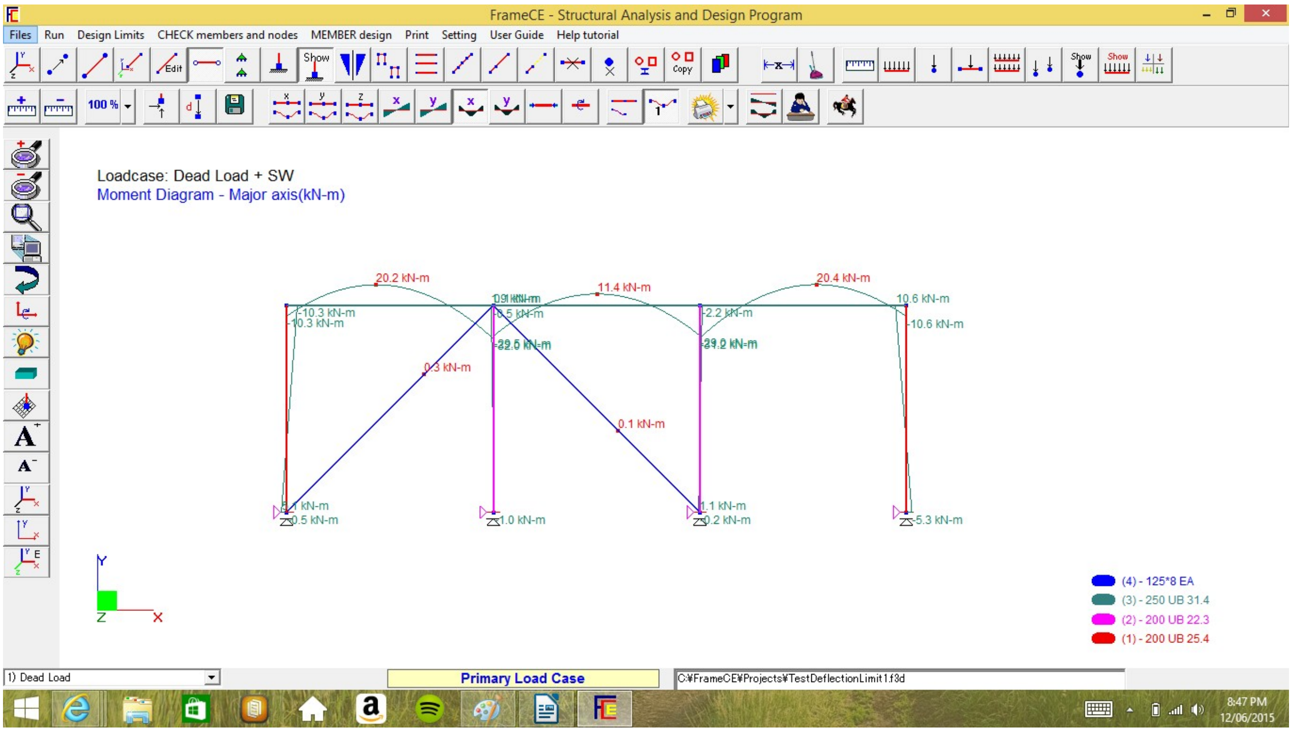


Figure 75

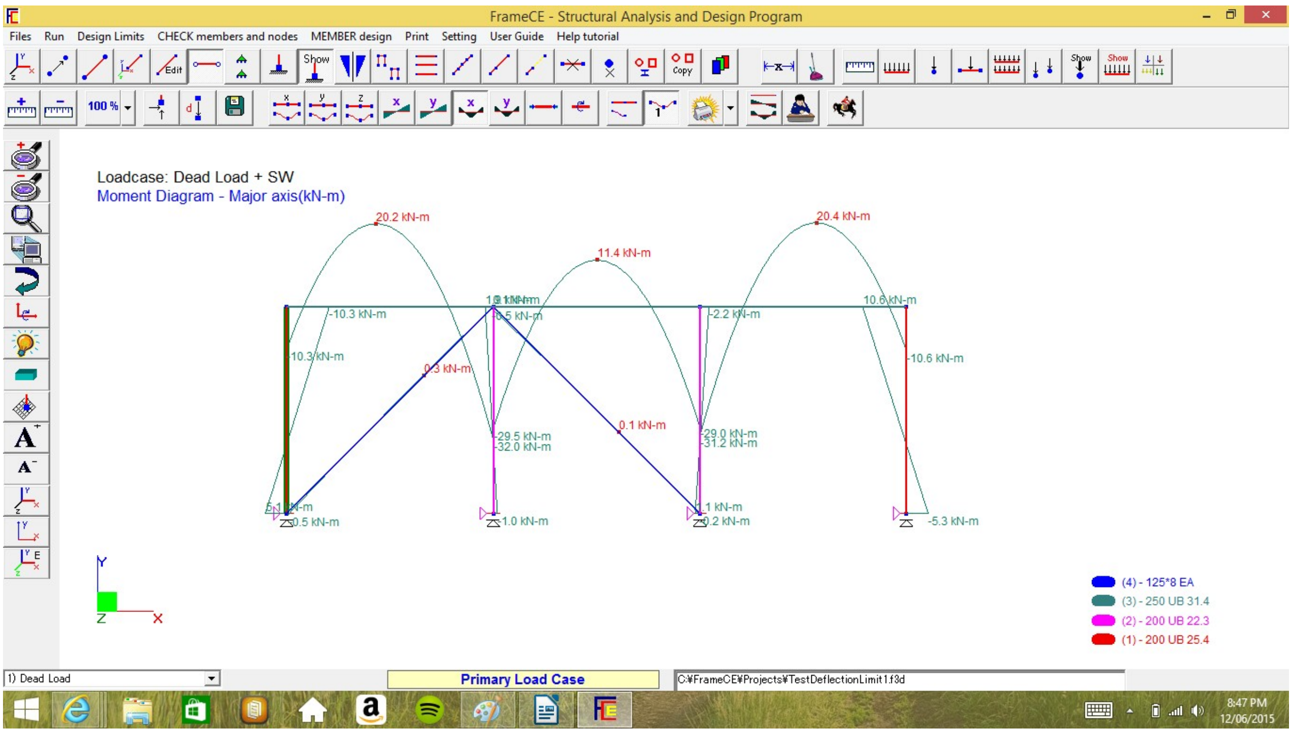
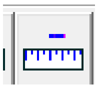
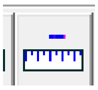
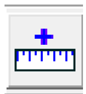
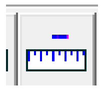
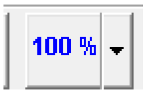


Figure 76

Decrease the scale of analysis results graphics

Step 1: Click button  repeatedly to get to the desired scale. For example, Figure 76 shows moment diagram. After clicking the button  twice, the moment diagram graphics is now shown in Figure 75 *(Observe the change in scale)*

Interval of graphic scale changes

The degree of change (increase or decrease) to the graphics when button  or  is clicked can be specified by clicking the downward triangle to the right of the button 

When this is clicked, the percentage interval options will be displayed as shown in Figure 77. Choose one of the interval options provided. Smaller values means smaller increase or decrease.

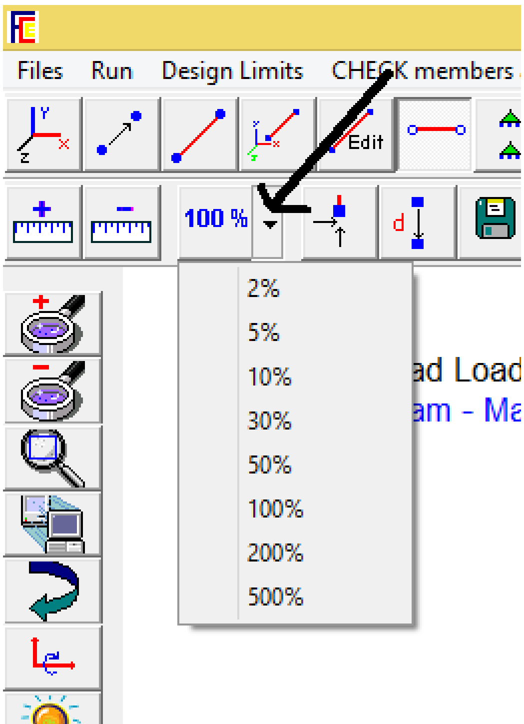


Figure 77

Showing analysis results of selected members

When the frame model is composed of many elements, viewing the analysis results can be complicated due to overlapping stress diagrams. For this reason, FrameCE introduced a command which allows the user to view the analysis results of selected members. The steps are described below.

Step 1: Run the analysis by clicking the button 

Step 2: View the analysis results. (For example the moment diagram) as shown in Figure 78;

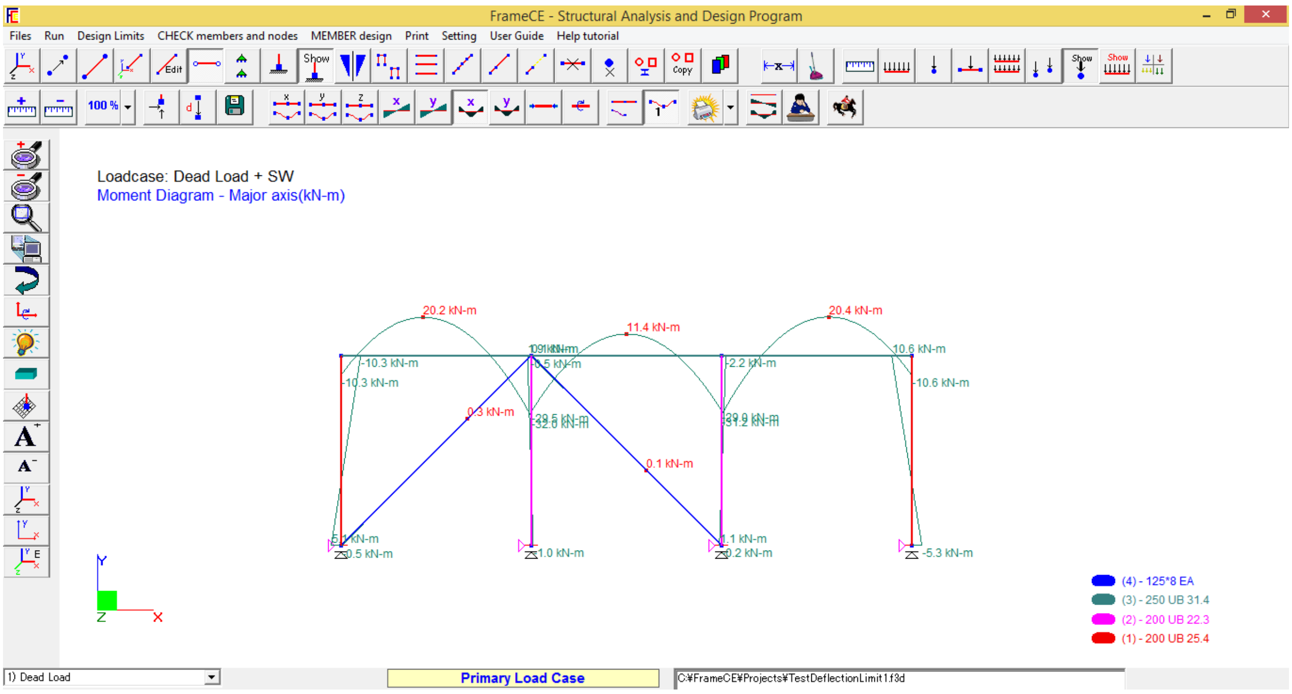


Figure 78

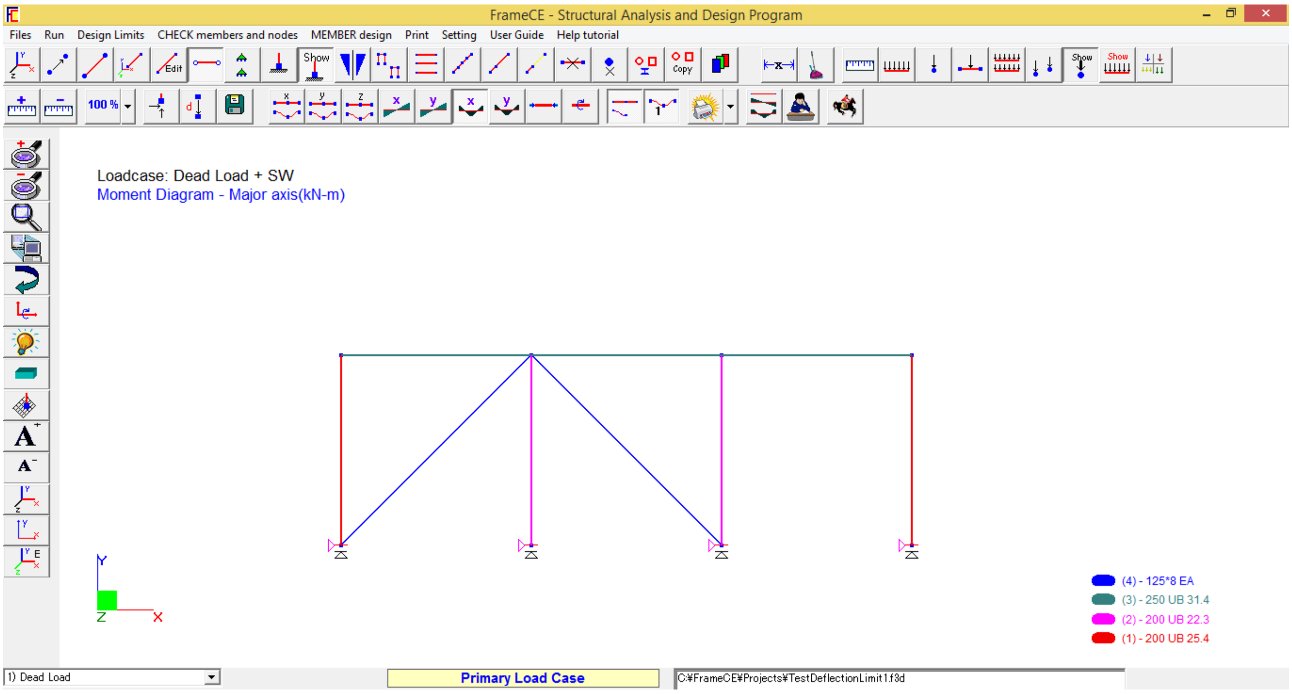


Figure 79

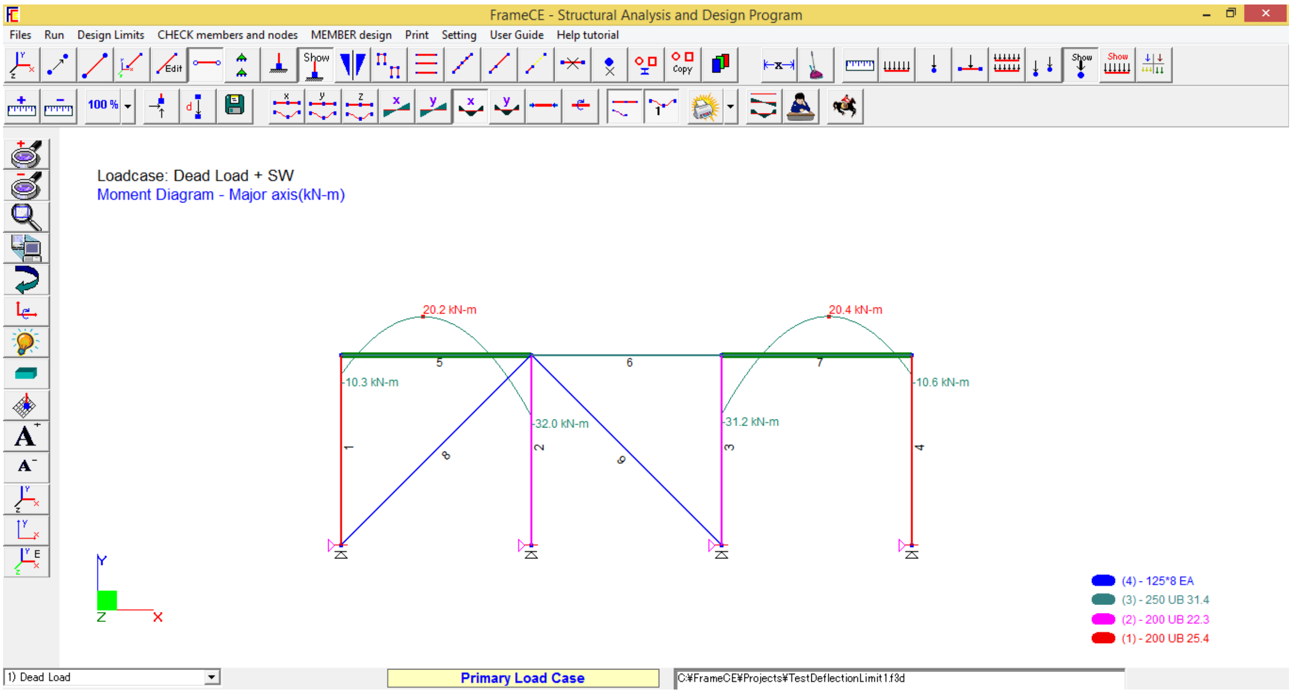
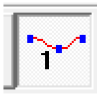


Figure 80

Showing the analysis results without the numbers

Analysis result graphics can be shown without the values by clicking the button.  This button is in pressed mode by default which means that the analysis results shown also show the values by default. Repeatedly clicking this button toggles between showing or hiding the values.

Dynamic frequency analysis

Dynamic frequency analysis gives the first 20 mode shapes and its corresponding Frequency (Hz) and Period (T) of the structure. The steps are given below:

Step 1: Go to menu 'Run' and select 'Frequency analysis'. Figure 82 Will then appear.

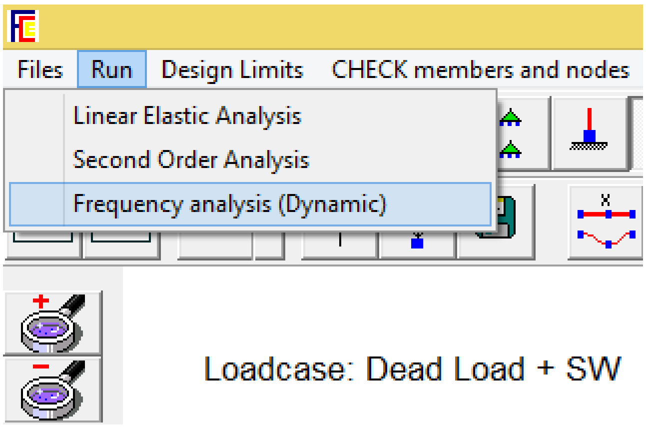


Figure 81

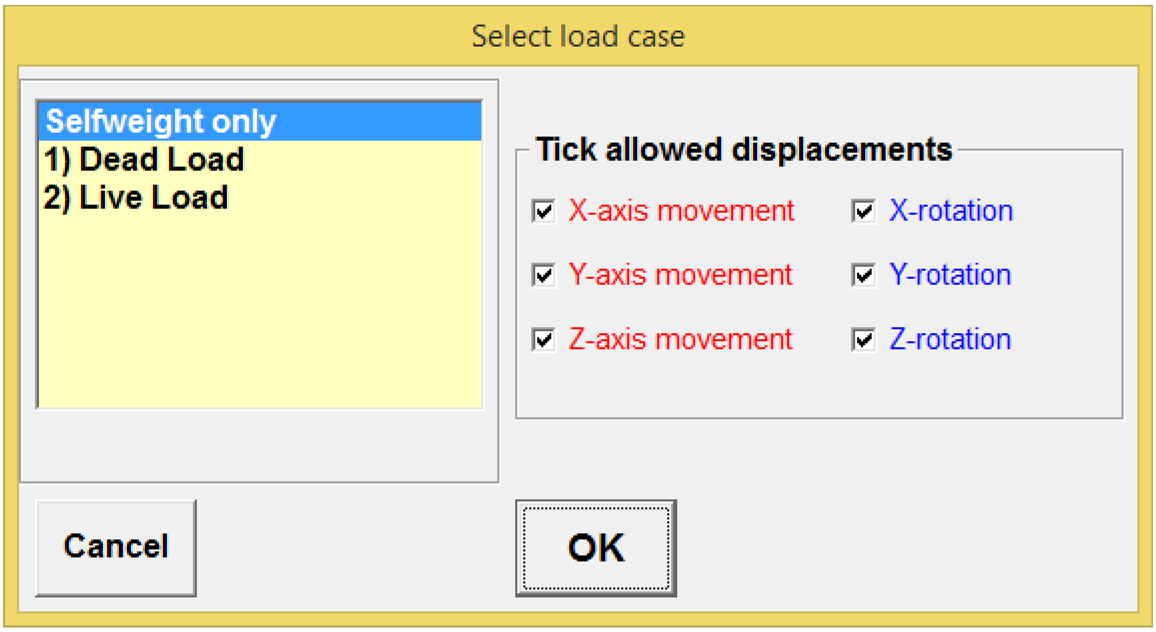


Figure 82

Step 2: Select the load case where the frequency analysis be done. Tick the allowed displacements. Un-ticking some of the displacements will increase the speed of the calculation since it will reduce the number of mode shapes thereby reducing the number of eigenvalues to be calculated. *(For example, in the analysis I did shown in Figure 83, I only ticked the x and y displacements because I am only interested on the mode shapes associated with x and y movements.)*

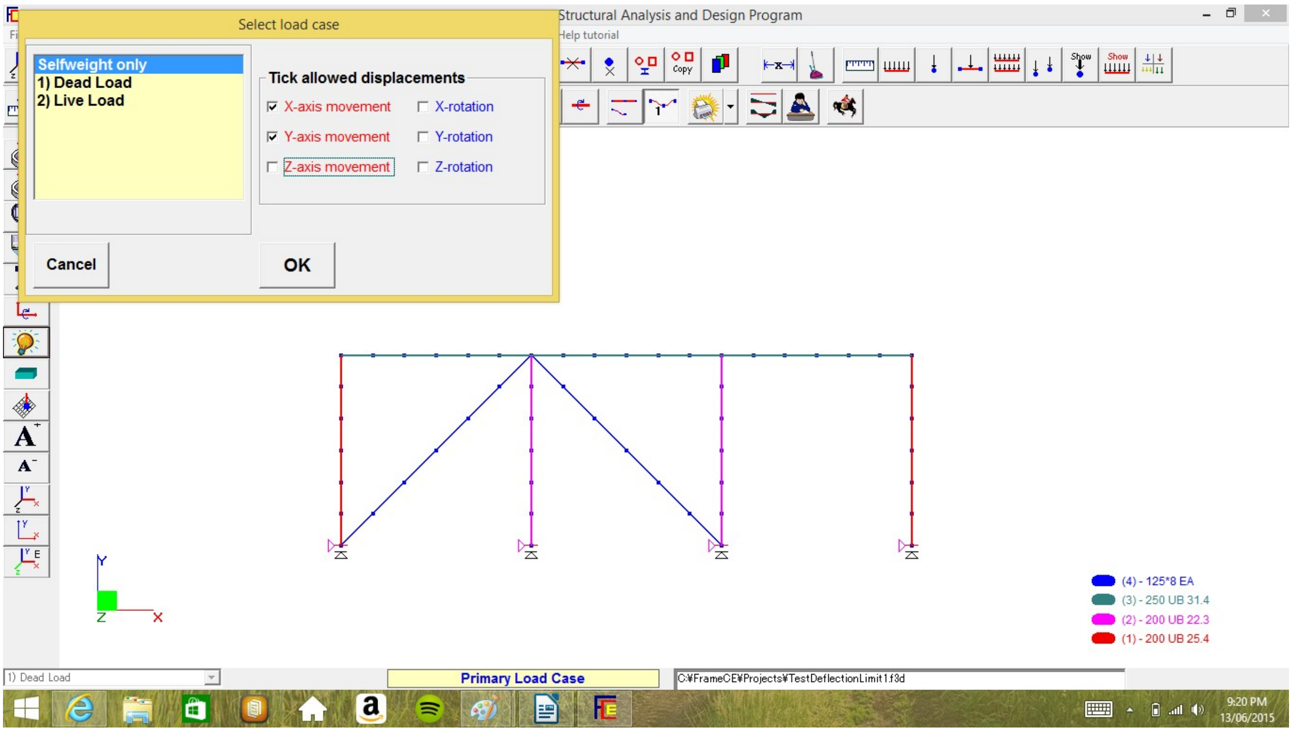


Figure 83

Step 3: Click 'OK' button. After the analysis is done, Figure 84 will appear. Select a mode number and the corresponding mode shape will be displayed.

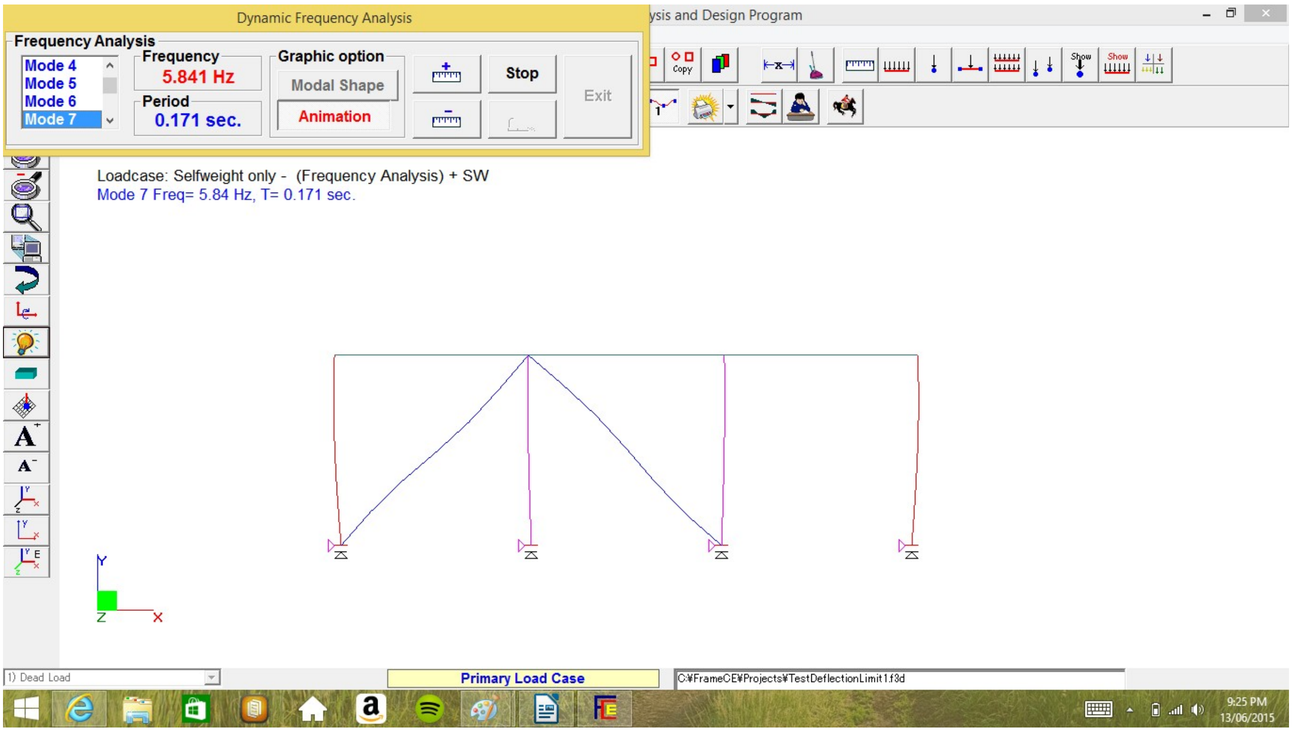

Dynamic Frequency Analysis
Frequency Analysis
Mode 4
Mode 5
Mode 6
Mode 7
Frequency
5.841 Hz
Period
0.171 sec.
Graphic option
Modal Shape
Animation
Stop
Exit
Loadcase: Selfweight only - (Frequency Analysis) + SW
Mode 7 Freq= 5.84 Hz, T= 0.171 sec.
Primary Load Case
1) Dead Load

Figure 84

Frequently Asked Questions (FAQs)

Q1) I want to use my own section preferences in the design. How can I do this?

Answer:

Step 1: Edit the section properties of the elements and change the section of the elements to your preferred sizes

Step 2: Go to menu 'Check members and nodes' then select 'Check members by both deflection and strength criteria' (see Figure 85). If all members are safe then proceed to **Step 3** otherwise edit the section properties and repeat **Step 1**.

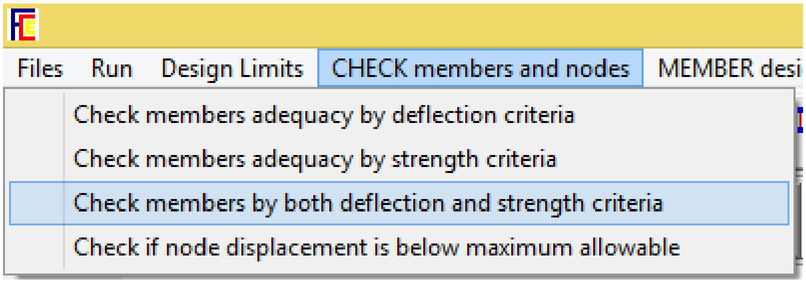


Figure 85

Step3: Check if node displacements are below maximum allowable (See Figure 86). If all nodes are below maximum allowable then design is complete and you can now print the design calculation. If some nodes fail then go to **Step 1.**

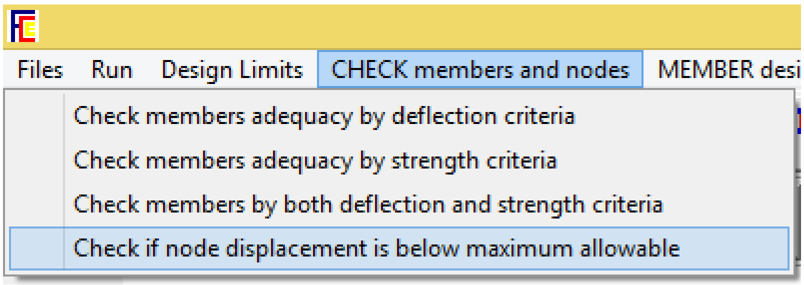


Figure 86

Q2) In the edit element properties, what is this Member Type (normal, tension or compression members) as shown in Figure 86 below?

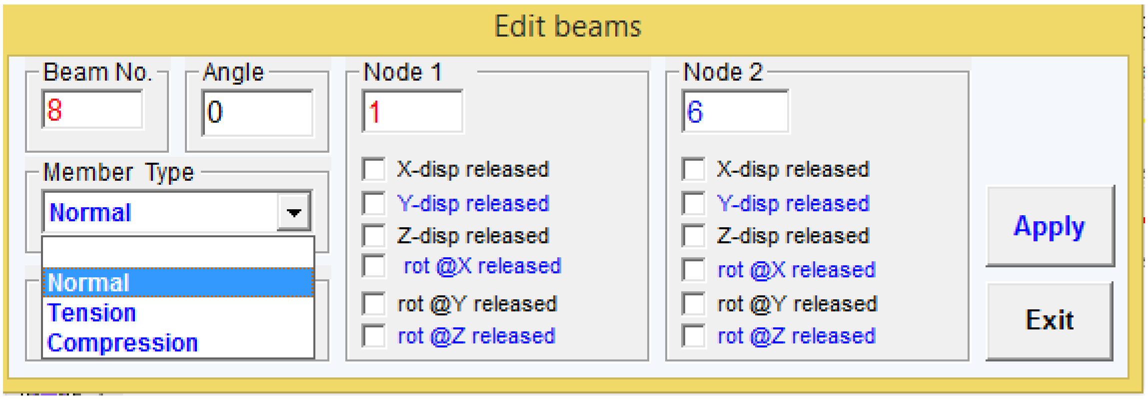


Figure 87

Answer:

Normal members can carry both compressive and tensile forces. Tension members can only carry tensile forces and compression members can only carry compressive forces. During the analysis, when FrameCE detects that a compression member is carrying a tensile force, its stiffness contribution is removed and FrameCE re-analyses the frame which makes the element carry zero stresses. Similarly, tensile member's stiffness contribution is removed when the member is carrying compressive stresses.