



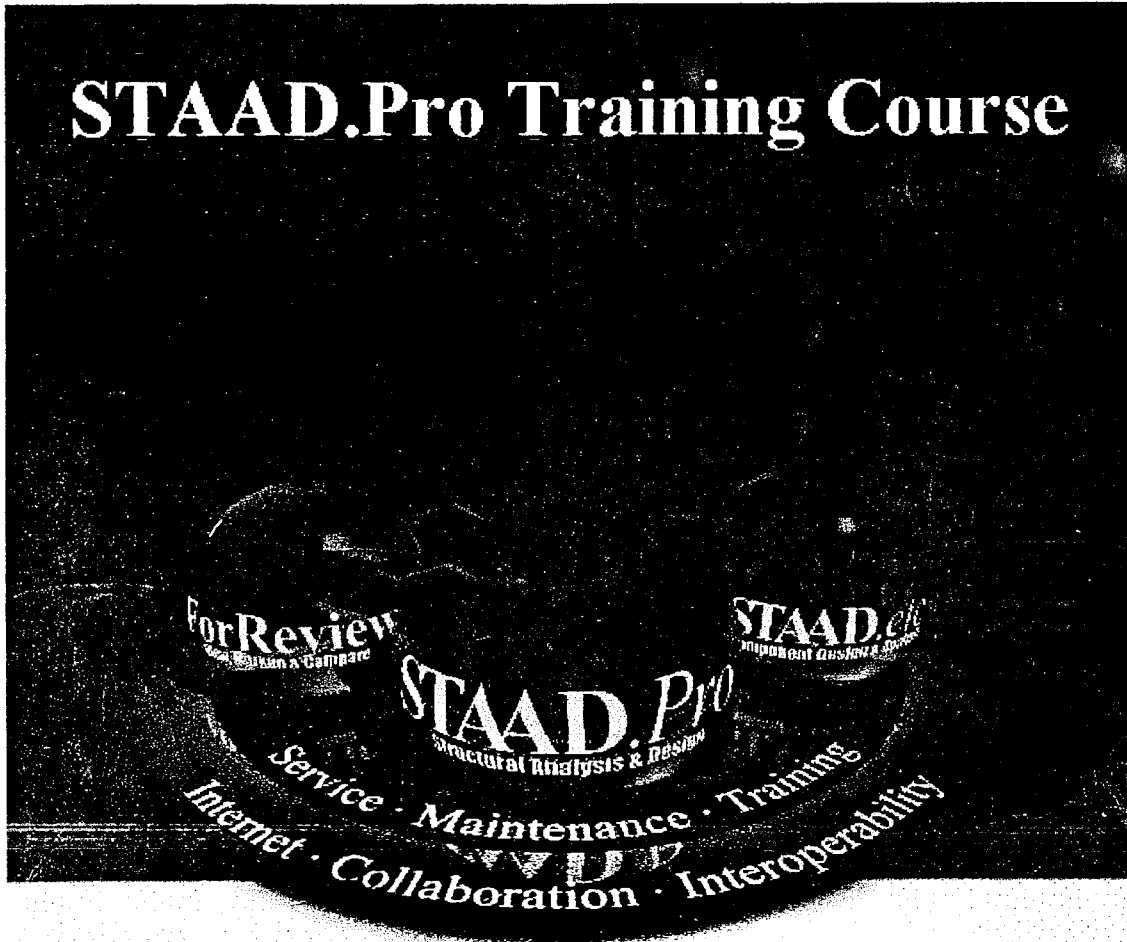
Research Engineers International

80, Marine Parade Road , #16-01 Parkway Parade, Singapore 449269

Tel: (65) 6225 6015 / 6 Fax: (65) 6225 5010

www.reiworld.com E-mail: support@reiasia.net

STAAD.Pro Training Course



Client	
Location	
Date	

Research Engineers is a subsidiary of

netGuru



STAAD.Pro:- STAAD Training

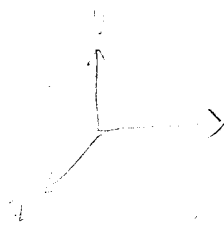
1. Introduction

This course is structured to look at the scope of the STAAD.Pro commands and capabilities. There are six exercises to complete, each one containing more STAAD commands and dealing with more complex problems.

This document has been produced for your benefit and assistance. This document is copyright and no part of it is to be copied, reproduced electronically or otherwise without the prior written consent of Research Engineers.

2. Revision History

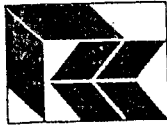
Date	Version	File Ref.	Details	By
17/06/98	1.0	STAAD Pro Training	First Issue	CA
10/03/99	1.0	STAAD Pro Training	Second and third examples added	CA
19/08/99	1.1	STAAD Pro Training	Updated for v3.1	CA
25/08/99	1.1	STAAD Pro Training	Dynamic example added	CA
06/09/99	1.2	STAAD Pro Training	RC Designer added	CA
01/05/00	1.3	STAAD Pro Training	Updated to STAAD 2000	CA
01/11/00	1.4	STAAD Pro Training	Detailed steel design examples added.	CA
24/09/01	1.5	STAAD Pro Training	Updated for STAAD 2001	ME
28/06/02	1.6	STAAD Pro Training	Updated for STAAD 2002	ME



- Fire
- (ent)

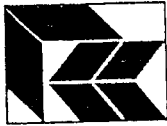
- Fire
- (new)

beam
- shear - moment



3. Contents

1. Introduction	1
2. Revision History	1
3. Contents	2
4. Principles of STAAD.Pro - STAAD	4
1) Modelling	4
2) Analysis	5
3) Post Processing	6
5. Important Windows Concepts	6
1) The Active Window	6
2) Menus	7
3) Toolbars	7
4) Selection Methods	7
5) Useful Keys	8
6. Example 1 – 2D Beam Model	8
1) Setup Model	9
2) Snap Node / Beam	10
3) Insert and Move a Node	11
4) Assign Properties (Database)	12
5) Assign Supports	12
6) Assign Releases	13
7) Assign Loads	13
8) Analyse	14
9) Results - Displacements	15
10) Results - Forces	16
11) Printout – Output File	16
12) Printout – User Report	18
7. Example 2 – 3D Beam and Plate Model with Steel Design	18
1) Generate 3D	18
2) Add Beam Elements	19
3) Add Plate Elements	19
4) Assigning Plate Properties	20
5) Creating Views	20
6) User Analysis Results Tables	21
7) Plate Element Contour Maps	21
8) Animations	22
9) Steel Design	23
10) Design Results	24
8. Example 3 – Using Cut and Paste and User Tables	24
1) Joints - Spreadsheet Input	25
2) Add Beams	25
3) Generate - Mirror	26
4) Cut/Copy and Paste Elements	27
5) User Tables	27
6) Add 1.5m x 2.0m Foundation Supports	28
7) Multiple Loadcase and Combinations	30
8) Analysis Results	30



STAAD.Pro:- STAAD Training

9. Example 4 – Structure Wizard and Dynamic Analysis.....	31
1) Structure Wizard.....	31
2) Add Properties and Supports.....	32
3) Dynamic Analysis Overview.....	33
<i>Mass Modelling</i>	33
<i>Mode Shapes</i>	33
2) Create Time History Graphs.....	34
3) Create a Time History Loadcase.....	34
<i>Masses</i>	34
<i>Time History</i>	34
4) Viewing Mode Shapes.....	35
10. Example 5 – Multiple Analysis.....	36
1) Defining Beams as Tension Only.....	36
2) Modify the Loading.....	36
3) Add Multiple Perform Analysis Commands.....	37
11. Example 6 – Beam, Column and Slab RC Design.....	38
1) Create Beam and Columns.....	38
2) Create First Floor Slab.....	38
3) Define Section Properties and Supports.....	38
4) Floor Loading.....	39
5) Slab Design.....	40
6) STAAD.Pro RC Design (Beam and Column design).....	41
1. <i>Create envelope</i>	41
2. <i>Create members</i>	41
3. <i>Create Design Briefs</i>	42
4. <i>Create Design Groups</i>	42
5. <i>Design the Group</i>	42
12. Detailed Steel Design Example to BS 5950: Part 1:2000.....	45
Example 1, Simply supported beam with full lateral restraint.....	45
Example 2, Simply supported beam with restraint at load points.....	49
Example 3, Simply supported beam without intermediate restraints.....	53
Example 4, Column with pinned ends and intermediate support.....	55
13. Other STAAD Features.....	57
Load Generators.....	57
Other Forms of Analysis.....	57
<i>P-Delta</i>	57
<i>Non-Linear</i>	57
Stardyne Analysis Engine.....	57
14. Other STAAD.Pro Components.....	58
STAAD.beava.....	58
STAAD.etc.....	58
Layout.....	58
Offshore Loading Program.....	58
Section Wizard.....	58

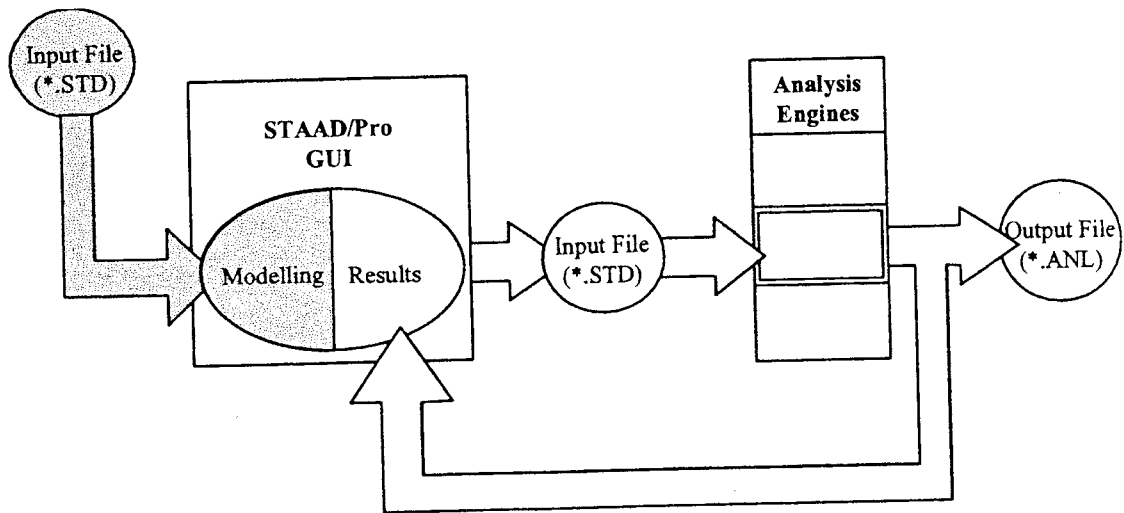


4. Principles of STAAD.Pro - STAAD

STAAD.Pro - STAAD is an analysis and design software that uses a graphical user interface to generate a data file that is processed by an analysis engine. It can also view the results in the post processing mode.

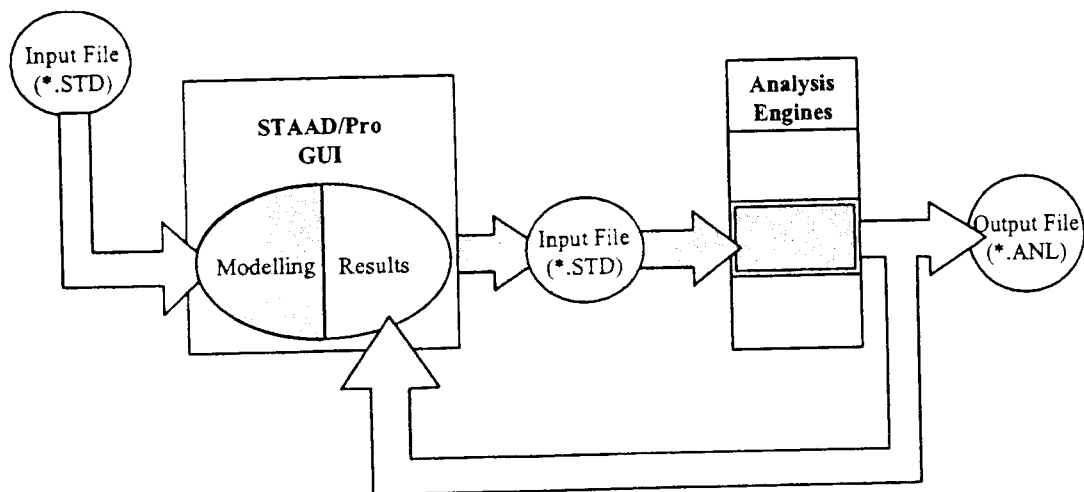
1) Modelling

The modelling stage of the process works with an ascii file format that has a file extension *.STD. This is opened by STAAD.Pro - STAAD and creates an internal database which the Graphical User Interface (GUI) then works with.



2) Analysis

When the modelling is complete and an analysis is required, the input file is updated and this is passed into the analysis engine. As there are so many design engines, when the analysis is started, STAAD.Pro - STAAD must be informed which analysis engine is to be used.

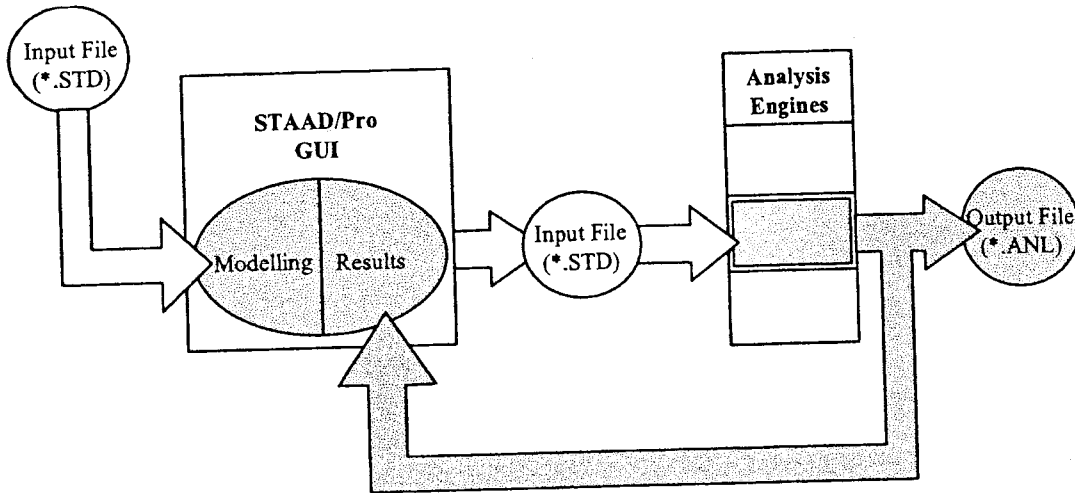




STAAD.Pro:- STAAD Training

3) Post Processing

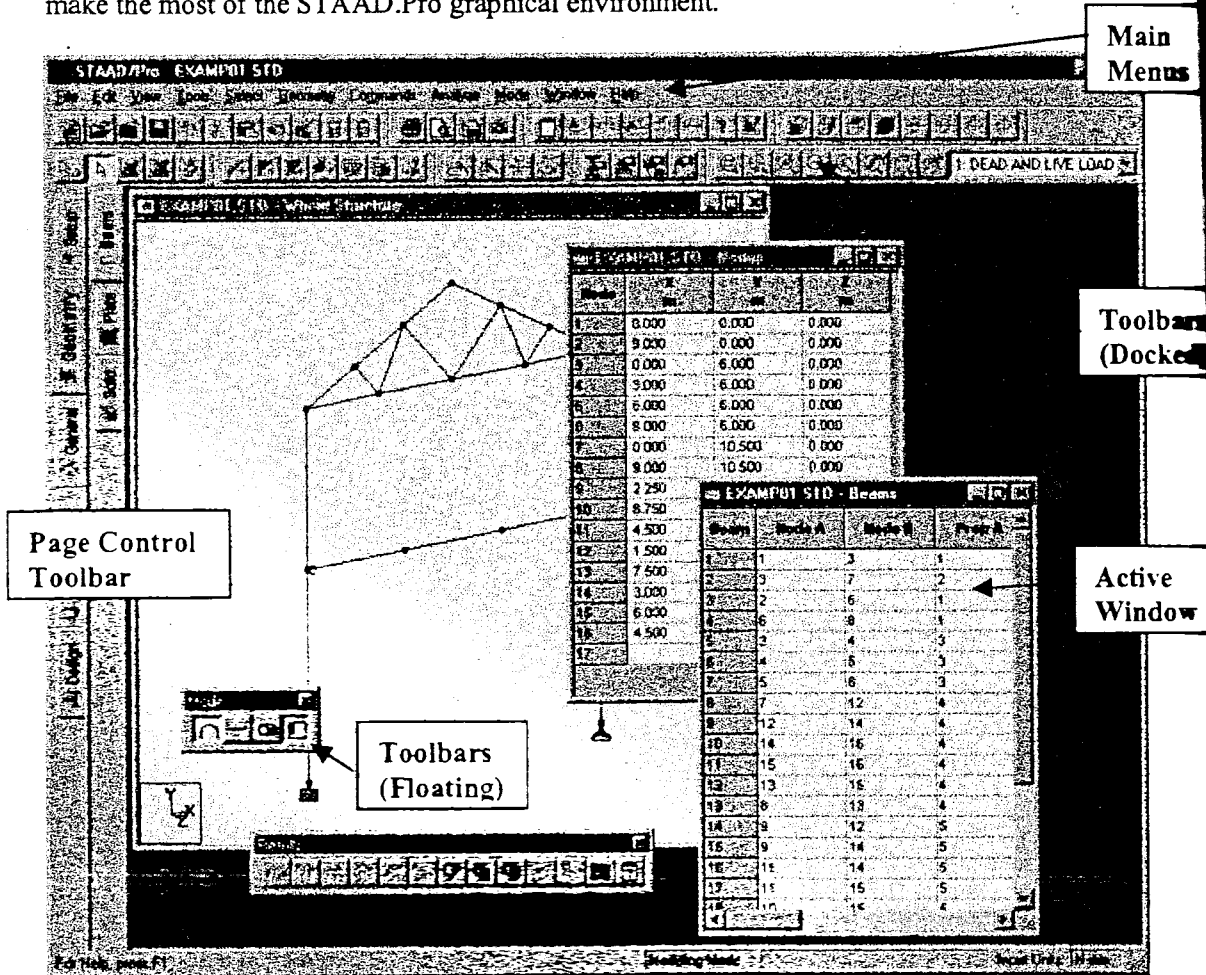
Completion of the analysis produces two sets of output. As the input file is processed, an output file with the same filename, but a .ANL extension is produced which can be viewed either within STAAD.Pro or an ascii file viewer. The analysis process also generates additional database files that are used with the GUI so that the user can interrogate the results.





5. Important Windows Concepts

There a number of general features of Microsoft Windows that are worth noting in order to make the most of the STAAD.Pro graphical environment.

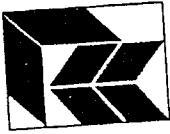


1) The Active Window.

STAAD.Pro is a MDI (Multiple Document Interface) program. That is the program operates with a number of 'child' windows within the main STAAD Window. The child window can be either text based, such as the tables, graphical such as the diagrams or dialogue boxes. Any operation is carried out on the 'Active' window. The active window can be identified by the dark title bar. (Note although this is true for standard Windows setup, the colour configuration can be altered from Windows Control Panel.) By clicking the mouse on a child window, that window becomes the active window. The menu and toolbars respond to the active window so that only those commands that are appropriate to the active window are available.

2) Menus

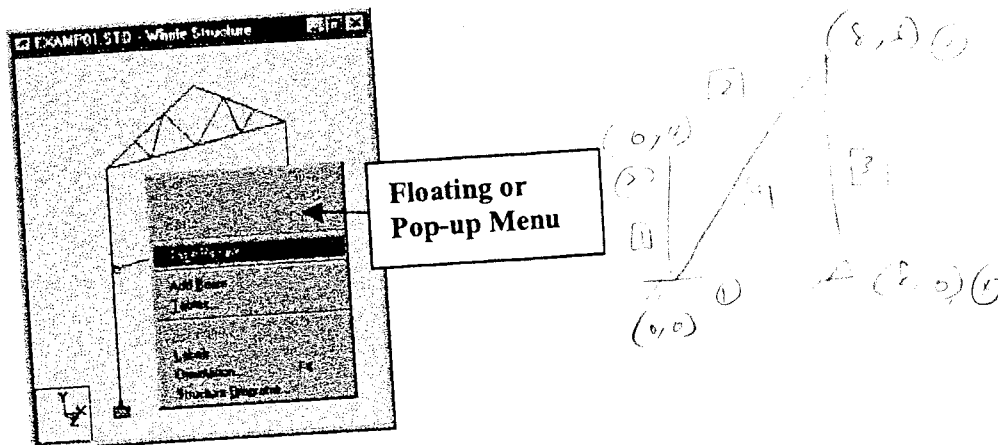
The primary method of accessing all the commands available in STAAD.Pro is from the 'Main menu' which runs across the top of the main application window. Note that this menu is sensitive to the active window. For example, if the active window is a table, then the



STAAD.Pro:- STAAD Training

command 'Copy Picture' does not appear in the EDIT menu. There are also features known as 'Floating Menus' available by clicking over a window with the right mouse button. These floating menus have commands that are available elsewhere but a right click can get to it quicker.

The 'Page Control menu' is unique to Research Engineers. This can be considered as a file management system. When turned on, the Page Control menu is fixed to the left of the main window and has two uses. Firstly it can be used to step through the stages of creating a model and secondly as a desk tidy, re-organising the child windows.



3) Toolbars

STAAD.Pro can access a number of the more common commands by clicking buttons on toolbars. These toolbars can be either 'Docked' to the sides of the main window or 'Floating' free. The buttons that each toolbar contains can be customised by selecting the 'Toolbars...' option from the View menu.

4) Selection Methods

There are several methods of making multiple selections in Windows that have been utilised by STAAD.Pro. Within a text window, such as the nodes or elements tables, it is possible to click in a label to select the row, and by holding the shift key while clicking on another label will also select all items in-between. Also, by holding the 'control' key down will add that item to the currently selected list. Within a graphical window, point and click will select either nodes, elements, etc. depending on the current selection mode, but by holding the mouse button down and dragging, a selection box will be created which will include all items whose greater part is within the box. As with text windows, by holding the 'control' key while making a selection, will add these to the current selection.

5) Useful Keys

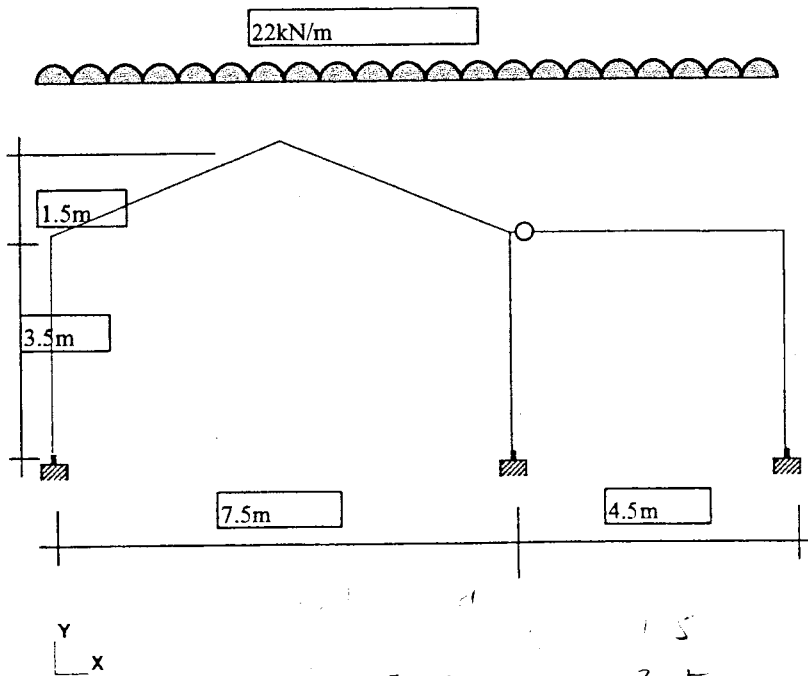
The graphical windows showing the structure can be rotated from the keyboard using the cursor keys. When entering data into dialogue boxes the 'tab' key changes between input fields.



STAAD.Pro:- STAAD Training

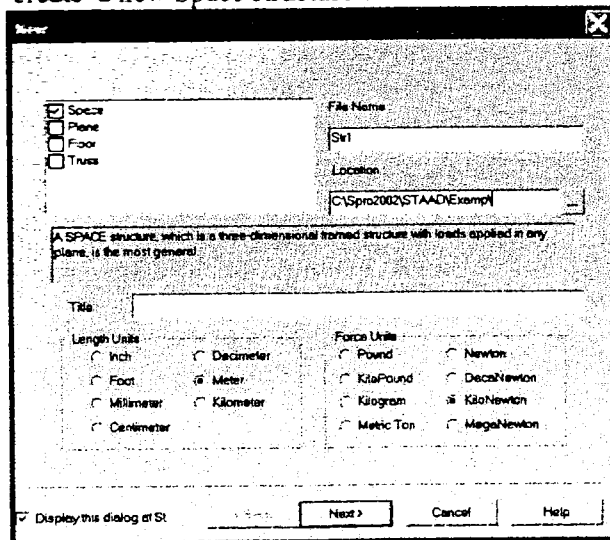
6. Example 1 – 2D Beam Model

This example shows some of the most basic functionality needed to quickly create, analyse and print a report.



1) Setup Model

Start STAAD.Pro & create a new Space structure based on Metres and KiloNewtons.




↓
- view
- Design
- load
- beam

These set the unit types of objects that are created such as plate thickness and loading.



2) Snap Node / Beam

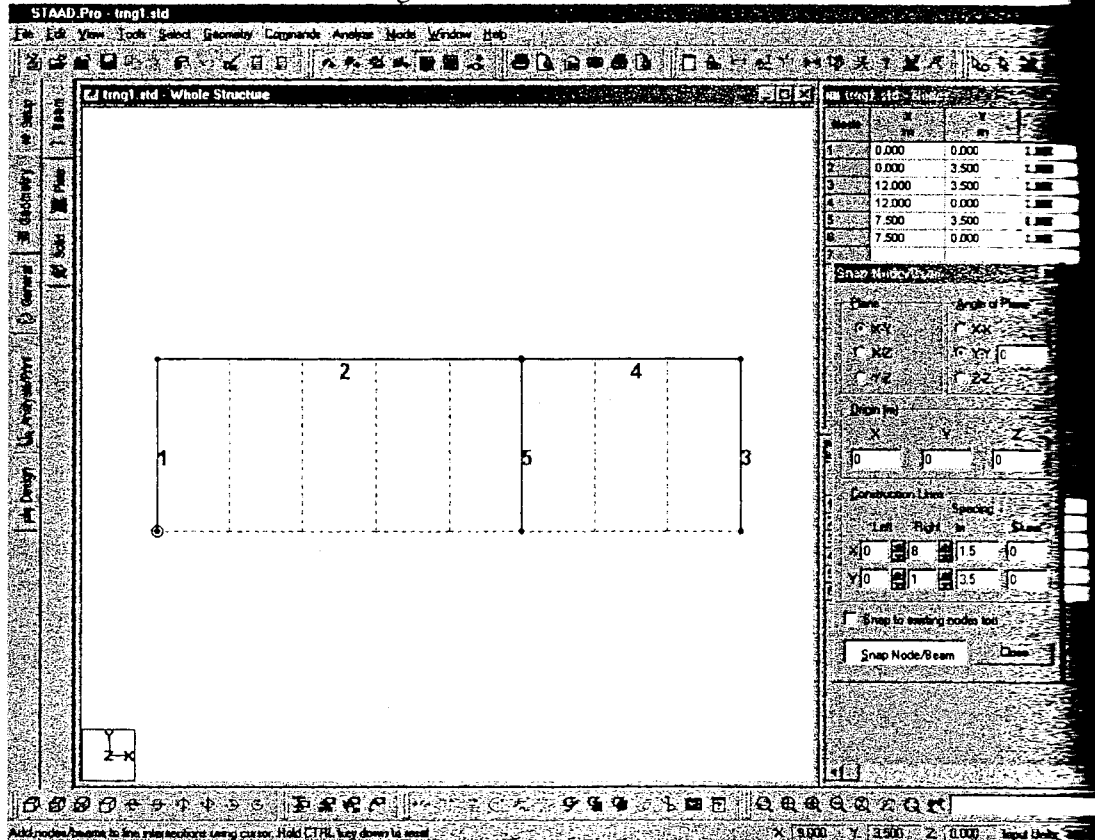
Go to the **Geometry Page**, staying on the **Beam** sub-page. This arranges a window to display the structure and nodes and beams tables. As there is no structure yet defined, the 'Snap Node/Beam' dialog is automatically opened displaying a grid on the structure so that nodes and beams can be added graphically.

1. In the Snap Node/Beam dialog, set the grid plane to the **X-Y** plane. Note the axes in the lower left corner of the structure window.
2. Click on the 'Front View'  button to get a front on view (X-Y plane) of the grid.
3. To set the grid to suit the dimensions of the structure modify the spacing to:-
X, 8 bays at 1.500m c/c
Y, 1 bay at 3.500m c/c
4. Note that the origin is set to 0, 0, 0. This is indicated on the grid as a circle. As you move the cursor over the grid, note the co-ordinate shown in the lower right corner which can be used as a visual check.
5. Click on the origin to generate a node.
6. Click on the grid line above to a new node and generate a column. Note the co-ordinate should read 0.000, 3.500, 0.000.
7. Click on the extreme top right corner of the grid, 12.000, 3.500, 0.000 , to form a horizontal beam. Follow this by clicking in the lower right corner, 12.000,0.000, 0.000, to form a column.
8. Now whilst holding down the CTRL key (only a node is required, not a beam) and click on the horizontal beam 5 bays from the left hand side at co-ordinate 7.500, 3.500, 0.000.
9. To insert the final column, without holding down the CTRL key, click on the grid below the last node to create a new node at 7.500, 0.000, 0.000, along with the column.




STAAD.Pro:- STAAD Training

The screen should look something like this:-




Click on the **Close** button to turn off the grid..

All the node co-ordinates and beam numbers have been automatically created. From the Structure toolbar click on the 'Symbols and Labels'  button and turn on the 'Node Numbers' and 'Beam-Numbers'.

The format and locations of labels can be modified by selecting **Options** from the *View* Edit menu.

3) Insert and Move a Node

From the Structure toolbar press the 'Insert Node'  button. Note that the cursor has changed shape when over the structure window. From the structure window select beam number 2, the top element of the larger span. In the resulting dialogue box press the 'Add mid point' button and 'OK'.

Change the selection cursor from the 'Select' menu to 'Nodes Cursor'.

Again note the change to the cursor when above the structure window. Select the newly created node and click with the right mouse button to get a Floating menu. Select 'Move' from the menu and enter 1.5 into the Y: field of the dialogue box that is presented. And click on OK.

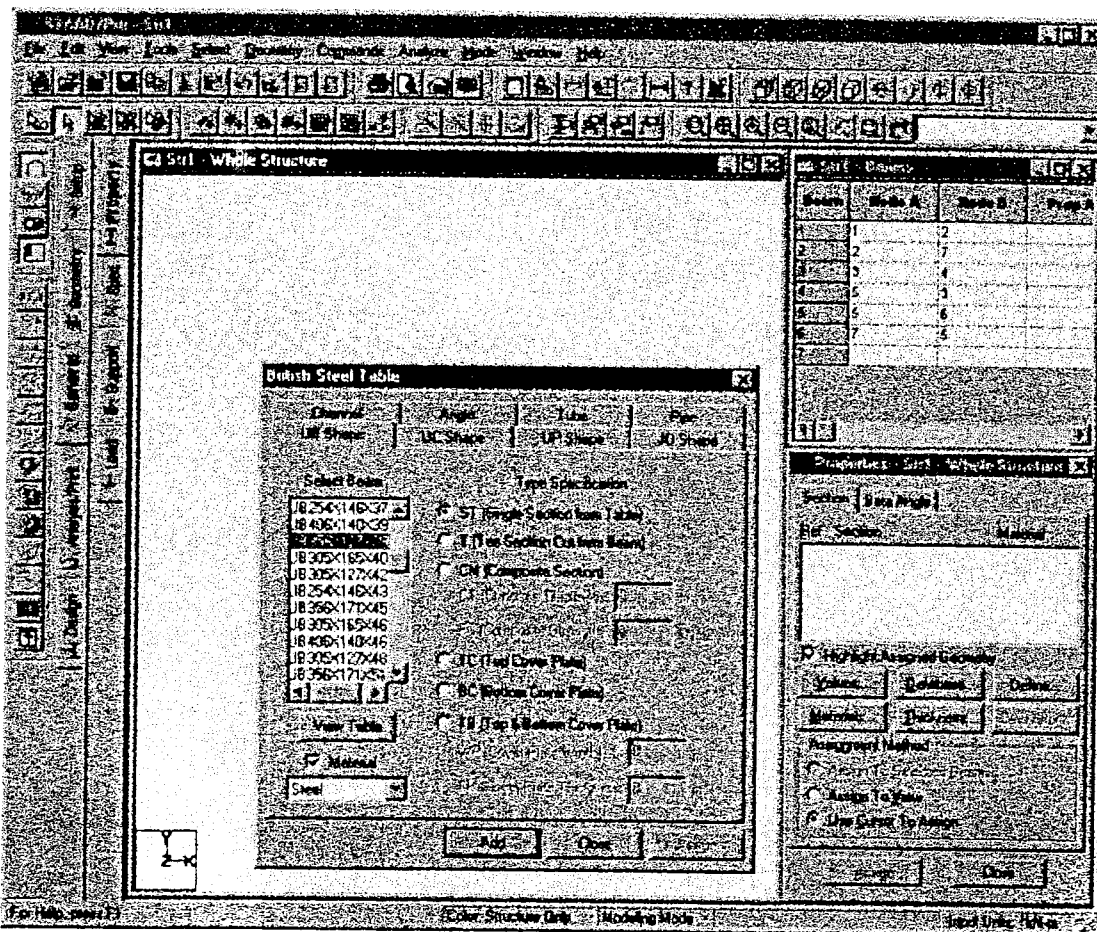


STAAD.Pro:- STAAD Training

4) Assign Properties (Database)

Change to the **General Page**, staying on the **Property** sub-page. From the 'Assign Properties' Dialogue box press the '**Database...**' button. Select a '**UB 356x127x33**' from the British database. This is the first item of a list of sections that will be used on the structure, more can be added in the same way.

The structure should now look like the following diagram;



Note that when a section is chosen from the database in this way, it has associated with it a material property that will also get assigned when this section is assigned to the beams of the structure.

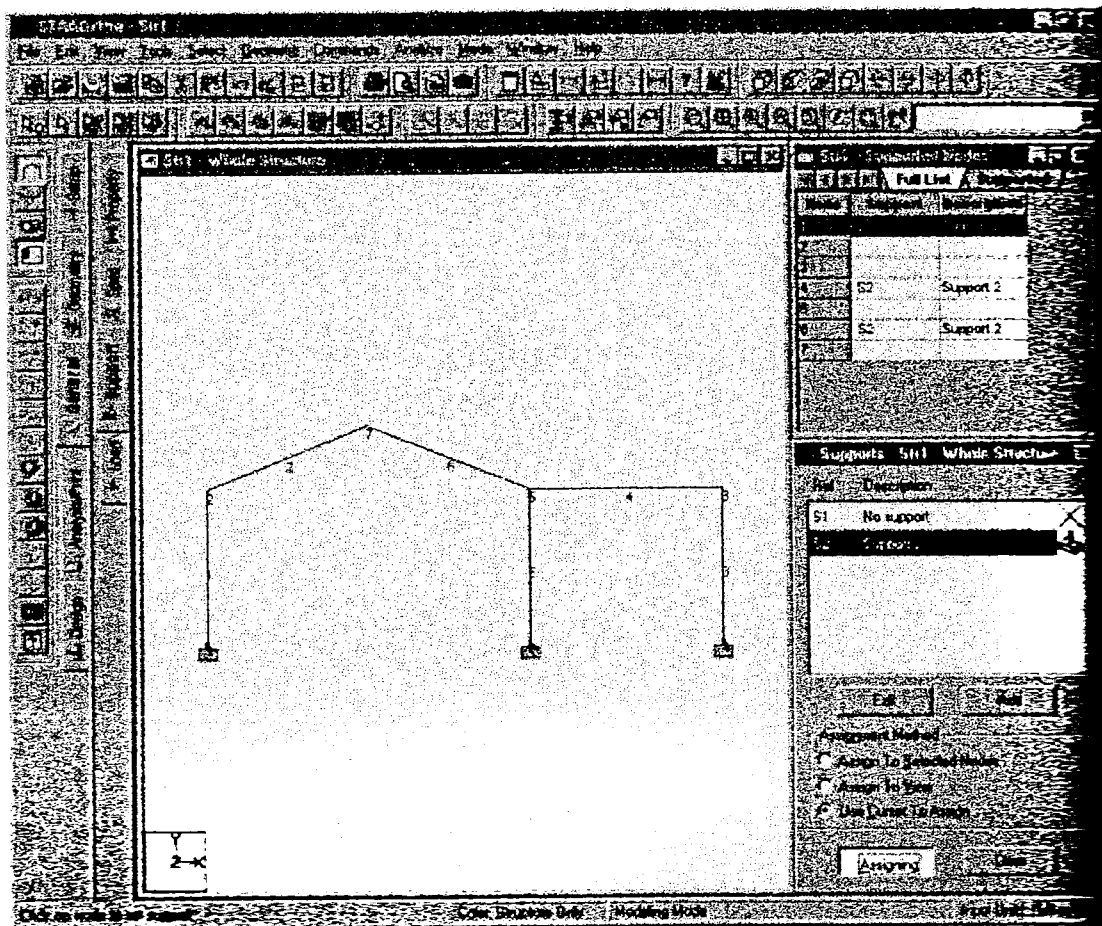
Select the 356UB from the list, select on the option 'Assign to View' and click on the 'Assign' button. All the elements in the structure now have this section property.



STAAD.Pro:- STAAD Training

5) Assign Supports

Staying in the **General Page** change to the **Supports Sub-Page**. Create a fully fixed support to the 'Support' dialogue by clicking on the **Add** button. To attach this support to the structure, select the support in the dialog box, check that the Assignment method is set to 'Use cursor to Assign' and click on the **Assign** button. The cursor changes to the support icon when over the structure window. Click over the bases of the columns to provide the support. The structure should now look like this.



6) Assign Releases

Staying in the **General Page** change to the **Spec Sub-Page**. Create a release by clicking on the **'Beam...'** button and define a release that is free in **MZ** and occurs at the start. Assign to member 4.




STAAD.Pro:- STAAD Training

7) Assign Loads

Staying in the **General Page** change to the **Loads Sub-Page**. When entering this page, a loadcase must either be selected or defined. Thus for the first time the loadcase must be created. Enter the title '**Dead**' and click on OK. The beam load needs to be created, thus click on the '**Member...**' button. Select the **Uniform Force** tab and enter the value **-22 kN/m** into the **W1** edit box and as a **Projected** load is required, set the direction to **PY** and click on the **Add** button.

This creates a load that can be applied to the structure. This time, pre-select the three roof beams 2, 4 and 6. With the assignment method set to '**Assign to Selected Beams**', click on the **Assign** button.

8) Analyse

To define the method of analysis and additional information that is required in the standard output file. Change to the **Analysis/Print** page and go to the **Analysis** sub-page. Click on the '**Define Commands...**' button. On the '**Perform Analysis**' tab, with **No Print** selected, click on the **Add** button to insert the command, followed by the **Close** button. The file is now ready to be analysed, but can be checked in the text editor by clicking on the **Editor**  button. At this point the file is saved to disk and is shown thus:-

```
STAAD.Pro
-----
START PAGE: IMPROVED
ENGINEER DATE: 10-10-00
MEMBER INFORMATION
INPUT FILE: 00
UNIT SYSTEM: MK
JOINT COORDINATES
1 0.000 0.000 0.000
2 0.000 7.000 0.000
3 10.000 0.000 0.000
4 10.000 7.000 0.000
5 0.000 0.000 10.000
6 0.000 7.000 10.000
MEMBER INFORMATION
1 1 2
2 2 3
3 3 4
4 4 5
5 5 6
MEMBER PROPERTY DEFINITION
1 TO 6: IZAL AT ENDCOINTEZED
CONTINUED
1 2: IZALGZ WANE 1 TO 4
PROPERTY SET NAME: 1 TO 4
LOADS: 1 TO 4
MEMBER PROPERTY DEFINITION
1 4: FINER
RELEASE RELEASE
4: DIAPHRAGM
LOADS: 1 TO 4
MEMBER LOADS
1 2 4: UNIF. PY -22
PERFORM ANALYSIS
FINISH
```

View the file and close it without modifications, or STAAD will reprocess the file returning to the graphical environment.

Click on the '**Analyze**' menu and select '**Run Analysis...**' On the '**Analysis Option**' tab, ensure that the '**STAAD Analysis**' option has been selected and click on the '**Run Analysis...**' button. Save the file if required.

Once the analysis has been completed, click on the **Done** button to close the analysis engine. The structure now has results which can be viewed.



STAAD.Pro:- STAAD Training

9) Results - Displacements

To view results, the program needs to be switched to the Post Processing Mode. This can be done either from the Mode menu or by clicking on the **Post Processing** button. If there are no results available for a structure, then this button is disabled.

When entering this mode, the results to be displayed in the tables has to be selected. Use the '>>' button to select loadcase 1 'Dead' and click on the **OK** button.

The first page that is entered in the Post Processing mode is the **Node** page with the **Displacements** sub-page. The structure is displayed with the displacement diagram. However the displacement scale needs to be set to an appropriate value. Click on the **Scale** button and set the displacement scale to **10 mm/m**.

Identify the displacements of nodes in the table by clicking on the nodes on the diagram and select a displacement in the table and note how it is located on the structure.

As this is a plane structure, the displacements in the Z direction and the rX and rY directions are all zero and thus not required. The widths of columns can be resized, or set to zero and thus the size of the table can be reduced.

The screenshot shows the STAAD.Pro interface with a structural model of a truss structure. The model has nodes labeled 1 through 6. Node 1 is at the bottom left, node 2 is at the top left, node 3 is at the bottom right, node 4 is at the top right, node 5 is at the bottom center, and node 6 is at the top center. The displacement diagram shows the structure with arrows indicating the direction and magnitude of displacements.

Two tables are displayed on the right side of the screen:

Table 1: Node Displacements

Node	LC	X	Y	Z	rx	ry	rz
1	1 Dead	0.000	0.000	0.000	0.000	0.000000	0.000000
2	1 Dead	-7.484	-0.348	0.000	7.502	0.000000	0.000000
3	1 Dead	0.783	-0.234	0.000	0.828	0.000000	0.001689
4	1 Dead	0.000	0.000	0.000	0.000	0.000000	0.000000
5	1 Dead	0.877	-0.491	0.000	1.005	0.000000	0.002673
6	1 Dead	0.000	0.000	0.000	0.000	0.000000	0.000000

Table 2: All Relative Displacement / Max Relative Displacement

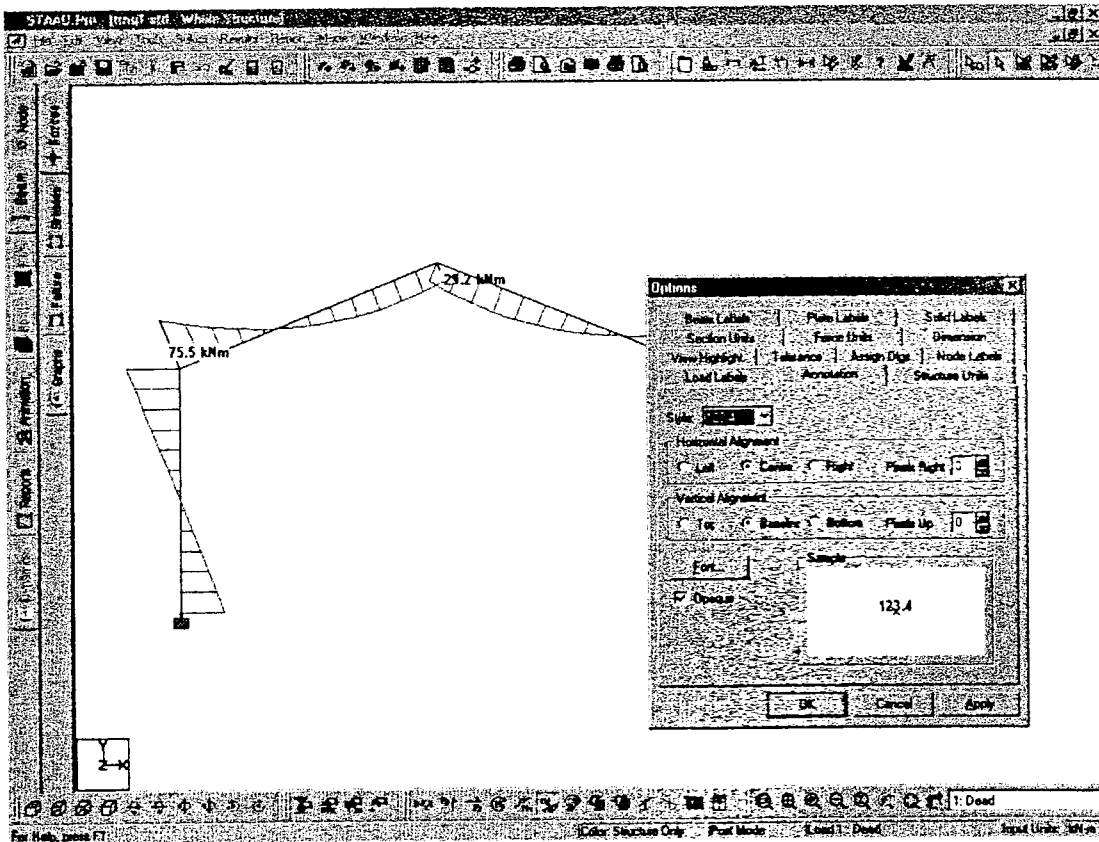
Node	LC	Max	Y	Z	Max	Max	Max
1	Dead	0.000	0.000	0.000	0.000	0.000	0.000
		0.975	0.454	-0.000	0.000	0.454	0.457
		0.750	-0.457	-0.000	0.000	0.457	1.139
		2.825	-1.139	0.000	0.000	1.139	0.000
		0.500	0.000	0.000	0.000	0.000	0.000
2	Dead	0.000	0.000	0.000	0.000	0.000	0.000
		1.810	-0.048	0.120	0.000	0.120	0.000
		2.019	0.398	-0.987	0.000	1.074	0.000
		0.028	0.538	-1.345	0.000	1.448	0.000
		0.000	0.000	0.000	0.000	0.000	0.000
5	1 Dead	0.000	0.000	0.000	0.000	0.000	0.000
		0.875	0.878	0.000	0.000	0.878	0.878
		1.750	0.743	0.000	0.000	0.743	0.743
		2.825	0.235	0.000	0.000	0.235	0.235
		0.000	0.000	0.000	0.000	0.000	0.000
4	1 Dead	0.000	0.000	0.000	0.000	0.000	0.000
		1.125	-0.000	-3.208	0.000	3.208	0.000
		0.350	-0.000	-4.165	0.000	4.165	0.000
		3.375	0.001	-2.520	0.000	2.520	0.000
		4.000	0.000	0.000	0.000	0.000	0.000
5	1 Dead	0.000	0.000	0.000	0.000	0.000	0.000
		0.975	1.353	0.000	0.000	1.353	0.000
		1.750	1.170	0.000	0.000	1.170	0.000



STAAD.Pro:- STAAD Training

10) Results - Forces

To view the forces on the beams, change to the **Beam** page and stay on the **Forces** sub-page. The structure diagram is displayed with the Mz moment (major axis) and the Beam End Forces Table. Note that clicking on the table or the diagram highlights the beam in the other window. To add values onto the diagram, maximise the Whole Structure window. Click on the Results menu and select the 'View Value...' option. On the 'Ranges' tab, select the 'Ranges' option and enter **2** into the 'Beams' edit box. Then change to the 'Beam Results' tab, click on the **Bending Ends** check box and click on the **Annotate** button. The labels have been added, but may not be sufficiently clear, so re-size the font. To do this click on the View menu and select the 'Options...' menu. On the Annotation tab, click on the **Font** button and set the font to **12pt, bold**. Click the **OK** button on the font and **Apply** on the Options dialog to show as below.

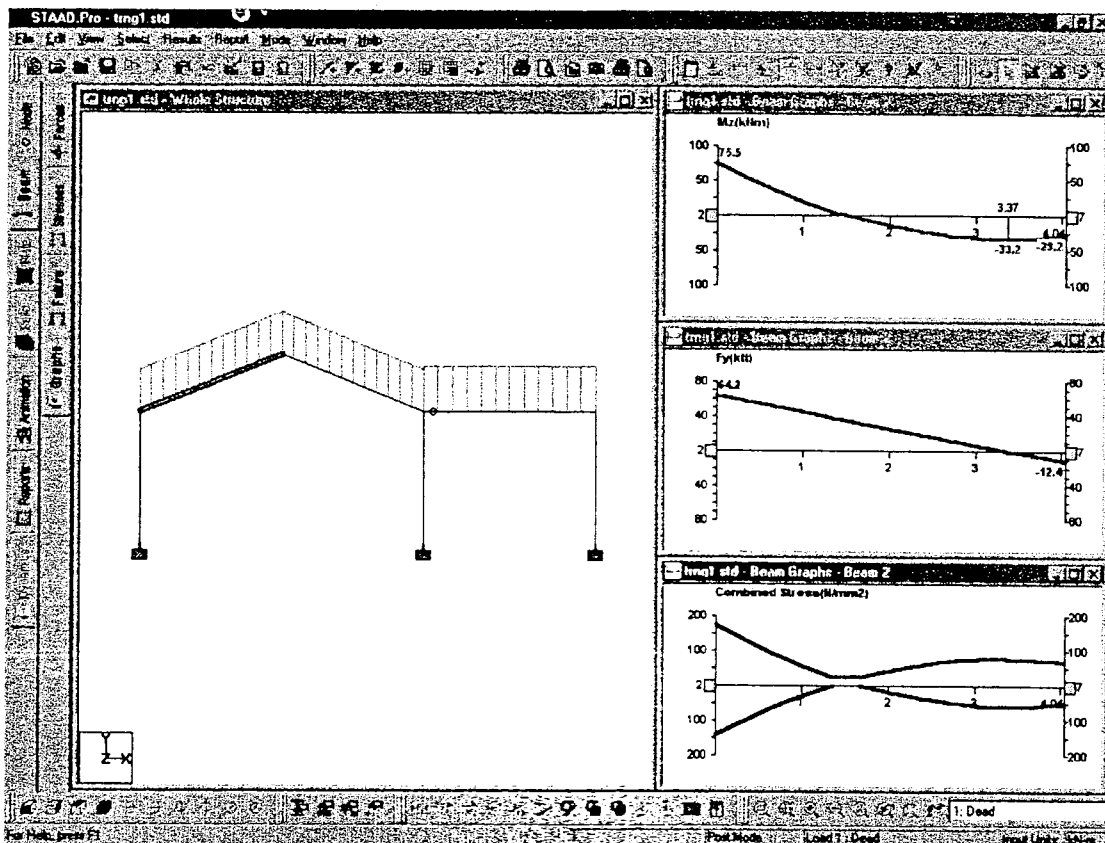





STAAD.Pro:- STAAD Training

To get individual bending moment graphs, change to **Graphs** sub-page. Click on the beams of the structure to view the bending moments, shear forces and axial force diagrams in the windows on the right hand side.


To view different diagrams, such as the beam combined stresses, click on bottom graph window to make it the active window. Click with the right mouse button over this window and select the '**Diagrams...**' option from the floating menu. Click on the **Stress** option to turn it on and on the **Axial** option to turn it off. Click on **OK** to accept the change to produce a page layout similar to this:-



11) Printout – Output File


There are two reports that can be printed. Firstly the Output file that is automatically created by STAAD as the analysis is performed. To view this file press on the **STAAD Output**  button. This is the form of output STAAD III users will be familiar with.

12) Printout – User Report

The second method is to use a User Report. With this method information can be selected and organised for a particular situation, including tables and pictures. Pictures can be any graphical view of the structure, including results. Take a picture now, ensure that the required window is active, and click on the **Take Picture**  button. Accept the default comment. This becomes Picture 1, any number of pictures can be taken and added to a User Report.



STAAD.Pro:- STAAD Training

To setup a User Report, click on the **Report Setup**  button. Select Nodes and Beams from the Input, select Node Displacements from the Output and Picture 1 from the Picture Album. Click on the 'Load Cases' tab and select the **Dead loadcase**. Click on **OK** to accept this report setup.

Finally click on the **Print Preview**  button to view the report.

The screenshot displays the 'Report Setup' dialog in STAAD.Pro. The left pane contains the following sections:

- Job Information:** Fields for Job Name, Date, and User.
- Nodes:** A table with columns for Node No., X, Y, Z, and a checkbox for selection.
- Beams:** A table with columns for Beam No., X1, Y1, Z1, X2, Y2, Z2, and a checkbox for selection.

The right pane contains the following sections:

- Node Displacements:** A table with columns for Node No., U, V, W, and checkboxes for selection.
- Picture Album:** A 3D wireframe model of a structure, likely a frame or truss, with a coordinate system (X, Y, Z) shown at the bottom left.

The dialog has a 'Print Preview' button at the bottom right and several tabs at the bottom: 'Job Information', 'Nodes', 'Beams', 'Node Displacements', and 'Picture Album'.

This concludes the exercise Example 1.



7. Example 2 – 3D Beam and Plate Model with Steel Design

This example expands on the model created in Example 1.

1) Generate 3D

There are a number of generation tools available in STAAD.Pro. They are accessible from the Geometry menu or the Generate toolbar.



Open the model from Example 1 and save as 'Example 2'. Go to the **Geometry** page. Use the **Beams** cursor to select all the beams. Click on the **Translational Repeat** button on the Generate toolbar or select the '**Translational Repeat...**' item from the Geometry menu. In the dialog box, set the direction to the **Z axis**, **5 steps @ 5m c/c**, but change the third bay to **6m**. Tick the **Link bays** and **Open base** options. Click on the **OK** button to generate the structure. Note how the section profiles, supports and loads have been copied onto the new frames, but properties have to be added onto the linking members.

2) Add Beam Elements

Beam elements can be added graphically onto the structure view windows.

Click on the **Add Beams**

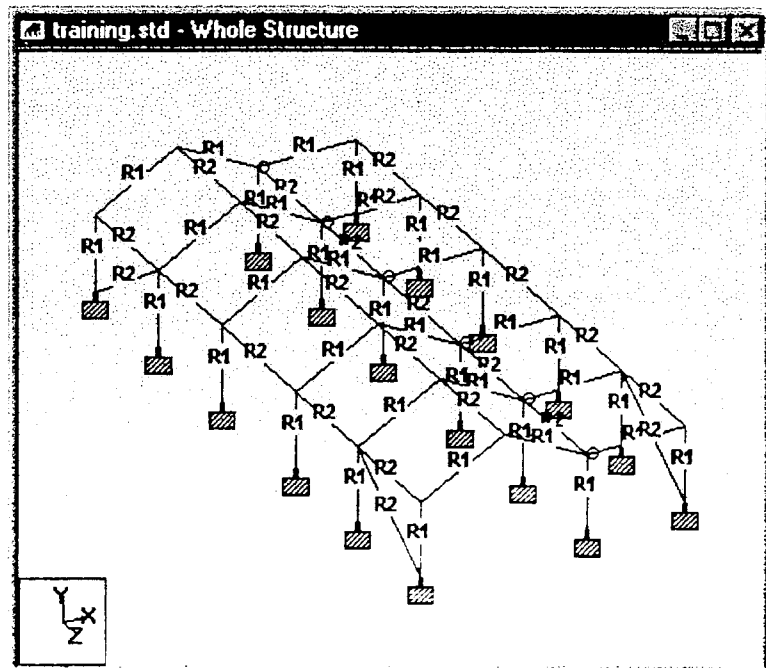
button on the Geometry  toolbar, Note how the cursor

changes when it is over structure view.

Add bracing elements by clicking on the start joint and end joints of the end frames as per the above diagram, rotating the structure if necessary.

As with the link beams from above, these new beams also need properties. Go to the **Property** sub-page and add a '**UB 254x146x31**' from the British Database.

To assign the new property, click on the **Side View** button, select the link beams and the diagonal bracing, highlight the new property and click on the **Assign** button.






STAAD.Pro:- STAAD Training

3) Add Plate Elements

STAAD.Pro models can include plate and solid elements. Set the plates to be filled. From the View menu, select Structure Diagrams. Select **Fill Plates/Solids** and **Sort Geometry**.

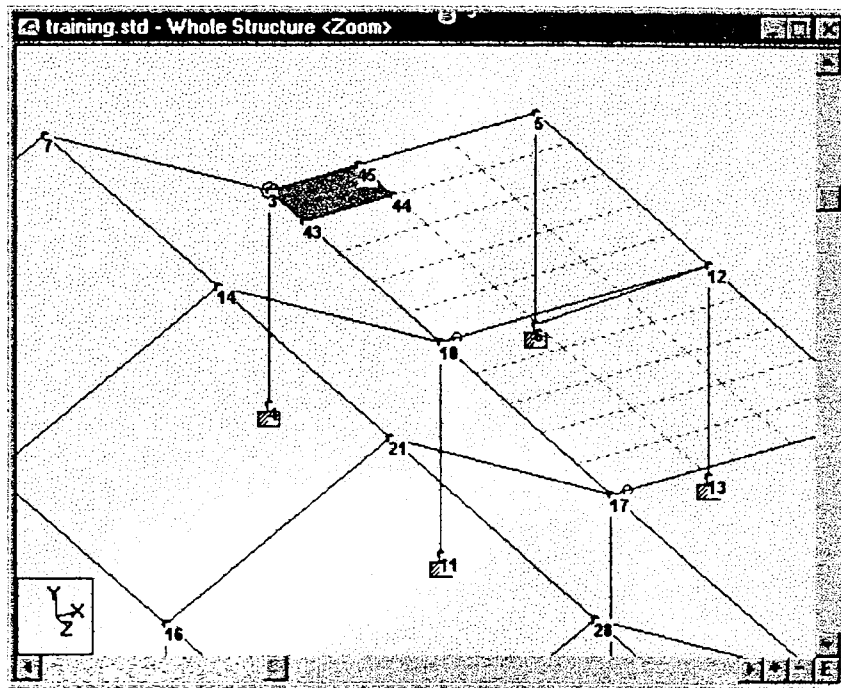
Start the Snap Node Plate dialog by clicking on the **Snap Node/Plate**  button on the Geometry toolbar.

Locate the origin to the top of the end bay, ie $X=7.5$, $Y=3.5$, $Z=0.0$

Set the grid to 3 bays @ **1.5m** along the X axis.

Starting at the origin click anti-clockwise on the grid to form a plate **1.0m x 1.5m**.

The last joint on the plate becomes the first joint on the next plate so that only three other joints need to be defined. Again click anti-clockwise to define a second and third plate along the lean-to main beam. Once the three plates have been created, close the Snap Node/Plate dialog.



To fill in the rest of the roof plane, select the plates using the **Plates Cursor**. Select the **Generate Repeat** from the toolbar or Geometry menu and set the direction to the Z axis and **25** steps of **1.0m**. Note that the **Link Steps** option should not be selected.

Note that where the generated plates have introduced joints along beam lines, beams will be split and new beams will be formed with the section and material properties of the original provided the tolerance is set to Test for added node breaking a beam.

4) Assigning Plate Properties

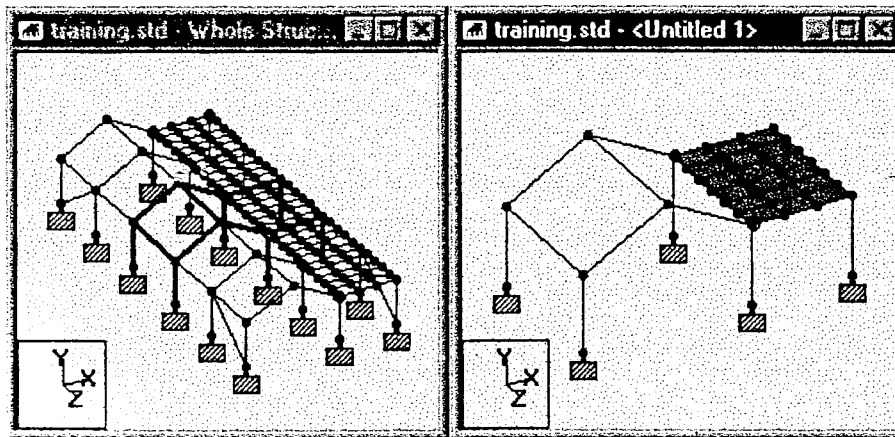
Plate properties are determined by the thickness and material properties which can be specified in the Properties sub-page. On the **Properties** dialog, click on the **Thickness...** button and specify a thickness of **0.25m**. Note that plates can be specified with varying thickness', but if only the node 1 value is specified, then a uniform thickness is assumed. Select the plate property, select the 'Assign to View' option and click on the **Assign** button.



STAAD.Pro:- STAAD Training

5) Creating Views

When working with larger models, it is possible that only a part of the structure is required. These can be defined as views. To create a view, select the part of the structure that is required and from the View menu, select the New View option.



To create the view as above, click on **Plan View**. Change the selection cursor to the **Geometry Cursor** so that both the beams and plates can get selected. Select the middle bay and select **New View** from the View menu. Select the option 'Create a new window for the view' and click **OK**. Note that a Whole Structure window will highlight beams of the current view as shown above.

The View Management, in the View menu, allows views to be named by selecting the **Save View** option and entering a name. With the new view (currently named as <Untitled 1>) as the current window, select the **Save View** and name it as 'Middle Bay'.

As the current view is used for the structure when pages are entered, ensure that the Whole Structure view is the active window when changing to the next page. If the Whole Structure view is closed it can be re-opened at any time from the View menu.

6) User Analysis Results Tables

It is possible to create results tables by selecting groups of joints, beams or plates which can be saved away and included in a User Report.

Perform an analysis as in Example 1, change to the Post Processing mode and select the Dead loadcase to display in the results tables.

Using the Joints cursor, select a number of joints by dragging a window across the structure or holding down the Ctrl key and clicking on a number of joints. From the Report menu select **Joint Displacements**. Select the sort method to **Absolute Displacement**. Change to the Report tab, select **Save Report** and in the Id box, enter the name 'First set of node displacements'. Click **OK**.

The table is now displayed on screen. Open the Report Setup dialog, and note that as the table was Saved, the table has been added to the user report. Close and view the report with **Print Preview**.



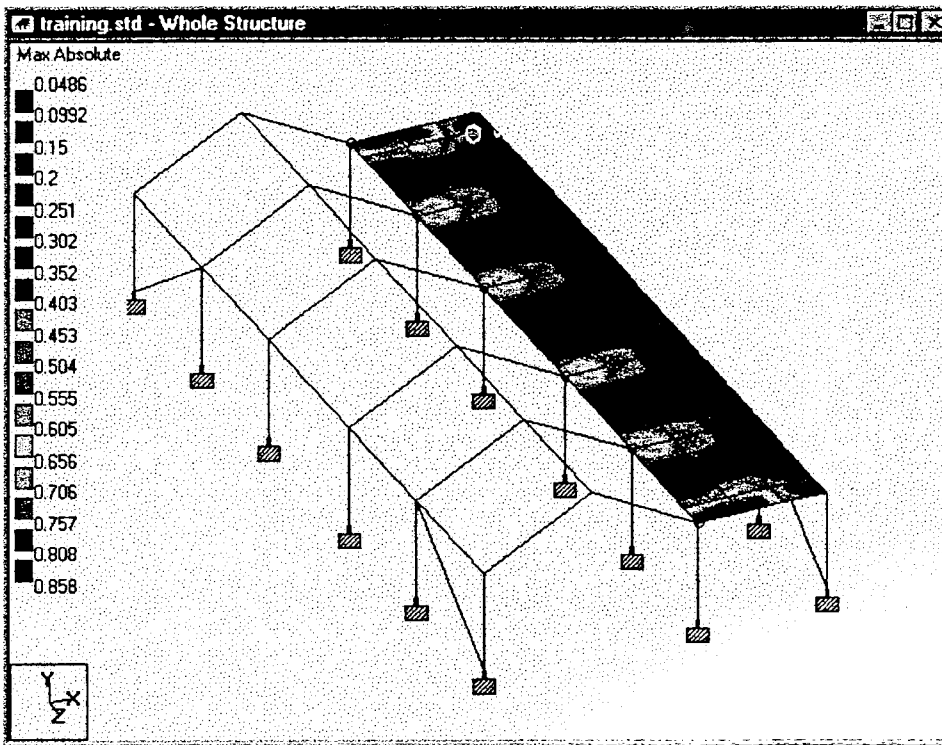
STAAD.Pro:- STAAD Training

7) Plate Element Contour Maps

Note that the Plate Page has been activated by the presence of plate elements.

From the View menu, select the Options item. Change to the Force Units tab and set the Stress to display N/mm^2 .

Change to the Plate Page. Select the Maximum Absolute stress type and click on OK.



8) Animations

Both displacements and contour maps can be animated.

To animate the displacements, change to the Animation Page. Select **Section Displacement**, set Target FPS (frames per second) to **10** and press **OK**. Press **Esc** on the keyboard to stop the animation. The second displacement option **Deflection**, animates the deflection of the joints only rather without including the curvature in the beams.

To animate the stress contours, select **Structure Diagrams** from the View menu. On **Plate Stress Contour** tab select **Max Absolute** stress type. Go to the Animation tab and select **Stress** and click on **OK**.

Press **Esc** to stop.



STAAD.Pro:- STAAD Training

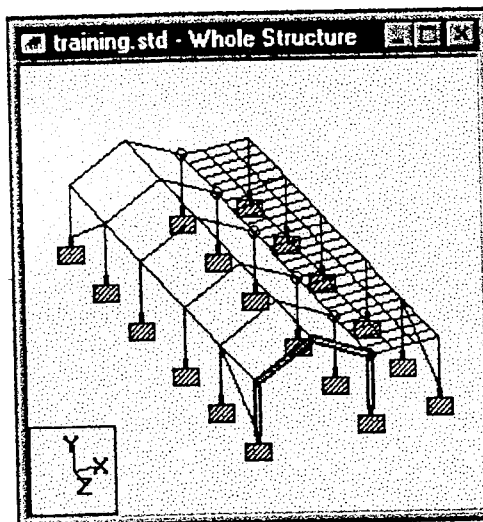
9) Steel Design

STAAD.Pro can perform a steel design on the analysis results of beam elements and concrete design on the results of beam and plate elements. The scope of design to be carried out are set as parameters in the Design page.

Change back to Modelling Mode.

Go to the Design Page. In the Steel Design dialog Select **BS5950**. Click on the **Select Parameters** button and remove all the parameters except the **Beam** and **Track** parameters and click **OK**.

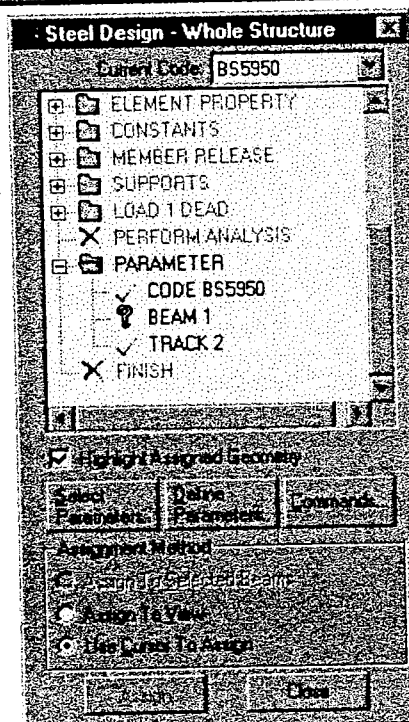
Holding the **Ctrl** key, select the beams that form the near portal frame.



Click on the **Define Parameters** button, set the **Beam** parameter, which sets what moment values should be used in the steel design, to **1.0** and press **Add**. This includes the parameter but does not define which beams it applies to.

Click on the **Track** tab, which sets the level of detail the design should produce in the output file, and set the parameter to **2.0** and press **Assign**. This includes the parameter and also defines which beams it applies to.

Note that a parameter block has been added with the code set to **BS5950**. The **Beam** parameter has a question mark to show that it has not been assigned and needs to be before it will get added to the file. To assign the **Beam** parameter, remove the **Highlight Assigned Geometry** option, and click on the **Beam 1** label to highlight it. The **Assign** button becomes available with the **Assignment method** set to **Assign To Selected Beams**. Click on the **Assign** button to assign the parameter.



Once all the parameters have been set, the command to perform the code check needs to be set. With the frame still selected, click on **Commands...** button and press **Assign** (or **Add** and **Assign to Selected Beams** as per the **Beam** parameter).

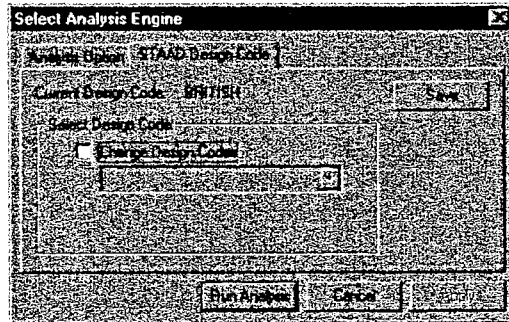
A steel design has now been setup.



STAAD.Pro:- STAAD Training


10) Design Results

Start analysis from Analyze menu. This time it is important to check the current Design Code setting. Click on the STAAD Design Code tab and ensure the current design code is set to **BRITISH**, then click on the **Run Analysis** button.

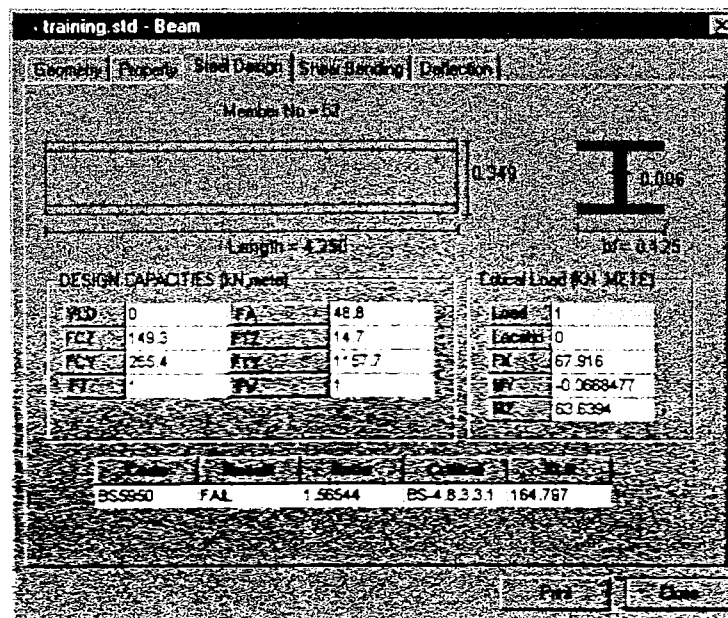


Note that design codes are optional extras, please check you have purchased a design code before trying to use it.

There are 3 ways to view the design results.

1. Firstly the View Output file. Click on the **STAAD Output**  button and go to page 6 onwards.

2. Secondly, using the member query by double clicking on one of the beams. When a member has been designed it has a Steel Design tab such as the one shown.



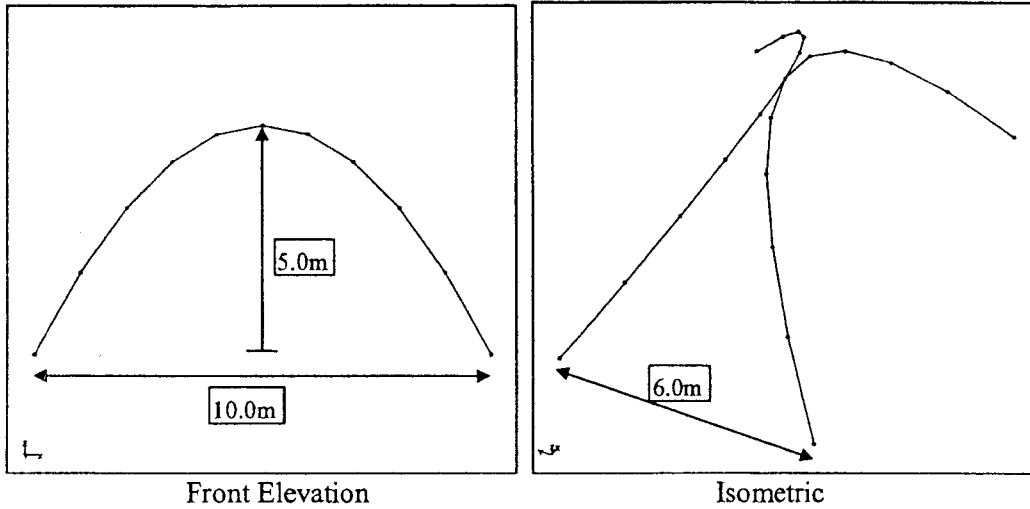
3. The third method is to display a Design Results table. Change to **Post Processing** mode, change to the **Beam Results** page and **Failure** sub-page. The structure diagram is colour coded and those beams that have been designed are labelled with their maximum design ratio.

This concludes the exercise Example 2.



8. Example 3 – Using Cut and Paste and User Tables

This example shows some of the more advanced features available with STAAD.Pro. This structure is based on two back to back parabolic arches.



Clearly the structural geometry can be defined using simple mathematical equations and thus we can utilise the power of a spreadsheet to calculate the co-ordinates for us.

1) Joints - Spreadsheet Input.

The vertical co-ordinates for each node on one arch is defined by the following equation:-

$$Y = 5 - ((X-5)^2 \times 5 / 25)$$

and the lateral offset:-

$$Z = 3 - ((X-5)^2 \times 3 / 25)$$

With these simple equations added into the spreadsheet, and dragging them for a range of values for X of 0 to 10 in 1 meter steps, all the co-ordinates for one arch will be created thus.

	A	B	C
1	X	Y	Z
2	"1-10"	"5-((A2-5)^2*5/25)"	"3-((A2-5)^2*3/25)"
3	0	0	0
4	1	1.8	1.08
5	2	3.2	1.92
6	3	4.2	2.52
7	4	4.8	2.88
8	5	5	3
9	6	4.8	2.88
10	7	4.2	2.52
11	8	3.2	1.92
12	9	1.8	1.08
13	10	0	0



STAAD.Pro:- STAAD Training

These values can now be highlighted and 'copied' to the clipboard using either the 'Copy' option from the 'Edit' menu or pressing the 'Ctrl and C' keys. They are now ready to insert into STAAD.Pro. Launch the program, if it is not already running, start a new structure and go to the **Geometry** page. Close down the 'Snap node/beam' dialogue box and make the **Nodes** table the current window. Go to the 'Edit' menu and this time select 'Paste'. The values have been inserted into the table and show on the diagram. Note that the nodes are automatically numbered from the point you selected in the table.

The screenshot shows the STAAD.Pro software interface. The main window is titled 'Str1 - Whole Structure' and displays a 2D structure diagram with 12 nodes. A table titled 'Str1 - Nodes' is open, showing the coordinates for each node. The table has columns for Node, X (m), Y (m), and Z (m). The data is as follows:


Node	X (m)	Y (m)	Z (m)
1	0.000	0.000	0.000
2	1.000	1.800	1.080
3	2.000	3.200	1.920
4	3.000	4.200	2.520
5	4.000	4.800	2.880
6	5.000	5.000	3.000
7	6.000	4.800	2.880
8	7.000	4.200	2.520
9	8.000	3.200	1.920
10	9.000	1.800	1.080
11	10.000	0.000	0.000
12			

Below the nodes table, there is a 'Str1 - Beams' table with columns for Beam, Node A, Node B, and Prop A. The structure diagram shows a series of nodes connected by beams, forming a curved path. The status bar at the bottom indicates 'Modeling Mode'.

2) Add Beams

Make the structure diagram current by clicking inside the window. From the Geometry toolbar, click on the **Add Beams** button. Note that the cursor has changed shape when in the structure window to show that it is in this mode. Click on the first joint, then the second to form a beam between the two. Click on the second joint, then the third to create a beam between these two joints. Repeat this to create all the beams.

3) Generate - Mirror.

To form the other half of the structure is simply a case of selecting those beams that are to be mirrored and clicking on the **Mirror**  button.



STAAD.Pro:- STAAD Training

In this case select all the beams using a window or holding the control key and click on all the beams. Click on the **Mirror** button. The plane of the mirror is the **X-Y** plane with an offset of **3m**. Also ensure that the **Copy** option is toggled rather than Move. Finally press '**OK**'. If the final structure does not appear correct, remember that you can generally 'Undo' your last operation. This can be done from the 'Edit' menu.

4) Cut/Copy and Paste Elements

The structure can be repeated by using 'Cut / Copy and Paste'.

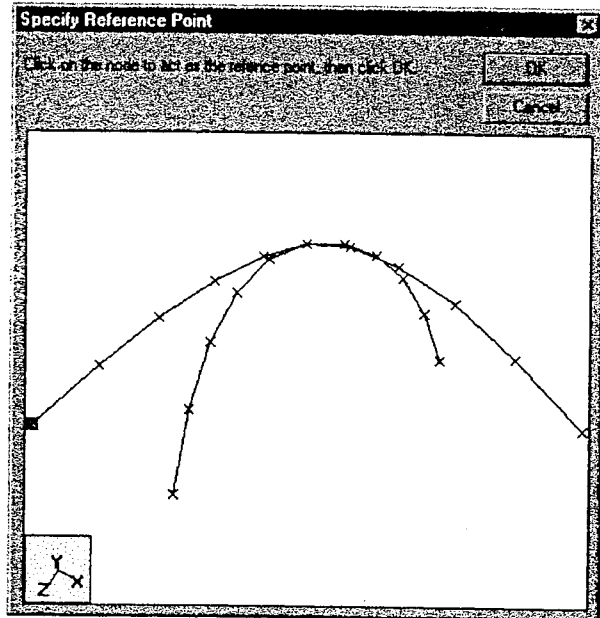
Once again make sure all the elements are selected, but this time select **Copy** from the 'Edit' menu. Once again go to the 'Edit' menu, this time select **Paste Elements**. This presents a dialogue box which can define an offset from the original location, or graphically using a reference point.

Press the **Reference Point** button and an image of the structure to be pasted in is displayed with a highlighted node which represents the insertion point. Select either one of the ends of the frame and press **OK**.

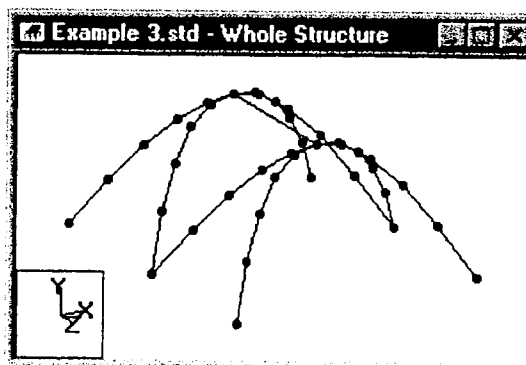
The location point on the current structure needs to be identified, so select the base of the other arch.

This calculates the offset to be 6m in the Z direction. Click on the OK button to accept the offset.

There is a warning regarding the section properties as none have yet been assigned, this can be ignored.



Finally, use the **Add Beam** command to draw a beam from the top of the first pair of arches, to the top of the second pair of arches.





STAAD.Pro:- STAAD Training

5) User Tables

In addition to the standard databases provided within STAAD.Pro, it is possible to create specialised sections using User Tables which are stored either 'Internally', within the file, or 'Externally', as a separate file in the same folder as the structure file.

Before creating a User Table, the current input units must be changed to centimetres. Do this by clicking on the units by clicking on the **Input Units** button.

Set the length units to **centimetres** and click on **OK**.

Select **Create User Table...** from the Tools menu. Select a **Wide Flange** as the section type. In the resulting dialog box, click on the **Add New Property** button.

In the following dialog box, enter the dimensional values as shown:-

Name: 127x76x13UB

$A_x = 16.5$ - no need to input

$D = 12.7$

$T_w = 0.4$

$W_f = 7.6$

$T_f = 0.76$

FW

Property	Value	Unit
Section Name	127x76x13UB	
Gross Section Area	16.024	cm²
Moment about X-axis	458.86	cm⁴
Moment about Y-axis	55.6632	cm⁴
Torsional Constant	2.46265	cm⁴
Shear Area	5.08	cm²
Shear Area	7.70133	cm²

Click on the **Calculate** button for STAAD.Pro to calculate the sectional properties based on the supplied dimensions. Note that any of these values can be modified for example to reduce the cross sectional area. Click on **OK** to complete the definition of the section. Click the **Close** button to complete the User table.

To assign this section to the structure, go to the General page and in the Property sub-page click on the **User Table...** button on the Property dialog box, select the property and click on the **Add** button. Notice that as with Standard Databases, a material can be associated with this section. Finally to assign the section to the structure, click on the property, select the Assign to View option and click on the **Assign** button.

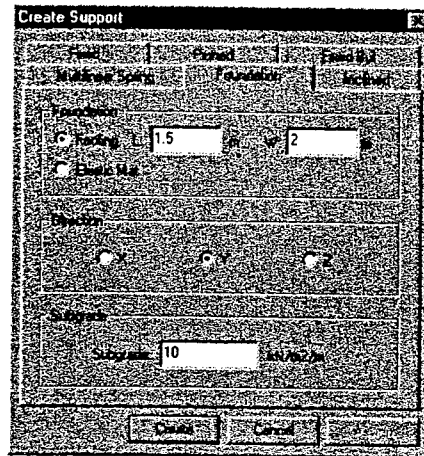
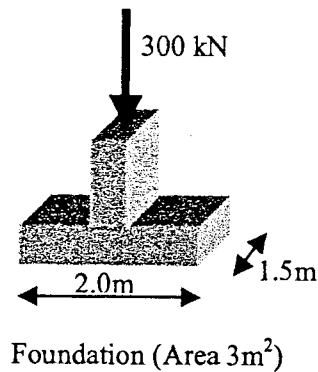
6) Add 1.5m x 2.0m Foundation Supports

Go to the **Supports** sub-page and change to the **Foundation** tab. There are two types of foundation support that can be created. **Footing**, which allows pad foundations of a specified size to be created and an **Elastic Mat** option where STAAD will establish a zone around depending on the spacing and location of surrounding nodes. Both commands create a spring support depending on the sub-grade stiffness.



STAAD.Pro:- STAAD Training

A sub-grade stiffness $10 \text{ kN/m}^2/\text{m}$ means that a load of 10 kN over a plan area of 1 m^2 will settle 1 m such as in the following example.



Note that this is analytically identical to creating the command:-

FIXED BUT KFY 10

Enter the values shown in the dialog box and click on the **Create** button. Assign the support to the base of all the arches.

Add

7) Multiple Loadcase and Combinations

The first two Examples used structures that only had one loadcase each. STAAD.Pro can analyse structures with many individual loadcases and combine them together with various multiplication factors.

First the *basic loadcases* must be defined.

Go to the Loads sub-page, create loadcase **1** and name it **Dead Loads**.

Click on the **Selfweight...** button and create the selfweight loadcase with a factor of **-1** in the **Y** direction.

Add a UDL of 1 kN/m on all the beams by clicking on the **Member...** button. In the Uniform Force tab, enter the value of **-1** for **W1** and click on the **Add** button. Select the Assign to View method and click on the **Assign** button.

To create the second loadcase, select the menu item **Commands|Loading|Create New Primary Load...**, select the option to Create a New Primary Load Case and name it **Live Load**.

Click on the **Member...** button. On the Uniform Force tab enter the values **-15** for **W1**, **1** for **d1** and **2** for **d2**. Click on the **Add** button and **Assign** it to the beam that spans between the two pairs of trusses.



STAAD.Pro:- STAAD Training

Create the third loadcase as above by selecting the menu item **Commands|Loading|Create New Primary Load...** , select the option to Create a New Primary Load Case and name it **Wind Load**.

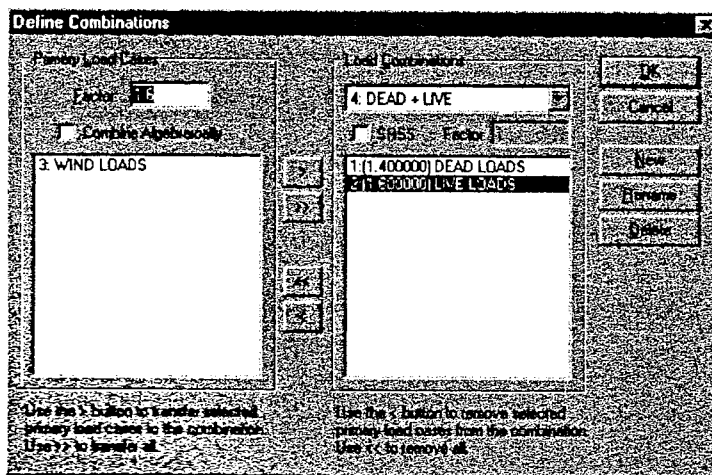
Click on the **Nodal...** button. On the Node tab enter the value 5 kN in the FX field and click on Add. Select the two joints (not the beam itself) at the ends of the beam that spans between the pairs of trusses and **Assign** it to those joints.

Now these loadcases can be combined into *combinations*.

Select the menu item **Commands|Loading|Load Combinations...** In the dialog box click on the **New** button, type the title of the combination as **Dead + Live** and click **OK** to create the combination.

To include the dead loadcase in this combination, set a factor of **1.4**, select **Dead** from the available list and click on the **>** button.

Repeat this with the **Live** loadcase with a factor of **1.6**.



Create a second combination by clicking on the **New** button and enter a title of **Dead + Live + Wind**. This time set the load factor to **1.2** and click on the **>>** button to include all the loadcases in the combination.

As before the structure needs the command to instruct the analysis to proceed. Select the menu item **Commands|Analysis|Perform Analysis...** and click on the **OK** button to accept the command with the 'No Print' option.

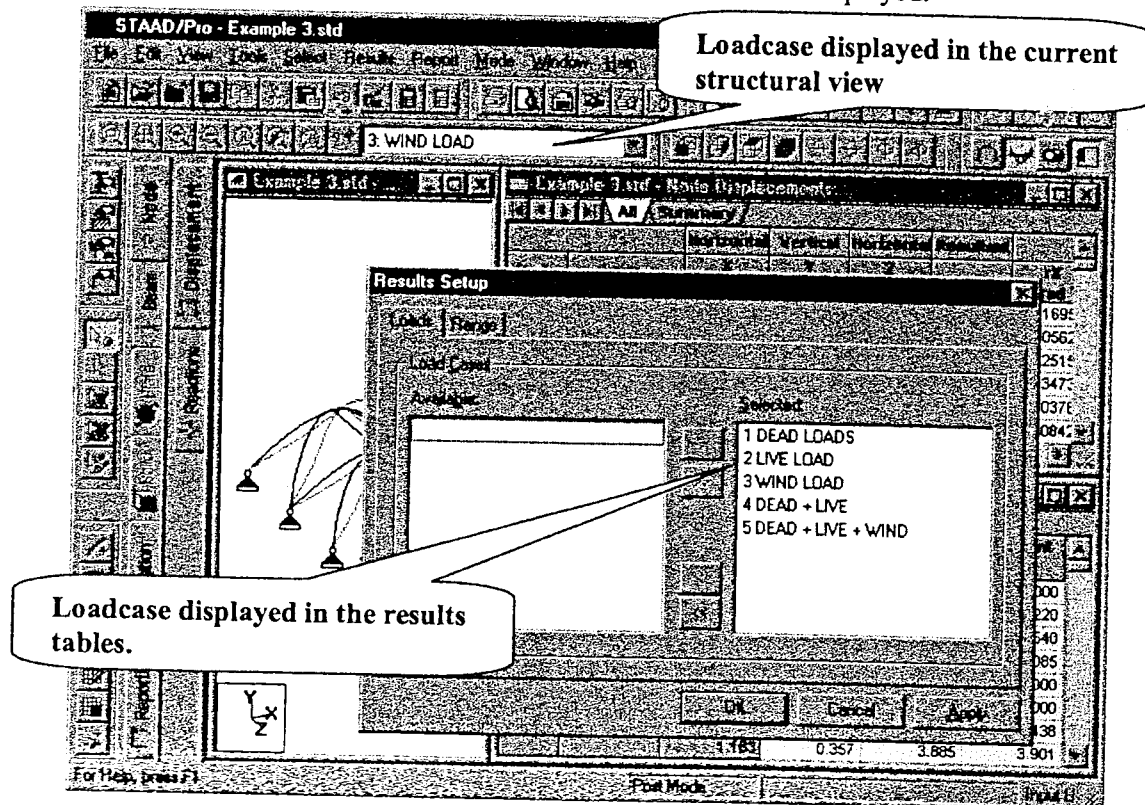
Run the analysis to produce a set of analysis results.



STAAD.Pro:- STAAD Training

8) Analysis Results

Now that the structure has more than one loadcase, it is important to recognise the fundamental difference between graphical views. That is that whilst the graphical views can display the results of the analysis, they can only display the results of one loadcase at a time. Results tables on the other hand can display the results of many loadcases and combinations simultaneously and establish which are the maximum and minimum values. Therefore, when entering into the Post Processing mode the loadcases which will be displayed.



Click on the Post Processing mode and select all the loadcases in the Results Setup dialog as above and click OK.

To change the loadcases that are displayed in the results tables either right click over the table and select **Results Setup** from the pop-up menu or it can be selected from the main menu **Results|Results Setup**.

This concludes the exercise Example 3.



9. Example 4 – Structure Wizard and Dynamic Analysis

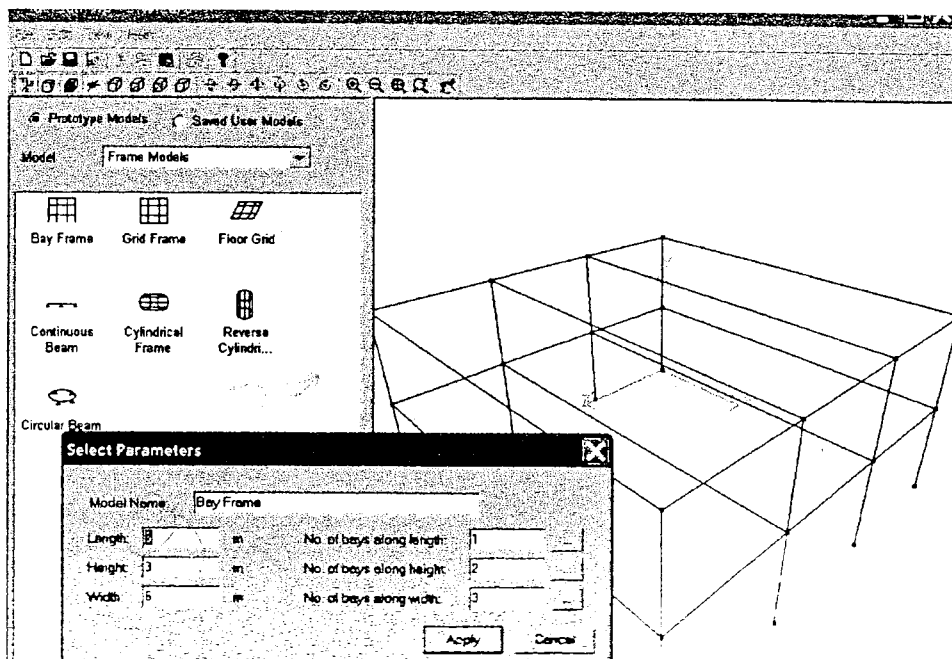
This example looks at other features within STAAD.Pro.

1) Structure Wizard

Yet another form of generating structural geometry is the Structure Wizard. Start a new structure, click on the Geometry menu and select the **Run Structure Wizard** option. This launches the Structure Wizard.

Select Frame models and double click the **Bay Frame** and set the parameters as shown and click **Apply**.

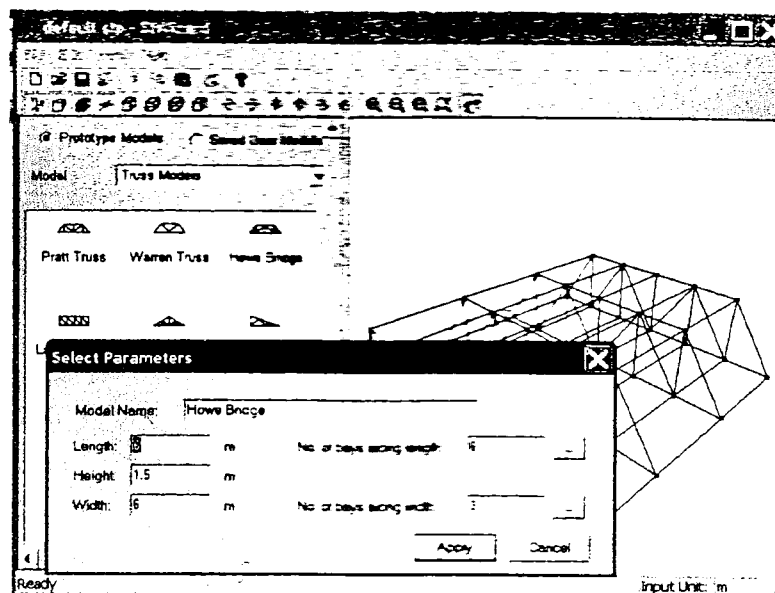
To transfer the model into the STAAD.Pro structure, click on the **Transfer Model** button. Locate the model without a shift from the origin.



paste
ilove

We wish to replace the four top beams with trusses. Highlight them in the structure view and click on the **Del** key.

Once again Start the Structure Wizard. This time select a and generate a **Howe Bridge Frame** with the dimensions as shown. Transfer it to STAAD.Pro as before, but this time locate this model 3m above the origin, i.e at the top of the columns. Note that where beams would be duplicated, they have not been created.





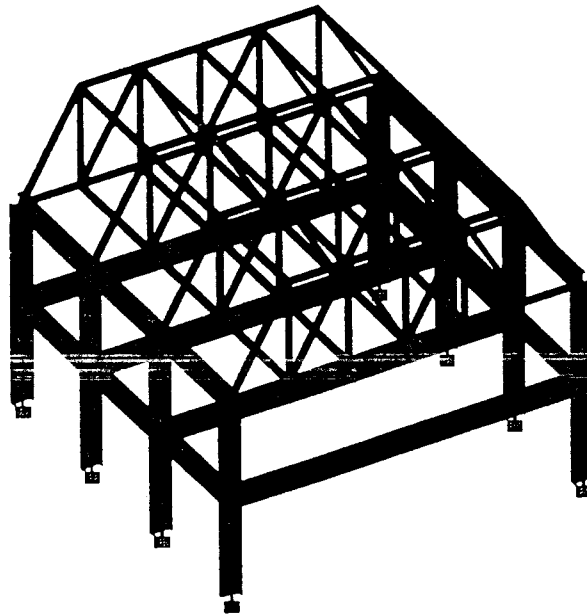
STAAD.Pro:- STAAD Training

2) Add Properties and Supports

Add the section properties as in Example 1 and 2. Go to the **General** page (**Property** sub-page), add the British section **UB305x102x33** and assign it to the columns, first floor and eaves beams. Then add the British section **UA100x75x8** and assign it to all the remaining roof beams.

Go to the **Supports** sub-page and add a **PINNED** support. Assign the support to the base of all the columns.

Right click on the Whole Structure window and select the **Structure Diagrams...** option from the pop up menu. On 3D sections, set the option to Full Sections. The diagram should look thus:-





10. Example 5 – Multiple Analysis

During a single analysis run, STAAD.Pro can modify the structure (therefore the stiffness matrix) and perform multiple analysis calculations. This is needed in situations such as when beams are defined as tension members and thus each loadcase needs to be analysed separately as beams may be removed under some load conditions, but not others. In such situations additional commands are required:-

SET NL Required to let STAAD know how much space will be required for analysis.
CHANGE To reset the stiffness matrix for the next analysis.

Use the Example 3 model which has 2 loadcases and 2 combinations already defined.
Open the file and SaveAs... Example 5.

1) Defining Beams as Tension Only

Go to the **General** page and the **Spec** sub-page.

With the beams cursor, select the cross beam that joins the pairs of arches together. Click on the **Beam...** button and select the **Tension** tab. Click on the **Assign** button to define the selected beam as a tension only beam.

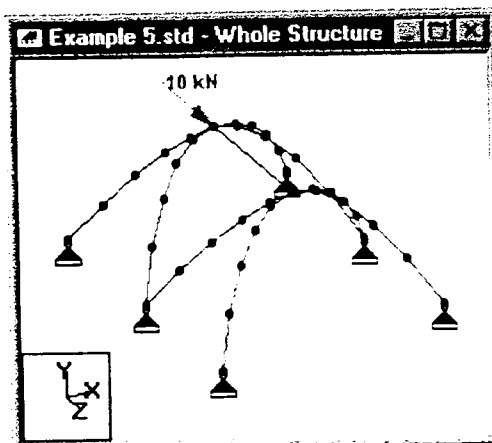
2) Modify the Loading

Recall that this structure currently has 3 basic loadcases (numbers 1, 2 and 3) and 2 combination loadcases (numbers 4 and 5). Combination numbers must follow all basic loadcase numbers. To add an additional loadcase, the current combination numbers need to be revised.

Firstly go to the **Load** sub-page and display the 1st loadcase. Select the menu item **Commands|Loading|Load Combinations...** and display load combination 4. Remove the combination by clicking on the **Delete** button. Click OK to close the Combinations dialog.

To add a new basic loadcase, click on the menu item **Commands|Loading|Create New Primary Load...**, select the **Create** option set the loadcase number to **4** and name it **Side Wind** and click on the **OK** button.

Create a **10 kN** Nodal load and assign it to **node 6** so the structure should look like this:-





STAAD.Pro:- STAAD Training

3) Add Multiple Perform Analysis Commands

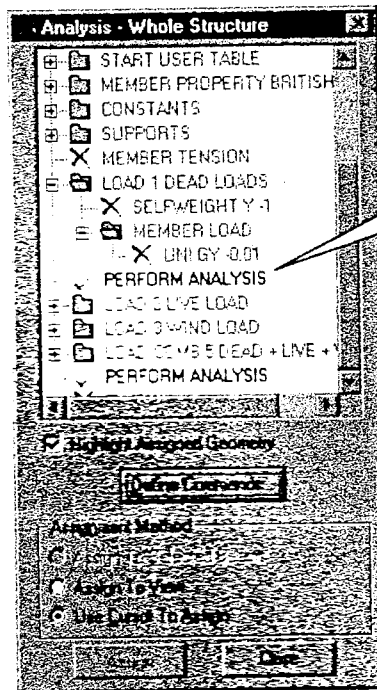
As discussed at the start of this example, multiple analysis commands have to be added and the number of loadcases.

Firstly, set the number of loadcases, select the menu item **Commands|Miscellaneous|Set NL...**, set the number to **4** and click **OK**.

Next, the commands to perform each analysis need to be added. To do this, go to the **Analysis/Print** page and the **Analysis** sub-page.

On the Analysis Tree control window, click on **Load 1 Dead Loads** to make it the current selection and click on the **Define Commands...** button.

Select the option to add the command **After Current** so that it is shown with a tick. Then, whilst on the Perform Analysis tab, click on the **Add** button.



Now select the command **PERFORM ANALYSIS** command in the tree control window and again click on the **Define Commands...** button. Select the **Change** tab, again ensure that the **After Current** option is set and click on the **Add** button.

Repeat the above two steps of adding the **PERFORM ANALYSIS** and **CHANGE** commands to follow loadcases 2 and 3. Then after the last **PERFORM ANALYSIS** that was defined from Example 3, add a **CHANGE** command.

Finally run the analysis, change to Post Processing Mode. Select the results of Loadcase 4 only. Change to the Beam page and note the forces on the beam between the frames that was defined as **TENSION** only is all zero as the applied load has forced it into compression.

This concludes the exercise Example 5.



11. Example 6 – RC Beam, Column and Slab Design

There are two methods to design beams, columns and slabs in STAAD.Pro. The original STAAD method involves including a number of parameters in the input file after the PERFORM ANALYSIS commands so that when the analysis is run a design is also performed. This gives a fixed set of design results and requires the whole analysis to be re-performed if there are tweaks required in the design. The second newer and more favoured method utilises the analysis results in the graphical interface allowing greater interactivity with the design and is described in the following example.

Design members are formed from the analytical beam elements and can be used to design beam and / or columns for a range of design codes. Also shell elements can be designed as slabs.

Note that slab designs are performed using the finite elements. The local MX and MY moments are resolved into the global axes and the reinforcement requirement calculated on those forces alone. Note that there is no check for shear. Also no account is taken for the torsional MXY moments.

1) Create Beam and Columns

Start a new structure based on kNs and metres.

Use the **Structure Wizard** and create the 3D frame in Example 4 (without the roof trusses).

2) Create First Floor Slab

Return to the Structure Wizard and create a Surface, Quadrilateral with the following parameters:-

	X	Y	Z
A	0	0	0
B	6	0	0
C	6	0	6
D	0	0	6

	Bias	Divn.
AB	1	6
BC	1	6
CD	1	6
DA	1	6

Click on the Transfer Model icon to incorporate the slab into the model. To raise the slab to first floor level, enter a value of 1.5m when prompted in the Paste Prototype Model dialog box. You can alternatively do the same using the mesh generator cursor available in the select menu and by clicking on the corner points for the mesh in the GUI itself.

3) Define Section Properties and Supports

Change to the **General** page and in the **Property** sub-page click on the **Define...** button to create the cross sections.

For the columns, create a **circle** with a diameter (YD) of 0.35m. Note that the material property associated with this section is concrete as defined in the INI file and displayed in the materials table. Assign the section to all the columns.

For the beams, create a **rectangle** with a depth (YD) of 0.40m and width (ZD) of 0.25m.

For the slab, define a plate property **thickness** of 0.175m made of concrete.

Go to the **Support** sub-page, create a **fixed** support and assign it to the base of all the columns.



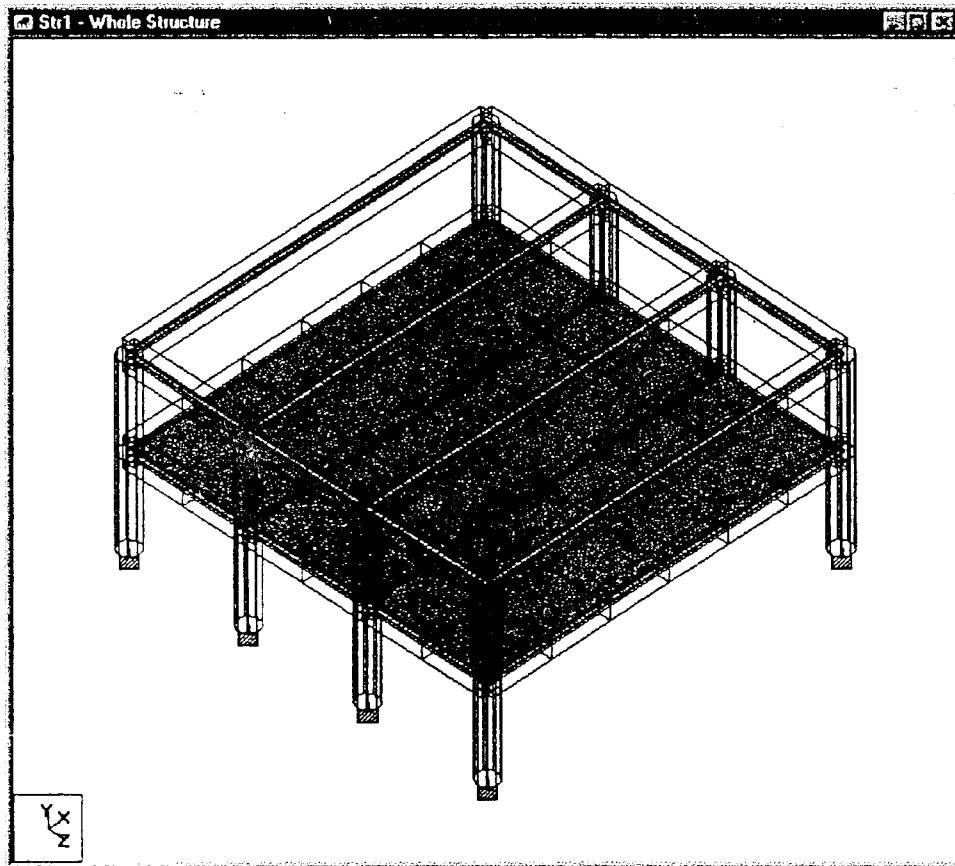
STAAD.Pro:- STAAD Training

In the Structure Diagrams dialog box,

- i) Set 3D sections to **Full Sections**.
- ii) Set View Fill Plates / Solids.
- iii) Set sort geometry and,
- iv) Set sort Nodes.

Section outline

the structure should appear thus:-



4) Floor Loading

Go to the **Load** sub-page and create the first loadcase naming it **Dead**.

Click on the **Selfweight...** button and create a selfweight load in Y with a factor of -1.

Click on the **Member...** button and select the 'Floor with Y range' tab. Set the load to **-3.5 kN/m²**, the Y range maximum to **3 m** (ie both floors), X range maximum to **6m** and Z range maximum also to **6m**. Click on the **Add** button to apply.

Note that this command is similar to the Selfweight command in that it contains all the information needed in itself and does not need to be assigned to the structure. Also this command only applies loads to the beams, not the plates.

Add an area load of **10 kN/m²** to the plates. Click on the **Plate...** button and on the 'Pressure' tab, set a load of **10 kN/m²** in the Local z direction. Note that the local z axis on these plates points downwards, hence the load is on the top acting down.



STAAD.Pro:- STAAD Training

From the menu item **Commands|Loading|Create New Primary Load...**, create a second loadcase and name it **Live 1**.

Click on the **Member...** button and select the 'Floor with Y range' tab. Set the load to **-15.0 kN/m²**, the Y range maximum to **3 m** (ie both floors), X range maximum to **6m** and Z range maximum also to **2m**. Click on the **Add** button to apply.

From the menu item **Commands|Loading|Create New Primary Load...**, create a third loadcase and name it **Live 2**.

Click on the **Member...** button and select the 'Floor with Y range' tab. Set the load to **-20.0 kN/m²**, the Y range maximum to **3 m** (ie both floors), X range maximum to **6m** and Z range minimum to **2m** and maximum to **4m**. Click on the **Add** button to apply.

From the menu item **Commands|Loading|Create New Primary Load...**, create a fourth loadcase and name it **Additional Live**.

Click on the **Nodal...** button and Add an **FY** joint load of **-150 kN**. Assign this load to the first floor level of the columns on the left of the structure after clicking on the **Isometric View** button.

Create a combination from the menu item **Commands|Loading|Load Combination...**, click on the **New** button and name the combination **ULS**. Set the load factors at **1.4** for the Dead loadcase and **1.6** for all the others.

Finally add the **Perform Analysis** command as before and run the analysis.

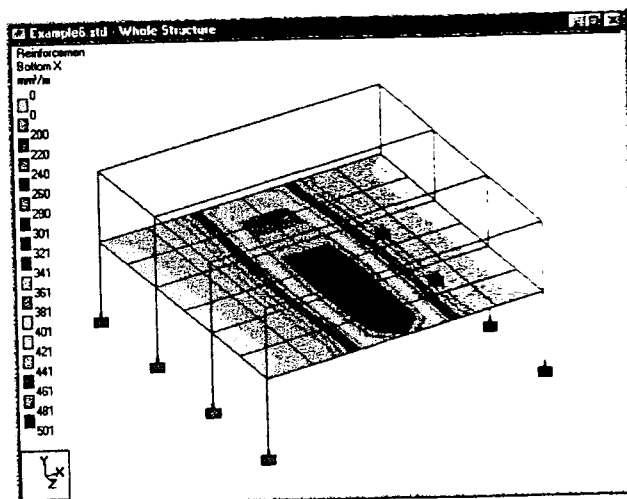
5) Slab Design

Change to the Post Processing mode.

Make the graphical view full screen. Select the menu option **Results|Slab Reinforcement...**

The dialog allows slab reinforcement design parameters to be set.

First set the loadcases that are to be designed by changing to the 'Loading' tab and selecting the loadcase '5 ULS'. Select the view of 'Bottom' reinforcement with an 'Enhanced' contour to get the following :-





STAAD.Pro:- STAAD Training

6) STAAD.Pro RC Design (Beam and Column design)

To start the RC Designer, select the menu option **File|Run StaadPro RC Design**. This starts another application and loads it with the current structure and analysis results.

There are 5 steps to then perform a concrete design.

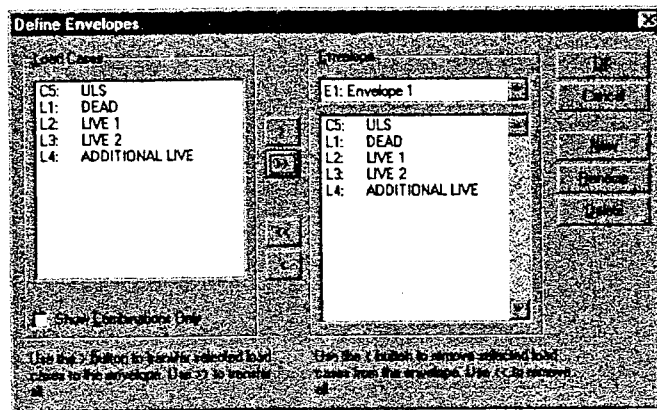
1. Create envelope

Beams are designed using envelopes of the analysis forces. Click on the **Envelopes** page. On the Envelopes table click on the **New Env...** button to open the Envelope dialog box.

Type a name for the envelope and click **OK** to open the following dialog box:-

To view all the loadcases as well as combinations, click on the **Show Combinations Only** option so that it is not selected.

Include all the loadcases in the envelope by clicking on the **>>** button and click **OK**.

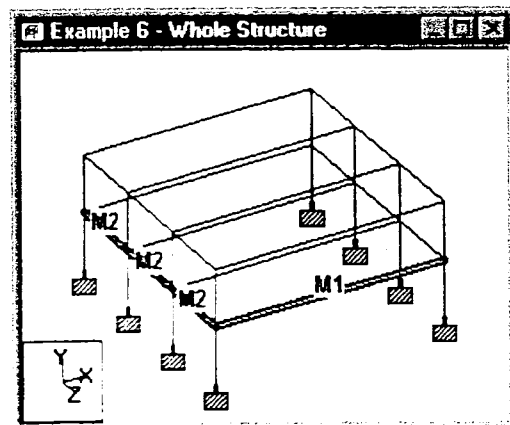


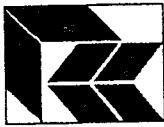
2. Create members

Design is performed on **Members**, formed from one or more beam elements that can be joined together. There are two methods of forming Members, **Form Member** or **AutoForm Member**. The first attempts to join all the selected elements into a single beam, the second uses internal algorithms to identify continuous beams and separates others beam elements into other members.


First select the nearest long span beam at first floor and click on the **Form Member** button.

Then select the elements that span along the short side at first floor level and click on the **Form Member** button. This should produce the following two members:-

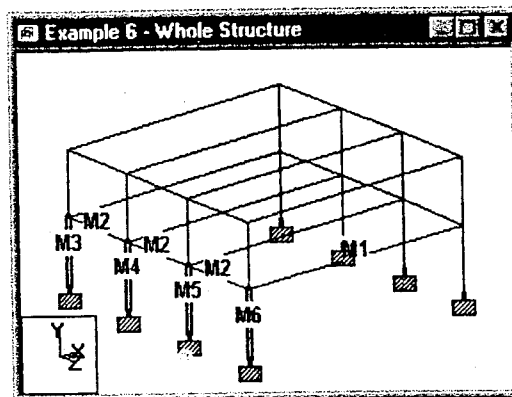




STAAD.Pro:- STAAD Training

Then select all the beam elements that form the columns under member M2 and click on the **AutoForm Member**  button.

This will create 4 new members, M3 to M6, thus:-



3. Create Design Briefs

The design is then carried out to a set of parameters that are defined in a **Design Brief**. A Design Brief is then used by a group of members known as a **Design Group**.

Go to the **Groups/Briefs** page. Check the Design Briefs table and ensure that the code is set to **BS8110 Beam**. Select it from the pulldown list if it is not currently selected. Click on the text '*Design Brief 1*' and rename it by entering the text '*BS8110 beam brief*'. Click on the **Edit Brief** button to view the parameters. Change the concrete grade to **C30** and click **OK** to accept this change and the rest of the default parameters.

Create a second design brief for designing columns. Click on the **New Brief** button, select **BS8110 Column** and name it '*BS8110 Column Brief*'. Click on the **Edit Brief** button and from the Member Combs tab, select the combination **ULS**. Click **OK** to accept this change and the rest of the default parameters.

4. Create Design Groups

A Design Group, as mentioned above, are members selected from the graphical window and associating them with a Design Brief. Note that a member can exist in more than one design group.

Select the beam members M1 and M2 and click on the **New Design Grp** button. In the dialog box, name the design group '*Beams*', ensure that the BS8110 Beam brief defined above has been selected and click **OK**.

Select the column members M3, M4, M5 and M6 and again click on the **New Design Grp** button. Name the group '*Columns*' and select the second design brief BS8110 Column Brief.

5. Design the Group

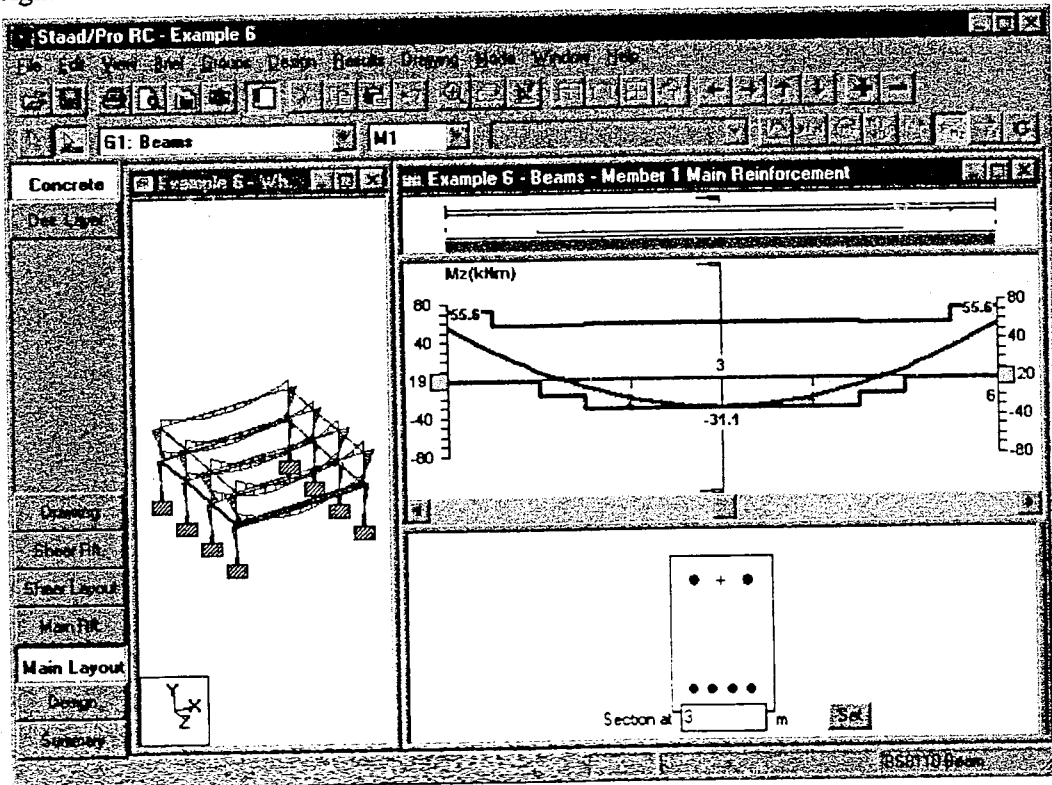
All that remains is to perform the design. Firstly, designing the beams.

From the pulldown list of Design Groups select G1:Beams. Change to the **Concrete Design Mode** by selecting it from the Mode menu or from the mode button on the page control. As with STAAD.Pro - STAAD, changing mode changes the pages on the page control. The pages and the layout of the pages is determined by the design code associated with the current design group. With this BS8110 Beam design there are seven pages and is entered on the **Summary** page. To design the group, click on the **Design** page which shows the Design Options dialog box. We wish to design both members of the design group, therefore click on the **Design** button.



STAAD.Pro:- STAAD Training

Both members are shown in green to indicate that they were designed successfully, as is confirmed in the Summary table. Click on the **Main Layout** page to view the result of the design.



The Main Reinforcement diagram shows the layout of the reinforcement above a capacity graph which is in turn above a cross section of the beam. To move the location of the cross section, either type in a distance in the section edit box and click on the set button or click and drag the cross section arrows. Also note that proportions of the windows can be resized by clicking and dragging the horizontal separation bars.

In the whole structure window, click on the member m2 to view the designed layout on that member. Alternatively select M2 from the pulldown list in the toolbar.

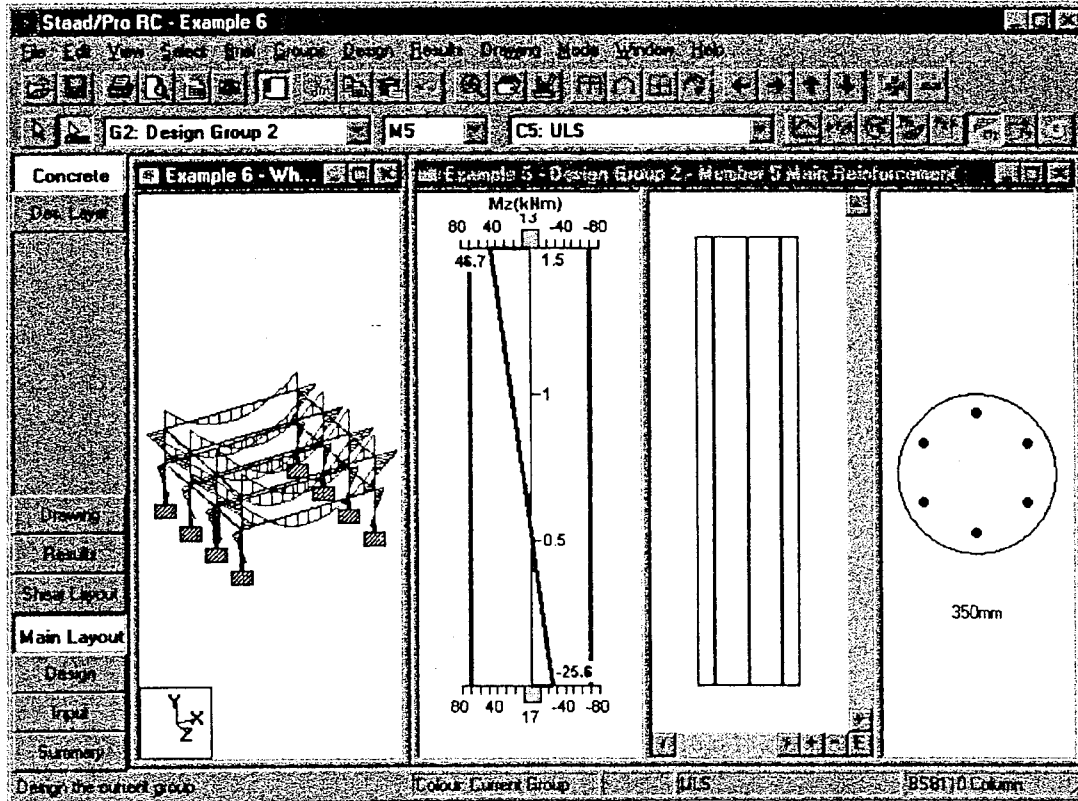
To view the details of the design, select the menu option **File|BS8110 Report setup...** Remove all the selected items from the report, add the item **Design Detail** and click OK. To view the report, click on the **Print Preview** button.

Next to design the columns, select the design group **G2: Columns** from the pulldown list in the toolbar.

Perform the design as for the beams by clicking on the Design page and click Design on the Design Options dialog box. Change to the Main Layout page to view the resulting reinforcement layout.



STAAD.Pro:- STAAD Training



This concludes the exercise Example 6



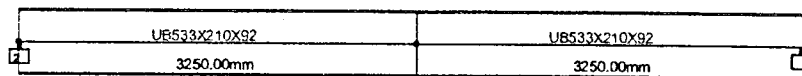
12. Detailed Steel Design Example to BS 5950: Part 1:1990

There are a number of example calculations produced by the Steel Construction Institute in their publication Steelwork Design Guide to BS 5950: Part 1:1990 Volume 2 Worked Examples (Revised Edition). The following examples demonstrate how those aspects which are within the scope of the STAAD.Pro can be implemented.

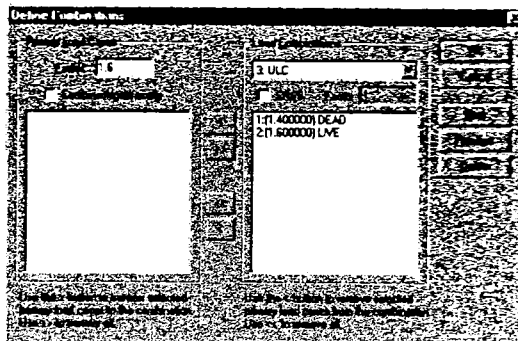
Example 1, Simply supported beam with full lateral restraint.

The first stage is to define the model (steps 1-8). Then check the section for forces (steps 9-14). Finally check for deflection (steps 15 to 21).

This is the model:-



1. Start a new 'Space' file with the default SI units of 'Metres' and 'KiloNewtons'.
2. Change to the **Geometry Page, Beam** sub page, change the grid so that the X spacing is 3.250m. Generate the two segments of the 6.5m beam.
3. Change to the **General Page, Property** sub page, select a 533x210x92 UB section and assign it to both beam segments.
4. Change to the **Supports** sub page. The supports need to be pinned about the global Z axis (not fully pinned as this will cause an instability about the global (and local) X axis. Therefore create a Fixed But support where it is released about the MZ axis. Assign this support to either end of the model.
5. Change to the **Loads** sub page and create the first loadcase, 'Dead'. Create the loads UDL -15 kN/m assigned to both beams and the point load -40 kN and assign it to the central node.
6. Create the second loadcase, 'Live'. Assign the UDL loads of -30 kN/m to both the beam segments and the point load of -50 kN to the central node.
7. Create a load combination by clicking on the 'Combine...' button. On the Combination dialog, click on the New button and name the combination 'ULS'. Include loadcase 1, 'Dead' with a factor of 1.4 and loadcase 2 'Live' with a factor 1.6.



8. Change to the **Analysis/Print** page, **Analysis** sub page. Add the command `PERFORM ANALYSIS PRINT STATICS CHECK` and run the analysis.

Check your analysis results and confirm that your statics check in the output file indicates that



STAAD.Pro:- STAAD Training

- I) SUM-Y for loadcase 1 is **137.5 kN**
- II) SUM-Y for loadcase 2 is **245.0 kN**.

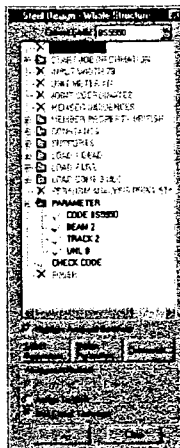
Also confirm by whichever method you prefer that the maximum moment in the combination is **585 kNm**.

Now to check the section to BS 5950. The first check will be for the forces, the second will be for deflection. The section is defined as having it's compression flange as fully restrained. This means that for the lateral torsional buckling check, the effective length is zero.

9. In the **Modelling Mode**, change to the **Design** page, **Steel** sub-page. On the dialog box, change the code to **BS5950**. Click on the **Select Parameters**, remove all the parameters from the selected list except for **BEAM**, **TRACK** and **UNL**.
10. Create a **BEAM** parameter of 2.0 and assign it to both beams. The **BEAM** parameter is used to define the locations where forces should be checked and calculations should be performed on a beam. The default value is 0.0 which means that the beams are designed using the forces at either end and the maximum result is reported. If there are forces applied along the length of a beam, then it is advised that a **BEAM 1.0**, **2.0** or **3.0** is used.
11. Create a **TRACK 2.0** command and assign it to both beams. This will generate a more detailed report in the output file which can be checked.
12. Create a **UNL 0.0** command and assign it to both beams. As stated above, the beam is restrained against lateral torsional buckling and thus it's effective length in buckling is zero.
13. Click on the **Commands...** button and create a **Check Code** command and assign it to both beams. This is the instruction to perform the design.

The Steel Design dialog should now look similar to this:-

Note that there are only ticks in the parameter section. If there are any '?' next to the parameters, then the program is indicating that commands have been defined, but not assigned to any members.



14. Save the file and re-run the analysis. With design parameters included, it is important that when running the analysis the correct design code is selected from the analysis dialog box. Ensure that it is set to **BRITISH** and not **BRITISH(EXTENDED)** as this is for BS 8007 and BS 5400.



STAAD.Pro:- STAAD Training

Check the output file for the pages of the code check and you should have the following.
Note in particular the highlighted items:-

STAAD-III CODE CHECKING - (BSI)

PROGRAM CODE REVISION V3.3_5950_90/2
STAAD SPACE

-- PAGE NO. 5

ALL UNITS ARE - KN METRE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
1	ST UB533X210X92	PASS 0.00	BS-4.3 / 4.9 0.00	0.902 585.41	3 3.25

MATERIAL DATA

Grade of steel = 43
Modulus of elasticity = 205 kN/mm2
Design Strength (py) = 275 N/mm2 Reduced = 275N/mm2

SECTION PROPERTIES (units - cm)

Member Length = 325.00
Gross Area = 117.00 Net Area = 117.00

	z-axis	y-axis
Moment of inertia	55229.996	2389.000
Elastic modulus	2360.000	356.000
Elastic modulus	2072.031	228.285
Shear Area	58.770	53.843
Radius of gyration	21.727	4.519
Effective Length	325.000	325.000

DESIGN DATA (units - kN,m)

Section Class : PLASTIC
Squash Load : 3217.5
Axial force Squash load : 1000

	z-axis	y-axis
Slenderness ratio (KL/r)	15.1	71.9
Compression Capacity	3217.5	3217.5
Tension Capacity	3217.5	3217.5
Moment Capacity	649.1	75.3
Reduced Moment Capacity	649.1	75.3
Shear Capacity	969.7	888.4

BUCKLING CALCULATIONS (units - kN,m)

Lateral Torsional Buckling Moment MS = 649.1
co-efficients m & n : m =1.00 n =1.00, Effective Length= 0.000

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m) :

CLAUSE	RATIO	LOAD	FX	VY	VZ	MZ	MY
BS-4.8.3.2	0.814	3	0.0	292.2	0.0	585.4	0.0
BS-4.8.3.3.2	0.902	3	0.0	292.2	0.0	585.4	0.0
BS-4.2.3-(Y)	0.329	3	0.0	292.2	0.0	0.0	0.0
BS-4.3 / 4.9	0.902	3	0.0	292.2	0.0	585.4	0.0

STAAD SPACE


-- PAGE NO. 6

Torsion and deflections have not been considered in the design.

The final stage is to perform the deflection checks.



STAAD.Pro:- STAAD Training

15. In the **Modelling Mode**, change to the **Analysis** page, **Post Print** sub-page. Click on the units icon  and select '*millimetres*'. Click on the **Define Commands...** button and on the **Load List** option, move the **Live** loadcase from the left to the right selection list. Click on the **Add** button, then close the dialog box. Note that both the unit command and the load list command are now displayed in the dialog box.
16. Click on the **Design** page, **Steel** sub-page. The code will already be set to BS5950. Click on the **Select Parameters** button and select **DFF**, **DJ1**, **DJ2** and **TRACK**.
17. Select both beams and create a **TRACK 4** command. As both beams are selected the **Assign** option should be available on the **Design Parameters** dialog box. Click on this button to both create the command and assign it to those beams. Note that a new parameter section has been started in the file.
18. With the beams still selected, create and assign **DJ1** as node 1 and **DJ2** as node 3, which defines the extent of the beam which should be checked for the local deflection.
19. Create and assign the command **DFF** with a value of 360 as the problem states that the beam is assumed to carry a plaster finish.
20. Click on the **Commands...** button, create and assign **Check Code**.
21. Finally save the file and re-run the analysis. The final page of the output file should be similar to this. Again take note of the highlighted values:-

CLAUSE BS-2.5.1 CHECKS

LENGTH UNITS - MMS

MEMBER	TABLE	RESULT	ACTUAL DEFL.	DEFL. LIMIT	DEFL. LEN/ DFF	LOAD/ LOCATION
1	ST UB533X210X92	PASS	9.228	18.056	6500.000 360.000	2 3250.00
2	ST UB533X210X92	PASS	9.228	18.056	6500.000 360.000	2 0.00

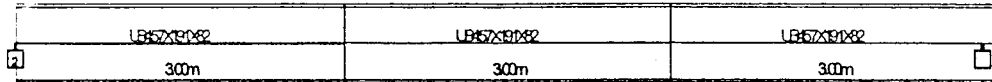
***** END OF TABULATED RESULT OF DESIGN *****

Note that there is a slight difference in the calculated deflection of 9.2mm (value in publication 8.66mm) is due to the I value differing between that used by STAAD of 55229cm⁴ and 55400cm⁴ used in the publication.



STAAD.Pro:- STAAD Training

Example 2, Simply supported beam with restraint at load points.
Create the model below using steps 1-8 in example 1.



Load Data:-

Loadcase 1:-

- UDL, -3 kN/m (GY) on all three beams.
- Point loads -40 kN (FY) on joint 2
- 20 kN (FY) on joint 3

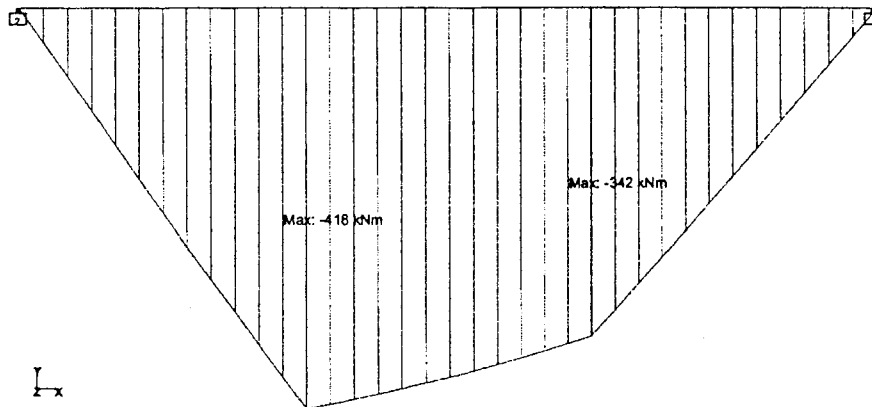
Loadcase 2:-

- Point loads -60 kN (FY) on joint 2
- 30 kN (FY) on joint 3

Load Combination 3:-

$$1.4 \times \text{Loadcase 1} + 1.6 \times \text{Loadcase 2}$$

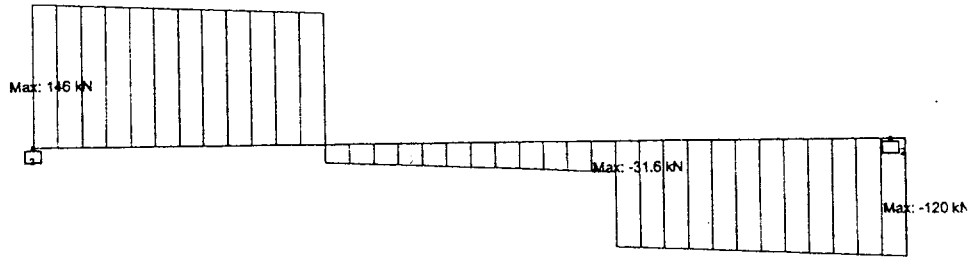
The bending moment should appear thus:-



and the shear force:-



STAAD.Pro:- STAAD Training



I_x

The structure is now ready to be designed.

There are essentially three checks that are carried out in this example,

- I) Shear load / shear capacity, (the maximum shear force is at the start of beam 1).
- II) Maximum moment / moment capacity, (the maximum moment is at the end of beam 1 or the start of beam 2).
- III) Equivalent moment / lateral torsional buckling capacity, (the critical section is between the two intermediate restraints, ie beam 2).

Therefore we need to set the following parameters:-

TRACK as 2.0 to get a detailed output

CMM to -1.0 to calculate 'm' for the major axis (otherwise it uses the default of 1.0).

CHECK CODE members 1 and 2.

Define the parameters as outlined in the previous example steps 9 to 14 and run the analysis.
The last pages of the output file should be as follows:-

```
34. PARAMETER
35. CODE BS5950
36. CMM -1 ALL
37. TRACK 2 ALL
38. CHECK CODE MEMB 1 2
```

Check your parameters to ensure the correct parameters are being used.



STAAD.Pro:- STAAD Training

STAAD-III CODE CHECKING - (BSI)

PROGRAM CODE REVISION V3.3_5950_90/2
STAAD SPACE

-- PAGE NO. 3

ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
1	ST UB457X191X82	PASS 0.00	BS-4.8.3.2 0.00	0.688 417.80	3 3.00

MATERIAL DATA

Grade of steel = 43
 Modulus of elasticity = 205 kN/mm²
 Design Strength (py) = 275 N/mm² Reduced = 275N/mm²

SECTION PROPERTIES (units - cm)

Member Length = 300.00
 Gross Area = 104.00 Net Area = 104.00

	z-axis	y-axis
Moment of inertia	: 37050.000	1871.000
Plastic modulus	: 1831.000	304.000
Elastic modulus	: 1610.869	195.609
Shear Area	: 55.093	45.540
Radius of gyration	: 18.875	4.242
Effective Length	: 300.000	300.000

DESIGN DATA (units - kN,m)

BS5950/1990
 Section Class : PLASTIC
 Squash Load : 2860.00
 Axial force/Squash load : 0.000

	z-axis	y-axis
Slenderness ratio (KL/r)	: 15.9	70.7
Compression Capacity	: 2860.0	2084.9
Tension Capacity	: 2860.0	2860.0
Moment Capacity	: 503.5	64.6
Reduced Moment Capacity	: 503.5	64.6
Shear Capacity	: 909.1	751.4

BUCKLING CALCULATIONS (units - kN,m)

Lateral Torsional Buckling Moment MB = 396.5
 co-efficients m & n : m = 0.57 n = 1.00 Effective Length = 3.000

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m :

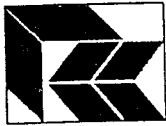
CLAUSE	RATIO	LOAD	FX	VY	VD	MZ	MY
BS-4.8.3.2	0.688	3	0.0	-133.0	0.0	417.8	0.0
BS-4.8.3.3.2	0.601	3	0.0	-133.0	0.0	417.8	0.0
BS-4.2.3 (Y)	0.194	3	0.0	145.6	0.0	0.0	0.0
BS-4.3 / 4.9	0.601	3	0.0	-133.0	0.0	417.8	0.0

Torsion and deflections have not been considered in the design.

Comparing the published results with those in the STAAD design:-

The published moment check shows the capacity as 503 kNm (STAAD design is 503.5 kNm)

The published shear check shows the capacity as 752 kN (STAAD design is 751.4 kN)



STAAD.Pro:- STAAD Training

STAAD SPACE

-- PAGE NO. 4

ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
2	ST UB457X191X82	PASS 0.00	BS-4.3 / 4.9 0.00	0.956 -417.80	3 0.00

MATERIAL DATA

Grade of steel = 43
 Modulus of elasticity = 205 kN/mm2
 Design Strength (py) = 275 N/mm2 Reduced = 275N/mm2

SECTION PROPERTIES (units - cm)

Member Length = 300.00
 Gross Area = 104.00 Net Area = 104.00

	z-axis	y-axis
Moment of inertia	: 37050.000	1871.000
Plastic modulus	: 1831.000	304.000
Elastic modulus	: 1610.869	195.609
Shear Area	: 55.093	45.540
Radius of gyration	: 18.875	4.242
Effective Length	: 300.000	300.000

DESIGN DATA (units - kN,m)

BS5950/1990
 Section Class : PLASTIC
 Squash Load : 2860.00
 Axial force/Squash load : 0.000

	z-axis	y-axis
Slenderness ratio (KL/r)	: 15.9	70.7
Compression Capacity	: 2860.0	2084.9
Tension Capacity	: 2860.0	2860.0
Moment Capacity	: 503.5	64.6
Reduced Moment Capacity	: 503.5	64.6
Shear Capacity	: 909.1	751.4

BUCKLING CALCULATIONS (units - kN,m)

Lateral Torsional Buckling Moment MB = 396.5
 co-efficients m & n : m = 0.91 n = 1.00, Effective Length= 3.000

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

CLAUSE	RATIO	LOAD	FX	VY	VZ	MZ	MY
BS-4.8.3.2	0.688	3	0.0	-19.0	0.0	-417.8	0.0
BS-4.8.3.3.2	0.956	3	0.0	-19.0	0.0	-417.8	0.0
BS-4.2.3-(Y)	0.042	3	0.0	31.6	0.0	341.8	0.0
BS-4.3 / 4.9	0.956	3	0.0	-19.0	0.0	-417.8	0.0

Torsion and deflections have not been considered in the design.

***** END OF TABULATED RESULT OF DESIGN *****

39. FINISH

Comparing the published results with those in the STAAD design:-
 The published equivalent moment factor m is 0.91 (STAAD design is 0.91)
 The published Buckling resistance moment Mb is 397 kNm (STAAD design is 396.5 kNm)



STAAD.Pro:- STAAD Training

Example 3, Simply supported beam without intermediate restraints.

As example 2, create the model below using steps 1-8 in example 1 or copy the example 2 model without the design parameters and change the section profile to:-

UB610X229X140.

As the beam is not restrained at the joint locations, the effective lengths of the beams need to be defined for the calculations of M_b in the lateral torsional buckling check. This can be done either by setting an appropriate factor on each beam using the UNL parameter, or specifying the effective length directly using the UNF parameter. In this example, the effective length of the beam is defined as $0.85 \times 9.000\text{m} = 7.65\text{m}$. This can be specified by assigning UNL 7.65 to the members or assigning the parameter UNF of $0.85 \times 3 = 2.55$. This can be awkward in situations where the beam is not made up of equal segments.

Therefore we need to set the following parameters:-

TRACK as 2.0 to get a detailed output

UNL 7.65.

CHECK CODE ALL members.

Define the parameters as outlined in the first example steps 9 to 14 and run the analysis. The last pages of the output file should be as follows:-

```
34. PARAMETER
35. CODE BS5950
36. TRACK 2 ALL
37. UNL 7.65 ALL
38. CHECK CODE ALL
```



STAAD.Pro:- STAAD Training

STAAD-III CODE CHECKING - (BSI)

PROGRAM CODE REVISION V3.3_5950_90/2
STAAD SPACE

-- PAGE NO. 3

ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
1	ST UB610X229X140	PASS 0.00	BS-4.3 / 4.9 0.00	0.923 417.80	3 3.00

MATERIAL DATA

Grade of steel = 43
Modulus of elasticity = 205 kN/mm²
Design Strength (py) = 265 N/mm² Reduced = 265N/mm²

SECTION PROPERTIES (units - cm)

Member Length = 300.00
Gross Area = 178.00 Net Area = 178.00

	z-axis	y-axis
Moment of inertia	111799.992	4505.000
Plastic modulus	4142.000	611.000
Elastic modulus	3622.812	391.399
Shear Area	91.571	80.853
Radius of gyration	25.062	5.031
Effective Length	300.000	300.000

DESIGN DATA (units - kN,m) BS5950/1990

Section Class : PLASTIC
Squash Load : 4717.00
Axial force/Squash load : 0.000

	z-axis	y-axis
Slenderness ratio (KL/r)	12.0	59.6
Compression Capacity	4717.0	3814.2
Tension Capacity	4717.0	4717.0
Moment Capacity	1097.6	124.5
Reduced Moment Capacity	1097.6	124.5
Shear Capacity	1456.0	1285.6

BUCKLING CALCULATIONS (units - kN,m)

Lateral Torsional Buckling Moment MB = 452.6
co-efficients m & n : m =1.00 n =1.00, Effective Length= 7.650

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

CLAUSE	RATIO	LOAD	FX	VY	VZ	MZ	MY
BS-4.8.3.2	0.145	3	0.0	-133.0	0.0	417.8	0.0
BS-4.8.3.3.2	0.923	3	0.0	-133.0	0.0	417.8	0.0
BS-4.2.3-(Y)	0.113	3	0.0	145.6	0.0	0.0	0.0
BS-4.3 / 4.9	0.923	3	0.0	-133.0	0.0	417.8	0.0

Torsion and deflections have not been considered in the design.

Comparing the published results with those in the STAAD design:-
The published Buckling resistance moment Mb is 450 kNm (STAAD design is 452.6 kNm)



STAAD.Pro:- STAAD Training

Example 4, Column with pinned ends and intermediate support.

As with the previous examples, create the model below using steps 1-8 in example 1.

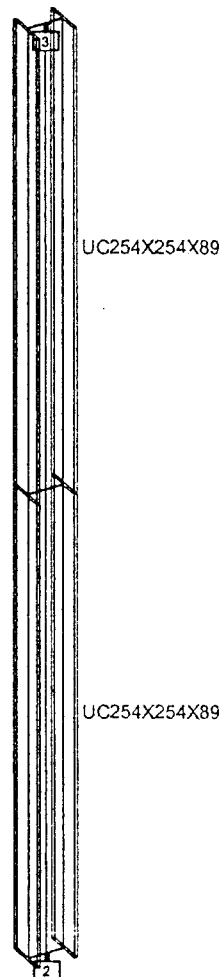
This is a 6m high column with a lateral restraint at mid-height.

Provide a support at the base which is 'Fixed But', MX and MY. At the top provide another support which is 'Fixed But' MX, MY and also FY.

Loading is applied as a single factored point load at the top of the column of -2500 kN (FY). As the load has already been factored, then no load combination is required.

Create the command PERFORM ANALYSIS as with all the other examples.

Check that there is the correct loading by changing to the **Post Processing Mode** and review the reactions. There should be none at the top and 2500 kN vertically at the base.



The structure is ready to be designed.

When beams are being designed as columns (ie compression members, clause 4.7), then the design lengths and factors that are used are defined with the parameters LY and LZ if defining the actual lengths or KY and KZ if defining a multiplication factor of the actual beam length. In this example, restraint is provided at mid height when bending about the local Y axis, but not the local Z axis. Therefore the effective length of both these beams when bending about Z axis is twice their actual length, or 6.0 m.

Therefore we need to set the following parameters to both members:-

TRACK 2.0
LZ 6.0
CHECK CODE



STAAD.Pro:- STAAD Training

The resulting design output should for either member should be:-

STAAD-III CODE CHECKING - (BSI)

PROGRAM CODE REVISION V3.3_5950_90/2
STAAD SPACE

-- PAGE NO. 3

ALL UNITS ARE - KN METE (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
* 1	ST UC254X254X89	FAIL	BS-4.7 (C)	1.003	1
		2500.00 C	0.00	0.00	0.00

MATERIAL DATA

Grade of steel = 43
 Modulus of elasticity = 205 kN/mm²
 Design Strength (py) = 265 N/mm² Reduced = 265N/mm²

SECTION PROPERTIES (units - cm)

Member Length = 300.00
 Gross Area = 113.00 Net Area = 113.00

		z-axis	y-axis
Moment of inertia	:	14269.999	4857.000
Plastic modulus	:	1224.000	575.000
Elastic modulus	:	1096.427	379.009
Shear Area	:	79.810	26.811
Radius of gyration	:	11.238	6.556
Effective Length	:	600.000	300.000

DESIGN DATA (units - kN,m)
 Section Class : BS5950/1990
 Squash Load : PLASTIC
 Axial force/Squash load : 2994.50
 : 0.835

		z-axis	y-axis
Slenderness ratio (KL/r)	:	53.4	45.8
Compression Capacity	:	2529.9	2492.2
Tension Capacity	:	2994.5	2994.5
Moment Capacity	:	324.4	120.5
Reduced Moment Capacity	:	60.3	51.9
Shear Capacity	:	1269.0	426.3

BUCKLING CALCULATIONS (units - kN,m)

Lateral Torsional Buckling Moment MB = 323.7
 co-efficients m & n : m =1.00 n =1.00, Effective Length= 3.000

CRITICAL LOADS FOR EACH CLAUSE CHECK (units- kN,m):

CLAUSE	RATIO	LOAD	FX	VY	VZ	MZ	MY
BS-4.7 (C)	1.003	1	2500.0	0.0	0.0	3.0	0.0
STAAD SPACE							

-- PAGE NO. 4

Torsion and deflections have not been considered in the design.

Comparing the published results with those in the STAAD design:-
 Published compression capacity is 2508 kN (STAAD design is 2492.2 kN)

The published design only passes as it uses the old British Steel section properties whereas STAAD uses those currently published by British Steel in particular the printed example uses $A_g = 114 \text{ cm}^2$, which is now published as 113 cm^2 .