detailing
Documentation
analysis
Design
Structures
detailing
analysis
documenta
modeling

MANUAL
STAAD.Pro v8i
2014

ALFREDO ZERTUCHE HERRERA
RAFAEL HERNANDEZ SOTELO

MANUAL DE STAAD.Pro v8i

TOMO I

ALFREDO ZERTUCHE HERRERA
RAFAEL HERNANDEZ SOTELO

ING. HECTOR ELIUD MARTINEZ ADAME

NUEVO LAREDO, TAM.   JUNIO 2014

MANUAL

DE

STAAD.Pro
V8i

MANUAL

DE

STAAD.Pro V8i

First Edition

Volume I

ALFREDO HERNANDEZ

RAFAEL HERRERA ZERTUCHE

Sotelo

ING. HECTOR MARTINEZ ELIUD DAMEA

Preface

Formerly, the analysis and design of structures has been a very extensive work that can take you hours, days, or months of calculations the most experienced designer. Such a feat has now been reduced in time, by the help of computer programs specializing in the design and structural analysis.

There are software packages that allow you to perform such work, such as STAAD.Pro, SAP 2000, ETABS, Tricalc, RISA, among others. Being the STAAD.Pro program, one of the most widely used by structural engineers worldwide.

This series of manuals for STAAD.Pro V8i program is aimed at students, professors, professionals such as civil engineers, architects, etc., who are interested in learning how to manipulate this great tool. Even if they have any history or already familiar with the program, this manual will enable them to delve into detail procedures or commands not as well known. It also serves as a complement to the structures related subjects taught in Civil Engineering.

The content of this document is based on and translated from the number four series of the English version of the "Getting Started and Technical Reference of STAAD.Pro" because it had no such detailed Spanish version of the manual. Along with this, excellent information obtained in a course taken by students in the 2010-2014 generation of Civil Engineering of the Technological Institute of Nuevo Laredo, where whose contribution was more attached to the labor application of this software approach was also collected; ie the most common problems that can occur in structural design as a profession and possible solutions thereof.

In the treatment of different instructions that appear throughout the text, we find Anglo technical terminology that immediately after; in some terms, its meaning, others said it is understood. As the software is of English origin, thus allowing a quick understanding of the procedures and / or commands used in the tutorials.

The book was written as a student work Advanced Structural Analysis subject belonging to the seventh semester of the race. It is deeply grateful to the Engineer Hector Adame Eliud Martínez, for their unconditional to pass on their knowledge, allowing us to look beyond our horizons contribution.

Content

Introduction

STAAD.Pro is a general purpose program for analysis and design of a wide variety of structures. The three basic activities that can be undertaken to achieve that goal - a) generation of the model b) calculations for verification of analytical results c) result - they are all provided by GUI tools contained in the program. This manual contains some tutorials showing a guide through the steps to create, analyze, process, and generate reports for each example.

The first of these tutorials shows these processes using a simple two-dimensional steel frame website. It is a good starting point for learning the program. If you are not familiar with STAAD.Pro, will greatly benefit through this first tutorial.

For the second tutorial, we have chosen a reinforced concrete structure. We can generate the model, the analysis and design of concrete beams and columns. It contains extensive information on the various facilities available for viewing and verification of results.

The modeling and analysis of a slab is shown in the third tutorial. Slabs and other surface entities such as walls are modeled with plate elements. Large surface entities will be defined using several elements and this sometimes requires a tool called mesh generator. This tutorial demonstrates the simple techniques as well as mesh generation method to generate the finite element model of the slab. It also explains the methods by which one can check the results for the elements of the plate.

Contains practical exercises as a warehouse consists of several frames repeated in an address, a seven-story building, a dome, in the which are summarized in a few steps all seen on the section of tutorials, plus using other commands to create other types of common load presented in a building being used in structural analysis and design as applicable, such as wind loads and loads due to earthquake. In seismic loads, the google earth program is used to obtain geographic coordinates and thus to perform the analysis of seismic spectrum given location and how to apply or generate a building.

About STAAD.Pro

STAAD.Pro is a general purpose program for structural analysis and design with applications primarily in the building industry - commercial buildings, bridges and structures, industrial structures, structures for chemical plants, dams, retaining walls, foundations of turbines, sewers and other embedded structures, etc. Therefore, the program consists of the following means for this task.

1. Utilities for generating graphical model and based text editor commands to create the mathematical model. Members of the beam and column are represented by lines. Panel type entities, slabs and walls are represented by triangular and rectangular finite elements. Blocks are represented using solid brick elements. These utilities allow you to create the geometry, assign properties, target cross sections as you wish, assign materials like steel, concrete, wood, aluminum, specify supports, apply loads explicitly and have the program generate loads, design parameters etc. ..

2. Analysis engine to perform linear elastic analysis of PDELTA, finite element analysis, extraction and dynamic response frequency (spectrum, time history, steady state, etc.).

3. Engine design for code verification and optimization of the members of steel, aluminum and wood. Calculations of reinforcement for concrete beams, columns, slabs and shear walls. Design of shear and moment connections for steel members.

4. Result visualization tools generation and verification result report to examine patterns of displacement, bending moment and shear force diagrams, beam, plate and solid efforts contours,

5. Peripheral tools for activities such as import and export data to and from other widely accepted formats, links to other popular programs for niche areas such as the design of slabs of reinforced and prestressed concrete, foundation design, steel connection design, etc. ..

6. A library of functions exposed OpenSTAAD call that allows you to access internal functions and routines STAAD.Pro and their graphics to access the database and link STAAD input and output third command software written using languages such as C, C ++, VB, VBA, FORTRAN, Java, Delphi, etc. .. So, can OpenSTAAD used to link internal or third-party applications with STAAD.Pro.

Document Conventions

Throughout this manual, the following typographic conventions are used and math. It is recommended to spend some time to familiarize yourself with these as to facilitate understanding of the content.

Notes, Tips and Warnings

Items to note are indicated as follows:

> **Note:** This is an element of general importance.

> **Tip:** This information is optional timesaving.

> **Caveat:** This is information about the actions that should not be done under normal operating conditions.

File Path / File Name.extension

A fixed-width font is used to indicate file names, file paths and file extensions (for example, C:. / SPROV8I / STAAD / STAADPRO (EXE)

Interface Control

A bold font is used to indicate the user controls. Menu items and submenus are indicated with a number of> characters to distinguish levels of menu. (For example, **File> Save As ...**)

Data Entry by User

A fixed-width font is used to indicate the information to be entered manually. (For example, type DEAD LOAD as the title for load case 1).

Page Control STAAD

A character "|" is used to represent control levels page between pages and subpages. (Eg, select the page **Design | Steel**).

Terminology

- Click - this refers to the action of pressing a mouse button to "click" a button on screen interface. When not specified, click means to press the left mouse button.

- Select - indicates that the command should be run on a menu or dialog (Click synonymous). Used when referring to an action in another control, drop-down list or list box

menu where multiple options are available to you.

- l popup menu – a popup menu typically appears with a right mouse click on an element of the interface.

- Window – describes the display element that can be manipulated independently. Multiple windows can be open simultaneously and interacted with.

- dialogue – is an element on the screen (typically) in what should be interacted before returning to the main window.

- Cursor – various selection tools are called "sliders" in STAAD.Pro. Select one of these tools will change the mouse icon to reflect the current selection mode.

Conventions Mathematics

Similar to orthographic conventions, as this program is of American origin, the American mathematical notation used throughout the documentation.

- Numbers greater than 999 are written using a comma (,) to separate every three digits. For example, US value of Young's modulus is taken as 29, 000,000 psi.

  Caveat: Do not use commas or spaces to separate digits in a number in a STAAD input file.

- Numbers with decimal fractions are written with a period to separate integer and fraction parts. For example, a beam with a length of 21.75 feet.

- Multiplication is represented by a high point, or medium, ($\cdot$). For example, $P = m \cdot A$.

- Separators are used operations in the following order:

  1. Parenthesis ()

  2. Brackets []

  3. Braces {}

For example,

$$Fa = Fy \; \frac{1 - \dfrac{2\,\dfrac{KL}{r}}{2 \cdot Cc_2}}{\dfrac{5}{3} + \dfrac{3\,\dfrac{KL}{r}}{8 \cdot Cc} - \dfrac{3\,\dfrac{KL}{r}}{8 \cdot Cc_3}}$$

Section 1

Getting Started ...

_____

1. Introduction

STAAD.Pro is a software package for analysis and design of structural engineering. This manual is designed to guide users who are new to this software, as well as experienced users seeking specific information on the basics of using the program.

Section 1 describes the following:

- system requirements

- installation and licenses

- Running STAAD.Pro

- Running Section Wizard

- Running Mesher

Section 2 contains tutorials on using STAAD.Pro. The tutorials guide the user through the process of:

- Creating a structural model. This consists of generating the structural geometry, specify a member property, material constants, loads, analysis and design specifications, etc.

- Visualization and verification of the model geometry.

- Running STAAD analysis engine for analysis and design. •

Verification results - graphically and numerically.

- Printing and reporting.

Section 3 contains some additional information about how to perform some common tasks in STAAD.Pro.

Section 4 contains some practical exercises on specific commands that are not addressed in Section 3 and / or a summary of the most commonly used commands.

2. System Requirements

The following minimum hardware requirements are suggested. Higher capacity systems provide improved performance.

- PC with Intel-Pentium or equivalent.

- graphics card and monitor with 1280 x 1024 resolution, 256-color display (16-bit color high recommended).

- System Memory: 512 MB minimum, 1 GB recommended better and 2GB.

- Windows NT 4.0 or higher operating system. (Windows 95, Windows 98 and Windows Me are not supported). The program works best on Windows 2000, XP, Vista or Windows 7 operating systems.

- enough free space on your hard disk to save the program files and data. The disk space requirement varies depending on the modules you are installing. The typical minimum is 500MB free space.

- A clever speaker and sound card multimedia system is required to run the tutorial movies and slideshows.

An additional RAM, disk space and video memory will improve the performance of STAAD.Pro. From STAAD.Pro 2001 version, the size of structures that can handle the program has increased significantly. As a result, the minimum amount of physical memory
Virtual + required by the program has also increased to more than 600 MB. You may need to ensure that an adequate amount of virtual memory available, and Windows NT and 2000 systems, parameters such as the size of the paging file should be large enough to span multiple units or other free space is exhausted in one of the disks .

Another aspect to consider is the location of the "TEMP" parameter as in the environment variable "TEMP SET" on Windows NT and 2000 systems While performing calculations, depending on the size of the structure, the program can create gigantic scratch files that are placed on the location of the associated "TEMP" folder parameter. Users may wish to point the variable "TEMP SET" to a folder on a disk that has disk space large enough to accommodate the requirements for large structures.

You should have a basic knowledge of Microsoft Windows systems to use the software.

3. Installation and Licensing

Please see the document Quick Start and Troubleshooting (Quickstart.pdf), which is included in the installation package.

Once installation is complete, select Quick Start and Troubleshooting (Quick Start & Troubleshooting Guide) on the Start menu in Windows **All Programs> Bentley Engineering> STAAD.Pro V8i**.

Figure 1-1: Quickstart

4. Starting STAAD.Pro

Select the STAAD.Pro icon from the STAAD.Pro V8i program group in the Windows Start menu. The home interface or STAAD.Pro window opens on the home screen.

Figure 1-2:

5. Running Section Wizard

Section Wizard is a set of utilities for generating custom sections for use in STAAD.Pr0.

- Select one of the program icons located in the folder assistant section (Section Wizard) in the program group STAAD.Pro V8i located in the Windows Start menu. It consists of a set of utilities for generating custom sections for use in STAAD.Pro.

Note: For help on using this program, see the help section of the wizard (Section Wizard) accessed from the same location or from within individual programs.

6. Running Mesher

- Select situated in STAAD.Pro V8i program group in the Windows Start menu Mesher icon.

Information on the use of this program is available from the Help menu of the program.

Section 2

Tutorials

Tutorial 1 - Portals Steel

This chapter provides a step by step tutorial for creating a framework of 2D portal STAAD.Pro.
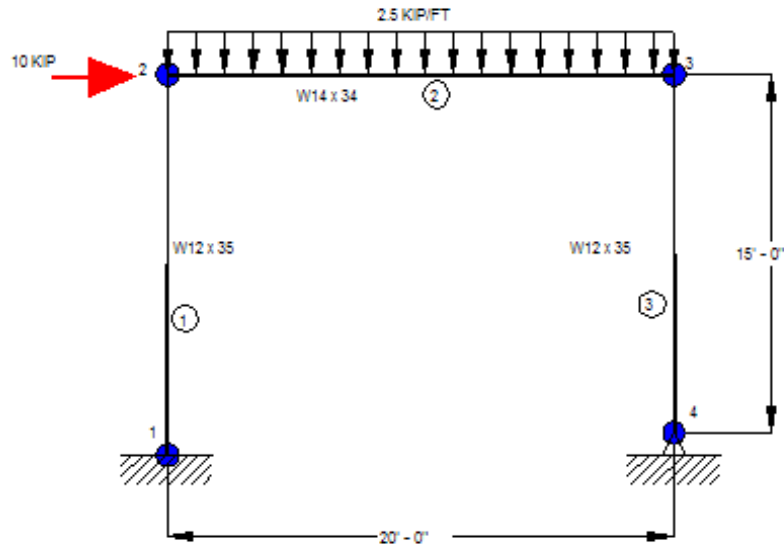
1.1 Model creation methods

There are two methods to create data structure:

1. Using the graphical model generation mode, or graphical user interface (GUI) that typically refers

2. Using the file command (Command File)

The File command is a text file that contains the data of the modeled structure. This file consists of simple English like commands. This script can be created directly using the editor built into the program, or at least, any editor that saves the data in text form (eg, Notepad, TextPad, Notepad ++, etc. ..).

This script is automatically created behind the scenes when the structure is generated using the GUI.

The graphical model generation mode and the script are integrated. So, at any time, you can temporarily exit the graph generation model and access the script. You will find that reflects all data entered through the mode graph generation model. In addition, to make changes to the command file and save it, the GUI immediately reflects the changes made to the structure through the script.

Both methods of creating our model are explained in this tutorial. Sections 1.3 to 1.6 explains the procedure to create the file using the GUI. Section 1.7 describes the creation of the script file using the text editor STAAD.Pro.

1.2 Problem description tutorial

The structure of this project is a steel frame with a clear and a floor that will be analyzed and designed with profiles according to American standards. You can style it with some other type of profile or other standard (IMCA).

The program has provided an input file called Tut-01-portal.std (C: \ SPro2007 \ STAAD \ examp \ US \ Tut__01__portal) containing the input data for the structure to be held in this tutorial.

Basic information about the structure

Attributes	Data
Properties for Members	Members 1 and 3: W12X35 Member 2: W14X34
Material Constants	Modulus of Elasticity: 29000 ksi Poisson ratio: 0.30
Offset Members	6.0 inches along global X for member 2 at both ends
Supports	Node 1: Built-Node 4: Articulated
Loads	Load case 1: Dead + Live Beam 2: 2.5 kips / ft along down load and overall Case 2: Wind left 10 kips point force at node 2 load case 3: 75 percent of (DL + LL + WL) Load Combination - L1 + L2 X 0.75 X 0.75
Type of analysis	Elastic-Linear (PERFORM)
Design of Steel	Considered load cases 1 and 3 only. Parameters: 40 ksi Length flexural compression flange without stand: 10 feet for 2 and 3 members, 15 members 1 feet. Creep of Steel: 40 ksi The selection of members for the members 2 and 3

1.3 Starting the program

1. Select the STAAD.Pro icon from the STAAD.Pro program group in the Windows Start menu.

STAAD.Pro window opens on the home screen.

Figure 2-1:

Note on the unit system:

There are two systems of base units in the program control unit (length, force, temperature, etc.) in which the values; specifically the results and other information presented in tables and reports are shown. The base system units also dictates what kind of default, the program will use when attributes such as modulus of elasticity, density, etc., based on types of materials are assigned: steel, concrete, aluminum - selected library program . These two systems are English units (foot, pound, etc.) and metric or International (KN, metro, etc.).

If you remember, one of the choices made when installing STAAD.Pro is this basic system configuration units. That choice will serve as the default until you change it specifically we value.

We can change this setting either by going to the menu **File> Configure** or by selecting Settings under project tasks. In the dialog box that appears, choose the unit system to your liking. For this tutorial, we will choose English units (Kip, feet, etc.).

Figure 2-2:
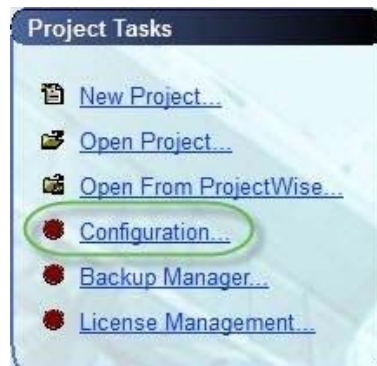
Figure 2-3:
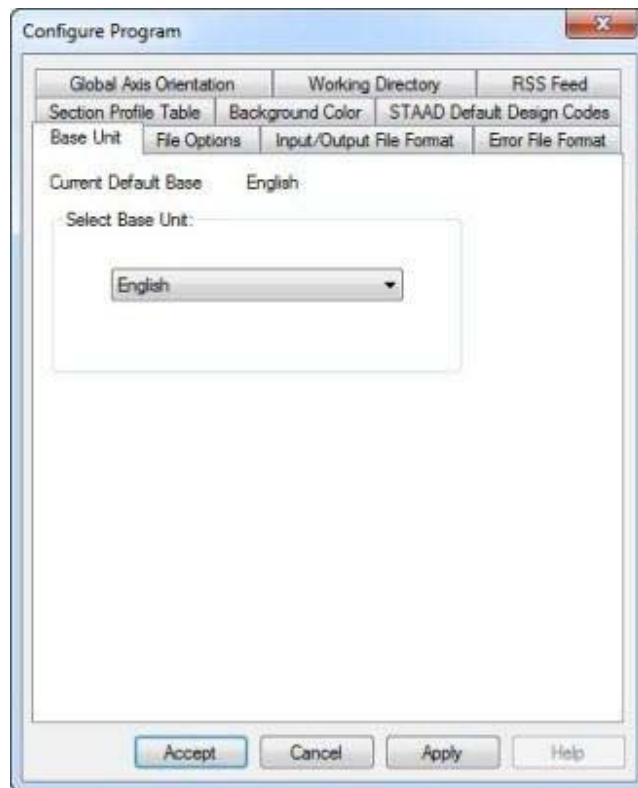
Click OK to close the dialog box.

1.4 Creating a new structure

In the New dialog box, offer some initial critical data needed to build the model.

1. Select **File> New** or select *new project* on project tasks.

Figure 2-4:
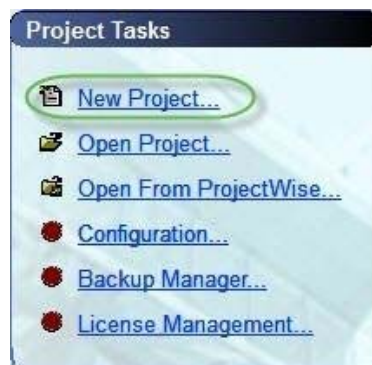
The New dialog opens.

Figure 2-5:

The structure type is defined as **Space** (**Space**), **Plane (Plane)**,**Floor (Ground)**, Or **Truss (Armor)**:

Space

The structure, charge, or both, cause the structure to deform in all three global axes (X, Y and Z).

Plane

The geometry of the load and the deformation is limited only to the overall plane XY.

Floor

A structure whose geometry is limited to the plane XZ.

Truss

The structure carries pure axial load action. The members of the truss are considered incapable of carrying loads cutting, bending and twisting.

2. Select **Plane**.

3. Select **Foot (Feet)** as unit length and **Kilo Pound (Pound Kilo)** as a unit of force.

Tip: The units can be changed later if necessary, at any stage of the creation model.

4. Specify the file name as PORTAL and specify a location where the STAAD input file will be located on your computer or network.

   Directly you can type a file path or click [...] to open the search dialog by folder, which is used to select a location using a Windows file tree. After specifying the previous entry, click Next. The next wizard page, Where do you want to go? (Where do you want it to go?) Opens.

Figure 2-6:
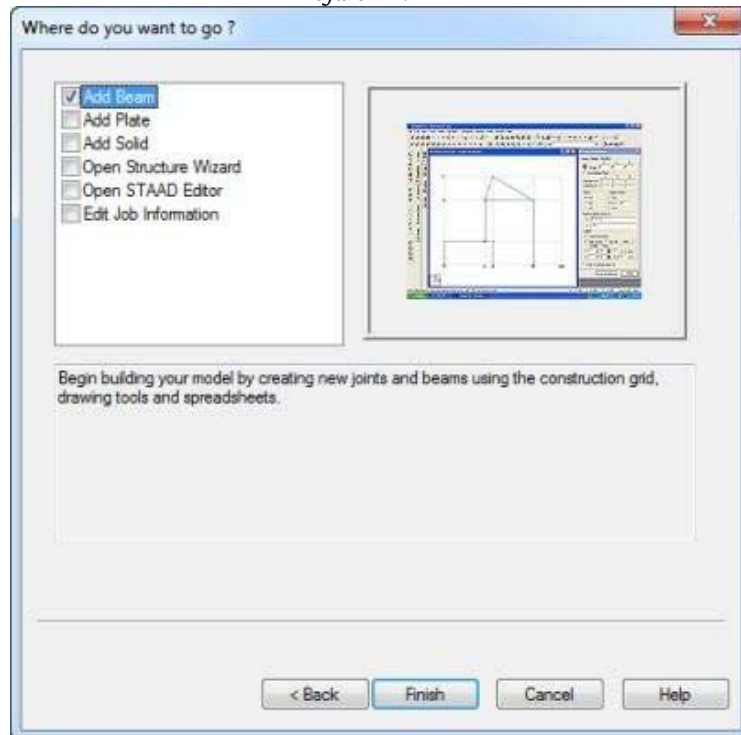
In the dialog, we choose the tools that will be used to initially build the model.

Add Beam (Beam Add), Add Plate (Add plate) or Add Solid (Add Solid)

Respectively, the choices you make tools are used in the construction of beams, plates or solids when the GUI (Graphical User Interface) is opened.

Open Structure Wizard (Wizard Open structure)

Provides access to a library of structural templates that comes equipped with the program. Those template models can be extracted and modified parametrically to reach the model geometry or some of its parts.

Open STAAD Editor

Used to create a model using STAAD command language in the STAAD editor. All these options are also available from the menus and dialog interface, even after this rule box

dialog.

> **Note**: To use the Editor to create the model, choose Open STAAD Editor, click Finish and proceed to section 1.7.

5. Select the Add Beam option and click finish. The dialog box will be dismissed and professional STAAD.Pro graphical environment is displayed.

1.5 Creating the model using the graphical user interface (GUI)

It is useful to take some time to familiarize yourself with the components of the STAAD.Pro window. A sample of the STAAD.Pro window shown in the figure below. The window has five major elements as described below:

Figure 2-7:
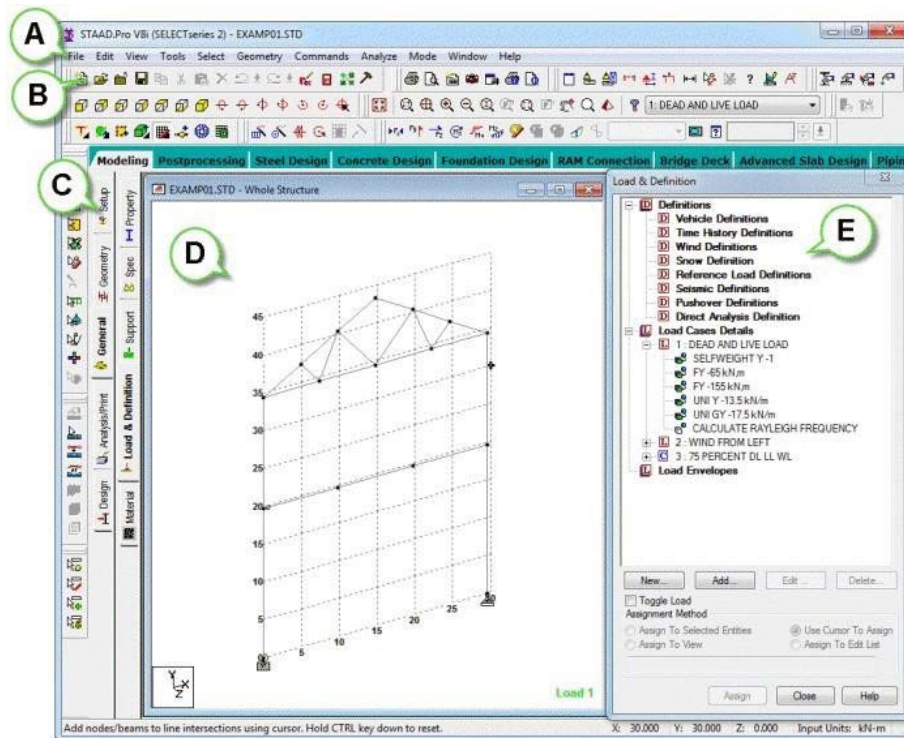
A) Menu Bar

> Located at the top of the screen, the menu bar provides access to all commands STAAD.Pro.

B) Toolbar

> The dockable toolbar provides access to the most frequently used commands can also create your own customized toolbar.

C) Main Window

> This is the largest area in the center of the screen, where the drawings and model

results are shown in pictorial form

D) Control Page

Control page is a set of tabs on the left side of the screen. Each tab in the Control page allows you to perform specific tasks. The pages are arranged from top to bottom, represents the logical sequence of operations, such as defining beams, specification of member properties, loading and so on.

Each tab has a name and an icon for easy identification. The name on the cards may or may not appear depending on your screen resolution and window size of STAAD.Pro. However, the icons always appear in the tabs Control page. The pages in the area of control of the page depends on the operating mode. The operating mode can be set from the mode menu bar

E) Data Area

The right side of the screen is called the **Data Area** Where different dialogues, tables, list boxes, etc. appear depending on the type of operation being performed. For example, when you select the **Geometry | Page Beam** The data area

contains the node-coordinate member-incidence and ironing board. When you're in **Load Page** The content of the changes in the data area shows the load cases assigned currently and icons for different types of loads.

The icons in the toolbar and in the area of *Page Control* ToolTip offer help. As we move the mouse over a button, the button name appears - called a ToolTip - above or below the button. This helps identify the floating Tooltip icon. A brief description of the icon also appears in the status bar.

We are now ready to start building the model geometry. The steps and, when possible, the corresponding STAAD.Pro commands (instructions that are written in the STAAD input file) are described in the following sections.

1.5.1 Generating the model geometry

The geometry of the structure consists of numbers of junctions or nodes, coordinates, member number, the member's connectivity information, numbers of element-plate, etc. From the point of view of the STAAD command file, the commands generated by the structure shown in section 1.2 are:

```
JOINT COORDINATES
1  0.  0 .; 2 0.  15 .; 3 20 15 .; 4   MEMBER
INCIDENCE 20 0
1 1 2; 2 2 3; 3 3 4
```

Procedure:

1. Select the Add Beam (Beam Add) option possible to facilitate the addition of beams to create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left corner of the drawing area.

Figure 2-8:
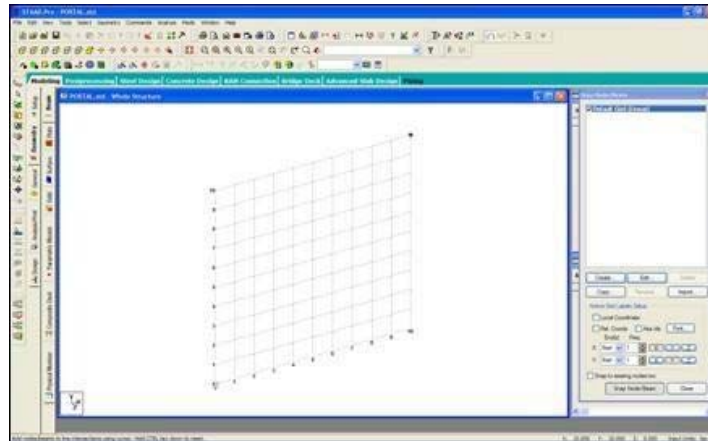
2. Dialogue Snap Node / Beam appears in the data area on the right side of the screen. Click Create.

A dialog box allows you to set a grid opens. Within this dialog box, there is a drop-down list from which you can select the linear, Radial or Irregular shape of the grid lines.

Figure 2-9:
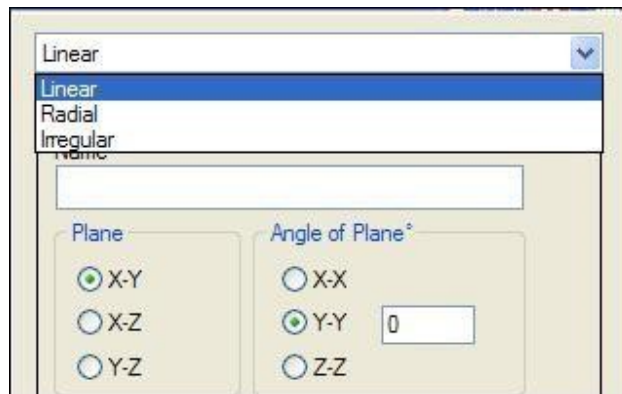
The linear profile is intended for the placement of construction lines perpendicular to each other along a pattern of "left to right - top to bottom", as in the lines of a chessboard.

The Radial tab allows construction lines appear on a web style that

makes it easy to create circular type models where members are modeled as piece-wise linear line segments into.

The Irregular tab can be used to create the grid lines with unequal spacing found in the world on a flat or incline.

Select line (linear), which is the default network.

In our structure, the segment formed by the members of 1-3 and 1-4 nodes, occurs in the XY plane. So in this dialog box, keep as the XY plane of the net. The size of the model that can be removed at any time is controlled by the number of construction lines left and right of the origin and spacing between lines of adjacent construction.

By setting 20 as the number of lines to the right of the origin along X, 15 on the origin Y and a 1 foot between lines along both X and Y (see figure below) we can draw a frame 20 ft X 15 m, suitable for our structure.

After entering the specifications, provide a name and click accept.

Figure 2-10:
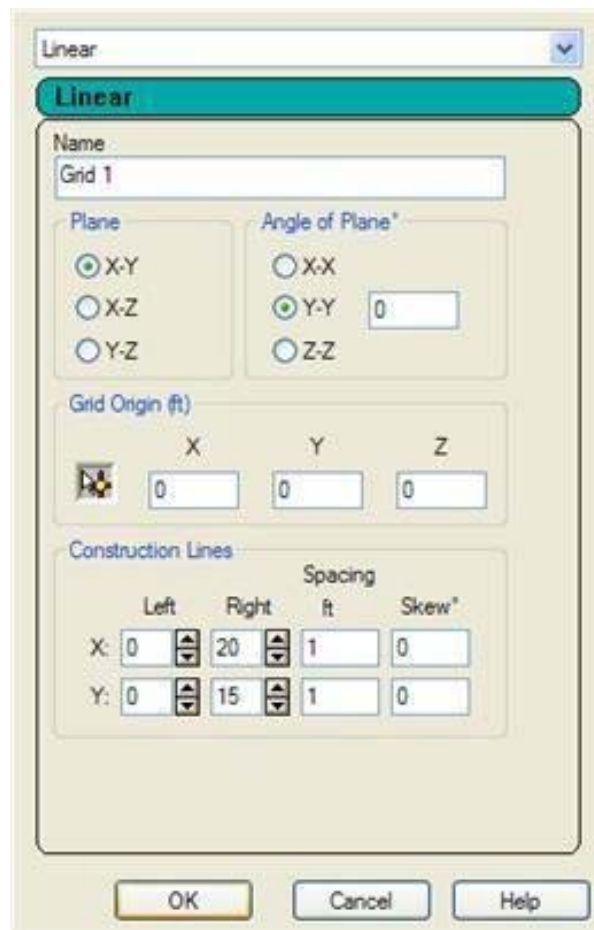
3. Note that these values are only starting grid setting, to enable us to draw the structure, and do not restrict our overall model these limits.

Figure 2-11:

In this way, we can create any number of grids. By providing a name, each new grid can be identified for future reference. To change the settings for this grid, click Edit.

4. Let's start creating the nodes. From the button Snap Node / Beam is active by default, with the mouse, click the origin (0, 0) to create the first node.

Figure 2-12:
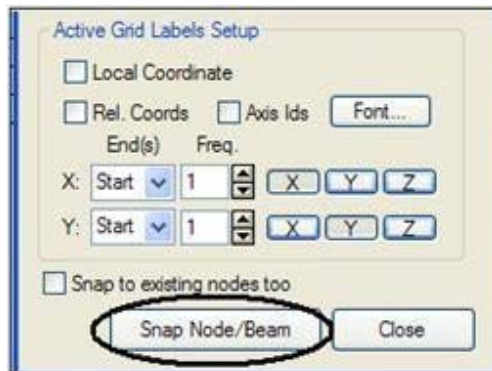
Figure 2-13:
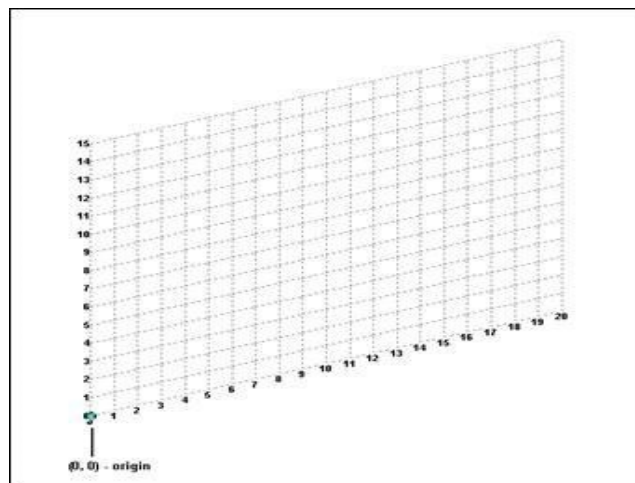
5. Similarly, click the following to create nodes and automatically successive nodes are joined by members of the beam

   (0, 15), (20, 15) AND (20, 0)

   The exact location of the mouse can be monitored in the status bar at the bottom of the window that continuously update the coordinates X, Y and Z of the current cursor position.

Figure 2-14: Status bar

When you have completed steps 1-4, the structure is displayed in the drawing area as shown below.
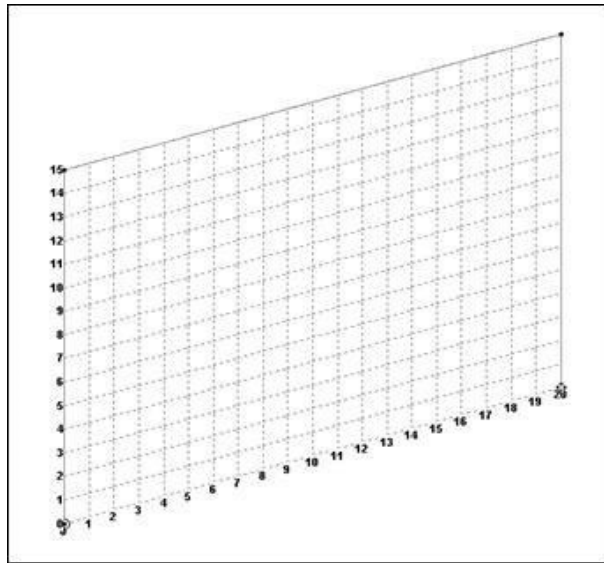
Figure 2-15:

6. At this point, remove the grid structure. To do this, click on the close box Snap dialog Node / Beam

Figure 2-16:

Now the grid and the structure of the main window should look like the figure shown below will be deleted.

Figure 2-17:
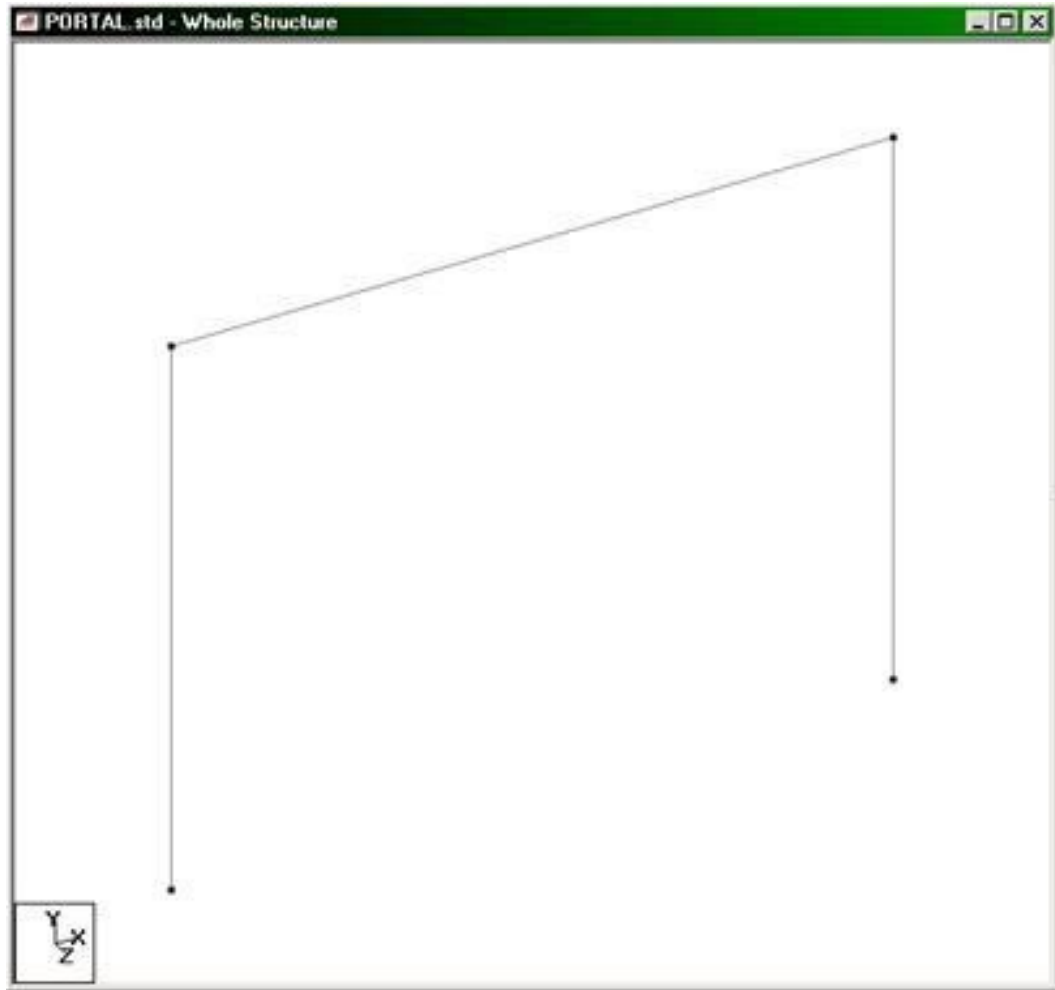
It is very important that you save your work often to prevent data loss and protect our investment of time and effort against power outages, system problems or other unforeseen events.

Tip: Remember to save your work or by selecting File> Save, save the tool or press **CTRL + S**.

1.5.2 Switching node labels and beam

The node and beam labels are a way to identify entities that have drawn on the screen.

1. Right-click anywhere in the drawing area, select the labels in the pop-up menu or select View> diagrams of structures. The dialog box diagrams opens in the Labels tab.

2. In the dialog box that appears diagrams, numbers selected node (Node Numbers) and number of beam (Beam Numbers) and click on OK.

Figure 2-18:
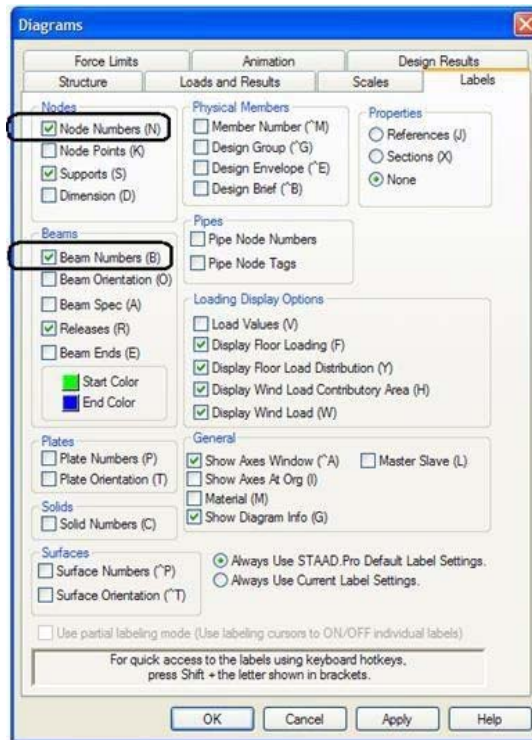
The following figure shows the node and beam numbers shown in the structure. The structure of the main window should look like the figure shown below.

Figure 2-19:
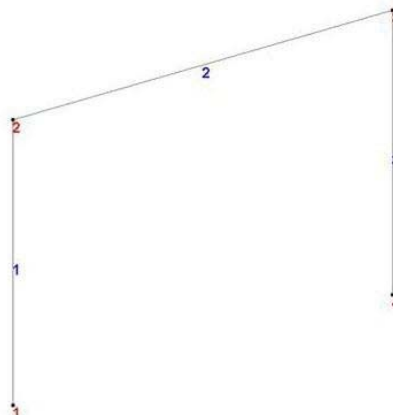
1.5.3 Specific properties of the Member

Our next task is to assign cross-section properties of beams and columns (see figure in section 1.2). For those who want to know the equivalent commands in the STAAD editor are:

```
MEMBER AMERICAN PROPERTY
1   3   TABLE ST W12X35
2 TABLE ST W12X34
```

Procedure:

1. To define the properties of members, select the tool properties page located on the top toolbar.

Figure 2-20:

Property Page

Alternatively, one can go to **General | Property** the tabs on the left side of the screen as shown below.

Figure 2-21:

2. In any case, the Properties dialog box opens (see figure). The type of property you want to create is the way W of the AISC table. This is available on the button of the database sections (**Section Database**) In the dialog box

   Properties as shown below. So click *Section Database*.

Figure 2-22:
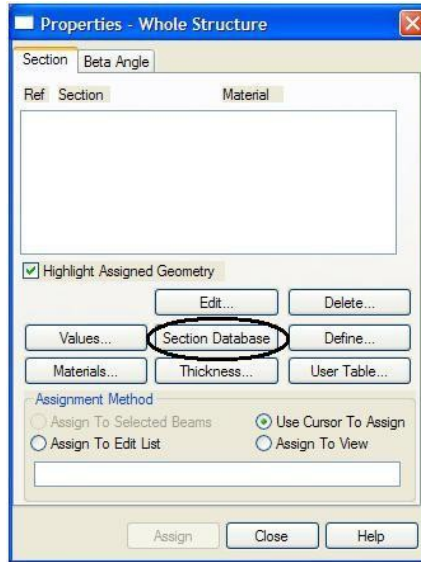
3. In the table dialog profile-section (Section Profile Tables) that appears, select the tab **W Shape** under the American option.

   The checkbox material (*Material* check box) is selected. Leave it as it is used to subsequently assign the material constants E, density, Poisson, etc, along with the cross-section, assigning default values.

   Choose **W12X35** (**IR310x52** according to MIAC) as the size of the beam and **ST** as such section. Then, click Add as shown in the following figure.

Figure 2-23:
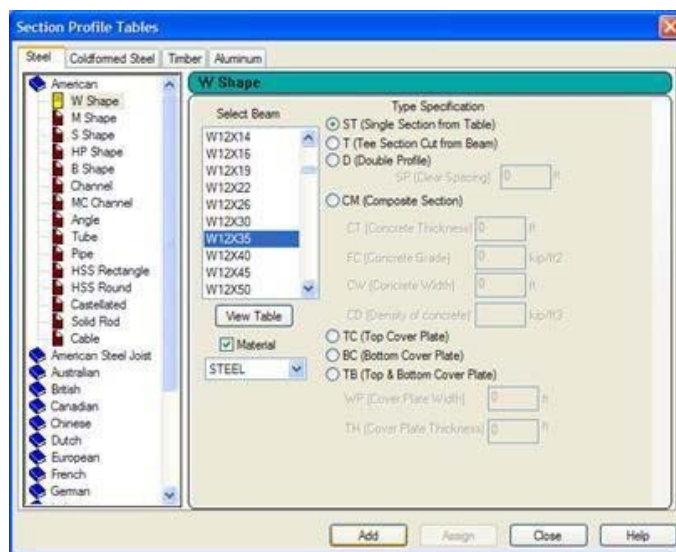
4. To create the second member property (ST W14X34), select the shape of **W14X34** (**IR360X51**) And click add.

 After you have created the member properties, click Close.

5. The next step is to associate the properties we just created with selected members in our model. Follow these steps.

 a. Select the first reference to the property in the Properties dialog box (W12X35).

 b. Choose "**Use Cursor to Assign**"Allocation method in the box.

 c. Click on **Assign**.

 The mouse cursor will change to ‘**I**

 d. Click on the members 1 and 3.

 e. To stop the allocation process, or select assign or press the ESC key.

Figure 2-24:
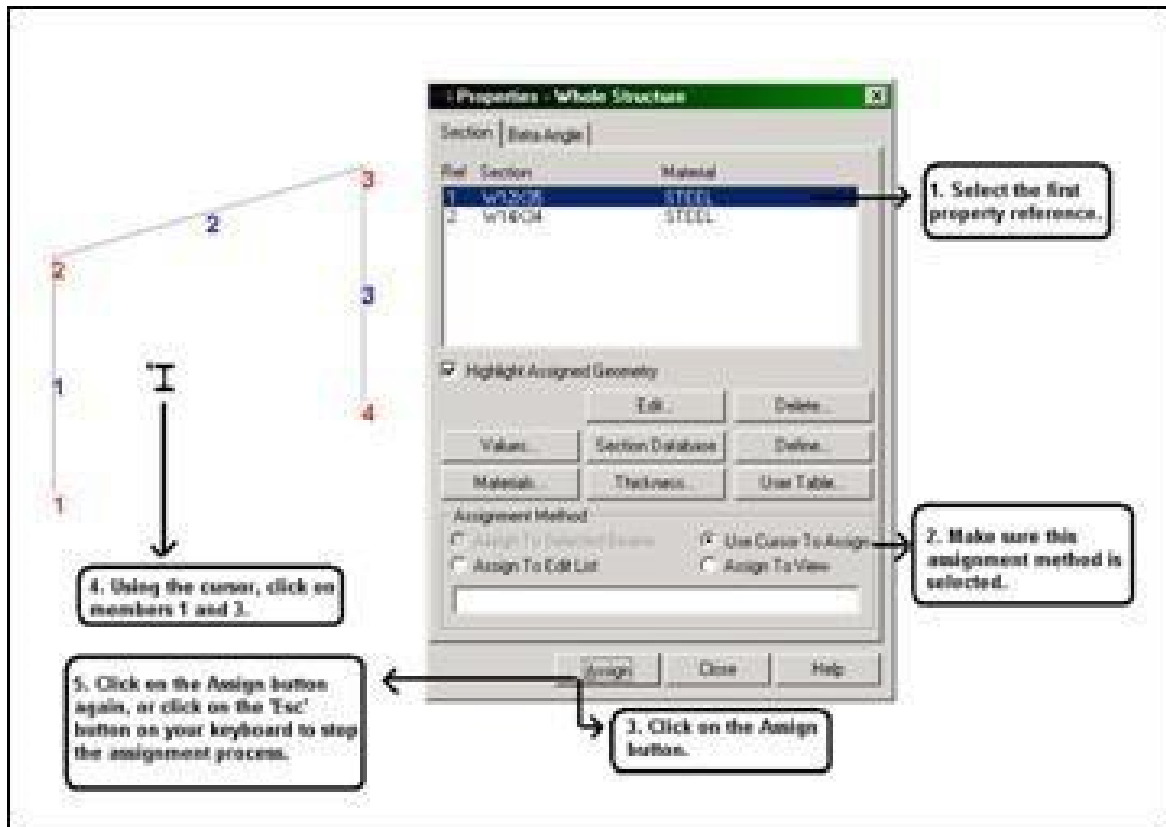
6. Similarly, assign the second property reference (W14X34) the Member 2.
    After both properties have been assigned to the respective members, our model should
    resemble the following figure.

Figure 2-25:
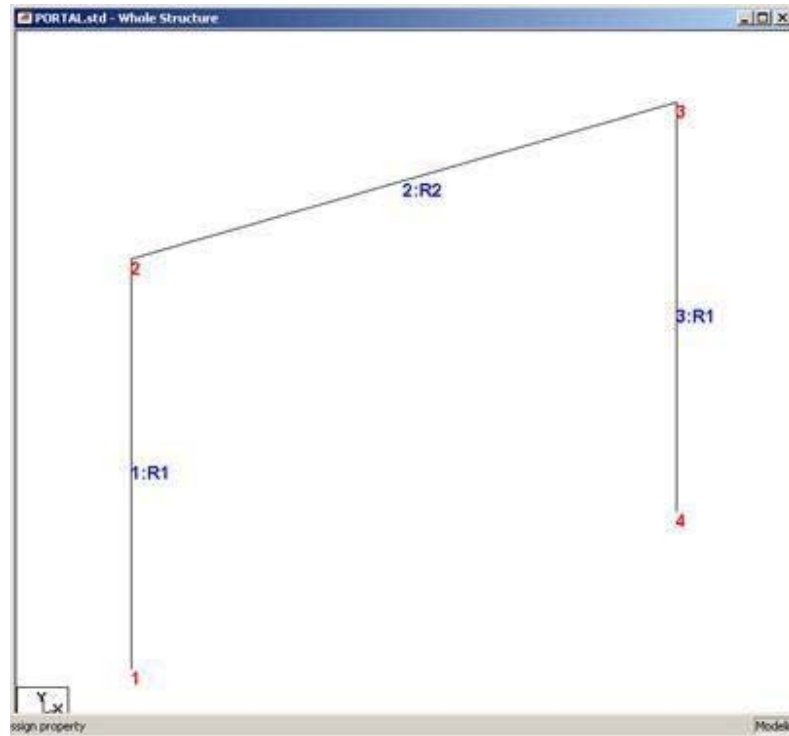
> **Tip**: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**

1.5.4 Specifying material constants

In Section 1.5.3, we kept the check box of "on" Material to assign the properties of the member.
Consequently, the material constants are assigned to members together with the properties, and
obtained the following commands in the command file:

```
CONSTANTS
29000 E MEMB 1 TO 3
0.3 POISSON MEMB 1 TO 3
DENSITY 0.000283 MEMB 1 TO 3
6.5e-006 ALPHA MEMB 1 TO 3
```

Therefore, there is no need to separately allocate constant. However, if these had not been previously assigned, selecting from the submenu items **C o m m a n d s> Material Constants,** used to make these assignments.

Figure 2-26:
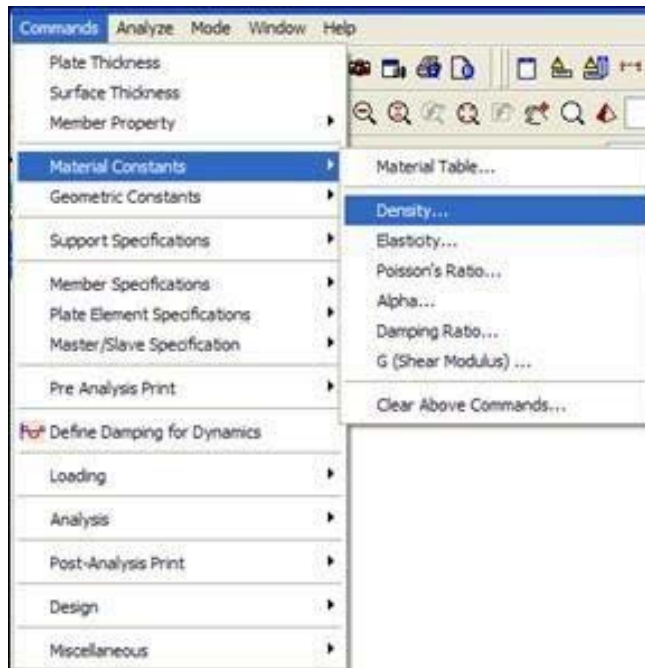
1.5.5 Changing input length units

To enter compensation values, as a matter of convenience, it is easier if our length units are inches instead of feet. The commands generated are:

```
UNIT KIP INCHES
```

Steps:

1. Select the tool or input units

Figure 2-27:

Input Units

o**Tools> Set Current Input Unit**.

The Units Setup dialog box opens current input.

2. Set the *units long* in **Inch** inches and click **OK**.

Figure 2-28:

1.5.6 Specifying offsets member or element

Since the beam 2 actually covers only the clear distance between the faces of the column and not the distance from center to center, we can take advantage of this aspect specifying offsets. The member 2 is offset to the initial set by 6 inches in the global X direction, 0.0 and 0.0 in directions Y and Z. The same member is offset by the negative 6.0 inches in the final joint. The corresponding STAAD commands are:

```
MEMBER OFFSET
2  START 6.0  0.0 0.0
2  END  -6.0  0.0 0.0
```

Procedure:

1. Since we know that 2 is the member to be assigned to the offset, first select this member before the offset itself. Select Member 2 by clicking on it with the tool **Beam Cursor** . Member is highlighted selected. (See 'Tasks performed frequently' of this manual to learn more about selecting members section).

2. To set the compensation of members, select the tool specification page (**Specification Page)** located on the top toolbar.

Figure 2-29:

Specification Page
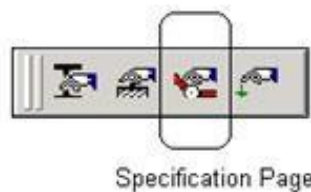
Alternatively, one can go to **General | Spec** from the left side of the screen.

Figure 2-30:
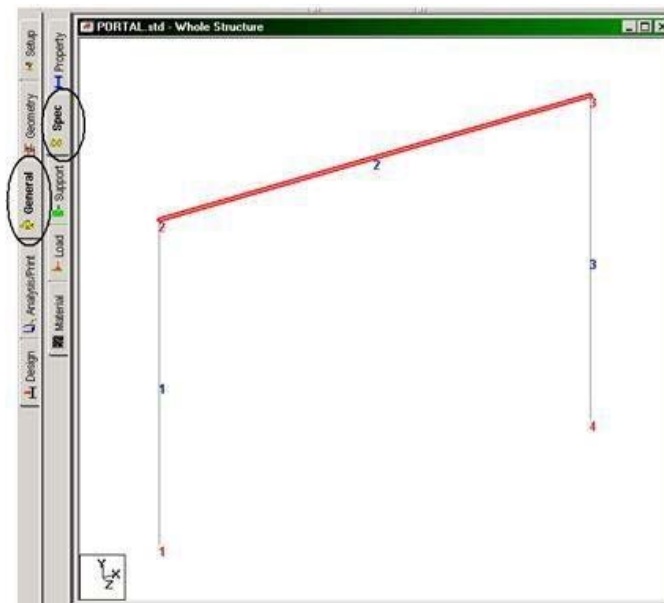
3. In any case, the specifications dialog shown below appears. The press members and offsets are defined through the button beam (**Beam)** in this dialog box as shown below.

Figure 2-31
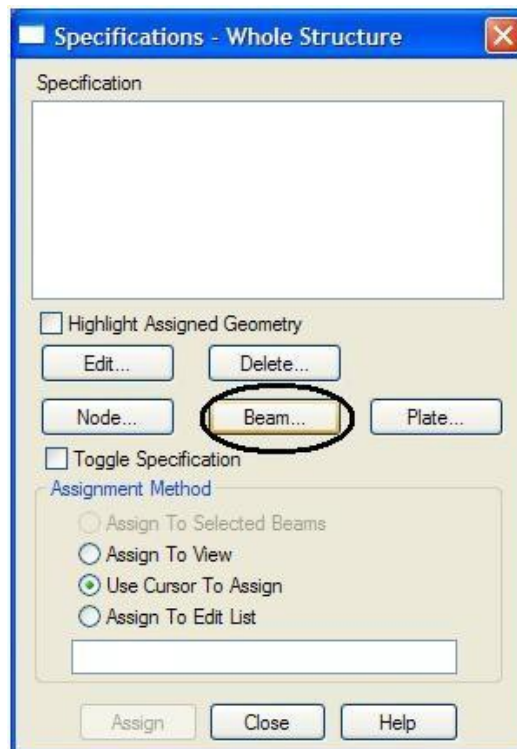
4. In the dialog box beam specifications that opens, select the tab for **Offset**. We define the offset in the starting node in the direction X. Therefore, make sure the option is selected **Start** Location option below.

   Specify a value of 6.0 for X. Since we have already selected the member, we click Assign.

Figure 2-32:

5. To apply the displacement at the end node, repeat steps 3 and 4, except for selecting the final option, providing a value of **-6.0** for X.

   After you have been assigned both the start and end offsets, the model will be as shown below.
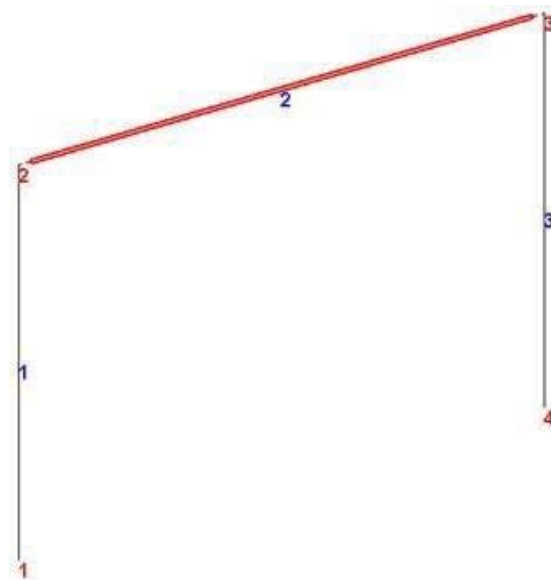
Figure 2-33:

Click anywhere in the drawing area to desrealzar member.

1.5.7 Printing member information in an output file

We would like to get a report that contains information on all members including initial and final numbers of connections (incidence), length of membership, angle beta (Beta Angle) and the final statement in the STAAD output file member. The corresponding STAAD command is:

```
PRINT ALL INFORMATION MEMBER
```

Procedure:

1. Select all members going to the menu **Select> By All> All Beams**

2. Select **Commands> Print Preview Analysis> Member Information.**

   The box of member information Print dialog (Print Member Information)

   Figure 2-34:
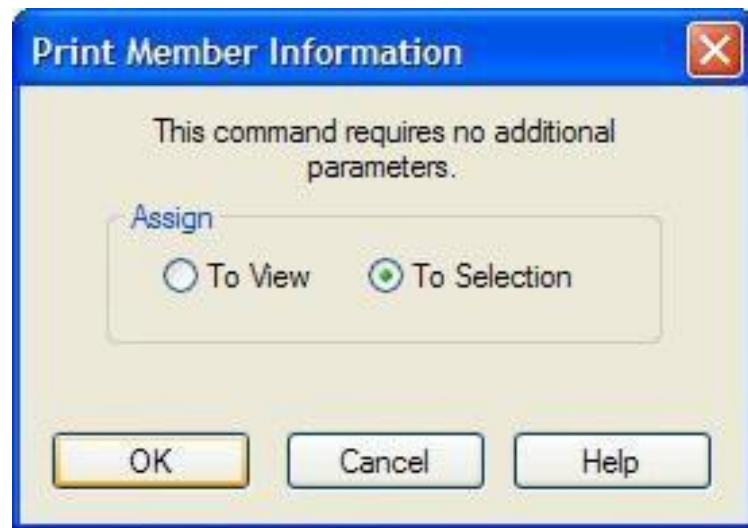
3. Make sure that the method of allocation is **To Selection**.

4. Press the button **OK** in this dialogue.

   Click anywhere in the drawing area to desrealzar member.

1.5.8 Specifying Support

The specifications of this problem (see section 1.2) require restrict all degrees of freedom at node 1 (embedded support "fixed") and an articulated support (Pinned) in node 4 (restricted against all translations, free to all rotations). The commands to be generated are:

SUPPORTS

1 FIXED; 4   PINNED

Procedure:

1. To create a support tool selected **Support Page** located in the top toolbar as shown below.

Figure 2-35:

Or select the tab **General> Support Page** from the left of the screen.

Figure 2-36:
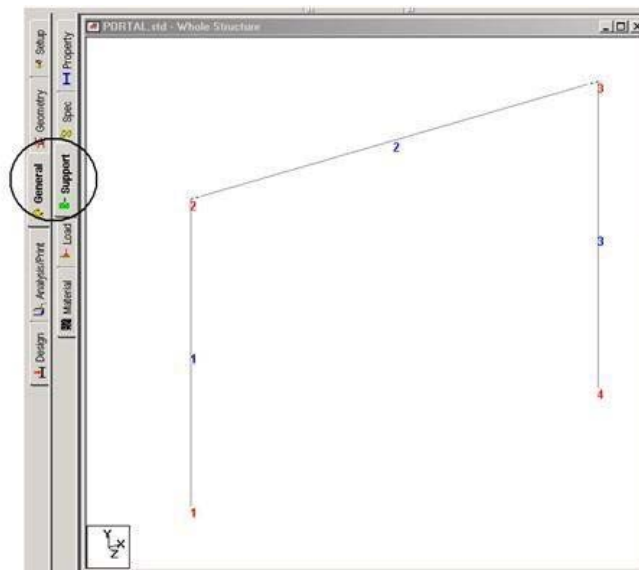
2. In any case, the dialog box *Supports* opens as shown in the following figure. Since we know that node 1 is to be asso ciated with a fitted stand, using the **Nodes Cursor**

, Select node 1 is

He highlighted. (See 'Tasks performed frequently' of this manual to learn more about selecting members section).

3. Click **Create** in the dialog *Supports* as shown below.

Figure 2-37:
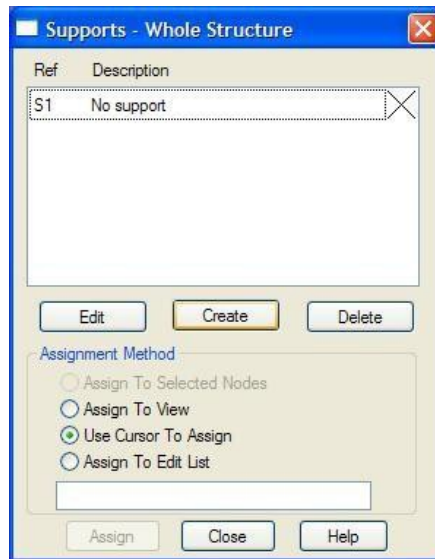
4. In the dialog box *Create Support* that opens, select the tab **Fixed** (Which also happens to be the default) and click **Assign** as shown below. This creates a kind of support FIXED on node 1 where all subjects are
    6 degrees of freedom.

Figure 2-38:
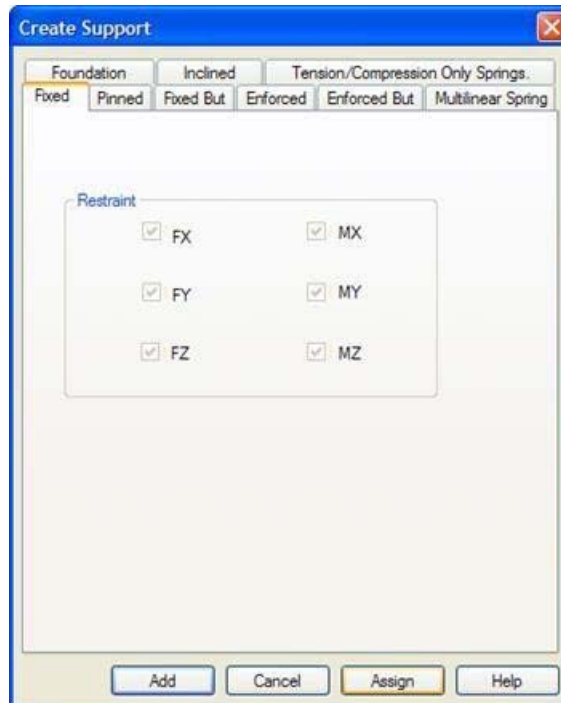
5. To create a PINNED support (articulated) in node 4, repeat steps 2-4, except

to select the node 4 and select the tab **Pinned** in the dialog box. After the supports have been assigned, the structure will look as shown below.

Figure 2-39:
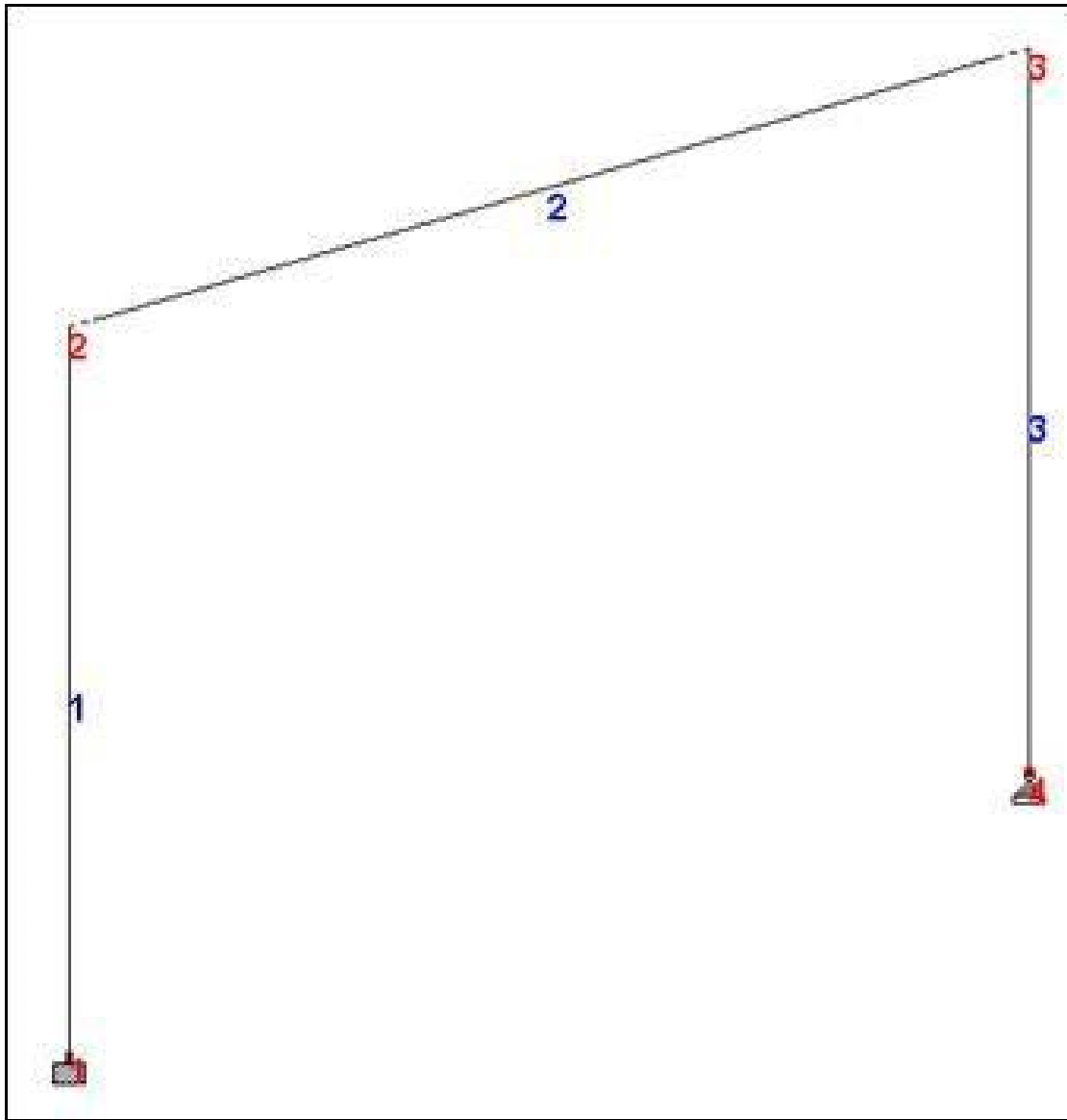
Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

1.5.9 Visualizing the 3D model

Let's see how we can show our 3D model.

1. Right-click and select **Structure Diagrams (**structure diagrams) from the popup menu or select **View> Structure Diagrams**.

The dialogue of the diagrams (*Diagrams*) Is opened in the flange *Structure*.

Figure 2-40:
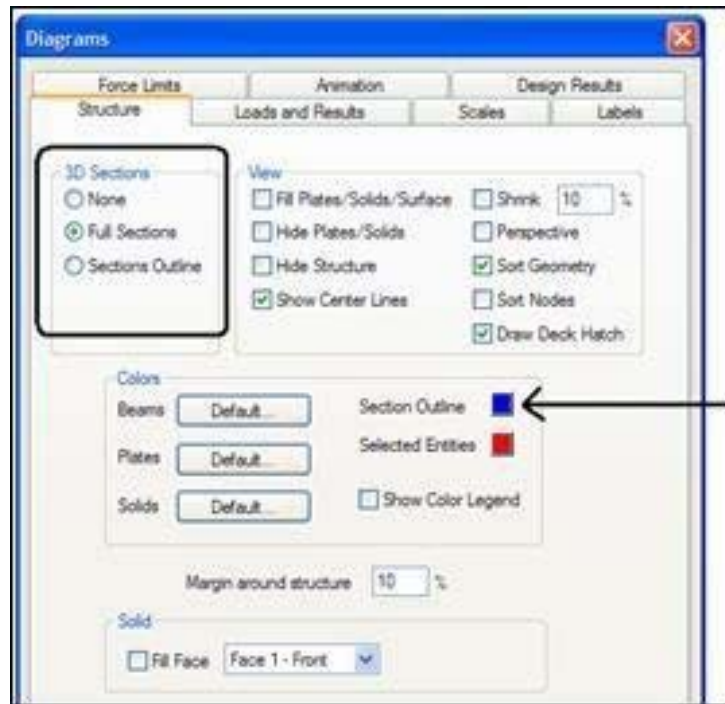
Sections 3D options control how members are displayed.

None (None)

Shows the structure without showing the properties of the members and cross members.

Full Sections (Complete Sections)

3D shows the cross sections of members, depending on the member properties.

Sections Outline (Outline of Sections)

Displays only the outline of the sections of the members

2. Select **Full Sections** and click **OK**.

Tip: You can also change the color of the sections by clicking the button color contour section (*Section Outline*) In color.

The resulting diagram is shown in the following figure.

Figure 2-41:

Fast 3D Rendering Option

One.   Right-click and select **3D Rendering** from the popup menu or select **View> 3D Rendering**.

A new view opens with the 3D model rendering, perspective

1.5.10 Specifying loads

Three load cases should be created for this structure. Earlier in this tutorial details of individual cases are explained. Then the corresponding commands generated are listed.

```
UNIT KIP FEET
LOADING  1  DEAD  +
        MEMBER LIVE LOAD
2 UNI GY -2.5
LOADING  2  WIND  FROM  LEFT
JOINT LOAD
2 FX 10.
March 75 PERCENT LOAD COMBINATION OF (DL +
LL + WL) 1 0.75 2 0.75
```

Procedure:

The creation and allocation of load cases involves the following two steps

I.   First, we will create all the three load cases

II.   Then assign to the respective members / nodes

Creating load cases 1 and 2

1. To create loads, first select the tool **Load Page** (Page load) on the top toolbar.

Figure 2-42:

Load Page
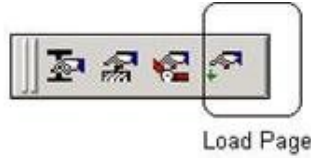
Alternatively, one can go to **General | Load** from the left side of the screen.
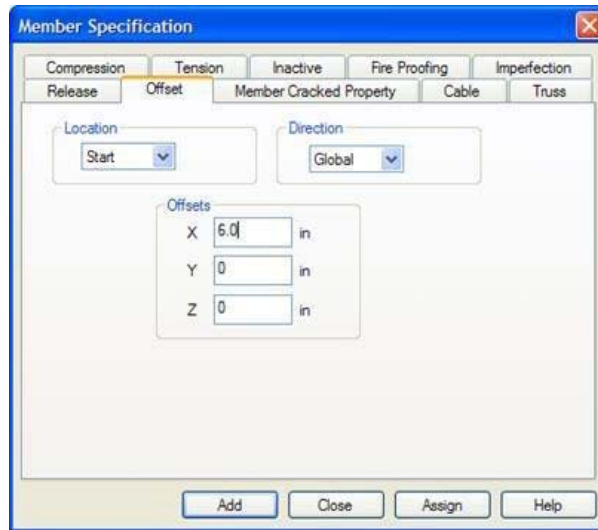
Figure 2-43:
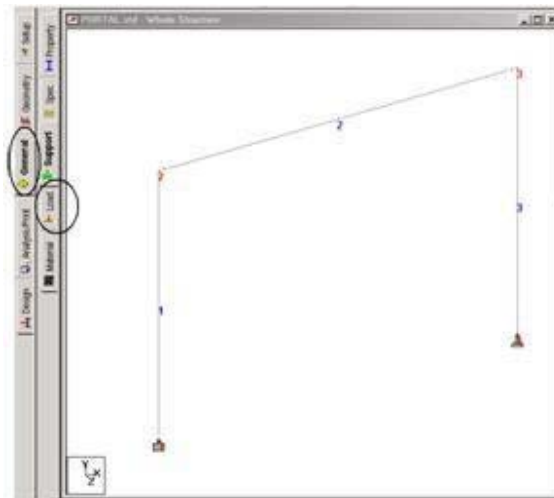
2. Before creating the first load case, we need to change our units of length in feet. This, as above, use the tool **input Units** "Input units" (see Section 1.5.5) Note that a window appears titled "Load" to

right of the screen. To create the first load case, highlight details of case load (**Load Cases Details**) And then click **add** (**Add**) In the dialog box.

Figure 2-44:
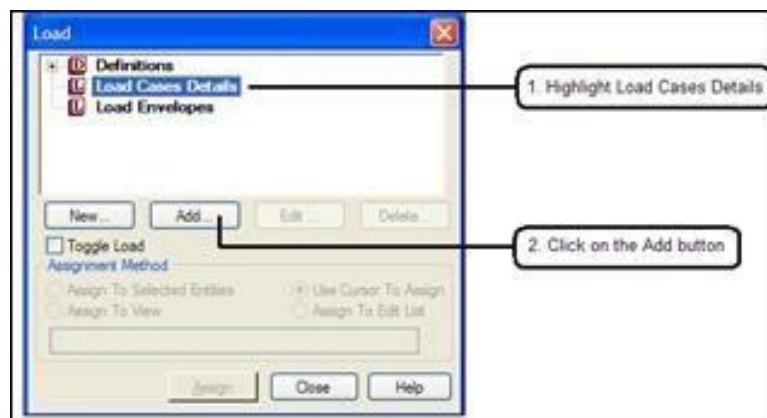
3. Dialogue *Add New Load Cases (r I added new load cases)* it opens.

The drop-down list for type of charge is available if you want to link the case load that we are creating with any of the definitions of ACI, AISC or IBC of dead, live, ice loads, etc. This type of association must be made if we use the installation program to automatically generate load combinations according to the codes. Note that there is a checkbox called reducible by UBC / IBC. This feature becomes active only when the load case is assigned a type called Live load at the time of creation of that case.

As we do not intend to use the option of automatic generation of load combination (*Auto Load Combination*), We will leave the type of load as **None** (*Loading Type*- None). Enter **DEAD + LIVE** as the title for the load case 1 and click **Add**.

Figure 2-45:

The newly created load case appears under the option *Load Cases Details* (Details of charging load cases).

To create the load member, first select input 1: DEAD + LIVE. You will notice that the dialog *Add New Load Items* (Add new items to load) now shows more options.

Figure 2-46:

4. In the dialog box *Add New Load Items*Select the tab **Uniform Force** under the loading point of the member. Specify **GY** as the address, type **-2.5** as strength and click **Add**.

Figure 2-47:
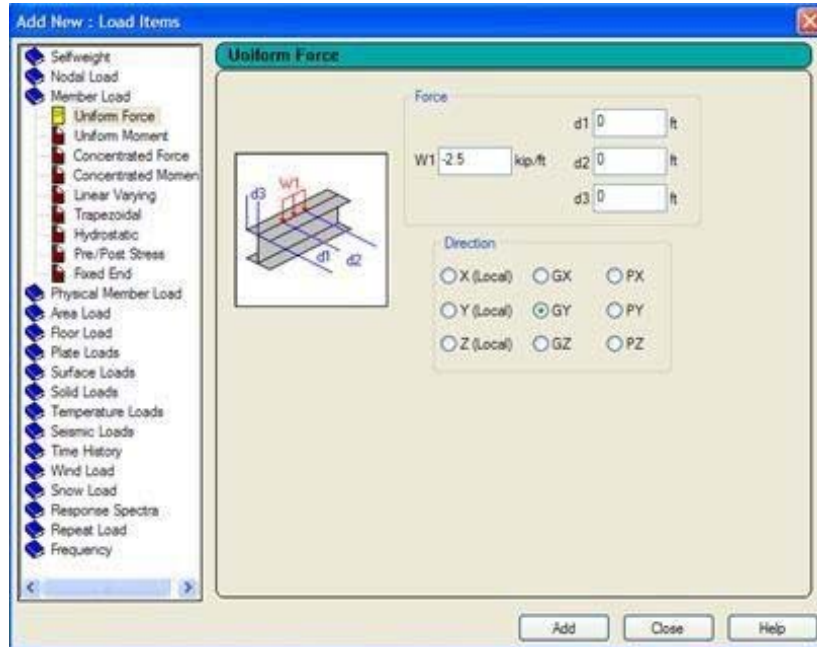
The next step is to create the second load case containing a common load

5. Perform step 3 again to create a second type of load. Enhancement **Load Case Details,** click **Add,** let the type of load as **None**. Specify the title of the second load case as **WIND FROM LEFT** (Wind from left) and click **Add**.

Figure 2-48:
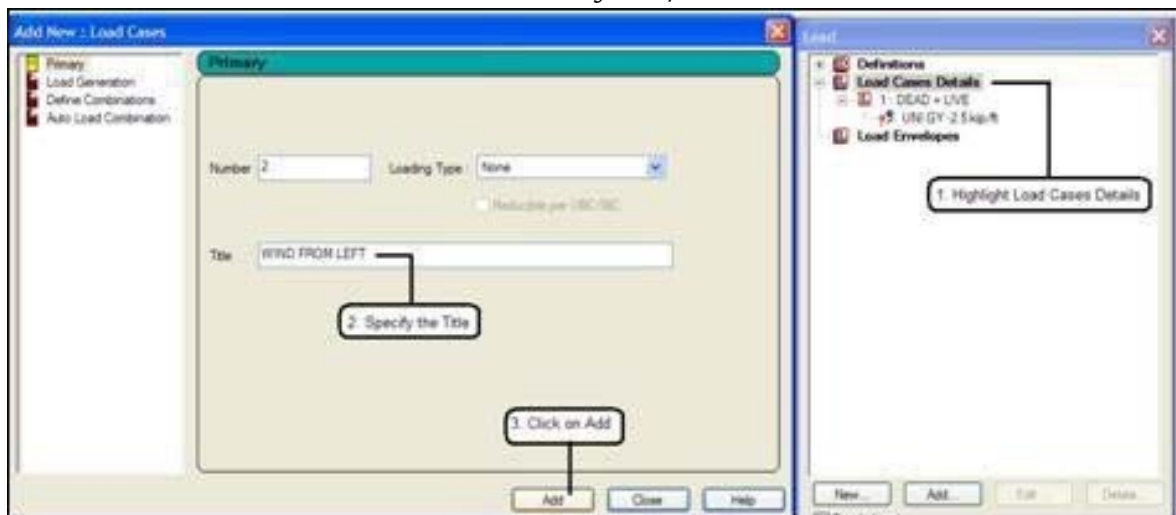
6. Then, to create the joint load, select **2: Wind From Left**.

Figure 2-49:

7. In the dialog box *Add New Load Items*Select the tab **Node** Nodal point for the load. Specify **10** for **Fx** and click **Add**.

Figure 2-50:
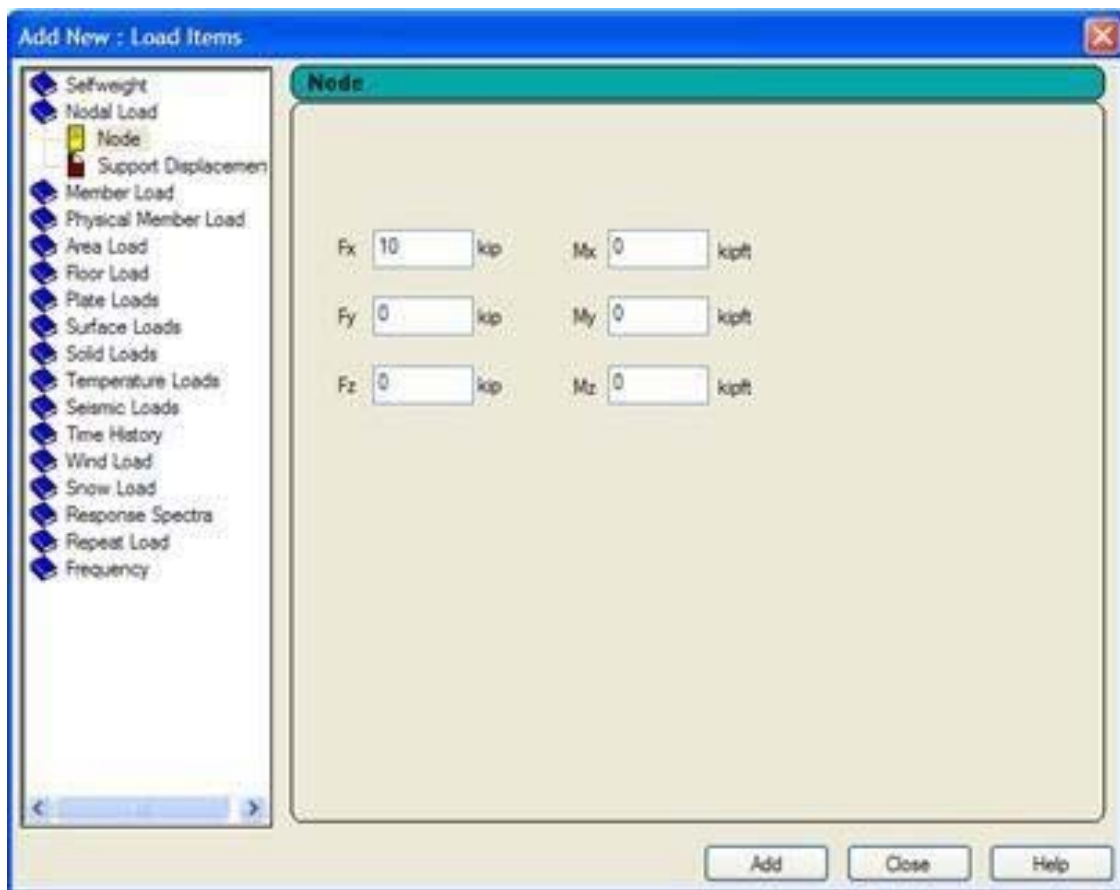
Creating the third load case

The Load Cases 1 and 2 were cases of main charge. Charging Case 3 was defined as a combination of load. Then the next step is to define the load case 3 and 0.75 x (1 load

Load + 2), which is a combination of load.

1. This, again, select **Load Case Details (**details of the case load) in the

    Loads dialog & Definition. In the dialog box *Add New Load Cases*Press
    the tab **Define Combinations** the left side. Specify the title as the **75 Percent of [DL + LL + WL]**.
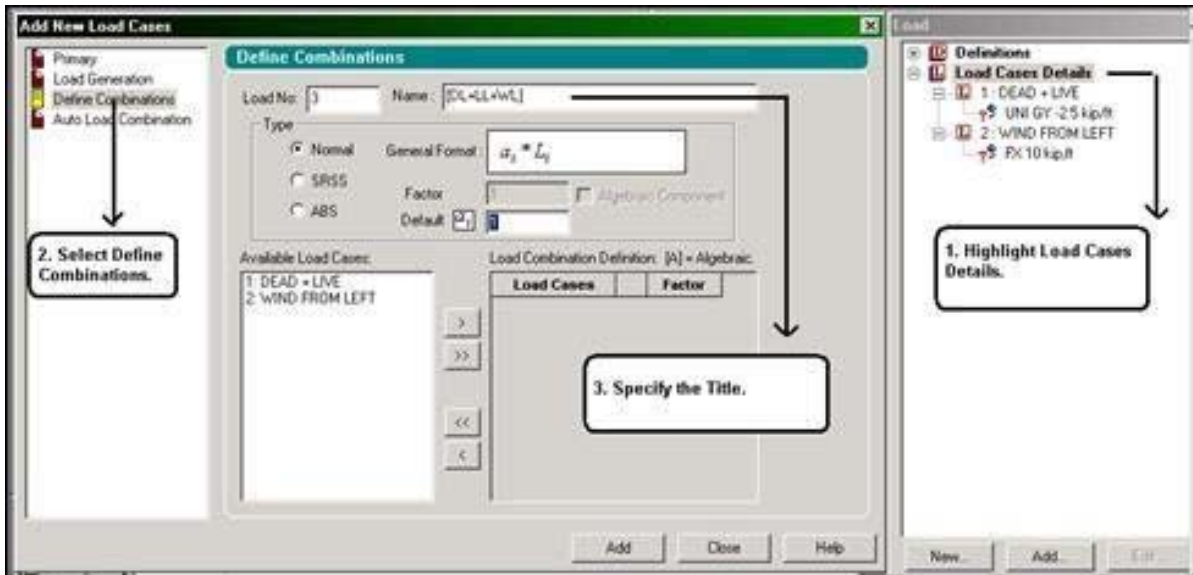
Table define combinations, the combination type is set to predetermined load **Normal (**default**)**, Which means an algebraic combination. The other types are called combination available **SRSS** ("Square root of sum of squares" - root

square of the sum of squares) and ABS (absolute). The SRSS type offers the flexibility of SRSS and algebraic part. That is, certain load cases are combined using the square root of the sum of the squares approach, and the result is combined with algebraic other cases, as in

$$A + SQRT (B * B + C * C)$$

Where A, B and C are primary individual cases.

We intend to use the type of algebraic combination default (Normal).

2. Table *defined combinations*Select both load cases in the list box on the left side (by holding down **CTRL**) And click [>]. The load cases appear in the list box on the right. Then, enter **0.75** in the edit box *Factor*. (These data indicate that the two cases are adding a

    multiplication factor loading of 0.75 and that the results of the load combination would be obtained by the algebraic sum of the results for individual load cases). Click **Add**.

Figure 2-52:
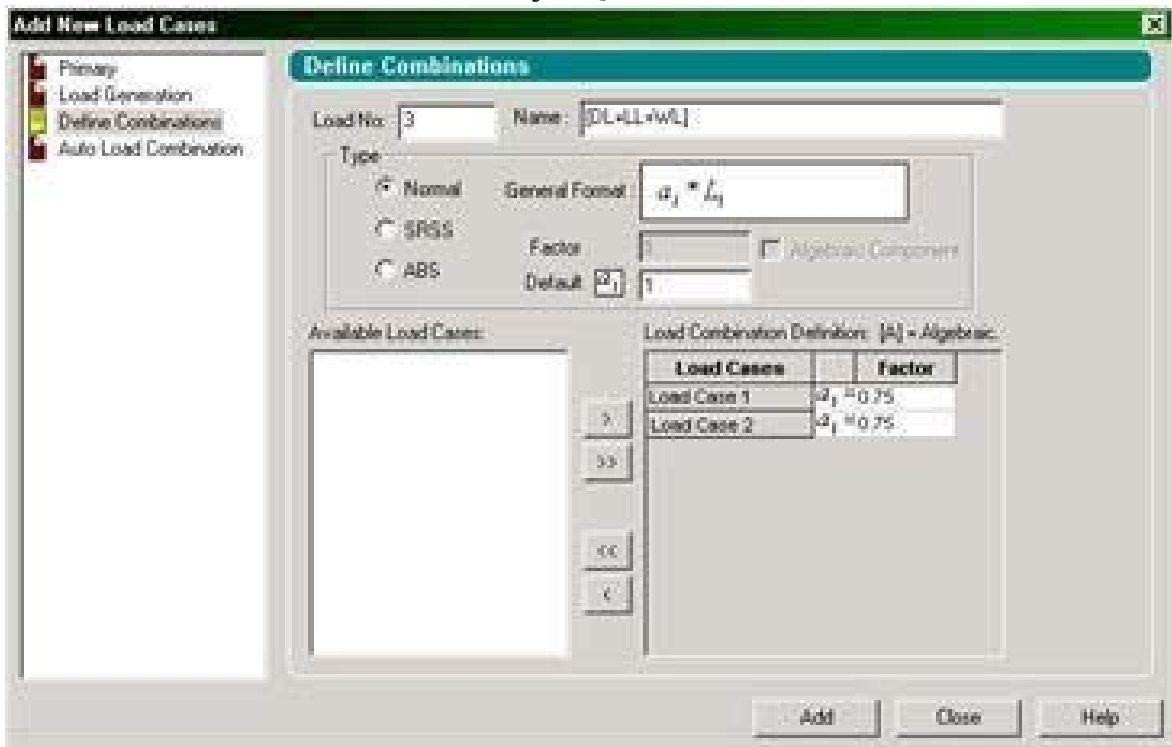
Now that we have completed the task of creating all load cases 3, click **shut down**.

Associating a load case with a member

Our next step is to associate the load case 1 with 2 member following steps.

1. Select the first reference load in the load dialog (UNI GY-2.5 kip / ft).

2. Select the option **Use Cursor to Assign** in the allocation method box.

3. Click on **Assign**.

The cursor will change to this form

4. Click the Member 2.

5. To stop the allocation process, click **Assign** or press **Esc**.

Figure 2-53
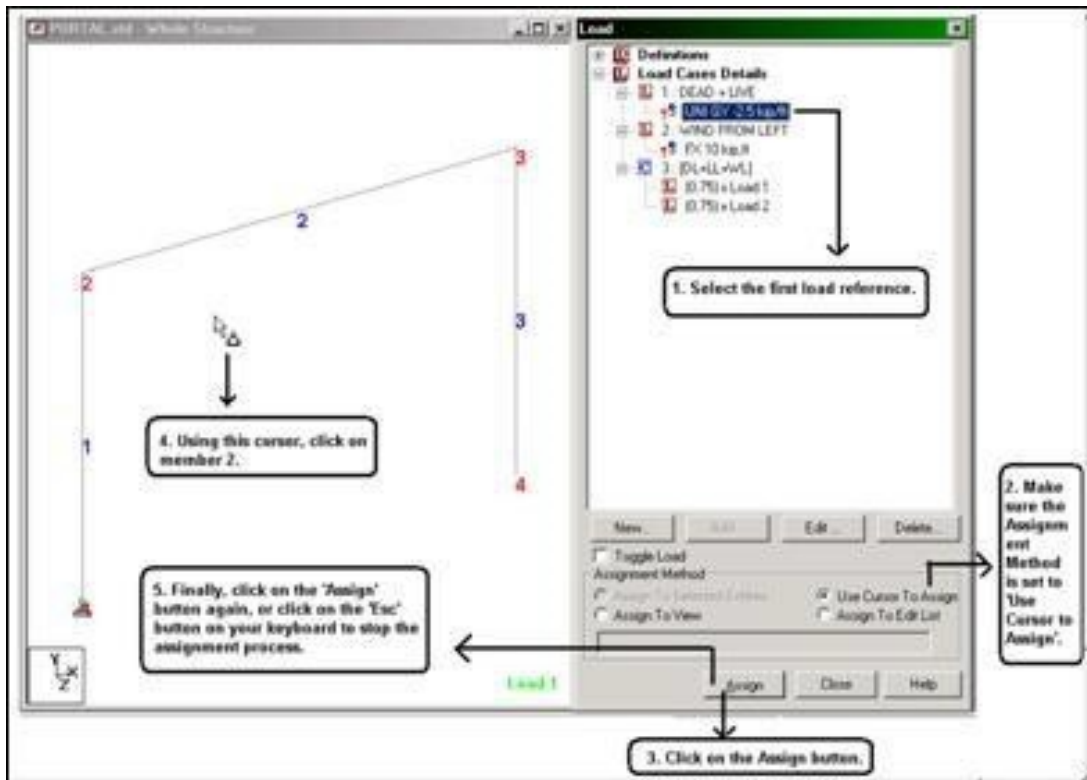
After charging member is assigned, the pattern will be as shown below.

Figure 2-54:
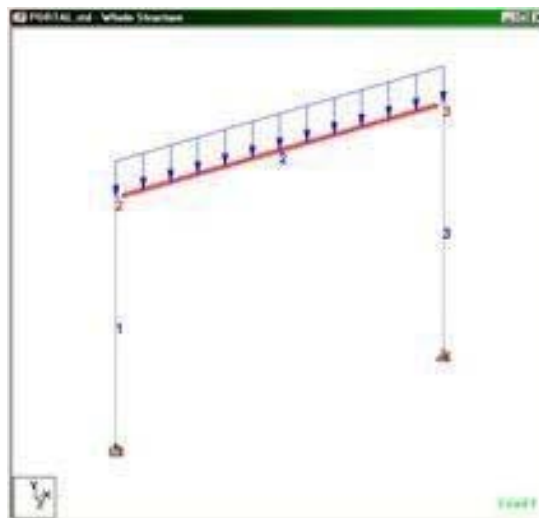
Similarly, the second assigning load case (FX 10 kip, ft) to node 2.

Figure 2-55:
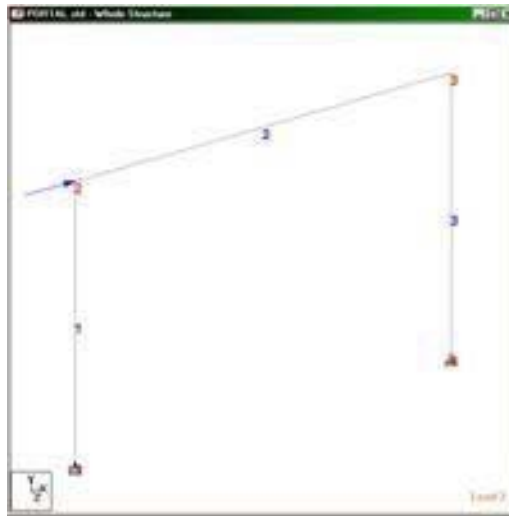
> **Tip**: Remember to save your work or by selecting File> Save, save the tool or press **CTRL + S.**

1.5.11 Specifying type of analysis

We are required to do the kind of analysis as a linear static type. We also need to get a report of static equilibrium. This requires the command:

```
PERFORM ANALYSIS PRINT CHECK statics
```

Procedure:

1. To specify the analysis command, go to **Analysis / Print | Analysis** from the left side of the screen. The Analysis / Print Commands dialog opens:

Figure 2-56:

2. In the dialog box *Analysis / Print Commands* that appears, make sure the tab is selected **Perform Analysis** (Analyzes). Then, check the Static check printing (**Check Statics** print option). Click Add and then Close.

Figure 2-57:
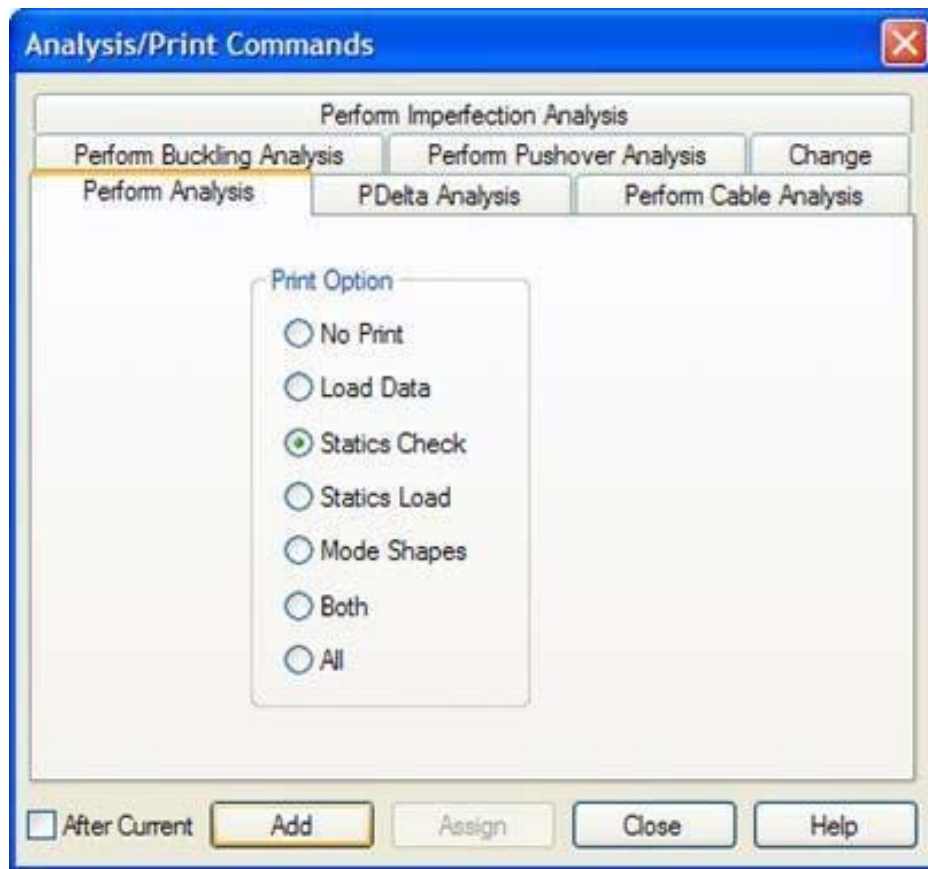
Tip: Remember to save your work or by selecting File> Save, save the tool or press **CTRL + S.**

1.5.12 Specifying print commands further analysis

We would like to get the final member forces and support reactions are written to the output file. This requires specification of the following commands:

```
PRINT ALL FORCES MEMBER
PRINT SUPPORT REACTION LIST April 1
```

Procedure:

1. The dialog box to specify the above opens when **Analysis / Print | Post-Print** is selected.

Figure 2-58:
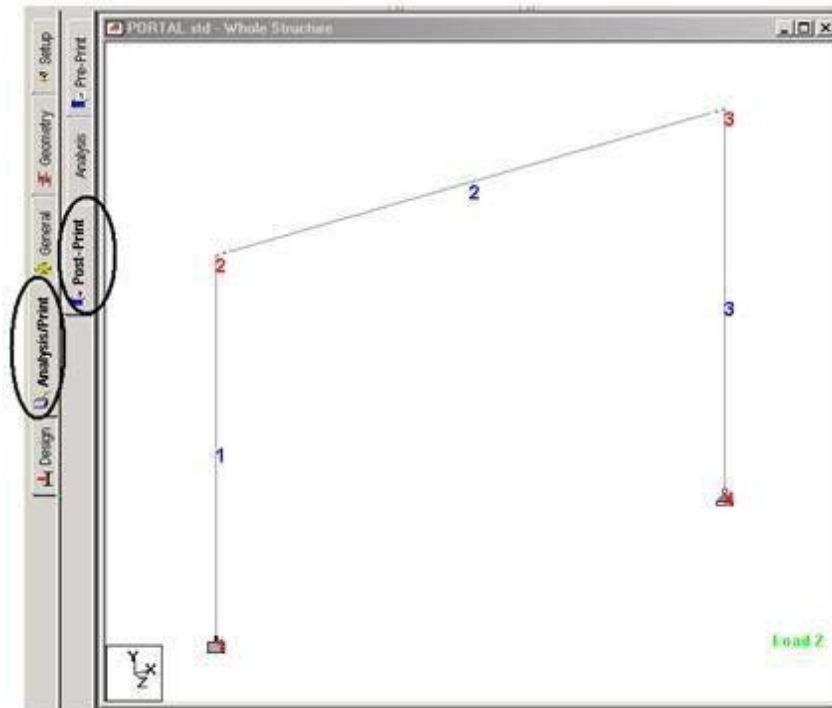
2. Then select all members.

3. Click **Define Commands** on the right side of the screen.

Figure 2-59:
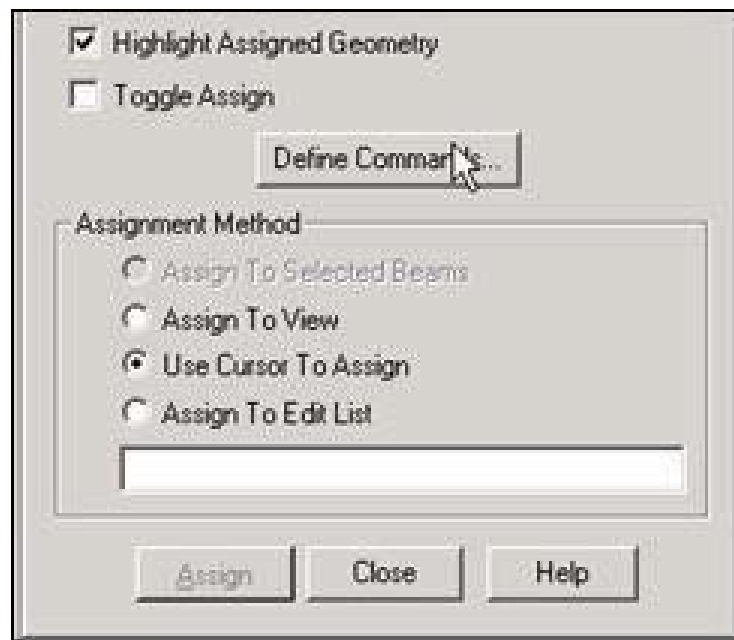
4. In the dialog box *Analysis / Print Commands* that appears, select the tab **Member Forces** and click **Assign** followed by the button **shut down**

Figure 2-60:

5. Repeat steps 2-4 except selecting both supports and support reactions tab in the dialog box analysis and printing commands. (Recall that

brackets can be selected by turning the *Nodes Cursor*, Hold the key '**Ctrl** ' and click

brackets). Click **assign** and then **shut down**. In this point, the Print dialog Post analysis should resemble the figure shown below.

Figure 2-61:
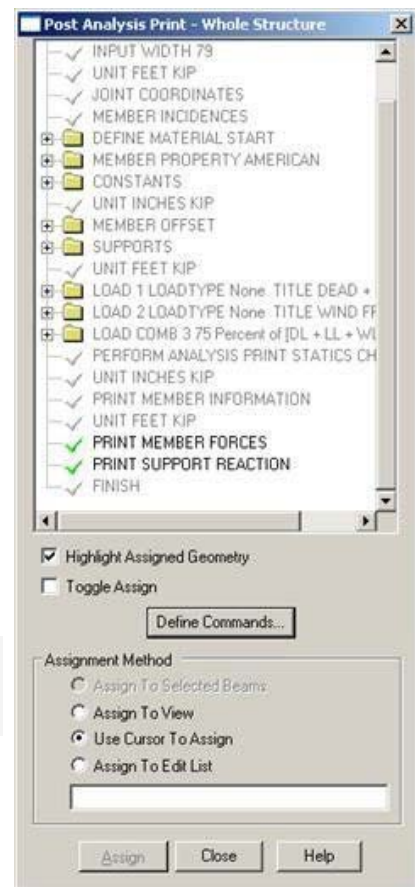
Tip: Remember to save your work or selecting **Archive > Save**The tool **save** or press **CTRL + S**.

1.5.13 Pre-selection of load cases to be used in steel design

The design of steel shall be carried out for load cases 1 and 3 only for the specification at the beginning of this tutorial. To tell the program to use these cases and ignore the rest, we have to use the LOAD LIST command. The command appears in the STAAD file as:

```
LOAD LIST March 1
```

Procedure:

1. In the menus at the top of the screen, go to the option **Commands> Loading> Load List** as shown below. The List cargo box opens.

Figure 2-62:
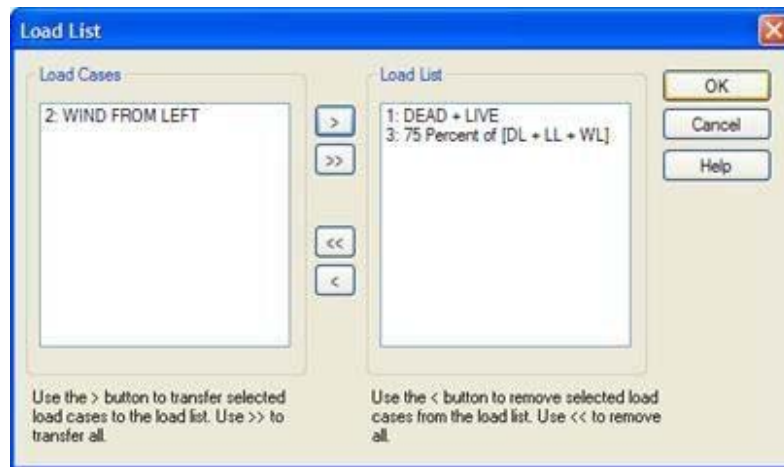
2. From the list box load cases on the left, double-click **1: DEAD + LIVE** and **3: 75 Percent of [DL + LL + WL]** to send to the list box on the right load, as shown below. Click on OK.

1.5.14 Specifying design parameters steel

The specifications listed in Section 1.2 of this tutorial require us to provide values for some of the terms used in the design of steel because the default values of these terms are not adequate. The corresponding commands are generated:

```
PARAMETER
CODE AISC
5760 ALL Fyld
MEMB UNT 2 3 10.0
UNB 10.0 MEMB 23
TRACK 2 MEMB March 2
SELECT MEMB March 2
```

Procedure:

1. To specify the design parameters of steel, go to **Design | Steel** from the left side of the screen. Ensure that selections **Current code** in the upper right side is **AISC ASD** (American Institute of Steel Construction- Allowables Stress Design)

Figure 2-63:
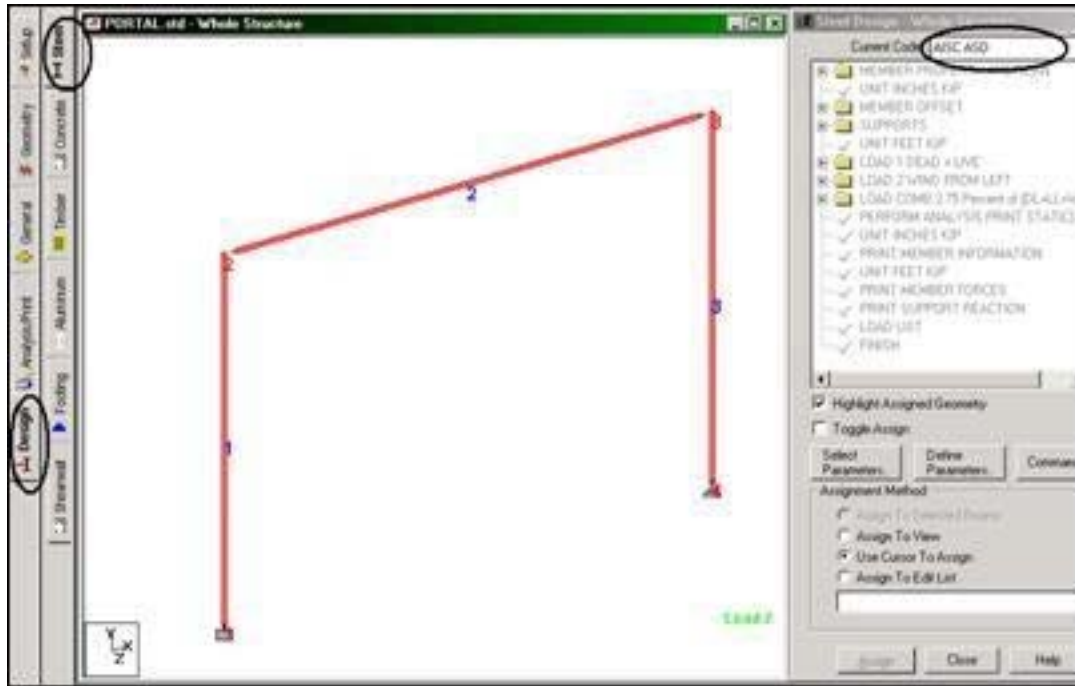
2. Click on **Define Parameters** in dialogue *Steel Design*.

Figure 2-64:
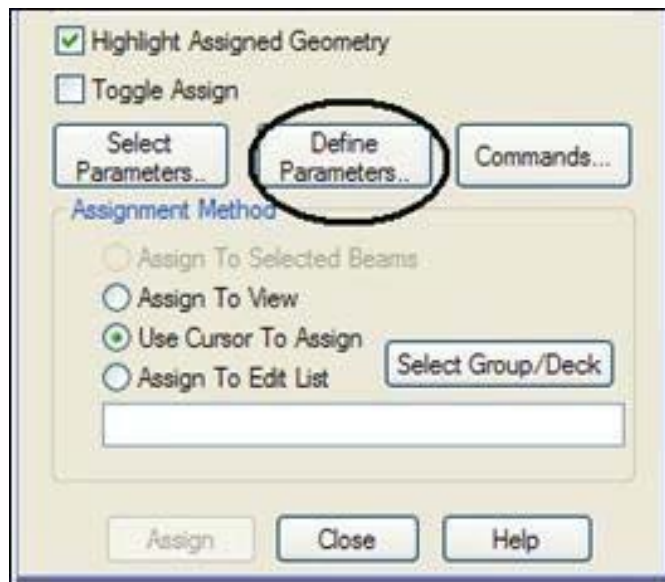
3. In the dialog box that appears design parameters, select the parameter

Fyld. Then, provided the **Yield strength of steel** (Yield strength steel) and 5760 Kip / ft2 and click **add**.

Figure 2-65:
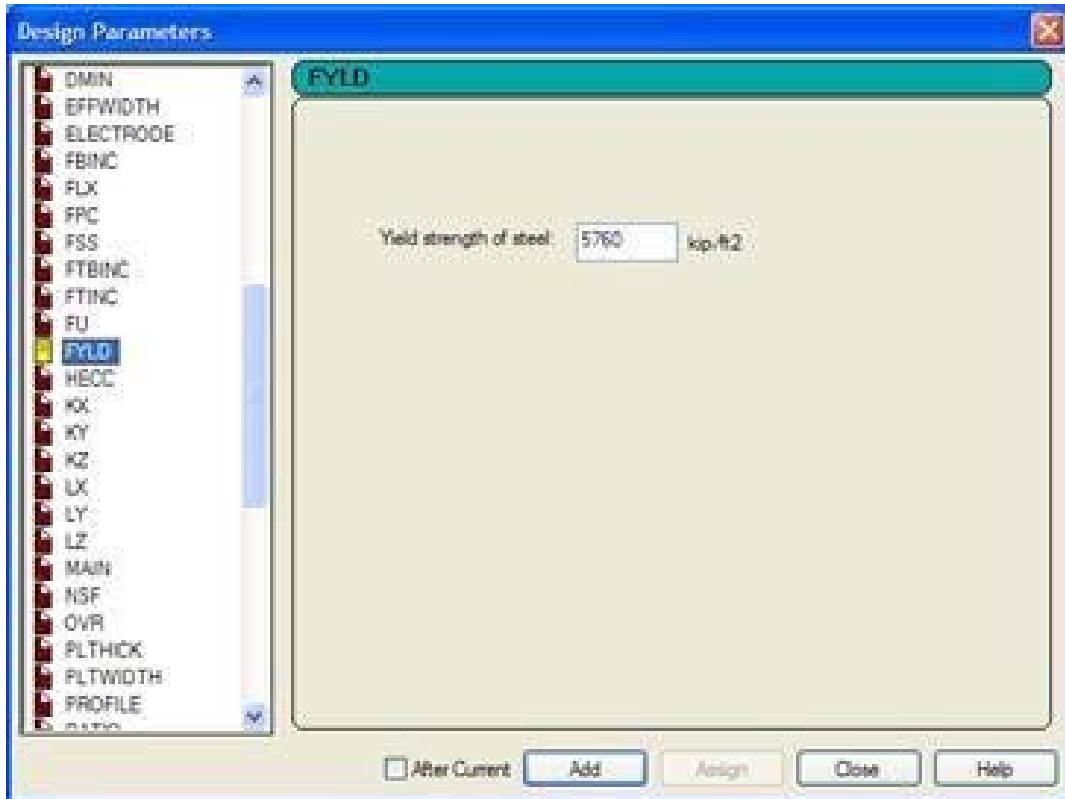
4. To define the remaining parameters, repeat step 3 except to select the parameters and provide the values listed below.

Parameter Value

UNT	1
UNB	1
TRACK	2

5. When you have added all the parameters, click the close button in the dialog box *design parameters*.

6. The next step is to assign those parameters for specific members of the model. Looking the requirements listed in the beginning of this tutorial, we know that the parameter Fyld should be assigned to all members, while the remaining parameters must be allocated to the members 2 and 3.

    As before, using the method **Use Cursor to Assign** to assign these parameters.

Figure 2-66:
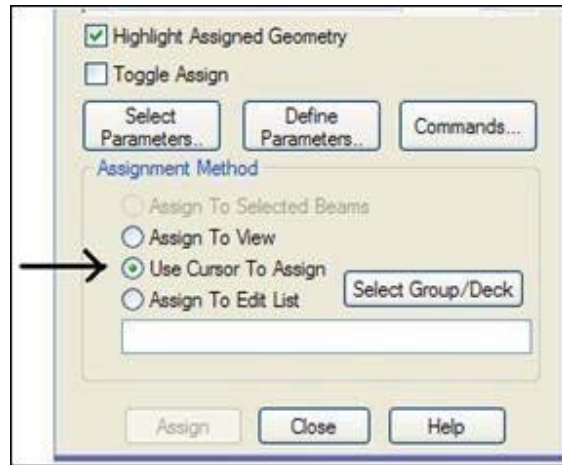
After all the design parameters have been assigned, the dialog *Steel Design* will be as shown below.

Figure 2-67:
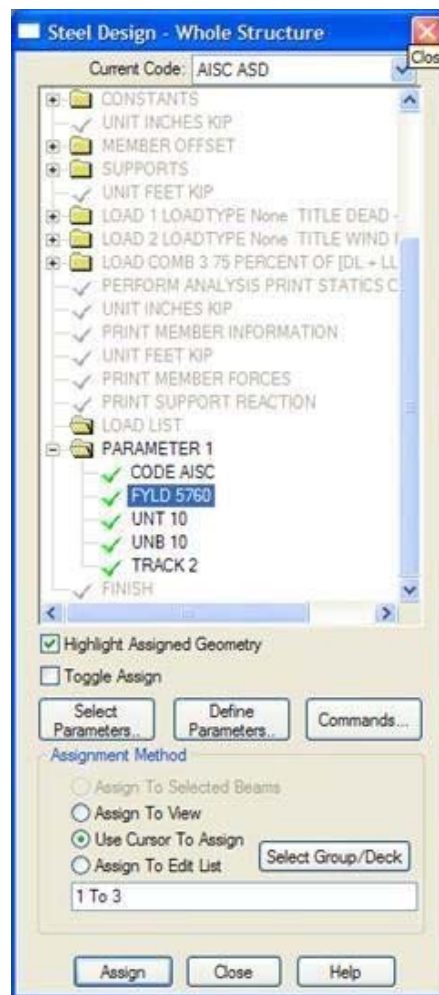
7. To specify the SELECT command, click **Commands** in the dialog box Design-steel as shown below. The SELECT command is an instruction to the program to get and set the cross section of less weight that meets all code requirements (AISC ASD) for the member.

Figure 2-68:

8. In the box *Dialog command* design that appears, click on the tab **select**. Then click on **add**, Followed by the button **shut down**.

Figure 2-69:
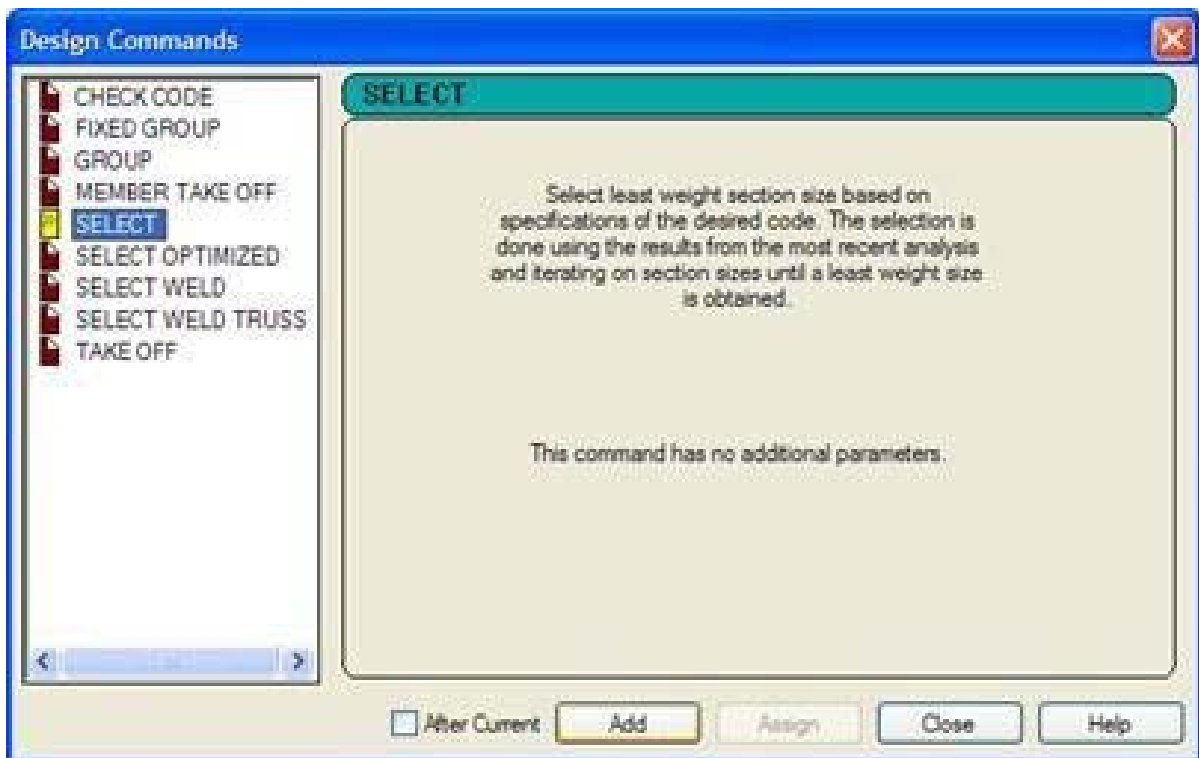
9. Again, this command must be associated with members of both 2 and 3 can be *use Cursor method to assign* or first select members 2 and 3 and then use the option **Assign to Selected Beams.**

After the parameters are assigned, click anywhere in the drawing area to desrealzar members.

Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

1.5.15 Re-specifying the command Analysis

When the analysis and design engine executes the member selection operation we specified in the previous step, a new set of properties will end up being allocated to the members. This has the effect of changing the distribution of stiffness of the structure whole. Since the structure is *statically indeterminate*We must go back if we analyze the nodal displacements, member forces, etc. to reflect the new distribution of stiffness. Therefore, it is the command to generate:

PERFORM ANALYSIS

Procedure:

1. To specify the analysis command, repeat step 1 of section 1.5.11 of this tutorial. In the dialog analysis and printing command that appears, select the tab **Perform Analysis**. Since we are not interested in a review report static

   once again setting option **No print**. Finally, click **add**, Followed by the button **shut down**.

   Tip: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S**.

1.5.16 Re-specifying parameter TRACK

The final calculation we have to do is make sure that the current set of member properties meet the requirements of codes by means of the forces in the members updated. This will require us to do a code check the operation again. To restrict the output produced at a reasonable level, the track parameter (TRACK) again is specified as:

```
TRACK 1 ALL
```

Procedure:

1. To define and assign 1.0 to the track parameter, repeat steps 1-4 of section 1.5.14 of this tutorial.

2. Then select all the members by clicking and dragging a window around them using the mouse. (See 'Tasks performed frequently' of this manual to learn more about selecting members section). Then assign this parameter to all members.

1.5.17 Specifying the CHECK CODE command

The scan operation carried out in response to the command in Section 1.5.15 will create a new set of forces members. It is likely that these forces are very different from those used in the operation of selecting members (see commands section 1.5.14). Consequently, we have to verify that the structure is able - from the point of view of the requirements of the design code - to bring these new forces. A trial run code that uses the updated sections of the members and the last forces of members, we will provide a report on this topic. Therefore, it is the command to generate:

```
CHECK CODE ALL
```

Procedure:
1. Select **Commands> Design> Steel Design**.

2. Click **Commands** in dialogue *Steel Design* dialog as shown below.

Figure 2-70:
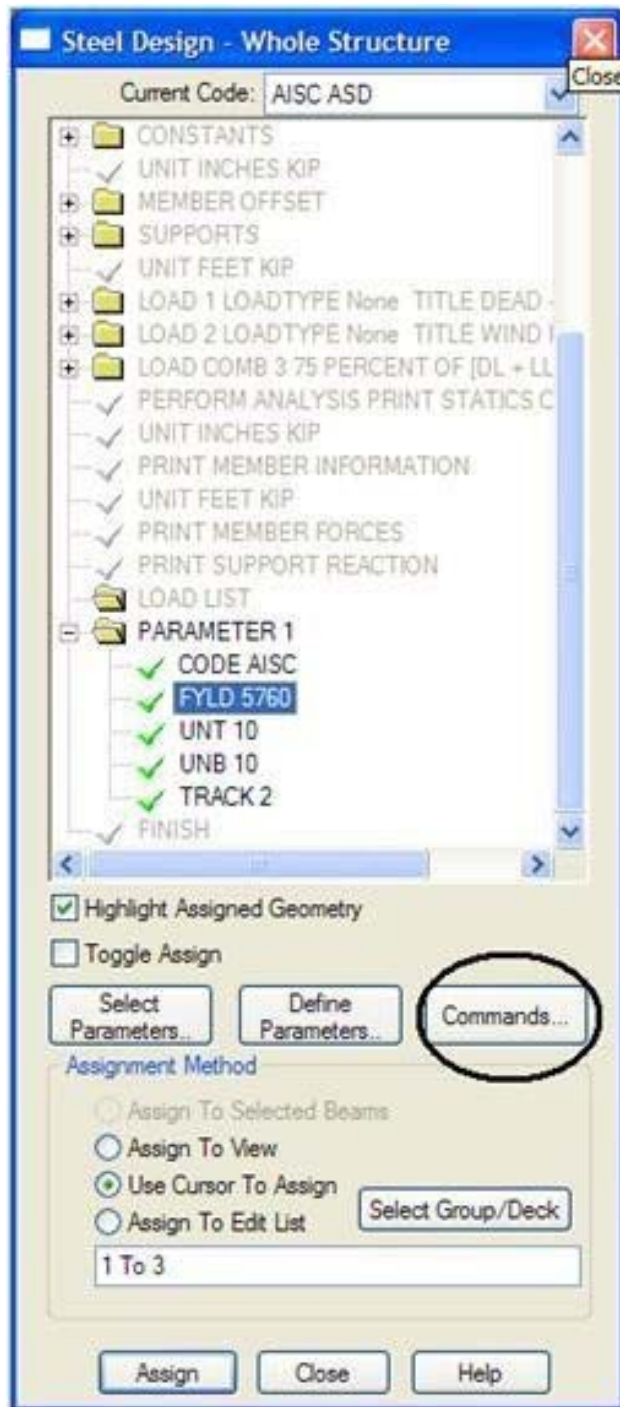
3. In the dialog box layout commands that appears, click the CHECK CODE (Check Code) tab. Then click on **add**, Followed by the button **shut down**.

Figure 2-71:
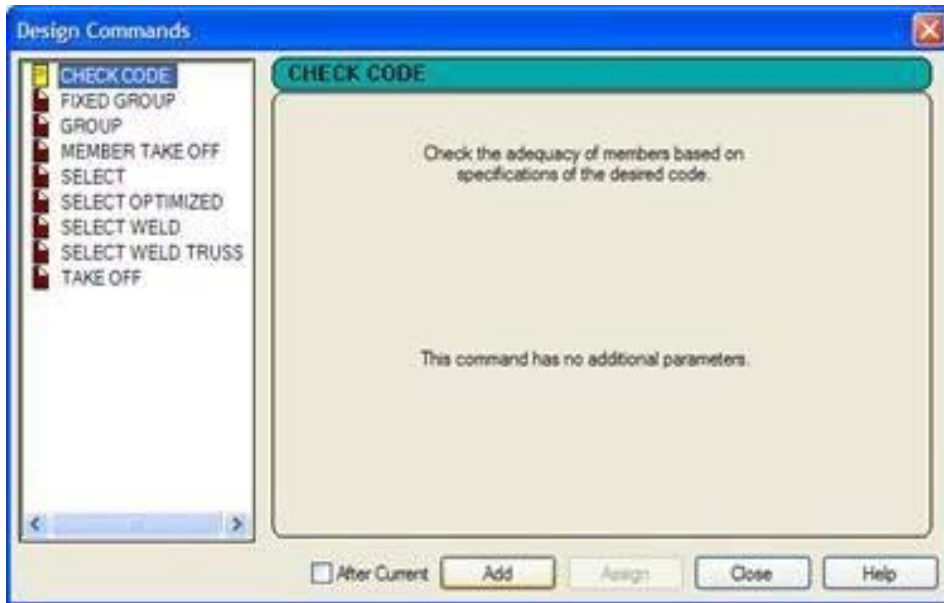
4. Since the verify command code must be assigned to all members, the easiest way to do is to click on map to view - **Assign to View**-.

Figure 2-72:
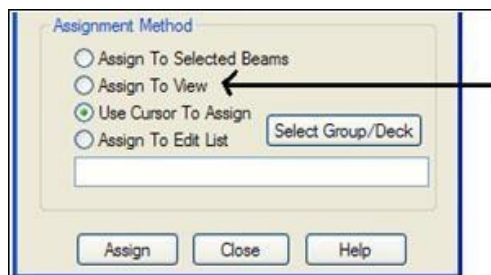
We have now completed the tasks for the assignment of the input for this model.

> **Tip**: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

1.6 Viewing the input command

Procedure:

Now take a look at the data that has been written to the file you just saved before. The contents of the file can be viewed by clicking on the tool **STAAD Editor** or, going to the Edit menu and selecting Edit command file input as shown below.

Figure 2-73:

Figure 2-74:

A new window with the data listed will open as shown here:

Figure 2-75:
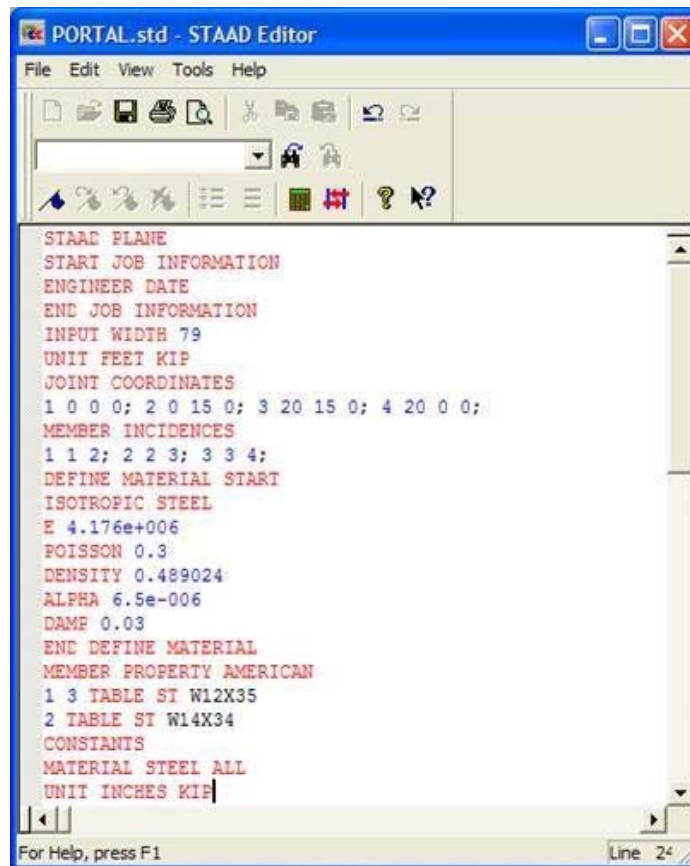
This window and the commands they contain are known as STAAD Editor.

We may make changes to the data of our structure in this *Editor* if we want to. To exit the Editor selects **File> Exit option** Menu Editor window of the output (not the file> exit the menu from the main window behind the editor window).

As we saw in Section 1.1, we could have created the same model also writing relevant STAAD commands in a text file using the STAAD editor, or any external editor of your choice. To understand this method, proceed to the next section. If you want to skip that part, proceed to Section 1.8 where we perform the analysis and design of this model.

1.7 Creating the model using the command-file

We now use the method of command-file to create the model for the above structure. The STAAD.Pro command file can be created using the built-in editor, the procedure explained below in this section. Any text editor such as Notepad or WordPad can also be used to create the command file. However, the Editor STAAD.Pro command file syntax has the advantage when writing the commands. The keywords, numeric data, etc. are shown in different colors in the STAAD.Pro Editor. Below is a typical screen editor is shown to illustrate the general appearance.

Figure 2-76:
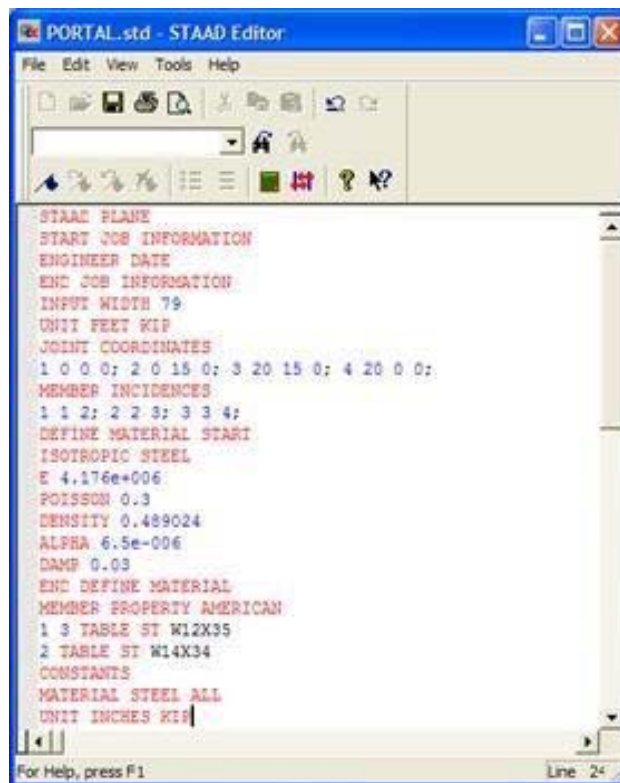
To access the built-in editor, first start the program using the procedure explained in section 1.3. Then follow step 1 of section 1.4.

Figure 2-77:
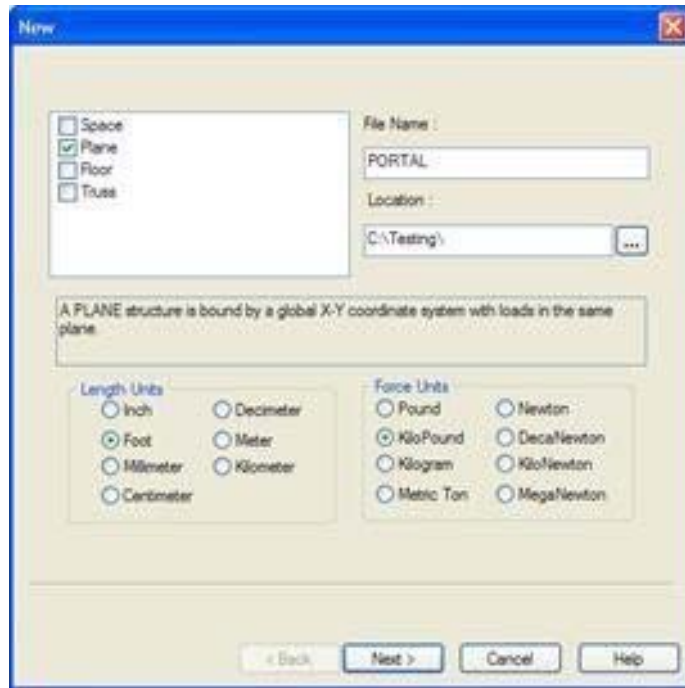
Then you will find the dialog box shown below. In this dialog box, set the check box **Open STAAD Editor** (Open STAAD Editor).

Figure 2-78:
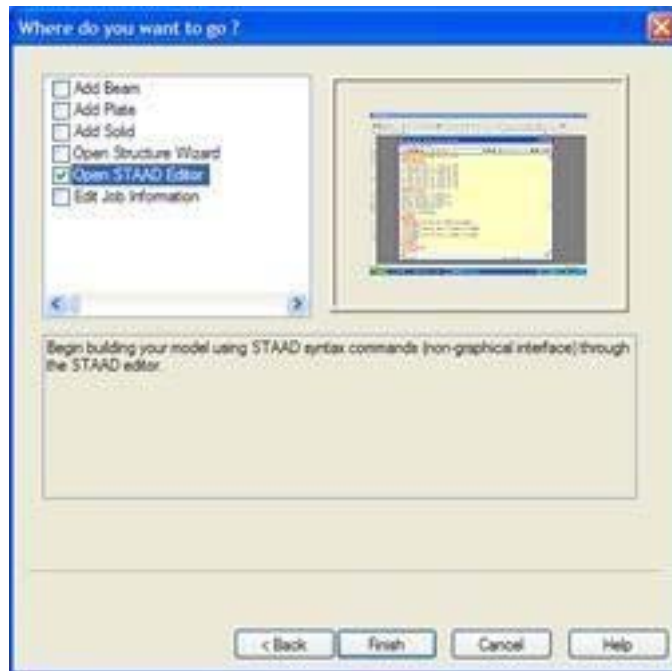
At this point, the editor screen like the one shown below is displayed.

Figure 2-79:
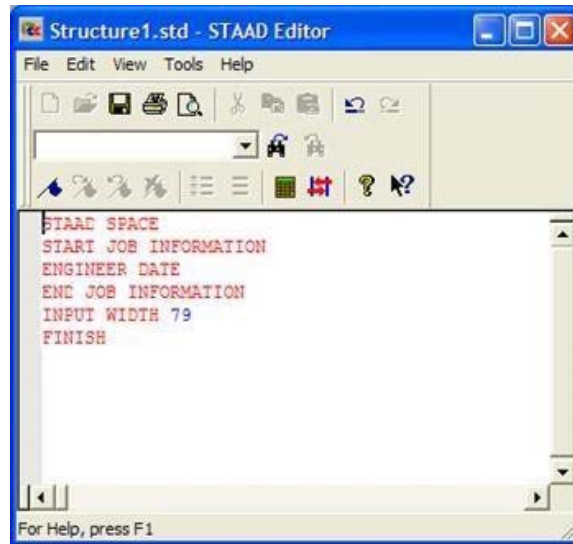
Deletes all lines of command in the editor window and the lines shown in bold below (do not have to delete the lines if you know what to keep and where to fill in the rest of the commands). The commands can be written in lowercase or upper letters. Usually all that is needed are the first three letters of a keyword, the remaining letters of the word are not necessary. The letters are underlined necessary. ("PLANE" = "PLA" = "plane" = "pla")

    STAAD PLANE PORTAL FRAME

Each STAAD.Pro input file must start with the word **STAAD**. Word **PLANE** means that the structure is a flat frame (in the XY plane). The rest of the words is the title of the problem, which is optional.

If you write a line with an asterisk in the first column means that the line is a comment line and should not be executed. For example, one could have put the optional title above a separate line as follows.

    *   PORTAL   FRAME
    UNIT FEET KIP

Specify the units of force and length commands to follow.

    JOINT COORDINATES
    1 0 0; 2 0 15; 3 20 15; 4 20 0.

...

Numbers boards and their corresponding global coordinate X and Y are provided above. For example, March 20, 15 indicates that node 3 has a 20 ft X coordinate and a Y coordinate of 15 feet. Note that the reason for not providing the Z coordinate is because the structure is a plane frame. If this were a spatial structure, the Z coordinate would also be necessary The semicolon (;) are used as line separators. In other words, they can put data normally put in multiple lines in a single line by separating them with a semicolon.

```
MEMBER INCIdence
1  1  2; 2   2  3; 3   3  4
```

The members are defined by the joints to which they are connected.

```
MEMBER PROAME PERTYRICAN
1  3  TABLE  ST  W12X35
2  TABLE ST   W14X34
```

The members 1 and 3 are assigned one table section W12X35 built American steel. The Member 2 has been assigned a W14X34. Word ST is defined as the standard single section. Sections 5.20.1 to 5.20.5 of the STAAD.Pro V8i Technical Manual
explain the Convention for naming the member properties.

```
UNIT
INWITH
CHESStant
s
E29000.0 ALL
POISSON 0.3
ALL
```

The unit of length is changed from feet to inches to facilitate the entry of modulus of elasticity (E). CONSTANTS keyword is needed before material properties that provide such as E, density, coefficient of Poisson's ratio, thermal expansion (ALPHA) etc. See Section 5.26 of the Technical Manual for details.

```
MEMBER OFFSET
2 START 6.0 0 0.
2 END -6.0 0 0.
```

The beam member is physically connected to the two columns on the face of the column and not on the center line of the column. This creates a rigid zone, about half of the depth of the columns, in the two ends of the beam 2 This rigid zone is exploited by

compensation to a member (it's your choice if you want to use these). Thus, the above commands define the member 2 is connected eccentrically or offset its initial meeting by 6 inches in the global X direction, 0.0 and 0.0 in Y and Z. The same member is offset by the negative 6.0 inches its final meeting. See section 5.25 of STAAD.Pro V8i Technical Manual for details.

```
PRINT MEMBER INFOrmation ALL
```

The above command is self-explanatory. The printed information includes first and last numbers together (incidence), length of membership, member releases beta angle and end.

```
SUPPORTS
1  FIXED; 4   PINNED
```

A flush mount is in the ring 1 and an articulated support (fixed for translations released for rotations) on the board 4 More information on the specification of support is available in Section 5.27 of the STAAD Technical Manual.

```
UNIT FT
```

The unit of length is changed to the feet to facilitate the entry of charges.

```
LOADING  1  DEAD  +  LIVE
MEMBER LOAD
2  UNI GY -2.5
```

The previous commands identify a load condition. DEAD + LIVE is an optional title to identify this load case. A uniformly distributed load in the Member 2.5 kips / ft is acting in the member 2 in the overall direction Y. The specification of negative charge members explained in section 5.31 of the STAAD Technical Manual.

```
LOADING WIND FROM LEFT
JOI 2NT LOAD
2  FX  In October.
```

The previous commands identified a second case load. This load is a load on the joint. A force of 10 kip is acting on node 2 in the overall direction X.

```
LOAD COMBination March 75 PERCENT OF (DL +
LL + WL) 1  0.752      0.75
```

This command identifies a load combination with an optional title. The second line gives the components of the load combination box - primary load cases and factors which must be multiplied individually.

```
PERFORM ANALStatics Ysis CHECK PRINT
```

This command instructs the program to continue the analysis and report verification of static equilibrium. Section 5.34 of the STAAD Technical Manual provides information on the various options available analysis.

```
PRINT MEMBER FORCES ALL
PRISUP NTPORT REACTION LIST April 1
```

The above print commands are self-explanatory. The forces in the members are in the local axes while the reactions of the supports in the global axes.

```
March 1 PAR
LOAD
LISTAMETERS
CODE AISC
UNT 10.0 MEMB March 2
UNB 10.0 MEMB
March 2 Fyld 5760
ALL TRACK 2.0 MEMB
March 2
SELECT MEMBER March 2
```

The script is used to start the design process of steel. Command parameters are followed by the different design parameters of steel. The parameters are normally specified when their values differ from the defects of the software supplied. AISC ASD code specifications must be followed. A list of parameters for the AISC code is available in Table 2.1 on page 80 of the Technical Manual STAAD.Pro V8i.

All members have 10 feet of unsupported length of the upper and lower flange (UNT and UNB). UNB UNT and used to calculate the allowable compressive stress in bending. The yield strength of steel is specified as 5.760 MPa (40 ksi), and which is different from the default value of 36 ksi. The TRACK parameter controls the level of description of the output, 2.0 is the most detailed. The SELECT command lists MEMBER load cases (1 and 3) to be used in the design. The member selection command calls the program to get the cheapest part for members 2 and 3 in the context of the above analysis

```
PERFORM ANALYSIS
```

When the analysis and design engine executes the operation selection of members was

specified in the previous step, a new set of properties will be allocated to the members. This has the effect of changing the distribution of stiffness of the entire structure. Since the structure is statically indeterminate, we must go back if we analyze the nodal displacements, member forces, etc. to reflect the new distribution of stiffness. The above command instructs the program to do another cycle of analysis.

```
1 ALL TRACK
PARAMETER
```

The TRACK parameter is re-specified. Controls the level of information produced in the output of steel design. We have dropped from 2.0 to 1.0 we specified before, since we are not interested in the maximum level of detail at this time.

```
CHECK CODE ALL
```

The analysis performed above operation will create a new set of forces in the members. These forces are likely to be very different from those used in the operation of selecting members. Consequently, we have to verify that the structure is safe - from the point of view of the requirements of the design code - able to bring these new forces. A trial run code that uses the updated sections of the members and the last forces of members, we will provide a report on this topic.

```
FINISH
```

The STAAD executor is terminated using the FINISH command. Save the file and return to the main screen.

This concludes the session in the generation of our model as a script using the built-in editor. To perform the analysis and design, you can proceed to the next section of this manual. The facilities at the screen post-processing are explained in Section 1.10.

Caveat: Remember that without completing the analysis and design of post-processing facilities are not accessible.

1.8 Performing the Analysis / Design

STAAD.Pro performs analysis and design simultaneously.

Tip: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S.**

In order to perform analysis and design, select **Analyse> Run Analysis ...**.

As the analysis progresses, several messages appear on the screen as shown in the following figure.

Figure 2-80:
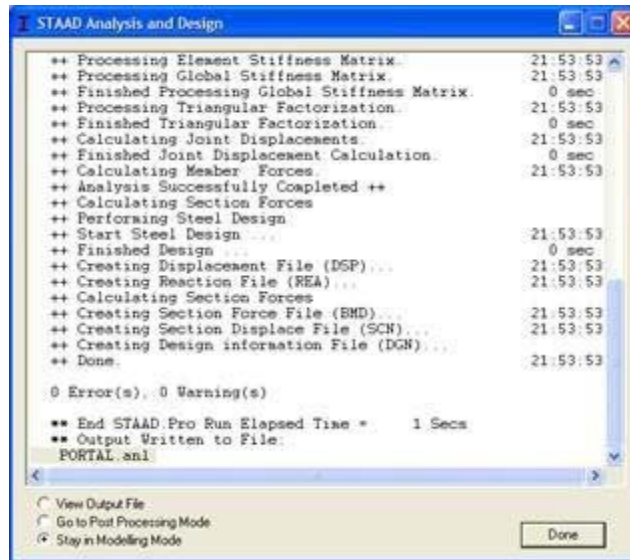
Note that we can choose from the three options available in the dialog box above:

Figure 2-81:
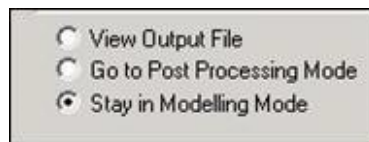
These options are indicative of what will happen after you click **Done**.

Option *View Output File (*see output file) allows us to see the output file created by STAAD. The output file contains the numerical results produced in response to different input commands you specified during the generation process model. It also tells us if errors were found and if so, whether the analysis and design was completed successfully or not. Section 1.9 provides additional details on viewing and understanding the contents of the output file.

Option *Go to Post Processing Mode (*go to the post-processing mode) allows us to go to the graphics part of the program known as the post-processor. This is where one can exhaustively verify the results, see the results graphically, draw diagrams of results, produce reports, etc. Section 1.10 explains the post-processing mode in more detail.

Option *Stay in Modelling Mode (*Staying in modeling mode) allows us to remain at the generation mode of the program model (in which it is currently) in case we want to make more changes in our model.

1.9 Viewing the output file

During the analysis process, STAAD.Pro creates an output file. This file contains important information about whether the analysis was successful. For example, if STAAD.Pro is a problem of instability during the analysis process will be reported in the output file.

We can access the output file using the method explained at the end of the previous section. Alternatively, you can select the option **File> View> Output File> STAAD Output** the top menu. The STAAD.Pro output file for the problem that we have

I found is shown in the following pages.

Figure 2-82:
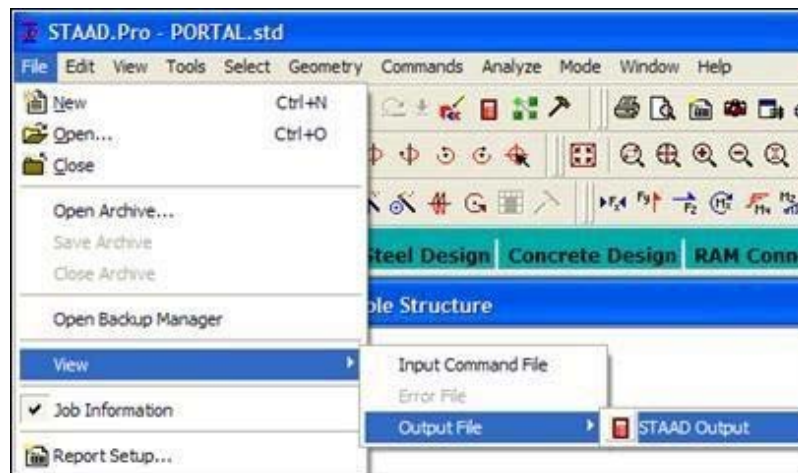
The output file is displayed through a file viewer called *SproView*. This display lets you set the text font for the whole file and print the output file to a printer. Use the appropriate menu option *Archive* from the menu bar.

Figure 2-83:
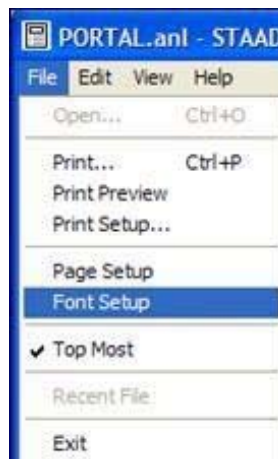
By default, the output file also contains a list of all the input. You can choose not to print the echo of the commands into the output file. Please select **Commands> Miscellaneous> September Echo** from the menu bar and

select the button **Echo Off**.

It is very important that we sail across the output file and ensure that the results seem reasonable, no error messages or warnings reported, etc. Errors found during the analysis & design can disable access to post-processing mode - graphical displays which can display results graphically. The information presented in the output file is a crucial indicator of whether or not the structure meets the requirements of engineering safety and ease of maintenance.

1.10 Post-Processing

STAAD.Pro provides ample facilities for verification and visualization of results. These commands are accessible from the Post-processing mode. The post-processing mode is used to check the results of analysis and design and generate reports.

For this tutorial, we must perform the following tasks:

1.10.1 Go to the post-processing mode

At the end of section 1.8, we saw how one could go directly from the scan window to screen post-processing. However, you can access the post-processing mode using the following procedure at any time.

Procedure:

1. Select the tool **Post Processing** or **Mode> Post Processing.** Opens the Output Settings dialog.
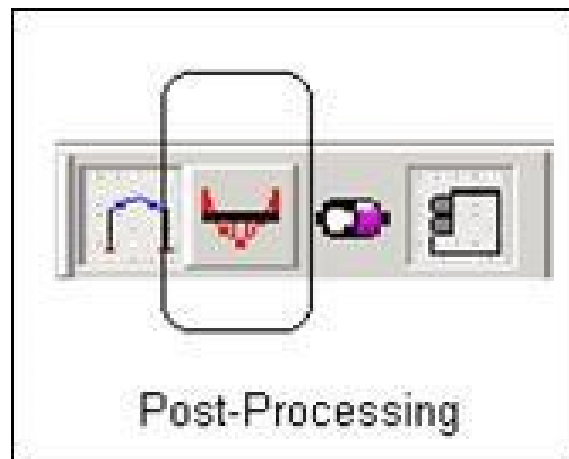
Figure 1.95

Figure 2-84:
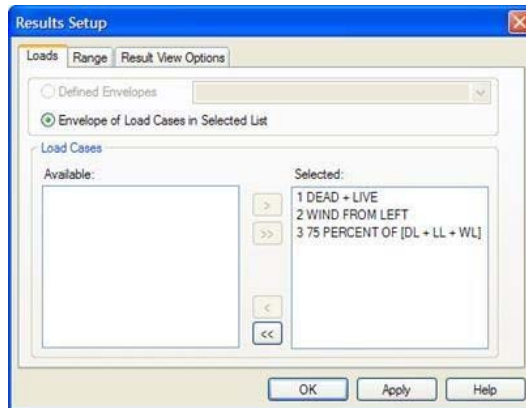
2. Select the load cases for which to display the results. For this tutorial, click [>>] To select all load cases and click accept.

Note that in the post-processing mode, change control bar tabbed page and the menu bar to offer the post processing functions:

Figure 2-85:
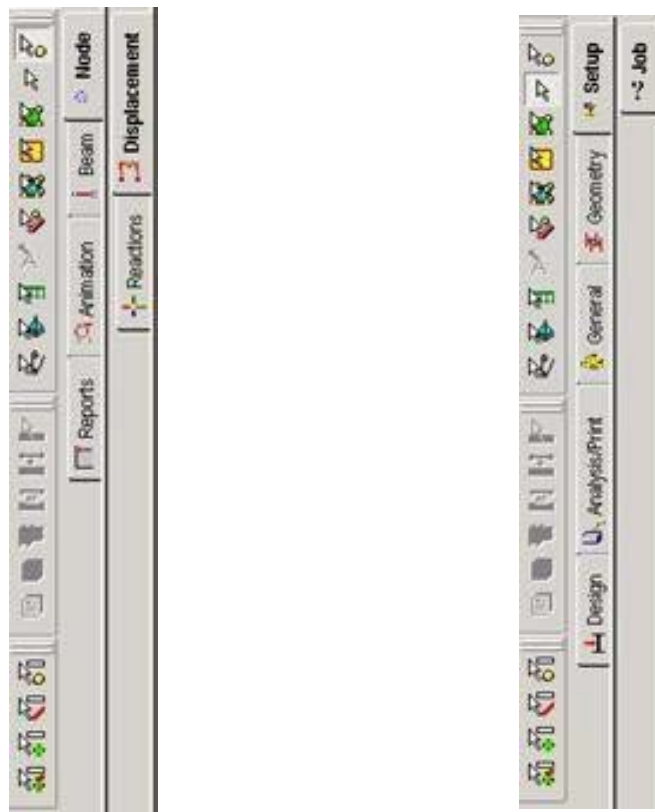
Page Control mode modeling

Control mode Page Post-Processing

Menu Bar mode modeling

Bar menu mode post-processing

1.10.2 Noting the displacement

Procedure:

The screen will now look like the figure shown below.

Figure 2-86:
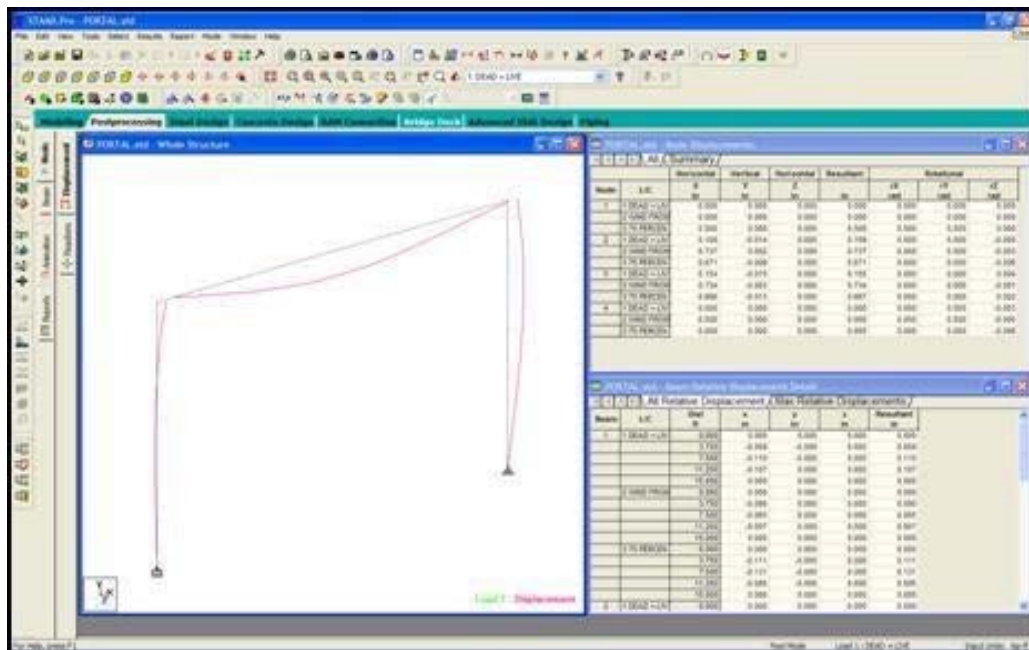
The chart is currently on display nodal deflection diagram or "elastic curve" for the case of load 1 (DEAD + LIVE). The title in the bottom of the diagram is indicative of that aspect. If you, say, he walked away to any other result diagram and wanted to return to the diagram

deviation, just select the tab **Node> Displacement** along the area
Page control on the left side.

Figure 2-87:

Annotation (Annotation) is the process of viewing the offset values on the screen. Select

Results> View Value …

The annotation dialog box opens

Figure 2-88:
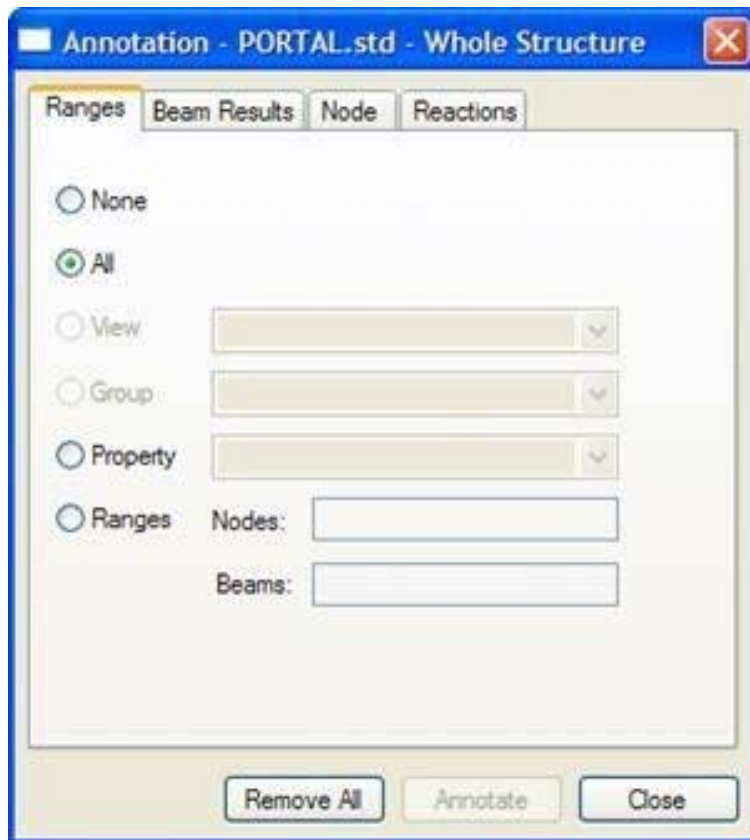
From the Record **Ranges**Select **All** all nodes. If you want to annotate deflection for few nodes, node numbers specified in the node list.

From the Record **Node**Set the checkbox **Resultant**. This represents the square root of the sum of the squares of the values of X, Y and Z displacements. Click the record button and note that the values appear in the structure and then click

shut
down.

Figure 2-89:
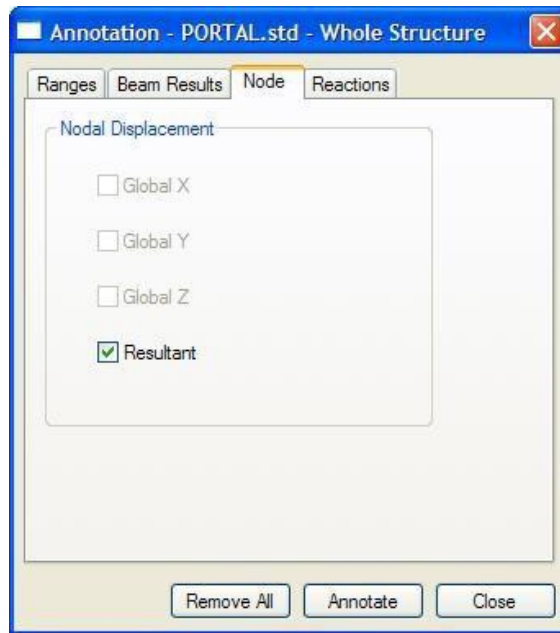
The following figure shows the annotated deflection diagram for load case 1.

Figure 2-90:
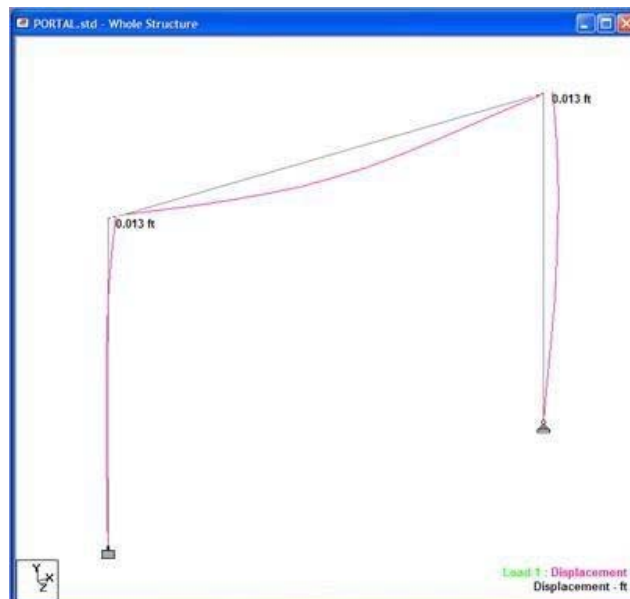
1.10.3 Viewing diagrams of forces / moment

Procedure:

The easiest way to access the facilities to display diagrams of force / moment is from the tab **Beam | Forces** along the page control area on the left side of the screen. The bending moment MZ will be checked by default, makes evident

which may be in the form of icon Mz shown in the diagram below which it becomes active.

Figure 1 106

Figure 2-91:
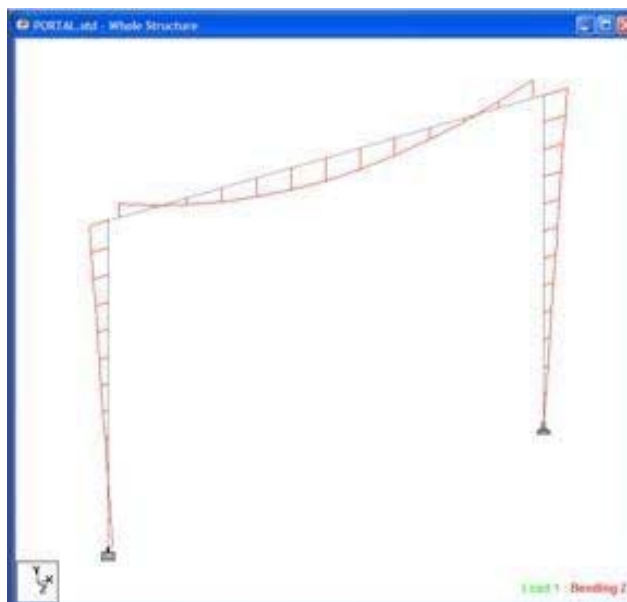
The option to select the scheme forces / time from another facility is available too - menu option **Results> Bending Moment**.

Figure 2-92:

1.10.4 Scoring diagram force / moment

Procedure:

Annotation is the process of displaying the values of force / moment on the screen. Select **Results> View Value ...**. The Annotation dialog.

Select the tab **Ranges** and then **All** all members. If you want to annotate the strength to few members, specify the numbers of the beam in the list of the beam.

Figure 2-93:
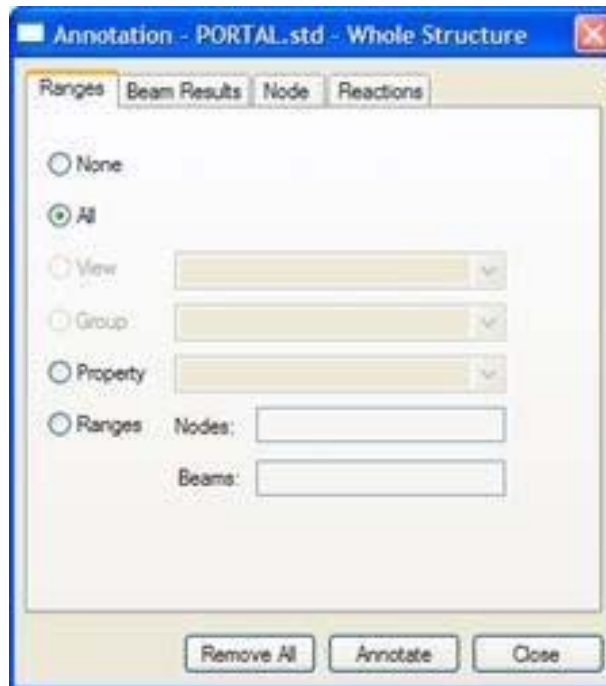
On the record **Beam Results** (Results of beam), set the checkboxes **Ends** and**Mid-point** section **Bending**. Click the record button and note that the values appear in the structure and click **OK**.

Figure 2-94:

The following diagram shows the MZ scored for the case of load 2.

Figure 2-95:
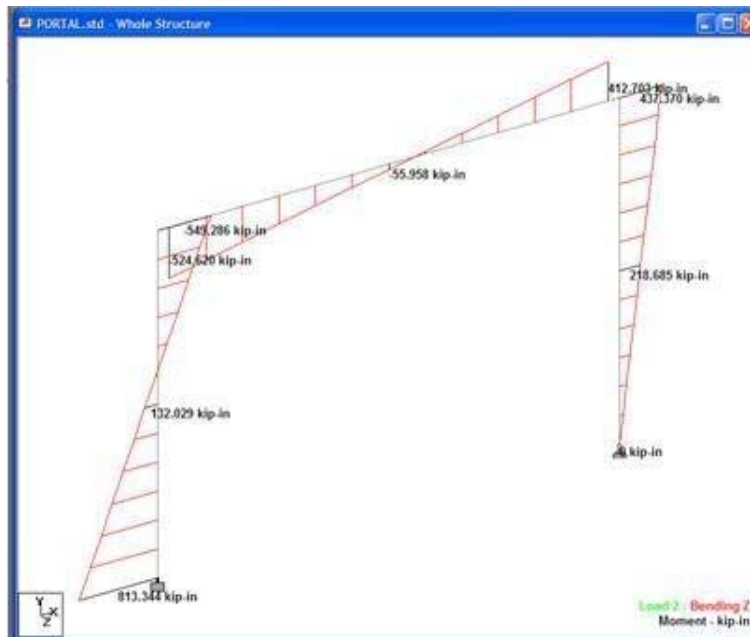
1.10.5 Changing the degree of freedom for the forces diagram trace

They can draw diagrams and moment of 6 degrees of freedom - Axial cut, shear-Y,-Z shear, torque, moment-Y,-Z time.

Select **View> Structure Diagrams> Loads and Results**.

Figure 2-96:
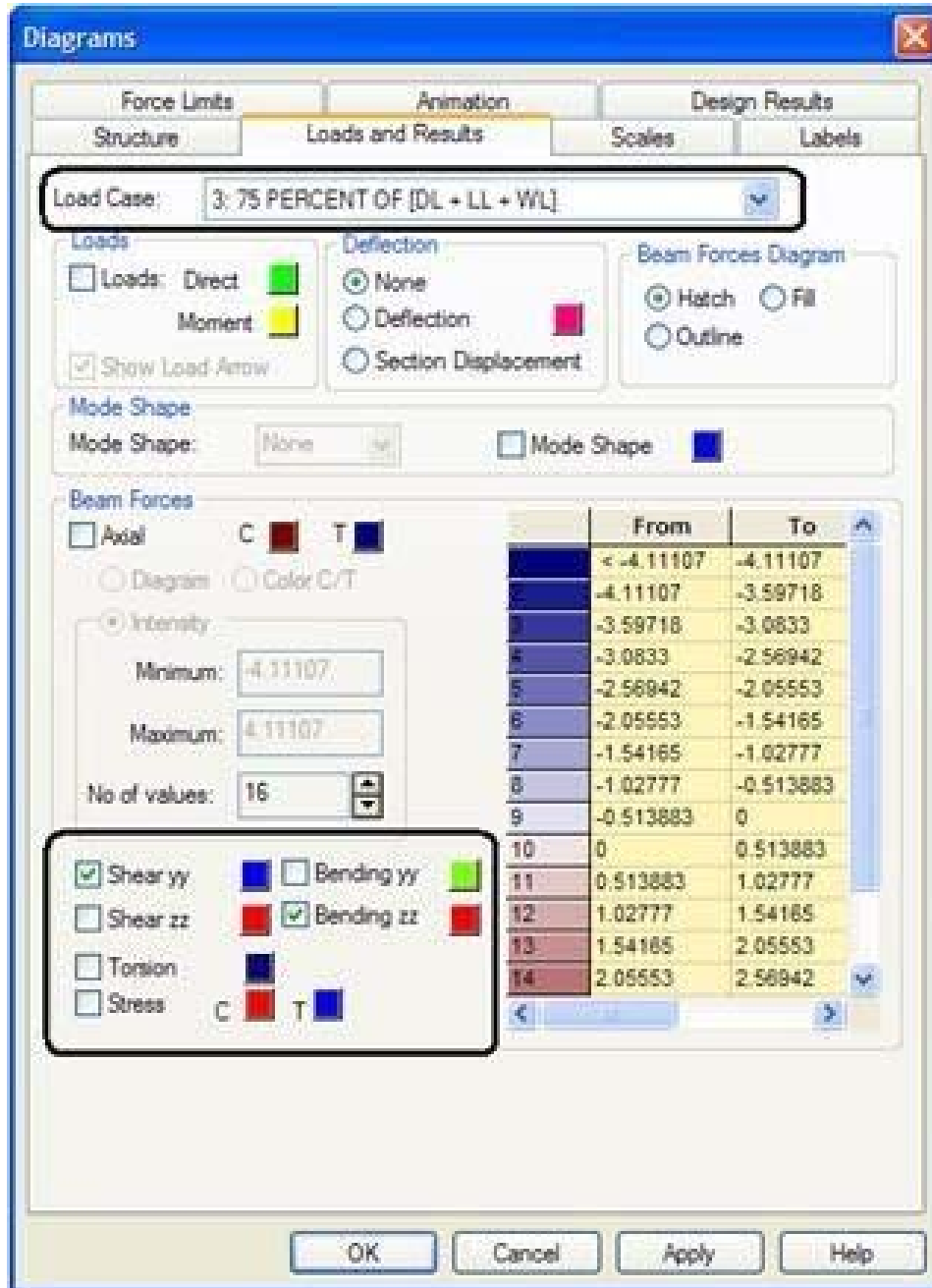
Select Load Case 3 (75 PERCENT OF [DL + LL + WL] and set the checkbox **Shear yy**.

The resulting figure is shown below.

Figure 2-97:
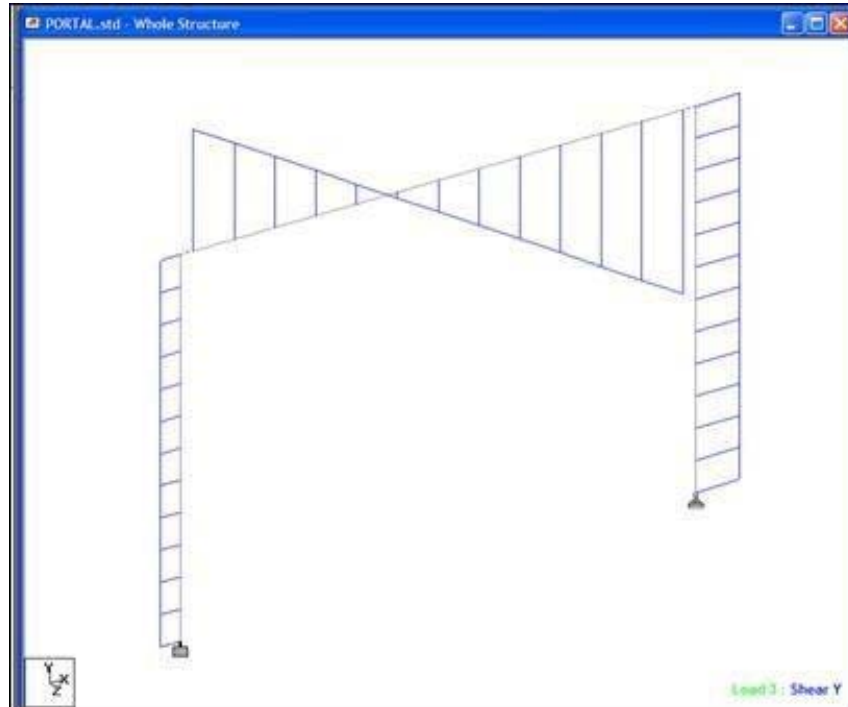
The degrees of freedom currently drawn are noted with a check mark in the dialog box diagrams.

The icons in the bar results can also be used to enable / disable certain degrees of freedom

Figure 2-98:

For the sake of easy identification, each degree of freedom (degree of freedom –dof–) has been assigned a different color (see diagrams above dialog). One can change the color of the "shallow" by clicking on the color button next to the "shallow" and make a new selection of the color palette.

Figure 2-99:
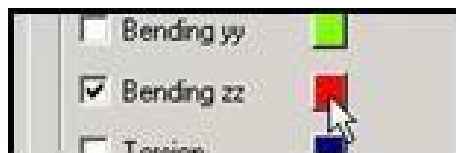
The appearance of the diagram can also be set to one of 3 - Grilles, fill and stroke by selecting the appropriate option in the dialog box shown above.

Figure 2-100:
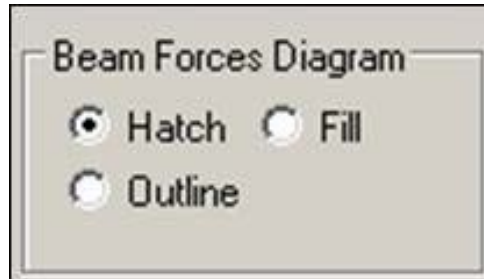
1.10.6 Showing the dimensions of the members

To display the dimension members, select the dimension tool. Alternatively, one can go to the command **Dimension** on the Tools menu. In the dialog box that option dimension to open the view is active. Click **Display** followed by the button **Close**And the dimensions of the members appear next to members.

Figure 2-101:
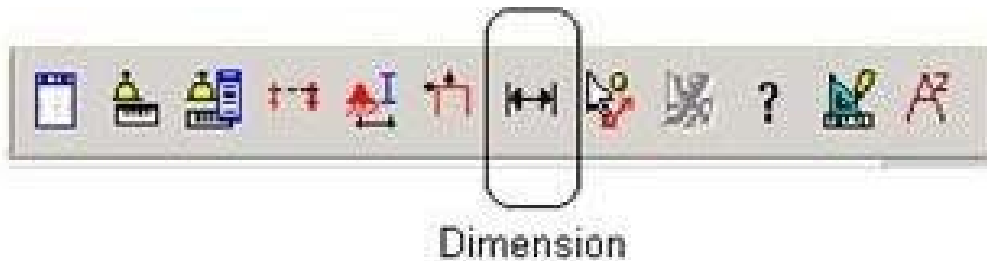
Figure 2-102:
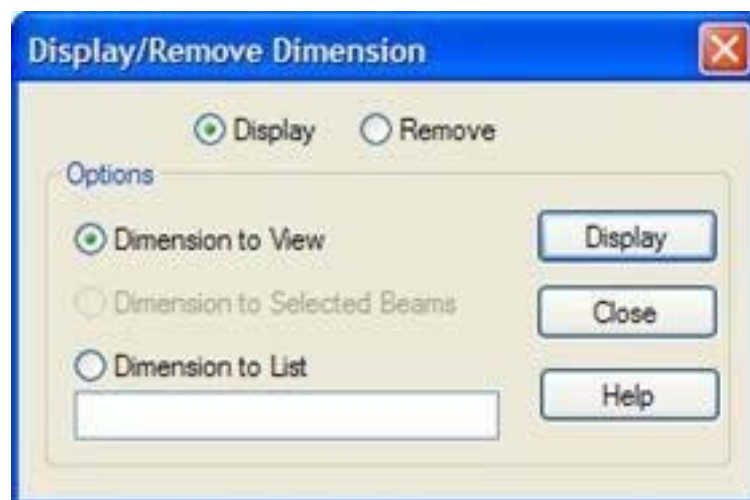
The diagram will be as shown below.

Figure 2-103:
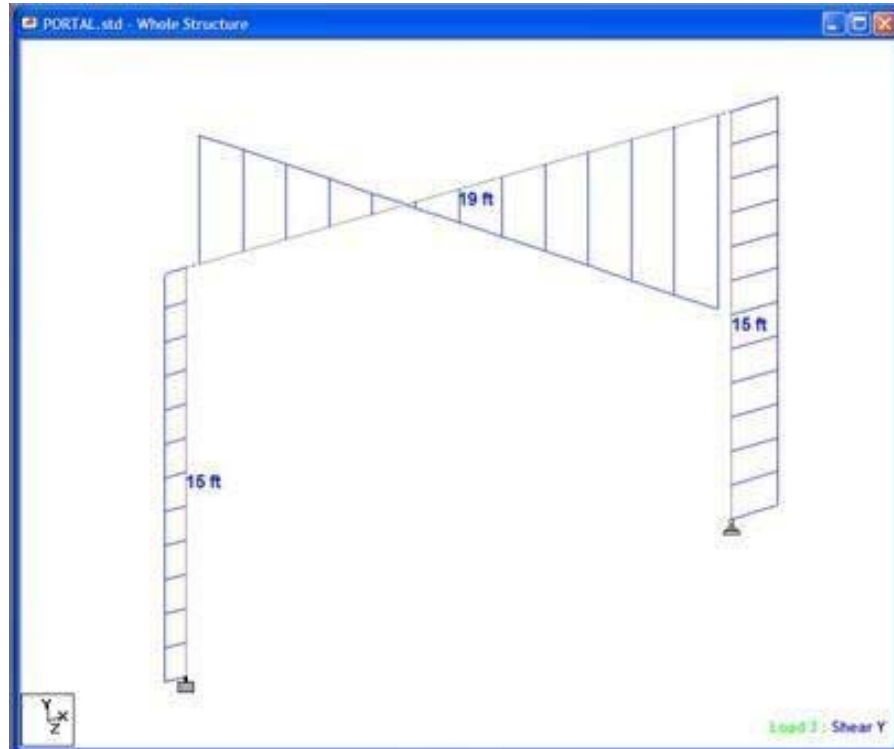
We can choose to save the screenshot by clicking on the tool **Take Picture** (Photo-taking shown below). This table can be included in custom reports.

Figure 2-104:

To get a quick impression of the plot on the screen, select the tool **Print Current View** (print the current view) as shown below.

Figure 2-105:

Tutorial 2 - Setting reinforced concrete

This tutorial provides step by step instructions to create the model of a reinforced concrete framed structure with STAAD.Pro.
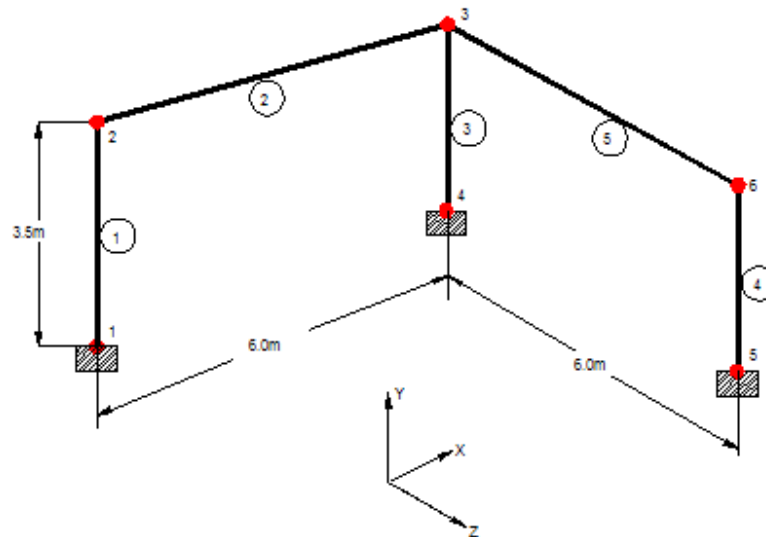
2.1 Model creation methods

As explained in section 1.1 of the tutorial 1, there are two methods for creating the data structure:

1. Using the graphical model generation mode, or graphical user interface (GUI) that typically refers.

2. Using the file command.

Both methods are explained in this tutorial too. The graphical method is explained first, in section 2.2 hereinafter. Section 2.8 describes the process of creating the model using the method of command file and text editor STAAD.Pro

2.2 Problem description tutorial

The structure of this project is a reinforced concrete structure with 2 bays and one level located in two Cartesian planes (XY, YZ), as shown in Fig. Our goal is to create the model, assign everything that requires input and perform the design and analysis of concrete.

Basic information about the structure

Attributes	Data
Member Properties	Beams 2 & 5: Rectangular, 275 mm Width 350 mm Depth Columns 1 & 4: Rectangular, 275 mm wide by 300 mm depth Column 3: Round, diameter 350 mm
Member Orientation	ALL members (except column 4): Default Column 4: rotated 90 degrees relative to the default
Material Constants	Modulus of elasticity: 22 KN / sq.mm Density: 25 kn / cu.m Poisson ratio: 0.17
Supports	Base of all columns: Built
Loads	Load case 1: Dead load with weight of the beams 2 & 5 structure: 400 kg / m in the global Y down load case 2: live load Beams 2 & 5: 600 kg / m in the global Y down load case 3: wind load Beam 1: 300 kg / m along X positive overall Beam 4: 500 kg / m along positive overall charge of X Case 4: DEAD + LIVE L1 + L2 X 1.2 X 1.5 (Use REPEAT LOAD not Load Combination) Load Case 5: DEAD WIND L1 + L2 + X 1.1 X 1.3 (Use REPEAT LOAD not Load Combination)
Type of Analysis	PDELTA
Concrete Design	Consider load cases 4 and 5 only. Parameters: Yield Strength: 415 n / sq.mm Concrete Strength: 25 n / sq.mm clear cover up: 25 mm clear cover to the bottom: 30 mm Cover the light side: 25 mm design of beams 2 and 5 Design of columns 1, 3 and 4

2.3 Starting the program

1. Select the STAAD.Pro icon from the program group STAAD.Pro V8i in the Windows Start menu.

   The STAAD.Pro window opens to the home screen.

   Figure 2-106:

> **Note**: See "1.3 Starting the Program" on page 22 for notes about changing the system default drive.

2.4 Creating a new structure

In the dialog box *New* Offer some initial critical data needed to build the model.

1. Select **File> New** or select **New Project** under *Project Tasks*.

figure 2-107:
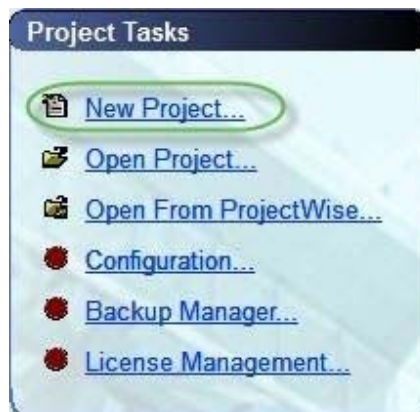
The New dialog
opens:

Figure 2-108:
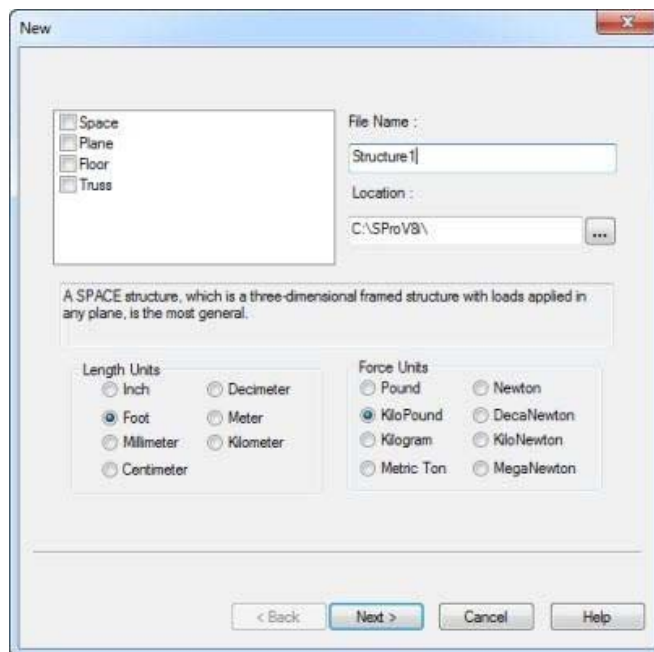
The structure type is defined as Space, **Plane**,**Floor**, Or **Truss**:

Space (Space Structure)

The structure, charge, or both, cause the structure to deform in the three global axes (X, Y and Z).

Plane (Flat Structure)

The geometry of the load and the deformation is limited only to the overall plane XY

Floor
(Floor)

A structure whose geometry is limited to the plane XZ.

Truss (Armor)

The structure carries pure axial load action. The members of the truss are considered incapable of carrying shear, bending and torsion.

2. Select **Space**.

3. Select **Meter** as the unit length and **Kilo Newton** co m u l a f orce nity of.

> **Tip**: The units can be changed later if necessary, at any stage of the creation model.

4. Specify the file name as **RCFrame** and specify a location where the STAAD input file will be located on your computer or network.

   Directly you can type a file path or click [...] to open the search dialog by folder, which is used to select a location using a Windows file tree.

   After specifying the previous entry, click **Next**. The next wizard page, Where do you want to go open.

Figure 2-109:
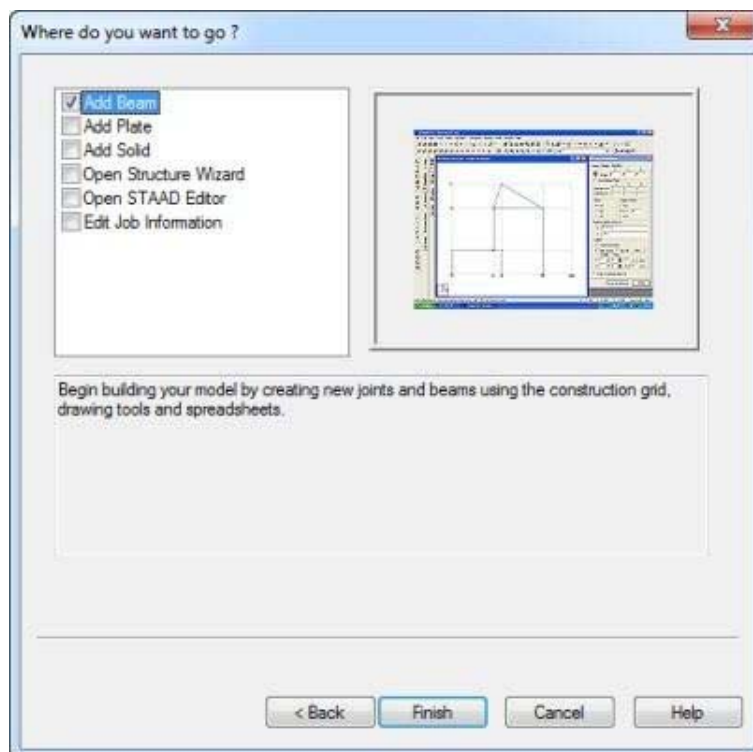
In the dialog, we choose the tools that will be used to initially build the model.

Add Beam, Plate Add, or Add Solid

Respectively, the selected tools are used in the construction of beams, plates or solids when the GUI is opened.

Open Structure Wizard

Provides access to a library of structural templates that comes equipped with the program. Those template models can be extracted and modified parametrically to reach the model geometry or some of its parts.

Open STAAD Editor

Usually created a model using STAAD command language in the STAAD editor.

All these options are also available from the menus and dialog interface, even after we discard this dialog.

> **Note**: To use the Editor to create the model, choose Open STAAD Editor, click **end** and proceed to section 2.8.

5. Select the option to add the beam and click finish. The dialog box will be dismissed and STAAD. Professional GUI displays.

2.5 Elements of the STAAD.Pro screen

STAAD.Pro The main window is the screen where the model generation process is performed. It is important to become familiar with the components of the window before embarking on the creation of the structure of reinforced concrete frame. In section 1.5 the tutorial problem 1 of this manual explains the components of that window in detail.

2.6 Building the STAAD.Pro model

We are now ready to start building the model geometry. The steps and, when possible, the corresponding commands (instructions that are written in the STAAD input file) are described in the following sections.

2.6.1 Generating the model geometry

The geometry of the structure is composed of node numbers, coordinates, member numbers, connectivity information of the member, plate element numbers, etc. From the point of view of the STAAD command file, the commands are generated by the structure shown in section 2.2:

```
JOINT COORDINATES
1 0.0 0.0 0.0; 2 0.0 3.5 0.0
3 6.0 3.5 0.0; 4 6.0 0.0 0.0

5 6.0 0.0 6.0; 6 6.0 3.5 6.0
MEMBER INCIDENCE
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 3 6
```

Procedure

1. Select the option *Add Beam* before to allow us to add beams and columns to create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X, Y and Z) are represented in the icon in the lower left corner of the drawing area.

Figure 2-110:
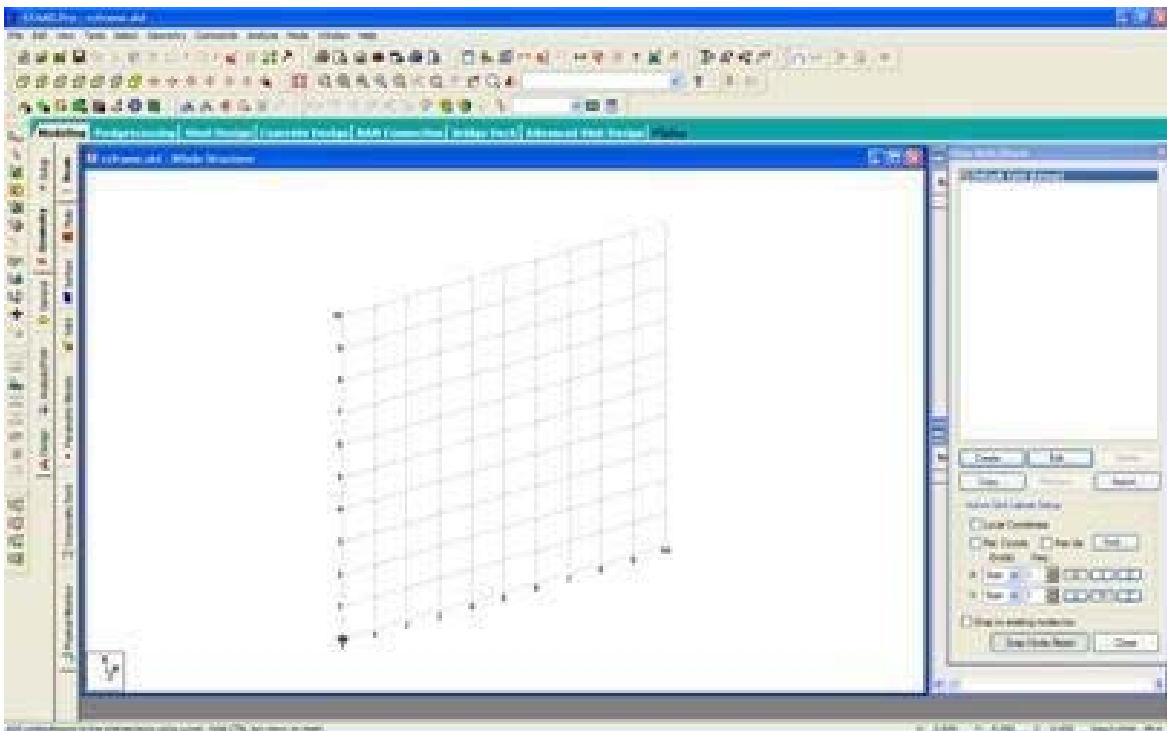
2. Dialogue appears in the data area on the right side of the screen *Snap Node / Beam*. Click **create**. A dialog box allows you to set a grid opens.

Within this dialog box, there is a drop-down list from which you can select Linear, Radial or Irregular shaped grid lines.

Figure 2-111:

Select **linear**, Which is the default network.

In our structure, the segment consists of members from 1-3 and 1-4 nodes happens in the plane **X-Y**. So in this dialog box, keep **X-Y** as the plane of the net. The size of the model that can be removed at any time is controlled by the number of construction lines left and right of the origin of

axes and the spacing between lines of adjacent construction. 12 by setting the number of lines to the right of the origin along **X**,**7**on the origin **And**and space **0.5** m between lines along both **X**and**And**(See figure below) we can draw a marquee 6m **X**3.5 m, suitable for our segment.

After entering the specifications, provide a name and click **OK**.

Figure 2-112:
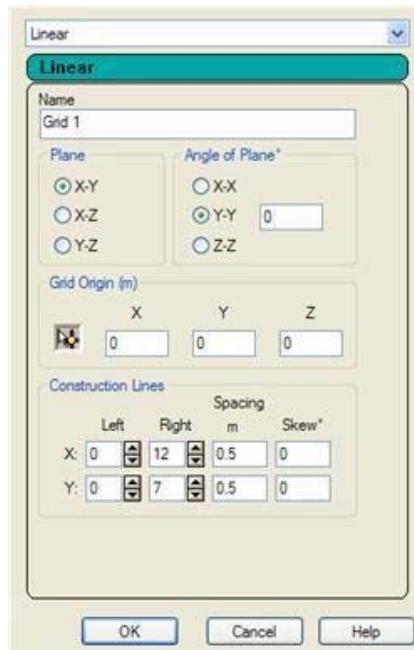
In this way, we can create any number of grids. By providing a name, each new grid can be identified for future reference.

Please note that these values are only starting grid setting, to enable us to draw the structure, and do not restrict our overall model these limits.

3. Let's start creating the nodes. From the Snap Node / Beam button is enabled by default, with the mouse, click the origin (0, 0) to create the first node.

Figure 2-113:
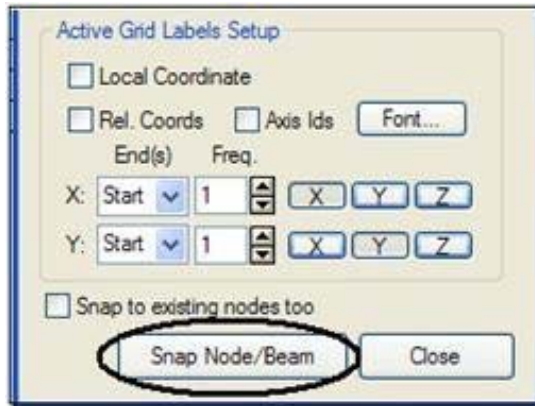
Figure 2-114:
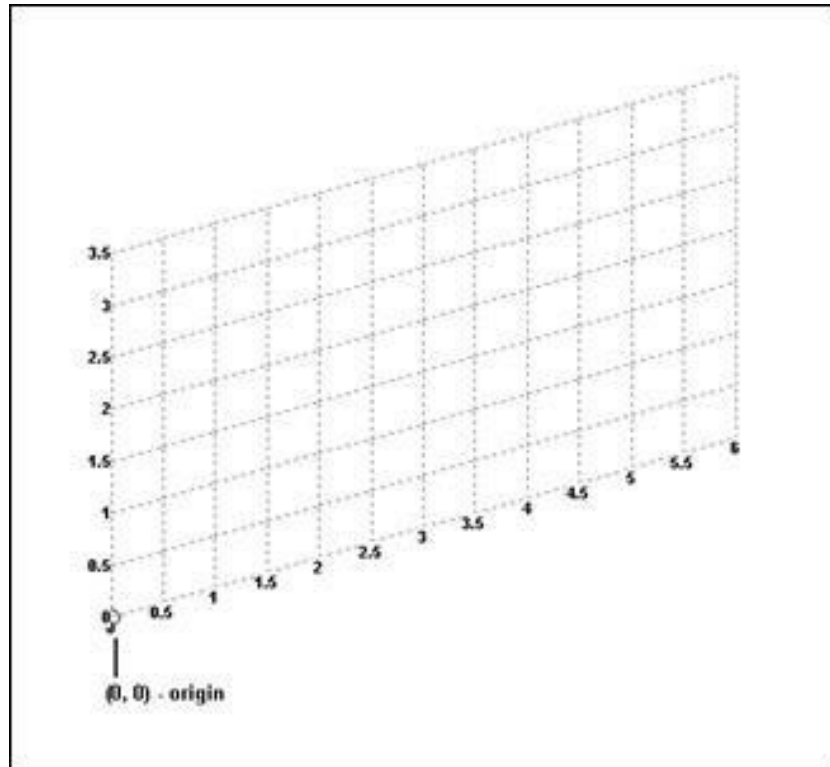
4. Similarly, click the following to create nodes and nodes by successive members of the beam is automatically attached.

(0, 3.5), (6, 3.5) and (6, 0)

The exact location of the mouse can be monitored in the status bar at the bottom of the window that continuously update the coordinates X, Y and Z of the current cursor position.

When you have completed steps 1-4, the frame will be displayed in the drawing area as shown below.

Figure 2-115:
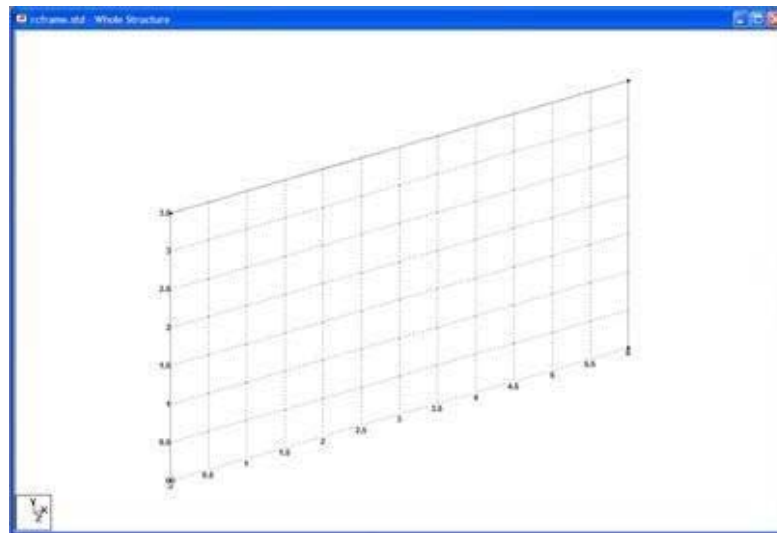
5. At this point, we remove the display of the grid structure. To do this, click Close in the dialog box *Snap Node / Beam*

Figure 2-116:
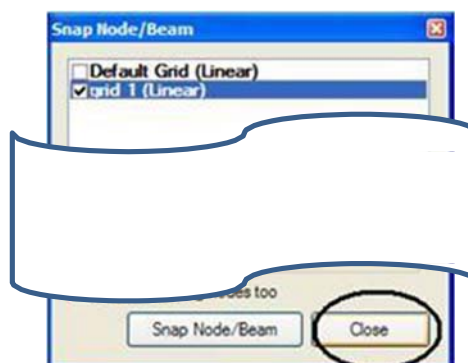
Now the grid and the structure of the main window should look like the figure shown below will be deleted.

Figure 2-117:
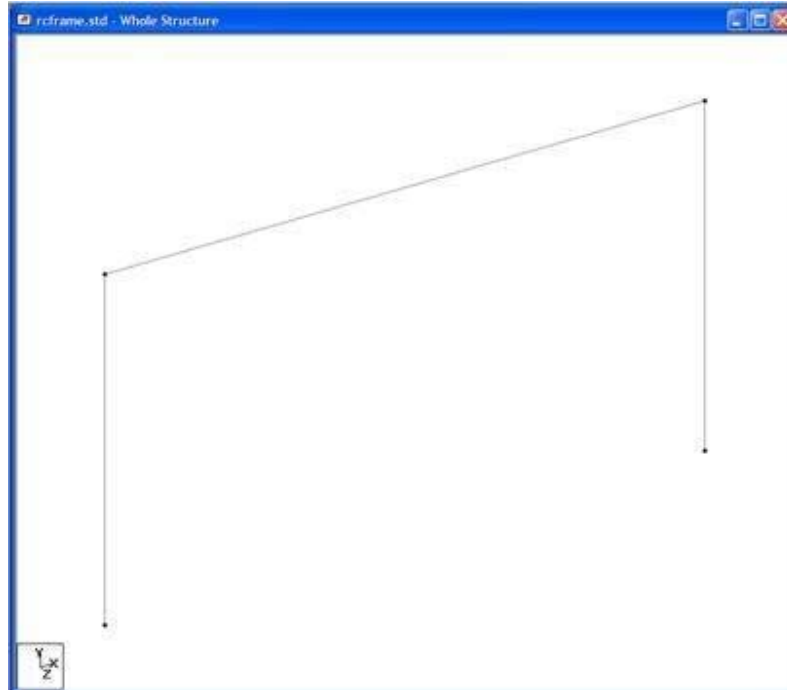
6. It is very important to save our work often to prevent data loss and protect our investment of time and effort against power outages, system problems or other unforeseen events.

> **Tip**: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

Switching node labels and beam

The node and beam labels are a way to identify entities that have drawn on the screen. In order to show the node numbers and beam.

1. Right-click anywhere in the drawing area and select **Labels** from the popup menu or select **View> Structure Diagrams**

    The Diagrams dialog opens in the Labels tab.

Figure 2-118:
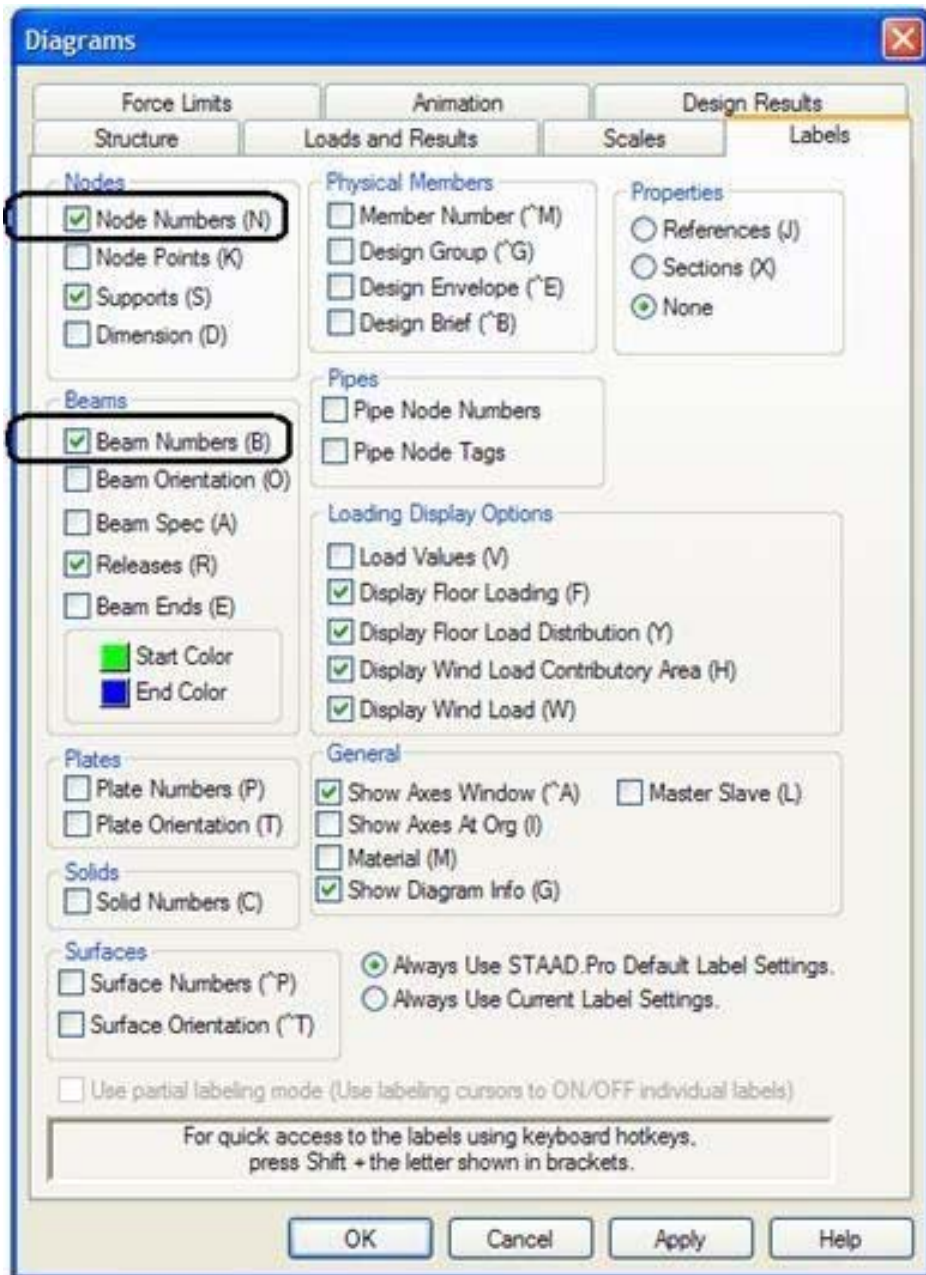
2. In the dialog box that appears diagrams, set the boxes node and beam numbers and click **OK**.

The following figure shows the node and beam numbers shown in the structure. The structure of the main window should look like the figure shown below.

figure 2-119:
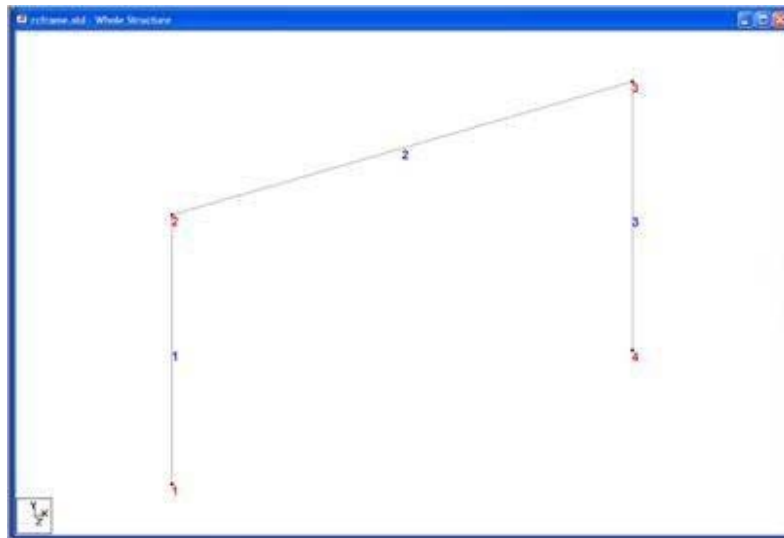
If you're feeling adventurous, here's a little exercise for you. Change the font of the labels / node beam selecting **View> Options** and then select the corresponding tab (node labels / tags beam) dialog box

Options.

3. Examining the structure shown in section 2.2 of this tutorial, you can see that the members of 4 and 5 can be easily generated if we could create a copy of first members 1 and 2 and then turn those units copied on a vertical line passing the point (6, 0, 0, that is, node 4) by 90 degrees. Fortunately, there are those

    facilities that can be executed in one step. It is called *Circular Repeat* (Repeat Loop) and is available from the menu **Geometry**.

    First, select the members 1 and 2 using the tool **Beams Cursor**

4. Select the tool **Circular repeat** from the bar proper tools

Figure 2 21
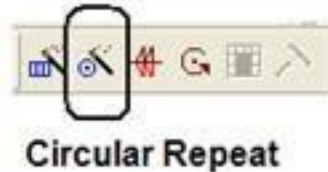
Circular Repeat

Or select **Geometry> Circular Repeat**. 3D Circular Table dialog box opens.

5. Specify the axis of rotation Y, Total angle 90 degrees and step numbers as 1 and the vertical line as step 4 of the node. O can be specified in place of X and Z coordinates as 6 and 0 respectively.

Clear the **Link Steps (**link steps) and click ok.

Figure 2-120:
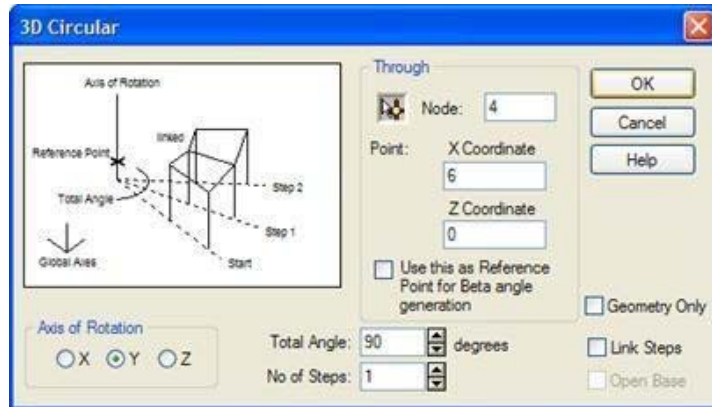
After completing the process loop repetition, the pattern will be as shown below.

Before
Figure 2-121:
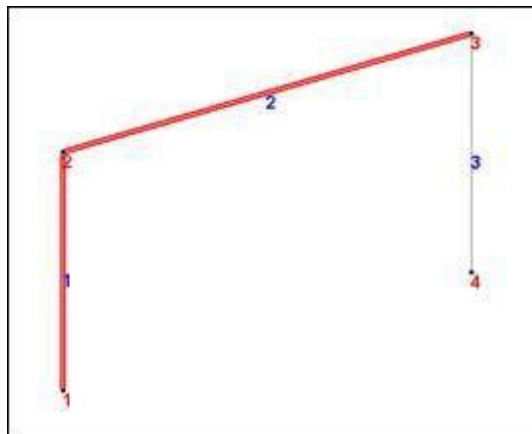
Then
Figure 2-122:
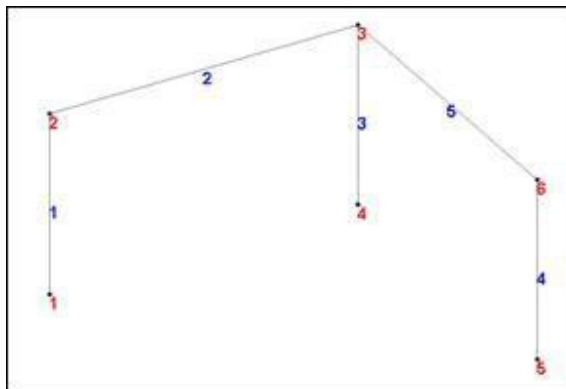
Tip: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S**.

2.6.2 Changing input length units

As a matter of convenience, to specify properties of members of our structure, it is easier if our length units are millimeters instead of meter. This will require changing the current length units input. The commands generated are:

```
UNIT KN MMS
```

Procedure:

1. Select tool or input units

Figure 2-123:

Input Units

o **Tools> Set Current Input Unit**.

The Units Setup dialog box opens current input.

Figure 2-124:

2. Sets the units of length **mm** and click **OK**.

2.6.3 Specifying the properties of members

Our next task is to assign cross-sectional properties of beams and columns (see section 2.2). For those of us know the equivalent commands in the STAAD command file are:

```
MEMB PROP
1 4 PRIS YD 300 ZD 275
2 5 PRIS YD 350 ZD 275
3 PRIS YD 350
```

Procedure:

1. Select Tool property page located on the toolbar of the structure.

Figure 2-125:

Property Page

Or select **General | Property page** from the left side of the screen shown below.

Figure 2-126:
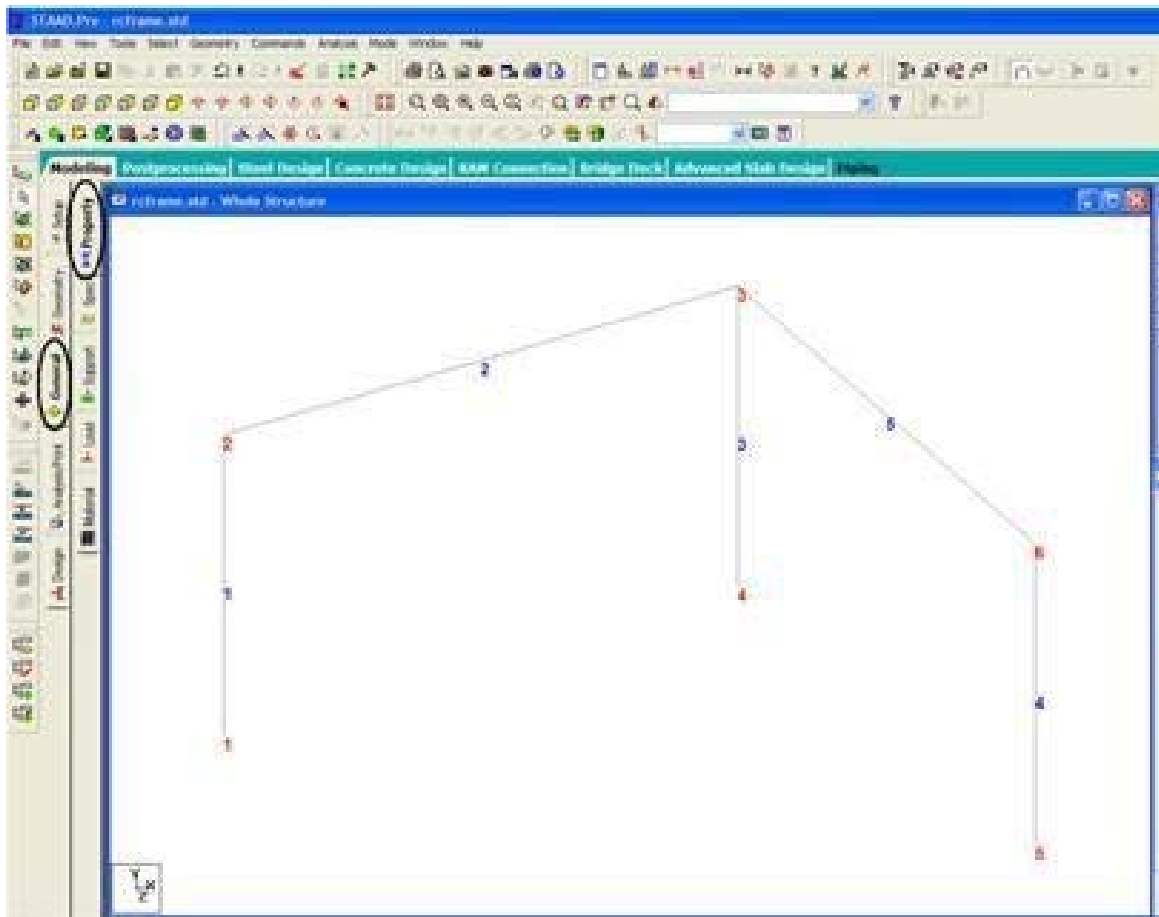
The Properties dialog - Whole Structure opens.

Figure 2-127:
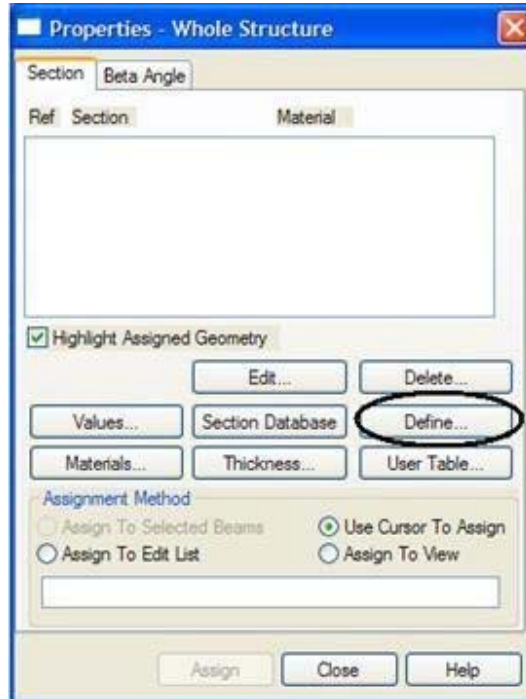
2. Click on **Define ...**

Open the Properties dialog box.

3. Select the tab **Rectangle**

> **Note**: The checkbox is set Material. By keeping the shape, the particular material properties (E, Poisson, density, alpha, etc.) along with the name of cross section is assigned. The material property values thus assigned shall be the default program. We do not want the default, however we will assign our own values later. Therefore, clear the check box Material. Then enter the following values:
>
> **Property Value**
>
> | | |
> |---|---|
> | YD | 300 mm |
> | ZD | 275 mm |

Finally, click the button **Add** as shown below.

Figure 2-128:
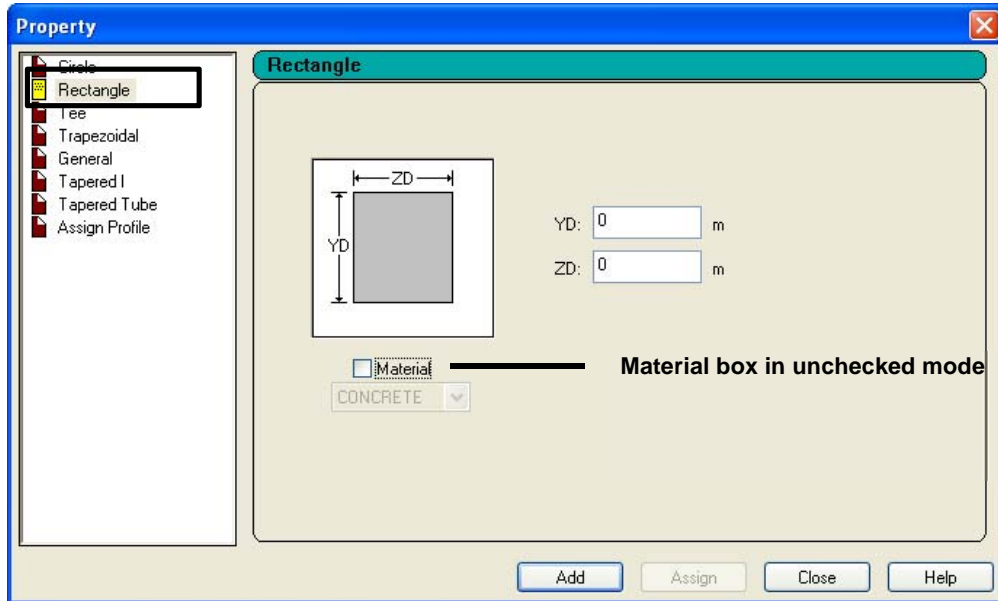
4. Repeat step 3 to create the second member property (PRIS YD 350 ZD 275) provide 350 **YD** and 275 to **ZD** (Instead of 300 and 275) and click **add**.

5. To create the third member property in the dialog *Properties* Select the option **Circle**. Specify diameter (YD) as **350** mm. So, clear the check box **Material** and click **add**.
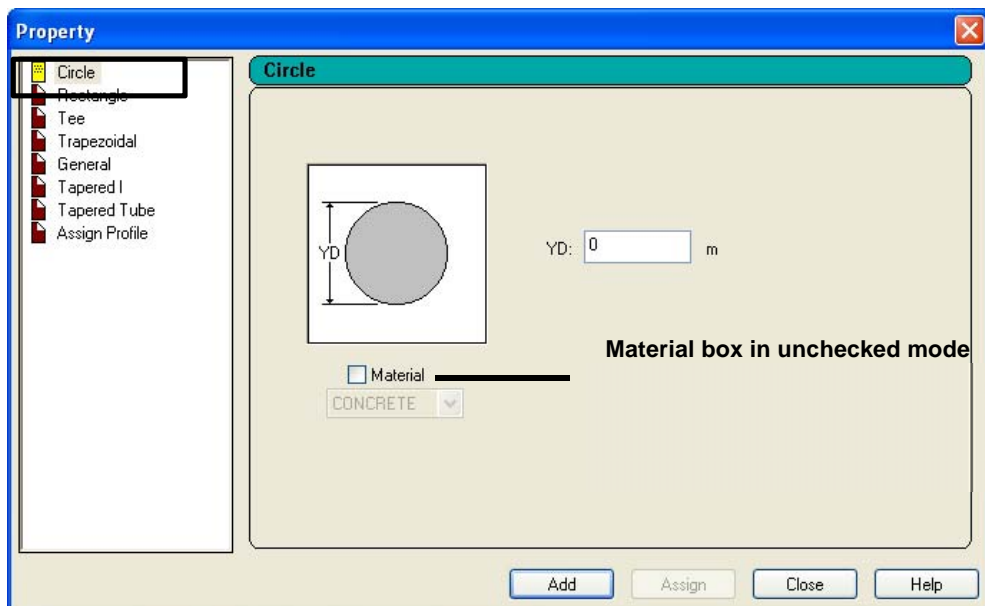
6. Click on **Close**.

The next step is to assign the properties of these members as follows: •

0.30x0.28 Rect - members 1 and 4

• 0.35x0.28 Rect - members 2 and 5

• Cir 0.35 - Member 3

To assign the member properties, follow these steps:

1. Select the first property reference in the dialog
   Properties (Rect 0.30x28).

2. Select the **Use Cursor to Assign** in the allocation method box.

3. Click on **Assign**.

The mouse cursor changes to 'I

4. Click members 1 and 4.

5. To stop the allocation process select **Assign** or press **Esc**.

Figure 2-130:
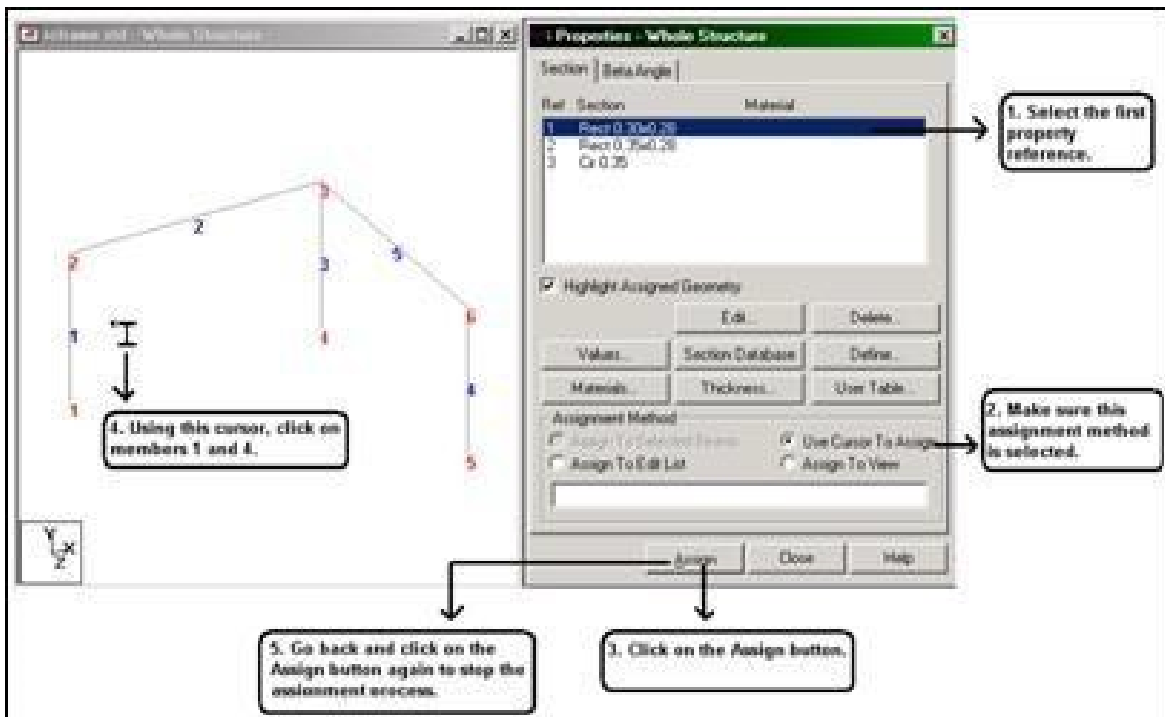
Similarly, assigning the remaining properties.

After all member properties have been assigned, the model will be as

below.

Figure 2-131:
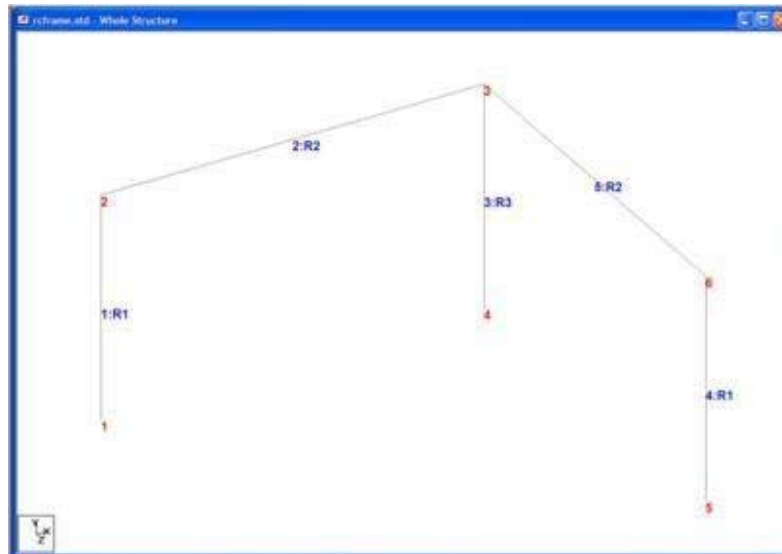
> **Tip**: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S**.

2.6.4 Specifying geometric constants

In the absence of any explicit instruction, STAAD guide the beams and columns of the structure of a default. Orientation refers to the direction that the width and depth of the cross section is aligned with respect to the global axis system. The rules that dictate this default orientation are explained in section 1 of the STAAD.Pro Technical Reference Manual.

We wish guiding member 4 so that their longer edges (local Y axis parallel to sides) are parallel to the global Z axis. This requires applying a beta angle of 90 degrees. The command has to be generated is:

```
BETA 90 MEMB 4
```

Steps:

1. Select the tab **Beta Angle** in the dialog box *Properties*.
2. Click on **Create Beta Angle**.

3. In the dialogue box Angle Beta, specify the **angle in degrees** 90.

Figure 2-132:
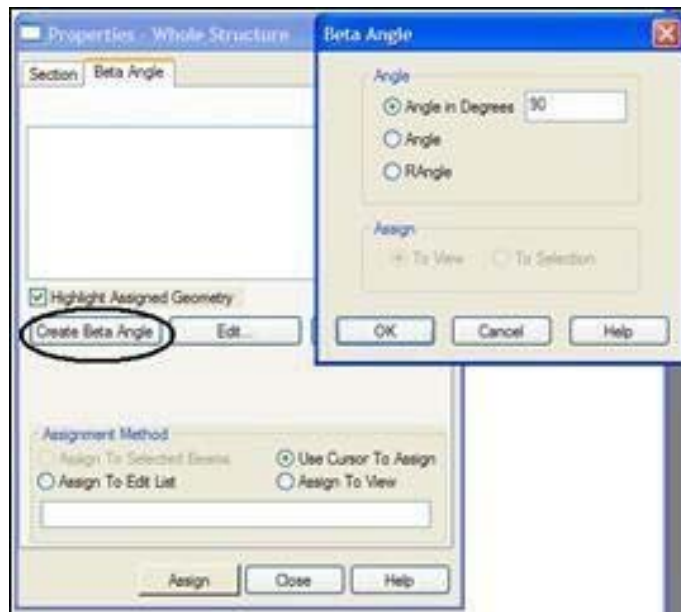
4. Highlight the expression *Beta 90* in the dialog box *Properties*.

5. Then select member 4 using the tool **Beams Cursor**   .

    Note that we select the member, the allocation method to automatically set *assigned to the selected beams*.

6. Click on **Assign**. Click anywhere in the drawing area to an enhanced member.

(An alternative method for allocating beta angles is as follows. First select the member for which you want to assign the beta angle. Then select **Commands> Geometric Constants> Beta Angle**. Specifies the *angle in degrees* 90, ensure that the allocation method is "*A selection*"And click **OK**.)

Figure 2 36

Check the orientation of the local axes of selecting members **View> Structure diagrams> Labels** and select the option for *orientation of the beam.*

figure 2-133:
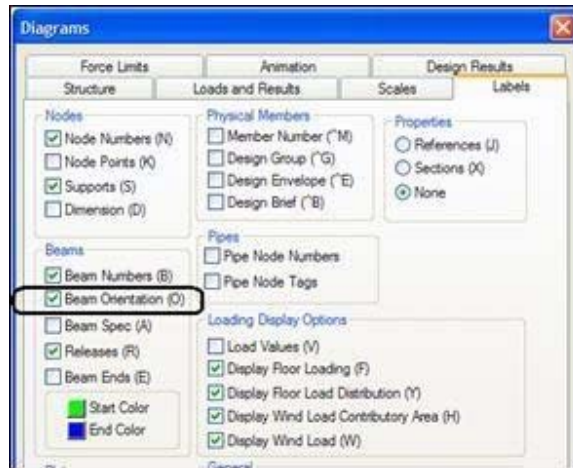
Figure 2-134:
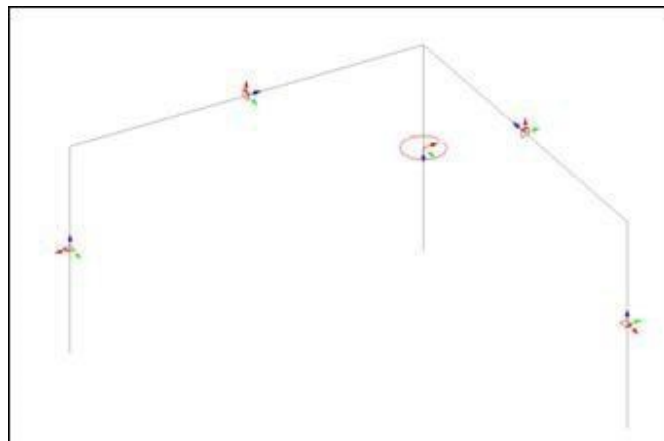
2.6.5 Specifying the material constants

When assigning member properties, we deliberately choose not to assign the same material constants as we wanted to specify values that are different from the default built. The desired values are listed at the beginning of this tutorial. The appropriate commands we want to generate in the STAAD input file are:

```
CONSTANTS
E   22   ALL
UNIT METER
```

```
DENSITY 25.0 ALL

POISSON 0.17 ALL
```

Procedure:

1. Select **Commands> Material Constants> Elasticity ...**.

2. In the dialog *Material Constant* that appears, enter **22** in Table *Enter Value*. Since the value has to be assigned to all members of the structure, the current configuration of the allocation method, ie, **To View** Allows us to accomplish this easily. Then click on **OK.**

Figure 2-135:

3. To specify the constant density, it will be convenient if we change our units of length in meters. To change the units of length, as before, select the tool **input Units** the structure toolbar, or select the menu option **Tools> Set Current Input Unit** unit of the menu bar top. In the current set input units dialog that appears, specify the units of length and meter.

Figure 2-136:

4. We follow steps 1 and 2 above, we choose **Commands> Material Constants> Density,** specify the value as 25 kN / m 3 and assign to the view.

5. To define the **Poisson's ratio**, Using a similar procedure as described above, providing the value **0.17** all members of sight.

2.6.6 Specifying Mounting

The basis of all columns nodes are restrained against translation and rotation of the 3 global axes (see section 2.2). In other words, fixed supports (wardrobes) must be specified in those nodes. The commands generated are:

SUPPORTS

1   4   5   FIXED

Procedure:

1. Select the tool **Support Page** located on the toolbar of the structure as shown below.

Figure 2-137:

Or select the tab **General | Support** from the left side of the screen

Figure 2-138:
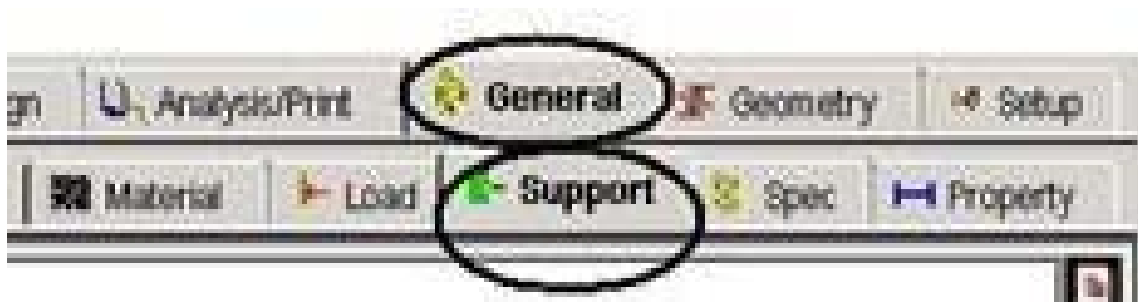
The media dialog opens.

figure 2-139:
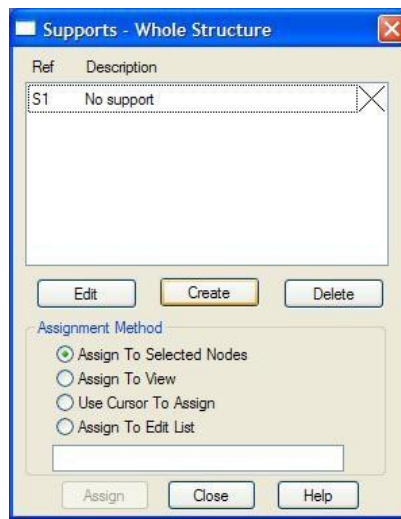
2. Since we know that the nodes 1, 4 and 5 must be associated with the fixed support, using the **Nodes Cursor** . Select these nodes.

3. Click **create**. Open dialog box build support.

Figure 2-140:
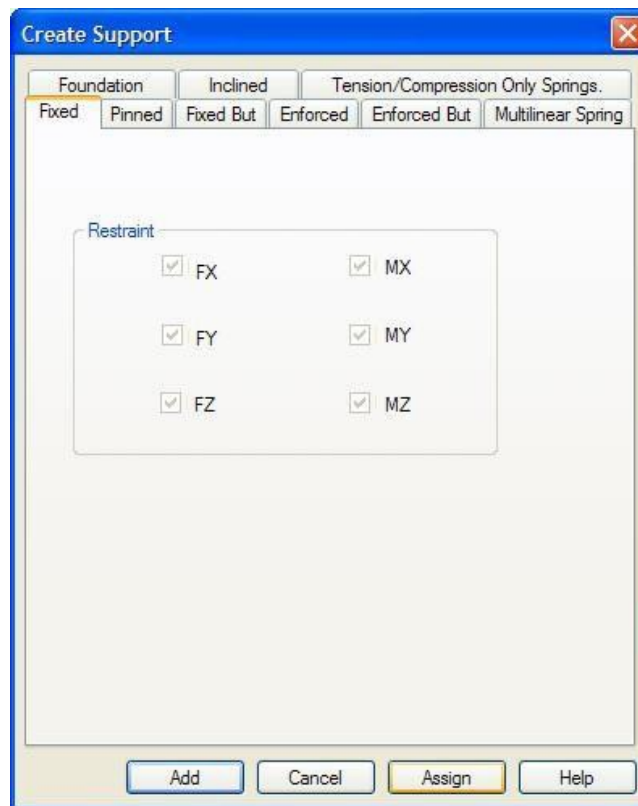
4. Select the tab **Fixed** and click **Assign**.

   After the supports have been assigned, the structure will look as shown below.

Figure 2-141:
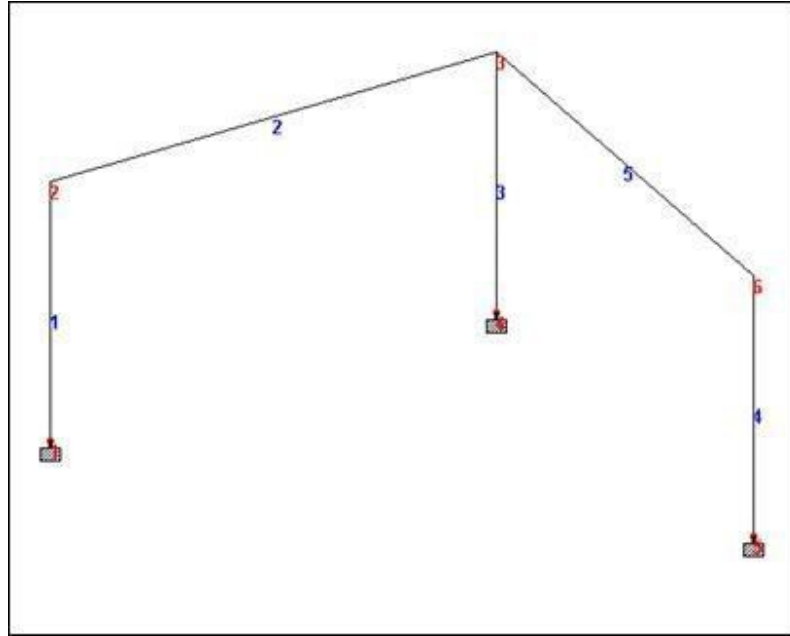
5. Click anywhere in the drawing area to deselect all selected nodes s and avoid accidentally assigning unwanted data to nodes. The pulse anywhere in the drawing area to deselect all selected nodes and avoid accidental data allowance unwanted nodes.

Tip: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S**.

2.6.7 Specifying Loads

The five load cases need to be created for this structure. Earlier in this tutorial details of individual cases are explained.

Then the corresponding commands generated are listed. Note that cases 4 and 5 are to be built not like the type of standard combination, but using a combination of load type called **REPEAT LOAD** (Repetitive) load. The instructions at the beginning

This tutorial require us to analyze this structure using a type of analysis called **PDELTA**. Analysis **PDELTA** It is a type of nonlinear analysis. In STAAD, states also account for the effects of the simultaneous PDELTA action horizontal and vertical loads

predefined; previous cases should be included as components of the combo box using the exchange **REPEAT LOAD**.

```
 KG METER UNIT
DEAD LOAD 1 LOAD
SELFWEIGHT -1 MEMBER
AND LOAD
May 2 UNI GY -
400 MEMBER LOAD
LOAD LOAD 2 Live
May 2 UNI GY -
600 3 WIND LOAD
LOAD LOAD MEMBER
1 UNI GX 300
4 UNI GX 500
LOAD 4 DEAD + LIVE
REPEAT LOAD
1 1.2 2 1.5
LOAD 5 DEAD +
      REPEAT LOAD WIND
1 1.1 3 1.3
```

LOAD CASE 1

STAAD has a limitation in that you can not change the unit load cases while editing. An error message appears if you try.

Before creating load cases, we have to change the unit of **kilogram**. See "2.6.2 Changing the units of the input length" on page 107 for related information on the required steps.

(At the beginning of this tutorial load values appear in units kg and $subway$. Instead of converting these values in the current input units, conform to those units. The current input units, while finally establish specify the density, are KN and METER).

1. To create loads, select the tool or **Load Page** located on the toolbar *Structure Tools*.

figure 2-142:

Load Page

Or select the **General | Load** General | Page Load from the left side of the screen.

Figure 2-143:
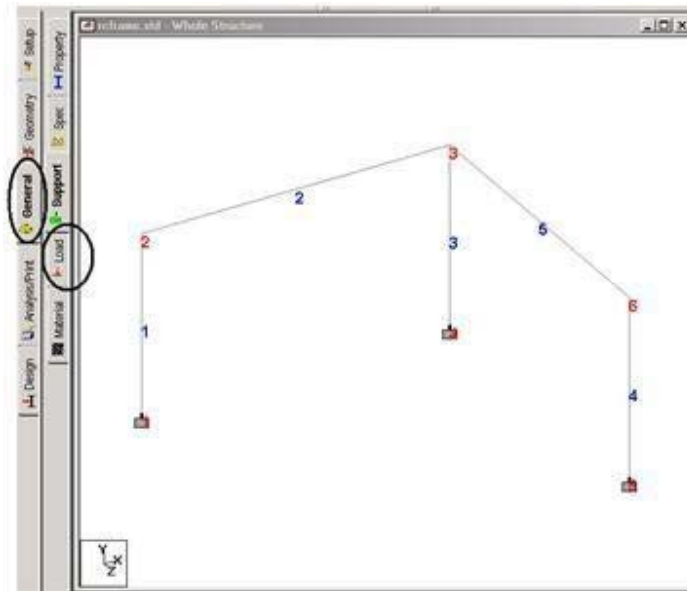
Definition dialog box & load on the right side of the program window opens.

2. To start the first load case, select the section **Load Case Details** in the list and click **add**.

Figure 2-144:
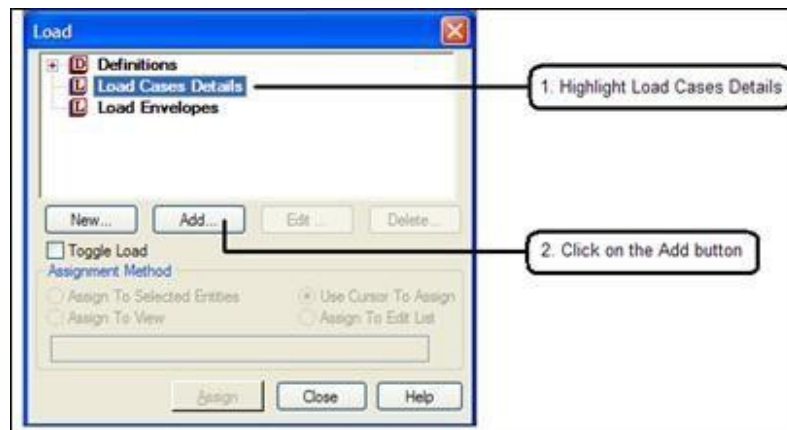
The Open dialog box *Add New Load Cases (*Add new load cases).

3. Select **loading type** if you want to associate the load case with any of the definitions of ACI, AISC or IBC dead load, live, ice, etc. This type of partnership should be done if you intend to use the facility for automatic program generation load combinations in accordance with the codes.

Note that there is a checkbox labeled *Reducible per UBC / IBC* (Reducible by UBC / IBC). This feature becomes active only when the load case is assigned a load type called *Live* at the time of creation of that case.

As we do not intend to use the option of automatic generation of load combination, we will leave the **loading type** how come **None**. Write **DEAD LOAD** as the title for *If load 1* and click **add**.

Figure 2-145:

The newly created load case appears under the option *Load Cases Details* (Load case data).

Figure 2-146:
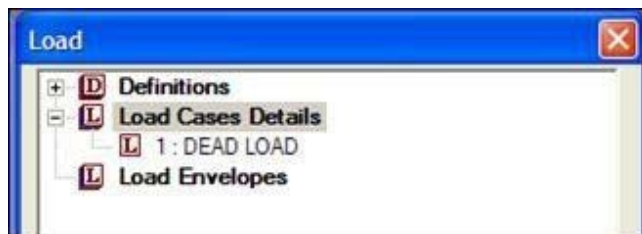
4. To generate and assign the type of selfweight load (own weight), first select 1: DEAD LOAD. You will notice that the dialog *Add new items loading* now shows more options.

Figure 2-147:

5. Specify the address as **Y,** and Factor as **-1.0**. A negative number means that the load acts against the positive direction of the overall axis (Y in this case) over which it is applied. Click the button **Add**. The load selfweight applies to all

members of the structure and can not be applied to a selected list of members.

Figure 2-148:
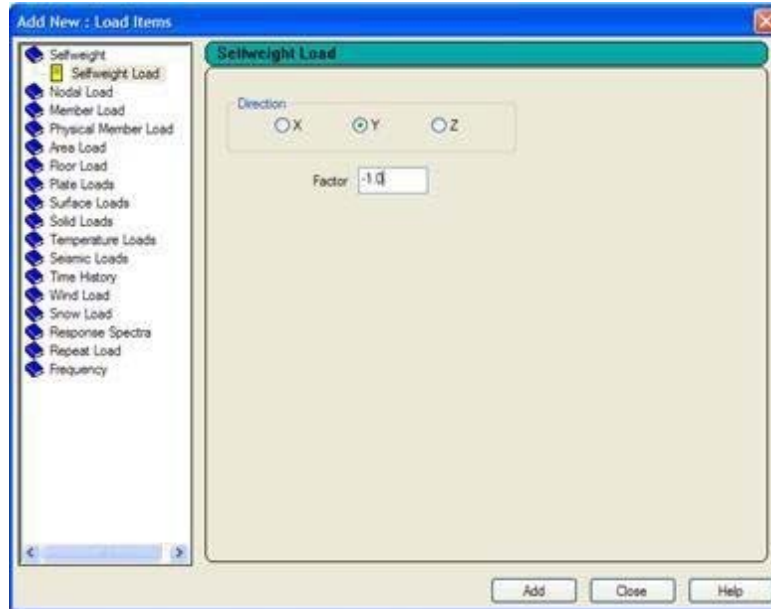
6. Load 1 contains a component of additional load, loads on the members of 2 and 5 To create the load member, first select 1: Dead Load button followed **Add**Then .... click Article **Member Load** in the dialog box *Add new items loading*.

Figure 2-149:
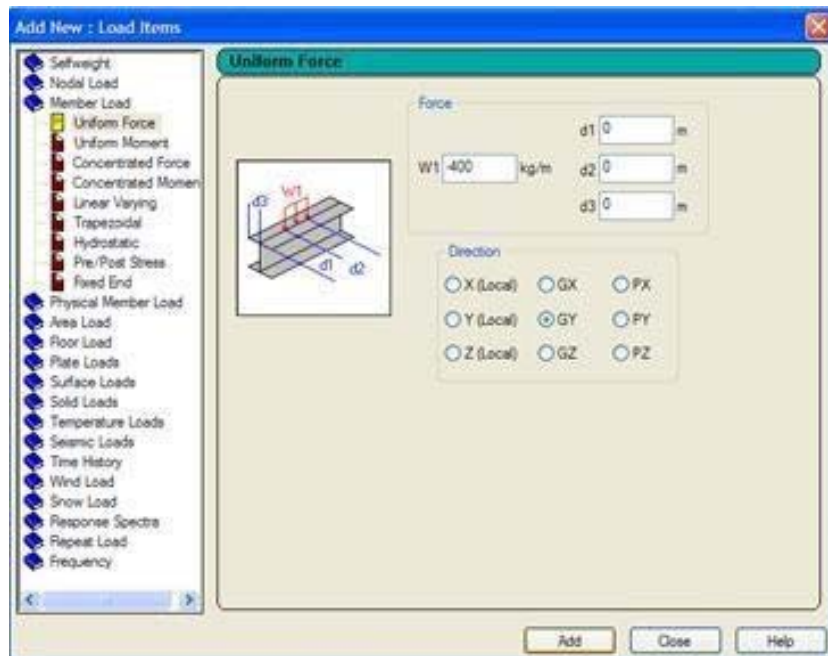
7. Select the option **Uniform Load** and specify **GY** as *Street address* and **-400** as *force*. For these members, because the local Y axis coincides with the axis Y global, you can choose the direction of the load as "Y" or "GY" both have the same effect.

   (You can see the orientation of the local axes members to go to **View> Structure Diagrams> Labels> Beam Orientation**.) A negative value means that the load acts in the negative direction of GY. Click **add** and then **shut down**.

8. The burden of the member who just created should be assigned to the members of 2 and 5.

   Select the input **UNI GY-400 kg / m** in the dialog box *Load & Definitions*.

Figure 2-150:

9. Then select members 2 and 5 with the tool **Beams Cursor** .

See 'frequently performed tasks' section at the end of this manual for learn more about the selection of members).

Then, select **Assign to Selected Beams** and then **Assign**.

Figure 2-151:

As you click on the button *Assign* The following dialog box appears. This message box appears only wish to confirm that indeed the case load associated with the selected beams. Click **Yes I do.**

Figure 2-152:

After the load has been assigned, the structure will look as shown below:

Figure 2-153:
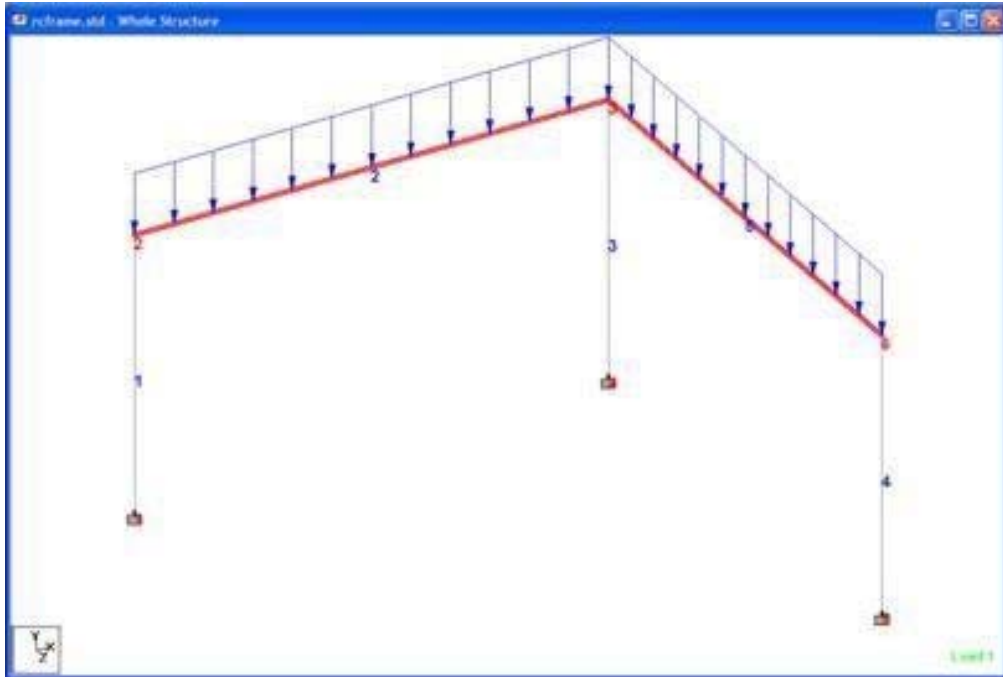
LOAD CASE 2

1. The next step is to start charging the second case containing a load over the element. Select **details of the case load** and click **add**. Again, the dialog box opens *Add new load cases*.

Figure 2-154:
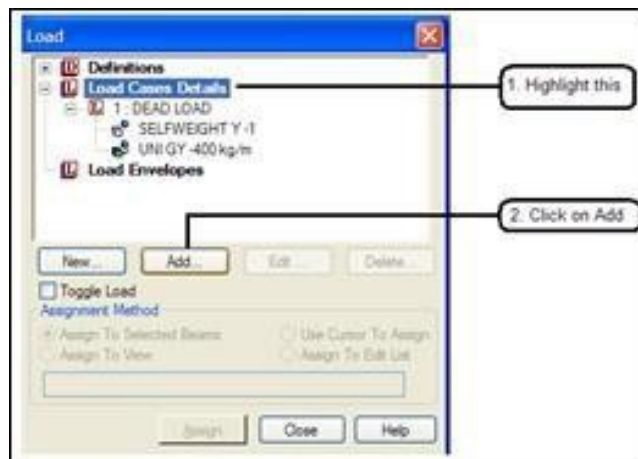
In this dialog box, once again, we are not we associating the load case, we are about to create with any load code base and so leave that box as **None**. Specify the title of the second load case as **Live Load** (Live load) and click **add**.

Figure 2-155:

2. To create the load member, point 2: **Live Load**.

Figure 2-156:

3. Follow steps 6-9 of the previous task to create and assign a uniformly distributed force **-600** Kg / m 2 and 5.

After loading the second case has been assigned, the structure will look as shown below:

Figure 2-157:
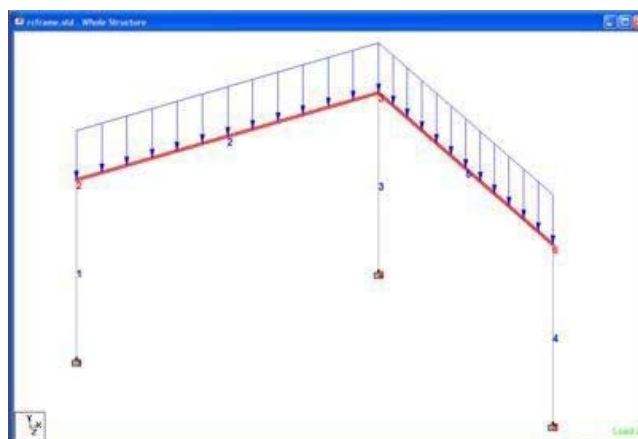
Click anywhere in the drawing area to an enhanced members.

CASE LOAD 3

1. The third load case, which has loads members again, the creation involves the same procedure as in the case of load 2 As before, first select **Load Case Details** in the *Load dialog box* to start charging the third case. Type **Wind Load** (Wind load) as the title for the case of load 3.

2. To apply the load to the member 1, follow similar procedure as in steps 6 to 9 The only differences are, member 1 is the recipient of this burden, *Street address* is **GX** and the force is **+300** Kg / m.

3. Also, for the member 4 and the third load case, specify the force **500** Kg / m and direction as **GX**.

After charging the third case has been assigned, the structure will look as shown below:

Figure 2-158:
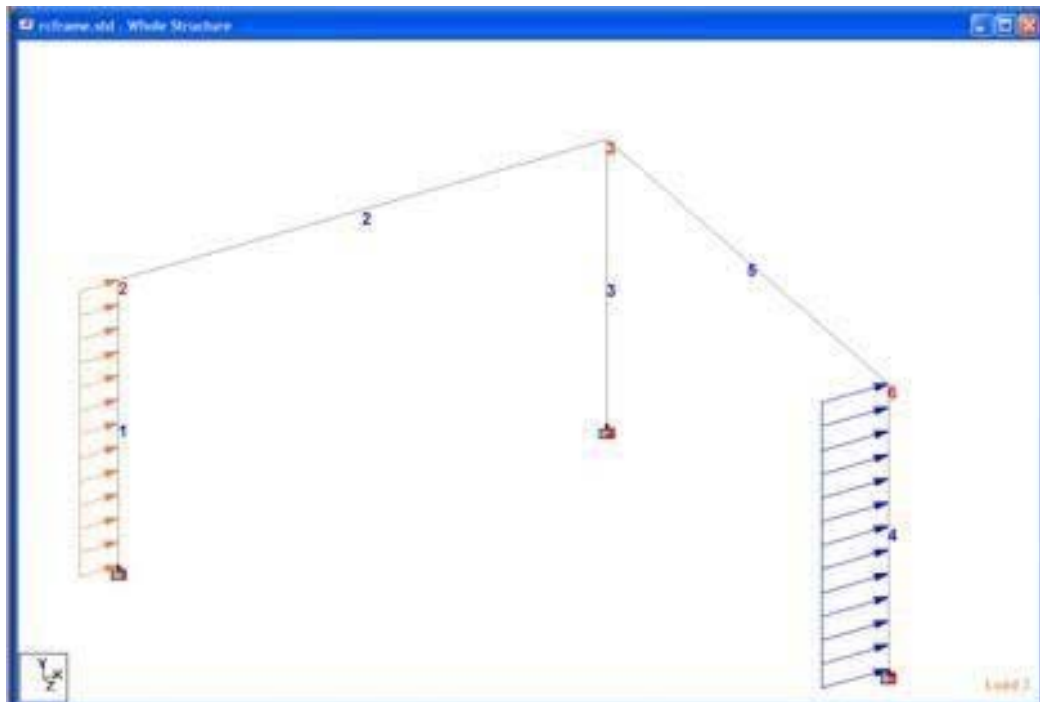
CASE LOAD 4

1. Now we come to the point where we have to create the load case 4 and (1.2 x Load 1) (1.5 x Load 2). We saw earlier in this section should be creating a type of "Repeat Load" of the combination and not the kind of "Load Combination". To start the load case 4, select **Load Case Details** in the dialog box and load

specify the title as **DEAD + LIVE**.

2. Then click on **4: DEAD + LIVE** in the dialog box definitions & loads as shown below.

Figure 2-159:
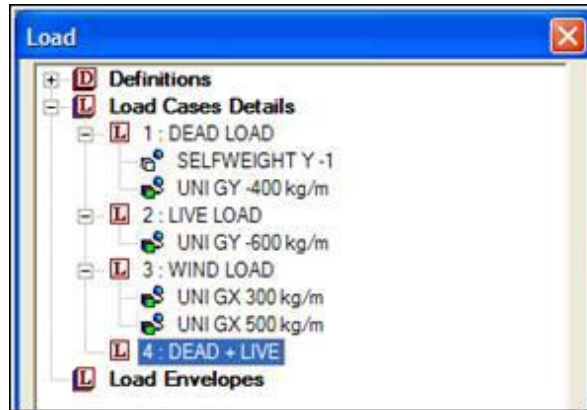
3. In the dialog box *Add new items loading* Select the option **Repeat Load**. Then select load case 1 (dead load), click [>] and enter the *Factor* how come **1.2**. (This indicates that the data values of one case cargo load

   are multiplied by a factor of 1.2, and the resulting values are used in load case 4)

4. Similarly, select box load 2 (live load), click the [>] button and enter the factor as **1.5**. You will now see the dialog *Add new items loading* as shown below. Click the button **Add**.

Figure 2-160:
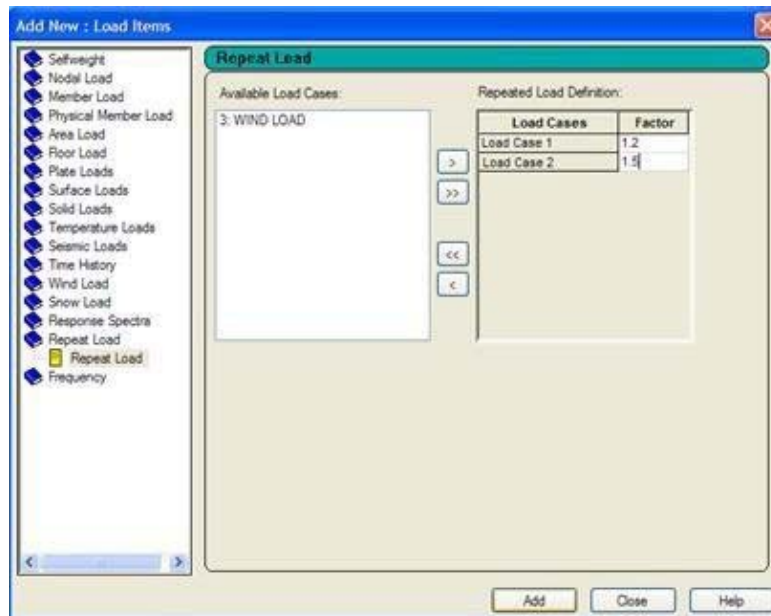
No additional operation is needed for load case 4 recipients (members) loads in load case 4 are automatically chosen to be the same that were assigned components REPEAT LOAD cases (repetition of load-loads 1 and 2).

The structure will now similar to the one shown below.

Figure 2-161:
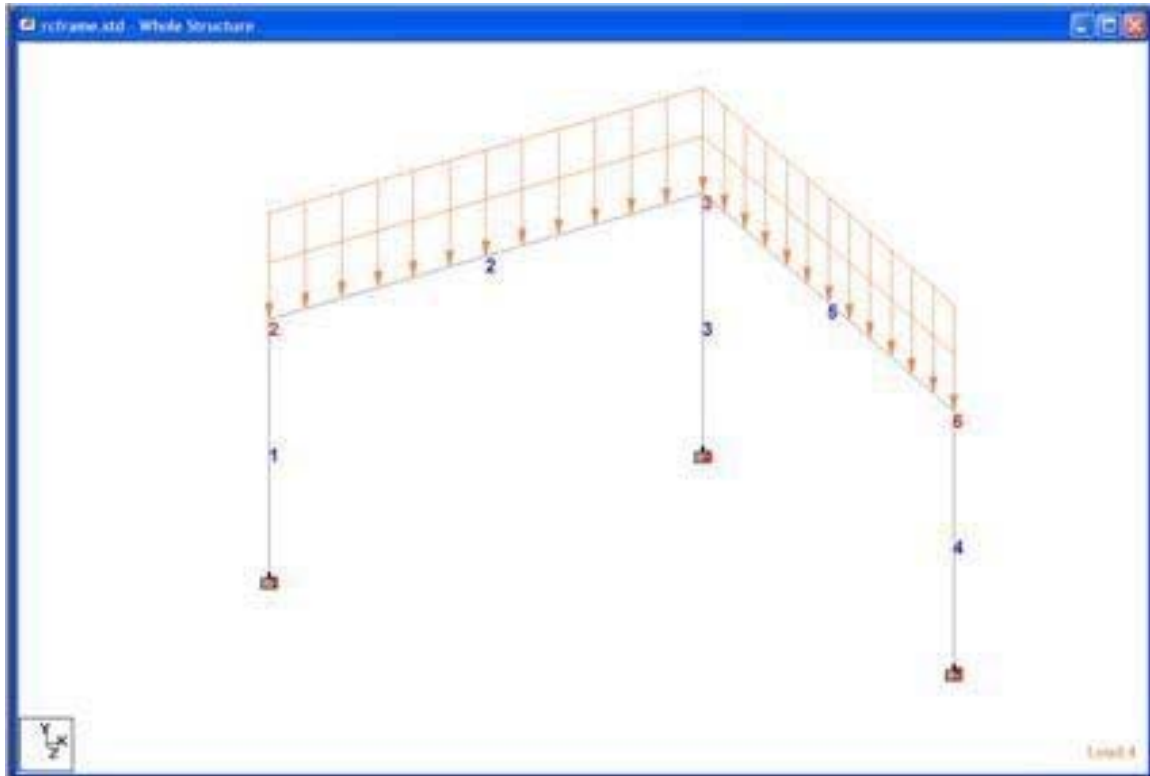
CASE LOAD 5

1. Since cases 4 and 5 are nearly identical in nature, the same procedure used in building load hypothesis 4 is also applicable for the case 5 Let us select **Load Case Details** in the dialog box to start the load

   fifth load case. Enter **Dead + Wind** as the title for the load case 5.

2. Follow steps 16-19 except for associate a factor of 1.1 for the first case and load factor 1.3 with the third load case. Follow steps 16-19 except associates a factor 1.1 if the first load and a factor 1.3 with the third load case.

You will now see the dialog *Add new items loading* as shown below. Click the button **Add**

Figure 2-162:
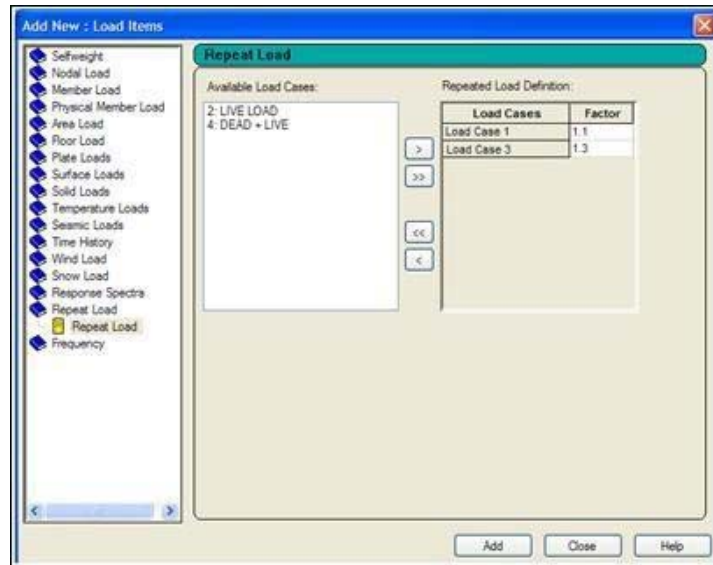
We've finished creating all load cases, we can now click **shut down**.

The structure will now similar to the one shown below.

Figure 2-163:
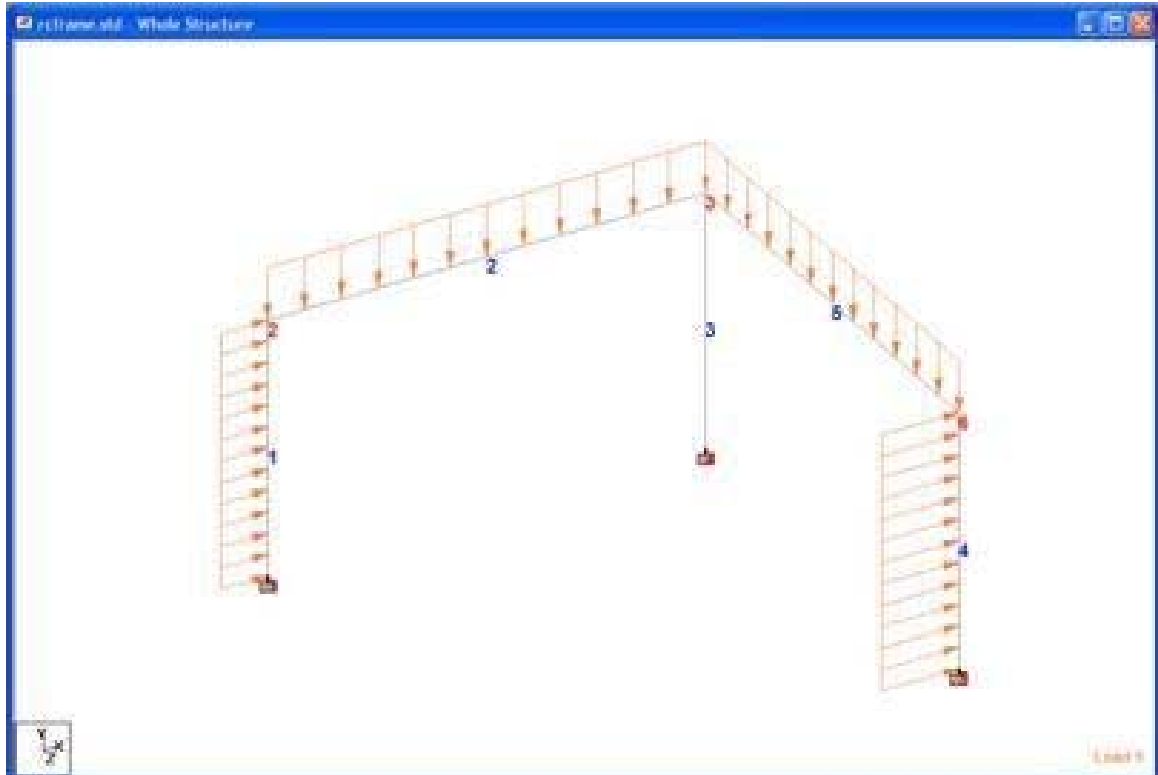
Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S.**

2.6.8 Specifying the type of analysis

The analysis of this type of structure is called PDELTA. Since this problem involves concrete beam and column design by the ACI code, the second-order analysis is required and must be made in the factored loads acting simultaneously. The factored loads previously created as cases 4 and 5 Now it's time to specify the type of analysis.

The command for a PDELTA analysis appear in the STAAD file as:

```
ANALYSIS PDELTA
```

Procedure

1. Select **Analysis / Print | Analysis** on the left side of the screen.

Figure 2-164:
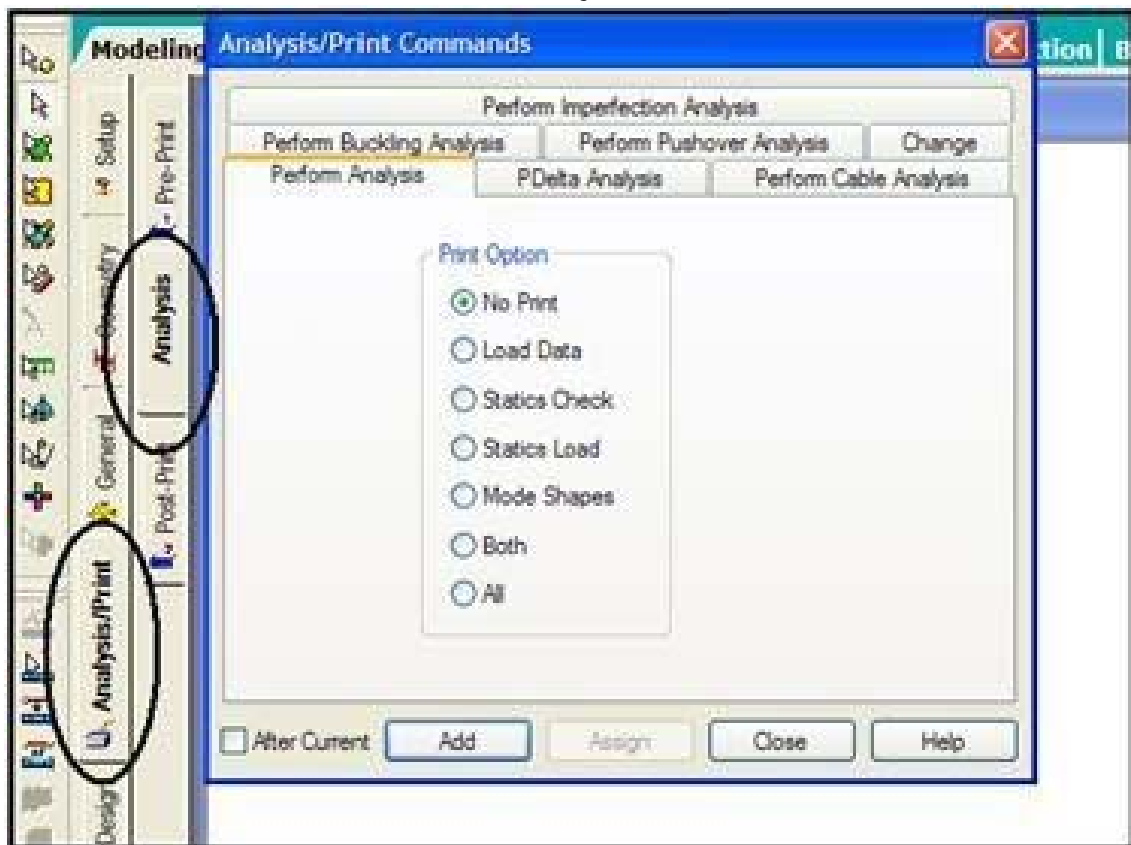
Dialog box opens *Analysis / Print Commands*.

Figure 2-165:
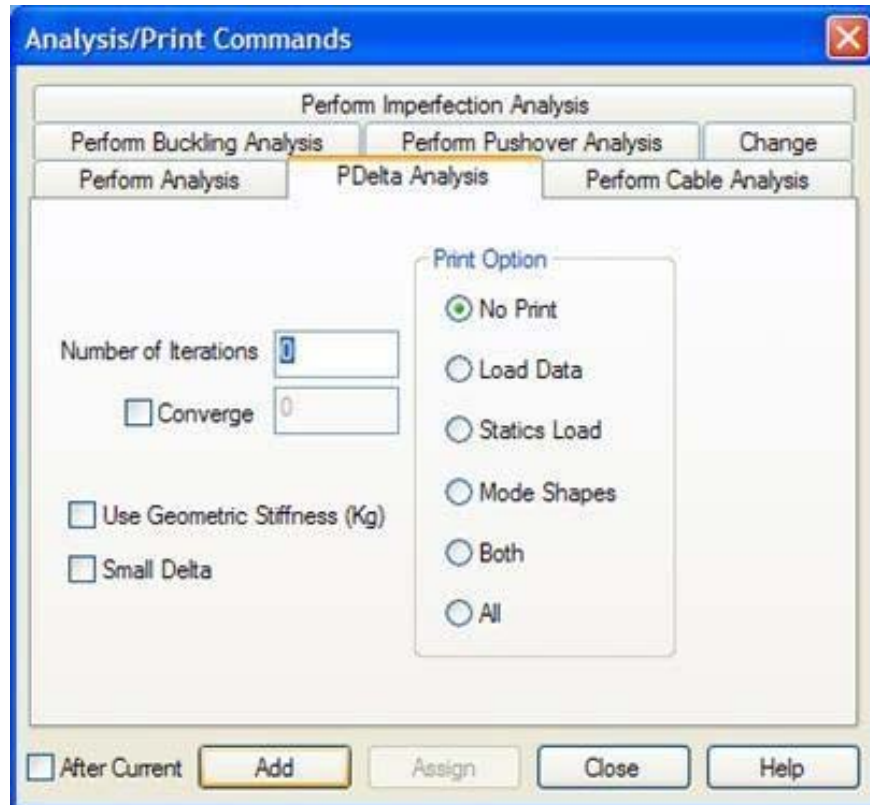
2. Select the tab **PDELTA Analysis**.

3. Click on **Add** and then **Close**.

> **Tip**: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

2.6.9 Preselection of load cases to be used in the design of concrete

The design of concrete shall be carried out for load cases 4 and 5 only because those are only the cases factored. To tell the program to use these cases and ignore the rest, we have to use the LOAD LIST command.

The command appears in the STAAD file as:

```
LOAD LIST May 4
```

Procedure:

1. Select **Commands> Loading> Load List**. The List Open dialog load.

Figure 2-166:
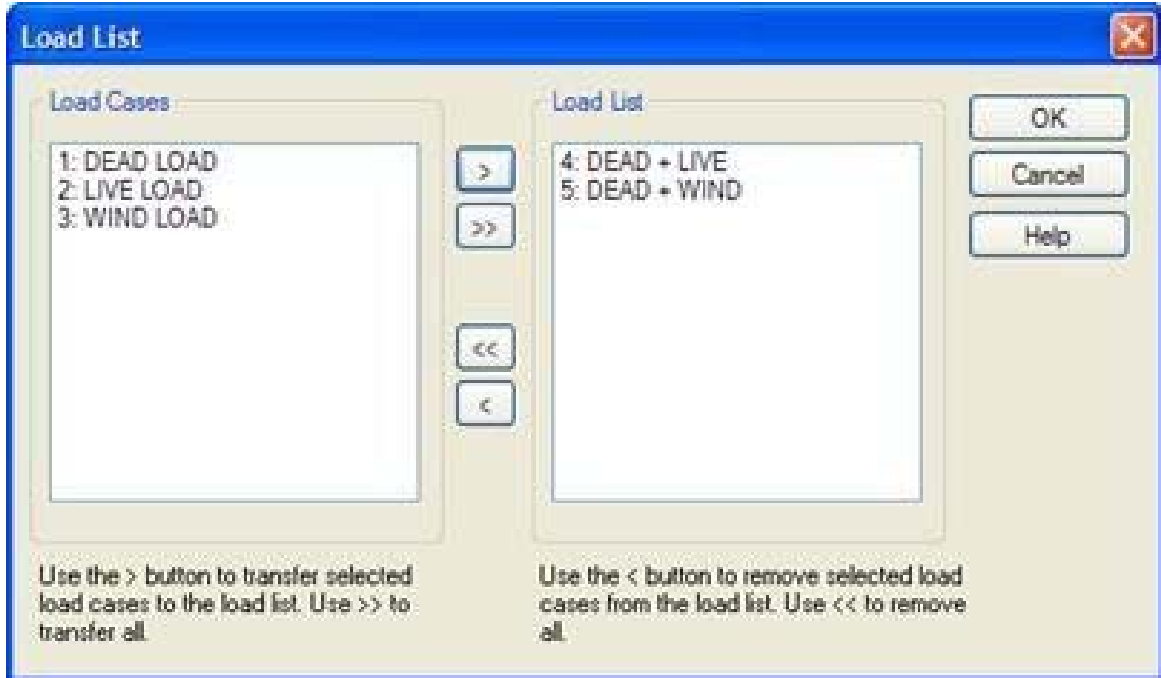
2. Press the key **CTRL** to select cases of load 4 (DEAD + LIVE) and 5 (DEAD + WIND).

3. Click [>]. Load cases 4 and 5 will be selected and placed in the cargo box selection list.

4. Click on **OK**.

2.6.10 Specifying particular design parameters

Among the various terms in the equations for the design of concrete beams and columns, some of these can be directly specified; such as the degree of concrete or the maximum size of the rebar that may wish to use. Such terms are called specific design parameters. For the ACI code, a list of these parameters is available in Section 3 of the Technical Manual STAAD.Pro. The parameters that we want to use and the associated command should appear in the STAAD input file are:

```
NEWTON MMS ACI
CODE UNIT
CLT 25 ALL
```

```
CLB 30 ALL

CLS 25 ALL

FC 25 ALL

FYMAIN 415 ALL

TRACK 1 ALL
```

Procedure

1. Define the units of force as **Newton** and the units of length and **mm**. See "2.6.2 Changing the units of the input length" on page 107 for details.

2. Select the tab **Design | Concrete** from the left side of the screen and select **ACI** as the current code in the dialog *Concrete design*.

Figure 2-167:
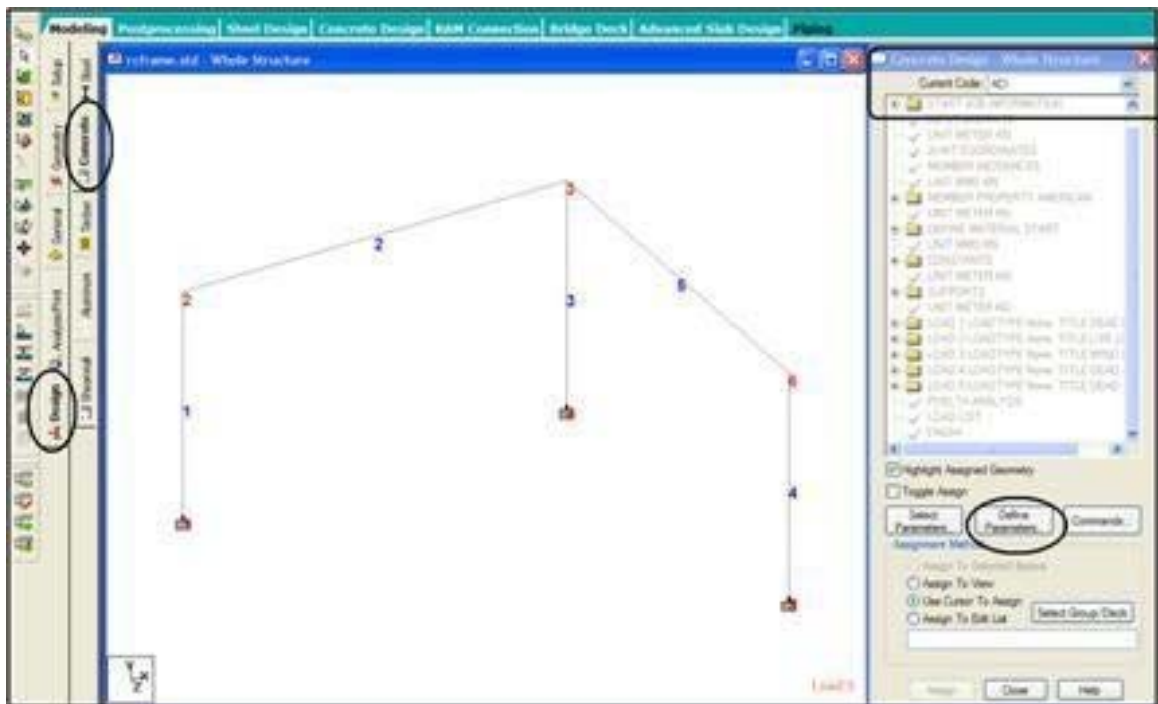
3. Click **define parameters** in the dialog concrete design.

   It opens dialogue *Design Parameters*.

Figure 2-168:
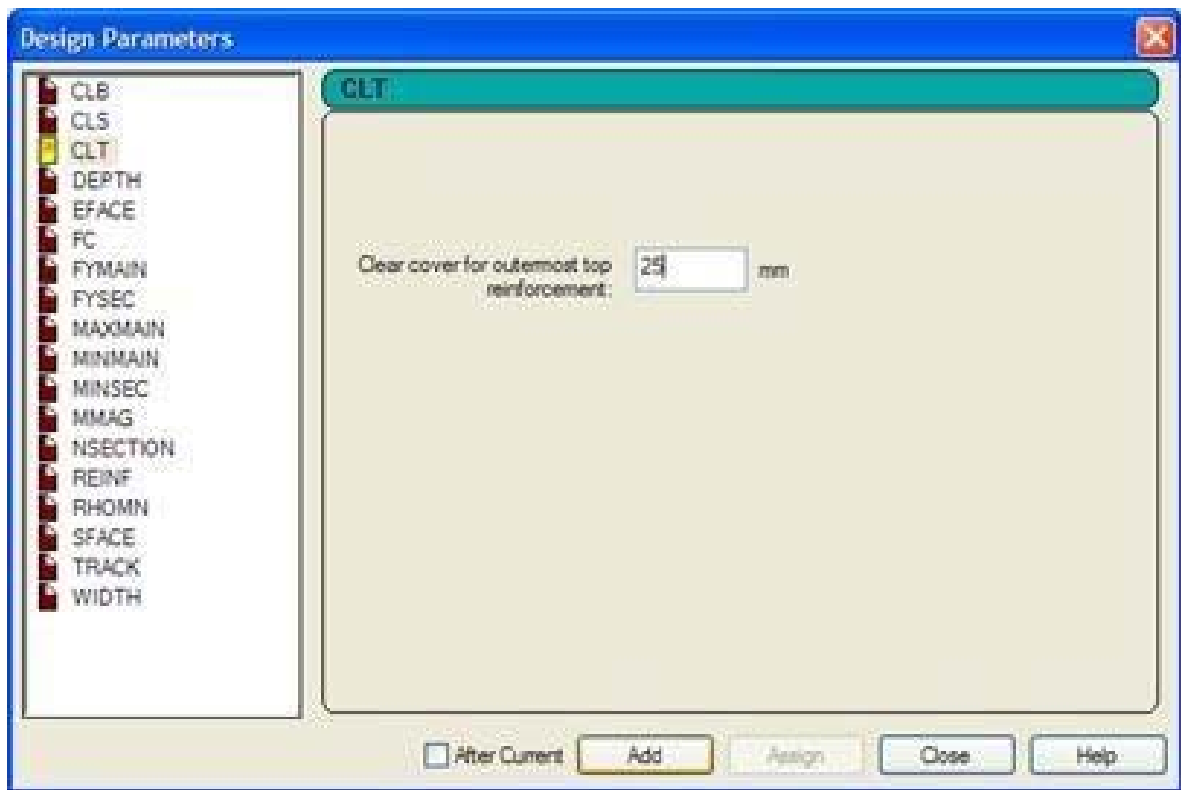
4. Select the parameter **CLT** (Clear Cover for Top- topcoat reinforcement). Then, provide the value as **25**mm and click **add**.

5. Repeat step 4 to set the remaining parameters to the following values:

Parameter	Value
Clb	30
Cls	2
Fc	2
Fymain	415
Track	1.0

When all the above parameters have been assigned, click Close in the dialog box *design parameters*.

After that all design parameters have been assigned, the dialog *Concrete design* will be as shown below.

Figure 2-169:
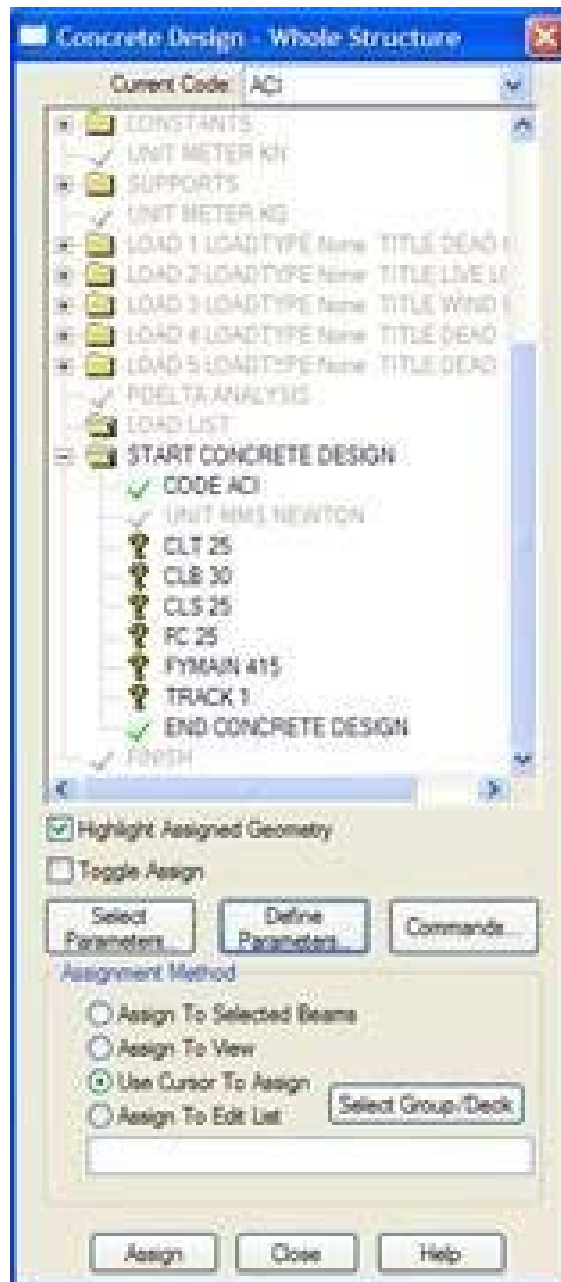
The next step is to assign these parameters to all members of our model. The easiest way is to use the method of assigning to the view:

1. Stresses the parameter in the Concrete Design | Whole Structure to be assigned to model elements.

2. Select the option **Assign to View**.

3. Click on **Assign**.

Figure 2-170:
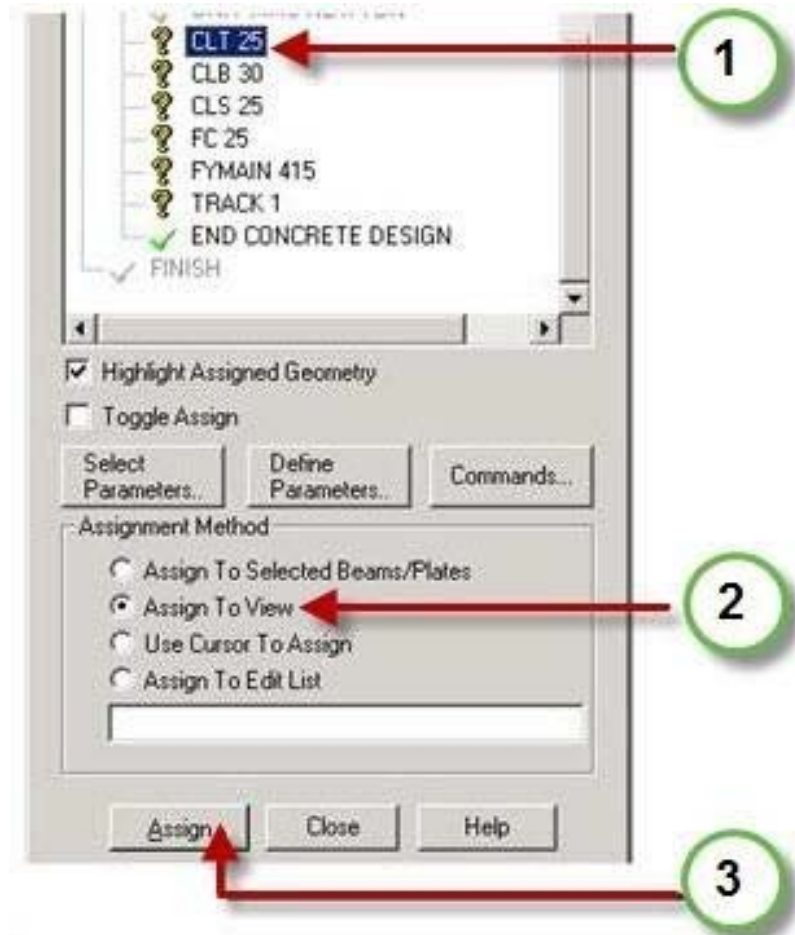
Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

2.6.11 Specifying design commands

Commands design are the actual instructions for the design of beams and columns. We intend to design beams 2 and 5 and columns 1, 3 and 4 commands generated are: Design commands are the actual instructions for the design of beams and columns. We intend to design beams columns 1, 3 and 4 and 5 and 2 commands generated are:

```
DESIGN BEAM May 2
COLUMN DESIGN 1 March 4
```

Procedure:

1. The design commands are generated through the dialog boxes available under the button **Commands** in the dialog box *Concrete Design*. So, we click **Commands** as shown below.

Figure 2-171:
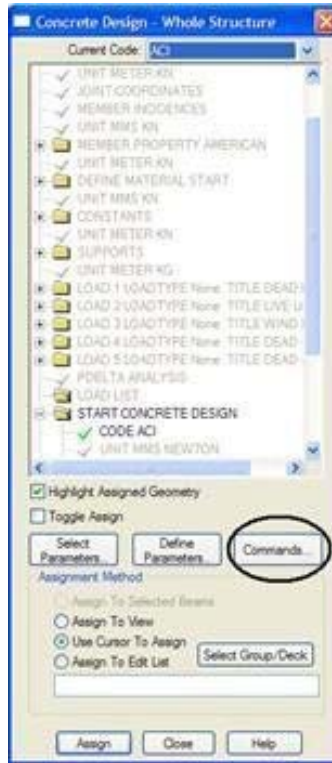
2. In the dialog box *Design Commands* that appears, select the option **Beam Design** and click **add**.

Figure 2-172:
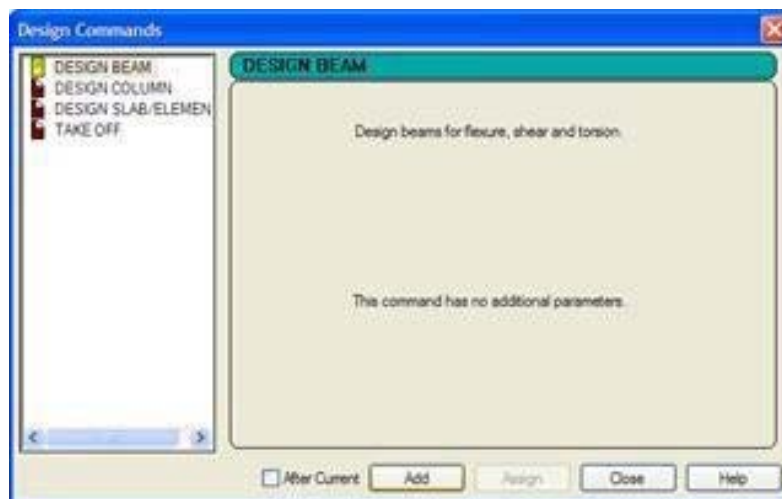
3. We need to add a command for column layout. Then, select the **Column Design** and click **to add.**

4. Click on **Close**.

The next step is to associate the layout command beam members 2 and 5 and the design command column members 1, 3 and 4.

1. Select the design and then the beam members 2 and 5 with the tool **Beams Cursor** .

2. Click on **Assign to Selected Beams** and then in **Assign**.

Figure 2-173:
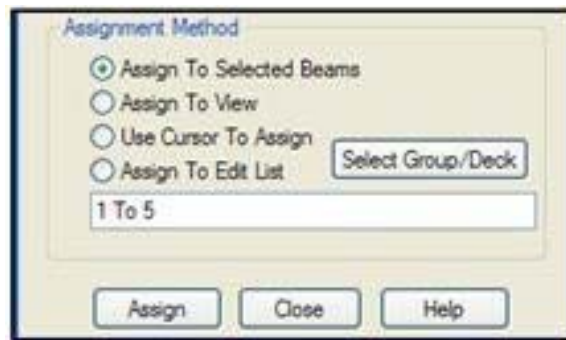
As we click **assign,** The following dialog box appears. This message box appears only to confirm that indeed we associate the design of the beams of the selected command. Click **Yes**.

Figure 2-174:

3. Repeat steps 1 and 2 to assign the command **column design** members 1, 3 and 4

Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

2.7 Visualizing the archive command input

Now take a look at the data that has been written to the file you saved above.

1. To view the contents of the file, select the tool **STAAD Editor**

Figure 2 82

Or select **Edit> Edit Input Command File**. STAAD Editor window opens.

Figure 2-175:
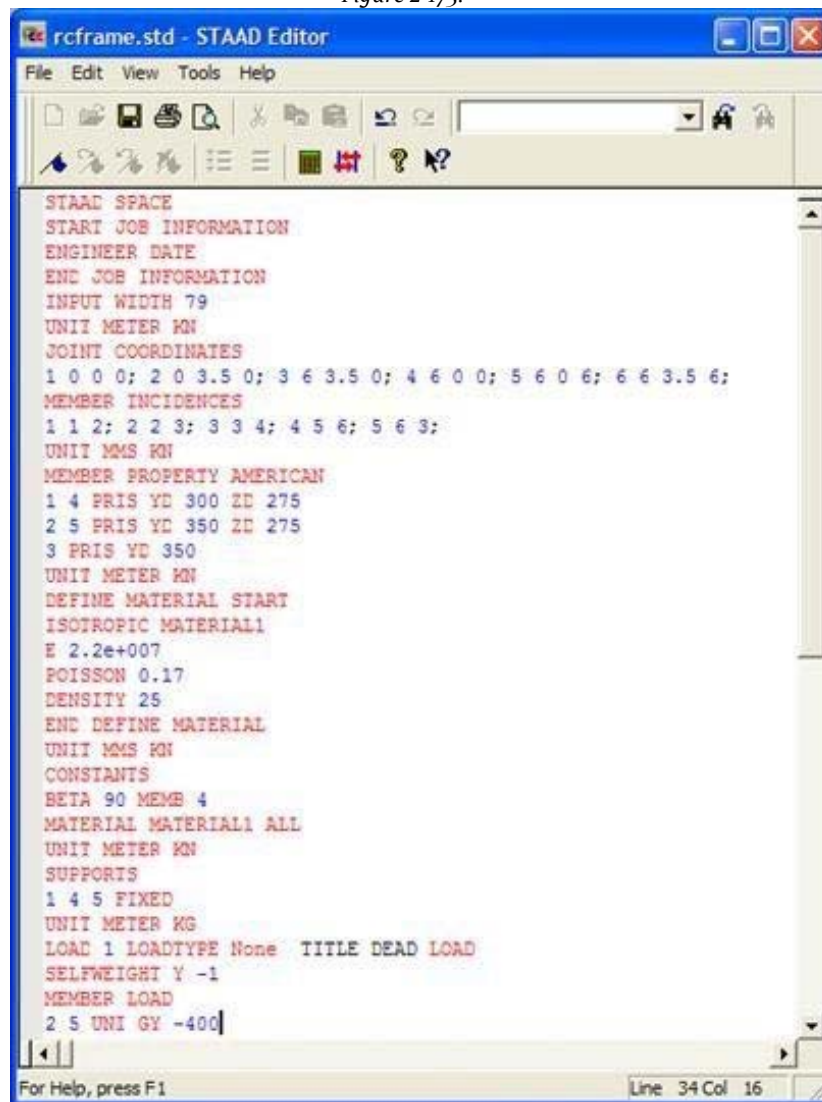
Changes to the data can be made in this window and saved.

2. Select **File> Exit** to close the Editor window.

As we saw in Section 2.1, we could also have created the same model relevant STAAD typing commands into a text file using the STAAD editor, or any external editor of your choice. To understand this method, proceed to the next section. If you want to skip that part, proceed to Section 2.9 where we perform the analysis and design of this model.

2.8 Creating the model using the file command

Now we use the method of file command to create the model for the above structure. The commands used in the script are described later in this section.

The STAAD.Pro command file can be created using the built-in editor, the procedure explained below in this section. Any text editor such as Notepad or WordPad can also be used to create the command file. However, the Editor STAAD.Pro command file syntax has the advantage as we write commands. The keywords, numeric data, etc. are shown in different colors in the STAAD.Pro Editor. Below is a typical screen editor is shown to illustrate the general appearance.

Figure 2-176:
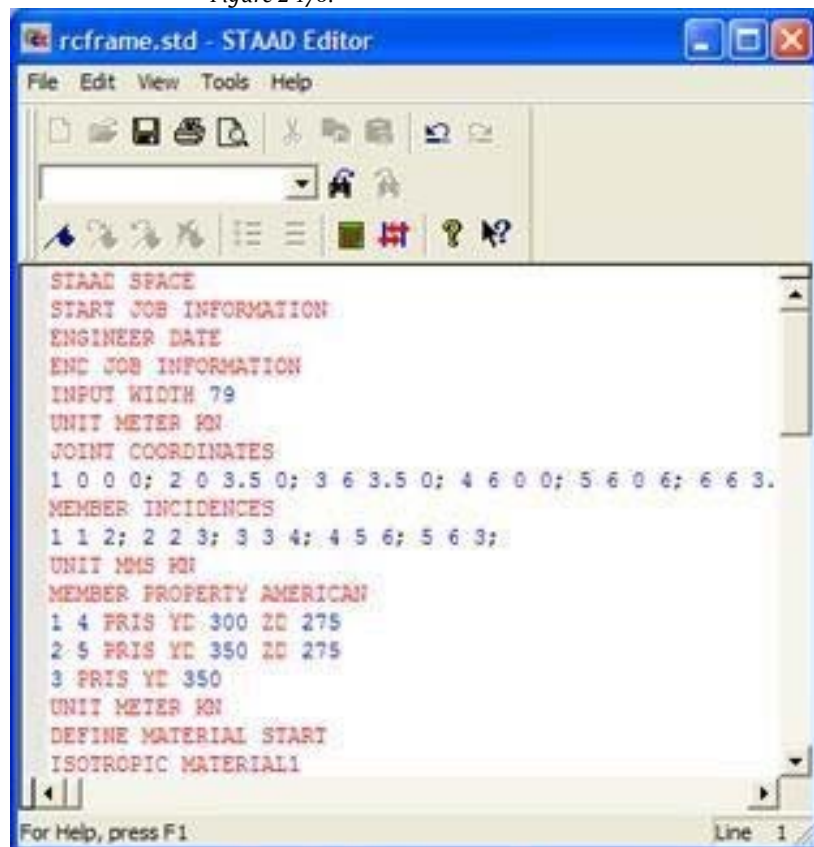
To access the built-in editor, start the program using the procedure explained in Section 2.2. Then follow step 1 of section 2.4.

Figure 2-177:
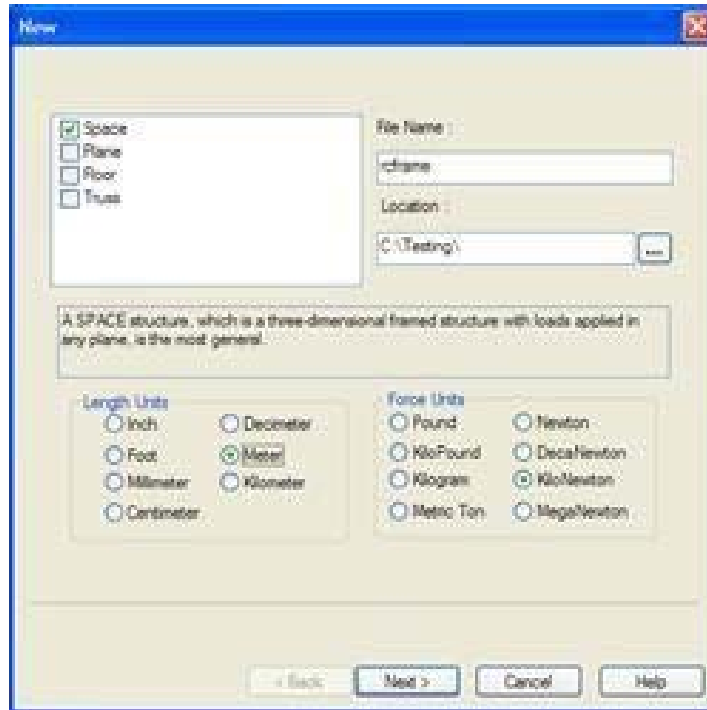
Then you will find the dialog box shown below. In this dialog box, choose Open STAAD Editor.

Figure 2-178:
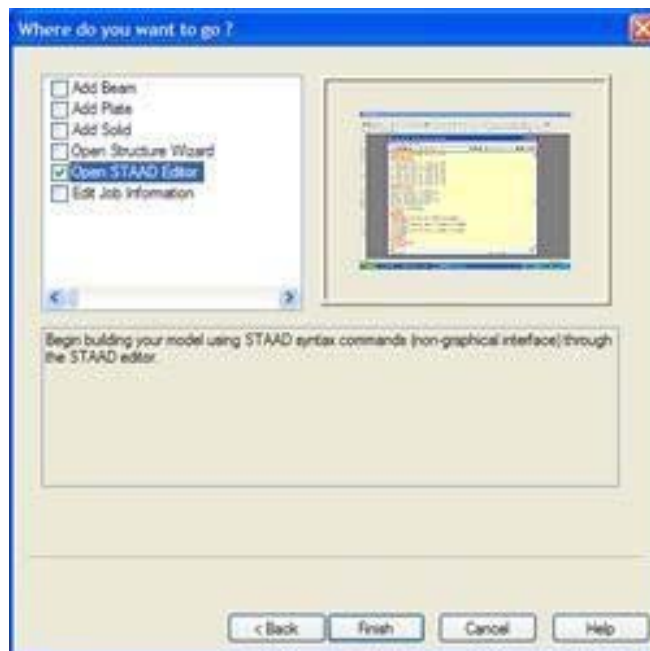
Editor screen opens as shown below.

Figure 2-179:
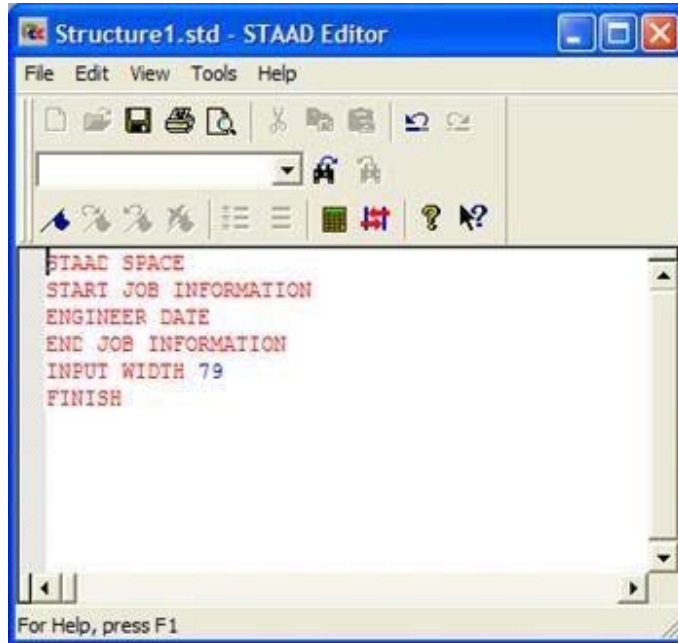
Deletes all lines of command in the editor window and the lines shown in bold below (do not have to delete the lines if you know what and where to keep the rest of the commands). The commands can be written in lowercase or uppercase. Usually all that is needed are the first three letters of a keyword, the rest of the Letters of the word are not necessary. The letters are underlined necessary. ("SPACE" = "SPA" = "space" = "spa")

Current entry points is shown in bold, followed by the explanation.

STAAD SPACE RC FRAMED STRUCTURE

Each entry must begin with the word STAAD. SPACE word means that the structure is a structure of space (three-dimensional) and the geometry defined by X, Y and Z coordinates.

UNIT METKN ER

Specifies the unit to use.

JOINT COORDINATES
1 0 0 0; 2 0 3.5 0; March 6 3.5 0
4 6 0 0; 5 6 0 6; June 6 3.5 6

Node number followed by the X, Y and Z are shown. Signs semicolon (;) are used as line separators. It allows us to offer multiple data sets in a single line.

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4
4 5 6; 5 6 3
```

Defines members are connected to the joints.

```
UNIT MMSKN
MEMBER PROAME PERTYRICAN
1 4 PRIS YD 300 ZD 275
2 5 PRIS YD 350 ZD 275
3 PRIS YD 350
```

The member properties defined above by the prismatic attribute to YD (D) and ZD (w) values are provided in MM unit. When have YD and ZD, STAAD considers the rectangular section. When one specified YD, the section is considered to be circular. Details are available in Section 5 of the Technical Manual STAAD.Pro V8i.

```
WITHStants
E22 MEMB 1 TO 5
```

The material constant E (modulus of elasticity) is specified as 22KN / sq.mm command followed CONSTANTS.

```
UNIT   METER   KN
WITHStants
DEN25.0 SITY ALL
POISSON 0.17 ALL
```

The unit of length is changed to METER MMS to facilitate entry of the density. Next, the Poisson ratio is specified.

```
BETA 90 MEMB 4
```

In the absence of any explicit instruction, STAAD guide the beams and columns of the structure in a predefined manner (see section 1 of the Technical Manual for more

information.) To guide the member 4 so that its longer edges (parallel to local Y axis sides) are parallel to the global Z axis, we have to apply a 90 degree angle beta.

SUPPORT

1 4 5 FIXED

Nodes 1, 4 and 5 are defined as recessed support.

UNIT   METER   Kg

LOADEAD LOAD D1

Force units are changed to KG KN to facilitate entry of loads. The load case 1 begins with an accompanying title.

SELFWEIGHT Y -1

One component of the load case 1 is the weight of the structure acting on the overall direction by a factor of -1.0. Since Y is overall vertically upward, the factor of -1.0 indicates that the load will act downward.

MEMBER LOAD

2 5 UNIGY -400

Load 1 contains member loads too. GY indicates that the burden is on the overall direction Y. The word means uni UNI loadformly distributed. Loads are applied in the Member 2 and 5.

LOAD2 LIVE LOAD

The load case 2 starts along with a companion title.

MEMBER LOAD

May 2 UNIGY -600

Charge 2 also contains loads of members. GY indicates that the burden is on the overall direction. The word UNI stands for uniformly distributed load. Loads are applied in the Member 2 and 5.

LOAD3 WIND LOAD

The load case 3 starts along with a companion title.

```
MEMBER
LOAD1 UNIGX
300
4 UNIGX 500
```

Charge 3 members also contains fillers. GX indicates load in the X direction is global. The word UNI is meant to evenly distributed load. Loads are applied to members 1 and 4.

```
LOAD 4  DEAD + LIVE
```

The load case 4 starts along with a companion title.

```
REPEAT LOAD
1 1.2 2 1.5
```

When loading 4 illustrates the technique used to Staad instruct to create a case consisting of charging data to be assembled from other load cases specified above. We are instructing the program to examine the structure loads of cases 1 and 2 acting simultaneously. The data values of load load case 1 multiplied by

factor 1.2, And the resulting values are used in the case of load 4 also values

Data load load case 2 are multiplied by a factor of 1.5, And the values resulting are also used in case of load 4.

```
LOAD 5 DEAD + WIND
```

The load case 5 starts with an accompanying title.

```
REPEAT LOAD
1 1.1 3 1.3
```

We are instructing the program to examine the structure loads of cases 1 and 3 acting simultaneously.

```
PDELTA ANALYSIS
```

The command is an instruction PDELTA analysis program to execute a second-order analysis and account for P-delta effects.

```
LOAD LIST May 4
```

The previous LOAD LIST command is a way of affirming that all calculations should be based

results of load cases 4 and 5 only. The intention is to restrict concrete design calculations for load cases 4 and 5 only.

```
START   WITHCRETE  DESIGN
CODE ACI

UNIT    MMSNEWTON
CLT 25 ALL
CLB 30 ALL
CLS 25 ALL
FC 25 ALL
FYMAIN 415 ALL
TRACK 1 ALL
```

The first thing is to align the command that starts the operation of concrete design. The values for the design parameters of the concrete defined in the above commands. The design is made according to the ACI code. Length units are switched from the meter to MMS to facilitate entry of the design parameters. Similarly, the units of force to change KG NEWTON. The TRACK value determines the scope of the information related to the project to be produced by the program at the start. The parameters specified include CLT (Clear cover for top surface - superior surface coverage), CLB (Clear cover for bottom surface - bottom cover), CLS (Clear cover for sides - side cover), FC (Strength of concrete - Effort to Concrete Compressive Strength at 28 days), and FYMAIN (Ultimate strength of steel - ultimate strength of reinforcing steel). These parameters are described in Section 3 of the Technical Manual.

```
DESIGN BEAM May 2
DESIGN AUUMN 1 April 3
```

The above commands indicate a program to design the beams 2 and 5 for bending, shear and torsion, and design the columns 1, 3 and 4 for axial and biaxial flexure.

```
ENDCONCRETE DESIGN
```

This command terminates the operation of concrete design.

```
EndISH
```

This command terminates the STAAD run. Let's save the file and exit the editor.

2.9 Performing analysis and design

STAAD.Pro performs analysis and design simultaneously. In order to perform analysis and design, select **Run Analysis** (Perform analysis) on the menu **Analyze**.

Figure 2-180:

> **Tip**: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

As the analysis progresses, several messages appear on the screen as shown in the following figure.

Figure 2-181:
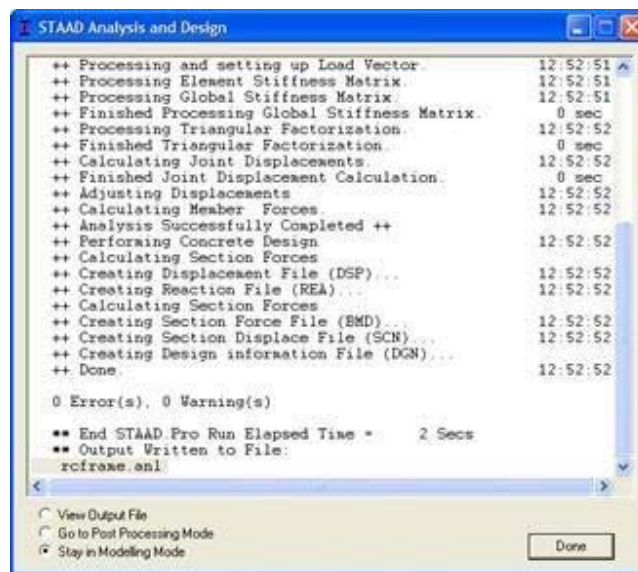
Note that we can choose from the three options available in the dialog box above:

Figure 2-182:
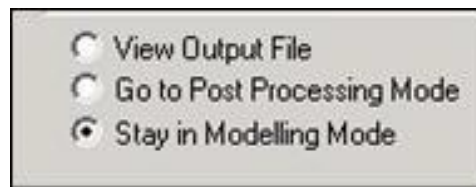
These options are indicative of what will happen after we click the button **Done**.

Option *View Output File* allows us to see the output file created by STAAD. The output file contains the numerical results produced in response to different input commands that were specified during model generation. It also tells us if errors are found, if the analysis and design was completed successfully or not. Section 2.10 provides additional details on viewing and understanding the contents of the output file.

Option *Go to Post Processing Mode* lets go to the graphic part of the program known as the postprocessor. This is where one can exhaustively verify the results, see the results graphically, draw diagrams of results, produce reports, etc. Section 2.11 explains how post-processing in detail.

Stay in Modelling Mode allows us to continue in the mode of generation of the program model (in which they are present) in case we want to make more changes in our model.

2.10 Viewing the output file

During the analysis process, STAAD.Pro creates an output file. This file contains important information about whether the analysis was successful. For example, if STAAD.Pro encounters an instability problem during the analysis process will be reported in the output file.

We can access the output file using the method explained at the end of the previous section. Alternatively, you can select the option **File> View> Output File> STAAD Output** the top menu or go directly to the command **STAAD Output** 🔲 placing in the toolbar. The output file for the problem we have encountered is shown in the following pages.

Figure 2-183:
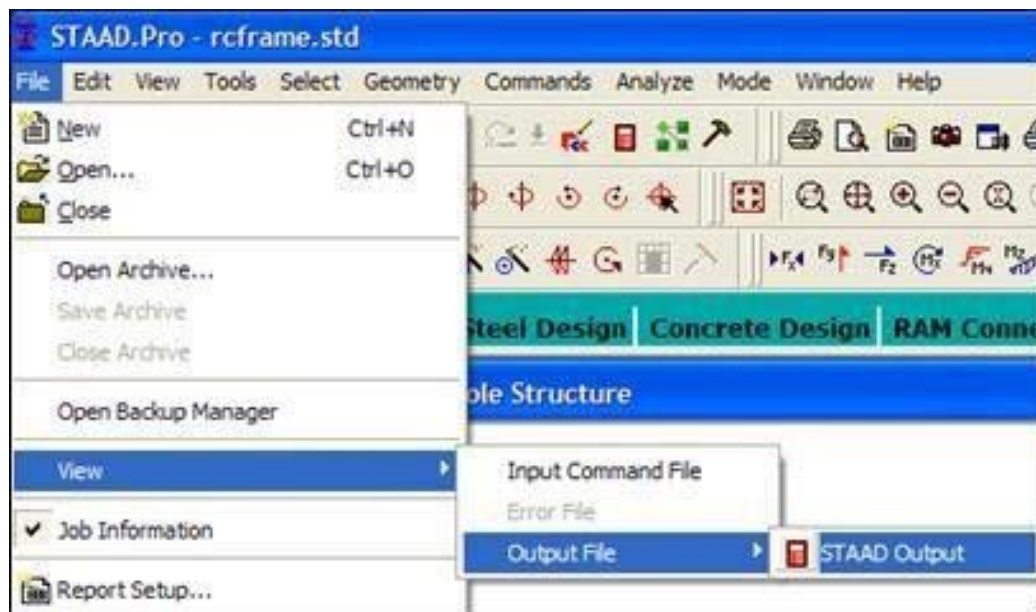
The output file is displayed through a file viewer called *SproView*. This display lets you set the text font for the whole file and print the output file to a printer. Use the appropriate menu option *Archive* from the menu bar.

Figure 2-184:
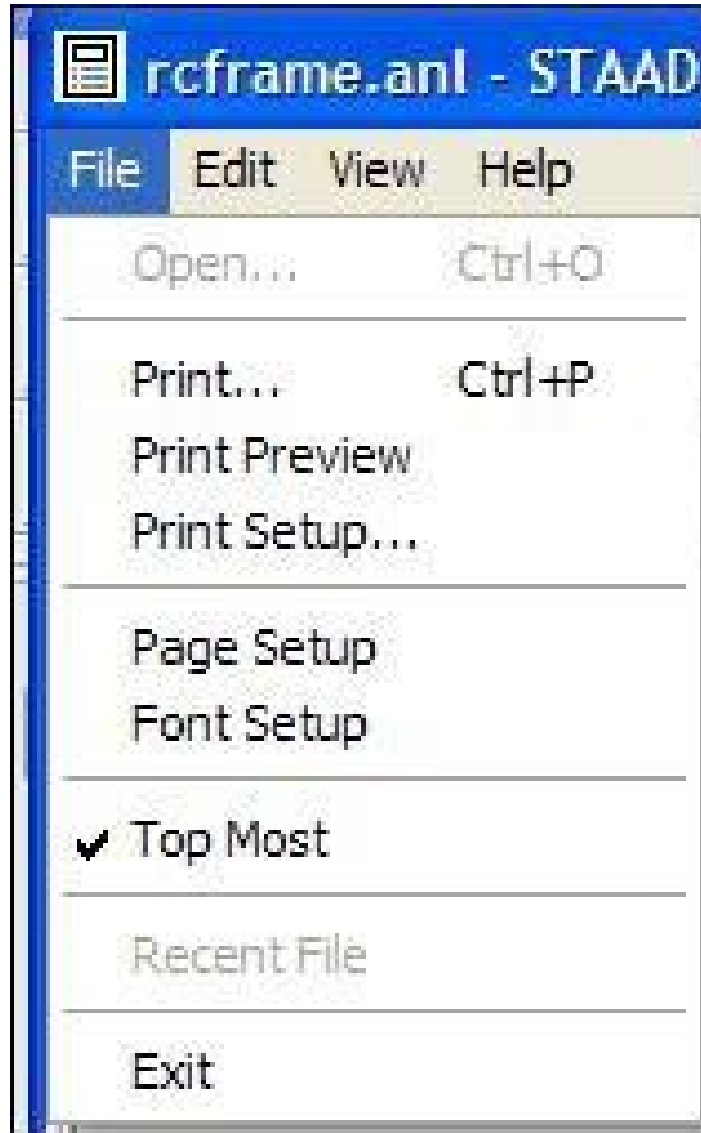
By default, the output file contains a list of all the input too. You can choose not to print the echo of the commands into the output file. Select **Commands> Miscellaneous> September Echo** from the menu bar and select the

button **Echo Off**.

It is very important that we sail across the output file and ensure that the results seem reasonable, no error messages or warnings reported, etc. Errors found during the analysis & design can disable access to post-processing mode - graphical displays which can graphically display the results. The information presented in the output file is a crucial indicator of whether or not the structure meets the requirements of engineering safety and ease of maintenance.

```
***************************************************
*                                                 *
*         STAAD.PRO V8I SELECTSERIES1             *
*                                                 *
*         VERSION  20.07.06.35                    *
*                                                 *
*         PROPRIETARY PROGRAM OF                  *
*                                                 *
*         BENTLEY SYSTEMS, INC.                   *
*                                                 *
*         DATE=   JUL  7, 2010                    *
*                                                 *
*         TIME=   11:37:11                        *
*                                                 *
*                                                 *
*         USER ID: BENTLEY SYSTEMS                *
*                                                 *
***************************************************

    1. STAAD SPACE RC FRAMED STRUCTURE
INPUT FILE: TUT_02_RCFRAME.STD
    2. START JOB INFORMATION
    3. ENGINEER DATE 16-FEB-02
    4. END JOB INFORMATION
    5. INPUT WIDTH 79
    6. UNIT METER KN
    7. JOINT COORDINATES
    8. 1 0 0 0; 2 0 3.5 0; 3 6 3.5 0; 4 6 0 0; 5 0 0 6; 6 6 3.5 6
    9. MEMBER INCIDENCES
   10. 1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 3
   11. UNIT MMS KN
   12. MEMBER PROPERTY AMERICAN
   13. 1 4 PRIS YD 300 ZD 275
   14. 2 5 PRIS YD 350 ZD 275
   15. 3 PRIS YD 350
   16. CONSTANTS
   17. E 22 MEMB 1 TO 5
   18. UNIT METER KN
   19. CONSTANTS
   20. DENSITY 25.0 ALL
   21. POISSON 0.17 ALL
```

```
22. BETA 90 MEMB 4

23. SUPPORTS

24. 1 4 5 FIXED

25. UNIT METER KG

26. LOAD 1 DEAD LOAD

27. SELFWEIGHT Y -1

28. MEMBER LOAD

29. 2 5 UNI GY -400

30. LOAD 2 LIVE LOAD

31. MEMBER LOAD

32. 2 5 UNI GY -600

33. LOAD 3 WIND LOAD

34. MEMBER LOAD

35. 1 UNI GX 500

36. 4 UNI GX 500

37. LOAD 4 DEAD + LIVE

38. REPEAT LOAD

39. 1 1.2 2 1.5

40. LOAD 5 DEAD + WIND

41. REPEAT LOAD

42. 1 1.1 3 1.3

43. PDELTA ANALYSIS


      P R O B L E M   S T A T I S T I C S
      ------------------------------------

   NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =     6/     5/     3

         SOLVER USED IS THE IN-CORE ADVANCED SOLVER

   TOTAL PRIMARY LOAD CASES =    5, TOTAL DEGREES OF FREEDOM =     18
++ ADJUSTING DISPLACEMENTS                         11:37:11


44. LOAD LIST 4 5

45. START CONCRETE DESIGN

46. CODE ACI

47. UNIT MMS NEWTON

48. CLT 25 MEMB 1 TO 5

49. CLB 30 MEMB 1 TO 5

50. CLS 25 ALL

51. FC 25 MEMB 1 TO 5

52. FYMAIN 415 MEMB 1 TO 5

53. TRACK 1 ALL
```

```
 54. DESIGN BEAM 2 5


 ========================================================================

        BEAM   NO.     2 DESIGN RESULTS - FLEXURE PER CODE ACI 318-05

 LEN -  6000. MM  FY -  415.  FC -  25.  MPA, SIZE -  275. X  350. MMS

 LEVEL    HEIGHT     BAR INFO        FROM          TO          ANCHOR
          (MM)                       (MM)         (MM)         STA  END

 _____

   1        51.    2 - 16MM       312.         5438.         NO   NO
 |-------------------------------------------------------------------|
 |   CRITICAL POS MOMENT=     39.24 KN-MET  AT  3000.MM, LOAD    4|
 |   REQD STEEL=   367.MM2, RHO=0.0045, RHOMX=0.0193 RHOMN=0.0033 |
 |   MAX/MIN/ACTUAL BAR SPACING=   273./   41./  184. MMS         |
 |   REQD. DEVELOPMENT LENGTH =   480. MMS                        |
 |-------------------------------------------------------------------|

   CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =    20964.0 CM^4

   2       306.    3 - 12MM        0.          860.         YES  NO
 |-------------------------------------------------------------------|
 |   CRITICAL NEG MOMENT=     33.29 KN-MET  AT    0.MM, LOAD    4|
 |   REQD STEEL=   302.MM2, RHO=0.0036, RHOMX=0.0193 RHOMN=0.0033 |
 |   MAX/MIN/ACTUAL BAR SPACING=   273./   37./   94. MMS         |
 |   REQD. DEVELOPMENT LENGTH =   360. MMS                        |
 |-------------------------------------------------------------------|

   CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =    19148.5 CM^4

   3       306.    3 - 12MM      4390.         6000.         NO   YES
 |-------------------------------------------------------------------|
 |   CRITICAL NEG MOMENT=     36.17 KN-MET  AT 6000.MM, LOAD    4|
 |   REQD STEEL=   329.MM2, RHO=0.0039, RHOMX=0.0193 RHOMN=0.0033 |
 |   MAX/MIN/ACTUAL BAR SPACING=   273./   37./   94. MMS         |
 |   REQD. DEVELOPMENT LENGTH =   360. MMS                        |
 |-------------------------------------------------------------------|

   CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =    19148.5 CM^4


       B E A M   N O.    2 D E S I G N   R E S U L T S - SHEAR

 AT START SUPPORT - VU=   43.94 KNS  VC=   69.67 KNS  VS=    0.00 KNS
 TU=    2.55 KN-MET  TC=    2.4 KN-MET  TS=    3.4 KN-MET  LOAD    4
```

```
    STIRRUPS ARE REQUIRED FOR TORSION.

    REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1.

    PROVIDE 12 MM 2-LEGGED STIRRUPS AT 124. MM  C/C FOR 2705. MM

    ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE =   0.79 SQ.CM.

    AT END   SUPPORT - VU=   44.90 KNS  VC=   69.19 KNS  VS=    0.00 KNS

    TU=    2.55 KN-MET  TC=    2.4 KN-MET  TS=    3.4 KN-MET  LOAD     4

    STIRRUPS ARE REQUIRED FOR TORSION.

    REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1.

    PROVIDE 12 MM 2-LEGGED STIRRUPS AT 124. MM  C/C FOR 2705. MM

    ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE =   0.79 SQ.CM.
  ___    2J_____    6000X 275X  350_____    3J____
  |                                                                         |
  ||========                                   ===================||
  | 3N012 H 306.   0.TO  860                    3N012|H|306.4390.TO 6000 |
  | 23*12C/C124                                 | | |23*12C/C124    |
  |   2N016 H  51. 312.TO 5438                  | | | | | | | |      |
  |  =================================================================  |
  |                                                                         |
  |_____|


   _____    _____    _____    _____    _____
  |       |  | |       |  | |       |  | |       |  | |       |
  |  OOO  |  | |       |  | |       |  | |  OOO  |  | |  OOO  |
  | 3#12  |  | |       |  | |       |  | | 3#12  |  | | 3#12  |
  |       |  | |       |  | |       |  | |       |  | |       |
  |       |  | | 2#16  |  | | 2#16  |  | | 2#16  |  | |       |
  |       |  | |  OO   |  | |  OO   |  | |  OO   |  | |       |
  |       |  | |       |  | |       |  | |       |  | |       |
  |_____|  |_____|   |_____|   |_____|   |_____|


  =====================================================================
        BEAM  NO.    5 DESIGN RESULTS - FLEXURE PER CODE ACI 318-05

  LEN - 6000. MM  FY - 415.  FC - 25.  MPA, SIZE - 275. X  350. MMS

  LEVEL   HEIGHT    BAR INFO      FROM          TO          ANCHOR
          (MM)                    (MM)          (MM)        STA  END
  _____

   1       51.   2 - 16MM      312.        5438.        NO   NO
  |-----------------------------------------------------------|
  |  CRITICAL POS MOMENT=     39.24 KN-MET  AT  3000.MM, LOAD    4|
  |  REQD STEEL=   367.MM2, RHO=0.0045, RHOMX=0.0193 RHOMN=0.0033 |
```

```
|   MAX/MIN/ACTUAL BAR SPACING=   273./   41./  184. MMS          |
|   REQD. DEVELOPMENT LENGTH =   480. MMS                         |
|----------------------------------------------------------------|

  CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =   20964.0 CM^4

  2      306.    3 - 12MM          0.          860.       YES  NO
|----------------------------------------------------------------|
|   CRITICAL NEG MOMENT=      33.29 KN-MET  AT    0.MM, LOAD   4|
|   REQD STEEL=   302.MM2, RHO=0.0036, RHOMX=0.0193 RHOMN=0.0033 |
|   MAX/MIN/ACTUAL BAR SPACING=   273./   37./   94. MMS          |
|   REQD. DEVELOPMENT LENGTH =   360. MMS                         |
|----------------------------------------------------------------|

  CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =   19148.5 CM^4
  3      306.    3 - 12MM      4890.          6000.       NO   YES
|----------------------------------------------------------------|
|   CRITICAL NEG MOMENT=      36.17 KN-MET  AT 6000.MM, LOAD   4|
|   REQD STEEL=   329.MM2, RHO=0.0039, RHOMX=0.0193 RHOMN=0.0033 |
|   MAX/MIN/ACTUAL BAR SPACING=   273./   37./   94. MMS          |
|   REQD. DEVELOPMENT LENGTH =   360. MMS                         |
|----------------------------------------------------------------|

  CRACKED MOMENT OF INERTIA IZ AT ABOVE LOCATION =   19148.5 CM^4


      B E A M  N O.    5 D E S I G N  R E S U L T S - SHEAR

AT START SUPPORT - VU=   18.56 KNS  VC=   69.78 KNS  VS=    0.00 KNS
TU=    4.74 KN-MET  TC=    2.4 KN-MET  TS=    6.3 KN-MET  LOAD    5
STIRRUPS ARE REQUIRED FOR TORSION.
REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1.
PROVIDE 12 MM 2-LEGGED STIRRUPS AT 124. MM  C/C FOR 2705. MM
ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE =   1.47 SQ.CM.
AT END   SUPPORT - VU=   19.11 KNS  VC=   69.12 KNS  VS=    0.00 KNS
TU=    4.74 KN-MET  TC=    2.4 KN-MET  TS=    6.3 KN-MET  LOAD    5
STIRRUPS ARE REQUIRED FOR TORSION.
REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1.
PROVIDE 12 MM 2-LEGGED STIRRUPS AT 124. MM  C/C FOR 2705. MM
ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE =   1.47 SQ.CM.
___  6J_____    6000X 275X  350_____    3J____
|                                                                |
||========                                         ==============||
| 3N012 H 306.   0.TO  860                 3N012 H 306.4890.TO 6000 |
```

```
| 23*12C/C124                                    23*12C/C124   |
|   2N016 H  51. 312.TO 5438                     | | | |       |
|   =================================================================   |
|                                                              |
|_____|

  _____   _____   _____   _____   _____
|        | |       | |       | |       | |       |        |
|  OOO   | |       | |       | |       | |       |  OOO   |
| 3#12   | |       | |       | |       | |       | 3#12   |
|        | |       | |       | |       | |       |        |
|        | | 2#16  | | 2#16  | | 2#16  | |       |        |
|        | |  OO   | |  OO   | |  OO   | |       |        |
|        | |       | |       | |       | |       |        |
|_____| |_____| |_____| |_____| |_____| |_____|


  ********************END OF BEAM DESIGN****************************


  55. DESIGN COLUMN 1 3 4

  =================================================================

      COLUMN  NO.    1  DESIGN PER ACI 318-05 - AXIAL + BENDING

  FY - 415.0  FC -  25.0 MPA,  RECT SIZE - 275.0 X 300.0 MMS, TIED
        AREA OF STEEL REQUIRED =   882.8  SQ. MM

  BAR CONFIGURATION      REINF PCT.   LOAD   LOCATION   PHI
  ------------------------------------------------------------

   8 - 12 MM              1.097       4      END    0.650
  (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
  TIE BAR NUMBER   12 SPACING 192.00 MM

     COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)

  -----------------------------------------------------------

   P0        PN MAX   P-BAL.    M-BAL.     E-BAL. (MM)
  2109.38   1687.50   785.43    97.75     124.5
   M0        P-TENS.  DES.PN    DES.MN    E/H
   47.51    -375.48    75.05    51.22    0.19501
  -----------------------------------------------------------

     COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)

  -----------------------------------------------------------
   P0        PN MAX   P-BAL.    M-BAL.     E-BAL. (MM)
```

```
  2109.38   1687.50    775.80     88.78     114.4
    M0        P-TENS.   DES.PN    DES.MN     E/H
   43.01     -375.48    75.05      3.92     0.01493
   ------------------------------------------------------


 ======================================================================

      COLUMN  NO.    3  DESIGN PER ACI 318-05 - AXIAL + BENDING

 FY - 415.0  FC -   25.0 MPA,  CIRC SIZE   350.0 MMS DIAMETER  TIED
        AREA OF STEEL REQUIRED =  1096.8  SQ. MM

 BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
 ---------------------------------------------------------

  10 - 12 MM                 1.176         4     STA    0.650
 (EQUALLY SPACED)
 TIE BAR NUMBER   12 SPACING 192.00 MM

     COLUMN INTERACTION: MOMENT ABOUT Z/Y -AXIS (KN-MET)
   --------------------------------------------------------

   P0        PN MAX    P-BAL.    M-BAL.    E-BAL. (MM)
  2489.81   1991.85    939.46    109.84     116.9
    M0        P-TENS.   DES.PN    DES.MN     E/H
   62.23     -469.35    153.05     73.16    0.09657
   --------------------------------------------------------


 ======================================================================

      COLUMN  NO.    4  DESIGN PER ACI 318-05 - AXIAL + BENDING

 FY - 415.0  FC -   25.0 MPA,  RECT SIZE - 275.0 X 300.0 MMS, TIED
        AREA OF STEEL REQUIRED =  1056.0  SQ. MM

 BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
 ---------------------------------------------------------

   4 - 20 MM                 1.523         5     STA    0.650
 (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
 TIE BAR NUMBER   12 SPACING 192.00 MM

     COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)
   --------------------------------------------------------

   P0        PN MAX    P-BAL.    M-BAL.    E-BAL. (MM)
  2247.93   1798.34    764.68    123.35     161.3
    M0        P-TENS.   DES.PN    DES.MN     E/H
   63.35     -521.51    43.92      10.51    0.06838
```

```
----------------------------------------------------------

    COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)

----------------------------------------------------------

   P0         PN MAX     P-BAL.    M-BAL.     E-BAL. (MM)
2247.93     1798.34     755.33    111.44     147.5
    M0        P-TENS.    DES.PN    DES.MN     E/H
  57.35     -521.51      43.92     47.49     0.30890

----------------------------------------------------------


*********************END OF COLUMN DESIGN RESULTS********************

 56. END CONCRETE DESIGN
 57. FINISH

         *********** END OF THE STAAD.PRO RUN ***********

          **** DATE= JUL  7,2010   TIME= 11:37:12 ****
         ***********************************************************
         *           FOR QUESTIONS ON STAAD.PRO, PLEASE CONTACT    *
         *       BENTLEY SYSTEMS OFFICES AT THE FOLLOWING LOCATIONS *
         *                                                         *
         *                TELEPHONE           WEB / EMAIL          *
         *                                                         *
         *  USA:       +1 (714)974-2500                           *
         *  UK         +44(1454)207-000                           *
         *  SINGAPORE +65 6225-6158                               *
         *  EUROPE     +31 23 5560560                             *
         *  INDIA      +91(033)4006-2021                          *
         *  JAPAN      +81(03)5952-6500    HTTP://WWW.CTC-G.CO.JP *
         *  CHINA      +86 10 5929 7000                           *
         *  THAILAND   +66(0)2645-1018/19 PARTHA.P@REISOFTWARETH.COM *
         *                                                         *
         * WORLDWIDE    HTTP://SELECTSERVICES.BENTLEY.COM/EN-US/   *
         *                                                         *
         ***********************************************************
```

2.11 Post-Processing

STAAD.Pro provides ample facilities for verification and visualization of results. These facilities are accessible from the post-processing mode. The post-processing mode is used to check the results of analysis and design and generate reports.

For this tutorial, we must perform the following tasks:

2.11.1 Go to the post-processing mode

At the end of section 2.9, we saw how one could go directly from the analysis window to the screen postprocessing. However, you can access the post-processing mode using the following procedure at any time.

Procedure:

1. Select Tool **Post-Processing** or**Mode> Post Processing**. Table Settings dialog opens results.

Figure 1.95

Figure 2-185:
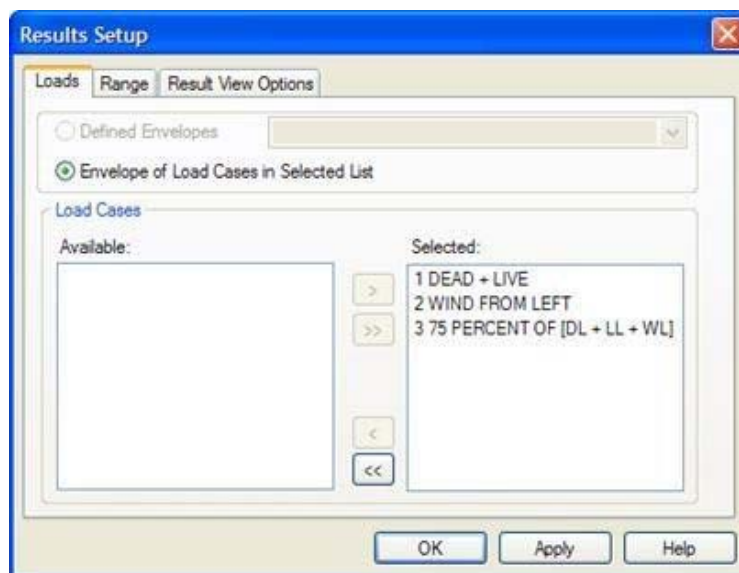
2. Select the load cases for which to display the results. For this tutorial, click [>] to select all load cases and click **OK**.

2.11.2 Viewing the chart deflections

The screen will now look like the figure shown below.

Figure 2-186:
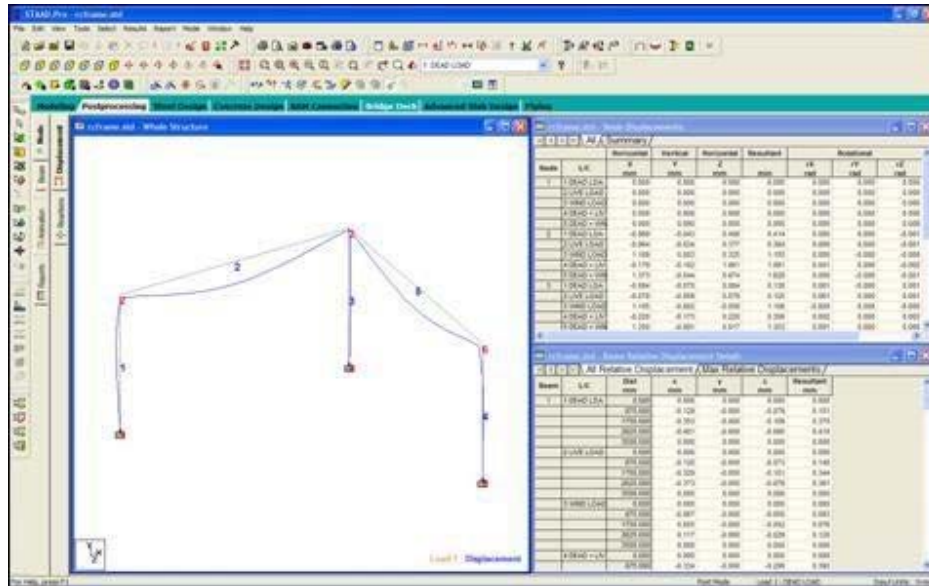
The chart is currently on display diagram displaced nodes and elements for load case (DEAD LOAD) 1.

To return to this particular scheme, select the page **Node | Displacement** along the page control area on the left side.

Figure 2-187:

or**Results> Deflection**.

2.11.3 Switching between load cases for diagram deflection

Steps:

1. To change the load to that seen in the deviation diagram, select the desired loading wishlist

   active load

Figure 2-188:

o
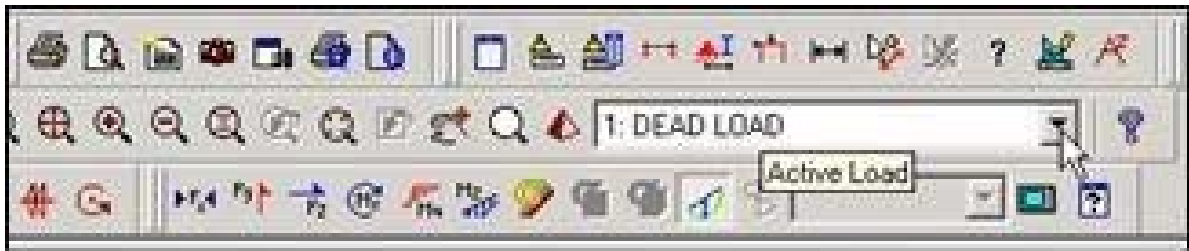
Select the tool **Symbols and Labels**

Figure 2 100

o

Select **View> Structure Diagrams**

For the latter two options, the dialog box opens diagrams.

2. Select the tab **Loads and Results** and choose the desired load case list box load case.

Figure 2-189:
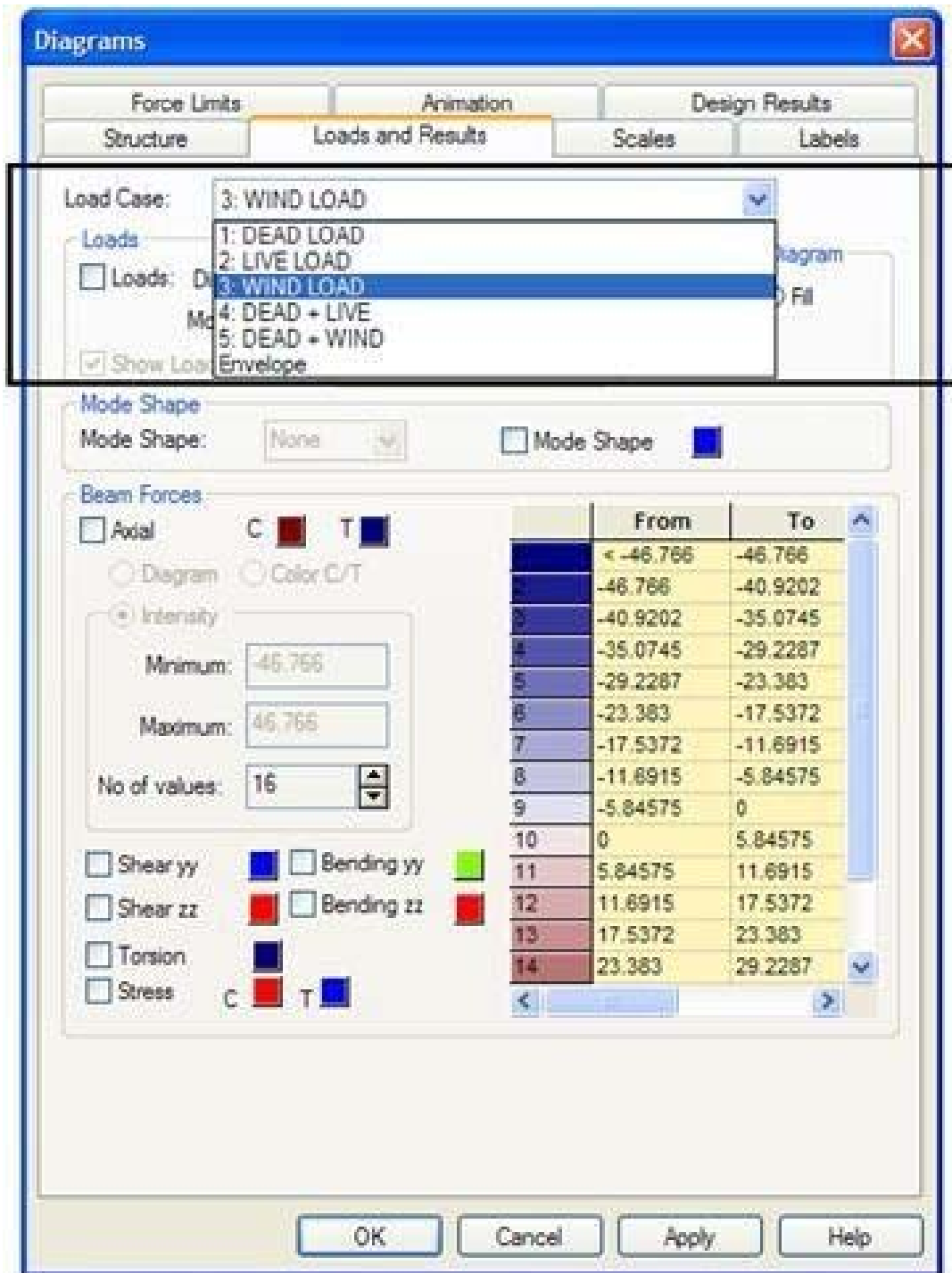
3. Click on **OK**.

    The following figure shows the deflected shape of the structure in the case of load 3.

Figure 2-190:
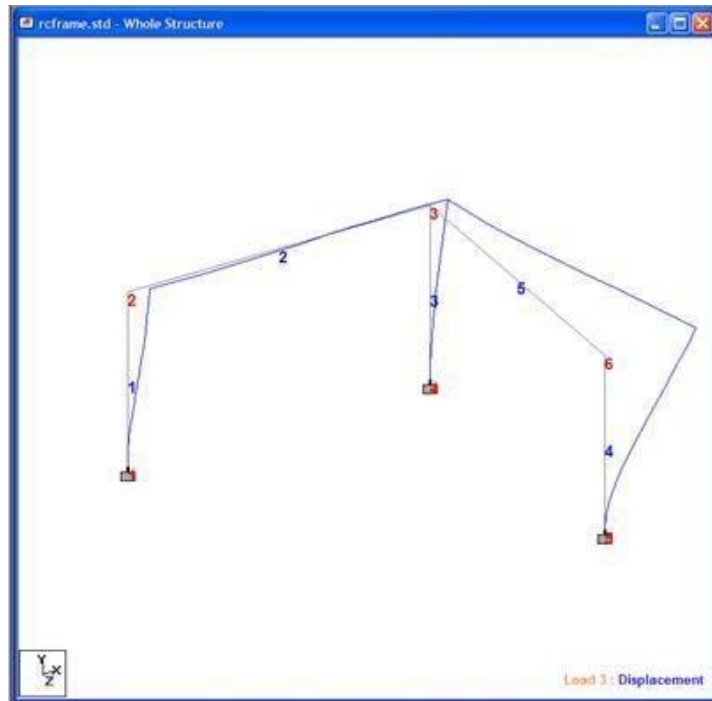
4. To show deflection say, load case 5 (DEAD + WIND), repeat steps 1-3 and select load case 5.

The deflection of the load case 5 will now be displayed in the model as shown in the following figure.

Figure 2-191:
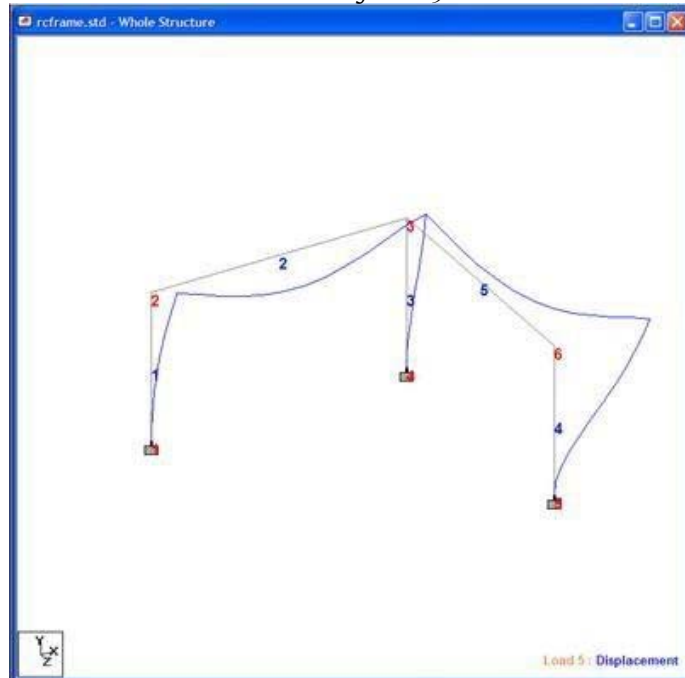
2.11.4 Resize deflection diagram

Steps:

If the chart appears very imperceptible, it may be because you can draw a very small scale. To change the scale of the plot of the deviation, you can

1. Select the tool **Scale**

   Or choose *Scale* from menu *Results*

Figure 2-192:

Figure 2-193:

Or select **View> Structure Diagrams> Scales**. The dialog box opens diagrams

the tab stops.

Figure 2-194:
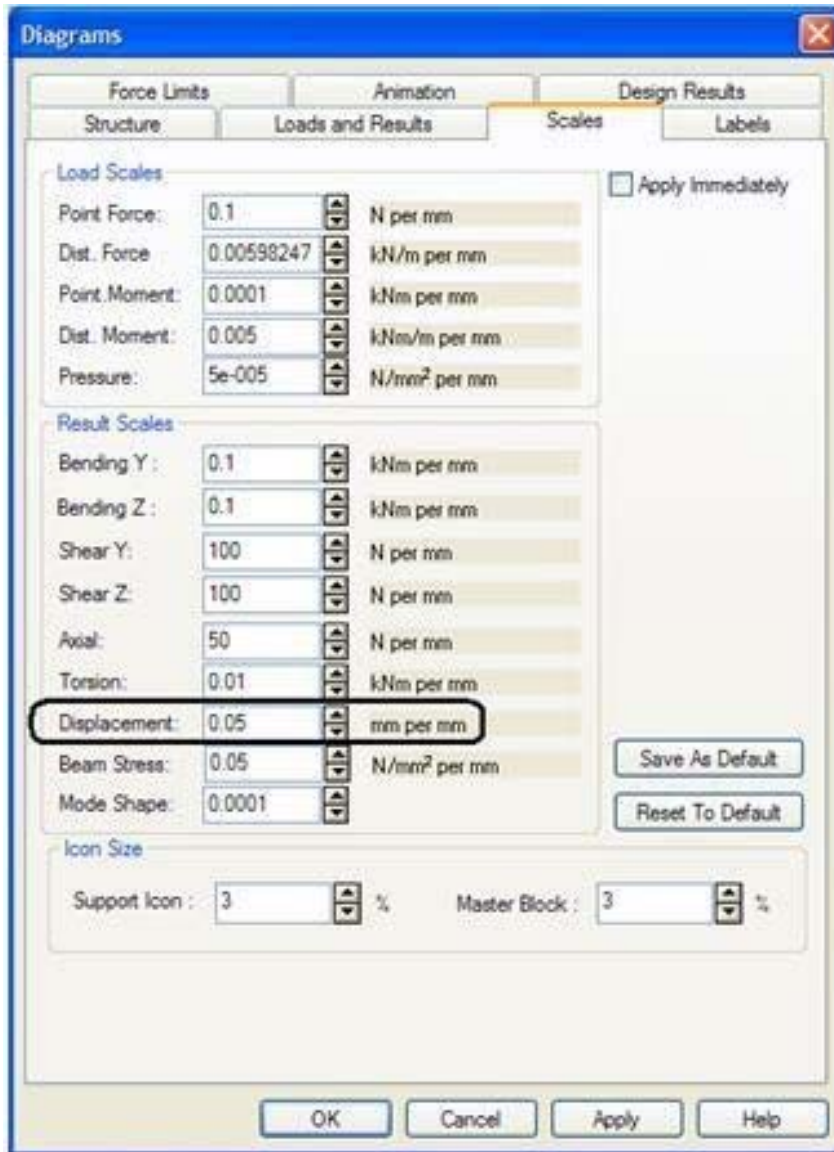
2. In the field of *Displacement* (Displacement), specify a smaller number of what is currently on the list and click **OK**.

The deflection diagram should now be larger.

In the tab *Scales* in the dialog box *Diagrams* If you set checkbox **Apply Immediately** Pressing the up or down buttons associated parameter will produce immediate results in terms of a larger retail or diagram (Preview).

Figure 2-195:
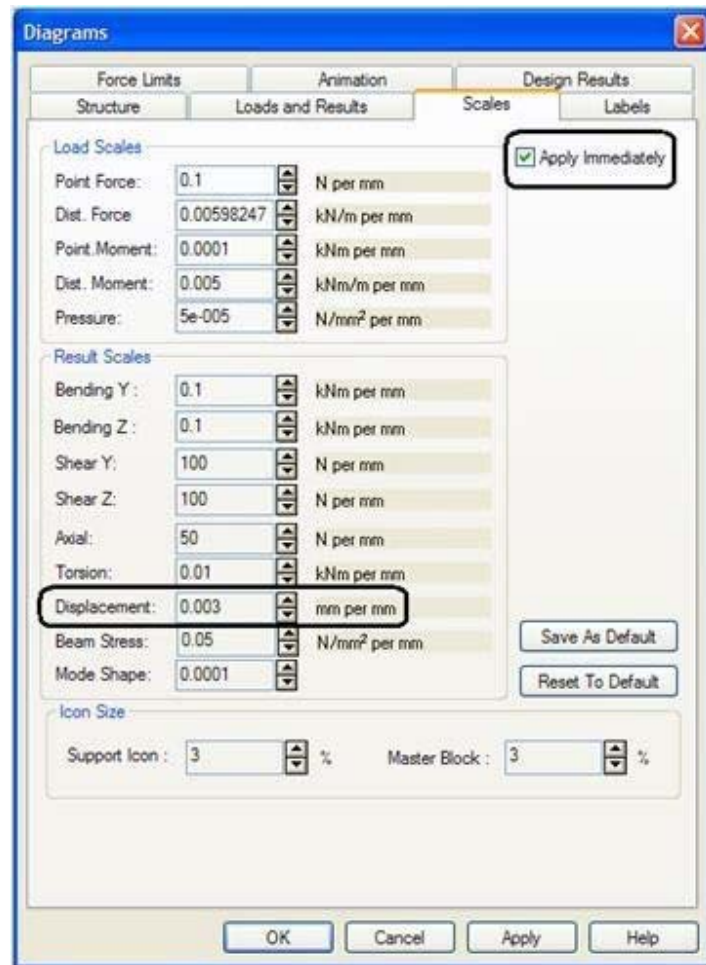
2.11.5 Annotation displacements

L to Annotation is the process of viewing the offset values on the screen.

Procedure:

1. Select **Results> View Value ...**.

2. The following dialog box opens. From the Record *Ranges*Select **all** nodes (all nodes)

   Tip: If you want to annotate deflection few nodes, node numbers specified in the node list.

Figure 2-196:
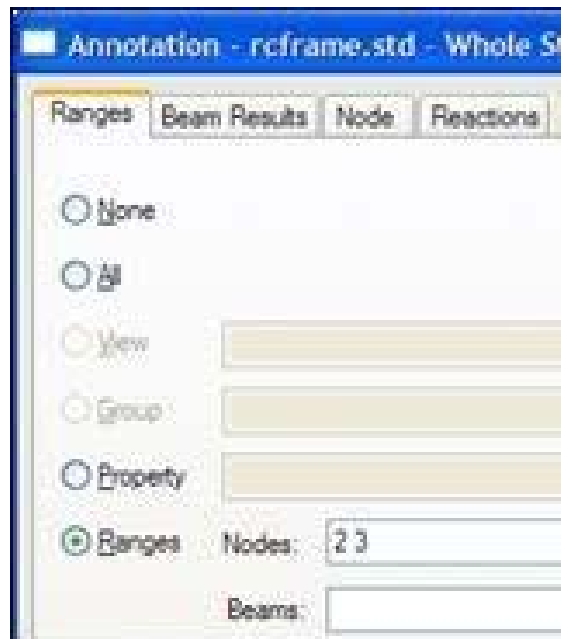
1. Select the tab **Node** and set the option **Resultant**.

Figure 2-197:
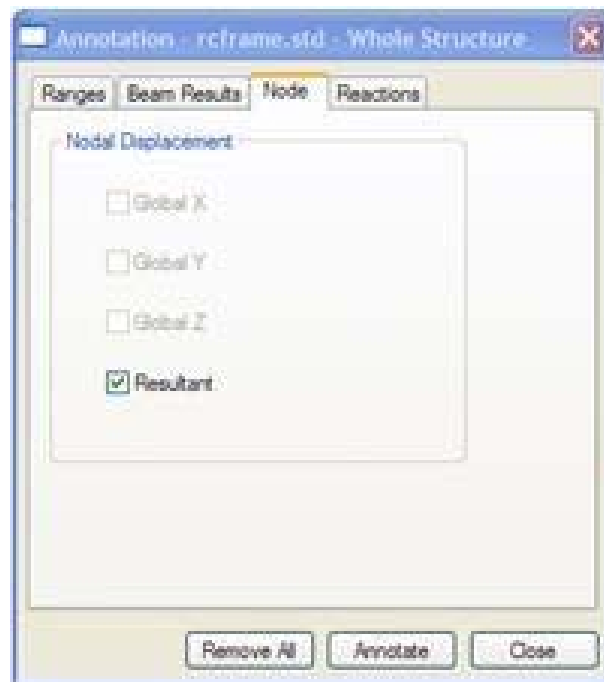
The result represents the square root of the sum of the squares of the values of X, Y and Z displacements.

2. Click on **Annotate** and then click **Close**.

The deflection pattern structure is recorded for the load case 2 as in the figure below.

Figure 2-198:
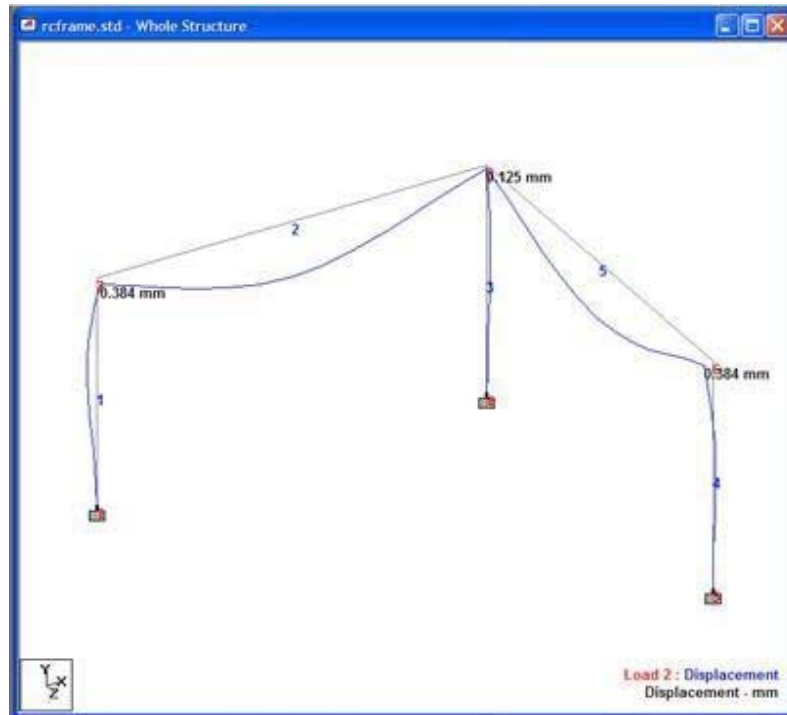
2.11.6 Changing the units in which the offset values are entered

The units in which the values of displacement are shown in post-processing mode called display units.

Procedure:

1. Modifies the display units by selecting the tool **Graphical Display Change Unit**

Figure 2-199:

171

Or select **Tools> Set Current Display Unit** or selecting **View> Options**. The Options dialog box opens.

Figure 2-200:
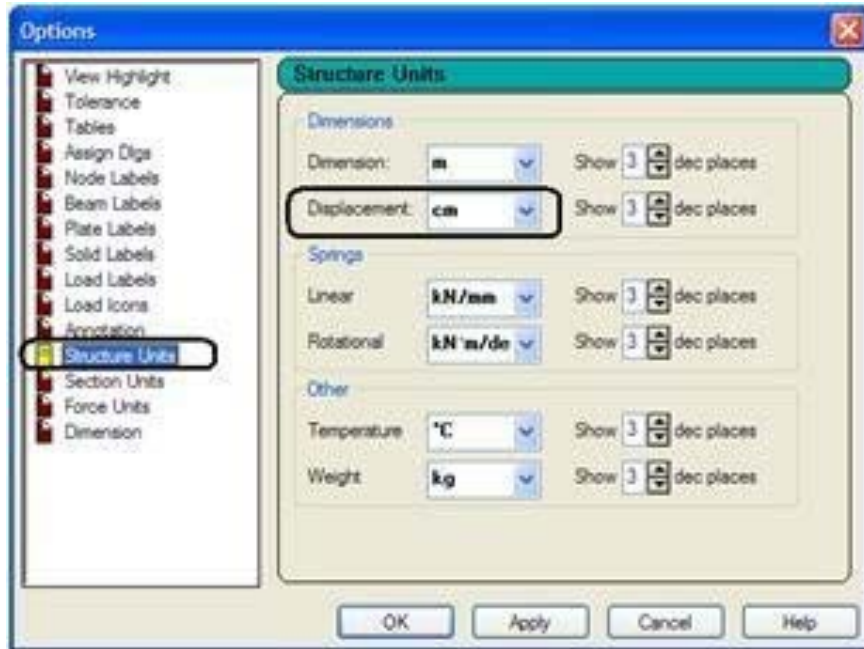
2. Select the tab **Structure Units** and changing the dimensions of the displacement of **mm** to**inches**.

3. Click on **OK**. The chart is updated to reflect the new units.

Figure 2 116
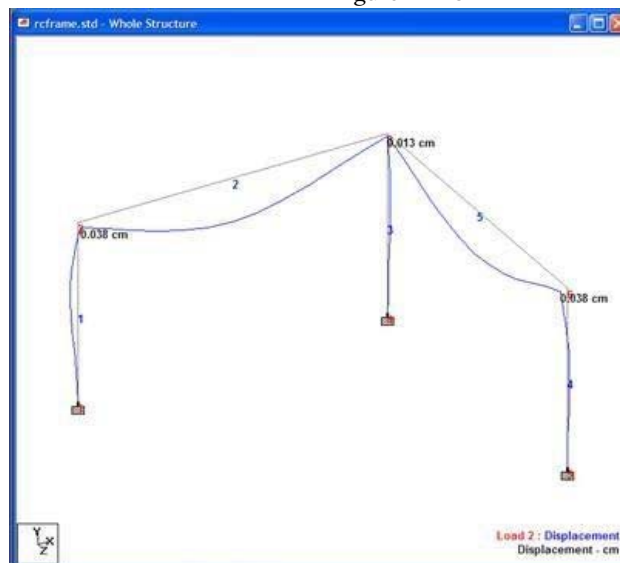
2.11.7 Sliding Table node

As we enter the post-processing mode, then the first screen that we are shown.

Figure 2-201:
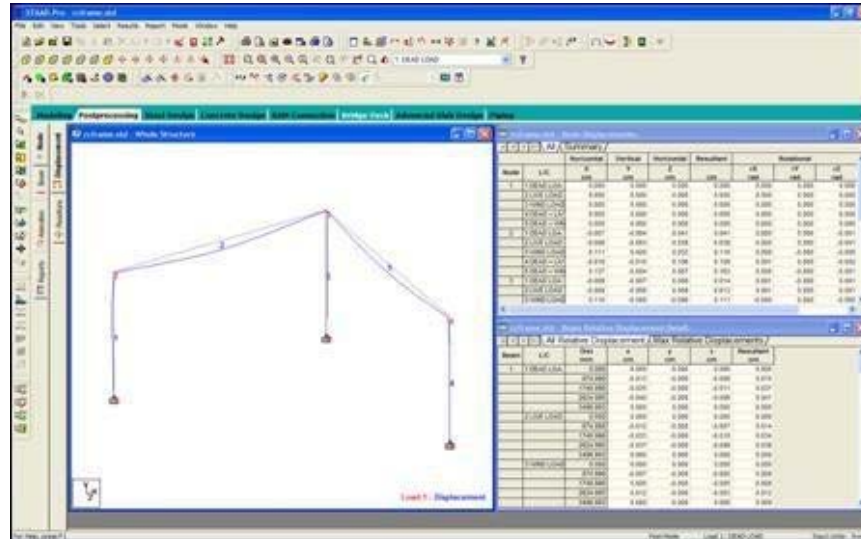
For page *Node> Displacement* on the left side, note that there are two tables appears along the right side. The upper table Displacements called Node table (table node displacement) lists offset values from each node for each selected load case. The load cases can be selected or de-selected in

purpose of this table from the menu *Results> Select Load Case*. (See 2/11/16 for
details) The table below is called Beam relative displacement table (table of relative displacement of the beam).

If you happen to any of these tables is closed, you can restore from the menu *View> Tables*.

Figure 2 118
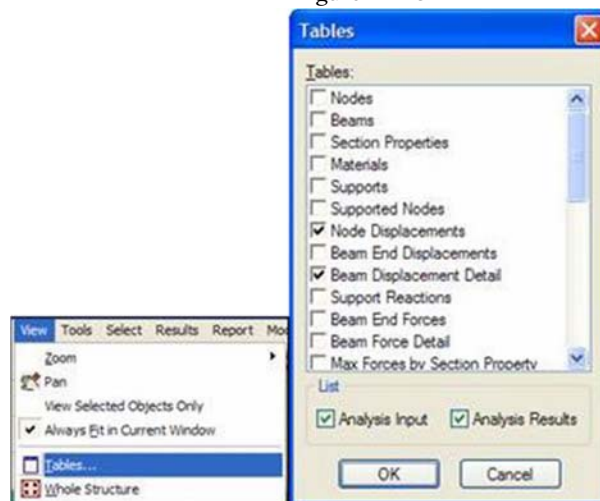
The window table nodal displacements has two tabs: *All* and *Summary* (See figure below).

Figure 2-202:
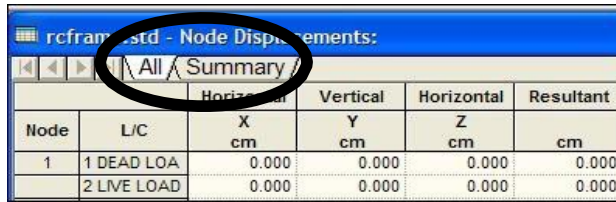
All

This tab displays all the nodal displacements in tabular form for all load cases and degrees of freedom.

Figure 2-203:

Summary

This tab, shown in the figure below presents the maximum and minimum nodal displacement (translation and rotation) for each degree of freedom. All nodes and load cases specified during configuration of results are considered. The maximum values are presented for all degrees of freedom with the corresponding node of occurrence and number of load case (L / C).

Figure 2-204:

Table for relative displacement of the beam, the details are as follows:

All

The tab *All* presents the displacements of the members at the points of the intermediate section. All specified members and all cases include the specified load. The table shows the displacements along the local axes of the members and their resultant.

Max Displacements

The tab *Max Displacements* presents the summary sectional maximum displacement (see figure). This table includes the values of maximum displacement and location of their occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the maximum resultant displacement member to member section.

Figure 2-205:

Subpages in the page *Node* described below in brief.

Page	Sub-Page	Purpose
Node	Displacement	Displays the nodal displacements with tabular results for nodal displacements and displacement sectional beams.
	Reactions	Shows the reactions of the supports in the drawing and in a tabular form.
	Modes	Modes shows the number of ways to form selected mode. The eigenvectors appear simultaneously in tabular form. This page appears only in case of dynamic analysis, response spectrum, time history, and modal calculations if requested.
	Time History	Shows the time history graphs for analysis over time. This sub-page also appear only if the consolidation analysis were performed.

2.11.8 Viewing diagrams force / time

Procedure:

1. The easiest way to access the amenities to display diagrams of force / moment is from page **Beam | Forces** along the page control area on the left side of the screen. The bending moment will be marked by MZ

   fault, evidence of which can be in the form of icon Mz shown in the diagram below which it becomes active.

Figure 2 123
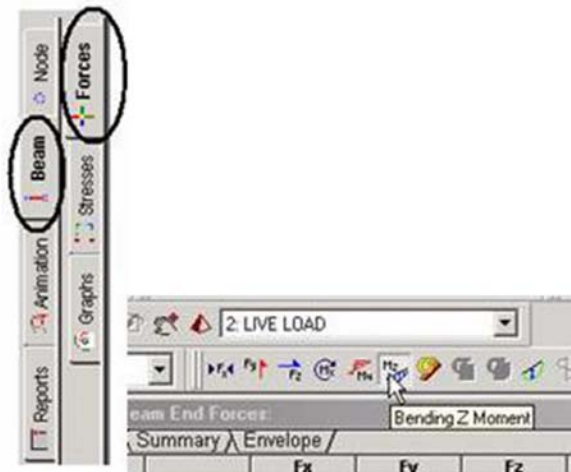
Figure 2-206:
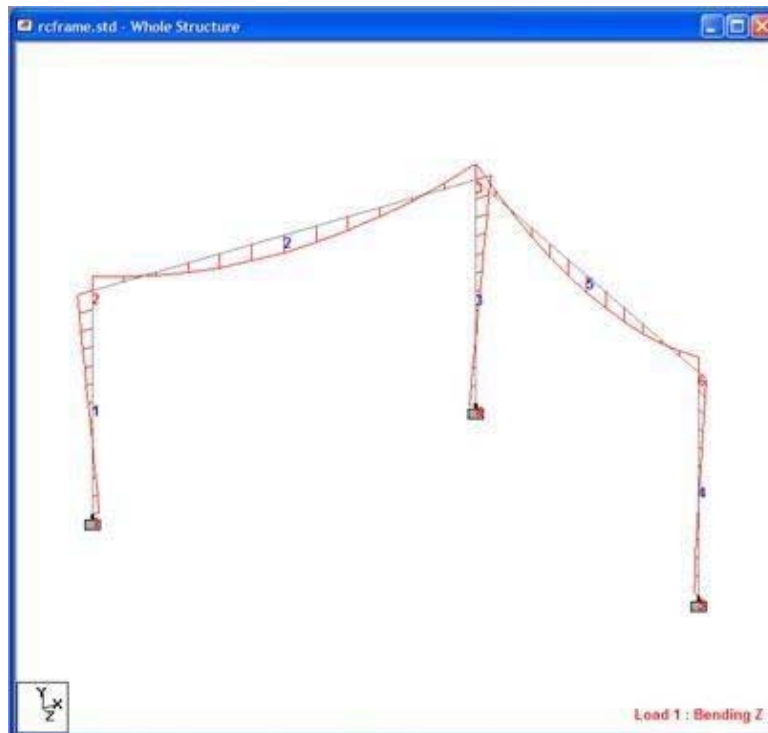
2. The option to select the diagram of force / time from another facility is available too - the menu **Results> Bending Moment** - As shown below.

Figure 2-207:

2.11.9 Switching between load cases for a diagram of force / moment

Procedure:

1. To change the load case for a diagram of force / time, you can click the list box called **Active Load** and choose the one you want.

Figure 2-208:
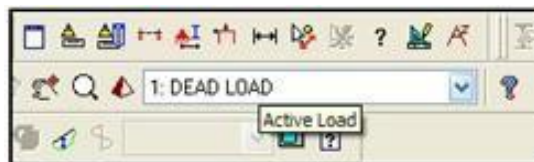
Or select the tool **Symbols and Labels** or select **View> Structure Diagrams**. Dialog box opens diagrams.

Figure 2-209:
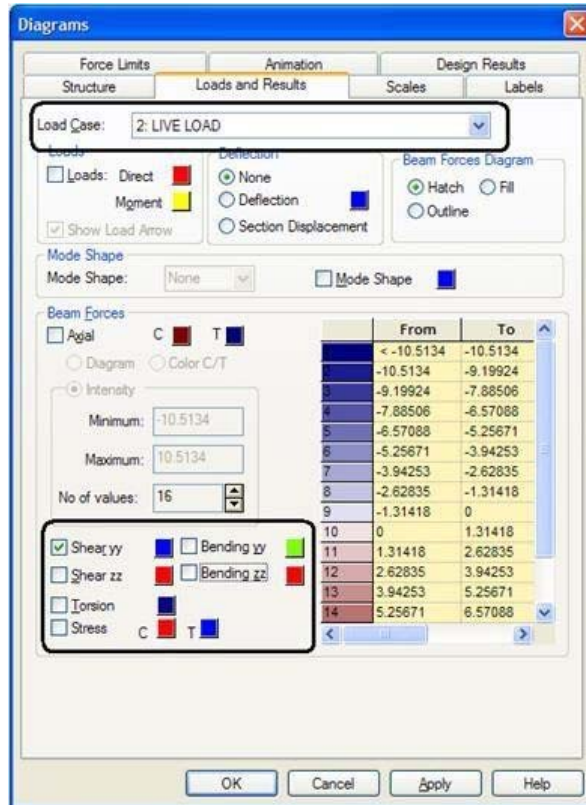
2. Select the tab **Loads and Results** and choose the second load case (LIVE LOAD) in the list box of the case load and check box **Shear yy**.

3. Click on **OK**.

The following diagram shows the shear forces for the load case 2.

Figure 2-210:
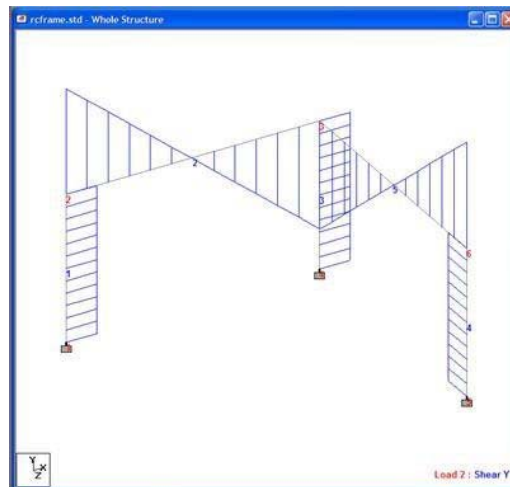
4. To show the diagram of bending moment to say, if load 4 (DEAD + LIVE), follow steps 1-3 and select the load 4.

    The following diagram should appear in the drawing area:

Figure 2-211:
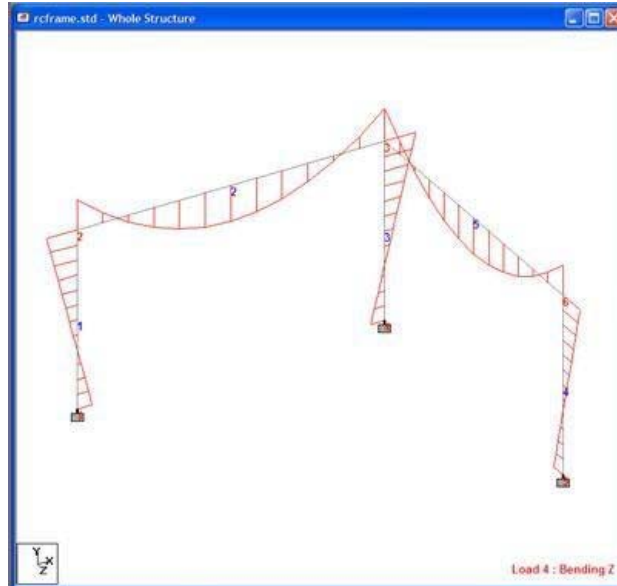
2.11.10 Resize diagram force / moment

Procedure:

If the chart appears very imperceptible, it may be because it is drawn to a very small scale. To change the scale of the plot of the moment, you can:

1. Select tool **Scale** or select **Results> Scale**

Or selecting **View> Structure Diagrams> Scales**

The Diagrams dialog opens on the tab stops.

Figure 2-214:
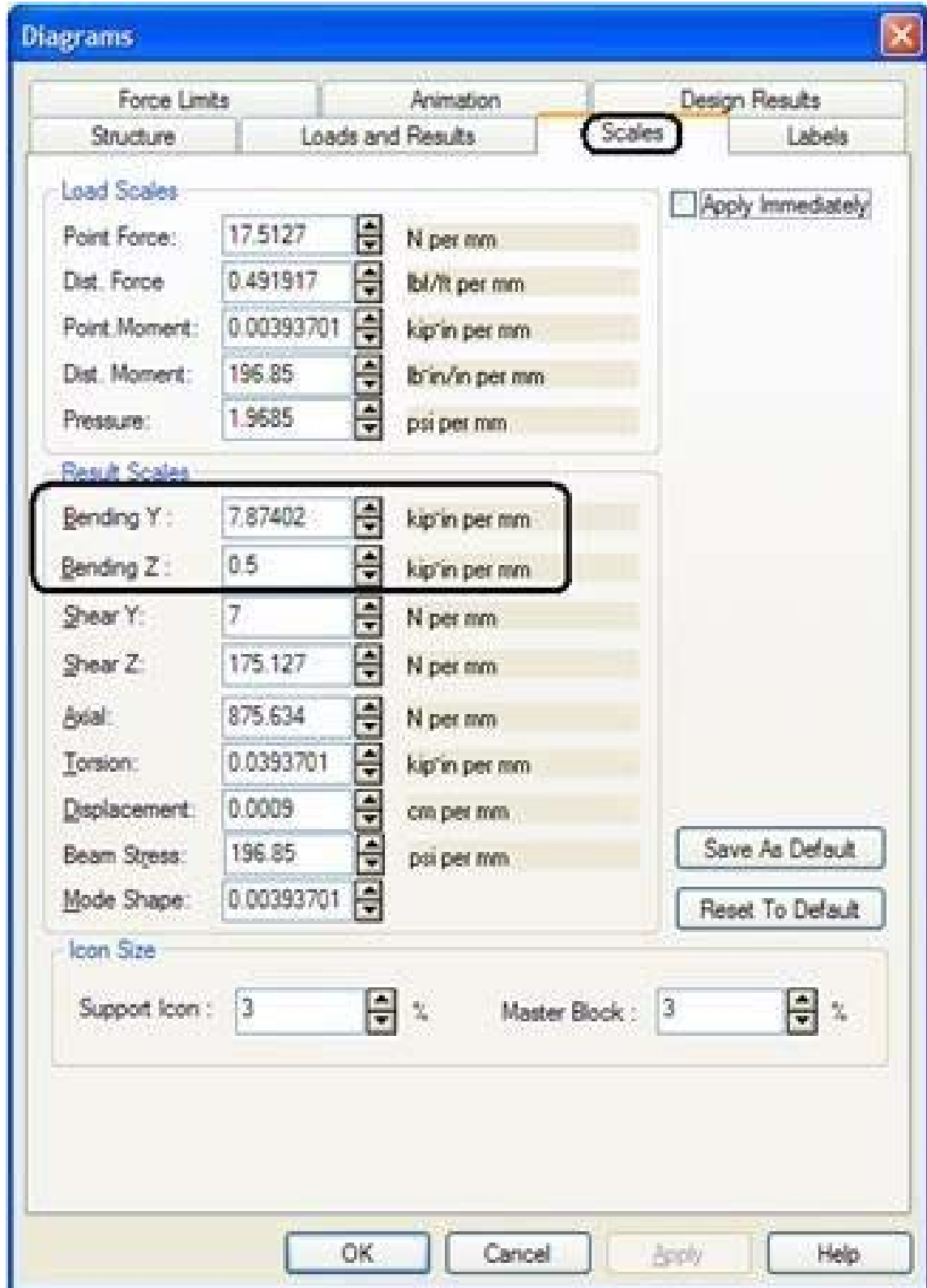
2. In the field of **Bending** Specify a smaller number of what is currently on the list and click **OK**. The moment diagram should now be larger.

3. In the above dialog box, if the box set **Apply Immediately** Pressing the up or down arrows next to the number will produce immediate results in terms of a smaller or a larger diagram.

Figure 2-215:
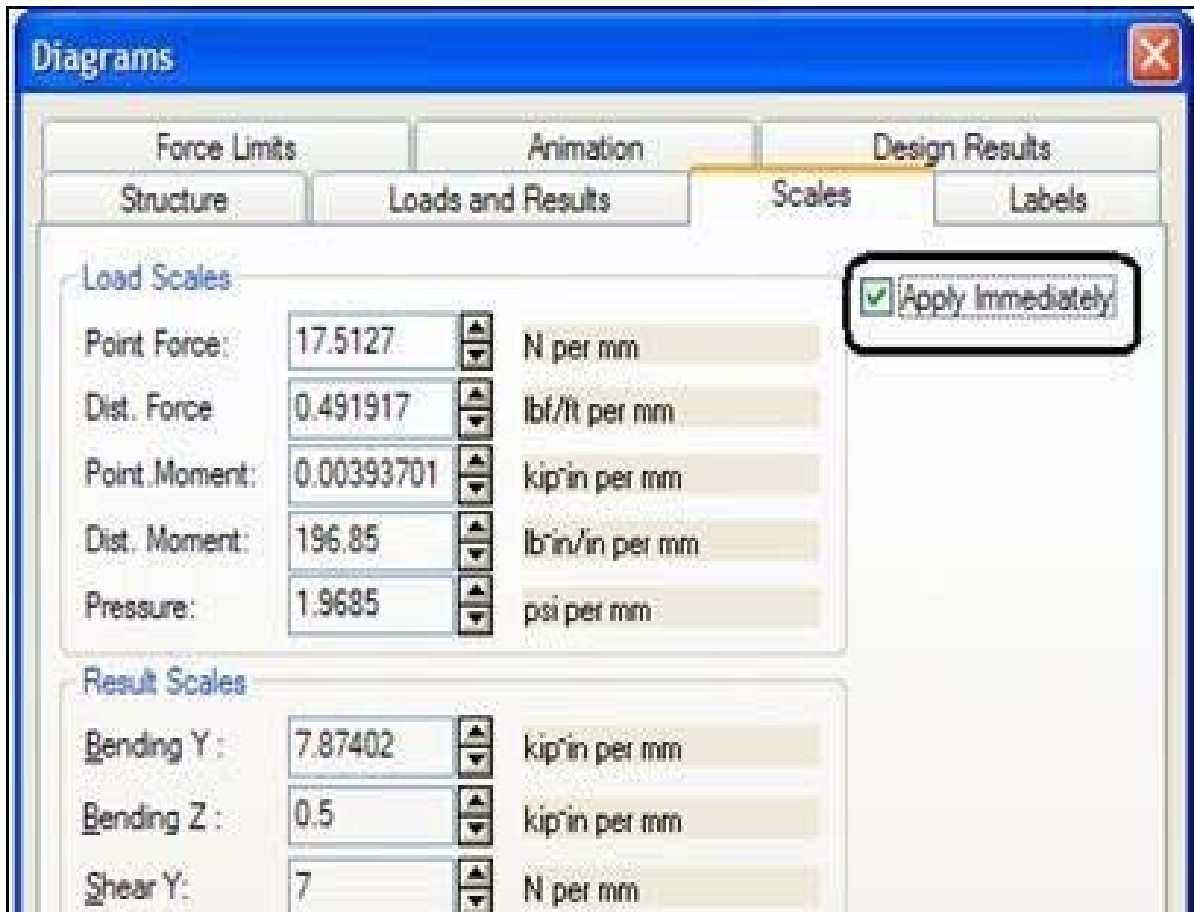
2.11.11 Changing the degree of freedom for the layout of the diagram of forces.

The force and moment diagrams can be drawn by six degrees of freedom: Axial cutting–Y, Shear-Z, torsional moment and moment-Y-Z.

1. Select **View> Structure Diagrams> Loads and Results.**

Figure 2-216:
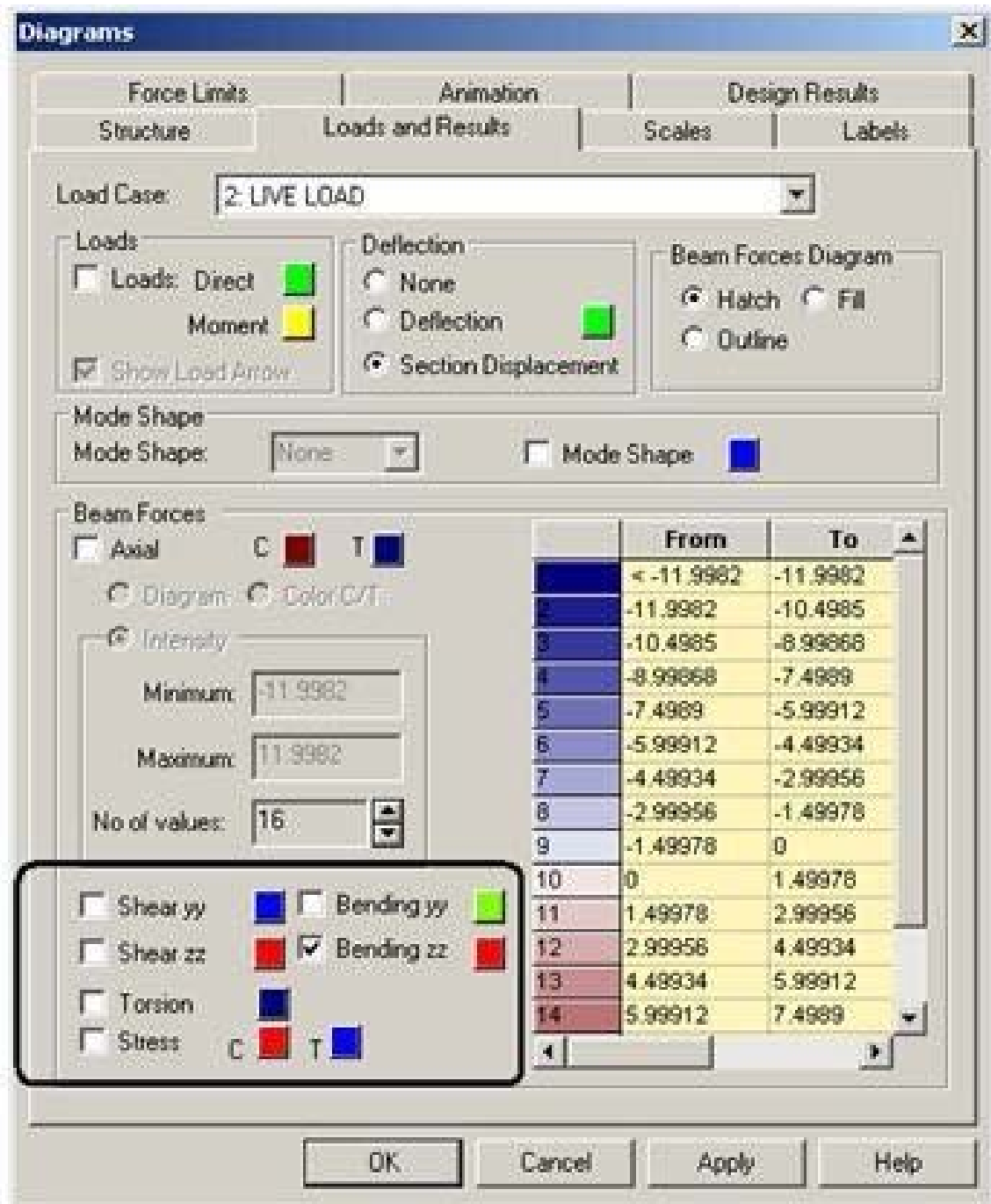
Occupies associated with each degree of freedom shown box. The icons in the bar results can also be used to enable / disable certain degrees of freedom.

Figure 2-217:

For easy identification, each degree of freedom (DOF) has been assigned a different color. You can change the color of that d.o.f. clicking the color button with the shallow and make a new selection of the color palette.

Figure 2-218:
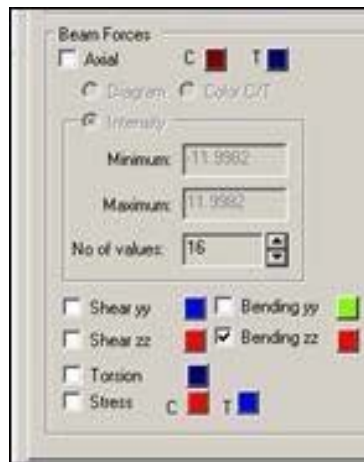
The appearance of the diagram can also be set to one of three options: grid, fill and stroke by selecting the appropriate option in the dialog box shown above.

Figure 2-219:
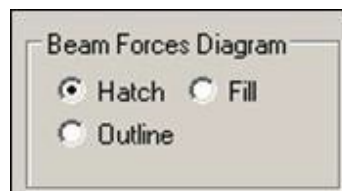
2.11.12 Scoring diagram force / moment

Annotation is the process of displaying the values of force / moment on the screen. Procedure:

1. Select **Results> View Value**. The annotation dialog box opens.

2. Select the tab **Ranges** and select **All**.

> **Tip**: To record the forces and moments for a few members, specify the numbers of the beam in the list of the beam.

Figure 2-220:
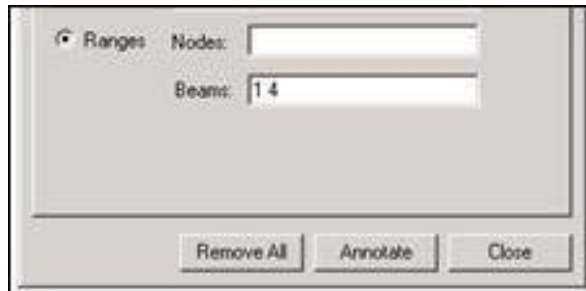
3. Select the tab **Beam Results**, Tick **Maximum** bending results.

Figure 2-221:
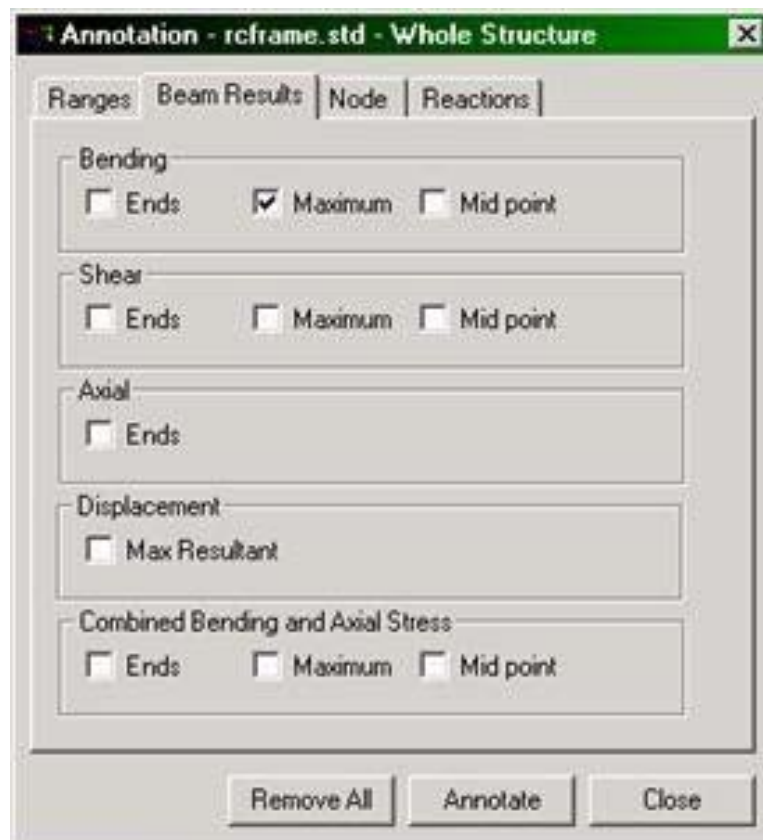
4. Click on **Annotate** and then **Close**.

    The maximum time, MZ, the values of load 5 are shown in the diagram bending structure, as shown in the following figure.

Figure 2-222:
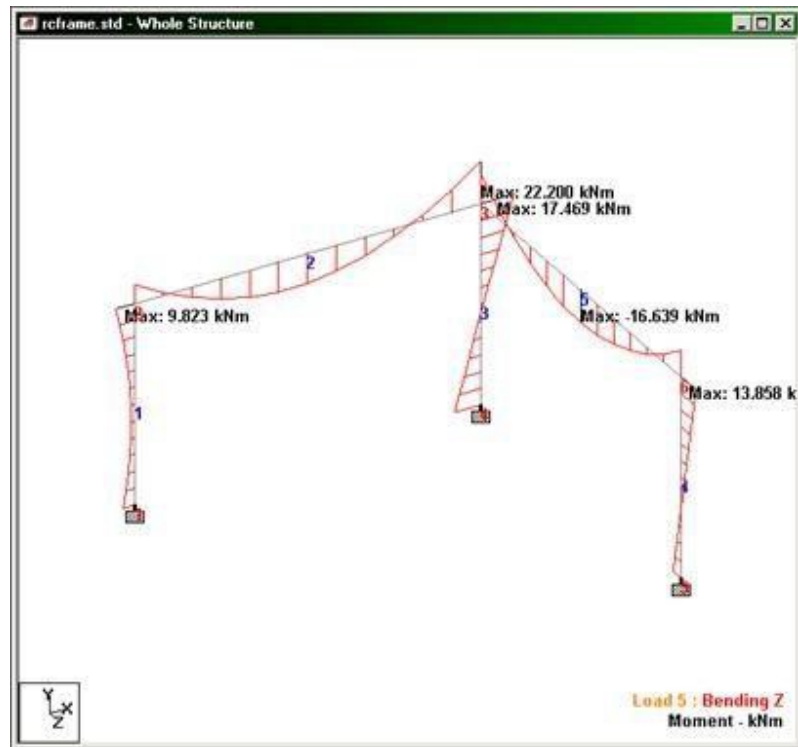
2.11.13 Changing the units in which are recorded the moments / forces

The units in which the values of force and moment in the post-processing mode appear are called the *display units*.

Steps:

1. Units can also be modified by selecting a tool **Graphical Display Change Unit**

Figure 2-223:
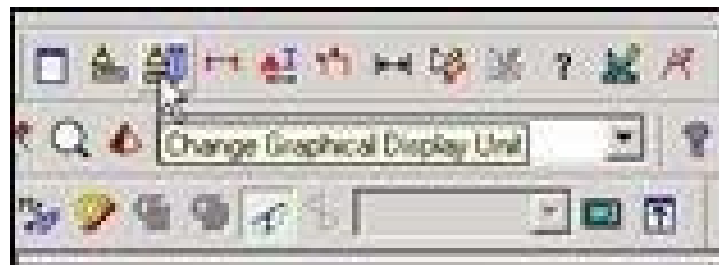
Or selecting **Tools> Set Current Display Unit** or selecting **View> Options**.

The Options dialog box opens

2. Select the tab **Force Units**. For bending moments change the unit of time of your current configuration **kip-ft**.

Figure 2-224:
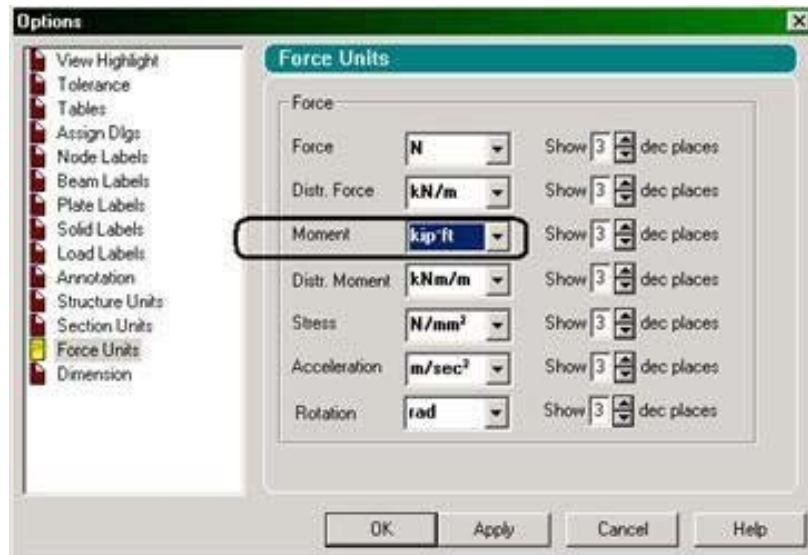
3. Click on **OK**.

The chart is updated to reflect the new units.

Figure 2-225:
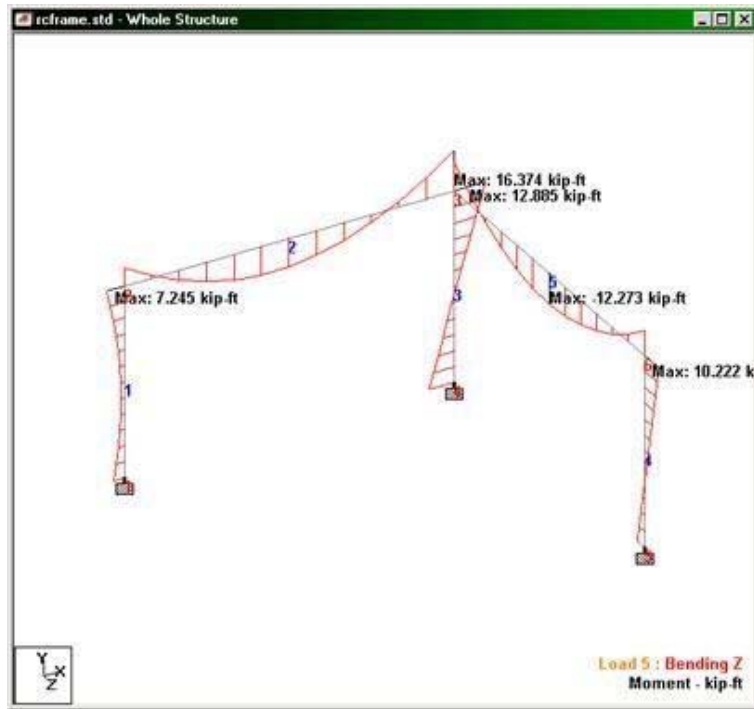
2.11.14 Table of forces in beam

When selecting the page **Beam | Forces** from the control area of the page on the left side, the screen shown below.

Figure 2-226:
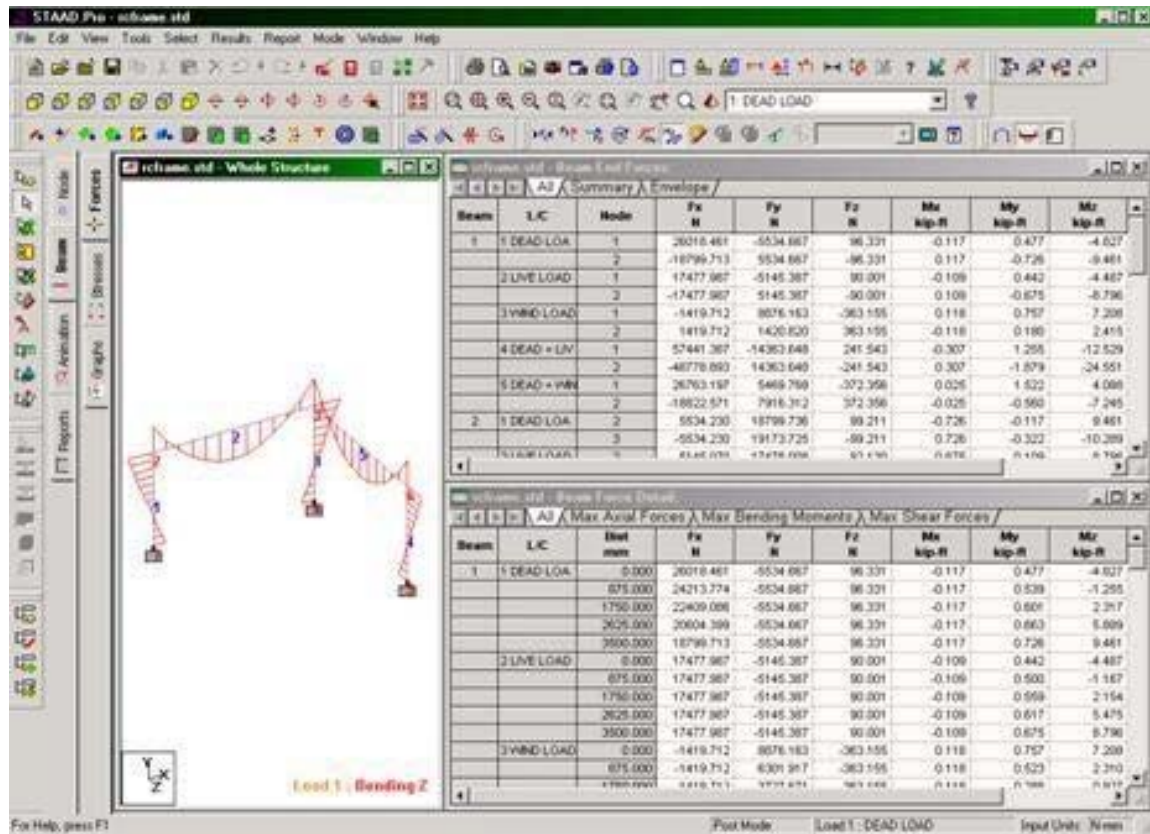
The axial forces and shear forces, bending moments and torsional all selected for all cases of load beams appear in a tabular selected along the right half of the screen form. The load cases can be selected or discounts

selected for the purpose of this table from the menu **Results> Select Load Case**
(See 11.2.16 for details)

If you happen to any of these tables is closed, you can restore from the menu **View> Tables**.

Figure 2 149
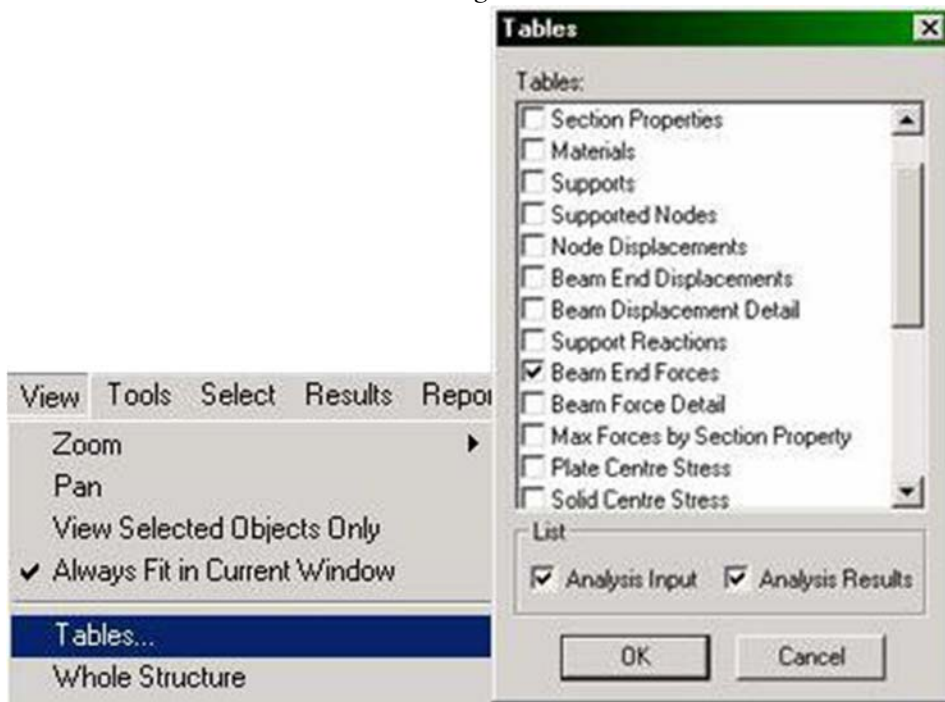
The table window *Beam End Forces* has three tabs: *All, Summary* and *Envelope*.

Figure 2-227:

All

This card has all the forces and moments corresponding to all six degrees of freedom at the beginning and end of each selected member for all selected load cases.

Figure 2-228:

Summary

This tab, shown in the figure below shows the maximum and minimum values (forces and moments) for each degree of freedom. All beams and all specified loading cases are considered when setting results. The maximum values are presented for all degrees of freedom with the corresponding node of occurrence and number of load case (L / C).

Figure 2-229:

Envelope

This tab displays a table consisting of the maximum and minimum for each degree of freedom for each member and responsible for each of these load values.

Figure 2-230:

2.11.15 Viewing diagrams force / moment from page Beam> Graphs

Page *graphics* in the *post-processing mode* is used to graphically display moments and forces as *Axial*,*zz bending*,*Shear yy* and*combined stresses* for individual members.

1. Select **Beam | Graph** on the left side of the screen as shown below.

Figure 2-231:
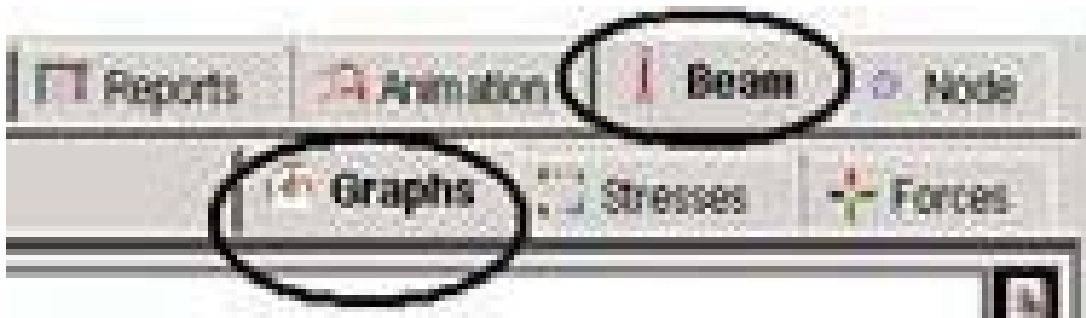
The view window shows the load of the structure. On the right side of the screen diagrams force / moment appear (see figure below).

2. Select a member in the main window and plotted graphs for members in the data area. The following figure shows the graphs for members 1 for the case of load 4.

Figure 2-232:
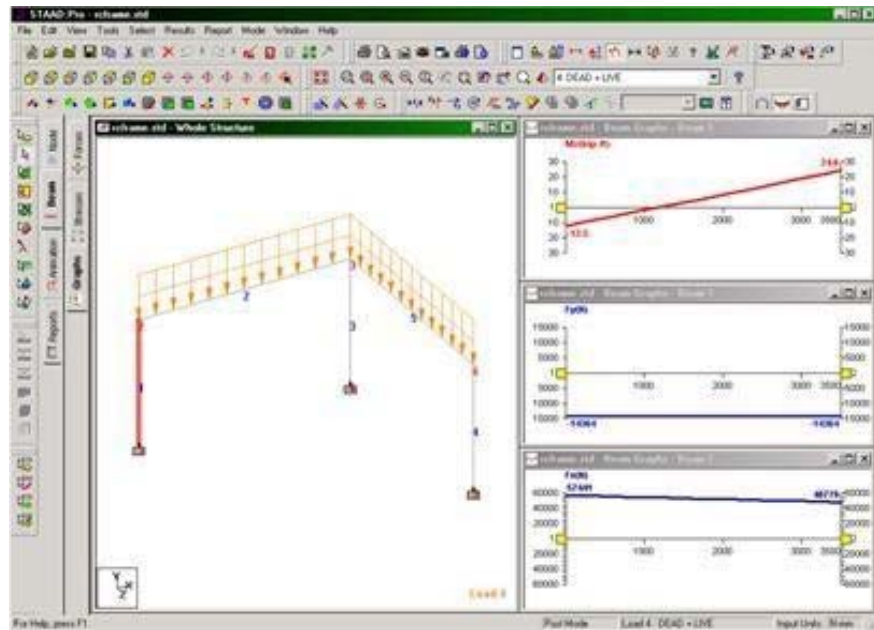
The following figure shows the graphs for the same member 2 charging.

Figure 2-233:
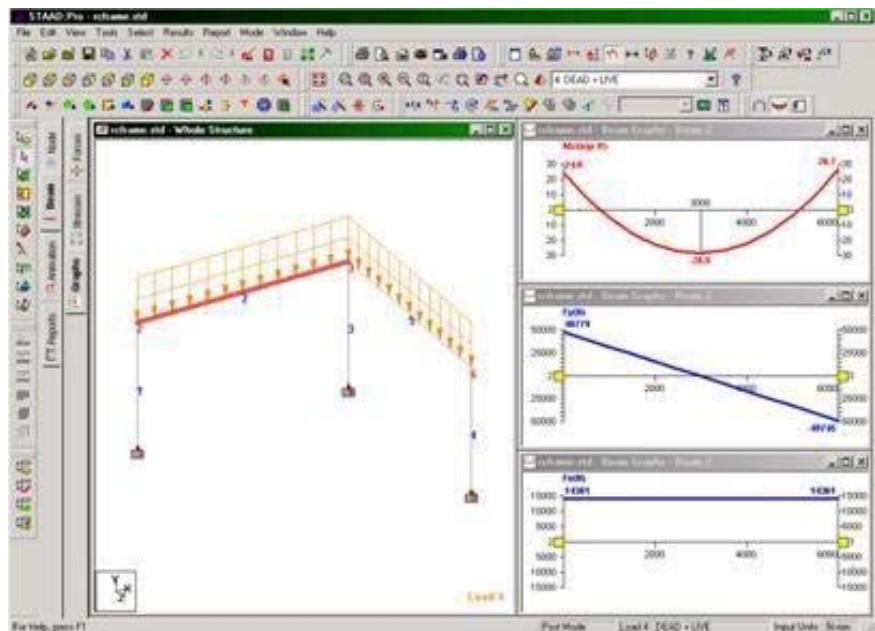
3. Right Click on any of the diagrams force / time and select
   Diagrams ... the popup menu.

Figure 2-234:
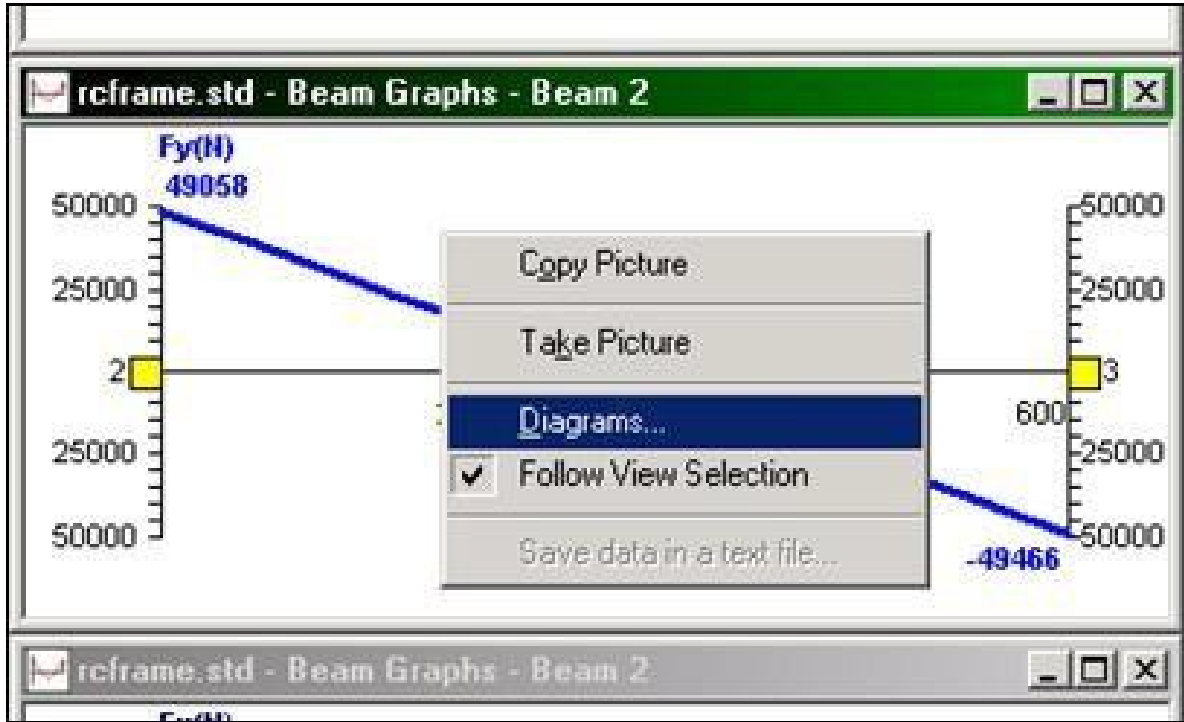
Open dialog box diagram.

4. Set the check box to the degrees of freedom you want to see in the diagram.

Figure 2-235:

5. Click on **OK** .

The selected degree of freedom is drawn in the window.

2.11.16 Restricting the load cases for which the results are seen

Steps:

1. To restrict the load cases for which the results are viewed, select the tool **Setup Results** 📖 or, go to the menu **Results> Select Load Case** as shown below.

Figure 2-236:

2. In the dialog box *Setup Results* that appears first case de-seleccionanamos load and selected by clicking the [<<].

Figure 2-237:
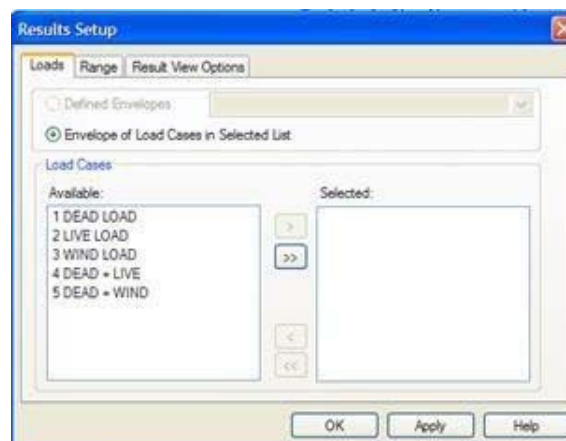
3. Hold down the 'Ctrl' key to select the load cases 1 (DEAD LOAD) and 3 (WIND LOAD). Click [>]. After the load cases have been selected, click **OK**.

Figure 2-238:
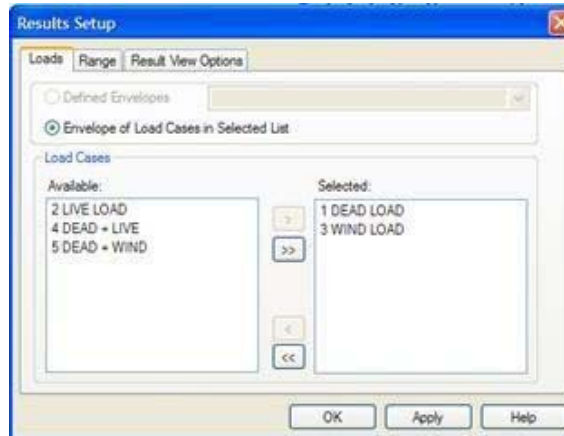
2.11.17 Using the Query command members

Member consultation (*Member query*) Is a facility where you can see several results for specific members simultaneously from a single dialog box. It is also a place where many of the attributes of the members as the definition of property specifications (release, armor, cable, etc.) and beta angle can be changed for purposes of entry.

Procedure:

To access this service, first select the member. Then, go to the menu **Tools> Query> Member** or double-click the member. Try to double-click Member 4.

Figure 2-239:
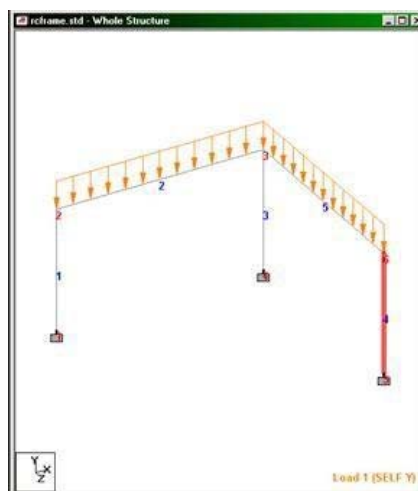
As we double-click the member 4, the following dialog box opens. Take a look at the record *Property*.

Figure 2-240:
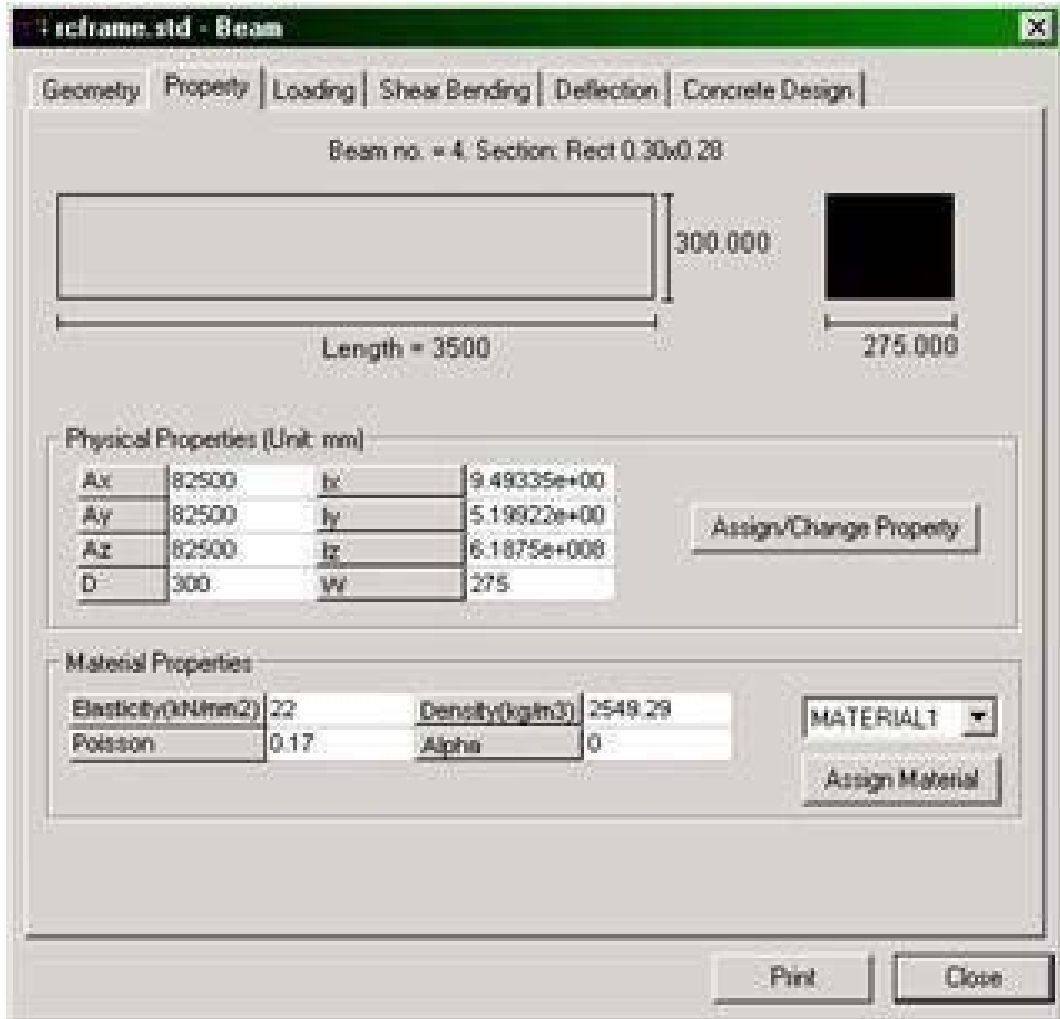
In the figure above shows where the buttons on the box member consultation. If the member contains output result sheets (shear / flexion, deflection, steel design, etc.) in the query box, it will cause these chips results disappear by changing the attributes of members and released. This is due to the fact that the current output does not reflect the new entry.

Note: If you assign or change property by clicking on the button **Assign / Change Property** in the previous dialog box, make sure to keep the check mark on "Apply to this Member only" in the next dialog. Otherwise, changing the attributes of an individual member to later change the attributes of all the other members belonging to the same attribute list. For example, if the property of the current members are also assigned to other members, changing the property of the current member will change the property of all members.

STAAD.Pro V8i MANUAL

We click on the tab *Shear / Bending*. The following dialog box appears.

Figure 2-241:
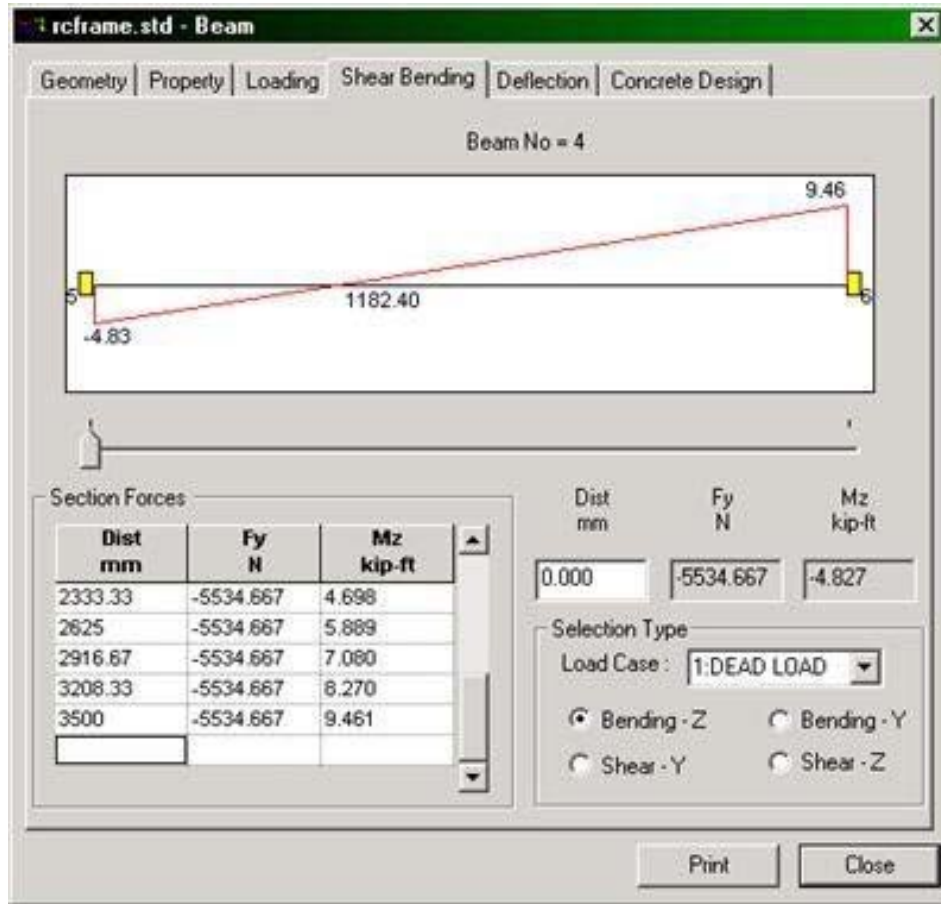
The above page contains amenities for displaying the values of shear and moments, selecting the load cases for which the results are presented a slide bar (see figure below) to see the values at specific points along the limb length and a print option for printing screen elements. Experiment with these options to see what kind of results you can get. Take the slider with the mouse and move it to get the values in specific locations.

Figure 2-242:
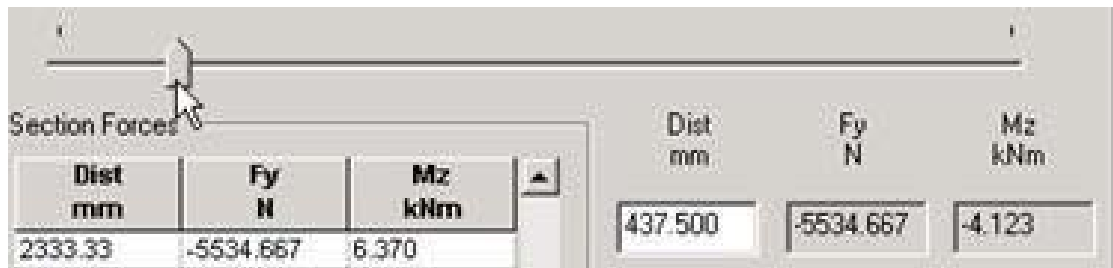
Then another page (deflection) of the previous dialog box is displayed.

Figure 2-243:
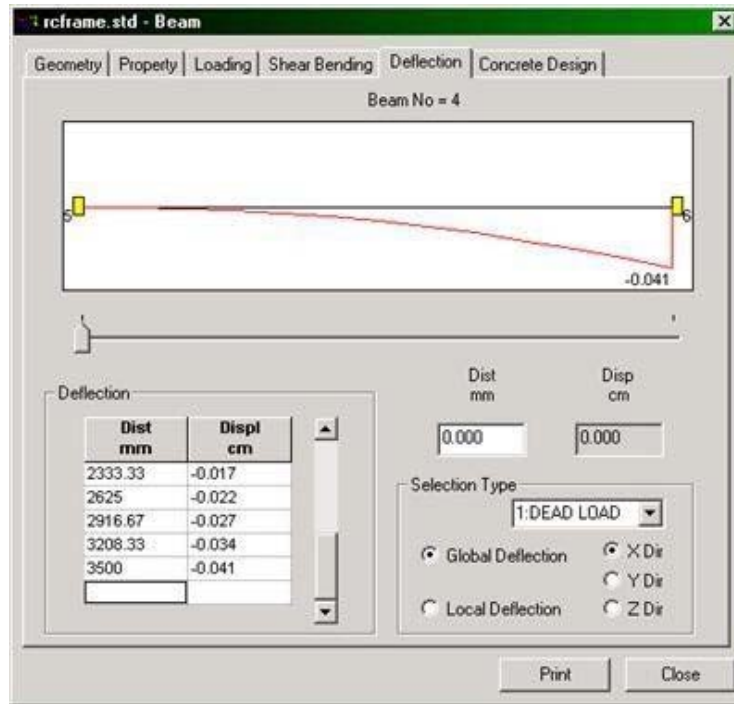
Then the page design concrete above dialog box is displayed.

Figure 2-244:
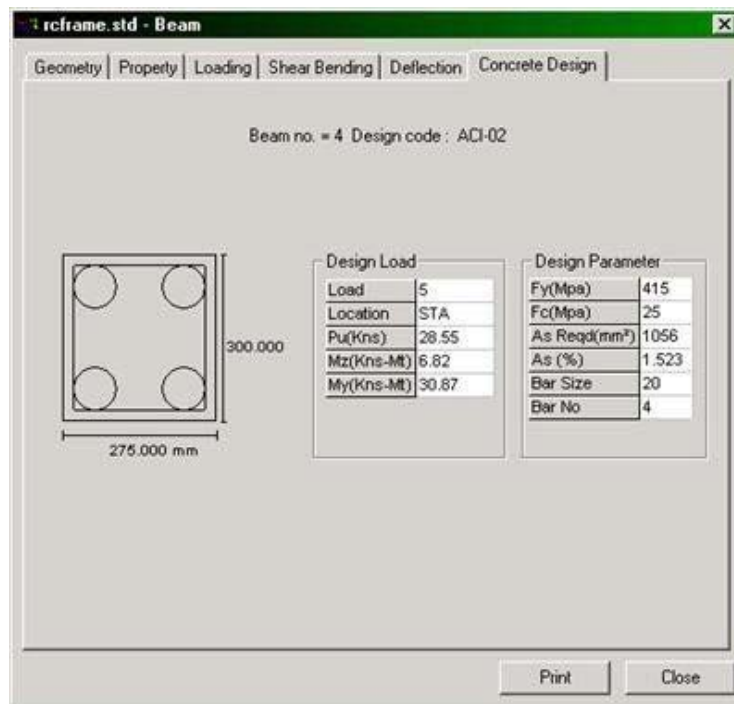
To see the results of another member of the use of this mechanism for consultation, simply close this dialogue box query and repeat the steps described earlier in this section for the desired member.

2.11.18 Producing a report on screen

Sometimes we find the need for results that conform to certain restrictions, such as, for example, resulting from node movements for a couple of selected nodes, for a few selected load cases, sorted in order of lowest to highest, with values reported in tabular form. The facility allows us to obtain these results as they appear on the screen is the menu Report in the top of the screen.

Here, a report that includes a table with values greater time shaft member (MZ) sorted from high to low, for members 1 and 4 for all load cases created.

Steps:

1. Select the members 1 and 4 of the structure using the **Beams Cursor**

    ○**Select> By List> Beams** and type 1 and 4 as the numbers of members

2. Select **Report> Beam End Forces**.

    Open dialog box beam end forces.

Figure 2-245:
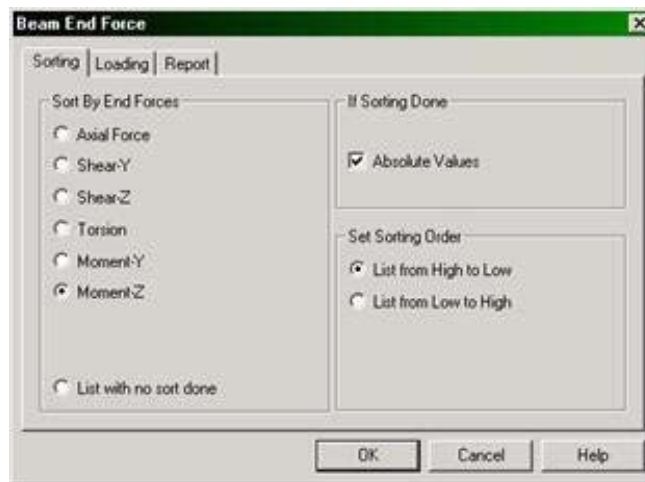
3. Select the tab **Sorting** (Classification).

4. Select **Moment-Z** as *final force* Establishes the *Sorting Order* to**List from High to Low** and set the checkbox **Absolute Values** low *If done Sorting*.

> **Tip** If you want to save this report for future use, select the tab **Report** Provides a title for the report and sets the checkbox **Save ID**.

5. Select the tab **Loading** and ensures that all the five load cases were selected.

6. Click on **OK**.

The leaderboard at the end of the member forces opens with MZ values sorted from highest to lowest based on absolute numbers.

Figure 2-246:

Beam	L/C	Node	Axial Force kN	Shear-Y kN	Shear-Z kN	Torsion kNm	Moment-Y kNm	Moment-Z kNm
4	4	6	-48.779	14.364	0.242	-0.416	2.547	-33.266
1	4	2	-48.779	14.364	-0.242	0.416	-2.547	-33.266
1	4	1	57.441	-14.364	0.242	-0.416	1.702	-16.966
4	4	5	57.441	-14.364	-0.242	0.416	-1.702	-16.966
4	5	6	-20.609	5.909	-0.982	2.210	4.731	-13.858
1	1	2	-18.800	5.535	-0.096	0.159	-0.984	-12.827
4	1	6	-18.800	5.535	0.096	-0.159	0.984	-12.827
1	2	2	-17.478	5.145	-0.090	0.148	-0.915	-11.905
4	2	6	-17.478	5.145	0.090	-0.148	0.915	-11.905
1	5	2	-18.823	7.916	0.372	-0.034	-0.780	-9.823
1	3	1	-1.420	8.876	-0.363	0.161	1.026	9.772
4	5	5	28.549	-5.909	-21.328	-2.210	30.874	-8.804
1	1	1	26.018	-5.535	0.096	-0.159	0.647	-6.544
4	1	5	26.018	-5.535	-0.096	0.159	-0.647	-6.544
1	2	1	17.478	-5.145	0.090	-0.148	0.600	-6.084
4	2	5	17.478	-5.145	-0.090	0.148	-0.600	-6.084
1	5	1	28.763	5.470	-0.372	0.034	2.083	5.542
1	3	2	1.420	1.421	0.363	-0.161	0.245	3.275
4	3	5	-0.054	0.136	-16.287	-1.822	24.188	0.286
4	3	6	0.054	-0.136	-0.874	1.822	2.785	0.192

7. To print this table, right click anywhere in this part of the table and select **Print** the popup menu.

2.11.19 Taking photographs

Several options are available to take pictures. The simplest of these is in the Edit menu is called *Copy Picture*. Transfer the contents of the active drawing window in the windows clipboard. Then we can go into any word processing program
Image as Microsoft Word or Microsoft Paint and paste the image into the program for further processing.

Another versatile option allows any "snapshot" or image of the drawing window in a report. It is called *Take Picture* and is under the menu *Edit*. Consider this feature.

Steps:

1. To take a picture, select the tool **Take Picture** or select **Tools> Take Picture**.

Figure 2 172
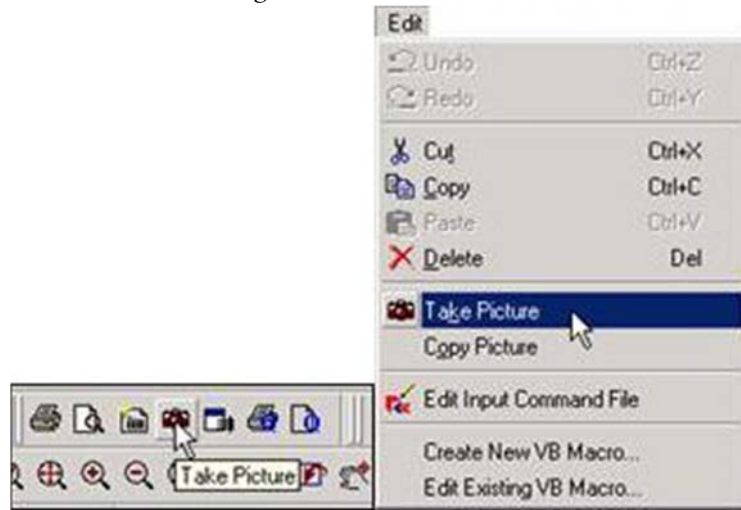
Picture box # dialog opens.

Figure 2-247:
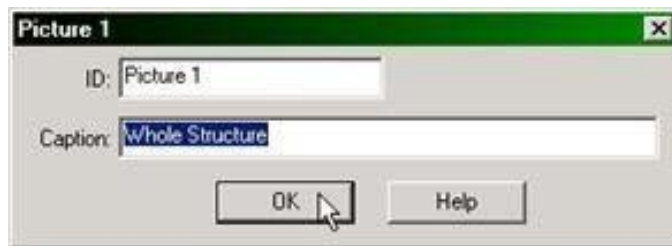
2. Provides a title for the box so that you can be identified when generating a report. Click **OK** to save the image.

This image is saved until we are ready to produce a personalized report of results.

2.11.20 Creating Custom

STAAD.Pro offers spacious facilities reporting. Articles that can be incorporated in the reports include information input numerical results steel design results, etc. One can choose from a select set of load cases, mode shapes, structural elements, etc ... We can include any "snapshot" or image of the

screenshot taken using the icon toolbar *Take Picture*. Other parameters include configurable font size, title block, headers, footers, etc.

1. Select the page **Reports | Reports** or tool **Report Setup** .

Figure 2-248:

The Report Settings dialog box opens.

Figure 2-249:
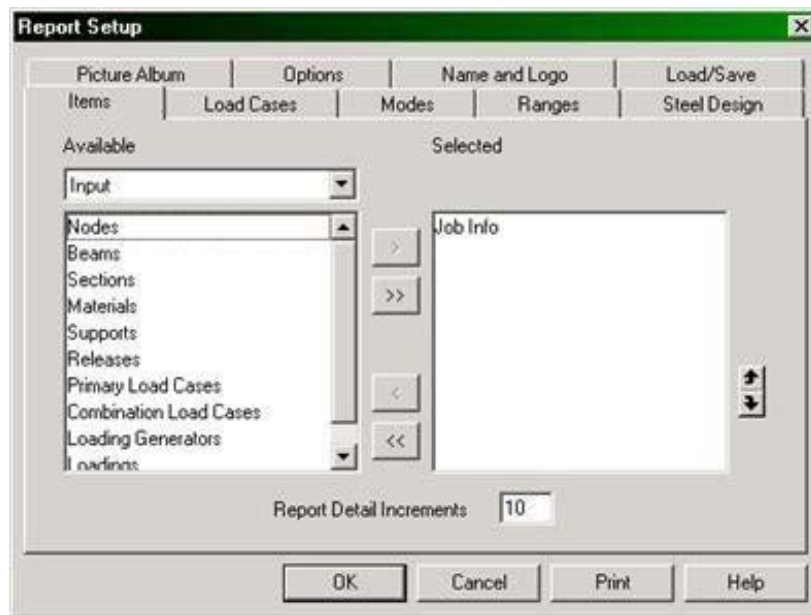
Different tabs of this dialog box offer different options. The Record *Items* lists all available data that can be included in the report. Note that the elements in the list *Selected* are those that have been selected by default.

Items available are classified into seven categories: *input*,*Output*,*Pictures, Reports, STAAD.etc output, Steel Design Output* and*Advanced Query Reports*.

Figure 2-250:

2. In our report, you will include *Job Information*,*Displacement Node Summary*,*Max Beam Moments*, And *Picture 1.*

   Job Information is selected by default.

   In the box *Available* list, select *Output*. Since the output elements available, select *Displacement Node Summary* and*Max Beam Moments*.

   Then select list boxes *Available* and select *Picture 1*.

   When all items have been selected, the dialog box *Report Settings* should appear as shown below.

Figure 2-251:
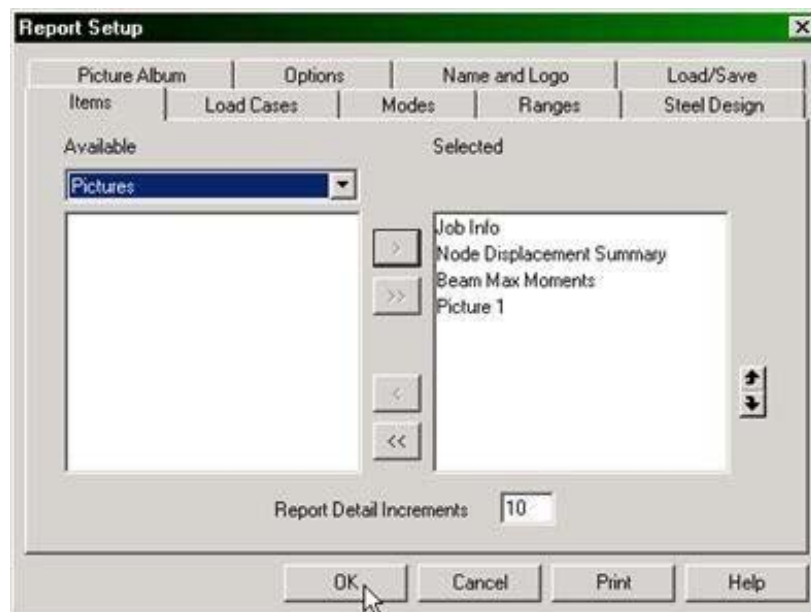
The **Report detail increases** indicates the number of segments in which a member Printable divide sectional displacements, forces, etc.

3. Select the tab **Load Cases** to select the load cases to be included in the report. Used options **Grouping** to group the data in the group node numbers / beam or load case number.

   In the first case, all the results of the load event appear below a certain node or beam. In the second case, results for all nodes or beams for a particular load case appear together.

Figure 2-252:
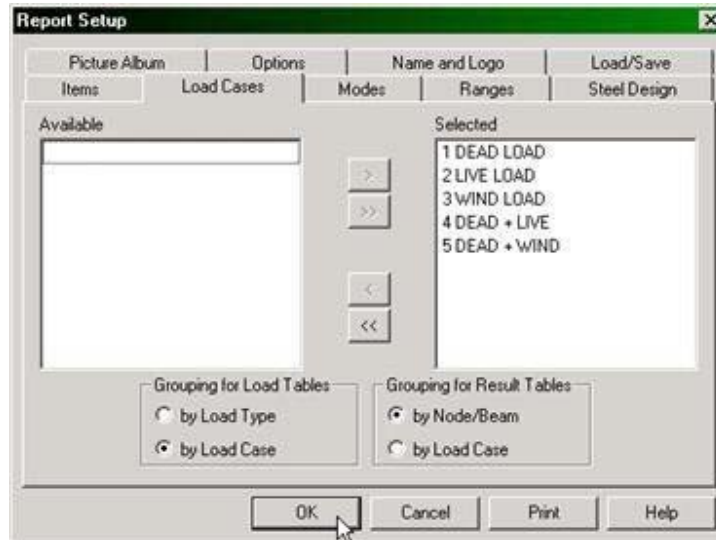
4. Select the tab **Picture Album** to visually identify previously taken photos. The following figure shows 1 photo stored by the program.

Figure 2-253:
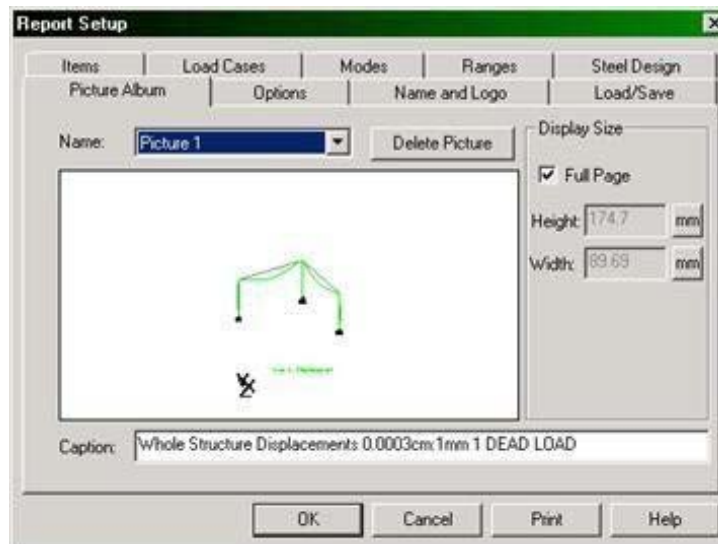
5. The Options tab allows you to include *header, footer, page numbers, tables grids*, Sources for *column header* and *data table* Etc. ..

6. The Record *Name and Logo* allows you to enter the company name and logo.
Click the blank area and type the name and address of the company. Click **Font** in the group *Text* and adjust the font to be Arial Bold 16 Pt. Click the button *right* in the group *Alignment* align text to the right of the name of the

compan

y.

Figure 2-254:

7. Select **OK** to end or select *Print* to print the report.

Tip: It is always a good idea first before printing the report. This is done by selection of the tool **Print Preview** (Print preview).

Figure 2-255:
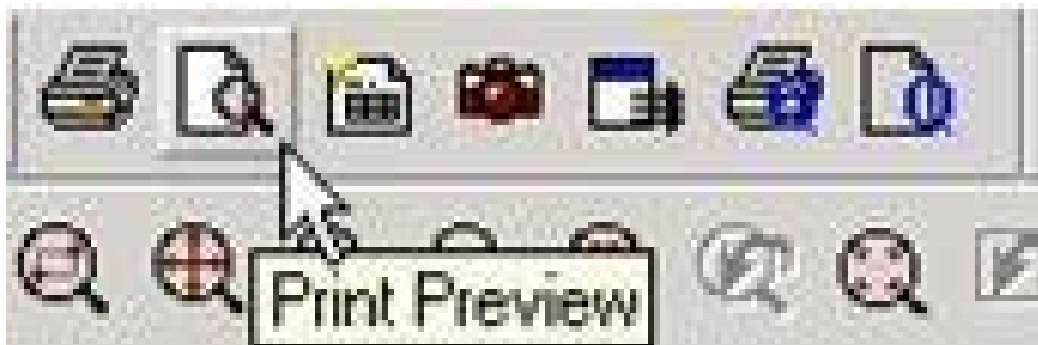
The first and last pages of the report are shown in the following two figures.

Figure 2-256:

Figure 2-257:

This brings us to the end of this tutorial. Although we have covered a lot of topics, many more missing features to explore in STAAD.Pro.

Tutorial 3 - Analysis of a Slab

This tutorial provides step-by-step modeling and analysis of a slab supported on two embedded edges.
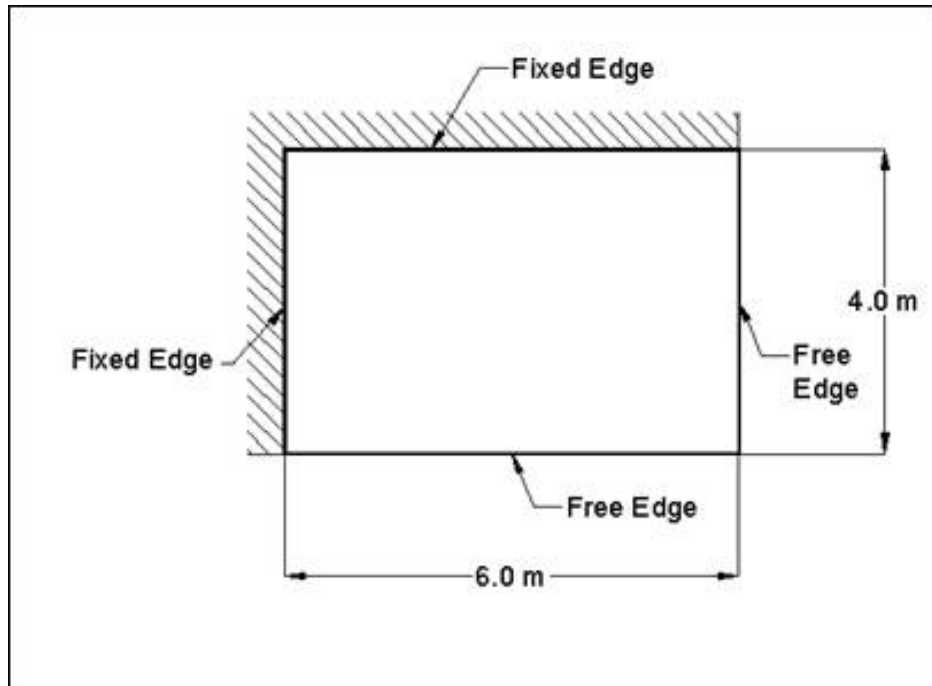
3.1 Model creation methods

As explained in section 1.1 of the tutorial problem 1, there are two methods for creating the data structure:

1. Using the graphical model generation mode or graphical user interface (GUI) (user GUI) that typically refers.

2. Using the file command.

Both methods to create the model explained in this tutorial. The graphical method is explained from section 3.2. The script method is explained in section 3.8.

3.2 Problem description tutorial

The structure of this project is a slab fixed along two edges. We are using 6 flat quadrilateral elements. The structure and the mathematical model are shown in the following figures. Is subjected to own weight loads and pressure loads temperature. Our goal is to create the model, assign all the required entries, perform analysis and review the results.

Figure 2-258:
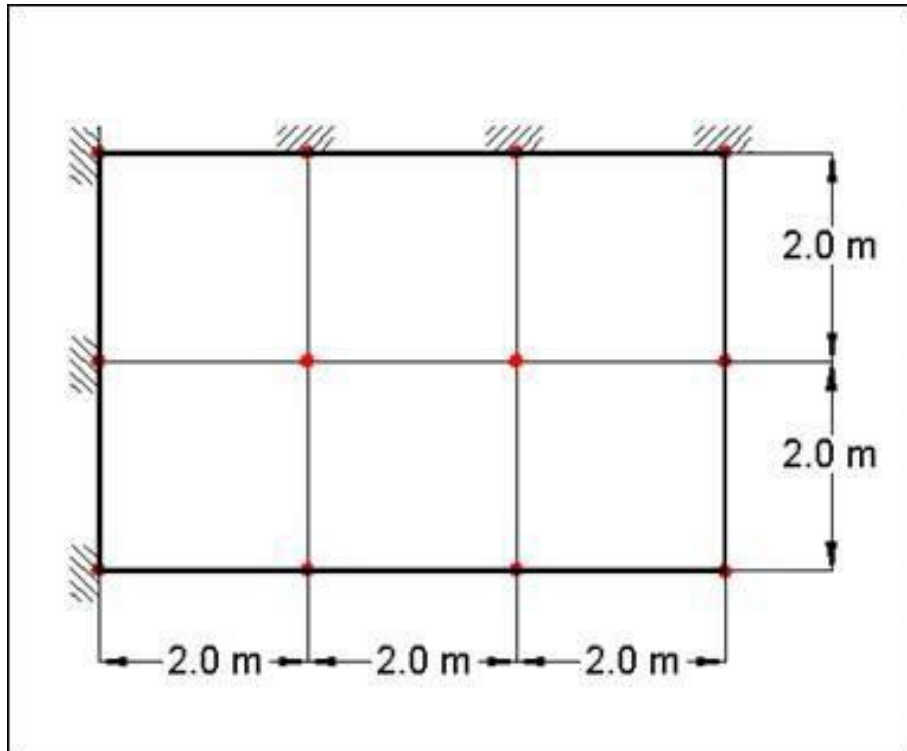
Basic Information of the structure

Attributes	Data
Item Properties	Slab is 300mm thick
Material Constants	E, Density, Poisson, Alfa - specific default values
Supports	The nodes along the two edges are fixed as shown in Figure 3.2
Primary Loads	Charge 1: Dead Weight Charge 2: Pressure Load 300kg / m2. act vertically downwards load 3: uniform expansion 75 degrees, plus 60 degrees upper surface is hotter than the bottom
Combined Loads	Case 101: Case 1 + Case 2 Case 102: Case 1 + Case 3
Type of Analysis	Elastic-Linear

3.3 Starting the program

1. Select the STAAD.Pro icon from the program group in the Windows Start menu or shortcut icon on the desktop (If you have one).

    The STAAD.Pro V8i window opens to the home screen

Figure 2-259:

> **Note**: See "1.3 starting the program" on page 19 for notes about changing the system default drive.

3.4 Creating the new structure

In the New dialog box, offer some initial critical data needed to build the model.

1. Select **File> New**

    or

    select **New Project** low *Project Tasks*.

Figure 2-260:
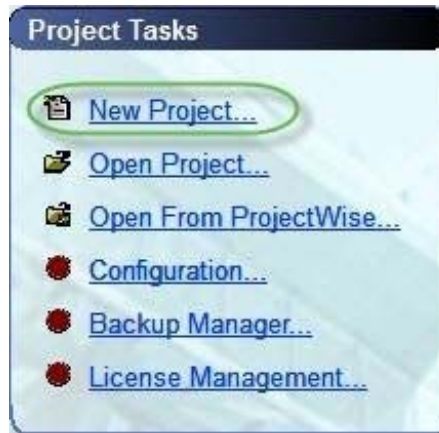
Opens the New dialog box.

Figure 2-261:

The structure type is defined as **Space**,**Plane**,**Floor**, Or **Truss**:

Space

The structure, charge, or both, cause the structure to deform in the three global axes (X, Y and Z).

Plan e

The geometry of the load and the deformation is limited only to the overall plane XY

Floor

A structure whose geometry is limited to the plane XZ.

Trus s

The structure carries pure axial load action. Truss members deemed incapable of carrying shear, bending and torsion.

2. Select **Space**.

3. Select **Meter** as the unit length and **Kilo Newton** as the unit of force.

> **Tip**: The units can be changed later if necessary, at any stage of the creation model.

4. Specify the file name as **Plates Tutorial** and specify a location where the STAAD input file will be located on your computer or network.

    Directly you can type a file path or click [...] to open the Search Folder dialog, which is used to select a location using a Windows file tree.

    After specifying the previous entry, click **Next**.

    The next wizard page, Where do you want to go ?, opens.

Figure 2-262:
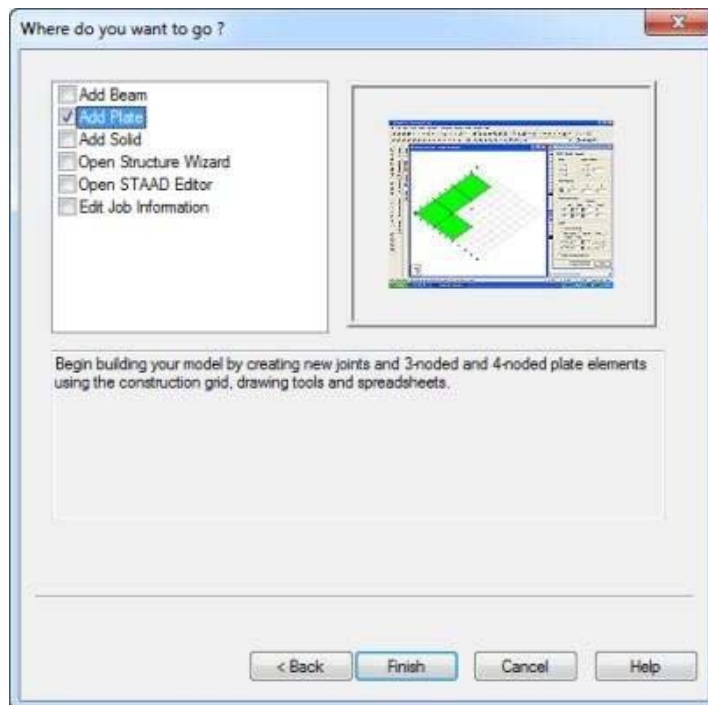
5. Set Checkbox **Add Plate**.

Figure 2-263:
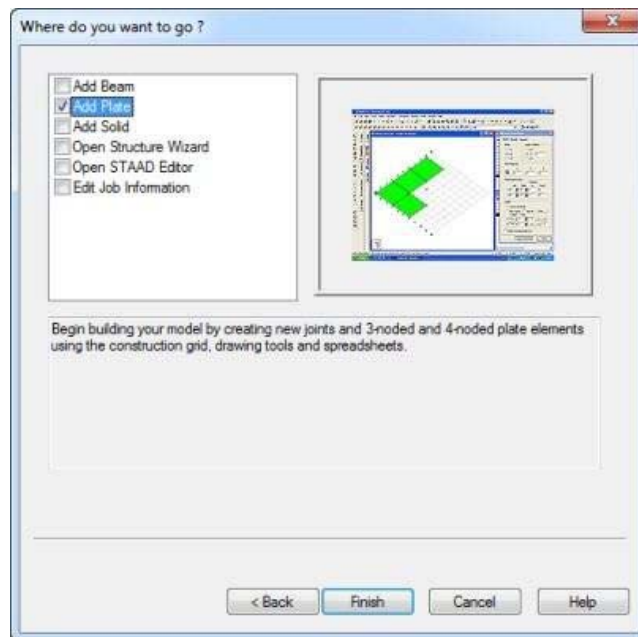
In the dialog, we choose the tools that will be used to initially build the model.

Add Beam, Plate Add, or Add Solid

Respectively, the tools you selected for use in the construction of beams, plates or solids when the GUI is opened.

Open Structure Wizard

Provides access to a library of structural templates that comes equipped with the program. Those template models can be extracted and modified parametrically to reach the model geometry or some of its parts.

Open STAAD Editor

It is customary to create a language model using STAAD commands in the STAAD editor.

All these options are also available from the menus and dialog interface, even after we discard this dialog.

Note: To use the Editor to create the model, choose Open STAAD Editor, click Finish and proceed to **Section 3.8**.

6. Click **end**. The dialog box will be dismissed and professional STAAD.Pro graphical environment is displayed.

3.5 Elements of the STAAD.Pro screen

The main window STAAD.Pro is the main screen where the generation process model is performed. It is important to become familiar with the components of the window before embarking on the creation of the structure. Section 1.5 in the tutorial problem 1 of this manual explains the components of that window in detail.

3.6 Building the STAAD.Pro model

We are now ready to start building the model geometry. The steps and, when possible, the corresponding commands (instructions that are written in the STAAD input file) are described in the following sections.

3.6.1 Generating the model geometry

The geometry of the structure is composed of numbers together, coordinates, membership numbers, the connectivity information of the member element plate numbers, etc. .. From the point of view of the STAAD command file, the commands generate are:

```
JOINT COORDINATES
1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
9 2 0 4 ; 10 0 0 4 ; 11 4 0 4  ; 12 6 0 4
ELEMENT INCIDENCES SHELL
1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6
4 4 3 9 10 ; 5 3 6 11 9 ; 6 6 8 12 11
```

In this tutorial, we will explore 4 different methods to create the model in section 3.2:

1. Using a mix of drawing an item and the ease of copy / paste.

2. Using a mixture of drawing and ease element translational repetition.

3. Through the comfort of the Geometry Structure Wizard menu.

4. Using the mesh generation facilities of the main graphical display.

Creating placas- Method 1

Steps:

Grid Settings

1. Select the option *add plate* to permit anterior plates add

create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X, Y and Z) are represented in the icon in the lower left corner of the drawing area. (Note that we could start this

selecting grid **Geometry> Snap / Grid Node> Plate** also menu option.)

Figure 2-264:
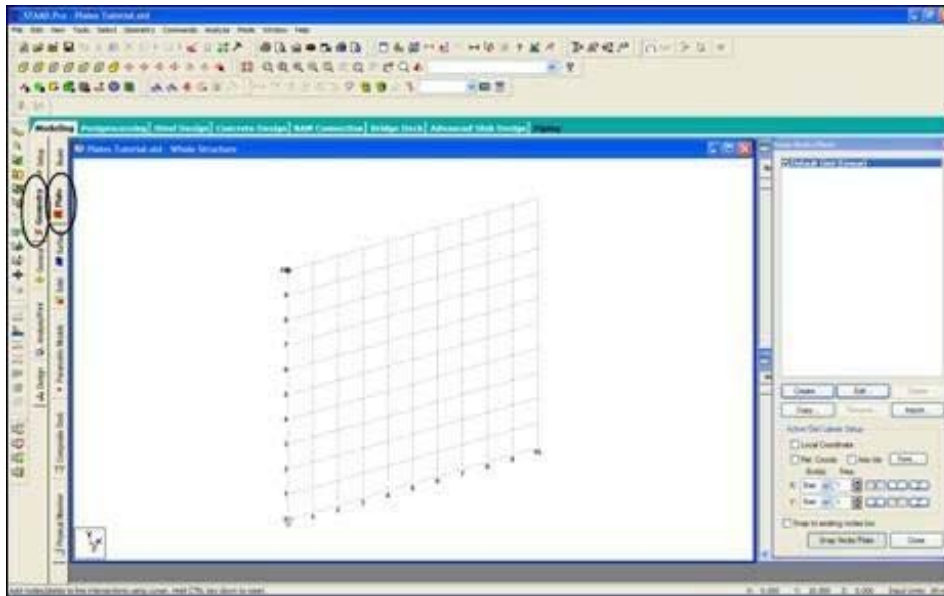
It is worth paying attention to the fact that when we choose the option of *Add Plate* in section 3.4, the control page **Geometry | Plate** is automatically selected.

Figure 2-265:

2. Dialogue appears in the data area on the right side of the screen *Snap Node / Beam*. Click **Create**. A dialog box allows you to set a grid opens.

   Within this dialog box, there is a drop-down list from which you can select the linear, Radial or Irregular shape of the grid lines.

Figure 2-266:

The Record *Linear* is intended for the placement of construction lines perpendicular to each other along a pattern of "left to right - top to bottom", as in the lines of a chessboard. The Record *Radial* allows lines

construction appear in a web style that makes it easy to create circular type models where members are modeled as linear pieces to line segments. The Record *Irregular* can be used to create the grid lines with unequal spacing

found in the overall design or an inclined plane. We will choose **Linear** is

the *default network*.

3. In our structure, the elements are in the X-Z plane. Thus, in this dialogue, let us choose **X-Z** as the plane of the net. The size of the model that can be removed at any time is controlled by the number of *construction lines* to

   left and right of the origin of the axes and the spacing between adjacent lines of construction. By setting **6** as the number of lines to the right of the origin along the X, **4** Z along and a distance of **1** meters between lines along both X and Z

   (See figure below) we can draw a frame 6 m X 4 m, suitable for our model.

   Please note that these values are only an initial grid setting, to enable us to draw the structure, and do not restrict our overall model these limits. In fact, do not even need this grid of 6 m X 4 m. Here we are using the

method requires only a grid of 2 m X 2 m since we are about to take one single component.

Figure 2-267:
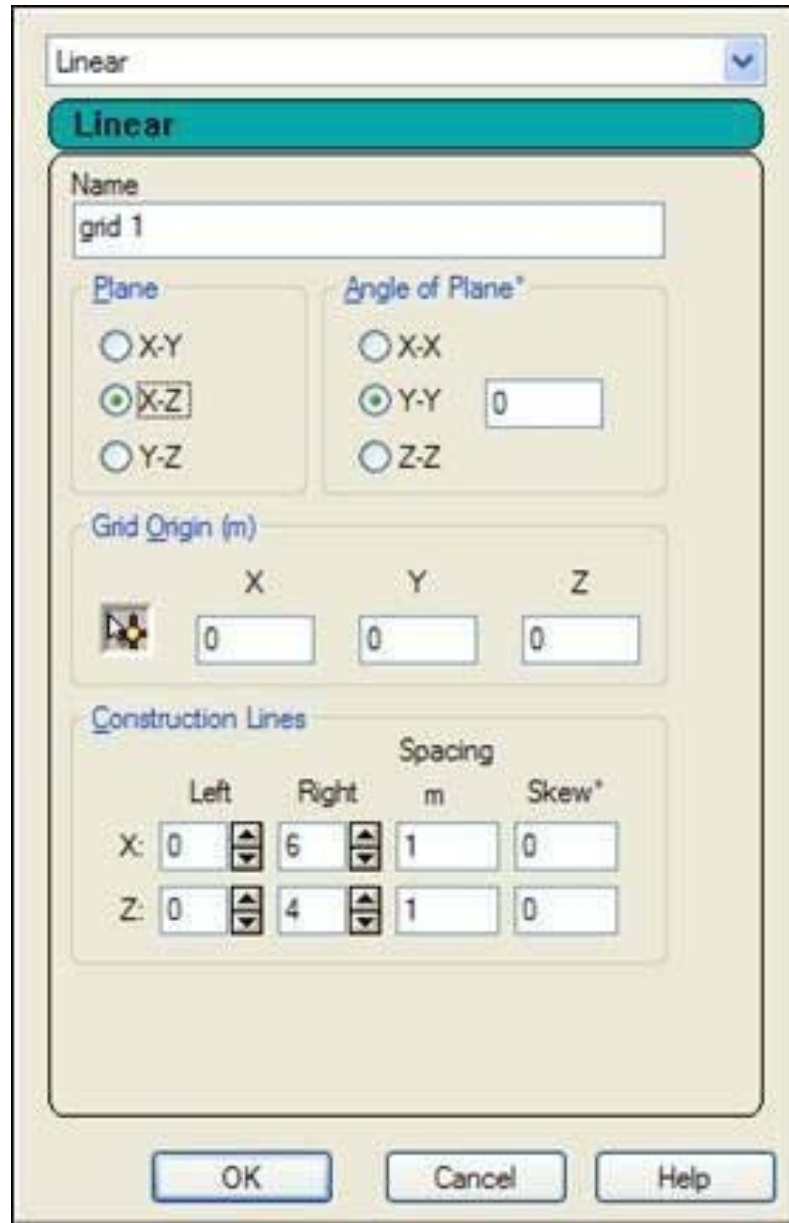
After entering the specifications, provide a name and click **OK**.

In this way, we can create any number of grids. By providing a name, each new grid can be identified for future reference.

Note that these values are only starting grid setting, to enable us to draw the structure, and do not restrict our overall model these limits.

4. In the dialog box *Snap Node / Plate* Check the box of red 1 (linear).

Figure 2-268:
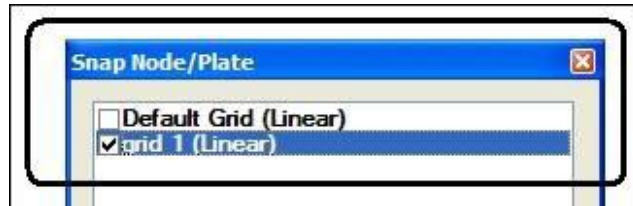
The newly defined network will be displayed.

Figure 2-269:
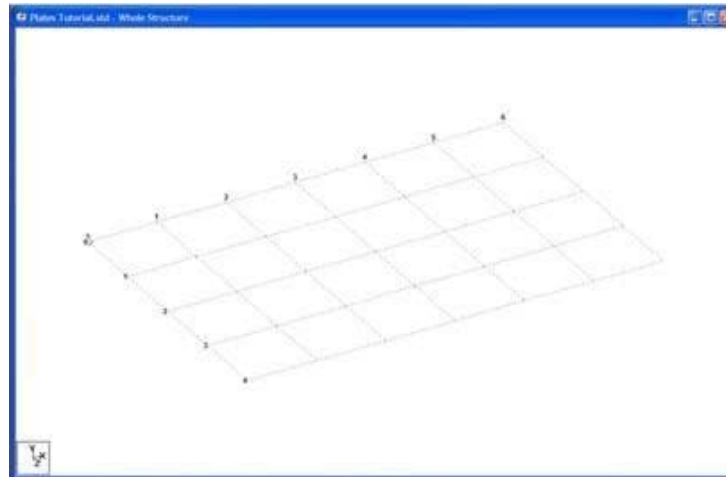
Creation of element 1

a. The four corners of the first element are the coordinates (0, 0, 0), (2, 0, 0), (2, 0, 2) and (0, 0, 2) respectively. Since the button **Snap Node / Beam** it is active by default, using the mouse, click the origin (0, 0, 0) to create the first node.

Figure 2-270:
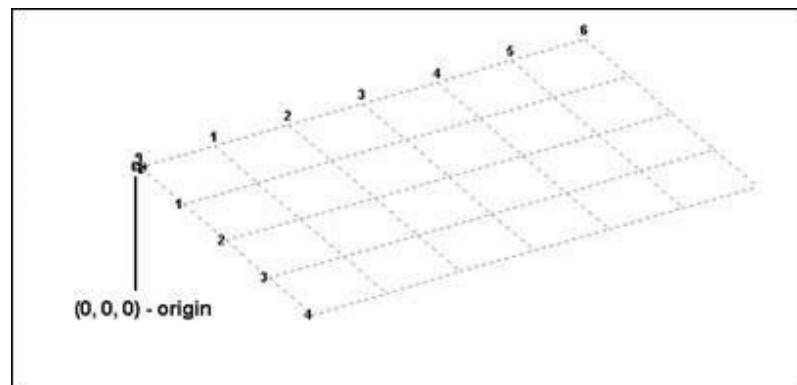
b. Similarly, click on the remaining three points to create nodes and automatically join successive nodes of a plate.

(2, 0, 0), (2, 0, 2) and (0, 0, 2)

The exact location of the mouse can be monitored in the status bar at the bottom of the window that continuously update the coordinates X, Y and Z of the current cursor position.

When you have completed steps 1 and 2, the item is displayed in the drawing area as shown below.

Figure 2-271:
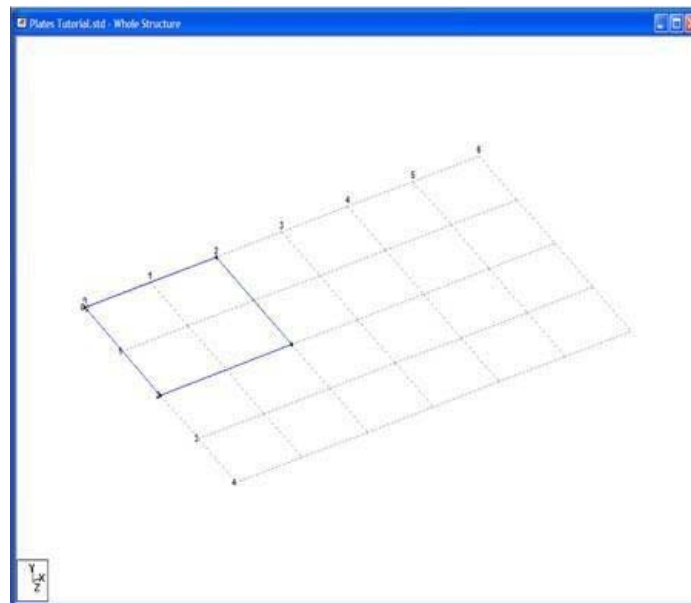
c. At this point, we remove the display of the grid structure. To do this, click Close in the dialog box *Snap Node / Plate*.

Figure 2-272:
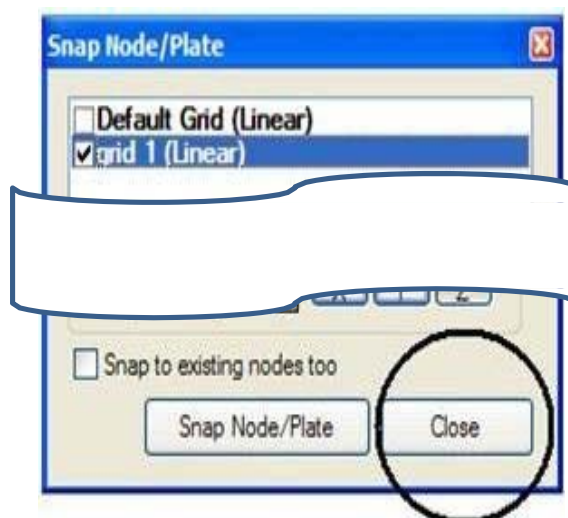
Now the grid and the structure of the main window should look like the figure shown below will be deleted.

Figure 2-273:
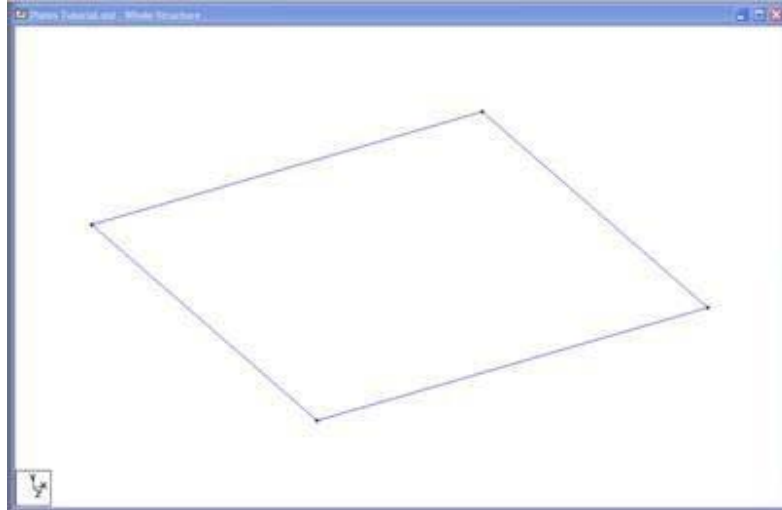
d. It is very important that you save your work frequently to avoid losing data and protect our investment of time and effort against power outages, system problems or other unforeseen events. To save the file, pull

down menu *Archive* and select the command *Save*.

e. To facilitate identification, entities drawn on the display can be labeled. Let us see the numbers on the board. (See 'frequently performed tasks' section at the end of this manual to learn more about the change node labels / beams / plates).

The following figure shows the plate number that appears in the structure. The structure of the main window should look like the figure shown below.

Figure 2-274:
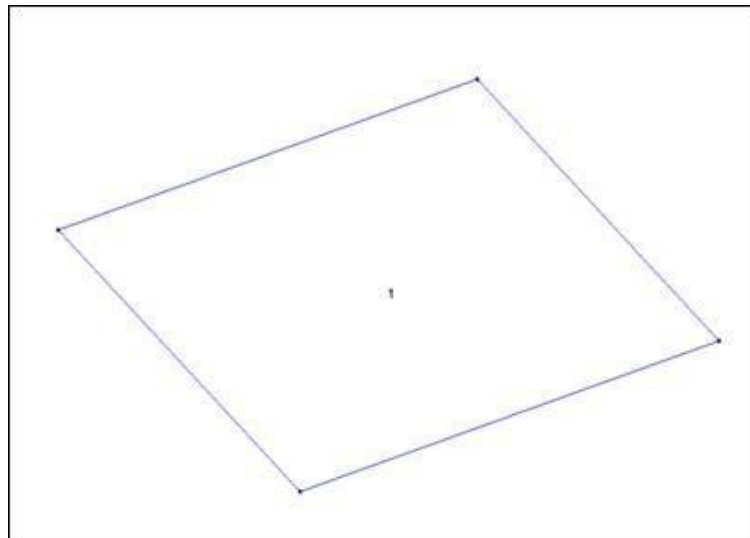
If you're feeling adventurous, here's a little exercise for you. Change the font of the labels by selecting plate **View> Options** and then selecting the appropriate tab (label plate) of the Options dialog box.

Creation of element 2

a. Examining the structure shown in section 3.2 of this tutorial, we see that the remaining elements can be easily generated if the existing board could copy and then paste the copied unit at specific distances. The program has made

    ease of copying and pasting is below the menu *Edit*.

    First, select Plate 1 using the **Plates Cursor** .

b. Click the right mouse button and choose *copy* the pop-up menu (or select **Edit> Copy**). Again, click the right mouse button and select *Paste Plates* (Or select **Edit> Paste Plates**).

c. From this place allows us to create a single copy at a time, all we can create from element 1 is 2 component The four nodes are element 2 distance X = 2, Y = 0 and Z = 0 of the element 1. Thus, in the dialog box that comes up,

    provide **2**,**0**, And **0**for X, Y and Z respectively and click **OK**.

Figure 2-275:
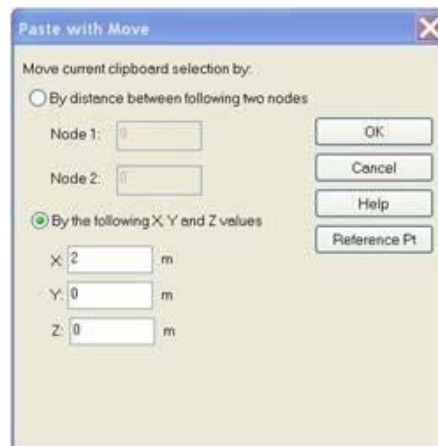
The model will now look like the one shown below.

Figure 2-276:
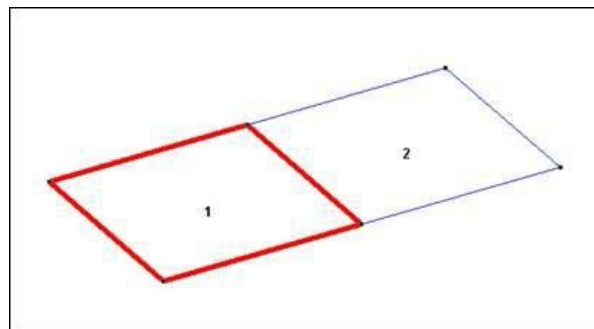
Creation of element 3

a. The nodes of the element 3 are X = 4m away from the element 1 Thus, we create the third element repeating steps 8-10 except X 4 m expected in the dialog *Paste with Move*.

Alternatively, we could use an element 2 as the basis for item 3, in which case it will be the increased X 2m. If you make a mistake and end up hitting the item in a wrong location, you can undo the operation by selecting

of *Undo* menu *edition*.

After creating the third element, the model should it look like the one shown below

Figure 2-277:
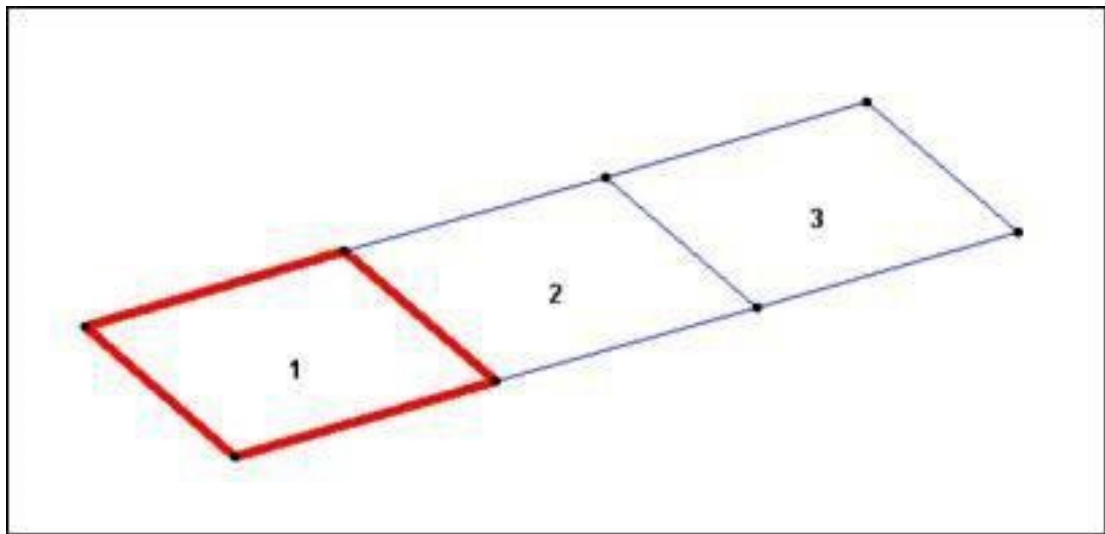
Click anywhere on the screen for the highlighted desrealzar plate.

Creation of the elements 4, 5 and 6

a. The elements 4, 5 and 6 are identical to the first three elements except that the nodes are at a distance of 2m Z distance elements corresponding nodes 1 through 3 therefore can use the technique of *copy-paste* and Z specify increasing

how come **2**m

Select the three existing boards doing a "rubber band" around them using the mouse.

b. Right-click and select **Copy** the pop-up menu (or select **Edit> Copy**). Again, click the right mouse button and select **Paste Plates** (Or select **Edit> Paste Plates**).

c. Enter **0,0**, And **2**for X, Y and Z respectively in the box *Paste with Move* that appears. Then click on **OK** and note that three new elements are created.

Since some elements still stand, tap anywhere on the drawing area for a desrealzado those elements.

The model, with all six plaques generated will now be as shown below.

Figure 2-278:
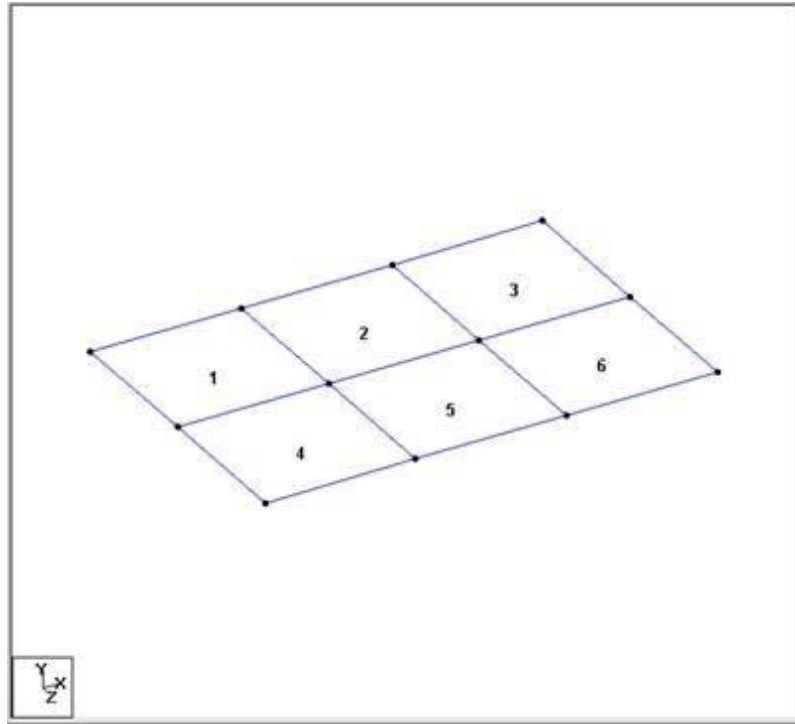
To continue with the rest of the data allocation, go to section 3.6.2. But if you want to explore the other methods of creating this model, the current structure will have to be removed completely. This can be done using the following procedure.

1. Select **Select> By All> By Geometry**. The complete structure will be highlighted (in elements in red)

2. Press the key **DELETE**.

   A message box appears to confirm deletion of the selected plates.

Figure 2-279:
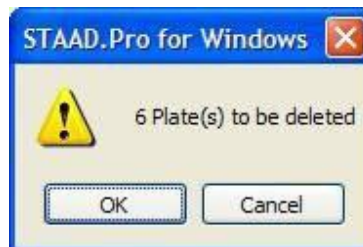
3. Click on **OK**

A message box opens indicating that you have created the solitary nodes; ie, disjoint, and confirm the deletion.

4. Click on **Yes**. Now the complete structure is eliminated.

Cr e d a tio n P la cas - M ét or d to 2

Steps

Creation of Element 1

a. In this method, we use the command *Translational Repeat* STAAD to create our model. To use this service, we need at least one existing entity to use as a base for translational repetition. Then, follow the steps 1-7 of 'Method 1' to create the first element. Once this is done, our model will look like the one shown below.

> **Note** If you have problems to get the dialog box network settings, go to the menu *geometry* and select *Snap / Grid Node Plate*.

Figure 2-280:
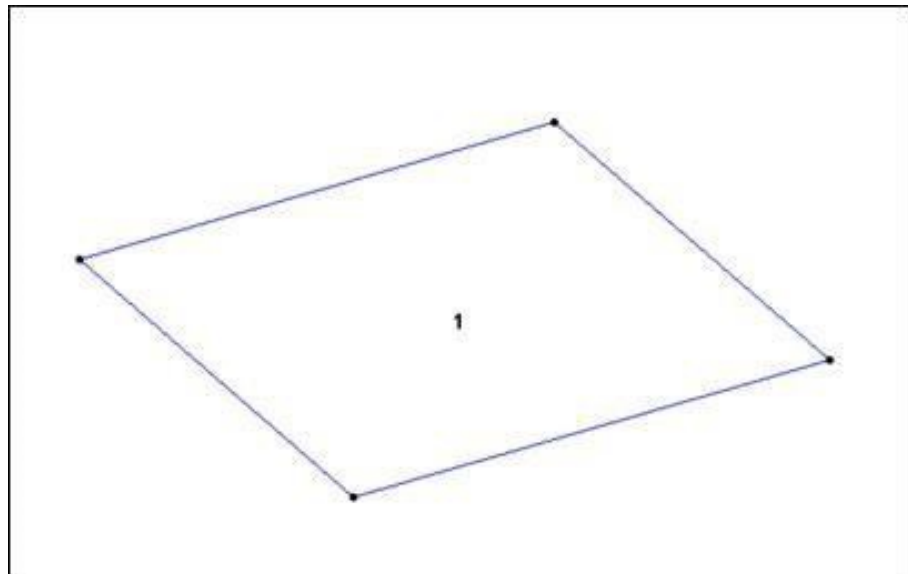
Creation of the elements 2 and 3

a. In method 1, it required two separate executions of the copy / paste to create the elements 2 and 3 That's because this facility does not contain a provision to specify the number of copies you would like to create. Translational Repetition is a command where that provision is available.

Select plate 1 using *Plates Cursor* [icon]. (See 'frequently performed tasks' section at the end of this manual to learn more about selection of plates).

b. Select the tool **Translational Repeat**

Figure 3 31
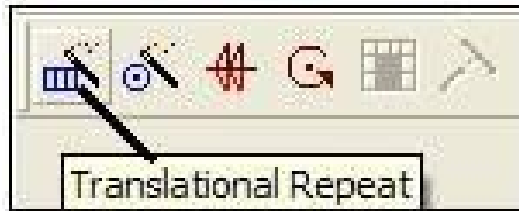
Or select **Geometry> Translational Repeat**.

Open the dialog *repeat 3D*. By default (when the option *Only Geometry* is not selected), all loads, properties, design parameters, freed member, etc. on the selected features are automatically copied along with the entities.

Making a new option *Only Geometry* The replay will be made using translational only the geometry data. In our example, it does not matter because there are other attributes that have been assigned yet.

c. To create the elements 2 and 3 in the X direction, specifies the *Global address* how come **X**,*No of Steps* how come **2** and *Default Step Spacing* (Along X) as **2m**. Option *Link Steps* applicable when the newly created units physically existing units are extracted,

and when you want to connect to those using the members. *Renumber Bay* enables us use our own numbering scheme for the entities to be created, rather than using a sequential numbering if the program does no instructions provided. Let these boxes unchecked. Then click on **OK**.

Figure 2-281:
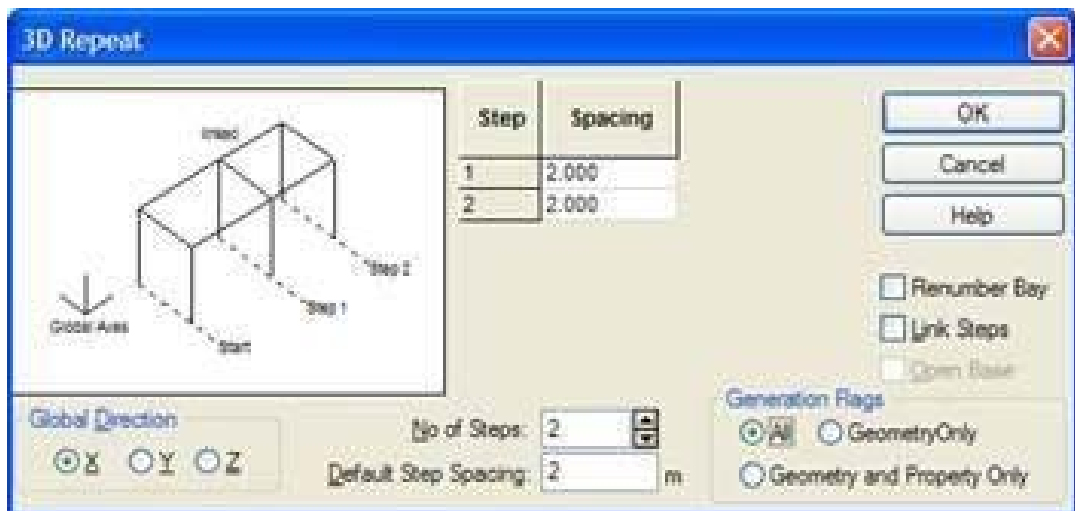
d. Since even the element 1 is highlighted click anywhere in the drawing area to desrealzarlo.

The model will now look as shown below.

Creation of the elements 4, 5 and 6

Figure 2-282:
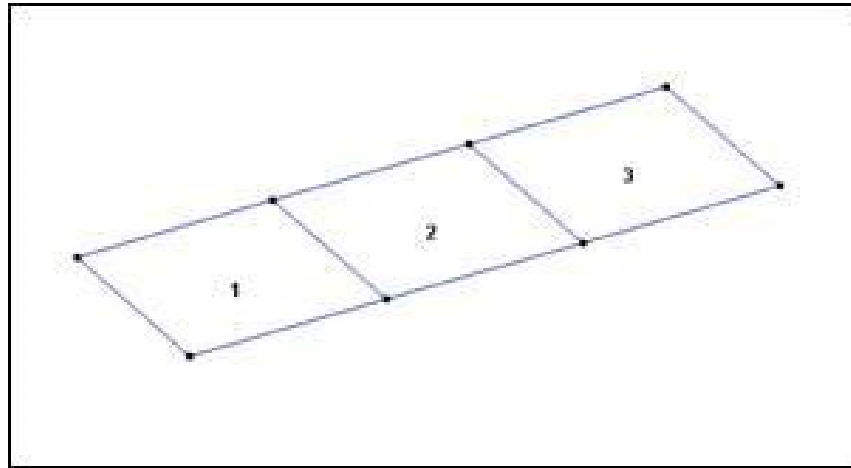
a. Let us follow the same approach translational repetition to create these items. Select all three sombreándolas existing plaques using the mouse. Make sure before you do this, the cursor type is *Course Plates* If not, will not be selected plates.

b. Repeat steps 3 and 4, but this time, specify the address as Global **Z**,*No of Steps* how come **1**and the default spacing as m step 2 Leave all boxes unchecked. Then do click **OK**.

Now all 6 elements are created.

Since some of the dishes stand out even click anywhere in the drawing area to desrealzarlos.

Our model will now look like the one shown below.

Figure 2-283:
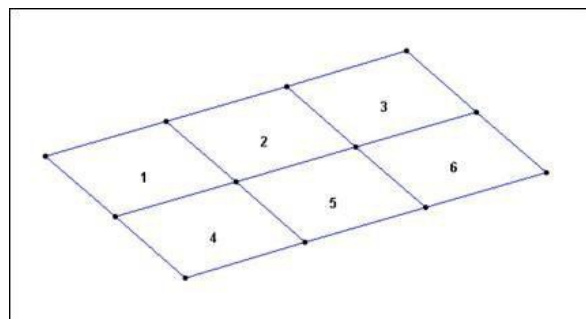
To continue with the rest of the data allocation, go to section 3.6.2.

But if you want to explore the other methods of creating this model, the current structure will have to be removed completely. This can be done using the following procedure.

1. Select > **By All**> **By Geometry**. The entire structure will be highlighted

2. Press the key **DELETE**.

   Opens a message box to confirm the deletion of the selected plates.

Figure 2-284:

3. Click on **OK**

   A message box opens indicating that orphans were created nodes and to confirm your deletion.

4. Click on **Yes**. Now the entire structure is eliminated.

Creating plates - Method 3

Procedure

There is a facility in STAAD called *Structure Wizard* that provides a library of predefined prototypes structures such as Pratt Truss Armor, Armor Northlight cylindrical frame, etc. .. A surface entity such as a slab or wall, which can be defined using elements with 3 or 4 nodes on the plate, is a prototype . We can also create our own library of prototype structure.

From this wizard, a structural model can be generated parametrically and then can be incorporated into our structure. *Structure Wizard* can therefore be thought of as a store where you can find various components and assemble a structure

complete. Even with *Structure Wizard* You can import models created from other packages software; eg AUTOCAD.

1. Select **Geometry**> **Structure Wizard Run**.

   Opens the wizard to the structure as shown below.

Figure 2-285:
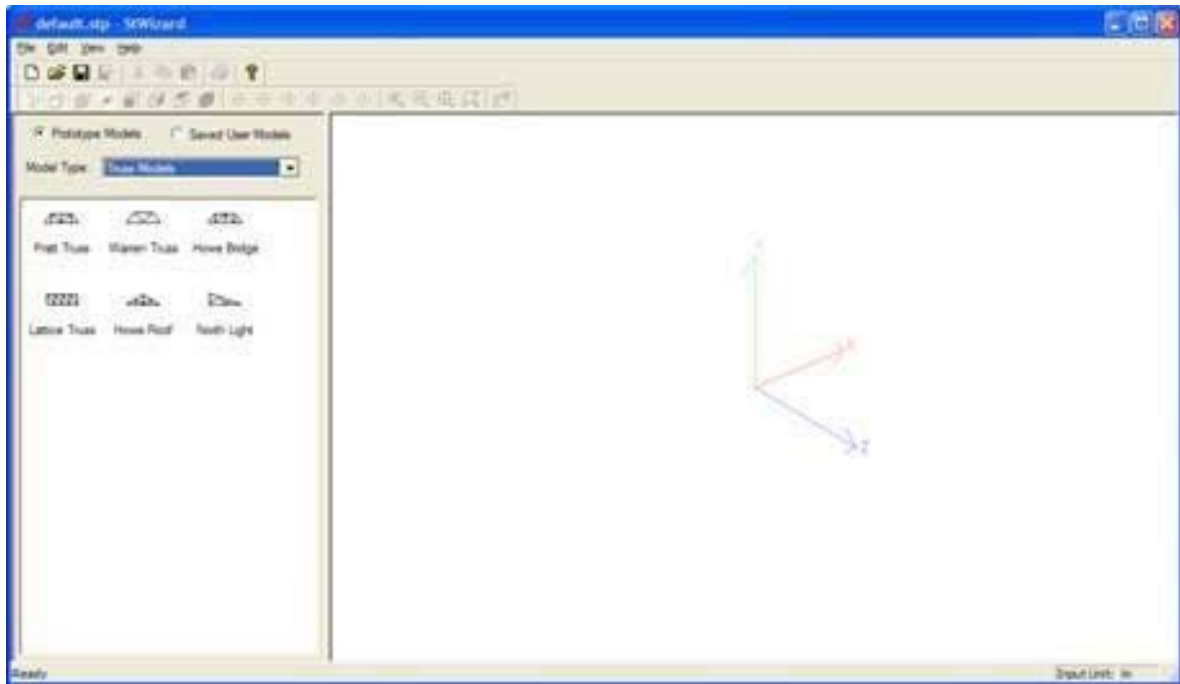
> **Note**: The option *Open Structure Wizard* In the dialogue *Where do you want to go?* in the initial stage of creating a new dialogue structure - see Figure 3.9 - also brings this facility.

2. It must be specified before the length unit of the generation of a model. Select **File> Select Units** the wizard window structure. In the dialog box to select units that opens, select **meters** and click **OK**.

Figure 3 40
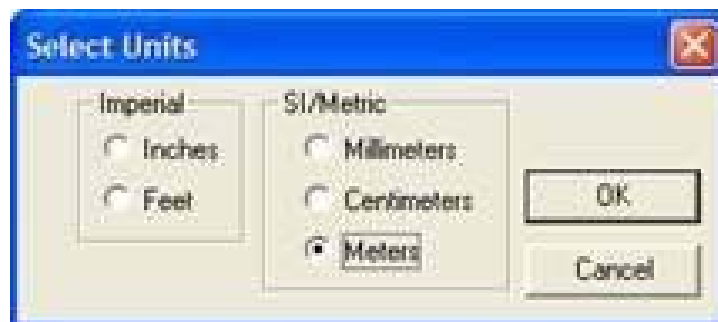
3. The box model type list, select **Surface / Plate Models** as shown below.

Figure 2-286:
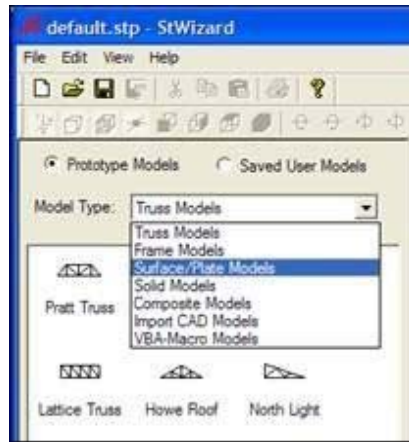
4. Double-click the option **Quad Plate** or drag the option **Quad Plate** on the right side of the assistant structure as shown below.

Figure 2-287:
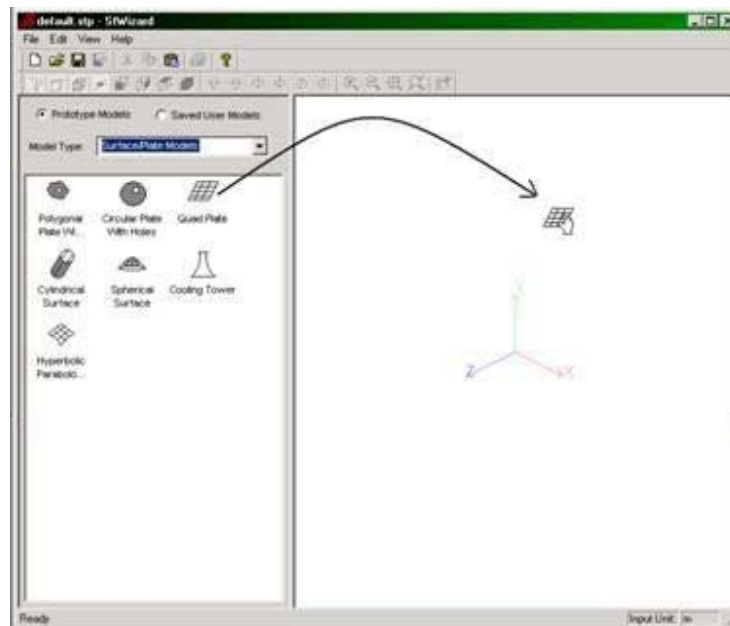
5. A dialog box appears with the name of *Select Meshing Parameters*. In this box, specify, inter alia, two major pieces of information - a) size limit (or super element as it is commonly known) that the individual elements are generated b) the number of individual elements that must be generated. (A) is defined in terms of the X, Y, Z of the corners A, B, C and D.

    (B) is defined in terms of the number of divisions along the sides AB, BC, etc. ..

    Let us provide the **Corners** (Corners) **Bias** (Bias) and **Divisions** model as shown in the following figure. Then click Apply.

Figure 2-288:
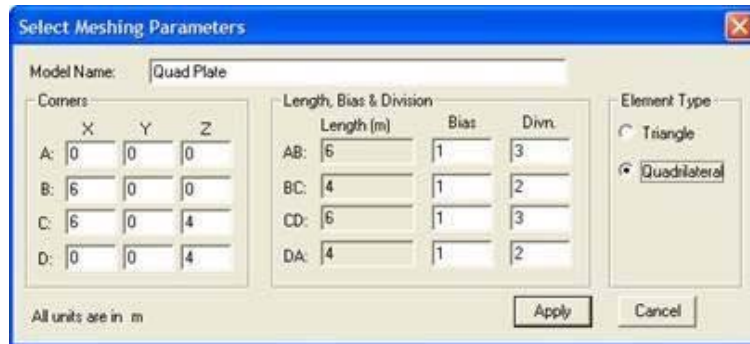
> **Tip** If you made a mistake and want the previous dialog box appears, right click in the drawing area and select **Change Property (**change of ownership) in the popup menu.

Figure 2-289:
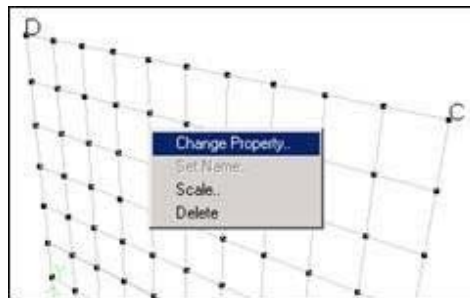
6. To transfer the model in the main window, select **File> Merge Model with STAAD.Pro Model** or click on the command **Transfer Model** located on the toolbar of the window as shown below.

7. When the following message box appears, confirms the transfer by clicking the button "**Yes**".

Figure 2-290:

The dialog box shown in the following figure appears. If we had an existing structure in the main window, in this dialog box, we will be able to provide the coordinates of a node in the structure in the main window you want to connect the piece being brought from the wizard.

If there is no existing structure, this box is a way to specify any distances along the X, Y and Z by which we want the unit (to be brought from the wizard) to be displaced.

Figure 2-291:

8. In our case, since we have an existing structure in the main window, or want to change the unit for any amount, we click **OK**.

The model will now be transferred to the main window.

Figure 3 50
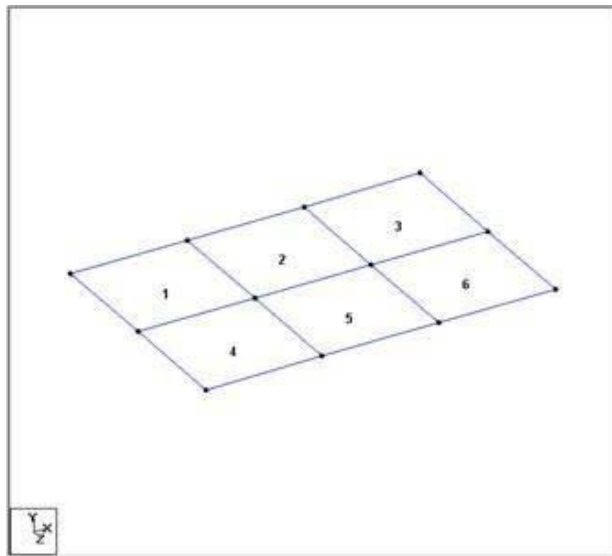
To continue with the rest of the data allocation, go to section 3.6.2. But if you want to explore the other methods of creating this model, the current structure will have to be removed completely. This can be done using the following procedure.

1. Select the menu option **Select> By All> By Geometry**. The entire structure is highlighted.

2. Press the key **DELETE**.

    Opens a message box to confirm the deletion of the selected plates.

Figure 2-292:

3. Click on **OK**

    A message box opens indicating that orphans were created nodes and to confirm your deletion.

4. Click on **Yes**. The whole structure is deleted.

Creating plates - Method 4

Procedure:

The graphical user interface STAAD.Pro contains a mechanism to generate a mesh of elements of a limit (or super element) defined by a set of corner nodes. This facility is in addition to what we saw in the method 3 The limit must form a closed surface must be a plane, but the plane can be inclined to any global levels.

1. The first step is to define the boundary selection of corner nodes. If there are no nodes must be created before they can be selected. Then, select Tool *Snap Node / Quad Plates*.

Figure 2-293:
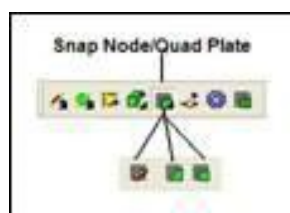
> **Note**In STAAD.Pro 2007, the amount of space occupied by a series of icons on the screen has recovered from a series of similar icons to collapse in a single icon. You can change the active icon while holding down the left mouse button or click the button. The icons that have this property are identified with a black triangle in the bottom right corner.

Icon Snap Grid Node supports three icons:

Figure 2-294:
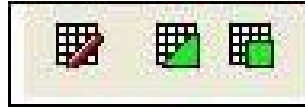
They are the tools: **Snap Node Beam**,**Snap Node Quadrilateral Plate**,

And

Snap Node Triangular Plate.

Alternatively, select **Geometry> Snap / Grid Node> Plate> Quad**.

2. Dialogue appears in the data area on the right side of the screen *Snap Node / Plate*. (We have seen this dialog in methods 1 and 2). As before, click **Create**. Opens the dialog line. In our structure, the elements are in the

X-Z plane. Thus, in this dialogue, let us choose **X-Z** as the plane of the net. Size model that can be removed at any time is controlled by the number of construction lines left and right of the origin of axes and the spacing between lines of adjacent construction. All we want is the 4 nodes of the corners of super–

element. Then, let **1**as the number of lines to the right of the origin to along X and Z, and a space of 6m between lines along X and **4m** along Z.

Figure 2-295:
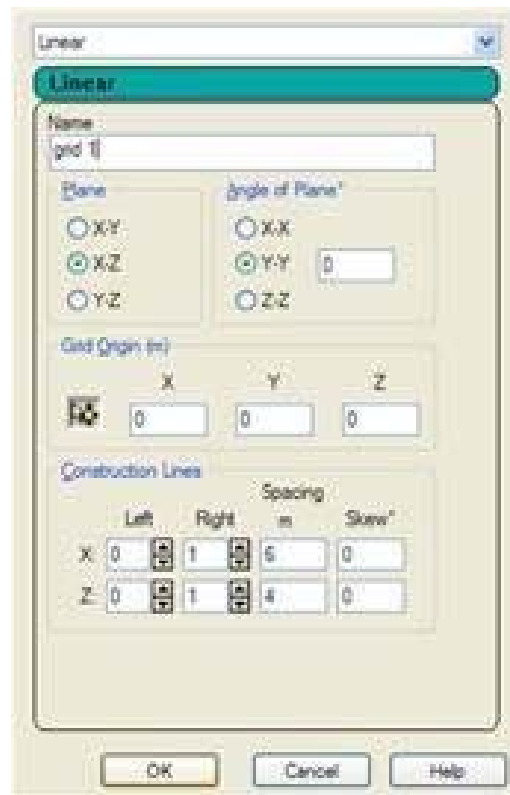
The main drawing area now look like the one shown below.

Figure 2-296:
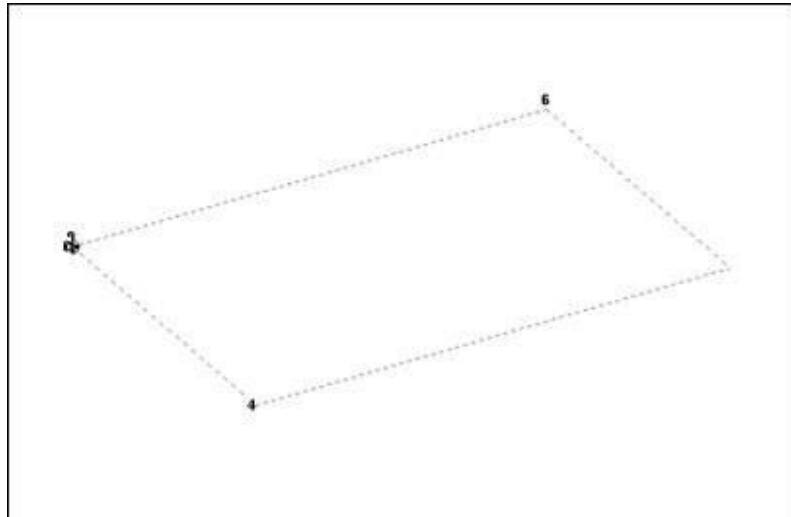
3. To start creating the nodes, first we activate the button **Snap Node / Plate** clicking on it. Hold down the 'Ctrl' key, click on the four corners of the grid, as shown below. These four points represent the four corners of the slab and are (0, 0, 0), (6, 0, 0), (6, 0, 4) and (0, 0, 4). Indeed, keeping the

key **Ctrl** He pressed and clicking on grid points, is a
how to create new nodes without connecting nodes with beams or plates. If you do not hold the key **CTRL**, The nodes would be connected.

Figure 2-297:
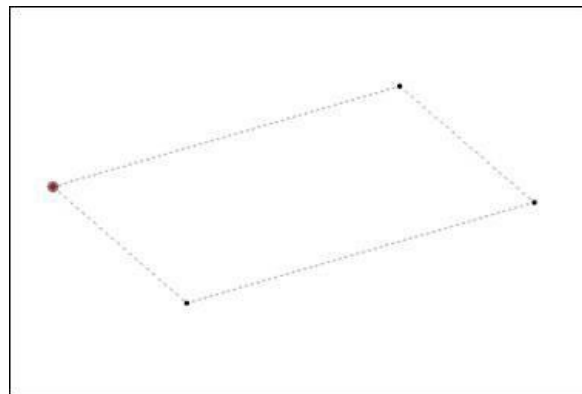
If nodes are not visible, press **SHIFT + K**.

It should be noted that the purpose of the above four steps was simply to create the four nodes. Consequently, any of the various methods available in the program could be used to create such nodes. We could write the data in the editor, or the grid tables Control Area page

geometry of the plate, or even select **Geometry> Grid Snap / Node Beam** for
graphically create the points.

4. Click on **Close** dialogue *Snap Node / Plate*.

Figure 2-298:
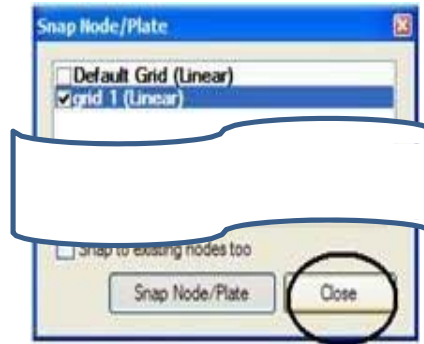
We are now ready to use the second method available in the mesh generation program.

5. Select the tool **Generate Surface Meshing**

Figure 2-299:

or

Geometry> Generate Surface Meshing.

6. Select the points that form the boundary of the super-element from which the individual elements are created. The four points that we just created are those four points. So we click the 4 points in succession as shown below. Finally, closes the cycle by clicking the start node (or the first click point) again.

Figure 2-300:
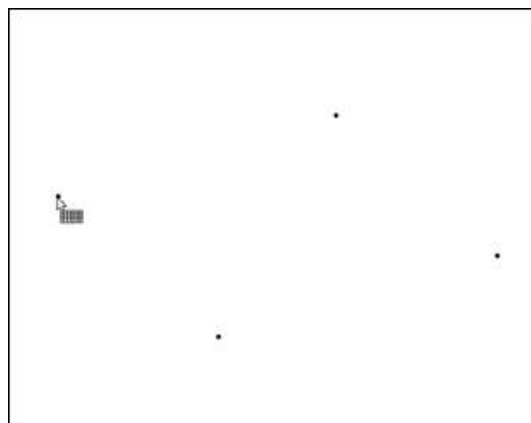
As you click on the start node of the second half, the following dialog box opens. Select the option **Quadrilateral Meshing** and click **OK**.

Figure 2-301:

7. Dialogue *Select Meshing Parameters* (As we saw earlier in method 3), appears. Note that this time however, the data from the four corners are automatically filled. The program uses the coordinates of the four nodes that have chosen to define A, B, C and D provide the **Bias** and division model as shown in the following figure. Click **Apply**.

Figure 2-302:

As We press **apply** Our model appears in the drawing area as shown below. Press the key **Esc** to exit mesh generation.

Figure 2-303:
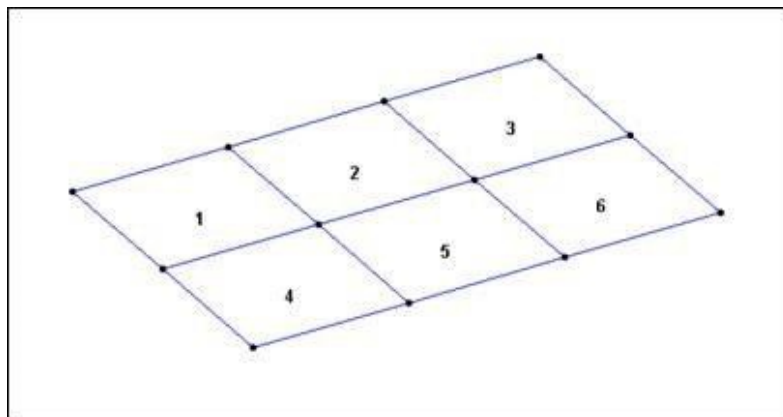
3.6.2 Changing input length units

As a matter of convenience, to specify the properties of the element of our structure, it is easier if our units of length are *inches* instead of *subway*. This will require changing the current length units input. The command to generate is:

```
UNIT CM KN
```

Steps:

1. Select the herrramienta **Input Units**
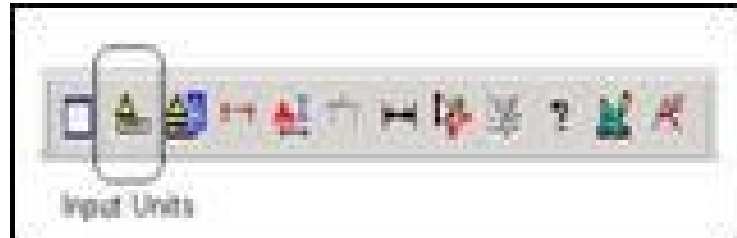
Figure 2-304:

or

Tools> Set Current Input Unit

Opens the Configure dialog units current input.

Figure 2-305:

2 Define the units of length and the centimeter click accept.

3.6.3 Specifying the properties of elements

As the properties are assigned to members, properties must also be assigned to elements of the plate. The property required for the plates is the plate thickness (or the thickness at each node if the slab element has a varying thickness).

The corresponding command to be generated in the STAAD command file is:

```
ELEMENT PROPERTY
THICKNESS 1 TO 6 30
```

Steps

1. Select the tool **Property Page** located on the toolbar *Structure Tools*.

Figure 2-306:
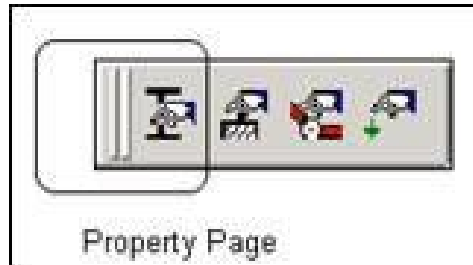
Or select **General | Property** the left side of the screen as shown below.

Figure 2-307:
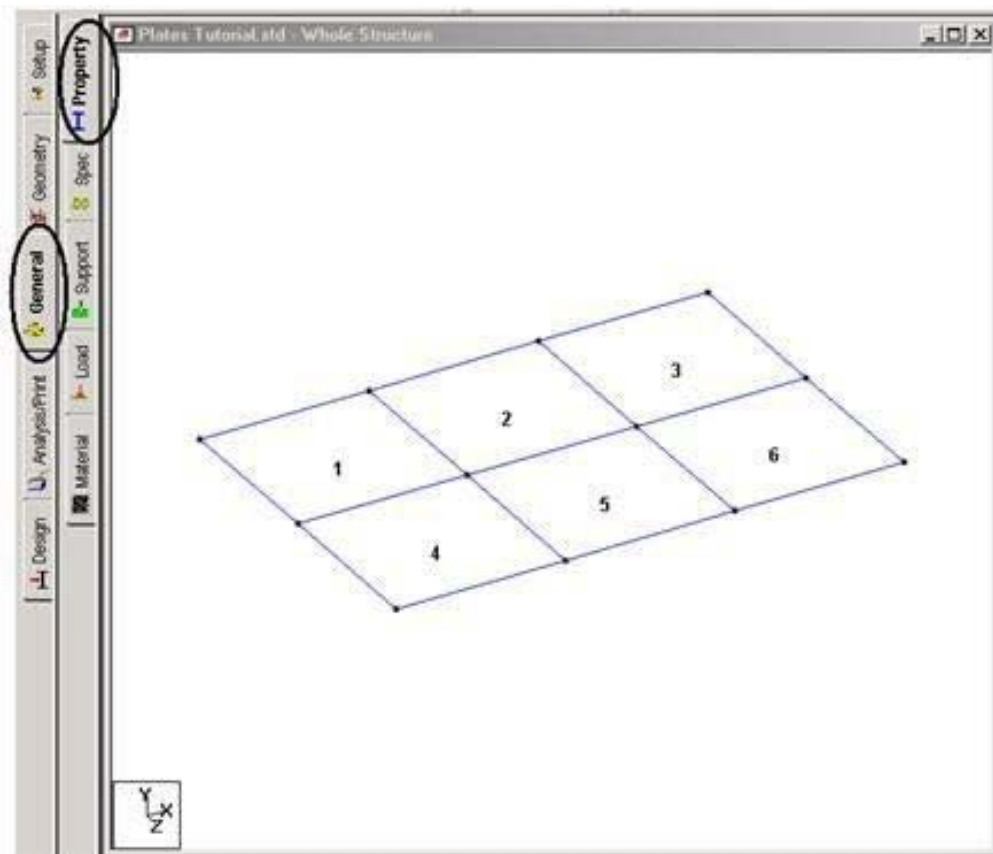
The Properties dialog - Whole Structure opens.

2. Click on **Thickness ...**.

Figure 2-308:
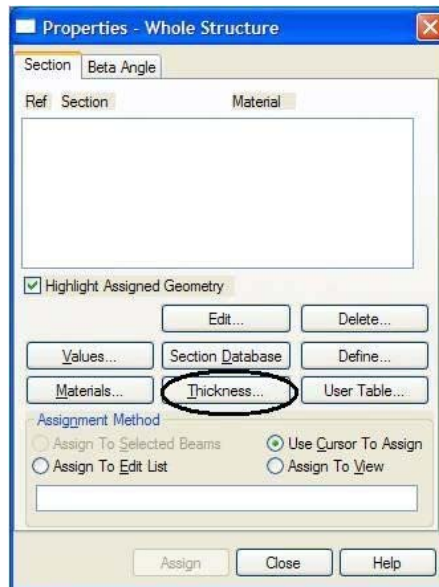
Dialogue Element Plate / Surface Property opens.

3. The dialog box shown below appears. Provide the bulk of plate as **30 cm**. Note that the field named *Material* is currently in the current mode. If we keep this way, the material properties are assigned

   concrete (E, Poisson, density, alpha, etc.) together with the plate thickness. The material property values thus assigned shall be the default program. (To see the default values, press **materials** in the dialog box shown in Figure

   above). Since we want to assign only the defaults, keep the material in the activated mode. Then click on **add**, Followed by the button **close** as shown below.

Figure 2-309:
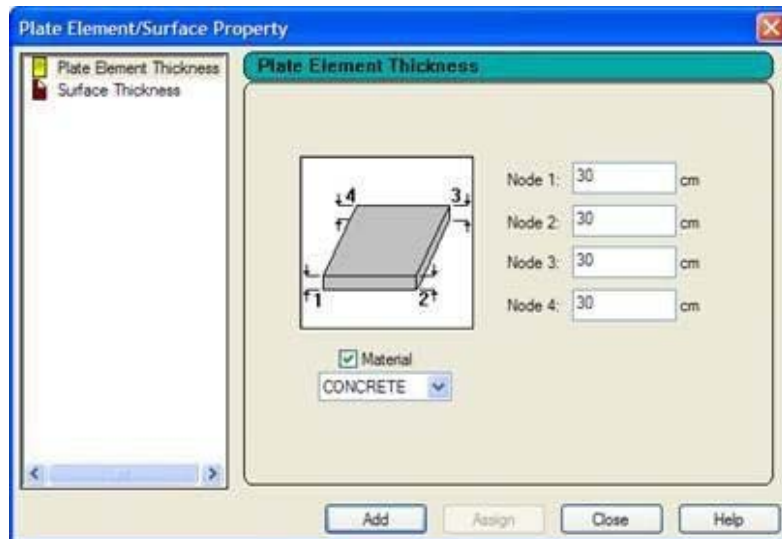
At this point, the Properties dialog box will be as shown below.

Figure 2-310:
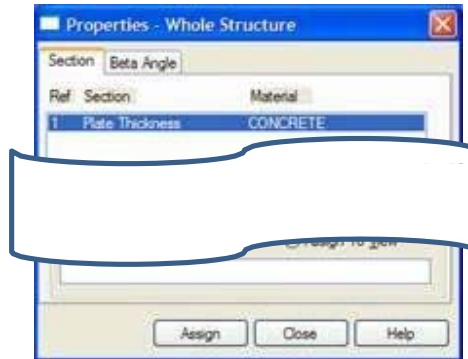
4. Since we want the bulk is applied to all elements of the structure, we select the *allocation method* call **Assign to View** and then click **Assign** as shown in the previous figure.

   The following message dialog box opens. Click the button "**Yes**"To confirm.

Figure 2-311:

The structure will now look as shown below.

Figure 2-312:
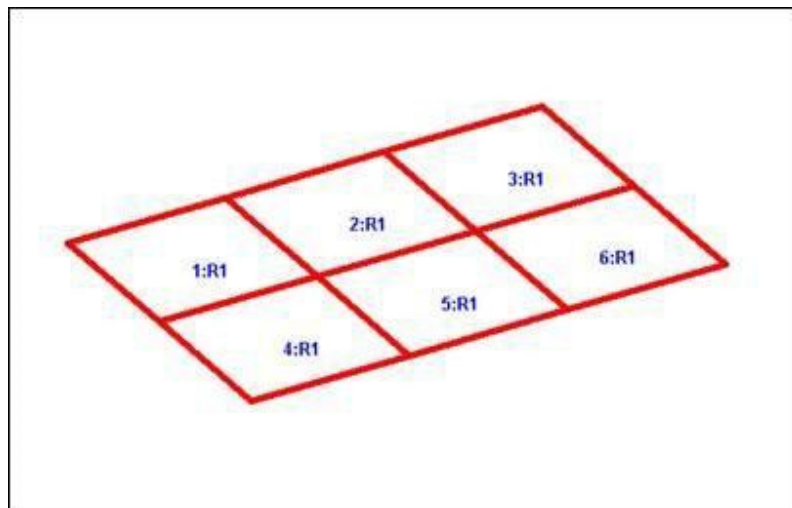
5. Click anywhere in the drawing area to desrealzar selected entities.

We do this only as a safety measure. When an entity is highlighted, clicking on any option to set it can cause an unwanted attribute being assigned to that entity.

3.6.4 Specifying the material constants

In Section 3.6.3, we kept the check box Material "on" to assign the properties of the element. Consequently, the material constants (E, density, Poisson's ratio, etc.) of concrete are assigned to the plates along with the properties, and obtained the following commands in the command file:

```
UNIT METER KN
CONSTANTS
E MEMB 2.17185E + 007 1
TO 6
POISSON 0.17 MEMB 1 TO 6
DENSITY 23.5616 MEMB 1 TO 6
ALPHA 1.0E-5 MEMB 1 TO 6
```

Therefore, there is no need to allocate constants separately. However, if we would not have assigned as before, we could go to the option menu **Commands> Material Constants** and explicitly assign as shown in the following figure.

Figure 2-313:
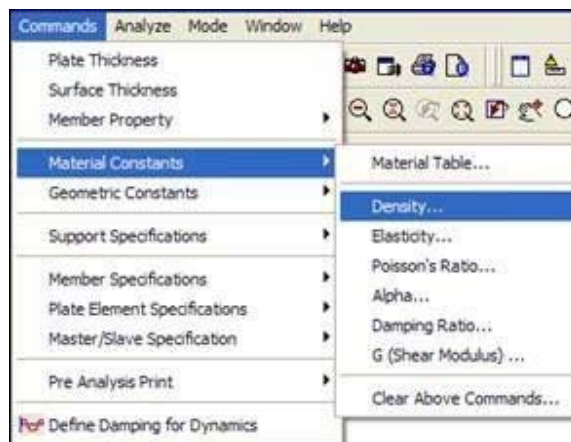
3.6.5 Specifying Support

The slab is supported fixedly (recessed) along the entire length of two sides. However, when modeled as elements of the plate, the supports can be specified only in the nodes along the edges and at any point between the nodes. Therefore, it is clear that if one is interested in better model boundary conditions, the slab must be modeled using a larger number of elements.

In our case, we have to generate the commands are:

SUPPORTS

`January 2 4 5 7 10 FIXED`

Steps:

1. To create media, select the tool **Support Page** located on the toolbar of the structure as shown below.
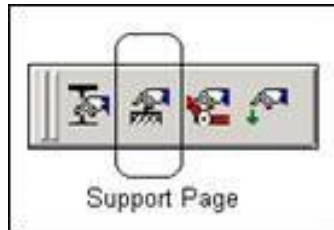
Figure 2-314:

Alternatively, one can go to **General | Support** from the left side of the screen.

Figure 2-315:

2. In any case, the dialog box opens aid as shown in the following figure.

3. To facilitate the identification of the nodes where we want to place the brackets, alternating the display of **node numbers**.

4. Since we know that the nodes **1**,**2**,**5**,**7**,**4**and**10** They are associated with the fixed support, using *Nodes Cursor* Select these nodes.

5. Click Create in the dialog brackets as shown below.

Figure 2-316:
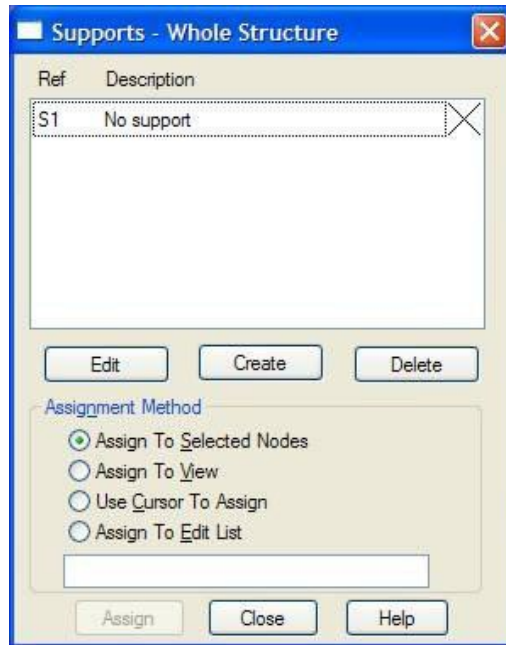
6. The dialog box shown below appears. The Record **Fixed** becomes the default value that is appropriate for this case. As shown below, click **assign**.

Figure 2-317:
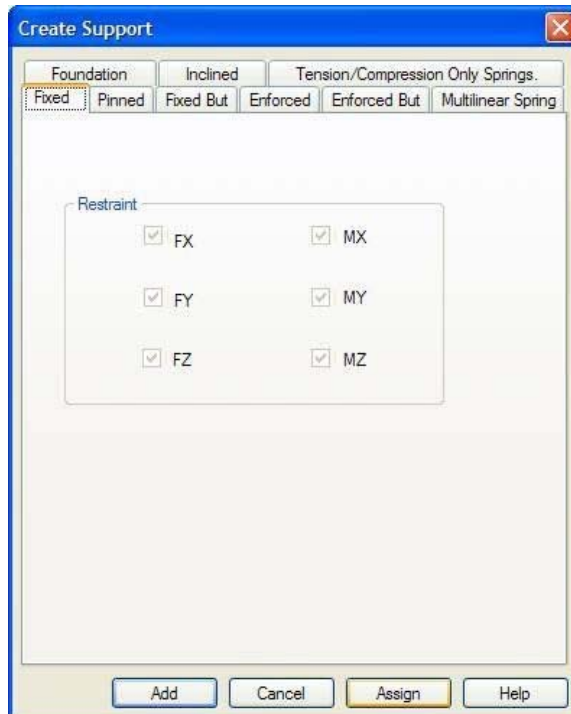
> **Note** It is important to understand that the button **Assign** asset is what we did in step 4 above. We had not selected nodes before reaching this point, this option would not have been activated.

After the supports have been assigned, the structure will look as shown below.

Figure 2-318:
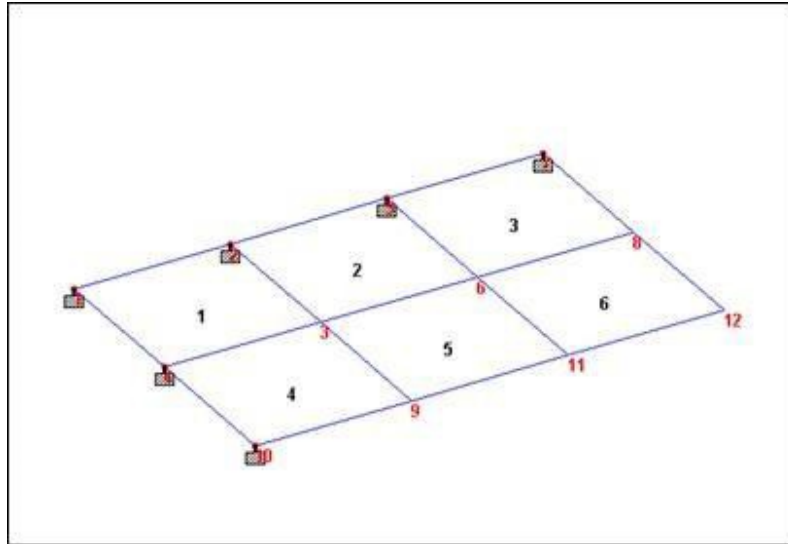
3.6.6 Load cases specifying primary

Three cases of primary charge should be created for this structure. Details of these load cases are available at the beginning of this tutorial. Ahead corresponding commands generated are listed.

```
KG   LOAD   METER
UNIT 1 DEAD LOAD
AND SELF -1.0
2 EXTERNAL PRESSURE LOAD LOAD
LOAD ELEMENT
1 TO 6 PR GY -300
3 TEMPERATURE
LOAD LOAD TEMP 1
TO 6 40 30
```

Procedure:

1. To create loads, select the tool **Load Page** bar located in

tools tools structure.

Figure 2-319:
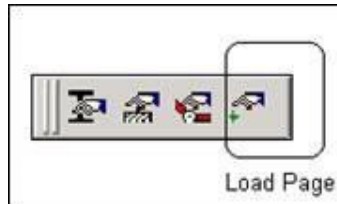
Alternatively, one can go to **General | Load** from the left side of the screen.

Figure 2-320:

2. Note that the enumerated pressure load at the beginning of this tutorial value is in units *KN* and*subway*. Instead of converting that value to units

   input current, conform to those units. The current input units, while finally establish specify the thickness was **centimeter**. We have to change the unit to **kilogram** and units of length

   Meters. To change the units, as before, select the tool **Input Units** from the bar above, or select the Tools menu **Tools> Set Current Input Unit**. In the dialog *Set Current Input Units* that appears, specify the units of length and **subway** and force units like **kilogram**.

3. A window titled "Load" appears on the right side of the screen. To start the first load case, highlight **Load Case Details** and click **add**.

Figure 2-321:
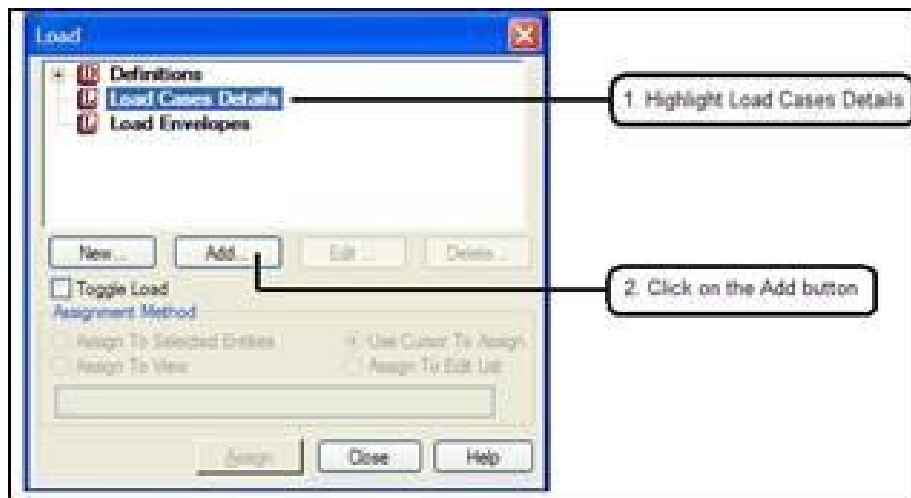
4. Open the dialog *Add new load cases*. The drop-down list against *Loading Type* is available if you want to link the case load we are creating with any of the definitions of ACI, AISC or IBC undead, ice, etc. This type of association must be made if we are to use installation program to automatically generate load combinations according to the codes.

   Note that there is a checkbox labeled *Reducible per UBC / IBC*. This feature becomes active only when the load case is assigned a load type called *Live* at the time of creation of that case.

   As we do not intend to use the option of automatic generation of load combination, we will leave the type of load as **none**. Write **Dead Load** as the title for load case 1 and click **add**.

Figure 2-322:

The newly created load case details appear under the load cases in the load dialog.

Figure 2-323:
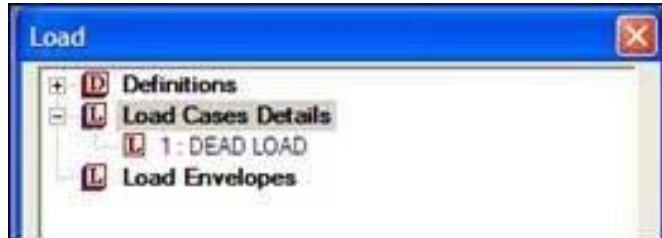
5. To generate and assign the first type of load, select **1: Dead Load**. You will notice that the dialog *Add new items loading* now shows more options.

Figure 2-324:
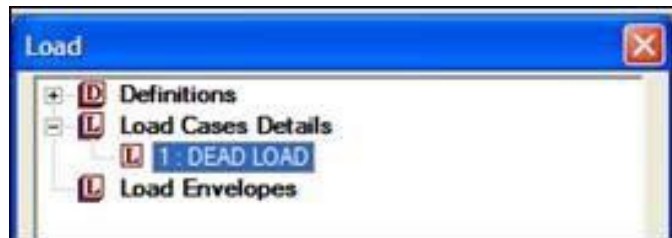
6. In the dialog box, select the **Load Selfweight** under Article *Selfweight*. Specify the address **And**Such as Factor **-1.0**. A negative number means that the load acts against the positive direction of the overall axis (Y in this case) over which it is applied. Click add. The

selfweight charge is applicable to all members of the structure and can not be applied to a selected list of members.

Figure 2-325:
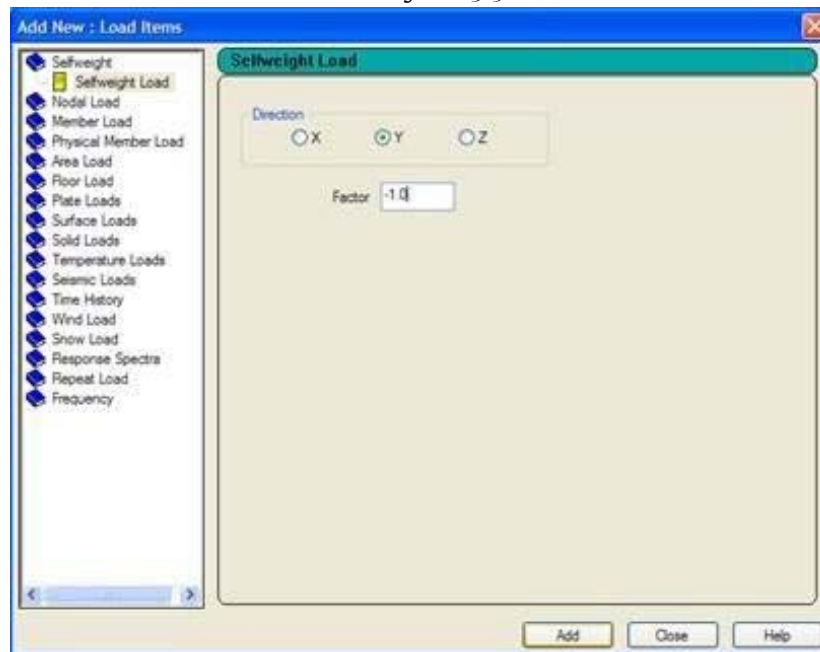
7. Then, initiate the establishment of the second load case is a pressure load on the elements. This highlights **Load Case Details** in the dialog box *Add New Load Cases*Once again, we are not we are associating the load based about to create code with any kind of load and so leave that box as **none**. Specify the title of the second load case as **Load External Pressure** (Loading of external pressure) and then

click add.

Figure 2-326:
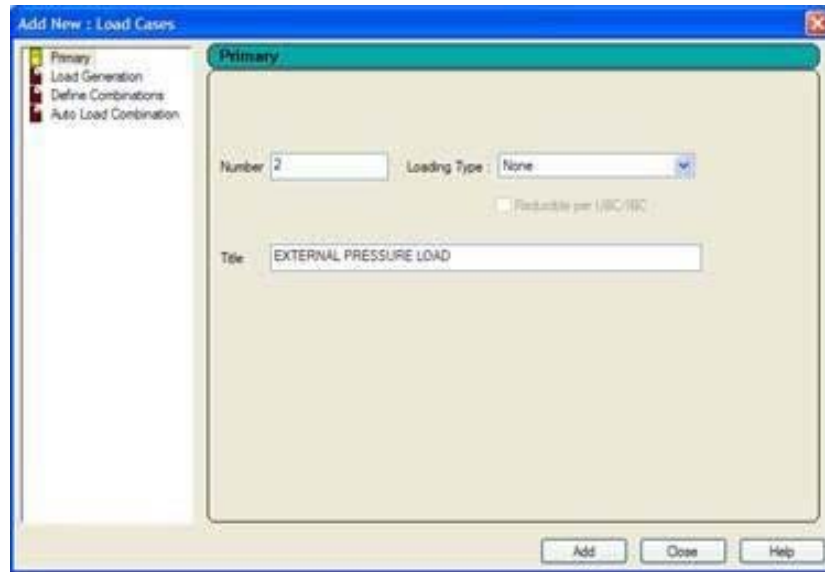
To generate and assign the second load type, highlight **2: External Pressure Load**.

Figure 2-327:
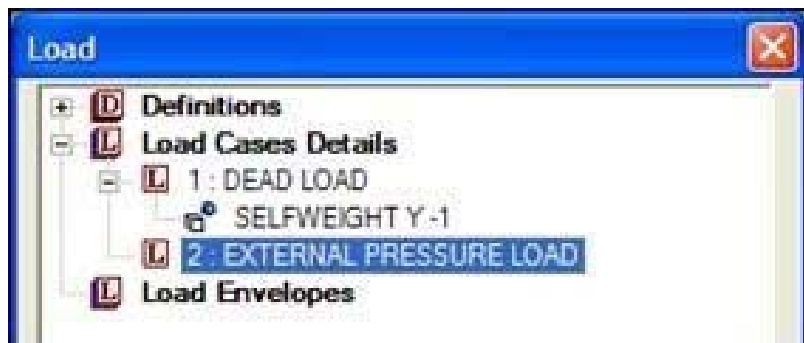
8. In the dialog box *Add New Load Items*Select the option **Pressure on Full Plate** under the theme of *Plate Loads* enables the load applied over the entire area of the element. (The concentrated load is to apply a force concentrated on the

element. Options *Trapezoidal*is e *hydrostatic* they are definable by the pressure with varying intensity from one point to another. The *Partial Pressure Plate Load -partial pressure loading plate*- It is useful if the load is to be applied as a "patch" located in a small portion of an element).

Provides **-300** kg / m^{2}for W_1 (force) **GY** as *Street address* and click **Add** followed by **Close**.

Figure 2-328:
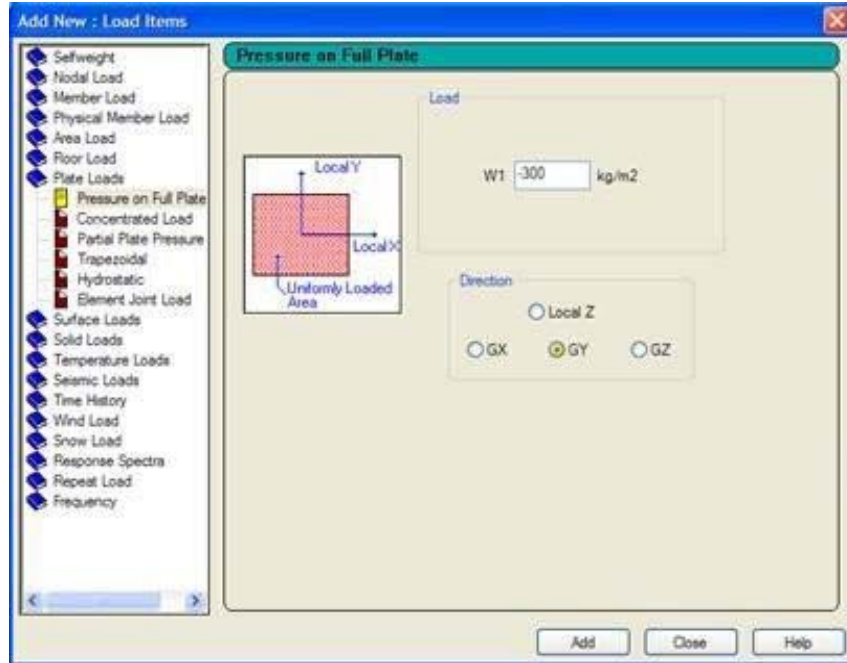
9. Since the burden of pressure should be applied to all elements of the model, the easiest way is to set the method of allocation **Assign to View**. Then click on **assign** in the upload dialog box as shown below.

Figure 2-329:
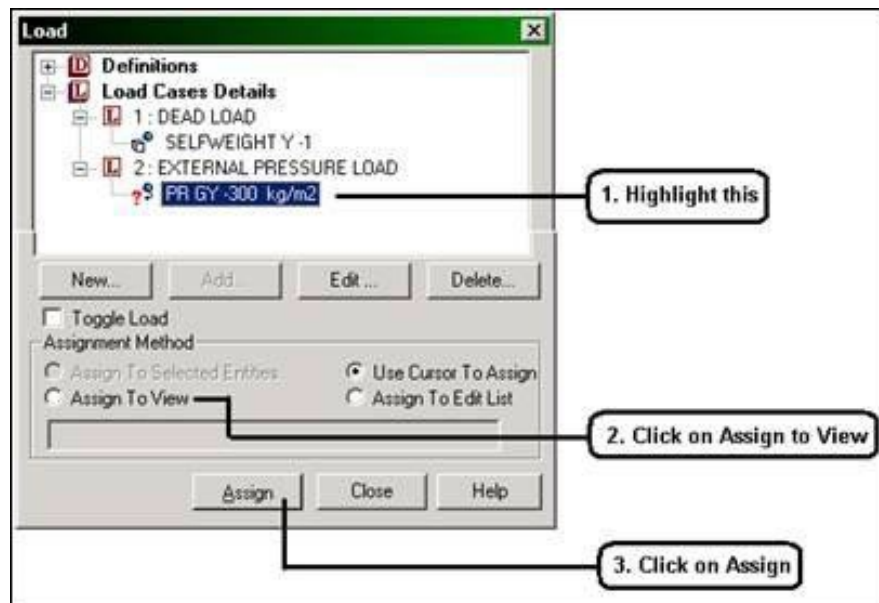
After the load has been assigned, the model will be as shown below.

Figure 2-330:
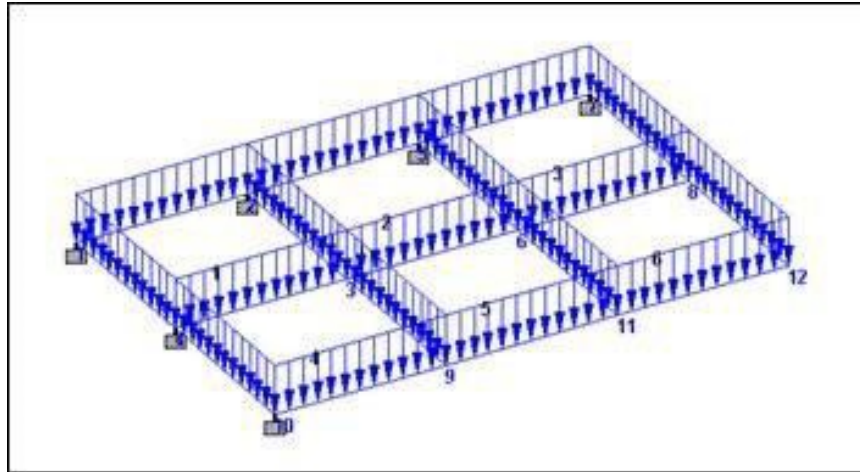
10. Then let us create the third load case is a temperature load.
The initiation of a new load case is performed by the procedure explained in Step 7 In the dialog box that appears, specify the title of the third load case as **Temperature Load** and click **Add**.

Figure 2-331:
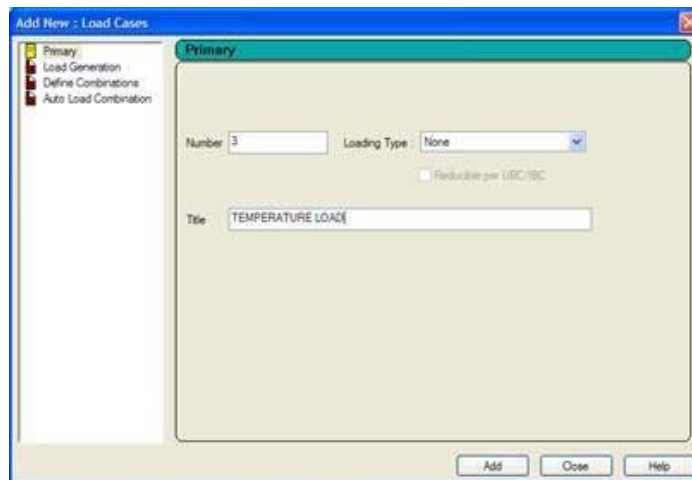
11. To generate and assign the third type of load, as before, select **3: Temperature Load**. Charges temperature created from the input screens are available on the option **Temperature** in the dialog box *Add New Load Items*.

12. In the dialog box, make sure the item is selected **temperature** under option *Temperature Loads*. Then provides **40** as the temperature change for Axial elongation (*Temperature Change for Axial Elongation)* and **30** as the temperature differential from top to bottom (*Differential Temperature from Top to Bottom)* and click **Add** and click **Close**.
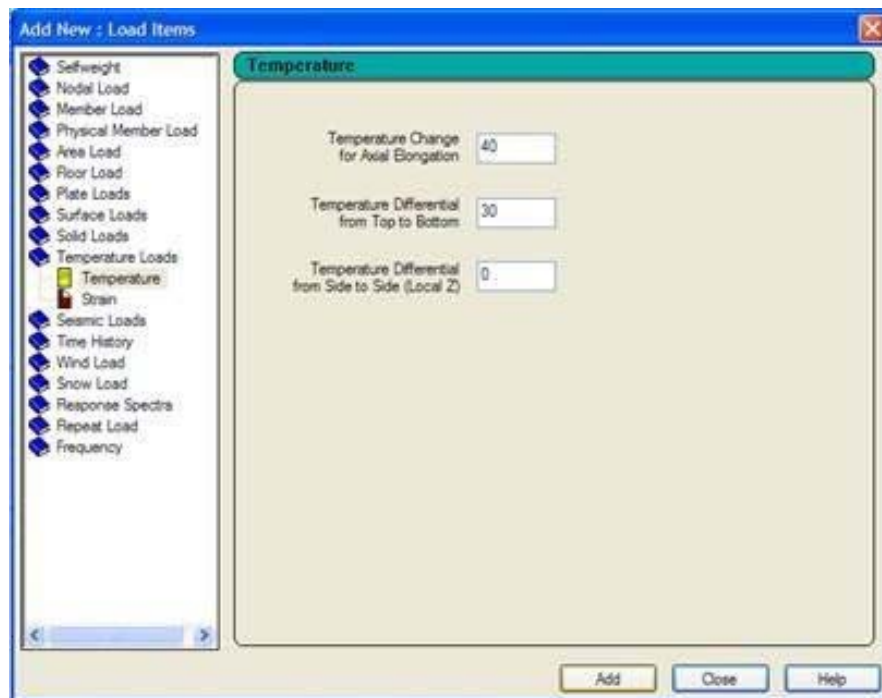
13. Since we intend to apply the temperature load on all plates, as before, choose **Assign to View** and click **Assign** loads in the dialog (see step 9 for explanation).

3.6.7 Creating load combinations

Specifications at the beginning of this tutorial require us to create two instances of combination. The required commands are:

```
101 CASE LOAD COMBINATION CASE 1 + 2
1 1.0 2 1.0
LOAD COMBINATION 102 CASE 3 CASE 1 +
1 1.0 3 1.0
```

Steps:

1. To initiate and define the load case 4 as a load combination, once again, highlight the option **Load Case Details**. In the dialog box *Add new load cases* Click the option **Define Combinations** the left side. Enter *No Load*: How come **101** and title as **Case 1 + Case 2**.

Figure 2-333:
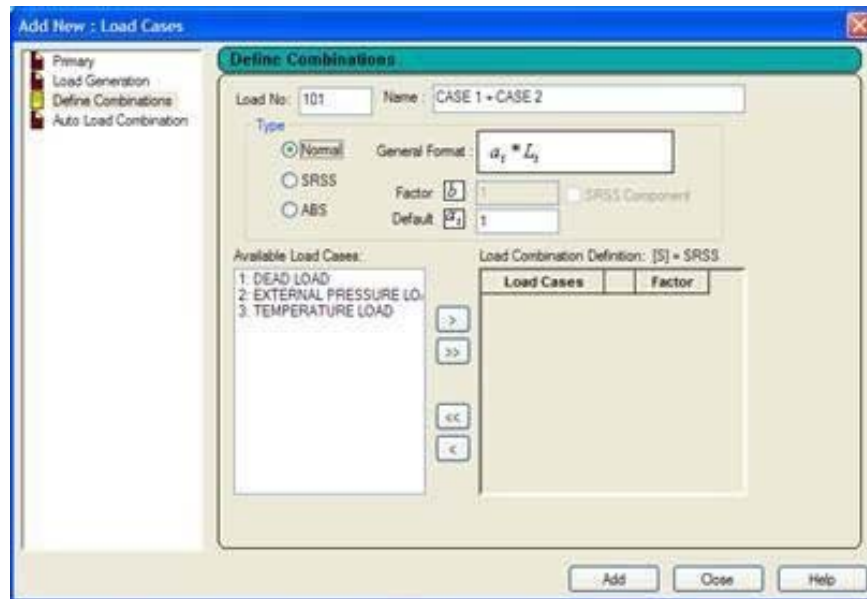
2. Then, in the box *define combinations*Select load case 1 on the left side of the list and click [>]. Repeat this step with the load case 2 also. The Load Cases 1 and 2 appear in the list box on the right as shown in the following figure. (These data indicate that the two cases are adding a multiplication factor of 1.0 of the load and the results of the load combination would be obtained by the algebraic sum of the results

for individual load cases). Finally, click **Add**.

Figure 2-334:
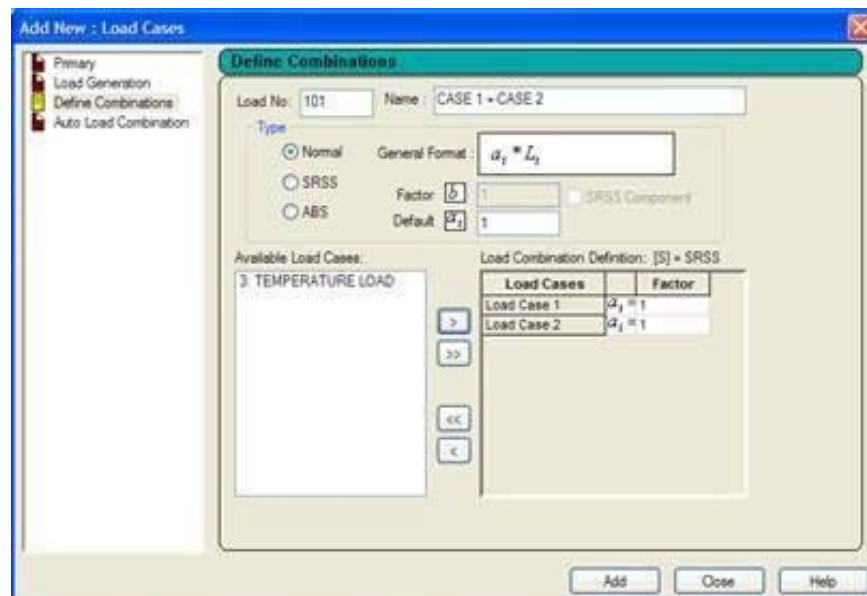
The Case 101 has now been created.

3. To initiate and define the load case 5 as a load combination, as before, enter the $No\ Load$:**102** and title as **Case 1 + Case 3**.

4. Then repeat step 2, except for selected load cases 1 and 3 instead of 1 and 2 cases.

Figure 2-335:
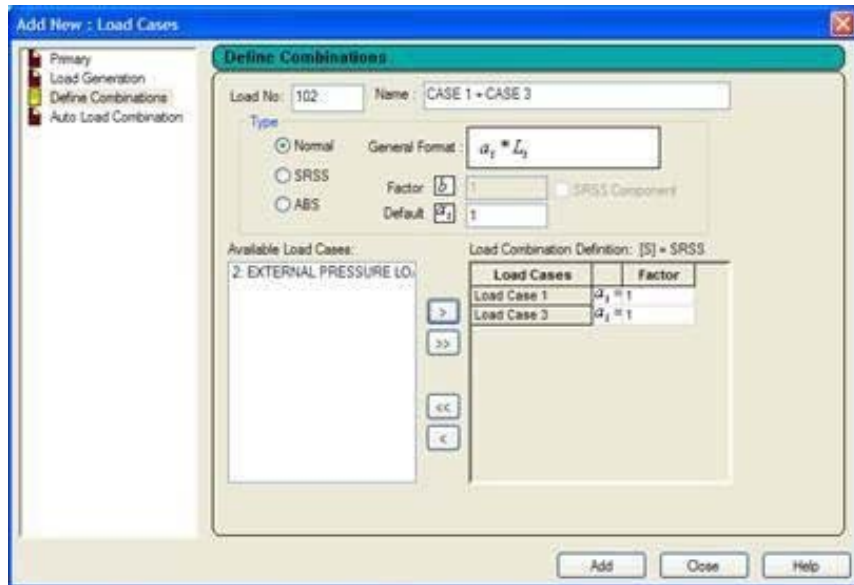
Therefore, load 102 is also created if we change our mind about the composition of any case of existing combination, you can select if you want to change and make the necessary changes in the instances or their constituent factors.

Figure 2-336:
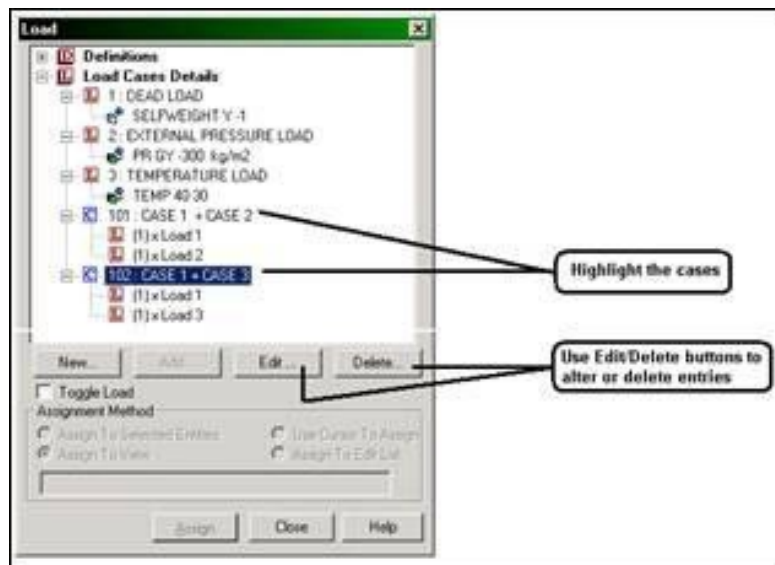
5. Let's get this dialog box by clicking the button **Close**.

   Also noteworthy is that as the load cases are created, a facility to quickly switch between the different cases is available on the top of the screen as a selection box load cases as shown below.

   Figure 2-337:
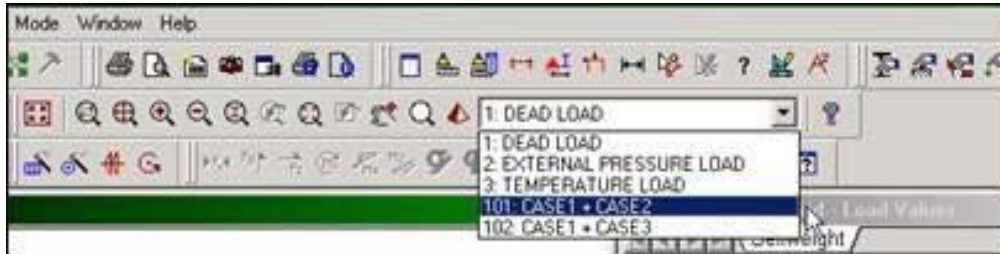
   We have now completed the task of creating all load cases.

   > **Tip**: Remember to save your work or selecting **File> Save** The tool to save or press **CTRL + S**.

3.6.8 Specifying the type of analysis

The type of analysis that is required is a linear static type. We will also obtain a report of static equilibrium. This requires the command:

```
PERFORM ANALYSIS PRINT CHECK statics
```

Procedure:

1. Select the tab **Analysis / Print | Analysis** from the left side of the screen.

   Figure 2-338:

   Opens the dialog analysis and printing commands.

2. Select the tab **Perform Analysis**. For static equilibrium report, select Print **Statics Check.**

Figure 2-339:
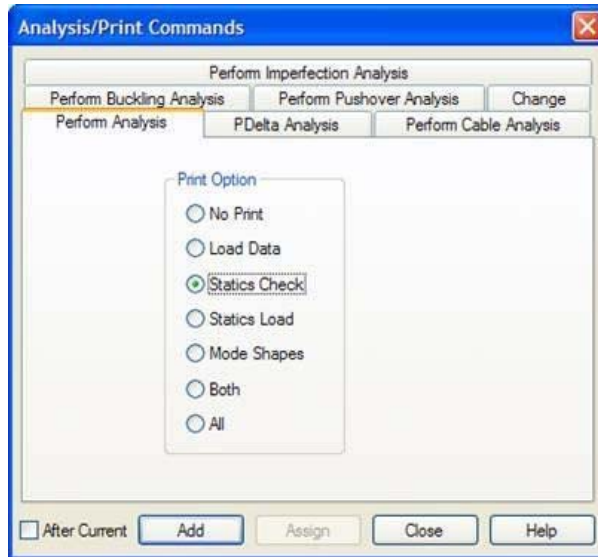
> **Note** In response to this option, a report is the Summary of the applied load and Summary of the support reactions for each load case will occur in the STAAD output file. See Section 3.10 for information on how to view this report.

3. Finally, click **add**, Followed by the button **shut down**. The dialog analysis in the data area with the newly added training will be as shown below.

Figure 3 103
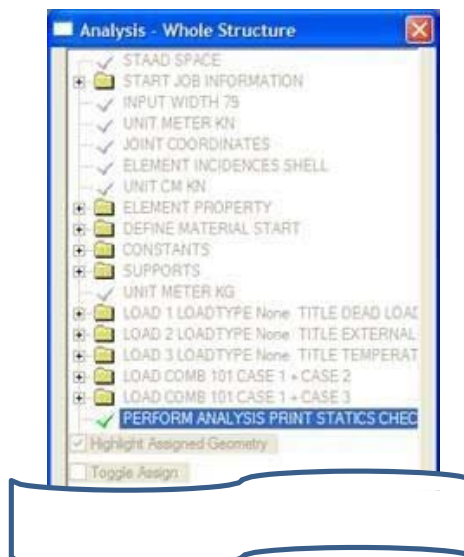
Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

3.6.9 Specifying print commands post-analysis

Two types of results of element may be obtained: a) efforts elements (ELEMENT STRESSES) at the centroid, or any point on the surface element b) the forces of elements (FORCE ELEMENT) nodes. The first consists of voltage and time width of the unit, as described in Sections 1.6.1 and 5.41 STAAD Technical Reference Manual. The latter consists of 3 forces and 3 moments on each node of the elements in the global axis system (see section 5.41 for details).

We'd like to get both these results. Also establish the units in which these results are printed and Metro KN element voltages and Kg and metro forces in the element. This requires specification of the following commands:

```
UNIT METER KN
PRINT LIST ELEMENT STRESSES 3 KG
METER UNIT
PRINT FORCE ELEMENT LIST June
```

These results will be written to the output file can be viewed with STAAD and the procedure explained in section 3.10.

Steps:

1. Set the length and strength units **subway** and**Kilonewton** respectively.

   See "3.6.2 changing the units of the input length" on page 232 for more information on this procedure.

2. Select the tab **Analysis / Print | Post Print**.

Figure 2-340:

3. Click **Define Commands** in the data area on the right side of the screen.

   Opens the Print command Analysis and dialogue.

Figure 2-341:
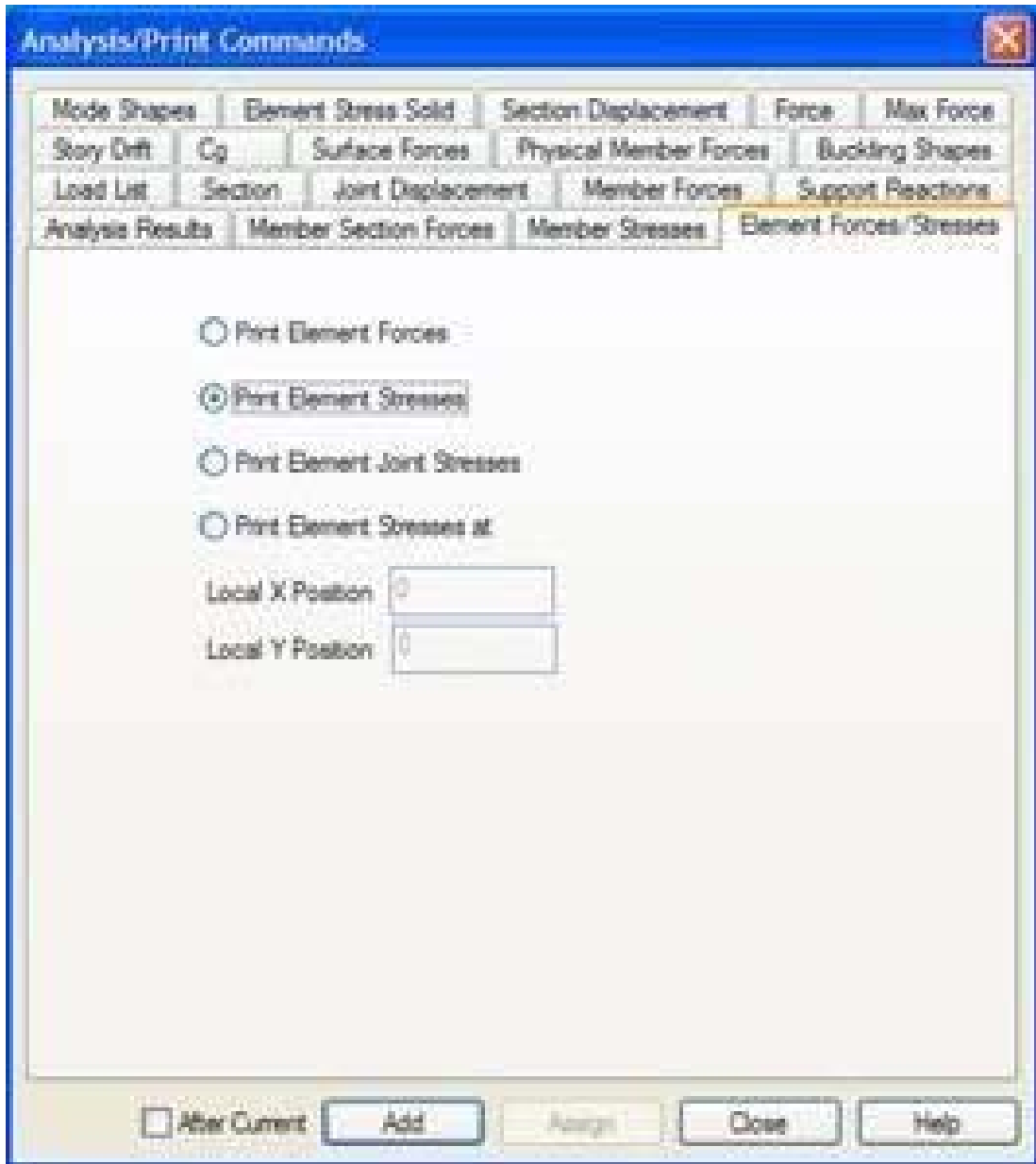
4. Select the tab **Element Forces / Stress.** Select the option **Print Element Stresses** and click **add** followed by **Close**.

5. Set the length and strength units **subway** and **kilogram** respectively.

6. Then, repeat steps 2 and 3 In step 3, select **Print Element Forces** and click **add**, Followed by the button **Close**.

At this point, the dialog *Print Post analysis* will be as shown below.

Figure 2-342:
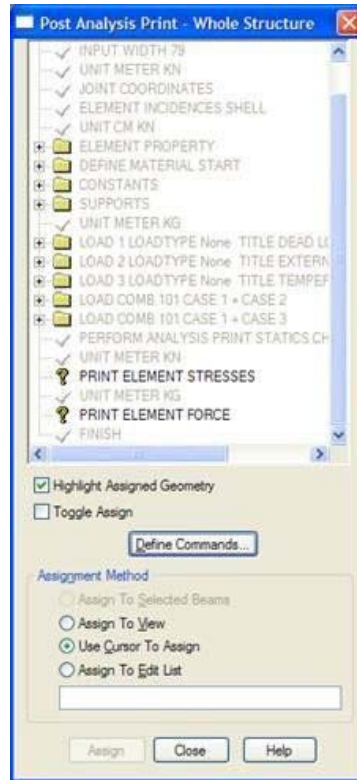
7. To associate the command **PRINT ELEMENT STRESSES** element 3, first select the command as shown in the figure above. Then, using Cursor Plates Click the selected item No.3 As plaque, *allocation method* automatically becomes *Assign to Selected Plates*. As shown below, click

   assign.

Figure 2-343:
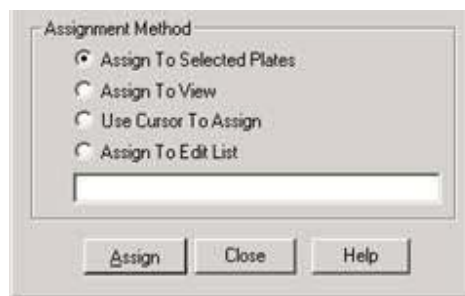
Eight. To associate the command **PRINT ELEMENT FORCE** to item 6, repeat step 7 except to select item No. 6 in place of No. 3 element.

We have now completed the task of mapping the input to this model.

Tip: Remember to save your work or selecting **File> Save**The tool to save or press **CTRL + S**.

3.7 Displaying the command file entry

Now take a look at the data that has been written to the file you just saved before. As seen in the previous tutorials, while the model is being created graphically, a corresponding set of commands that describes that aspect of the model simultaneously written to a script file is a simple text file. A command summary was also mentioned under "commands are generated ..." at the beginning of each section of this tutorial.

The content of that text file can be viewed in its entirety or by clicking the tool **STAAD Editor** or by going to the menu *edition* and selecting *Edit Input Command File* as shown below.

Figure 2-344:
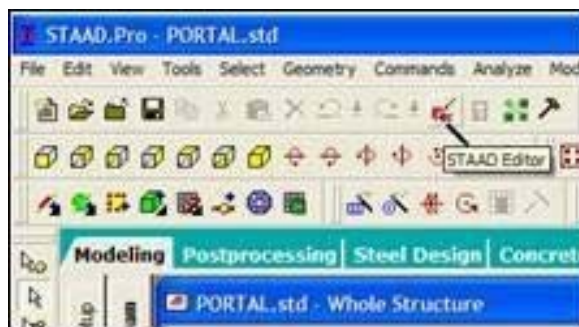
Figure 2-345:
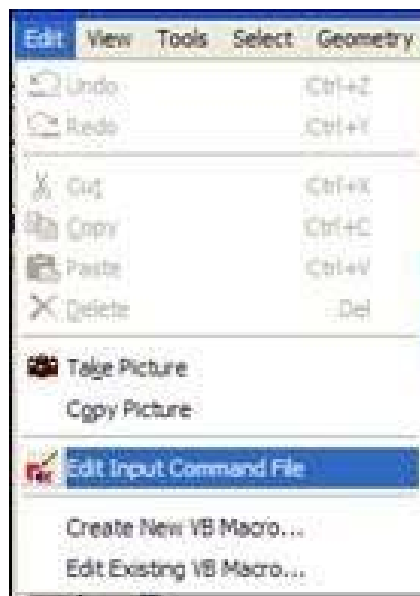
A new window with the data listed will open as shown here:

Figure 2-346:
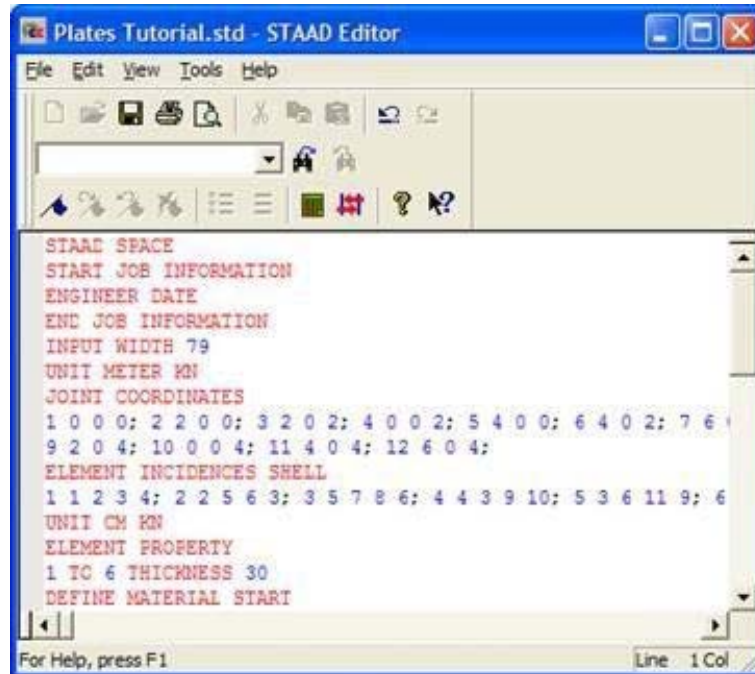
This window contains the facilities and is known as the *STAAD Editor*.

We may make changes to the data of our structure in this *Editor* if we want to. Let's get out of *Editor* do without selecting the menu option **File> Exit** the output editor window (not the **File> Exit** menu of the main window behind the Editor window).

Instead of using graphical methods explained in the previous sections, we could have created the entire model writing these specific commands in the editor. This was one of the methods mentioned in paragraph 3.1 of this tutorial. To understand this method, proceed to the next section. If you want to skip that part, proceed to Section 3.9 where we reviewed this model.

3.8 Creating the model using the script

Now we use the method of the command file to create the model for the above structure. The commands used in the script are described later in this section.

The STAAD.Pro command file can be created using the built-in editor, the procedure explained below in this section. Any text editor such as Notepad or WordPad can also be used to create the command file. However, the File Editor command syntax has the advantage as we write

commands. Keyword, numerical data, comments, etc. are shown in different colors in the STAAD.Pro Editor. Below is a typical screen editor is shown to illustrate the general appearance.

Figure 2-347:
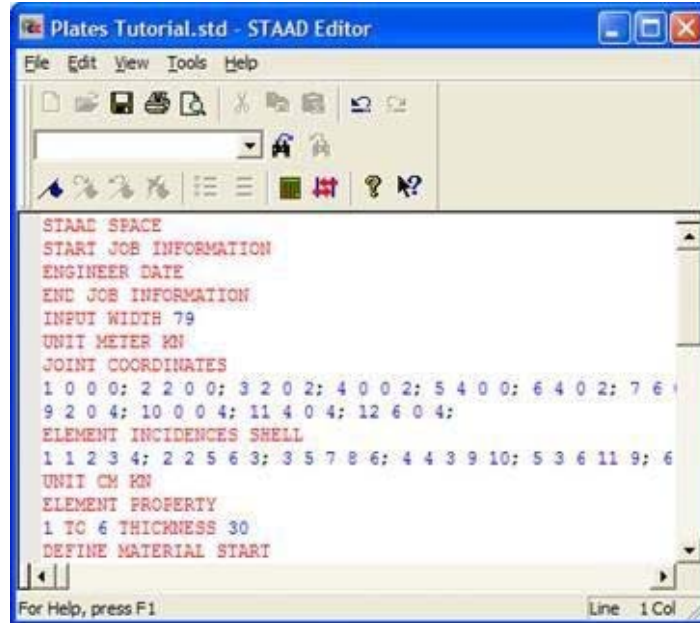
To access the built-in editor, start the program using the procedure explained in section 3.3. Then follow step 1 of Section 3.4 (also see examples below).

Figure 2-348:
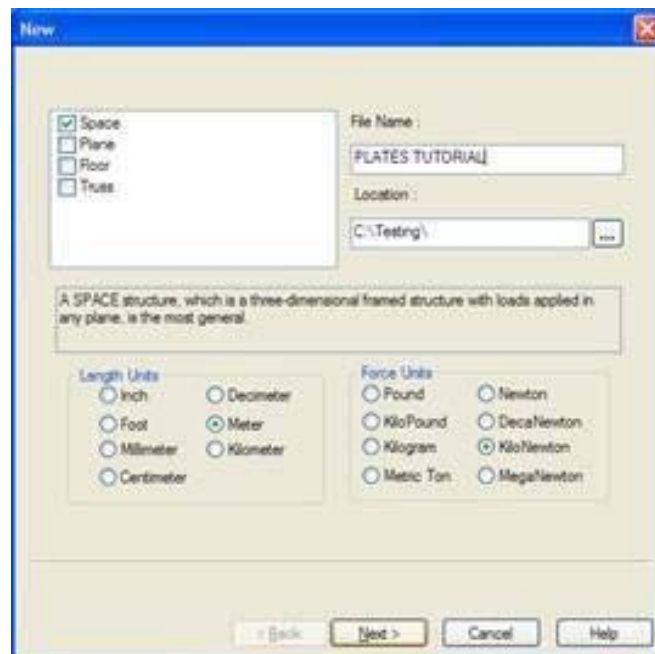
Then you will find the dialog box shown below. In this dialog box, choose *Open STAAD Editor*.

Figure 2-349:
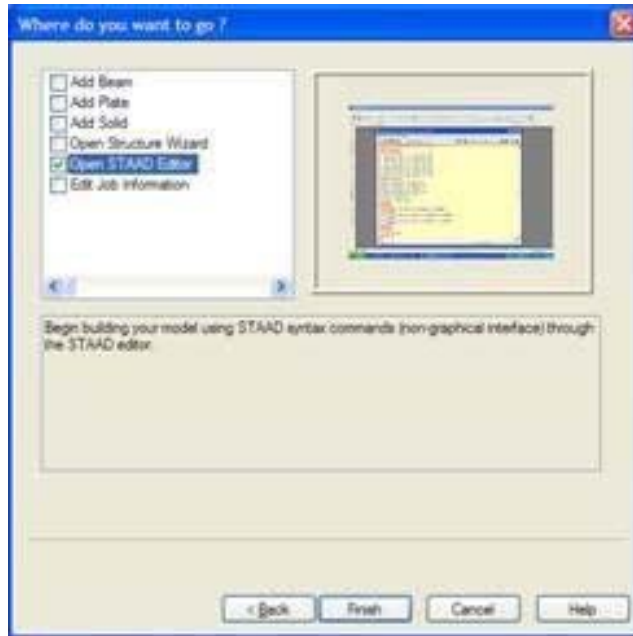
At this point, the editor screen will open as shown below.

Figure 2-350:
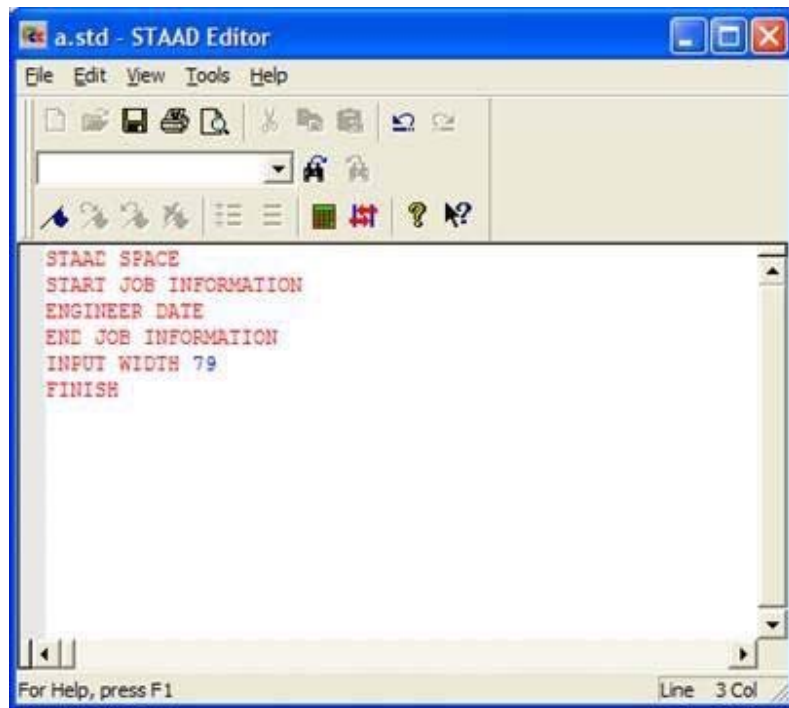
Deletes all lines of command appears in the editor window and the lines shown in bold below (do not have to delete the lines if you know what to keep and where the rest of the commands). The commands can be written in lowercase or upper letters. Usually all that is needed are the first three letters of a keyword, the rest of the Letters of the word are not necessary. Necessary letters are underlined. ("SPACE" = "SPA" = "space"

=

"Spa")

The current input is shown in bold, followed by explanation Letters.

<u>STA</u>AD SPA2 EC SLAB SUPPORTED ALONG EDGES

Each entry must begin with the word STAAD. The word space means that the structure is a structure of space (three-dimensional) and the geometry defined by the X, Y and Z.

<u>UNI</u>T METKN ER

Specifies the unit to be used for data to follow.

<u>JOI</u>NT COORDINATES
1 0 0 0 ; 2 2 0 0; March 2 0 2 ; 4 0 0 2

5 4 0 0 ; 6 4 0 2; June 7   0 0 ; 8 6 0 2

9 2 0 4 ; 1O O O 4; 11   4 0 4 ; June 12 O

The number of Union followed by X, Y and Z coordinates are indicated. Signs semicolon (;) are used as line separators. It allows us to offer multiple data sets in a single line. For example, node 6 has coordinates (X, Y, Z) of (4, 0, 2).

<u>ELE</u>MENT INCIDENCES SHELL
1 1 2 3 4; 2 2 5 6 3; 3 5 7 8 6; April 4th September 3, 10;
March 5 6 11 9; 6 June 12 August 11

The incidences of the elements are defined above. For example, the element 3 is defined as a connection between nodes 5, 7, 8 and 6.

<u>UNI</u>T   <u>CMK</u>N
ELEMENT
PROPERTY
1 TO 6 THICKNESS 30

Unit length is changed from meters to centimeters. Element specifying properties of the

elements are 30 cm wide are then provided.

```
UNIKN     ERmet     T
WITHStants
E2.17185E + 007    ALL
POISSON 0.17 ALL
DENSITY  23.5616  ALL
ALPIt has 1E-005 ALL
```

The constants of the materials, which are E (modulus of elasticity), density, and Poisson's ratio alpha specified command following constants. Before this, the input units to Metro and KN are changed.

```
SUPPORTS
January 2 4 5 7 10 FIXED
```

Unions of 1, 2, 4, 5, 7 and 10 are defined as embedded support. This will make all 6 degrees of freedom in these nodes are restricted.

```
UNIT KG
LOADEAD LOAD D1
```

Units of force change to KG KN to facilitate entry of loads. The Load Case 1 then starts with a companion.

```
SELFWEIGHT Y -1
```

The Load Case 1 consists of the weight of the structure acting in the Y direction by a factor of overall **-1.0**. And overall is vertically upwards, the factor **-1.0** indicates that the load will act downward.

```
LOAD 2 LOAD EXTERNAL PRESSURE
```

The Load Case 2 starts along with a companion title.

```
ELEMENT LOAD
1 TO 6 PRGY -300
```

Charge 2 is a pressure on the elements. A uniform pressure is applied $300kg / m2$ on all elements. GY indicates that the load is in the global address Y. The negative (-300) indicates that the load acts against the overall positive direction.

```
LOAD3 TEMPERATURE LOAD
```

The Case Load 3 starts along with a companion title.

```
TEMPerature LOAD
1 TO 6 TEMP 40
30
```

Charge 3 is a temperature load. All six elements are subjected to a temperature rise in the plane of 40 degrees and a temperature variation throughout the thickness of 30 degrees. This increase is in the same units as the temperature alpha value specified above under CONSTANT.

```
LOADCOM101 B 1 + CASE 2 CASE
1 1.0 2 1.0
```

The Load Combination 101 starts along with a companion title. The Load Cases 1 and 2 are taken into account individually so a value of 1.0, and algebraic factored values are combined.

```
LOAD COMB 102 + CASE 3 CASE 1
1 1.0 3 1.0
```

The Load Combination 102 starts along with a companion title. Load cases 1 and 3 are individually factored by a value of 1.0, and the values are combined algebraically factored.

```
PERANA FORMPRI LYSISSTA NTTICS CHECK
```

The above command instructs the program to proceed with the analysis. A report of static equilibrium with the help of words CHECK PRINT statics are also required.

```
UNIT METKN ER
PRINT ELEMENT STRESS LIST 3
```

The tensions and moments of unit width requested in the centroid of the element 3 in KN and meter units.

```
UNIT KG METER
PRINT ELEMENT FORCE LIST 6
```

The forces and moments for all 6 degrees of freedom at the corner nodes of the element 6

are ordered in units of KG and the subway.

EndISH

This command terminates the STAAD run. Let's save the file and get out of the editor.

3.9 Carrying out the analysis and design

In order to obtain the displacements, forces, stresses and reactions in the structure due to the applied loads, the model must be analyzed. If the Member State simplistic elements and the code requirements for steel and concrete is to be determined, which

involves a process called design. Both processes are started by selecting **Analysis> Run Analysis**.

Figure 2-351:

If the structure is not saved after the last change was made, you must save the structure first using the command *Save* Menu *Photo*.

As the analysis progresses, several messages appear on the screen as shown in the following figure.

Figure 2-352:
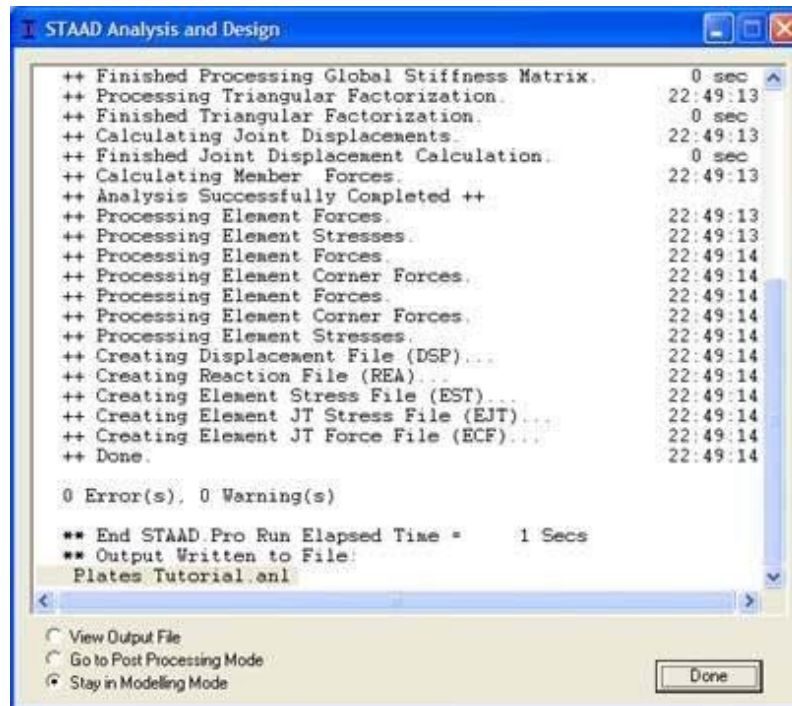
At the end of these calculations, two activities take place. a) a button **Done** becomes active b) Three options are available in the lower left corner of the info window.

Figure 2-353:

These options are indicative of what will happen after you click **Done**.

Option *View Output File* allows us to see the output file created by STAAD. The output file contains the numerical results produced in response to different input commands you specified during the generation process model. It also tells us if errors were found and if so, whether the analysis and design was completed successfully or not. Section

3.10 (see also section 1.9) provides additional details on viewing and understanding the contents of the output file.

Option *Go To Post Processing Mode* allows us to go to the graphics part of the program known as the postprocessor. This is where one can exhaustively verify the results, see the results graphically, draw diagrams of results, produce reports, etc. Section 3.11 explains the post-processing mode in more detail.

Option *Stay in Modelling Mode* allows us to remain in the mode of generation of the program model (in which it is currently) in case we want to make more changes in our model.

3.10 Viewing the output file

During the analysis stage, an output file containing the results, warnings and messages associated with errors in the output if any, occurs. This file has the extension .anl and can be viewed using the display output. Section 1.9 of this manual contains information on how to view this file.

In sections 3.6.8 and 3.6.9, we have supplied instructing the program to write some very specific results in the output file. Examine the results.

```
**********************************************************
*                                                        *
*               STAAD.PRO V8I SELECTSERIES1              *
*               VERSION  20.07.06.35                     *
*               PROPRIETARY PROGRAM OF                   *
*               BENTLEY SYSTEMS, INC.                    *
*               DATE=    JUL  7, 2010                    *
*               TIME=    11:44:10                        *
*                                                        *
*          USER ID: BENTLEY SYSTEMS                      *
**********************************************************

     1. STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES
INPUT FILE: TUT_03_SLAB.STD
     3. UNIT METER KN
     4. JOINT COORDINATES
     5. 1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2
     6. 5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2
     7. 9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4
     9. ELEMENT INCIDENCES SHELL
    10. 1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6 ; 4 4 3 9 10
    11. 5 3 6 11 9 ; 6 6 8 12 11
    13. UNIT CM KN
    14. ELEMENT PROPERTY
    15. 1 TO 6 THICKNESS 30
    17. UNIT METER KN
    18. CONSTANTS
    19. E 2.17185E+007 ALL
    20. POISSON 0.17 ALL
    21. DENSITY 23.5616 ALL
    22. ALPHA 1.0E-005 ALL
    24. SUPPORTS
    25. 1 2 4 5 7 10 FIXED
```

```
27. UNIT KG

28. LOAD 1 DEAD LOAD

29. SELFWEIGHT Y -1

31. LOAD 2 EXTERNAL PRESSURE LOAD

32. ELEMENT LOAD

33. 1 TO 6 PR GY -300

35. LOAD 3 TEMPERATURE LOAD

36. TEMPERATURE LOAD

37. 1 TO 6 TEMP 40 30

39. LOAD COMB 101 CASE 1 + CASE 2

40. 1 1.0 2 1.0

42. LOAD COMB 102 CASE 1 + CASE 3

43. 1 1.0 3 1.0

45. PERFORM ANALYSIS PRINT STATICS CHECK


        P R O B L E M   S T A T I S T I C S
        ---------------------------------------

  NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =    11/     6/     6

        SOLVER USED IS THE IN-CORE ADVANCED SOLVER

  TOTAL PRIMARY LOAD CASES =    3, TOTAL DEGREES OF FREEDOM =    36



        STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.    1
        DEAD LOAD


        CENTER OF FORCE BASED ON Y FORCES ONLY (METE).
        (FORCES IN NON-GLOBAL DIRECTIONS WILL INVALIDATE RESULTS)

                  X =  0.3000000230E+01
                  Y =  0.0000000000E+00
                  I =  0.1999999983E+01

***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING    1 )
    SUMMATION FORCE-X =           0.00
    SUMMATION FORCE-Y =       -17298.83
    SUMMATION FORCE-Z =           0.00

   SUMMATION OF MOMENTS AROUND THE ORIGIN-
   MX=      34597.65  MY=           0.00  MZ=       -51896.48


***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING    1 )
    SUMMATION FORCE-X =           0.00
    SUMMATION FORCE-Y =       17298.82
```

```
     Z = 0.000000+00        0

     RX= 4.00045E-04        12

     RY= 0.000000+00        0

     RZ= -2.70258E-04        9


          STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.    3
          TEMPERATURE LOAD


***TOTAL APPLIED LOAD ( KG   METE ) SUMMARY (LOADING    3 )

    SUMMATION FORCE-X =  5.1567403E-11

    SUMMATION FORCE-Y =  2.4152937E-27

    SUMMATION FORCE-Z = -1.0313497E-10


    SUMMATION OF MOMENTS AROUND THE ORIGIN-

    MX= -4.0357990E-11  MY=  3.3531239E-10  MZ=  1.5953626E-13


***TOTAL REACTION LOAD( KG   METE ) SUMMARY (LOADING    3 )

    SUMMATION FORCE-X =  4.5770903E-02

    SUMMATION FORCE-Y =  0.0400481E-05

    SUMMATION FORCE-Z =  1.5280378E-02


    SUMMATION OF MOMENTS AROUND THE ORIGIN-

    MX=         0.00  MY=        -0.07  MZ=          0.00

MAXIMUM DISPLACEMENTS (  CM  /RADIANS) (LOADING      3)

          MAXIMUMS    AT NODE

    X =  2.01178E-01       12

    Y =  8.97375E-01       12

    Z =  1.66258E-01       11

    RX= -3.51267E-05       12

    RY= -2.41818E-04       11

    RZ=  2.62397E-03       12



************ END OF DATA FROM INTERNAL STORAGE ************


 47. UNIT METER KN

 48. PRINT ELEMENT STRESS LIST 3

ELEMENT STRESSES    FORCE,LENGTH UNITS= KN   METE
-------------------

        STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT  LOAD      SQX         SQY         MX          MY          MXY
```

```
                    VONT      VONB       SX        SY        SXY
                   TRESCAT    TRESCAB

        3     1     -18.13     72.86     -3.56     -28.42     -3.35
                   1388.65    1388.65     0.00      0.00      0.00
                   1404.84    1404.84
       TOP :  SMAX=  -220.35  SMIN= -1404.84  TMAX=  592.24  ANGLE= -11.1
       BOTT:  SMAX=  1404.84  SMIN=   220.35  TMAX=  592.24  ANGLE=  78.9
              2      -7.54     38.33     -1.65     -8.50     -1.39
                    544.68    544.68      0.00      0.00      0.00
                    584.71    584.71
       TOP :  SMAX=   -91.71  SMIN=  -584.71  TMAX=  248.50  ANGLE= -11.1
       BOTT:  SMAX=   584.71  SMIN=    91.71  TMAX=  248.50  ANGLE=  78.9
              3      96.73    -59.42    -38.45    -14.83     18.43
                  18779.74   5388.57  -5044.91  -2389.56   3890.86
                  18912.86   5585.64
       TOP :  SMAX=   289.70  SMIN=-10642.36  TMAX= 5456.83  ANGLE=  55.1
       BOTT:  SMAX=   624.60  SMIN= -4960.95  TMAX= 2792.82  ANGLE=  53.8
            101     -25.67    103.19     -5.61    -28.92     -4.74
                   1853.33   1853.33     0.00      0.00      0.00
                   1989.55   1989.55
       TOP :  SMAX=  -312.06  SMIN= -1989.55  TMAX=  838.74  ANGLE= -11.1
       BOTT:  SMAX=  1989.55  SMIN=   312.06  TMAX=  838.74  ANGLE=  78.9
            102      78.61     13.44    -34.41    -35.24     15.86
                  18643.84   5713.28  -5044.91  -2389.56   3890.86
                  11874.60   6408.65
       TOP :  SMAX=  -923.25  SMIN=-11874.60  TMAX= 5875.67  ANGLE=  52.7
       BOTT:  SMAX=  1848.79  SMIN= -4559.86  TMAX= 3204.33  ANGLE=  57.9


         **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
             MAXIMUM      MINIMUM      MAXIMUM      MAXIMUM      MAXIMUM
            PRINCIPAL    PRINCIPAL     SHEAR       VONMISES     TRESCA
             STRESS       STRESS       STRESS      STRESS       STRESS

          1.989540E+03 -1.187460E+04 5.450830E+03 1.077974E+04 1.187460E+04
  PLATE NO.      3            3            3            3            3
  CASE NO.     101          102          3            3          102


  ******************END OF ELEMENT FORCES******************

    50. UNIT KG METER
```

```
   51. PRINT ELEMENT FORCE LIST 6

  ELEMENT FORCES      FORCE,LENGTH UNITS= KG   METE

  ----------------

  **NOTE- IF A COMBINATION INCLUDES A DYNAMIC CASE OR IS AN SRSS OR ABS COMBINATION
          THEN RESULTS CANNOT BE COMPUTED PROPERLY.
    GLOBAL CORNER FORCES
 JOINT    FX          FY           FZ           MX          MY          MZ

              ELE.NO.       6 FOR LOAD CASE    1
   6  0.0000E+00  4.5324E+02  0.0000E+00 -1.1313E+03  0.0000E+00  7.5002E+02
   8  0.0000E+00  5.0615E+02  0.0000E+00 -3.2050E+02  0.0000E+00  2.3379E+02
  12  0.0000E+00 -7.2070E+02  0.0000E+00  7.5724E-12  0.0000E+00 -3.9254E-12
  11  0.0000E+00 -2.3060E+02  0.0000E+00 -4.6605E+02  0.0000E+00 -6.0134E+02
              ELE.NO.       6 FOR LOAD CASE    2
   6  0.0000E+00  1.0064E+02  0.0000E+00 -4.7007E+02  0.0000E+00  3.2315E+02
   8  0.0000E+00  2.1067E+02  0.0000E+00 -1.3540E+02  0.0000E+00  9.9004E+01
  12  0.0000E+00 -3.0000E+02  0.0000E+00 -5.0699E-13  0.0000E+00  1.0010E-12
  11  0.0000E+00 -9.9310E+01  0.0000E+00 -1.9435E+02  0.0000E+00 -2.5029E+02
              ELE.NO.       6 FOR LOAD CASE    3
   6 -2.9880E+05  6.6191E+02 -3.0717E+05  6.3684E+03  2.7911E+03 -5.5444E+03
   8  3.0633E+05 -9.9012E+02 -3.2773E+05  4.3051E+03 -3.7440E+03  4.3520E+03
  12  3.2019E+05  6.4459E-12  3.2019E+05 -6.0036E+03  3.0675E-11  6.0036E+03
  11 -3.2773E+05  3.2821E+02  3.1470E+05 -4.0134E+03  9.5379E+02 -2.8310E+03
              ELE.NO.       6 FOR LOAD CASE   101
   6  0.0000E+00  6.4180E+02  0.0000E+00 -1.6022E+03  0.0000E+00  1.1200E+03
   8  0.0000E+00  7.1682E+02  0.0000E+00 -4.5390E+02  0.0000E+00  3.3059E+02
  12  0.0000E+00 -1.0200E+03  0.0000E+00 -8.3210E-04  0.0000E+00  8.8595E-05
  11  0.0000E+00 -3.3792E+02  0.0000E+00 -6.6131E+02  0.0000E+00 -8.5163E+02
              ELE.NO.       6 FOR LOAD CASE   102
   6 -2.9880E+05  1.1151E+03 -3.0717E+05  5.2371E+03  2.7911E+03 -4.7536E+03
   8  3.0633E+05 -4.0397E+02 -3.2773E+05  3.9046E+03 -3.7440E+03  4.5918E+03
  12  3.2019E+05 -7.2070E+02  3.2019E+05 -6.0036E+03  7.4911E-03  6.0036E+03
  11 -3.2773E+05  8.9607E+01  3.1470E+05 -4.4004E+03  9.5378E+02 -3.4324E+03
   53. FINISH


           *********** END OF THE STAAD.PRO RUN ***********

              **** DATE= JUL  7,2010   TIME= 11:44:11 ****

           *************************************************************
              *       FOR QUESTIONS ON STAAD.PRO, PLEASE CONTACT      *
```

```
*    BENTLEY SYSTEMS OFFICES AT THE FOLLOWING LOCATIONS    *
*                                                          *
*                TELEPHONE            WEB / EMAIL          *
*                                                          *
*  USA:       +1 (714)974-2500                             *
*  UK         +44(1454)207-000                             *
*  SINGAPORE +65 6225-6158                                 *
*  EUROPE    +31 23 5560560                                *
*  INDIA     +91(033)4006-2021                             *
*  JAPAN     +81(03)5952-6500    HTTP://WWW.CTC-G.CO.JP    *
*  CHINA     +86 10 5929 7000                              *
*  THAILAND  +66(0)2645-1018/19 PARTHA.P@REISOFTWARETH.COM *
*                                                          *
* WORLDWIDE    HTTP://SELECTSERVICES.BENTLEY.COM/EN-US/    *
*                                                          *
************************************************************
```

3.11 Post-Processing

If no entry errors, the analysis is completed successfully. The spacious comforts of post-processing mode can then be used to

1. view results graphically and numerically

2. assess the adequacy of the structure from the point of view of safety, service and efficiency

3. create custom reports

The procedure to enter the post-processing mode is explained in Section 2.11.1 of this manual.

The results such as nodal displacements and support reactions are available for all models. The methods are explained in the first two tutorials - see sections 2.11.2 to 2.11.7. If the beams are present in the model, the results will also be available (see sections 2.11.8 to 2.11.18 for information on these). In this example, we see the support reactions. We have no beam in our model, so these results are not available for this type of entity.

For boards, the available results are tensions and moments "unit width". Various methods for displaying the results, as explained in the following sections.

3.11.1 Visualizing stress values in a tabular form

1. Select **View> Tables** or the right mouse button in the window view and select **Tables** the popup menu. Opens the dialog table.

Figure 2-354:
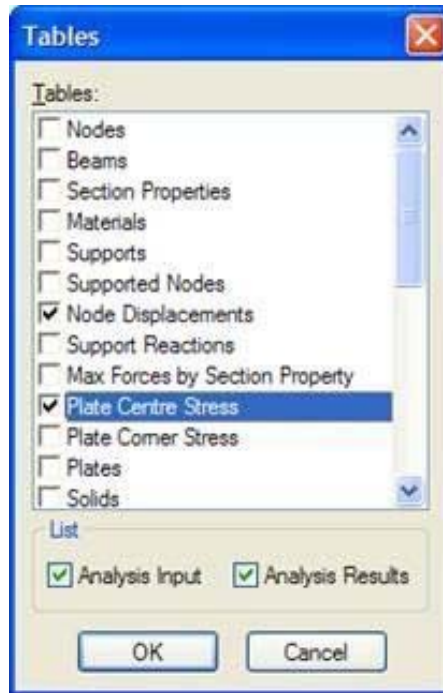
2. Select **Plate Stress Center** and click **OK**. Opens the busy city center of the plate.

Figure 2-355:

The table has the following tabs:

Shear, Membrane and Bending

Individual values for each plate for each selected load case is.

Summary

This card contains the maximum for each of the 8 values in the shear tab, membrane and bending.

Principal and Von Mises

Individual values for each plate for each load case selected for the top and bottom surfaces of the elements are shown.

Summary

This card contains the maximum for each of the 8 values in the Main tab and Von Mises.

Global Moments

The tab provides the overall time on the X, Y and Z axes in the center of each element.

3.11.2 Printing tables

These tables can be printed by clicking the right mouse button on the table area and selecting *Print out.*

Figure 2-356:
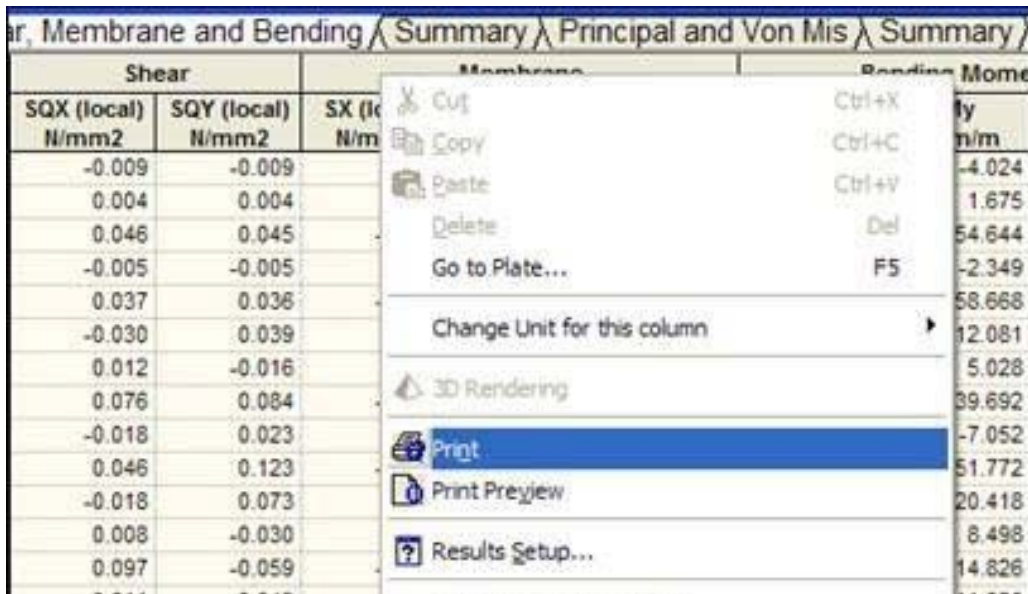
3.11.3 Changing the units of the values listed in the tables above

The units of length and strength of tensions and moments appear beside individual column headers to the terms.

1. Select **Tools> Set Current Display Unit ...**.

    The Options dialog opens

Figure 2-357:
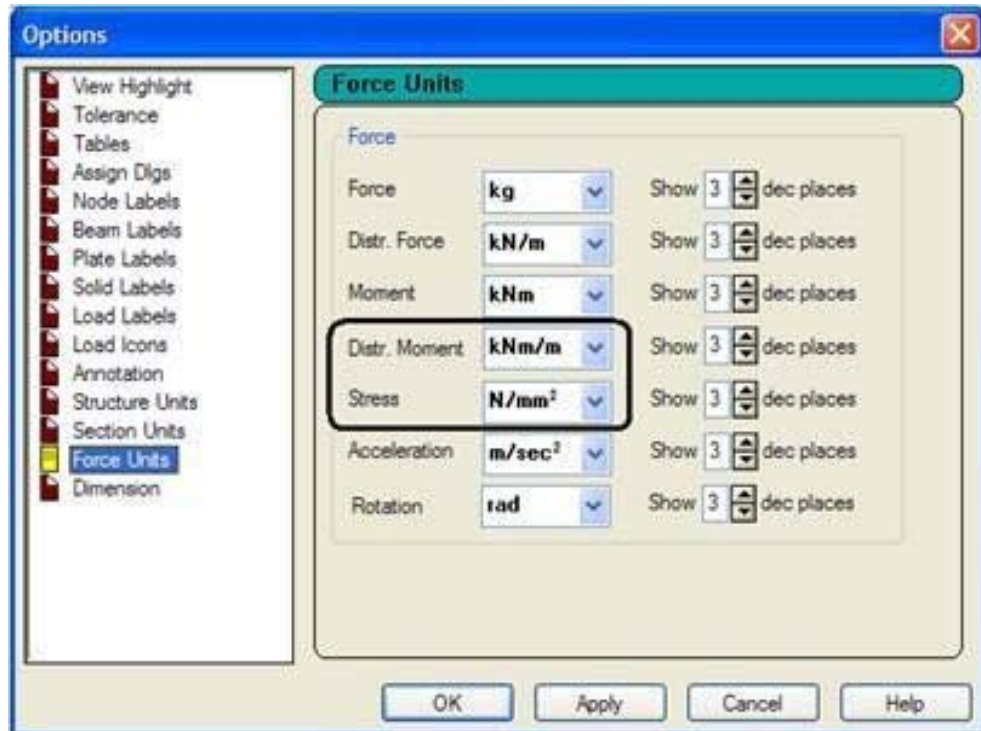
2. Select the tab **Force Units** unit and specify the required fields *Stress* and *moment*.
3. Click **Apply** for the changes to take effect immediately. Once you're sure you have chosen the right combination of drive, click **OK**.

3.11.4 Limiting the load conditions for the results shown

When we entered the post-processing mode, we choose that all cases in the dialog box *Setup Results*. Therefore, the table contains results for all load conditions.

Use the following procedure to change the loading list.

1. Select **Results> Select Load Case ...**.

Opens the Output Settings dialog.

Figure 2-358:
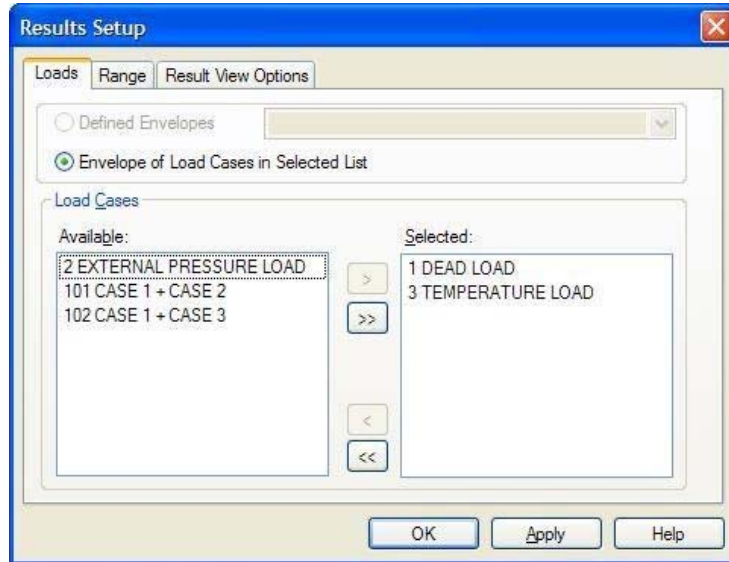
2. Choose the load cases from the list of **available** and click [>]. Selected load cases are transferred from the list **Available** to the list of selected

3. Click on **OK**.

3.11.5 Efforts curves

Efforts curves are a plot based on the color of the voltage variation or time via the surface of the slab or a selected part of it. There are two ways to turn graphics these curves:

1. Select the tab **Plate | Contour**

Figure 2-359:

Or select **Results> Plate Stress Contour**. Dialog box opens diagrams

Figure 2-360:
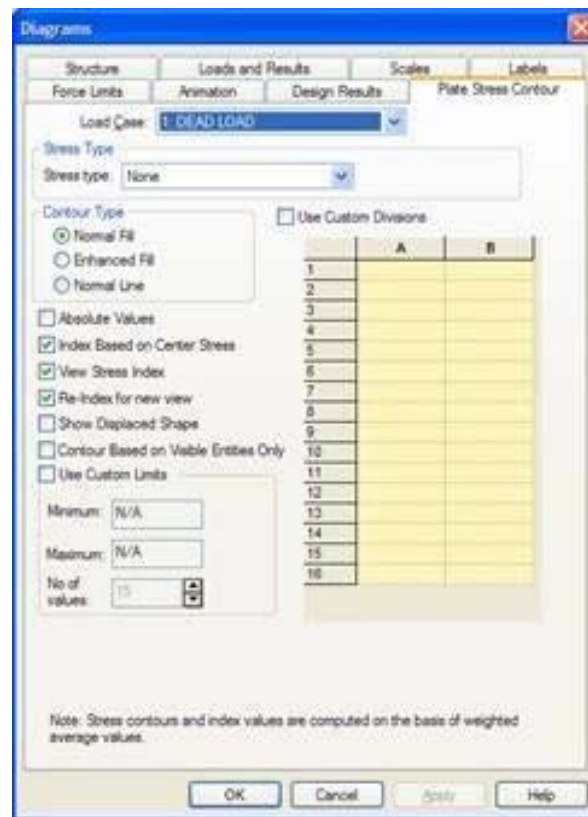
2. Field *Stress type* Select the specific type of effort that you want to get the outline drawn.

3. In the selection box *Load Case* Select the number of load case.

   The voltage values are known only at locations exactly centroid of the plate. Everywhere, calculated by linear interpolation between the values of voltage center point of adjacent plates. The contour type *Enhanced*
   (Improved) chooses a higher number of points from the outline of the type *Normal* in determining the voltage variation.

4. *View Stress Index* consisting of a small table is displayed in the numeric range of low to high values that are represented in the plot.

   Establish the following:

   - Load case: – 102

   - Stress Type: – My Top Von

   - Contour Type: – Normal Fill

   - Index based on Stress Center

   - View Stress Index

   - Re-Index for new view

5. Click on **Apply**.

Figure 2-361:
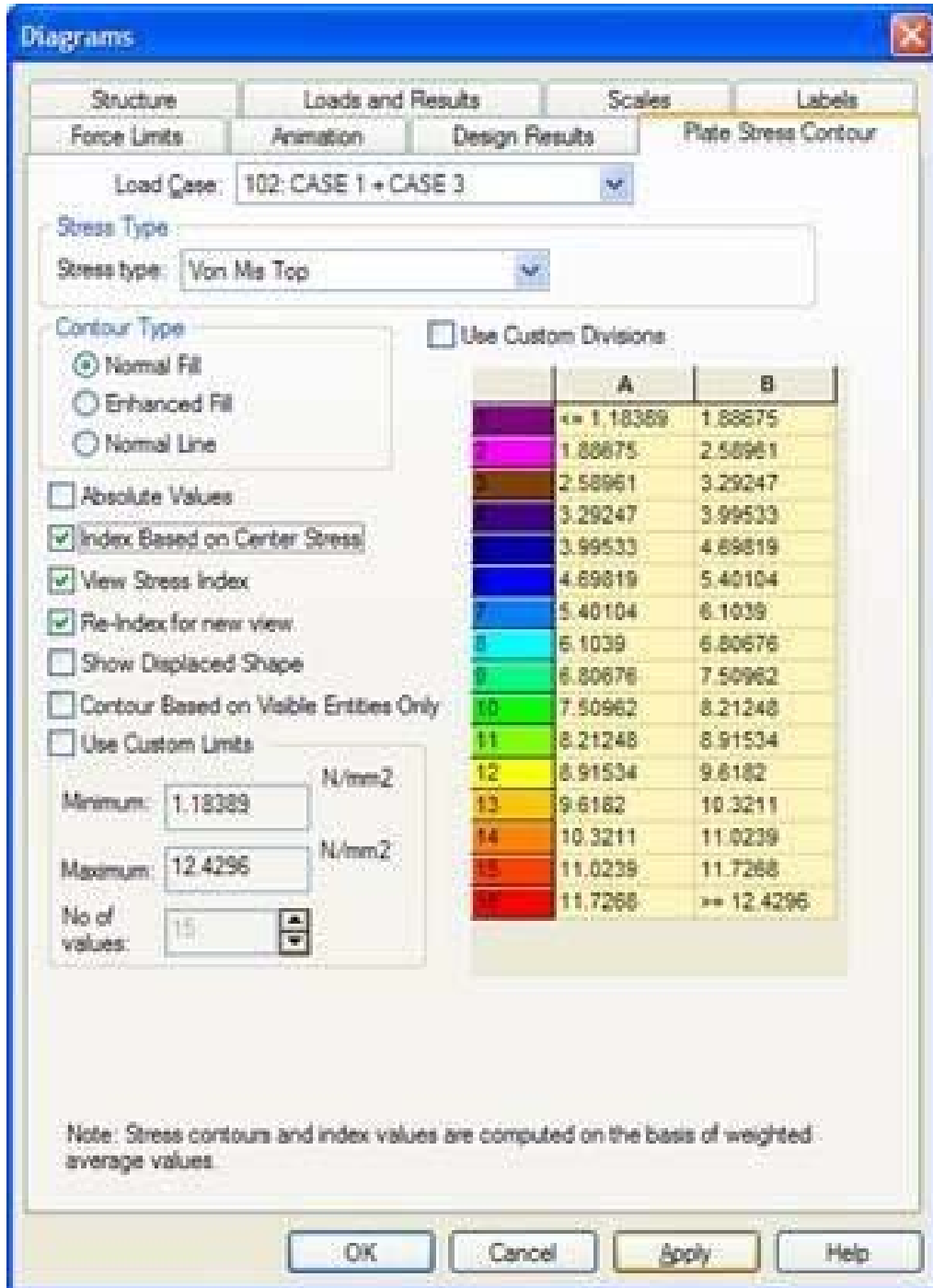
The following diagram displays. We can still change the settings and click *apply* to see all the different possible outcomes in the previous facility.

Figure 2-362:
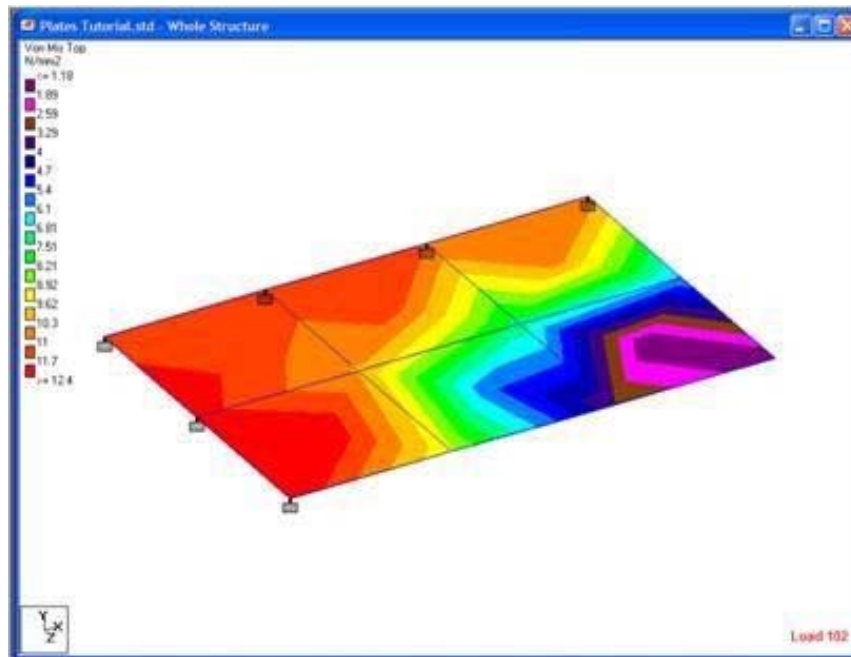
6. It opens the dialog box to examine the (animation) function explained in the next section.

   If any portion of the structure is truncated, we can bring that portion given by choosing one of the following methods:

   a. Select the tool **Zoom Out** to reduce the size of the drawn region.

Figure 2-363: Before and After using the Zoom Out tool
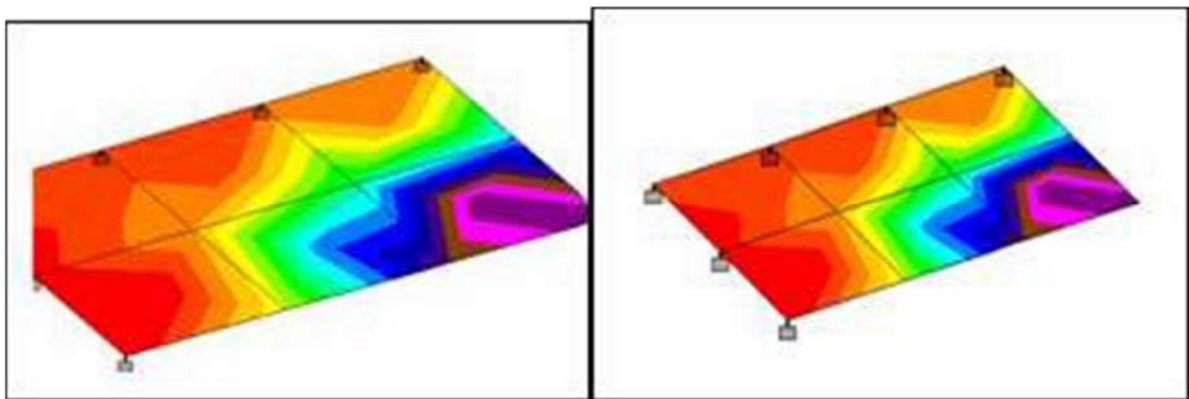
b. Select the tool **Pan** to change the position of the index structure.

Figure 2-364: Before and after using the Pan tool
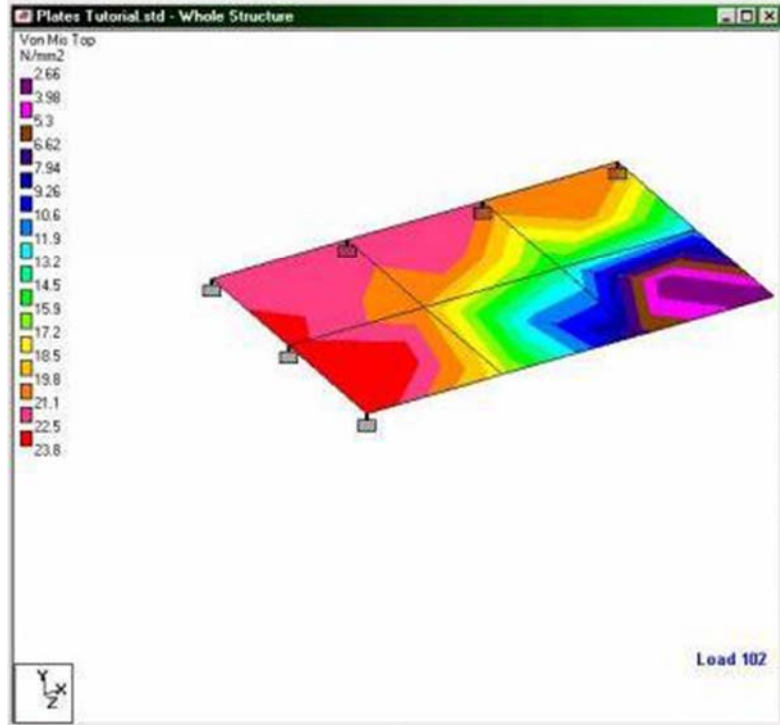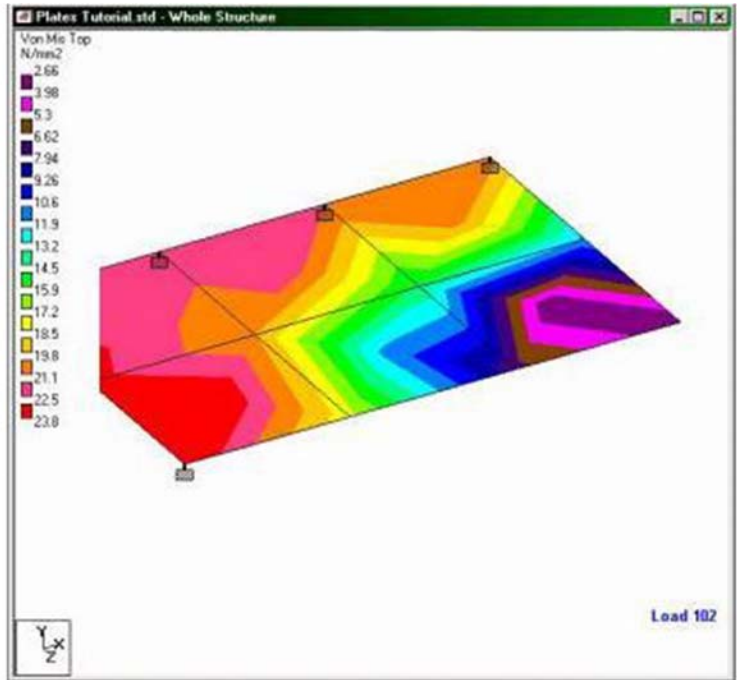
3.11.6 Animation curves efforts

The same dialog box as shown in the previous section can be used to obtain the stress contours in an animated view. This is a method to achieve a "dynamic" rather than static representation of the graph.

1. After holding elections as explained in that section, click on the animation tab of diagrams dialogue.

2. Select the option **Stress** and click **Apply**.

Figure 2-365:
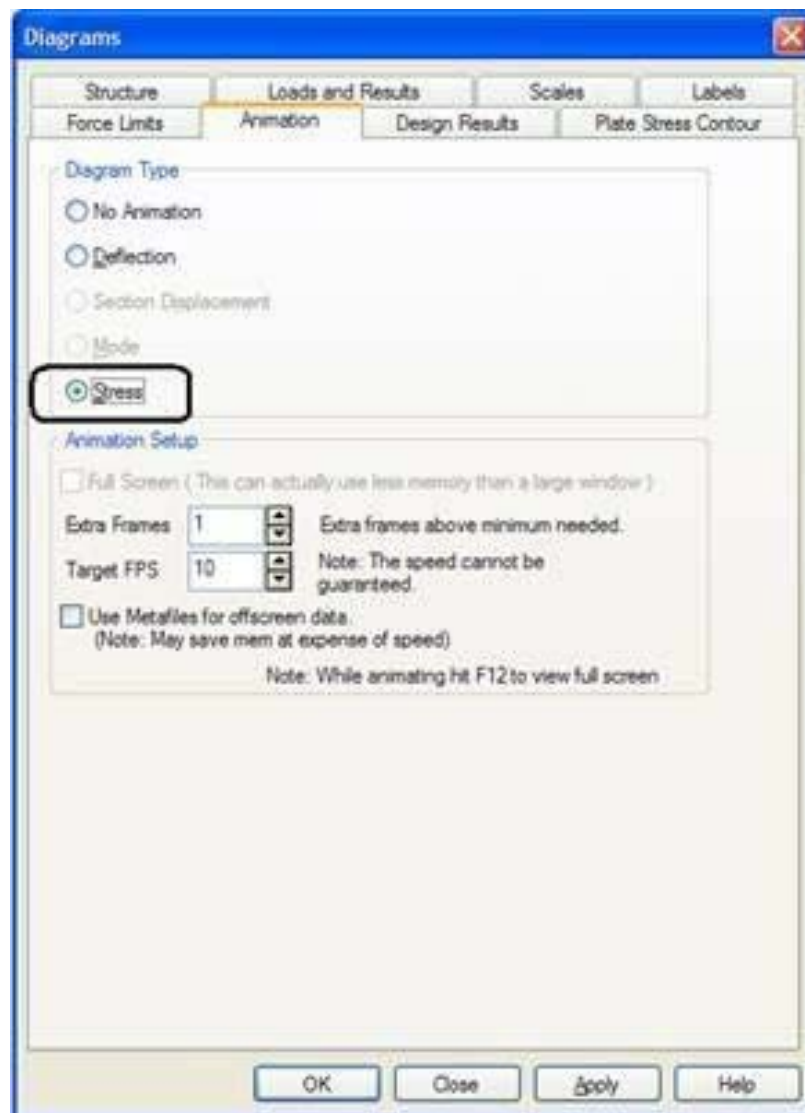
3. To stop the animation, select **No Animation** and click **Apply** again.

3.11.7 Creating AVI files

The video files (*Animation Video Files*) Are a mechanism by which a dynamic such as a deviation in the diagram animation result may be captured and recorded. Currently, this service is available in STAAD for deviation node beam displacement section plate mode shape and stress contour plots. These files can then be viewed using Adobe video software like Windows Media Player.

1. Select **Tools> Create AVI File**.

   Create AVI File dialog opens

Figure 2-366:

In an animated view, the movement from one end to the other is captured as multiple frames. The number of frames that make a movement is controlled by specifying a value for the total number of frames. The speed of movement is controlled by the rate of frames / sec. The rest of the options in the dialog above are for the type of diagram which is the video file to be created. Certain items such as modal shape and curves efforts on the plate does not become active (still dimmed) if the required data such are not present in the STAAD file, such as finite element or modal extraction.

2. After making the appropriate selections, click **OK**. A dialog box to specify the filename Opens.

3. Provide a filename and location for the video file and click **OK**.The Video Compression dialog box Opens.

Figure 2-367:
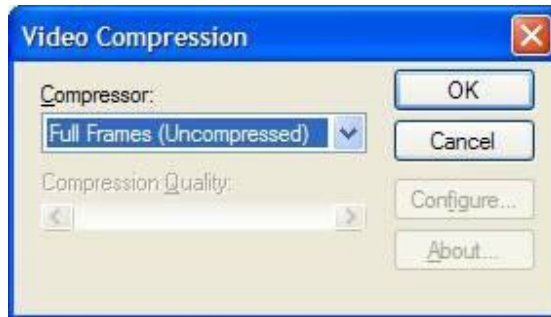
4. Select a compression option and set the value of quality. Click **OK** to start creating the video file.

> **Note**: Video files can be quite large, and compression is a technique used to reduce the size of these files, but the softness of some videos is lost in this process.

When the file is generated, a message indicating that the operation was successful appears.

Figure 2-368:
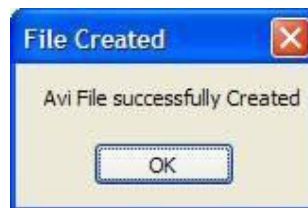
5. Click **OK** to close.

The AVI file with the extension is saved in the same folder where the STAAD input file is located.

3.11.8 Viewing results by consulting the plate element

The Query element is a command where you can see several results for a specific element while a single dialog box. Exploring this facility for item 4.

1. Select the tool **Cursor Plate**.

2. Double-click the item 4 or 4 and select select element **Tools> Query> Plate**. Plate dialogue opens

The various tabs in the query box that let you view various types of information such as the geometry of the plate, constant property, voltages, etc., for various load cases and print those values. Some tablets example of this dialog box shown in the following figures.

Figure 3 141
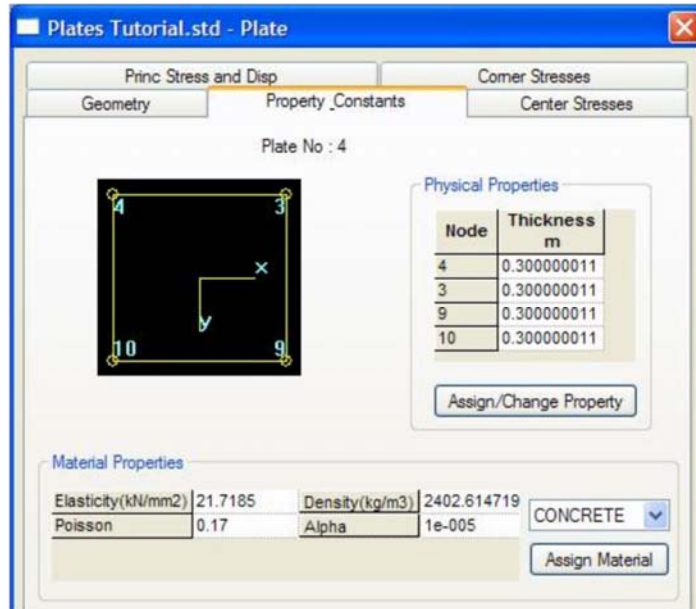
Figure 3 142
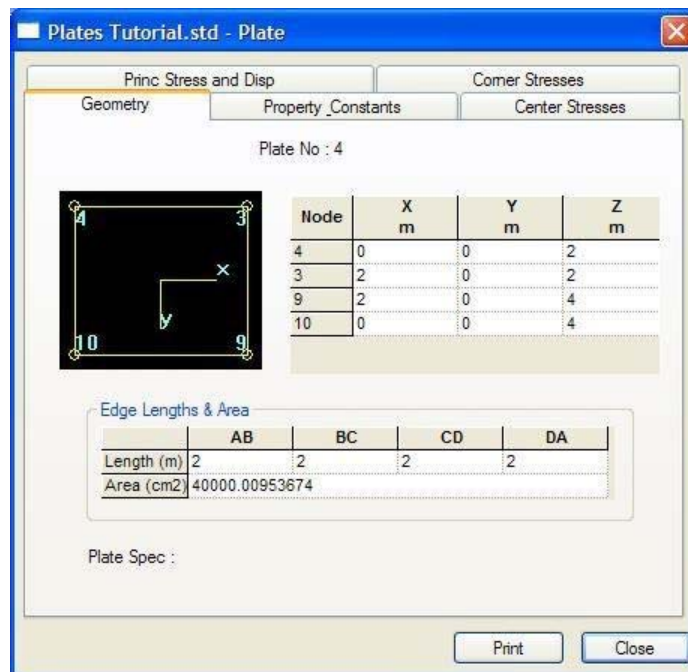
Figure 3 143
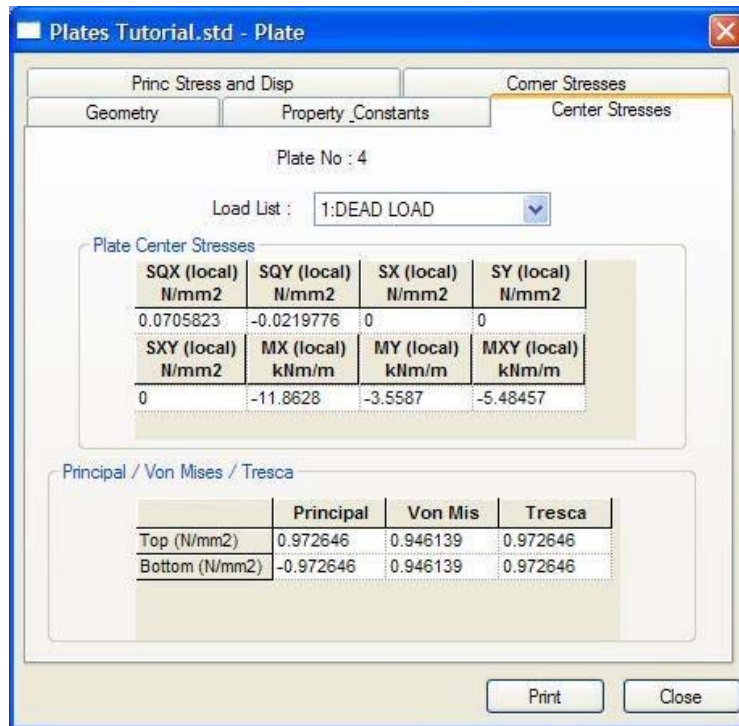
Figure 3 144
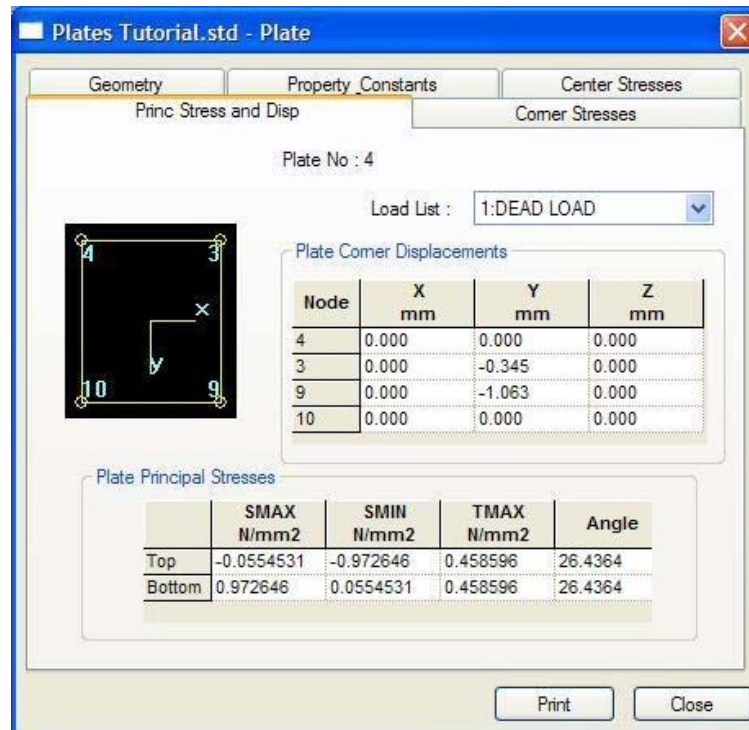
Figure 3 145
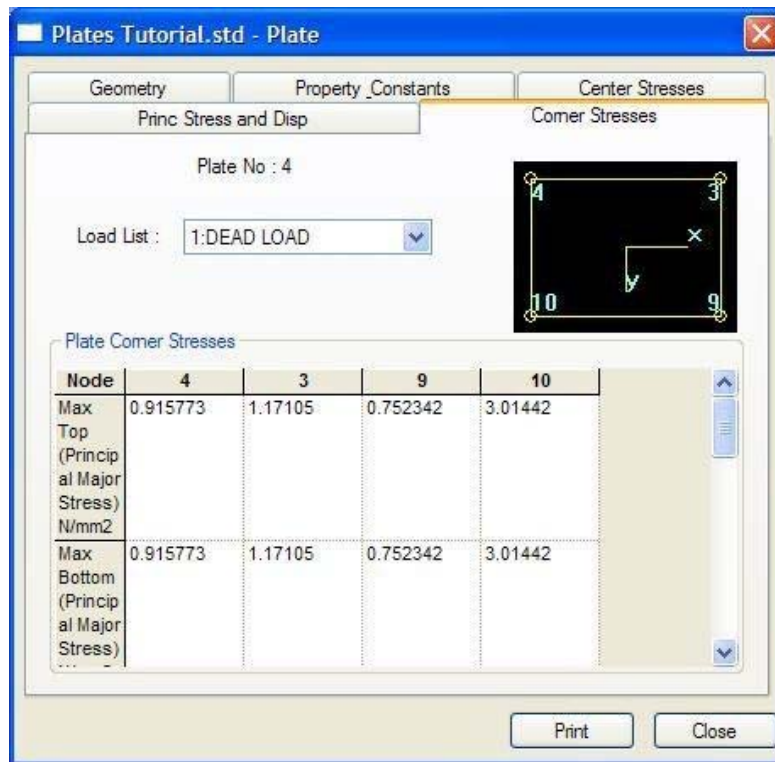
3.11.9 Preparing a report on screen

Sometimes we will come across the need for results under certain restrictions, such as displacement of the resulting node to a few selected nodes, for a few selected load cases, ranked in order from low to high, with values reported in tabular form. The service allows us to obtain such a modified

customized on the screen is the menu results *report (Report)* above screen.

Let us elaborate a report consisting of the principal stresses of the plate, for all boards, classified in the order from low to high voltage, high-Principal (SMAX) for load cases 101 and 102 (load combinations).

1.   Select all plates using *Plates Cursor*

2.   Select **Report> Plate Results> Principal Stresses**.

     The dialog box forces Opens plate.

Figure 2-369:
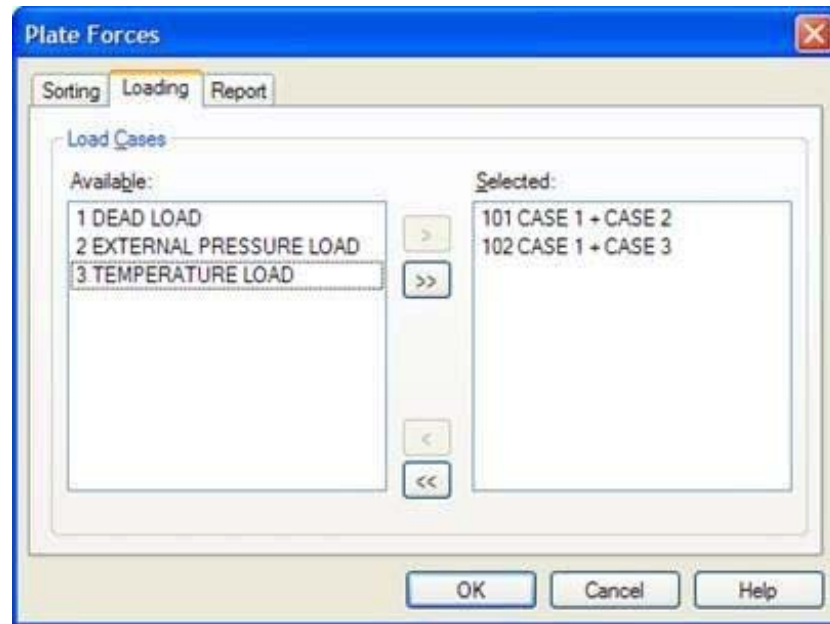
3. Select the tab **Loading**.

4. Select load cases **101** and **102** available in the list and click [>] to add to the short list.

5. Select the tab **Sorting**. Choose **SMAX** low *Sort by Plate Stress* and select **List from Low to High** how come *September Sorting Order*.

Figure 2-370:
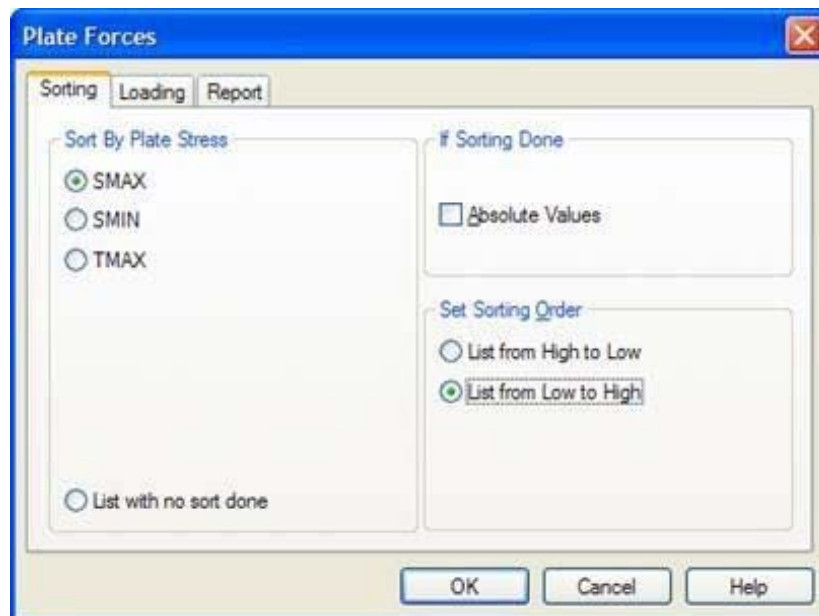
6. (Optional) If you want to save this report for future use, select the tab **Report**Provides a title for the report and set the checkbox **Save ID**.

7. Click on **OK**.

    The following table shows the maximum principal stress with SMAX values ranked from low to high.

Figure 2-371:

Plate	L/C		SMAX N/mm2	SMIN N/mm2	TMAX N/mm2	Angle
1	102	Top	-7.700	-13.323	2.812	-39.107
4	102	Top	-2.889	-13.620	5.365	-41.610
3	102	Top	-0.923	-11.075	5.076	-37.347
5	102	Top	-0.450	-5.770	2.660	-18.993
3	101	Top	-0.129	-0.820	0.346	78.932
1	101	Top	-0.045	-0.267	0.111	45.073
4	101	Top	-0.032	-0.568	0.268	26.436
2	102	Bottom	0.000	0.000	0.000	0.000
2	102	Top	0.000	0.000	0.000	0.000
6	102	Top	0.000	0.000	0.000	0.000
2	101	Bottom	0.000	0.000	0.000	0.000
6	101	Bottom	0.000	0.000	0.000	0.000
2	101	Top	0.000	0.000	0.000	0.000
6	101	Top	0.000	0.000	0.000	0.000
6	102	Bottom	0.000	0.000	0.000	0.000
1	102	Bottom	0.091	-5.486	2.789	-38.968
1	101	Bottom	0.267	0.045	0.111	45.073
5	101	Top	0.324	-0.330	0.327	54.271
5	101	Bottom	0.330	-0.324	0.327	54.271
4	101	Bottom	0.568	0.032	0.268	26.436
5	102	Bottom	0.740	0.005	0.367	38.036
3	101	Bottom	0.820	0.129	0.346	78.932
4	102	Bottom	0.958	-6.449	3.703	-47.958
3	102	Bottom	1.849	-4.560	3.204	-32.091

8. To print this table, right-click anywhere within the table and select Print from the menu.

Select the option to print a hard copy of the report to transfer the contents

of this table to a Microsoft Excel file.

1. Click the top left corner of the table with the left mouse button.
    Entire table appears checked.

2. Right-click and select **Copy** the popup menu.

3. Open an Excel spreadsheet, click on the desired cell and **paste** content.

3.11.10 Viewing the reactions at the supports

Since the supports are in the nodes of the structure, the results of this type are available along with other node results as displacement.

1. Select the tab **Node | Reactions** on the left side of the screen.

Figure 2-372:
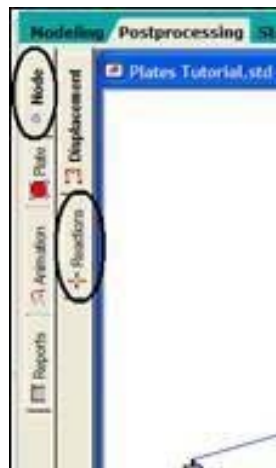
The support reactions are shown on the drawing as shown below.

Figure 2-373:
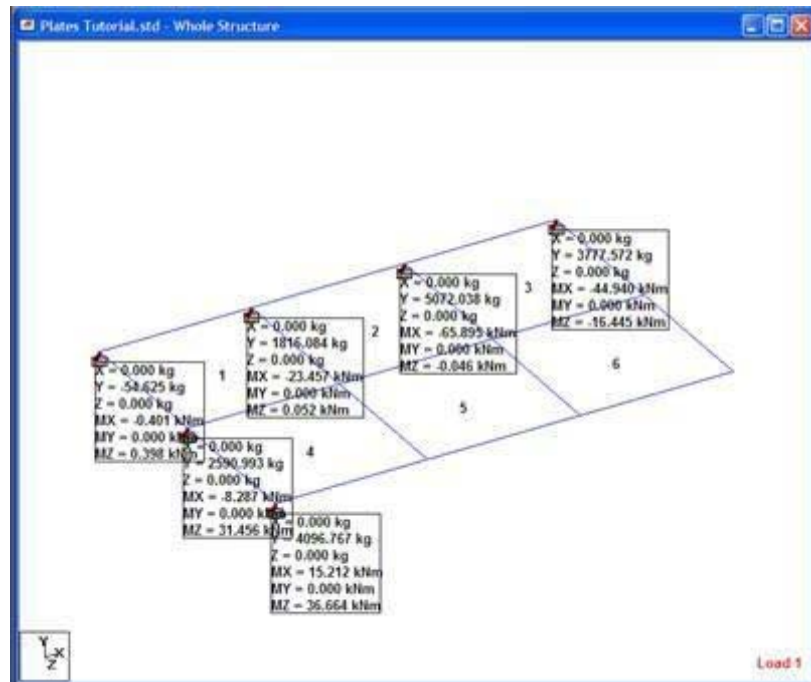
The six values - that is, the three forces overall along X, Y and Z and three Mx, My and Mz moments in the global axis system - shown in a box for each support node.

Displays one or more of the six terms of each support node can be set off in the following manner.

1. Select **Results> View Value ...**.

Open dialog annotation.

Figure 2-374:
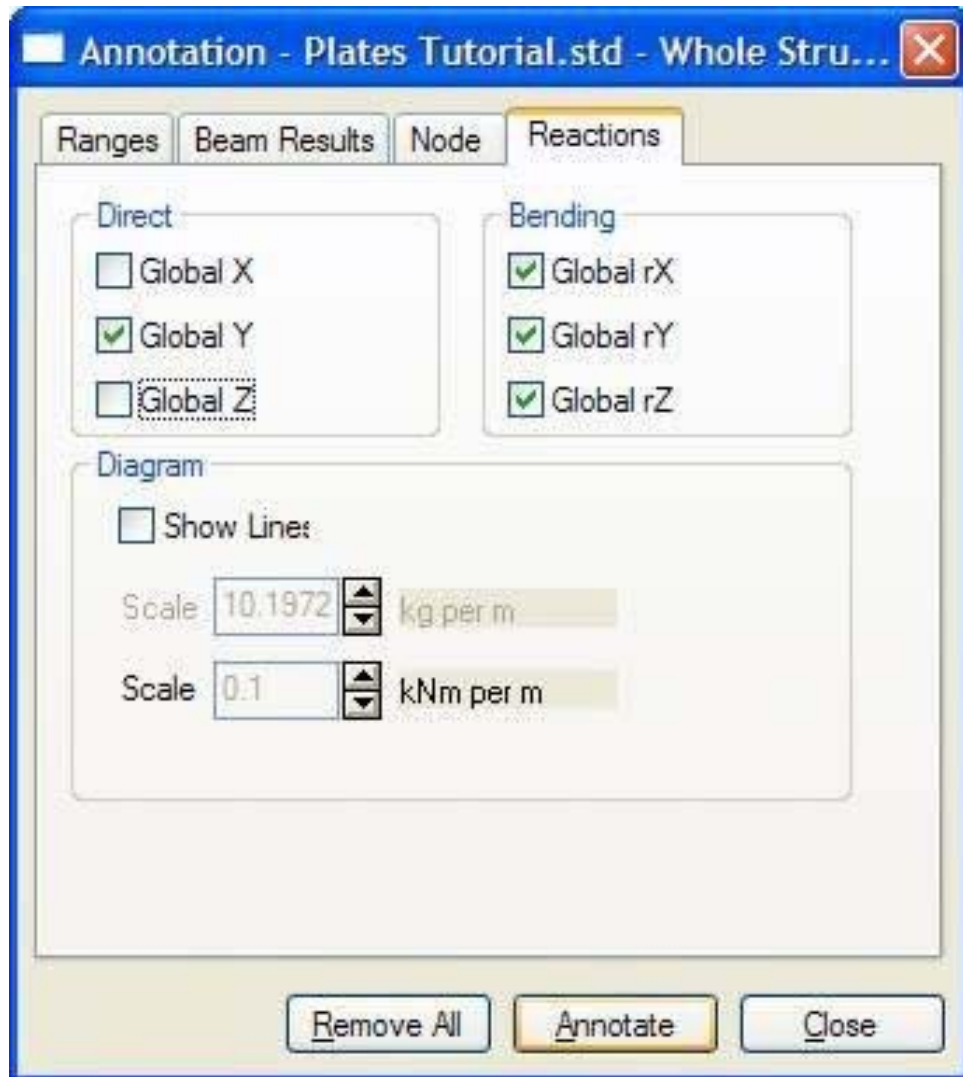
2. Select the **Reactions** tab. Clear the **Global X** and **Global Z** check boxes in the *Direct* category. Select the tab **Reactions.** Uncheck the boxes **Global X** and**Global Z** category *Direct*.

3. Click **Annotate** and then **Close**.

The drawing will contain only the remaining 4 terms (see figure below).

Figure 2-375:
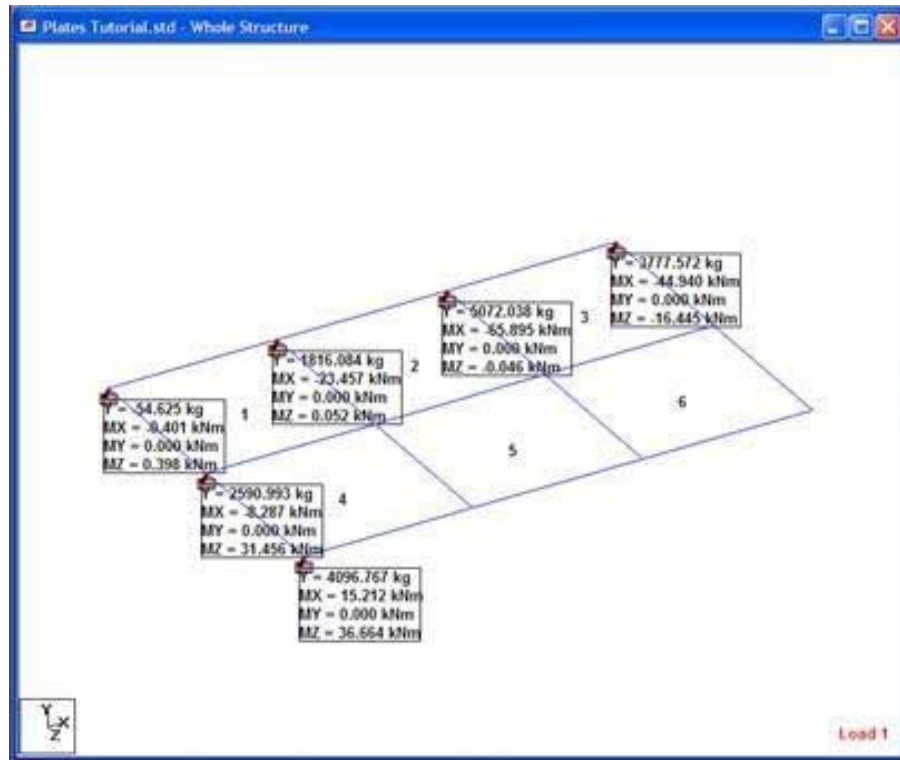
To change the load case for which the reactions are displayed, select the desired selection box load case.

Figure 2-376:
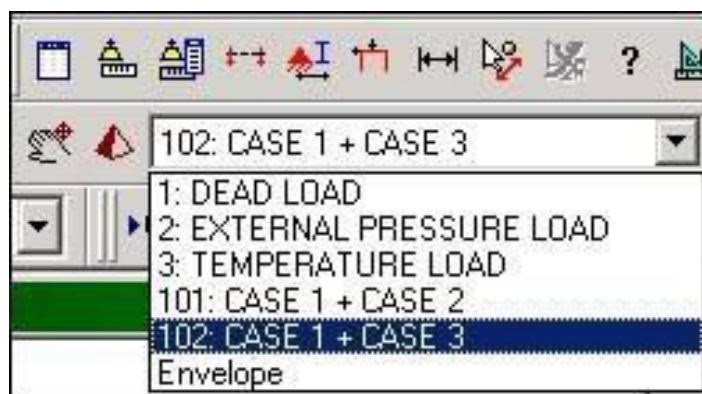
For clarity in displaying the results in the drawing area (and to reduce clutter on the screen), a variety of methods are available. For example, hold the mouse button down on the **Zoom In** and see the drawing progressively larger. Use

button **Pan** to physically change the drawing around. Also usable other

options as buttons *Print / dynamic reduction* (*Dynamic Zoom)* and**zoom in / out window** (**Zoom Window)**. To restore the original view, click **Display Whole Structure** (Show all structures). (Some of these options are explained in more detail in the section '

Tasks often done 'at the end of this manual).

Icon	Name
	Zoom In
	Pan
	Dynamic Zoom
	Zoom Window
	Display Whole Structure

The table to the right of the screen contains the values for all reaction media for all selected load cases.

Figure 2-377:

Plates Tutorial.std - Support Reactions

All / Summary / Envelope /

Node	L/C	Horizontal Fx kg	Vertical Fy kg	Horizontal Fz kg	Moment Mx kNm	My kNm	Mz kNm
1	1 DEAD LOA	0.000	-54.624	0.000	-0.401	0.000	0.399
	2 EXTERNAL	0.000	-22.735	0.000	-0.167	0.000	0.166
	3 TEMPERAT	172.8E3	2649.422	152.5E3	-78.401	70.559	78.382
	101 CASE 1 +	0.000	-77.359	0.000	-0.568	0.000	0.565
	102 CASE 1 +	172.8E3	2594.799	152.5E3	-78.802	70.559	78.781
2	1 DEAD LOA	0.000	1816.168	0.000	-23.458	0.000	0.053
	2 EXTERNAL	0.000	755.913	0.000	-9.763	0.000	0.022
	3 TEMPERAT	-208.18E3	5522.880	177.2E3	-148.111	361.463	28.141
	101 CASE 1 +	0.000	2572.082	0.000	-33.221	0.000	0.075
	102 CASE 1 +	-208.18E3	7339.049	177.2E3	-171.568	361.463	28.194
4	1 DEAD LOA	0.000	2591.007	0.000	-8.287	0.000	31.456
	2 EXTERNAL	0.000	1078.411	0.000	-3.449	0.000	13.092
	3 TEMPERAT	295.97E3	3055.179	-241.91E3	18.182	-38.685	106.888
	101 CASE 1 +	0.000	3669.418	0.000	-11.737	0.000	44.548
	102 CASE 1 +	295.97E3	5646.186	-241.91E3	9.895	-38.685	138.344

This table can also be viewed from any mode by clicking on the menu *see*, Choosing *tables*, And on the *support reactions*.

The method explained in Section 3.11.3 can be used to change the units in which these values are shown. The Summary tab contains the maximum value for each of the 6 degrees of freedom along with the case number responsible for this charge.

Figure 2-378:

	Node	L/C	Horizontal Fx kg	Vertical Fy kg	Horizontal Fz kg	Moment Mx kNm	My kNm	Mz kNm
Max Fx	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Min Fx	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Max Fy	5	102 CASE 1 +	-267.41E3	9209.981	42301.568	-150.203	-163.894	-12.020
Min Fy	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Max Fz	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Min Fz	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Max Mx	10	102 CASE 1 +	339.42E3	-3669.513	-367.75E3	38.652	32.043	32.681
Min Mx	2	102 CASE 1 +	-208.18E3	7339.049	177.2E3	-171.568	361.463	28.194
Max My	2	3 TEMPERAT	-208.18E3	5522.880	177.2E3	-148.111	361.463	28.141
Min My	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Max Mz	4	102 CASE 1 +	295.97E3	5646.186	-241.91E3	9.895	-38.685	138.344
Min Mz	7	102 CASE 1 +	-332.6E3	-3821.673	237.67E3	-20.462	-261.486	-37.400

Section 3

Frequently Performed Tasks

1. Selection toolbar

Nodes Cursor

Beams Cursor

Plates Cursor

Solids Cursor

Geometry Cursor

Text Label Cursor
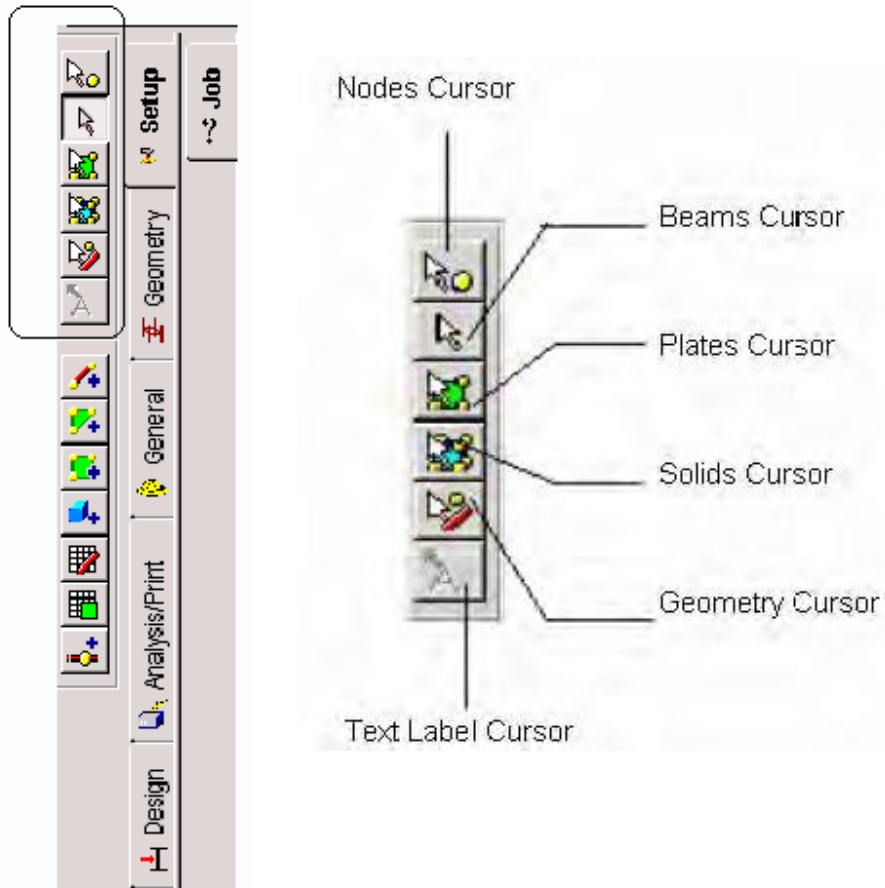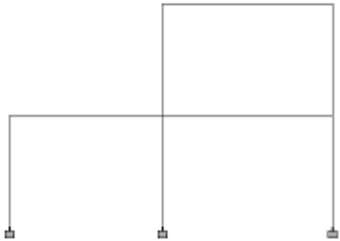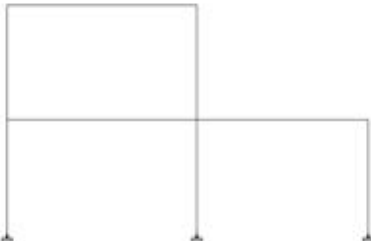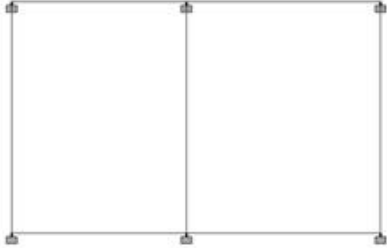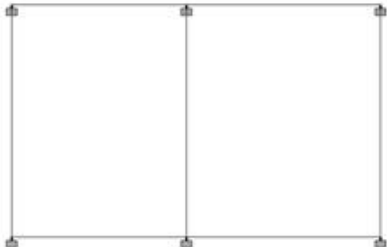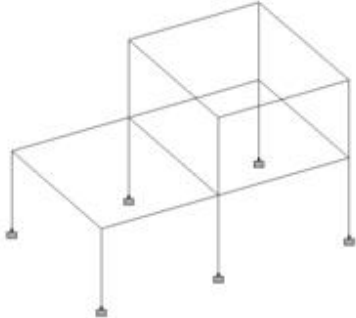
Icon	Item Correspondent menu	Purpose	Description
Nodes Cursor	Select> Cursor Nodes	It is used to select the nodes graphically	First, select the Nodes Cursor tool. Then, click to select the nodes. To select multiple nodes, hold down the Control key while you select or create a window style elastic band around the desired nodes.

Beams Cursor	Select> Cursor Beams	It is used to graphically select the beams	First, select the Cursor beams. Then, click to select the members you want. To select multiple members, hold down the Control key while you select or create a window style band around the desired beams.
Plates Cursor	Select> Cursor Plates	It is used to select the plates graphically	First, select the Cursor plate. Then click dishes to select. To select multiple plates, hold down the Control key while you select or create a window rubber band around the desired style plates.
Surface Cursor	Select> Cursor Surface	It is used to select the surface graphically	First, select the Cursor surface. Then click the area you want to select. To select multiple areas, hold down the Control key while you select or create a window style rubber band around the desired surfaces.
Solids Cursor	Select> Cursor Solids	It is used to graphically select solids	First, select the Cursor solids. Then, click the solids to be selected. To select multiple solids, hold down the Control key while you select or create a window style elastic band around the desired solid
Geometry Cursor	Select> Geometry Cursor	Select any geometry graphically. It is a mechanism for selecting the nodes, beams, plates, solid or any combination thereof, at the same time.	First, select the Cursor geometry. Then click the feature you want to select. To select multiple institutions, hold down the Control key while you select or create a window style rubber band around the desired entities.

Load Edit Cursor	Select> Load Edit Cursor	Used to graphically select a component of an existing load case for modification.	First, select the edition cursor load. Then double-click the component diagram of load you want to edit.
Edit Cursor Support	Select> Edit Cursor Support	Used to graphically select a medium whose parameters one wishes to modify.	First select the Cursor editing support. Then double-click on the help icon on the node where you want to edit the details of existing aid.
Release Edit Cursor	Select> Edit Cursor Release	Used to graphically select and modify an existing member specification release	First, select the Edit Cursor release. Then double-click the member in which a condition of initial or final release is currently defined to be modified.
Select Text	Select> Text Cursor	Use to enter the edit mode text labels previously created	To edit any previously created text, first select the text cursor label. Then double-click the text to edit.
Filtered Selection	Select> Filtered Selection Cursor	Used to select various types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in a single pass.	This is used to select various types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in a single pass.
Select Joints	Select> Cursor Joints	Used for select together, when you use the RAM interface connection from within STAAD.	This is used to select joint. With this cursor as the active cursor selection, one can assign joint written design or you can just double-click to launch joint RAM connection, etc.

2.Toolbar rotation

Icon	Description	Example
View From Z +	Shows the structure as seen from the front. When the global axis is vertical, this is the elevation view, and looking towards the negative direction of the z axis.	
View From - Z	Shows the structure as seen from the rear. When the global axis is vertical, it is the opinion of elevation, as seen looking toward the positive direction of the z axis.	
View From + X	Shows the structure as seen from the right side. When the global axis is vertical, that is the side elevation, as seen looking toward the negative direction of x axis.	
View From - X	Shows the structure as seen from the left side. When the global axis is vertical, that is the side elevation, as seen looking toward the positive direction of x axis.	

Icon	Description	Example
View From + Y	Shows the structure as seen from the top looking down. When the global axis is vertical, this is the plan view, as seen from the sky looking down.	
View From – Y	The structure is shown as if one is looking to the sky.	
Isometric View	The structure shown in isometric view. The angle defining the isometric view is generally X = 30, Y = 30, Z = 0	

3.Switching labels for nodes, beams, plates, etc.

Tags are a way to identify the entities that have drawn on the screen. To demonstrate the ease, open EXAMP01.STD as shown in the following figure.
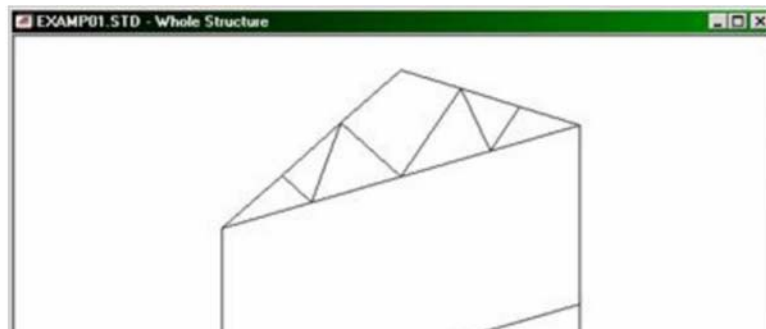
Figure 3-1:

1. To change node labels and beam

- Select the tool of symbols and labels (**Symbols and Labels)**.

Figure 3-2:

or
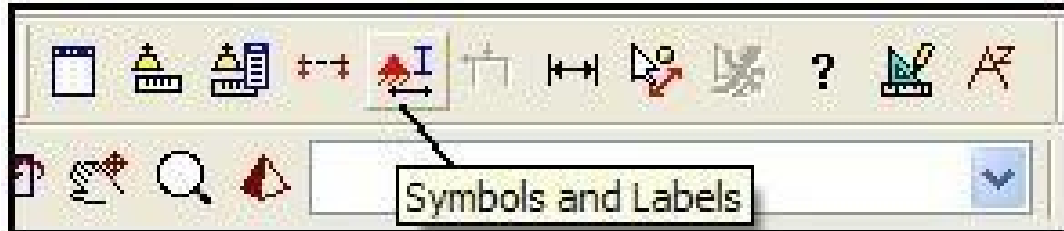
- Right-click anywhere in the view area and select from the popup menu labels of readjustment

Figure 3-3:

or
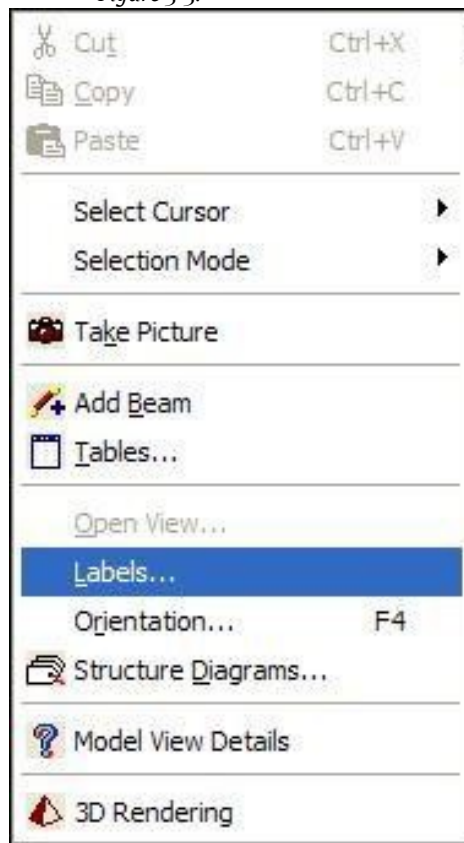
- Select **View> StructureDiagrams**. Open dialog box diagrams.

2. Select the Labels tab and select the options for the appropriate tags (examples shown in the following figure). Click **OK** to update the view window and close the dialog box.

Figure 3-4:
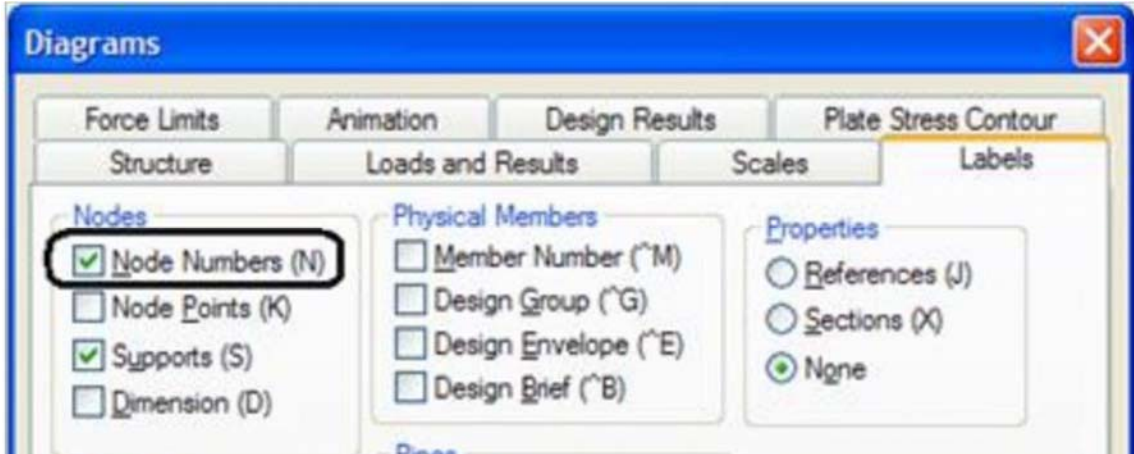
The following figure shows the structure with node and beam labels appear on it.

Figure 3-5:
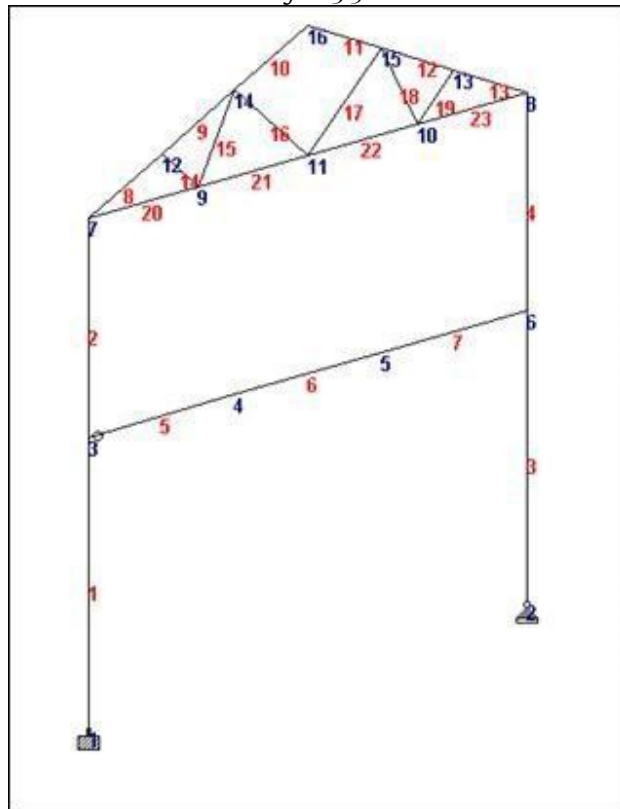
3. To change the font of the labels of node / beam, select **View> Options**. The Options dialog box opens.

Figure 3-6:
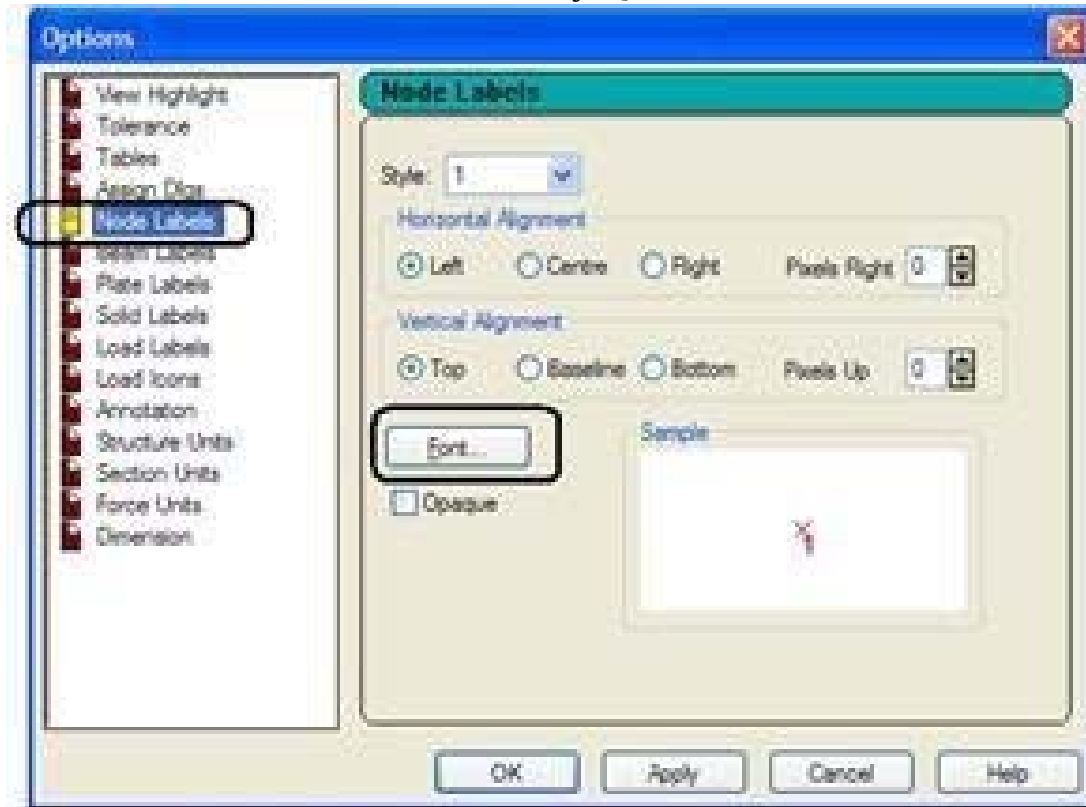
4. Select the appropriate tab (Node Labels / Beam labels) and click **Font**... To make the desired changes. Click **OK** to close the Font dialog box.

4. Displaying a part of the model

Sometimes, the large number of entities that are drawn on the screen can make it difficult to clearly see the details on any particular region of the structure. In such cases, one is faced with the task of removing cluttering the screen or in specific regions or entities while the rest of the structure is removed from the point of view.

There are different methods by which you STAAD.Pro can see a part of the structure. The following pages show three different methods.

Method 1 - View> New View

1. To demonstrate this method, open **EXAMP08.STD** (C: \ SPro2007 \ STAAD \ examp \ AU) as shown in the following figure.

Figure 3-7:
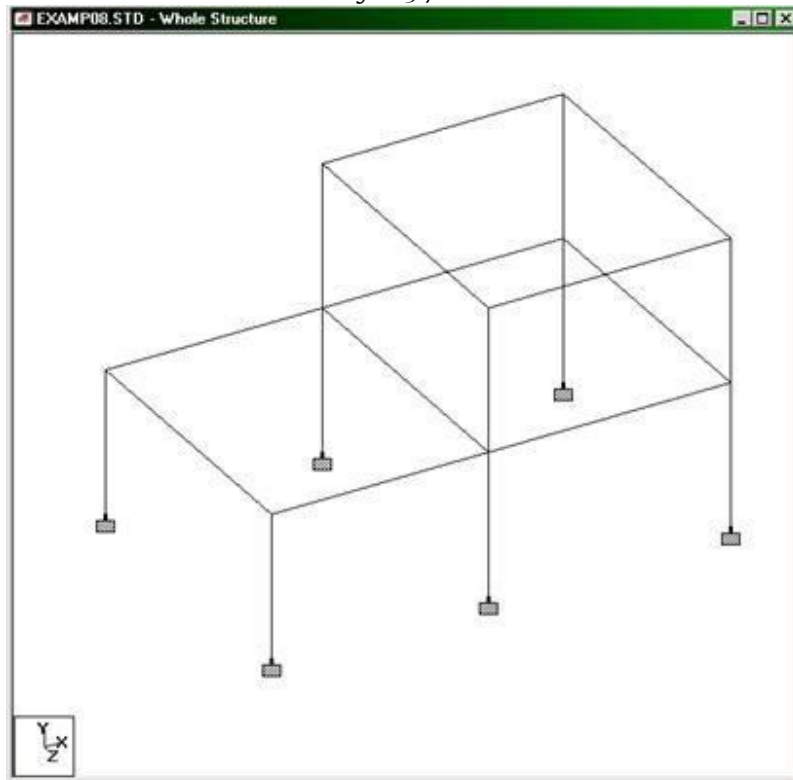
2. To see the members who are halfway up, first select the tool by selecting members **View From**

   Z + and dragging a rubber band around these members in the view window.

> **Tip**: Ensure that the tool **Beams Cursor** is the selection tool is active.

Figure 3-8:
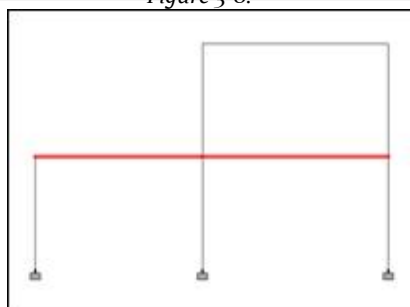
3. To visually check that they have been selected the correct members, select the tool

   Isometric View .

   Selected members are highlighted.

Figure 3-9:
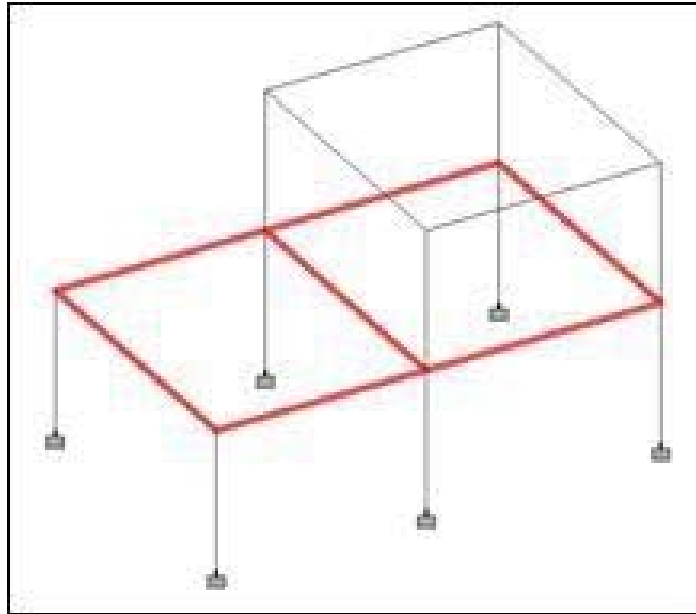
4. Create a new view by selecting **View> New View ...**.or

   Right-click anywhere in the viewing window and select

   New View ... the popup menu. Opens the New View dialog.

Figure 3-10:
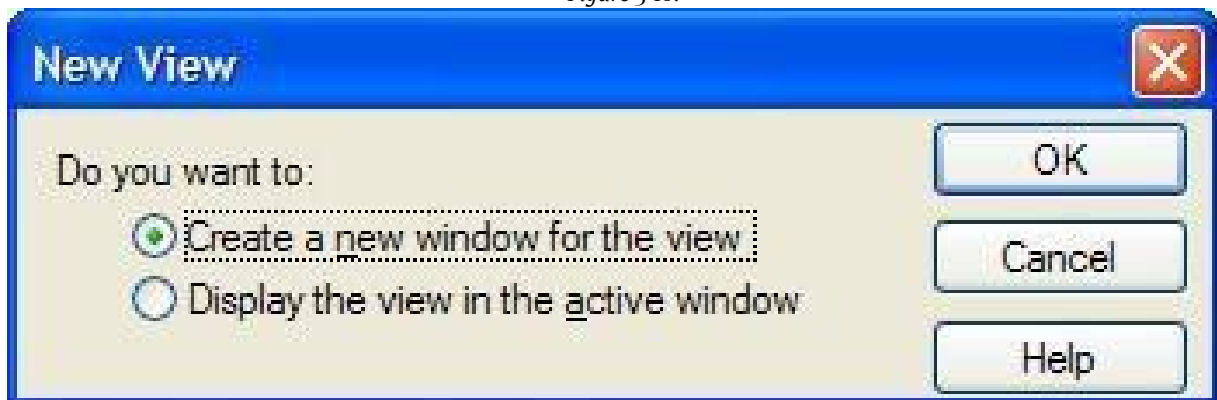
The dialog box includes options to open the new view in a new window or to replace the current window view.

5. Select the option **Create a new window for the view** and click **OK**.

   The portion of the structure that we have selected will now be displayed in a new window as shown in the following figure. Auxiliary view multiple windows can be created in this way.

Figure 3-11:
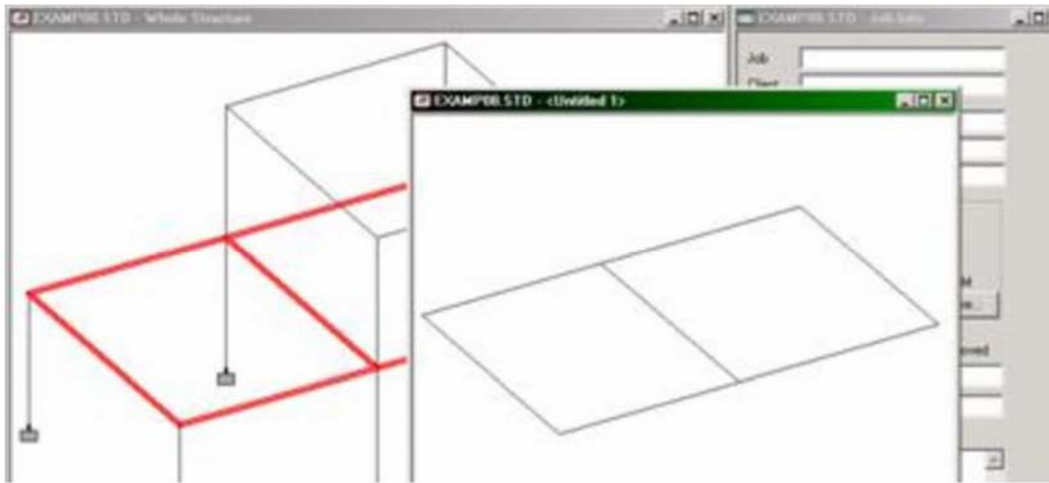
> **Tip**: The new windows are useful for performing operations such as adding and removing members, assigning properties, loads, stands and more. A new view of a selected portion has the advantage of clearing the screen and limit the few elected bodies visualized objects.

6. To return to the main view window, close the new window by clicking secondary view close [X] in the upper right corner.

7. Repeat steps 2-4 to start another new view.

8. Select the option **Display the view in the active window** and click on OK.

   The original structure and the portion of the structure we have in place selected will be displayed in the current view main window as shown in the following figure is hidden.

Figure 3-12:
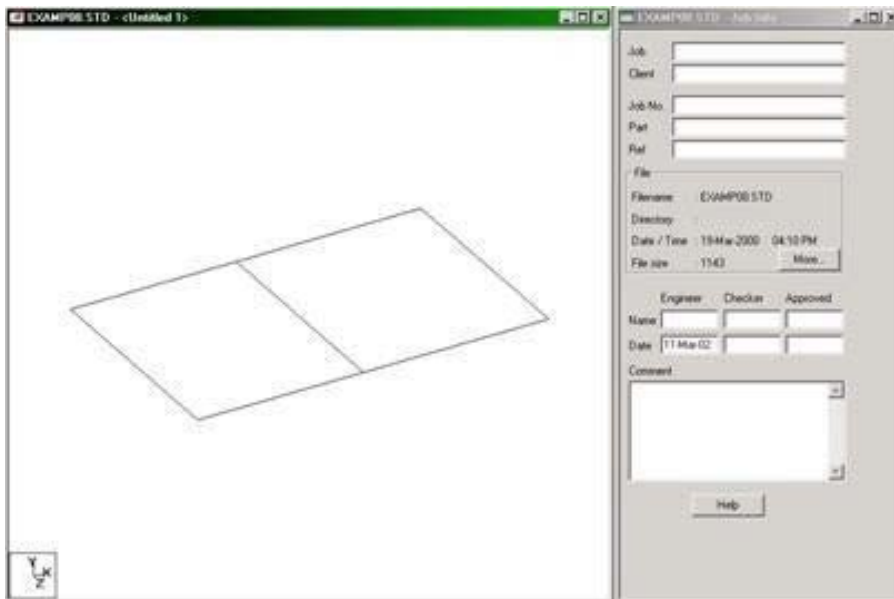
9. Select the tool **Display Whole Structure** to restore the original view.

Figure 3-13:
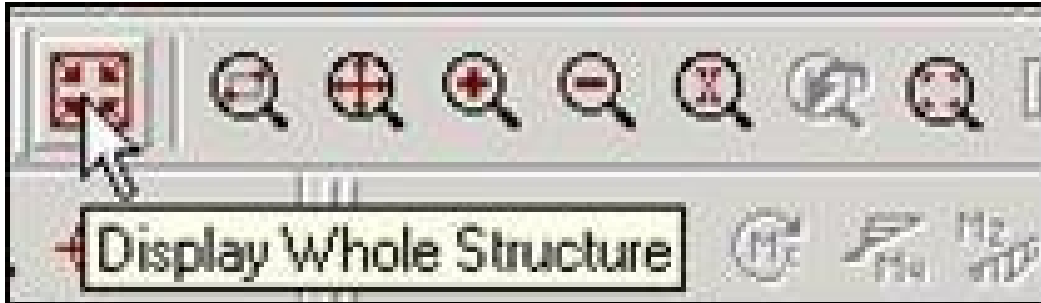
> **Note**: These new insights can be saved by going to the menu **View> View Management> Save View**. Provide a title for the new view. Later these saved views can be opened by going to the menu **View> Open View**.

Method 2 - Tools> Cut Section

1. To demonstrate this method, open **EXAMP08.STD** (C: \ SPro2007 \ STAAD \ examp \ AU) as shown in the following figure.

Figure 3-14:
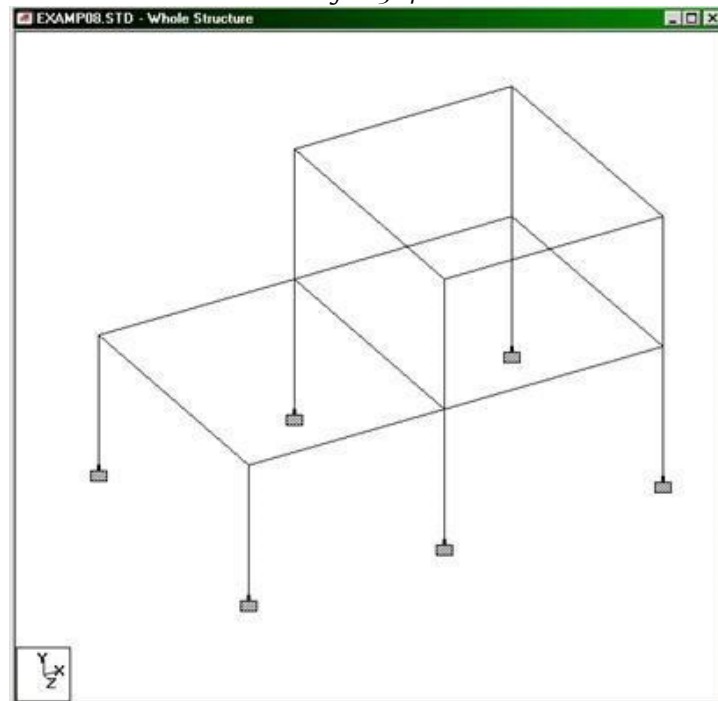
2. Select the tool **Symbols and Labels**.

   The Diagrams dialog opens. Select the option **Node Numbers** and click on OK.

This will aide to specify a number of nodes in the next step.

Figure 3-15:
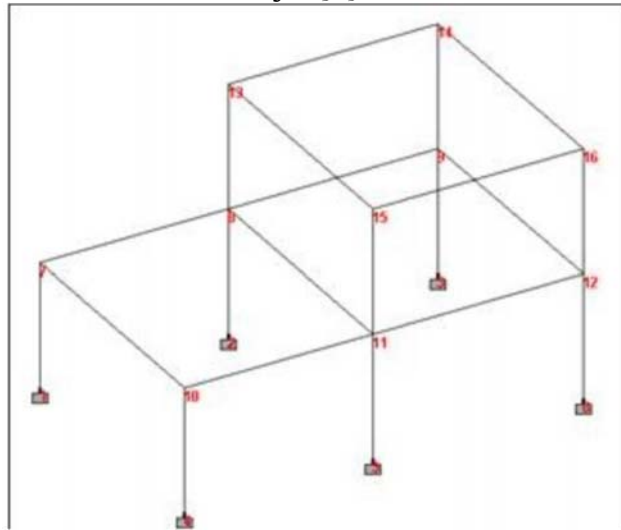
3. Select **Tools> Cut Section**. Open dialog section.

Figure 3-16:
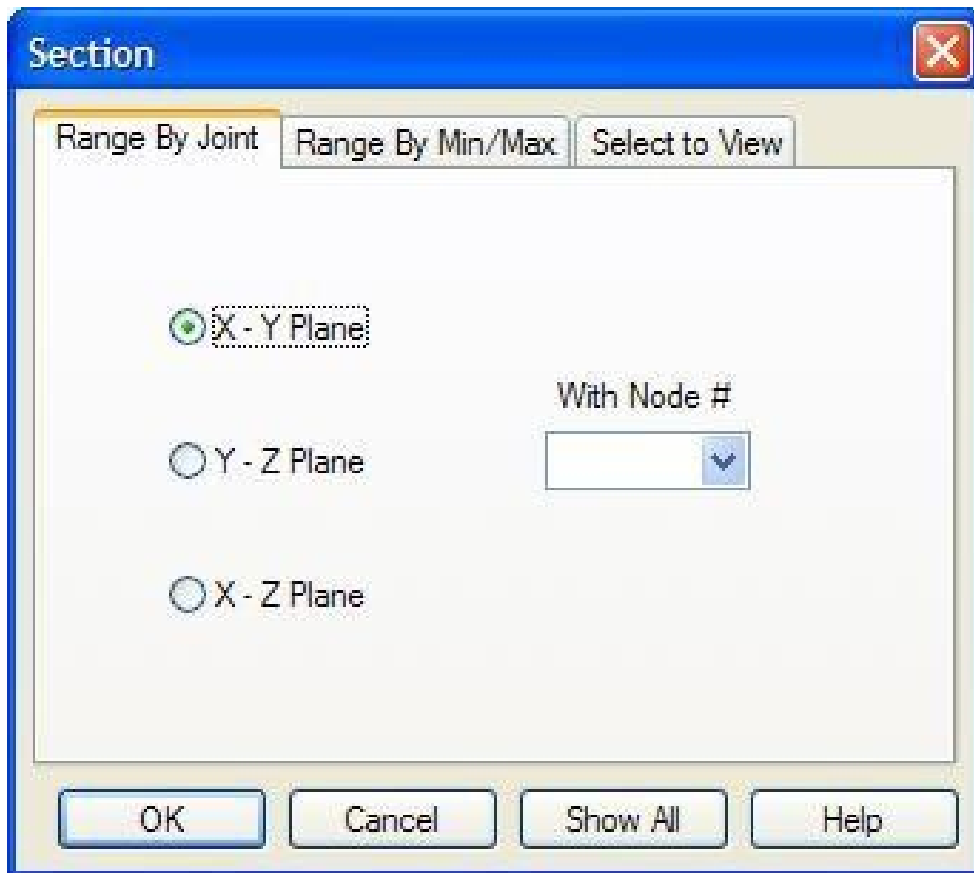
By The Joint Range
method

Since the beams level is average height containing meeting # 10 (or any of the joints 7-12), it would be desirable if we can tell the program to display the entities in the XZ plane and going through one of those joints 7 to 12, Facilities of

record *range of joint* allow us to do.

4. We select that tab (it is the default). Here, you can select the plane of the section by clicking on one of the options - XY, YZ, or XZ. We have to choose the **X-Z plane** for our example. Then in the drop down list *Node #*,

provide a joint that is located in the sectional plane. In our example, we can choose any of the nodes between 7 and 12 We choose **With Node # 10**. Click **OK**.

The following figure shows the cut part of the original structure shown in the drawing area.

Figure 3-17:
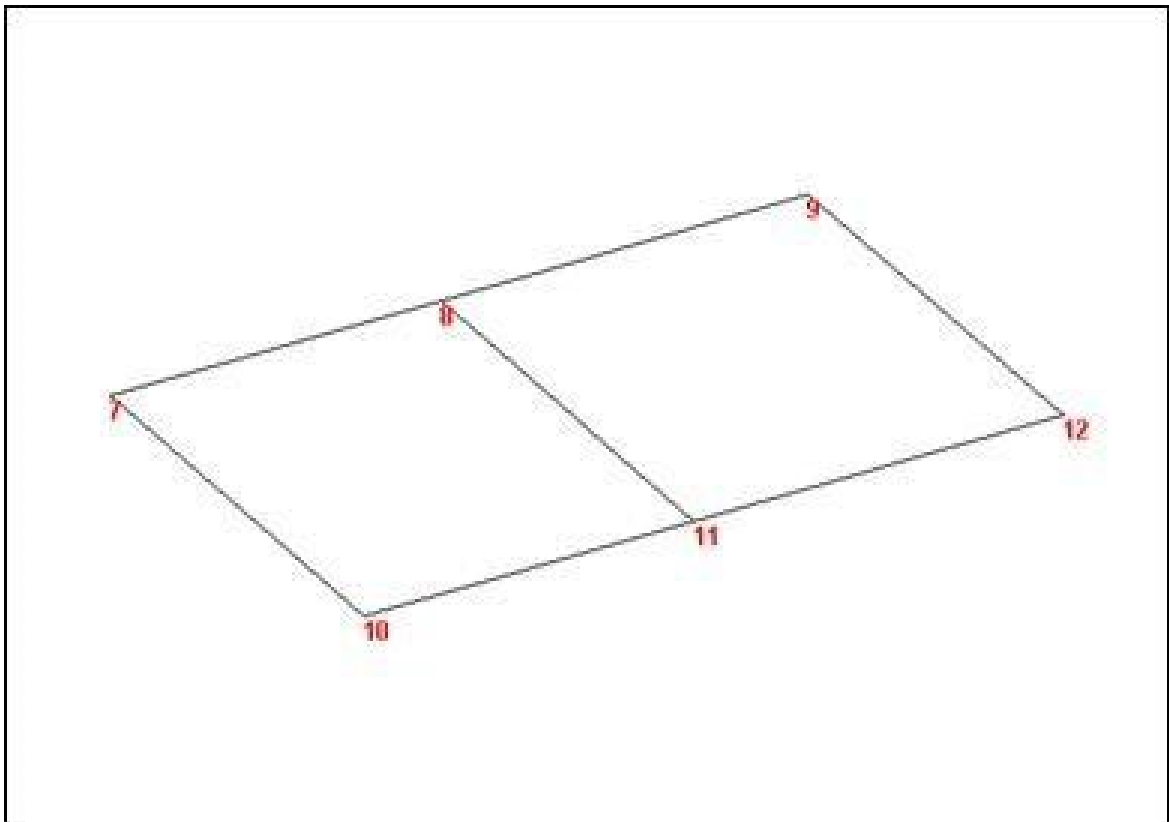
5. Select the tool **Display Whole Structure** to restore the original view.

Moreover, the dialog box shown in step 4 and click **Show All**. The Range By Min / Max method

Procedure:

Another convenient way to choose the beams in the level of average height specify that the beams are in the XZ plane between Y values range from 12 ft - 12 ft over and + a, where "a" is an arbitrary value as say 2ft

6. To do this, we select the tab **Range By Min / Max**. Here again we can select the plane of the section by clicking on one of the options - XY, YZ, or XZ. We have to choose the plane **X-Z** for our example. Edit boxes *Minimum* and

   Maximum represent the limit of the distances along the axis perpendicular to the plane Sectional. All objects between these two distances are displayed. In our example, we provide **10** how come *Minimum* and**14** and the distance *Maximum*. (Before doing this, make sure the current entries units of length, appears in the bottom corner

   right of the window STAAD program are feet). Then click on **OK**.

Figure 3-18:
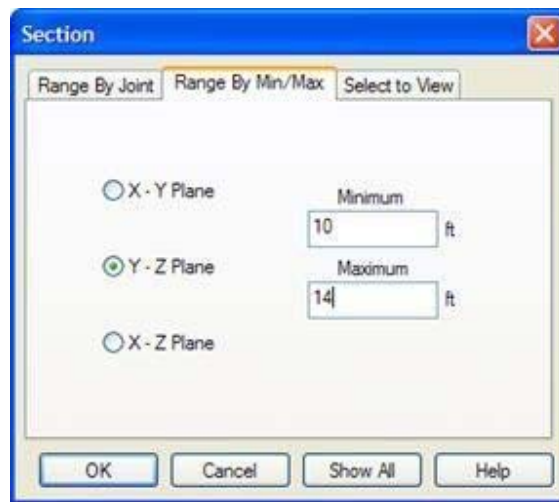
The following figure shows the cut part of the original structure shown in the drawing area.

Figure 3-19:
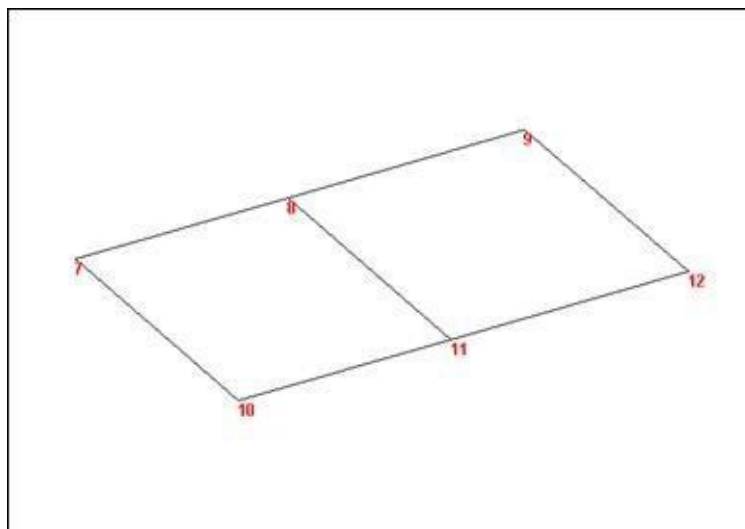
7. Select the tool **Display Whole Structure** to restore the original view.

The Select method to View

To demonstrate this method, instead of viewing specific beams, look at the nodes in place.

Steps:

Using this option, you can select the portion of the structure that we wish to specify the objects that are in that part. Option *Window / Rubber Band* allows to select the part of the structure for specifying a window

rubber around it. Option *Highlighted View Only* only shows only the objects selected (highlighted) on the screen and the members and elements that we want to see must be selected before choosing this option. Option *Select To View* allows us to see

only beams, plates, or solid, depending on the check boxes that are activated.

8. In our example, let us choose **Select To View** and setting the checkbox **Node**. Then click on **OK** as shown below.

Figure 3-20:
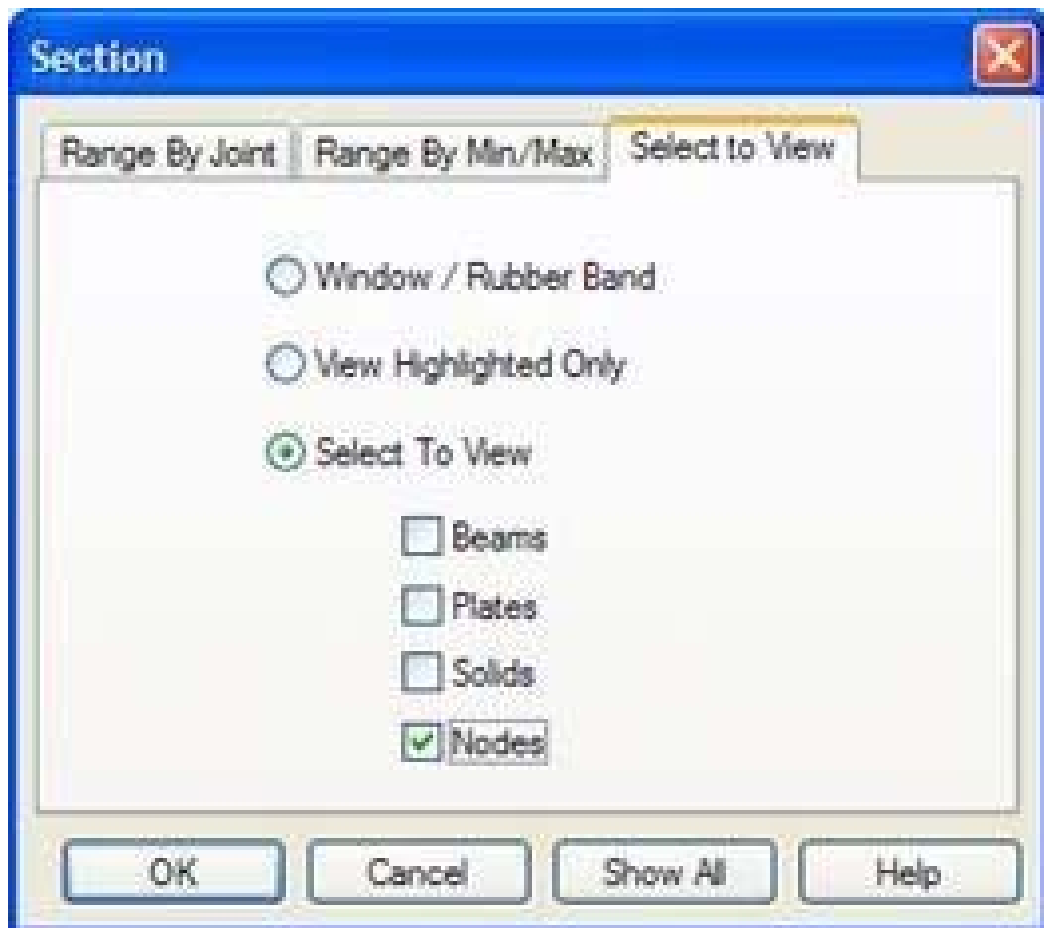
The following figure shows the view of the structure with only nodes that appears.

Figure 3-21:
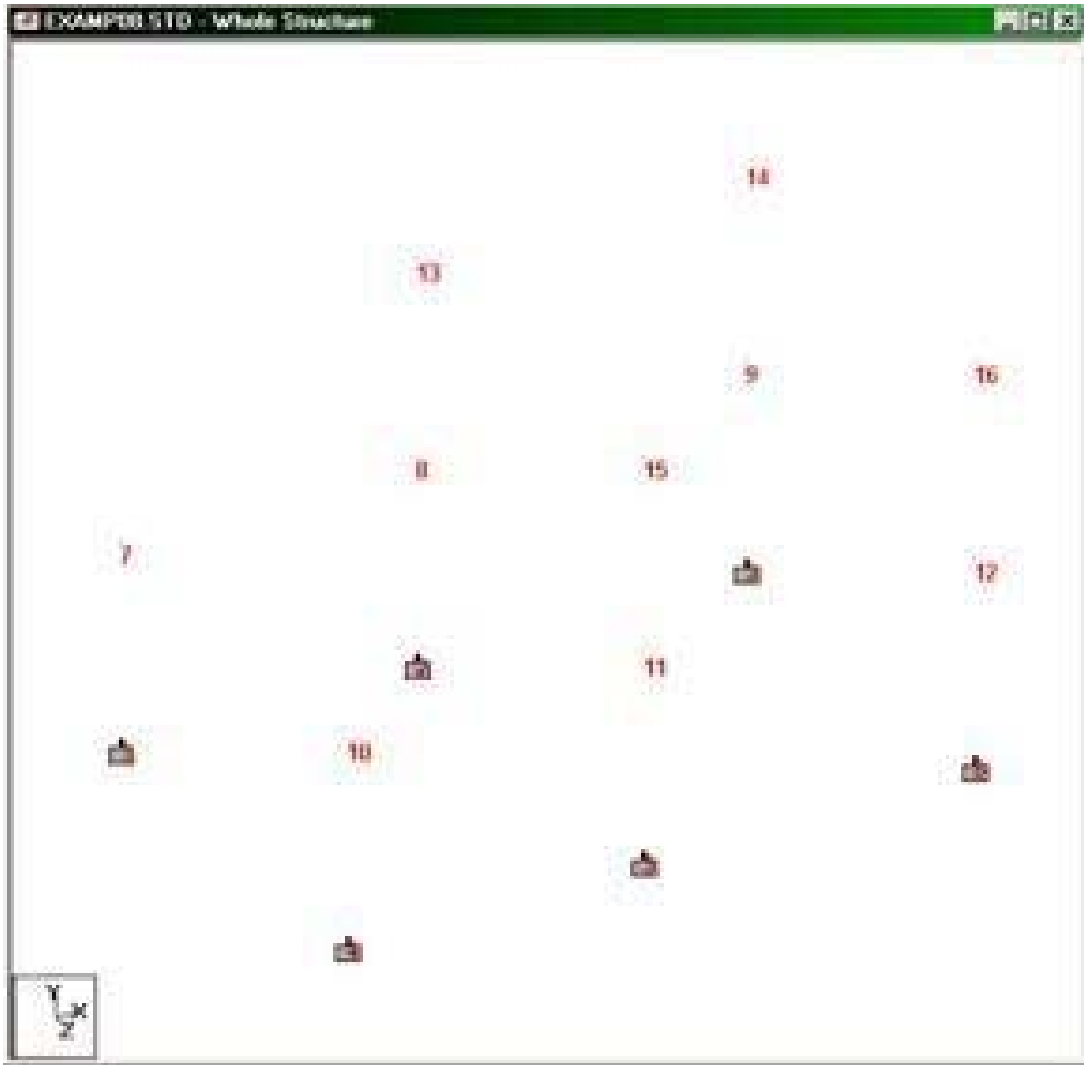
9. Select the tool **Display Whole Structure** to restore the original view.

Note: These new insights can be saved by going to the menu **View> View Management> Save View**. Provide a title for the new view. Later these saved views can be opened by going to the menu **View> Open View**.

Method 3 - View> Zoom

The menu option *Zoom* allows us to zoom (zoom in) or out (decrease) in one portion of the structure, as one would using a telescope.

1. To demonstrate this method, open EXAMP09.STD as shown in the following figure.

Figure 3-22:
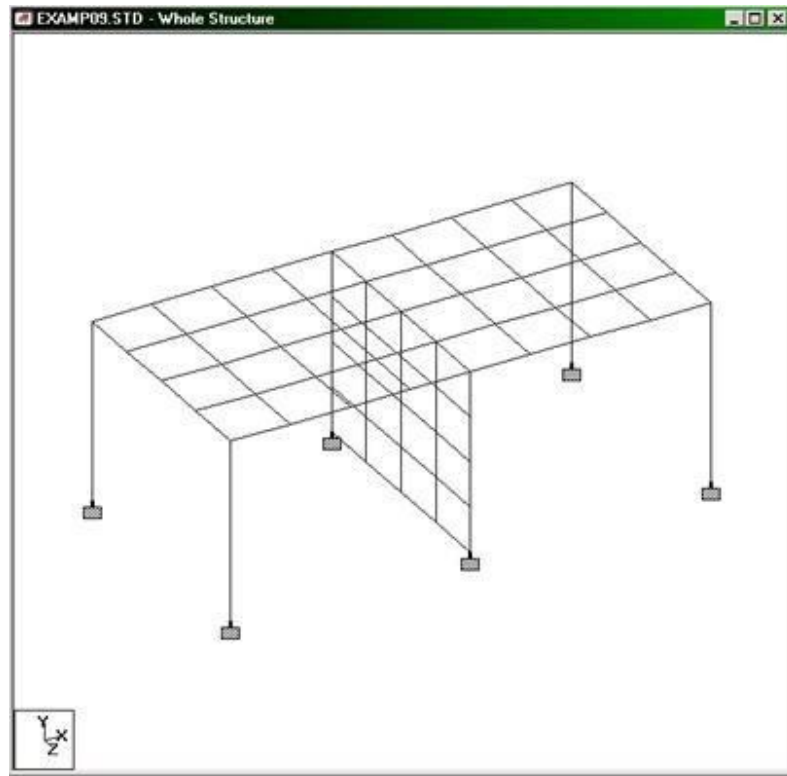
2. Say, we want to expand the portion of the structure shown in the figure below.

Figure 3-23:
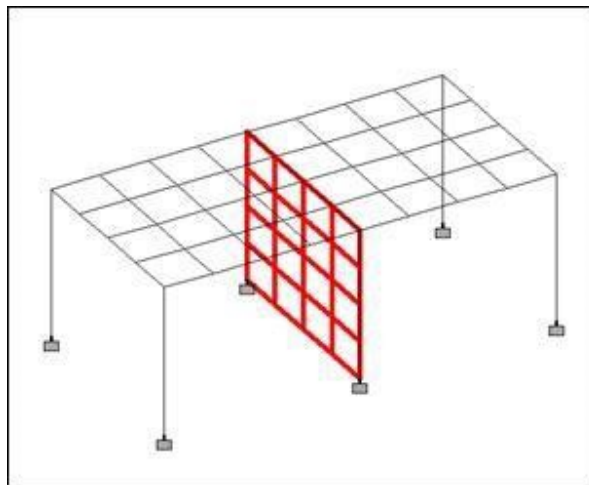
3. To select the plates as shown in the preceding figure by selecting the first tool **View From Z +**. Then select the tool **Plates Cursor**

    and drag a window around the elements of the plate in the view window as shown in the following figure.

Figure 3-24:
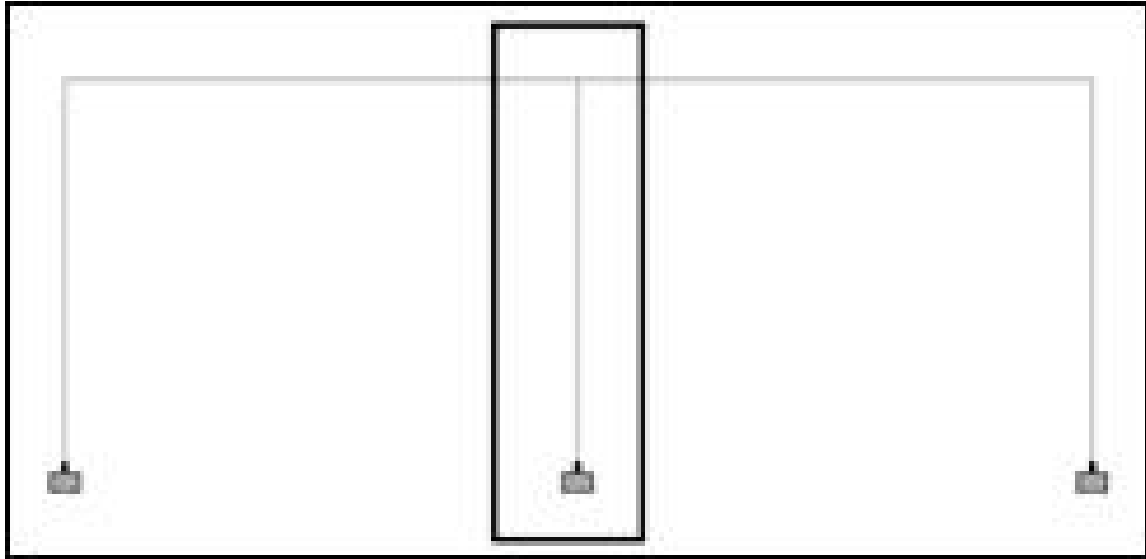
By the way, seeing the model *And from the view +*, Or, *from the view X* and using the methods of rubber window can be as effective in the selection of these elements.

4. Select the tool **Isometric View**.

    This is useful to ensure that the right entities are selected, as in the previous figure.

5. Expand the selected portion of the structure either by selecting the tool **Zoom Window**

Figure 3-25:

o

r

selecting **View> Zoom> Zoom Window**.

The mouse pointer changes to a crosshair in the view window.

6. Click and drag a window around the selected portion of the structure. magnified view of the selected structure.

Figure 3-26:
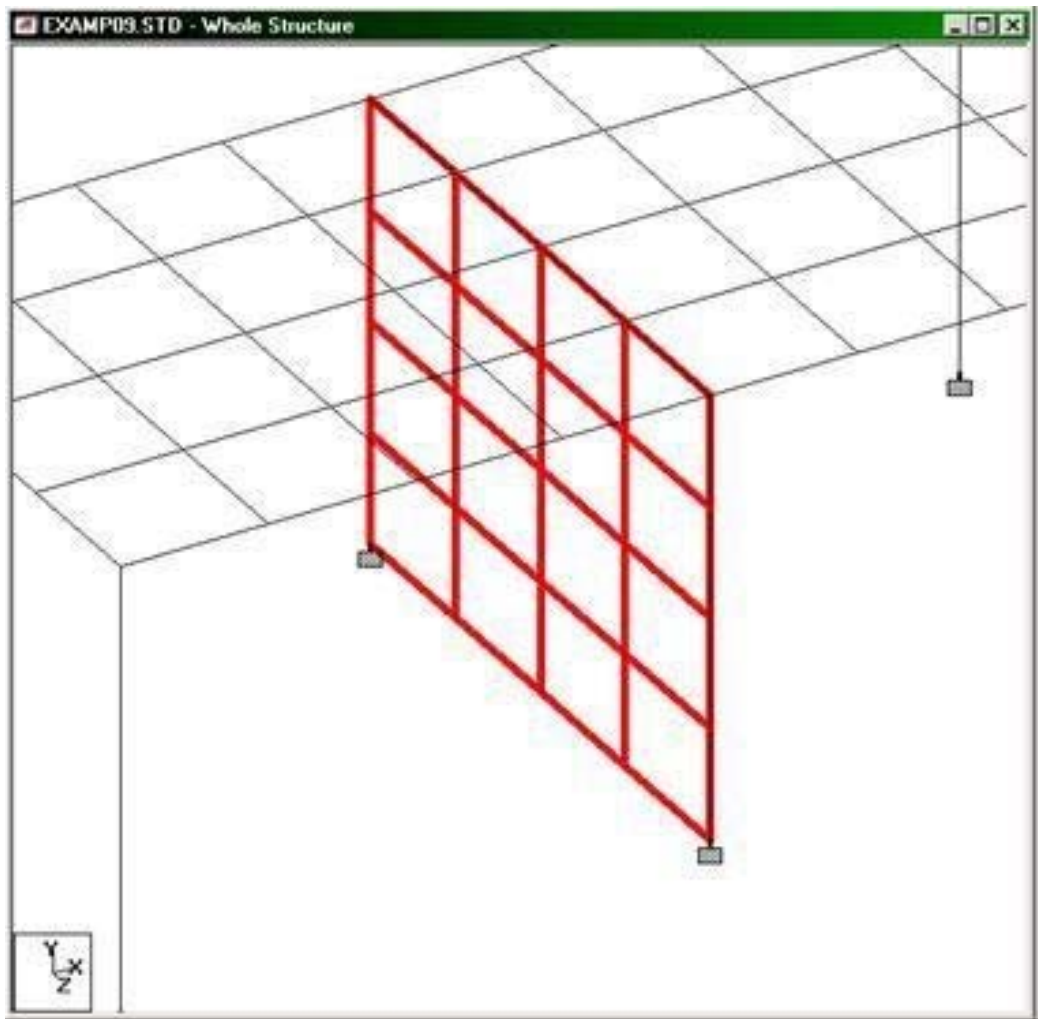
Alternatively, the tool is used **Zoom In** to expand the view by a predefined factor.

The tool **Dynamic Zoom** is used to show the enlarged view in a new window.

Figure 3-27:

Select a region by clicking and dragging in the view window, similar to the use of the tool *zoom in / out window*. That region will now be displayed in a new window in the drawing area as shown in the following figure.

Figure 3-28:
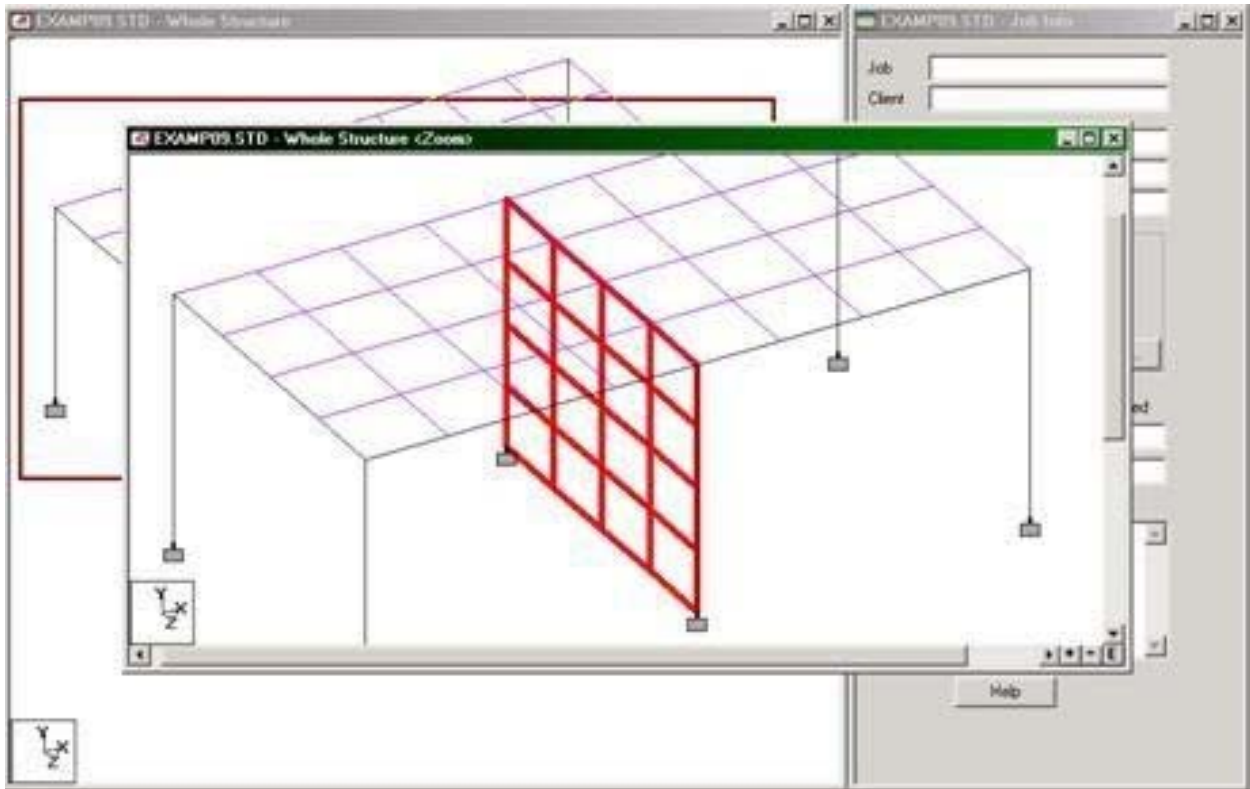
7. To restore the view of the entire structure, select the tool **Display Whole Structure**.

    Note: These new insights can be saved by going to the menu **View> View Management> Save View**. Provides a title for the new view. Later these saved views can be opened by going to the menu **View> Open View**.

5. Creating groups

Group names are a means to easily identify a collection of entities such as beams, plates or solids with a simple nickname. Grouping these entities, we must assign attributes as member properties and material constants for the group, a simple process compared to the task of assigning them to individual members.

1. To demonstrate this, open EXAMP01.STD (C: \ SPro2007 \ STAAD \ examp \ AU) as shown in the following figure.

Figure 3-29:
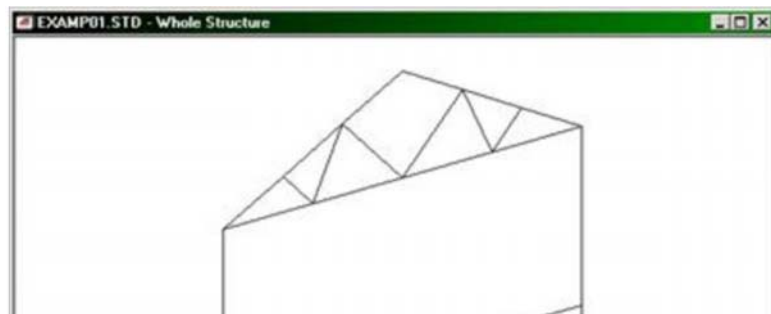
Consider the members forming the armor is in one of three groups: Chords Top, Bottom Chords and Web Members.

2. Select the tool **Beams Cursor** and select the inclined in the view window as shown in the figure below members.

Figure 3-30:
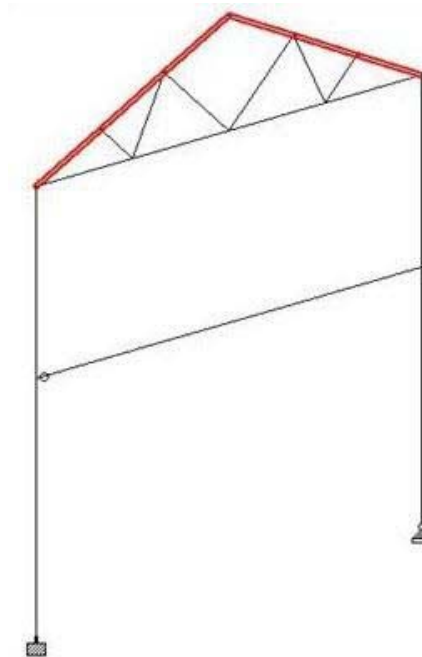
3. Select **Tools> Create New Group**. Opens the Define Name dialog group.

4. Enter **_TOPCOR** on behalf of the group.

> **Note**: Group names must begin with the character '_'.

Since group names can be assigned to nodes, beams, plates, solids, and a general category called geometry, it is very important to choose the appropriate type of entity. Therefore, specify **Select Type** how come **Beam** and then

click **OK**.

Figure 3-31:

Opens the Create Group dialog box.

5. Click on **Associate**.

The methods of allocating the dialog *Create group* currently activated *Associate Selected to Geometry*.

Figure 3-32:

6. Select the members forming the chord of the bottom as shown in the following figure.

Figure 3-33:
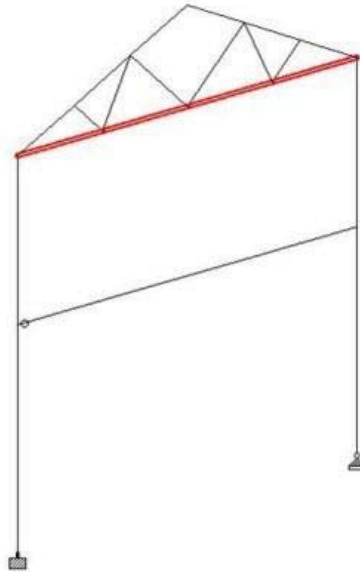
7. In the Create Group dialog box, click **Create**. Follow steps 4 and 5 and assign the group name **_BOTCOR** these members.

8. Now, you can group network members in the armor. Using the mouse, select members of the transverse reinforcement as shown in the following figure.

Figure 3-34:
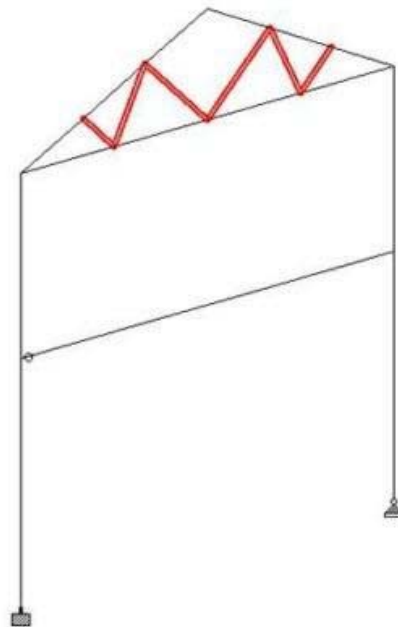
9. Then follow step 7 except to specify the group name as **_TRNTRUS**.

Figure 3-35:
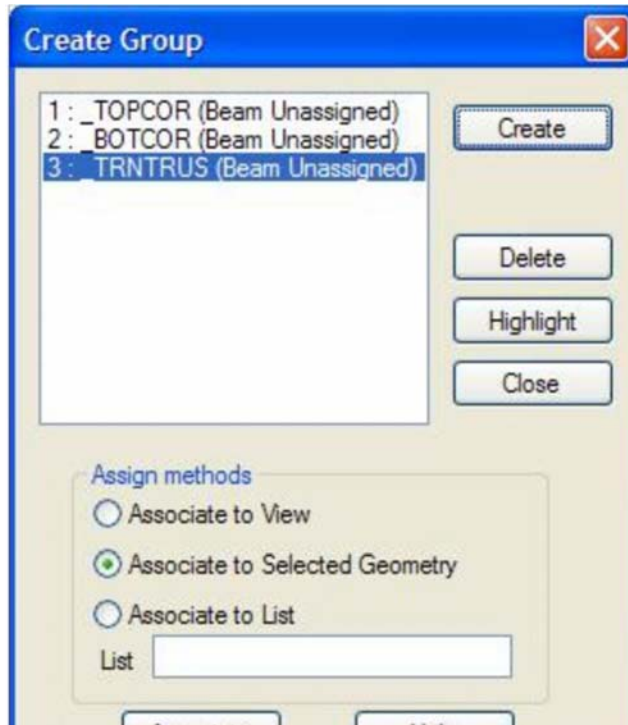
10. Click **Close** in the dialog box *Create group*.

Now they have created three groups. Select **Select> By Group Name** to open the Select Groups dialog box, which can be used to select a group call.

Figure 3-36:

Note: If you disabled the option to display the label, labels such as beam numbers will not show these specific members, even if the icon is lit beams numbering for the entire structure.

6. Showing screen loads

1. To demonstrate this feature, open EXAMP08.STD (C: \ SPro2007 \ STAAD \ examp \ AU) as shown in the following figure.

Figure 3-37:
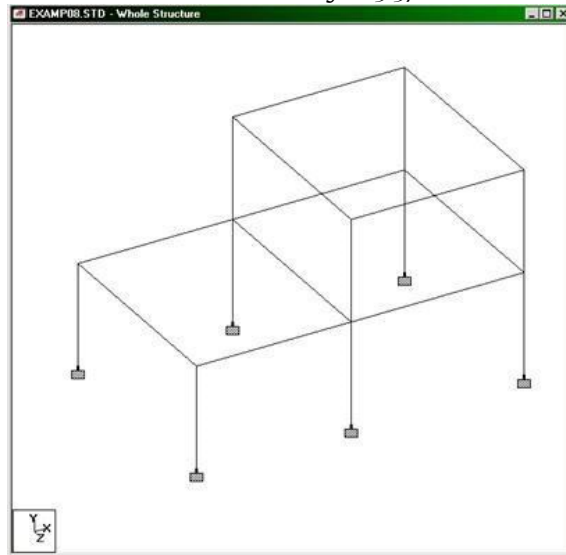
2. Select the tool **Loads** and then a specific load as **Active Load** the drop-down list of all loads in the input file.

Figure 3-38:

o
r
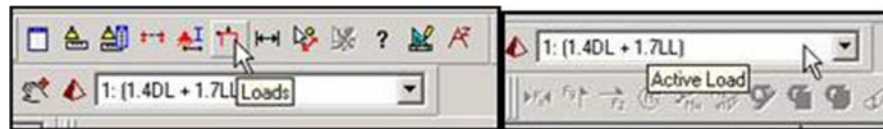
Select **View> Structure Diagrams**. In the dialog box *diagrams* that opens, select the tab *Loads and Results*. select the option *Loads* and select if you want to load from the list box in the cargo box. Then do
click **OK**.

Figure 3-39:
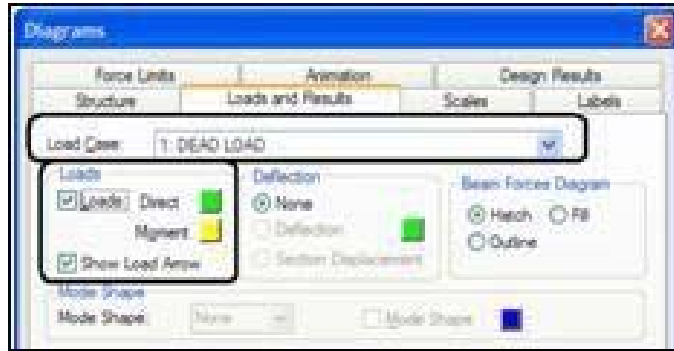
The following figure shows the load case 1 for this example model.

Figure 3-40:
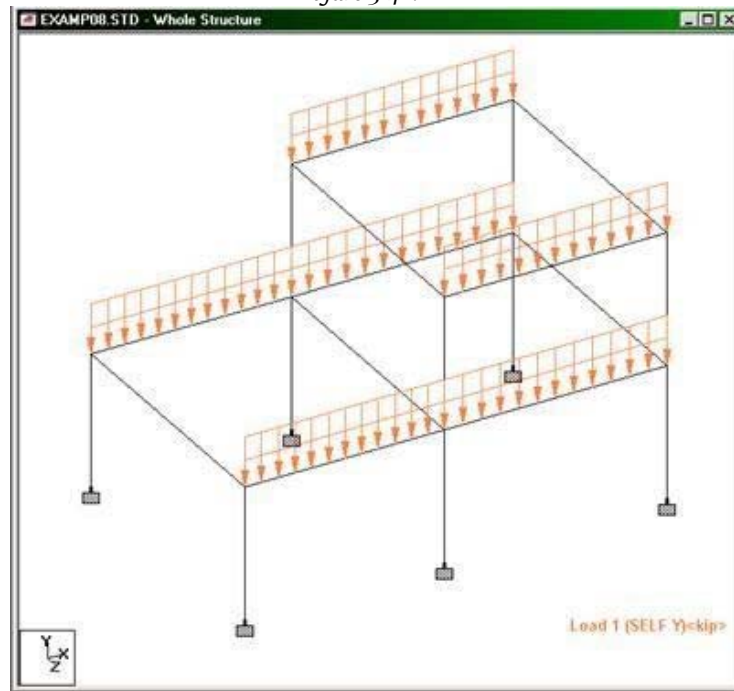
7. Showing the load values on the screen

1. In the previous exercise, we saw the method to turn the icon loads, but the numerical value of loads. To see load values, open EXAMP08.STD as shown in

the figure below.

Figure 3-41:
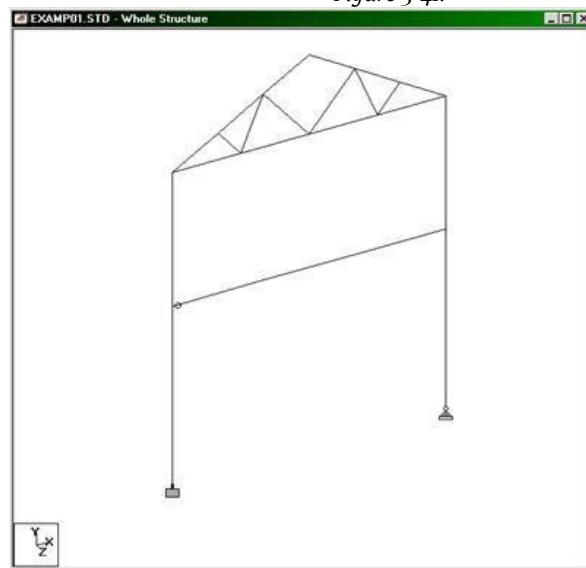
2. Select the tool **Loads** and then right-click anywhere in the view window and select **Labels** the popup menu.

    or

    select the tool **Symbols and Labels**

    or

    select **View> Structure Diagrams** and select the tab *Labels* in the dialog box diagrams.

    The dialog box opens diagrams Record *Labels*.

3. Set the check box **Load Values** category *Loading Display Options* as shown in the following figure.

Figure 3-42:
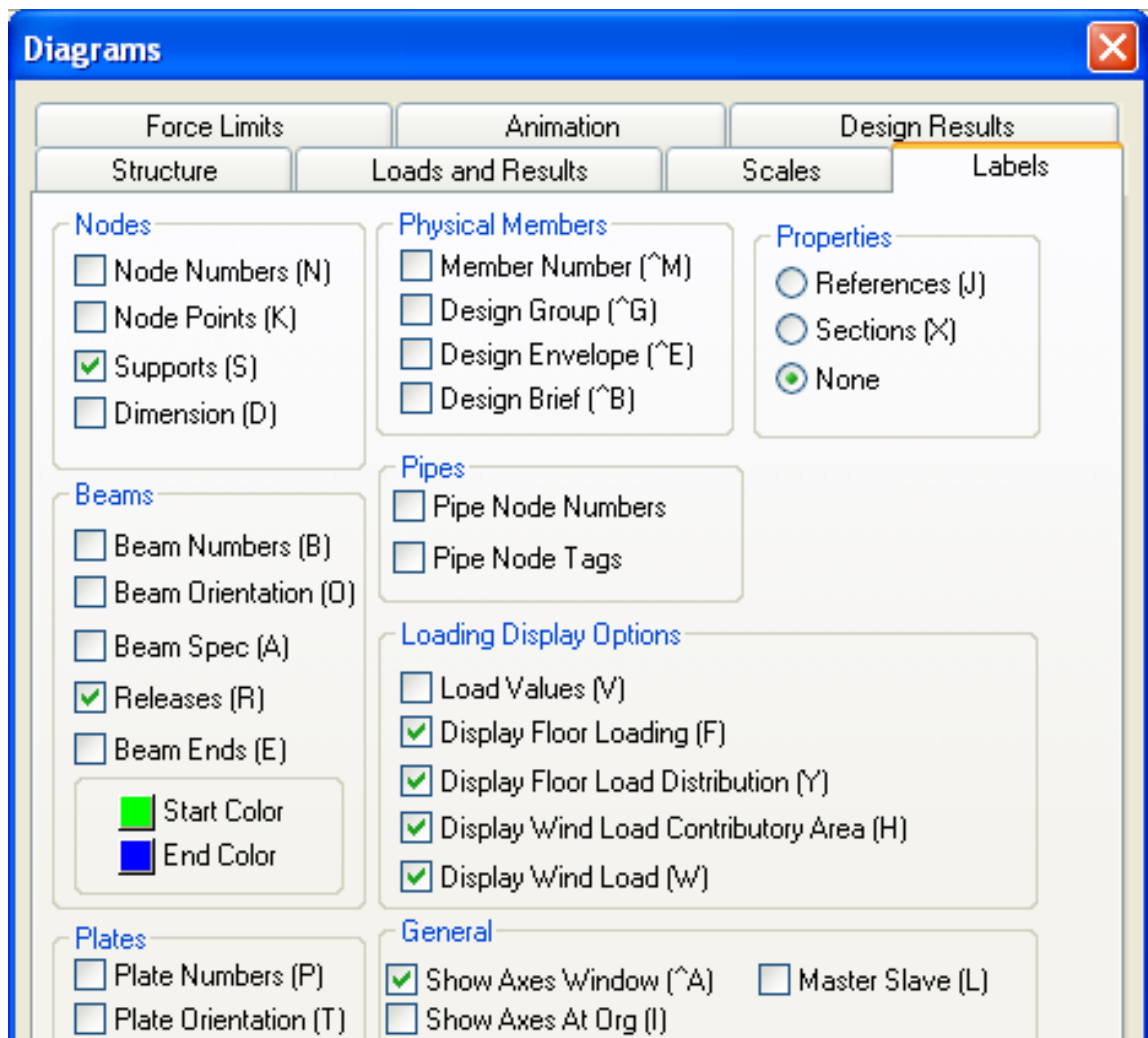
The following figure shows the example structure with the load values listed in it.

Figure 3-43:
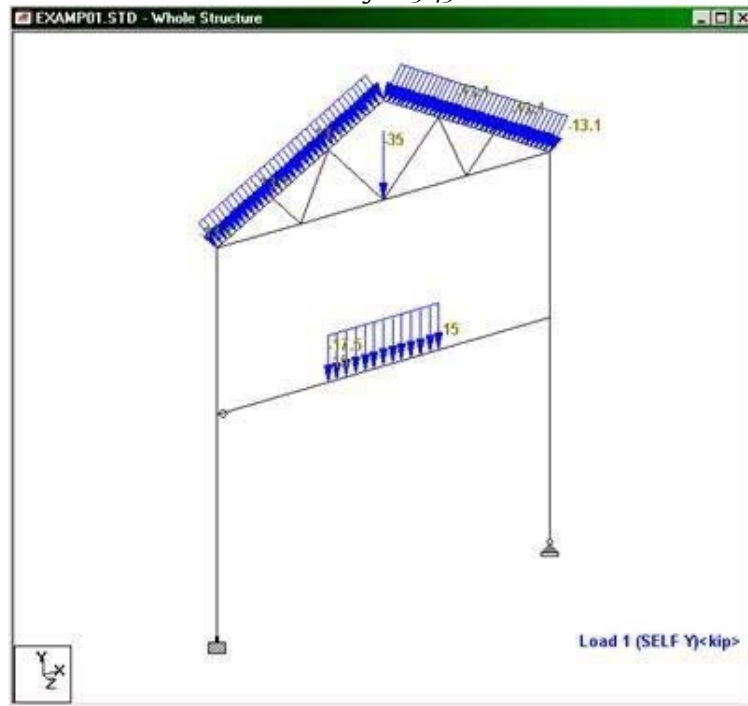
4. Select **Tools> Set Current Display Unit**.

The Options dialog box opens.

Select the tab **Force Units** and then change the selection under force to **Kip** and *distributed force* to**kN / m**.

Figure 3-44:
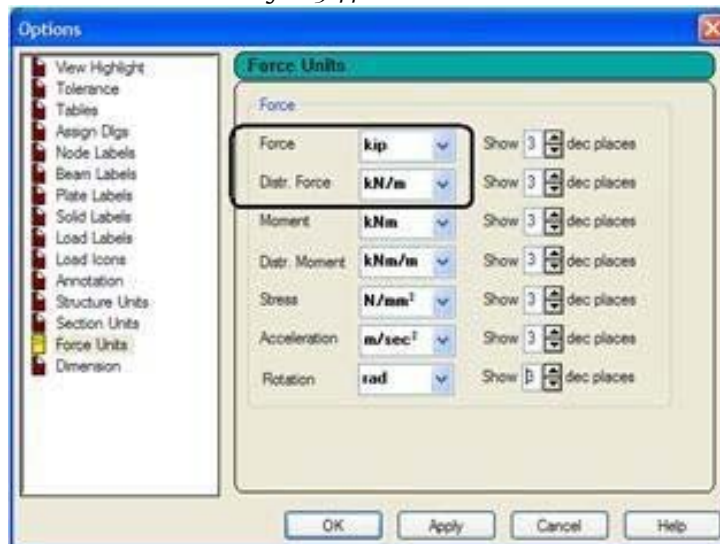
5. Click on **OK**.

The values are updated with the new units.

8. Structural tool options advice

Figure 3-45:
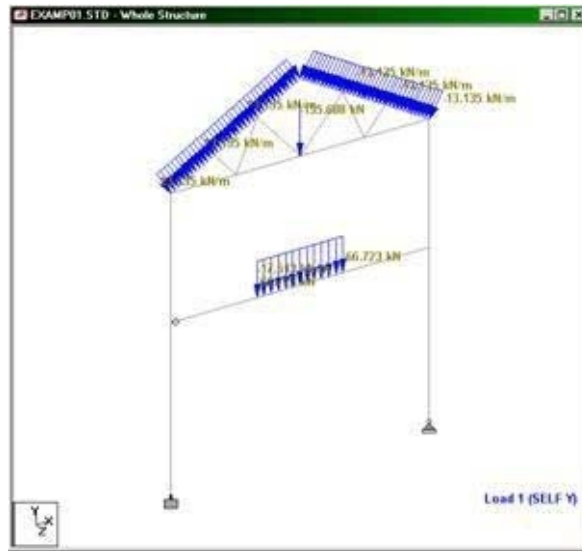
Tips structural tool provides a facility to display any custom input custom or output information about a node, beams, plate or solid element when the mouse cursor over the structural entity is placed. The tool tip is similar to the one shown when the mouse cursor over an icon in the toolbar moves. When the mouse cursor is moved away from the site, the information displayed is also off.

1.Select **View> Structural Tool Tip Options**.

Opens the Options dialog tool tip.

Figure 3-46:
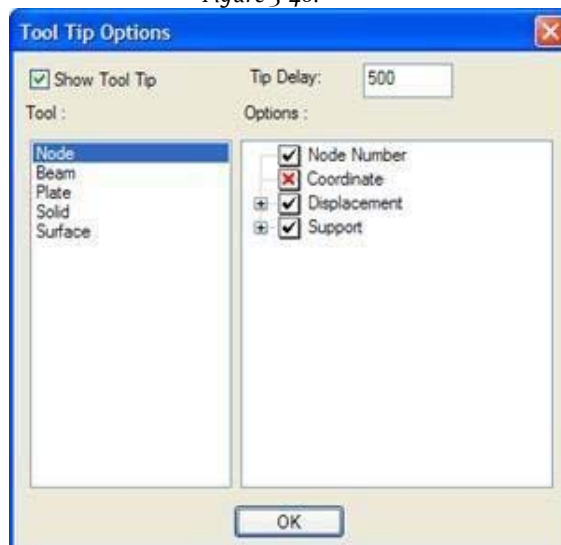
2. Select the option **Show Tool Tip** is to ensure that the tooltip can be alternated on.

Box *Tip Delay* means the amount of time it takes from when the mouse cursor over an entity appears first when actually the tooltip appears. This number is expressed in milliseconds (that is, 1000 = 1 second).

The options (items that can be displayed) for each entity are shown in the table *options*. A check mark indicates that the particular data item is displayed in the tool tip. An option with a "+" next to it means that other options can be enabled or disabled. A red "X" indicates that data are not displayed in the tool tip. Just click the check box to enable or disable the option.

Tips structural tool can be configured to display a wide variety of information about an entity model, as shown in the following figure.

Figure 3-47:

#4 (20.000,0.000,20.000)
Displacement for Load Case #1 ...
  Y = -0.000 in Z = 0.000 in rY = 0.000 rad
  rZ = 0.000 rad
Node Reaction For Load Case #1 ...
  FX = 0.000 kip FY = 234.917 kip FZ = -75.287 kip
  MX = 467.881 kip-in MY = -0.000 kip-in MZ = -0.000 kip-in
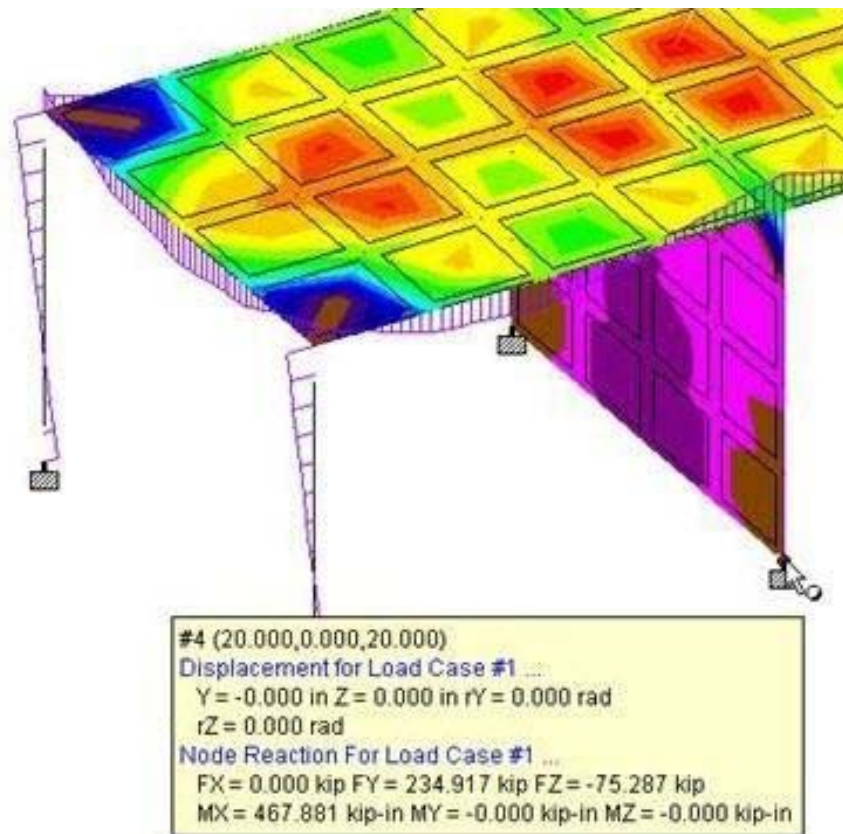
Note: The tool automatically shows results for the case of active load. All values are reported in the current display units.

9. Identifying start and end of the beam

When assigning attributes as member freed or compensated member, an additional element of information that accompanies this information is if this attribute is applied to the "Start" node of a member or the "end" node. Therefore, there must be a way to quickly identify these two points without doubt one another.

1. To demonstrate this facility, open EXAMP01.STD as shown in the following figure.

Figure 3-48:
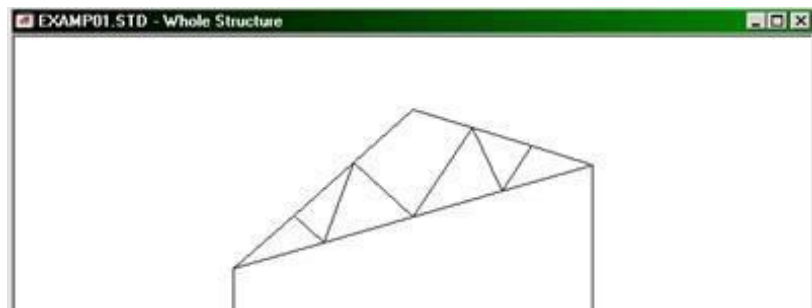
2. There are two ways to identify the start or end of a beam:

   a. Place the mouse pointer over a specific beam. The resulting tool tips show the start node and end node of the beam in two different colors along with their corresponding coordinates. Of course, you must have screen tool tips enabled for this method to work.

Figure 3-49:
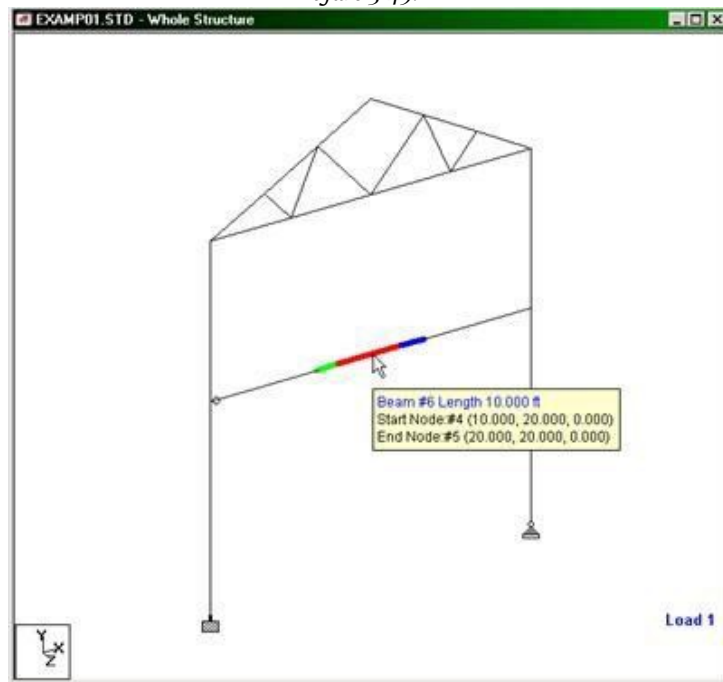
b.   Right–click anywhere in the view window and select **Labels** the popup menu reset.

The Diagrams dialog opens in the Labels tab.

Set the check box **Beam Ends**. To change the color at the beginning or end is displayed, simply click on the color palette with options *Initial Color* or*Final color* and choose the color you want. Then click on

Apply to update the changes. Click **OK** to close the dialog box.

Figure 3-50:
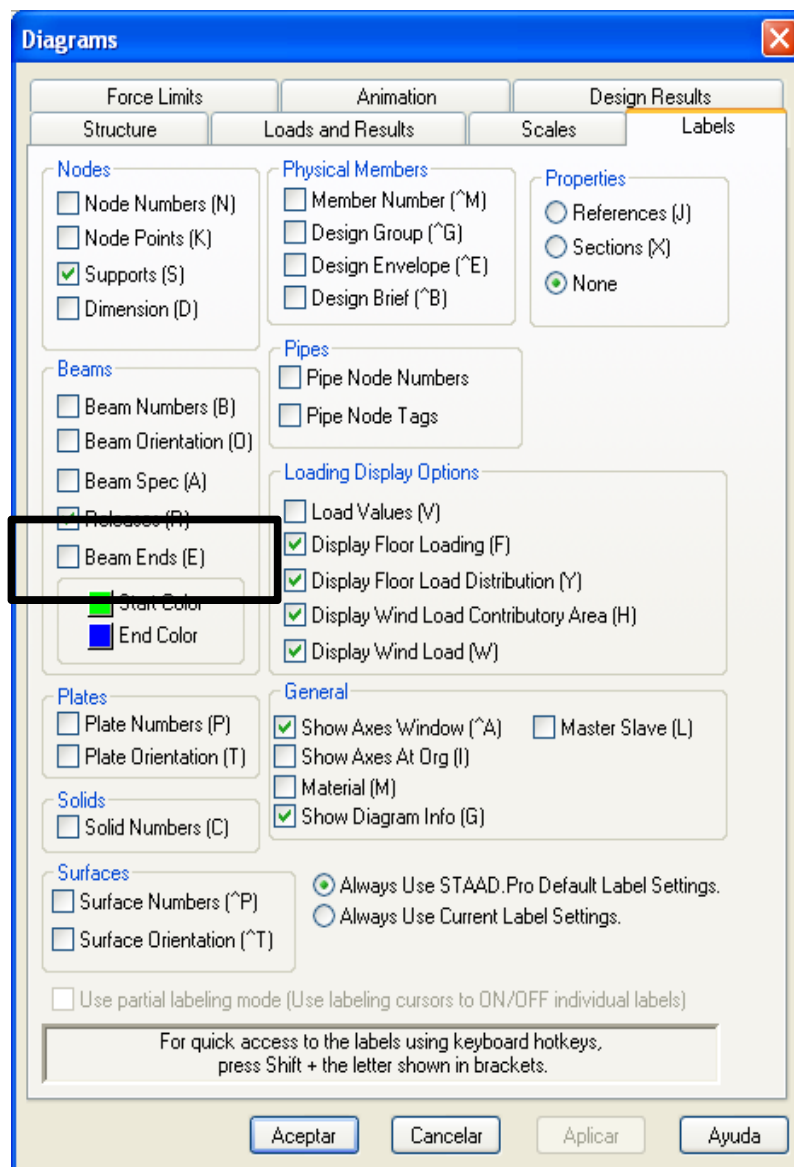
The following figure shows the structure with the beginning and end of all beams shown in two separate colors.

10. Charting from STAAD.Pro

Figure 3-51:
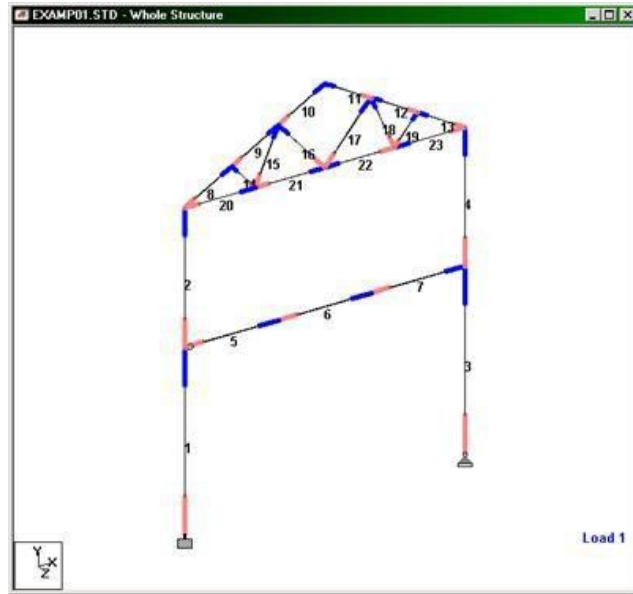
Explanation The following are five methods to trace the STAAD model plane and STAAD result diagrams.

Method 1: Using the current option to print view

1. To demonstrate this facility, open EXAMP14.STD (C: \ SPro2007 \ STAAD \ examp \ AU) as shown in the figure below.

Figure 3-52:
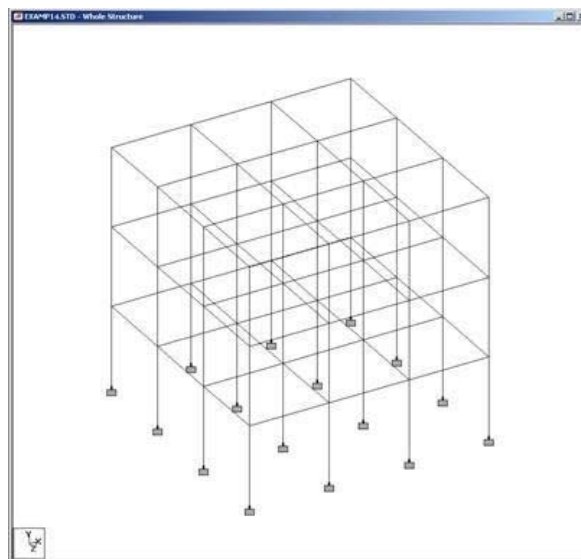
2. Select the tool **Preview Print Current View** to preview the image as it will appear on paper.

Figure 3-53:

Opens the print preview to show the view in a report format.

Figure 3-54:
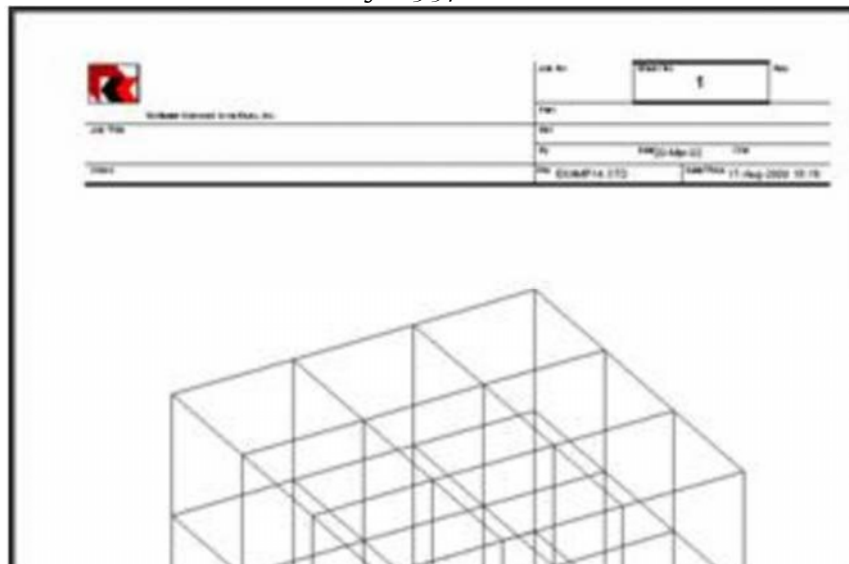
3. Select the tool **Print Current View** to draw the image.

Figure 3-55:

The Print dialog box Windows standard, which is used to select the printer or plotter you want to send the image opens.

4. Select the desired printer and click **OK**.

Figure 3-56:
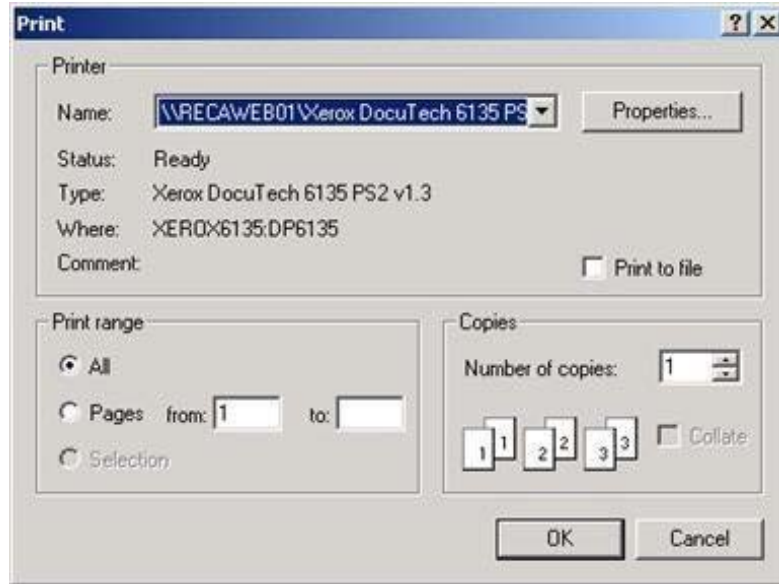
Method 2: Using the option of taking photo

1. Select the tool **Take Picture**.

○**Edit> Take Picture**. Opens the Picture # dialogue.

2. Enter a picture ID and subtitles.

Figure 3-57:
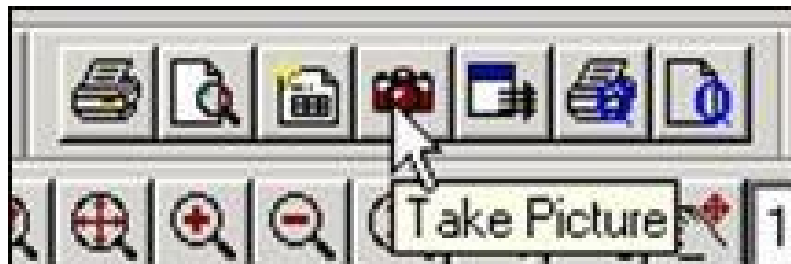
Figure 3-58:
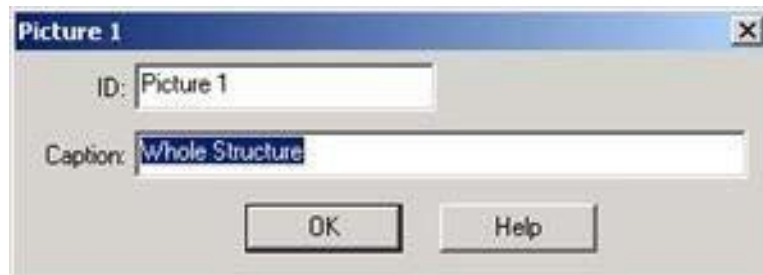
3. Select **File> Report Setup ...** Select the tool or **Report Setup**.

Figure 3-59:

Opens the Report Settings dialog.

4. On the record **Items** Select **Pictures** in the list of available items.

Figure 3-60:
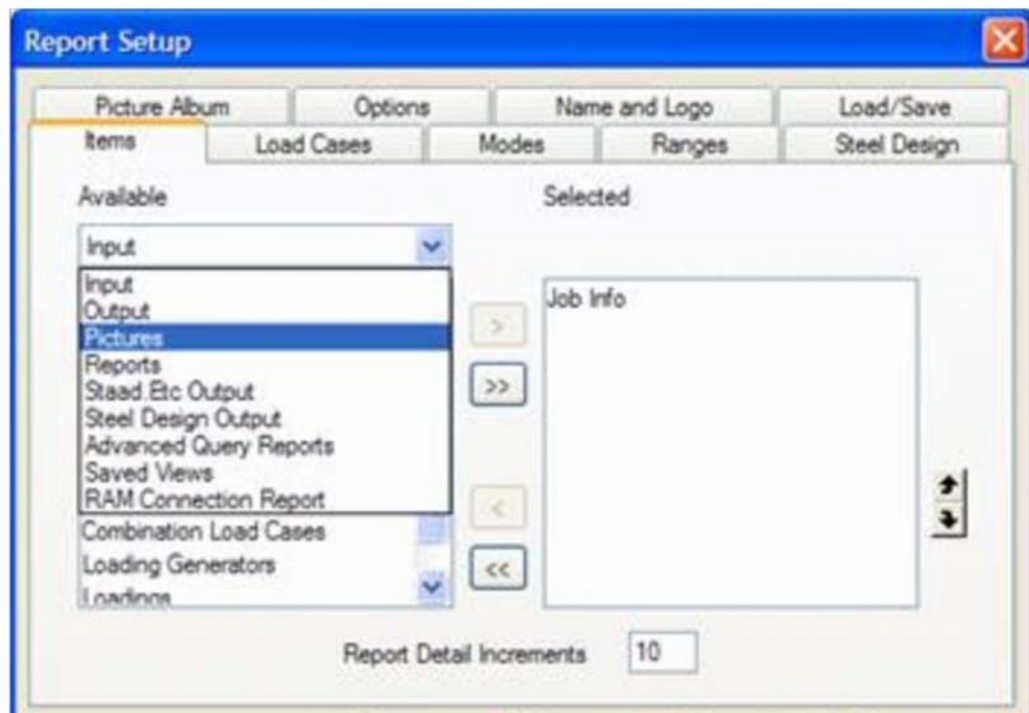
If there is more than one image, select the desired trace by pressing [>].

5. Select the tab **Picture Album** and setting the checkbox **Full Page**. Click **OK**.

Figure 3-61:
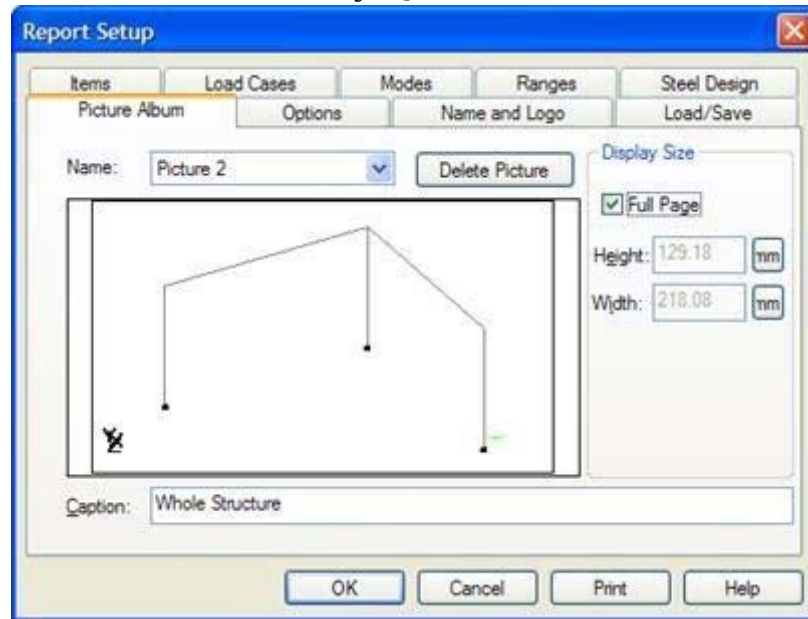
6. Select **File> Print Report**.

The outline will be drawn.

Alternatively, select **File> Export Report> MS Word File** to create a report you can print from Microsoft Word.

In the dialog *Save as* Specify a name for the file and click **Save**. In the dialog box *Templates*, Select the *Normal template*.

Figure 3-62:
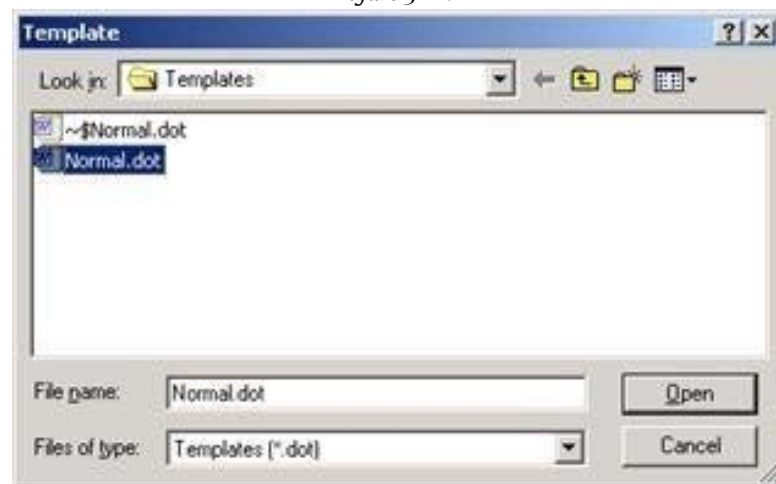
Microsoft Word file creation begins with the image on it.   Once this is complete, the new file opens in Microsoft Word.

Method 3: Using the view option to export

STAAD.Pro has a command to export a drawing to a graphic image file.

1. With the diagram you want to plot is displayed, select the tool **Export View** in the toolbar print

Figure 3-63:

Opens the Save As dialog.

2. Select the chart format in which to save the file (ie, Bitmap, JPEG, TIFF, GIF, etc.)

Figure 3-64:
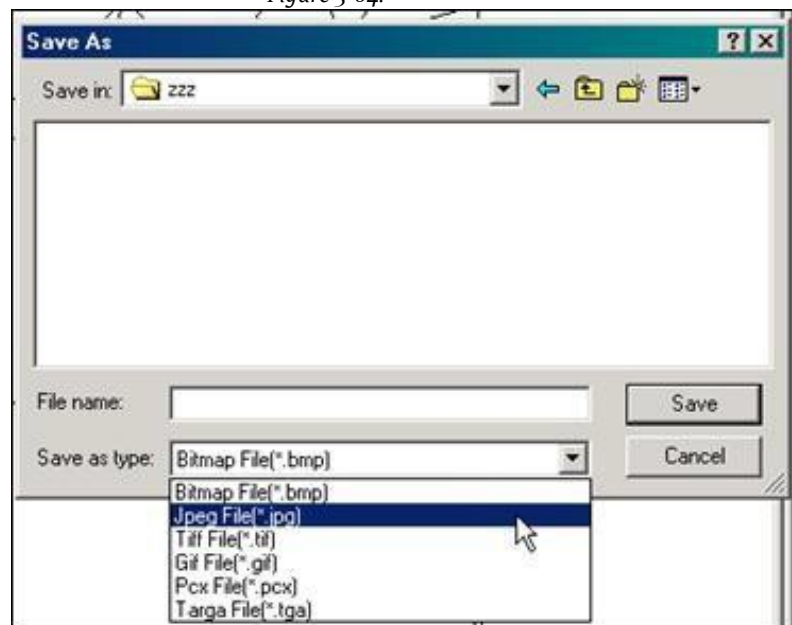
Provide a file name and save the file.

3. Open an image editing program text or graphics processing (eg, Microsoft Word or Adobe Photoshop).

4. Import or insert the drawing from the previously saved file. Now you can print the image file from the graphics program.

Method 4: Using the Copy Image

1. Stresses the window containing the diagram you want to plot. This can be done by ensuring that the title bar of that window is the color that indicates that it is in focus.

2. Select **Edit> Copy Picture**.

3. Open a graphics program able to trim the graphic image to their needs (eg, Microsoft Paint, Corel PaintShop Pro, Adobe Photoshop, etc.).

    Paste the clipboard contents into an empty file in this program. You will notice that the drawing window now STAAD sticks in that program. Using the tools provided by the graphics program, you can modify the design if you wish. Now you can print the image file from the graphics program.

Method 5: Using the Windows clipboard

1. When the diagram you want to plot on the screen is displayed, press **PRINT SCREEN or PRINT SCREEN** (For some keyboards, this functionality requires pressing **SHIFT or SHIFT + PRINT SCREEN**+**PRINT SCREEN**). Currently in view the entire image is copied to the Windows clipboard.

2. Open a graphics program able to trim the graphic image to their needs (eg, Microsoft Paint, Corel PaintShop Pro, Adobe Photoshop, etc.). Paste the clipboard contents into an empty file in this program. You will notice that all STAAD screen (including the diagram, menus, icons toolbar, tables, etc.) is part of the pasted image.

3. Using the tools provided by the graphics program, crop the image in order to retain only the drawing and remove unwanted parts. Now you can print the image file from the graphics program.

Section 4
Practical Exercises

4.1 Geometric Generating a seven-story building

The "New" popup window appears where we enter:

Workspace:
- o   Space - Design in space (x, y, z).
- o   Plane - Design in a plane (x, y) or porch.
- o   Floor - Design in a plane (x, z) or floor.
- o   Truss - Designed Armor.

File Name (File Name).

Location (location where the file is saved.) Units of Length and Force Units. (Meter and kilogram.)

Select Space, we entered the relevant data and we click Next.

Will run into the "Where do you want to go?" (Where to go?) Window with the options:

- o   Add Beam - Add Beams.
- o   Add Plate - Add Plates or Slabs
- o   Add Solid - Add Solid
- o   Open Structure Wizard - Open Standar library of structures
- o   Open STAAD Editor - No graphical interface commands STAAD
- o   Edit Job Information - Edit job information.

We will generate different types of geometries, like a number of structural frameworks that make a building of 7 levels, a building of circular cross section and a dome.

For the 7-story building, will draw originally a 5x5 frame with open base, for that select the "Snap Node / Beam" tool and begin to draw from the point of origin (0, 0, 0). Essentially this represents each of our homes. The general idea is to create a society governed by frames and modulated structure under these measures.

We will select the "AddBeams" option and we will work with beams or members and give you the option
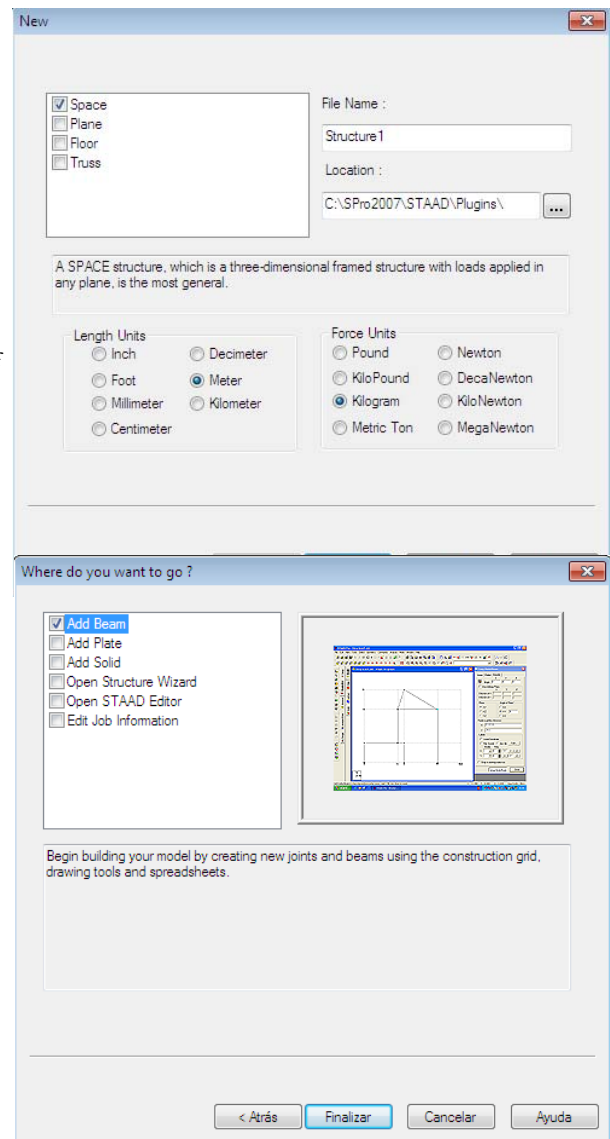
end.

Workspace where we start to handle the geometry of our project, a building opens.

It is important to consider using "Snap Node / Beam" (Network Node / Beam) as this creates for us a grid where we can enter our data, activate the icon This Snap Node / Beam and select the "Geometry" option the left side tab
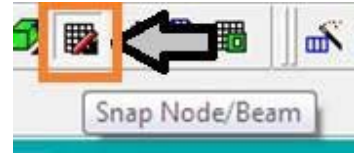
This Grid can be edited to increase the number of spaces in "x" or "y"
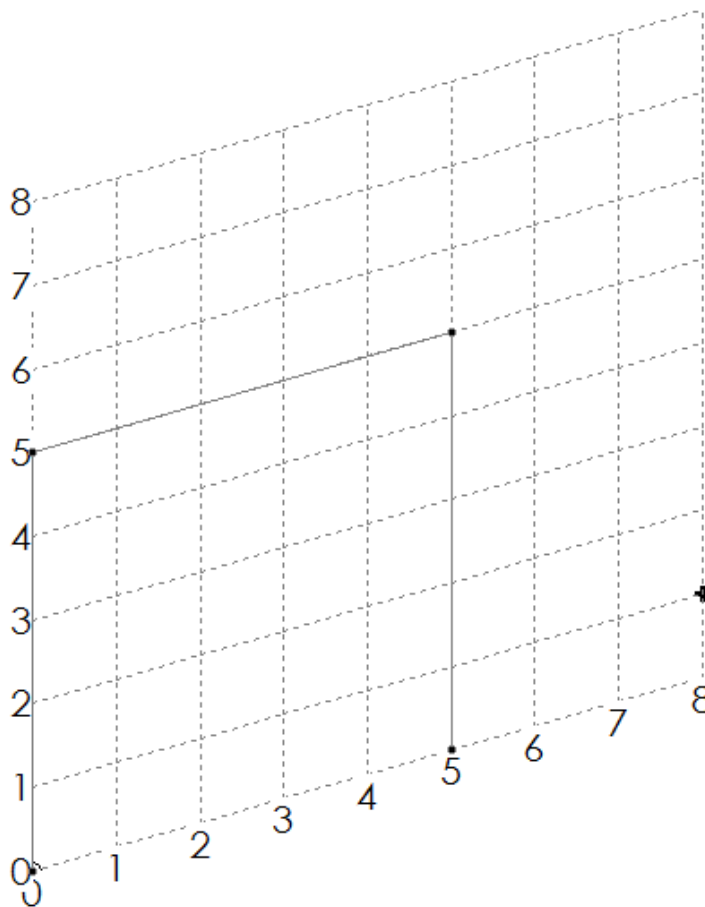
Select it and Translational Repeat the tool we will create a number of copies of our element, we start by creating frames one side of our modulation.
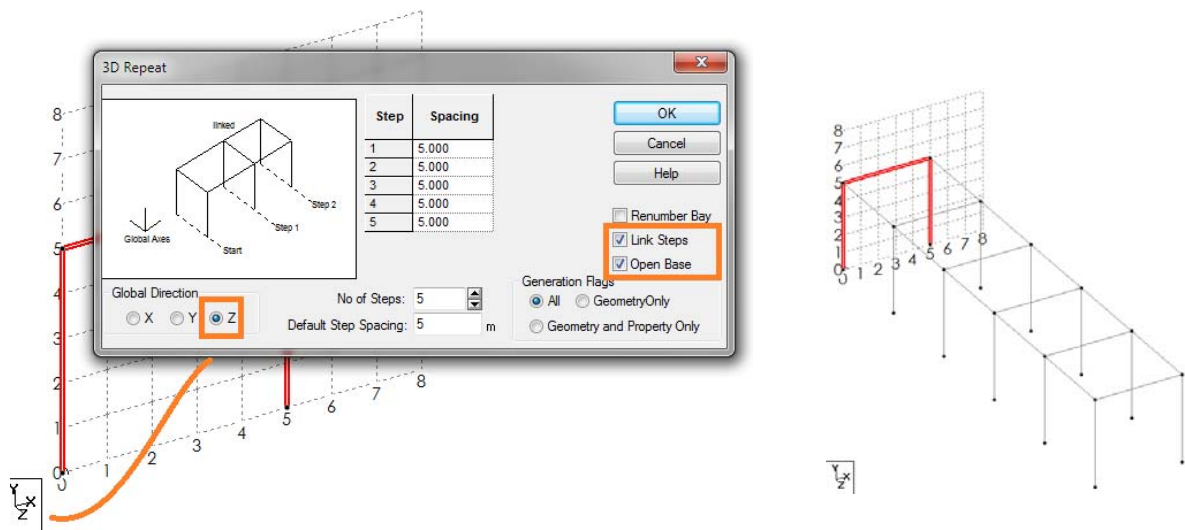
We do the same in the other direction, but with all the elements to create cloned frames downstairs of our project, it is important to always point where we want to deploy our copies, in this case on the "x" axis.
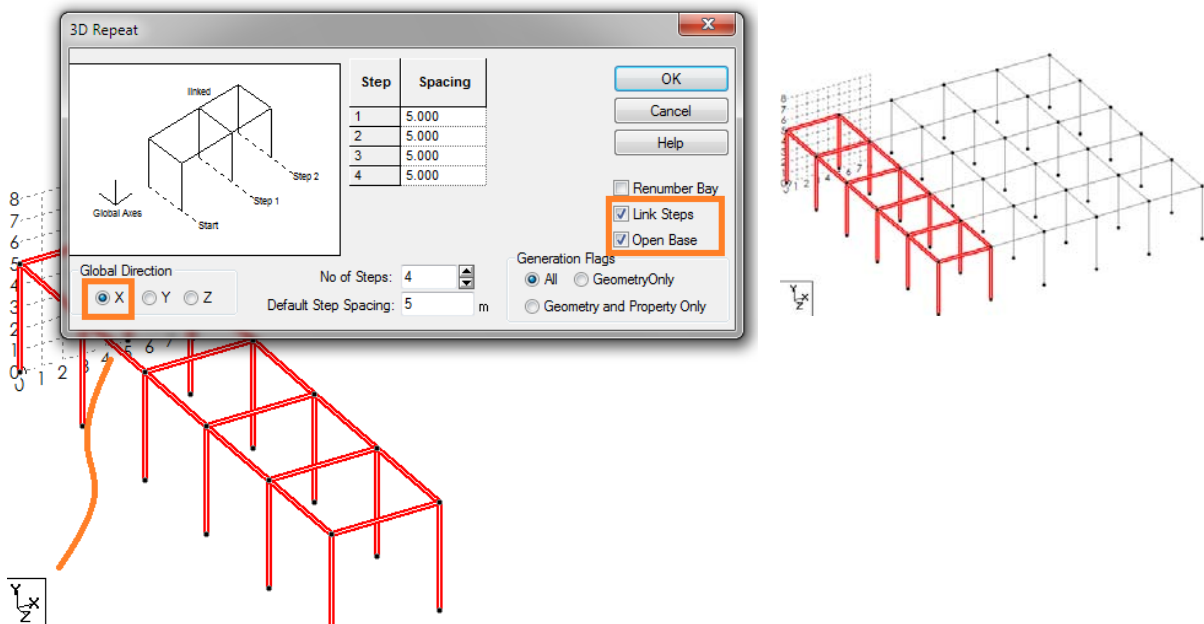
Finally do the same but now on the "Y" in order to give height to our building. Modify only the number of copies that will be the number of floors above that we have in this case will be 6 and the shaft,

unmarked boxes "link spaces" and therefore "Open Base". We get our building and

.
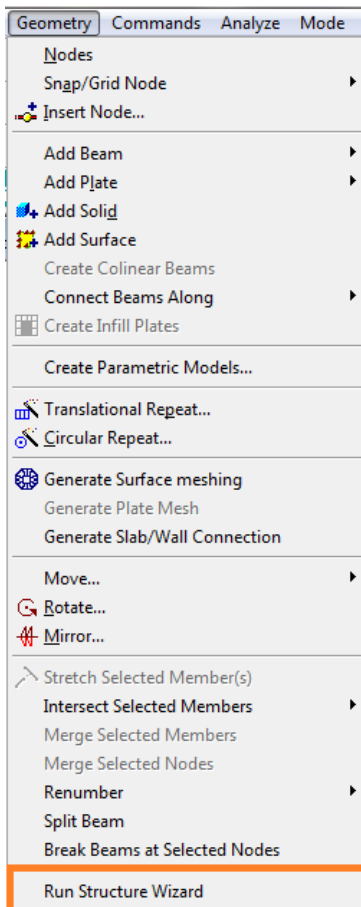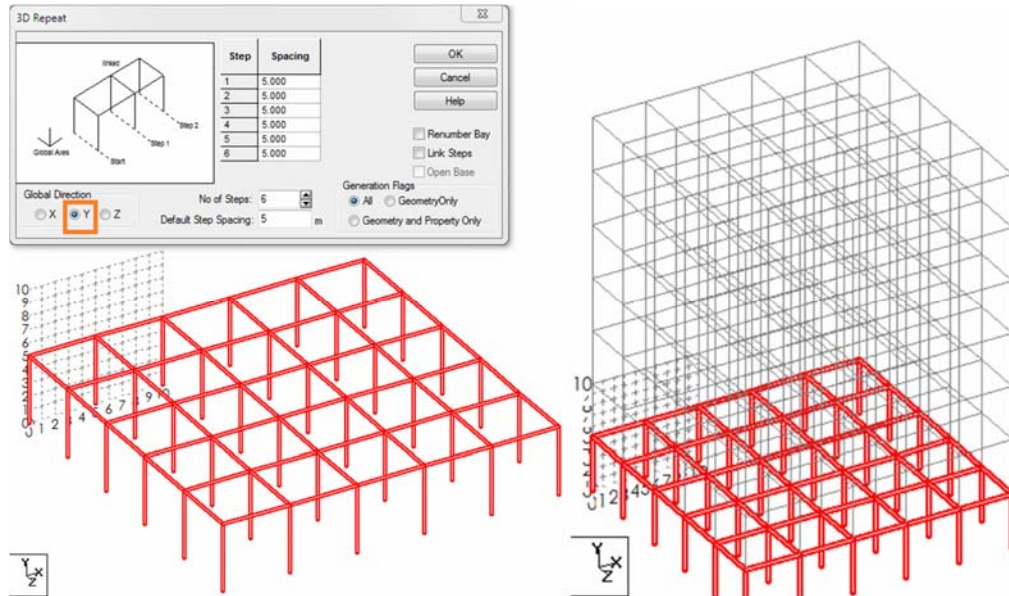
Another quickest way to obtain the same result by "Structure Wizard" or "Wizard Structures" tool, this option provides a library of predefined prototype structures, such as:

- Truss Models: Models Armour.

Frame Models: Models Marcos.

- Surface / Plate Models: Models of surfaces or plates.

- Solid Models: Models of Solids.

- Composite Models: Models of compound shapes.

- CAD Models Import: Import models in DXF format.

For location we go to the "Geometry" (Geometry) and at the bottom we find the "Run Structure Wizard" option (Run Wizard structures).

We select from among the options "Frame Models", will go to the "File" menu select units "Select Units" menu and change the imperial to SI / Metric or metric marking the Meters "m" option. Now, the options that allows us

select "Bay Frame" and configure the pop-up screen, it will ask us the length, width and height of our building and the number of "bays" or spaces in each of these elements, the configure as shown in the image.
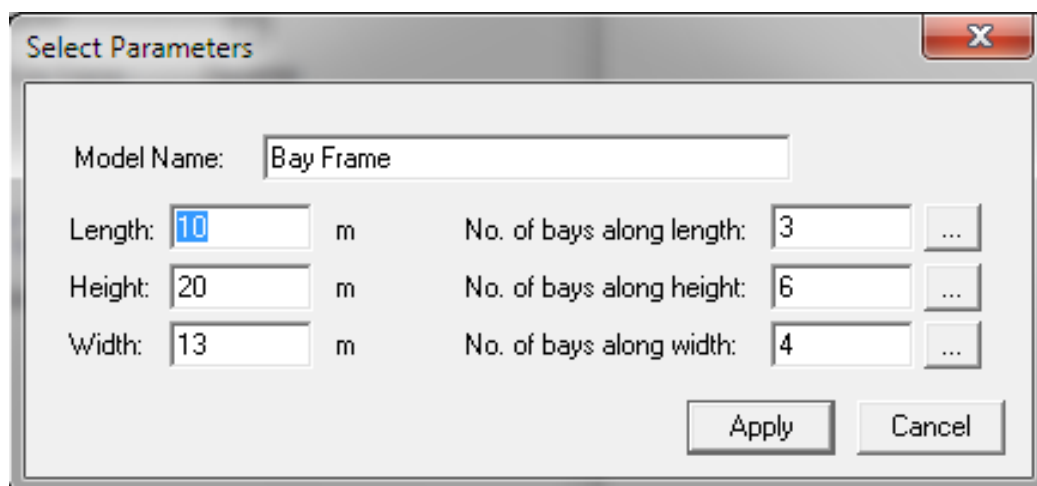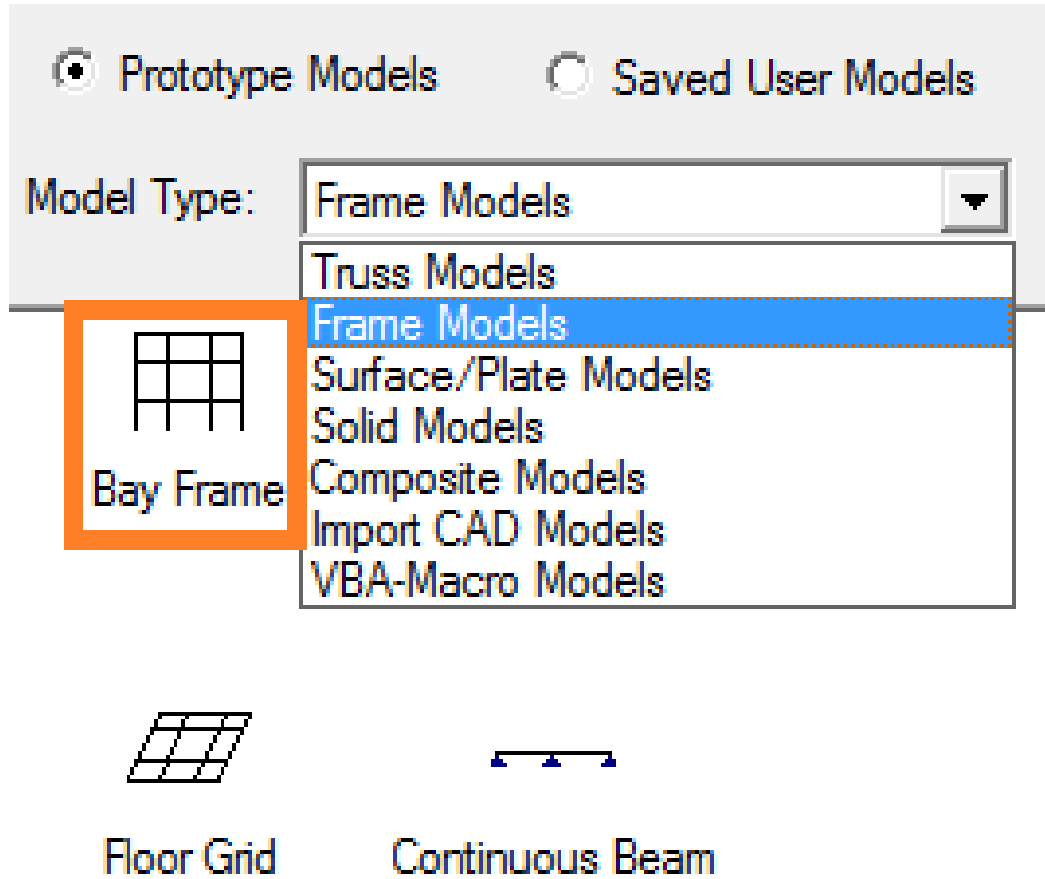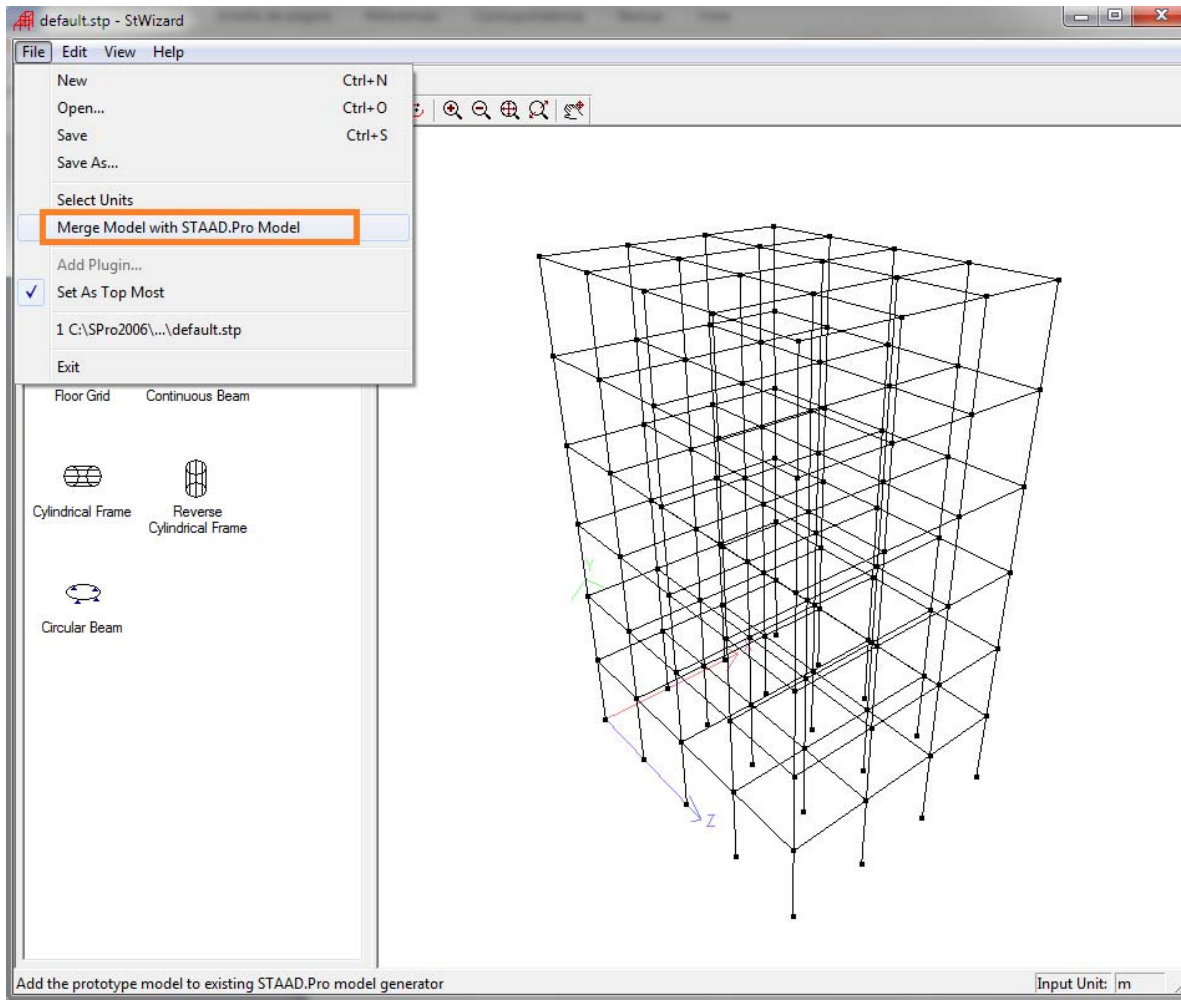
We model the path to the desired configuration, the task now is to migrate our Editor STAAD interface for that we go to the file menu from the "Merge Model with STAAD appears and. Pro Model "

A popup will open to confirm the order to migrate the model, we will click on "Yes" and we will get a Paste Options window where we will give indications of where we want to be the origin of our drawing, select "By the following X, Y and Z values "(for the following values in X, Y, and Z) we will change the value of zero at all, send it to the origin. And we are ready to apply loads in STAAD model.

4.2 Geometric Generating a domed tower

Now chart a circular building plant and for this we will rely again on the "Snap Node / Beam" tool and chart a frame 5 x 5 on our network, once this is done we will select our frame and go to the tool "Circular Repeat "which we will create a repeat of our figure based on an axis located in a center, as a compass which will place on the" Y "and we will place it in X and Z as zero. We assign a 30 ° angle and only one repetition. When generating the geometry manually close the figure with the "Beam" tool or we can make a "Link Steps" option - Link steps and "Open Base" - Base Open ".
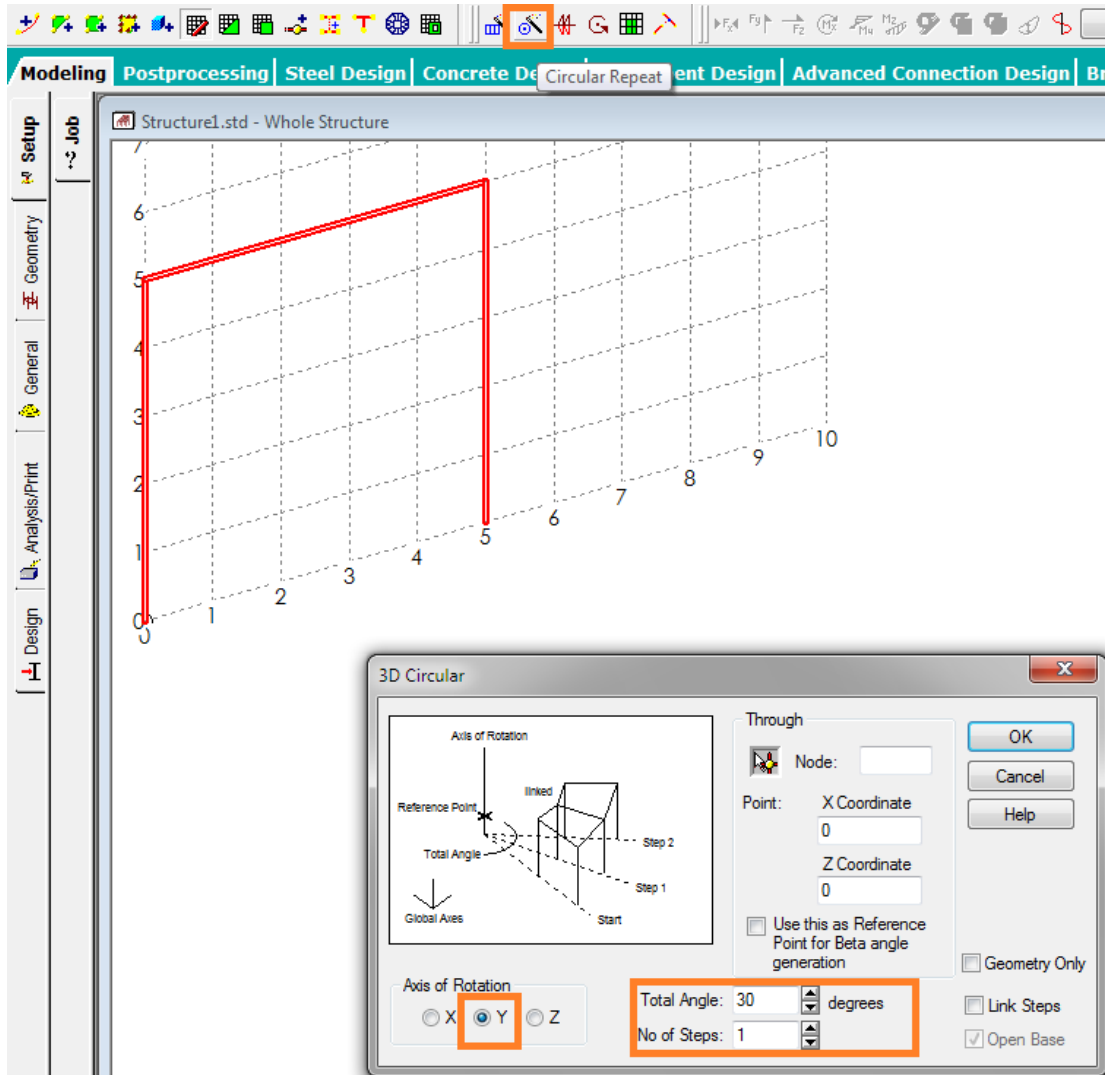
We select our new items and select "Repeat Circular," Circular configure 3D window as shown in the image, that make a circle 360, resulting in the first floor of our building with a ground loop.
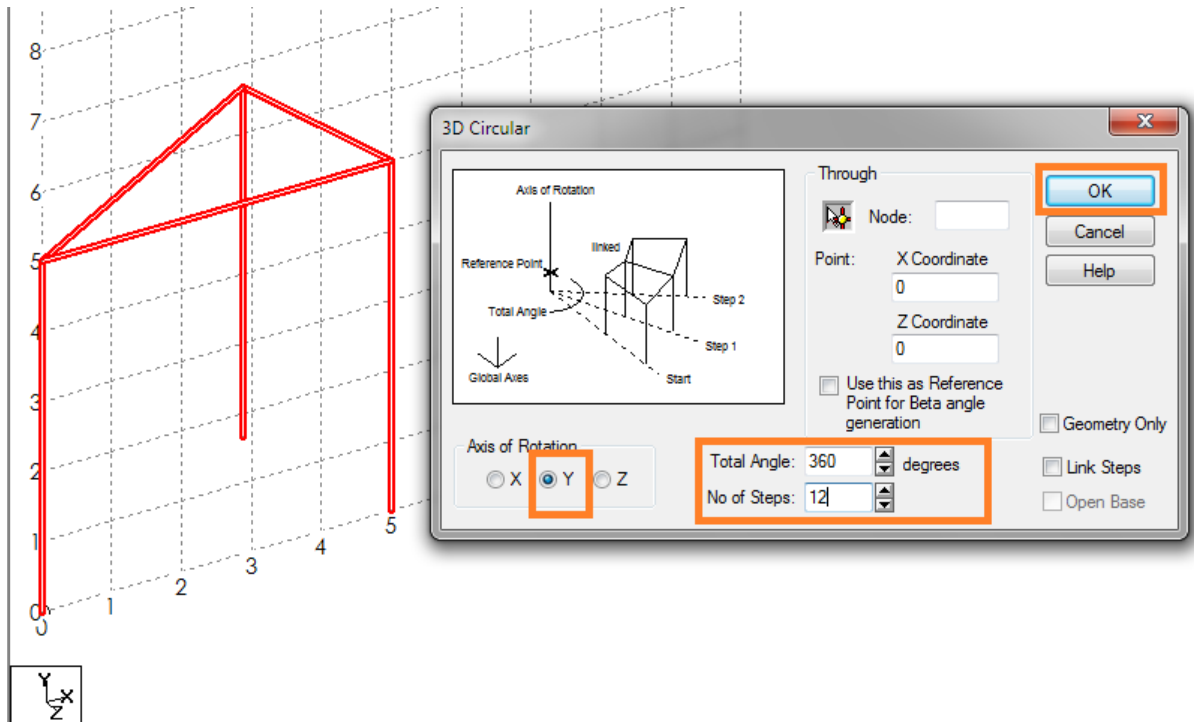
Now we will give up our building, for it will select all our geometry and proceed to the "Traslational Repeat" tool will configure the popup window to perform 3D Repeat us copies of our framework on the "Y" axis at a distance of 5 m. which is the height of our frames.
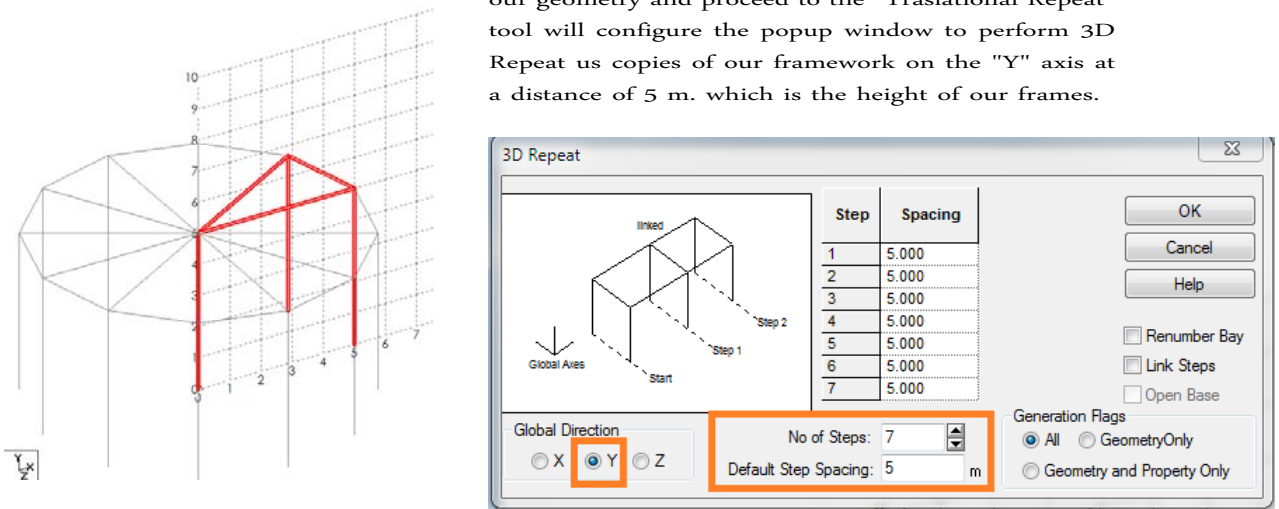
We click OK and we generate the floors that make up our building and with it all our geometry. We can experiment with the settings and get the tools varied geometries. Now as an example and create a dome.
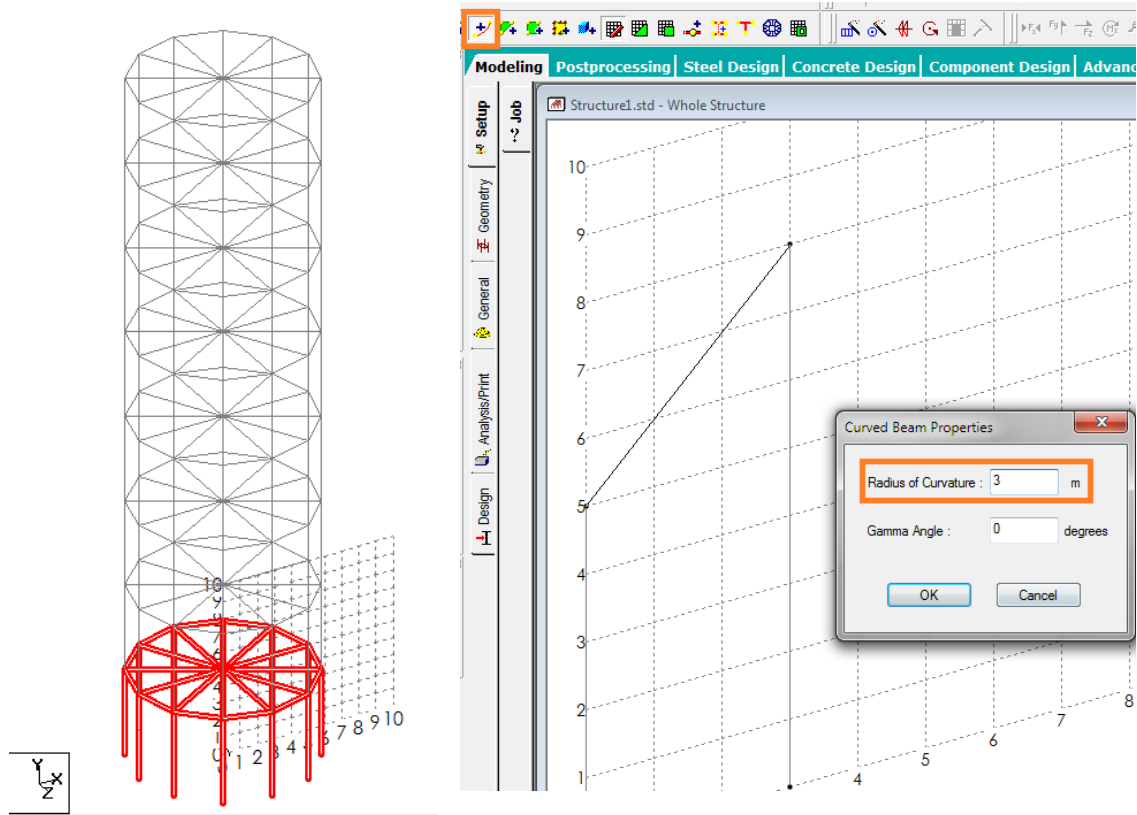
With "Snap Node / Beam" tool we will draw the section of our dome, we stand at the origin and draw as shown in the figure, but ahore resort to "Add Curved Beam" tool (Add Beam curved) giving click Left sustained on the icon "Add Beam", ask us a radius of curvature, we will enter 3 as the value and we will draw our curved beam based on the radius of curvature.
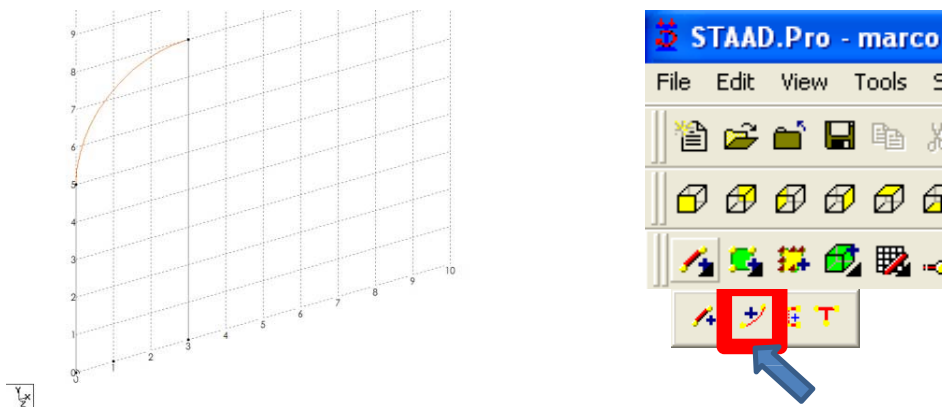
Now select all elements of your screen and activate the "Circular repeat" tool. The configure to rotate about the axis "Y", and the fulcrum is placed at the center of our dome that would be the case in point three "X", for a total of 360 and 11 as

number of times to repeat, configure link for replays and keep the base open, we can see in the picture
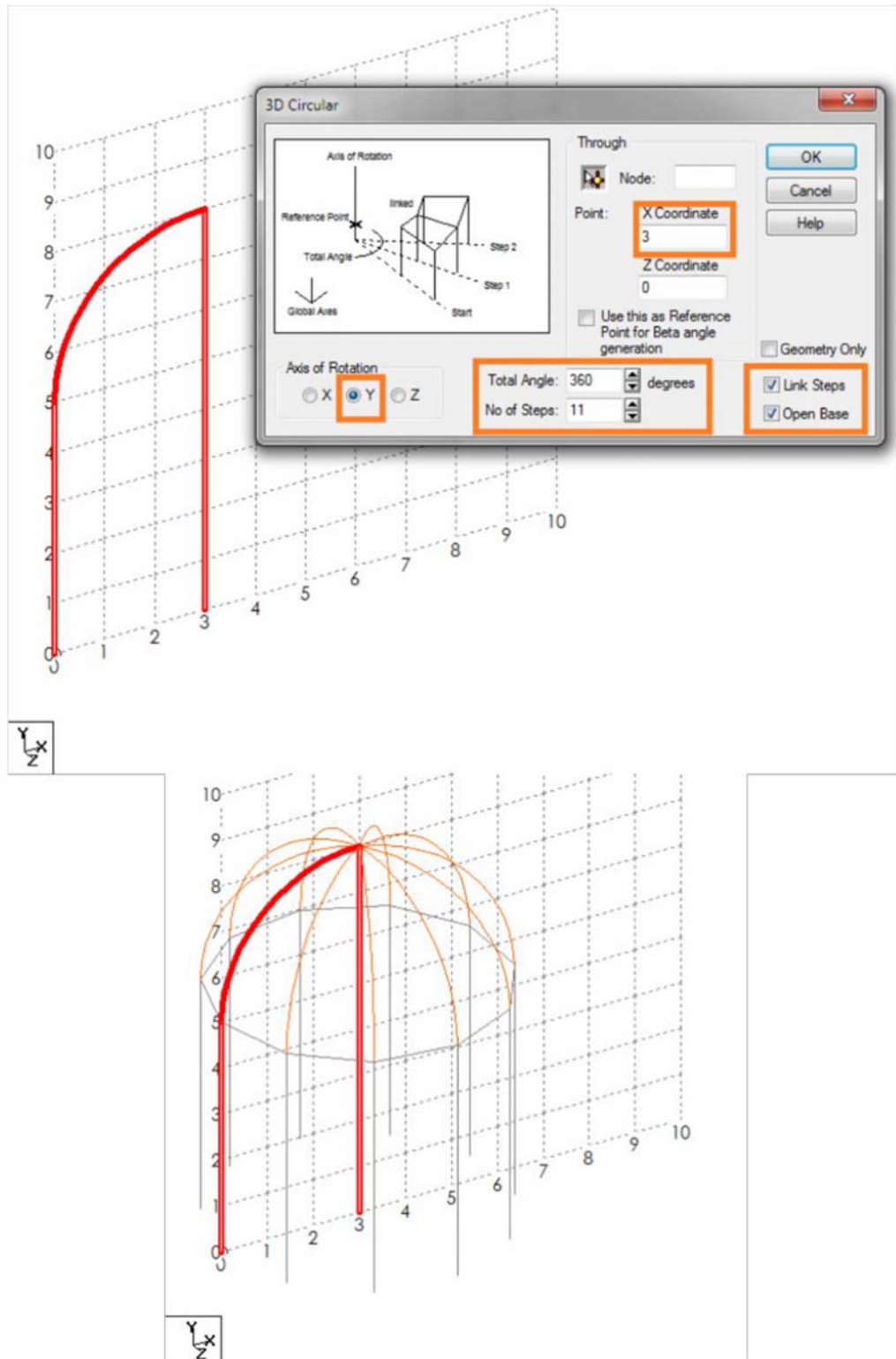
There are also a number of tools that can aid us, as is the case of the measurement tools, the first is "Dimension"

This tool will generate the dimensions of all items displayed on screen, we click on "Display" and "Dimension to view" - Dimensions for truth and we click the Display button.
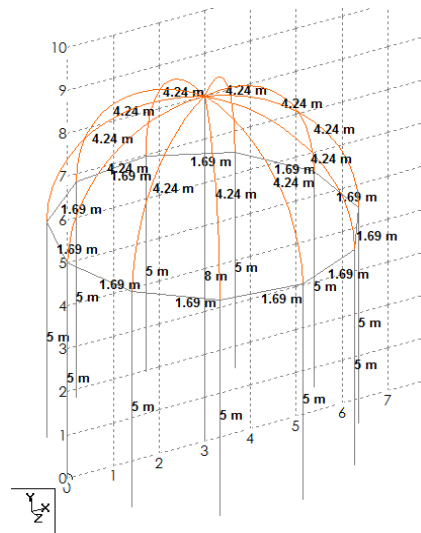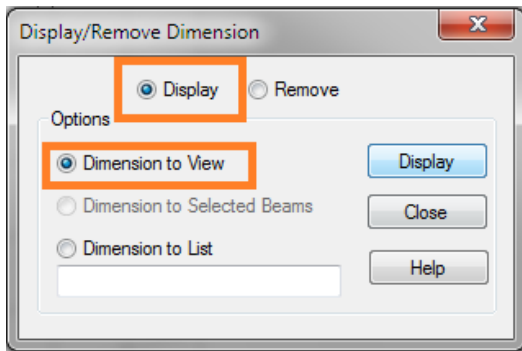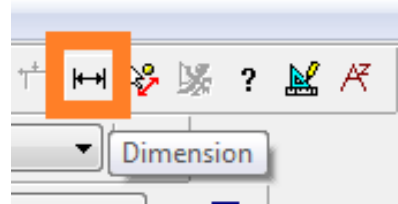
To remove the procedure is the same except that you select the "Remove" option - Remover to remove all the legends.
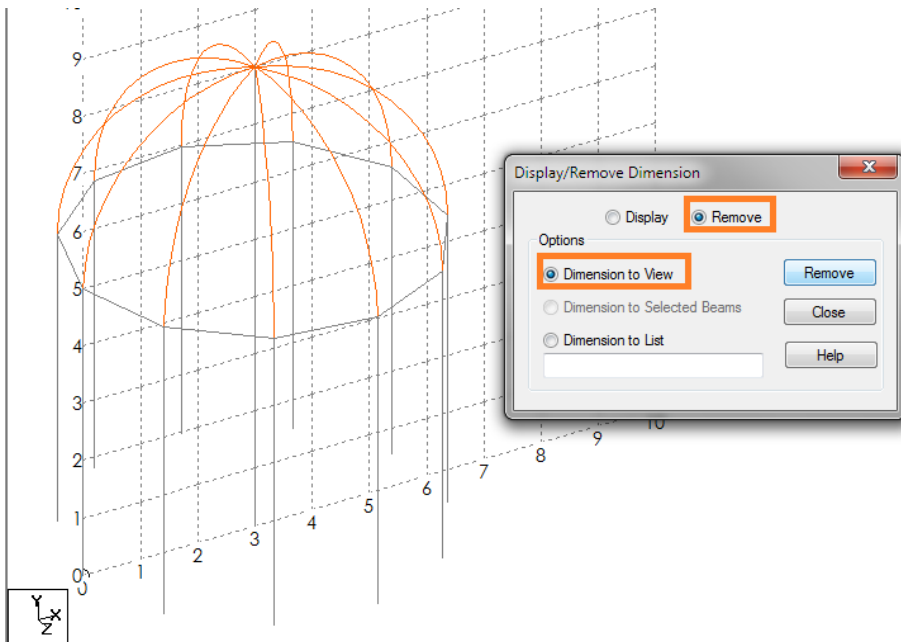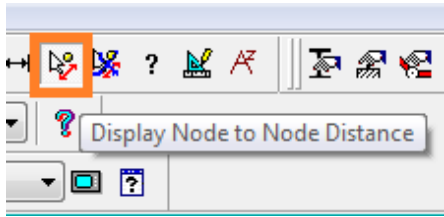
If we require only the measurement of a single item select the "Display Node to Node Distance" tool Shows the distance from node to node-which will cause your cursor cursor automatically switch to Nodes, select the node to node distance we need .
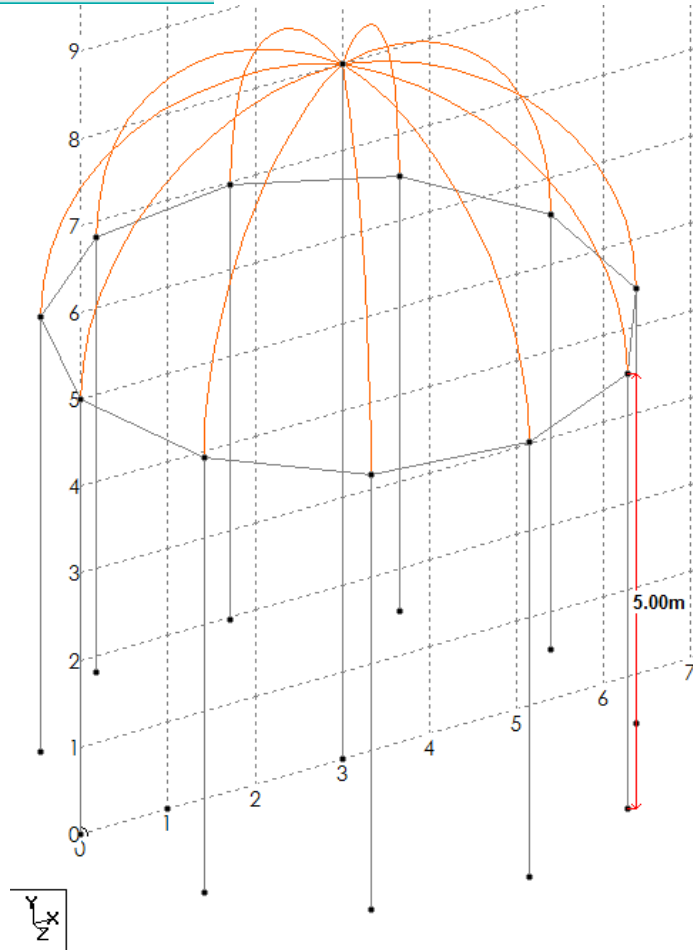
To remove them you just have to click on the "Remove All Node To Node Distances Display" button - Remove all views distances from node to node-
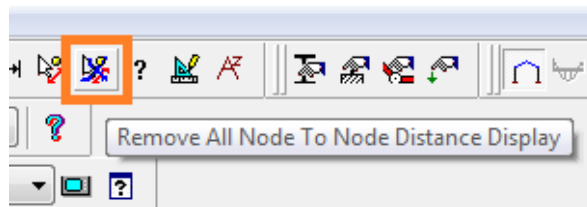
4.3 Geometric Generation, Analysis and Design of an industrial building

We will now work on developing a geometry to a warehouse for further analysis.

We will select the "AddBeams" option and we will work with beams or members and we end option.

Workspace where we start to handle the sampling geometry project, a warehouse opens.

It is important to consider using "Snap Node / Beam" (Network Node / Beam) as this creates for us a grid where we can enter our data, activate the icon This Snap Node / Beam and select the "Geometry" option the left side tab
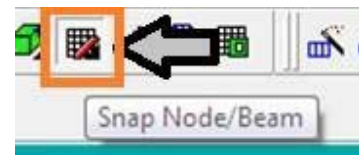
This Grid can be edited to increase the number of spaces in "x" or "y"
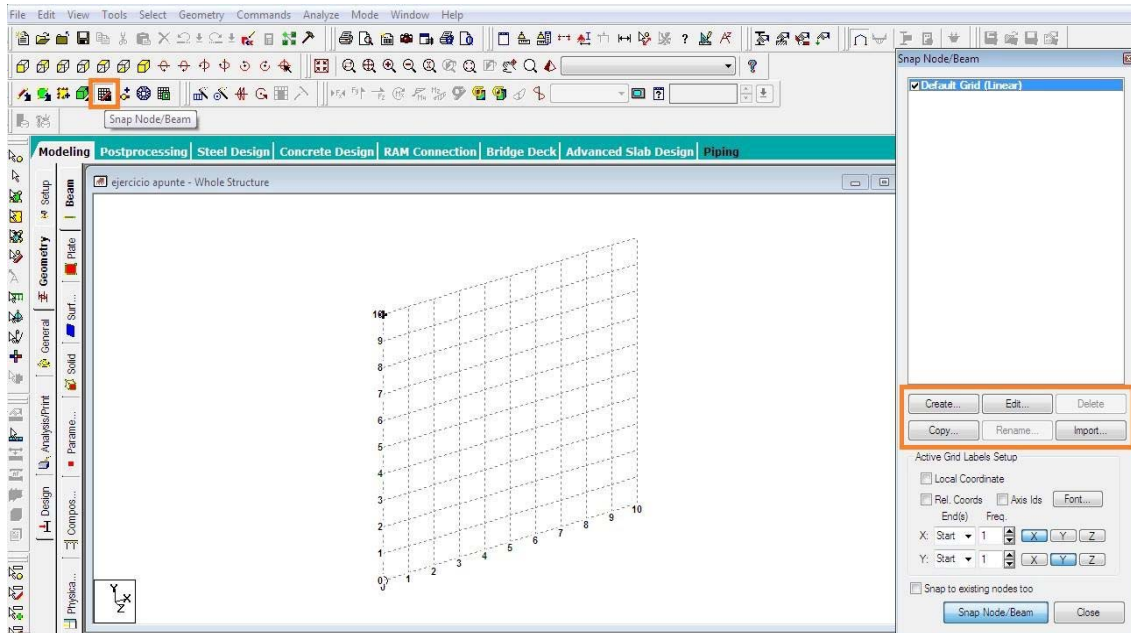
We are presented with the window opening STAAD initial work, with the options the grid, mesh or grid enabled on the right we can see the options to edit the mesh (Create, Edit, Copy, Import). We are going to click on the "Edit" button to add our Malla spaces. The idea is to draw a profile of a warehouse of 16 m. wide by 7 feet high.
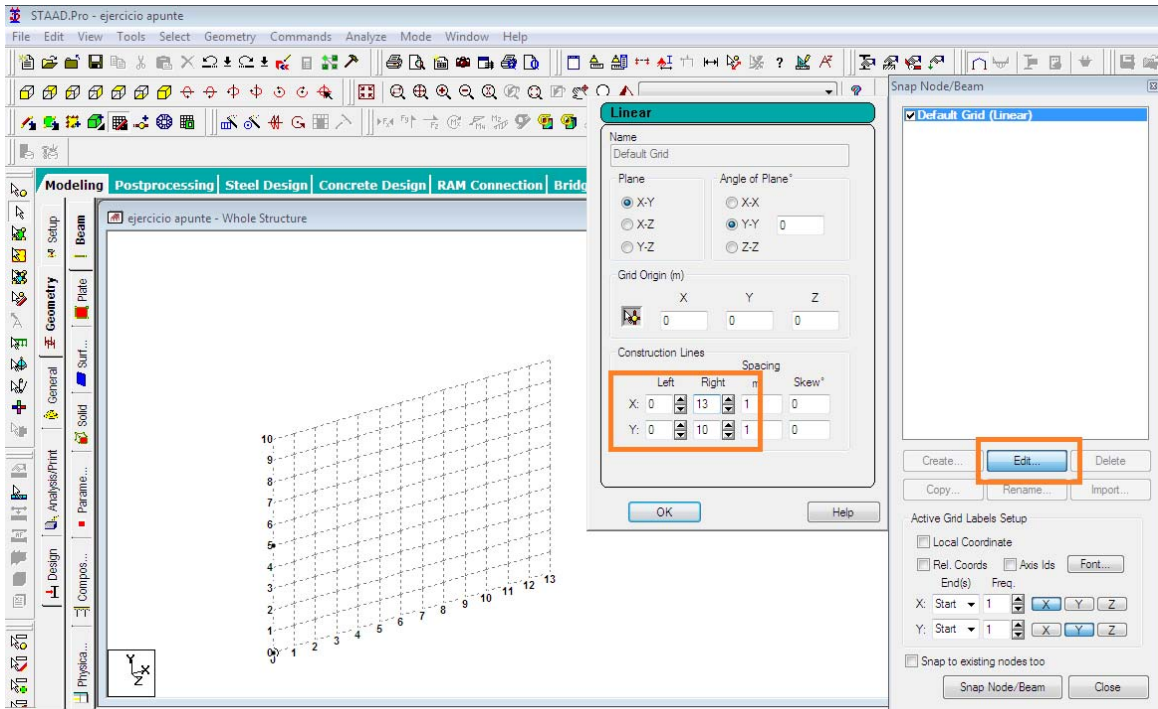
At the bottom we find the "Snap Node / Beam" button to start tracing the profile of our ship must be activated (turns blue) and draw the profile as shown in Fig. Press "Escape" (Esc) to end.
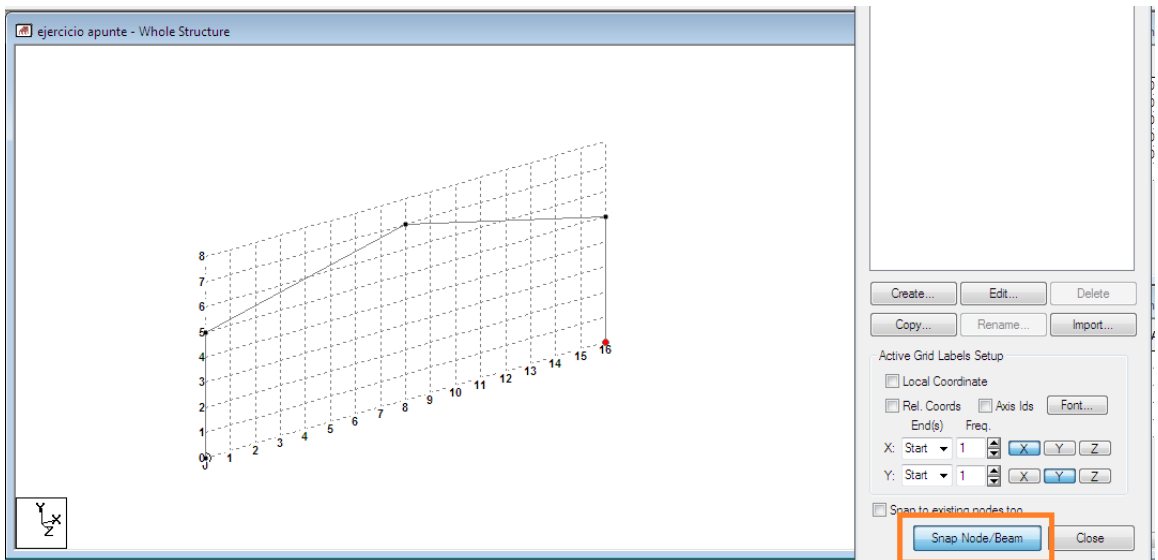
Already with the first section that will shape our ship, we proceed to the 3D geometry, we can do this in several ways, by pressing Ctrl + C to copy and Ctrl + V to paste, the software will ask how far and on what axis we glue the copy of our profile.

Importantly, on which axis will move our element, "x", "y" or "z" is "z" where we will copy our element and moving to create our ship, we will enter the number 5 **ok.** With the "Add Beams" option (Add beams) we will add the beams that it needs our

structure
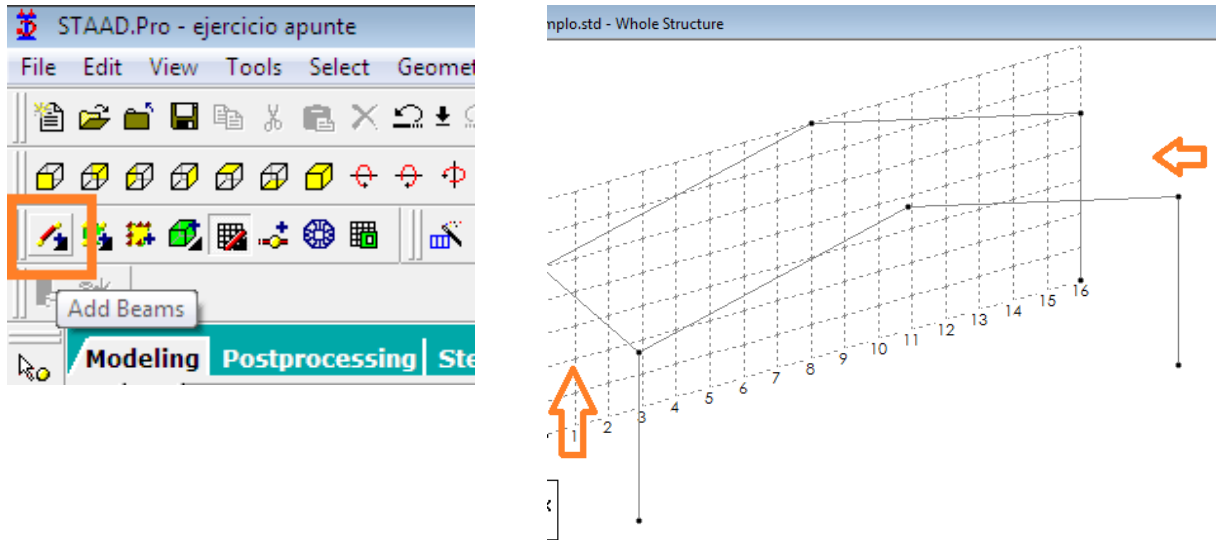
To create a representation of us let our rails segment our roof with "Insert Node" (Insert Node) tool. Insert six intermediate nodes in each of our covers, for that select the beam where we enter the nodes and we click on "Insert Nodes", a popup window where we are asked at what points we want to insert our nodes appear, we will select for number of nodes to add, we will enter 5, and we will press the "Add n Points" button (Add n points) the software will place them equidistant on the selected beam. We OK and do the same with the other three beams.
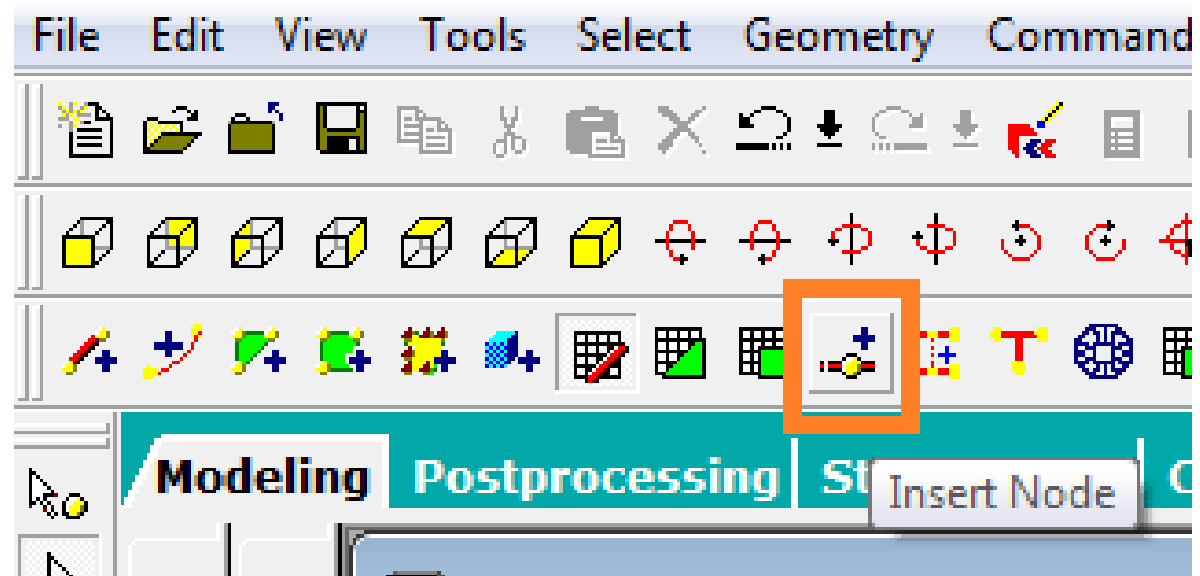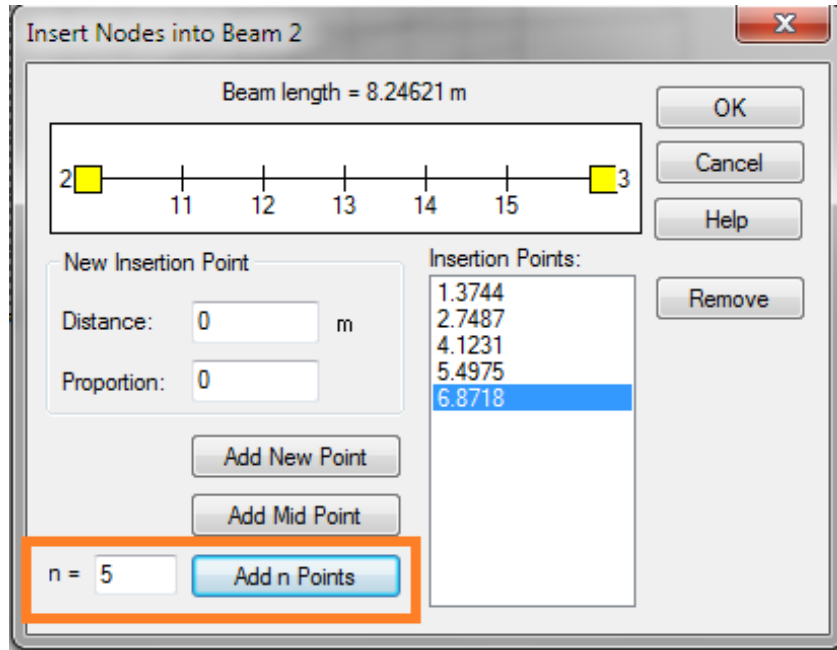
Once this process will join the dots to have the desired geometry. We can do it by "Add Beams" tool which is based on the location of the nodes we enter the cover to generate the beams and get a graphical representation of our structural elements.
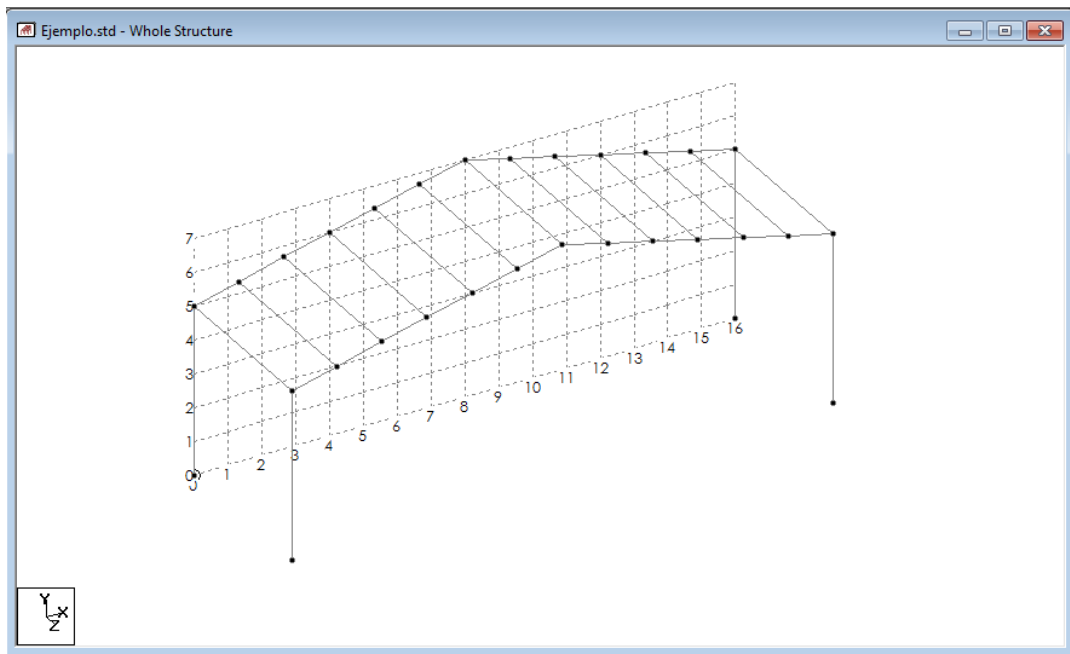
Another option to perform this task is to select the "Translational Repeat" tool (translational repeat). This tool will create a copy or set of copies of the selected items and stick to the distance on the axis indicate. We will select all the elements that make up our ship and select the tool, we a popup titled "3D Repeat" (3D Repeat) where we asked that we enter the data that we will need appears, we will need 7 copies of our profile to a distance of 5 m. on the axis "z", we'll configure from the popup.
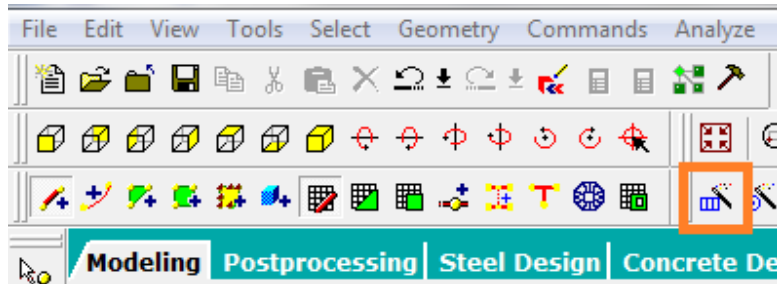
The configure as shown in the image, taking into account the axis on which we want the displacement copying our elements, which in this case is on the axis "Z" is made
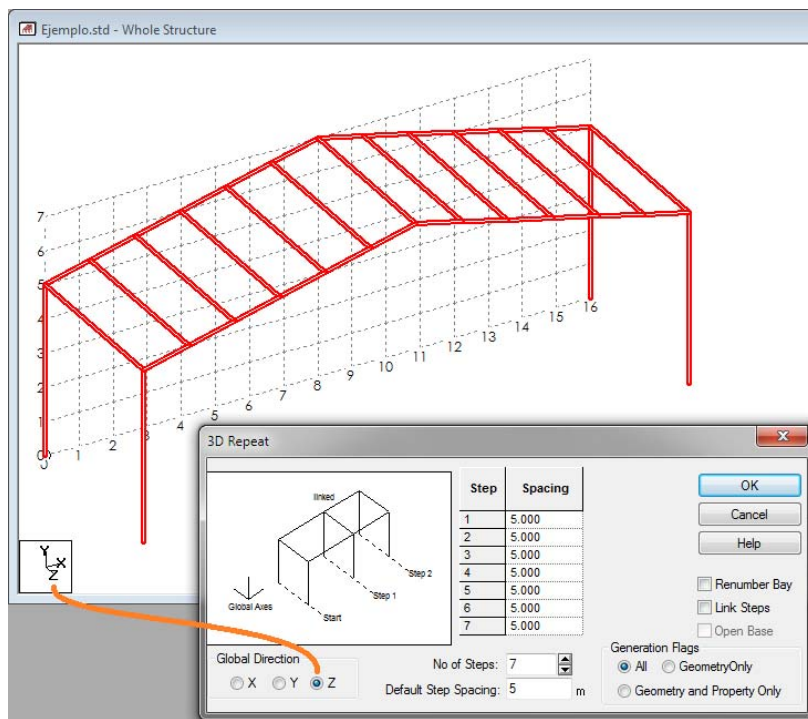
And we will prove a more composed our warehouse geometry, the last step

would add contraventeos upper and side to provide stability.
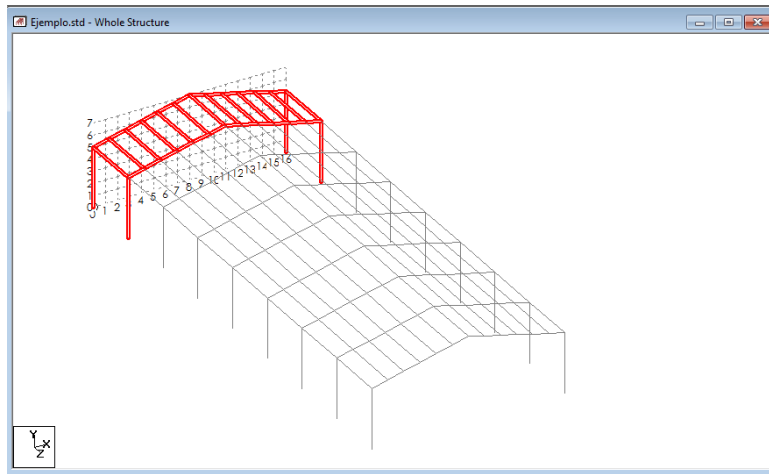
Another option is to perform a certain way and fastest way would be from zero to trace the profile of our warehouse without making any copies from our deck beams to the appropriate nodes from this point using the tool "translational repeat" configuring in the same way but instead of adding 8 and 7 copies selecting different "Link steps" and "base Open" to generate an open base. Once this geometry only add Add contraventeos with Beams tool.
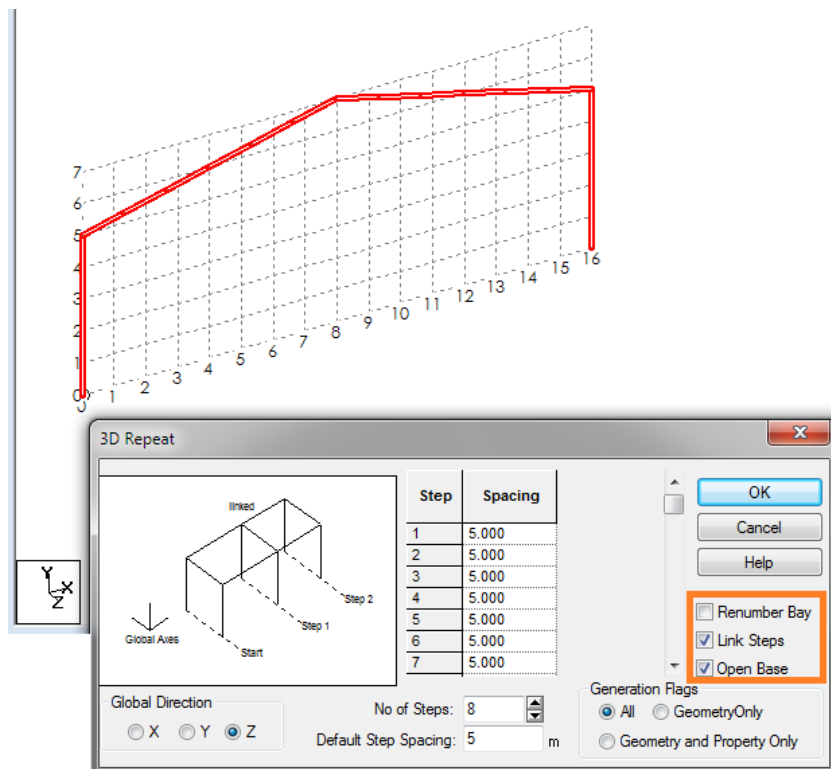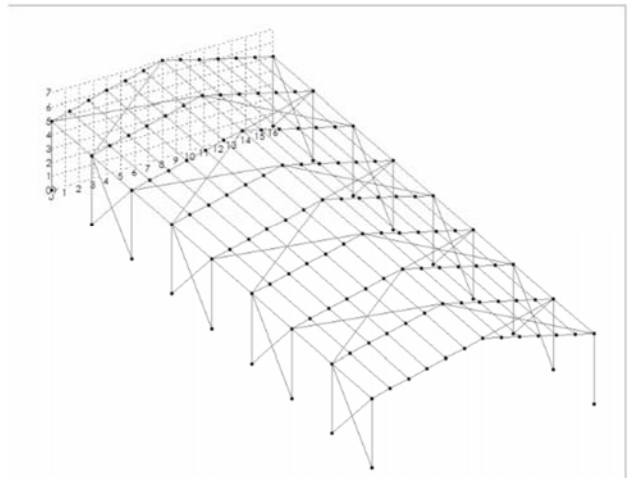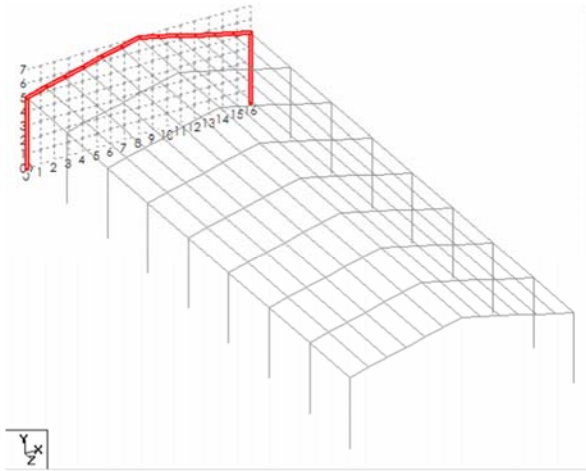
After adding the side and top contraventeos we have completed our ship geometry.

We begin by applying media, use the Support Page tool located at the top right of the interface program.
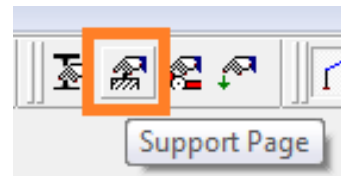
We popup window appears, select the "Create" Create option to create a new kind of support, we will have different types of support, Built (Fixed), Pinned (Articulated), Etc.
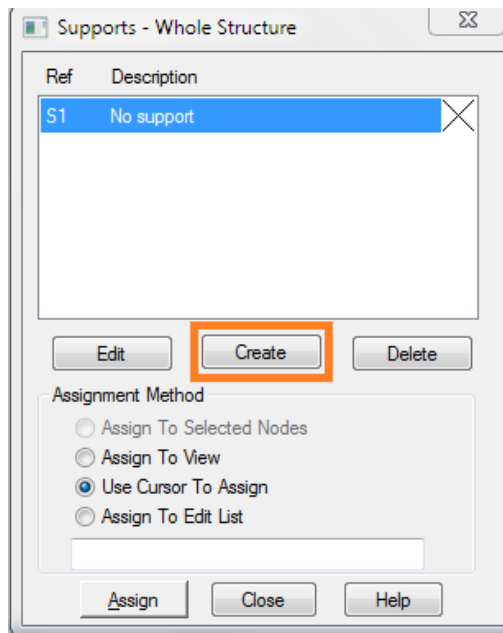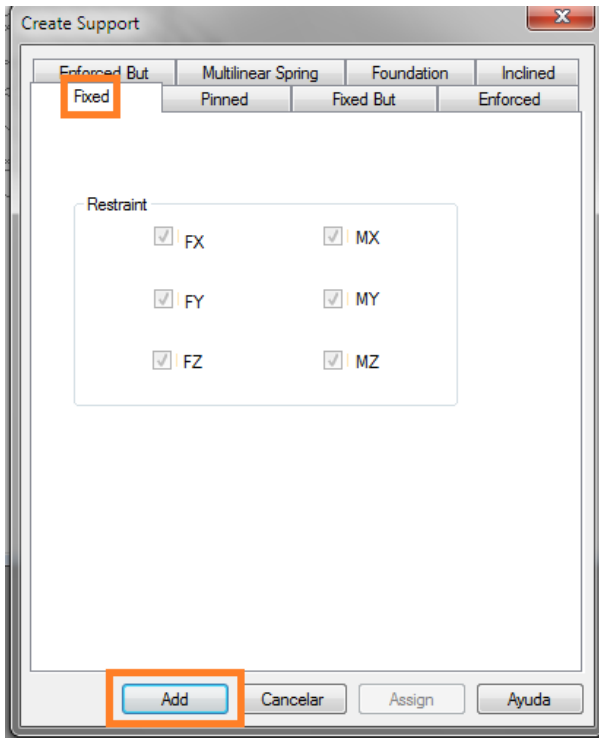
Fixed selected and click the "Add" button.

It appears we are now on the menu a support 2 "Support 2" with a drawing that represents the type of support. We will place the cursor on the support and select the option "Use cursor to Assign" (Use cursor to assign), automatically your cursor will change to cursor node, we go to the icon "View from Z" to see our ship from a front view, (icon, along with all the views are in the upper left of the window), this will help us to select all nodes through a single window selection, once made the selection nodes are shown in red and will enable and autoseleccionará Option "Assign to selected Nodes" (Assign Selected Nodes) We click on "Assign" (Assign).
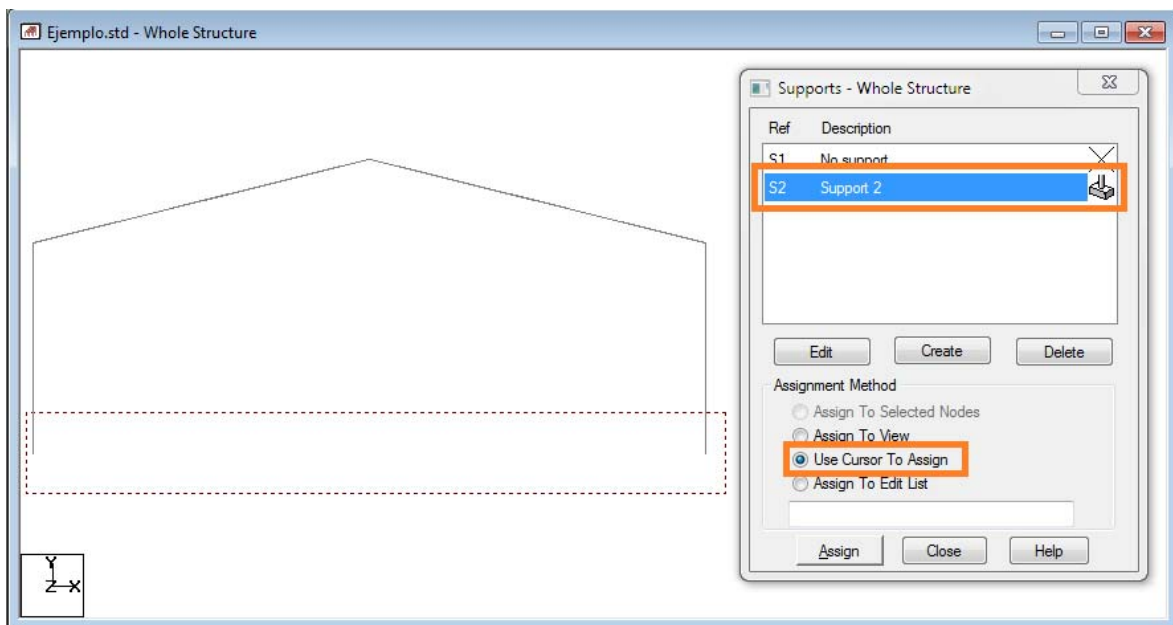
A popup will open to confirm the selection method, we will click "Yes" and we have the support conditions.
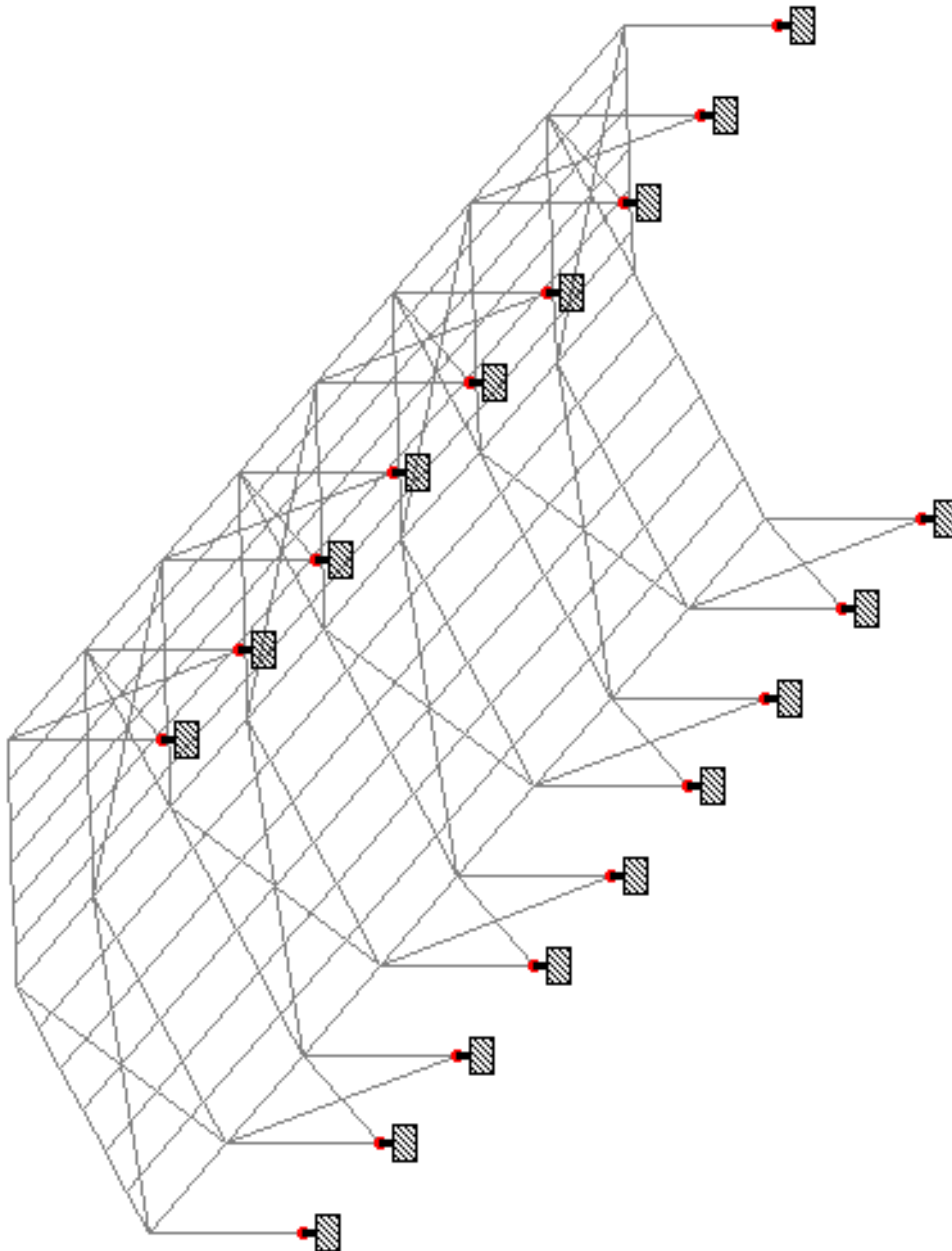
4.3.1 Creating Groups (for such elements).

Within the selected toolbar TOOLS rear tab, select this option the CREATE NEW GROUP. It can also be used as a short command Ctrl + G.
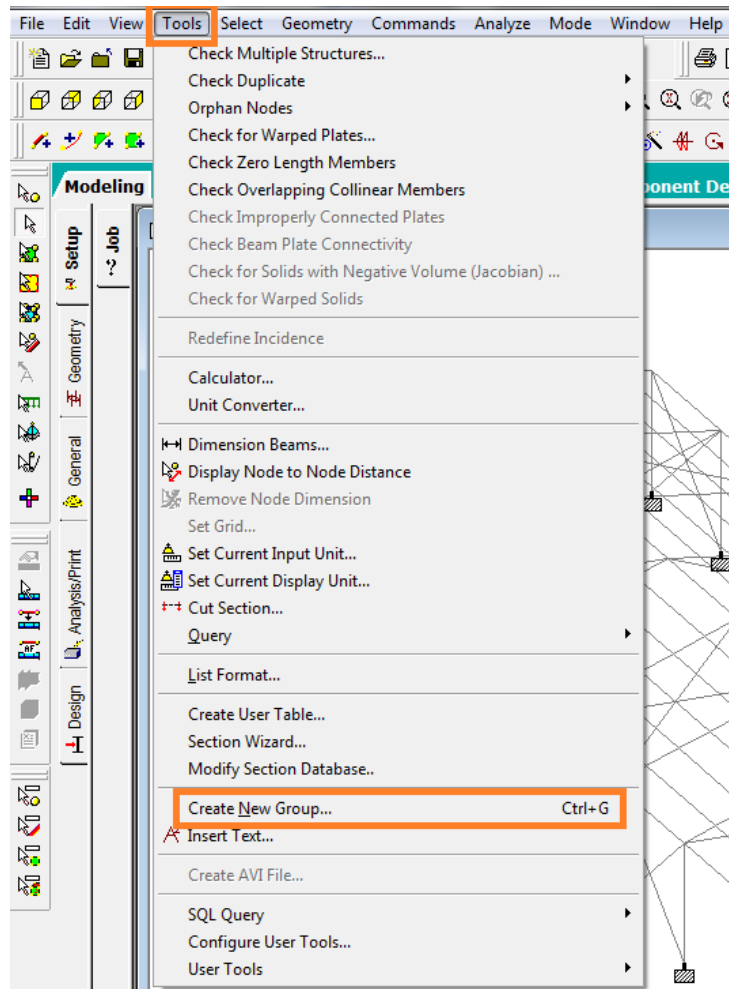
If you are creating a group for the first time within the project, the following dialog appears.
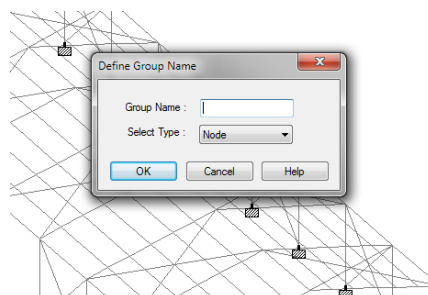
Where we enter the name of the group that we create as well as the type of team that should be recognized STAAD, depending on our project; For the year to use as the name of COLUMN group and type of selection we seek will be beamed, referring to the lines of your drawing.
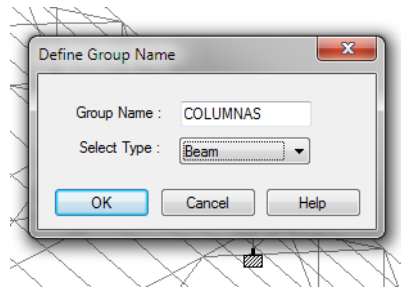
Once you have entered the data, we click OK and it will open a new dialog.
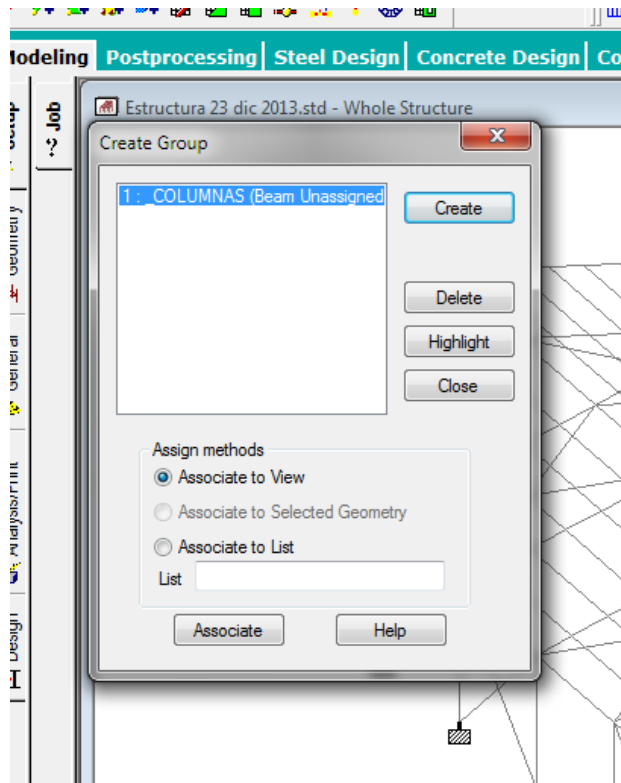
Here we realize that the name of our group was created correctly, but has not been selected beams (lines) that form this group in the drawing, to do so, we return to the interface of our model and selected lines formed the group created, once selected we will click ASSOCIATE TO SELECTED GEOMETRY, as a last step will click in the box ASSOCIATE.
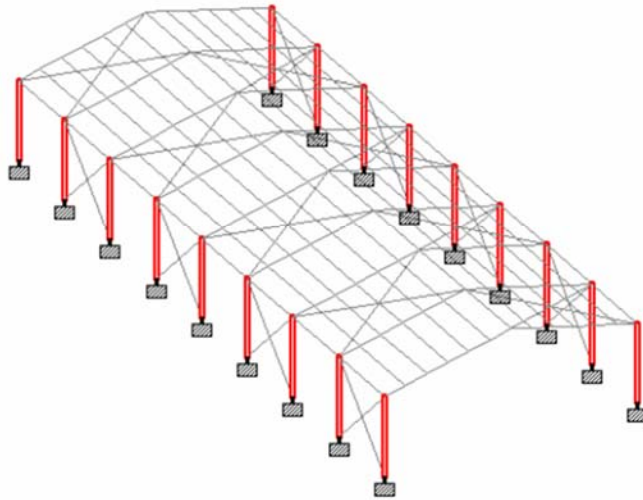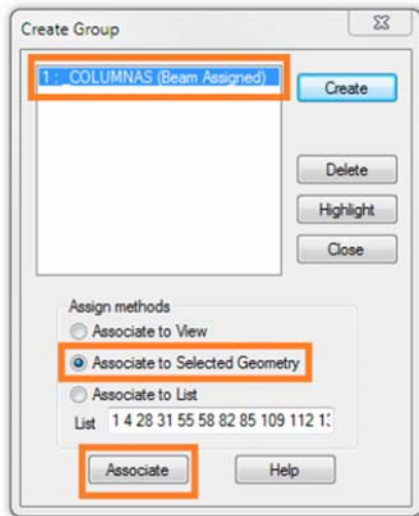
It is important to note that when you click ASSOCIATE, Box LIST numbers of lines that have been associated with this group appear.

Do the same with all groups that want to create within the project by clicking on the box and deposit data CREATE explained above for each of the cases.
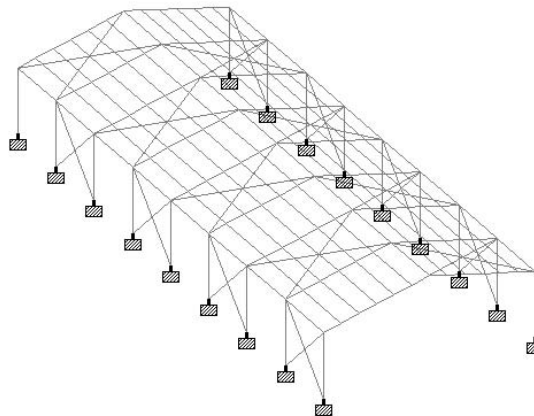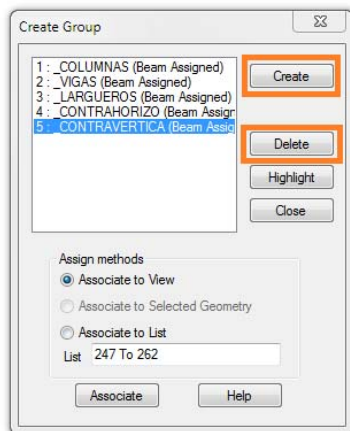
In the window building groups will reflect the groups we create in our project; if we remove one of the already created groups, select the group in the list (displayed in blue) and you click on the DELETE box as shown in the figure above.

The utility of creating groups is to select faster than doing it line by line, and when any changes are made within a group (change of material properties), the change is reflected on all lines belonging this group.

To select a group, within the selected toolbar
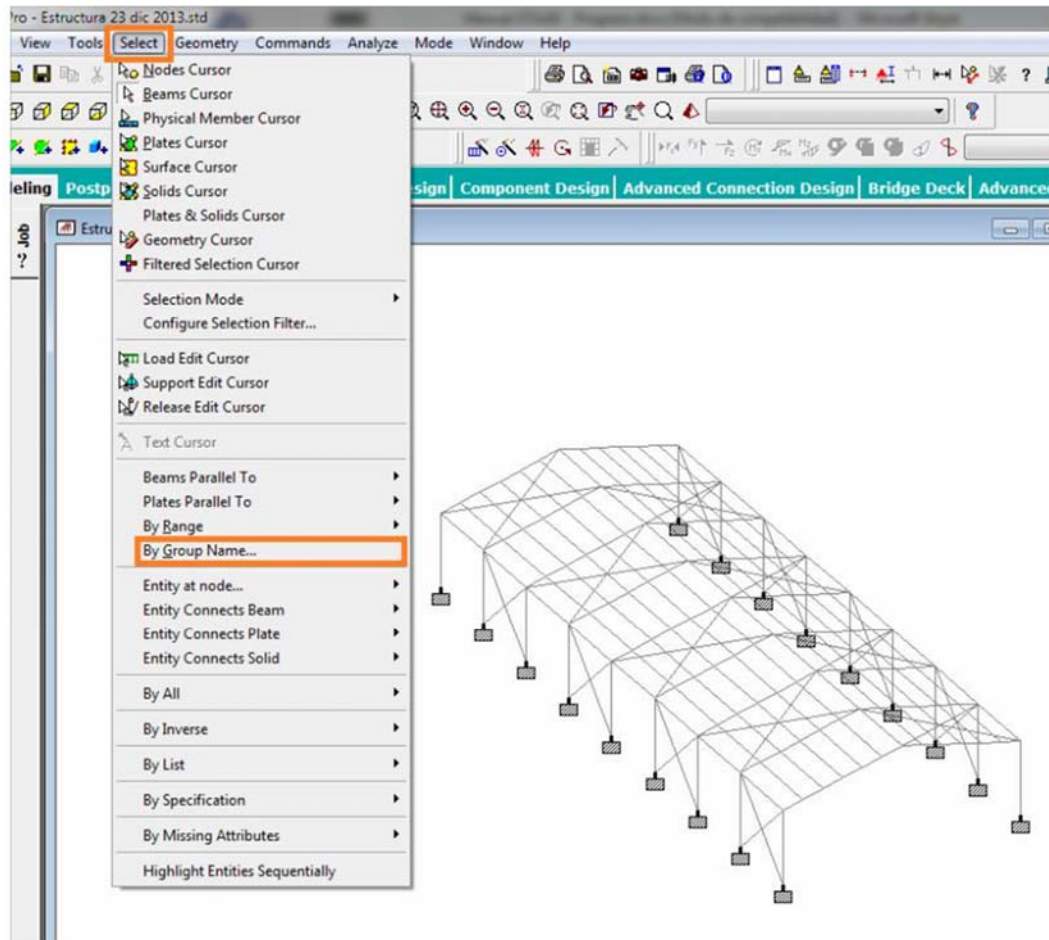
tab that says SELECT and then the option GROUP BY NAME. Then we click on the

group you want it to be selected in the model.
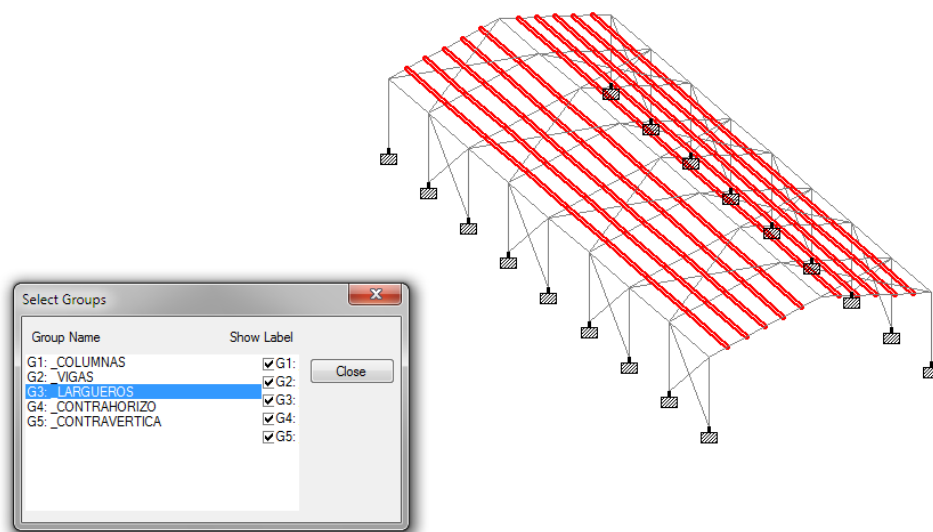
4.3.2 Assigning materials.

For the placement of materials within our project PAGE PROPERTY choose the option that is within our toolbar with the following logo and will click.
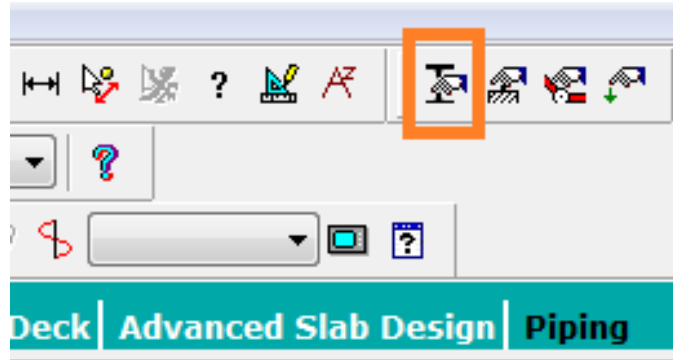
After this we will open a new window, where we select the database section where the existing profiles, we will click on the DATABASE SECTION option.
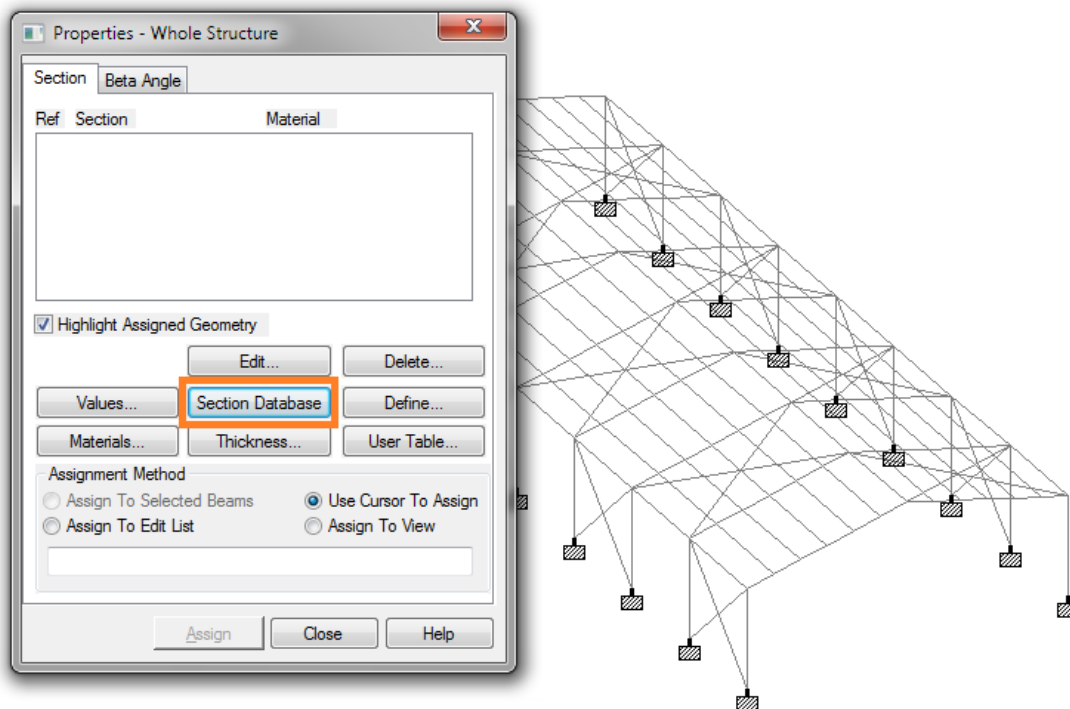
We will generate a new window, in the toolbar, select the STEEL, where the different countries and their books unfold database option, select the Mexican book and the types of industrial profiles in the market will be deployed in the right side shows the various dimensions of profiles depending on the characteristics selected from the list; select the size you require for the project and then we click on the ADD window.
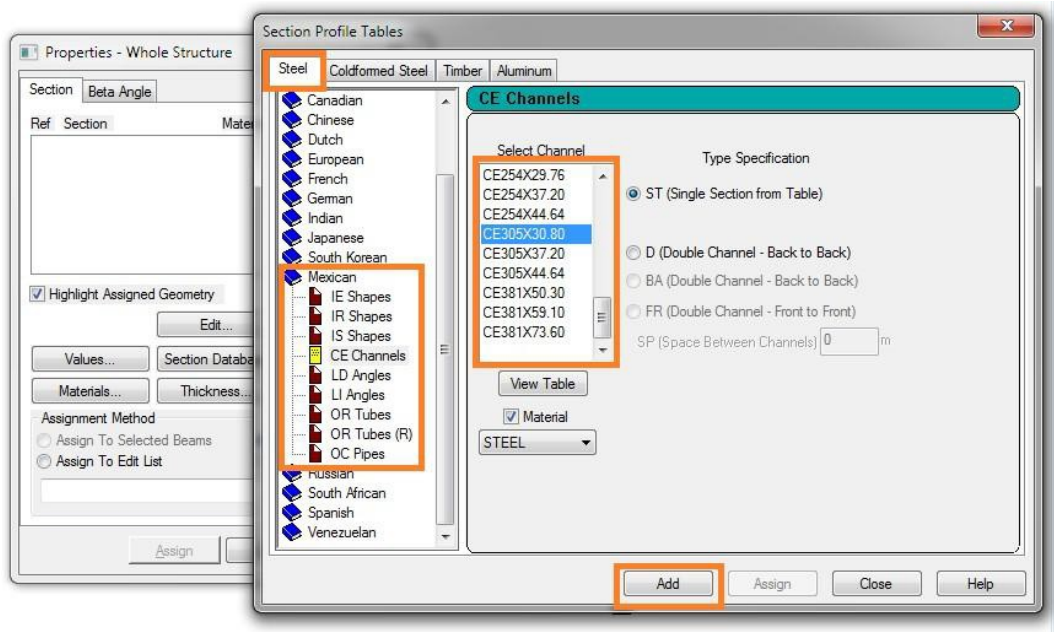
And continue with the selection of materials and dimensions of all the different profiles to use in the project. Materials that have chosen appear in the List window PROPERTIES - WHOLE ESTRUCTURE, which at the beginning was blank.
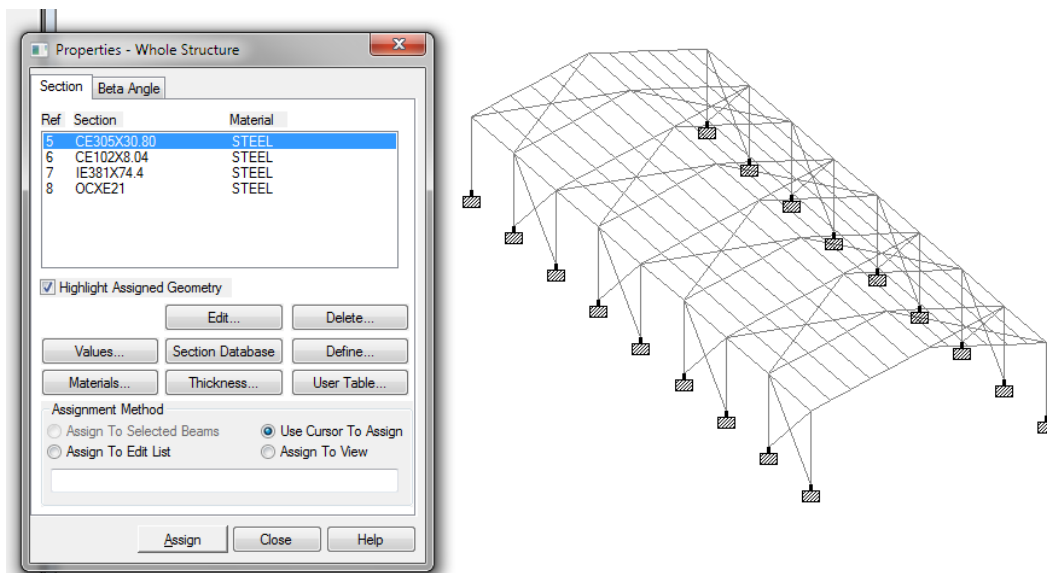
For the allocation of materials, select the material you want to apply and then select the group you are going to apply that material, group selection we stand in the SELECT option, located in the toolbar and choose the GROUP BY NAME option.
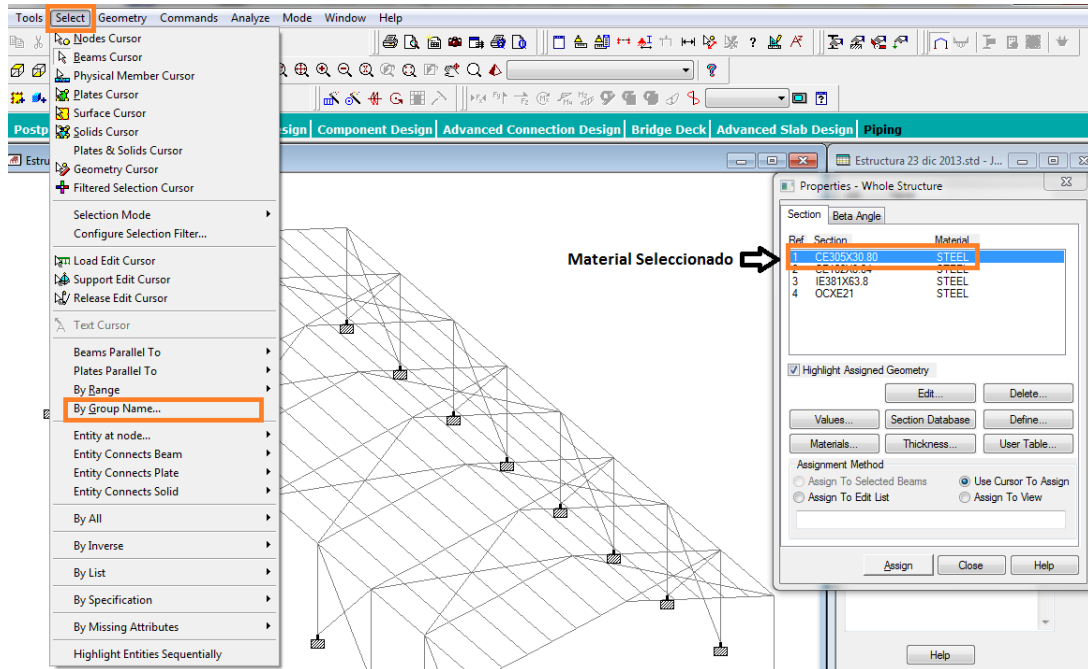
Once you select the group to which we will apply the material within the PROPERTIES window - select the ASSIGN WHOLE STRUCTURE BEAMS TO SELECTED option and finally we click on ASSIGN.
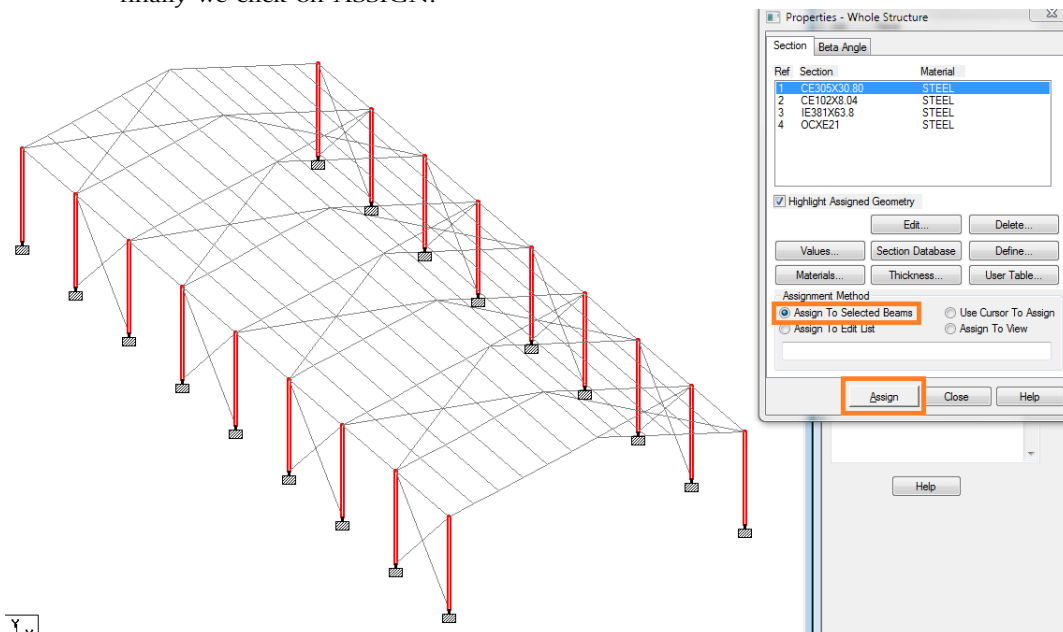
4.3.3 Assigning Loads

Once assigned materials, determine the loads applied to the structure and the elements that will be affected by such charges.

Importantly, the charges will be varied depending on the roof system is applied, the number of floors that contain our element and the area in which the project is located.

To determine the loads will click on PAGE LOAD icon and we immediately open a side window called LOAD, within this window will place the burdens be allocated. We stand by clicking on LOAD CASES DETAILS tab and once it is highlighted blue we click the ADD option.
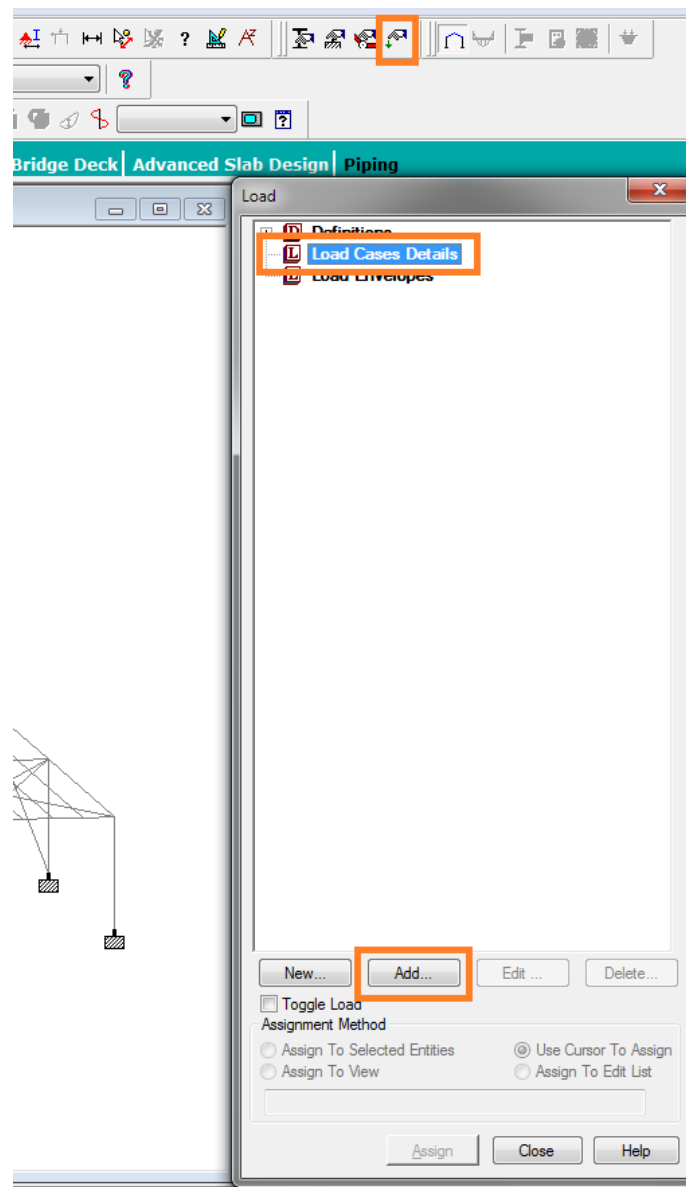
Once this is done, we will open a new popup window called ADD NEW LOAD CASES, which give high loads that will be considered within the project and the name of these charges; as a first step enter the name of the load, then the option TYPE LOADING deploy options and select live, dead or wind, depending on the type of cargo that we are adding, and finally we will click ADD.
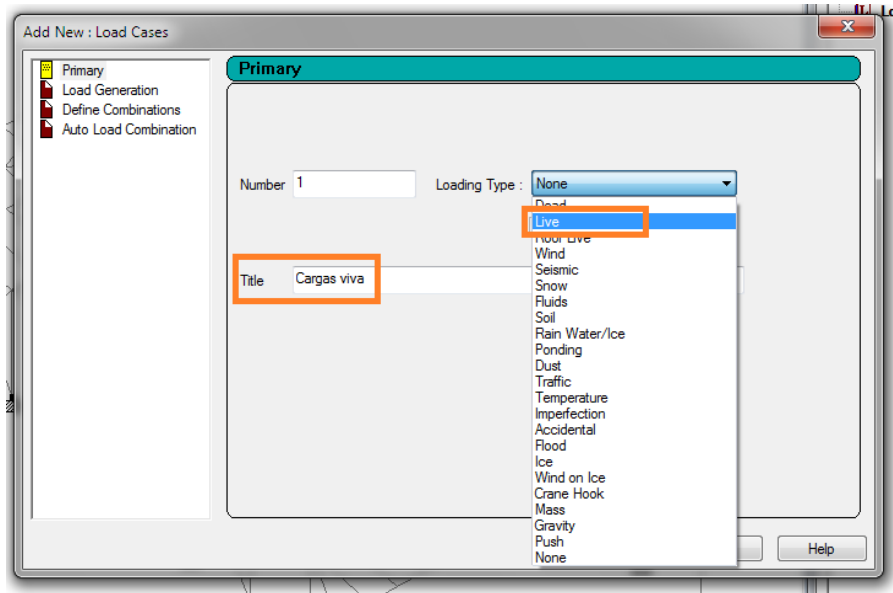
Once we gave high loads, appear LOAD CASES DETAILS tab. We have not added any value to the load placed on it (alive) will click on ADD.
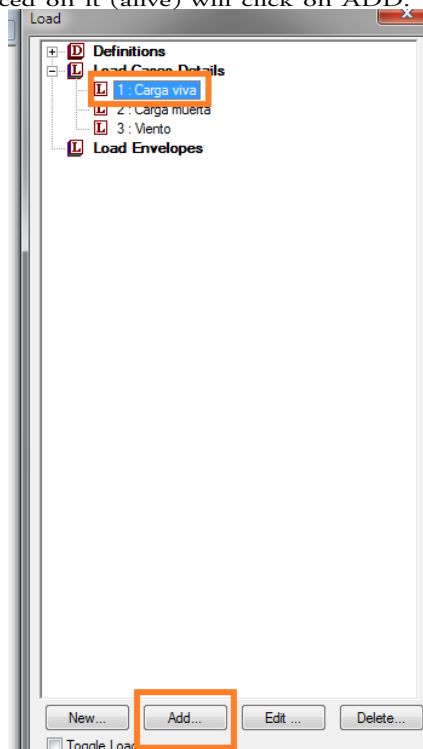
After this we will open a new popup window called ADD NEW LOAD ITEMS here assign the value of the charges and the manner in which they were generated in our elements. To this list we will appear with the following options:

- o   Selfweigth: Dead weight.
- o   Nodal Load: load nodal.
- o   Member Load: Load the member.
- o   Physical Member Load: Load the physical member.
- o   Area Load: Loading area
- o   Floor Load: floor or floor load.
- o   Plate Loads: Loads of plaque.
- o   Surface Loads: Loads surface.
- o   Solid Loads: Loads solid.
- o   Temperature Loads: Loads of temperature.
- o   Seismic Loads: Seismic loads.
- o   Time History: History of time.
- o   Wind Load: Wind load.
- o   Snow Load: Snow load.
- o   Response Spectra: Response spectra.
- o   Repeat Load: Loads of repetition.
- o   Frequency: Frequency.

In this case, for our select SELFWEIGTH live load, which is handled as the weight of the building, it only once, so the factor will be -1 as it will negatively on the Y axis, and we click ADD.
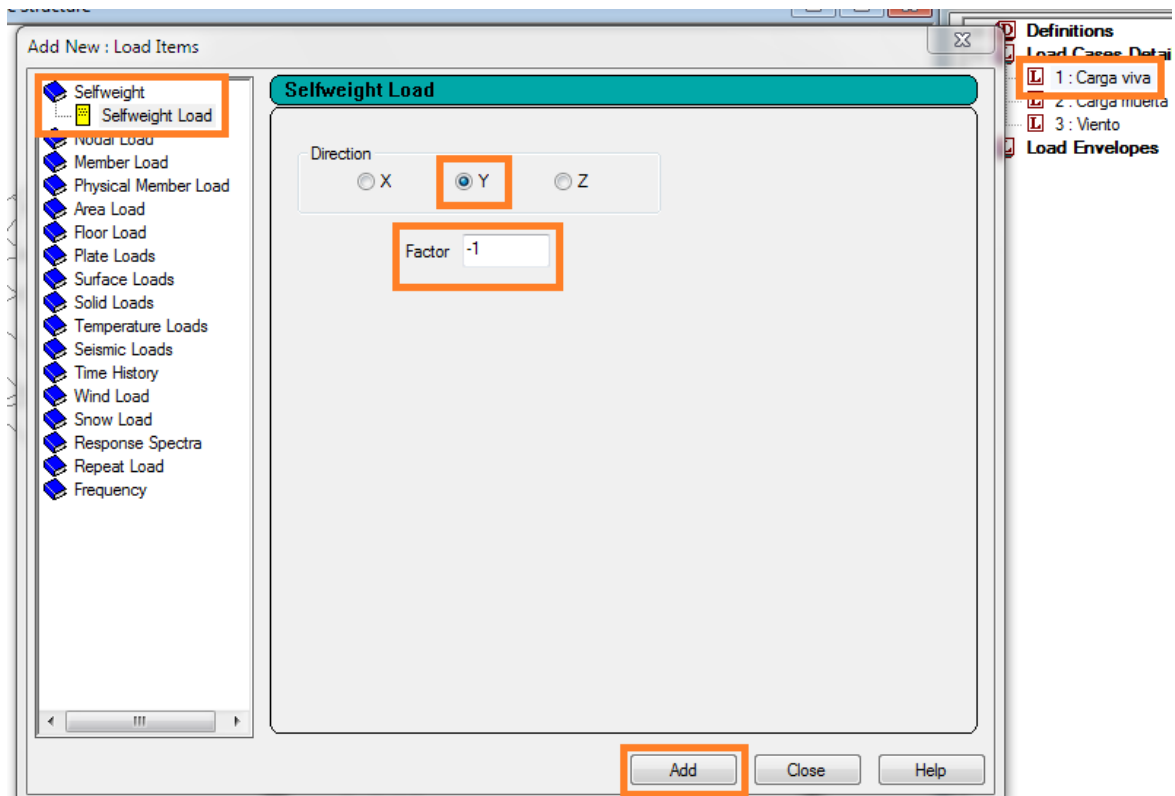
In the case of the dead load, which in this case refers to the weight of the cover and the way this will rest on our members; will click on the NODAL LOAD option, if the armor; and select Fy which is on the axis will reload our deck, this option will add the amount of KG coming down on node negative way, because as we explained the axis will be positive, while the charge is negative and finally we click ADD.
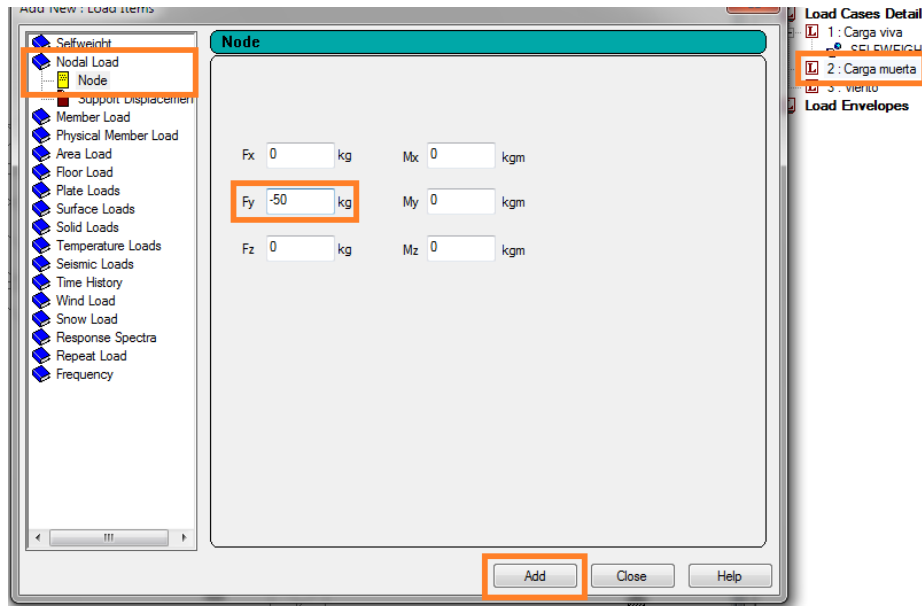
To assign this burden, we stand in the list of charges, select the load assigned as dead, we select the nodes to which we apply the burden and we click on the ASSIGN TO SELECTED NODES option and finally click ASSIGN.
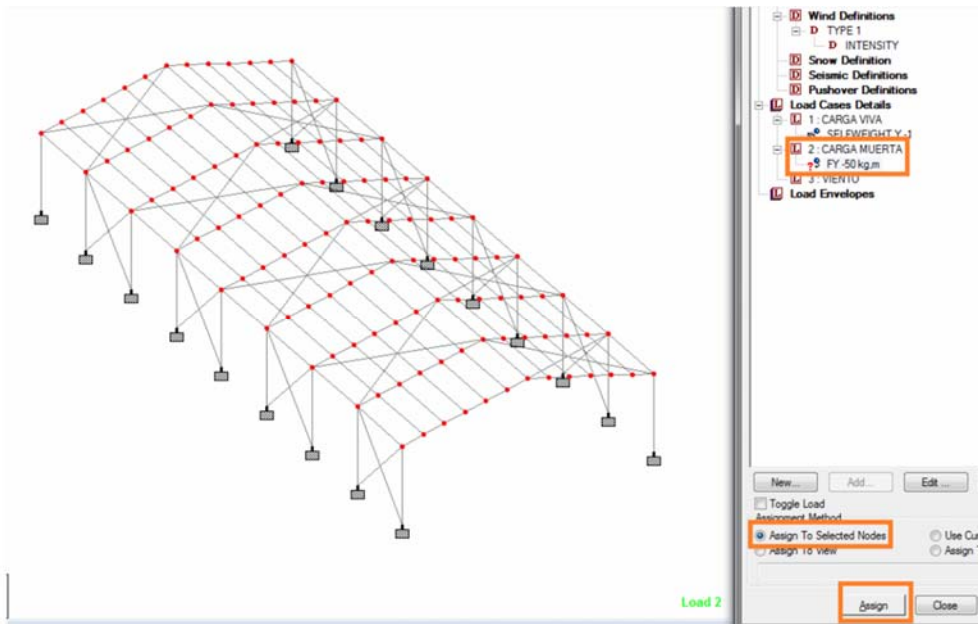
In order to add wind load, you must first enter a factor of intensity, it depends on the place where the project is located.

To assign this value, give DEFINITIONS Double-click the option, where we will display a list of options and then we put on WIND DEFINITIONS and we click on ADD.
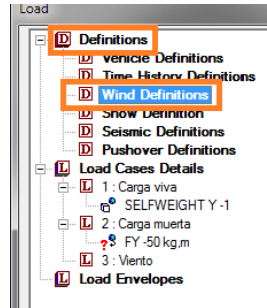
Once we ADD, opens a popup window called ADD NEW: WIND DEFINITIONS, where the number of load generating wind that are generated, it will give equal and just click ADD.
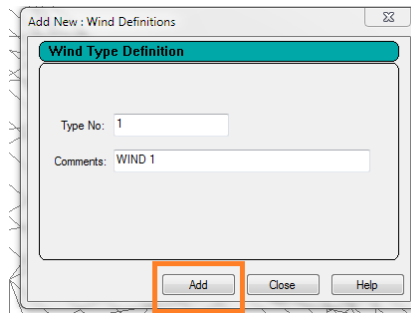
Now in our list of charges in the WIND DEFINITIONS option, a new listing that called TYPE 1 which is the kind of wind that just add, as a next step we put ourselves in the TYPE 1 and then we click on ADD appears.
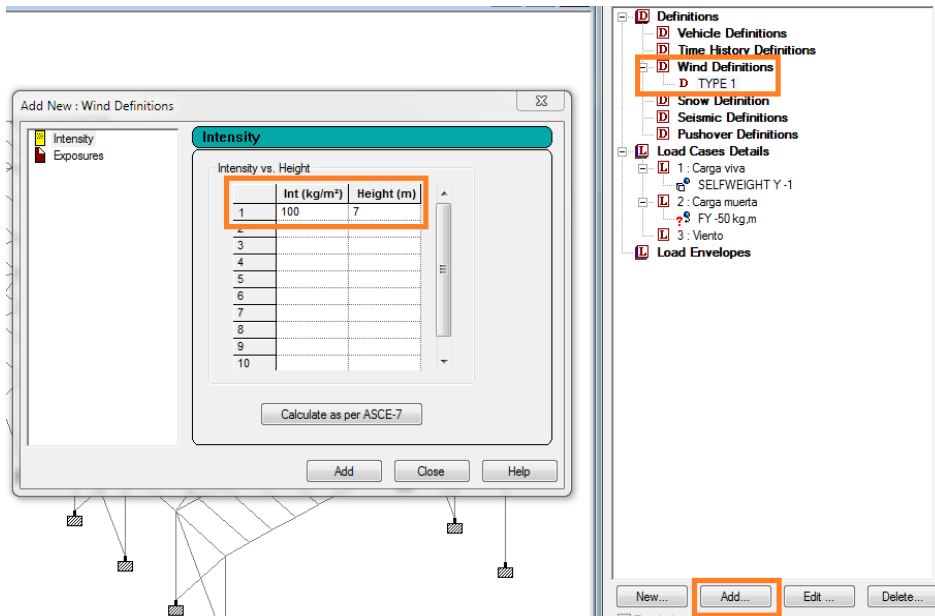
After this procedure, we stand on the load 3 (wind) and we click ADD, we will open a popup window called ADD NEW LOAD ITEMS which contains the list explained above, select the WIND LOAD option which will bring us a new window called ADD NEW LOAD ITEMS, within this range we assign to which we want to take the wind into our building; what we do is set the range on the Y axis, to take this as the minimum range 0 (from the floor) and a maximum range will enter the greater height of our element in this case is 5 m. and finally we click on ADD.
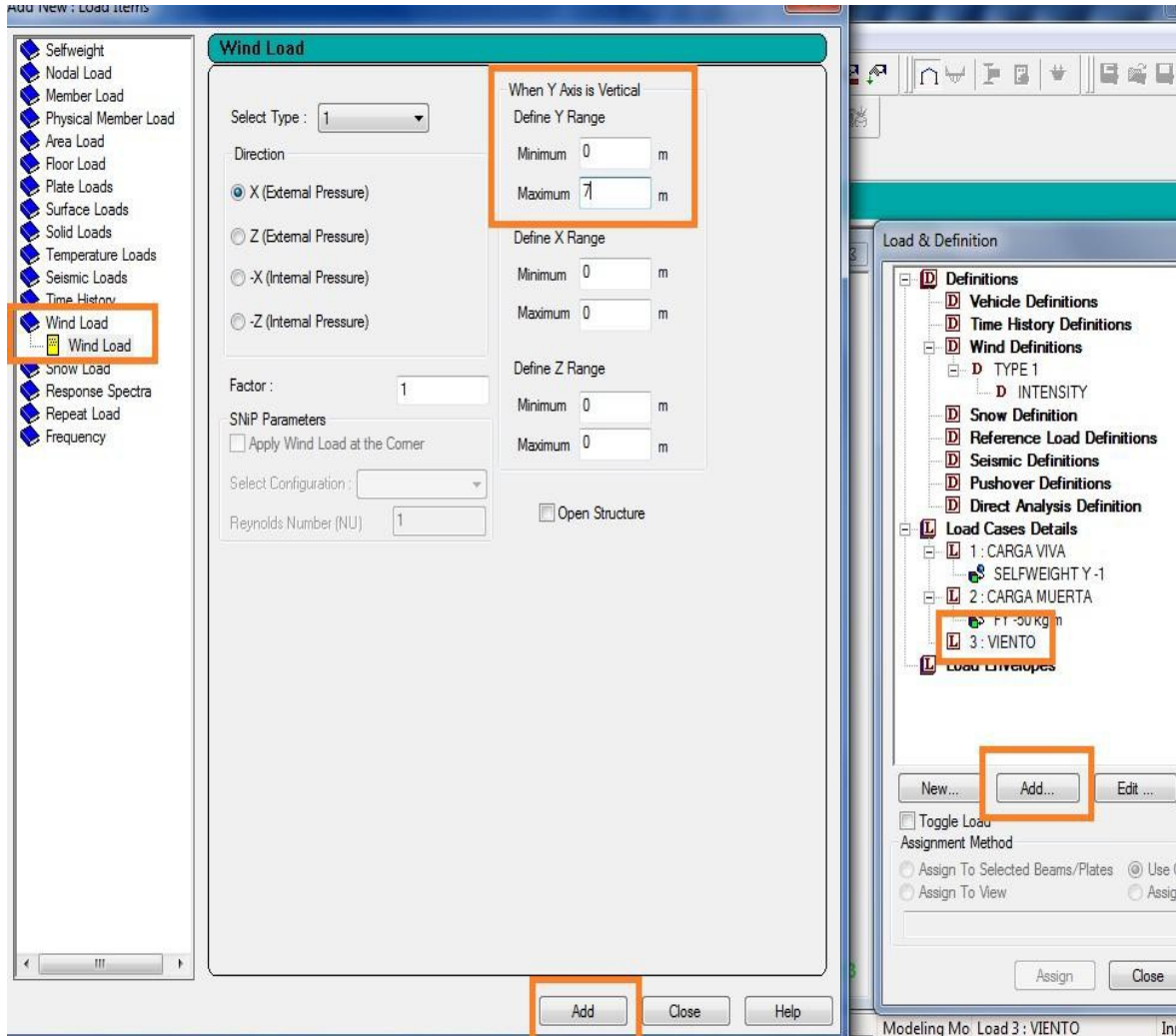
Having done this for wind load, should not be assigned to any of the elements, as this is determined by an address which was declared earlier in the screen shown above. Importantly, this happens only in the case of wind load, the two previous loads and you want to show in the project, whether they should be assigned to the components of the geometry of the project.
This part ends all matters relating to the placement of loads within the project, the next step is to determine the deformations that have been generated in our element by force loads that have been assigned.

Finally we generate a combination of loads, ie generate a single charge which included all that we have previously generated.

For this, we have to select LOAD CASES DETAILS tab will click on the button with the ADD option.
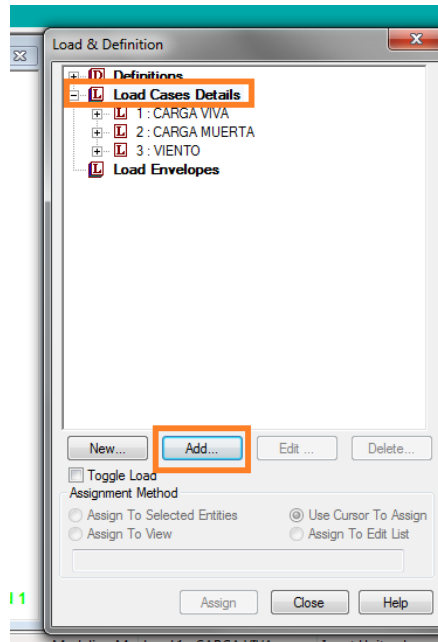
Where we can generate a new window called ADD NEW LOAD CASES, will click on the DEFINE COMBINATIONS option to do this change the interface from which we add the combination of loads within the dialog box called CASES AVAILABLE LOAD must appear the previously generated loads (live load, dead load and wind).
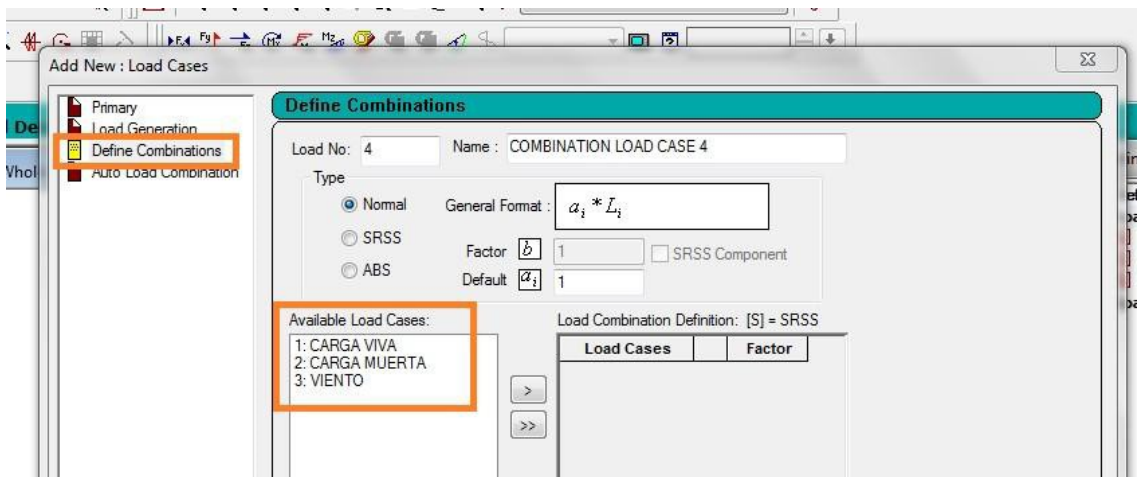
To define the loads that are within our combination must enter them with the icon on the right side of the listing of our loads (>) for this first select the load we want within our combination and once labeled (giving a click) give one more click the (>) icon.
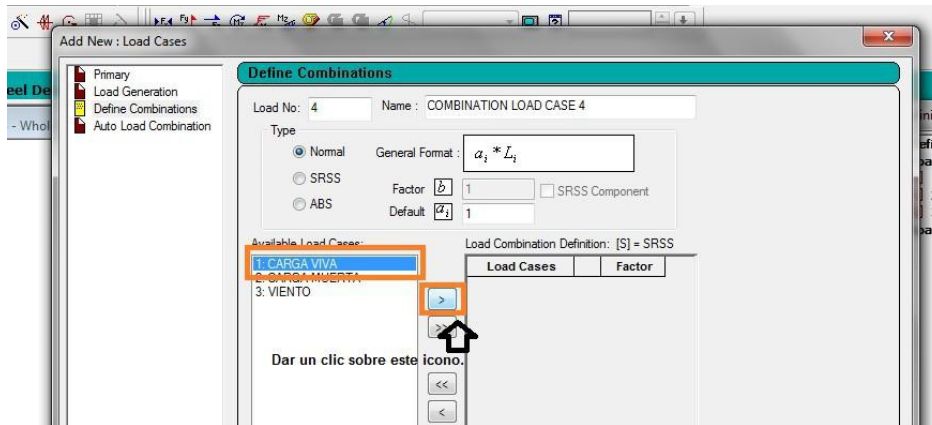
Once placed on the right side loads that will make up our combination, we include a factor, which refers to dictate we Complementary Technical Norms for this specific case because our steel factor is 0.9, so we will include that FACTOR value in the column for each of our burdens, once the procedure done will click on the ADD option.
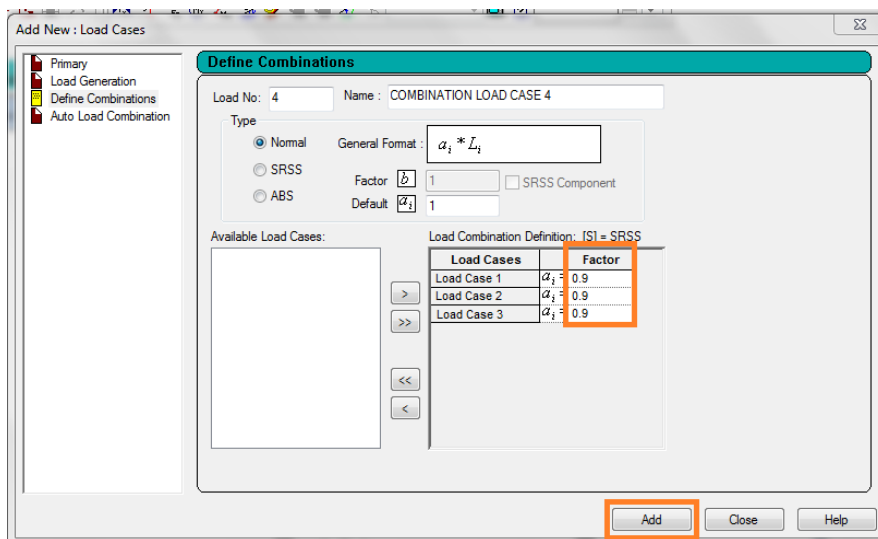
And now, our combination will appear in our list of charges.
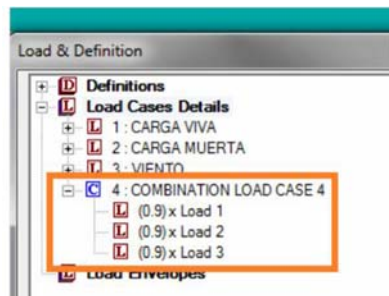
4.3.4 View Element Deformation

To make this process as a first step we analyze our structure, which is necessary to run an analysis within the program, because what has been done so this step is only allocation of materials and loads, without actually reflect whether these Elections are right or wrong.

As a first step we will go to the COMMANDS option found within our toolbar, select the ANALYSIS option and select the option PERFORM ANALYSIS.
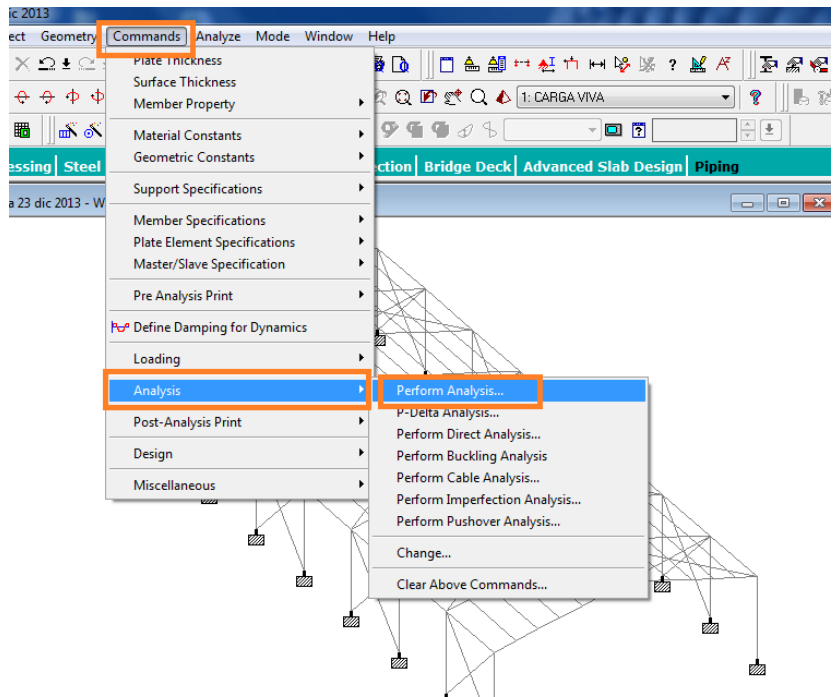
Once this done, we will open a popup window called PERFORM ANALYSIS, where we indicate what we want to analyze, we will click on the ALL option click OK.
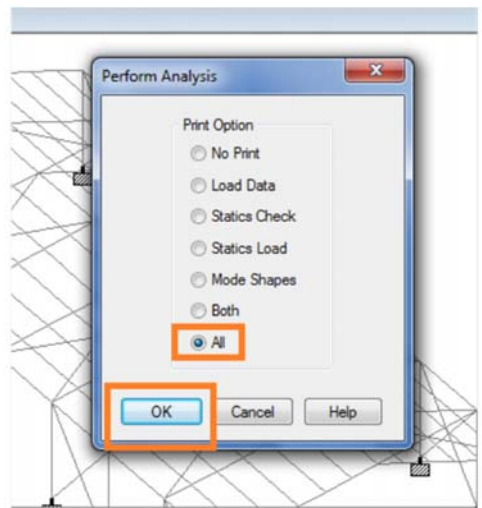
This completes the indication of an analysis, although no analysis has been run for this, on our toolbar we click on the ANALYZE option and click on the only option is RUN ANALYSIS exists or can be generated also CTRL +5.
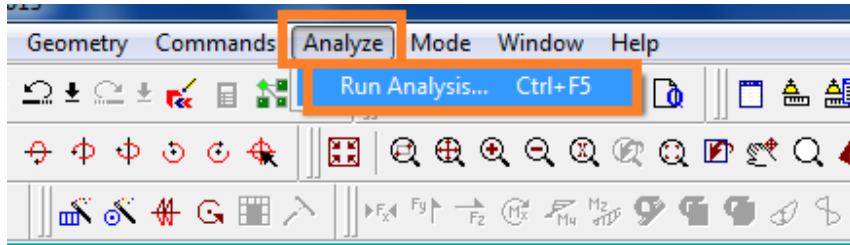
In doing so, the analysis sheet, where the elements that we contain within our drawing shown is generated, and the procedure we have done to reach those results. This window is called STAAD AND ANALYSIS DESING. It is important to verify that in this result sheet 0 number of errors are handled, otherwise you will have to go back to the drawing geometry and correct the irregularities.
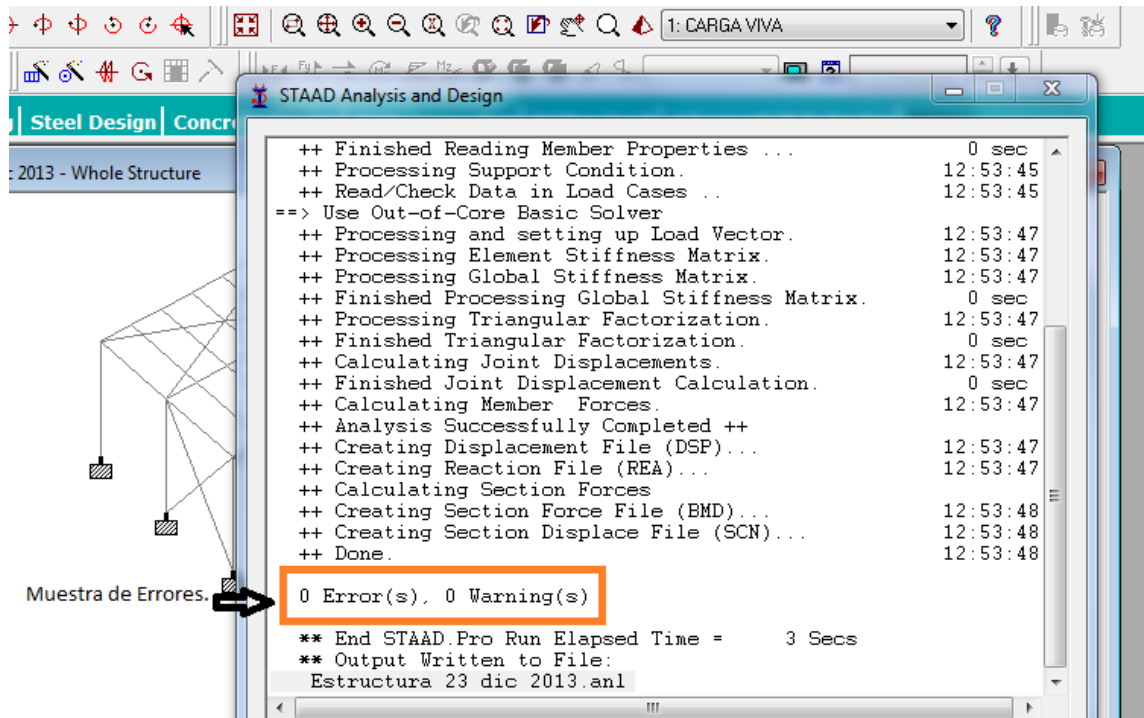
With no errors, we will continue with the development of the work, which is solely to show the results which gives us the program in terms of deformations once we applied loads, to determine if the structure is having an allowable deformation within our regulations.

Which depends on the type of material, type of construction and dimensions of this; where we apply the Regulation of Complementary Standards. (Table 7.1. Maximum permissible vertical displacements in structural elements).

The next step is to see the dimensions of movements that were generated in the structural elements when implementing different loads above, for this we must change the display mode within our interface; we are currently in the MODELING interface to switch to postprocessing select this option from the options bar we have at the top of our drawing.
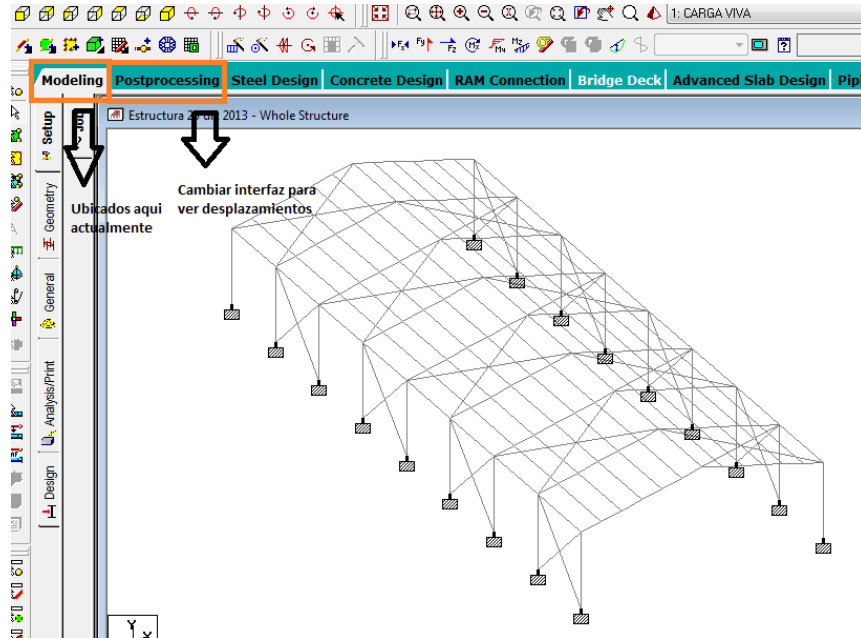
Once we change our working interface, the window that is generated has also changed, as this will allow us to do other things that were previously impossible.

Once we got to this interface, we can now see if the maximum offset values that were generated by assigning different loads.

For this we go to the RESULTS option that appears in our taskbar and then we click on the VIEW VALUE option.
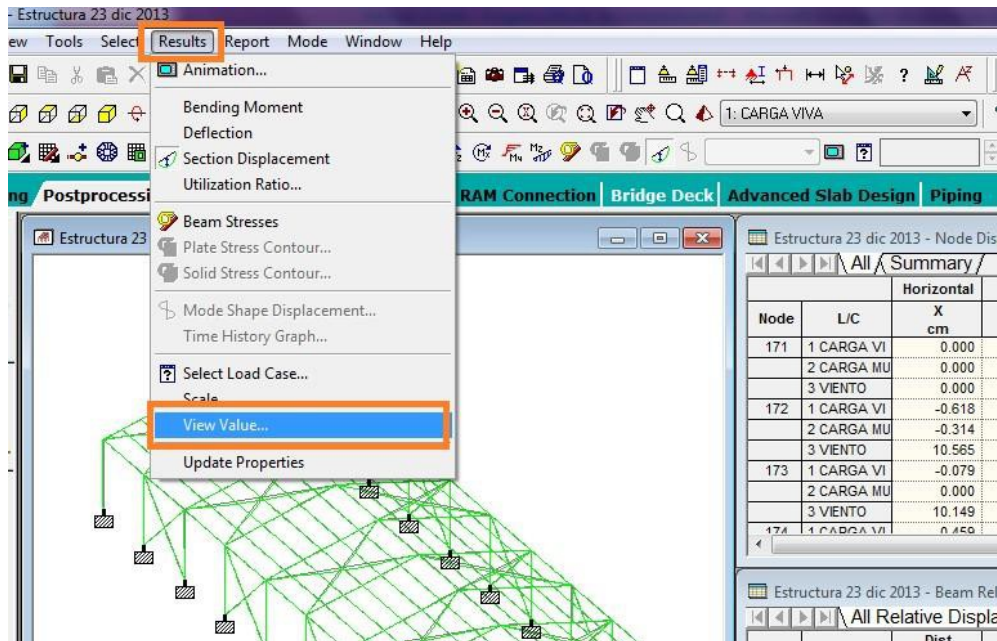
When you select this option we will generate a popup called ANNOTATION, and to display the values will click on the BEAM RESULTS tab, and within DISPLACEMENT options, we will click on the box Resultant MAX (maximum performance) and finally click Annotate.
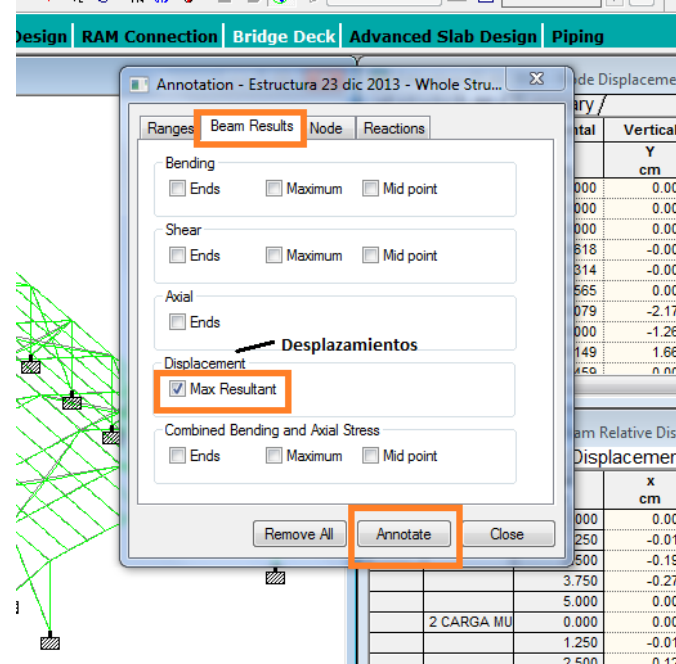
Once the above procedure we perform our construction is shown with the maximum displacements (which were selected) both in the columns and beams of our element.
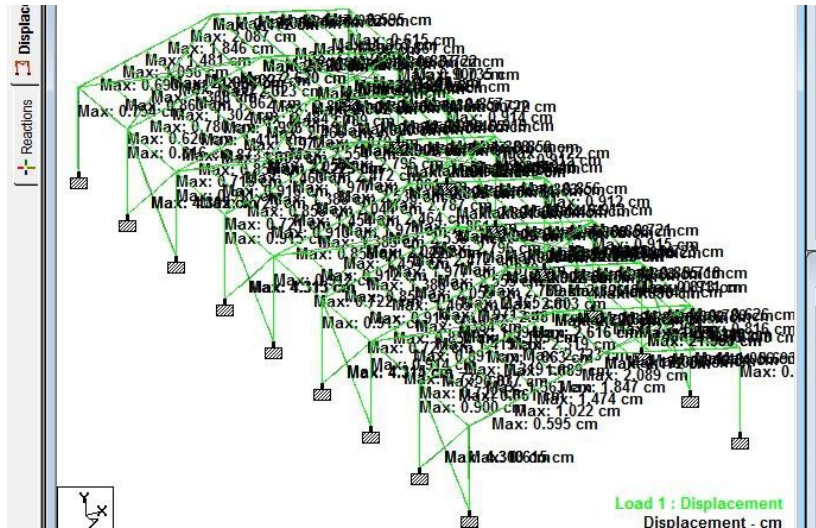
We can realize that the information we are getting is too much, so our data are pasting with each other, so we are looking only appreciate the vertical displacements (columns) that are positioned on the axis of the Y` s; for this we select, for this we will go to the SELECT tab in the tool bar and then select BEAMS PARALLEL TO without clicking the three axes we have (x, y, z) were deployed and select to see the columns axis Y.
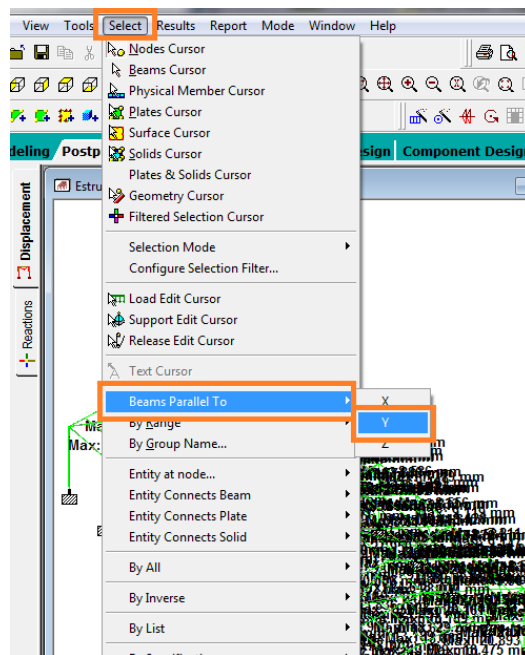
Once you have selected the columns, which are the parallel beams on the Y axis, we will go to the VIEW option found within our toolbar and will click on the VIEW ONLY SELECTED OBJECTS option; and so the lines are not selected, which are those that are not currently using went out.
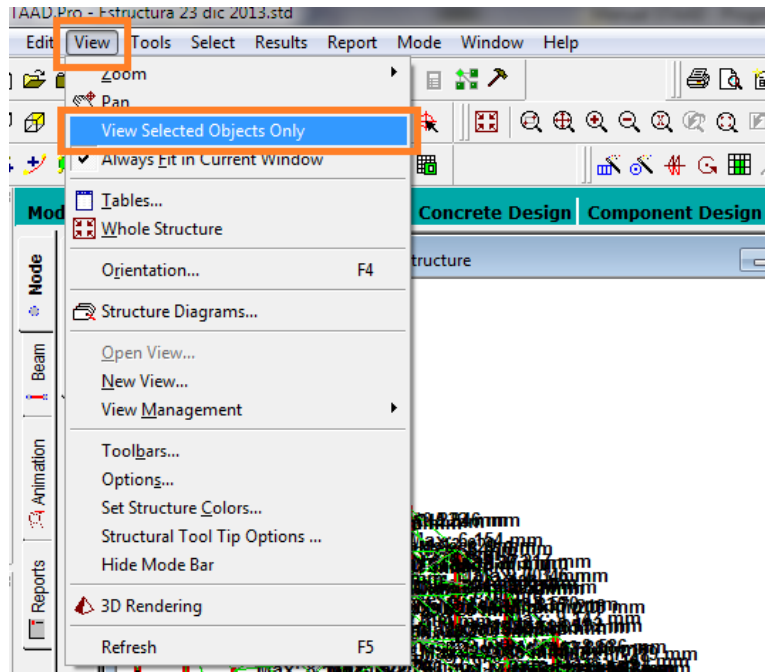
Now, each time I made this process we are seeing in our interface, the maximum displacement that are taking the columns. Achieved which follows seen.
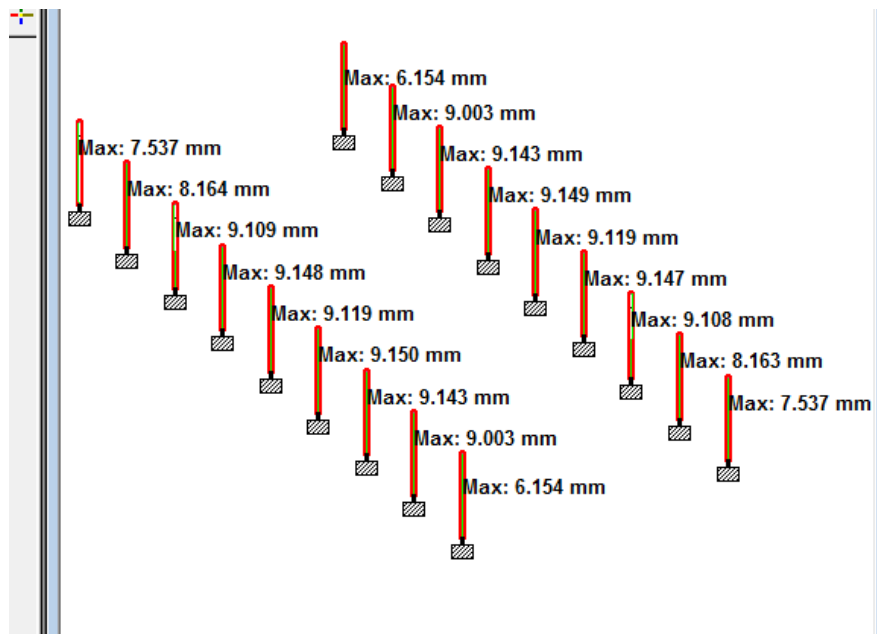
4.4. Application of seismic structures factors

In this exercise, a way to apply seismic factors in STAAD PRO V8i structure without the need for a study of soil mechanics, or direct reading with GPS, on the specific point of study show.

This method should be taken with reservation because while methods that are acceptable under certain criteria used are not 100% accurate but for practical purposes we can generate approximate results rather reality.

In this case, make use of two tools is to use one window **Response Spectrum** and the other will support the program overall geo-positioning **Google Earth** in its latest version.

Next step will describe the design of a tipocolumna element analyzed with consideration of finite elements and the element concerned with seismic spectra.

Procedure

1. First we proceed to generate the geometry of our structure or element in this case as stated for our exercise we will work with a guy in the option element to plane geometry with the following characteristics:

❖ Three 15 meter vertical elements one placed above the other.
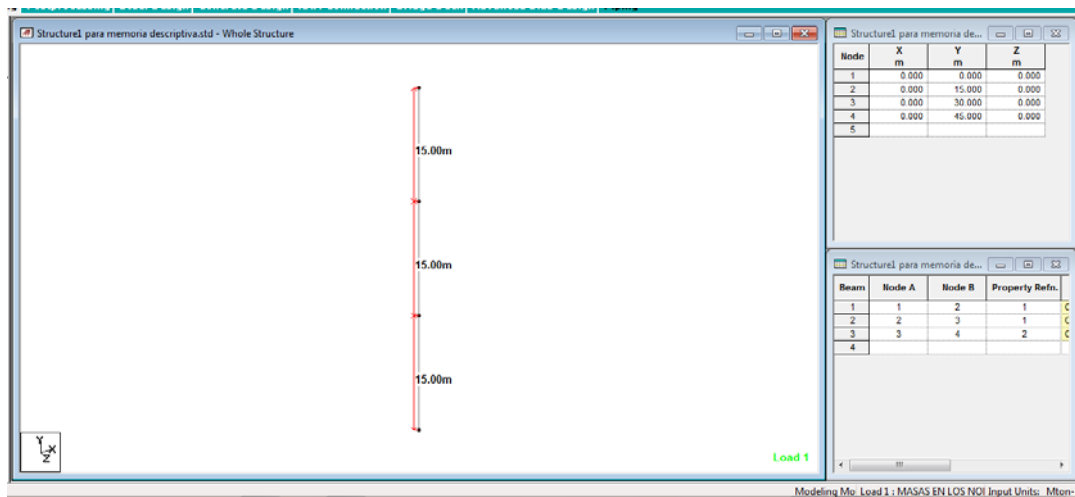
FIG. 1 geometric element composition.

Since we have our well-defined geometry continue with the process of defining the material of our items, sections and assigning media.

This action is done from the Menu Structure Tool submenu Property Page, click the button

Sets the dialog screen will be displayed, in this define the geometric shape we want for our element and size we require.
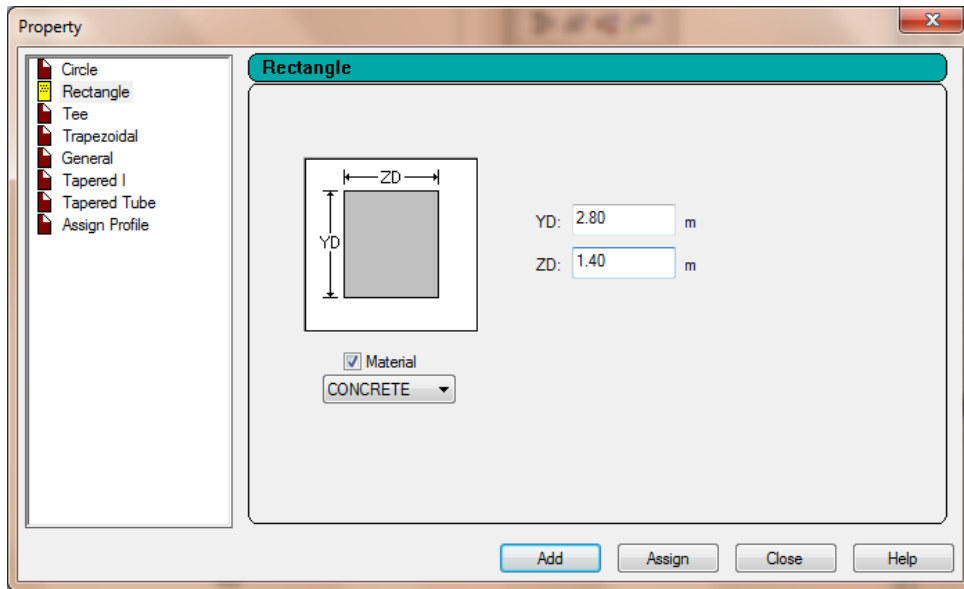
Fig. 2.-dimensional assignment of particular profiles.

2. Assign the material; for this specific example, will be as follows:

❖ Sections 1 and 2 of concrete with dimensions YD = 2.8m, 1.4m ZD =

❖ Section 3 of concrete with dimensions YD = 2.22, ZD = 1.11m
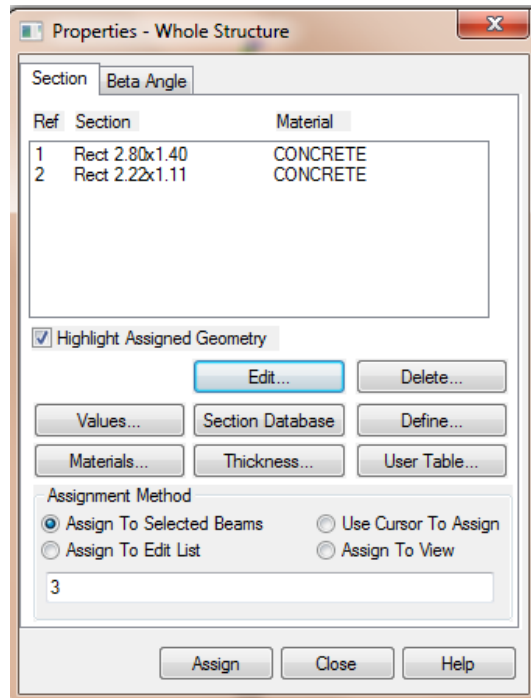
Fig.3. Definition of material elements.

If the material allocation process was successful our element should look like the following figure.
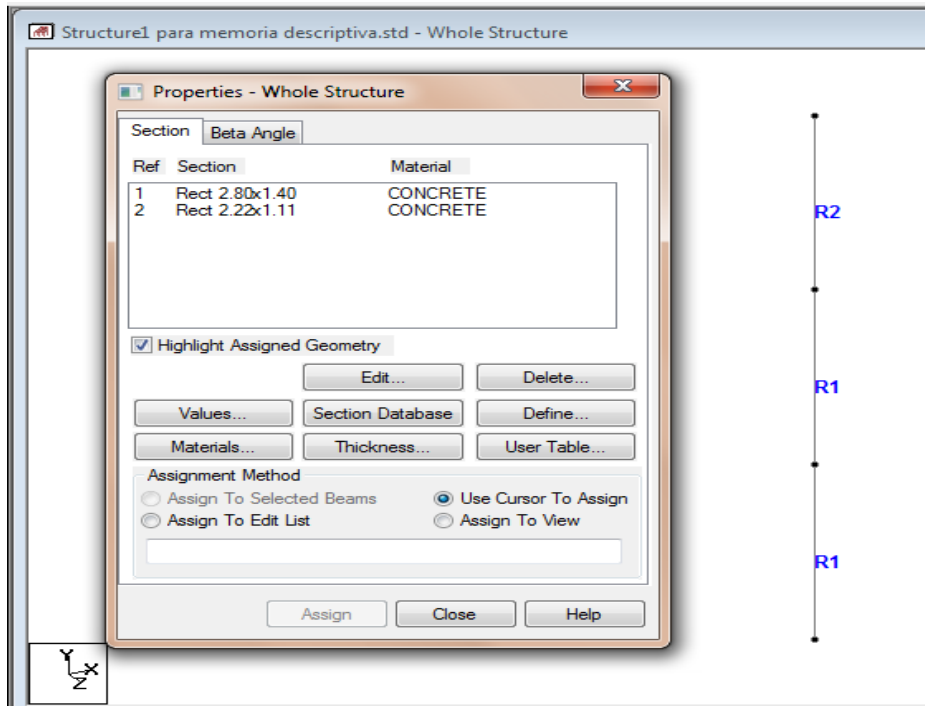
Fig. 4. material Assignment elements.

3. After that proceed to the allocation of support in a way very similar to the allocation of our material structural elements. This procedure is conducted from the Structure Tools menu, sub menu Support Page.

    For the particular case of this exercise two different types of media were used one will be a predefined type of support **FIXED** and three more will be the type **FIXED BUT,** this type of support is assigned to a node when it wants to establish degree of freedom

    shift and we can choose the window **Create Support,** Fixed sub menu

    But, for our exercise the degree of freedom is assigned in **FX,** but you can assign any direction or direction required.

If the allocation procedure was performed properly supports our exercise should look like the following figure.
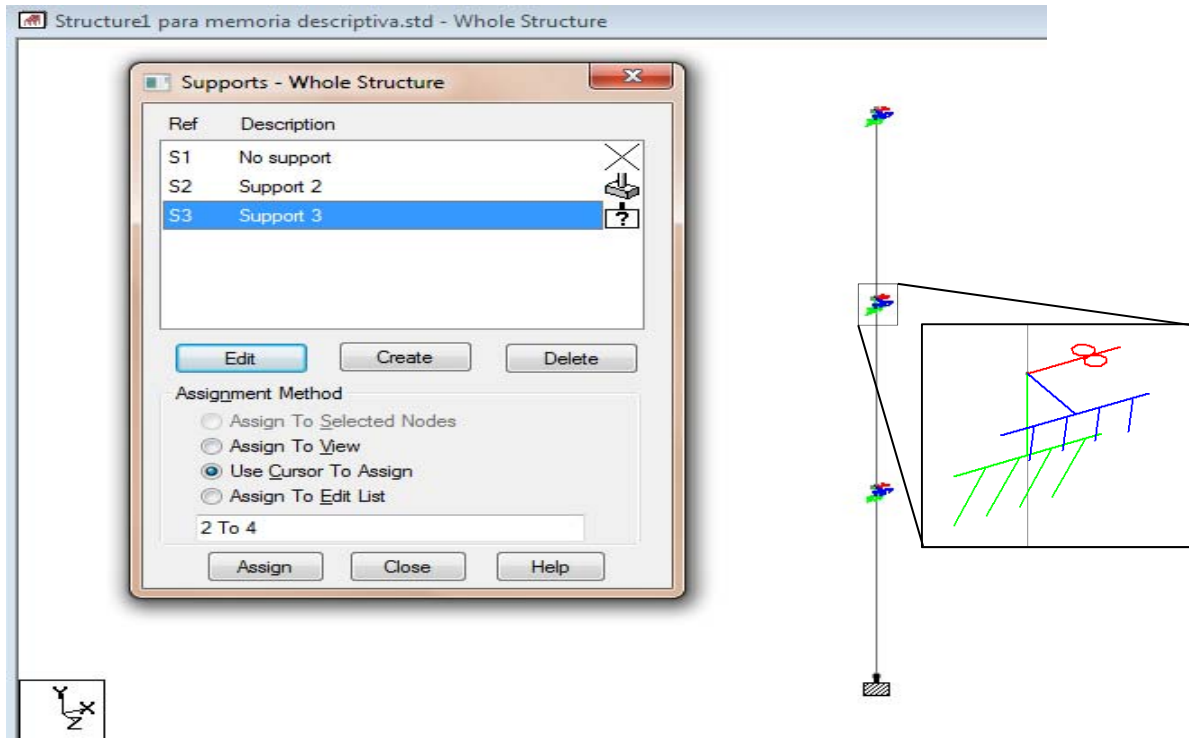
FIG. 5 Assignment supports our elements.

Note: we can see that at nodes 2, 3 and 4 a symbol of three colors is placed with a particular symbology, they are indicating that it is a node with freedom of movement, should not be viewed just enough to activate the label Nodes, Support.

4. Now that we have our well defined both geometrically in props and materials, proceed to the definition and application of forces and loads acting on different elements of our structure.

In the case of our particular exercise will only use lateral loads on the nodes which represent our structure seismic forces. Later these forces will be affected by factors of seismic forces.

Following are the steps for performing the method of the above actions.

1. Assign the following nodal forces:

❖ Nodes 2 and 3 = FX 400
❖ Node 4 = FX 200

For this, we first define our first charge which we call MASS NODES (FORCES AS), do this from the Structure Tools menu, submenu **Load Page-**Load Cases Details-New-Load Case. If done correctly the action will see a screen

as in the figure below.
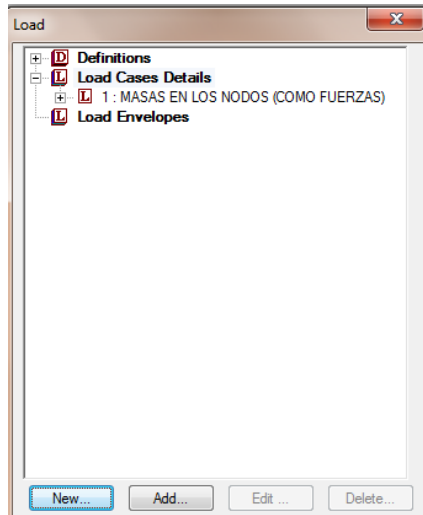
FIG. 6 Defining our group of forces.

2. After our arguments in the group of forces that define and give just click the button **New** to create the forces that we act on our structure, our exercise we will create two nodal forces:

❖ In nodes 2 and 3 FX = 200
❖ In node 4 FX = 400

This can be done from the window **Load**-New Items-Nodal-Load Load.

3. Immediately after this step we have to do is specify our analysis parameters, which can be determined from **Load**-New-Frequency Load Items- and the two options are Rayleigh Frequency and Modal Calculation will be elected.
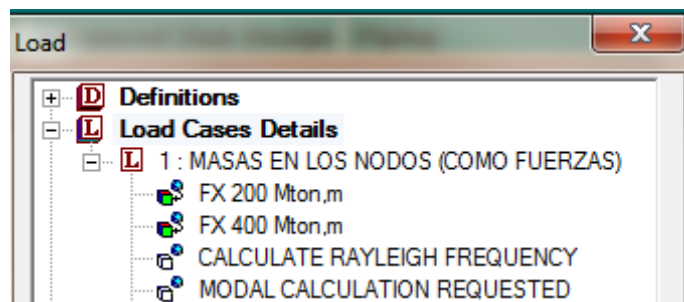
Fig.7. Assigning nodal forces and calculation parameters.

From this point the only thing we lack is our define seismic spectrum. We do this from the window **Load**-New Items-Load-Response Spectra, correctly follows the procedure will see a screen like the one shown below.
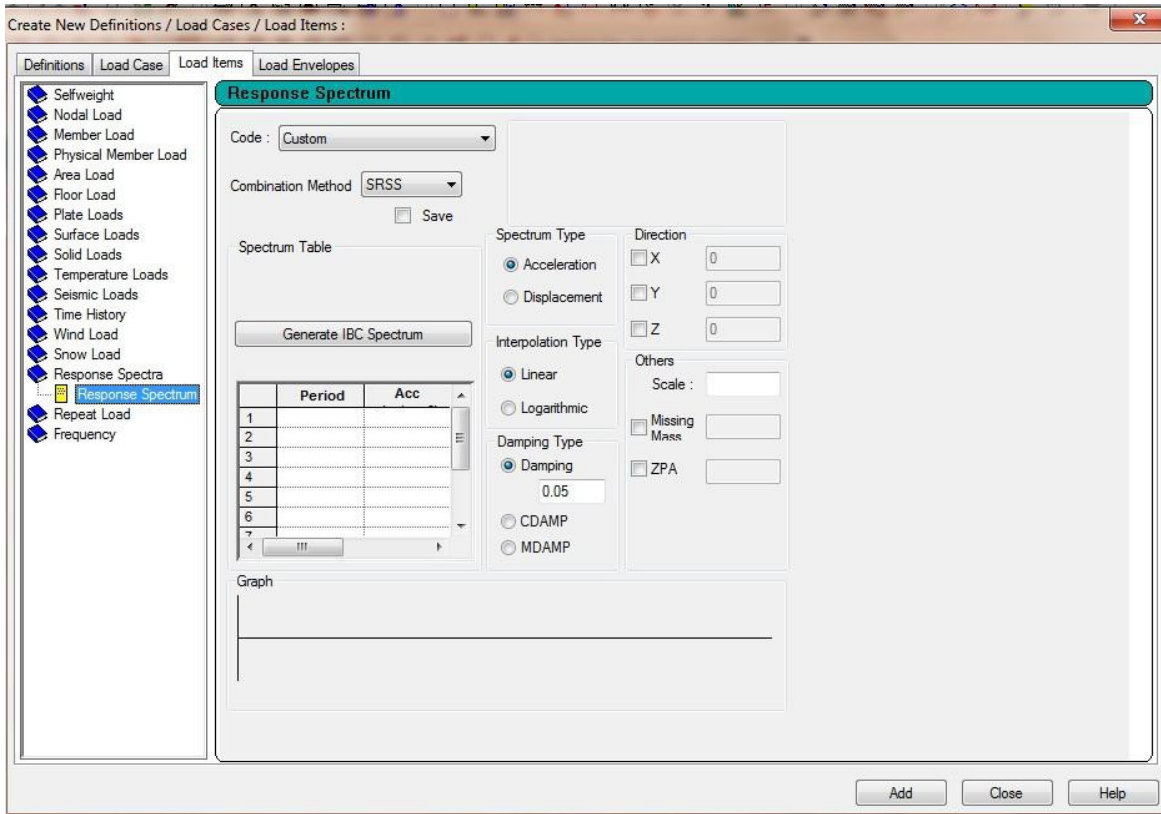
FIG. 8 Calculation of Response Spectra.

For this step the STAAD PRO V8i program asks us to introduce seismic spectra obtained from a study of soil mechanics or give us the option we generated a list from the same program with the IBC Spectrum Generate button, so we click on that button and we were presented a screen like the following.

FIG. 9 Seismic Parameters Generation Spectrum.

We can see that on this screen shows one **Latitude** and **Length** and a predetermined **Select Zip** already preset in this screen you can modify the Select Zip and we automatically generates a latitude, longitude, and values for the parameters S1 and S2;
also have an opportunity to change the interval at which we want our results to generate spectra and the duration of the seismic event.

To start this procedure first thing we have to do is modify the geographic coordinates since we are not assigned by default we want; *But* *how we can get this data?*

There are several ways to obtain the geographical coordinates we need to generate the data (longitude and latitude) STAAD.Pro V8i program that asks us as a requirement for our seismic spectrum, there are direct methods and indirect methods.

The most common direct methods are to conduct a survey with a total station, being the owner of a Garmin GPS or a satellite, or know how to perform an astronomical orientation for which a theodolite must be taken first and a current almanac and neither mentioning that depend on it is not a cloudy day.

Because of the complications that can result in the implementation of any of these direct methods above are indirect methods which in most are programs that provide the coordinates depending on the region you want, some of these programs are free and some others we can buy. In this case the use of a free license program and extensive use is proposed, this program is **Google Earth**.

Fig. Presentation 10 Display Google Earth

If you already know this program or are somewhat familiar know that in the bottom of the main screen is going to allow you to see the coordinates of the point you're pointing the pointer but default gives the coordinates as Rumbo, but what we want is the coordinates in decimal degrees.

Below are examples of the coordinates in the decimal form and shape Degrees, Minutes and Seconds (Rumbo) are given.

Coordinates in decimal.

Latitude   19.432620,   -99.133218
Long

Coordinates as Degrees, Minutes and Seconds (Rumbo).

19 ° 25'57 "N, 99 ° 07'59" W (west)

For us, the coordinates that will help us be the coordinates in decimal and to make the Google Earth we generate this type of coordinates we have to set it up very easily just have to open the Tools menu, select options and view 3D to show lat. / long and change of Degrees, Minutes, Seconds to Decimal Degrees.
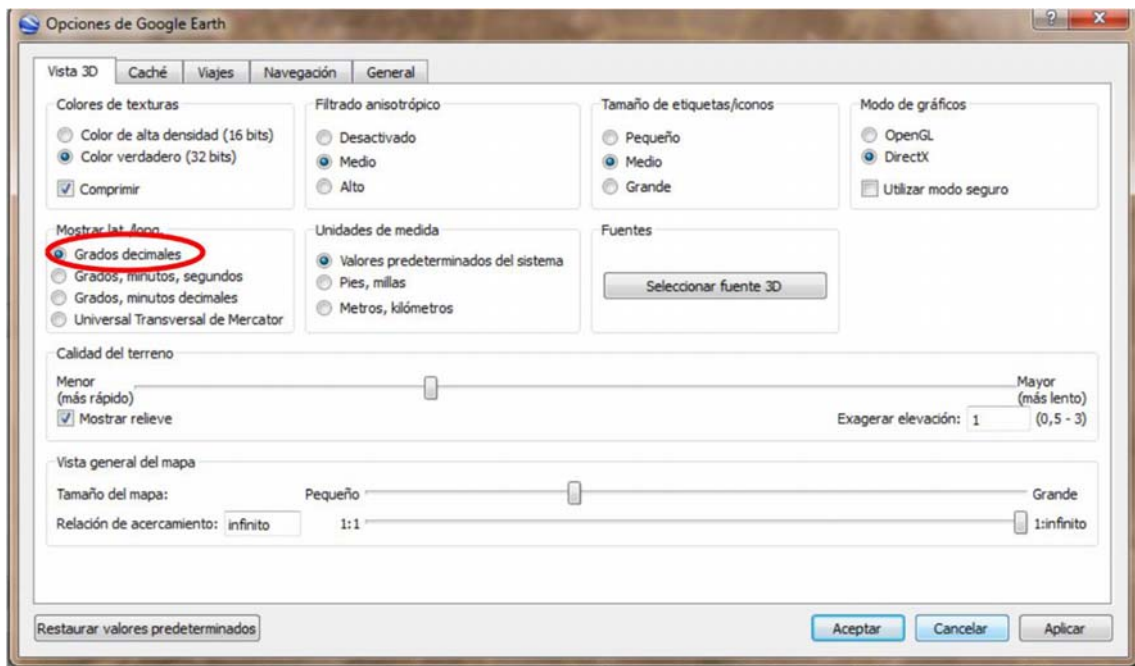
Fig. Configuration 11 coordinates in decimal degrees.

Since we have configured the way coordinates that remains is selecting the area where our structure is found and get the coordinates that Google Earth creates us is.

This does not have to be as accurate as we know that seismic frequencies do not vary much in small sections, so just simply choose an approximate point.

Fig. 12. location coordinates in Google Earth.

The next step is to capture the coordinates in the Spectrum Parameters window so in our case should be.

Latitude and Longitude = 19.435738 = -99.143957m, we click on the Calculate S1, SS button, modify the duration of the seismic event at 60 seconds and 1 second intervals so that we generate only 60 points of the spectrum, then you click on the Generate button Spectrum.

After these steps STAAD generate a graph to show the intensities (displacement, velocity and acceleration) of the seismic event throughout its duration.

The chart shown below is generated corresponding to the spectrum for our particular exercise.
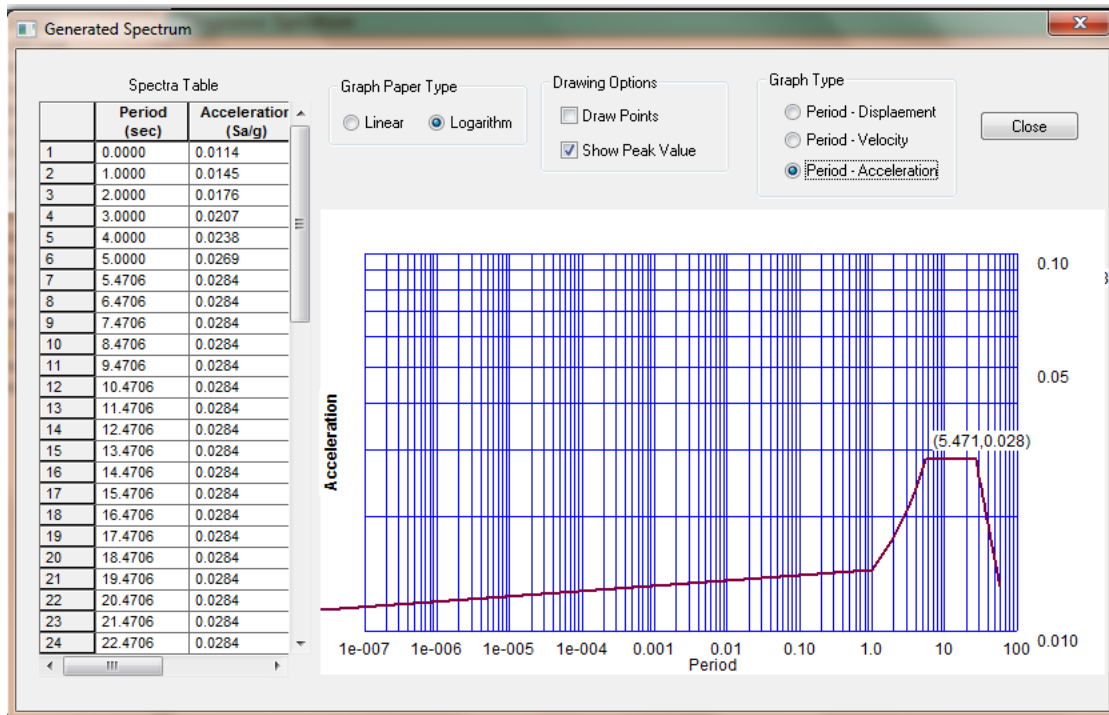
Fig. Grafica 13 seismic acceleration spectrum.

This chart is for informational purposes only so that we proceed to close it.

After close back to the window Response Spectrum with Spectrum Table table and with data and a chart on this window you can also choose the Combination Method spectrum with the forces being the most common SRSS and CQC, the Code section is in Custom , choose the type of interpolation between Linear and Logarithmic, note the X direction with a value of 1 and we click on the Add button.

This is the last step as we can see after assigning spectrum applied in our window **Load.**
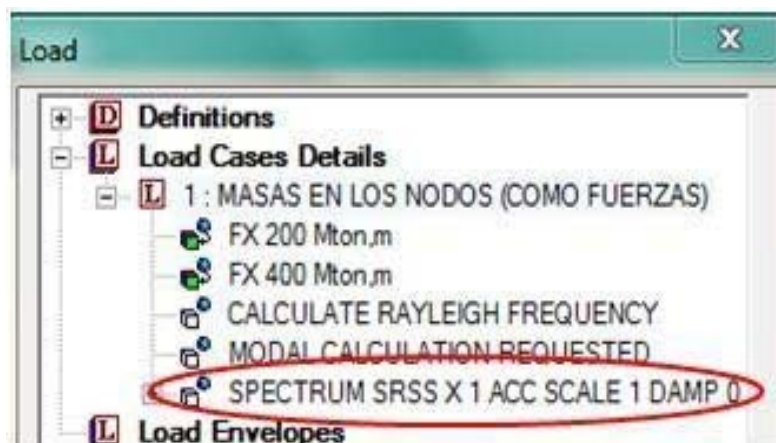
Fig.14 seismic spectrum assigned.

This concludes the process of assigning seismic, our structure spectrum.

The process just described throughout this exercise for spectrum allocation is the same for any kind of structure.

In this exercise only aims to show the process of spectrum allocation in the simplest form so no load combinations or other forces are carried as air, or a variety of combinations of loads that may occur include snow in our structures.

To complete the analytical text editor STAAD attached. (In the case of editor spectra are generated at intervals of 0.05 sec. Duration of the event and 10 sec.'s Why there are many interpolation points of the spectrum.)

STAAD SPACE
ENGINEER JOB INFORMATION START DATE 09-Aug-12
END JOB INFORMATION
INPUT WIDTH 79
JOINT
COORDINATES UNIT
METER MTON
1 0 0 0; 2 0 15 0; 3 0 30 0; 4 0 45 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
DEFINE MATERIAL START
ISOTROPIC 2.02147e
CONCRETE E + 006 0.17
POISSON
DENSITY 2.40262
ALPHA DAMP 0.05
1e-005
END DEFINE MATERIAL
AMERICAN MEMBER
PROPERTY January 2nd PRIS YD
ZD 2.8 1.4
PRIS YD ZD 3 2.22 1.11
CONSTANTS
SUPPORTS ALL MATERIAL CONCRETE
1 FIXED
2 FIXED BUT TO 4 FX
1 MASS LOAD NODE (AS FORCES)
JOINT LOAD
4 FX 200
March 2 FX 400
CALCULATE FREQUENCY RAYLEIGH
MODAL CALCULATION REQUESTED

SRSS SPECTRUM SCALE 1 x 1 ACC 0 LIN MY DAMP
0 0.106863; 0.05 0.108356; 0.1 0.109848; 0.15 0.111341; 0.2 0.112834;
0.25 0.114326; 0.3 0.115819; 0.35 0.117312; 0.4 0.118804; 0.45
0.120297;
0.5 0.12179; 0.55 0.123282; 0.6 0.124775; 0.65 0.126268; 0.7 0.12776;
0.75 0.129253; 0.8 0.130746; 0.85 0.132238; 0.9 0.133731; 0.95
0.135224;
1 0.136716; 1.05 0.138209; 1.1 0.139701; 1.15 0.141194; 1.2 0.142687;
1.25 0.144179; 1.3 0.145672; 1.35 0.147165; 1.4 0.148657; 1.45 0.15015;
1.5 0.151643; 1.55 0.153135; 1.6 0.154628; 1.65 0.156121; 1.7 0.157613;
1.75 0.159106; 1.8 0.160599; 1.85 0.162091; 1.9 0.163584; 1.95
0.165077;
2 0.166569; 2.05 0.168062; 2.1 0.169555; 2.15 0.171047; 2.2 0.17254;
2.25 0.174033; 2.3 0.175525; 2.35 0.177018; 2.4 0.178511; 2.45
0.180003;
2.5 0.181496; 2.55 0.182989; 2.6 0.184481; 2.65 0.185974; 2.7 0.187466;
2.75 0.188959; 2.8 0.190452; 2.85 0.191944; 2.9 0.193437; 2.95 0.19493;

> The last point on this line can be interpreted as an error by the system so the editor will have to move to a new line to avoid error in the run

3 0.196422; 3.05 0.197915; 3.1 0.199408; 3.15 0.2009; 3.2 0.202393; 3.25 0.203886;
3.3 0.205378; 3.35 0.206871; 3.4 0.208364; 3.45 0.209856; 3.5 0.211349;
3.55 0.212842; 3.6 0.214334; 3.65 0.215827; 3.7 0.21732; 3.75 0.218812;
3.8 0.220305; 3.85 0.221798; 3.9 0.22329; 3.95 0.224783; 4 0.226276;
4.05 0.227768; 4.1 0.229261; 4.15 0.230754; 4.2 0.232246; 4.25 0.233739;
4.3 0.235232; 4.35 0.236724; 4.4 0.238217; 4.45 0.239709; 4.5 0.241202;
4.55 0.242695; 4.6 0.244187; 4.65 0.24568; 4.7 0.247173; 4.75 0.248665;
4.8 0.250158; 4.85 0.251651; 4.9 0.253143; 4.95 0.254636; 5 0.256129;
5.05 0.257621; 5.1 0.259114; 5.15 0.260607; 5.2 0.262099; 5.25 0.263592;
5.3 0.265085; 5.35 0.266577; 5.36944 0.267158; 5.41944 0.267158; 5.46944
0.267158;
5.51944 0.267158; 5.56944 0.267158; 5.61944 0.267158; 5.66944 0.267158;
5.71944 0.267158; 5.76944 0.267158; 5.81944 0.267158; 5.86944 0.267158;
5.91944 0.267158; 5.96944 0.267158; 6.01944 0.267158; 6.06944 0.267158;
6.11944 0.267158; 6.16944 0.267158; 6.21944 0.267158; 6.26944 0.267158;
6.31944 0.267158; 6.36944 0.267158; 6.41944 0.267158; 6.46944 0.267158;
6.51944 0.267158; 6.56944 0.267158; 6.61944 0.267158; 6.66944 0.267158;
6.71944 0.267158; 6.76944 0.267158; 6.81944 0.267158; 6.86944 0.267158;
6.91944 0.267158; 6.96944 0.267158; 7.01944 0.267158; 7.06944 0.267158;
7.11944 0.267158; 7.16944 0.267158; 7.21944 0.267158; 7.26944 0.267158;
7.31944 0.267158; 7.36944 0.267158; 7.41944 0.267158; 7.46944 0.267158;
7.51944 0.267158; 7.56944 0.267158; 7.61944 0.267158; 7.66944 0.267158;
7.71944 0.267158; 7.76944 0.267158; 7.81944 0.267158; 7.86944 0.267158;
7.91944 0.267158; 7.96944 0.267158; 8.01944 0.267158; 8.06944 0.267158;
8.11944 0.267158; 8.16944 0.267158; 8.21944 0.267158; 8.26944 0.267158;
8.31944 0.267158; 8.36944 0.267158; 8.41944 0.267158; 8.46944 0.267158;
8.51944 0.267158; 8.56944 0.267158; 8.61944 0.267158; 8.66944 0.267158;
8.71944 0.267158; 8.76944 0.267158; 8.81944 0.267158; 8.86944 0.267158;
8.91944 0.267158; 8.96944 0.267158; 9.01944 0.267158; 9.06944 0.267158;
9.11944 0.267158; 9.16944 0.267158; 9.21944 0.267158; 9.26944 0.267158;
9.31944 0.267158; 9.36944 0.267158; 9.41944 0.267158; 9.46944 0.267158;
9.51944 0.267158; 9.56944 0.267158; 9.61944 0.267158; 9.66944 0.267158;
9.71944 0.267158; 9.76944 0.267158; 9.81944 0.267158; 9.86944 0.267158;
9.91944 0.267158; 9.96944 0.267158; 10.0194 0.267158; 10.0694 0.267158;

> The last point on this line can be interpreted as an error by the system so the editor will have to move to a new line to avoid error in the run

10.1194 0.267158; 10.1694 0.267158; 10.2194 0.267158; 10.2694 0.267158;
10.3194 0.267158; 10.3694 0.267158; 10.4194 0.267158; 10.4694 0.267158;
10.5194 0.267158; 10.5694 0.267158; 10.6194 0.267158; 10.6694 0.267158;
10.7194 0.267158; 10.7694 0.267158; 10.8194 0.267158; 10.8694 0.267158;
10.9194 0.267158; 10.9694 0.267158; 11.0194 0.267158; 11.0694 0.267158;
11.1194 0.267158; 11.1694 0.267158; 11.2194 0.267158; 11.2694 0.267158;
11.3194 0.267158; 11.3694 0.267158; 11.4194 0.267158; 11.4694 0.267158;
11.5194 0.267158; 11.5694 0.267158; 11.6194 0.267158; 11.6694 0.267158;
11.7194 0.267158; 11.7694 0.267158; 11.8194 0.267158; 11.8694 0.267158;
11.9194 0.267158; 11.9694 0.267158; 12.0194 0.267158; 12.0694 0.267158;
12.1194 0.267158; 12.1694 0.267158; 12.2194 0.267158; 12.2694 0.267158;

12.3194 0.267158; 12.3694 0.267158; 12.4194 0.267158; 12.4694 0.267158;
12.5194 0.267158; 12.5694 0.267158; 12.6194 0.267158; 12.6694 0.267158;
12.7194 0.267158; 12.7694 0.267158; 12.8194 0.267158; 12.8694 0.267158;
12.9194 0.267158; 12.9694 0.267158; 13.0194 0.267158; 13.0694 0.267158;
13.1194 0.267158; 13.1694 0.267158; 13.2194 0.267158; 13.2694 0.267158;
13.3194 0.267158; 13.3694 0.267158; 13.4194 0.267158; 13.4694 0.267158;
13.5194 0.267158; 13.5694 0.267158; 13.6194 0.267158; 13.6694 0.267158;
13.7194 0.267158; 13.7694 0.267158; 13.8194 0.267158; 13.8694 0.267158;
13.9194 0.267158; 13.9694 0.267158; 14.0194 0.267158; 14.0694 0.267158;
14.1194 0.267158; 14.1694 0.267158; 14.2194 0.267158; 14.2694 0.267158;

14.3194 0.267158; 14.3694 0.267158; 14.4194 0.267158; 14.4694 0.267158;
14.5194 0.267158; 14.5694 0.267158; 14.6194 0.267158; 14.6694 0.267158;
14.7194 0.267158; 14.7694 0.267158; 14.8194 0.267158; 14.8694 0.267158;
14.9194 0.267158; 14.9694 0.267158; 15.0194 0.267158; 15.0694 0.267158;
15.1194 0.267158; 15.1694 0.267158; 15.2194 0.267158; 15.2694 0.267158;
15.3194 0.267158; 15.3694 0.267158; 15.4194 0.267158; 15.4694 0.267158;
15.5194 0.267158; 15.5694 0.267158; 15.6194 0.267158; 15.6694 0.267158;
15.7194 0.267158; 15.7694 0.267158; 15.8194 0.267158; 15.8694 0.267158;
15.9194 0.267158; 15.9694 0.267158; 16.0194 0.267158; 16.0694 0.267158;
16.1194 0.267158; 16.1694 0.267158; 16.2194 0.267158; 16.2694 0.267158;
16.3194 0.267158; 16.3694 0.267158; 16.4194 0.267158; 16.4694 0.267158;

16.5194 0.267158; 16.5694 0.267158; 16.6194 0.267158; 16.6694 0.267158;
16.7194 0.267158; 16.7694 0.267158; 16.8194 0.267158; 16.8694 0.267158;
16.9194 0.267158; 16.9694 0.267158; 17.0194 0.267158; 17.0694 0.267158;
17.1194 0.267158; 17.1694 0.267158; 17.2194 0.267158; 17.2694 0.267158;
17.3194 0.267158; 17.3694 0.267158; 17.4194 0.267158; 17.4694 0.267158;
17.5194 0.267158; 17.5694 0.267158; 17.6194 0.267158; 17.6694 0.267158;
17.7194 0.267158; 17.7694 0.267158; 17.8194 0.267158; 17.8694 0.267158;
17.9194 0.267158; 17.9694 0.267158; 18.0194 0.267158; 18.0694 0.267158;
18.1194 0.267158; 18.1694 0.267158; 18.2194 0.267158; 18.2694 0.267158;
18.3194 0.267158; 18.3694 0.267158; 18.4194 0.267158; 18.4694 0.267158;
18.5194 0.267158; 18.5694 0.267158; 18.6194 0.267158; 18.6694 0.267158;
18.7194 0.267158; 18.7694 0.267158; 18.8194 0.267158; 18.8694 0.267158;
18.9194 0.267158; 18.9694 0.267158; 19.0194 0.267158; 19.0694 0.267158;
19.1194 0.267158; 19.1694 0.267158; 19.2194 0.267158; 19.2694 0.267158;

19.3194 0.267158; 19.3694 0.267158; 19.4194 0.267158; 19.4694 0.267158;
19.5194 0.267158; 19.5694 0.267158; 19.6194 0.267158; 19.6694 0.267158;
19.7194 0.267158; 19.7694 0.267158; 19.8194 0.267158; 19.8694 0.267158;
19.9194 0.267158; 19.9694 0.267158; 20.0194 0.267158; 20.0694 0.267158;
20.1194 0.267158; 20.1694 0.267158; 20.2194 0.267158; 20.2694 0.267158;
20.3194 0.267158; 20.3694 0.267158; 20.4194 0.267158; 20.4694 0.267158;
20.5194 0.267158; 20.5694 0.267158; 20.6194 0.267158; 20.6694 0.267158;
20.7194 0.267158; 20.7694 0.267158; 20.8194 0.267158; 20.8694 0.267158;
20.9194 0.267158; 20.9694 0.267158; 21.0194 0.267158; 21.0694 0.267158;
21.1194 0.267158; 21.1694 0.267158; 21.2194 0.267158; 21.2694 0.267158;
21.3194 0.267158; 21.3694 0.267158; 21.4194 0.267158; 21.4694 0.267158;

21.5194 0.267158; 21.5694 0.267158; 21.6194 0.267158; 21.6694 0.267158;
21.7194 0.267158; 21.7694 0.267158; 21.8194 0.267158; 21.8694 0.267158;
21.9194 0.267158; 21.9694 0.267158; 22.0194 0.267158; 22.0694 0.267158;
22.1194 0.267158; 22.1694 0.267158; 22.2194 0.267158; 22.2694 0.267158;
22.3194 0.267158; 22.3694 0.267158; 22.4194 0.267158; 22.4694 0.267158;
22.5194 0.267158; 22.5694 0.267158; 22.6194 0.267158; 22.6694 0.267158;
22.7194 0.267158; 22.7694 0.267158; 22.8194 0.267158; 22.8694 0.267158;
22.9194 0.267158; 22.9694 0.267158; 23.0194 0.267158; 23.0694 0.267158;
23.1194 0.267158; 23.1694 0.267158; 23.2194 0.267158; 23.2694 0.267158;
23.3194 0.267158; 23.3694 0.267158; 23.4194 0.267158; 23.4694 0.267158;

23.5194 0.267158; 23.5694 0.267158; 23.6194 0.267158; 23.6694 0.267158;
23.7194 0.267158; 23.7694 0.267158; 23.8194 0.267158; 23.8694 0.267158;
23.9194 0.267158; 23.9694 0.267158; 24.0194 0.267158; 24.0694 0.267158;
24.1194 0.267158; 24.1694 0.267158; 24.2194 0.267158; 24.2694 0.267158;
24.3194 0.267158; 24.3694 0.267158; 24.4194 0.267158; 24.4694 0.267158;
24.5194 0.267158; 24.5694 0.267158; 24.6194 0.267158; 24.6694 0.267158;
24.7194 0.267158; 24.7694 0.267158; 24.8194 0.267158; 24.8694 0.267158;
24.9194 0.267158; 24.9694 0.267158; 25.0194 0.267158; 25.0694 0.267158;
25.1194 0.267158; 25.1694 0.267158; 25.2194 0.267158; 25.2694 0.267158;
25.3194 0.267158; 25.3694 0.267158; 25.4194 0.267158; 25.4694 0.267158;
25.5194 0.267158; 25.5694 0.267158; 25.6194 0.267158; 25.6694 0.267158;

25.7194 0.267158; 25.7694 0.267158; 25.8194 0.267158; 25.8694 0.267158;
25.9194 0.267158; 25.9694 0.267158; 26.0194 0.267158; 26.0694 0.267158;
26.1194 0.267158; 26.1694 0.267158; 26.2194 0.267158; 26.2694 0.267158;
26.3194 0.267158; 26.3694 0.267158; 26.4194 0.267158; 26.4694 0.267158;
26.5194 0.267158; 26.5694 0.267158; 26.6194 0.267158; 26.6694 0.267158;
26.7194 0.267158; 26.7694 0.267158; 26.8194 0.267158;
PERFORM ANALYSIS PRINT ALL
FINISH

4.5 Generation of earthquake loads in the Z direction and earthquake loads in the X direction

This exercise will be carried out by entering data written to the STAAD editor. Yes you have not yet opened a STAAD file, open a new file in the footsteps of past tutorials and practical exercises in this section. With the difference in choosing the "Open STAAD Editor" option in the window of "Where do you want to go?", As shown in the following figure.
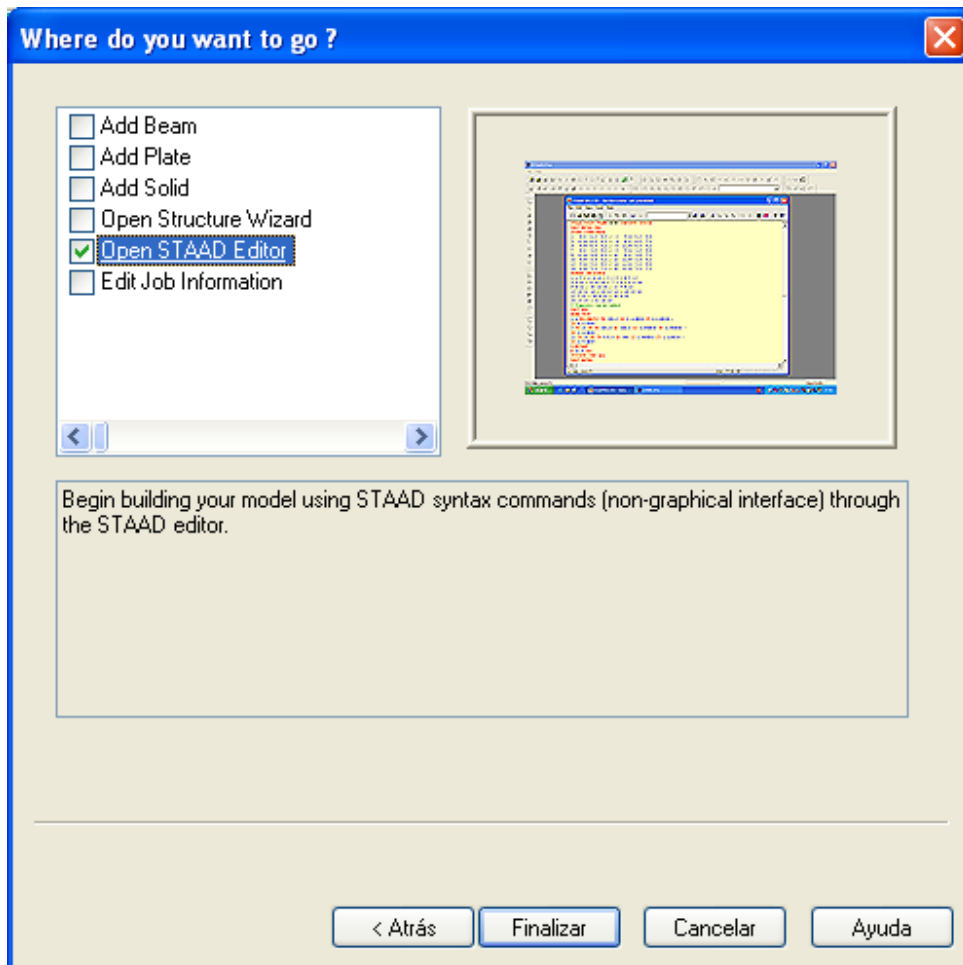
Dale click Finish to enter the editor interface. Yes you have a file open, you can access the editor directly from the STAAD Editor command located in the toolbar.
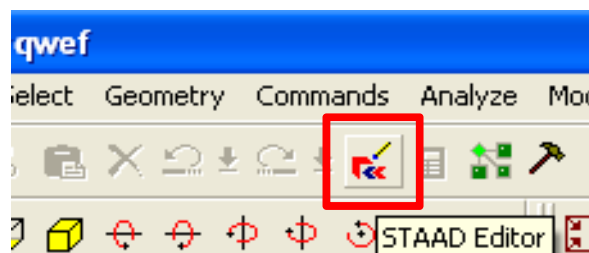
A window like the following appears. This window is the STAAD Editor, where we will enter the commands in order of relevance, as the tutorials described in your procedure.
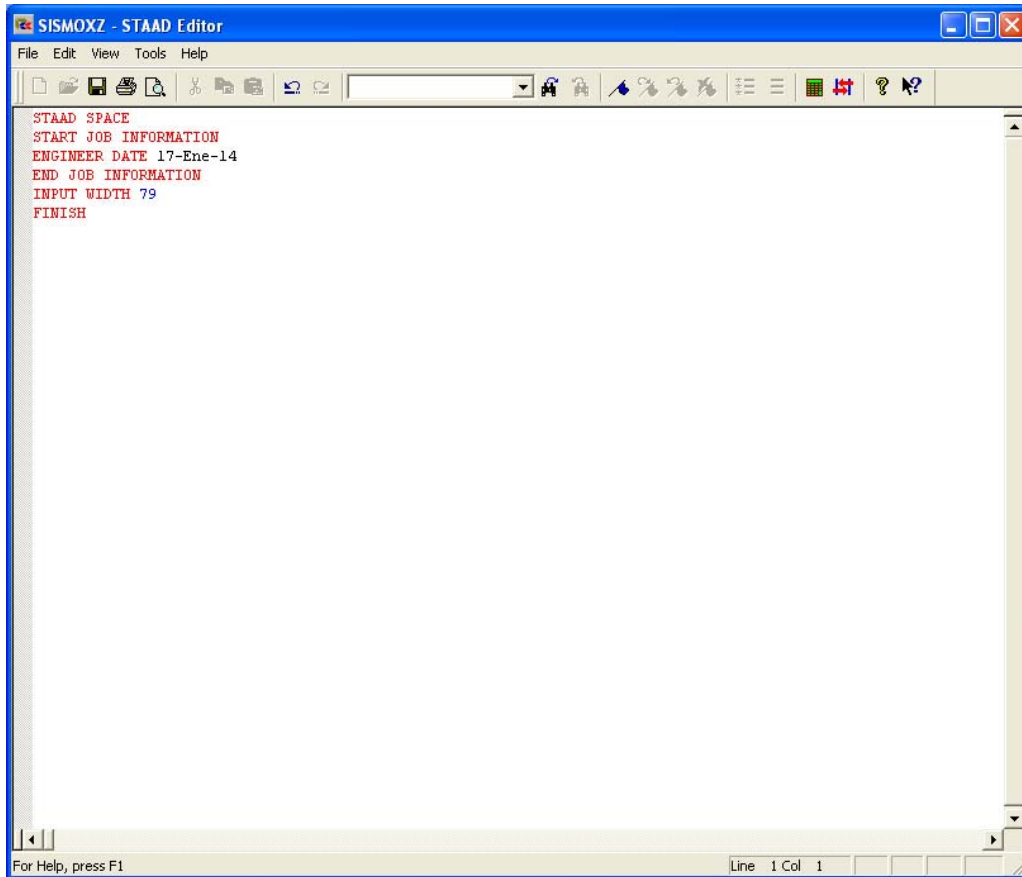
The STAAD editor text reflects the step tutorials and prior years typically using the graphical user interface (GUI) with graphical methods or structures assistant. So the first step to write to the editor are the units (Before sure to delete the last line "FINISH" command:

<div align="center">UNIT METER MTON</div>

The next step is to set the coordinates where the nodes are located in the structure:

JOINT COORDINATES

1 0 0 0; 2 3 0 0; 3 0 4 0; 4 3 4 0; 5 0 8 0; 6 3 8 0; 7 0 0 3; 8 3 0 3;
9 0 4 3; March 10 April 3; 11 0 8 3; 12 3 8 3;
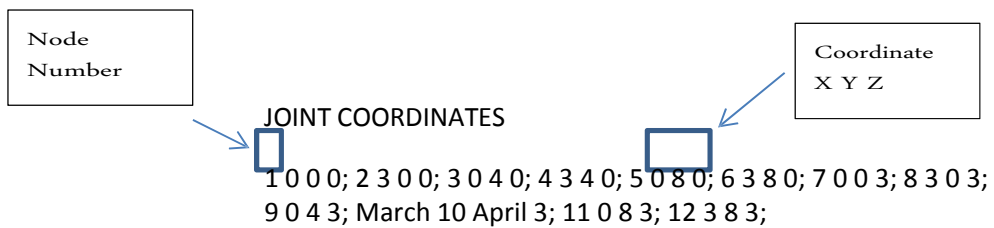
The first number indicates the number of the node, the following indicates the coordinate on X, the following indicates the Y coordinate so on down to the Z coordinate, the data nodes are separated by a semicolon (;).

It continues to tell the program which few beams and node to which node will be linked as follows:

Initial node (home of the beam)

Item number

MEMBER INCIDENCES
March 1 4; 2 5 6; 3 1 3; 4 2 4; 5 3 5; 6 4 6; September 7, 10; August 11, 12; 9 7 9; August 10, 10;
November 9, 11; October 12, 12; March 13 9; April 14, 10; May 15, 11; June 16, 12;

End node (end of the beam)
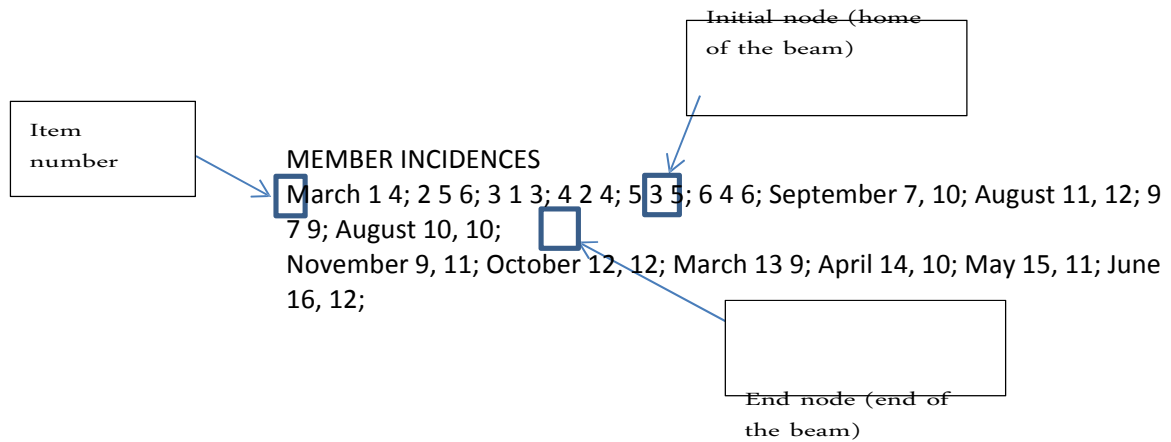
In a similar way as the nodes, the first number indicates the number of the element, followed by the initial node; ie that point begins our beam in question and the following number indicates where it ends, each element separated by a semicolon.

Once given the geometry of the structure, the material to be used in the analysis is defined. It is important to tell the program that will define a title (DEFINE MATERIAL START), followed by the type of material involved and the constants of the same (constants can vary and take into account the insertion units), as shown below :

```
DEFINE MATERIAL CONCRETE
START ISOTROPIC
E 2.21467e +006
POISSON 0.17
DENSITY 2.40262
ALPHA DAMP 0.05
1e-005
END DEFINE MATERIAL
```

Then it is necessary to indicate the properties of the elements; that is, the dimensions of the section and of which the material is composed. In this case the properties of American standards and concrete material to set the whole structure.

```
MEMBER PROPERTY AMERICAN
PRIS 1 TO 16 YD 0.2 ZD 0.2
CONSTANTS
ALL MATERIAL CONCRETE
```

Once defined the properties, we proceed with braces indicate starting with writing the word "SUPPORTS" in which embedded support (FIXED) was placed in all lower nodes (1, 2, 7, 8)

```
SUPPORTS
```

1 2 7 8 FIXED

The next step is to propose the load cases to be presented in the structure, the order of integration is to define the number and load case followed by the title we assign. Types of loads (distributed load, concentrated, etc.) followed by the listed items to be assigned to these loads are then written. The order of the types of loads is indistinct, it is important to be careful with data entry.

```
LOAD 1 LOADTYPE None TITLE C. M.
SELFWEIGHT Y -1 1 TO 16 MEMBER
LIST LOAD
 1 2 7 8 13 to 16 UNI GY -0.5
LOAD 2 LOADTYPE None TITLE C.V.
MEMBER LOAD
 1 2 7 8 13 to 16 UNI GY -0.2
LOAD 3 LOADTYPE None EARTHQUAKE IN TITLE X
SELFWEIGHT X -1 1 TO 16 MEMBER
LIST LOAD
1 2 7 8 13 to 16 -0.5 UNI GX
MEMBER LOAD
 1 2 7 8 13 to 16 -0.2 GX UNI
LOAD 4 LOADTYPE None EARTHQUAKE IN
TITLE SELFWEIGHT Z -1 Z LIST 1 TO 16
MEMBER LOAD
1 2 7 8 13 to 16 -0.5 UNI GZ
MEMBER LOAD
 1 2 7 8 13 to 16 -0.2 GZ UNI
```

In addition to physical loads, a load combination where some factors increase to these loads are applied and combined with other defined. As an example, an increase of 10% (1.1) as a combination of load for all cases was assumed, you can follow the specific rules; either ACI, LRFD or NTC:

```
LOAD COMB CASE LOAD COMBINATION 5 5
1 1.1 2 1.1 3 1.1 4 1.1
```

Within the analysis, only need to define what type of analysis must be used. Because concrete is governed by the plastic theories or ultimate strength theories, these theories are not elastic-linear (as in the case of steel), so that the analysis used is that PDELTA:

```
ANALYSIS PDELTA
```

Finally we proceed to design the concrete elements, for that the initial command, followed by the code to which the design is based (in this case will be ACI), then by the design parameters, such as writes coatings reinforcement The compressive stress, the yield stress, the type of printing (TRACK). The values used in this exercise are the default, you can

another basis. For the output file sections with steel reinforcement is displayed, you must indicate the "DESIGN BEAM" and "DESIGN COLUMN" to corresponding elements, finalizing the design of specific commands. For more information, see section 5 of the technical manual.

```
START CONCRETE DESIGN
ACI CODE
ALL 0.0381 CLB
ALL CLS 0.0381
ALL CLT 0.0381
ALL FC 2812.3
FYMAIN 42184.4 ALL
TRACK 1 ALL
 DESIGN BEAM 1 2 7 8 13 to 16
DESIGN COLUMN 3 June 9 TO 12 TO
END CONCRETE DESIGN
```

To finish the analysis and design, type FINISH as last line. Yes you correctly followed all the above steps, the final text will look like below (the difference is only in the date):

```
STAAD SPACE
ENGINEER JOB
INFORMATION START
DATE 17-Jan-14 END JOB
INFORMATION INPUT
WIDTH 79
UNIT METER MTON
JOINT COORDINATES
1 0 0 0; 2 3 0 0; 3 0 4 0; 4 3 4 0; 5 0 8 0; 6 3 8 0; 7 0 0 3; 8 3 0 3;
9 0 4 3; March 10 April 3; 11 0 8 3;
12 3 8 3; MEMBER INCIDENCES
March 1 4; 2 5 6; 3 1 3; 4 2 4; 5 3 5; 6 4 6; September 7, 10; August
11, 12; 9 7 9; August 10
10;
November 9, 11; October 12, 12; March 13 9; April 14,
10; May 15, 11; June 16, 12; DEFINE MATERIAL START
ISOTROPIC 2.21467e
CONCRETE E + 006
0.17 POISSON
DENSITY 2.40262
ALPHA DAMP 0.05
1e-005
END DEFINE MATERIAL
MEMBER PROPERTY AMERICAN
PRIS 1 TO 16 YD 0.5 ZD 0.4
CONSTANTS
ALL MATERIAL CONCRETE
SUPPORTS
1 2 7 8 FIXED
LOAD 1 LOADTYPE None TITLE C. M.
SELFWEIGHT -1 LIST AND 1 TO 16
```

```
MEMBER LOAD
1 2 7 8 13 to 16 UNI GY -0.5
LOAD 2 LOADTYPE None TITLE C.V.
MEMBER LOAD
1 2 7 8 13 to 16 UNI GY -0.2
LOAD 3 LOADTYPE None EARTHQUAKE IN TITLE X
SELFWEIGHT X -1 1 TO 16 MEMBER
LIST LOAD
1 2 7 8 13 to 16 -0.5 UNI GX
MEMBER LOAD
1 2 7 8 13 to 16 -0.2 GX UNI
LOAD 4 LOADTYPE None EARTHQUAKE IN TITLE
SELFWEIGHT Z -1 Z LIST 1 TO 16
MEMBER LOAD
1 2 7 8 13 to 16 -0.5 UNI GZ
MEMBER LOAD
1 2 7 8 13 to 16 -0.2 GZ UNI
LOAD COMB CASE LOAD COMBINATION 5 5
1 1.1 2 1.1 3 1.1 4 1.1
ANALYSIS PDELTA
ACI CONCRETE DESIGN CODE
START
ALL 0.0381 CLB
ALL CLS 0.0381
ALL CLT 0.0381
ALL FC 2812.3
FYMAIN 42184.4 ALL
TRACK 1 ALL
DESIGN BEAM 1 2 7 8 13 to 16
DESIGN COLUMN 3 June 9 TO 12 TO
END CONCRETE DESIGN
FINISH
```

You give save button located in the toolbar of the editor window STAAD. Then, you set yourself on the FILE tab, select the EXIT option to exit the editor.
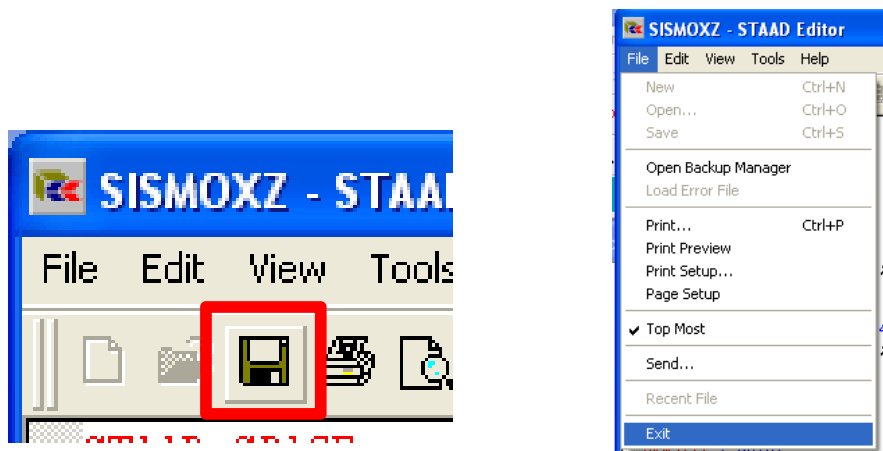
The structure with details of load cases written with STAAD editor should be displayed like the following figure:
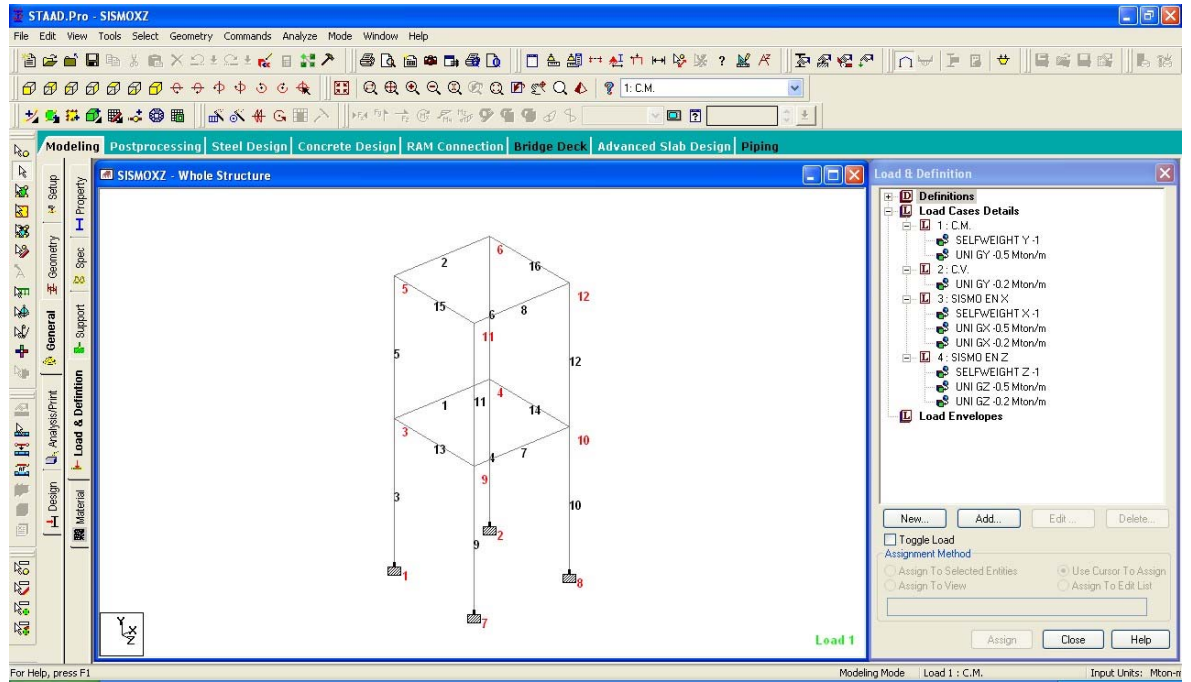
The next step is to define the seismic spectrum (Response Spectrum) that occurs at the site of the building that is being designed. For this we place ourselves in the case of load EARTHQUAKE IN X, then give ... Add / Response Spectra / Response Spectrum and the following window will open.
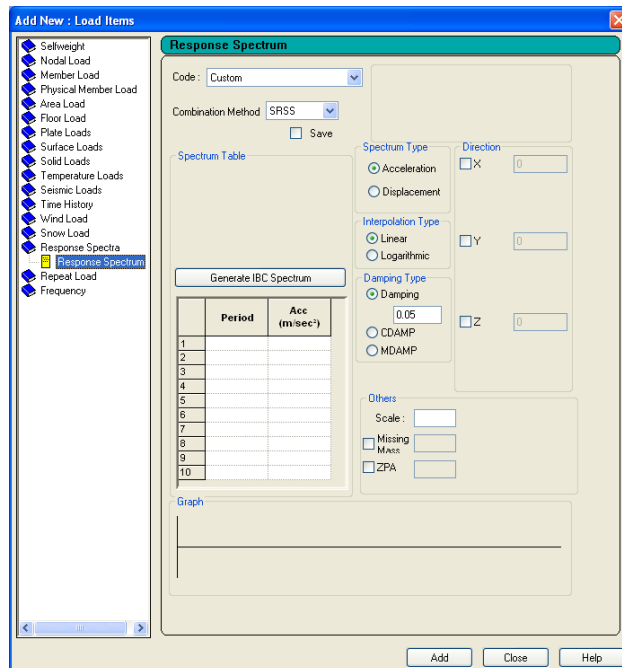
Unlike the previous practical exercise, we will not use any other program to obtain the required parameters. This time we will use values previously collected by the Environmental Attorney and Zoning of the Federal District in its regulation of the additional technical standards (NTC-DF) as the location where "build" building example in the Mexican capital. The values are shown in the following table:

T	C	ao	Ta	Tb	r	to	Q	Q '	a / Q '
0.00	0.16	0.04	0.20	1.35	1.00	0.04	2.00	1.6	0,025
0.20	0.16	0.04	0.20	1.35	1.00	1.08	2.00	1.6	0,675
0.40	0.16	0.04	0.20	1.35	1.00	0.54	2.00	1.6	0,338
0.60	0.16	0.04	0.20	1.35	1.00	0.36	2.00	1.6	0,225
0.80	0.16	0.04	0.20	1.35	1.00	0.27	2.00	1.6	0,169
1.00	0.16	0.04	0.20	1.35	1.00	0.22	2.00	1.6	0,135
1.20	0.16	0.04	0.20	1.35	1.00	0.18	2.00	1.6	0,113
1.40	0.16	0.04	0.20	1.35	1.00	0.15	2.00	1.6	0,096
1.60	0.16	0.04	0.20	1.35	1.00	0.14	2.00	1.6	0,084
1.80	0.16	0.04	0.20	1.35	1.00	0.12	2.00	1.6	0,075
2.00	0.16	0.04	0.20	1.35	1.00	0.11	2.00	1.6	0,068
2.20	0.16	0.04	0.20	1.35	1.00	0.10	2.00	1.6	0,061
2.40	0.16	0.04	0.20	1.35	1.00	0.09	2.00	1.6	0,056
2.60	0.16	0.04	0.20	1.35	1.00	0.08	2.00	1.6	0,052
2.80	0.16	0.04	0.20	1.35	1.00	0.08	2.00	1.6	0,048
3.00	0.16	0.04	0.20	1.35	1.00	0.07	2.00	1.6	0,045
3.20	0.16	0.04	0.20	1.35	1.00	0.07	2.00	1.6	0,042
3.40	0.16	0.04	0.20	1.35	1.00	0.06	2.00	1.6	0,040
3.60	0.16	0.04	0.20	1.35	1.00	0.06	2.00	1.6	0,038
3.80	0.16	0.04	0.20	1.35	1.00	0.06	2.00	1.6	0,036
4.00	0.16	0.04	0.20	1.35	1.00	0.05	2.00	1.6	0,034
4.20	0.16	0.04	0.20	1.35	1.00	0.05	2.00	1.6	0,032
4.40	0.16	0.04	0.20	1.35	1.00	0.05	2.00	1.6	0,031
4.60	0.16	0.04	0.20	1.35	1.00	0.05	2.00	1.6	0,029
4.80	0.16	0.04	0.20	1.35	1.00	0.05	2.00	1.6	0,028
5.00	0.16	0.04	0.20	1.35	1.00	0.04	2.00	1.6	0,027

In the section of the popup in STAAD called Spectrum Table, put the values shown in the table above, using nothing but green columns. The column named T sticks called Period column and the last column named a / Q ' is

paste in the column entitled Acc (m / sec 2). As shown below.

After only activate the option in paragraph X direction, giving a value of 1 We write 9.81 as the scale in paragraph Others, leaving all the default options. As shown in the figure below:
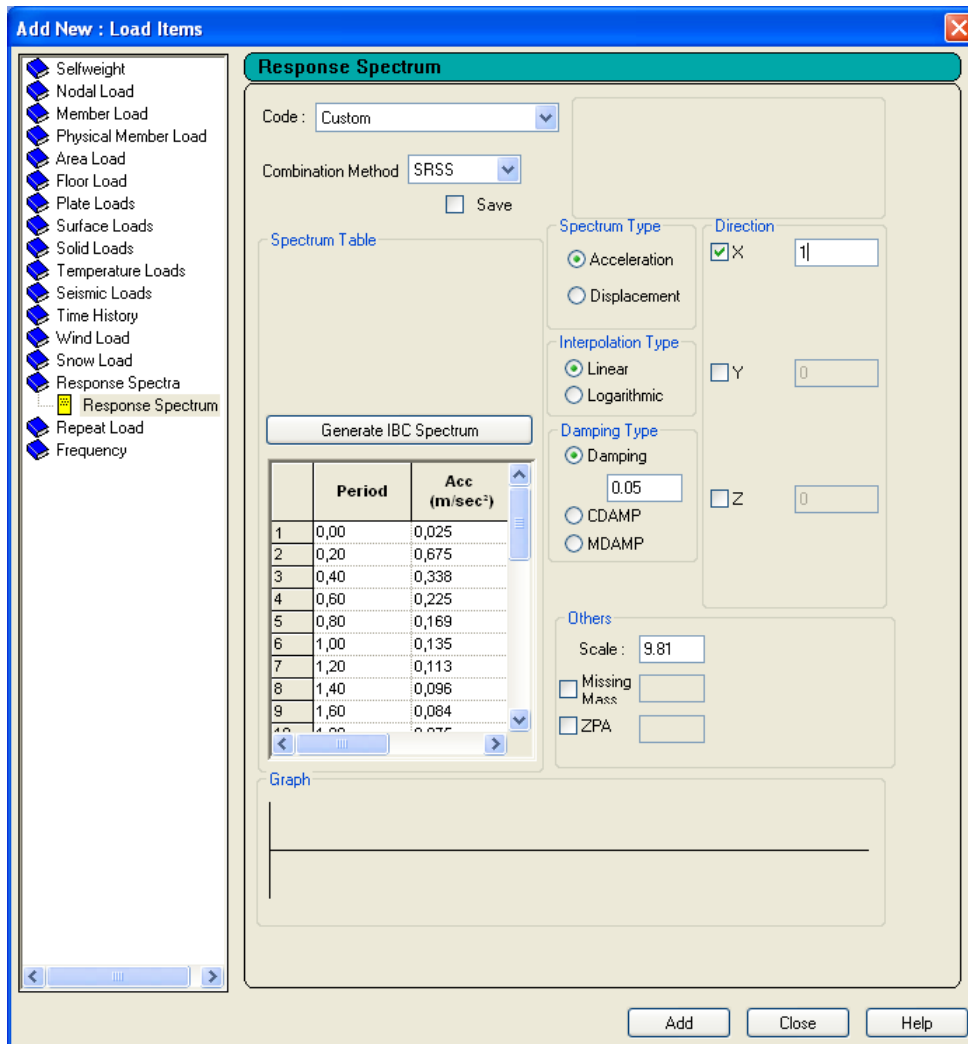
Then we give Add, then click Close to close the window and perform the same step for the load case EARTHQUAKE IN Z, with the only difference that activate the option in the Z direction section also giving a value of 1.
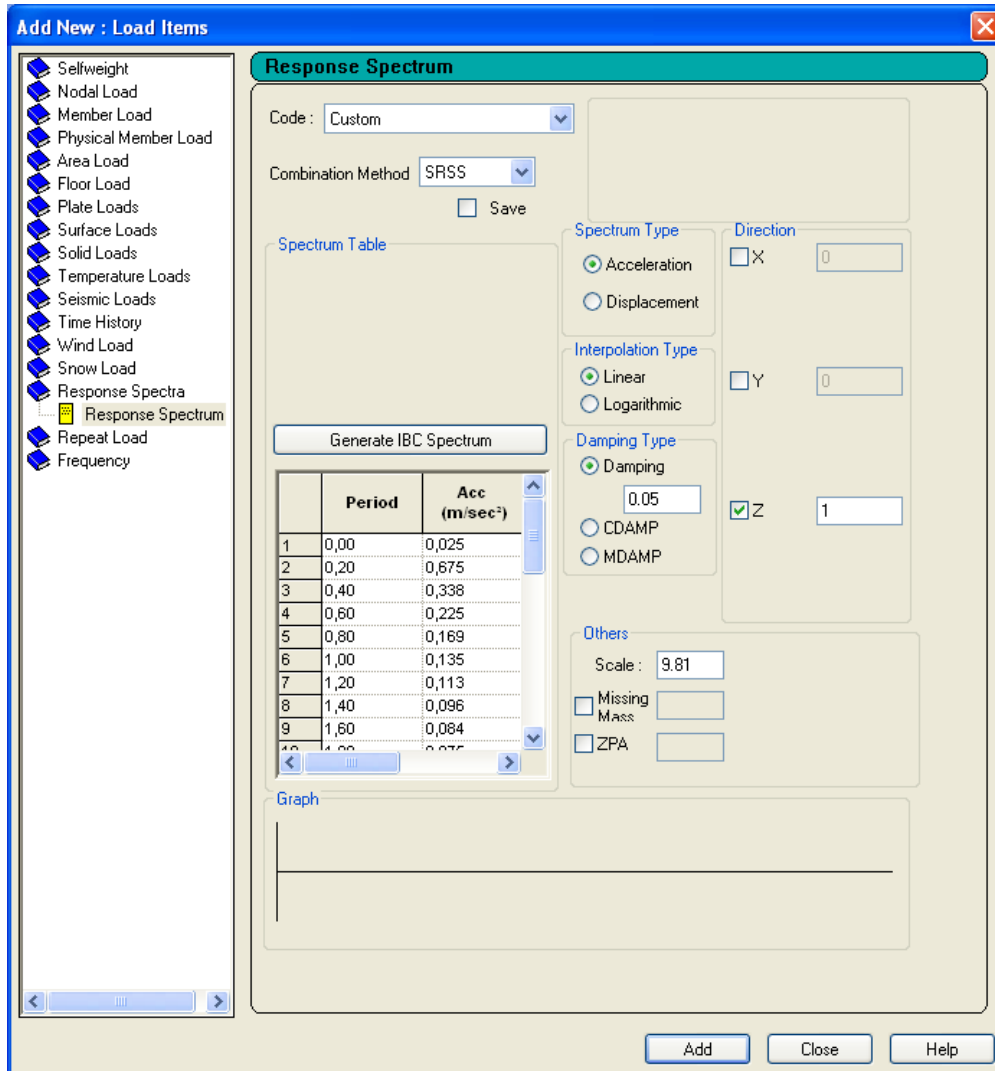
Once entered the data of seismic response spectrum in both horizontal directions (X and Z), the next step is to run the analysis and program design and engine following steps outlined in both tutorials as previous years. See section 5 of STAAD.Pro V8i Technical Manual for more information.
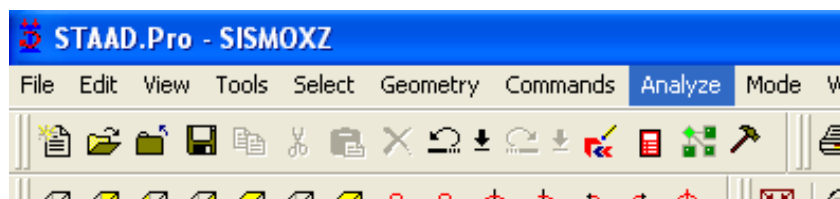
After giving click the located in the menu bar of the program Analyze command, a popup window will appear, as shown below.
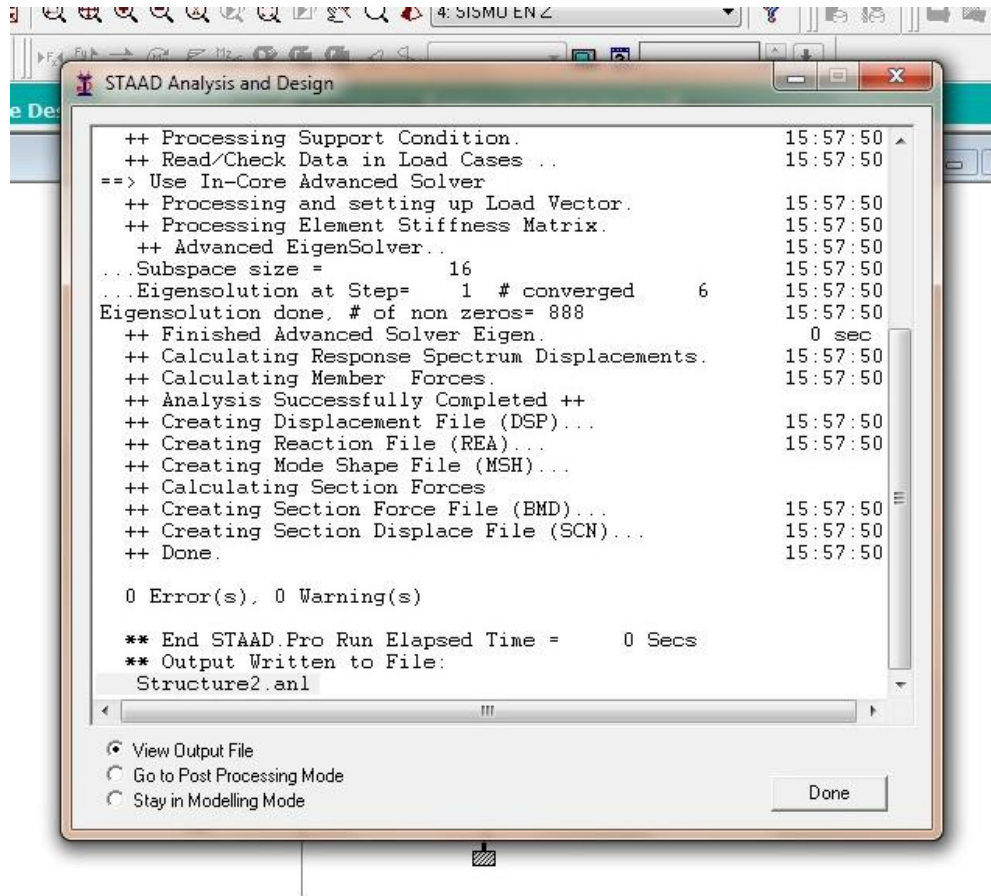
Then we click on the Command Output file located in the toolbar
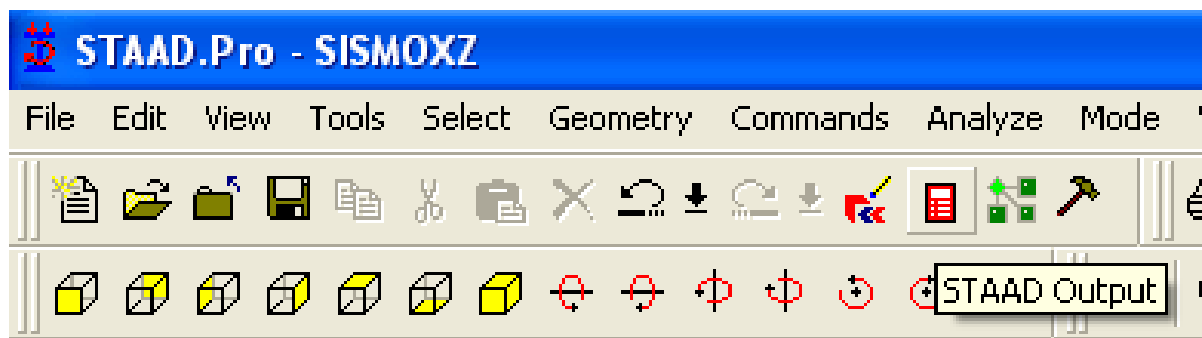
Or directly in the popup window "STAAD analysis and design", we click on the View Output file option. Window shown in the following figure appears:

In the Output File shown in the results section of "Participation Factors" shows the percentage of the mass of the structure which is subjected to the stresses caused by the "trembling" that may occur at the site. A "mode" refers to the different vibration modes in which you can present in the structure.

To interpret and understand perfectly the values shown in the STAAD output file. You need to know the theories, fundamentals and principles on which, studies on the behavior of natural phenomena in the different types of existing buildings are based. Also the commands used and the captured results are explained in detail in the Technical Manual (Volume II).

Tip: For more information about the efforts that cause tremors in buildings and how they behave for further design. See the book "*Buildings Seismic and wind resistant, criteria for calculation and design*"José Creixell i ll i the Edi tor the

Limusa

(UNAM)

Citing Suggested text book (Creixell, 1993):

"To simplify problems, it is often assumed that an earthquake occur, the building will be subject to a horizontal or vertical forces, and the different elements of it are calculated statically to support them, regardless of their movements. This type of calculation is called static, but in fact this name is not correct, because the calculation itself can not be dynamic not static.

It is designated as "dynamic analysis" that is used for the elements of the structure, if it considers the actions of the seismic forces are not constant and movements that cause construction. The dynamic calculation, besides giving a clear view of the real problem, shows the position to assess to what extent and with what limitations you can use the static calculation. "

MANUAL DE STAAD.Pro v8i

TOMO II

ALFREDO ZERTUCHE HERRERA
RAFAEL HERNANDEZ SOTELO

ING. HECTOR ELIUD MARTINEZ ADAME

NUEVO LAREDO, TAM.   JUNIO 2014

MANUAL

DE

STAAD.Pro
V8i

TECHNICAL MANUAL

MANUAL

DE

STAAD.Pro

V8i TECHNICAL

MANUAL First Edition

Volume II

ALFREDO HERNANDEZ

RAFAEL HERRERA ZERTUCHE

Sotelo

ING. HECTOR MARTINEZ ELIUD DAMEA

Content

Section 1

General Description

1.1   Introduction

The STAAD.Pro V8i Graphic User Interface (GUI) is normally used to create all the input specifications and display all output reports. These input specifications of structural modeling and saved in a text file with the ".STD" extension. When the GUI opens a file to start a session with an existing model receives all this information file STD. A user can edit / create this file and have the STD GUI and engine analysis, both reflect the changes.

STD file is processed by the STAAD analysis "engine" to produce results that are stored in multiple files with extensions such as ANL, BMD, TMH, etc. ANL text file containing the printout created by the specifications in this manual. The other files contain the results (displacements, forces in the members / elements, mode shapes, forces / moments / movements of the section, etc.) that is used by the GUI mode processing.

This section of the manual contains an overview of the availability of the analysis and design of the STAAD engine. Specific information on steel and concrete are available in Sections 2 and 3 manual, it respectively. The details of the STAAD engine and commands Archives STD and other specific information for the user are presented in Section 5.

The aim of this section is to familiarize the user with the basic principles involved in the application of the various means of analysis / design offered by the STAAD engine. As a general rule, the sequence in which the facilities are discussed following the recommended use of the input file data stream STD.

1.2   The   Generating input data

The GUI (or user) communicates with the STAAD analysis engine via the input file STD. This input file is a text file consisting of a series of commands that are executed sequentially. The orders contain instructions or data that allow analysis and / or design. The elements and conventions of language STAAD commands are described in Section 5 of this manual.

The STAAD input file can be created through the text editor or the GUI ease of modeling. In general, any text editor can be used to edit / create the input file STD. Ease of Modeling GUI creates the input file through an interactive menu-oriented graphical procedure.

1.3 Types   Structures

A STRUCTURE can be defined as a collection of elements. The STAAD is capable of analyzing and designing structures consisting of frames and / or plate / shell and solid elements. Almost any type of structure can be analyzed by STAAD.

- Structure **SPACE (SPACE) which is a frame structure in three dimensions with loads applied at any level, is the most usual.**

- Structure **FLAT (PLANE) is limited by an XY coordinate axis in a global system loads in the same plane.**

- A kind structure **ARMOR (TRUSS) consists of members armor type that can have only axial forces in the members and no bending members.**

- A kind structure **FLOOR (FLOOR) is one of two or three dimensions that have no horizontal motion (global X or Z) in the structure [FX, FZ and MY restrict in each joint]. The floor frame (in global coordinates in the XZ plane) of a building is a perfect example of a structure of type. They also modeled the columns with floor condo in a structure provided that the structure has no horizontal load. If there is any horizontal load should be analyzed as a SPACE structure.**

The specification of the correct type of structure reduces the number of equations to be solved during analysis. This results in a faster and more economical solution to the user. The degrees of freedom associated with the frame members of different types of structures are shown in Figure 1.1.

Types of Structure
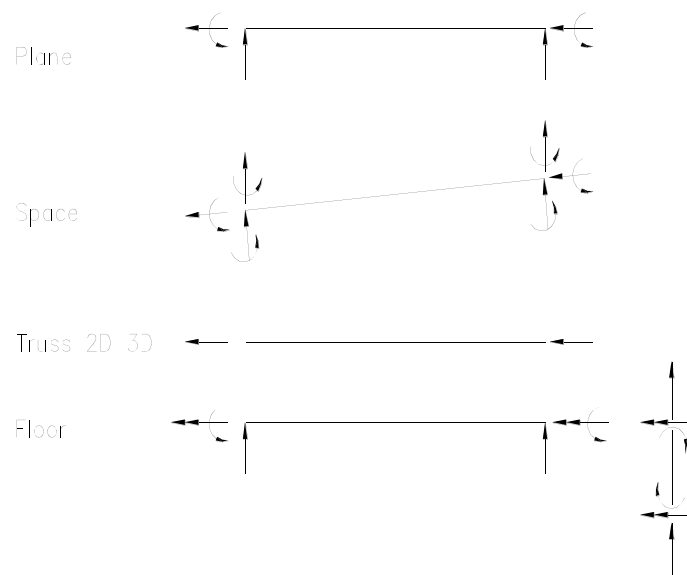
Figure 1.1

1.4   Systems   Units

FPS is usually the user allowed to enter data and receive results in almost all commonly used unit systems engineering including MKS, SI. In the input file, the user can change the units as many times as required. Mix and match between length and force units of the systems of different units is also allowed. The input units for angles (or rotations) are degrees. However, the output of the SEALS OFFSET, rotations are given in radians. For all outputs, the units are clearly specified by the program.

1.5   Structure and Geometry of Coordinate Systems

A structure is an assembly of components such as beams, columns, slabs, plates etc. To Staad, may use elements of the frame and floor elements to model the structural components. Typically, modeling the geometry of the structure consists of two steps:

    A.   The identification and description of joints or nodes.

    B.   Modeling of members or elements through the specification of its connectivity (incidents) between joints.

In general, the term MEMBER (MEMBER) is used to refer to elements and the end frame member (element) is used to refer to the plates / shells and solids. Connectivity can be provided to Members through the MEMBER command of IMPACT (MEMBER INCIDENCES) while connectivity for elements can be provided through the command of IMPACT ELEMENT (ELEMENT INCIDENCES).

Staad uses two types of coordinate systems for defining the geometry of the structure and load patterns. GLOBAL coordinate system is an arbitrary coordinate system in space that is used to specify the overall geometry and loading pattern of the structure. A LOCAL coordinate system is associated with each member (or element) and used at the output of the EXTREME FORCES MEMBERS or local load specification.

1.5.1   Global Coordinate System

The following coordinate systems are available for the specification of the geometry of the structure.

    A.   The conventional Cartesian coordinate system: This coordinate system (Fig. 1.2) is a rectangular coordinate system (X, Y, Z) which follow the rule of the orthogonal right hand. This coordinate system can be used to define the location of the meeting and the directions of the loads. The translational degrees of freedom are denoted by u1, u2, u3 and the degrees of freedom of rotation are denoted by the u4, u5 and u6.

    B.   The cylindrical coordinates system: In this coordinate system (Fig. 1.3) the coordinate axis X and Y of the conventional Cartesian system is replaced by R (the

radio) and Ø (angle in degrees). The Z coordinate is identical to Z of the Cartesian coordinate system and its positive direction is determined by the right hand rule.

C.   Inverse System Cylindrical Coordinates: This is a cylindrical coordinate system type (Fig 1.4.) Where R is in the plane corresponding to the plane Ø X Z of the Cartesian system. The right hand rule is followed to determine the positive direction of the Y axis
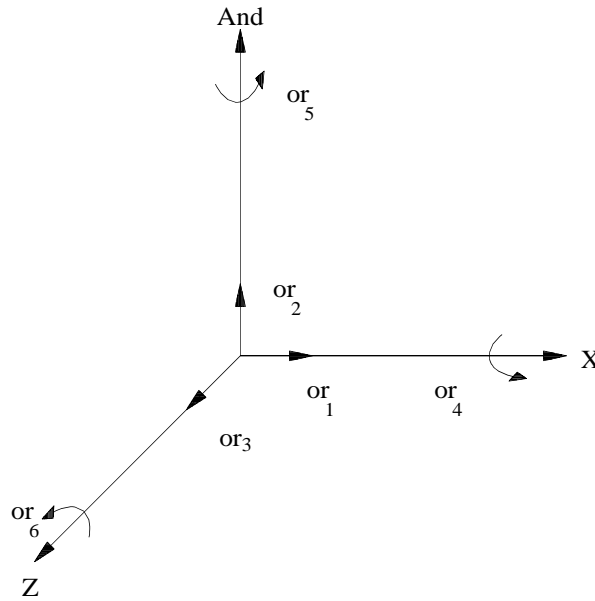
Cartesian Coordinate System (Rectangular) Figure 1.2
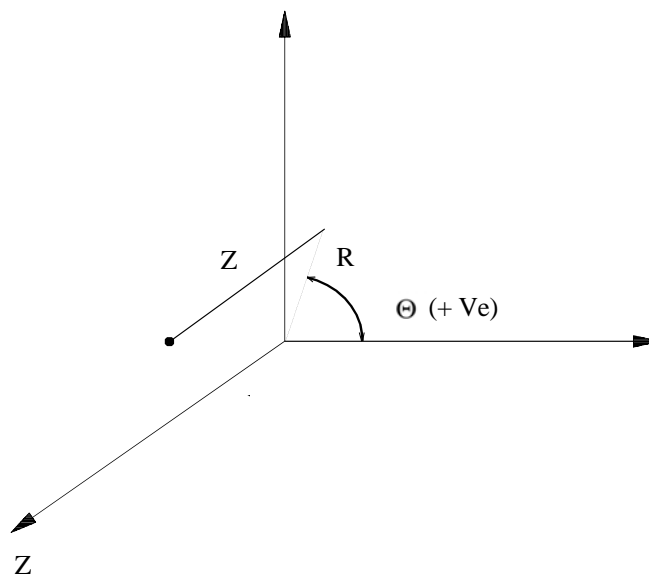
Cylindrical Coordinates System

Figure 1.3
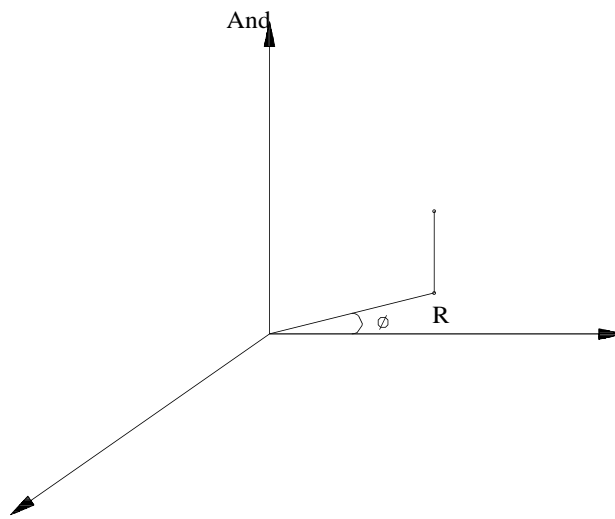
Inverse System Figure 1.4 Cylindrical Coordinates

1.5.2 Local Coordinate System

A local coordinate system is associated with each member. Each axis orthogonal system of local coordinates is also based on the rule of the right hand. Fig. 1.5 shows a member of the board of the beam with the start 'i' and the final meeting 'j'. The positive direction of the x local axis is determined attaching 'i' to 'j' and projecting in the same direction. The right hand rule can be applied

for the positive directions of the local axes "y" and "z". Local "and" axis and "z" match
with the axes of the two principal moments of inertia. Note that the local coordinate system is always rectangular.

A wide range of sectional shapes can be specified for analysis. These include rolling steel, prismatic forms that the user specified, etc. The Fig. 1.6 shows the local axes system for these forms.

When Global-Y is Vertical

When Global-Z is Vertical (Z SET UP is used)
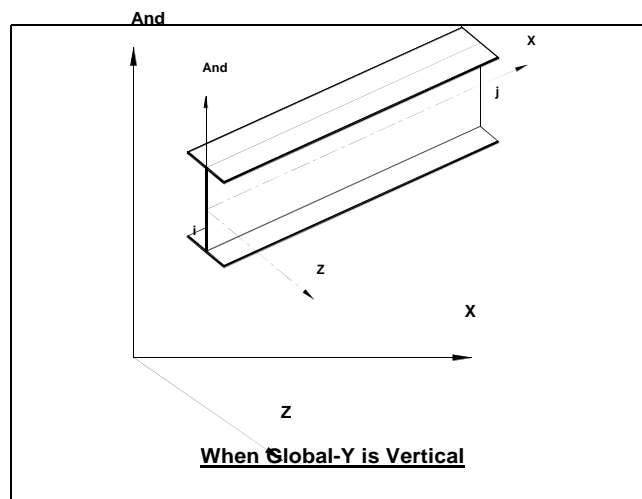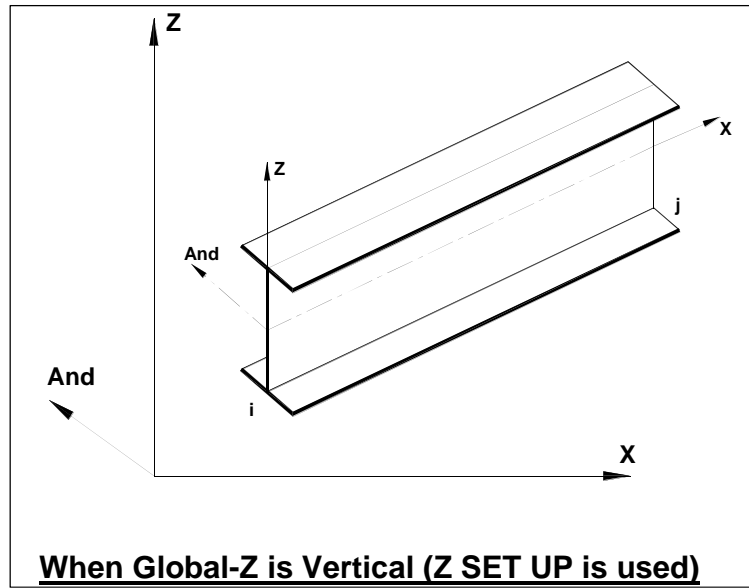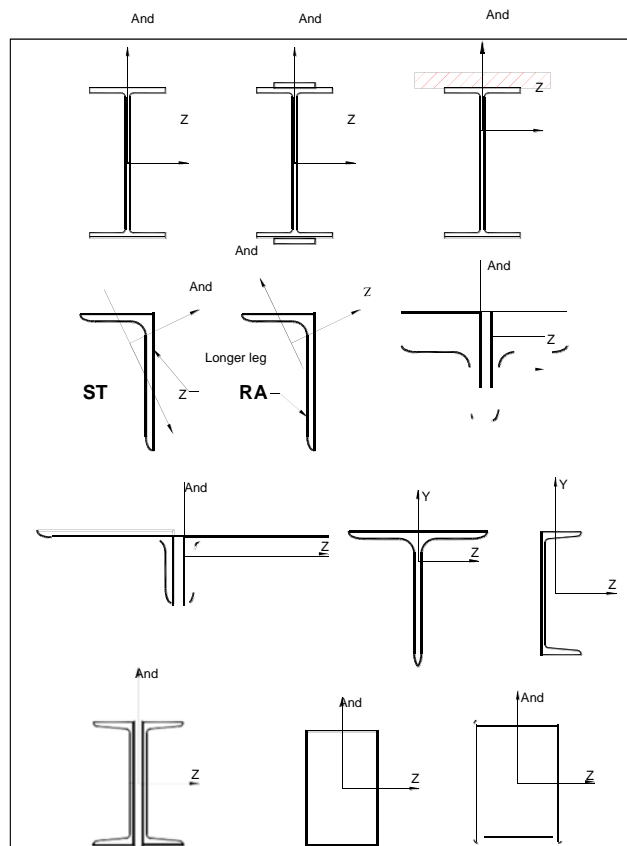
Figure 1.5

Local axes for different cross sections Figure 1.6
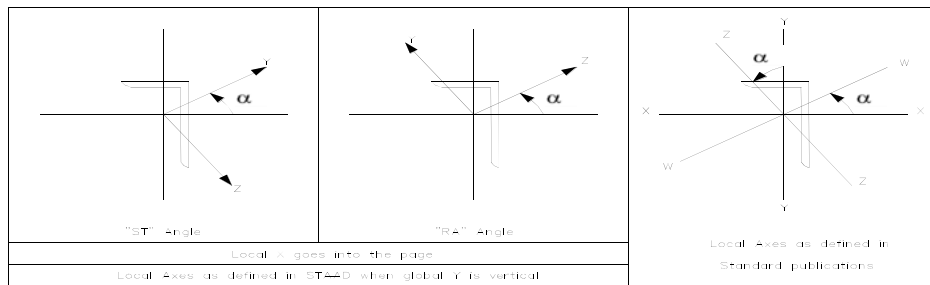
Local Axis System

For the different sections when the "Z UP" command is specified.

The Local X axis goes into the page.
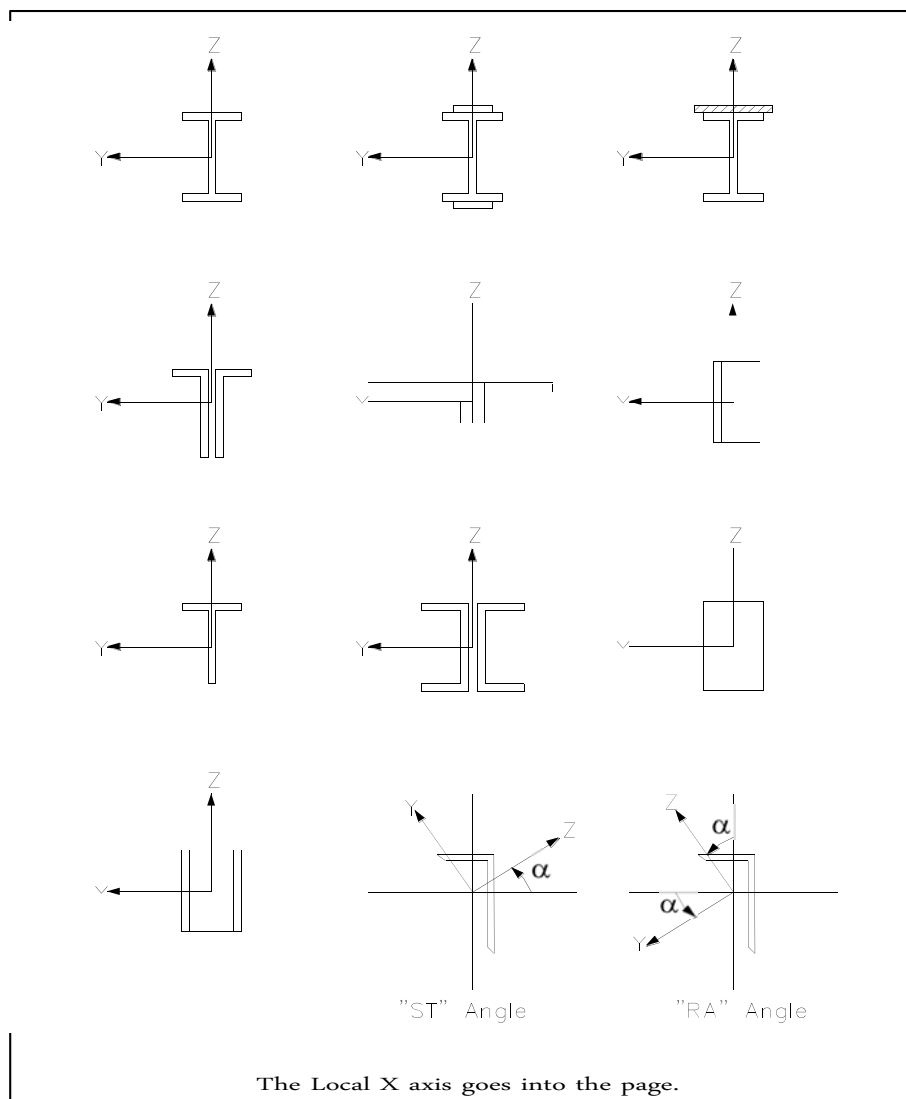
1.5.3   The relationship between Global and Local Coordinates

From the entrance to the loads of the member may remain in the system of local and global coordinate and the output of the extreme forces of the Member will be printed in the local coordinate system, it is important to know the relationship between the systems of local coordinates and Global. This relationship is defined by an angle measured in the following manner. This angle is defined as beta (). For members of the beta displaced reference angle indicate that the features are based on the position of the displacement of the local axis, and not positions of the joint.

Beta Angle

For input, see section 5.26

When the local x-axis is parallel to the Y axis global, as in the case of a column in a structure, the angle beta is the angle through which the local z-axis is rotated about the local axis x from a position parallel and in the same positive direction of the global axis Z.

When the local axis x is parallel to the overall axis, the angle beta is the angle through which the local coordinate system is rotated about the local axis x of the position of having the local z axis parallel to the plane of Global and local XZ axis in the same direction as the overall positive axis Y. Figure 1.7 outlines the beta positions equal to 0 degrees or 90 degrees. By providing loads of member local axis of the member, it is useful to refer to this figure for a rapid determination of the local axis system.

The Benchmark

An alternative to provide Member orientation is get the coordinates (or a number of the board) which will be a reference point located in the xy plane of the member but not in the axis of the member. From the location of the reference point, the program calculates the xy plane orientation of the member automatically.
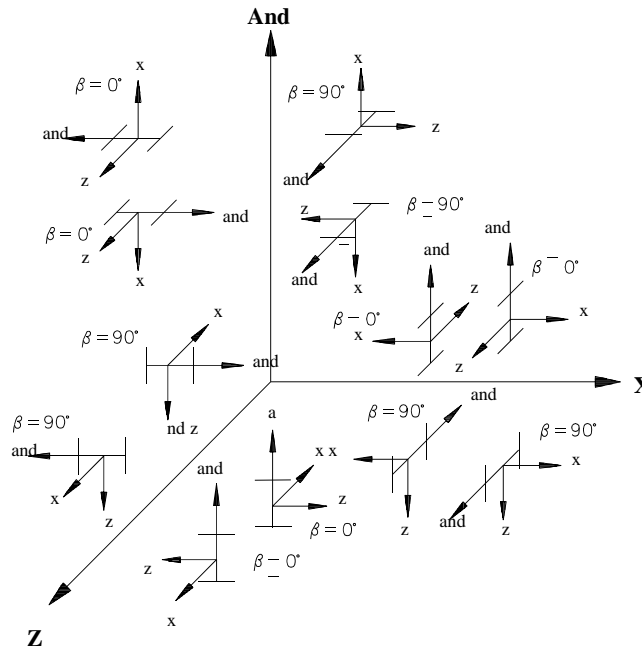
Relationship between Global and Local Axes

Figure 1.7

BETA-ROTATION OF EQUAL & UNEQUAL LEGGED 'ST' ANGLES

NOTE: THE ORDER OF THE JOINT NUMBERS IN THE MEMBER INCIDENCE COMMAND DETERMINES THE DIRECTION OF THE MEMBER'S LOCAL X-AXIS

Member Orientation for various Beta Angles
Global Y-Axis is vertical

Member Orientation for various Beta Angles
Global Y-Axis is vertical
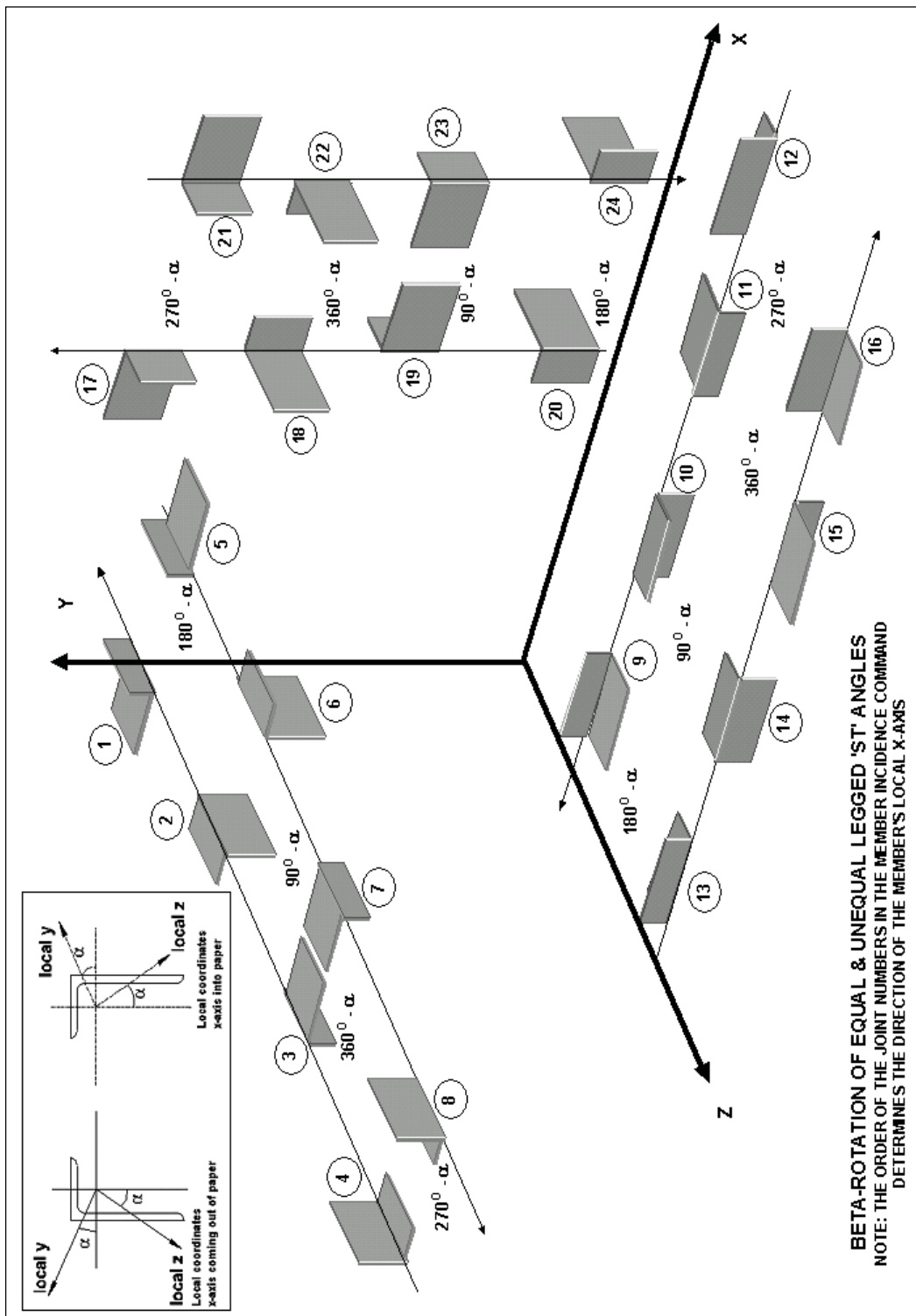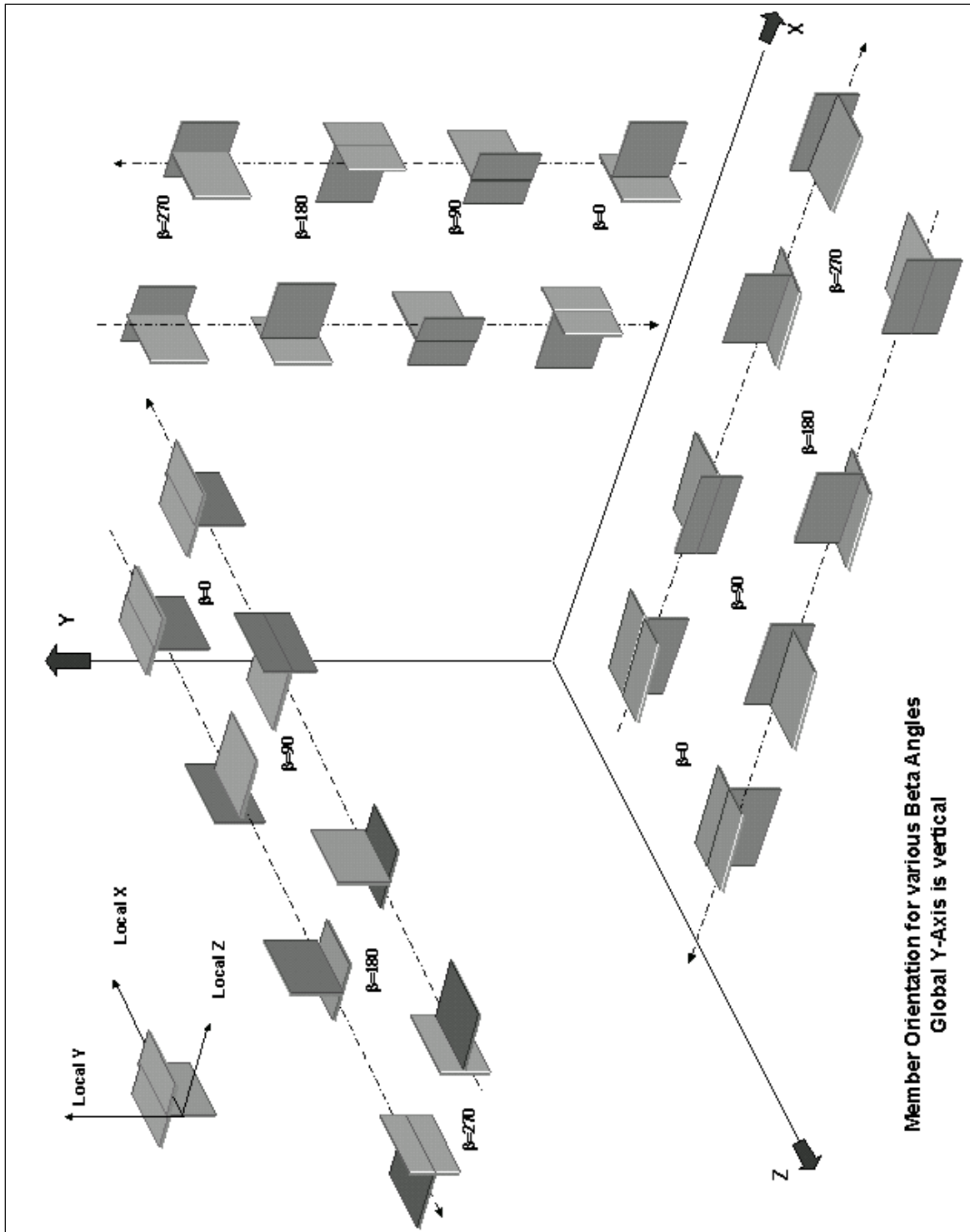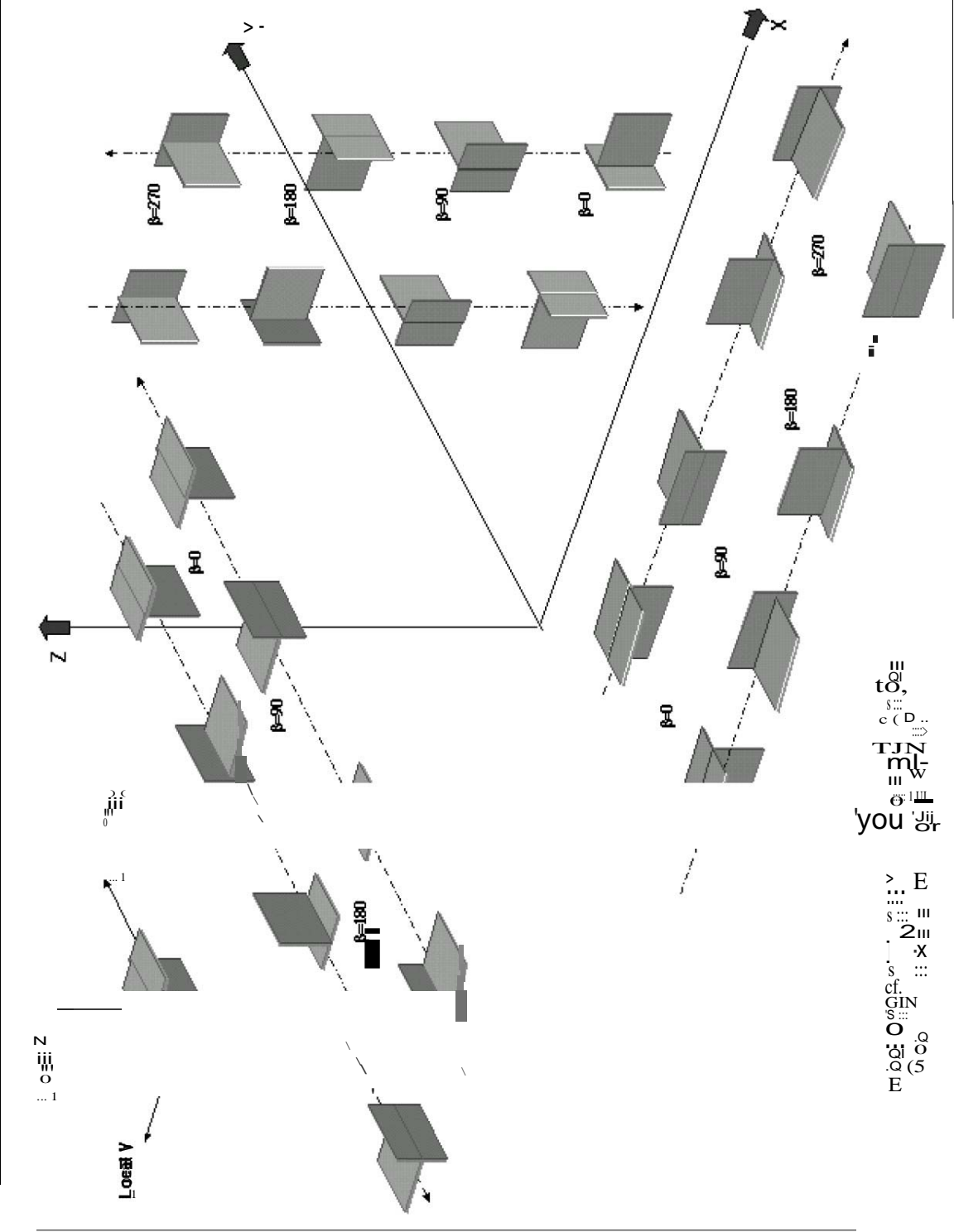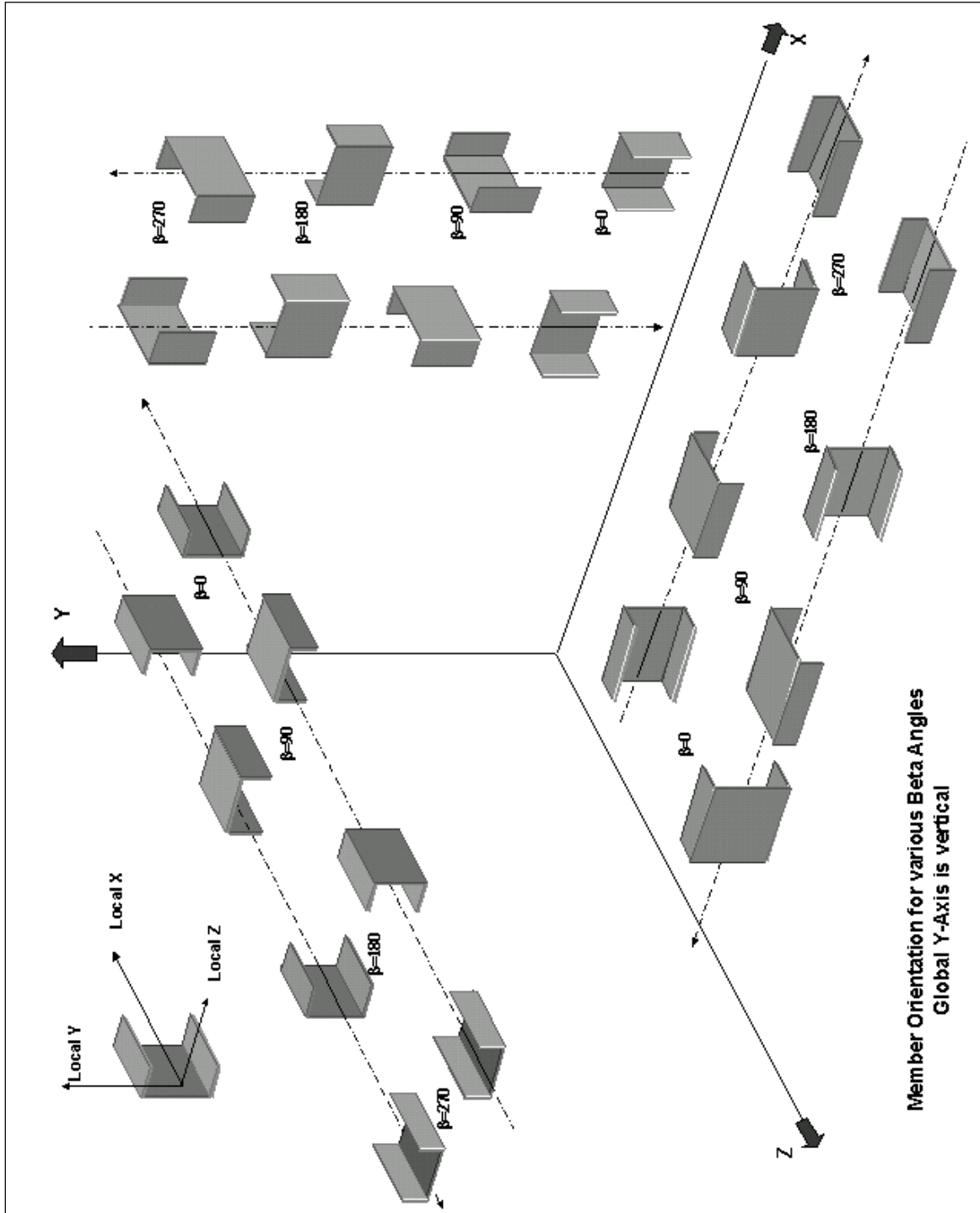
1.6   The   Information finite element

STAAD is equipped with an innovative condition plate / shell and solid finite element. The characteristics of each are explained below. For input, see Sections 5.11, 5.13, 5.14, 5.21, 5.24, and 5.31.3

1.6.1   Element Plaque / Smudge

The finite element Plaque / Smudge is based on the formulation of the hybrid cell. The element can be 3 knots (triangular) or 4 knots (ring). If all four nodes of a quadrilateral element does not lie in a plane, it is advisable to model as triangular elements. The thickness of the element may differ from one node to another.

The "surface structures" such as walls, slabs, plates and shells can be modeled using finite elements. For convenience in generating a finer mesh of plate / shell within a large area, a choice of mesh generation is available. The option is described in detail in Section 5.14.

The user can also use the element for PLANAR EFFORTS action only (for example the rigidity of the membrane / in one plane only). The command EFFORTS IN PLANAR ELEMENT should be used for this purpose.

Considerations in the Modeling of Geometry

The following rules related to geometry modeling to remember when you are using the element plate / shell

1) The program automatically generates a fifth knot "O" (the center knot - see Fig. 1.8) to the center of the element.

2) While the nodes are allocated to an element in the input data, it is essential that the nodes are specified either in the direction of clockwise or counter-clockwise (Fig. 1.9). For efficiency, similar elements must be numbered sequentially.

3) The aspect ratio between elements should not be excessive. They should be in the order of 1: 1, and preferably less than 4: 1.

4) No individual elements are distorted. The angles between adjacent sides of the element must not be larger than 90 and never exceeding 180 degrees.

Specifying Element Loading

The following specifications of charges are accepted:

1) Loads on the boards at the nodes of the elements in the global addresses.

2) Concentrated loads at any point specified by the user within the element in global or local addresses.

3) Uniform pressure on the surface of the element in the global or local addresses

4) Uniform partial pressure the user to specify a portion of the surface of the element in the global or local addresses

5) Linearly varying pressure on the surface of the local addresses.

6) Temperature load due to an increase or decrease in temperature.

7) The load due to temperature difference in the temperature between the upper and lower surfaces of the element.

Figure 1.8

Figure 1.9

Figure 1.10

Figure 1.11
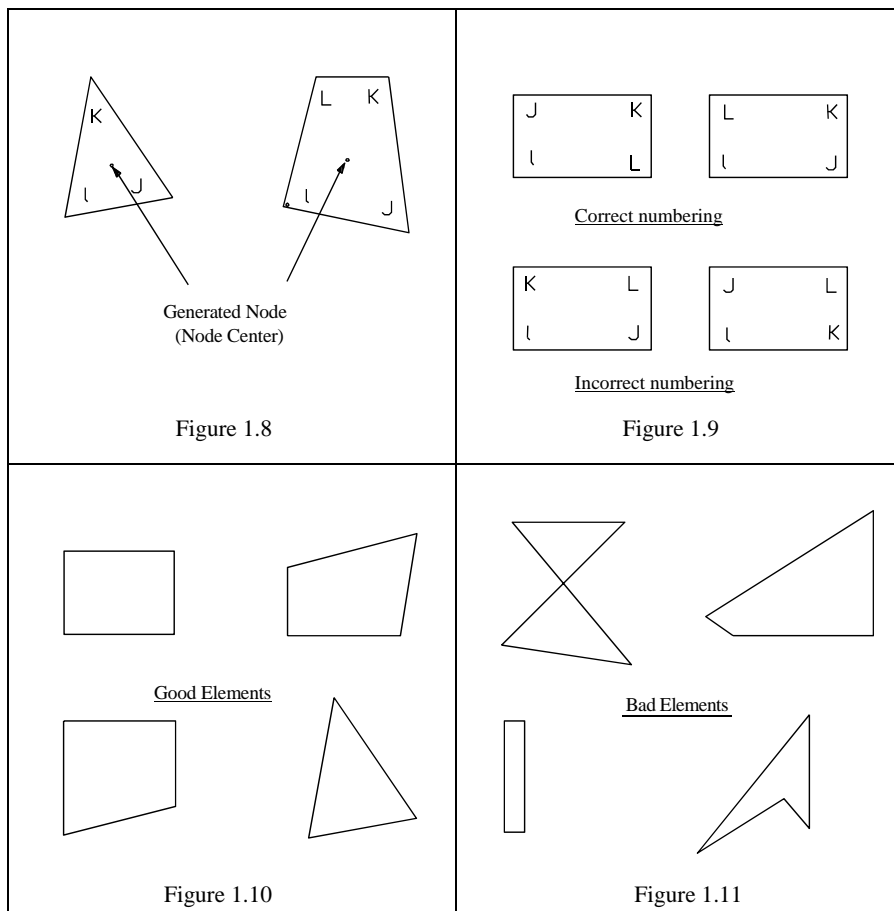
Theoretical
Bases

The finite element Staad plate is based on the finite element hybrid formulations. Full distribution is assumed quadratic efforts. For the action of forces in a plane, the stress distribution is supposed to be as follows.
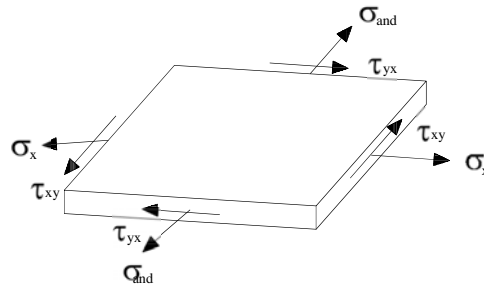
The complete quadratic distribution of alleged efforts

$$
\begin{pmatrix} \sigma_x \\ \sigma_{and} \\ \tau_{xy} \end{pmatrix} = \begin{bmatrix} 1 & x & and & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ y & 0 & 0 & 0 & 1 & x & and & 0 & 0 & 2xy \\ 0 & -a & 0 & 0 & 0 & -x & 1 & -2xy & -and & -x^2 \end{bmatrix} \begin{pmatrix} to_1 \\ to_2 \\ to_3 \\ \vdots \\ to_{10} \end{pmatrix}
$$

a1 to a10 = constant polynomial efforts.

The following distribution of quadratic action efforts to bending of the plate is assumed:
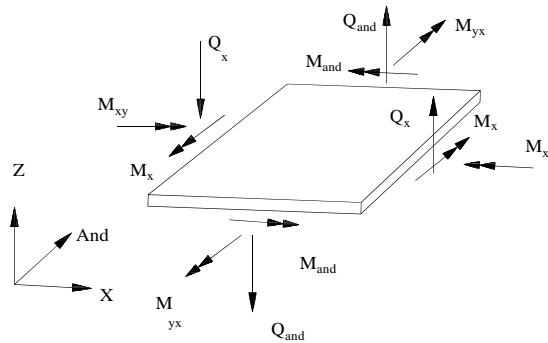
The distribution of full quadratic alleged efforts:

$$
\begin{pmatrix} M_x \\ M_{and} \\ M_{xy} \\ Q_x \\ Q_{and} \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & and^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & and & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & and & 0 & -x \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & - & and & 0 & x & and \end{bmatrix} \begin{pmatrix} to_1 \\ to_2 \\ to_3 \\ \vdots \\ \vdots \\ to_{13} \end{pmatrix}
$$

The distinctive features of this finite element are:

1) Compatibility between the component displacement efforts in an element plane and the bending component of an adjacent plate element which forms an angle with the first (see Fig. Below) is reached. This compatibility requirement is usually ignored in most of the flat elements of shell / plate.

2) The rotational stiffness out of the plane of the planar portion of the efforts of each item is properly built and not treated as a simulation as is usually done in most commonly available commercial software.

3) Despite the incorporation of the aforementioned rotational stiffness, the elements satisfy the verification test completely.

4) These elements are available as triangles and quadrilaterals with knots at the corners only, and six degrees of freedom at each node

5) These elements are the simplest forms of shell / plate planes possible with knots at the corners only six degrees of freedom per node. Nevertheless the solutions to solve the problems converge quickly and accurate even with large mesh size responses.

6) These elements may be connected to the planar members / or spatial frame with full support of displacement. No restrictions / additional relaxation is required.

7) Cutting out of the plane of the shear strain energy is incorporated into the formulation of component plate bending. As a result, the elements correspond to the Poisson boundary conditions which are considered more accurate than the usual boundary conditions Kirchoff

8) The bending portion of the plate can be considered thick and thin plates, thus extending the usefulness of the plate elements a number of problems. Further, the thickness of the plate is taken into account by calculating the cutting out of the plane.

9) Efforts triangular plane behaves almost on par with the well-known triangle of linear stresses. Triangles of the flat elements of the most similar triangular shells incorporate constant effort convergence has very low proportions. Thus the triangular shell element is very useful in problems where the quadrilateral double-curved element can not be used.

10) The recovery of nodal forces at any point within the element.

Local Coordinate System Element

The precise orientation of the local coordinates is determined as follows:

1) Denoting the midpoints of the four or three sides of the IJ, JK, KL, LI by M, N, O, P element respectively.

2) The vector pointing from P to N is defined to be the x axis locally (in a triangle, that is always parallel to IJ).

3) The vector product of PN and MO vectors (for a triangle, ON and MK) defines the local z axis, ie, z = PN x MO.

4) The vector product of vectors z and x defines the local axis, that is, y = (z)

(x). The sign convention of the force and moment resulting protrusion is shown in

Fig. 1.13.
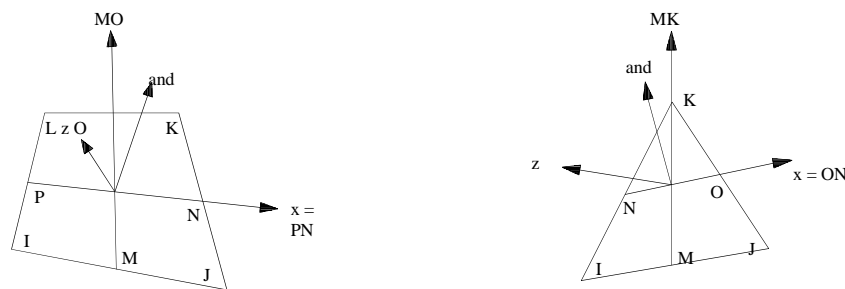
Figure 1.12

Output Element Forces

The results of the command FORCE ELEMENT (ELEMENT FORCES) are available in the following situations:

A.   At the center point of the element.
B.   In every corner of the element nodes.
C.   At any point specified by the user within the element.

The following are the items included in the output EFFORTS ELEMENT. SQX, SQY

Shear stress (force / unit Long. / Thickness)

SX, SY, SXY   Efforts Membrane (Strength / Long unit. / Thickness)
MX, MY, MXY bending moments per unit width (Time / Long drive.) SMAX,

SMIN   Principal stresses (force / unit area)

TMAX   Maximum shear (force / unit area) ANGLE

The orientation of the principal plane (Grades)

VONT, Von Mises VONB efforts where

$$VM = 0.707\sqrt{\left(SMAX - SMIN\right)^2 + SMAX^2}$$

Notes:

1) All results of the efforts of the member are in the local coordinate system. The direction and sense of the efforts of the element in Fig. 1.13 are explained.

2) For efforts element at a specified point in the element, the user must provide the coordinate system of the element. Note that the origin of the local coordinate system coincides with the center of the element node.

3) The main efforts (the SMAX and SMIN), the maximum shear (TMAX), the orientation of the principal plane (ANGLE), and efforts Von Mises (VONT and VONB) are also printed to the upper and lower surfaces of the . The upper and lower surfaces are determined based on the z axis of the local address.

Signs Convention Forces the Elem ents

Figure 1.13

STRESS CAUSED BY M_X

M_x is the Bending Moment on the local x face, the local x-face is the face perpendicular to the local x-axis;
M_y is the Bending Moment on the local y face, the local y-face is the face perpendicular to the local y-axis

Stress Caused by M_{and}

Membrane Stresses Sx and Sy

In plane shear stresses Sxy and Syx

Out-of plane shear stresses (SQX and SQY)
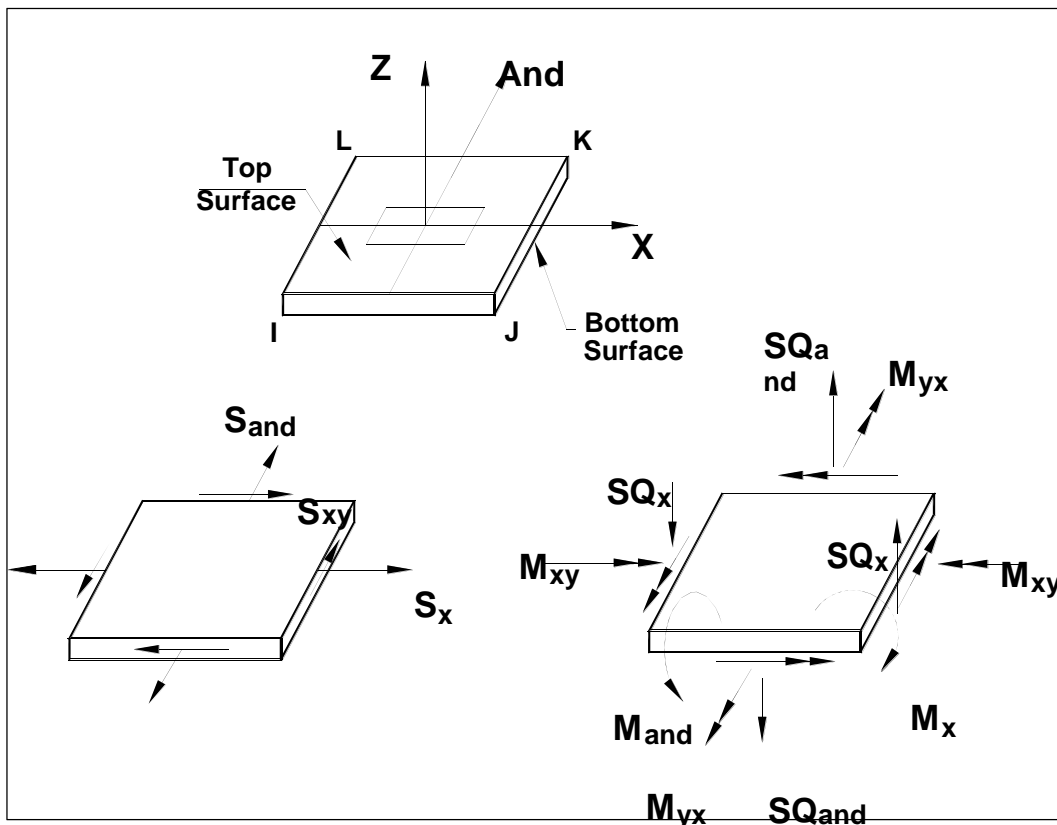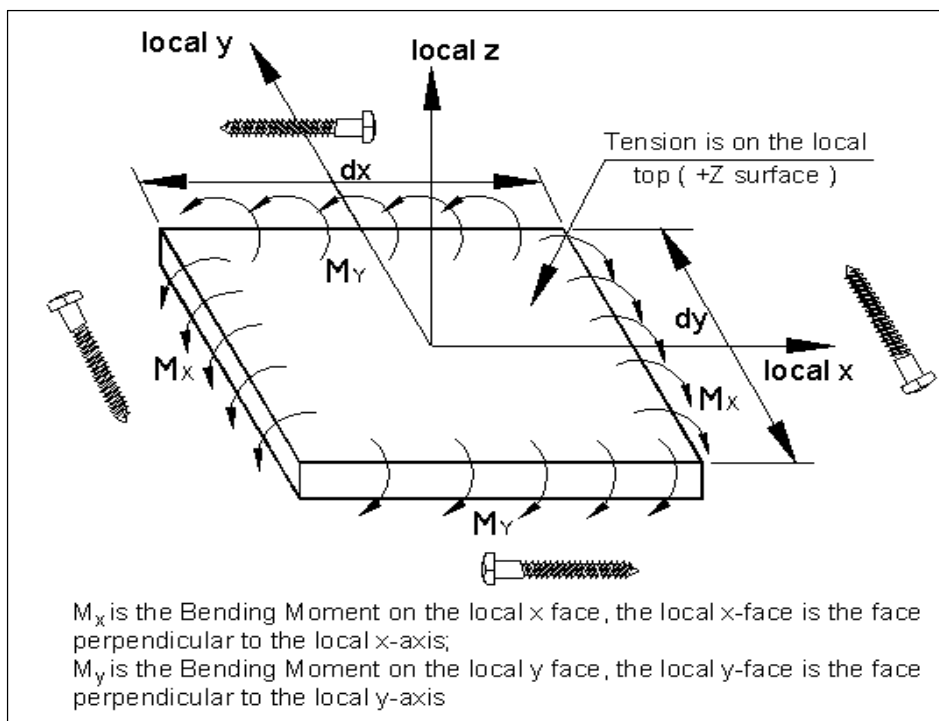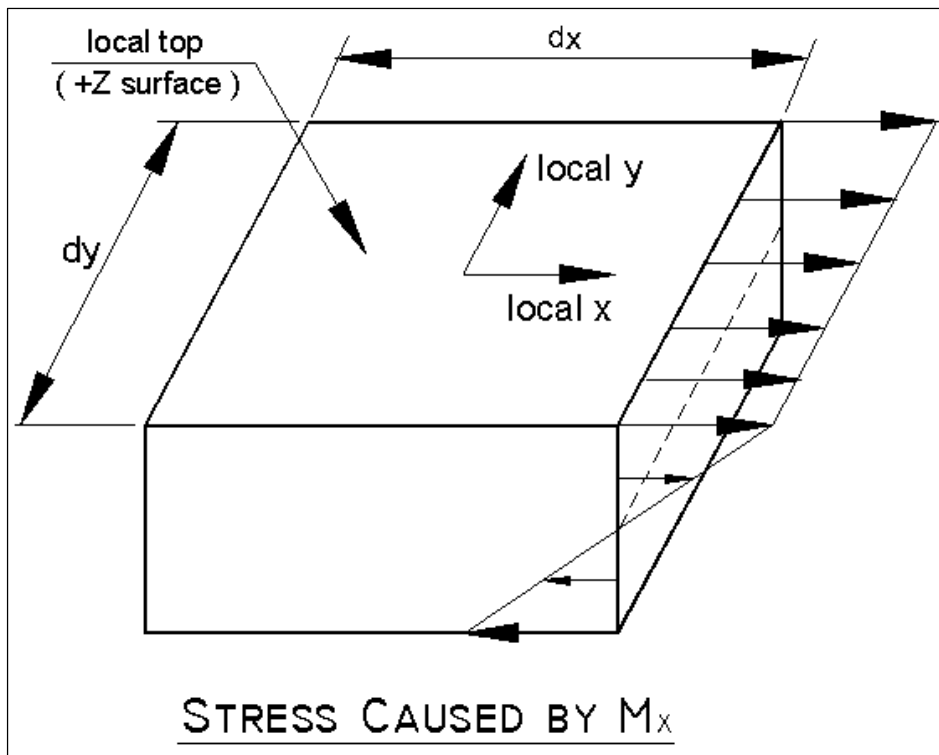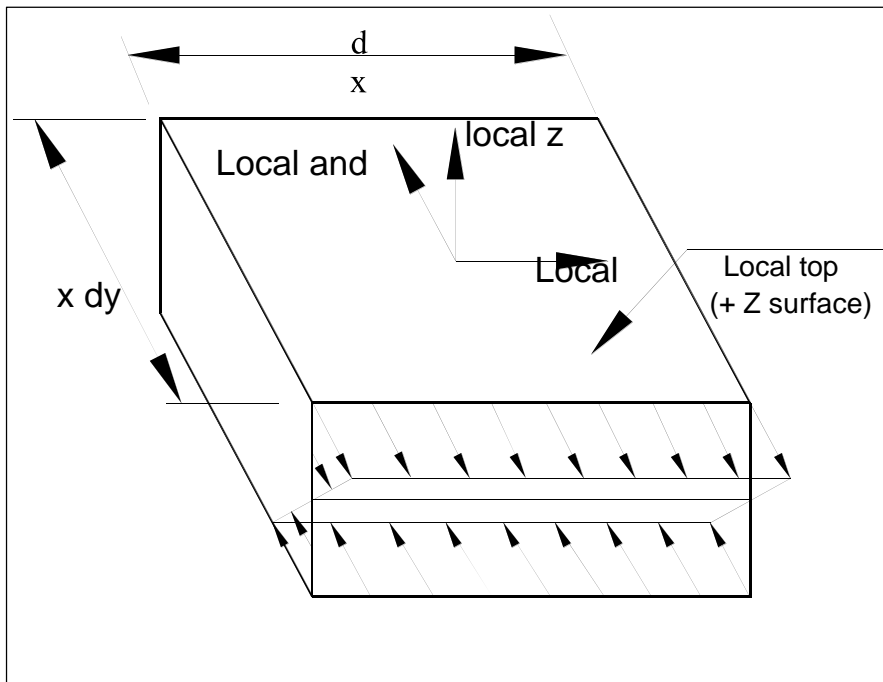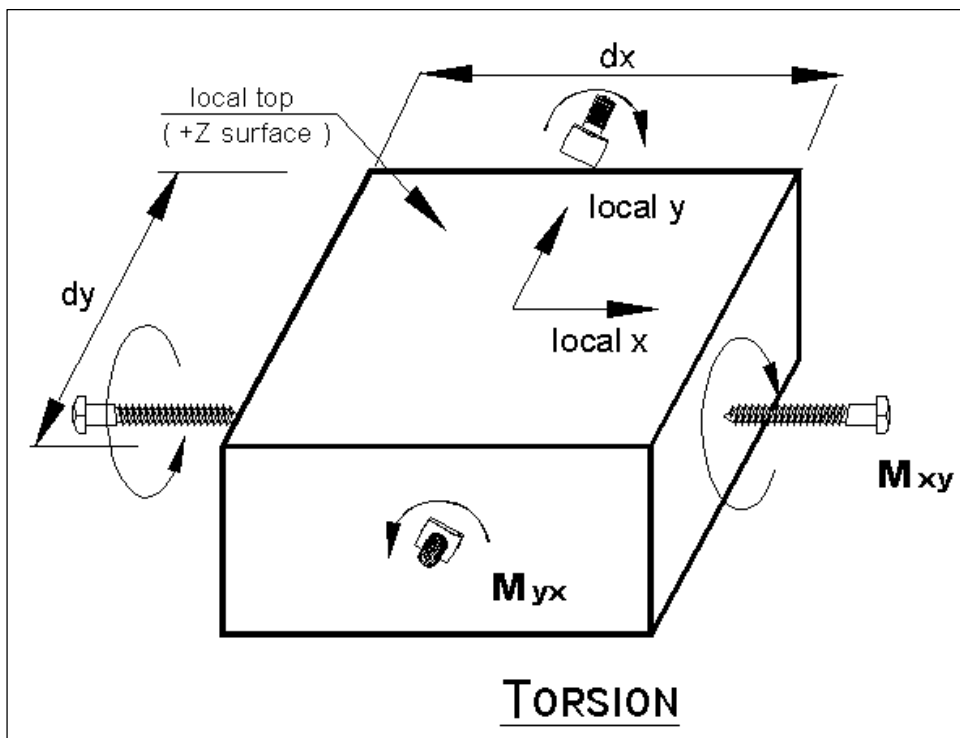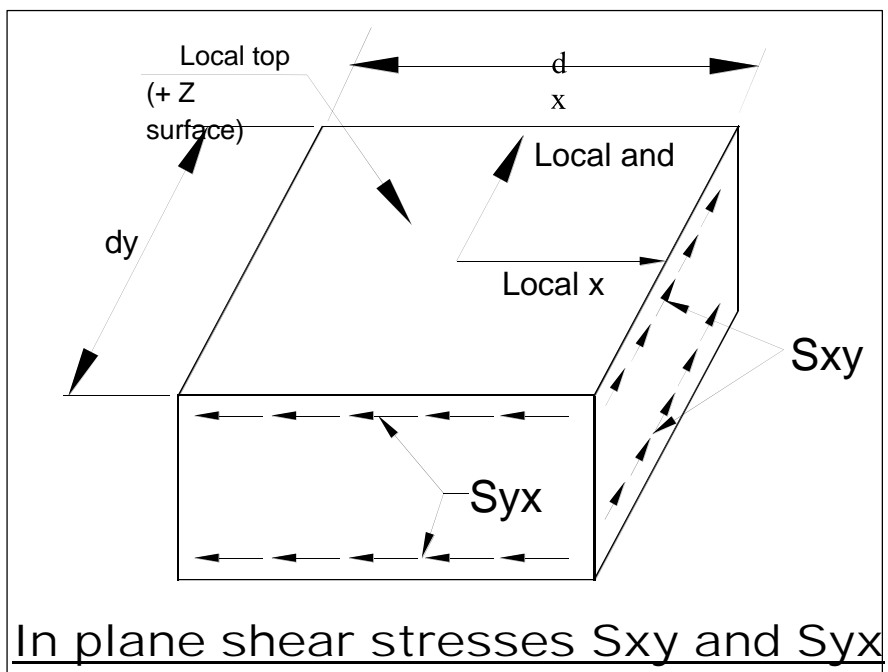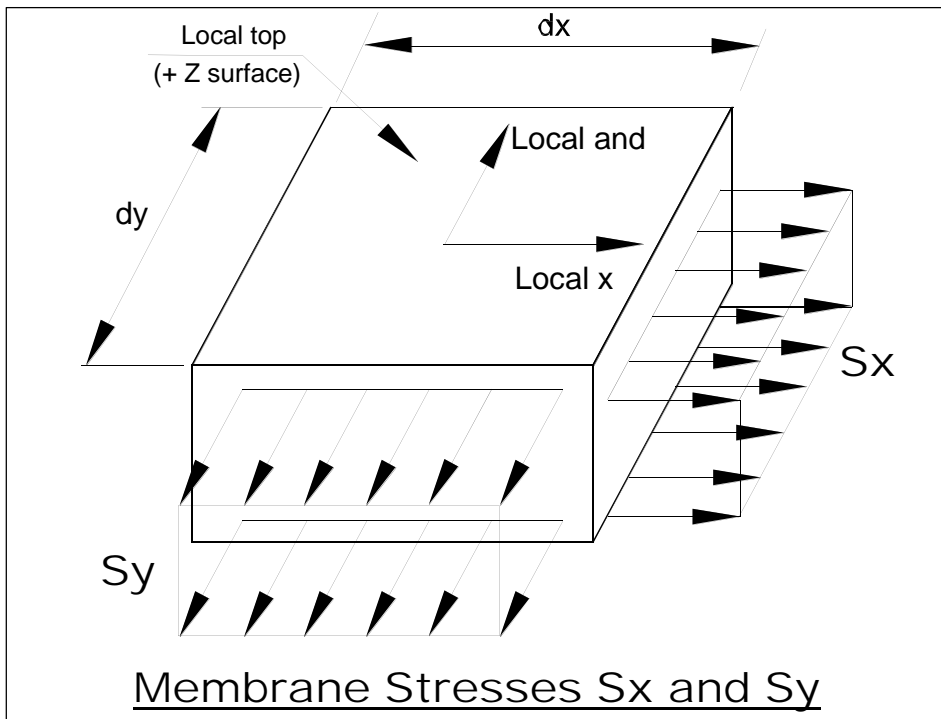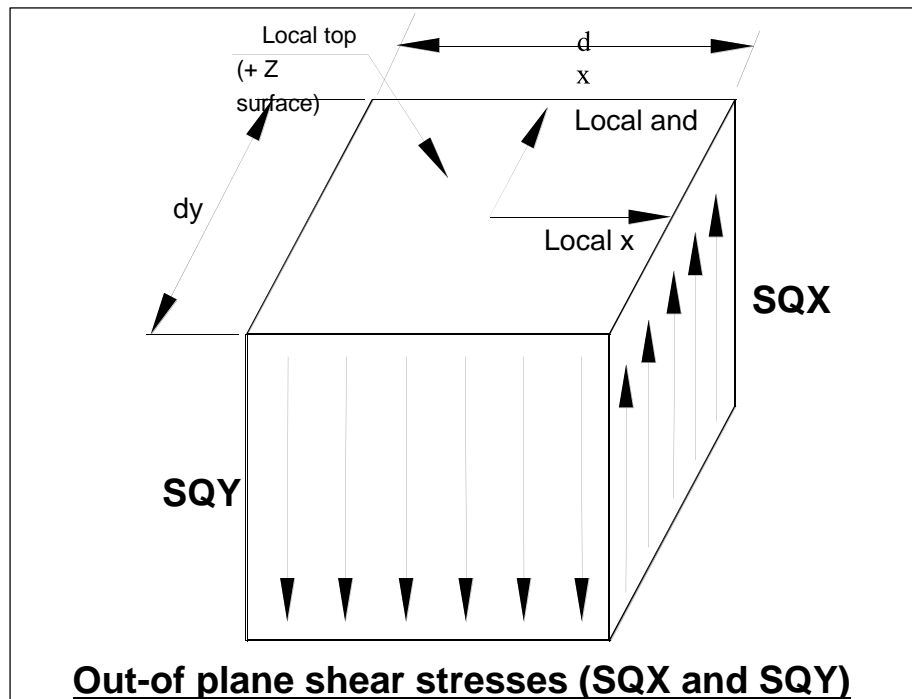
Keep in mind the following restrictions on the use of finite element STAAD:

1) Frame members and the finite element can be used together to Staad analysis. The mandate of ELEMENT INCIDENTS entry must follow MEMBER INCIDENTS immediately.

2) The dead weight of the finite element becomes loads in the joints in the connected nodes and not used as a pressure element.

3) Element forces are printed on the centroid and joints, but not along any edge.

4) In addition to the forces shown in Fig. 1.13 Von Mises forces on the part of the upper and lower surfaces of the element are also printed.

Numbering Elements

During the generation of the element stiffness matrix, the program checks if the element is the same as above. If it does not repetitive calculations. The sequence in which the element stiffness matrix is generated is the same as the sequence in which the elements enter in the incidences of the element.

Therefore, to save some computer time, must be sequentially numbered like elements. Fig. 1.14 shows examples element numbering efficient and inefficient.

However, the user must decide whether to adopt a numbering system which reduces the computation time with a numbering system that increases the ability to define the geometry of the structure.
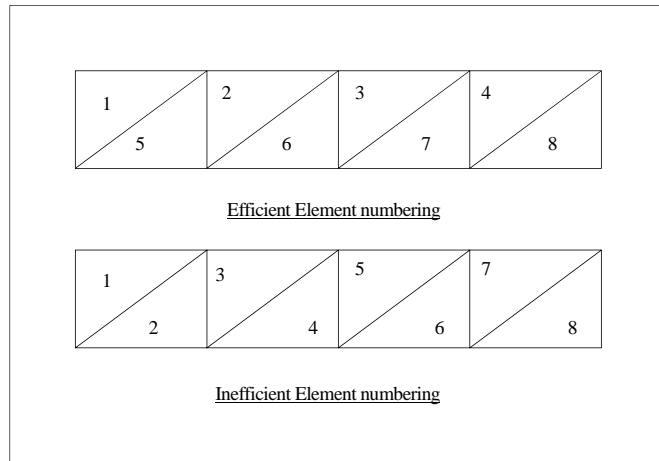
Figure 1.14

1.6.2 Solid Elements

The solids give the solution to the structural problems involving three-dimensional efforts. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where the finite element analysis using solid elements providing a powerful tool.

Rationale

The solid element is used to Staad isoparametric type eight knots. These elements have three translational degrees of freedom per node.
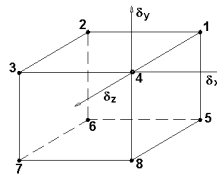
Collapsing several nodes together, a solid element of eight nodes can degenerate to the following forms with four to seven knots.
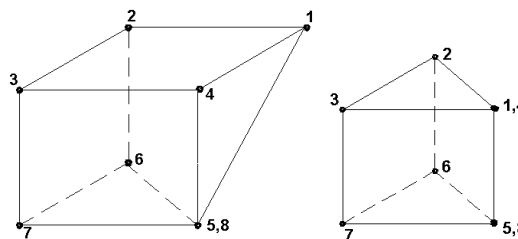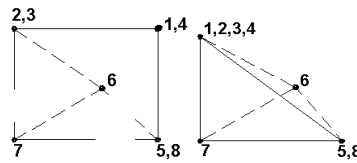
The stiffness matrix of the solid element is evaluated by numerical integration with eight points Gauss-Legendre. To facilitate numerical integration element geometry interpolating function is expressed using the natural coordinate system (r, s, t) of the element with its origin at the center of gravity. The interpolation functions are shown below:

$$x = \sum_{i=1}^{8} h_i x_i, \quad \text{and} \sum_{i=1}^{8} h_i \text{and}_i, \ z = \sum_{i=1}^{8} h_i z_i$$

Where x, y and z are the coordinates of any point on the element and xi, yi, zi, i = 1, .., 8 are the coordinates of nodes defined in the global coordinate system. Interpolation functions hi defined in the natural coordinate system, (r, s, t). Each of r, s and t ranges from -1 to +1. The fundamental property of the interpolation functions hi is unknown values in the coordinate system of natural units are at node i, and 0 in the other nodes of the element. The displacement of the element is also interpreted in the same way as the geometry. For completeness, the functions are given below:

$$u = \sum_{i=1}^{8} h_i \text{or}_i, \ v = \sum_{i=1}^{8} h_i v_i, \ w = \sum_{i=1}^{8} h_i w_i$$

Where u, v and w are the displacements at any point in the element and ui, vi, wi, i = 1,8 correspond to the nodal displacements in the coordinate system used to describe the geometry.

Three functions of "bubble" with zeros additional displacement surfaces and displacements in each direction is added for improving the development of shear to form a 33x33 matrix are added. The static condensation is used to reduce this matrix to a 24x24 matrix in the corners of the joints.

Local Coordinate System

The local coordinate system used in the solid element is equal to the overall system shown below:
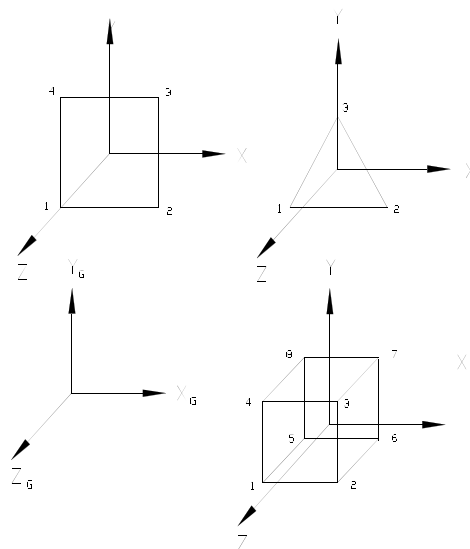
Figure 1.15

Properties and Constants

In the different members and shell elements (plate), any property to the solid elements is required. However, the constant as the elastic modulus and Poisson's ratio must be specified.

Also, the density needs to be provided if the weight is included in any load case.

Results Forces of the Elements

Element forces can be obtained at the center and on the boards of the solid element. The items to be printed are:

Normal stresses:	SXX, SYY and
SZZ cutting efforts:	SXY, SYZ and
SZX Major Efforts:	S1, S2 and S3.
Von Mises Stress:	

$$SIGE = \overline{\ S_1 - S_2\ ^2 +\ S_2 - S_3\ ^2 +\ S_3 - S_1\ ^2\ }$$

Direction cosines: 6 are printed cosine addresses following the DC term, corresponding to the first two directions of principal stress.

1.7   Member Properties

The following types of specifications of member properties are available in STAAD:

See section 5.20

A.   The specifications of prismatic properties

B.   Steel profiles standard library

C.   Steel tables that the user created

D.   The TAPERED sections (variables)

E.   Through the ASSIGN command

F.   The specification CURVE

1.7.1 Prism Properties (Prismatic Properties)

Prismatic the following properties are required for analysis:

AX =	Area of the cross section IX	
=	Torsional constant	
IY =	Moment of inertia about the axis. IZ	
=	Moment of inertia about the z axis.	

Additionally, the user can choose to specify the following properties:

AY = The effective cutting area for cutting force parallel to the axis and local. AZ = The effective cutting area for cutting force parallel to the local z axis. YD = The depth of the section parallel to the axis and local.

ZD = The depth of the section parallel to the local z axis.

To specify a T beam or a trapezoidal beam, the following additional properties to be provided.

YB = The depth of the soul of the T section [See figure below]
ZB = The width of the core section or the width T of the bottom of the trapezoidal section.

To specify the T beam, the user must specify YD, ZD, YB and ZB. Similarly for Trapezoidal sections, YD, ZD and ZB be provided.

If the cutting areas are given, the program will consider shear deformations automatically in the analysis, and if omitted, the shear deformation is ignored. In a frame structure, the ratio of shear strain at bending deformation is so small that in most cases can be ignored. The depths in the two major directions (YD and ZD) are used in the program to calculate the section modulus. They need only to calculate efforts or members for the design of concrete. You can skip YD and ZD values if efforts or the design of these members are not of interest. The default value is 254 mm (10 inches) for YD and ZD. Each prismatic properties are given in the local coordinates of the member.
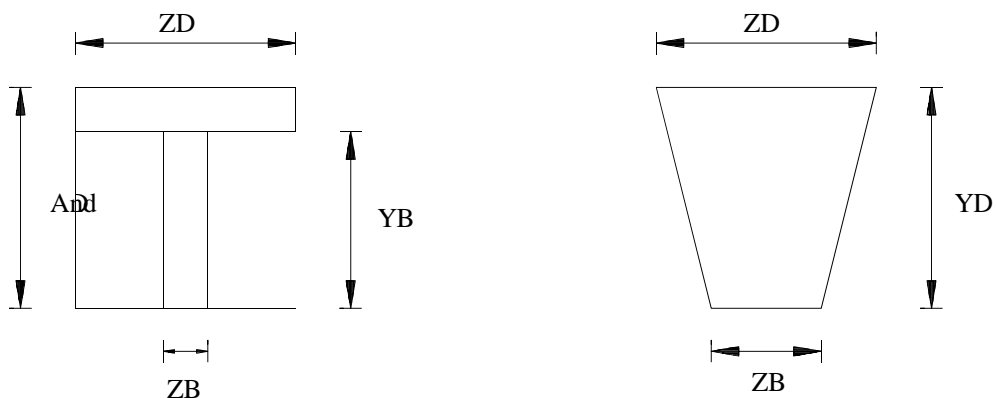
To define a concrete member, you should not provide the AX, But instead should provide YD and ZD for a rectangular section and YD only for a circular section. If no moment of inertia or cutting areas are provided, the program will calculate them automatically YD and ZD.

Table 1.1 provides the user with help in specifying required values section. List by structural type, the section properties required for any analysis. For FLOOR PLAN or analysis type, the choice of the moment of inertia required depends on the beta angle. If BETA equals zero, the desired property is IZ.

Table 1.1 Required Properties Type

Structure	Properties Structure
Required ARMOUR	AX
FLAT Structure	AX, IY or IZ
Structure FLOOR	IX, IY or IZ
Spatial Structure	AX, IX, IY, IZ

1.7.2   Library Rolled Steel Sections

See Section 2.2.1 and 5.20.1

This feature of the program allows the user to specify the names of the standard section steel profiles manufactured in different countries. Information concerning the forms of American steel is available in Section 2.

Since the cutting areas of the sections are built in tables, the shear deformation is always considered for these sections.

1.7.3   Steel Table Provided by the user

See sections 5.19 and 5.20.4

The user can provide custom steel table designated proper names and corresponding properties. The program can then find the member properties of those tables. The member selection can also be done with the program that selects the members of the tables that are provided only.

These tables can be provided as a part of the STAAD input files created separately or as of which the program can read the properties. The user does not use the standard forms shifty or who uses a limited number of specific forms can create permanent archives of member properties analysis and design can be limited to sections in these files.

1.7.4   Sections variables (Tapered Sections)

I variables section properties can be provided through the MEMBER PROPERTY specifications. Giving the key dimensions of the section, the program can calculate the cross-sectional properties used in the analysis result. The VARIABLES specification sections described in Section 5 of this manual.

1.7.5   Command Assign (Assign)

Through this command, the user can instruct the program to automatically select a steel section of the table for subsequent analysis and design. Section types that can be assigned include BEAM, COLUMN, CANAL, DOUBLE ANGLE and ANGLE. When a BEAM COLUMN or specified, the program will assign a section of I-beam (WF for AISC) and subsequently the selection of the member or optimization is performed with a section of similar type.

1.7.6   Curved Members

Any section can be assigned to a curved member. Elbows can also be assigned to a pipe ASME code based on the reduction of flexibility.

1.8   Degrees of freedom of Members

STAAD allows members freedom and stop elements. They may be left at liberty one or both ends of a member or element. It is assumed that members / elements are rigidly connected at joints with structural type specified. When this rigidity is not complete individual components of the forces, either at either end of the member may be reset with the statutes of relaxation of the member. Relaxation specifying components, the individual degrees of freedom are removed from the analysis. Relaxation components are given in the local coordinate system for each limb. Note that the relaxation time is also allowed PARTIAL.

Only one of the properties of sections 1.8 and 1.9 can be assigned to a Member. The last entry will be used.

1.9   Armor / Tension / Compression - Members Only

For analyzes involving members having only axial loads, ie the reinforcement members, there are two ways to specify this condition. When all members in the structure are members armor, the type of structure is declared as ARMOR (TRUSS) while when only some of the members are the armature members (eg bracing of a building), the ARMOR order MEMBER can be used where those members are identified separately. A considerable amount of analysis and design time saves declaring an axial member as a member ARMOR, the specification of such members as the members of the articulated frame with both ends should be avoided wherever possible.

In STAAD, commands the MEMBER TENSION (MEMBER TENSION) and MEMBER in COMPRESSION (MEMBER COMPRESSION) can be used to constrain the direction of the load that the member may have. The analysis will be performed accordingly. Refer to Section 5.23.3 for details of this option.

1.10   Tension / Compression - Solo Springs

In STAAD, the commands in TENSION SPRING (SPRING TENSION) and SPRING in COMPRESSION (COMPRESSION SPRING) can be used to limit the direction of the load that can be supported. The analysis will be performed accordingly. Refer to Section 5.23.4 for details of this facility.

1.11   Member Type Cable

1.11.1   Member Type Linear Cable

The wire-like members can be specified using the command MEMBER CABLE (MEMBER CABLE). When members are specifying type cable, the initial tension in the cable must be provided. The following paragraph explains how the stiff wire is calculated.

The increase in the length of a live wire is a combination of two effects. The first component is the elastic stretch, and is governed by the common ratio springs:

$$F = \text{Kx where K}_{elastica} = \frac{EA}{L}$$

The second component of elongation is due to a change in the geometry (such as a cable is pulled, the buckling is reduced). This relationship can be described by

$$\underline{F} = Kx \text{ but here } K_{sag} = \frac{12T^3}{w^2L^3} \left[\frac{1.0}{COS^2\alpha} \right]$$

Where

w = weight per unit length of the cable

T = tension in the cable

α = The angle to the cable axis makes with a horizontal plane (= 0, the cord is horizontal; = 90, the wire is vertical).

Therefore, the "stiffness" of a cable depends on the initial voltage applied (or buckling). These two effects can be combined as follows:

$$K_{comb} = \frac{1}{1/K_{sag} + 1/K_{elastic}}$$

$$K_{comb} = \frac{\frac{EA}{L}}{\left[1 + \frac{w^2L^2EA\,COS^2\alpha}{12T^3} \right]}$$

Where T = infinity, Kcomb =
EA / L where T = 0, Kcomb =
0

It may be noted that increasing the pressure (decrease the buckling) the combined stiffness approaches the pure elastic situation.

The following points need to consider using cable STAAD member:

1) The cable member is just a frame member whose properties buckling factor and voltage can accommodate. Member conduct cable is identical to that of an individual armor. You can only carry axial loads. As a result, the basic rules involved in modeling armor members have to be followed when members of the cable model. For example, when two wire members are in a common seal, if there is no support or 3rd member connected to that board, is a point of potential instability.

2) Because the reasons specified in 1) above, apply a transverse load on a cable member is not advisable. The burden will become two concentrated loads in the 2 ends of the cable and the true model of the deflection of the cable will never get done.

3) A wire member does not resist a compressive force applied to its ends. When the end of the member together with a compressive force are attached, they yield causing the wire to buckle. Under these circumstances, the

Cable has zero stiffness member and this situation has to be considered in the stiffness matrix and displacement have to be recalculated. But in STAAD, declaring the member merely to be a cable member is no guarantee that behavior will be considered. It is also important that you declare the member to have single stress using the STRESS MEMBER command after command CORD. This will ensure that the program will test the nature of the force in the member after analysis and compression if the member is off and the stiffness matrix is recalculated. Please read the requirements TENSION MEMBER command (MEMBER TENSION) recalling the RUN ANALYSIS (PERFORM ANALYSIS) and commands CHANGE (CHANGE) with each load case; SET command and NL.

4) Because of potential instability problems explained in item 1 above, users should also avoid modeling a catenary breaking it in several straight line segments. Member to Staad cable can not be used to simulate the behavior of a catenary. For the overhead, we are referring to these structural components and having a curved profile and develop axial forces due to its own weight. This behavior is in reality a non-linear behavior where the axial force is caused due to a change in the profile of the member or is induced by large displacements no such assumptions are valid in an elastic analysis. A typical example of a catenary is the U shape of the main wire used in suspension bridges.

5) Increased stiffness of the cable as the tension on it increases under applied load that is updated after each iteration if the members of the cable are also declared to be in TENSION MEMBER. However, the iteration stops when all the tension members are in tension or loose; When no wire tension converges. For integer multiple iterations, the RUN command consecutive ANALYSIS; will make a further iteration then for each entered command.

1.11.2 Members cable type Nonlinear

See section 5.23

The wire members for Nonlinear Analysis of Cable can be specified using the command type LEAD MEMBER. While wire-like members, the initial tension in the cable or cable length effortlessly be provided are specified.

1.12 Member of the Board was

See section 5.25

Some members of a structure can not concur with the incidents together creating a member off the board. This distance outside the ring is specified in the system of global or local coordinates (ie X, Y and Z distances from the incident together). Induced secondary forces due to off the board this connection, are taken into account when analyzing the structure and also to calculate individual member forces. The new centroid outside the gasket can be in the incidence of start or end point of the new asset will also be the new start or end of the member. Therefore, any reference beginning or end of that member will always be new points off the board.

MEMBER OFFSET
1   START 7
1   END -6
2   END -6 -9
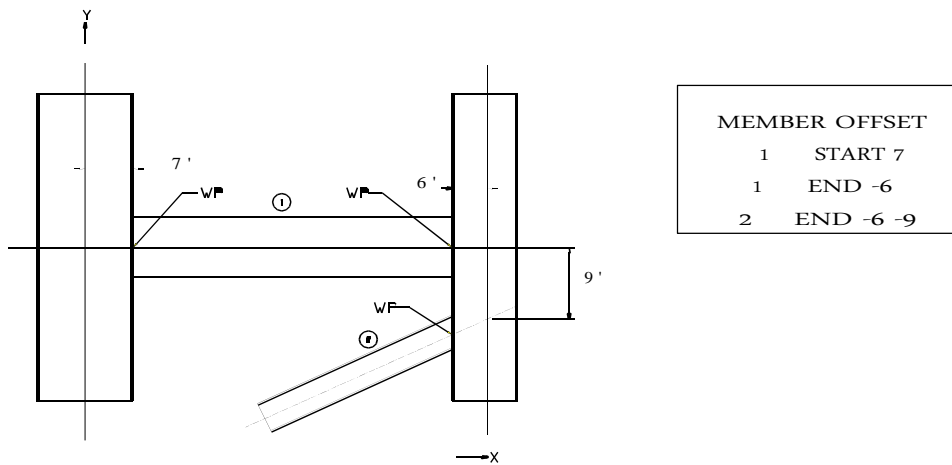
Figure 1.15

1.13   Constants Materials

The constants of the materials are: the modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); coefficient of thermal expansion (ALPHA), Composite damping ratio and angle beta (BETA) coordinates for any point of reference (REF).

See section 5.26

The value of E shall be provided for analysis or fail. The density (DEN) is only used when the weight (selfweight) structure is taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (normally referred to as G) by the formula,

$$G = 0.5 \times E / (1 + POISS)$$

If Poisson's ratio is not provided, STAAD assume a value for this amount based on the value of E. The coefficient of thermal expansion (ALFA) is used to calculate the expansion of the members if the temperature loads are applied. The temperature unit for loading ALFA temperature and must be the same.

The composite damping ratio is used to compute the damping ratio for each mode in a dynamic solution. This is only useful if there are several materials with different damping ratios.

The BETA angle and point of reference are discussed in Section 1.5.3 and they get as part of the constant member.

Note: The Poisson ratio must always be defined after the Modulus of Elasticity for a given element or member.

1.14   Stands

STAAD allows media specifications that are parallel and inclined to the global axes.

Braces are specified as ARTICULATED (PINNED), wardrobes (FIXED) or wardrobes (FIXED) with different relaxations. An articulated support has restrictions against any translational motion and none against rotation. In other words, an articulated support for the reactions have all forces but resist any time. A recessed holder has restrictions from all directions of motion.

They can also relax the constraints of a fixed support in any desired direction as specified in Section 5.

You can also specify translational and rotational springs. The springs are represented by the relation to their spring constants. A translational spring constant is defined as the force for replace a board supporting a unit of length in the specified global address. Similarly, a rotational spring constant is defined as the force to rotate the support board a degree about the global address specified.

For the static analysis supports Multi-linear spring may be used to model the variation, nonlinear resistance of a medium (eg soil). See section 5.27 for descriptions of options for shoes and foundation slabs in an elastic medium.

1.15   Joints Master / Slave (Master / Slave Joints)

The choice of master / slave is provided to allow the user to model the rigid links in a structural system. This can be used to model the elements
special structural and rigid floor diaphragm. Some boards working as a slave which can be given to the same displacement as the board of the master will be assigned. The user also has the flexibility to choose the specific degrees of freedom for movement restrictions are imposed on the boards worked as a link. If all degrees of freedom (Fx, Fy, Fz, Mx, My and Mz) are provided as the constraints, it is assumed that the joints are rigidly connected.

1.16   Loads (Loads)

Loads can be specified in a structure as loads on the board, member loads, temperature loads and loads of cupboard at one end member. STAAD can also generate the weight of the structure and can use it as uniformly distributed loads member in the analysis. Any part of this own weight can be applied in any desired direction.

1.16.1 Loads on Boards (Joint Load)

The loads on the joints, forces and moments can be applied to any free board structure. These loads act in the global coordinate system of the structure. Positive forces act in the directions of positive coordinates. Any number of loads can be applied in a single meeting, in which case the charges will be add-on that board.

1.16.2 Loads on Members (Member Load)

Three types of load member can be applied directly to a member of a structure. These charges are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). The Standard loads act in the full or partial length of a member. The concentrated loads act at any intermediate point specified. Linearly varying loads acting on the total length of a member. V linearly varying loads acting on the full or partial length of a member. Trapezoidal loads in a uniform load and concentrated loads some converted.

Any number of loads can be specified to act on a member in any independent loading condition. Member loads can be specified in the coordinate system of the member or the global coordinate system. Uniformly distributed loads can be specified in the member in the global coordinate system to act along the whole or the projected length of the member. Refer to Fig. 1.3 to find a member's relationship to the global coordinate system to specify the member loads. Positive forces acting on the directions of the positive, local or global coordinates, as appropriate.
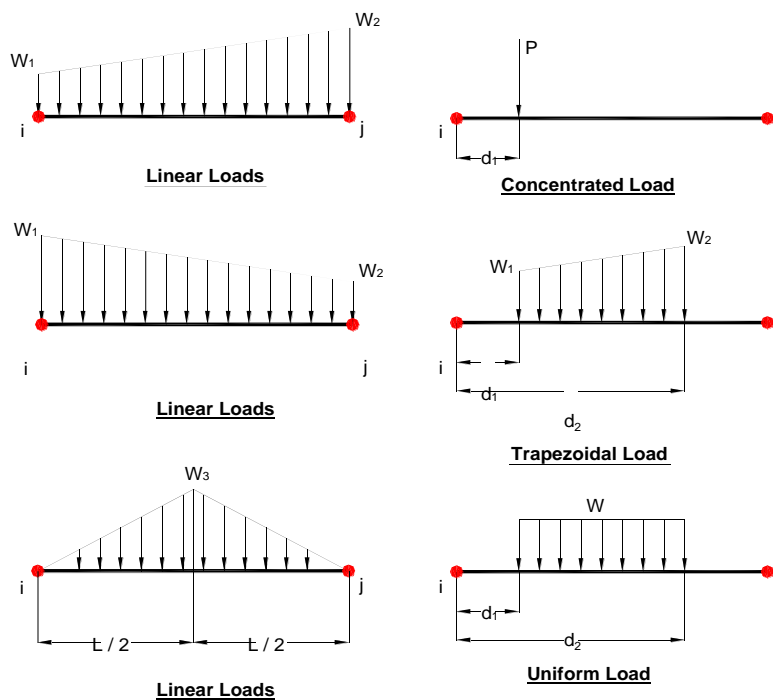
The Load Settings Member - Figure 1.16

1.16.3 Cargo Area or Floor (Area / Floor Load)

Often a floor (limited by the XZ plane) is subjected to a uniformly distributed load. It could require a lot of work to estimate the burden of membership for individual members on that floor. However, with the LOAD command or LOAD FLOOR AREA, the user can specify the load area (unit load per unit of area) for members. The program will calculate the tax area for these members and provide loads the appropriate member. The load area is used for a distribution in one direction and the load floor is used for distributions in two ways.

The following assumptions are made when loading cargo area and floor-to-member load is transferred:

1) It is assumed that the load of the member is a linearly varying load values for start and end may be of different magnitude.

2) The tributary area of a member with a load area is calculated based on half the space nearby on both sides approximately parallel members. If space is greater than or equal to the length of the member, the load area is ignored.

3) Do not load specified Area / Flat for members declared MEMBER CABLE ARMOR MEMBER MEMBER in TENSION, COMPRESSION MEMBER in or CURVO.

Figure 1.17 shows a floor structure with a specification of 0.1 area load.
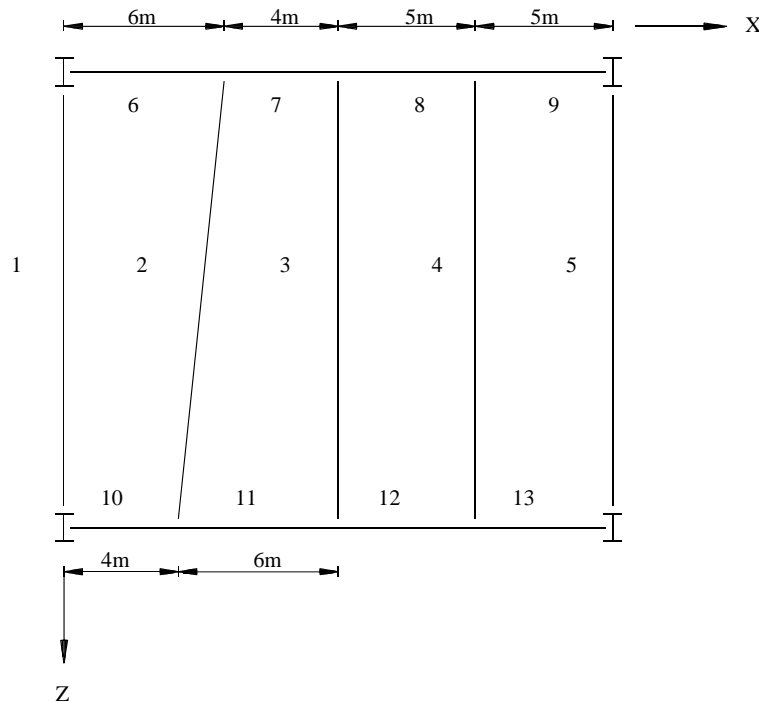
Figure 1.17

The member 1 has a linear load of 0.3 to 0.2 at one end and other end. The members 2 and 4 have a uniform 0.5 of the total length up load. Member 3 will have a load

0.45 and 0.55 line in the respective ends. Member 5 will have a uniform load of 0.25. The remaining members, 6-13, will not have the burden on taxpayers because the area closest parallel members are separately over each of the lengths of the member. However, the reactions of the members shall be considered to the beam. Solid plates and these commands are ignored.

1.16.4   Built-in member loads in Far

You can also specify loading effects on a member as it relates to their charges fixed end. These charges are defined in terms of the coordinate system of the Member and directions will be opposite to the actual load on the member. Each end of a member can have six forces: axial; cutting; cutting z; torsion; time and time and z.

1.16.5   Loads for Prestressed and Post Stressed in Members

Members in a structure may be subject to prestressing loads so the load distribution in the structure to be investigated. Prestressing loads can be applied in a Member axially or eccentrically. Eccentricities be provided to the board start, middle, or end. These eccentricities are only on the local axis. A positive eccentricity is positive Y. Only local address in the local axis eccentricities are provided and care should be taken to provide the prismatic properties or to specify the correct angle BETA when necessary rotate the coordinates of the member. Two types of prestressing load specification are available; PRESTRESSING (prestress) where the effects of the load the rest of the structure is transmitted, and (POST-TENSIONING) POSTSTRESS where charging effects are only experienced by members in that it is applied.

1) It is assumed that the cable has a generalized parabolic profile. It is assumed that the equation of the parabola is

$$\text{and } ax^2 + bx + c$$

Where:

$$t = \frac{1}{L2} \left(2es - 4em + 2ee \right)$$

o

$$b = -\frac{1}{L} \left(4em - ee L - 3es \right)$$

$$c = is$$

Where   is   =   eccentricity of the cable to the beginning of the member
ee   =   the eccentricity of the cable through the member (local axis)
e   =   eccentric cable end member (local axis)
L   =   Limb length

2) The tilt angle of the cable with respect to the local x axis (a straight line connecting the start and end joints of the limb) the start and end points are small which gives rise to the presumption that

$$\text{run out} \theta = \theta = dy / dx$$

Hence, if the axial force in the cable is P, the vertical component of the force to the end is P (dy / dx) and the horizontal component of the rope force is,

$$P\sqrt{1 - \left(\frac{dy}{dx}\right)^2}$$

Users to ensure that your profile meets this requirement cable is advised. Less than 5 degrees are recommended

3) The member is tested for the effects of prestressed or post-tensioned using the equivalent load method. This method is well documented in most books recognized in the Analysis and Design of Prestressed Concrete. The magnitude of the uniformly distributed load is calculated as

$$udl = \frac{8\,Pe}{L^2}$$

$$and \quad = \quad \frac{+}{2} \frac{s\,ee}{} \quad em$$

Where:

P = axial force on the wire L = length of the member

4) It is assumed that the force on the cable is the same along the length of the member. No reduction is taken into account in the cable due to friction or other losses.

5) PRESTRESSED MEMBER The term as used in STAAD means the following condition. The structure is built first. Then, the prestressing force is applied to the relevant members. As a result, members are deformed depending on the conditions of its ends, the forces are transmitted to other members in the structure. In other words, "PRE" refers to the time of placing the member on the structure relative to the time of pre-stressing.

6) The POST-TENSIONING MEMBER term as used in the following condition Staad means. members in which such charge is applied are first molded in the factory. After this, the prestress force is applied to them. Meanwhile, the rest of the structure is built on the building site. So prestressing members are brought and placed in position on the partially built structure. Because this sequence, the effects of prestressing are "experienced" by prestressing members only and not transmitted to the rest of the structure. In other words, the "POST" refers to the time of placing the member on the structure relative to the time of esforzamiento.

7) As may be evident from Article (6) above, it is not possible to estimate the displacements of the ends of the members post-tensioned to the effects of post-tensioning and is assumed to be zero. As a result, intermediate displacements of sections l (See SECTION OFFSET command) are measured by relating as a straight line connecting the start and the end of the boards of member defined by its coordinates in the initial BOARD.

1.16.6 Load Temperature / Voltage (Temperature / Strain Load)

Uniform temperature differences along the members and items can be specified. Temperature differences across both sides of the members and through the thickness of plates can also be specified (the temperature is uniform only for solids). The program calculates the axial strain (elongation and shrinkage) due to the different temperatures of the members. This calculates the induced forces in the member and the analysis is done accordingly. They can get the voltage ranges elongation and shrinkage directly.

1.16.7 Load by Displacement the Support (Support Load Displacement)

A static structure thus refers to the displacement of the supports may be applied loads. The displacement can be translational or rotational. Translational movements in units of length are provided while rotational movements are always in degrees. Note that you can only specify the displacements in directions in which the support has a specification "force" in the Command Support.

1.16.8 Loading in the Elements (Loading on Elements)

The elements of plate / Eggshell, load Allowable types are:

1) Pressure loads consisting of loads acting perpendicular to the surface of the element. Pressure loads can be of uniform or trapezoidal intensity and intensity variation on a small or on the entire surface of the element portion.

2) The loads on the board are forces or moments are applied at the joints in the direction of the global axes.

3) The loading temperature may be constant throughout the plate element (causing only elongation / or shortening) may vary through the depth of a plate bending element causing the plate element. The coefficient of thermal expansion for the material of the element shall be provided to facilitate the calculation of these effects.

4) The weight of the elements can be applied using the load condition SELFWEIGHT. The density of the elements has to be given to facilitate the calculation of the own weight.

In the **Solid Elements**, The only three available charging types are

1) The weight of the solid elements using load condition SELFWEIGHT may apply. The density of the elements has to be given to facilitate the calculation of the own weight.

2) The load on the joints which are forces or moments are applied at the joints in the direction of the global axes.

3) The loading temperature may be constant along the solid elements (causing only lengthening / shortening). The coefficient of thermal expansion for the material of the element should be to facilitate the calculation of these effects.

> **Note: Only the translational stiffness is supported on solids. Thus, at the joints where solids only, no moments are applied. For best efficiency, the rotational supports should be used in these joints.**

1.17 Load Generator (Load Generator)

STAAD is equipped with built-in algorithms to generate moving loads and seismic lateral loads (Uniform Building Code and the 1893 IS Code) in a structure. Using the application load generation consists of two parts:

1) The definition of the system (s) load.

2) Primary charge generation using the system (s) predefined load).

The following sections describe the salient features of the moving load generator, the generator of seismic load and wind load generator available.

1.17.1 Loads Generator Mobile (Moving Load Generator)

See Sections 5.30.1 and 31/05/12

This feature allows the user to generate moving loads in the members of a structure. The system (s) of moving load is concentrated load to specified fixed distance in both directions in a plane and can be defined by the user. If a user specified number of primary load cases as a result will be generated by the program and will be considered in the analysis. Loads of The American Association of State Highway and Transportation Officials (AASHTO, 1983) are available within the program and can be specified using normal AASHTO designations.

1.17.2 Seismic Loads Generator UBC / IBC (UBC / IBC Seismic Load Generator)

See Sections 5.30.2 and 31/05/12

In STAAD seismic load generator follows the UBC or UBC procedure equivalent lateral load analysis. It is assumed that the lateral loads shall be applied in the X, Y and Z be the direction of the gravity loads. Thus, for a model of the building, the Y axis is perpendicular to the floors and point upward (positive coordinates of all joints Y). The user must provide their model accordingly. The total lateral seismic force or base shear is calculated automatically by using STAAD UBC appropriate equation.

To generate load codes, requires the user to provide the coefficients of the seismic zone, importance factors, periods, etc. See Section 5.31.2 for detailed input required for each code.

Instead of using UBC approximate formulas for estimating the period of the building at a certain address, the program calculates the period using the Raleigh ratio technique. This period is then used to calculate the seismic coefficient C.

After the base shear is calculated from the appropriate equation is distributed among the various levels, extending UBC specifications. The cutting basal are then distributed as lateral loads applied to the structure. These loads can then be used as normal load cases for analysis and design.

1.17.3 Wind Loads Generator (Wind Load Generator)

See Sections 5.30.3 and 31/05/12

The loading STAAD Wind is able to calculate wind loads on the boards of a structure when the user specified Wind intensities and exposure factors. Can be specified for different intensities of wind different areas of the structure height. They can be shaped openings in the structure using the exposure factors. An exposure factor is associated with each joint of the structure and is defined as the fraction of the catchment area in which they operate wind loads. Built-in algorithms calculate the exposed area based on automatically limited by the Member (not considered plates and solid), then calculates wind loads of intensity and exposure and tucked distributes loads and lateral loads on the joints areas.

1.18 Options Analysis

ANALYSIS The following options are available in STAAD.

1) Stiffness Analysis / Linear Static Analysis

2) Static Analysis of Second Order

   a. P-Delta Analysis

   b. Nonlinear Analysis

   c. Linear Multi Spring Bracket

   d. Only Member / Spring Tension / Compression

3) Dynamic Analysis

   a. Through Time

   b. Response Spectrum

Salient features of each type of analysis in the following sections are discussed. The detailed theoretical treatments of these features are available in the standard textbooks of structural engineering.

1.18.1 Analysis of Stiffness

The stiffness tests performed on Staad is based matrix displacement method. In the analysis of the array of structures to the displacement method, the first idealized structure an assembly of discrete structural components (members of the frame or finite element). Each component has a scroll shape assumed in a manner which satisfies the balance of power and compatibility of the displacement of the joints.

The structural system as slabs, sheets, approximately square foundations, etc., that transmit loads in two directions have to be discretized in finite element 3 or 4 nodes connected to each other. Charges may apply in the form of distributed loads on the surfaces of the element or as concentrated loads at the joints. The effects of voltage levels and the effects of bending of the plate taken into account in the analysis.

Analysis The consideracione s

For a complete analysis of the structure, the necessary matrices are generated based on the following assumptions:

1) The structure is idealized into an assembly of beams, plates and solid type elements joined at their vertices (nodes). The structure is loaded and reacts to concentrated loads acting at the knots. These loads may be forces and moments that can act in any direction specified

2) A beam member is a longitudinal structural member having a constant, a doubly symmetrical or near symmetrical doubly through its cross-sectional length. Beam members almost always carry axial forces. They can also be subjected to shear and bending in two arbitrary perpendicular planes, and can also be subjected to torsion. From this point of view these beam members are called "members" in the manual.

3) A plate element is a flat element that three or four nodes have the constant thickness. A three-dimensional solid element is an element of 4-8 knots. These plates and solid elements are called "elements" in the manual.

4) Internal and external loads acting at each node are balanced. If the torsional or flexural properties are defined for any member, six degrees of freedom are considered at each node (that is, three translational and three rotation) in the generation of the relevant matrices. If the member is defined as reinforcement member (ie carrying only axial forces) then only the three degrees (the translation) of freedom is considered at each node.

5) Two types of coordinate systems in the generation of the required matrices are used and are called local and global systems.

Local coordinate axes are assigned to each individual element and is oriented such that the computation effort for the element stiffness matrices generalizes and minimized. The axes of the world coordinates are a common data set for all idealized elements to the element force and displacement can be related to a common frame of reference.

The   Basic Equation

The matrix of complete rigidity of the structure is obtained by systematically adding the contributions of the various members and stiffeners. External loads on the structure are shown as discrete action of concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the movement of the nodes by the equation:

$$A_j = A_j + S_j x \ D_j$$

This formulation includes all joints of the structure, either if they are free to move or restricted for the supports. These components of displacements of the board you are free to move are called the degrees of freedom. The total number of degrees of freedom is the number of unknowns in the analysis.

The Resolve method pa ra zamientos Desplà

There are many methods to solve the unknowns in a set of simultaneous equations. An approach which is particularly suitable for the structural analysis method is called decomposition. This method was selected for use in STAAD. Since the stiffness matrices of all elastic structures are always linearly symmetrical, a particularly efficient form of decomposition called Modified Cholesky method can be applied to these problems. This method is very accurate and very suitable for the Gauss elimination process by solving the simultaneous equations.

Consideration Bandwidth

The decomposition method is particularly effective when applied to an array of symmetric band. For these types of matrix calculations are required less due to the fact that outside of the band elements are all zero.

STAAD takes full advantage of the concept of bandwidth during the solution, to seek to have the lowest bandwidth for the most effective solution. For this purpose, STAAD offers features for which the program can internally reorder the numbering of the joints to provide better bandwidth.

Integrity ructural Est

The integrity of the structure is a very important requirement that must be satisfied for all models. Users should ensure that the developed model represents only one structure, not two or more separate structures.

A structure "whole" or "a" structure can be defined as a system where appropriate "rigid connections" between members / elements. The functions of the complete model are taken as a single integrated load resisting system. Two or more independent structures within a model result in erroneous mathematical formulations and therefore generates numerical problems. Staad structural integrity check using a sophisticated algorithm and provides reports of multiple structures in the model.

Numerical Modeling Issues and Instability

Instability problems can occur due to two primary reasons.

1) **Modeling Problem: There are a variety of problems in modeling can lead to unstable conditions. They can be classified into two groups.**

   a. **The local instability - A local instability is a condition where the conditions of embedment in the end (s) of an individual is such that cause instability in the member about one or more degrees of freedom. Examples of local instability are:**

      i. **Relaxation member: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) will be subject to this problem.**

      ii. **A designed with columns and beams structure where the columns are defined as members "ARMOR." Such a column has no ability to transfer shear or moments of the superstructure to supports.**

   b. **The Global Instability - It is caused when the supports of the structure are such that they can not resist to slide or overturn the structure in one or more directions. For example, a 2D (a frame in the XY plane) structure that is defined as a SPATIAL FRAME with articulated supports and subjected to a force in the direction of Z will dump over the rotating axis X. Another example is that of a spatial framework with all the supports released for FX, FY and FZ.**

2) **Precision Mathematics: An error occurs when mathematical precision numerical instabilities occur in the decomposition of the matrix (investment) process. A condition of equilibrium equation takes the form 1 / (1-A), where A = k1 / (k1 + k2); k1 and k2 being the stiffness coefficients of two adjacent members.**

When a member very "rigid" is adjacent to a very "flexible" member, vis., When the k2 >> k1, k1 + k2 or k1, A = 1 and, hence 1 / (1-A) = 1 / 0. So, do not allow large variations in the stiffness of adjacent members. Artificially high values of E or I should be reduced when this occurs.

It also causes error mathematical precision when the units of length and strength to properly lengths member is not defined, member properties, constants, etc.

Users also have to ensure that the defined model structure represents only one, not two or more separate structures. For example, in an effort to model an expansion joint, the user may end up defining separate structures within the same input file. Multiple structures defined in the same file can lead to grossly erroneous results.

1.18.2   Analysis of Second Order (Second Order Analysis)

See section 5.35

STAAD provides the ability to perform stability analysis of second order. Two methods are available a simplified method called the P-Delta analysis and a detailed method called Nonlinear Analysis. Both methods are explained below.

1.18.2.1   P-Delta Analysis

See section 5.35

Structures subjected to lateral loads often experience secondary forces due to moving the point of application of the vertical loads. This side effect, commonly known as the P-Delta effect, plays an important role in the structure analysis. In STAAD, a single procedure has been adopted to incorporate the effect of P-Delta analysis. The procedure involves the following steps:

1)   First, the primary deflection is calculated based on the application of external loads.

2)   Are then combined primary loads deviations originally applied to create the secondary loads. The load vector is then revised to include the side effects

Note: The side load must be submitted concurrently with the vertical load for proper consideration of the effect of P-Delta. The option REPEAT LOAD command (REPEAT LOAD) (see Section 5.32.11) was created with this requirement in mind. This option allows the user to combine the cases of pre-defined primary charge to create a new primary charging case.

3)   A new analysis of rigidity is carried out based on the revised load vector to generate new deflections.

4)   The forces in the Element / Member and reactions are calculated based on the new deflections.

It may be noted that this method yields very accurate results with small displacement problems. Allows the user to Staad through multiple iterations of the procedure of P-Delta if necessary. Allows the user to specify the number of iterations depending on the requirement. To set the convergence tolerance of displacement between the SET DISP command before the coordinates of the Board. If the change in the rule of displacement from one iteration to another is less than f then converge.

The P-Delta analysis is recommended by various design codes such as ACI 318, LRFD, IS456-1978, etc. instead of time amplification method to calculate more realistic forces and moments.

The P-Delta effects are only calculated for framing members and elements of the plate only. They are not calculated for the solid elements. The P-Delta and Nonlinear analysis restricted to structures where members and plate elements carry the vertical load from one level to the next structure.

1.18.2.2 Nonlinear Analysis

STAAD also offers the ability to perform nonlinear static analysis based on the geometric nonlinear analysis. The algorithm of nonlinear analysis incorporates both geometric corrections member stiffness and secondary loads.

See section 5.35

The Nonlinear analysis methodology is generally adopted for structures subjected to large displacements. As large displacements generally produce significant moves the point of application loads, consideration of secondary loads becomes an important criterion. In addition, the geometric stiffness corrections to take into account the modified geometry apply. Since geometric stiffness corrections are based on the generation of movement are different for different load cases. This makes the analysis option is non-linear load dependent. Staad algorithm of nonlinear analysis involves the following steps:

1) First, the primary movements are calculated for the applied load.

2) Stiffness corrections are applied in the Member / plate based on stiffness matrix shifts observed in the element. The new global stiffness matrix is assembled based on the stiffness matrices revised member / element which uses the coordinates of the board updated.

3) Load vectors are revised to include the side effects due to the primary movement.

4) The new set of equations is solved to generate the new displacement.

5) Forces on Elements / Members and the support reactions are calculated from these new displacement.

6) STAAD algorithm of non-linear analysis allows the user to go through multiple iterations of the above procedure. The number of iterations can be specified by the user based on the requirements. It may be noted, however, that multiple iterations may increase the requirements of computer resources and execution time substantially.

Note: The following points should be noted with respect to the choice of non-linear analysis

7) Since the procedure is dependent on the load, the user is required to use the SET NL command and the command CHANGE properly. NL SET command be provided to specify the total number of cases of primary charging. CHANGE command should be used to restore the stiffness matrices after each load case.

8) When geometric corrections are based on displacement, all loads that are capable of producing significant shifts should be part of the case (s) charge for the non-linear analysis.

9) To set the tolerance of divergence of displacement, a SET DISP f is entered before the coordinates of the Board. If any shift in any iteration exceeds f, then the solution is diverging and terminated. Value

f is preset for the largest overall width, height, or depth of the structure divided by 120.

The P-Delta and Nonlinear analysis restricted to structures where members or plates carry the axial load from one level to the next structure.

1.18.2.3   Multi-Linear Analysis

When the ground is modeled as spring supports, the resistance change due to external loads can be modeled using this option, as well as their behavior differs from tension to compression behavior. The characteristics of soil stiffness-displacement can be represented by a multi-linear curve. The amplitude of the curve will represent the ground spring characteristic different offset values. The load case on an analysis of Multi-linear springs must be separated by the CHANGE command and should be done with the ANALYSIS EXECUTE command. The SET-NL command be provided to specify the total number of cases of primary charging. There can be no PDELTA, NONLINEAR, dynamic, or membership in cases TENSION / COMPRESSION. The command of Multi-linear spring start an iterative analysis continues until convergence.

1.18.2.4   Analysis of Tension / Compression Only

When members or support springs are linear but carry only tension (or compression only), then this analysis can be used. This analysis is automatically selected if any member or spring has been characteristic only tension or compression. This analysis is an iterative analysis continues until convergence. Any member / spring fails in his opinion be inactivated (not shown) in the next iteration. The iteration continues until all members have the appropriate load address or are inactive (the predefined iteration limit is 10).

This is a simple method that can not be worked in some cases because the members are removed at internal iterations because they are needed for stability. If messages appear instability at 2 and subsequent iterations that did not appear in the first cycle, then use the solution. If this happens in cases where the springs are the only entities tension / compression, then use the analysis of Multi-linear spring.

The load cases on an analysis of tension / compression must be separated by the CHANGE command and should be performed with the RUN command from the ANALYSIS. NL SET-command be provided to specify the total number of cases of primary charging. There can be no case of Multi-linear springs, NONLINEAR, or dynamic cases.

1.18.2.5   Nonlinear Analysis Cable / Armor

When all members and supports spring elements are linear except the cable and truss members, then this analysis can be used. This analysis is based on applying the load balancing steps with iterations until convergence in each step. The step size starts small and gradually increases (15 steps is the default), the iteration continues until each step change in strain is

little before proceeding to the next step. If there is no convergence, then the step is repeated with step size low (once). If you have not converged, then the solution is stopped. The user can then select more steps or alter the structure and reanalizarla. Conversely, if several consecutive steps easily converge, then the step size is increased automatically.

Structures artificially stabilized during the first load steps if the structure is initially unstable (in the small linear static displacement, in a theoretical sense).

The user has control over the number of steps, the number of iterations for step, tolerance of convergence, and artificial stiffness stabilizer

This method assumes small displacement theory for all members / armor / items rather than cables and preloaded reinforcement. Cables and preloaded armor can have large or moderate / great efforts displacements. It is assumed that the wires only carry loads of shear stress to the wire at each point. Preloaded armor may only carry only tension and compression or tension. The claim is necessary to stretch the cable / armor without straining its length to allow it to fit between the two end joints strength. Alternatively, you can enter longitude for cables without straining.

The cable / preloaded course armature is automatically loaded by the weight, and optionally temperature. Moreover loads applied to the cables / Preloaded armor are ignored. Is it right to interrupt the cable / armor several members and apply forces to the intermediate joints.

It is assumed and
upward.

In this analysis, the geometrical beams have armor and rigidity.

The strength of the member for printed wire is tangent to Fx and deformed wire board (not along the line of the cord).

The load cases in a nonlinear analysis of the cable must be separated by the CHANGE command and must be made by the RUN command ANALYSIS CABLE. The SET-NL command be provided to specify the total number of cases of primary charging. There can be no case of Multi-linear spring, compression only, PDELTA, NONLINEAR, or dynamic cases.

1.18.3 Dynamic Analysis

The means of dynamic analysis currently available include solving the problem of free vibration (the eigenproblem), response spectrum analysis and forced vibration analysis.

Solution of Eigen problem

The eigenproblema is solved for the frequencies of the structure and the mode shape considering a diagonal matrix of concentrated masses, masses possible in all

possible assets d.o.f. included. The subspace iteration method for all sizes of the problem (the default), and the optional method of determining search for small problems: They can be used two methods of solution.

Mass Modeling

The natural frequencies and mode shape of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forces function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. The results of calculations of the frequency and mode shapes can vary significantly depending on the modeling of the masses.

This variation, in turn, affects the response spectrum analysis and the results of forced vibration. Thus, extreme caution must be exercised in modeling the masses into a problem of dynamic analysis.

In STAAD, modellation all bodies that are able to move as loads applied in all possible directions of movement. Although it is known that the load will be in one direction there is usually mass movements in other directions in some or all joints and these directions of the masses (the "charges" in weight) must be entered to be correct . Moments of the board that are admitted are

shall consider are moments of inertia weight (in units of force-length2   ).

Please enter the weight and loads on the joints and elements in the global addresses with the same sign as much as possible so that the "masses" are not canceled. Member loads / element may also be used to generate translational masses joints.

Note: The moments in joints extreme members generated by the load on the member (including concentrates now) is dismissed as irrelevant to the dynamics. Enter the moments of inertia, if necessary, at meetings of the board as moments.

STAAD uses a diagonal matrix mass of 6 equations for mass concentrated together. The weight or loads evenly member 50% focus on each end joint without rotational mass moments of inertia. The other elements are integrated, but talking real sense the weight is equally distributed between the joints of the element.

The members / elements in the theory of finite element are simple mathematical representations of deformation applied over a small region. FEA procedures converge if you subdivide the elements and run the analysis again; then subdivides the elements that have significantly changed the results and run the analysis again; etc. until significant results converge to the accuracy needed.

An example of a simple beam problem that needs to subdivide the real members to better represent the mass distribution (and the dynamic response and the force distribution along the member) is a single-beam floor between 2 columns

ization

ground motion at the base. This analysis is performed using the modal superposition method.

Hence, all masses must be modeled as active loads to facilitate the determination of modal shapes and frequencies. Please refer to the section above on "modeling mass" for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the lowest modes "p". The equilibrium equations are written as

$$[M]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{P_t\}_{...} \qquad ...(1)$$

Using the transformation

$$x = \sum_{i=1}^{p} \phi_i q_i$$

Equation 1 reduces to "p" to separate equations of the form

$$\ddot{q}_i + 2\xi_i\omega_i\dot{q}_i + \omega_i^2 q_i = R_i(T)$$

Where ξ is the modal damping ratio and the natural frequency for the ith mode.

These are solved by the Wilson method - θ which is an unconditionally stable scheme step by step step. The passage of time for response is chosen as 0.1 T where T is the period of the highest mode to be included in the response. The qis is substituted into equation 2 to obtain the displacements {x} at each time step.

Analysis through Time pa ra d a a struc aa ra Sujet Harmonic Loads

A harmonic load is one in which can be described using the following equation

$$F(t) = F_0 \sin(t\omega + \phi)$$

In the above equation,

F (t)   = The value of the function of force at any instant of time "t" F0
            = Maximum value of the power function
ω = Frequency of the forcing function
ϕ = The phase angle

A drawing in the above equation is shown in Figure below.

$$F = F_0 \sin(\omega t + \phi)$$
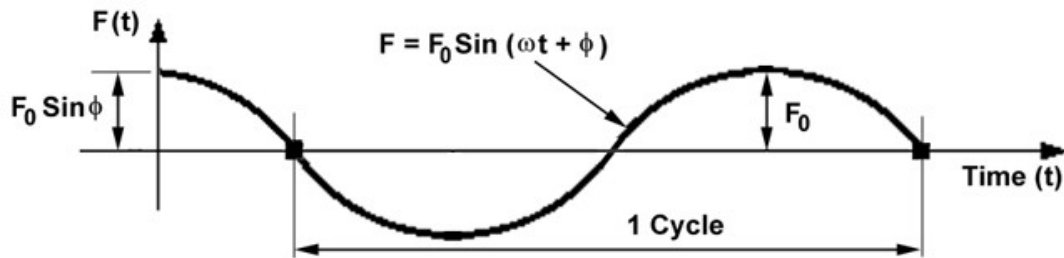
Note: Results are the maximum of the total time, including transients at first, not tied to any results-constant response.

Defining Entry into STAAD tion for Fun Force Prve r

As can be seen from its definition, a function of force is a continuous function. However, Staad, a set of pairs of discrete time-force generated by the force function and an analysis was performed using these discrete pairs of force over time. What it means is that based on the number of cycles that the specific user for loading, Staad generates a table consisting of the magnitude of the force to the various time points. Time values are chosen from time 0 to n * tc in steps of "STEP" where n is the number of cycles and tc is the duration of a cycle. STEP is a value that the user can provide or choose the preset that is built into the program. STAAD to adjust the STEP ¼ cycle is divided into one or more steps. Users can refer to section 5.31.4 of this manual for a list of input parameters that need to be specified for Analysis Over Time in a structure subjected to a harmonic load.

The relationship between variables in STAAD entries and the corresponding terms in the equation shown above is explained below.

F0	WIDTH =
ω	= FREQUENCY
ϕ	PHASE =

1.19   Forces in the ends of the member

See section 5.41

The forces and moments on the end of a member are a result of loads applied to the structure. These forces are in the coordinate system of local member. Figure 1.18 shares samples at the ends of members with their addresses.

And

Global Y is Vertical

Global Y is Vertical

Global Z is Vertical (Z SET UP)

Global Z is Vertical (Z SET UP)
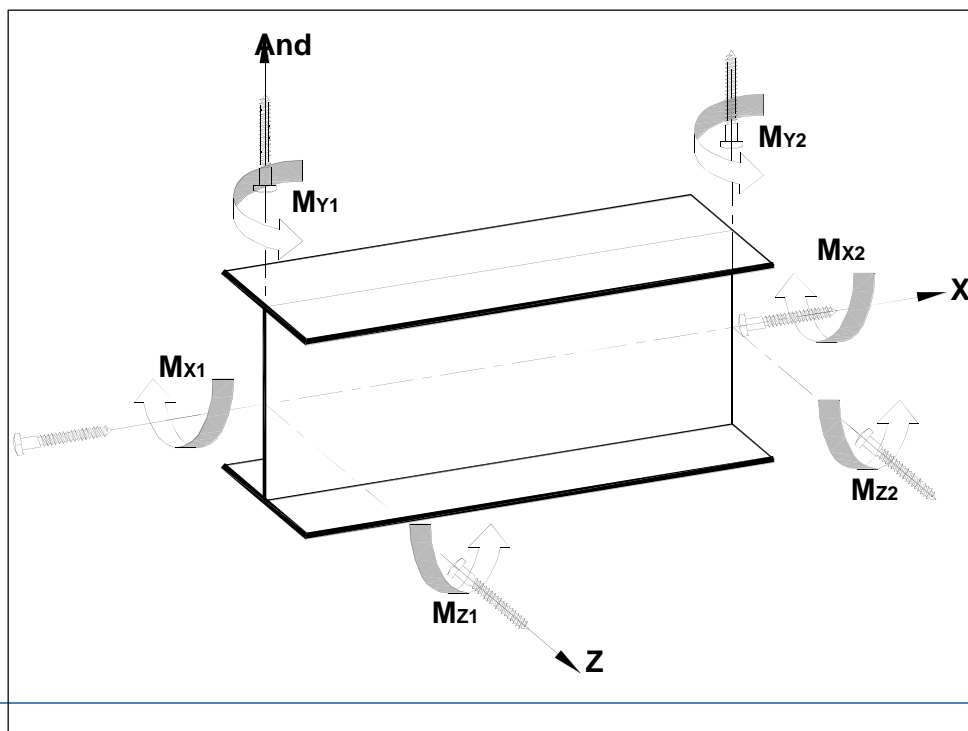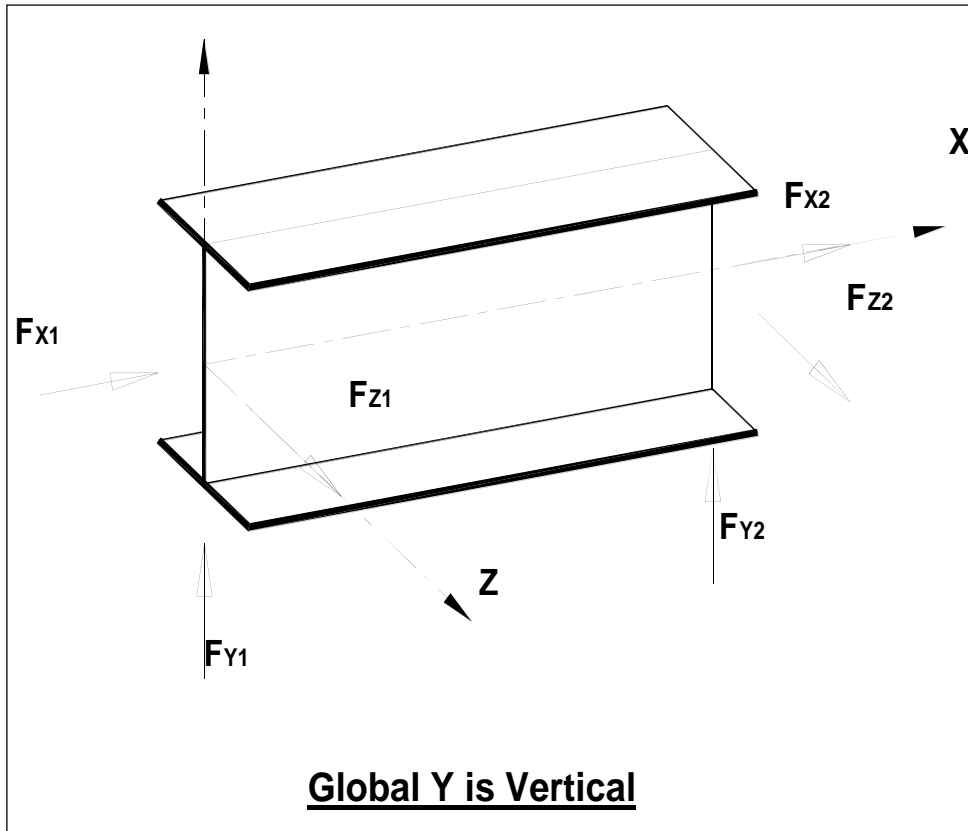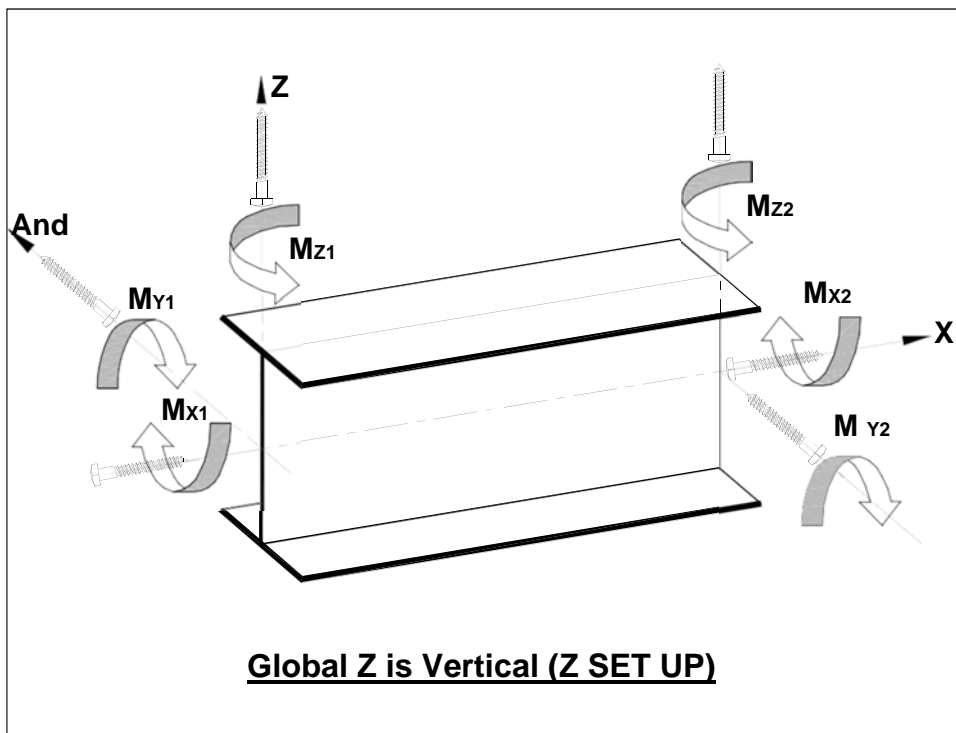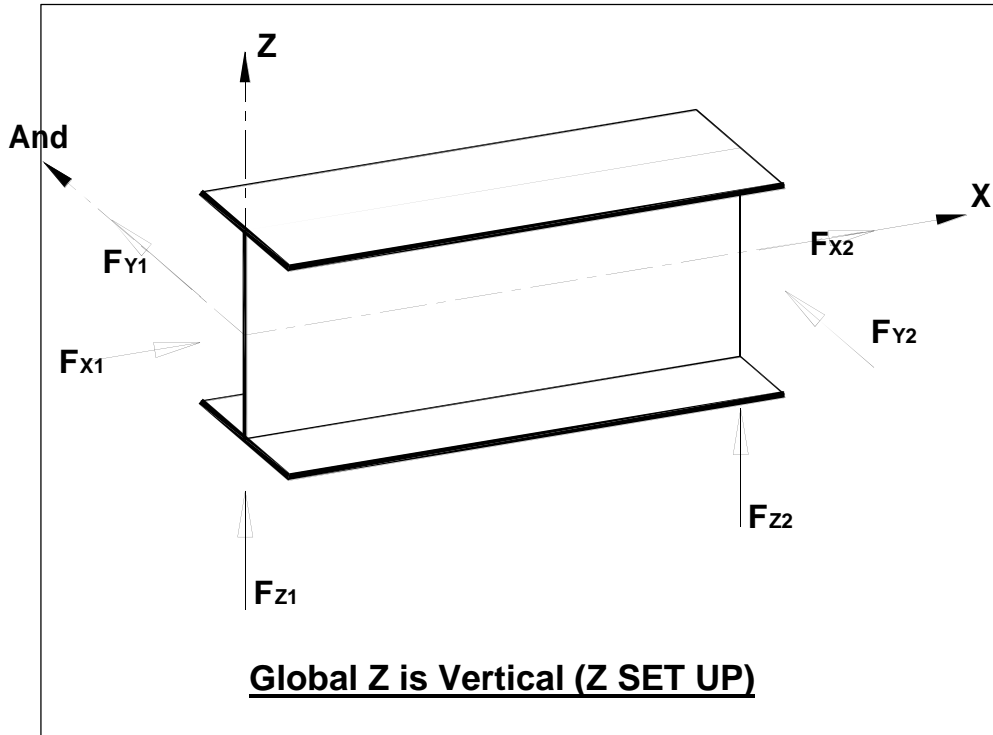
Figure 1.18
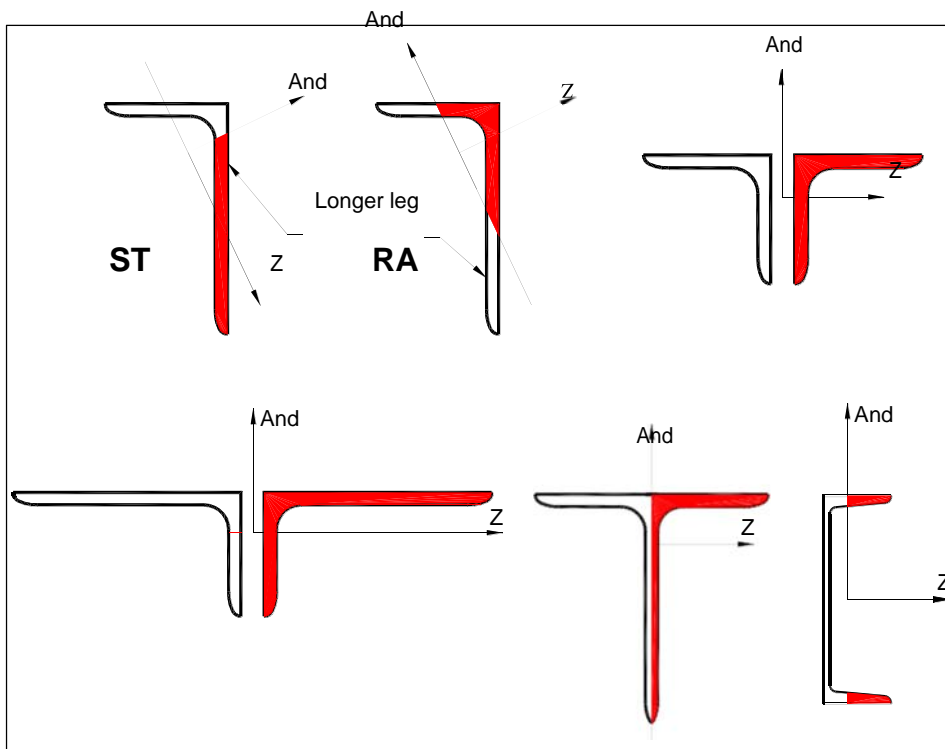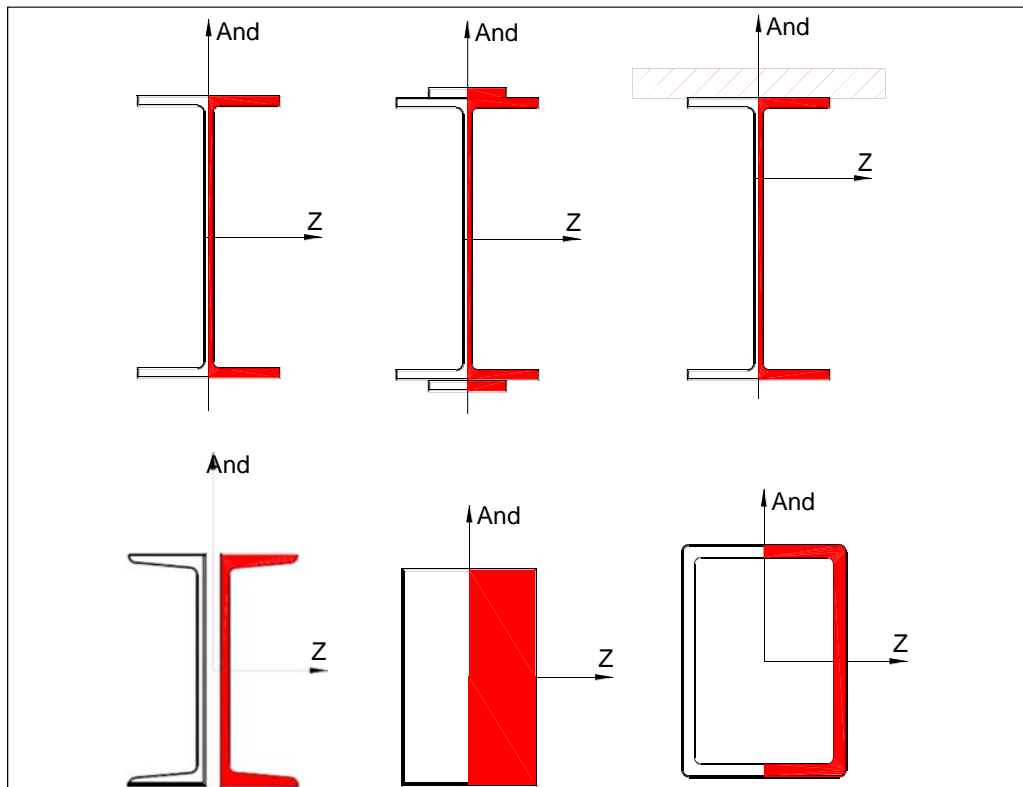
Effort Zones due to bending about the Y axis (MY)

Note: The local X axis goes into the page; And overall is vertically upwards; The shaded area indicates the area under compression; Non-shaded area indicates the area under the stress.
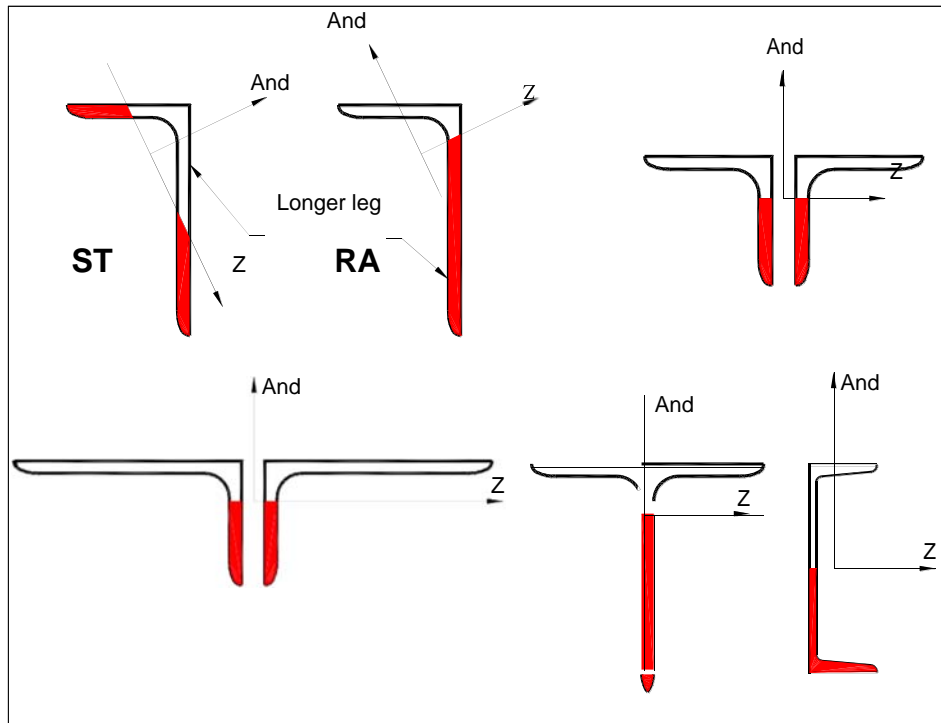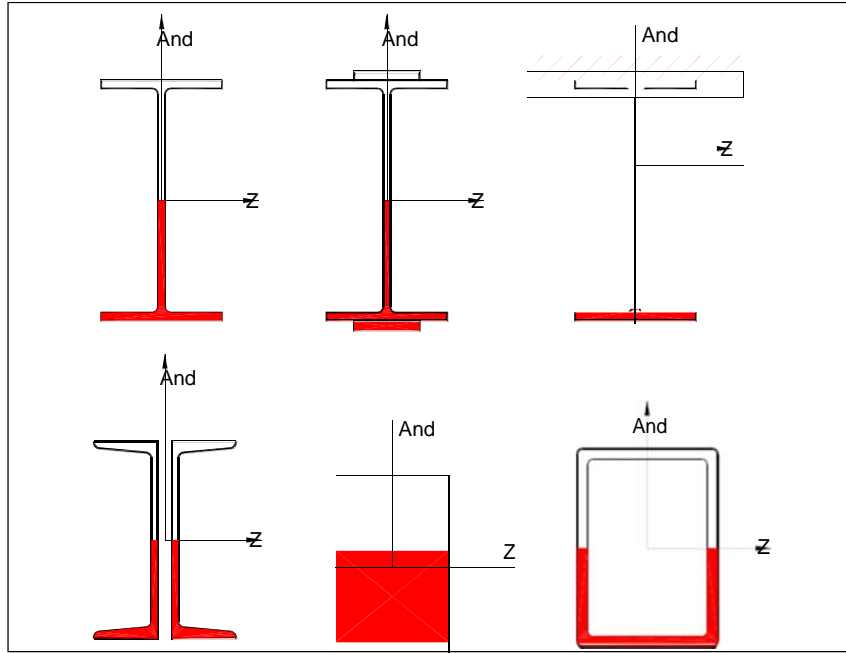
Effort Zones due to bending about the Z axis (MZ)

Note: The local X axis goes into the page; And overall is vertically upwards; The shaded area indicates the area under compression; Non-shaded area indicates the area under the stress

1.19.1   The Secondary Analysis

See Sections 5.38, 5.41, 5.42 and 5.43

The solution of the equations of stiffness resulting displacements and forces in the joints or end points of the member. STAAD is equipped with the following options for secondary analysis results at intermediate points within a Member.

1) Member forces in the middle sections.

2) The movement of the member in the middle sections.

3) Efforts in the member specific sections.

4) Grouping loads (Load Envelope) to

The following sections describe the options for secondary analyzes in detail.

1.19.2   Member forces in the Middle Sections

See Sections 5.39 and 5.41

With the SECTION command, the user can choose any intermediate section of a member where the forces and moments need to be calculated. These forces and moments can also be used in the design of the members. The maximum number of specified sections can not exceed five, including those who are at the beginning and end of a member. If no intermediate section is required, the program will consider the forces at the beginning and end of the member for design. However, only the sections that are proportionate, will be taken into consideration in the design.

1.19.3   Travel by Members in the Intermediate Sections

See Sections 5.39 and 5.41

As the forces, displacements can be printed or plotted onto intermediate sections of the members. This command can not be used for reinforcement or wire-like members.

1.19.4   Efforts on Specific Sections Members

See Sections 5.39 and 5.41

Efforts can be printed in the members specified intermediate sections as well as the start and end joints of the limb. These efforts include:

a) Axial stress is calculated by dividing the axial force by the area of the particular cross-section,

b) And bending stress is calculated by dividing the time by the local address and the section modulus in the same direction,

c) Z bending stress is the same as above except that it is in the local z direction,

d) The shear stresses (in directions y, z), and

e) Combined effort is the sum of efforts axial, bending and bending and Z. All

efforts are calculated by taking the absolute value.

1.19.5 Grouping Loads

The envelopes of forces of a force in the FX (axial force), FY member (Cut in-y) and MZ (the moment about the local axis z, that is the major axis) may be printed for any number of intermediate sections . The force values include the maximum positive and maximum negative value. The following is the sign convention for maximum and minimum values:

FX: A positive value is compression, and negative voltage.
FY: A positive value is the cut in the positive y direction, and negative in the negative direction.

FZ: Same as above except the local z direction.
MZ: A positive time means a time that causes the tension in the upper member. Conversely, when a negative voltage will cause the bottom of the member. The top of a member is defined as one that is in the positive direction of the local axis.

MY: Same as above, except that it is on the local z axis.

1.20 Multiple Analysis

The analysis / structural design may require multiple analyzes in the same run. STAAD allows the user to change the input data as member properties, support conditions etc. in an input file to facilitate multiple analyzes in the same run. They can be combined the different analysis results for design purposes.

For shoring structures, it may be necessary to make certain inactive participants for a particular load case and subsequently activate them to another. STAAD contains INACTIVE option for this type of analysis. The INACTIVE option is discussed in detail in the following paragraph.

<u>Inactive Members</u>

With the IDLE command (INACTIVE), inactive members do. These inactive members are not considered in the analysis of stiffness or in any of the prints. Members made inactive by the INACTIVE command are made active again with the command CHANGE (CHANGE). This can be useful in an analysis where only the desired tension bracing, this way only a certain group members will be inactive for certain load cases. This can be achieved as follows:

a) making the desired inactive members;

b) providing relevant load cases for which the members are inactive;

c) performing the test;

d) using the CHANGE for all active command inactive members;

e) and making another set of inactive members and providing appropriate load cases to which members are to take inactive, and performing analysis by repeating the above procedure as many times as required

1.21 Design of Steel / Concrete

See Sections 2 3 and 4

STAAD has extensive design capabilities for steel sections, concrete and wood (the latter will not be seen). The details about the design of steel and concrete information is presented in Sections 2 and 3 respectively.

1.22 Design of Foundations

The option for the design of foundation is able to design individual foundations for support (s) the user specified. All cases of active load are verified and the design is made for support reactions requiring the maximum size of the foundation. Parameters can control the design. The output includes the dimensions of the foundation and reinforcement details. The anchor rods and development lengths are also calculated and included in the result of design. A description and specification (s) for the design of the command you are available in section 5.48 of this manual.

1.23 Printing Facilities

All input and output data and the results can be printed using the PRINT command (PRINT) available in STAAD. The input data are usually repeated in the output file. This is important from a viewpoint of the documentation. However, if required, they can be omitted.

Extensive options listed are provided in almost all PRINT commands allow the user to select boards or members (elements) for which values are required.

1.24   Plotting Facilities

Please refer to STAAD.Pro Graphical Environment Manual for a complete description of the wide screen and printing graphic media available and how to use them.

1.25   Miscellaneous Facilities

See section 5.17

STAAD offers the following miscellaneous means for troubleshooting:

Perform ROTATION

This command can be used to rotate the shape of the structure through any desired angle for any global axis. The rotated configuration can be used for further analyzes and designs. This command can get after the coordinates of the Board or between two commands Coordinate Board or after all Trouble Member / Element are specified.

SUBSTITUTE   (S   ustit ution)

See section 5.15

You can redefine numbering Staad members together and through the use of SUBSTITUTE command. After that a new set of numbers is assigned, the values of input and output agree with the new numbering scheme. This option allows the user to specify numbering schemes that produce input specifications simple and easy interpretation of the data.

The calcu ass of Cen ter G ravity

Staad is able to calculate the center of gravity of the structure. The CG PRINT command can be used for this purpose.

1.26   Post Facilities   Process

All outputs of the STAAD run can be used in subsequent processes by STAAD.Pro GUI. Please refer to STAAD.Pro Manual (Volume I) for a complete description of the extended screen and printing graphic media available and how to use them.

Section 2

American Steel Design

2.1 Operations   Design

STAAD contains a wide variety of resources for the design of structural members, as individual components of a structure analyzed. The resources which account for the design of members, give the user the ability to perform many different operations design. These options can be selectively used according to the requirements of the design problem. The operations to make a design are:

- Specify members and load cases to be considered in the design.

- Specify whether run either code check or selecting members.

- Specify the values of the design parameters when they are different from the values assigned by default.

These operations can be repeated by the user as often as necessary, depending on the specific requirements of the design.

The steel design can be performed based on the following codes: AISC-ASD, AISC-LRFD and AASHTO. A brief description of each is presented in the following pages.

STAAD currently supports the design of steel wide runner (W), S, M, HP, angle, double angle, channel, dual channel with cover plate beams, joists and verification code prismatic properties.

2.2 Properties Member (Member Properties)

For specification of member properties of standard American steel sections, the library of available steel sections can be used in STAAD. The syntax for specifying the names of built-in steel forms are described in the next section.

2.2.1 Made of Steel Sections Library

The following sections describe the specifications of the AISC steel sections (9th Edition, 1989) Tables of Steel.

A IS C T ab l A ce ro

Most forms of AISC steel are available for entry. Then described are all types of sections are available:

Pa ti ne s an ch or s (Pe rfi l W)

All wide-flange sections listed in AISC / LRFD-89 are available from the way they are written, for example W10X49, W21X50, etc.

```
20 TO 30 AT ST W10X49
ST 33 36 TA W18X86
```

F o rm a s C, M C, S, M, HP

The above forms are available as listed in AISC (9th Edition) without the decimal points. For example, C8X11.5 be entered as C8X11 and S15X42.9 to be entered as S15X42, omitting the decimal points. (Except: MC6X151 as MC6X15.1 and MC6X153 as MC6X15.3.)

```
10 TO 20 BY 2 TA ST C15X40
February 1 TA ST MC8X20
```

Double Channels

Double channels back to back, with or without space between them, are available. The letter D in front of the section name specified by a double channel.

```
21 22 24 TA D MC9X25
55 TO 60 TA D C8X18
```

Angles

Specifications in STAAD angles are different from those in the AISC manual. The following example illustrates the specifications of the angle.
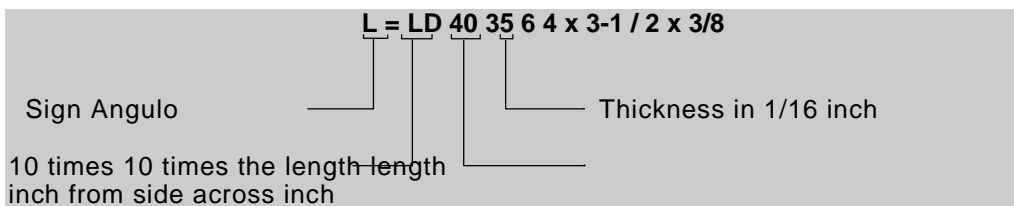
Similarly, L505010 = L 5 "x 5" x 8.5 "and L904016 = L 9" x 4 "x 1"

At present there are two ways to define the local axes and Z axes for a local angle section. To make the transition from AISC Manual data easily to program the standard section for an angle specified:

```
ST TA 51 52 53 L40356
```

This specification has the local z-axis (that is, minor axis) corresponding to the axis specified in Tables ZZ steel. Many engineers are familiar with a convention

used by some other programs in the local y-axis is the minor axis. STAAD adhere to this convention accepting the command:

> **TA RA 54 55 56 L40356**   (RA denotes the reverse angle)

Angles is Dobl

Double angles with short side back to back with the long side or back to back can be specified by entering the word SD or LD, respectively, before the size of the angle. In case of an angle with equal sides or LD or SD will serve the purpose.

> **14 TO 20 TA 0.5 SP LD L35304 Long Side Back to Back L3 -1/2 "x3" x1 / 4 "**
>   with 0.5 "space
>
> **SD 23 27 TA L904012**   Short side back to back L9 "x4" x3 / 4 "

Profile T (Tees)

T profiles are not paid by their real names, as listed in the AISC manual, designating instead of the shape of the beam (W and S) which were cut. For example,

> **1 2 5 8 TA T T W8X24 W8X24 the cut of what WT4X12**

Pipes (Pipes)

Two types of specifications for sections of this type may be used. Generally tubes (pipes) can be defined by its outer and inner diameters. For example,

> **1 TO 9 TA ST PIPE OD 2.0 ID 1875**   shall mean a tube with O.D. 2.0 and I.D. 1,875 units in the system in use.

Sections of pipe listed in the AISC manual can be specified as follows.

>   **5 TO 10 ST TA PIPX20**

>   **PIP X 20 denotes extra strong pipe 2 "diameter**
>
> Symbol Tube ——————⌐ ⌐ └—— 10 x diameter in inches
>   Specify   only   portion   before the decimal point
> Spec. Resistance ————
> (S = Standard, X = Extra-strong, D = Double extra strong)
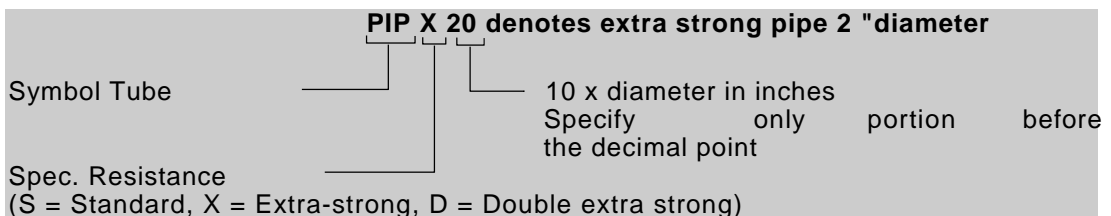
Rectangular Tubular (Tube)

The rectangular hollow section AISC tables can be specified as follows.

5 TO 10 ST TA TUB120808

TUB 120 80 8

Symbol Tub. Rec. ———————— Thickness in 1/16 inch

Alto x 10 (in.) ———————— Width x 10 (in.)
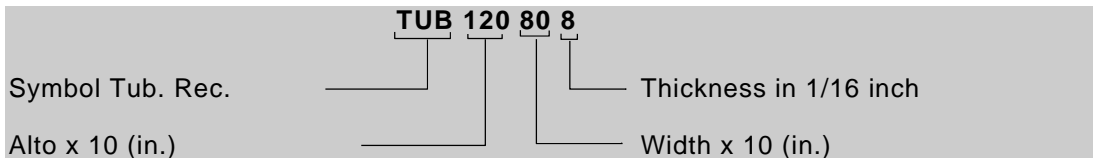
The rect tubular profiles. (Tubes) and tube (pipe), can be defined by its dimensions (height, width and thickness) as follows.

6 TA DT ST TUBE WT 8.0 6.0 0.5 TH
rect is a tubular profile. having a height of 8 ", a width of 6" and a wall thickness of 0.5 ".

The team member can not be performed at specified tubes latest fashion. Only code checks may be made in these sections.

Sun Beams Plates given

Beams welded plates AISC manual can be specified as follows.

B 61 October 20

Symbol section manufactured ———————— Flange thickness in inches x 10
(Only the portion used before point decimal)

Nominal superelevation in inches ———————— Nominal width shoe (inches)
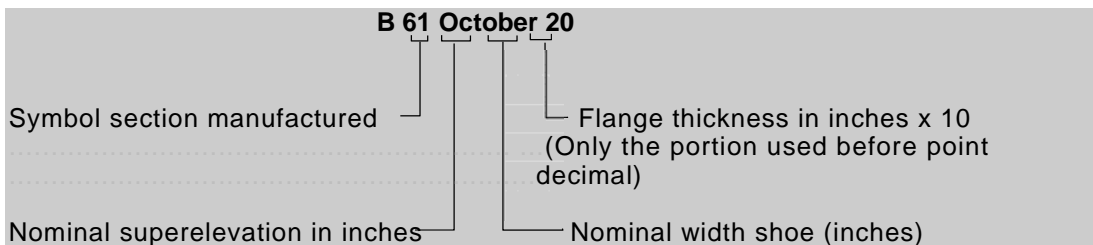
Example:
    1 TO 10 ST TA B612017
    ST RT 15 16 B682210

2.3   Permissible tolerances AISC Code

For steel design, STAAD compares actual efforts as allowable stresses are defined by the Code of the American Institute of Steel Construction (AISC). The ninth edition of the AISC Code, published in 1989, is used as the basis of this design (except for the tensile stresses). Due to the large number and complexity of the AISC Code, it would be impractical to describe every aspect of the design of steel in this manual. Instead, a brief description of some of the largest allowable stress described below.

2.3.1 Tension Effort

Efforts allowable tension in the net section is calculated as 0.60 Fy.

2.3.2 Cutting Effort

Allowable shear stress on the gross section,

$$F_v = 0.4F_{and}$$

For the shear in the soul, the entire section is taken as the product of the total depth and web thickness. For shear (cut) in the skates, the total cross section is taken as 2/3 of the total areas of the skates.

2.3.3 Compressive stress due

The allowable compressive stress in the overall section axially loaded compression members are calculated based on the formula E-1 AISC Code, when most slenderness ratio (Kl / r) is less than Cc. If Kl / r Cc exceeds the allowable compressive stress is decreased by the formula 1E2-2 Code.

$$C_c = \sqrt{2\pi^2 E / F_{and}}$$

2.3.4 Flexural Strength

The allowable bending stress, tension and compression, for a symmetrical member loaded in the plane of its minor axis, as given in section 1.5.1.4 is:

$$F_b = 0.66F_{and}$$

Yes meets the requirements of this section:

a) bf / 2tf is less than or equal to
   65 / Fy

b) bf / tf is less than or equal to
   190 / Fy

c) d / t is less than or equal to 640 (1-3.74 (f_{to}/ F_{and})) / F_{and} when (f_{to}/ F_{and}) <0.16, or 257 / F_{and} if
   (f_{to}/ F_{and})
   > 0.16

d) The laterally unsupported length does not exceed 76.0bf / Fy (except for tubes or pipes rectangular), or 20,000 / (dFy / Af)

e) The ratio of diameter to thickness of pipes shall not exceed 3300
   / Fy

Yes to these symmetric members, bf / 2tf exceeds 65 / Fy, but less than 95 / Fy, Fy = Fb (0.79 to 0.002 (bf / 2tf) Fy)

For other symmetric members who do not comply with the above, Fb is determined as the greatest value based formulas or F1-7 F1-6 AISC and F1-8, but not greater than 0.60Fy. A rigid member not subjected to axial compression due to compression or bending is considered fully effective when the width-to-thickness ratio is not greater than:

76.0 $\sqrt{\text{Fy}}$, for single or double angles with dividers. 95.0 / Fy for double contact angles.

127.0 $\sqrt{\text{Fy}}$, for the soul of beams T.

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by B5 AISC code.

Tension and compression for doubly symmetric sections (I and H) with bf / 2tf less than 65 / Fy $\sqrt{}$ and folded about its minor axis, Fb = Yes 0.75Fy bf / 2tf exceeds 65 $\sqrt{\text{Fy}}$, but less than 95 / Fy $\sqrt{}$, Fb = Fy (1075-0005 (bf / 2tf) $\sqrt{\text{Fy}}$)

For tubes, b and c are subsections of this section folded over the minor axis, Fb = 0.66Fy; if subsections b and c are not met that the width-thickness ratio is less than 238 / Fy, then Fb = 0.6Fy. $\sqrt{}$

2.3.5 Combined Compression and Bending

Members subjected to both axial efforts compression and bending are provided to meet the formula and H1-2 H1-1 AISC when fa / Fa is greater than 0.15 otherwise H1-3 formula is used. It should be noted that during code review or member selection, yes fa / Fa exceeds unity, the program will calculate the second and third part of the formula H1-1, because this would result in a virtually misleading reason. Cm coefficient value is taken as 0.85 and 0.6 for contraventeados members - 0.4 (M1 / M2), but not less than 0.4 to not contraventeados.

2.3.6 Sections   Symmetric alone

For double Tee's angles and having only one axis of symmetry, the ratio KL / r about the YY axis is determined locally using the terms specified on pages 3-53 AISC Manual.

2.3.7 The Twist by Publication T114

The code 89 to the AISC Steel Design currently has no prevention for designing specific sections for the torsion. However, AISC has published a separate document called the "Torsional Analysis of Steel Members" that provides the standard for torsional moments transform normal stresses and shear stresses can be incorporated into the equations of interaction explained in Chapter H AISC code 89. publication guidelines are now incorporated in the 89-AISC steel design modules STAAD.

To consider the stresses due to torsion code review or member selection procedure, TORQUE parameter with a value of 1.0 is specified. See Table

2.1 for more details.

Methodolog
y

If the user were to ask the design for torsion, torsional properties required to calculate the warping normal stresses, the shear buckling and pure cutting efforts are first determined. These depend on the conditions of "limit" prevailing at the ends of the member. These boundary conditions are defined as "Free", "Articulated" or "Flush". Are explained below:

Free: "Free" is the boundary condition that there is an open-end beam flown. It means that no other member connected to the beam at that point.

Articles: "Articulated" represents the condition corresponding to a pinned support defined in the board through the support (SUPPORT) command or relaxation of any of
moments in the joint through a specification of relaxation Member (MEMBER
RELEASE).

Built "Built" represents the condition where there is a support embedded in the board. In the absence of a support on that board, represents a condition where a connection between the rigid frame member given me at least one other member connected to that board. Also, any relaxation of the member must be present at that meeting in the given member.

After the boundary conditions are determined, the normal stresses and shear stresses are calculated. The guidelines in the publication T114 concentrated torsional moments acting at the ends of the specific member to calculate these efforts are used.

The warping normal stresses are added to the axial forces caused by the axial load. These are then substituted into the equations of interaction in Chapter H of AISC 89 code to determine the relationship. Cutting forces in the plane of roll to shear forces due to actual cutting and compared against the allowable shear efforts on the cross section is added.

Restrictions

This facility is currently available for wide forms of skate (the W, M and S), Flat, forms T, Rectangular Pipes and Tubular. Not available for Single, Double Angles Angles, members with PRISMATIC specification property, Composite sections (wide-flange shapes with concrete slab or plate on top), or Double Channels. Also, efforts are computed based on the rules for concentrated torsional moments acting on the ends of the member.

2.3.8 Sections of Alma Design Variable

Appendix F of AISC-89 contains the design specifications members Alma variable section. These specifications have been incorporated in STAAD for verification code in the forms of variable wide-flange web (web-tapered). Please note that the selection of a member can not be performed on members of variable section Alma.

2.4 Design Parameters

The program contains a large number of names of parameters needed for the design and code review. These parameter names with their default values are listed in Table 2.1. These parameters communicate design decisions to software engineer.

The default values of the parameters have been selected such that they are the most frequently used in the conventional design. Depending on the requirements of the analysis of a particular design, some or all of these parameter values have to be modified to get the exact model of the physical structure. For example, the default value of KZ (k value in the local z axis) of an individual is set as 1.0, while in the actual structure may be 1.5. In that case, the value in the program KZ can be changed to 1.5, as shown in the directions of the entry (Section 6). Similarly, the value of a member of TRACK is set to 0.0 which means that any allowable stress of the member are not printed. If the allowable stresses to be printed, the value should be set to 1.0 TRACK.

Note that the names of PROFILE, DMAX and DMIN parameter is only used in the member selection.

2.5 Code Review

The purpose of code review is to verify whether the properties of the sections provided to members are appropriate. The sufficiency is verified according to AISC-89. Code review is performed using the forces and moments in the specified sections of the members. If no section is specified, the program uses the start and end forces member for code review.

When reviewing the code is selected, the program calculates and prints if the members have passed or failed the code; critical code AISC state (as either AISC specifications compressive stress, cutting, etc.); the value of the ratio of the critical condition (overstrain to a value greater than 1.0 or any other value of RATIO (RATIO) specified); governing load case, and the situation (distance from the start of the member) of forces in the member where the critical condition occurs.

The code review can be made of any type steel section in Section List 2.2 of this manual.

2.6 Selecting Members (Member Selection)

Staad is capable of performing operations specified design on members. Once the analysis has been run, the program can select the most economical section, ie the lightest section, which meets the requirements of the code for the specified member. The selected section will be the same type section of the member who originally is being designed. A wide–flange, will be selected to replace a wide–flange, etc. .. Several parameters are available to guide you in the selection. If the parameter PROFILE (PROFILE) is provided, the search for the lightest section is restricted to that profile. Up to three (3) different types of profiles can be set for when the section of a member is

being selected. The member selection can also be restricted by DMAX and DMIN parameters that limit the maximum and minimum depth of the members. Note that if the parameter PROFILE (PROFILE) is provided in specific members, DMAX and DMIN parameters are ignored by the program in the selection of these members.

The member selection can be executed with all types of steel sections listed in Section 2.2 of this manual. Note that for beams with cover plates, the size of the cover plates are kept constant while the beam section is iterated.

The selection of members for those properties that are originally given from a self-generated catalog will be limited thereto.

The member selection can not be made to members whose section properties are given as prismatic.

2.6.1 The selection member for Optimization

The properties of a catalog for a complete steel structure can be optimized with STAAD. This method involves optimization technique, which requires multiple automated analysis. You can start without defining the properties for the members, but provided the kind of profile you have (eg BEAM, COLUMN, CHANNEL, ANGLE, etc.). Member sizes are optimized according to their contribution of stiffness and the number of loads they are receiving. Based on this, a measured size for each member is selected. This method requires a large amount of computer time, therefore, should be used with caution.

2.6.2 Review of Deflection in Steel Design

This facility allows the user to consider the deflection as a criterion in the process REVIEW CODE (CODE CHECK) and SELECT MEMBER (SELECT MEMBER).
The revision of the deflection can be controlled by using three parameters described in Table 2.1.

> **Note: The deflection also uses and other efforts related to the stability criteria. The calculation of the local deflection is based on the latest test results.**

2.7 Members Armour (Truss Members)

As mentioned at the beginning, a reinforcement member is able to sustain only axial forces. So in the design, no time is spent calculating the allowable bending or shear, greatly reducing design time. Therefore, if there is any member armor in an analysis (upwind or as a prop, etc.), it is wise to declare it as a member of armor instead of as a member of the regular frame with both ends hinged.

2.8 Asymmetrical Sections

For asymmetric sections (like the angle, bend angle and T), Staad considered the smallest module to verify bending section. For some cases, this approach may produce slightly conservative results.

Appendix C of the AISC code has also been incorporated to reduce the effort of compression elements without stiffness. Furthermore, AISC specification for the design of simple ANGLES fully included the effects of lateral torsional buckling.

Table 2.1 - Parameters of AISC

Name parameter	Default Value	Description
KY	1.0	K value in the local y axis. Usually this is the minor axis.
KZ	1.0	K value in the local z axis. Usually, this is the major axis.
LY	The length of the Member	Length on the local axis and calculate the slenderness ratio for buckling.
LZ	The length of the Member	Same as above, except that on the z axis (major axis).
FYLD	36 KSI	Yield strength of steel units in use.
FU	60 KSI	The last effort of steel stress in the units in use.
NSF	1.0	Factor for net section tension members.
UNT	Length of Member	Unsupported length of the top flange in compression to calculate the allowable bending stress. Only used if the compression bending is in the top flange.
UNB	Length of Member	Unsupported length of the bottom flange in compression to calculate the allowable bending stress. Only used if the compression bending is in the bottom flange.
CB	1.0	Cb value as used in Section 1.5 of AISC. Cb = 0.0 means that the value will be calculated. Any other value will be used in the design.
SSY	0.0	0.0 = Bracing in the local y axis. 1.0 = Not contraventeado
SSZ	0.0	Same as above except that the local z axis.
CMY CMZ	0.85 for contraventeado and calculated	The value of Cm in local y and z axes

	Non contraventeado	
MAIN	0.0	0.0 = slenderness review 1.0 = suppress slenderness review
STIFF	Length of Member	Spacing of stiffeners for the design of plate girders
TRACK	0.0	Controls the level of detail with which the results are reported. 0 = Detail 1 = Intermediate Level 2 = Maximum detail (See Figura2.1)
DMAX	45 inches	Maximum allowable superelevation.
DMIN	0.0 inches	Minimum allowable superelevation.
RATIO	1.0	The relationship between forces acting against allowable stresses.
WELD	1 for closed sections 2 for open sections	Welding type, as explained in Section 2.11. A value of 1 means that only one side welded except for skates and wide sections where T is always assumed that the soul is welded on both sides. A value of 2 means that both sides are welded. For closed sections such as rectangular pipes and tubes, welding will be on one side.
BEAM	1.0	0.0 = only design end moments or those locations specified by the SECTION command. 1.0 = extreme and designed for each twelfth point along the length of the member. (The default)
WMIN	1/16 inch	Minimum weld thickness.
WSTR	0.4 x Fyld	Allowable Effort welding.
DFF	None (Required for reviewing deflection)	"Deflection Length" / Maximum allowable deflection Local
DJ1	Start Nudo member	Board No. denotes the starting point for the calculation of "Deflection Length" (See Note 1)
DJ2	Knot the end of the member	Board No. denotes the end point for the calculation of "Deflection Length" (See Note 1)
TORSION	0.0	0.0 = No review performed torsion. 1.0 = Make twisting reviewing rules based on AISC T114.
TAPER	1.0	0.0 = The I-beam design variable section just based on the rules of Chapter F and Appendix B. Do not use the rules of Appendix F. 1.0 = The I-beam design variable based on the rules section of Appendix F of AISC-89.

* The top and bottom represent the positive and negative side of the local axis (local Z if Z SET UP (up) is used).

NOTES:

1) The "deflection length" is defined as the length used for the calculation of local deflections within a Member. It may be noted that for most cases the "Length Deflection" may be equal to the length of the member. However, in some situations, the "Length Deflection" may be different. For example, it refers to the figure below where a beam that has used four and three members together has been modeled. Note that the "length of the deflection" for the three members be equal to the total length of the beam in this case. The parameters to be used are DJ1 and DJ2 to model this situation. Also the straight line joining DJ2 DJ1 and is used as the reference line at the local deflections are measured. Thus, for all three members here, DJ1 should be "1" and DJ2 should be "4".
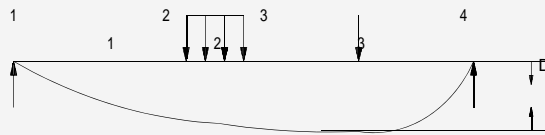
D = Local Maximum deflection for members January 2 and 3.

EXAMPLE: PARAMETERS
DFF 300 ALL
DJ1 1 ALL
DJ2 4 ALL

2) If not used DJ2 DJ1 and the "Deflection Length" will have as the default length of the limb and the local deflection of the line of the original member is measured.

3) It is important to note that unless a specified value of DFF, STAAD not conduct a review of the deflection. This agrees with the fact that no default for DFF (see Table 2.1).

4) A critical difference between the UNT / UNB parameters and parameters LY and LZ. The parameters UNT / UNB represent the length skate without lateral support compression. Defined in Chapter F, page 5-47 of the specifications in the 1989 AISC ASD manual as the distance between cross sections secured against buckling or lateral displacement of the compression flange. Parameters UNT / UNB are used to calculate the allowable compressive stress (FCZ and FCY) for behavior as a beam. LY and LZ are, moreover lengths without contraventear for behavior as a column and used to calculate the proportions of KL / r and the allowable axial compressive stress FA.

SSY and CMY are two parameters that are based on 2 values defined on page 5-55, Chapter H of AISC 9th edition manual. SSY is a variable that allows the user to define either if there is contraventeado or member in the local address Y. CMY is a variable used to define the term called Cm in AISC manual. When SSY is set to 0 (which is the default) means that the bracing member is subject to local Y. When SSY address is set to 1.0, means that the member is not subject to local bracing direction . SSY The only effect that has is causing the program to calculate the appropriate value of CMY. If SSY is set to 0 and not provided CMY, CMY STAAD calculated as 0.85. If SSY is set to 1 and not provided CMY, CMY STAAD calculated from the equation on page 5-55. However, if the user provides CMY, the program will use this value rather CMY calculated without taking into account at all, what you define for SSY.

Terms used to calculate slenderness ratios KL / r for local Y and Z
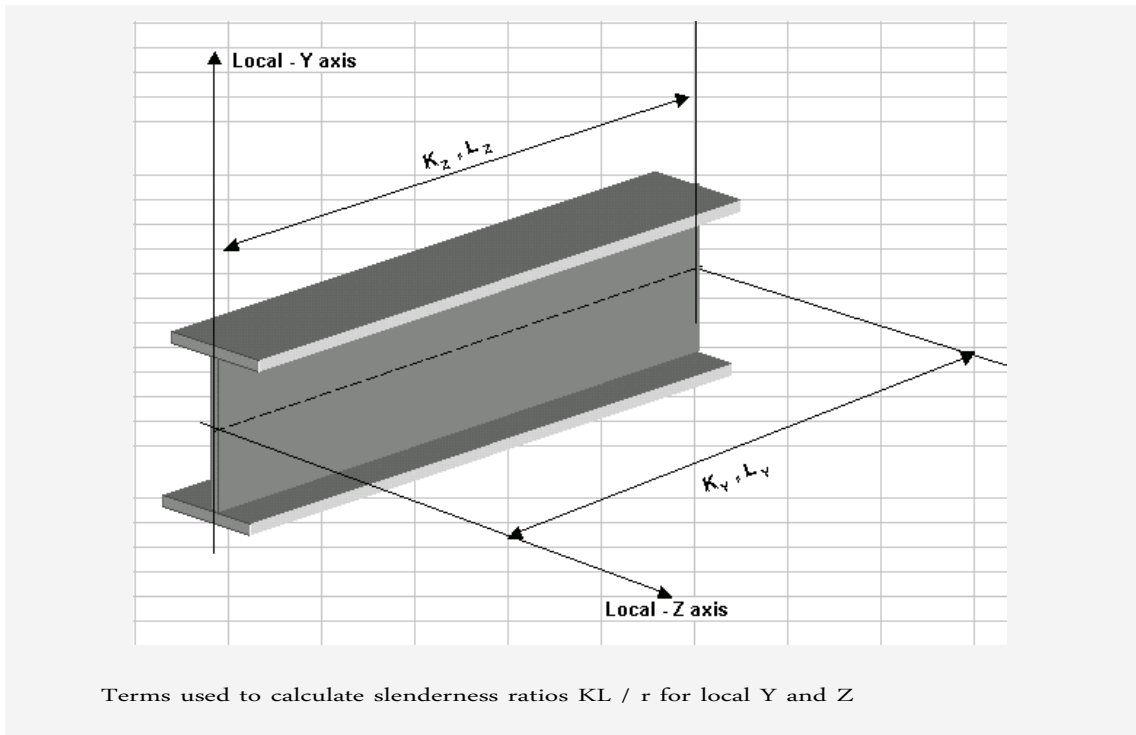
2.9 Composite Beam Design AISC-ASD

Description

STAAD now supports the design of a flexural member composed by the AISC-ASD code. Full composite action and are assumed uniformly distributed load. Shear connectors will be continued in the area of negative moment. Steel deck formed is assumed to be perpendicular to the member. If it is parallel to the member, the number of shear connectors shall conform.

The following additional parameters have been introduced to support the design of the composite member.

Table 2.2 Parameters for AISC-ASD joists

Name Parameter	Value Defaults	Description
CMP	0	The composite action with the connectors 0 = no composite action 1 = composite action
DIA	0.625 inch	The diameter of shear connectors
HGT	2.5 inch	The height of shear connectors after

		welding
DR1	0.4	The proportion of time due to dead load applied before the concrete hardens the total time
WID	0.25 times the length of the member	Effective width of the concrete slab
FPC	3.0 ksi	Compressive strength of concrete at 28 days
PLT	0.0	Thickness of the cover plate welded to the bottom flange of the composite beam
PLW	0.0	Width of cover plate welded to the bottom flange of the composite beam
RBH	0.0	The height of the rib of the steel deck
RBW	2.5 inch	The width of the rib of the steel deck
SHR	0	The underpinning temporal during construction 0 = no shore 1 = underpinned
THK	4.0 inch	The thickness of the concrete slab or the thickness of the concrete slab on steel deck steel.

```
UNIT
PARAMETER
CODE AISC
INCH BEAM 1
ALL
TRACK 2 ALL
DR1 ALL 0.3135
WID 69 525 ALL
FPC 3.0 ALL
THK 4.0 CMP
ALL 1 ALL
CHECK CODE ALL
SELECT ALL
```

2.10 Armed Trabes

Plate girders can be designed according to Chapter G of the AISC specifications. Widespread ISECTION option specification in the catalog can be used to specify sections of plate girders. Forms for welded plate girders AISC (pages 2-230 and 2-231 AISC 9th Edition) are contained in the library of steel sections of the program. Both the review of the code as member selection can be implemented in a catalog of AISC standard sections for armed or through widespread ISECTION option girders. The STIFF parameter (see Table 2.1) can be used to specify the spacing of the reinforcements, such that appropriate shear

are calculated allowable. The program automatically calculates efforts skates according to the requirements of the G2 section.

2.11 Tabulated Results of Steel Design

To review the code or member selection, the program produces the results of a free mode. Items in the output table are explained as follows:

a) MEMBER (MEMBER) refers to the number of member for which the design has been executed.

b) TABLE (TABLE) refers to the name of the AISC steel section has been compared with the steel code, or has been selected.

c) Impressions RESULT (RESULT) if the member has past (PASS) or failed (FAIL). If the result is FAIL, there will be an asterisk (*) in front of the member number.

d) CRITICAL CONDITION (CRITICAL COND) refers to the section of the AISC code, which governs the design.

e) RELATIONSHIP (RATIO) prints the ratio of actual to allowable stresses for the critical condition efforts. Usually a value of 1.0 or less means that the member has passed.

f) LOADING (LOADING) defines the number of load case governed the design.

g) FX, MY and MZ defined axial force, when the local time in the y-axis and z local axes respectively. Although STAAD considers all the forces and moments on the member for the design, only FX MY and MZ are printed, because these are the most important in most cases.

h) The LOCATION (LOCATION) specifies the actual distance from the beginning of the member to the section where the design forces govern.

i) If the TRACK parameter is set to 1.0, the program block of the table and print efforts allowable bending compression (FCY and FCZ) and tension (FTY and FTZ), the allowable axial stress in compression (FA), and Allowable shear stress (FV), all in kips per square inch. Besides print, limb length, area, section modulus, which govern the rate of KL / r CB.

j) At the exit to TRACK 2.0, Fey and Fez items is as follows:

$$Fey = \frac{12\pi^2 E}{23\left(K_{And}L_{And}/r_{And}\right)^2}$$

$$Fez = \frac{12\pi^2 E}{23\left(K_Z L_Z r_Z/\right)^2}$$

```
STAAD-[1] MEMBER SELECTION - (AISC)
*****************************
```

```
                                         Y        PROPERTIES
                                         |        IN INCH UNIT

MEMBER   2                                |        AX =   10.00
                                          |    —Z  AY =    3.98
                ST  W14 X34               |        AZ =    4.09
DESIGN CODE                               |        SY =    6.91
AISC-1989                                 |        SZ =   48.64
                                                   RY =    1.53
           |«———LENGTH (FT)=   19.00 ——»|          RZ =    5.83


           77.4 (KIP-FEET)
PARAMETER      ┬           ■1  ■1                  STRESSES
IN KIP  INCH   |                   ■1              IN KIP  INCH

KL/R-Y= 149.37 |         ■3                        FA  =    6.69
KL/R-Z=  39.10 |                            ■3     fa  =    0.35
UNL   = 120.00 |      ■3            ■1              FCZ =   21.60
CB    =   1.00 |                                   FTZ =   21.60
CMY   =   0.85 |   ■3                               FCY =   27.00
CM2   =   0.85 ├■1                                  FTY =   27.00
FYLD  =  36.00 |                                    fbz =   19.10
NSF   =   1.00 |                        ■1  ■3      fby =    0.00
DFF   = 250.00 20.3 ┼─┼─┼─┼─┼─┼─┼─┼─┼─┼─┼─          FU  =   14.40
dff   = 522.27         ABSOLUTE M2 ENVELOPE         Fey =    6.69
                          (WITH LOAD NO.)           Fez =   97.67


              MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)
              ─────────────────────────

              AXIAL     SHEAR-Y    SHEAR-Z   MOMENT-Y    MOMENT-Z

    VALUE      4.4       24.3       0.0        0.0        77.4
  LOCATION     7.9        9.5       0.0        0.0         9.5
  LOADING      3          1         0          0           1
```

```
                  DESIGN SUMMARY  (KIP-FEET)
                  ─────────────────

      RESULT/    CRITICAL COND/    RATIO/      LOADING/
      FX              MY            M2         LOCATION
      ──────────────────────────────────────────────

      PASS       AISC- H1-3        0.937           1
      3.50 C        0.00         -77.43         9.50
```

Figure 2.1

2.12 Welding Design

STAAD is capable of selecting the thickness of solder connections and tabulate the various efforts. The design of welding is limited to only those members who have properties catalog as rectangular sections wide-flange, flange beam, single angle, single channel, tubular and tubular loop. The WELD, and WSTR WMIN parameters (as described in Table 2.1) governing the design of welds.

Since the thickness of a weld is very small compared to its length, the
weld properties can be calculated as if they were linear members. Therefore, the cross-sectional area (AZ) of the solder actually be the length of the weld. Similarly, the units for the modules section (SY and SZ) will square length and polar moments of inertia (JW) is cubic length. The following table shows the different linear welds available; their types and their coordinate axes.

WELD TYPE	ANGLE	WIDEFLANGE	TEE	CHANNEL	PIPE	TUBE
	and	and	and	and	and	and
1						
2	and	and	and	and	—	—

Actual efforts, calculated from the forces in the members, can be defined by three names, according to their directions.

- Horizontal stress - caused by the cutting force in the local z direction and the torsional moment.

- Vertical load - caused by the shear force in the axial direction, and the torsional moment.

- Direct stress - the axial force produced by bending moments and local y and z in directions.

Combining efforts as the square root of the sum of the squares of the above three principal stresses is calculated.

Then the equations are: <u>Forces</u>

<u>MX</u> = Torsional Moment
<u>MY</u> = Deflection local axis
and <u>MZ</u> = Bending local z
axis <u>FX</u> = Axial Force
<u>VY</u> = Shear in the local y axis <u>VZ</u>
= Shear in the local z axis

<u>Welding Properties</u>

<u>AX</u> = The area of the weld as a linear member <u>SY</u>
    = Modulus of Section around the local y axis
<u>SZ</u> = Modulus of Section around the local z
axis <u>JW</u> = Polar moment of inertia
<u>CH</u> = Distance from the extreme fiber to the horizontal forces (local z)
<u>CV</u> = Distance from the extreme fiber to the horizontal forces (local
and)

Efforts equations:

$$\text{Effort Horizontal, Fh} \quad = \frac{VZ}{AX} + \frac{MX \cdot CH}{JW}$$

$$\text{Vertical Effort, Fv} \quad = \frac{VY}{AX} + \frac{CV \cdot MX}{JW}$$

$$\text{Direct Effort, Fd} \quad = \frac{FX}{AX} + \frac{MZ}{SZ} + \frac{MY^{*}}{SY}$$

* Note: MY and MZ are taken now in its absolute value, which could lead to conservative results for asymmetric as the angle sections, T and channel.

$$\text{Combined Forces } F_{comb} \quad = \sqrt{F_h^2 + F_v^2 + F_d^2}$$

$$\text{Weld Thickness} \quad = \frac{F_{comb}}{F_w}$$

Where:

Fw = Allowable Effort in welding, the default value is 0.4 Fyld (Table 2.1).

The thickness t is rounded to the nearest value 1/16 and all efforts are recalculated. The printout shows the latest efforts calculated. If the TRACK parameter is defined as 1.0, the printout will include the properties of the weld. Note that the program does not calculate the minimum thickness of welding as required by some codes, but only compared to the minimum thickness you defined (or every 1/16 inch if not provided).

When the ARMOR (TRUSS) command with the command to select Welding (WELD SELECT) is used, the program designs welds required for the angle of armor and double angles are attached to the splice plates. The program reports the number of welds (two for simple angles and four for double angles), and the required length for each weld. The thickness of the weld is taken as 1/4 inch (6 mm) to 1/4 inch members (6 mm) thick, 1/16 inch and less (1.5 mm) thickness for the largest angle members 1/4 inch (6 mm) thick. The minimum weld length is taken as four times the thickness of the weld.
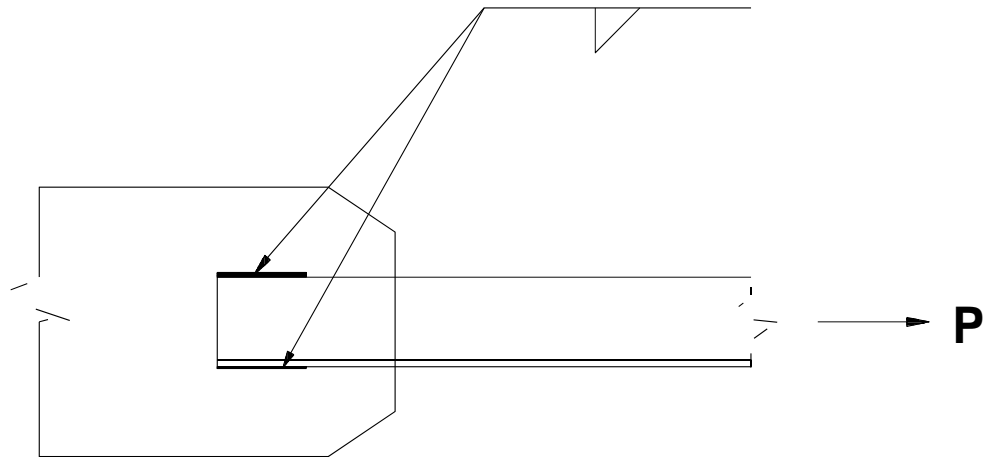
Figure 2.2 - Design WELD Welding SELECT TRUSS.

2.13   The steel design by AASHTO Specifications

2.13.1   General Comments

This section presents some general guidelines regarding the adjustments to the American Association of State Highway and Transportation AASHTO (American Association of State Highway and Transportation Officials) for STAAD steel design. The design philosophy and logistical process for the selection of members and code review is based on the principles of design allowable stress. Two major shortcomings are recognized: deficiency due to over-exertion and deficiency due to stability considerations. The following sections describe salient features of the calculated allowable stress and stability criteria used. Members are defined in such a way as to withstand design loads without exceeding the allowable stresses and is the most economical section is selected based on the criterion of minimum weight. Part of the program for the revision of the code, it also reviews the slenderness requirements, minimum metal thickness and width-thickness. Usually it is assumed that the designer will take due care in the detailed requirements, such as definition of stiffness and reviewing local buckling effects skates, tearing of the beam web, etc. ..

2.13.2   Eligible efforts according to AASHTO Code

As mentioned above, the design and code review members Staad is based on the design method of allowable stress. Which is the definition of structural members using the design loads and forces, allowable stresses and design limits for the appropriate material under service conditions. Due to practical reasons, is beyond the scope of this manual describe each of the aspects of the AASHTO specifications for structural steel design. This section will discuss only the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A AASHTO code specifies the allowable stresses.

Axial Effort

The allowable tension stress, as calculated in AASHTO, is based on the net section. This tends to produce a slightly conservative result. The allowable stress in the net section stress is given by,

$$F_t = 0.55F_{and}$$

The allowable stress in the overall compression section subject to compression members axially loaded is calculated based on the following formula:

$$F_{and}\!\!\neq\!\!F_{to} \frac{F_{and}}{F.\,S.} \frac{(1- (Kl / r)^2)}{4\pi^2 E} \qquad \text{when } (Kl / r) < C_c$$

$$F_{to} = \frac{\pi^2 E}{F.\,S.\ (Kl / r)^2} \qquad \text{when } (Kl / r) > C_c$$

$$\text{with } C_c = (2\pi^2 E / F_a)^{\text{Hal}} f \quad \text{and} \quad \frac{\pi}{F.\,S.} = 2.12$$

It may be mentioned here, that lacks AASHTO no option to increase the allowable stress for a secondary member as 1 / r exceeds a certain value.

Effort ection d e Fl

The allowable stress for compression deflection profiles for sections of beams whose armed skates compressions are supported laterally by means of its total length drowned in concrete is given by

$$F_b = 0.55F_{and}$$

For similar length members supported or not supported partially skates, the permissible bending stress of compression is given by

$$F_b = 0.55F_{and}(1- \frac{(1/r)^2 F_{and}}{4\pi^2 E}) \qquad \text{when } (kl / r) < C_c$$

$$\text{with } (r^2) = b^2/ 12$$

The AASHTO code does not have a specification for the maximum stress allowable stress for members subject to bending. The AISC specification is relevant for compact symmetric 0.66Fy assembled sections or hot forged, and loaded in the plane of the minor axis and meeting other

requirements of Section 1.5.1.4.1 for AISC specifications. A common practice among designers of bridges, 0.55Fy is used as the tensile stress by permissible for members who meet other requirements of Section 1.5.1.4.1 of the AISC specifications flexion. This practice is slightly conservative and used to adapt the code in STAAD AASHTO steel.

Effort d e Court

The effort allowable cut in the total cross section is given by

$$F_v = 0.33F_{and}$$

For shear in the soul, the total cross section is defined as the product of the total depth of the beam and the web thickness. The AASHTO code does not specify anything about the allowable stress for the shear on skates. The program assumes the same effort allowable cut (0.33Fy) to shear on souls and skates. For shear skid, the total cross section is taken as 2/3 times the area of the total skate.

Interaction between Esf uerzos Axial and Bending

Members subject to both axial forces and bending are provided pursuant to section 10.36 of steel AASHTO code. All members subjected to bending and axial compression shall satisfy the following formulas:

$$\frac{f_{to}}{F_{to}} + \frac{C_{mx}f_{bx}}{(1-f_{to}/F_{ex})\,F_{bx}} + \frac{C_{my}f_{by}}{(1-f_{to}/F_{ey})\,F_{by}} < 1.0$$

At intermediate points, and

$$\frac{f_{to}}{0.472F_{and}} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

In support points.

AASHTO steel code provides no specification for the combined interaction of axial tension and bending. The AISC review procedure for the interaction of axial-bending stress is used in the adaptation of AASHTO in STAAD. All members subject to combined axial tension and bending are required to satisfy the conditions
1.6.2 AISC.

2.13.3 Stability Requirements for the AASHTO Code

According to the AASHTO code for compression members, the slenderness ratio, KL / r, will be limited to 120 to 140 for main members and secondary members. The AASHTO code defines child members as those whose primary purpose is to contraventear structure against longitudinal and lateral force, or to reduce the length that no other bracing members, primary or secondary. For tension members, KL / r is 200 for the main and side members to 240 members.

2.13.4 Requirement Minimum thickness of Metal

The AASHTO code has a minimum thickness requirement for the whole structure
steel. According to this requirement, all steel structures, except for certain souls of rolled shapes, ribs closed in orthotropic decks, fillers and handrails, shall not be less than 0.3125 inches. The thickness of the web of the beam, glulam

grooved or should not be less than 0.23 inches. These requirements have been incorporated into the adaptation of AASHTO in STAAD.

2.14   Design of Steel   by AISC Specification / LRFD

2.14.1   General Comments

The design philosophy involved in the LRFD (Load and Resistance Factor Design) specification is grounded in the concept of limit state design, the innovative power in structural engineering. The structures are designed and proportionate taking into account the yield point of the total load which would become unfit for use on intended. Two main categories of limit state are recognized; ultimate load and serviceability. The primary considerations in the design yield strength of the ultimate load are the strength and stability, while for the serviceability deflection is used. Appropriate load and resistance factors are used to achieve a uniform operational safety for all steel structures under various load conditions while boundary changes that may be exceeded are considerably remote.

In adapting to LRFD STAAD, members are provided to withstand design loads without exceeding the limit states of resistance, stability and serviceability. Also, the cheapest section based on the criterion of lowest weight is increased by the designer in specifying allowable depth of the member, type of desired section, or similar parameter set. Part of the code review, review the code requirements for each section are identified and selected under the ruling consistent criterion.

The following sections describe the salient features of the adaptation of the LRFD specifications in the design of steel STAAD. A detailed description of the design process with their implicit concepts and assumptions are contained in the LRFD manual. However, since the design philosophy is drastically different from the conventional ASD (Allowable Stress Design), presented here, a brief description of the basic concepts to start it in the design process.

2.14.2   Fundamentals of LRFD

The primary objective of the LRFD specification is to provide a uniform operational safety for all steel structures under various loading conditions. This uniformity can not be obtained in the form of allowable stress design (ASD).

ASD method can be represented by the inequality

$$\sum Q_i < R_n / F. S.$$

The term on the left side of the inequality is the required strength, which is the sum of the effects of load Q_i (forces and moments). The term on the right side, the

design strength is the nominal resistance and resistance Rn divided by a safety factor. When divided by the appropriate property of the section (area or section modulus), the two sides of the inequality become the actual effort and effort

permissible respectively. ASD, then, is characterized by the use of fillers in "work" no factored in conjunction with a single safety factor applied to the resistor. Due to the high variability and hence the unpredictability of the live load and other loads compared to the dead load, security uniform operating is not possible.

LRFD, as its name implies, uses separate files for each load and resistance factors. Since different factors reflect the degree of uncertainty of different loads and load combinations and precision for predicting resistance, smoother operation safety is not possible. The LRFD method can be synthesized by the inequality

$$and_i Q_i R_n \varnothing$$

On the left side of the inequality, the required strength is the sum of the effects of loads, Qi, multiplied by their respective load factors, he yi. The design strength, the term of the right hand, is the nominal resistance or Rn, multiplied by a resistance factor, Ø.

In adapting LRFD in STAAD, it is assumed that the user will use the appropriate load factors and load combinations will create necessary for analysis. The share of program design takes into account the effects of load (forces and moments) obtained from the analysis. Calculations for resistance elements (beams, columns, etc.), resistance (the nominal resistance) and strength factor will be considered applicable automatically.

2.14.3 Requirements Analysis

Construction types recognized by AISC specifications have not changed, except that both the "semi-rigid" "simple frame" (formerly Type 2) and frame (type 3 above) have been combined into the same category, Type PR (partially fixed). "The rigid frame" (formerly Type 1) is now the (fully fixed) FR type. The type of construction FR is allowed unconditionally. The type of construction PR, it may require some inelastic deformation of steel structural member. Thus, when you specify type PR construction, the designer should consider the effects of partial restriction on the stability of the structure, lateral deflections and bending moments of second order. As proposed in Section C1 LRFD Specification, it requires an analysis of the effects of second order. Thus, when the LRFD code is used for the design of steel, the user must use option P-Delta deSTAAD analysis.

2.14.4 Classification of Sections

The specification allows LRFD inelastic deformations considered in sections of the elements. Thus, the local buckling becomes an important criterion. Steel sections are classified according to their characteristics of local buckling, as compact, noncompact or slender element sections. This classification is based on the geometrical properties of the section. Design procedures will differ depending on the

class section. Staad is able to determine the type of section to the standard shapes and forms that have been user-defined and designed according to these.

2.14.5 Axial Tension

The criteria governing the tensile strength of members is based on two boundary conditions. The limit yield condition in the overall section is intended to prevent the over-extension of the limb. The second boundary condition involves the broken section in the minimum effective net area. The net section area can be specified by using the NSF parameter (see Table 2.2). STAAD calculates the voltage rating of a given member, based on these two types of boundary conditions and proceeds with the selection of the member or code review in due form.

2.14.6 Axial Compression

Resistance equations flambeo LRFD were reviewed to take into account the inelastic deformation and other recent research related to the behavior of columns. Two equations governing flambeo resistance are available, one for inelastic buckling and another for the elastic or Euler buckling. Both equations include the effects of residual stresses and initial deflection. The compressive strength for a particular member to Staad is calculated according to E outlined in Chapter LRFD Specification procedure. For slender elements, the method described in Appendix B5.3 used.

Individual members with symmetric or asymmetric compression are designed based on the boundary conditions of torsional bending and torsional. Appendix procedure E3 is adapted for determining the design strength to these boundary conditions.

The effective length for calculating the compressive strength can be defined through the KY, KZ and / or LY, LZ parameters. If none of them is defined, the total length of the member shall be taken into consideration.

To the criterion of resistance to compression, the compression members are required to satisfy the constraints of slenderness, which are a function of the nature of use of the member, (the main component resisting load bracing members, etc.) . In the process of member selection and verification code, STAAD immediately reviews the slenderness appropriate members before continuing with other procedures to determine the adequacy of a given member.

2.14.7 Flexural Design

In LRFD, the design of the flexural strength of a member is determined by the boundary condition at the lateral torsional buckling. Inelastic flexure is permitted and basic measure of bendability is the plastic moment capacity of the section. The flexural strength is a function of the capacity of the plastic moment, laterally braced not effective length, non-limiting length braced laterally bending moment and flexural modulus. The length Lr does not laterally braced and buckling moment Mr are functions of the geometry of the section and are calculated according to the procedure

Chapter F. The purpose of bending coefficient Cb is to consider the influence of the gradient of the moment about the torsional buckling. This coefficient can be specified by the user through the use of CB parameter (see Table 2.2) or calculated by the program (if CB is specified as 0.0). In the absence of CB parameter, a default value of 1.0 is used. The procedure for calculating design resistance to bending, also considers the presence of residual stresses in the bearing. To specify the not laterally braced length, the UNT / UNB parameters can be used (see Table 2.2).

2.14.8 Combined Axial Force and Bending

The interaction of axial forces and bending in simple and double symmetrical shapes, it is governed by the formulas and H1-1b H1-1a. These formulas interaction cover the general case of biaxial flexural strength combined with axial. Which are also valid for uniaxial bending and axial force.

2.14.9 Design by Court

The method of Section F2 of the LRFD specification used in STAAD to design shear forces in the members. The shear strength, as calculated in LRFD, is governed by the following boundary conditions: The F2-1a equation for the provenance of the beam web; F2-2a equation for the inelastic buckling; F2-3a equation for elastic buckling. The shear or shear skates for wide channels and sections is resisted by the web area of the beam, which is taken as the time of the total depth to the thickness of the beam web.

2.14.10 Design Parameters

See Table 2.2 and Section 5.44.1

For LRFD design specifications required by using the parameter CODE (CODE) (see Section 5.44). Other applicable parameters are summarized in Table 2.2. These parameters communicate the decisions of the engineer to the program, allowing you to control the design process to meet a particular application.

The default values of the parameters have been selected such that are most frequently used in the conventional design. Depending on the particular design requirements, some or all values of the parameters can be changed to accurately model the physical structure.

The DMAX and DMIN parameters apply to the selection of the member only.

2.14.11 Code Review and Team Member

The two options for code verification and selection of members are contained in adapting to STAAD LRFD. For general information on these options, refer to Sections 2.5 and 2.6. For information on specifying these commands, refer to Section 5.44.1.

2.14.12 The Tabulated Results of Steel Design

Results of code verification and selection of members are presented in a tabular format. A detailed discussion of the format is provided in Section 2.11. Note

the following exceptions: CRITICAL CONDITION (CRITICAL COND) refers to the section of the LRFD specifications that governed the design.

If TRACK is set to 1.0, the design strength of members will be sent to print.

Table 2.2 Parameters of the LRFD

Name parameter	Default Value	Description
KX	1.0	K value for the bending-torsional buckling.
KY	1.0	K value Y. Usually local axis is the minor axis.
KZ	1.0	K value Z. Usually local axis is the major axis.
LX	Limb length	Length for flexural-torsional buckling.
LY	Limb length	Length to calculate the aspect ratio of the local buckling axis Y.
LZ	Limb length	Length to calculate the aspect ratio of the local buckling axis Z.
FYLD	36.0 ksi	Creep resistance of the steel.
FU	60.0 ksi	Tensile Strength Steel Last.
NSF	1.0	The factor of net section stress members.
UNT	Limb length	Unsupported length (L_b) * Skate superior to the bending force calculated. Only used if the compression bending is in the top flange.
UNB	Limb length	Unsupported length (L_b) Of the bottom flange * to the bending force calculated. Only used if the compression bending is in the bottom flange.
STIFF	Limb length	Spacing of stiffeners for beams for shear design.
CB	1.0	Coefficient C_b Chapter by F. If C_b will please to 0.0, calculated by the program. Any other value will be used directly in design.
TRACK	0.0	0.0 = Deletes all design forces. 1.0 = Print all design forces. 2.0 = Print out the extended design.
DMAX	45.0 inch.	Allowable maximum cant
DMIN	0.0 inch.	Minimum allowable superelevation.
RATIO	1.0	The allowable ratio of actual load and resistance design.
BEAM	1.0	0.0 = design at the ends and those situations specified by the SECTION command. 1.0 = design at the ends and at each point 1/12 along the length of the member. (The default)

* The top and bottom represent the positive and negative side of the local axis (local Z Z ZET UP if used).

> **Note: For review of the deflection, the DFF, DJ1 and DJ2 parameters of Table 2.1 can be used. All requirements remain the same.**

2.15 Design by the American Code of Cold Formed Steel

General

The projections of the AISI Specification for the Design of Structural Steel Members Cold Formed, 1996 Edition has been implemented. The program allows the design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations using the LRFD method. For members in bending, Section Nominal Resistance is calculated based on the initiation of provenance in the effective section (Method I). The increase in strength due to manufacturing Cold Work is a user option.

Cross Section Properties

The user specifies the geometry of the cross section by choosing one of the designations shaped section Tables Steel STAAD for sections cold formed reflecting Tables Property Section Thick published in the "Manual of Steel Design Formed in cold "AISI, 1996 Edition.

Tables are now available for the following forms: • Canal

with Lips

- Canal without Lips

- Angle with Lip • Angle without Lips • Z with Lips

- Z without Lips

- Hat

The shape selection can be done using the properties page of a member of the graphical user interface (GUI) or by specifying the section designation symbol in the input file. Details of the latter are explained below.

The AISC Steel Section Library:

The syntax of the command to assign line-steel sections AISC library is explained below:

<u>S e ction C co n s i o L ab</u>

```
20 TO 30 AT ST 14CS3.75X135
ST 33 36 TA 12CS1.625X102
ST 42 43 TA 4CS4X060
```

<u>S e ction C i s i o n s L ab</u>

```
50 TO 60 AT ST 10CU1.25X071
ST 32 33 TA 3CU1.25X057
TA 21 28 ST 1.5CU1.25X035
```

<u>S e ction co Z i o n s L ab</u>

```
January 3 4 TA ST 12ZS3.25X135
33 45 TA ST 10ZS3X060
December 13 TA ST 6ZS2X048
```

<u>S e ction Z i s i o n s L ab</u>

```
March 2 TA ST 8ZU1.25X105
May 4 TA ST 4ZU1.25X036
June 7 TA ST 1.5ZU1.25X048
```

<u>Á L ngu s what I Ig or ale ad co n s L ab ios</u>

```
August 9 TA ST 4LS4X105
November 10 ST TA 3LS3X060
December 13 TA ST 2LS2X075
```

<u>Á L ngu s what I Ig or ale ad wWithout L ab ios</u>

```
May 1 ST TA 4LU4X135
July 8 TA ST 2.5LU2.5X105
April 9 TA ST 2LU2X060
```

S and S cci or ne s o m b e ro n i s i or s L ab

```
4 8 TA ST 10HU5X075
5 6 TA ST 6HU9X105
January 7 TA ST 3HU4.5X135
```

Current Limitations:

Currently, only simple sections are available for standard specification. Options such as double angles, double channels, and sections provided by the user even tubes and rectangular tubes available shortly. Additionally, sections combined, as an angle above as a channel, or a plate welded on the top flange, bottom or side of the above forms are not available at this time.

STAAD.Pro uses the properties of the section without any reduction in the analysis phase of the structure. Both sections without cutting and the effective properties of the section in the design phase, as applicable.

Design Procedure

The following two design modes are available:

1. The Code Review
   The program compares the strength members to the effects of applied load, according with Method of AISI LRFD code. Code review is conducted for situations specified by the user via the SECTION (SECTION) command or the BEAM (BEAM) parameter. The results are presented in the form of an identifier PASS / FAIL and a RELATIONSHIP (RATIO) of the effects of load resistance for each verified member. The user can choose the level of detail in the data output by setting the TRACK parameter.

2. The Member Selection
   The user can ask the program to search the forms of cold-formed steel base data (AISI standard sections) for alternate members who pass code review and are at the discretion of lower weight. In addition, a minimum / maximum acceptable depth of member can be specified. Initially the program will then evaluate all sections of the specified database type (ie, channel, angle, etc.) and, if a suitable replacement is located, presents the results of the design for that section. If any section that satisfies the constraints of depth or lightness in the home can be found, the program stops the member unchanged, regardless if it passes code review or not.

The program calculates the properties of the effective section in accordance with the following sections:

> B2.1 Uniformly Compressed Elements stiffened. B2.3, Almas and
>
> stiffened elements with stress gradient B3.1, Uniformly
>
> Compressed Elements without Atiesamiento
>
> B3.2, without stiffening elements and Edge Stiffeners with Stress Gradient B4.2,
>
> Uniformly Compressed Elements with stiffener at the Edge

Cross-sectional properties of members to comply with the following Sections are checked:

> B1.1 (a) The radii of Plano-width-to-thickness maximum, and
>
> B1.2, The maximum depth within thick soul -a

The program verifies the strength of the member in accordance with Chapter C of the specification as follows:

Members in tension.

The resistance is calculated in accordance with Section

C2. Flexural Members.

a) C3.1, only bending efforts:
   C3.1.1, the nominal Efforts Section C3.1.2 Procedure I, lateral
   buckling Efforts

b) C3.2, efforts by the Court only.

c) C3.3, Efforts to Cut Combined Bending and
Compression Members Charged Concentric.

a) C4.1, Sections not subject to torsion or twist of flexion, and

b) C4.2, Single or Double Symmetrical Sections subject to Bending Torsion or Twist by. Combined Bending and Axial Load.

b) C5.1, Axial Tensile and Combined Bending and

d) C5.2, Axial Compressive Load and Bending Combined.

The following table lists the input parameters to specify the values of the design variables and selecting design options.

The design parameters of cold-formed steel AISC		
Name Parameter	Default Value	Description
BEAM	1.0	When this parameter is set to 1.0 (the default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the value is 0.0 BEAM, 13 locations checks are not done, and instead, verification is done only at the locations specified by the order of the SECTION (SECTION) (See manual for details STAAD). If neither parameter BEAM (BEAM) or any specified SECTION command, STAAD end the run and ask the user to provide one of those 2 commands. This rule does not gives to members of WEAVE (TRUSS) type.
CMZ	1.0	The coefficient extreme bending moments on the axis Z. See AISI C5.2.2. Used for the design of axial load and bending combined. The values range from 0.4 to 1.0.
CMY	0.0	The coefficient extreme bending moments about the axis Y. See AISI C5.2.2. Used for the design of axial load and bending combined. The values range from 0.4 to 1.0.
CWY	0	Specify whether the job cold rolling should be included in the calculation of the resistance. See AISI A7.2. Values: 0 -the effect is not to be included 1-the effect should be included
DMAX	1000.0	The maximum allowable depth for the section during the selection of the member. This value must be provided in the current units.
DMIN	0.0	REQUIRED The minimum depth for the section during the selection of the member. This value must be provided in the current units.
FLX	1	Specify if either flexural or torsional buckling is not necessary for the member is restricted. See AISI C4.1 Values: 0-Section subject to buckling flex no torsional constraint is provided 1-provided or unnecessary restriction
FU	58 ksi	The ultimate tensile strength of steel in the current units.
FY L D	36 ksi	Yield stress of steel in the current units.
KT	1.0	The factor of effective length for buckling

The design parameters of cold-formed steel AISC		
Name Parameter	**Default Value**	**Description**
		torsional. It is a fraction without units. The values can range from 0.01 (for a completely prevented torsional column buckling) to any specification of a large value for the user. It is used to calculate the proportion of KL / R for bending for the ability to axial compression.
KY	1.0	The effective length factor for column buckling about the local Y axis. It is a fraction and is unitless. The values can range from 0.01 (for a completely prevented column buckling) to any large value that the user specified. It is used to calculate the ratio of KL / R for determining the axial
KZ	1.0	The effective length factor for column buckling in the local Z .. and it's a fraction without units. The values can range from 0.01 (for a completely prevented column buckling) to any large value that the user specified. It is used to calculate the ratio of KL / R for the ability to axial compression.
LT	Length of Member	The length without contraventear for twisting. You enter in the current length units. The values can range from 0.01 (for a completely prevented torsional column buckling) to any large value that the user specified. It is used to calculate the proportion of KL / R by twisting for the ability to axial compression.
LY	Length of Member	The buckling length for the column in the local Y axis is entered in the current length units. The values can range from 0.01 (for a completely prevented column for buckling) to any large value that the user specified. It is used to calculate the ratio of KL / R to determine the ability to axial compression.
LZ	Length of Member	The effective length for buckling column in the local Z axis. You enter in the current length units. The values can range from 0.01 (for a completely prevented column buckling) to any large value that the user specified. It is used to calculate the ratio of

The design parameters of cold-formed steel AISC		
Parameter Name	Default Value	Description
		KL / R to determine the allowable stress in axial compression.
NSF	1.0	Factor for the net section tension members, see AISI C2.
ST I FF	Length of Member	The spacing in the longitudinal direction of shear stiffener elements for reinforced soul. You enter in the current length units. See section C3.2 AISI
T RA C K	0	This parameter is used to control the level of detail in the design output is reported in the output file. Acceptable values are: 0 - Prints only the member number, section name, relationship, and state PASS / FAIL (PASS / FAIL). 1 - Additionally print the design summary in addition to the printed by TRACK 1 2 -Print Members and material properties in addition to the printed by TRACK 2.
T SA	1	Specifies whether supports and intermediate transverse if atiezadores are present. If this is true, the program uses the most liberal group in AISI interaction equations C3.3.2. Values: 0-Beams reinforced with soul 1-The beams with stiffeners soul transverse

Section 3
American Concrete Design

3.1 Operations Design

STAAD has the necessary means for the design of concrete. It will calculate the reinforcement required for specified concrete section. All concrete design calculations are based on the current ACI 318.

3.2 Types Sections for the Design of Concrete

The following types of sections for concrete design can be defined. For

Beams   Prismatic (Rectangular and Square), Keystone and forms T

To Columns Prismatic (Rectangular, Square and Circular)
Slab   Finite element with specific thickness.

Walls / Boards
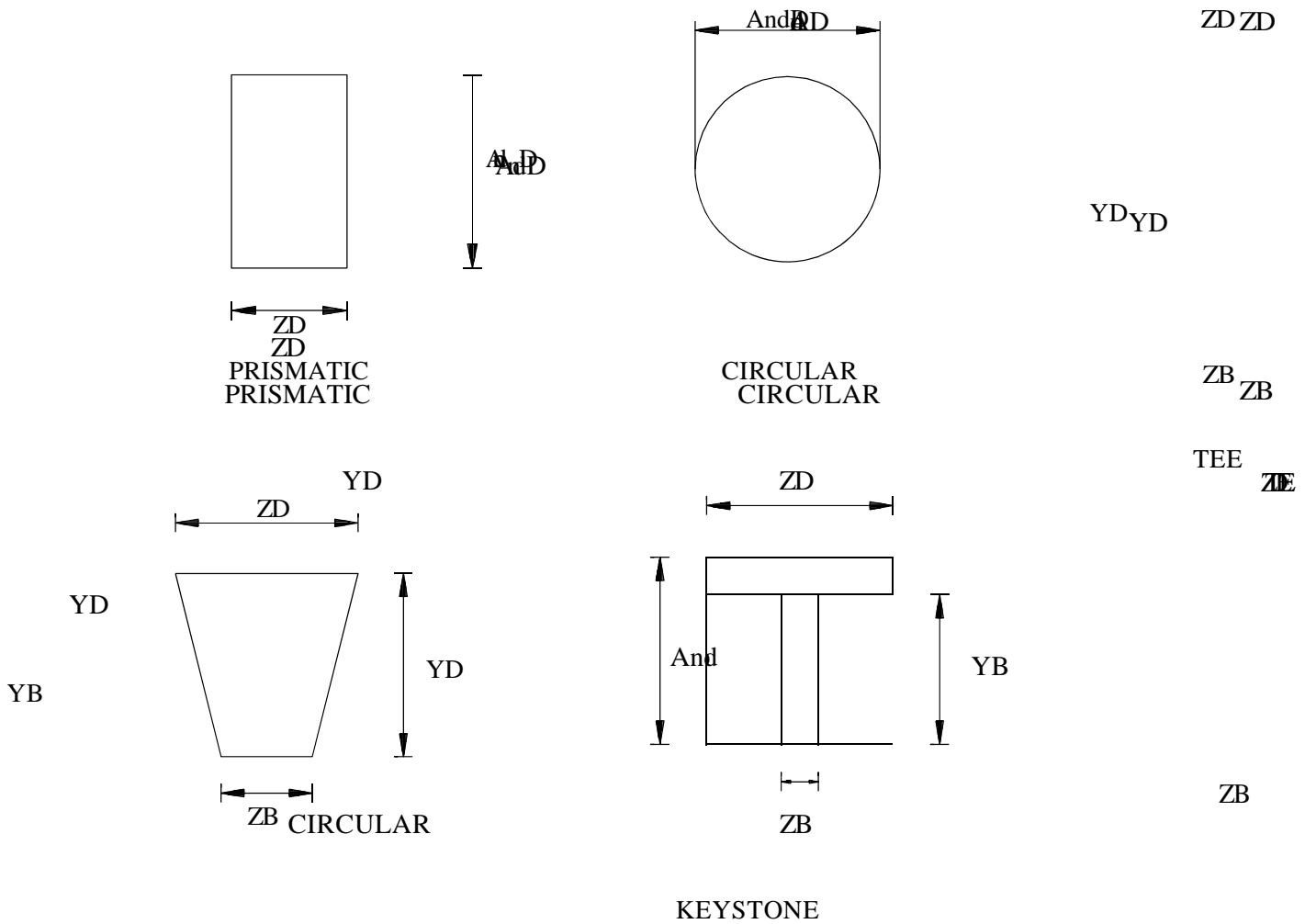
Figure 3.1 Different types of concrete sections

3.3 Dimensions of Members

For concrete members that will be designed with the program, they must define certain properties of the section, under the command MEMBER PROPERTY (MEMBER PROPERTY). The necessary input data are shown in the following example:

```
INCH UNIT
MEMBER PROPERTY
March 1 TO 18 YD July 9 PRISM 2916 ZD 12 IZ IY 1296
November 13 PR 12 YD.
TO PRIS YD 14 16 24 18 YB ZB ZD 48 12.
PR 17 TO 19 YD 24 ZD 18 ZB 12.
```

In the previous post, the first group defines a rectangular members (18 inches deep and 12 inches wide) and the second group, only provided depth and no width defines a circular member 12 inches in diameter. Note that the area (AX) is not defined for these members. For the design of concrete, this property does not have to be provided. If the cutting areas and moments of inertia are not provided, the program calculates these values of YD and ZD. Consider, as in the previous example, the values provided to IY and IZ are really 50% of the values calculated using YD and ZD. This is conventional practice, taking into account parameters revising section due to cracking of the section.

Note that the third and fourth group of the above example, defining T-shaped members and trapezoidal, respectively. Depending on the properties include (YD, ZD, YB, ZB, etc), the program will determine if the section is rectangular, trapezoidal or U shape and the design of beam (BEAM) will be made accordingly.

3.4 Design Parameters

The program contains a number of parameters that will be needed to implement the design by the ACI code. The default values of the parameters have been selected such that they are most often used by numbers of conventional design requirements. These values can be changed to suit a particular design. Table 3.1 is a complete list of the available parameters and their default values.

See
Section 5.47

Section 5.47.2 of this manual describes the commands necessary to define these parameters in the input file. For example, the distances of the faces of the supports of the end nodes of a beam, SFACE and EFACE (parameters that are used in the design section), have default values of zero, but can be modified depending on the actual situation . Similarly, the beams and columns are designed based on the moment analysis obtained directly without any magnification. MMag factor that can be used for purposes of magnification. For the beams, the user can generate load cases containing loads magnified by the appropriate load factors.

3.5 Slenderness Effects and Analysis Considerations

The slenderness effects are extremely important in the design of compression members. The ACI-318 code specifies two options with which the effect of slenderness can be adapted (Section 10.10 and 10.11 ACI-318). One option is to run a precise analysis that takes into account the influence of axial loads and variable moment of inertia about the stiffness of the limbs and times fixed ends, the effect of deflections on moments and forces, and the effect of the duration load. Another option is to amplify approximately design times.

STAAD has been drafted in such a way that allows the use of both. To perform the first type of analysis, use the command PDELTA ANALYSIS (PDELTA ANALYSIS) instead of the RUN ANALYSIS (PERFORM ANALYSIS). This method of analysis set requirements as specified in Section 10.10 of ACI-318 code, except for the effects of load duration. It is presumed that this effect can be ignored without risk since experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If desired, Staad can also accommodate any arbitrary time amplification factor (second choice) as an input, in order to provide some security for the effects of the duration of loading.

Name Parameter	Default Value	Description
FYMAIN	* 60,000 psi	Creep effort main reinforcing steel.
FYSEC	* 60,000 psi	Creep stress secondary reinforcement steel.
FC	* 4,000 psi	Compressive Strength of Concrete.
CLT	* 1.5 inch	Free coating for superior reinforcement.
CLB	* 1.5 inch	The free coating to the bottom gusset.
CLS	* 1.5 inch	The free coating for side reinforcement.
MinmAIN **	Rod Issue 4	Minimum size main reinforcement rod (No. 4-18)
MINSEC **	Rod Issue 4	Minimum size secondary reinforcement rod.
MAXMAIN **	Rod Number 18	Maximum size of main reinforcement rod
SFACE	* 0.0	Supporting face at the start location of the beam. If specified, the cutting force at the beginning at a distance of d + knot SFACE Start member is calculated.
EFACE	* 0.0	Locating the support face at the end of the beam. (Note: SFACE & EFACE are given positive value) If specified, the cutting force is calculated at the end at a distance of d + EFACE knot the end of the member.
REINF	0.0	Column with stirrups. A value of 1.0 means spiral stirrups.
MMAG	1.0 (For single columns)	A factor by which the moments of column design will be magnified.

WIDTH	* ZD	Width of the concrete member. The default ZD according to how it is taken is provided in the MEMBER PROPERTIES.
RipTH	* YD	Depth of concrete member. The default YD taken according to as provided in the MEMBER PROPERTIES.
NSECTION	12	Number of equally spaced sections to be considered in finding the critical points for the design of beams.
TRACK	0.0	DESIGN OF BEAM: With TRACK 0.0, Critical Moment will not be printed with the report design beam. A value of 1.0 means a print. A value of 2.0 will print the areas of steel required for all intermediate sections specified by NSECTION. COLUMN DESIGN: TRACK 0.0 the results of detailed design will be printed. TRACK 1.0 the result of the analysis of interaction in addition to the previous column printing (TRACK 0.0) is printed. TRACK 2.0 schematic diagram of interaction and interaction intermediate values is printed in addition to all the above

* These values should provided in the system of units that is you are driving.

** When using metric units for ACI design, hold their values
parameters in units 'mm' instead of the number of the rod. The following metric sizes rod are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25

mm, 32 mm, 40 mm, 50 mm and 60
mm.

While ignoring the effects of the duration of charge is, in a sense, an approach to be understood that the approximate evaluation of effects of slenderness is an approximate method. In this method, when the magnification is based on an empirical formula and assuming contraventeado. Considering all this information, it is our opinion that an PDELTA ANALYSIS, as implemented by STAAD, is very suitable for the design of concrete members. However, it should be noted, that to take advantage of this analysis, all load combinations should be provided as primary load cases and load combinations not like. This is due to the fact that the load combinations are purely algebraic combinations of forces and moments, while a primary charging case is reviewed during the P-Delta analysis based on the deflection. Also, note that the appropriate load factors (like 1.4 for DL, etc.) must be provided by the user. STAAD not factored loads automatically.

3.6   Design of Beams (Beam Design)

The beams are designed for bending, shear and torsion. For all these forces, all active loads on the beam are pre-analyzed to identify possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with NSECTION parameter. All these equidistant sections are analyzed to determine the time and cut envelopes.

3.6.1 Design by Flex

Reinforcements for positive and negative moments are calculated based on the properties of the section the user are defined. If the dimensions of the section are inadequate to withstand the applied load, that is, if the required reinforcement is greater than the maximum allowable for the section, the program reports the failure beam of reinforcement.

The effective depth is chosen as the total depth - (free coating + + stirrup diameter half the diameter of the main reinforcement), and an assay value obtained by adopting suitable rods sizes of the stirrup and the main reinforcement. The relevant clauses of sections 10.2 to 10.6 of ACI 318 is used to obtain the actual amount of steel required, as well as, allowable and minimum required maximum steel. These values are reported as ROW, and ROWMN ROWMX at the start and can be printed using the TRACK parameter 1.0 (see Table 3.1). Furthermore, the maximum and minimum actual spacing of the rod is also printed.

It is important to note that the beams are designed only for the bending moment MZ. The MY time is not considered in the design of flexion.

3.6.2 Design by Court

The shear reinforcement is calculated to resist both shear forces and torsional moments. Cutting forces are calculated at a distance (d + SFACE) and (d + EFACE) taken from the end nodes of the beam. EFACE SFACE and have default values of zero, unless otherwise specified (see Table 3.1). Note that the value of the effective depth "d" used for this purpose, is the ultimate value and features for the center of gravity (cg) of the main reinforcement actual calculated under the design bending. The relevant provisions of ACI 318 are used to calculate the reinforcement for shear forces and torsional moments. Based on the total requirement of the reinforcement stirrup, the size of rods, the spacing, the number of rods and the distance over which they are defined, the calculation takes place. The stirrups are always assumed 2 branches.

3.6.3 Design by Anchor

Anchor details are provided in the flexure design output. At any particular level, the coordinates of the START (START) and FIN (END) of the distribution of the main reinforcement is described along with information on how to stitch continuous anchorage, required or not, at the point of START and FIN. Note that the coordinates of these points HOME and END are obtained after taking into consideration the requirements of the anchor. The anchor length is calculated based on the clauses described in Chapter 12 of ACI 318.

3.6.4 Description of Output for the Design of Beams

Table 3.2 shows an output of a model of the real reinforcement developed by the program. The following notes apply to Table 3.2:

1) LEVEL: The serial number of the level of the rod which may contain one or more rods of group

2) HEIGHT (HT): height of the rod from the bottom of beam.

3) BAR INFO: About rebar detailing the number and size of rod.

4) FROM: Distance from the start of the beam at the beginning of the rebar.

5) TO: Distance from the beginning of the beam at the end of the rebar.

6) ANCHOR: Raises if the anchor, either a hook or (STA / END) continuum, is needed at the beginning (STA) or end (END).

7) ROW: Percentage of actual bending reinforcement required (As / bd) where b = width of the section (ZD for rectangular and square section) and d = effective depth of section (YD - the distance from the extreme tension fiber to cg main reinforcement).

8) ROWMN: Percentage required minimum flexural reinforcement (Amin / bd)

9) ROWMX: Percentage of maximum bending reinforcement required (Amax / bd)

10) SPACING: The distance between the centers of adjacent main reinforcement rods

11) Vu: Cutting force factored in the section.

12) Vc: nominal shear strength provided by concrete.

13) Vs: nominal shear strength provided by the shear reinforcement.

14) You: The factored torsional moment at section

15) Tc: nominal torsional moment strength provided by concrete.

16) Ts: nominal torsional moment strength provided by the reinforcement of torsion or pads (STIRRUP).

Table 3.2
(Real Output Design)

```
================================================= BEAM
================= N O.      14        D E S I G N    R
              E S U L T S - Flexure

   LEN - 20.00FT.  FY - 60000.  FC - 4000.  SIZE - 15.00 X 21.00 INCHES

   LEVEL    HEIGHT      BAR INFO      FROM           TO        ANCHOR
            FT.   IN.              FT.     IN.FT.  IN.    STA END
     -------------------------------------------------- ------------------
  1    0 + 2-5 / 8  3-Num.9      2 + 4 to 5/8  0-0 + 20/0    Do not     YES
     |-------------------------------------------------- ----------------|
     |     POS = CRITICAL MOMENT   189.77 12.00 AT KIP-FT FT, LOAD    1 |
     | REQD STEEL IN2 = 2.48, ROW = 0.0090, 0.0214 ROWMX ROWMN = 0.0033 = |
     |    MAX / MIN / CURRENT BAR SPACING = 7.50 / 2.26 / 4.94 INCH      |
     |    BASIC / REQD. DEVELOPMENT LENGTH = 37.95 / 31.42 INCH          |
     |-------------------------------------------------- ----------------|
  2    1 + 1.6 / 8  4-No.11      + 0-0 0/0    18 + 10-0 / 0 YES    Do not
     |-------------------------------------------------- ----------------|
     |     NEG = CRITICAL MOMENT    369.86 KIP-FT  AT  0.00 FT, LOAD   1 |
     | STEEL = REQD 5.47 IN2, ROW = 0.0198, 0.0214 ROWMX ROWMN = 0.0033 = |
     | MAX / MIN / CURRENT BAR SPACING =                    7.50 / 2.82
     |                      / 3.20 INCH
     |    BASIC / REQD. DEVELOPMENT LENGTH = 59.20 / 153.91 INCH         |
     |-------------------------------------------------- ----------------|
  3    1 + 3.6 / 8  3-No.6      8-1 + 16/4    0-0 + 20/0    Do not     YES
     |-------------------------------------------------- ----------------|
     |     NEG = CRITICAL MOMENT    105.87 KIP-FT  AT 20.00 FT, LOAD   1 |
     | STEEL = REQD 1.31 IN2, ROW = 0.0047, 0.0214 ROWMX ROWMN = 0.0033 = |
     | MAX / MIN / CURRENT BAR SPACING =                    7.50 / 1.75
     |                      / 5.13 INCH
     |    BASIC / REQD. DEVELOPMENT LENGTH = 16.70 / 27.75 INCH          |
     |-------------------------------------------------- ----------------|

      B E A M  N O.  14  D E S I G N  R E S U L T S - SHEAR

  START AT SUPPORT - Vu =  KIP 83.90  Vc =  KIP 35.10  Vs =  KIP 63.61
      PROVIDE NO. 4   AT BARS  7.0 IN. C / C FOR        108 IN.
  AT END   SUPPORT - Vu =  KIP 57.51  Vc =  KIP 35.10  Vs =  KIP 32.56
        PROVIDE NO. 4   AT BARS  9.3 IN. C / C FOR 84 IN.
```
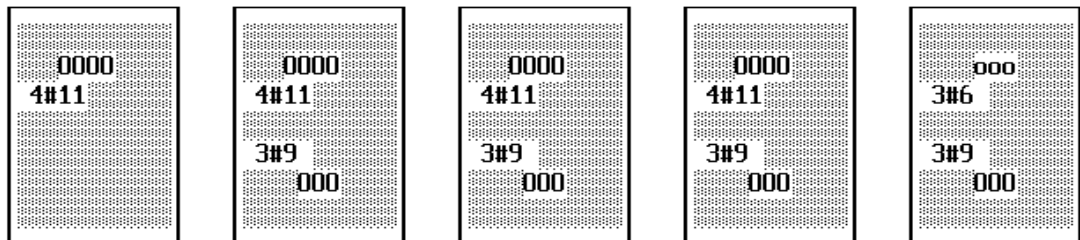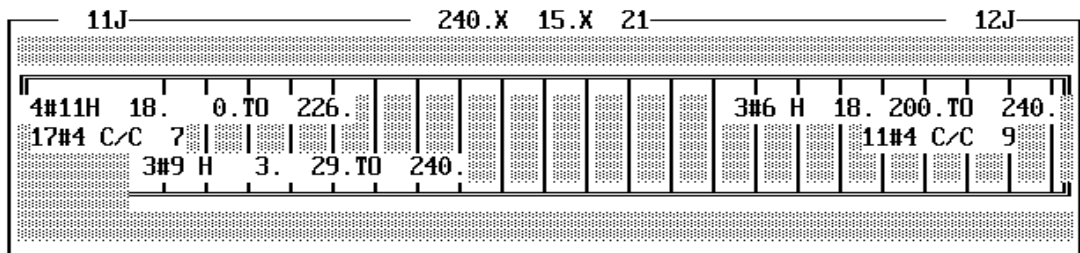
3.7 Design of Columns

The columns are designed by the ACI code STAAD for axial forces and uniaxial and biaxial moments. All active loads are examined to calculate the reinforcement. The load that produces the maximum reinforcement is called the critical load. The column layout is made for square, rectangular and circular sections.

For rectangular and square sections, the reinforcement provided is assumed to be evenly distributed with respect to each of the sides. This means that the total number of rods for these sections always be a multiple of four (4). If MMag parameter is specified, the column moments are multiplied by the value of MMag to reach the last moments in the column. Since the ACI code no longer requires any minimum eccentricity as a condition to be satisfied, such checks are not performed.

Method Used: The Method Bresler load contour

Known values: Pu, Very, Muz, B, D, free coating, Fc, Fy, The ultimate strain for concrete: 0.003

The steps involved:

1) Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.

2) Find an approximate arrangement of reinforcement rods for the course.

3) Calculate PNMAX = 0.85 Po where Po is the maximum axial load capacity of the section.
Ensure that the actual nominal column load does not exceed PNMAX. If PNMAX is less than Pu / PHI (PHI is the strength reduction factor) increased reinforcement and repeat steps 2 and 3 if the reinforcement exceeds 8%, the column can not be designed with its current dimensions.

4) In the event reinforcing rod under axial load and, find capabilities upon uniaxial column to the Y and Z independently. These values are respectively called MYCAP and MZCAP.

5) Solve the equation of Interaction

$$\left(\frac{Mny}{Myca_p}\right)^{\alpha} + \left(\frac{Mnz}{Mzca_p}\right)^{\alpha} \leq 1.0$$

where α = 1.24. If the column is subjected to a uniaxial moment α is chosen as 1.0

6) If the equation is satisfied Interaction, find an arrangement with the sizes of the rods available, find the uniaxial capacity and solve the equation of the interaction again. If the equation is now satisfied, reinforcement details to the output file is written.

7) If the interaction equation is not satisfied, the assumption reinforcement increases (ensuring that it is under 8%) and repeat steps 2 to 6.

Interaction of the column

The interaction values of the column can be obtained using the design parameter TRACK TRACK 1.0 and 2.0 for the column member. If a value of 2.0 is used for parameter TRACK, 12 different Pn-Mn pairs, each representing a different point in the curve of Pn-Mn will be printed.

Each of these points represents one of several Pn-Mn combinations of this column is able to carry on the given axis, to the actual reinforcement for the column is designed. In the case of round columns, the values are for any of the radial axes. Values printed in the output for the TRACK 1.0 are:

P0   = Maximum capacity purely axial load bearing column (zero time).

PNMAX = The maximum allowable axial load on the column (Section 10.3.5 of

ACI 318). P-bal   = Capacity to axial tension load balanced condition.

M-bal   = The ability to time the condition Uniaxial tension balanced. e-bal = M-bal

/ P-bal = Eccentricity to the condition of balanced tension.

M0   = Moment capacity with zero axial load. P-tens

               = Maximum load allowable stress on the spine.

Des. Pn = Pu / PHI PHI where is the Strength Reduction Factor and Pu is the axial load in the case of critical load.

Des. Mn = Mu * mmag / PHI PHI where is the Strength Reduction Factor and Mu is the bending moment for the proper axis for the case of critical load. For round columns,

$$M_{or} = \sqrt{M_{uy}^2 + M_{uz}^2}$$

e / h   = (Mn / Mn) / h where h is the length of the column.

Output Column design

The examples show different levels of production column design. The following output without any definition of TRACK (ie, using the default value TRACK 0.0) is generated:

```
=================================================================
    COLUMN  NO.    5  DESIGN PER ACI 318-05 - AXIAL + BENDING
FY - 60000  FC - 4000 PSI,  SQRE SIZE - 12.00 X 12.00 INCHES, TIED
        AREA OF STEEL REQUIRED =   7.589  SQ. IN.
BAR CONFIGURATION        REINF PCT.   LOAD   LOCATION   PHI
-----------------------------------------------------------
   8 - NUMBER  9            5.556       2      STA     0.650
(PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
TIE BAR NUMBER   4 SPACING   8.00 IN
```

TRACK 1.0 generates the following additional output over the previous output (**TRACK 0.0**).

```
      COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)
  ----------------------------------------------------------
   P0           Pn max     P-bal.     M-bal.    e-bal.(inch)
  942.40       753.92     179.59     170.75    11.41
   M0           P-tens.    Des.Pn     Des.Mn     e/h
  148.52       -480.00     350.15     10.47     0.00249
  ----------------------------------------------------------
      COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)
  ----------------------------------------------------------
   P0           Pn max     P-bal.     M-bal.    e-bal.(inch)
  942.40       753.92     179.59     170.75    11.41
   M0           P-tens.    Des.Pn     Des.Mn     e/h
  148.52       -480.00     350.15     136.51    0.03249
  ----------------------------------------------------------
```

TRACK 2.0 generates the following additional output over all previous outputs.

```
                              Pn       Mn       Pn       Mn     (@ Z )
                  |         695.93    77.23   347.96   148.53
          P0  |*           637.93    93.16   289.97   157.71
              | *          579.94   107.06   231.98   164.41
      Pn,max|__*           521.94   118.23   173.98   170.18
              |    *        463.95   129.01   115.99   163.66
      Pn    |     *         405.96   139.03    57.99   156.37
    NOMINAL|        *        Pn       Mn       Pn       Mn     (@ Y )
      AXIAL|          *     695.93    77.23   347.96   148.53
  COMPRESSION|          *   637.93    93.16   289.97   157.71
         Pb|-------*Mb      579.94   107.06   231.98   164.41
            |        *      521.94   118.23   173.98   170.18
  _____|____*_____   463.95   129.01   115.99   163.66
            |  * M0   Mn,   405.96   139.03    57.99   156.37
            | *    BENDING
      P-tens|*     MOMENT
            |
```

3.8 Design Elements, Slabs and Walls Cutters

STAAD currently provides facilities for the design of three types of entities associated with the type of surface structures.

   a) Individual plate elements - these are designed from the viewpoint that an element is independent of this element.

   b) Walls Cortantes- structural components modeled using the SURFACE command INCIDENCE (incidence of the surface) can be designed as shear walls support.

The design elements are made only by MX and MY time in the center of the element. The design will not be effected by SX, SY, SXY, SQX, SQY or MXY. Also, the design is not done at any other point on the surface of the element.

A typical example of the output of a finite element design is shown in Table 3.4. The reinforcement required to resist moment Mx is denoted as longitudinal reinforcement and reinforcement necessary to resist My time is denoted as transverse reinforcement (Figure 3.1). The FYMAIN, FC and CLEAR parameters are listed in Table 3.1 are relevant in the design of a slab. Other parameters listed in Table 3.1 are not applicable to the design of the slab. Note that the default value of the coating - CLT and CLB parameters - for the elements of the plate is 0.75 inches in the English system and 3 cm in the international system, as shown in Table 3.1.

Figure 3.2: Sign conventions for a plate-loaded element
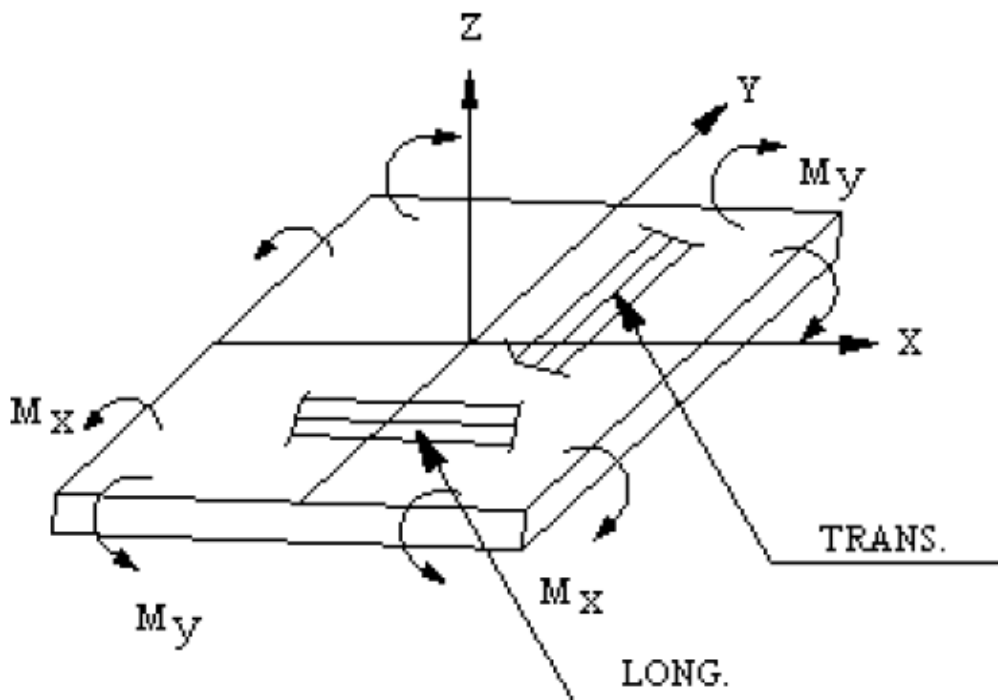
Table 3.4 Output element design plate-

```
ELEMENT DESIGN SUMMARY
----------------------
ELEMENT    LONG. REINF      MOM-X /LOAD     TRANS. REINF      MOM-Y /LOAD
           (SQ.IN/FT)       (K-FT/FT)       (SQ.IN/FT)       (K-FT/FT)
     FY:    60.000 KSI FC:     4.000 KSI COVER (TOP):     0.750 IN
     COVER (BOTTOM):     0.750 IN TH:      6.000 IN
  47 TOP : Longitudinal direction - Only minimum steel required.
  47 TOP : Transverse direction   - Only minimum steel required.
  47 TOP :     0.130            0.00 /    0           0.130            0.00 /    0
     BOTT:     0.562           11.60 /    1           0.851           14.83 /    1
```

Section 4

Mexican Design

4.1 Concrete Design by NTC MEX 1987

STAAD.Pro is able to perform the specific design based on the Mexican code NTC 1987 Complementary Standards for Design and Construction of Concrete Structures.

The design of the NTC members by 1987 requires selecting package STAAD "Latin American design codes" code.

4.1.1 Operations Design

STAAD has the capabilities for the design of concrete. It will calculate the reinforcement required for the specified section of concrete. All concrete design calculations are based on the most current: additional technical standards for the design and construction of structures

Concrete - November 1987 building code for the Mexican Federal District

-Aug. 1993 (Building Code for the Federal District).

4.1.2 Types sections for designing concrete

They can define the following types of cross sections for the particular design. Columns -

Prismatica (Rectangular, Square and Circular)

Beams - Prismatica (Rectangular and Square), Keystone, and forms T

Walls -Elements finite thickness with a specific

Figure 4.1.2 - Nomenclature concretely for beams and columns
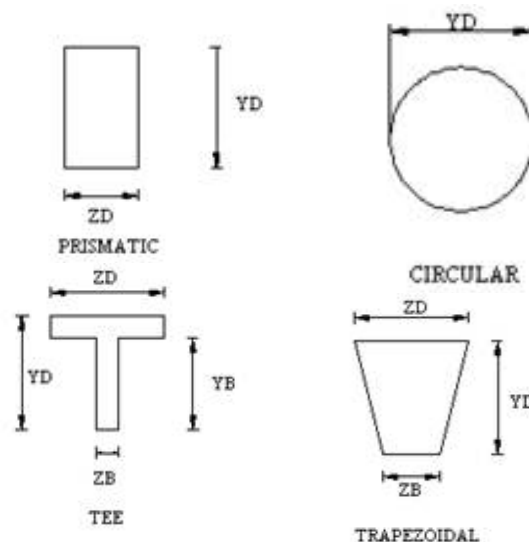
4.1.3 Dimensions of members

Members of concrete that will be designed by the program must have some income under Section Properties command MEMBER PROPERTY. The following example shows the required input:

```
UNIT CM
MEMBER PROPERTY
PRISM TO 79 YD 13 40 ZD 20 IY IZ 53333 13333
November 13 PR YD 20.
TO PRIS YD 14 16 24 18 YB ZB ZD 48 12.
PR 17 TO 19 YD 24 ZD 18 ZB 12.
```

In the previous post, the first set of members are rectangular (40 cm deep and 20 cm wide) and the second set of members, not only with depth and width provided, is assumed to be circular 20 cm in diameter. Note that no area (AX) is provided for these members. For the design of concrete, should not be provided at this property. If cutting areas and moments of inertia are not provided, the program calculates the values of YD and ZD. Note that in the above example the IZ and IY values provided are actually 50% of the calculated values using YD and ZD. This is a conventional practice to take into account the revised parameters due to cracking section of the section.

Note that the third and fourth set of members in the previous example represent a T shape and form KEYSTONE respectively. Depending on the properties provided (YD, ZD, YB, ZB, etc), the program will determine if the section is rectangular, trapezoidal or U shape and therefore the design of the beam is performed.

4.1.4 Design Parameters

The program contains a number of parameters that are necessary to realize the design by the Mexican code. They have been selected default parameters that are used for conventional design requirements numbers. These values can be changed to suit to perform the particular design. Table 3.1 is a complete list of the available parameters and their default values.

The manual describes the commands necessary to provide these parameters in the input file. For example, and EFACE SFACE values (parameters used in the design of cutting), the distances from the face of the mounting end nodes of a beam, are assigned a default value of zero, but may be modified depending of the real situation. Similarly, the beams and columns are designed to now obtained directly

analyzes without any increase. MMY factors and MMZ be used to increase column moments. For beams, the user can generate load cases containing fillers magnified by the appropriate load factors.

Note: When a parameter is specified, the value is maintained in the specified number until specified again. This is the way STAAD works for all codes.

Table 4.1.4- concrete design parameters Mexican

Parameter Name	Default Value	Description
CODE	–	Must specified how come **MEXICAN**. Design code to follow.
CCL	1	Class of concrete according 1.4.1d) to define the modulus of elasticity 1. Class 1 Concrete 2. Concrete class 2
CFB	FALSE	Classification of Cold Formed bar to define multipliers development according to Table 3.1 NTC • FALSE -Cold bar unformed • TRUE -Cold formed bar
CLB	3 cm	Lower reinforcement coating
CLS	3 cm	Lateral reinforcement coating
CLT	3 cm	Coating of top reinforcement
DAG	2 cm	Maximum aggregate diameter, in current units.
DCP	TRUE	Loads and reactions in beams direct compression • FALSE - loads indirectly applied • TRUE =Direct Compression
DEPTH	YD	Depth of the concrete member in current units. This default **YD** under **MEMBER PROPERTIES**.

DIM	TRUE	EALSE: No precautions - reduced from section to section 1.5 NTC Concrete TRUE: the precautions taken to ensure dimensional
DSD	TRUE	Ductile frames in accordance with Article 5 of the code. Some design conditions are considered (excluding, for the moment, or a geometric confinement) • FALSE - no ductile Marcos • TRUE -Marcos Ductile
EFACE	0	Location facing the tail end of the supporting beam. If specified, the beginning Shear force is computed at a distance of EFACE + d from the start of the joint member. Positive number.
EXP	FALSE	Exposure time to the ground or to set the coating and the steel reinforcement minimum • FALSE - Not exposed to soil or climate • TRUE - Exposed soil or climate
FC	200 Kg / cm2	Resistance to the compression concrete
FYMAIN	4,200 Kg / cm2	Yield stress for steel reinforcement principal
FYSEC	4,200 Kg / cm2	Yield stress for secondary steel reinforcement (stirrup)

LSS	0	Considered part of the longitudinal steel to reduce shear. 0 (zero) is conservative. Value between 1 and 0.
LTC	FALSE	Lightweight concrete development to define multipliers according to Table 3.1 NTC • FALSE - Regular concrete • TRUE -
MAXMAIN	12	Size Maximum main reinforcement bar (Issue 2 -18)
MINMAIN	2.5	Size rebar principal minimum (Issue 2 -18)
MINSEC	2.5	Size minimum secondary reinforcement bar (Number 2 -18)
MMY	1.0	Moment magnification factor for columns about My
MMZ	1.0	Moment magnification factor for columns on Mz
MOE	198,000 Kg / cm2	Modulus of elasticity of concrete
NSECTION	12	Number of equidistant sections should be considered in the search for critical points for the design of the beam
PHI	90 degrees	Stirrups angle with the axis of element
PSS	TRUE	Supported slab perimeter. To calculate the minimum steel to 2.1.2
REINF	0	Zunchada column (spiral reinforcement). A value of 1 means spiral.

SFACE	0	Location faces the leading end of the supporting beam. If specified, the beginning Shear force is computed at a distance of SFACE + d from the start of the joint member. A positive number
TEQ	FALSE	Beam balance required for torsional Cl.2.1.6a) 2nd paragraph • FALSE – No •
TRACK	0	**Design of Beams** 1. The critical moment is not printed with the design brief of the beam. 2. Print means a critical time. 3. It will print steel areas required for all intermediate sections specified by NSECTION. **Design of columns** 1. It will print results of detailed design. 2. It means that the interaction of the test results column will be printed with TRACK 0 output 3. It will print a schematic diagram of the interaction and intermediate values in addition to all the above.

WIDTH	ZD	Width of the concrete member in current units. This defaults to ZD under MEMBER PROPERTIES

* These values can be provided in the current unit system is used.

> **Note: When rods design metrics are used, provided values for these parameters in actual units 'mm' instead of the bar number. Are available the following metric sizes: 4.2 mm, 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32**

mm, 40 mm, 50 mm and 60 mm.

4.1.5 Design of beams

The beams are designed for bending, shear and torsion. For all these forces, all active loads are pre-scanned to identify possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with NSECTION parameter. All these equidistant sections are analyzed to determine the envelope of the time and shear.

4.1.5.1 Bending Design

Reinforcement for positive and negative moments are calculated based on the section properties provided by the user. If the dimensions of the section are insufficient to carry the load applied, if the required reinforcement is greater than the maximum allowed by the cross section, the program reports that beam failure of reinforcement. The rectangular sections are also designed with compression strengthening.

The effective depth is chosen as depth Total - (+ coating + stirrup diameter half the diameter of the main reinforcement), and a test value was obtained by adopting suitable rod sizes for stirrups and main reinforcements.

The relevant clauses in sections 1.5, 1.6, 2.1.1-2-5, 3.10 and 5.2.2 NTC concrete are used to obtain the actual amount of the maximum and minimum steel required and necessary. These values are reported as ROW, and ROWMN ROWMX on output and can be printed using the TRACK parameter 1.0 (see Table 4.1.4). Furthermore, the maximum, minimum and actual space bar also printed.

Importantly, the beams are designed for bending moment MZ only. The MY time is not considered in the design bending.

4.1.5.2 Design shear

The shear reinforcement is calculated to resist shear forces and torsional moments. These forces at a distance (d + SFACE) and (d + EFACE) is calculated away from the end nodes of the beam. EFACE SFACE and have default values of zero, unless it is established under other parameters (see Table 4.1.4). Note that the value of the effective depth "d" used for this purpose is to update accounts for the actual center of gravity of the main reinforcement calculated under bending design.

The concrete 2.1.5-6 and 5.2.4 NTC clauses are used to calculate the reinforcement shear forces and torsional moments. Based on total overall reinforcement stirrup required, the size of the bars, the spacing is calculated, the number of bars and the distance over which they are provided. The Stirrups due to geometric conditions are assumed to be 2 feet, due to design conditions could be 2 or 4 legs.

4.1.5.3 Anchorage Design

At the exit to design flexural strength, details of anchor are also provided. For any particular level, the initial end of the path of the main reinforcement coordinates and described in conjunction with information whether shaped anchor hook or a continuation or not the start and end points are required. Note that the coordinates of the start and end points are obtained after taking into account the anchorage requirements.

The anchorage length is calculated based on the terms described in Section 3.1 NTC concrete. In case the program selects 2 different diameters or principal compression reinforcement, anchoring analyzed only for the larger diameter.

4.1.5.4 Output description for the design of beams

The following list shows which of the various parameters used in the design of concrete beams by the Complementary Technical Norms do not vary at all with respect to the American Standards ACI (See Section 3 - American Concrete Design).

1) LEVEL: The serial number of the level of the rod which may contain one or more rods of group

2) HEIGHT (HT): height of the rod from the bottom of beam.

3) BAR INFO: About rebar detailing the number and size of rod.

4) FROM: Distance from the start of the beam at the beginning of the rebar.

5) TO: Distance from the beginning of the beam at the end of the rebar.

6) ANCHOR: Raises if the anchor, either a hook or (STA / END) continuum, is needed at the beginning (STA) or end (END).

7) ROW: Percentage of actual bending reinforcement required (As / bd) where b = width of the section (ZD for rectangular and square section) and d = effective depth of section (YD - the distance from the extreme tension fiber to cg main reinforcement).

8) ROWMN: Percentage required minimum flexural reinforcement (Amin / bd)

9) ROWMX: Percentage of maximum bending reinforcement required (Amax / bd)

10) SPACING: The distance between the centers of adjacent main reinforcement rods

11) Vu: Cutting force factored in the section.

12) Vc: nominal shear strength provided by concrete.

13) Vs: nominal shear strength provided by the shear reinforcement.

14) You: The factored torsional moment at section

15) Tc: nominal torsional moment strength provided by concrete.

16) Ts: nominal torsional moment strength provided by the reinforcement of torsion or pads (STIRRUP).

Example output file of the design of a beam


```
========================================================================

        BEAM   NO.     5 DESIGN RESULTS - FLEXURE

PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES,DDF

 LEN -  6096.00(mm)  FY -   412. FC -   20. SIZE -  400.00 X  500.00(mm)

 LEVEL     HEIGHT     BAR INFO       FROM          TO          ANCHOR
           (mm)                      (mm)         (mm)        STA   END
 _____


     1        47.    7 -  6MM        0.         3556.        YES   NO
     1        47.    2 -  4MM        0.         3556.
     1       453.    8 -  3MM        0.         3556.
     2        86.    3 -  6MM        0.         3556.        YES   NO
```

4.1.6 Design of columns

The design of columns in STAAD by the Mexican code is made by uniaxial axial force and biaxial moments like. All active fillers are made for calculating the reinforcement. The load that produces the greatest amount of reinforcement is called the critical load.

The column design is performed to square, rectangular and circular sections. For rectangular and circular sections, the reinforcement is always assumed to be equally distributed on all sides. This means that the total number of bars to these sections will always be a multiple of April 4th. If parameters are specified and MMAGx - MMAGy, column moments are multiplied by the corresponding value MMag to reach the last moments in the column. The conditions to satisfy minimum eccentricity are checked under Section 2.1.3.a of NTC.

Method Used: The Method Bresler load contour

Known Values: Pu, Muy, Muz, B, D, free coating, Fc, Fy

The ultimate strain for concrete: 0.003

The steps involved:

1. Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.

2. Find an approximate arrangement of reinforcement rods for the course.

3. Calculate PNMAX = Po, where Po is the maximum axial load capacity of the section. Ensure that the rated load in the column does not exceed PNMAX. If PNMAX is less than the axial force Pu / FR (FR is the force reduction factor) increases the reinforcement and repeat steps 2 and 3 If the reinforcement is greater than 6% (or 4% for the ductile design), the column can not be designed with its current dimensions.

4. For the assumed reinforcement arrangement and axial load capacities uniaxial found when the column to the Y and Z independently. These values are called MYCAP and MZCAP respectively.

5. Solve the equation of Interaction

$$\left(\frac{Mny}{Myca_p} \right)^{\alpha} + \left(\frac{Mnz}{Mzca_p} \right)^{\alpha} \leq 1.0$$

Where $\alpha = 1.24$. If the column is subjected to a uniaxial moment α is chosen as 1.0

6. If the interaction equation is met, a match is found with rod sizes available, is uniaxial to search capabilities and interaction solve equation again. If now the equation is satisfied, the reinforcement details are written to the output file.

7. If the interaction equation is not satisfied, the assumed reinforcement increases (to ensure it is below 6% and 4% respectively) and repeat steps 2 to 6.

To check the shear and torsion in columns, sections and beams must be checked and stricter reinforcement adopted adopted both shear and torsion.

4.1.7 Interaction column

The interaction values of the column can be obtained using the design parameter TRACK TRACK 1.0 and 2.0 for the column member. If a value of 2.0 is used for parameter TRACK, 12 different Pn-Mn pairs, each representing a different point in the curve of Pn-Mn will be printed.

Each of these points represents one of several Pn-Mn combinations of this column is able to carry on the given axis, to the actual reinforcement for the column is designed. In the case of round columns, the values are for any of the radial axes. Values printed in the output for the TRACK 1.0 are:

- P0 = the maximum capacity purely axial load bearing column (zero time).

- PNMAX = Maximum permissible axial load on the column

- P-bal = axial capacity at balanced strain loading condition.

- M-bal = The ability to time the condition Uniaxial tension balanced. • e-bal = (M-bal / bal-P) = Eccentricity to the condition of balanced tension. • M0 = The moment capacity with zero axial load.

- P-tens = maximum load allowable stress on the spine.

- Des. Pn = Pu / PHI PHI where is the Strength Reduction Factor and Pu is the axial load in the case of critical load.

- Des. Mn = Mu * mmag / PHI PHI where is the Strength Reduction Factor and Mu is the bending moment for the proper axis for the case of critical load. For round columns,

$$M_{or} = \sqrt{M_{uy}^2 + M_{uz}^2}$$

- e / h = (Mn / Mn) / h where h is the length of the column.

4.1.8 Output column layout

The examples show different levels of production column design. The following output without any definition of TRACK (ie, using the default value TRACK 0.0) is generated:

```
======================================================================

         COLUMN   NO.     1  DESIGN PER MEX NTC-87 - AXIAL + BENDING



  FY - 411.9  FC -  19.6 MPa     RECT SIZE   30.0 x  50.0 (mm) TIED

          AREA OF STEEL REQUIRED =4194.256

  BAR CONFIGURATION        REINF PCT.    LOAD    LOCATION    PHI
  -----------------------------------------------------------

    4 - NUMBER 12             3.040         1       STA     0.800
  (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)



======================================================================
```

TRACK 1.0 generates the following additional output over the previous output (**TRACK 0.0**).

```
     COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)

  -----------------------------------------------------------
    P0           Pn max        P-bal.        M-bal.       e-bal.(mm)
   6755.03       6755.03       2720.32        446.01        16.4
    M0           P-tens.       Des.Pn        'Des.Mn       e/h
     713.93      -4224.23      -273.12        663.14        -.398
  ---------------------------------------------------------------------

     COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)

  -----------------------------------------------------------
    P0           Pn max        P-bal.        M-bal.       e-bal.(mm)
   6755.03       6755.03       2644.48        338.35        12.8
    M0           P-tens.       Des.Pn        'Des.Mn       e/h
     530.87      -4224.23      -273.12          0.00        0.000
  ---------------------------------------------------------------------
```

TRACK 2.0 generates the following additional output over all previous outputs.

```
                 --------------------------------------------------------------------
                               Pn         Mn          Pn         Mn      (@ Z )
              |                6235.42     95.93      3117.71     535.85
          P0  |*               5715.80    173.57      2598.09     589.09
              |  *             5196.18    271.44      2078.47     641.01
    Pn,max|__ *                4676.56    354.07      1558.85     692.55
              |    *           4156.94    423.13      1039.24     732.33
          Pn  |       *        3637.33    480.06       519.62     732.52
       NOMINAL|          *     Pn         Mn          Pn         Mn      (@ Y )
         AXIAL|            *    6235.42     70.99      3117.71     402.20
   COMPRESSION|             *   5715.80    130.40      2598.09     441.31
            Pb|-------*Mb       5196.18    206.06      2078.47     479.31
              |          *      4676.56    267.63      1558.85     516.96
   _____|_____*_____    4156.94    318.56      1039.24     552.82
              |  * M0    Mn,    3637.33    360.97       519.62     551.77
              | *     BENDING
       P-tens|*       MOMENT
              |

       ========================================================================
```

4.1.9 Design of slabs

The slabs are designed according to specifications of the Mexican NTC. To design a slab should be modeled using finite elements.

The design elements are made only by MX and MY time in the center of the element. Such a design will not be made by FX, FY, FXY, MXY. Also, the design is not done at any other point on the surface of the element. The cut is checked with Q.

A typical example of output element design is shown. The reinforcement required to resist moment Mx is denoted as longitudinal reinforcement and reinforcement necessary to resist My time is denoted as transverse reinforcement. Parameters **FYMAIN, FC, CLB, CLS, CLT, DIM, and EXP listed in Table 4.1.4 are relevant to the design of the slab. Other mentioned parameters are not used in the design of the slab.**

Figure 4.1.9-moments item: Longitudinal (L) and transverse (T)
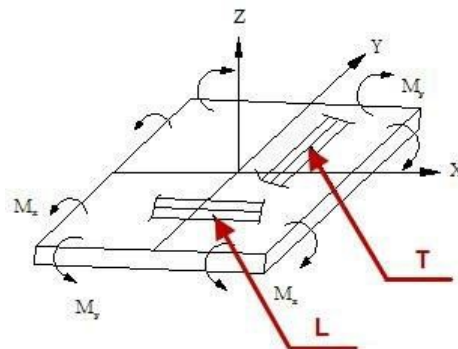
4.1.9.1 Example of an output file of a plate-like element

```
DESIGN ELEMENT SUMMARY
----------------------
ELEMENT   LONG. REINF   MOM-X / LOAD      TRANS. REINF   MOM-AND /
          (SQ.MM/MM)   (KN-MM / MM)       (SQ.MM/MM)     (KN-MM /
                                                          MM)
       47 TOP: Longitudinal direction - Only minimum required steel.
       47 TOP: Transverse steel direction- Only minimum required.
       47 TOP:   0.205      0.00 /      0        0.205   0.00 / 0
          BOTT:   0.254     10.44 /      1        0.362  13.35 / 1
       47 SHEAR CAPACITY  57.06 KN PASS *** 3 *** FOR LOAD CASE
            Indicates EXCEEDS MAXIMUM REINFORCEMENT *****
*************************** END  OF  ELEMENT  DESIGN ******************
********
```

4.2 Steel Design by NTC MEX 1987

STAAD.Pro is able to perform steel design based on the Mexican code NTC 1987 (Complementary Technical Norms for Design and Construction of Steel Structures) or Building Regulations for the Federal District.

The NTC members by Design 1987 is required to select the package STAAD Latin American design codes code.

4.2.1 General

The design philosophy considers that the method of load cases and design strength or limit states design usually known as factor load and resistance design (LRFD).

The structures are designed and proportionate taking into account the boundary condition which would become unfit for its intended use. Two major categories are recognized limit state - ultimate and serviceability. The main considerations in ultimate limit state design are strength and stability, while in service is deviation. The appropriate load factors and resistance are used as well, uniform for all steel structures under different load conditions while the chances of being overwhelmed reliability limits are acceptably remote is achieved.

In implementing STAAD Mexican standards for steel structures, members are provided to withstand design loads without exceeding the limit states of strength and stability. Allows you to check the deformation to verify serviceability.

Consequently, the economic section is selected based on the criteria of less weight gained by the designer in specifying allowable depths members, type of desired section or other such parameters. The check code of the program verifies that the requirements of the main code for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the Mexican specifications as implemented in the design of steel STAAD. A brief description of the key concepts is presented.

4.2.2 Fundamentals of limit state design

The primary objective of the design specification limit states is to provide a uniform basis for all steel structures under different loading conditions reliability.

The design method used to limit state independent factors for each load and resistance. Due to various factors reflect the degree of uncertainty of different loads and load combinations and the accuracy of estimated strength, more uniform reliability is possible.

The method can be summarized by the inequality

$$And_i Q_i \leq R_n FR$$

On the left side of the inequality, the required force is the sum of the various effects of load (Qi), multiplied by their respective load factors (Yi). The strength of the design on the right side is the nominal resistance and resistance (Rn), multiplied by a factor of resistance (FR).

In implementing STAAD Mexican standards, assumes that you use appropriate load factors and load combinations creates necessary for analysis. The part of the program design should consider the effects of load (forces and moments) obtained from the analysis. In the calculation of the resistor components (beams, columns, etc.), resistance (nominal strength) and resistance factor automatically deemed applicable.

4.2.3 Moments and forces members past

The last moments and forces members in the result of members of the loads applied to the structure. These forces are in the local coordinate system of the members. The following figures show the final actions of the member with their addresses. See section 1.19 of the Technical Reference Manual for details.

4.2.4 Classification section

The specification limit state design allows inelastic deformation of section elements. Thus, the local buckling becomes an important criterion. The steel sections are classified as compact (type 2), no compact (type 3) or threadlike element (type 4), depending on the characteristics sections of local buckling, in addition to one type of plastic sections are able design. This classification is a function of the geometric properties of the section. The design procedures are different depending on the class section. Staad is able to determine the classification of the section to the standard design methods accordingly.

4.2.5 Effort Axial members

The criteria governing the capacity of tension members is based on two limit states. The limit state performance in the gross section is intended to prevent excessive limb lengthening. The second involves the fracture limit state in the section with the lowest effective net area. The net area of the section may be specified by the user by using the NSF parameter (see Table 13B.1), which always refers to the gross section. STAAD calculates the voltage capacity of a Member based on these two limit states and proceeds accordingly with the selection of members or verification code.

Besides the criterion of tensile strength, the user defines whether the tension members are required to meet the slenderness limitations are a function of the nature of use of the member (main load resisting component, reinforcing members, etc.) . While the selection of member checking code and process it checks immediately STAAD slenderness to appropriate members before continuing with other procedures for determining the suitability of a particular member.

4.2.6 Axial Compression

Force equations take into account column inelastic deformation and other recent research on the behavior of the column. Two equations governing the pressure head are available, one for and one for inelastic buckling or elastic Euler buckling. Both equations include the effects of residual stresses and out-of-straightness initial. The compressive strength of a particular member STAAD is calculated by the method described in section 3.2 of the NTC. For thin elements, the procedure described in section 2.3.6 of the NTC is also used.

The procedures in section 3.2 of the reviews, design aids and examples of Complementary Standards for Design and Construction of Steel Structures, DDF (Comment - July 1993) were applied to the determination of the design strength for these states limit.

The effective length for calculating the compressive strength can be provided by using parameters KY, KZ or LY, LZ. If not provided, the entire length of the member is taken into consideration.

Further criteria compression strength, compression members are required to satisfy the constraints that are a function slenderness of the nature of use of the member (main resistance load component, reinforcing members, etc.). While the selection of member checking code and process it checks immediately STAAD slenderness to appropriate members before continuing with other procedures for determining the suitability of a particular member.

4.2.7 Bending Strength Design

In the method of limit state design, the design flexural strength of a member is determined primarily by the limit state of lateral torsional buckling. The inelastic bending is permitted and basic measure of resilience to bending is plastic moment capacity of the section.

The flexural strength is a function of capacity, actual long sideways without support, limiting unbraced length, the time button and the coefficient of plastic bending moment. Limiting laterally unsupported length (Lu) and flexural strength (Mr) are functions of the geometry of the section and are calculated according to the procedure in Section 3.3.2 of the NTC.

The purpose of bending coefficient Cb is to take into account the influence of the time gradient of the lateral torsional buckling. This coefficient may be specified by the user by using CBx or CBy parameter (see Table 11B.1) or can be calculated by the program (as LRDF USA specification) if CB is specified as 0.0. In the absence of CB parameter, a predetermined value of 1.0 is used.

To specify the unbraced length, none of the parameters and UNF UNL (Table 10B.1) can be used. It takes into account the reduction in flexural strength due to the thin web under section 4.5.8 of the NTC

The LRDF USA specification was used for sections where the soul and skate are thin. Areas of stress due to bending about the axis (MY)
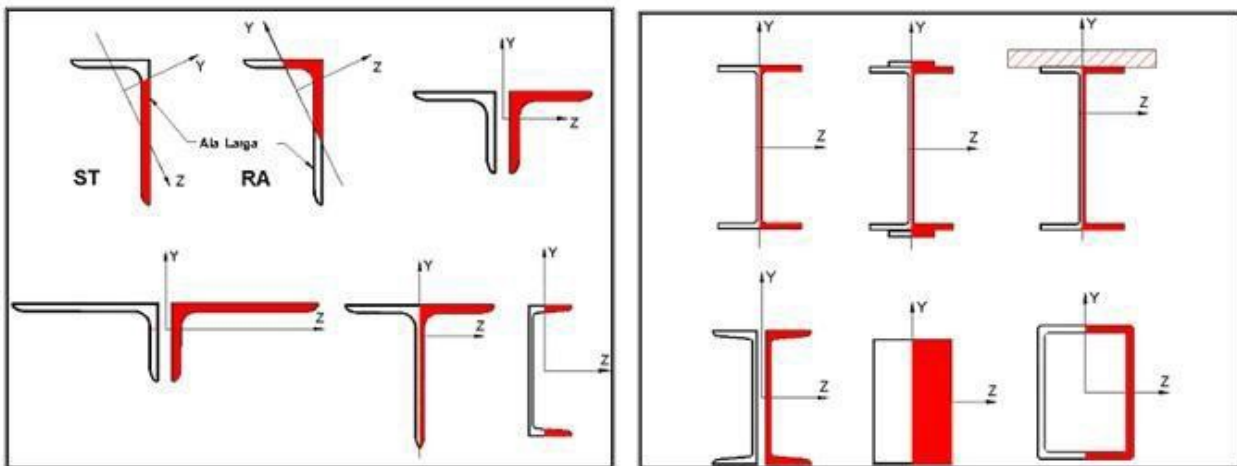
Note: The local X axis goes into the page; Global Y axis is vertically upwards; the shaded area indicates the area under compression; the shaded area indicates not energized

Areas of stress due to bending about the Z axis (MZ)
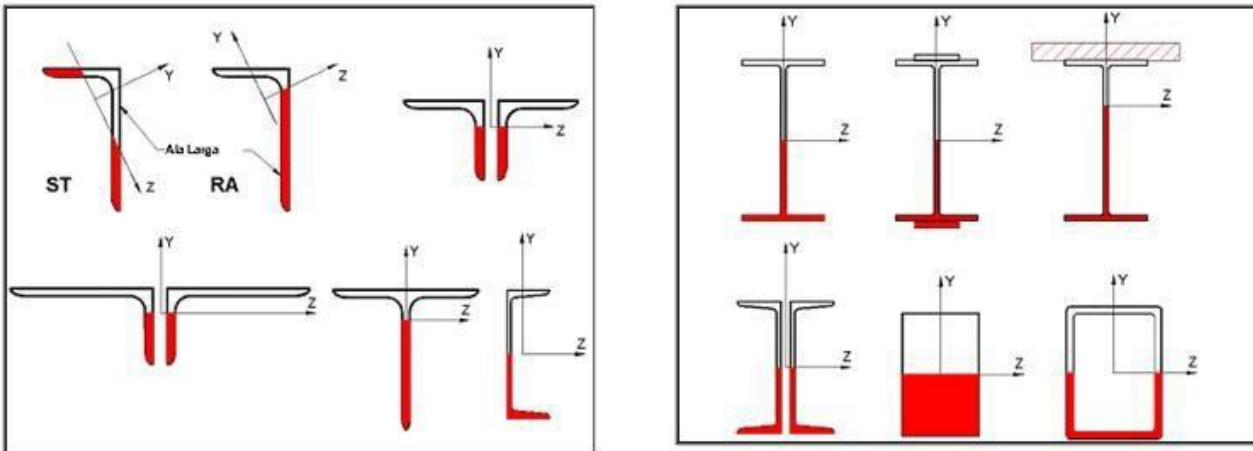
4.2.8 Design shear

The method of Item 3.3.3 Staad NTC used to design the cutting forces members. In addition to the combination of bending and shear are checked according to section 3.3.4 of the NTC, also considering the limits for stiffeners under Sections 4.5.6 / 7 NTC. The cut sections with wide channels skates and is resisted by the web area.

4.2.9 Moment and axial forces combined compression

The interaction of bending and axial forces individually and doubly symmetric shapes is governed by the formulas of section 3.4 of the NTC. These formulas of interaction cover the general case of biaxial flexural strength combined with axial. They are also valid for uniaxial bending and axial force.

It is considered that the frames are part of the structures having shear walls or rigid elements that could be considered the lateral displacements of a flat. The program has included formulas that include structures with lateral displacement in the future by considering the columns B2 individually and not full floor analysis.

It takes into account whether the elements have transverse loads and if the ends are angularly restricted.

4.2.10 Moment and axial forces combined tension

Based on Section 3.5 4 of the NTC.

4.2.11 Design Parameters

This design is sought by Mexican standards using the CODE command. Other applicable parameters are summarized in Table 4.2.11. These parameters communicate design decisions engineer for the program and thus allow the engineer to control the design process to suit the specific needs of an application.

The parameter values were selected default numbers which are frequently used in the conventional design. Depending on the particular design requirements, some or all of these parameter values can be changed to accurately model the physical structure.

The DMAX and DMIN parameters can only be used for the selection of members.

When a parameter is specified, the value remains in specifying the number until specified again. This is the way STAAD works for all codes.

Table 4.2.11-design parameters of steel design according to Mexican standards NTC

Parameter name	Default Value	Description
CODE	–	Must be specified as **MEXICAN**. Design code to follow. See section 5.48.1 of the Technical Reference Manual.
BEAM	0	0: design at the ends and the locations specified by the command of the section. 1: design at the ends and at each point along twelfth member
CB	1	The coefficient Cb defined under section NTC 3.3.2.2. If Cb is 0.0 shall be calculated by the program as LRFD USA (CbMex = 1 / CbUSA). Any other value will be used

CMB	1	The combined forces CMB factor when transverse loads on the limbs. 3.4.3.3.ii section NTC CMB 1.0 = The end members are angularly restricted. CMB = 0.85 The end members are not restricted angularly.
DFF	None (required for verification deflection TRACK 4.0)	"Length of deflection between the maximum local deflection permitted. See Note 1 below.
DJ1	Initial node member	Number of node denoting the starting point for the calculation of "Deflection Length". See Note 1 below.
DJ2	End node of the member	Number of node denoting the end point for the calculation of "Deflection Length". See Note 1 below.
DMAX	114 cm	Maximum allowable depth
DMIN	0.0 cm	Minimum allowable depth
DSD	T	Contents ductile seismic design according to Article 11 (True or False). Main design conditions are considered (excluding, for the moment, geometric)
FU	4,230 Kg / cm2	Ultimate tensile steel Effort

Fyld	2,530 kg / cm2	Minimum yield stress of steel
IMM	0	Primary or secondary member in order to check the slenderness 1. Senior Fellow 2. Secondary or contraventeos
INO	0	Definition of curve according NTC.3.2.2.1a, I defined for profiles or tubes 1. n = 1.4, rolled I-profiles, tubes or constructed with 3 or 4 plaques obtained welded wider cuts oxygen plates. 2. n = 1, I, built with tubes or 3 or 4 plates welded
IRR	0	Variable defined for the whole structure indicating whether it is regular or irregular according to section 3.4 of the NTC. 1. Columns that are part of regular structures 2. Columns belonging to
KX	1.0	Effective length factor torsional bending buckling
KY	1.0	Effective length factor for the local Y axis . Generally the minor axis
KZ	1.0	Effective length factor for the local axis Generally the major axis Z..

LDR	T	Defines if the structure has elements for wind load (shear walls, or rigid elements contraventeos) that restrict lateral movement and allow us to ignore the effects of slenderness. (True or False)
LX	Length element	Length to determine the resistance to torsional bending buckling
LY	Length element	Length to calculate slenderness ratio on the local Y axis.
LZ	Length element	Length to calculate the aspect ratio for the local Z-axis.
NSF	1	Net area factor for tension members
RATIO	1.0	Allowable ratio of the effect of the actual load and design effort
STIFF	Length or depth of the longest element	Spacing reinforcement beams for cutting design.
TRACK	0	Controls the level of detail of the output 0 = Delete all design forces 1 = Print all design forces 2 = Print output expanded design
UNB	Length element	Free length (L) of the lower * skate for calculating flexural strength. They will only be used if the compression is in the bottom flange. See note 2 below.

UNT	Length element	Free length (L) of the top * for calculating flexural skate. They will only be used if the compression is in the top flange. See note 2 below.

4.2.11.1 For verification of deflection parameters DFF, DJ1, DJ2 table and can be used. All requirements remain the same.

4.2.11.2 Top and bottom represent the positive and negative side of the local y-axis (local Z axis Z SET UP if used).

4.2.12 Member Selection Options and verification codes

Both verification code and member selection options are available in the application of Mexican standards STAAD.

See Section 2.5 of the Technical Guide for general information about the verification code. See section 5.44.2 of this manual for details on specifying the verification code check command.

See section 2.6 of the Technical Manual for general information on selecting members. See section 5.44.3 of this manual for details of the specification of the commands for selecting members.

4.2.13 Results Table steel design

The results of the verification code and member selection are presented in a tabular format. CRITICAL COND refers to the section governing NTC Mexican design. If TRACK is set to 1.0, the design forces members are printed.

Section 5

Commands and Input Instructions

___

Introduction

This section of the manual describes in detail various commands and instructions related to STAAD. The user uses a format command language to communicate instructions to the program. Each of these commands provides some information either instructs the program or to perform some calculations using the specified data. Format command language and conventions in Section 5.1 is described. This is followed by a description of the available commands.

Although the STAAD input can be created through the Modeling mode, is very important to understand the language of commands. With the knowledge of this language, it is easy to understand the problem and add or comment data when needed. The general sequence in which the commands must appear in an input file should ideally follow the same sequence presented in this section. Commands are executed in the input sequence. Obviously the data necessary for the proper execution of a command must precede the command (eg printing results after Perform Analysis). Moreover, commands can be provided in any order with the following exceptions.

i)   All data related to the design must be provided after the analysis command.

ii) All cases of load and load combinations together, except where the CHANGE and RESTORE commands are used must be provided. Additional load cases may be provided in the last part of the inlet.

All input data are provided are kept by the program. Data can be added, deleted or modified within existing data file.

5.1   Command Language Conventions

This section describes the language of the commands used in STAAD. First, the various elements of the language are discussed, then the command format is described in detail.

5.1.1   Elements of Commands

a)   **Whole Numbers:** Whole numbers are numbers written without a decimal point.

These numbers are designated as i1, i2, etc., and can not contain any point decimal. Signs (+ or -) are allowed in front of the numbers. If the sign is omitted, it is assumed to be (+).

b)   **Floating Point Numbers: These are real numbers that can contain a decimal portion. These numbers are designated as f1, f2 ... etc. .. The values can have a decimal point and / or exponent. When specifying numbers with magnitude less than 1/100, you should use the format E to avoid errors related to the precision limit of these are 16 characters.**

Example

```
5055.32   0.73   -8.9   732
5E3   -3.4E-6
etc.
```

When the sign is omitted, is assumed to be positive (+). Also note that the decimal point can be omitted if the decimal portion of the number is zero.

c)   **Alphanumeric: These are characters that are used to construct the names of the data, titles or commands. They alphabetic characters can be entered in uppercase or lowercase. No quotation marks to frame is needed.**

Example

```
MEMBER PROPERTIES
TABLE 1 TO 8 ST
W8X35
```

d)   **Repetitive data:** Repetitive numerical data can be provided using the following format:

$$n * f$$

Where   n = the number of times the data has to be
         repeated f = numeric data, integer and floating
         point

Example

```
JOINT COORDINATES
1 3 * 0.
```

This specification of the coordinate of the board is the same as:

```
1 0 0 0.
```

5.1.2 Command Formats

a)   **Free Entry Formats: All data entered STAAD are free-form style. The input data must be separated by whitespace other input data. Quotation marks are not required to remove any alphabetic word as data, commands or titles. An input limited to 16 characters.**

b)   **Comments Entry: To document a data file STAAD, the option to provide feedback is available. Comments can be included providing an asterisk (*) marks it as the first character in any line blank. The line with the comment echoes the output file but has not been processed by the program.**

Example

```
JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT FY Load 2
March 7 35.0
etc.
```

c) **Meaning of Emphasis in Manual: The exact format of the commands are described in the last part of this section. They can abbreviate many words in the commands and data. The whole word is given for the purpose of description of the command, with the required portion (abbreviation) underlined.**

For example, if the word is used in a MEMBER command, only the necessary portion enters MEMB. Although it is more clear for others who are reading the results if the word is used whole, however an experienced user may wish to use the abbreviations.

d) **Meaning of Brackets and Parentheses: Some formats of commands, brackets enclose a number of options, which are placed vertically. They can select one and only one of the options. However, some of the listed options may be selected if an asterisk (*) is located outside the brackets.**

Example

$$\begin{Bmatrix} \text{XY} \\ \text{YZ} \\ \text{XZ} \end{Bmatrix}$$

In the example above, the user must make a choice between XY or XZ or YZ.

Example

$$* \begin{Bmatrix} \text{FX} \\ \text{FY} \\ \text{FZ} \end{Bmatrix}$$

Here the user can choose one or all of the registration (FX, FY and FZ) in any order. The parentheses () indicates that the enclosed portion of the command is optional. The presence or absence of this portion affects the meaning of the command, as explained in the description of the particular command.

Example

```
PRINT (MEMBER) FORCES
PERFORM ANALYSIS (PRINT LOAD DATA)
```

In the first line, the word MEMBER can be omitted without changing the meaning of the command. In the second line,

LOAD DATA PRINT

The command can also be omitted in which case the load data will not be printed.

e) **Multiple Data Separator: Multiple data can be provided on a single line, if they are separated by the semicolon (;). One restriction is that a semicolon can not separate the consecutive commands. They should appear on separate lines.**

Example

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4
etc.
Possible Error:
PRINT FORCES; PRINT STRESSES
```

In the above case, only the PRINT command is processed FORCES and STRESSES PRINT command is ignored.

f) **Displaying the Data: In some descriptions STAAD commands, the word "list" is used to identify a list of meetings, members / elements or load cases. The format of a list can be defined as follows:**

$$\text{list} = \begin{cases} * & i_1, i_2, i_3....... \\ & i_1 \underline{\text{TO}} \, i_2 (\text{BY} \, i_3) \\ & X \text{ or } Y \text{ or } Z \end{cases}$$

Where TO means all integers from the first (i1) to the second (i2) inclusively. BY means that the numbers are increased by an amount equal to the third item (i3). BY i3 If omitted, the increase will be considered one. Sometimes the list may be too long to fit on one line in this case the list can be continued on the next line by providing a hyphen preceded by a blank. Also note, that this applies only to continue a list and can not be used for any other data

Instead of a number list, the X (or Y or Z) specification can be used. This specification will include all members are parallel to the specified global address. Note that this does not apply to joints or elements.

ALL, BEAM, PLATE, SOLID. Do not use them unless the documentation for the command specifically mentioned as available for that command. ALL means all members and elements, BEAM means all members, etc.

g) **Continuing a command on the next line: They can only be continued to the next line lists the finishing line with a space and hyphen (see above) with the few exceptions: multilinear spring supports, the supports, Master / Slave. Others who have special types of continuations. Please follow the descriptions of the command.**

Example

```
2 4 7 TO 13 TO 22 BY 19 February -
31 TO 33 FX 28 10.0
This list of items is the same as:
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
Possible Error:
3 5 TO 11
September 15 -
FX 10.0
```

In this case, the mark below for list data is used when the list of data is not continuous. This will produce an error message or possibly unpredictable results.

5.1.3 Listing of Meetings / Members / Elements by Specification Global Ranks

This command allows the user to specify lists together / members / elements providing global ranges. The general format of the specification is as follows.

General Format:

$$
\begin{Bmatrix} \underline{XR}ANGE \\ \underline{YR}ANGE \\ F_2\underline{ZR}ANGE \end{Bmatrix} \quad f_1,
$$

Where,
Xrange, yrange, ZRANGE = address range (parallel to the global X, Y, and Z respectively)

f1, f2 = values (in the current unit system) that defines the specified range

Notes

1. Only one address range (xrange, yrange etc.) is allowed per list. (Except: Loading Area / Floor and Master / Slave).

2. They can not be other data in the list.

3. The values that define the range (f1, f2) must be in the current drive system

Example

```
Xrange TRUSS
MEMBER
CONSTANTS 20 70
E STEEL yrange 10 55.
```

In the above example, a XRange specified with values 20 and 70, this range will include all members that are completely inside parallel to the overall axis X range bounded by X = 20 and X = 70.

5.2 Initiation of the Problem and Title

Purpose

The initial run commands in the STAAD allows the user to specify the type of structure and an optional title.

General Format:

$$STAAD \quad \begin{Bmatrix} \underline{PLA}NE \\ SPACE \\ to_1) \ TRUSS \\ \underline{FLO}OR \end{Bmatrix} \quad (Any \ title$$

Description

Any input STAAD must begin with the word STAAD. The following type specifications are available:

PLANE = Frame structure Plano
SPACE = Space Frame Structure
TRUSS = armor type Flat Structure or Space
FLOOR Floor type = structure

a1 = Any securities to the problem. This title will appear at the top of each page of results. To include additional information in the header of the page, use a line containing the relevant information as the second input line.

Notes

1)   The user should be careful to choose the type of structure. The choice is dependent on the number of degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedom considered in various type specifications. Detailed discussions are available in Section 1.3.

STRUCTURE TYPES

PLANE

SPACE

2D 3D TRUSS

FLOOR
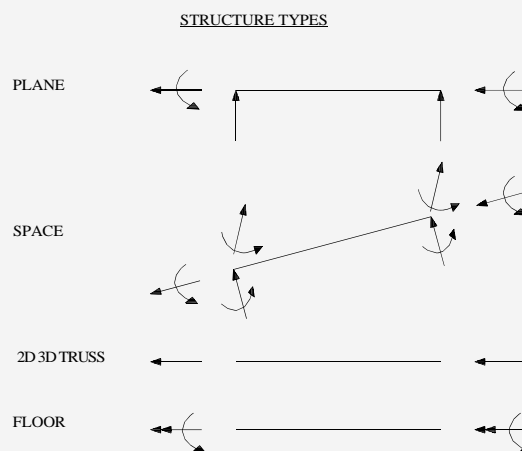
2) Optional user-supplied title is printed on the top of each page of output. The user can use this option to customize your results.

Limits

1) Number of Meetings: 1-999999
2) Number of meetings: 100000 *
3) Membership number / Elements: 1-999999
4) Number of members and Elements 100000 *
5) Load Case Number: 1-99999
6) Number of primary cases and combinations: 1000

STAAD * Some copies are available with much smaller limits, please verify what limits you have purchased.

5.3 Specifying Units

Purpose

This command allows the user to specify or change length and force units for input and results.

General Format:

```
            *   force-
   UNIT
               ⌈length-unit ⌉
               {            }
               ⌊unit        ⌋


                   ⌈INCHES    ⌉
                   |FEET or FT |
                   |Cm        |
   length-unit =   {METER     }
                   |MMS       |
                   |DME       |
                   ⌊KM        ⌋


                   ⌈KIP    ⌉
                   |POUND  |
                   |KG     |
   force-unit =    {MTON   }
                   |NEWTON |
                   |KNS    |
                   |MNS    |
                   ⌊DNS    ⌋
```

> **Note: DME denotes Decimeters. MNS denotes the Mega Newtons (1000 Newtons) and DNS denotes decaNewtons (10 Newtons). MTON denotes Metric ton (1000 kg). All other units are self explanatory.**

Description

The UNIT command can be specified any number of times during a run. It is assumed that all data is contained in the latest specification of the unit that precedes data. Also note that the input unit for angles is always degrees. However, the result unit for rotations of the board (the displacement of the joint) are radians. For all results, the units are clearly specified by the program.

Example

```
UNIT UNIT INCH
KIP FT
KNS METER UNIT
UNIT CM MTON
```

> **Notes: This command can be used as often as necessary to specify the data or generate the results in the desired length and / or units of force. Note that the mixture and tie between systems of different units (Imperial, Metric, SI etc.) are allowed.**
>
> This command can be anywhere as a primary command (eg, JOINT COORD, MEMBER INCIDENCE, etc.). Furthermore, it can also be a first level wherever load command is
>
> Exceptions: The second command level load (MEMBER LOAD, LOAD and FLOOR JOINT LOAD), allowed to enter the UNIT command on a separate line, not in a continuation or between the semicolons.

5.4 Specifying Width Input / Output

Purpose

These commands can be used to specify the width (s) of lines of file (s) output.

General Format:

$$
\begin{Bmatrix} \underline{\text{INPUT}} \\ \underline{\text{OUT}}\text{PUT} \end{Bmatrix} \quad \underline{\text{WID}}\text{TH} \quad i_1
$$

OUTPUT WIDTH,
i_1 = 72 or 118 depending on how narrow or wide departure.

Description

The user can specify the required width of the input / output when using this command. Width for INPUT is used almost always 79 The program can create outputs using two different widths of 72 results (the default) and 118 width of 72 characters can be used for deployment on most CRTs and print a "portrait" (vertical) on wide paper. The width of 118 characters can be used to print the "landscape" (horizontal) paper width.

Note: This is a custom option that can be used to improve the quality of presentation of the documents of the run.

5.5   September Command Specification

Purpose

This command allows the user to sort several general specifications for the run of the analysis / design.

General Format:

$$
\text{SET}
\begin{cases}
\underline{\text{NL}} & i_1 \\
\underline{\text{DISPLACEMENT}} & \\
& i_2 \\
\underline{\text{SDAMP}} & i_3 \\
\underline{\text{WARP}} & i_4 \\
\underline{\text{ITERLIM}} & i_5 \\
\underline{\text{PRINT}} & i_7 \\
\underline{\text{SHEAR}} & \\
\underline{\text{ECHO}} \begin{cases} \underline{\text{ON}} \\ \underline{\text{OFF}} \end{cases} & \\
\underline{\text{GUI}} & i_6 \\
\underline{\text{Z}} & \underline{\text{UP}}
\end{cases}
$$

Where,

i1 = The maximum number of elementary charges (NL)

i2 = The maximum acceptable tolerance for any joint displacement in the structure.

i3 = The ratio of buffer to be used for the calculations of all the levers of modal damping compound in dynamics.

i4 = The ratio of the roll to be restricted to members used in section I calculating the torsional rigidity. If it is 0.0 then there will be no restriction of bank, normal default option ,; If it is 1.0 then the restriction of roll is complete. Cw, the constant twist will be calculated and used in the calculation of torsional rigidity. Values between 0.0 and 1.0 will produce a partial warping restraint.

i5 = The maximum number of iterations of tension / compression.

i6 = 1 Deviation forming data and files needed for post-processing. i7 = 1, which

prints messages Deviation Zero Stiffness.

Description

SET NL command is used in a multiple analysis run if the user wants to add more cases of primary load after an analysis has been performed. Specifically, for those examples, which use the RESTORE or CHANGE command, if the user wants to add more cases of primary load, the value of NL should be the maximum number with the command SET NL. The program can then put the additional memory space next to the information to be added later. Note that this command must be given before any board member or load specifications. The value for i1 should not be greater than the maximum number of cases of primary charging.

For PDELTA ANALYSIS with option CONVERGE DISPLACEMENT SET command is used to specify the tolerance of convergence. If the RMS displacement of two consecutive iterations change less than the entered value, then that load case converges. This command must be placed before specification JOINT COORDINATE.

For NONLINEAR ANALYSIS SET DISPLACEMENT command is used to specify a limit value of displacement. If any movement after any iteration exceeds the value entered here, then the solution is diverging and ends. This command must be placed before specification JOINT COORDINATE.

The SET ECHO ON command activated and the SET ECHO OFF command will disable operation commands repeat (echo) input in the output file. In the absence of the SET ECHO command, the command repeat return input file to the output file.

By definition, the Y axis is the vertical axis. However, the SET Z UP command can be used to model situations where the Z axis represents the vertical axis (the direction of gravity load) of the structure. This can be built if the geometry of the entry is created through some CAD software. Note that this command will affect the specification of predefined angle BETA. However, the BETA can be set to a certain value for all parallel to a particular global axis using the MEMBER X (or Y or Z) type in the list members. For additional information, see the specification of the constants (Section 5.26).

Notes for SET Z UP

The SET Z UP command directly it influences the values of the following entries:

1) JOINT COORDINATE
2) The input for the command PERFORM ROTATION
3) BETA ANGLE

The following characteristics of STAAD can not be used with the SET Z UP command

1) Generation of Wind Loads
2) Generation Cargo Floor
3) Automatic Generator Mounts Springs for Slabs Foundations
4) Loads of UBC

The SET command SDAMP allow the damping spring can be considered in the calculation of the compound for each mode in a dynamic modal damping solution. This command is not used unless the proportions of CDAMP for members and elements in the command CONSTANTS also mess. Damping

compound is generally only used if there are many ways in the dynamic solution and there is a wide range of damping ratios for spring / members / elements.

The WARP SET command will allow the member end section warping that I restricted to be considered when calculating the torsional rigidity stiffened Total or partial or no restrictions warping is not allowed.

ITERLIM SET command is to increase the maximum limit of the iteration on the default of 10 iterations in tension / compression value. Since this iterative procedure does not necessarily converge, this option can not help more iterations and should be used with care.

After any analysis of tension / compression in the output file (. ANL) should be considered non-convergence warnings. Do not use the results of cases not converged.

The SET PRINT command 1 is to delete messages Zero Stiffness.

The command SET SHEAR is to omit the additional conditions to form shear distortion rigidities member-beam. With this command you can match exactly the results of the theory of beams textbooks.

Other rarely used SET command:

SET DATA CHECK	IGNORED
SET RUN	IGNORED
SET COMPRESS	FILE COMPRESSION OFF
SUN SETUTION INCORE	Use the search for determining the frequencies for small problems
SUN SETShow	be ignored
SET CORE	be ignored (Report)
SET EXM	be ignored (EXTENDED MEMORY)
SET NJ	be ignored
SET NM	be ignored
SET CONNECTIVITY	be ignored
MAS SETS	= 1 uses the generator times like masses.
SET MODAL	be ignored
SET THISTORY	= 2 Use forces exact integration over time.
INT SETERPOLATION	Lin or Log for spectra
SET DISPLACEMENT METHOD	be ignored
SET BUBBLE	Fns = 1 does not use bubbles in solids
SET NOSECT	no results sections will be calculated
SET TMH	be ignored

5.6 TheCommand Separator

Purpose

This command can be used to specify the desired separator character that can be used to separate multiple lines of data in a single input line.

General Format:

<u>SEP</u>Arator   **to**₁

See Section 5.1.2

Description

The semicolon (;) is the default character that serves as the separator for multiple data on a line. However, this can change separator character by a1 SEPARATOR command any character except comma or asterisk.

Note: The Coma (,) or asterisk (*) should not be used as a separator character.

5.7 Page New Command

Purpose

This command can be used to instruct the program to start a new output page.

General Format:

<u>P AG</u>E N EW

Description

With this command, a new page of results can begin. This command provides the flexibility, the user needs to design the format of the results.

Note: The quality of the submission of the results can be improved using this command properly.

5.8 Command Page Length / Eject

Purpose

These commands can be used to specify the length of the search results page and the desired character page eject.

General Format:

$$\text{PAGE} \quad \begin{cases} \text{LENGTH} \quad i \\ \text{SHAFTCT} \quad to_1 \end{cases}$$

The length of the page in Staad result is based on a default value of 60 lines. However, the user can change the page length to any number i (the number of lines per page) desired

Description

The character expels the normal page (CNTRL L for PCs and 1 for Mini / MFRM) is included in the program STAAD. The PAGE EJECT command to the input of character a1 eject eject alter the default page in the program. A blank character deleted ejection page.

5.9 Specifications Ignore

Purpose

This command allows the user to provide lists of members in a convenient manner without activating appropriate error messages to members nonexistent numbers.

General Format:

IGNORE L I ST

Description

LIST ignor be used if the user wants the program to ignore any absent member could be included in a specification list member. For example, for the sake of simplicity, a list of members can be specified as MEMB 3 A 40 where members 10 and 11 do not exist. An error message in this situation can be avoided by providing ignor anywhere LIST command at the beginning of the entry. A message of warning, otherwise nonexistent appear for each member.

5.10 Design Specification No

Purpose

This command allows the user to declare that no design operation will be performed during the run. The memory reserved for the design will be released to accommodate larger work analysis.

General Format:

INPUT NODESIGN

Description

STAAD always assumed that at some point in the input, the user may wish to make the design for steel or concrete members. These design processes require more computer memory. If memory availability is a problem, the above command can be used to remove the extra memory requirements.

5.11   Specification Joint Coordinates

Purpose

These commands allow the user to specify and generate the coordinates of the joints of the structure. The JOINT COORDINATES command starts the coordinates specification. The REPEAT ALL and REPEAT commands allow easy generation of coordinates using repetitive patterns.

General Format:

<u>JOI</u>NT   <u>COO</u>RDINATES   (CYLINDRICAL   (R EVERSE))   (NOT CHECK) **band-spec**

i_1, X_1, And$_1$, Z_1, (I_2, X_2, And$_2$, Z_2, I_3)
<u>R</u>EPEAT   n, xi_1, Yi_1, Zi_1(Xi_2, Yi_2, Zi_2, ..., Xi_n, Yi_n, Zi_n)
<u>R</u>EPEAT ALL   n, xi_1, Yi_1, Zi_1(Xi_2, Yi_2, Zi_2, ..., Xi_n, Yi_n, Zi_n)

<u>JT O</u>RIG   xOrigin   yOrigin   zOrigin
band-spec = (NOREDUCE BAND)

NOCHECK =   No place check for multiple structures or orphaned seals.

Description

The JOINT COORDINATES command specifies a Cartesian coordinate system (see Figure 1.2). Boards and are defined using the X, Y Z Global. The CYLINDRICAL JOINT COORDINATES command specifies a Cylindrical Coordinate System (see Figure

See Section 1.5.1

1.3). Meetings are defined using the r, θ and z coordinates. The JOINT command CYLINDRICAL COORDINATES REVERSE specifies a system Cylindrical Coordinates Reverse (see Figure 1.4). Meetings are defined using the coordinates r, y.θ

JTORIG causes the program to use a different origin (0, 0, 0) for all inputs together with this JOINT COORDINATES command. In JTORIG command must be entered on a separate line. Basically after the joint coordinates are entered or generated, then xOrigin, yOrigin and zOrigin values are added to the coordinates. For example a cylinder would then be generated on the Y axis driven by this command to its proper place. To create multiple structural parts out of the joints, the JOINT COORDINATES, each with a command JTORIG command goes further.

Note that the concept of multiple JOINT COORDINATES command allows you to change the ROTATION UNIDADADES and PERFORM (PERFORM ROTATION) between them; thereby

these commands are applied to a selected portion of the joints. However, the PERFORM ROTATION command applies to all predefined together, not only those in the previous COORDINATE JOINT command.

BAND NOREDUCE causes the program to run without making a reduction bandwidth.

Example

<div style="background-color:#d0d0d0">

JOINT COORDINATES NOREDUCE BAND

</div>

The REPEAT command causes the previous input line is repeated 'n' number of times with increases in the specified coordinates. The REPEAT ALL command functions are similar to the REPEAT command repeats only the most recent entries previously specified before pre REPEAT ALL command, or all data if no board REPEAT All previous command has been given. Note that using the "REPEAT ALL 0" command will start a data section to be repeated if necessary. (When using the REPEAT ALL and REPEAT commands, together numbering must be consecutive and must start with 1)

- $i1$ = The number of the board in which the coordinates are provided. Any integer within the limits (see Section 5.2 for the limit) are allowed.

- $x1$, $y1$ and $z1$ = X, Y and Z (R, θ and Z for cylindrical or R, Y and θ for cylindrical inverse) of the board.

For analysis PLANAR $z1$ is an optional input data for defining the individual joints. The $z1$ is always required for the generation of the joint. The following is only used if the seals are to be generated.

- $i2$ = The number of the second meeting at which the board coordinates are generated.

- $x2$, $y2$, and $z2$ = X, Y and Z (R, θ and Z for cylindrical or R, Y and θ for reverse cylindrical) the coordinates of the $i2$ board.

- $i3$ = number increased by the board which together generated increase. The default is 1 if omitted.

- n = The number of times that will be carried out. Note that "n" can not exceed 989 in a single one of the REPEAT command.

- x_{ik}, y_{ik} and z_{ik} = X, Y and Z (R, θ and Z [R, Y]) increases the coordinate for the k th replicate.

The X, Y and Z (R, and Z [R, Y]) coordinates are spaced equally between $i1$ and $i2$.

* Note that the REPEAT command uses the highest number of board entered so far plus one for the generation of the intermediate board

Example 1

```
JOINT COORDINATES
1 10.5 2.0 8.5
2 0.0 0.0 0.0
3 5.25 0.0 8.5 6 50.25 0.0 8.5
```

In this example, the XY coordinates of the joints and Z 1 are provided seals 6 Note that 3 to 6 will be generated with equally spaced joints 3 to 6 Hence, the ring 4 will have coordinates 20.25 8.5 and 0.0 will coordinate joint 5 0.0 8.5 35.25.

Example 2

```
JOINT COORDINATES
1 0.0 0.0 0.0 4 45 0.0 0.0
REPEAT 4 0.0 0.0 15.0
REPEAT ALL 10 0.0 10.0 0.0
```

Here, the coordinates of 220 joints of a structure 10 levels of 3 X 4 clear generated. The REPEAT command repeats the first input line 4 times, while increasing each coordinate Z 15 Thus, the first 2 lines are enough to create a "floor" of twenty together.

```
1 0 0 0; 2 15 0 0; 3 30 0 0; 4 45 0 0.
5 0 0 15; 6 15 0 15; 7 30 0 15; 8 45 0 15.
.....       .....      .....     .....
17 0 0 60; 18 15 0 60; 19 30 0 60; 20 45 0 60.
```

The REPEAT ALL command repeats all previous data (ie 20avo "floor" of the board) ten times, increasing the Y by 10 each time. This creates the remaining 200 seals the structure:

Example 3

```
21 0.0 10.0 0.0        ; 22 15.0 10.0 0.0; ...;

40 45.0 10.0 60.0; 41 0.0 20.0 0.0; ...;
200 45.0 90.0 60.0; 201 0.0 100.0 0.0; ...;
219 30.0 100.0 60.0; 220 45.0 100.0 60.0
```

The following examples show various applications of the REPEAT command.

```
10 5 10 5 REPEAT.
```

The REPEAT above command repeats the last input line 10 times using the same type of increments (ie x = 5, y = 10, z = 5)

> **REPEAT 3 2 10 5 3 15 3 5 20 3.**

The REPEAT command above will repeat the last input line three times. Each use a different repetition rate increases.

> **REPEAT 10 0 12 0 15 0 0 10 0 * 9 * 0**

The above REPEAT command repeats the last line of the input 10 times; six times using x, y and z increases from 0, 12 and 0, and four times using increments of 0, 10 and 0 every x, y and z values of 0 represents has not been changed the previous increase. To create the second to the sixth repeat, five sets of 0, 0 and 0 (15 * 0) are provided. The seventh repeat blocks increases in 0, 10 and 0 from the eighth to the tenth repeat are made with the same increases as the 7th, and is represented as 9 * 0

Note: JOINT COORDINATE The PRINT command can be used to verify the coordinates of the board proportionate or can be generated by the REPEAT ALL command. Also, use the Post Processing option to verify geometry graphically.

5.12 Specifying Member Incidences

Purpose

This set of commands is used to specify a member defining the connection between the boards. The REPEAT ALL and REPEAT commands are available to facilitate the generation of repetitive patterns.

Incidences member / element must be defined so that the developed model represents a single structure, not two or more separate structures. STAAD is able to automatically detect multiple structures.

General Format:

> M EMI BER NCIDENCES
> i_1, I_2, I_3, (I_4, I_5, I_6)
> **REPEAT** n, m_i,
> J_i**REPEAT ALL** n, m_i,
> J_i

Description

The REPEAT command causes the previous input line is repeated 'n' number of times with increases in members and specified together. The REPEAT ALL command functions are similar to the REPEAT command only repeating all previously specified entries back to the most recent REPEAT ALL command or at the beginning of the specification if no previous REPEAT ALL command was issued. (When REPEAT ALL and REPEAT commands are used, members must be numbered consecutively).

See Section 1.5.2

- i1 = member number to which incidents are maintained. Any whole number (up to six digits) are allowed.

- i2 = number of kickoff meeting.
- i3 = number of end seal.

Note: Use "REPEAT ALL o" to begin with a set of members to be repeated if you do not want to repeat back the last REPEAT ALL.

The following data are only used for the generation of membership:

- i4 = Second member number which members are generated. i5 =
- Increased number of member for generation.
- i6 = Increase in number of board to be added to the incidents together. (The i5 and i6 will default to 1 if omitted.)
- n = The number of times to repeat.
- mi = Increased number of member
- ji = The increase in the number of board

Example

```
MEMBER INCIDENCES
1 1 2
May 2 May 7
7 11 13 13 2 3
```

In this example, the member of the board 1 is 1 to 2 member 2 is connected between joints 5 to 7 the number of member 3-5 is created with increased membership number 1 and a number of board increased by 1 (default). That is, member 3 goes from 6-8, member 4 7-9, member 5 8 10 Similarly, in the next line, the Member 9

will be from 14 to 16, 11 and 13 17 to 19

20 to 22.

Additional Examples

```
MEMBER INCIDENCES
1 January 21 20
21 21 22 23
REPEAT 4 3 4
36 21 25 39
REPEAT 3 4 4
REPEAT ALL 20 September 51
```

This example creates the 510 members of a structure of ten levels, with clear 3 X 4 (this is a continuation of the example started in Section 5.12). The first line of the input creates twenty columns on the first floor:

```
January 1, 21; February 2, 22; March 3, 23; ...; 19 19 39; 20 20 40
```

The two commands (21 21 22 23 4 3 4 and REPEAT) 15 members who are creating the second floor, the joists run "floor", for example, in the east-west:

```
21  21  22; 22   22  23;  23  23  24
24  25  26; 25   26  27;  26  27  28
...  ...  ...
33  37  38; 34   38  39;  35  39  40
```

The next two commands (36 21 25 39 3 4 4 and REPEAT) operate similarly to the previous two commands, but here creates the sixteenth floor beam "floor" running in the north-south direction:

```
36  21  25; 37 22   26; 38  23  27;  39  24  28
40  25  29; 41 26   30; 42  27  31;  43  28  32
...  ...  ...  ...
48  33  37; 49 34   38; 50  35  39;  51  36  40
```

The preceding commands have created a unit floor beams and columns, a total of 51 members. The REPEAT ALL this unit now repeated nine times, generating 459 new members and ends ten-story structure. The number of members is increased by 51 (the number of members in a repeating unit) and the number of the board is increased by 20 (the number of joints in a floor).

Note: MEMBER INFO PRINT command can be used to verify the impact of the member provided or generated by REPEAT ALL and REPEAT commands. Also, use the Post Processor to verify geometry graphically

5.13 Incidence Element Specification

Purpose

This set of commands is used to specify the ELEMENT defining connectivity between the boards. The REPEAT ALL and REPEAT commands are available to facilitate the generation of repetitive patterns.

The incidences of the element must be defined so that the developed model represents only one structure, not two or more separate structures. STAAD is able to automatically detect multiple structures.

Plate / Shell Element Incidences

General Format

ELEMENT   INCIDENCES (SHELL)

i_1, I_2, I_3, I_4, (I_5),   (TO i_6, I_7, I_8)
REPEAT   n, and_i, J_i
REPEAT ALL   n, and_i, J_i

General Format

The incidences of the element for solid elements will be identified using the term to distinguish SOLID PLATE / SHELL elements.

```
ELEMENT INCIDENCES SOLID

i₁, I₂, I₃, I₄, I₅, I₆, I₇, I₈, I₉(TO        i₁₀, I₁₁, I₁₂)

REPEAT n, eᵢ, JᵢREPEAT
        ALL  n, andᵢ,
Jᵢ
```

Description

SOLID ELEMENT INCIDENCES should be provided immediately after INCIDENCES MEMBER (if any) may be specified as well as following SHELL ELEMENT INCIDENCES (if any).

- i1 = Number of Element
- i2 ... i9 = Numbers board solid element
- i10 = Last number of the element to be generated i11
- = Increasing the number of item
- i12 = Increased number of joint
- n = The number of times the REPEAT will be carried out
- ei = Increasing the number of item
- ji = The increase in the number of board

Specify the four nodes either side of the solid element in the opposite direction clockwise as seen from the outside of the element and then goes to the opposite side and the four nodes specified that face in the same direction used to specify the nodes of the first side. The opposite side must be behind the first side, defined by the right hand rule, ie the points of the other side (back) to the first (front) face of which point to the observer.

Use "REPEAT ALL o" to start a solid game that will be repeated if you do not want to repeat back to the last REPEAT ALL.

Example

```
INCIDENCE SOLID ELEMENT
1  1   5   6   2   21  25  26  22  TO  3
4  21  25  26  22  41  45  46  42  TO  6
```

5.14 Element Mesh Generation

Purpose

This set of commands is used to generate the finite element mesh. The procedure involves the definition of super-elements are then divided into smaller elements.

Description

This is the second method for generating the element incidents. If the user needs to divide a large element in several small items, you can use this option to generate the numbers and the coordinates of the board, the number and the impact of the element automatically. The use of this feature consists in two parts:

1. The definition of the limit points of super-element: A super-element can be defined either for 4 or 8 boundary points (see Figure on next page). A boundary point is denoted by a single letter of the alphabet (AZ az upper or lowercase) and their corresponding coordinates. Hence, it can be used in any 4 or 8 to 52 characters to define the boundary of the super-element. If 4 points are used to define the super-element, each side of the super-element is assumed that you have a straight edge connecting the 2 points defining that side. When 8 dots are used, each side is a smooth curve connecting the three points that define that side.

2. The generation of sub-elements: define the use of limit points of the super-element (4 or 8 as explained above) and specifies the total number of sub-elements required.

General Format:

```
DEFINE  MESH      ⎡CYL   ⎤
Aᵢxᵢandᵢzᵢ     ( ⎨       ⎬  (X_or, And_or, Z_or))
. . .             ⎣RCYL  ⎦
Aⱼxⱼandⱼzⱼ

                     ⎡(QUADRILATERAL)   ⎤
GENERATE  ELEMENT    ⎨                  ⎬
                     ⎣TRIANGLE          ⎦


MESH    AᵢAⱼN .....₁(N₂)
MESH    AₘAₙN .....₃(N₄)
..... .....
```
$A_i x_i$ and$_i z_i$

$A_j x_j$ and$_j z_j$

$\text{MESH} \quad A_i A_j N \dots_1 (N_2)$

$\text{MESH} \quad A_m A_n N \dots_3 (N_4)$

    (Above 20 MESH input lines)

Where

Ai, Aj = Alphabet letters A - Z or alphabets a - z. This is maximum 52.
xi, yi, zi = The coordinates for the boundary point Ai. If RCYL CYL or defined, the above coordinates will in the cylindrical system of coordinates or reverse. Optional coordinates xo, me and zo be the Cartesian coordinates for the origin of the cylindrical coordinates. The values will default to 0, 0, 0, if not provided. The 3 fields (x, y, z) can be replaced by a number of the board whose coordinates are defined in the JOINT COORDINATE command to the input

Ai JOINT jn instead of:

  Ai, Aj, Ak ... = A rectangular super-element defined by four or eight breakpoints.
  n1 = The number of elements along the side Ai Aj of the super-elements. (Do not exceed 20).

n2 = number of elements along the side of Aj Ak super-elements. (Do not exceed 20). If n2 is omitted, provided only n1, n1 then indicate the total number of elements within the super-element. In this case, n1 must be the square of an integer.

Limits

There is a limit of 20 commands Mesh. Up to 523 boards can be generated and up to 700 elements. The total number of joints in the model after this command is completed can not exceed 20,000.

Notes

All coordinates are in the system of the current drive. While using this option the user must have the following points:

1) All super-elements must be 4 or 8 knots. The generated elements to 4 knots retain the super-elements-straight line on the sides of the super-elements, while joints were generated elements 8 knots be super-element in a curved path.
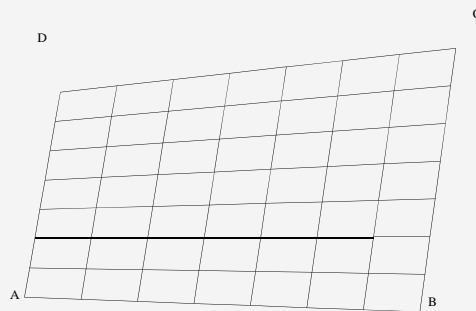
Mesh generated for a super-element 4 knots
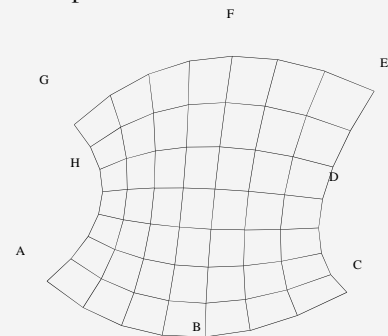
Mesh generated for a super-element 8 knots

2) Two super-elements, which have a common boundary must have the same number of elements along their common boundary.

3) The sequence of commands MESH super-elements define the super-elements. The sequence of this MESH command must be such that once one is defined, the next super-elements must be connected to it. Therefore, for convenience, the first super-element must be one that is connected to the largest number of super-elements. In the example shown here to the tank, the super-element's background is specified first.

4) This command should be used after the MEMBER INCIDENCE and INCIDENCE ELEMENT section before MEMBER ELEMENT PROPERTIES and PROPERTIES section. Elements that are created internally sequentially with an increment of one that starts the last member / element numbered numbered. Similarly additional homegrown sequentially together with an increase of one who begins the latest issue of the board plus one numbered. It is recommended that users keep the numbering of joints and member / element in a sequence with an increment of one starting one.

5) If members surrounding a super-element being entangled, users will have to take care of the additions / modifications required in section MEMBER INCIDENCE as more new gaskets may appear in the existing common boundary as a result of the super-tangle element. See the following figure:
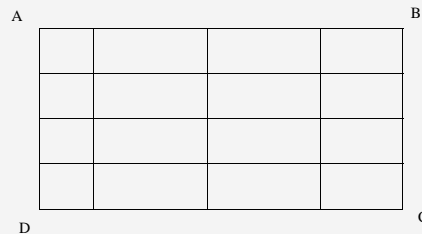
Note: If a member exists between points A and B, you must cut it into 4 parts member. Members are not snared automatically.

6) The sub-elements will have the same direction (in the direction of clockwise or counter clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. is in the sense of clockwise direction, or DCBA CBad etc. is in reverse clockwise direction smarter. If the super-special element is denoted as ABCD, all the sub-elements in it have the sense of clockwise incidence element in this example.

7) The incidences of the element of the sub-elements generated can be obtained by providing the 'PRINT ELEMENT INFORMATION' command after 'MESH ...' in the input file command.

8) If the input file contains commands for STAAD JOINT COORDINATES, MEMBER INCIDENCES, ELEMENT MESH GENERATION INCIDENCES and should specified in the following order:

> STAAD SPACE
> UNIT. . .
> JOINT COORDINATES ...
> MEMBER INCIDENCES ...
> INCIDENCES ... DEFINE
> ELEMENT MESH ...
> GENERATE ELEMENT ...

Example

The following input section illustrates the use of the MESH GENERATION option, the user can compare this example with the entries in the geometry of Prob # 10 in the manual for examples of STAAD.:
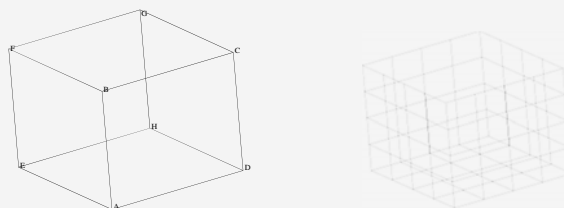
```
          STAAD SPACE STRUCTURE WITH TANK
          * MESH GENERATION
          UNIT  . . . DEFINE
          MESH
          A 0 0 0; B 0 20 0; C 20 20 0
          D 20 0 0; E 0 0 -20; F 0 20 -20
          G 20 20 -20; H 20 0 -20 GENERATE
          ELEMENT
          MESH AEHD 16
          MESH EABF 16
          ADCB MESH 16
          MESH HEFG 16
          MESH DHGC 16
```

Typical Generation Foursquare and triangular elements:
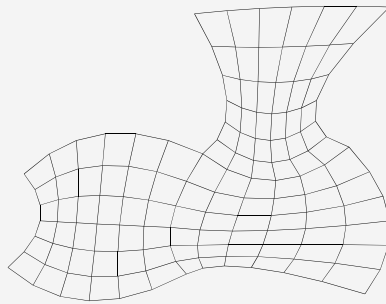
Typical generation
Quad elements
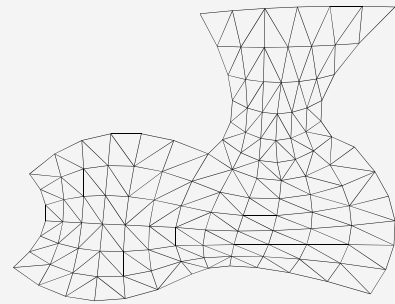
Typical generation
Triangular elements

5.15 Redefinition Boards and Numbering Members

Purpose

This command can be used to redefine the numbers and BOARD MEMBER. The numbers in the BOARD MEMBER and the original are replaced by the new numbers.

General Format:

```
       ⎡ ⎡JOINT    ⎤   ⎡XRANGE ⎤   ⎤
       ⎢ ⎧         ⎫   ⎧YRANGE ⎫   ⎢   f₁, F₂START i
SUBST  ⎨ ⎣MEMBER  ⎦   ⎣ZRANGE ⎦   ⎬
       ⎢ ⎢                           ⎢
       ⎣ COLUMN                      ⎦
```

Where f1 and f2 are two values in the range of x, y, ozei is the new starting number.

Description

Numbers can be redefined board member and STAAD through the use of SUBSTITUTE command. After a new set of numbers are assigned, the input and output values agree with the new numbering scheme. The user can design numbering schemes that produce a single input specification and the easy interpretation of results. For example, all together on the first floor of a building can be renumbered as 101, 102 ...., all joints of the second floor can be renumbered as 201, 202 ..... etc.

Example

```
UNIT METER
SUBST START JOINT YR 9.99 10.0 101 901
START SUBST COLUMN
```

The coordinates Y joints ranging from 9.99 to 10 meters will have a new start from number 101 columns starting with the new number 901 renumbered.

Note: The meaning of the re-specification of numbers and BOARD MEMBER can improve significantly the interpretation of results.

This command can be between incidence commands: MEMBER

> INCIDENCE
> SUBSTITUTE
> ELEMENT INCIDENCE

5.16 Displaying Boards of Members / Elements / Specification for GROUPS

This command allows the user to specify a group of joints / members / elements and save the information using a 'group-name'. The 'group-name' can be used consecutively in the input file instead of a list of boards / members / elements to specify other attributes. This extremely useful feature prevents multiple specifications of the same list of board / member. Following the general format required for the command GROUP

General Format:

> START GRODEF UPInition
> (GEOMETRY) _ (group-name) member / element / solid-list ......
>   (default) OR
> JOINT_ (group-name) joint-list ...... MEMBER_ (group-
> name) member-list ......

> **Element_ (group-name) element-list ...... SOLID_ (group-name) solid element-list ......**
>
> **END GROUP DEFINITION**

Where,

group-name = an alphanumeric name specified by the user to identify the group. The group name must start with the character '_' (underscore) and is limited to 16 characters.

member-list = the list of members / elements / solids belonging to the group. TO, BY, ALL, BEAM, PLATE, and SOLID allowed. ALL means all members + plates + solids; BEAM mean all beams; PLATE all plates; SOLID and all solids.

joint-list   = List together that belong to the group. TO, BY, and ALL are allowed.

Notes

1) The GROUP definition must start with the START GROUP DEFINITION command must end with the END command.

2) More than one GROUP name can be specified within the same definition of the specification.

3) JOINT words, MEMBER, SOLID ELEMENT and may be provided if the user wishes to identify the name of the group and lists those specific items. However, the group name and the list is merely a means of grouping more than one type of structural component under a single title, the word GEOMETRY be provided. In the absence of any of those five words (GEOMETRY, JOINT, MEMBER, or SOLID ELEMENT), it is assumed that the list is for GEOMETRY.

4) The same board or member number / element can be included in up to 4 groups. Multiple definitions are useful for output but can be ambiguous for data input as constants, the property of the section, relaxations, etc.

5) If two or more groups of consecutive entries with the same name, then they will join. If they are not consecutive, the second entry of the same name is ignored.

Example

```
START GROUP DEFINITION
_TRUSS 1 TO 20 25 35
_BEAM 40 TO 50
END
MEMBER PROPERTIES
_TRUSS TA LD L40304
ST TA _BEAM W12X26
```

Example

```
START JOINT GROUP
DEFINITION
_TAGA 1 TO 10
MEMBER
_TAGB 40 TO 50
```

```
GEOMETRY
_TAGC 101 TO 135
END

MEMBER PROPERTIES
_TAGB TA LD L40304
ST TA _TAGC W12X26
```

5.17 Rotating Geometry of Structure

Purpose

This command can be used to rotate the coordinates of the currently defined (and members / elements attached) on the global axes joint.

General Format:

$$\text{PERFORM ROTATION} \quad {}^* \begin{bmatrix} X & d_1 \\ A & d_2 \\ n & d_3 \end{bmatrix}$$

Where, d1, d2, d3 are the rotations (in degrees) about the global axes X, Y and Z respectively. This command can be entered after or between two Coordinates Joint Command or Joint Coordinates after all Trouble Member / Element specified. This command can be used to rotate the geometry of the structure (defined a priori to this command) to any desired angle for any global axis. The rotated configuration is used for analysis and design. While you are entering this command, note the direction of rotation must be under the rule of the right hand.

Description

This command can be used to rotate the geometry through any desired angle for any global axis. The rotated configuration can be used for analysis and design.

Example

```
ROTATION PERFORM X 20 Z -15
```

5.18 Specifying Inactive / Delete

Purpose

This set of commands can be used to temporarily inactivate (inactivate) or permanently DELETE (DELETE) the GASKETS or specified MEMBER.

General Format:

IN ACTIVE   M EMBERS   member-list

$$
\text{THE ETE} \quad
\begin{Bmatrix}
\text{M EMBERS} & \text{JOI-member} \\
\text{listNTS} & \text{joint-list}
\end{Bmatrix}
$$

Description

These commands can be used to specify that certain joints or members are disabled or completely nullify a structure. The INACTIVE command causes the temporarily inactive members; the user must re-activate them during the later part of the input for further processing. The DELETE command to completely void structure members; the user can not re-activate them. The Joint Delete command must be immediately after Joint Coordinates. Commands member should be provided immediately after all occurrences of the member / element are provided.

Notes

a) The DELETE command MEMBER void all associated together with automatically canceled, provided that the joints are not connected by any other member or members active element.

b) The DELETE MEMBER command also void all together which is not connected to the first structure. For example, such boards may have been generated to facilitate the entry of coordinates of the board and intended to be overridden. Hence, if a DELETE MEMBER command is used, a JOINT DELETE command should not be used.

c) The DELETE MEMBER command is applicable for clearing members as well as elements. If the list of members to be canceled extends beyond one line, should be continued on the next line by providing a space followed by a dash (-) at the end of the current line.

Example

```
INACTIVE MEMBERS    July 5 TO 10
DELETE MEMBERS      29 TO 34 43
```

d) The INACTIVE MEMBER command can not be used in situations where one day inactivating member arising that are not connected together in space.

e) The INACTIVE MEMBER command should not be used if the MEMBER TENSION / COMPRESSION command is used.

f) Inactivated members can be restored for future processes (such as analysis or design for a 2nd set of load cases) using the CHANGE command. See Section 5.37 and Example 4 for more information.

g) The DELETE MEMBER command must be used to override the elements too.
    Enter the command as DELETE MEMBER j, where j is the number of the item that you wish to delete. In the example above, 29 to 34 and 43 are the numbers of the element.

h) Charges have been defined as those declared INACTIVE members will not be considered in the analysis. This applies to SELFWEIGHT, MEMBER LOADS, LOADS Prestress and POSTSTRESS, TEMPERATURE loads, etc.

i) The JOINT DELETE command must precede commands incidence.

5.19 Steel Specification Table User

Purpose

STAAD allows users to create customized tables using steel sections for specifications of property, check the Code and Member Selection. This set of commands can be used to create the table (s) and provide the necessary data.

General Format:

```
START USER TABLE
TABLE   i₁(Fₙ)
section-type
section-name
property-spec
END
```

Where,

i1 = the number of the table (1-20). During analysis, the data in the table provided for each user are stored in a corresponding file with an extension. U01. for example, data table 5a is stored in U05. The first part of the name of the input file is the same as in the STAAD input file. These files are located in the same active directory the input file. Hence, they can then be used as external user input files to other tables.

fn = name of the external file containing the section type, name and properties corresponding (up to 16 characters)

section-type = name steel sections including: WIDE FLANGE, CHANNEL, ANGLE, ANGLE DOUBLE, TEE, PIPE, TUBE, GENERAL, ISECTION & PRISMATIC.

section-name = Any section name designated by the user, use 1 to 12 characters.
The first three characters of Pipes and Tubes should be respectively PIP and TUB. Only alphanumeric characters and digits defining section names are allowed. (Blank spaces, asterisks, question marks, commas, semicolons etc. are not allowed.)

property-spec = properties for the section. The requirements are different for each type of section as follows. Note that the cut areas AY and AZ should be provided to ensure proper cutting effort or cutting force calculations during design. They should also be provided SY, SZ, IZ and IY for design.

Length units predefined properties are the current units. If in UNIT command is then caught by the User table in the file those units become input the current units. However, a command UNIT in an external file that only affects file and has no effect on the units in the subsequent command input file. The user can specify the unit of desired length using the UNIT command as the first command in the table (see example that follows this description).

If the data is in the input file, then use up to 3 lines of input for each specification property (all done with a script, -). If the data is from the external file, then use only one line, but can be up to 250 characters long.

Description

The following section types are available under this option.

Wide Flange (Skate Width)

1) AX = Area of cross section
2) D = Superelevation section
3) TW = Web thickness
4) WF = width skate
5) TF = thickness Skate
6) IZ = Moment of inertia about the local Z (usually the major axis)
7) IY = Moment of inertia about local y-axis
8) IX = Torsional moment of inertia
9) AY = cutting area in the local y axis. If zero, the shear deformation is ignored in the analysis.
10) AZ = Same as above except the local z axis.

Channel (Canal)

1) AX, 2) D 3) TW, 4) WF 5) TF 6) IZ, 7) IY, 8) IX, 9) CZ, 10) AY, 11) AZ
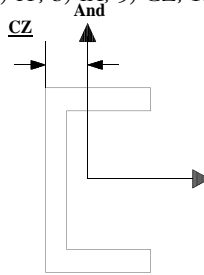
Angle (Angle)

1) D 2) WF, 3) TF, 4) R 5) AY, 6) AZ

R = radius of gyration about the principal axis, shown as r (ZZ) in the AISC manual. There should be zero.

Double Angl e (Double Angle)

1) D 2) WF, 3) TF, 4) SP 5) IZ, 6) IY, 7) IX, 8) CY, 9) AY, 10) AZ
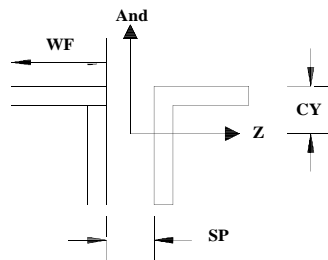
T (T)

1) AX, 2) D 3) WF 4) TF 5) TW, 6) IZ, 7) IY, 8) IX, 9) CY 10) AY, 11) AZ
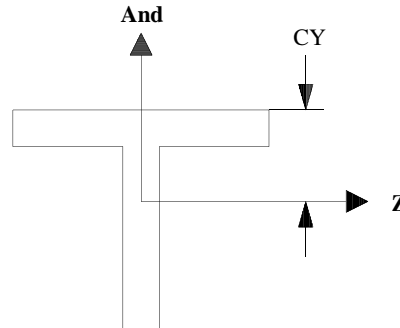
Pip   and (Tube)

1) OD   =   Outside
   Diameter
2) ID = Inside Diameter
3) AY 4) AZ 5) AX, 6) D 7) WF 8) TF, 9) IZ, 10) IY, 11) IX, 12) AY, 13) AZ

General

The properties of these cross sections should be used for this type section. Note that this option allows the user to specify a steel section catalog or unconventional. Provide both parameters Y and Z to design or code checking.

1) AX = Area of the cross section.
2) D = Superelevation
   section.
3) TD = Thickness associated with the element of the section parallel to the depth (usually the soul). to be used to verify the proportion cant / thickness.
4) B = width of the
   section.
5) TB = thickness associated with the element of section parallel to skate, to be used to verify the ratio of width / thickness.
6) IZ = Moment of inertia about local z axis.
7) IY = Moment of inertia about local y-axis.
8) IX = Torsional moment of inertia.
9) SZ = Section Modulus around the local z axis.
10) SY = Section Modulus around the local y axis.
11) AY = cutting area for cutting parallel to the local y
    axis.
12) AZ = cutting area for cutting parallel to the local z
    axis.
13) PZ = Plastic Module around the local z axis.
14) P = Plastic Module around the local y axis.
15) HSS = Torsional constant for calculation of lateral torsional buckling.
16) DEE = The depth of the soul. For the shifts sections, the distance between the bevels

must be provided.

Note: Property to be PZ, PY, HSS and DEE defined for the review / selection code member by plastic theory and limit state based on codes (AISC LRFD, British, French, German and Scandinavian codes). For systems based on allowable stress design (ASD-AISC, AASHTO, and NTC) codes should be zero values for these properties.

Profile
I

This type of section may be used to specify a general I-shaped section. The properties required of the cross section are listed below. Note that this option can be used to specify the variables forms I

1) DWW = Superelevation section at node startup.
2) TWW = Web thickness.
3) DWW1 = Superelevation section in the final knot.
4) BFF = Width of top flange.
5) TFF = Thickness of top flange.
6) BFF1 = Width of bottom flange.
7) TFF1 = thickness of bottom flange.
8) AYF = cutting area for cutting parallel to the Y axis
9) AZF = cutting area for cutting parallel to the Z axis
10) XIF = Torsional Constant (IX or J).
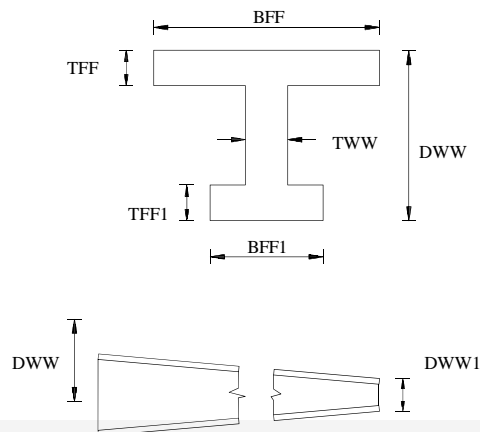
Notes:

1) DWW should never be less than DWW1. The user must provide the incidences of the member accordingly.
2) The user is allowed the following to the AYF, AZF and XIF values.

   a) If positive values are given, they are used directly by the program.
   b) If zero is supplied, the program calculates the properties using the following formula.

   AYF TWW x = D (where D = Superelevation section under consideration) AZF = 0.66 ((TFF BFF x) + (x BFF1 TFF1))

   XIF = 1/3 ((BFF x TFF3) + (DEE TWW3 x) + (x TFF13 BFF1))
   (Where the soul DEE = Superelevation section)

If negative values are given, they are applied as factors in the corresponding value (s) calculated by the program using the above formula. The factor is always applied provided the absolute value, ie if the user provides the value of XIF as -1.3, then the program multiplies the value of XIF, calculated by the formula above, by a factor of 1.3.

Prismatic

Property specifications for type PRISMATIC section is as follows:

1)   AX = Area of cross section
2)   IZ = Moment of inertia about local z axis
3)   IY = Moment of inertia about local y axis
4)   IX = Torsional moment of inertia
5)   AY = cutting area for cutting parallel to the local y axis.
6)   AZ = cutting area for cutting parallel to the local z axis.
7)   YD = depth of section in direction of the local axis.
8)   ZD = depth of section in direction of the local z axis.

Example

```
START USER TABLE
TABLE 1
UNIT. . .
WIDE FLANGE
W14X30
13.84 8.85 6.73 .27 .38 .385 291 19.6 0 0
W21X50
14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0
W14X109
14.605 32 .525 .86 14.32 7.12 7.52 447 1240 0
TABLE 2
UNIT. . .
ANGLES
L25255
2.5 2.5 0.3125 .489 0 0
L40404
4. 4. .25 .795 0 0
END
```

* Note that these section names must be provided in ascending order by weight, since the process of selecting the member uses these tables and the iteration starts at the top. The above example can also be entered as follows:

```
START USER TABLE
TABLE 1 TFILE1
END TABLE 2
TFILE2
```

Where are TFILE2 TFILE1 and file names to be created before the STAAD run, and where the TFILE1 file will contain the following:

```
UNIT. . . WIDE
FLANGE
W14X30
13.84 8.85 6.73 .27 .38 .385 291 19.6 0 0
W21X50
```

```
14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0
W14X109
32 14.32 0.525 14.605 1240 .86 447 7.12 7.52 0
```

and TFILE2 file will contain:

```
UNIT. . .
ANGLES
L25255
2.5 2.5 .3125 .489 0 0
L40404
4. 4. .25 .795 0 0
```

Notes: Table (s) Steel Provided by the user can be created and maintained as a separate file (s). The same files can be used for all models using sections of these tables. These files must reside in the same directory where the input file is located. In each file the first table must contain a UNITS command.

5.20 Specifying Properties Member

Purpose

This set of commands can be used for specifying properties of the section frame members.

General Format:

$$
\text{MEMBER PROPerties} \quad
\begin{Bmatrix}
\text{AUSCAN} \\
\text{TRALIANAdia} \\
\text{n EUROpean} \\
\text{FRENCH} \\
\text{INDIAN} \\
\text{AMEBRI} \\
\text{RICANTISH} \\
\text{CHINA} \\
\text{KORCOL} \\
\text{EAD} \\
\text{ALUMINUM} \\
\text{DUTCH} \\
\text{SPARUS} \\
\text{INSIA} \\
\text{GERMAN} \\
\text{JAPANESE}
\end{Bmatrix}
$$

$$\text{member-list} \begin{Bmatrix} \underline{\text{TA}}\text{BLE spec type-table-name} \\ \qquad \text{(additional-spec)} \\ \underline{\text{PR}}\text{ISMATIC property-spec} \\ \underline{\text{T}}\underline{\text{AP}}\text{ERED argument-list} \\ \text{UPT ABLE } i_1 \text{section-name} \\ \text{ASSIGN profile-spec} \end{Bmatrix}$$

The AMERICAN, BRITISH, EUROPEAN options (etc.) will instruct the program to take the properties of the steel table appropriately. The default depends on the country of distribution.

Description

This command starts the specification MEMBER PROPERTY. Following are the various options available:

a) Steel Specification user tables. (Section 5.20.1)
b) Specification of prismatic properties. (Section 5.20.2)
c) Specifying variable section members. (Section 5.20.3)
d) Specification for steel user tables. (Section 5.20.4)
e) Specification for assigning profile. (Section 5.20.5)

Each specification is described in detail in the following sections. Examples are available in Section 5.20.6.

MEMBER PROPERTY command can span multiple lines lines ending with a space and hyphen (-).

5.20.1 Specifying Table Properties of Steel

Purpose

The following commands are used to specify the properties of the table section (s) of steel catalog.

General Format:

type-spec. table-name additional-spec.

$$\text{type-spec} = \begin{Bmatrix} \underline{\text{ST}} \\ \text{RA} \\ \text{D} \\ \text{LD} \\ \text{SD} \\ \underline{\text{I}} \\ \text{C} \\ \text{m} \\ \text{TC} \\ \text{BC} \end{Bmatrix}$$
$$\text{TB}$$

ST section specifies simple standard catalog tables.
RA   specifies single reverse angle YZ axis (see Section 1.5.2). D
           specifies dual channel.

LD specifies double angle long side, back to back. SD specifies

double angle short side, back to back. T specifies tee beam section

cut so I.

Cm   specifies the composite section beams available so I. TC
           beams with specified upper plate covers.

BC specifies beams with bottom plate covers.

TB beams specified upper and lower plate covers.

table-name = The name of the table section as W8X18, C15X33 etc. The documentation steel design codes for each country includes information according to your specification steel section. For details on specific tables American steel sections, see Section 2.2.1 of this manual.

$$
\text{additional-spec} = {}^{*}\left\{\begin{array}{ll}
\underline{\text{SP}} & f_1 \\
\underline{\text{WP}} & f_2 \\
\underline{\text{TH}} & f_3 \\
\underline{\text{WT}} & f_4 \\
\underline{\text{DT}} & f_5 \\
\underline{\text{OD}} & f_6 \\
\underline{\text{ID}} & f_7 \\
\underline{\text{CT}} & f_8 \\
\underline{\text{FC}} & f_9 \\
\underline{\text{CW}} & f_{10}
\end{array}\right\}
$$

SP f_1 = This game describes the space (f_1) between angles or angles or double channels when channels are used. The f_1 is default to 0.0 if not given.

WP   Width = f_2 (f_2) of the cover plate cover plate if it is used with beam section I. TH Thickness = f_3 (f_3) of rectangular plates or tubes.

WT f_4 = W (f_4) of rectangular tubing, where TUBE is the name of the table.

DT   Depth = f_5 (f_5) of rectangular tubes.

OD   OD = f_6 (f_6) of tubes, wherein PIPE is the name of the table.

ID f_7 = Inside diameter (f_7) tube.

CT f_8 = thickness of the concrete (f_8) for composite sections.
FC f_9 = Compressive strength (f_9) of concrete for composite sections. CW
          f_{10} = width of concrete (f_{10}) for composite sections.

See Section 1.7.2

Notes

All F1-9 must provide values in current units. Some important points to note in the case of the composite section are:

1) The width of the concrete slab (CW) (if not entered) is assumed to be the width of the top flange of the steel section + 16 times the thickness of the table.
2) To calculate the properties of the cross-section, the ratio is calculated assuming modular this:

> ES = modulus of elasticity of steel = 29,000 ksi. Ec
> = Modulus of elasticity of concrete = 1802.5 ksi
>
> Where FC (in ksi) defined above.

5.20.2 Prism Specification Properties

Purpose

The following to specify the section properties for the prismatic cross sections are used commands.

General Format:

For PRISMATIC specification, properties are provided directly (At the end of each line with a hyphen "-") as follows:

T-SECTION   KEYSTONE SECTION
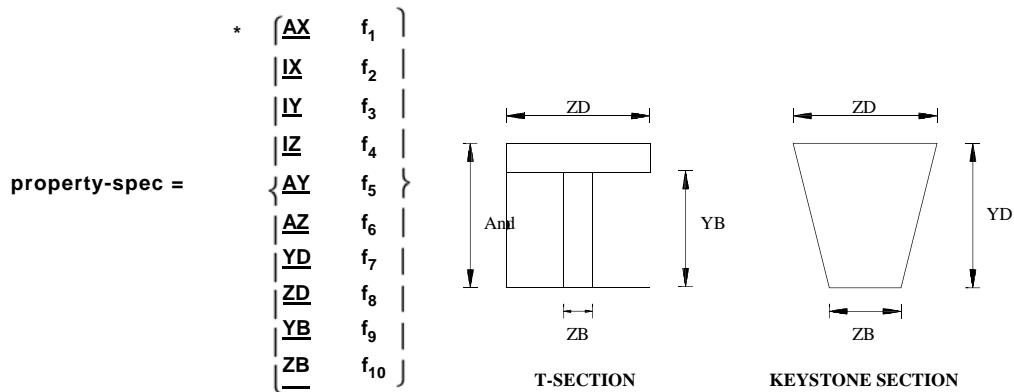
AX   f1 = Cross-sectional area of the member.

IX   f2 = Torsional moment of inertia.

IY   f3 = Moment of inertia about local y-axis.

IZ   f4 = Moment of inertia about local z axis (usually the largest). AY

       f5 = effective shear area in local y axis.

AZ   f6 = effective shear area in local z axis.

See Section 1.7.1

If any of the above 6 parameters is omitted, it is calculated YD, ZD, YB, and / or dimensions of ZB.

YD   f7 = The depth of the member in the local address. (The diameter of the round section members).

ZD   f8 = The depth of the member in the local z direction. If not provided ZD and YD is provided, it is assumed that the section is round.

YB   f9 = The depth of the soul to Section T.

ZB   f10 = The width of the web for T section or width below for KEYSTONE section.

5.20.2.1 Property Specification Prismatic Tapered Tube

Purpose

The following commands are used to specify the properties of the section to the cross sections of prismatic tubes variables. For property types shown below, additional information can be obtained from Table 2.1 of the document ASCE 72, 2nd edition.

General Format:

$$\text{property-spec} = \left\{ \begin{array}{l} *\underline{RO}UND \\ \text{HEXDECAGONAL} \\ \underline{DOD}ECAGONAL \\ \underline{OCT}AGONAL \\ \text{HEXAGONAL} \\ \text{SQUARE} \end{array} \right\} \text{START } d_1 \text{END } d_2 \text{THICK } t$$

START d1 = Depth of section at the beginning of the member. END d2 = Depth of section at the end of the member.

THICK t = thickness of the section (constant along the length of the member).

ROUND
And

SQUARE
And

HEXDECAGONAL (16 SIDES) Y

HEX (6 SIDES) AND

Dodecagonal (12 SIDES)

Octagonal (8 SIDES)

And

And
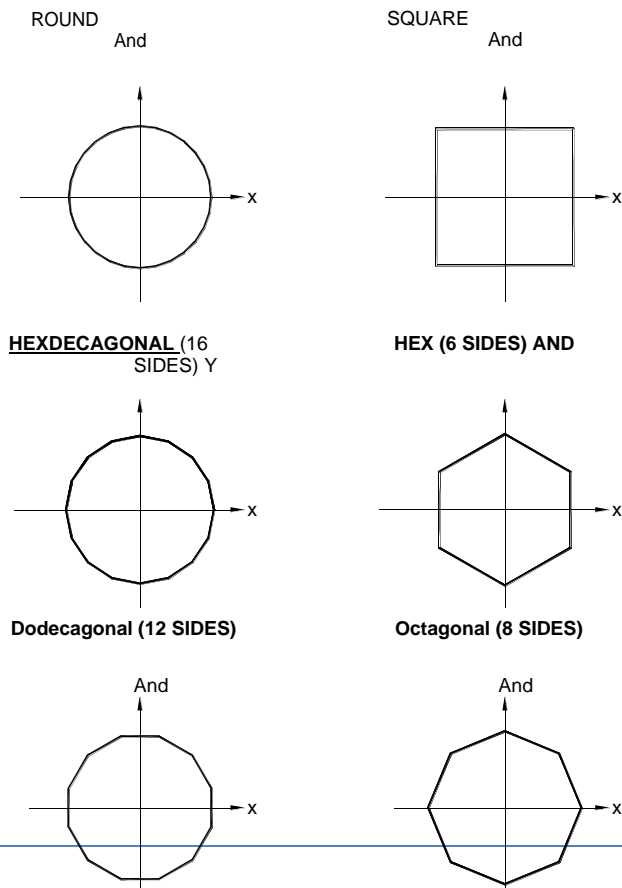
Example

```
UNIT. . .
MEMBER PROPERTY
1 PRIS STA 10 END ROUND 8 THI 0.375
2 PRIS HDC    STA 10 15 END THI 0.375
3 PRIS DOD    STA 12 12 END THI 0.375
```

Tapered Prism Forms Tube

5.20.3 Specification Tapered Member

Purpose

The following commands are used to specify the properties of the section for forms I variables.

General Format:

List-argument = $f_1 f_2 f_3 f_4 f_5 (F_6 f_7)$

Where,

f1 = Depth of section at node startup. f2 = Web

thickness.

f3 = The depth of the section at the end node.

f4 = Width of top flange.

See Section 1.7.4

f5 = Thickness of top flange.
f6 = Width of bottom flange. Defaults to f4 if it was omitted. f7 =

Thickness of bottom flange. Defaults to f5 if omitted

Example

```
MEMBER PROPERTY
TAPERED 1 TO 5 13.98 13.98 0.285 6.745 6.745 .455 .455
```

Notes:

1. All dimensions (f1, f2, f7 .....) must be in the current units.
2. f1 (The depth of the section in the start node) must always be greater than f3 (The depth of the section in order knot). The user must provide the incidences of the member accordingly.

5.20.4 Specification of properties from the user steel table

Purpose

The following commands are used to specify the properties of the section of a steel table that the user previously created with USER-PROVIDED STEEL TABLE.

General Format:

member-list <u>UPT</u>ABLE **I₁** **section-name**

UPTABLE to refer to the tables provided by the user.

i1 = table number as previously specified (1-20)
section-name = name of the section as specified in the table. (Refer to Section 5.19)

See Section 1.7.3

Example

See Section 5.20.6

5.20.5 Assign Profile Specification

Purpose

The ASSIGN command can be used to instruct the program to assign a member of the framework based on the profile specification shown below a steel section convenient.

General Format:

See Section 1.7.5

$$
\text{profile-spec} =
\begin{cases}
\underline{BEA}M \\
\underline{COL}UMN \\
\\
\underline{CHA}Nnel\ ANGLE \\
(DOUBLE)
\end{cases}
$$

Note: The sections are always chosen from the relevant steel table catalog. To find out the details of the sections are selected, the PRINT MEMBER PROPERTIES commands must be provided after the specification of all the properties of the member.

5.20.6 Examples of Member Property Specification

This section illustrates the various options available for specification MEMBER PROPERTY

Example

```
UNIT. . .
MEMBER PROPERTIES
TABLE 1 TO 5 ST W8X31
```

```
September 10 TABLE 0.25 SP LD L40304
12 TO 15 PRISMATIC 10.0 IZ AX 1520.0
ST RT 17 18 PIPE OD 2.5 ID 1.75
20 TO 25 AT ST TUBE 8 TH DT 0.5 WT 12
27 29 32 TO 40 42 PR 5 AX 400 IY IZ 33 -
YD 9 IX 0.2 ZD 3.
43 TO 47 UPT one W10X49
UPT 2 50 51 L40404
COLUMN ASSIGN 52 TO 55
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CT W14X34 CM 5.0 FC 3.0
```

This example shows each input type member property. The members 1 to 5 are wide skids beams I selected from Tables AISC; 9 and 10 are selected double angles tables AISC; 12-15 are binoculars members without shear deformation; 17 and 18 are the tube sections; 20 to 25 are of rectangular tube sections; 27, 29, 32-40, and 42 are members binoculars shear deformation; 43 to 47 are I-beams of a wide skids selected user table entry number one; 50 and 51 are simple angles chart 2 that the user entered; are designated as members 52 to 55 using the ASSIGN COLUMN specification. I assigned program table suitable for each member a steel section.

The Member # 56 is a wide-flange covers W12X26 plate with a width of 4.0 units 0.3 units higher thickness. The member 57 is a composite section with a concrete slab of thickness of 5.0 units in the top of a wide-flange W14X34. The
compressive strength of the concrete in the table is 3.0 Force /
length2

5.20.7 Specification curved Member

Purpose

The following commands are used to specify that a member is curved. The curve is a segment of a circle arc of length less than 180 degrees. Any no varying cross-section is allowed.
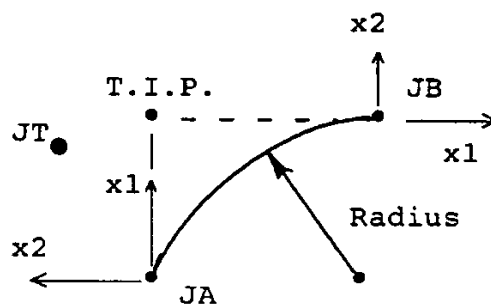
Signs Convention curved beams

T.I.P. = Tangent Intersection Point

General Format:

M EMBER C

URVE

$$\text{member-list CURVED (} \left\{ \begin{array}{l} \text{f1 f2 f3} \\ \underline{\text{ZMI}}\text{NUS} \\ \underline{\text{REF}} \text{ f4 f5 f6} \\ \underline{\text{REFJT}} \text{ f7} \end{array} \right\} \text{) BENDR f8 } \underline{\text{FLE}}\text{X f9}$$

Where,

The parameters in parenthesis are curved member by definition parameters in the local coordinate system (and orientation of the curve) that are optional. Default value: The local coordinate system is one in which a straight member between the two boards and the curve are locally in the XZ + Z side. To inhibit the default local system, from 1 to 4 parameter sets:

See Section 1.7.4

[The reference point along the two together define a half-plane containing the axis of

Local and + Z plane of the

curve.]

f1 f2 f3 = x, y, z global distances from the joint output of the reference point; f4 or

f5 f6 = x, y, z global coordinates of the reference point; or

f7 = the number of the board whose global coordinates x, y, z are the point of reference;

or

ZM = indicates that the local system is the default but that the plane of the curve will be in the XZ plane of the local Z-side of a line connecting the 2 together the member.

Other

parameters:

f8 = the radius of curvature of the centroidal axis of the limb (radius of curvature). See the figure.

f9 = flexibility parameter for the curvatures of the pipes. See notes.

FLEXIBILITY parameter. This only applies to members of curved pipes (elbows) and (OD and ID as entered). These members will bend over due to ovalization which depends on the internal pressure. The ASME-Boiler and Pressure Vessel Code (Pressure Vessel Code), Section III, NB-3687.2, 1971 The Class I components are used to calculate the reduction factor of flexibility.

- **September f9 = 0 is ignored for this calculation will occur flexibility increased internal pressure equal to zero.**
- **September f9 > 0 to specify the internal pressure for use in calculating this flexibility.**
  The increased pressure reduces flexibility.
- **September f9 = -9999 ignore this additional flexibility calculation and use only the theory of**
  beam.
- **September f9 = reduction factor flexibility (- FLEXF below); what should be a**

negativ e nu mb er less tha n - 1.0.

FLEXIBILITY FACTORS THEORY OF ELBOW PIPE ASME [ASME Section NB-3687.2]

—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—
—

This section only applies if (Radius Bending / Medium Haul) ≥ 1.70

O

If (arc length) > (Medium 2 * Radio)

$$\text{FLEXF} = (\text{Flexibility Factor}) = \frac{1.65 * (\text{Mean Radius})2}{t * (\text{Bend Radius})} * \frac{1}{1 + (\text{Press}) (\text{Mean Radius}) (\text{FACT.})}$$

Where

$$\text{FACT.} = \frac{6 * (MR / t)^{4/3} * (BR / MR)^{13}}{Et}$$

MR = Mean radius elbow wall   BR = Bending Radius

Press = Internal Pressure
t   = Wall thickness of the elbow   E   = Modulus of Elasticity

If the calculated Flexibility Factor is less than 1.0, then use 1.0. The Flexibility Factor is multiplied directly or contributes more to the conditions of non-cut on the elbow flexibility matrix.

Example

> **MEMBER PROPERTY**
> **1 TO 5 CURVED BENDR ZM 10.0**

Notes:

1) All dimensions and pressures must be in the current units.
2) For loads, relaxations, and outputs the local x-axis is tangent to the curve; local z axis is radially outside of the curve in the plane of the curve; and the local axis is perpendicular to the plane of the curve.
3) **This item is not complete** except stiffness, relaxations and displacements. Truss, tension / compression, should not be specified. Loads at each end of the member based on the distance instead of using beam theory recessed provided. The results of section ignore the actual distribution of charge and should be ignored.

5.21 Specifying Element Properties

Purpose

This set of commands can be used to specify properties of plate finite elements.
Unequal members and plate elements / shell not require any property to the solid elements. However, as the constant modulus and the Poisson ratio are specified.

General Format:

THE ELEMENT PROPERTY

element-list   T HICKNESS f$_1$ (F$_2$, F$_3$, F$_4$)

f$_1$ =   Thickness of the element.
f$_2$F ...$_4$ =   Thickness of other nodes of the element, if different

Description

See Section 1.6   Elements of uniform or linearly varying thickness can be modeled using this command. Note that the thickness value should be provided in the current units.

Example

```
UNIT. . .
ELEMENT PROPERTY
1 TO 14 August 16 TH 0.25
```

5.22   Member / Element Releases

STAAD relaxation allows specification of degrees of freedom in the frame members and plate elements. Section 5.22.1 describes the options for release of MEMBER and Section 5.22.2 describes the options ELEMENT release.

5.22.1   Member Specification Release

Purpose

This command set may be used to relax completely specific degrees of freedom for the ends of the frame members. Can also be used to describe a mode of attachment where the end of the member is connected to the board for specific degrees of freedom by means of springs.

General Format:

M EMBER R ELEases

```
                                *⌈ FX ⌉      *⌈ KFX  f1 ⌉
                     ⌈ START ⌉   | FY |       | KFY  f2 |
member-list  {        }   { EZ }      { KFZ  f3 }
                     ⌊ END   ⌋   | MX |       | KMX  f4 |
                                 | M  |       | KMY  f5 |
                                 ⌊ Y  ⌋       ⌊ KMZ  f6 ⌋
```

Where to MZ and FX KFX KMZ to represent the strength-x through the degrees of freedom of the moment in local z-axis of the member and f1 to f6 are the spring constants for these degrees of freedom. If FX is used by MZ means total relaxation for that shallow, and if KFX through KMZ are used, means a spring attached.

Example

> **MEMBER RELEASE**
> **January 3 September 11 12 TO START MY MZ KFX 1000.0**
> **January 10, 11 13 to 18 END MZ 200.0 KMX**

In the above example, for members 1, 3-9, 11 and 12, the moments about the local axes Y and Z relaxing together in their home (as specified in INCIDENCES MEMBER). Later, these board members to its INITIAL tied along its local axis x through a spring whose stiffness is 1000.0 units of force / length. For members 1, 10, 11 and 13-18, the time on the local Z axis is released its final meeting. Also, your EXTREME board members are tied on its local axis x through a spring-time whose rigidity is
Force-length / Grades 200.0 units. Note that members 1 and 11 together relaxing on both start and end, but not necessarily in the same degrees of freedom.

Partial Moments mind Freed

They can be partially released moments at the end of a member. This can be used to model partial embedment connections. The following format can be used to provide partial relaxation time. Note that this option is provided in the MEMBER option RELEASE and is added to the capabilities of existing RELEASE.

General Format:

M EMBER R ELEASE

$$
\text{member-list} \quad
\begin{Bmatrix} \underline{\text{STA}}\text{RT} \\ \underline{\text{END}} \end{Bmatrix}
\quad
{}^{*}\begin{Bmatrix} \underline{\text{MP}} & \text{f1} \\ \\ \underline{\text{MPX}}\ \text{f4} \\ \underline{\text{MPY}}\ \text{f5} \\ \underline{\text{MPZ}}\ \text{f6} \end{Bmatrix}
$$

See Section 1.8

Where f1 = relaxation factor for all 3 phases; or enter f4, f5, and / or f6 separately as relaxation factors for each time. The timing related to the relative stiffness coefficient is multiplied by a factor of (1-f1) to the specified end.

Example

> **MEMBER RELEASE**
> **15 25 START TO MP 0.25**

The RELEASE command above will apply a factor of 0.75 at the time of the shear modulus on the START of 15 to 25 members.

Notes: It is important to note that the factor f₁ indicates a reduction in stiffness corresponding to the rotational degrees of freedom of MX, MY and MZ. In other words, you should not expect that when the member is reduced by a factor of f₁. It may be necessary for the user to perform a test to reach the correct value of f₁ to produce the desired reduction in moment.

Also, note that the START and END is based on the specification of MEMBER INCIDENCE.

At either end of the member, for any particular a complete or partial relaxation DOF spring and can not be applied simultaneously. Allowed only one of the three.

If MY (or MZ) completely relaxes at both ends, then VZ (or VY) can not be transmitted to the member. The final cuts in the Member shall completely due to loads applied directly to the member.

5.22.2 Element Specification Release

Purpose

This set of commands can be used to relax specific degrees of freedom at the corners of plates of finite elements.

General Format:

THE ELEMENT RELEASE

```
                              *  FX
                   J1           [ FY ]
                  [ J2 ]        | FZ |
   element-list   { J3 }        { MX }
                  | J4 |        | MY |
                  [    ]        | MZ |
                               [    ]
```

See Section 1.8

Keywords where J1, J2, J3 and J4 mean joints in the order of specifying the effect of the term. For example, if the impact element 76 63 and 35 42 were defined, representing 35 J1, J2 represents 42, 76 represents J3 and J4 represents 63 Please note that relaxation of the element of the multiple joints can not be specified in a single line. Those must be specified separately as shown below.

FX through MZ represents forces / moments to be relaxed by the local axis system.

Example

Proper Use Misuse

10 ELEMENT TO RELEASE MY MX 50 J1	ELEMENT TO RELEASE 10 50 J1 J2 MX MY

10 TO 50 MY MX J2
MY 10 TO 50 J3
MY 10 TO 50 J4

10 TO 50 MY J3 J4

Notes:

1. All relaxations are in the local axis system. Generally, not above relaxation. The item must still behave like a plate after relaxations.

2. Selfweight is applied to each of the nodes as if there were no relaxation.
3. Thermal stresses will include pre-built strain at the end as if there was no relaxation.

4. Can not be used with Plane Stress Element or Element Ignore inplane commands Rotation in the same element.
5. **Note that the usual definitions of Mx and My local is reversed here.**
6. Relaxing Fz, Mx, My relax all bendability. Relaxing Fx, Fy, Mz relax all the rigidity in the plane.

5.23 Specifying Member Truss / Cable / Tension / Compression

A member can have one of the following specifications: MEMBER

TRUSS

TENSION MEMBER
COMPRESSION MEMBER
MEMBER RELEASES

If the same member multiple specifications are applied, use only the last entry (Warnings are printed).

TRUSS MEMBER, MEMBER TENSION, COMPRESSION MEMBER and MEMBER WIRE are only axial rigidity.

LEAD MEMBER are special members–armor type can also be specified as–voltage only.

Sections 5.23.1 to 5.23.3 describe these specifications.

5.23.1 Specifying Member Truss

Purpose

This command can be used to model a specified set of members as members TRUSS.

Description

This specification can be used to specify the type members PLANE TRUSS structures, or FLOOR SPACE. The TRUSS members are able to withstand axial forces only. Upwind type Typically, members or SPACE PLANE frameworks of this nature will.

General Format:

M EMBER T RUSS

member-list **T IN**SION f$_1$

Where f1 = Optional Initial Voltage type truss members (in current units)

Note that this command is superfluous when TRUSS structure type is already specified.

See Sections 1.9 and 1.10

Example

```
MEMB TRUSS
1 to 8 10 12 14 15
```

Note: TRUSS member has only one degree of freedom, the axial deformation. It is equivalent to a frame member with the time relaxed on both ends. Also note that Member Releases not allowed. Selfweight and transverse loads can induce distributions cutting / time member.

5.23.2 Cable Specification Member

Purpose

This command can be used to model a specific set of members as LEAD members.

Use in all analyzes described except Nonlinear Analysis of Cable:
The WIRE members besides the elastic axial deformation, are also capable of accommodate the stiffening effect of initial tension and stress due to static charge. LEAD members Theoretical discussions are presented in Section 1.11 of this manual.

General Format:

M EMBER C ABLE

member-list **T IN**SION f$_1$

See Sections 1.9, 1.11.

Where f1 = initial voltage on the wire member (in current units)

Example

```
MEMB WIRE
TENSION 20 TO 25 15.5
```

Notes: TENSION WIRE specified in the member is applied to the structure as well as external load is used to modify the stiffness of the member. See Section 1.11 for details. If TENSION or a minimum voltage value is omitted it will be used.

This is a truss member but not a member-only voltage unless you also include this member in input TENSION MEMBER. See section 5.23.3. Also note that Member Releases not allowed.

The preload tension is not the last cable tension after deformation due to preload it.

The description for use in the Nonlinear Analysis of Cable:

The WIRE members besides the elastic axial deformation, are also able to accommodate large displacements.

General Format:

<u>M EM</u>BER C ABLE

member-list <u>T IN</u>SION f_1
--- OR ---
member-list <u>L IN</u>GTH f_2

See Sections

*1.9, and
1.11.*

Where

f1 = Initial voltage wire members (in current units) f2 = unstressed length of cable (in current units)

Example

```
MEMB WIRE
TENSION 20 TO 25 15.5
```

Note: TENSION WIRE specified in the member is applied to the structure as an external load as well as modify the stiffness of the member. See Section 1.11 for details. The voltage is used to determine the unstressed length. That length is shorter than the distance between the joints for the distance that the tension stress the cable.

5.23.3 Member Specification Tension / Compression

Purpose

This command can be used to designate certain members as members tension-only or compression-only.

*See Section
1.9*

General Format:

<u>M EM</u>BER T INSION
member - list

M EMBER COMPRESSURE
member - list

Description

Members only are truss-voltage cable / members are only able to withstand tensile forces. Thus, they become inactive for cases automatically creating compressive load on them.

LINEAR ANALYSIS TENSION / COMPRESSION: Compression-only members are truss members that are capable of supporting only compression forces. Thus, they become inactive for automatically loading conditions that create stress on them. Note that Member Releases are not allowed. The procedure for the analysis of tension-only members or only-Compression requires iterations for each load case and therefore are quite involved. The user may also consider using the INACTIVE time spec if the solution becomes unacceptably high. It is important to recognize that the input data should be provided so that only one instance of the primary load is maintained in each PERFORM ANALYSIS. The SET and CHANGE commands NL also be used to carry STAAD to multiple analyzes and structural conditions are involved.

NO-LINEAR ANALYSIS LEAD: cables are automatically assumed to be only voltage (except only will always selfweight) without this command. They can only trusses preloaded as specified by this command only tension, moreover these trusses can take compression too. In this type of analysis, trusses without bias are assumed to be linear tension members carrying compression regardless of this command.

Example

```
TENSION
MEMBER TO 25 30
35 36
```

Example

```
MEMBER COMPRESSION
43 57 98 102 145
```

Example

```
MEMBER TO
TENSION
December 17 19
37 65
COMPRESSION MEMBER
May 13 46 TO 53 87
```

Notes LINEAR ANALYSIS TENSION / COMPRESSION:

1) Loads that are defined in the declared MEMBER MEMBER TENSION COMPRESSION members or even be active when the member becomes INACTIVE during the analysis process. This applies to SELFWEIGHT, MEMBER LOADS, LOADS Prestress & POSTSTRESS, TEMPERATURE LOAD, etc ,.

2) A TENSION declared as a member or member only COMPRESSION, bear only axial forces. It will not take moments or shear forces. In other words, it is a truss member.

3) They should specify the MEMBER TENSION and COMPRESSION MEMBER command if the command is specified INACTIVE MEMBER. The following is the general sequence of commands in the input file or if the MEMBER TENSION COMPRESSION MEMBER commands are used. This example is for the MEMBER TENSION command. Similar rules apply for the MEMBER COMPRESSION command. Dots indicate other items of input data.

```
STAAD ...
UNITS
SET NL ...
...
JOINT COORDINATES
...
MEMBER
INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
...
MEMBER
PROPERTY
...
ELEMENT PROPERTY
SUPPORTS
...
...
TENSION
MEMBER
LOAD 1
...
...
PERFORM ANALYSIS
CHANGE
TENSION
MEMBER
LOAD 2
...
...
PERFORM ANALYSIS
CHANGE
TENSION
MEMBER
LOAD 3
...
...
PERFORM ANALYSIS
CHANGE

TENSION MEMBER
LOAD 4
...
...
PERFORM ANALYSIS
CHANGE
```

TENSION MEMBER
5 LOAD
...
...

```
LOAD COMBINATION 6
...
LOAD COMBINATION July
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT
PARAMETER ...
...
CODE CHECK MEMBER
... SELECT ... FINISH
```

a) See Section 5.5 for explanation of the SET NL command. The number following this command is the total number of primary load cases in the file.

b) STAAD automatically above 10 iterations, stopping if converged. If not converged, a warning message will be output. Between SET ITERLIM i (i> 10) command before the first load case to increase the default number of iterations. Since the convergence may not be possible using this procedure, do not put the limit too high.

c) The principle used in the analysis is the following.

   The program reads the list of members declared TENSION MEMBER and / or COMPRESSION.

   - The analysis was performed for the whole structure and member forces are calculated.
   - For declared MEMBER TENSION / COMPRESSION members the program checks the axial force and forces determine whether tension or compression. If the member
   - can not take that one load, the member is "off" of the structure.

   The analysis was performed without shutdowns new members.
   Top 10 iterations of the above steps are done for each PERFORM ANALYSIS
   - command.
   - This method does not always converge and can be unstable. Check output for messages of instability. Do not use the results if it is unstable.
   -

d) In the example shown, only if specified for LOAD ANALYSIS. This is because a member who is under stress for a load case may be in for another compression load case. The stiffness matrix for analysis can answer the structural condition of one of these two cases of load.

e) Note that the MEMBER TENSION / COMPRESSION command can be provided and its membership after each (except the last) of the CHANGE command. If entered, the new MEMBER TENSION / COMPRESSION supersedes all previous commands. This is because every CHANGE command means that the forelimbs in TENSION / COMPRESSION command is active all over again. If these commands do not come after a CHANGE, then the above commands will still apply.

f) The TENSION MEMBER command should not be used if the following load cases are present: load case Response Spectrum, Time History load case, load case UBC, Moving load case. If used, MEMBER TENSION / COMPRESSION will be ignored in all load cases.

5.23.4 Specification Spring Tension / Compression

Purpose

This command can be used to designate certain media such as springs or tension springs only-only-compression springs.

General Format:

SPRING T ONSION
joint - list   spring-spec

SPRING COMPRESSURE
joint - list   spring-spec

$$\text{spring-spec} = \ast \begin{bmatrix} KFX \\ KFY \\ KFZ \end{bmatrix}$$

Description

Voltage-only springs are only able to withstand tensile forces. Thus, they become inactive for cases automatically creating compressive load on them. Only- Compression springs are only able to withstand the compression forces. Thus, they become inactive for automatically loading conditions that create stress on them.

See Section 1.10

If no specification spring then enters all translational spring at that meeting will be only tension (or compression). This command does not create a spring entry only if there is a support spring then the board at the address specified voltage will also be only (or compression). See Section 5.27.1 to 5.27.4 to define springs.

The procedure for the analysis of tension-only springs or compression only requires iterations for each load case and therefore are really involved.

It is very important to recognize that the input data must be provided so that only one case of primary load is maintained for each PERFORM ANALYSIS command. The SET and CHANGE commands NL also be used to carry STAAD to multiple analyzes and multiple structural conditions are grouped.

Example

```
SPRING TENSION
December 17 19 TO
37 65
COMPRESSION
SPRING May 13 46 TO
53 87                KFY
```

Notes

1. A spring-declared only TENSION-COMPRESSION or just take only axial forces. It will not take the time.
2. The SPRING TENSION / COMPRESSION command should not be specified if the command specified INACTIVE MEMBER.
3. The following is the general sequence of commands in the input file if COMPRESSION SPRING TENSION or commands are used. This example is for the SPRING TENSION command. Similar rules apply for the SPRING COMPRESSION command. Dots indicate other items of input data.

```
STAAD ...
UNITS
SET   NL
... ...
JOINT COORDINATES
...
MEMBER
INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
...
MEMBER
PROPERTY
...
ELEMENT PROPERTY
SUPPORTS
...
...
SPRING TENSION
LOAD 1
...
...
PERFORM ANALYSIS
CHANGE
SPRING TENSION
LOAD 2
...
...
PERFORM ANALYSIS
CHANGE
SPRING TENSION
LOAD 3
...
...
PERFORM ANALYSIS
CHANGE
SPRING TENSION
LOAD 4
...
...
PERFORM ANALYSIS
CHANGE
```

SPRING TENSION

```
5 LOAD
...
...
LOAD COMBINATION 6
...
LOAD COMBINATION July
...
PERFORM ANALYSIS

CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER
...
CODE CHECK
MEMBER ... SELECT
... FINISH
```

a) See Section 5.5 for explanation of the SET NL command. The number following this command is the total number of primary load cases in the file.

b) STAAD automatically above 10 iterations, stopping if converged. If not converged, the warning message will be output. Between SET ITERLIM i (i> 10) command before the first load case to increase the default number of iterations. Convergence may not be possible using this procedure, do not put the limit too high. If not converged, a message will be output.

c) The principle used in the analysis is the following.

- The program reads the list of declared SPRING TENSION springs and / or COMPRESSION.

- The analysis is performed for the whole structure and spring forces are calculated.

- For springs declared SPRING TENSION / COMPRESSION, the program checks the axial force to determine whether tension or compression. Positive displacement is STRESS. If the spring load can not make that, the spring is "off" of the structure.

    The analysis was performed without the springs back off.
-
    Top 10 iterations of the above steps are done for each PERFORM ANALYSIS
- command.

    This method does not always converge and can be unstable. Check output for
- messages of instability. Do not use the results if it is unstable. You may need to include some support in each global address that is not tension (or compression) only to be stable in each iteration.

d) In the example shown, only if specified for LOAD ANALYSIS. This is because a tension spring that is under load to a case the compression may be another case of load. The stiffness matrix for analysis can answer the structural condition of only one of these two cases of load.

e) Note that the SPRING TENSION / COMPRESSION command can be provided and list together after each (except the last) of the CHANGE command. If he came, new commands SPRING SPRING supersede all previous commands. This is because, each CHANGE command means that springs above a voltage / COMPRESSION back command are all active. If not entered after a CHANGE, then the definitions of the above springs are used.

f) SPRING TENSION The command should not be used if the following load cases are present: load case Response Spectrum, Time History load case, load case UBC, Moving load case. If used, SPRING TENSION / COMPRESSION are ignored all load cases.

5.24   Specifications   Plane Stress Element   Rotation and inplane

Purpose

These commands allow the user to model the following conditions in the plate elements

a) Condition PLANE STRESS
b) Rotational stiffness reformulated plane to be rigid or zero.

General Format:

$$
\text{THE EMENT} \left\{ \begin{array}{l} \underline{\text{PL}} \text{AST NE RESS} \\ \underline{\text{RIG}} \text{ID} \quad \text{(IN PLANE R OTATION)} \\ \underline{\text{IGN}} \text{ORE (IN PLANE R OTATION)} \end{array} \right\}
$$

   element-list

Description

See Section 1.6

The PLANE STRESS specification allows the user to model elements selected for actions only in the plane [No bending stiffness or cross section].

The inplane RIGID ROTATION command causes the program to turn the corner on Mz "flat rotation" action in the other corner stiffens Mz rotations. The formulation of plate elements Staad normally produces a very smooth stiffness Mz involving shear deformation in the plane. However when Mz board is connected to a beam with bending moment as the only way of charging for that time, then the inplane RIGID option can be used to move the element that moment Mz rigidly to another ring to prevent instability end of the beam. Usually only items attached to beams in this way would this specification.

The IGNORE inplane ROTATION command causes the program to ignore the actions "of rotation in the plane." The formulation of the plate element STAAD normally include this important action automatically. However, it may be noted that some formulations ignore this action element default. The user can use this option to compare the results of STAAD with solutions of these programs.

These options are unique to us and also the relaxation of the element. Not a single element can have more than one of these options.

Example

```
PLANE STRESS ELEMENT
1 TO 15 October20 25 35
ELEMENT IGNORE
30 50 TO 55
```

5.25   Member Offset Specification

Purpose

This command can be used for a rigid frame member offset from end to model mismatch condition exists at the ends of the frame members.

General Format:

M EMBER OFFSETS

member-list $\left\{ \begin{array}{l} \text{START} \\ \text{F}_3\text{END} \end{array} \right\}$ (**LOC**AL) f_1, F_2,

Description

See Section 1.12

f_1, f_2, f_3 and correspond to the distance measured in the global coordinate system, or Located board (START or END as specified) to the centroid of the start or end point of the listed members. The MEMBER OFFSET command can be used for any member whose start or end points are not coexisting with the incident given board. This command allows the user to answer the secondary forces which are induced due to the eccentricity of the member. Offsets can be specified in any of the Member

address, including the address can match the x axis of the member. WP in the diagram

refers to the situation of the centroid of the start point or end of the

member.
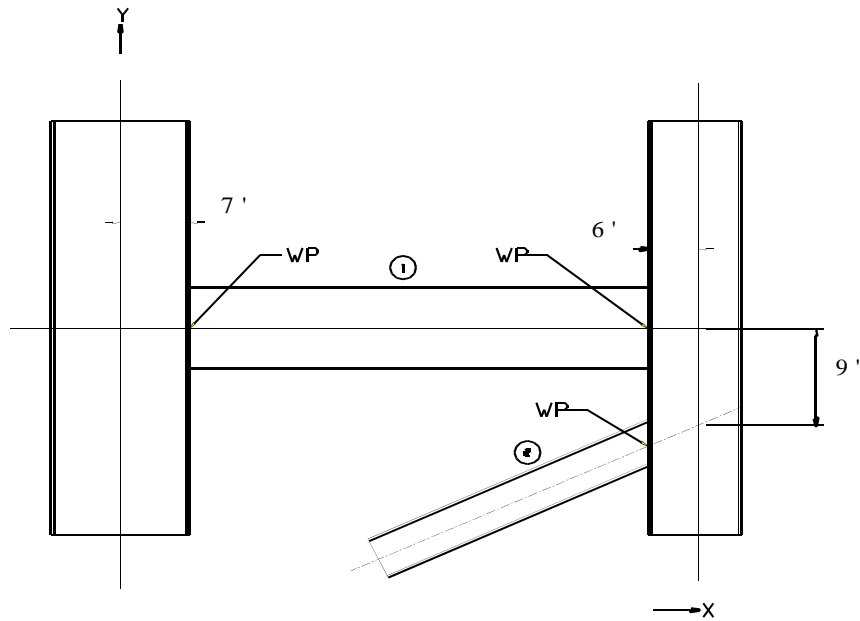
LOCAL is an optional parameter, if not put it in the f1, f2, f3 is then assumed to be global. LOCAL means that the distances f1, f2, f3, are in the same coordinate system that would result if the member was not offset member and BETA = 0.0.

Example

```
MEMBER OFFSET
1 START 7.0
0.0 -6.0 1 END
-6.0 -9.0 2 END
```

Notes :

1) If a MEMBER LOAD (see specification MEMBER LOAD) is applied to an individual for which the MEMBER OFFSETS specified, the location of the load does not measure the coordinates of the kickoff meeting. Instead, it is measured from the offset location of the kickoff meeting.
2) START and END is based on user specification MEMBER INCIDENCE for particular member.

5.26 Specification and Mapping Constants

The material constants are attributes such as modulus of elasticity and density required for operations like generating the stiffness matrix, compute themselves and for the design of steel and concrete weights.

In STAAD, there are two ways in which this data can be specified:

a) A two step process involving:

1. Creating material data by defining MATERIAL labels specified in section DEFINE MATERIAL (see "5.26.1 define Material")

2. Assigning them to individual members, plates and solid under the title CONSTANTS (see "Specifying constants for the member and Elements")

This creates commands as shown below:

```
DEFINE MATERIAL                                          PART 1
... NAME

...

...

END MATERIAL
CONSTANTS                                                PART 2
MATERIAL NAME ...
```

b) Assign material attributes explicitly specifying individual constants (see "Specifying constants for the members and elements").

```
CONSTANTS
E ...
POISSON .
```

5.26.1 Definition of Material Constants

Purpose

This command can be used to specify the properties of materials by the name of the material. Then assign members to these names and items of material in the CONSTANTS command.

General Format:

DEFINE MATERIAL

ISOTROPIC name

$\begin{Bmatrix} E \\ G \\ POISSON \end{Bmatrix}$

```
⎧ DENSITY  ⎫
⎪          ⎪  f₁
⎪ ALPHe    ⎪
⎩ DAMPING  ⎭
  CDAMP
```

ISOTROPIC Repeat the name and values for many materials desired then:

END MATERIAL (DEFInition)

- **Name: name of item (name up to 16 characters).**

- **E: Young's modulus specified.**

- **G: specifies the module Court. Only for beams between when the Poisson's ratio is not of 0.01 to 0.499.**

- **POISSON Specifies the Poisson ratio. If G is not entered, then this value is used to calculate the Court module (G = 0.5xE / (1 + POISSON)). Must be from .01 to .499.**

- **DENSITY:** specifies the weight density.

- **ALPHA: Coefficient of thermal expansion.**

- **DAMPING or CDAMP: damping ratio to be used in calculating modal damping method for damping compound.**

- **f₁: the value of the corresponding constants. For E, G, POISSON, DENSITY, ALPHA and damping.**

5.26.2 Specifying Constants

Purpose

This command can be used to specify the properties of materials (and Section Modulus, Poisson's ratio, density, linear expansion coefficient, and damping material) of the members and elements. In addition, this command also
can be used to specify the orientation of the member (BETA angle or point of reference).

General Format:

WITHStants

MATERIAL   *name* ⎡ **MEM**BER memb / elem-list ⎤
                    ⎣ (ALL)   ⎦

Where name is the name of the material as specified in the DEFINE MATERIAL command (see section 5.26.1).

No use with the previous post below. Use a separate command CONSTANTS.

```
⎡E       ⎤
⎢G       ⎥        ⎡MEMBER memb / elem-list  ⎤
⎢POISSON ⎥        ⎢                         ⎥
⎨DENSITY ⎬   f₁  ⎨BEAP                     ⎬
⎢BETA    ⎥        ⎢LA                       ⎥
⎢ALPlt   ⎥        ⎢SOLID                    ⎥
⎣has     ⎦        ⎣(ALL )                   ⎦

⎡REF f₂, F₃, F₄ ⎤     MEMBER memb / elem-list
⎣REFJT f₅       ⎦
```

Specification List: MEM, BEA, PLA, SOL, ALL. MEM only be followed by a list. Blanks mean ALL. ALL means all members and elements; BEA means all members; PLA, all plates; SOL, all solids.
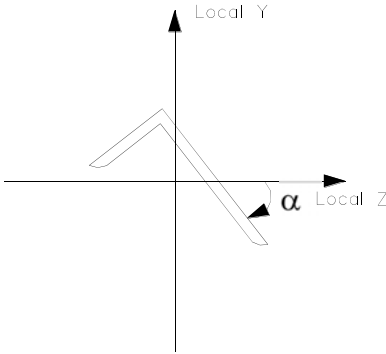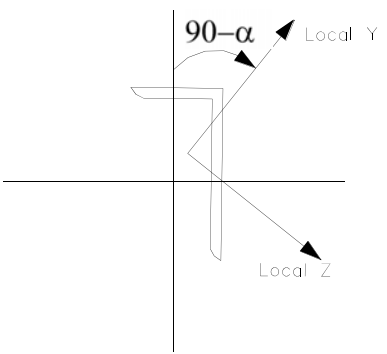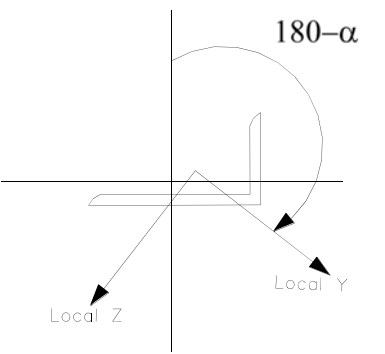
- **E-Specifies the Young's module. This value must be provided before POISSON for each member / element in the list of Constants.**

- **G: specifies the module Court. Among beams only when non–Poisson 0.01 to 0.499.**

- **POISSON Specifies Poisson's ratio. If G is not entered, then this value is used to calculate the Cutter Module (G = 0.5xE / (1 + POISSON)). Must be from .01 to .499.**

- **DENSITY:** specific gravity weight.

- **ALPHA: Coefficient of thermal expansion.**

- **CDAMP: damping ratio to be used in calculating modal damping method for damping compound.**

- **BETA: Specifies the rotation angle in degrees of the member (see Section 2).**
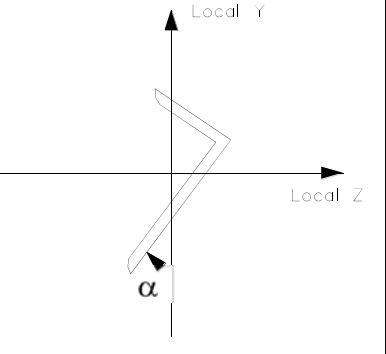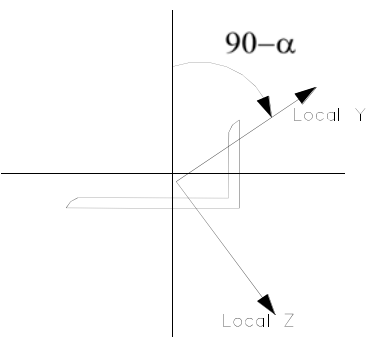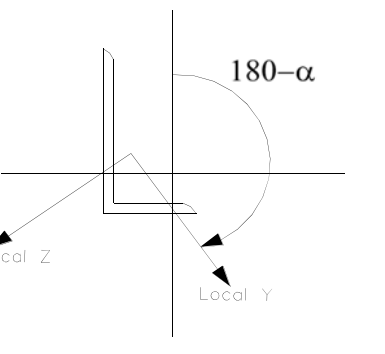
Note: single angle sections are oriented by default according to their main axes. If necessary, thereby orienting the sides are parallel to the overall axis BETA specification must be used. STAAD offers the following additional specifications for this purpose:

BETA BETA
ANGLE Rangle

The above two options will produce an orientation with sides parallel to the global axis. The 'ANGLE' option section rotates through the angle (90 - "alpha") (where "alpha" = the angle between the principal axis system and the global axis system). [Please check the figures on the next page.] The 'Rangle' option section rotates through an angle equal to (180 - "alpha"). Both

choices will work the same way for the same angles. For unequal angles, the correct option should be used based on the required orientation.

"ST" Angle	"ST" Angle	"ST" Angle
Beta = 0	Beta = ANGLE	Beta = RANGLE
Local X goes into the page, global Y-axis is vertical		

"RA" Angle	"RA" Angle	"RA" Angle
Beta = 0	Beta = ANGLE	Beta = RANGLE
Local X goes into the page, global Y-axis is vertical		
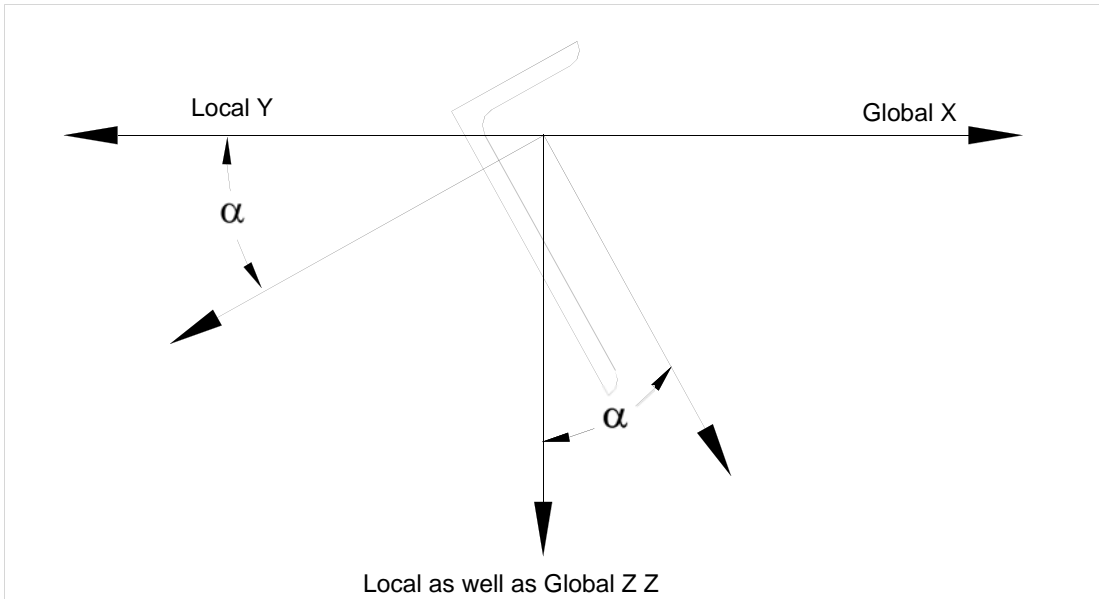
Orientation of a column (vertical element) for BETA = 0; Local X axis and Y Global depart from "paper" (Local X axis is parallel to the Y axis Global)
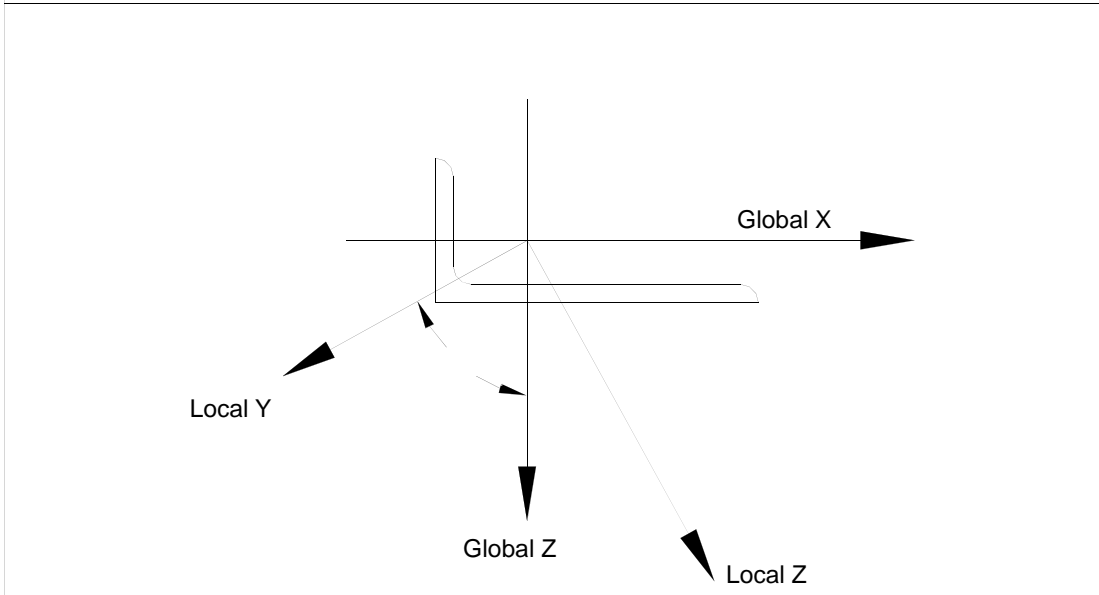
Orientation of one column (Item vertical) corresponding BETA = ANGLE; Local X axis and Y Global depart from "paper" (Local X axis is Global parallel to the Y axis)

The value f1 corresponding constants. For E, G, POISSON, DENSITY, and ALFA CDAMP, names can be entered in the material incorporated in place of f1. The names included are STEEL, ALUMINUM and CONCRETE. Appropriate values are assigned automatically (in units of the ALPHA degrees F) for catalog names.

CONSTANT FPS units				
Constants	Steel	Concrete	Aluminum	Units
E (US)	29000	3150	10000	Kip / in^2
Poisson	0.30	.17	.33	
Density	000,283	.0000868	000,098	Kip / in^3
Alpha	6.5e-6	5.5e-6	12.8E-6	L / L / deg-
CDAMP	.03	.05	.03	Ratio
E (nonus)	29732.736			Kip / in^2
CONSTANT in MKS units				
Constants	Steel	Concrete	Aluminum	Units
E (US)	199,947,960	21718455	68947573	kN / m^2
Poisson	0.30	.17	.33	
Density	76,819 541	612 23.561	26,601 820	kN / m^3
Alpha	6.5e-6	5.5e-6	12.8E-6	L / L / deg-
CDAMP	.03	.05	.03	Ratio
E (non-US)	205,000,000			kN / m^2

E (US) is used if the codes are installed US or "Member Properties American" is specified for analysis; otherwise E (non-US) is used.

f2, f3, f4 Global coordinates X, Y and Z for the benchmark or
f5 f5 use location point to the reference point, for which the angle BETA will be calculated by STAAD (see Section 1.5.3).

Example

```
DEFINE MATERIAL
ISOTROPIC CFSTEEL
E 28000.
POISSON
DENSITY 0.25
0.3E-3 ALPHA
DAMP 0.075
11.7E-6 END
MATERIAL

CONSTANTS
MATERIAL CFSTEEL MEMB 1 TO 5

ALL
CONSTANTS
E 2.1E5
45.0 BETA MEMB May 7 DENSITY
TO 18 TO 29 14 STEEL MEMB
BETA 90 X MEMB
```

Note that the last command in the above example will BETA as 90 ° for all members parallel to the axis X.

**Notes
:**

1) The value for E must always be first in the list of constants for each member / element.

2) Should be provided all numeric values in current units.

It is not necessary or possible to specify temperature units or ALPHA. The user must ensure that the value provided is consistent ALPHA thus refers to units with the value provided for the temperature (see Section 5.32.6).

If POISSON RATIO is not specified, STAAD assume Poisson's ratio based on the elastic modulus, E.

5.26.3 Modal Damping Information

Purpose

Define the only modal damping ratios for each mode. If all modes with the same buffer, then buffer between the Define Time History Load command or Dynamic Loading.

The damping here can be entered explicitly for some or all modes; EVALUATE STAAD or specifying that each mode based on buffer mode frequency and the minimum and maximum damping in here. The formula used for evaluating the buffer is given below.

The damping entered will be used in the time history load cases; and where Spectrum Response Charging using methods CQC / ASCE4 and / or spectra vs. Period curves against damping.

General Syntax

<u>DEF</u>INE DAMPING INFOrmation

$$
\begin{Bmatrix}
\underline{\text{EVAL}}\text{uate dmin (dmax)} \\
\\
\underline{\text{EXP}}\text{LICIT d1 (d2 d3 d4 d5 ...).}
\end{Bmatrix}
$$

END

1. Between dmin and dmax and the minimum and maximum ratio of buffer to be used in the formula below, respectively, or

2. Alternatively among d1, d2, d3, etc. as damping ratios for each mode. With each EXPLICIT option value can be preceded by a repeat factor (rf * damp) without the spaces.

    For example: EXPLICIT 7 * 0.05 0.04 0.03 <= buffer mode 1 is 0.03, modes 2-8 are 0.05, Mode 9 is 0.04. If there are fewer entries modes, then the damping which came last apply to other modes. This entry may be continued to 10 input lines with word EXPLICIT more only

line 1; end all lines but the last time with a space and a hyphen. There may be additional sets of lines before EXPLICIT PURPOSE.

Damping ratios can be used in the range of 0.0 to 1.0.

The formula used to EVALUATE (to evaluate the modal damping frequency) is: Damping for the first 2 modes is placed at the entrance to dmin. Damping modes for i = 3 to N given dmin and the first two frequencies 1 and 2 and the nth frequency modal i.

$$A_1 = Dmin / (\omega_1 + \omega_2) \quad A_0 = A_1 * \omega_1 * \omega_2$$

D (i) = (A0 / i) + (A1 * i) buffer for modes 3 and i = N.

If the resulting buffer is greater than the maximum value dmax quenching, dmax then be used.

Example: (for dmin = 0.02, 0.12 and dmax = 0 given below)

Mode	i	damping ratio
1	3	0.0200
2	4	0.0200
3	6	0.0228568
N	100	0.1200

(Calculated as 0.28605 then reset to the maximum input)

5.27 Supports Specifications

Specifications Staad support may be parallel or inclined to the global axes. The specification of a global axes parallel support is described in Section 5.27.1. Slants specification described in Section 5.27.2.

5.27.1 Specifying Global Support

Purpose

This set of commands can be used to specify conditions SUPPORT for global axes parallel to the supports.

General Format:

SUPPORTS

joint-list
```
    { PINNED                                    }
    { FIXED (BUT release-spec [spring-spec.])   }
    { ENFORCED (BUT release-spec)               }
```

$$
release\text{-}spec = \quad * \begin{Bmatrix} \underline{EX} \\ FY \\ FZ \\ \underline{MX} \\ \underline{MY} \\ MZ \end{Bmatrix}
$$

$$
spring\text{-}spec = \quad * \begin{Bmatrix} \underline{KFX} & f_1 \\ \underline{KFY} & f_2 \\ \underline{KFZ} & f_3 \\ \underline{KMX} & f_4 \\ \underline{KMY} & f_5 \\ \underline{KMZ} & f_6 \end{Bmatrix}
$$

Pinned and Fixed description

The PINNED support is a support having translational constraint, but no rotating. In other words, the support does not have capacity to bear now. A FIXED support has translational and rotational constraints. FIXED support can relax in the global addresses as described in the specification-relaxation (FX to force X through Z MZ to time). Also, a recessed support may have spring constants as described in the specification-spring (the spring in the global translational axis X and through the rotary KFX spring in the global Z and KMZ).

See Section 1.14

The corresponding spring constants are f1 to f6. Note that the rotational spring constants are always in degrees of rotation. All six relaxations can be provided as may be required to use the CHANGE command. If both the specifications of relaxation and spring specifications will be provided for the same joint support, relaxing specifications must come first.

Example

```
SUPPORTS
1 TO 4 July PINNED
June 5 FIXED BUT FX MZ
September 8 FIXED BUT MZ 50.0 KFY KFX 75.
18 21 FIXED
27 FIXED BUT KFY 125.0
```

In this example, the joints 1-4 and 7 are articulated joint. No moment is supported by those media. The seals 5 and 6 are fixed for the entire except DOF force in X and time Z. The gaskets 8 and 9 are fixed to all the DOF except when Z and has springs respectively in the X and Y global address with the corresponding spring constants of 50 and 75 units.

The seals 18 and 21 are fixed for all translational and rotational degrees of freedom. 27 At the meeting, all DOF are fixed except where FY DOF has a spring with spring constant 125 units.

Notes

:

1) Users are urged to refer to Section 5.36 for information on specifying SUPPORTS with specifications CHANGE command.

2) the spring constants must be provided at existing units.
3) All DOF springs must be entered after the last non-specified DOF spring if both are on the same line.

4) If there are two entries for the same board, then ,:
   a. any direction is pinned / fixed will be fixed at that address.
   b. any relaxed in one direction and a spring on the other it uses the spring.
   c. Any direction is pinned / fixed one and a spring on the other will use the pinned / fixed.

Enforced description

Support "Enforced" defines the translational and rotational directions, at a meeting, which may have an offset of tax support. Movements of support are defined for each load case in section 5.31.8. If no scrolling support is hard, then zero displacement will be imposed, as if that address was FIXED. The directions of movement of forces will be fixed to the dynamic load cases.

If there are two entries for the same board, then any direction will be given in force at that address, overwriting any other specification support for that direction of the board.

5.27.2 Specification Supports Inclined

Purpose

These commands can be used to specify media which are inclined with respect to the global axes.

General Format:

SU PPORT

joint-list INClined

```
⎧ f1 f2 f3      ⎫ ⎧ PINNED                                    ⎫
⎨               ⎬ ⎨                                           ⎬
⎪ REF f4 f5 f6  ⎪ ⎪ FIXED (BUT release-spec [spring-spec.])   ⎪
⎩ REFJT   f7    ⎭ ⎩ ENFORCED (BUT release-spec)               ⎭
```

Where:

f1, f2, f3 are the global distances x, y, z from the board to the "benchmark" or

f4, f5, f6 are the global coordinates x, y, z of the "benchmark" or

f7 is a number whose global coordinates x, y, z are the "benchmark".

A vector of the joint located at the location of the reference point used to define a local coordinate system (same as the member with BETA = 0). Addresses are inclined support this local "Axis System Support Tilt" (see below).

Note that the specification of relaxation and spring is the same as in the previous section (5.27.1). However, please note that FX to MZ and KMZ KFX to refer to the constant forces / moments and spring in the "Axis Tilt System Support" (see

below)

.

Reference point (1, -1, 0)
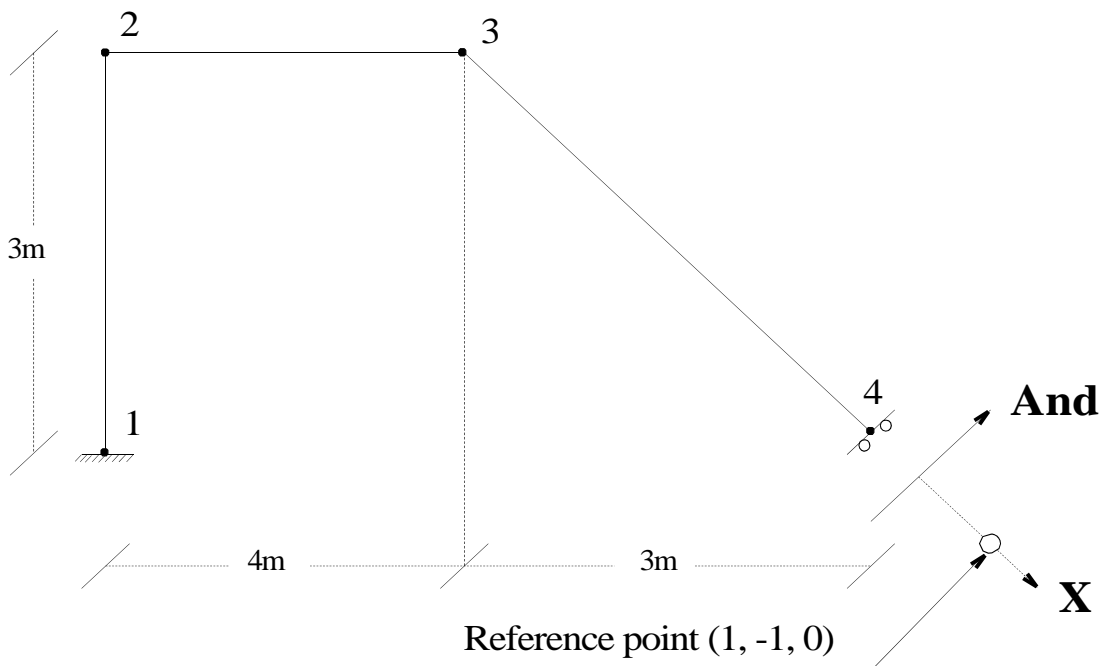
Mounting System Inclined Axes

SUPPORT INCLINED specification is based on the "axis system Mounting Inclined". The local x-axis of this system is defined assuming the inclined support joint as the source and linking it with a "point of reference" with coordinates f1, f2 and f3 (see figure) measured from the inclined support joint the global coordinate system.

The Y-axis and Z axis system tilt stand has the same orientation as the local Z axis and an imaginary member whose BETA ANGLE is zero and whose incidents are defined as inclined to support joint benchmark. Users can refer to section 1.5.3 of this manual for more information on these concepts.

Example

```
SUPPORT
4 INCLINED 1.0 -1.0 0.0 FIXED BUT MY M FY MX Z
```

Notes: addresses a tilt stand is assumed to be the same as global to calculate some dynamic results and intermediate results by UBC (eg global participation factors). If the mass and / or forces in the directions of the inclined free carriers is a relatively small portion of the overall force, then the

effect must be very small.

5.27.3 Automatic generator type supports Spring for Foundations

STAAD has the option for automatic generation of media springs to model foundations and slab foundations. This command is specified under the SUPPORT command.

General Format:

SU PPORT

THE A joint
$\left\{ \begin{array}{l} \underline{FOO}ting \ f1 \ (f2) \\ \underline{M \ AT} \end{array} \right\}$ $\underline{DIR}$ $\left\{ \begin{array}{l} X \\ \underline{XO}nly \\ \underline{And} \\ \underline{YO}nly \\ \underline{Z} \\ \underline{ZO}nly \end{array} \right\}$ $\underline{SUB}grade \ P \ f3 \ RINT$

Where

f1, f2 = length and width of the foundation. If f2 is not given, the foundation is assumed as a square with sides f1

f3 = Modulus of sub-grade soil in units of force / area / length X, Y, Z

= Global Direction in which the soil springs are to be generated

Do not use this commando with Z SET UP.

The choice of ELASTIC FOUNDATION: If you want to specify the area of influence of a board and has STAAD simply multiply the area you specified by the modulus of sub-grade, use the FOOTING option. Situations where this may be appropriate are as scattered when a foundation is located on a board where you want to specify a spring support. Please note that it is absolutely essential that you provide the f1 (and f2 if it is a non-square foundation) if you choose the option FOOTING.

The choice of ELASTIC BED: If you want to make STAAD calculate the area of influence for the board (instead of specifying an area) and use the area along with the module sub-degree to determine the value of spring stiffness, use the MAT option. Situations where this may be appropriate are as when a slab is on the earth and carries the weight of the structure above. You may have shaped the entire slab with finite elements and want to generate props springs at the nodes of the elements.

The choice of Address: The DIR keyword is followed by one of the letters of the alphabet X, Y or Z (or XONLY, YONLY or ZONLY) which indicates the direction of resistance of spring supports. If X or Y or Z is then select a spring support is generated at that address plus 3 other addresses receiving a fixed support, eg. If Y is selected, then FY is supported by a

spring; FX and FZ and MY are the fixed support; and MX and MZ are free. If XONLY, YONLY or ZONLY are selected then only one spring support at that address is generated.

Subgrade option: A keyword followed by the subgrade reaction value of the subgrade. Please note that the value should be provided in the current drive system meaning the most recent previous statement to SUPPORT UNIT command.

The PRINT option: Print the area of influence of each joint.

Example

```
SUPPORTS
1 TO 126    ADDRESS AND ELASTIC MAT 200 subg.
```

The above command tells STAAD supports generating internally to all nodes from 1 to 126 with elastic springs. STAAD calculates perpendicular first area of influence the overall axis of each joint and then multiplies the area of influence for the respective sub-module 200.0 soil degree calculated spring constant to be applied to the knot.

Notes
:

- A closed surface is generated based on the list of the board that accompanies the ELASTIC MAT command program. The area within this closed surface is determined and the portion of this area for each node in the list is then calculated.

- Hence, while you are specifying the list of boards, one must ensure that these together constitute a closed surface. Without proper closed surface, the calculated area for the region may be indeterminate and values of spring constants may be erroneous. Therefore, the list should have at least 3 knots ,.

- The interior angle formed by two adjacent segments connecting 3 consecutive nodes in the list must be less than 180 degrees. In other words, the region will have the form of a convex polygon. The example below explains the method can be used to reach a situation where a convex polygon is unavailable.

For model comprised of plate elements 100 to 102 in the figure below, one may want to generate spring supports 1 to 8 knots with one ELASTIC MAT command will not be enough, because the angle between the inner 1-8 and 8-7 edges to node 8 are 270 degrees that violate the requirements of a convex polygon. So, one should interrupt it in 2 commands:

- January 2nd August 3 ELASTIC MAT 200 DIREC And subg.
- 3 4 5 6 7 8 ADDRESS AND ELASTIC MAT 200 subg.
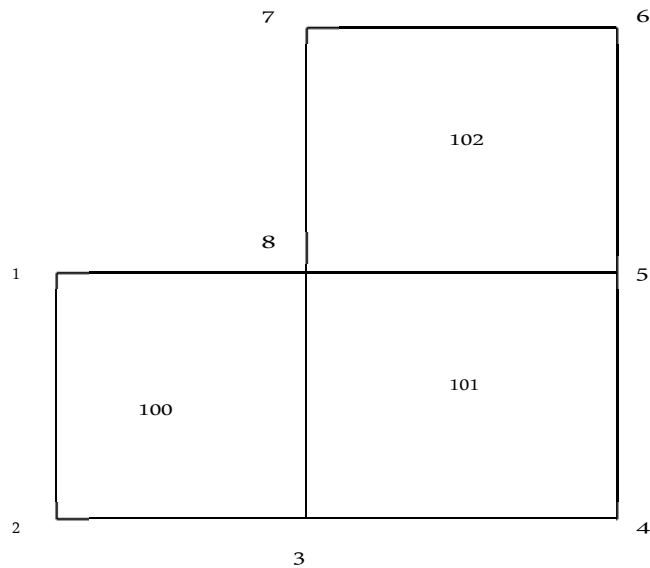
Boards 3 and 8 will reach here to contribute to both commands above. The command only works when the plane of the closed region is parallel to one of the global planes XY, XZ or YZ. For regions that are inclined to one of the global planes, the spring constant must be evaluated by hand and using the specified type of support FIXED BUT spring.

5.27.4 Specification Support Multi-Linear Type Spring

When the soil is modeled as spring supports, can be modeled variable resistance offered by external loads using this option, much as when its behavior is tension or compression.

General Format: MULSPR

    TILINEARINGS

    joint SPRINGS d-list$_1$s$_1$d$_2$s$_2$D ......$_n$s$_n$

Where (di si) pairs represent displacement and spring constant of the pairs (**Yup** is zero or positive). The first pair defines the large spring constant negative displacement offset above the second pair. The last pair defines the spring constant for displacements greater than the shift in the last couple.

Example

```
UNIT. . .
SUPPORT
1 PINNED; February 4 FIXED BUT SPRINGS KFY 40.0
multilinear
April 2 SPRINGS -0.51 40.0 -0.50 50.0 0.5 65.0
```

The load-displacement characteristics of the ground can be represented by a multi-linear curve. The amplitude of this curve represent the soil spring characteristics to different displacement values. A typical floor spring characteristic curve can be represented as step as shown in Figure below. In the above example

the command specifies multi-linear spring spring soil on boards 2 and 4 [Note that
the amplitude of the pitch curve does not change after the first point.]

Each load case in a multi-linear analysis should be separated by a CHANGE command and should have its own PERFORM ANALYSIS command. There can be no PDELTA, NONLIN, dynamic, or analysis of TENSION / COMPRESSION included. The command of multi-linear spring start an iterative cycle analysis and convergence check. The cycles continue until the RMS or effective proportions of used springs remain virtually the same for two consecutive cycles.

You must have entered previously SUPPORT springs during each spring in here. During the first cycle, the value used is the value of the spring in the spring (non zero displacement here). Use a realistic and stable value. All values in a spring board will support multi-linear curve with this one.

Note: This command may be continued up to 11 lines running all but the last with a hyphen. Dots and commas and xrange, yrange, ZRANGE listed items may not be used.
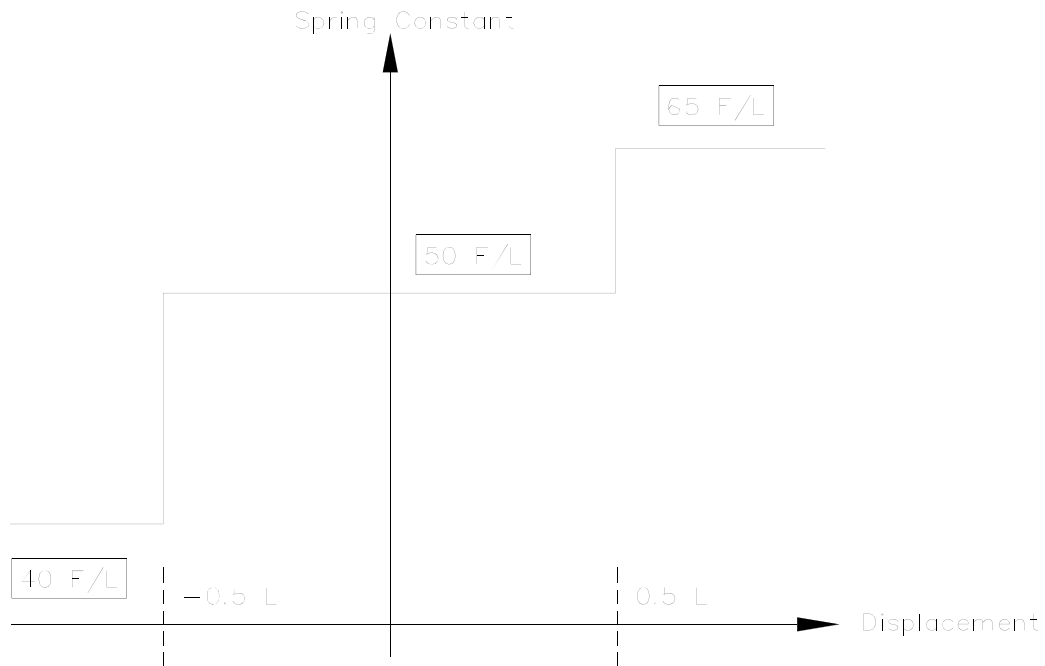
F = Force Units

L = Length Units

The spring constant is always positive or zero.

5.28 Specification Master / Slave

Purpose

This command set may be used to model the special joints (displacement ligation, the rigid links) through joints specification MASTER and SLAVE. Please read the release notes for restrictions.

General Format:

$$
\text{SLAVE} \quad * \begin{Bmatrix} \underline{XY} \\ YZ \\ \underline{ZX} \\ RIGID \\ \underline{FX} \\ FY \\ \underline{FZ} \\ MX \\ \underline{M} \\ \underline{Y} \end{Bmatrix} \quad \underline{MAS}\text{TER} \quad j \quad \underline{JOI}\text{NT} \quad \text{joint-spec}
$$

$$
\text{joint-spec} = \begin{Bmatrix} \text{joint-list} \\ *\underline{XRA}\text{NGE YRANGE} \quad f_1, \\ F_2 \underline{ZRANGE} \end{Bmatrix}
$$

Description

The choice of master / slave provided in STAAD allows the user to model the special junctions (linkage of displacement, rigid links) in the system. For example, SLAVE FY ... connecting the two together so that the displacement of the slave will be Y

And sum of displacements of the master over the rigid rotation R SIN.
θ

Note that instead of maintaining a list of boards worked like a slave, a range of values of the coordinate (in the global system) can be used. It is assumed that all joints whose coordinates are within the range will be worked as slaves together. For convenience, the range of the specified coordinate for the joints worked as slaves on that entry may include the board of the master. However, master and slave boards of other inputs should not be included in the range of the coordinate. All 2 or 3 ranges can be entered into a "tube" or "box" for selected region on the tube or box joints.

Fx, Fy etc. are the addresses which are slaves (any combination can be input).

If two or more entries have the same master, the slave list will be mixed. It is also OK to have different specified addresses.

Specifiers direction (XY, YZ, ZX) are combinations of the basic directions, XY is equal to log FX, FY, MZ ,; etc. Any combination of address specifiers can be entered. An example of using this format is a floor rigid diaphragm which still retains the flexibility of the entry ... ZX SLA flexion.

If RIGID or if all addresses are provided, the boards are supposed to be rigidly connected.

Restrictions

- Solid elements can not be connected together slaves. Together
- Teachers can not be slaves in another entry.
- Boards Slaves can not be master or slave in another post.
- Directions Slaves in the joints can not be addresses media or have imposed displacements.
- Together Teachers and / or slaves may not be inclined supports.
- The master / slave specification is only for linear static and dynamic analysis.

Example Fully Rigid and Rigid Floor diaphragm Day

```
RIGID JOINT MASTER SLAVE 22 10 TO 45
RIGID JOINT MASTER SLAVE 70 YR 25.5 27.5
SLA  ZX  MAS  80  JOINT  YR  34.5  35.5
```

5.29 Switching frequency vibration modes

Purpose

These commands are used in conjunction with dynamic analysis. They can be used to specify the highest frequency or the number of mode shapes need be considered.

General Format:

See Section 1.18.3

$$
\begin{cases}
\underline{\text{FREQ}}\text{uency} \\
f_1 \underline{\text{CUT}} \ \text{(OFF)} \\
\underline{\text{MODE}} \text{ SHAPE } i_1 \\
\underline{\text{TIME}} \qquad\qquad t_1
\end{cases}
$$

Where,

- f1 = the highest frequency (cycle / s) to be considered for dynamic analysis.
- i1 = The number of mode shapes to be considered for the dynamic analysis. If the command is not cut off frequency provides the cut off frequency will default to 108 cps. If the cut off form so no command is given, the first six modes were calculated. These commands must be provided above the specifications of loads.

t1 = Ending time for analysis over time. If zero (the default), the time history will end when the last show of force over.

Note that maximum i1 mode shapes is calculated despite f1. If during the test of convergence of 0 to f1 f1 frequencies converge, the modal calculation will be completed before the mode shapes of i1 is calculated.

5.30   Definition   charging systems

Purpose

This section describes the specifications necessary to define the various charging systems for automatic generation of Mobile Uploads, UBC Seismic loads and wind loads. In addition, this section also describes the specification of load time history for time history analysis.

Description

STAAD has incorporated algorithms to generate moving loads, the lateral seismic loads (the Code Uniform Building Code) and wind loads on a structure. The use of load generation option consists of two parts:

1) System definition (s) of cargo.
2) Case generation system using primary load (s) as defined above load.

See Section 1.17

The definition of the system (s) loading must be provided before any primary load case specified. This section describes the system specification (s) of cargo. Information on how to generate primary load cases using the system (s) defined load is available in Section 5.32.12.

UBC loads do not consider the effects of forces entirely inclined support addresses or addresses of the slave board. The forces applied to these situations can introduce errors which are generally small.

5.30.1   Definition of cargo moving system

Purpose

This set of commands can be used to define the charging system in motion.

General Format:

DEFINE M OVING AS AD   (FILE file-name)

$$\text{TYPE } j \begin{cases} \text{LOAD } f_1, F_2, .f_n (\text{DISTANCE } d_1, D_2, .. D_{n-1} \ (\text{WIDTH } w)) \\ \text{load-name} \quad (F) \end{cases}$$

(DISTANCE d_1, D_2, .. D_{n-1} (WIDTH w))

See Section 31.5.12

Note that LOAD MOVING system can be defined in two ways directly into the input file or an external file.

The FILE option should be used only in the second case when the data will be read from an external file. The file name should be limited to 16 characters.

Mobile charges may be generated for frame members only. They can not be generated for finite element. Note: All loads and distances are in units of the current system.

Define Loads moved tr or give them the input file

Use the first specification TYPE. The input data must be all on one line (as shown above) or two sets of lines. If two games, then the second game should start with DIS as shown below. If two games, then the Load and Dis lines can end each line but the last of each game with a script (see example).

T YPE j LO AD f_1, F_2, F ...$_n$ (Set 1) DISTANCE d_1, D_2, ... $D_{(N-1)}$ (WIDTH w)
**   (Set 2)**

Where,

 j   = Number of mobile charging system (integer limit of 100 types) n = number of loads (eg axles), 2-50.

 fi = value. ith load conc.

 di = the distance between the (i + 1) th and the ith charging load in the direction of movement

 w = space between loads perpendicular to the direction of motion. If omitted, a dimensional load is assumed. (For example the width of the vehicle). Note that this parameter will double the total load as the fi is applied to each wheel.

Definition of Mobile Charging using an external file

Use the second specification TYPE.

T YPLoad-name E j (f)

Where,

Load = name The name of the mobile charging system (maximum 12 characters).
and   f =   Optional Factor multiplied to scale up or down the value of loads. (Default = 1.0)

The following is a typical file containing the data.

 CS200   ----- Name system load (load-name must begin in column 1)

 ---- 50 80 90 100 loads (79 characters in the input line)
 7 7 9.   ----- Distance between charges (one line)
 6.5   ----- Wide

50   80   100

90

6.5   7.0   7.1   9.0
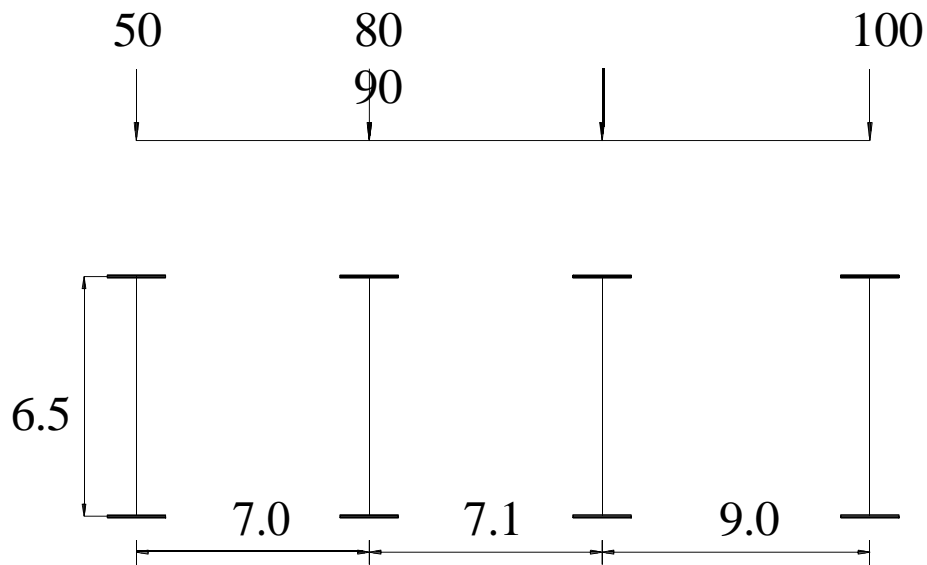
Note that some charging systems can be repeated within the same file. The mobile generator charges STAAD involves:

1) All loads are acting on the overall direction vertical (Y or Z) negative. The user is advised to fix the structural model accordingly.
2) The resultant in the direction of movement is determined by the increments of movements of X, Y and Z as they were provided by the user.

<u>Loads of rock references to</u>

First concentrated load specified in mobile charging system as the reference load is designated. While primary load subsequent instances are generated, the initial position of the charging system and the movement direction is defined with respect to the reference load condition. Also note that selecting the load situation with a positive reference value specified width, then the following two views define the reference load situation.
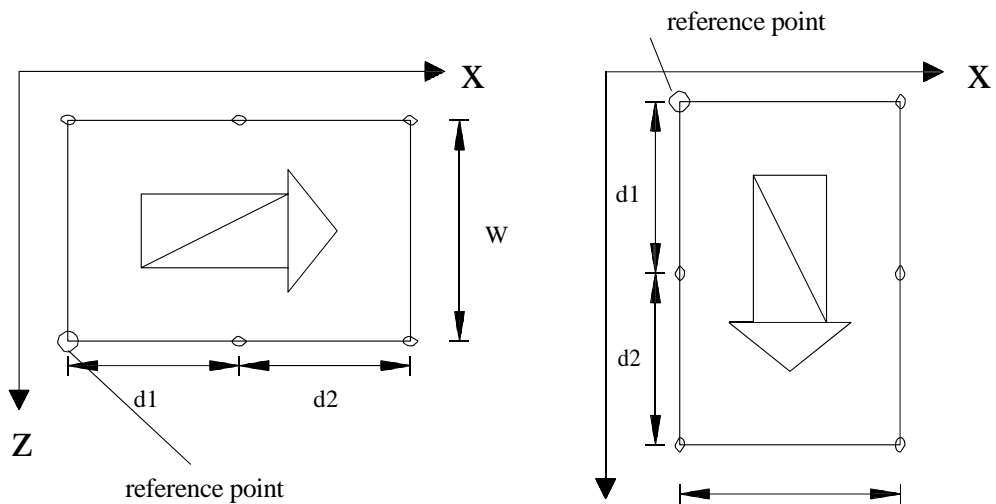
Z

Movement parallel to the overall axis X   *Movement parallel to the global Z*

Z

Note that in the left view, the point of reference is in the wheel track of the positive side of Z; while in the right view, the point of reference is the smallest positive footprint X side of the wheel.

Specifying standard AASHTO loads

General Format:

$$\text{TYPE i} \quad \begin{Bmatrix} \underline{HS2}0 \\ \underline{HS1}5 \\ H20 \\ \underline{H1}5 \end{Bmatrix} \quad (F) \quad (V)$$

where,

i = Mobile Charging System type number. (Integer).

f = Optional multiplication factor (default = 1.0)

vs = spacing of the variable as defined by AASHTO, stop HS series trucks (default = 14 ft.)

Example

```
DEFINE  MOVING  LOAD
TYPE  1  LOAD  10.0  20.0  -
15.0  10.0
DISTANCE  5.0  7.5  -
6.5  WIDTH  6.0
TYPE  2  HS20  0.80  22.0
```

Example:

When the data is provided through an external file called MOVLOAD

```
Data in input file

UNIT  . . .
DEFINE MOVING LOAD FILE TYPE 1
MOVLOAD AXLTYP1
TYPE 2 AXLTYP2 1.25

Data in external file MOVLOAD

AXLTYP1
10 20 15
5.0 7.5
6.0
AXLTYP2
20 20
10
7.5
```

5.30.2 Definitions for Static Force Procedure for Seismic Analysis

STAAD provides the means to determine the lateral loads acting on structures due to seismic forces, using rules codes available in several national publications and widely accepted. The codes allow publications to use what is called static force equivalent methods instead of more complex methods such as response spectrum analysis and history through time. Among the many such codes is supported by STAAD UBC, IBC, IS 1893, AIJ, etc.

Once the lateral loads are generated, the program can then analyze the structure for those loads using the rules explained in the code documents.

5.30.2.1   UBC load definition 1997

Purpose

This feature allows you to generate seismic loads by 97 UBC specifications using a static approximation equivalent.

Description

The seismic load generator may be used only to generate the lateral loads in the X and Z directions and is the direction of gravity loads. This option has not been developed for cases where the Z axis is positioned to be the vertical direction using the "SET Z UP" command.

Methodology

The design base shear is calculated in accordance with the code section UBC 1630.2.1 The primary equation 1997 called, 30-4, is checked as shown below.

$$V = \frac{C_v I}{RT} W \quad \text{...................... UBC Ecuación30-4 1997}$$

In addition, the following additional equations are checked: Equation

30-5 - The overall design of the base shear should not exceed:

$$V = \frac{2.5 C_{to} R}{W R}$$

Equation 30-6 - The overall design of the base shear should be no less than:

$$V = 0.11 C_{to} IW$$

Equation 30-7 - In addition, for Seismic Zone 4, the total base shear shall not be less than:

$$V = \frac{0.8ZN_vR}{W\ R}$$

For an explanation of the terms used in the above equations, please refer to the 1997 UBC code.

There are 2 phases of specification command to generate lateral loads. This is the first phase and is activated through the DEFINE UBC LOAD command.

General Format

> **DEFINE UBC (ACCIDENTAL) LOAD**
> **ZONE f1 ubc-spec**
> **SELFWEIGHT**
> **JOIWEI NTGHT**
> *Joint-list* **WEIGHT w**
> **MEMBER WEIGHT**
> *Mem-list (UNI ....)*
>
> *ubc-spec = {I f2, f3 RWX, RWZ f4, f5 STYP, NA f6, f7 NV (CT f8), (PX f9), (PZ f10)}*

Where:

- Seismic coefficient f1 = f2 = area
- Importance Factor
- f3 = Numerical coefficient R for lateral loads in the X direction
- f4 = Numerical coefficient R for lateral loads in the Z direction
- f5 = Type of Soil Profile
- Nearby f6 = source factor Na =
- f7 source factor Nv Near
- Optional f8 = CT value for calculating the period of time based on Method A
- f9 = Optional period structure (in sec) in the X direction to be used in Method B
- f10 = Optional period structure (in sec) in the Z direction to be used in the
  Method B

The type parameter STYP soil profile can take values from 1 to 5 These are related to the values shown in Table 16-J of the 1997 UBC code as follows:

Value STAAD	Code 1997 value UBC
1	S_A
2	S_B
3	S_C
4	S_D
5	S_E

Soil type SF profile is not supported.

The seismic zone factor (ZONE) along with the type of soil profile (STYP) factor of the near source (Na), and the Middle power factor (Nv), are used to determine the values of seismic coefficients Ca Cv and Tables 16-Q and 16-R of the 1997 UBC code.

Examples

```
DEFINE UBC LOAD
0.38 ZONE  I  1.0  STYP  2 RWX RWZ 5.6 5.6 NA 1.3 1.6 CT NV 0.037
JOINT
SELFWEIGHT
        WEIGHT
WEIGHT 51 56 93 100 1440
WEIGHT 101 106 143 150 1000
151 156 193 200 720 WEIGHT
```

The steps for calculating the basal cutting are:

1. The time period of the structure is calculated based on the clause 1630.2.2.1 (Method A) and 1630.2.2.2 (Method B).
2. The user can overwrite the program period calculated using the B method specifying a value for PX or PZ (the f9 and f10 items) depending on the direction of the burden of UBC. The specified value will be used instead of the calculated using Method B.
3. The Time Period ruler of the structure is then chosen between the two periods mentioned above-on the basis of the guidance provided in clause 1630.2.2.2.
4. The coefficients Ca and Cv are calculated in Table 16-Q and 16-R.
5. The Basal Shear Design is calculated based on the clause 1630.2.1 and distributed on each floor using the rules of clause 1630.5.
6. ACCIDENTAL If the option is specified, the program calculates the time of additional torque. The lever arm to calculate the torque is obtained as 5% of the dimension of the building at every floor level perpendicular to the direction of loading of UBC (clause 1630.6). Each board where a weight is located, the lateral seismic force acting on that board it is multiplied by the lever arm for the torque of that meeting.

The following example shows the commands required to enable the program to generate lateral loads. Users can refer to the section LOAD GENERATION Reference Manual for this information.

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
UBC LOAD X 0.75
LOAD 2 (LOAD IN SEISMIC Z
DIRECTION) UBC LOAD   Z 0.75
```

UBC / IBC input can be provided in two or more lines using the following symbol (hyphen) as shown in the following example:

```
ACCIDENTAL DEFINE UBC LOAD
```

```
ZONE 3000 -
I RWX 1.00 1.100 -
RWZ 1.40 NA 5.000 1.200 NV STYP 1.50 CT -
PZ PX 1,300 2,100 2.20
```

5.30.3 Defining Loads Of Wind

Objective

This group of commands is used to define the parameters for the automatic generation of wind loads on the structure.

General Format

DEFINE WIND LOAD

TYPE j

INTEnsity p1 p2 p3 ... pn HEIGHT h2 h3 h1 ... hn

EXP{E1 OSURE JOINT joint-list | YRANGE f1 f2 | ZRANGE f1 f2}

Where:

- J: number system load due to wind (integer)
- p1, p2, p3, ... pn wind intensity (pressure) force over area.
- h1, h2, h3, ... hn Elevations (on the absolute coordinate) to which the above intensities occur
- e1, e2, e3, ... in Exposure Factors. The default is 1.0, which means that due to wind force is applied to the entire area of influence associated with the node or nodes
- *joint-list: List of nodes associated with the exposure factor*
- f1 and f2 values of the (global) vertical coordinates that specify the vertical range of exposure.

If you do not specify the EXPOSURE command, the selected exposure factor is 1.0.

Description

All loads and elevations are defined in the system of units in use. In the list of the intensities, the first intensity value acts from the ground level to the first elevation. The second intensity (P2) acts on the absolute direction, between the two first heights (h1 and h2) and so on. The program assumes that the level of the floor has the lowest coordinate of axis Y.

The exposure factor (e) is the fraction of the area of influence associated with the nodes or on which the load acts. The total load acting on a particular node is calculated as follows:

Load at node = (exposure factor) X (area of influence) X (wind strength).

The exposure factor can be specified by a list of nodes, or giving a range for Y, within which all nodes have the same exposure. When the exposure factor is not specified, the default value is 1.0, in which case will be considered complete influence area associated with the nodes.

For plane frames, the area of influence for each of the nodes is calculated considering the perpendicular to the plane of the structure unit width. Note that you can tailor the actual width, when incorporated into the exposure factor as follows:

Exposure Factor (Spec. By User) = (fraction of the area of influence) X (width of influence for node)

Note: All currents, heights and ranges must be defined in the system of units in use.

Example:

```
UNIT FEET
DEFINE WIND LOAD TYPE 1
INTENSITY 0.1 0.15 HEIGHT
December 24
EXPOSURE 0.90 yrange November 13
EXPOSURE JOITN 0.85 17 20 22
WIND LOAD 1 LOAD IN X-DIRECTION
WIND LOAD TYPE 1 X 1.2
```

5.30.4 Definition of analysis "over time" (land consolidation)

Objective

This group of commands is used to define the parameters for the step load of the structure analysis.

General Format

DEFINE TIME HISTORY (DT x)

TYPE i { ACCELERATION | FORCE | MOMENT } (SCALE f_7) (SAVE)

{ t_1 p_1 t_2 p_2 ... t_n p_n | *function-spec* | *spectrum-spec* | READ *filename* (f_8) }

Repeat the TYPE command and places Amplitude vs Time until all commands are entered, then:

ARRIVAL T I ME

to$_1$to$_2$to$_3$To ....$_n$

(D AMPING d | CDAMP | MDAMP)

The time history (time history) can be explicitly defined using pairs of time and acceleration values, force or moment, where:

- Acceleration indicates the type of load time variation is an earthwork.

- Strength indicates a forcing function

- Time indicates a forcing function of time

Variable or Command	Default Value	Description
x	–	Solving time step used in the step of integration coupled equations. Values smaller than 0.00001 is placed at DT default 0.0013888 seconds.
i	–	Type varying number of load versus time (integer). They can provide up to six different types.
Scale f7	1.0	The choice of the scale factor multiplies all were, accelerations and amplitudes entered, read or generated within this type. Used primarily to convert acceleration g's to current units (9.80665, 386.08858, etc)
Save	–	The save option is to create a file (input file with TIM extension) that contains the history of the displacement of each node of the structure at each time step. Syntax: TYPE 1 FORCE SAVE
t_1, P_1; t_2, P_2	–	Values of time (sec) and corresponding force or acceleration dependent on whether the load variation with time is a forcing function or ground movement. They can provide up to 499 pairs for each type, values

		ascending time. It may use more than one line if required.
$to_1 to_2 to_3 ... A_n$	–	Values of the different possible arrival times (seconds) for the various types of dynamic loads. The arrival time is the time when a guy starts acting load on a node (forcing function) or at the base of the structure (ground motion). The same type of load may have different arrival times for different nodes, where these values should be specified here. Arrival times and the couple force - time to load types are used to create the necessary load vector for each time step of the analysis. See 5.32.10.2 for specifying information input in the application of loads or loads force functions ground motion. You can specify up to September 99 arrival times.
d	0.05	Modal damping ratio. Can only use a damping ratio for the entire structure.

Note that the option "function-spec" is used to specify harmonic loads. You can specify harmonic sine and cosine functions. The program will automatically calculate the harmonic load history based on the following specifications.

For the amplitude function and
option:

> *function-spec =*
> { <u>SINE</u> | <u>COSINE</u> }
> <u>AMPLITUDE</u> f_0 { <u>FREQUENCY</u> | <u>RPM</u> } f_2 (<u>PHASE</u> f_3) <u>CYCLES</u> f_4 { <u>SUBDIV</u> f_5 | <u>STEP</u> f_6 }

Where:

- f_1 - Range of motion in units in use.

- f_2 - If FREQUENCY then circular frequency (cycles / sec) If RPM, then rpm.

- f3 - Phase angle in degrees, or default.

- f4 - No. of load cycles.
- f5 - Time load step, default = one tenth of the circular frequency.

Example:

```
UNIT FT
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
TYPE 2 ACCELERATION
0.0 2.5 0.5 2.7 1.0 3.2 1.5 3.8
2.0 4.2 2.5 4.5 3.0 4.5 3.5 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Note: The 'READ THFILE' command will be provided only if the history of load variation with respect to time is read from an external file. THFILE Where is the file name. The data in the external file must have in each line of the torque - time as shown in the following example.

Data in the input file UNIT KIP

```
FEET
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
DAMPING 0.0 0.075

```

Data in the external file "THFILE" 0.0

```
1.0
1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6
5.0 2.8
```

Example of generation of harmonic loads

```
UNIT KIP
DEFINE TIME HISTORY TYPE 1 FORCE
* Following lines for Harmonic Generator Loading SINE
FUNCTION
60 AMPLITUDE FREQUENCY CYCLES 100 6.2831 0.02 ARRIVAL
TIME STEP
0.0
DAMPING 0.075
```

Note: The response (. Displacements, forces etc.) contain only the contribution of those modes whose frequency is less than or equal to 60 cps. The contribution of modes with frequency greater than 60 cps is not considered. For more information about the physical behavior of the soil from which the values entered in this section

see the book of Soil Mechanics Volume I of Juárez-Badillo (*Chapter 10: The phenomenon of one-dimensional consolidation of soil*)

5.30.5 Defining snow loads

This set of commands can be used to define some of the parameters for the generation of snow loads in the structure. See 5/31/13 snow loads Generation section for the definition of additional parameters and surfaces that will be charged.

General Format

> **DEF**INE SNOW LOAD
>
> **TYP**E f1 **PG** f2 **CE** f3 **CT** f4 **IM** f5

Where:

- f1: Type number (limit 100). The "type number" is an integrated value (1, 2, 3, etc.) denoting a number by which the type of snow load will be identified. Multiple types of snow load can be created in the same model.

- f : Snow load on the ground (default = 0.0). The pressure or weight per unit area, 2 is used to calculate the design snow load. Use a negative value to indicate that the load is acting above the roof (down).

- f3: exposure factor (default = 1.0). Depends on the type of exposure the roof (fully exposed / exposed partially / fully refugee) and category of land.

- f: thermal factor (default = 1.0). Depends on the thermal condition.
  4

- f: importance factor (default = 1.0). This value depends on the category in which five the structure belongs.

Example

```
FLOOR START GROUP
DEFINITION
_ROOFSNOW TO 102 TO 153 159 160 170 179 195 TO 197
DEFINITION GROUP END
UNIT DEFINE POUND
FEET SNOW LOAD
TYPE 1 PG 50 EC CT 0.7 1.1 1.1 IM
```

5.30.6 Definition of types of reference loads

Larger models can include multiple load cases that require analysis in their own right and are simply the building blocks for inclusion in the main load cases. Thus reference loads may be defined for this purpose. This is similar to

REPEAT LOAD command (see Section 5.31.11), but has the advantage of not being resolute in his

own right.

This is a real case of the burden becomes something similar to a load case definition. A reference load case is solved only when later he is called in a load case. The advantage is that you can define as many load cases as you like, but tell the program that actually solves only a limited number of cases of "real" charge number, which limits the number of results should be examined.

See "Load Cases Reference - Application" on page 272 for a description of the procedure to specify the information of the reference load in cases of active load.

General Format

```
REFERENCE DEFINE LOADS

LOAD R (i) LOADTYPE (type) TITLE load_title

(Load items)

.
.
.

REFERENCE END DEFINE LOADS
```

Example

```
REFERENCE DEFINE LOADS

LOAD R1 LOADTYPE SELFWEIGHT DEAD DEAD

AND TITLE REF -1

JOINT LOAD

FY 4071 4083 4245 4257 -4.04

4090    FY    -

0.64 ELEMENT LOAD

378 TO 379 406 TO 410 422 TO 426 PR GY -1.44

MEMBER LOAD

5006 TO 5229 UNI GY -0.64 PMEMBER

LOAD

1 TRAP GY -0,347 -0,254 35.5 42 LIVE

LOAD R2 LOADTYPE JOINT TITLE LIVE

LOAD REF

FY 4209 -6.63

FY 4071 4083 4245 4257 -1.71

SNOW LOAD R3 LOADTYPE JOINT TITLE SNOW

LOAD REF

FY 4109 -8.69

FY 4071 4083 4245 4257 -3.29

LOAD REF R4 LOADTYPE SOIL SOIL ELEMENT TITLE TO

LOAD 1367 1394 1396 1398 1522 1539 TO 1574 -

TRAP 1575 JT -0.78 -0.78 -0.719167 -0.719167

R4 LOADTYPE LOAD MODEL MASS MASS TITLE

SELFWEIGHT X1

SELFWEIGHT Y 1

SELFWEIGHT Z1

JOINT LOAD

FY 17 TO 48 -2.5

FY 49 TO 64 -1.25

REFERENCE END DEFINE LOADS
```

5.30.7 Definition of direct analysis of members

This set of commands can be used to define which members the bending stiffness and the axial stiffness are considered to contribute to the lateral stability of the structure. Also   b the initial value for each member can be placed here. See paragraph 7 of ANSI / AISC 360-05.

Members listed with FLEX have their EI to 0.80 times the factored   b while the overall solution is made. The final member forces and with verification code will be 100% of the bending stiffness. Members listed with SMD have their EA factored to

1.80 while the overall solution is made. The final member forces and code
100% verification will the axial stiffness. The value Fyld only be used for the calculation of   b.

General Format

```
DEFINE DIRECT
FLEX (f₁) list-spec
AXIAL list-spec
FYLD (f₂) list-spec
NOTIONAL LOAD FACTOR (f₃)
list-spec = * { XR f₄ f₅ | YR f₄ f₅ | ZR f₄ f₅ | MEMBERS mem-
list | LIST memb-list | ALL }
```

Where:

F1: τb value. Default = 1.0. See Appendix 7 of ANSI / AISC 360-05

F2: yield stress in the current units.

F3: nodal load factor. The default is 0.002. itself is entered a value of 0.003 or higher, then   b be equal to 1.0 and no iterations are performed.

F4, F5: upper and lower range values, respectively, used when a range is specified by FLEX or Fyld.

To specify nominal loads, please see Section 31.5.14. Nominal loads and factor used is entirely specified load data.

Notes :

a) the notional LOAD FACTOR f3 command instructs the program not create nodal loads. Instead, it is a parametric control using only to make the following decision:

- Yes f3 is 0.002, the calc program ula τb iteration on a base.

- Yes f3 is greater than 0.00299, the program sets the limit to one iteration and does not perform an additional iteration.

b) Tb is the value entered by the FLEX command. Tb omitted to 1.0 itself is not entered.

5.31 Loads Specification

Objective

This section describes the different options available load in STAAD. The following commands can be used to start a new load case.

General format:

<u>LOA</u>I DING$_1$(Any load title)

Where:

- i1 = any integer (up to five digits) that identifies the case load. This number need not be sequential with respect to numbers of previous load.

The LOADING command starts a new load case. Under this heading, you can give all the different charges related to this charge number. These different types of loads are described below.

5.31.1 Specification of nodal loads

**Objectiv
e**

This group of commands is used to specify the loads of nodes in the structure.

General Format:

<u>JOI</u>NT LOAD

*joint-list * {FX f7 | FY f8 | FZ f9 | MX f10 | MY f$_{11}$ | MZ f$_{12}$}*

Where:

- FX, FY and FZ: specify an absolute force in the appropriate direction. MX,
- MY and MZ: specify a time corresponding absolute address f1, f2, ... f6: load
- values

Example

```
JOINT LOAD
3  TO  7  9  11  FY  -17.2  MZ  180.0
5  8  FX  15.1
12  MX  180.0  FZ  6.3
```

Note: The numbers of nodes can be repeated, in the place where loads are to be added to the node.

5.31.2 Specification of cargo in the Member

Objective

This group of commands is used to specify the loads on frame members.

General Format:

MEMBER LOAD

member-list {{UNI | UMOM} dir-spec f1 f2 f3 f4 | {WITH | CMOM}
dir-spec f4 f5 f6 | LIN dir-f7 f8 f9 spec | f spec dir TRAP 10 ^{f11}

f12 f13}

Where:

dir-spec = {X | Y | Z | GX | GY | GZ | PX | PY | PZ}

- X, Y, and Z specifies the address of the load in the x, y, and z local (member). •

GX, GY and GZ specifies the address of the load in the X,

Y, and Z Global.

- PX, PY, PZ can be used and the burden is projected along the length of the member in the corresponding global address.

- UNI or UMOM Specifies a uniformly distributed load or a moment with a value of f1, f2 at a distance from the start of the member at the beginning of the load; f3 and distance from the start of membership at the end of the load. The load is assumed to cover the entire length of the member when f2 and f3 are omitted.
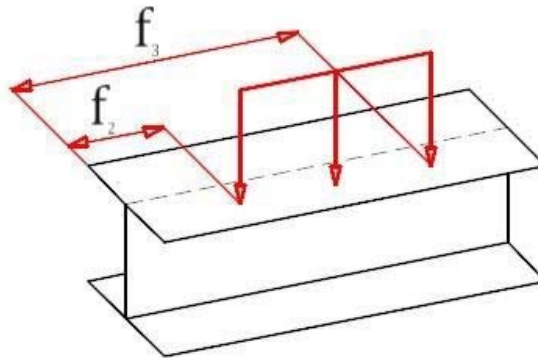
- WITH or CMOM Specifies a concentrated force or moment worth f5 f6 applied to a distance from the start of the member. f6 have as default, half the length of the member.

- f4 perpendicular distance from the shear center to the loading member. The value is positive in the general direction of the parallel (or nearly parallel) local axis
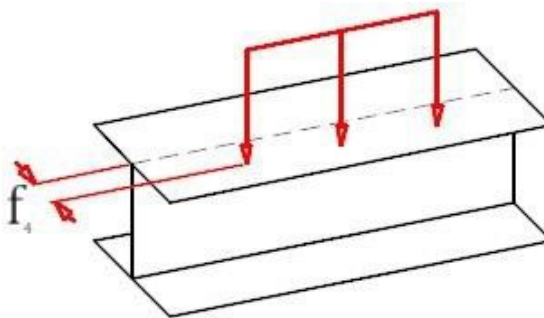
- LIN specifies an increase or decrease linear or a triangular load. If the load increases or decreases linearly, then f7 is the value at the beginning of the member and f8 is the value at the end. If the load is triangular, then f7 and f8 are given as f9 is zero and the value of half the loading member.

- Specifies a trapezoidal load TRAP linear variation, which can act on the partial or total length of a member and a local, absolute address or projected. The load value at the start is given by f10 and the load value at the tail end by f11. The location of the load is given by f12, the start point of the load, and f13, the endpoint. Both are measured from the start of the member. When n is not specified, f12 and f13, the load is assumed to cover the entire length of the member.

Note: The specification of the global axes is not permissible for linear loads (LIN option). If the member is loaded lag distance (see the specification of MEMBER OFFSET), the location of the load is not measured from the coordinates of the start node but the displaced distance.

5.31.3 Specifying load on an element

Objective

This group of commands is used to define the different types of loads on plate elements.

General Format

ELEMENT LOAD

PRESSURE element-list {{GX | GY | GZ} p1 (x1 y1 x2 y2}}

element-list <u>TRA</u>P *{GX | GY | GZ | LX | LY} {X | Y} f1 f2*

The PRESSURE option should be used when you need to specify a pressure UNIFORM. The uniform pressure can be set on the entire item, or a specific portion (defined by x1, y1 and x2, y2. See description below). If you do not provide x1, y1 and x2, y2, pressure is applied to the entire item. If only provided x1, y1, assume that the load is concentrated load applied to the specified point.

Note that the pressure can be defined either in the global addresses (GX, GY, GZ) or (normal to the element) Local Z direction. If the global address is omitted, the applied load is assumed at the local address is Z.

Where:

*   GX, GY, GZ: denotes overall direction for the pressure in the X, Y and Z respectively
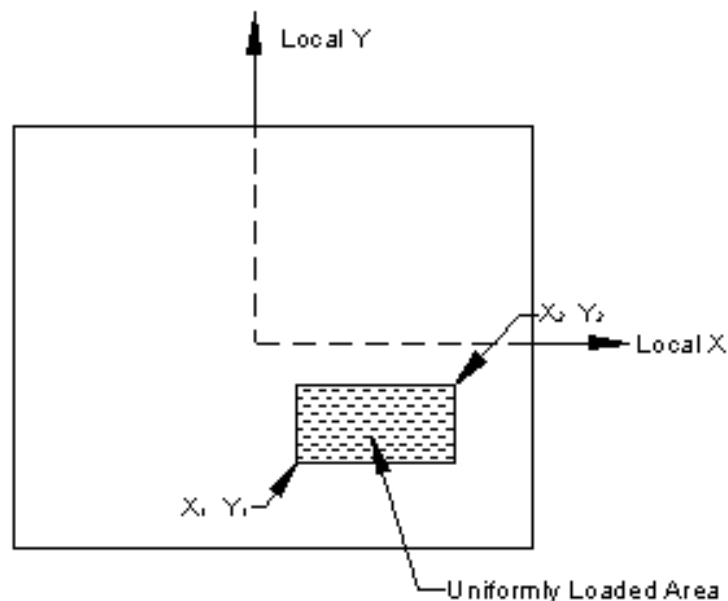
- f1: Pressure element (force / square length) or concentrated load (force).

  Note that f1 is considered as a concentrated load when x2 and y2 are omitted

- x1, y1 & x2, y2: coordinate points in the local coordinate system (the central node is the source) defining a rectangular area on which the pressure is applied.

- The TRAP option should be used when needed to define a linearly varying pressure. The variation must be defined on the whole element.

- X or Y address of variation pressure element. The TRAP option X / Y indicates that the variation of the trapezoid is the Local direction X or Local direction Y. The load always acts in the local Z axis

- f2: pressure intensity first part

- F3: Intensity of pressure at the end of
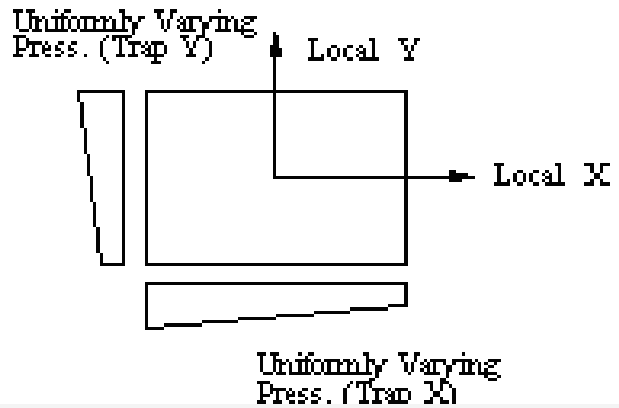
Note:

1. The start and end defined in the previous part is based on the positive directions of the local axes X or Y.
2. While X or Y direction indicates the variation of the V, the load acting on the local direction Z.

Example

```
4 ELEMENT
LOAD LOAD
1  7  TO  10  PR  2.5
11  12  PR  2.5  1.5  2.5  5.5  4.5
15  TO  25  TRAP  X  1.5  4.5
34  PR  5.0  2.5  2.5
35  TO  45  PR  -2.5
```

5.31.4 Specifying load floor and surface charge

**Objectiv
e**

These commands are used to specify Area Loads Loads condo or a structure. The surface charge is used to model a distribution in one direction and the load distribution to model a floor in two directions.

General format for LOAD AREA:

AREA LOAD

memb-list **ALO**AD f1 {GX | GY | GZ }

- f1 = value of load acting on an area (unit weight of unit length squared). This load is always acting along the Y-axis positive local members for analysis FLAT, this direction coincides with the axis in the absolute majority of cases. (For a detailed description refer to Section 2).

Example

```
AREA LOAD
February 4 TO 8 ALOAD -0.250
December 16 ALOAD -0.500
```

Note: The loading area should not be specified as members declares two MEMBER CABLE, TRUSS MEMBER or MEMBER TENSION.

General Format for FLOOR LOAD:

FLOOR **LOAD**

YRANGE   f1   f2   **FLOF**3 {AD XRANGE f4 f5 ZRANGE f6 f7}

Where:

- f1 f2 values in global vertical coordinates, to specify the vertical range. The floor load will be calculated for all members who are included in the global horizontal plane vertically within the specified range overall.

- The value of f3 floor loading (weight per unit length of the square). This load always acts parallel to the vertical axis absolute. A positive value means

the load   is acting in the overall positive direction Y. A negative value indicates an overall negative charge in the direction Y.

- f4-f7 absolute values of xyz coordinates that define the corner points of the area over which the load specified da floor (f3) act. If not specified, the load floor will be calculated for all members on all floors within the overall vertical range.

Notes:

1. The structure must be shaped such that the vertical axis remains perpendicular to the overall plane of the floor.

2. Specifying Load Floor considered for load distribution in two directions, The specification of the surface charge considered an action in one direction.

3. The load floor slab is distributed on four sides as trapezoidal or triangular loads depending on the length of the sides as shown in the diagram. Internally, these charges are converted to multiple loads.
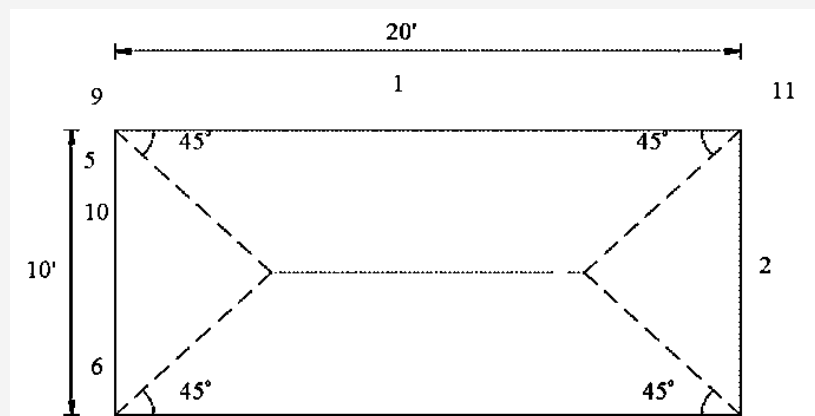
4. The total charge applies only to quadrilateral and triangular slabs or panels. The load per unit area can not be varied for a particular panel and then be assumed to be continuous without holes.

5. The FLOOR LOAD option is not available if the SET Z UP command is used (See Section 5.5.)

Example

The input LOAD FLOOR explained through an example. Consider the

following home plan at y = 12 '.
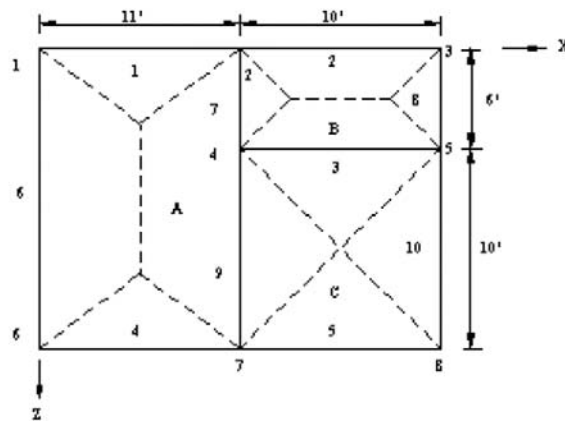
If the floor has a full load of 0.25 (force / unit area), then the entry would be as follows:

```
. . .
LOAD 2
FLOOR LOAD
YR 12.0 12.0 0.250 FLOAD
. . .
```

If in the above example, panel A has a charge of 0.25 and panel B and C have a charge of 0.5, then the entry will be as follows: Note the format of the xrange and yrange specifications.

```
. . .
LOAD 2
FLOOR LOAD
YR XR 12.0 0.0 12.0 11.0 0.250 FLOAD ZR 0.0 16.0
YR XR 12.0 12.0 11.0 21.0 0.5 FLOAD ZR 0.0 16.0
LOAD 3
. . .
```

The program identifies internally panels (shown as A, B and C in the figure above). The floor loads are distributed as trapezoidal and triangular loads as shown by the dotted lines in Fig. The negative sign to the charge means is applied downward on the global address Y.

5.31.5 Prestressing load specification

Objective

This command is used to define prestressing loads acting on the structure members.

General Format

$$\underline{\text{MEM}}\text{BER } \{\text{PRESTRESS } | \text{ POSTSTRESS}\} \text{ (LOAD)}$$
$$\textit{memb-list } \underline{\text{FOR}}\text{CE } f_1 * \{\text{ES } f2 \mid \text{EM } f_3 \mid \text{EE } f4\}$$

Where:

- $f1$ = prestressing force. This value is positive according to the direction of the local axis x

- ES = Specifies eccentricity prestressing force member to start a centroid distance $f2$

- EM = Specifies eccentricity prestressing force to the member to a half of the distance from the centroid $f3$

- EE = Specifies eccentricity prestressing force at the end of the member at a distance from the centroid $f4$

Description

The first option, LOAD memeber Prestress, considers the effect of prestressing force during application. Thus, the transverse shear generated at the ends of the member subject to pre-stressing forces is transferred to the adjacent members.

The second option (POSTSTRESS MEMBER LOAD) considers the effect of existing prestressing load after prestressing operation. Thus the cross-cutting at the ends of the member subject to pre-stressing force is not transferred to adjacent members.

Example

```
Prestress 2 MEMBER
TO FORCE July 11
50.0

MEMBER POSTSTRESS
8 FORCE IS 30.0 -6.0 3.0 EM EE 3.0
```

In the first instance, a force of pre-stress of 50 kips is applied through the centroid (i.e., no eccentricity) of the members 2 to 7 and 11 In the second example, a force of post - stress of 30 kips is applied with an eccentricity of 3 inches in the initial part, -6.0 half inches and 3 inches at the end of the end 8.

One of the limitations in the use of this command is that under any load case, any member can only apply a load of prestressing and post - effort only once. If the member has given multiple prestressing cables or have a condition of prestressing or post load - effort, this situation will have to be specified through multiple load cases for that member. See the example below.

Incorrect Entry

Entrada Correcta

```
LOAD 1
Prestress MEMBER
July 6 FORCE 100 GB 2 -3 EM EE 2
6 FORCE EM 150 GB 3 3 -6 EE PERFORM
ANALYSIS
```

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
6 FORCE 150 ES 3 EM -6 EE 3
LOAD COMB 3
1 1.0 2 1.0
PERFORM ANALYSIS
```

Examples of Modeling Techniques

The following examples describe the partial data entry for member profiles and cables shown below.

Example 1

```
JOINT COORD
1 0 0; 2 10 0
INCI MEMB
1 1 2
...
UNIT
LOAD 1
...
POSTSTRESS LOAD
1 FORCE EM 100 GB 3 3 -3 EE
 PERFORM ANALYSIS
```
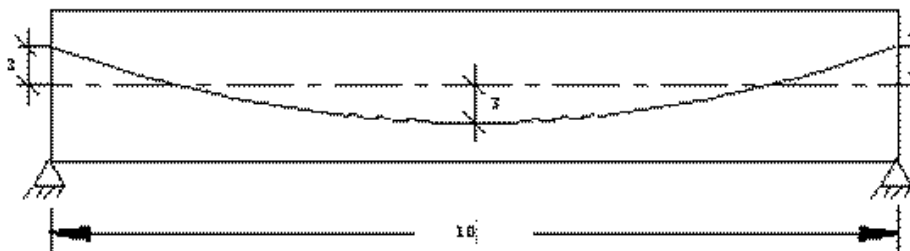
Example 2

```
JOINT COORD
1 0 0; February 20 0
INCI MEMB
1 1 2
...
Prestress INCH
UNIT LOAD 1
LOAD
1 FORCE MS 100 EN -3 -3 -3 EE
PERFORM ANALYSIS
```
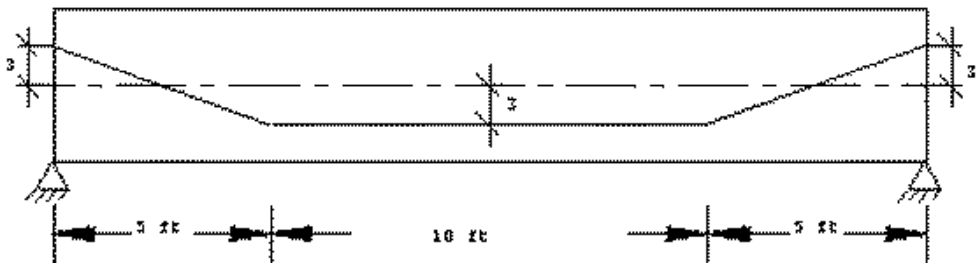

Example 3

```
JOINT COORD
1 0 0; 2 5 0; 3 15 0 0; April 20 0
INCI MEMB
1 1 2; 2 2 3; 3 3 4
...
1 INCH UNIT
LOAD
Prestress LOAD
1 FORCE MS 100 EN 3 0 -3 EE
2 FORCE MS 100 EN -3 -3 -3 EE
3 FORCE MS 100 EN -3 0 3 EE
PERFORM ANALYSIS
```

Example 4
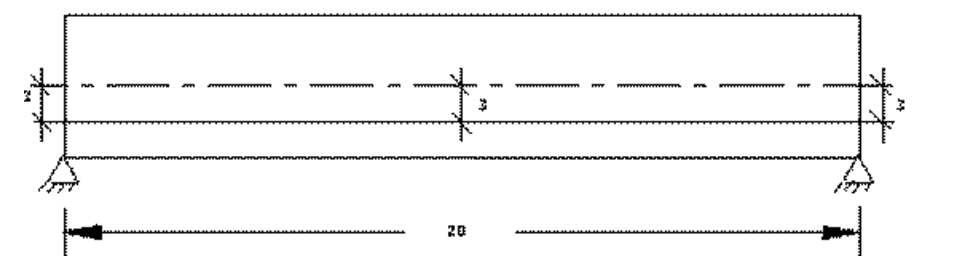
```
JOINT COORD
1 0 0; 2 10 0; March 20 0 0
INCI MEMB 1 1 2; 2 2 3
...
Prestress INCH UNIT
LOAD 1 LOAD
1 FORCE MS 100 EN 3 0 -3 EE
2 FORCE MS 100 EN -3 0 3 EE
PERFORM ANALYSIS
```
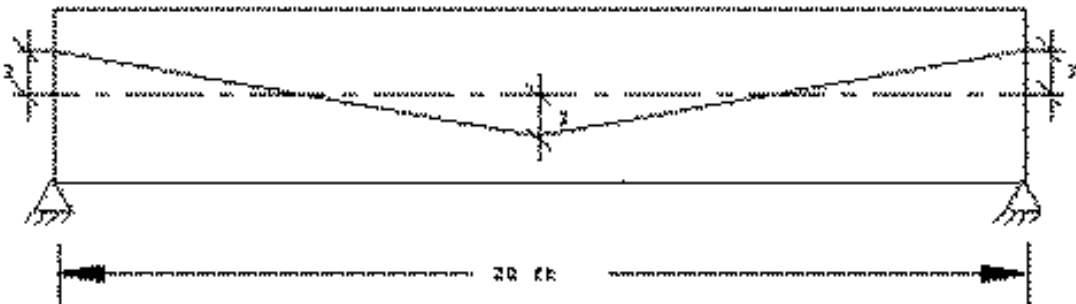

Example 5

```
JOINT COORD
1 0 0; 2 10 0; March 20 0 0
INCI MEMB
1 1 2; 2 2 3
...
1 INCH UNIT
LOAD
Prestress LOAD
1 FORCE MS 100 EN 3 3 -3 EE
2 FORCE MS 100 EN 3 -3 3 EE
PERFORM ANALYSIS
```
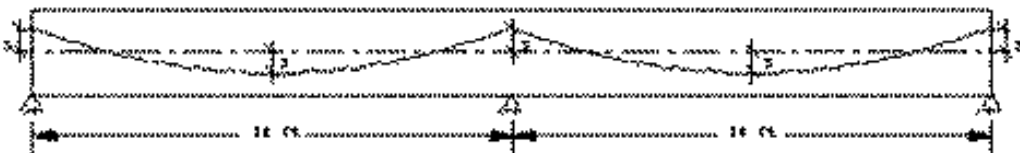
5.31.6 Load Specification Due to Temperature

Objective

This command is used to specify the loads due to temperature or strain loads on the members and elements.

General Format

$$memb\ /\ elem\text{-}list\ \{TEMP\ f_1 f_2 f_4\ |\ STRAIN\ f_3\ |\ STRAINRATE\ f_5\}$$

Where:

- $f1$ = The temperature change that produces axial expansion in Member or uniform volumetric expansion elements. The unit of temperature is the same as that selected for the thermal expansion coefficient ALPHA under command CONSTANT

- f_2= Temperature difference from top to the bottom element of a member or part. (T $_{upper\ surface}$ -T $_{lower\ surface}$). The bending will not be considered when f is omitted$_2$

- $f3$ = initial axial elongation (+) / contraction (-) in members due to misalignment, etc. in the unit length

Example

```
TEMP LOAD
1 TO 15 September 17 TEMP 70.0
TEMP TO 23 90.0 18 66.0
8 TO 13 0.45E-4 STRAIN
```

Note: It is not necessary or possible to specify the units of temperature or ALPHA. The user must ensure that the value provided to ALPHA is consistent in terms of units provided for charging temperature. (See Section 5.26)

5.31.7 Specifying Load On A Far Recessed

Objective

This command is used to define the loads acting on a flush-end framing members.

General Format

FIXED (END)   LOAD

member-list f_1, F$_2$, ... F$_{12}$

Where:

- f1 ... f6 = Force x, shear and z shear, torsion, time and time z (all in local coordinates) at the start of the member.

- f7 ... f12 = Same as above except that the final end of the member

5.31.8 Specifying Load Shift Support

Objective

This command is used to specify the load displacement of supports of the structure.

General Format

SUPPORT DISPlacment

joint-support-list {FX | FY | FZ | MX | MY | MZ F1}

Where:

- f1 = value of the corresponding displacement. For translational displacement, the units are specified das according to the unit of length in use, while for rotational displacement units will always be degrees.

- FX, FY, and FZ specifies n translational displacements in the global addresses X, Y and Z, respectively. MX, MY and MZ n specified rotational displacement absolute addresses X, Y and Z.

With this command, the shift in support is provided as a burden. Note that the offset can not be specified in a direction in which the support is free.

Exampl
e

```
SUPPORT UNIT
INCHES displ
5 TH November 13 FY -0.25
19 21 TO 25 MX 15.0
```

In this example, the nodes in the first list of supports will be displaced 0.25 inches in overall negative direction, while supporting the second list will be rotated by 15 degrees around the axis X. Overall

> **Notes: The offsets can be applied only in 4 load cases. Displacing support should not be applied in a finite structure containing elements.**

5.31.9 Specifying Load Due To Dead Weight

**Objectiv
e**

This command is used to calculate and apply the weight of the structure in the analysis.

General Format

```
SELFWEIGHT ({X | Y | Z} f1) (LIST member-list | ALL | _group name)
```

Where:

- f1 = factor used to multiply the weight of the structure

- X, Y and Z represents the overall direction in which acts the weight of the structure.

This command is used when you are going to consider the weight of the structure. The weight of each active member is calculated and applied as a uniformly distributed load member.

This command can also be used without specifying factor and address. So, if specified as "SELFWEIGHT" loads will be applied in the negative absolute direction with a unit factor.

> **Notes: The density shall be indicated for calculating the weight of the structure. The own weight of finite elements is converted to charge on nodes connected nodes and not used as a load element pressure.**

5.31.10 Dynamic Load Specification

**Objectiv
e**

Specifying command that is needed to perform the spectrum analysis and step is explained in the following sections.

5.31.10.1 Specifying the Response Spectrum

**Objectiv
e**

This command is used by the user to specify and implement a response spectrum in the dynamic load analysis.

General Format

```
SPECTRUM {SRSS | ABS | CQC | ASCE | TEN | CSM | GRP * {X}
f1 | Y f2 | Z f3 {ACC} | DIS } (SCALE f4)
```

```
{DAMP f5 | CDAMP | MDAMP} ({LIN | LOG}) (MIS f6) (ZPA f7) (FF1
f8) (FF2 f9) ({DOMInant f10 | SIGN}) (SAVE) (IMR f11) (STARTCASE
f12)
```

Where:

- $= f1 \ldots f3$ are the factors for the input spectrum, which will be applied in the X, Y, and Z. You can specify some or all of the addresses. When an address is provided its default value is zero.

- $f4 =$ Scale factor for which the response spectrum is modified. If not specified, the default value is 1.0.

- $f5 =$ Damping factor. The default value is 0.05 (5% damping). This value is required only for the CQC method.

- ACC or DIS = na is referred acceleration or displacement.

- P1, V1, P2, V2, ...; Pn, Vn = Default periods (seconds) and the corresponding acceleration (unit length / sec 2) or displacement (using length in use) as the case requires. More than one line may be used if necessary. The use of the hyphen (-) at the end of line is not allowed to continue on the next line data. Pairs of spectra must be provided in ascending period value with a maximum of 99 pairs of spectrum.

Description

Note that if the SPECTRUM SRSS, manners used combinations are made according to the method of SRSS (square root of the sum of the squares). Otherwise, the CQC (full quadratic combination) method is used.

This command must appear as part of a load specification. If the first, you must be accompanied by the load value that will be used to calculate the frequency and vibration modes. The Additional instructions do not require additional information. The maximum response spectrum load cases allowed execution is 4.

The results of calculations of frequency and vibration modes can vary significantly depending on the mass modeling. All active mass must be modeled as loads. All masses which are capable of movement, should be modeled as loads applied in all directions of movement.

In response spectrum analysis all bodies that are able to move in the direction of the spectrum should be provided as loads acting in that direction.

Example
:

```
2 LOAD SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0 Z 1.0 JOINT
LOAD SELFWEIGHT FX 10 17.5
FY 10 17.5
FZ 10 17.5
SPECTRUM SCALE ACC SRSS 1.0 X 32.2
0.20 0.2; 0.40 0.25; 0.60 0.35; 0.80 0.43; 1.0 0.47
1.2 0.5; 1.4 0.65; 1.6 0.67; 1.8 0.55; 2.0 0.43
```

Multiple Response Spectra

When defined in the input file more than one response spectrum, the load data must accompany only the first group of spectral data. Where subsequent load, only the spectrum must be defined. See the example below.

```
1 LOAD SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
FY 10 17.5
FZ 10 17.5
SPECTRUM SCALE ACC SRSS 1.0 X 32.2
0.20 0.2; 0.40 0.25; 0.60 0.35; 0.80 0.43; 1.0 0.47
1.2 0.5; 1.4 0.65; 1.6 0.67; 1.8 0.55; 2.0 0.43
*
2 LOAD SPECTRUM IN Y-DIRECTION
SRSS 1.0 ACC SPECTRUM AND SCALE 32.2
0.20, 0.1; 0.40 0.15; 0.60 0.33; 0.80 0.45; 1.00 0.48
1.20 0.51; 1.4 0.63; 1.6 0.67; 1.8 0.54; 2.0 0.42
*
3 LOAD SPECTRUM IN Z-DIRECTION
SPECTRUM Z SRSS 1.0 SCALE 32.2 ACC
0.20, 0.2; 0.40 0.25; 0.60 0.35; 0.80 0.43; 1.00 0.47
0.5 1.20; 1.4 0.65; 1.6 0.67; 1.8 0.55; 2.0 0.43
```

5.31.10.2 Applying Loads for Variable Time History Response Analysis

**Objectiv
e**

This command set is used to model the registration period the load on the structure for analysis step. Note that registration can provide both nodal and period ground motion.

General Format

> TIME LOAD
>
> *joint-list * {FX | FY | FZ | MX | MY | MZ }* $I_t I_{to} f_2$ GROUND
> MOTION {ABS | (REL)} {X | Y | Z} $I_t I_{to} f_2$

 Where:

It = type number for the load varying over time (see Section 5.30.4) Ia = number of time of arrival (Section 5.30.4) (integer).

This is the sequential number of response time in the list provided in section 6.31.4. Thus, the number of arrival time of a3 is 3 and is of an n.

Note that whether or GROUND MOTION TIME LOAD, or both may be specified under a load case. It's not allowed for a load case for the "over time" analysis.

Example

```
LOAD 1
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
1.0 MEMBER LOADS
SELFWEIGHT Z
5 WITH GX 7.5 10.0
5 CON GY 7.5 10.0
GZ 5 WITH 7.5 10.0
LOAD TIME
2  3  FX  1  3
5  7  FX  1  6
GROUND MOTION
X  2  1
```

In the above example, permanent masses in the structure are specified in the form of das own weight and charges for membership mode vibration and frequency. The rest of the data is input to the application of loads that vary with time in the structure. A force function type 1 is applied to the nodes 1 and 3 beginning with the time of arrival number 3 (the number 3 is the time of arrival of 1.8 seconds in the example presented in Section 6.31.4). Similarly, the forcing function of type 1 is applied to the nodes 5 and 7 starting with the

number six arrival time (4.4 seconds). Ground movement (type 2) acts on the structure in the x direction beginning with the arrival time number 1 (0.0 seconds).

5.31.11 Repetitive Loads Specification

**Objectiv
e**

This command is used to create a case of primary load cases using the combination of predefined primary charge.

General Format

$$\underline{REP}EAT \quad \underline{LOA}D$$

$$i_1, \ F_1, \ I_2, \ F_2 I \ \ldots_n, \ F_n$$

Where:

- i1, i2, ... in = number of cases of primary charge •
f1, f2, ... fn = corresponding factors

Description

This command can be used to create a primary load case combinations using predefined load cases. The repeated load command differs from the load merge command (see Section 5.34) in two ways:

1. a repetition of charge is treated as a new primary charge. Therefore, P-Delta analysis, reflecting the correct side effects. (The combinations of load, on the other hand, the effects of combined algebraically predefined primary loads, independently assessed).
2. In addition to primary loads previously defined, the user can add new load conditions.
3. The REPEAT LOAD option is available with load cases with JOINT LOADS and LOADS MEMBER containing specifications UNI and CON UMOM only. It is not available to MEMBER LOADS LIN specifications and TRAP. It can be used to load cargo cases ELEMENT PRESSURE (without the TRAP specification). These charges should not be used in repetitive loads (REPEAT LOAD): scrolling support load (LOAD SUPPORT DISPLACEMENTS) load temperature (TEMPERATURE LOAD) members or elements, load response spectrum (RESPONSE SPECTRUM LOAD) and TIME HISTORY LOAD. It is also available for the generated charges using the build options of the program and UBC generator loads, wind loads generator, generator charges in motion.

Example

```
LOAD 1 DL + LL
SELFWEIGHT -1.4
MEMBER AND LOAD
UNIFORM AND 1 TO 7 -3.5
LOAD 2 DL + LL + WL
REPEAT LOAD
1 1.10
JOINT LOAD
```

5.31.12 Generating Loads

**Objectiv
e**

This command is used to generate moving charges, UBC seismic loads and wind loads using the definitions previously defined load.

The primary load cases can be generated using predefined charging systems. The following sections describe the generation of moving charges, UBC seismic loads and wind loads.

5.31.12.1 Moving charges

The types of charging systems on predefined motion can be used to generate the desired number of primary load cases, each representing a particular position to the cargo moving on the structure. This procedure simulates the movement of a component in a specific direction on a specific plane of the structure.

General Format

LOAGEN DEration n (ADD LOAD i)

TYPJ E {x1 y1 z1 * XINC f1 | YINC f2 | ZINC f3} ({YRANGE | ZRANGE} r)

Where:

- n = total number of cases of primary charge to be generated

- i = number of load case, in the case of predefined load, to be added to the generated charges.

- j = number of type system previously defined load.

- x1, y1, z1 = coordinates X, Y and Z (global) the starting point of the reference load.

- f1, f3 = f2 increments X, Y and Z (overall) of the position of the loading system to be used to load subsequent instances.

- r = (optional) defines the section of the structure along the vertical axis Y. This absolute value of r is added to the reference vertical coordinate (y1 or z1) in the positive direction Y. The overall vertically moving charge externally will be distributed among all members thus generated between the vertical range. r should always be a positive number. In other words, the program seeks members who are in range Y1 and Y1 + ABS (r) or Z1 and Z1 + ABS (r).

The ADD LOAD specification can be used to add a load case previously defined for all load cases generated by the LOAD command GENERATION. In the example below, do the weight specified in the load case 1 is added to all load cases generated.

Sequential Numbers load cases will be assigned to case series generated primary load. The numbering will begin with the case number above prior load. They are allowed to specify the load cases after generation of load case.

> **Note: The primary load cases can be generated from the system moving charges only for members of frames.**

Exampl

e

```
LOAD ONLY 1 DL
SELFWEIGHT
LOAD LOAD GENERATION 20 ADD 1
TYPE  1 0.   May.In October.  XI    In October.
TYPE  2  0.  In October.    In
October. ZI     15 LOAD    22 LIVE
      LOAD    ON PAVEMENT MEMB LOAD
10  TO  20  30  TO  40  UNI  GY  -5.0
LOAD  COMBINATION   31
10  0.75  22  0.75
PERFORM ANALYSIS
```

5.31.12.1.1 UBC seismic loads

Algorithms included automatically distribute the base shear between the appropriate levels and ceiling according to the specifications of UBC (Universal Building Code). The

following format shall be used to generate a load of UBC type on a particular direction.

General Format

LOAD i

Code LOAD {X | Y | Z} (f1) (DEC f) (ACC f)
Code = {UBC | IBC | 1893 | JIA | COL | CFE | NTC | RPA}

Where:

- i = number of load case

- f1 = factor used to load the UBC, the default value is 1.0

- f2 = multiplier Natural torsion, which occurs due to static eccentricity is the difference between the center of mass and center of stiffness of a diaphragm rigid floor, to be used to multiply the UBC, IBC, 1893, charging Horizontal torsion etc. (Default = 0.0). Must be a positive or zero.

- f3 = Accidental torque multiplier should be used to multiply the UBC, IBC, 1893, accidental torsion loading etc. (Default = 1.0). May be negative (otherwise default signal MY is based on the direction of the lateral forces generated)

Example

```
DEFINE UBC LOAD
K ZONE 0.2 0.5 1.0 I 1.5 TS
SELFWEIGHT
JOINT WEIGHT
1 TO 100 WEIGHT 5.0
WEIGHT TO 200 101 7.5 1
LOAD IN X-DIRECTION UBC
UBC LOAD X
JOINT LOAD
May 25, 30 FY -17.5
2 LOAD IN Z-DIRECTION UBC
UBC LOAD Z
DEAD LOAD LOAD 3
SELFWEIGHT
LOAD COMBINATION 4
1  0.75  2  0.75  3  1.0
```

In the above example, note that the first two load cases belong to UBC type, and which are defined before any other load case.

Notes:

1. UBC load cases must be defined as the first set of loading cases.
   No primary load cases specified UBC before a load case UBC are not accepted. Additional loads as members MEMBER LOADS and LOADS loads JOINT nodes can be specified by the UBC load under the same load case.

Misuse

```
LOAD 1 LOAD
2 -1
SELFWEIGHT
AND JOINT
LOAD 3 FX
45
LOAD 3
UBC JOINT LOAD
LOAD X 1.2
3 FY -4.5
LOAD 4
UBC LOAD LOAD 3
Z 1.2 MEMBER
UNI GY -4.5
PERFORM ANALYSIS
```

Uso Correcto

```
LOAD 1
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 2
UBC LOAD Z 1.2
MEMBER LOAD 3
UNI GY -4.5
LOAD 3
SELFWEIGHT Y -1
LOAD 4
JOINT LOAD 3 FX 45
PERFORM ANALYSIS
```

2. All load cases involving UBC load generation must be given before the ANALYSIS specification. In other words, the multiple analysis in which UBC load generation is performed in the separate analysis gives not allowed.

Misuse

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT -1 JOINT
AND LOAD
3 FY -4.5
PDELTA LOAD
ANALYSIS 2
UBC LOAD Z 1.2
SELFWEIGHT -1 JOINT
AND LOAD
3 FY -4.5
ANALYSIS PDELTA
```

Uso Correcto

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
```

3. The REPEAT LOAD specification can not be used for load cases involving charge generation UBC. For example

Misuse

```
LOAD 1
UBC LOAD 2 LOAD X
1.0
SELFWEIGHT Y -1
LOAD 3
REPEAT LOAD 1
1.4 2 1.2
ANALYSIS PDELTA
```

4. If UBC load generation is performed given for X and Z directions, the command for the X direction must precede the command to the Z direction

Misuse   **Uso Correcto**

```
LOAD 1                    LOAD 1
UBC LOAD Y Z              UBC LOAD X 1.2
SELFWEIGHT 1.2 -         SELFWEIGHT Y -1
1 LOAD 2                 LOAD 2
UBC LOAD X 1.2          UBC LOAD Z 1.2
-1 ANALYSIS AND        SELFWEIGHT Y -1
PDELTA SELFWEIGHT      PDELTA ANALYSIS
```

5.31.12.2 Wind Loads

The option to generate wind load can be used to calculate wind loads based on the parameters defined in Section 5.30.3. The following general format shall be used for the generation of wind load.

General Format

<u>LOA</u>D i

<u>WIN</u>LOA DD (-) {X | Y | Z} (f) TYPE j (OPEN) {XR $f_1 f_2$| YR $f_1 f_2$| ZR $f_1 f_2$| LIST memb-list | ALL}

Where:

- i = Number of load case

- X, X or Z, Z = Wind direction with respect to the global axes system • **j =**

number of type system previously defined

- f = factor that will be used to multiply the wind load. The negative sign is used to indicate the opposite direction of the wind (the default is 1.0)

- f1, f2 = global coordinate values to specify the range of X or Y or Z to select members.

- OPEN = option word to be used if the load will be generated in such open structures. If this is not specified, load will be generated assuming that the panels are "closed".

Using X, X, Z or Z and f factor. To the axis, the negative sign indicates that suction occurs on the other side of the selected structure. If all members are selected and X (or Z) is used and the factor is positive, then the surfaces exposed to

within the leadership - x (or - z) will be charged to the positive x (oz) direction (normal to the wind

positive) direction. See the diagrams below. If you use a negative factor X and then exposed surfaces facing in the + x direction will be charged on the negative x direction (normal wind in negative direction). [If introduced - X and a negative factor,

then exposed surfaces in the x - direction will be charged to the negative x direction (suction). If - X is introduced and a positive factor, then the exposed surfaces facing in the + x direction will be charged in the positive x direction (suction)].
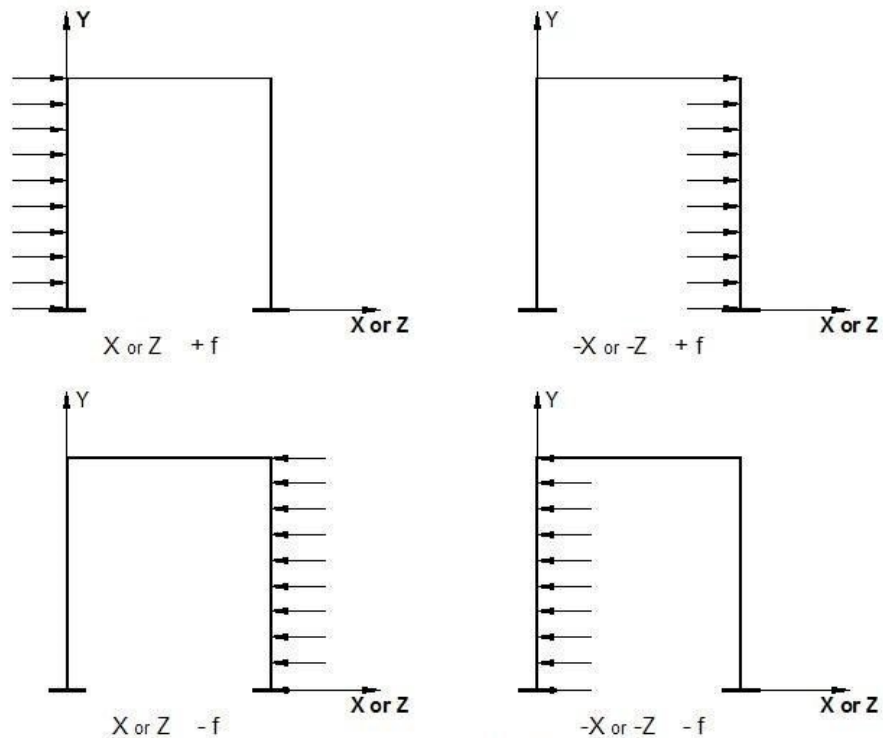
Figure 5.31.12.2 (a): Conventions signs for internal and external pressures

A membership list or a range of values of coordinates (global system) can be used. All members who have both end coordinates within the range are assumed to be candidates (for closed-type structures) to define a surface that can be charged if the surface is exposed to the wind. The charge will be in the form of common charges (no load on the member). Ranks 1, 2 or 3 can be introduced to form a "layer", "tube" or "box" for the selection of members in the combined ranges. Using ranges to accelerate calculations of the larger models.

Example

```
DEFINE WIND LOAD TYPE 1
INTENSITY 0.1  0.12 HEIGHT 100   200
EXP  0.6  JOI  1  TO  25  BY  7  29  TO  37  BY  4  22  23
TYPE  2
INT  0.1  0.12 HEIGHT 100   900
0.3 YR EXP 0 TO 500
LOAD 1
SELF AND -1.0
LOAD 2
WIND LOAD TYPE 2 Z 1.2
LOAD 3
WIND LOAD TYPE 1 X
```

To open structures

```
WIND LOAD 1 LOAD IN Z DIRECTION
WIND LOAD TYPE 1 2 -1.2 OPEN
```

Notes:

1. For closed-type structures, panels or closed surfaces are generated based on the members of the specified ranges and end joints program. The area within each closed surface is determined and then the area ratio (area of influence) for each node in the list is calculated. The individual must be bounded areas will not be charged flat surfaces, a tight tolerance, or. Therefore, one must ensure that members / joints that are exposed to wind form a closed surface (soil may form a closed surface edge). Without proper closed surface, the calculated area for the region may be indeterminate and joint force values may be erroneous. Consequently, the number of exposed joints should be at least three.

2. Plaques and solids are not considered for the generation of wind load. In such institutions, the wind must be applied using pressure load commands for plates and solid.
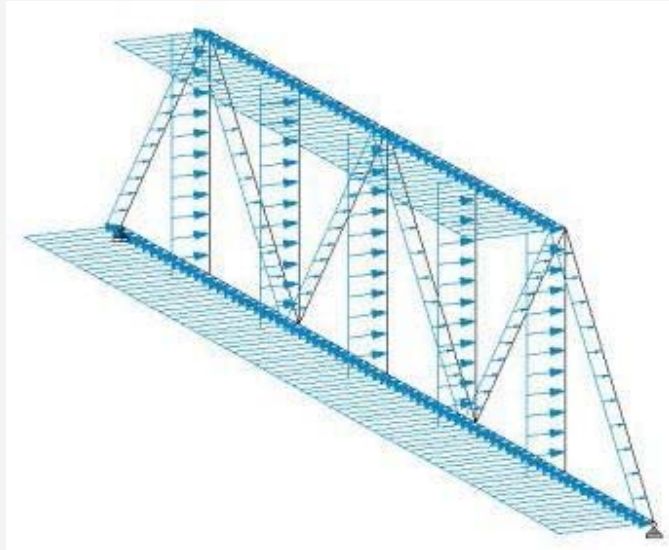
Figure 5.32.12.2 (b): Load diagrams for wind open structures.

> **Tip: To learn more about the behavior of structures due to the effects of natural phenomena (wind) and the grounds or theories in which the program is built to perform the necessary analysis, see "Effects of Wind" in the**
> book " *Di Stru cture l lis* " Meli Piralla 2da. Editing Editorial Limusa.

5.31.13 Generating snow loads

This command is used to generate snow loads using definitions previously specified snow load. This entry must be part of a load case.

General Format

> **SN<u>OW</u> LOAD**
>
> **_flr_group TYPE j CS f1 {BALA | UNBA } {OBST | ONEB} {MONO | HIP | GABLE}**

Where:

- *_flr_group = members that form the roof and must be loaded by snow load must be included in a group of soil (see Section 5.16).*

- j = load type system previously defined snow.

- f1 = roof slope factor (CS). Default = 0.0. For sloping roofs, the roof slope factor described in Section 7.4 of SEI / ASCE-7-2. A value of 0 indicates that the roof is horizontal.

- BALA or UNBA = Snow load balanced or unbalanced. Default is balanced (BALA). These terms are described in Section 7.6 and Figures 7.3 and 7.5 of ASCE 7-02.

- OBST or UNOB = obstructed or unobstructed. By default it is not clogged. •

MONO or HIP GABLE = type or ceiling. Default value is MONO.

Use as many groups and floor types as needed for each load case.

5.31.14 Nominal charges

A nominal load is a load side (horizontal load) that is derived from an existing vertical load case. This type of charging is introduced to accommodate a requirement in design codes. AISC 360-05 specifications define nominal loads such as lateral loads are applied at each level of structure and specified in terms of gravity loads.

Description

The cases of both major burden for reference can be selected and moved within the definition of nominal load factor can be specified where required and address. Rated loads are calculated and applied as loads on nodes.

> **Note: the actual values of the applied load in the user interface until after the analysis is done are not displayed.**

General Format

```
Nload LOADTYPE type LOAD TITLE title
...
Load items
...
Notional LOAD
n {X | Z} (f1)
```

Where:

- n = number if the primary load case or load case reference that contains the elements of vertical load the load.

- f1 = factor by which the content of CO is increased. Typically, codes recommend 0.2 to 0.3% (0002-0003). The default value is 0.002 and the factor specified in the definition block DIRECT ANALYSIS (scans).

You can specify multiple elements under any notional LOAD load (nominal load) and may have one or more blocks of the command of the rated load at any load case.

Example 1

```
LOAD  1  :  DEAD  JOINT
LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO 64 FY -100 LOAD 2: DEAD LOAD
notional
Notional LOAD  1  X
0.002
...
LOAD 10: Imposed JOINT
LOAD
13 TO 16 29 TO 32 45 TO 48 61 TO FY 64 -50 LOAD
11: Imposed Notional
Notional LOAD LOAD 10 x
0.002
```

Example 2

Yes we combine loading assumptions in the example shown above, the correct syntax would be

```
LOAD 3: LOAD LOAD 1 + 2 + Notional REPEAT LOADS LOAD
1 1.0 2 1.0
Notional LOAD
1 X 2 X 0002 0002
```

The following syntax does not include the contributions of nominal load cases for each component as nominal charges will not be repeated.

```
LOAD 3: LOAD LOAD 1 + 2 + Notional REPEAT
LOADS LOAD
1 1.0 2 1.0
```

Example 3

Similarly

```
LOAD TEST TITLE 1 LOADTYPE SELFWEIGHT NONE Y -
1.15
MEMBER LOAD
CON GY 143 145 -16
CON GY 502 515 -95
LOAD LOADTYPE NONE TITLE 2-Notional Notional LOAD
TEST 1 X 1.0
LOAD TEST 3 LOADTYPE NONE-COMBO TITLE
REPEAT LOAD
2 1.0
```

The Load Case 3 in the above example would produce loads.

Example 4

Using nominal loads with reference loads

```
REFERENCE DEFINE LOADS
LOAD R1 LOADTYPE DEAD DEAD LOAD REF TITLE AND SELFWEIGHT
LIST 1 -1 120 FLOOR
TO LOAD
Yrange FLOAD -0.05 19 21 GY
Yrange FLOAD -0.05 39 41 GY
LIVE LOAD R2 LOADTYPE TITLE FLOOR LOAD LIVE LOAD REF
yrange FLOAD -0.03 19 21 GY
Yrange FLOAD -0.03 39 41 GY
                                        REF
ERENCE END DEFINE LOADS
*******************************************
* LOAD COMBINATIONS INCLUDING LOADS * notional
*******************************************
*
* DEAD + LIVE LOAD +/- notional
*
```

```
LOAD    1    REFERENCE
LOAD  R1 R2 1.2 1.6
Notional LOAD
R1 R2 X 0.0024 X 0.0032
LOAD    2    REFERENCE
LOAD  R1 R2 1.2 1.6
Notional LOAD
R1 R2 X X -0.0024 -0.0032
LOAD    3    REFERENCE
LOAD  R1 R2 1.2 1.6
Notional LOAD
Z Z R1 R2 0.0024 0.0032
LOAD    4    REFERENCE
LOAD  R1 R2 1.2 1.6
Notional LOAD
R1 R2 Z Z -0.0024 -0.0032
```

If a PRINT or PRINT CHECK STATIC LOAD DATA with any analysis specification command input file is added, the theoretical loads generated would be printed as follows:

```
NOTIONAL LOAD - (POUN,FEET)

  LOADING    4  DL+NDLX

    JOINT          DIRECTION                LOAD

…

      6117              X     -0.00200 X      74124.266 =        -148.249

      6118              X     -0.00200 X     115504.569 =        -231.009

      6119              X     -0.00200 X      38397.157 =         -76.794

      6120              X     -0.00200 X      15551.155 =         -31.102

      6121              X     -0.00200 X      17347.718 =         -34.695

      6122              X     -0.00200 X      29341.723 =         -58.683

      6123              X     -0.00200 X      39771.147 =         -79.542

      6124              X     -0.00200 X      39712.750 =         -79.426

      6125              X     -0.00200 X      34006.254 =         -68.013

                           ==============================================

48554642.872      -97109.290
```

5.32 Load cases reference- Application

Describe how you can call the data specified under those types in cases of actual load. Reference load types are described in Section 5.30.6 of this manual.

General Format

The format of a reference to a reference of the load in a case of the primary load (j) looks like this:

```
LOAD LOADTYPE j (type) load_title
REFERENCE LOAD
R (i) 1.0
```

Example

```
LOAD   1   LOADTYPE   NONE   TITLE   D +
L REFERENCE LOAD
R1 R2 1.0 1.0
LOAD 2 LOADTYPE NONE TITLE REFERENCE
DEAD SNOW LOAD +
R3 R1 1.0 1.0
LOAD 3 LOADTYPE NONE TITLE
REFERENCE LOAD D + H
R1 R4 1.0 1.0 LOAD ELEMENT
TRAP GY JT 1212 1267 -0.54 -0.44 -0.44 -0.54.
```

5.33 Specification   of   Frequency   Natural

**Objectiv
e**

This command is used to calculate the natural frequency of the structure corresponding to the general direction of the deflection caused by the load case preceding this command vibration. So then, this command is typically used after a load case.

General format:

```
CALculate  NATuRAL  (FREQUENCY)
```

Description

This command is specified after all other load specifications of any primary load case for which the natural frequency is calculated. This natural frequency calculation is based on Rayleigh iteration method. If a full scale Eigen solution is required, the MODAL CALCULATION command (see next section) may be used. Note that a solution Eigen runs automatically if a response spectrum specified for a load case.

Example

```
1 DEAD AND LOADING AREA LOAD
LIVE LOAD
1 TO CALCULATE 23 ALOAD -200.0
NATURAL WIND LOAD FREQ 2
LOADING
```

In this example, the natural frequency for load case 1 will produce the output value of the natural frequency in cycles per second (cps), the maximum deflection together with the absolute address and the node number where this occurs is calculated.

Note: This command is based on the Rayleigh iteration method. The frequency estimates calculated frequency for vibration mode corresponding to the deflected shape generated static loads within the load case.

Calculating modal command 5.33.1

Objectiv
e

This command is used to obtain a full scale Eigen solution for the calculation of the relevant frequencies and vibration modes.

General format:

MODAL (CALCULATION REQUESTED)

This command is typically used in a load case after all specified load cases do have. The charges will be handled as masses for Eigen solutions. The static displacements and forces calculated for the load case in particular.

This command may be used only for a load case, and may NOT be used in conjunction with a response spectrum analysis.

Note: Eigen solution initiated by this command will cause the loads specified load cases are treated as masses. Users are advised to specify loads keeping this in mind.

5.34 Specification of Loads Combined

Objectiv
e

This command is used to combine the results of the analysis. The combination may be algebraic square root of the sum of squares (SRSS) or by combination of both.

General format:

LOAD COMBination (SRSS) i to$_1$

i$_1$, F$_1$, I$_2$, F$_2$... (F$_{SRSS}$)

Where:

- i = Number of combined load (any less than 100,000 different integer but any number previously defined for cases of primary load).

- a1 = Any title for the combined load.

- 1i, i2, ... = Represents the load case numbers which are to be combined.

- f1, f2, ... = represent the relevant factors to be applied to the loads.

- fsrss = optional Factor to be applied as a multiplier on the combined results of the load combination SRSS.

If the minus sign precedes any loading case number in the LOAD COMBINATION SRSS option, then that load case will be combined algebraically with the SRSS combination of the rest. The total number of cases of combination of charge can not exceed 150.

Description

Test results may be combined using either algebraically or SRSS method (square root of the sum of the squares). The combination scheme can be mixed when required. For example, if the same combination of load, the results of the load conditions can be combined as SRSS and then algebraically combined with other load cases. The following example illustrates this.

Example

Different examples of combination are provided to illustrate the possible combination schemes.

Simple algebraic combinations and the SRSS method

> **LOAD COMBINATION 7 DL + LL +**
> **WL 1 0.75 2 0.75 3 1.33**
> **LOAD COMBINATION SRSS 8 SEISMIC DL**
> **+ 1 1.0 2 0.4 3 0.4**

The first line above (LOAD COMBINATION 7) illustrates a simple algebraic combination. The second line (LOAD COMBINATION 8) illustrates a combination of pure SRSS load a default factor of 1 for SRSS combination The following diagram will be used.

$$v = 1.0 \sqrt{1 \times L1^2 + L2^2 + 0.4 \times 0.4 \times L3^2}$$

Where:

- v = combined value and l1 - l3 = values of load cases 1, 2 and 3.

Note that because a factor of SRSS, the default value used by the program is 1 is indicated.

Algebraic and SRSS combination within the same load case combination of Example 1

```
LOAD COMBINATION SRSS 9
 -1 0.75 2 1.0 3 1.0 0.75
```
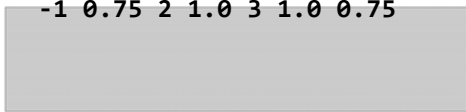
The combination formula is as follows

$$v = 0.75 + 0.75 \times L1 \sqrt{1.0 \times 1.0 \times L2^2 + L3^2}$$

Where:

- v = combined value

- L2 and L3 = values of load cases 2 and 3

In the above specification, note that the case load has a minus sign at the beginning. As the load 1 is algebraically combined with the results obtained from the combination of load cases 2 and 3 of SRSS. Note that a factor of 0.75 for the SRSS combination for 2 and 3 applies.

Example 2

```
LOAD COMBINATION SRSS 10
-1 0.75 -2 0.75 3 1.0 4 1.0 0.75
```

Here, both the load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4 Observe the SRSS factor of 0.75. The combination formula is

$$v = 0.75 \times 0.75 \times L1 + L2 + 0.75 \quad \sqrt{1.0 \times 1.0 \times L12 + L22}$$

Notes

1. This option combines the results of the analysis in the manner specified. No analyzes the structure for the combined load

2. If the side effects of the combination of load cases will be obtained through a PDELTA analysis, COMBINATION LOAD command is inappropriate for this purpose. See for details the REPEAT LOAD (section 6.32.11) command.

3. In a combination of load specification, a value of 0 (zero) as a load factor is not allowed. In other words, a specification as follows:

```
LOAD COMB 7
1  1.35  2  0.0  3  1.2  4  0.0  5  1.7
```

It is not allowed. What happens is that because of the way the program processes data as soon as it encounters a 0.0, it stops at 0.0 and not read further. Because of this, in the case of previous load results only consist n forces due to load 1 multiplied by a factor of 1.35. The contribution of 1.2 * 1.7 * Load 3 and Load 5 is not worth because the program simply does not read the data.

4. All cases of combination of loads must be provided immediately after the last case of primary charge.

5.35 Specification Analysis

Objectiv
e

The analysis option STAAD.Pro includes linear static analysis, P-Delta (or second-order analysis), analyzes and various linear dynamic analysis.

This command is used to define the type of analysis required. Additionally, this command is used to obtain various types of data related to the analysis, such as load information, static verification, vibration modes, etc.

General Format

```
                                                    ⌠ LOAD DATA
        ⌠PERFORM      ⌡                              | STATICS CHECK
        {PDELTA (n)    }    ANAPRI LYSISNT           { STATICS
        ⌊LOADNONLINEAR (n)                           | MODE SHAPES
                                                     | BOTHAL
                                                     ⌊ L
```

Where:

- n = number of desired iterations (the default is n = 1).

This command instructs the program execution analysis, which includes:

a) Check that all relevant information has been provided for the analysis

b) Generate the stiffness matrix of the nodes

c) Check the stability of the structure

d) Solving simultaneous equations, and

e) Calculate the member forces and displacements

f) If the P-Delta analysis, the forces and displacements are specific recalculated, taking into account the effect P-Delta

g) Nonlinear analysis will take into account the nonlinearity and P-Deltas effects. (Refer to Section 2.17.2.2).

h) When a response spectrum within a specified load case, or MODAL CALCULATION command, the program will perform a dynamic analysis is used.

i) For each of the n iterations of the P-Delta analysis, the load vector will be modified to include the side effects generated by the displacement caused by the previous analysis.

With the lack of any of these commands, the analysis will not be executed. These commands can be repeated when analyzing several analyzes may be required in different phases.

Note that a P-Delta correct analysis reflects the side effects of the combination of load cases, only when they are defined by the REPEAT LOAD (Section 5.31.11) specification. The side effects will not be evaluated correctly for LOAD COMBINATIONS.

If the LOAD DATA command PRINT is specified, the program will print an interpretation of all data load.

- Statics PRINT CHECK, provide a summary of the applied loads and support reactions, as well as a summary of the moments of the loads and reactions taken around the origin.

- Statics LOAD PRINT prints the same as PRINT CHECK statics, but also prints a summary of all internal and external forces at each node, generating large outputs.

- MODE SHAPES PRINT prints the vibrational modes in the nodes for all modes of vibration calculated.

- PRINT BOTH, is equivalent to more PRINT PRINT LOAD DATA CHECK statics.

- PRINT ALL is equivalent to more PRINT PRINT LOAD DATA LOAD statics.

Notes

STAAD.Pro allows multiple analysis within a single execution, which could be used for the following purposes:

1. Subsequent analysis and design cycles within a single execution result in a optimized design. The active database of STAAD.Pro sizes automatically updated cross-sections of the members, so that the whole process is automated.

2. The multiple analysis can be used to load dependent structures. For example, structures with bracing members are tested at various stages. Bracing members are considered which can only withstand tensile load. As will require activation and deactivation depending on the load laterally.

3. The entire process can be modeled within a single program execution through several PERFORM ANALYSIS command. The STAAD.Pro database automatically stores the results for different runs and is capable of generating a layout based on the load combinations defined.

4. Also can be used to model multiple analysis changes for other features such as SUPPORTS, RELEASES, etc. SECTION PROPERTIES

5. The use of multiple analysis may require the use of other commands such as SET and CHANGE NL.

5.36 Specification Change

Objective

This command is used to reset the stiffness matrix. This command is generally used when it is required to perform multiple analyzes within the same run.

General format:

<u>**CHA**NGE</u>

This command indicates that expected by those input data that will change the stiffness matrix. This command should be used only when an analysis has already been executed. The CHANGE command does the following:

a) Place the stiffness matrix to zero,

b) Transforms active members, when these are placed as inactive INACTIVE by a previous command, and

c) Allows specification of the supports again another SUPPORT command, which causes the former supports are ignored. The SUPPORT specification must be such that the number of relaxations before CHANGE must be greater or equal to the number of relaxations after CHANGE. Also, the supports must be specified in the same order before and after the CHANGE command.

Example

<u>Before CH ANGE</u>

```
                              1 PINNED
                              2 FIXED BUT MY MZ FX
      3 FIXED BUT FX MX MY MZ
```

<u>After CH ANGE</u>

```
                              1 PINNED
                              2 FIXED
                              3 FIXED BUT FX MZ
```

The CHANGE command is not necessary when we review only the member properties for further analysis. This is the typical case in which the user has asked for a selection of member and then use the PERFORM ANALYSIS command to analyze again based on the new member properties structure.

Notes

1. If new load cases are specified two after the CHANGE command as in the case of a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option.

2. Multiple Analysis using the CHANGE command should not be performed if the input file contains load cases involving UBC Analysis, Analysis and Response Spectrum Analysis HISTORY TIME.

5.37 Specification of loading lists

**Objectiv
e**

This command allows you to define a group of cases of active loads. All load cases defined as active using this command will remain so until a new list of charges specified.

General Format

```
LOAD LIST {load-list | ALL }
```

Description

This command is used to activate the cases listed in the command load and, to some extent, disables all other load cases that are not listed in this command. In other words, the loads listed are used for printing and design for the implementation of specific calculations. Note that when the PERFORM ANALYSIS command is used, the program uses internally all load cases, regardless of the LOAD LIST command, except after a RESTORE or CHANGE command. In these two cases, the LOAD LIST command causes the program to run the analysis on only those charges that are in the list. If the LOAD LIST command is never used, the program will assume that all load cases are active.

Example

```
LOAD LIST ALL
PRINT LOAD FORCES MEMBER LIST March 1
PRINT CHECK CODE ALL SUPPORT
REACTIONS
```

In this example, the forces in the members will be printed for all load cases, while loads 1 and 3 will be used to print the support reactions and review in accordance with codes of all members.

> **Note: The LOAD LIST command can be used in situations where multiple analysis required to execute a set of load cases only. Note that all load cases are automatically activated before a RESTORE or CHANGE command is used.**

5.38 Grouping Loads

"Load Envelopes" are a means of grouping a set of load cases under one name (number). If one or more tasks have to be performed by a set of load cases, such as: Service test on steel design for a set of load cases, design verification efforts steel under another set of cases, etc. This feature is desirable.

It is an alternative to the LOAD LIST command described in section 5.37.

The pool can be labeled with optional keywords to specify the qualitative nature of the load or combination cases included in the definition of the load.

Example

In the following example, the keyword associated with serviceability envelope 2.
Keywords can be any word (with or without spaces) of your choice.

```
DEFINE ENVELOPE

1 TO 8 1 ENVELOPE TYPE CONNECTION

September 15 ENVELOPE TO TYPE 2 Serviceability

April 16 ENVELOPE TO 28 END

DEFINE TYPE ENVELOPE STRESS
```

The first line in the DEFINE command ENVELOPE loading cases numbered 1 to 8 make up one group of type CONNECTION loads. Load cases 9-15 also defines the grouping of two kinds serviceability loads.

To print the reactions corresponding to the grouping of load 1 support, the following commands should be entered in the input file

```
LOAD LIST ENV 1

PRINT SUPPORT REACTIONS
```

5.39 Specification Sections

Objective

This command is used to specify the areas along the members of a frame-like structure for the forces and moments are required.

General Format

```
SECTION f₁(F₂) (F₃) ... (F₅) {MEMBER memb-list | (ALL)}
```

Description

This command specifies the sections in terms of fractional lengths of the members, in which the forces and moments are considered for further processing.

- f1, f2, ..., f5 = Section provided for members (in terms of the fraction of the length of the member). The maximum number of sections is 5, including those taken at the beginning and end of the member. In other words, more than three intermediate sections for the SECTION command is not allowed.

This command is used to rotate to any desired geometry around any global axis angle. May be used in the analysis and design rotated pattern.

Example

```
SECTION  0.0  0.5  1.0  MEMB  1  2
SECTION  0.25  0.75  MEMB  3  TO  7
SECTION  0.6  MEMB  8
```

In this example, the values of the first section of the members 1 and 2 and 0.0, 0.5 are placed, and 1.0, ie, at the beginning, in the middle and at the end. The members 3 and 7 are defined in the next SECTION command, where the sections are set as 0.25 and 0.75.

In the next SECTION command, member 8 has a specified section 0.6. The rest of the members will not have intermediate sections for them. If the value of the section to a member is not specified, the default values are 0.0 1.0 (ie, the beginning and end). For example, if the section is not defined, the initial and final forces members will be used in the design. As mentioned above, no more than three intermediate sections allowed for the SECTION command. However, if more than three intermediate sections are involved, they may be checked by repeating the SECTION command after completing the required calculations. The following example will help clarify this.

Example

```
SECTION  0.2  0.4  0.5
      PRINT ALL   SECTION
        SECTION FORCES
      0.6    0.75  0.9
      PRINT ALL   SECTION
        FORCES
```

In this example, first intermediated forces in three sections (that is, 0.2, 0.4 and 0.5) are printed and then three additional forces into sections (that is, 0.6, 0.75 and 0.9) are printed. This provides the user forces over three intermediate sections.

Notes :

1. The SECTION command specifies only the sections. SECTION FORCES Use the PRINT command after this command to print the forces and moments of the specified sections.

2. This is a secondary analysis command. Note that the analysis must be run before using this command.

5.40 Print Specification

Objective

This command is used to tell the program to print various test results and information modeling. STAAD.Pro offers a number of versatile printing commands used to format the output.

General Format

To print command data:

PRIJOI NT {NT COORDINATES | MEMBER PROPerties | ELEMENT INFOrmation (SOLID) | MEMBER PROPerties | SUPPORT **INFO**rmation | ALL} {(ALL) | LIST item / joint / Member-list}

To print CG location

PRINT CG (_group_name)

To print the test results

```
PRI{NT (JOINT) DISPLACEMENTS | MEMBER FORCES (GLOBAL) |
ANALYSIS RESUTLS | MEMBER SECTION FORCES | MEMBER
STRESSES | ELEMENT (JOINT) STRESSES (AT f₁ f₂) |
ELEMENT FORCES | ELEMENT (JOINT) STRESSES SOLID | MODE
SHAPES spec-list}
```

List-spec = {(ALL) | LIST joint / member / elements-list}

For printing the support reactions

```
PRISUP NTPORT REACtions
```

For offset printing or stiffness

```
PRINT STORY DRIFT (f₃PRINT STORY STIFFNESS
```

Print the location of the center of rigidity at each floor

```
PRINT DAYPHRAGM CR
```

Description

Note that the list of terms is not applicable for RESULTS ANALYSIS PRINT, PRINT and PRINT MODE SUPPORT REACTIONS SHAPES commands.

The PRINT JOINT COORDINATES, command prints all the coordinates of the nodes interpreted.

The PRINT MEMBER INFORMATION, command prints all the information of the members, including limb length, incidence of members, beta angles, if a member is not armor or conditions of release and the Member at the beginning and end (1 = free, 0
=   Not
free).

The PRINT command prints INFORMATION ELEMENT all incidents nodes, thicknesses of the elements and Poisson modules for flat / crack ng elements. The PRINT command prints INFORMATION ELEMENT similar information for solids.

The PRINT MEMBER PROPERTIES command prints all properties including cross-sectional area, moments of inertia, and section modulus members on both axes. The units for the properties are always inches or centimeters INCH CM (depending on the system or METRIC units FPS) without considering the drive specified in the UNIT command.

The following designation is used for property names of members: •   AX

-         Cross Section Area

•   AY -   Area used to calculate the shear deformation at the local Y axis

- **AZ -** Area used to calculate the shear deformation in the local Z axis

IZ – Moment of Inertia about the local Z axis

- **IY -** Moment of inertia about local y axis

IX – Torsional Constant

- **SY -** Smaller section modulus about the local Y axis

SZ – Smaller section modulus about the local Z

MATERIAL PROPERTIES The PRINT command prints all the properties of the materials to members, including E (elastic modulus), G (shear modulus), specific gravity and thermal expansion coefficient (alpha) for frame members. This command is available for members only.

SUPPORT INFORMATION The PRINT command prints all the information regarding your support stability, freedom and spring constant, if any.

The PRINT ALL command is equivalent to the combination of the last five commands. This command prints the coordinates of the nodes, the information of members, member properties, properties of materials and information support in that order.

CG The PRINT command prints the coordinates of the center of gravity of the structure. Only the weight of the structure is used to calculate the center of gravity. The charges in the nodes and the user-defined members are not considered in calculating the center of gravity.

The PRINT (JOINT) command prints DISPLACEMENTS displacements at the nodes in a tabulated form. Offsets for all six directions be printed for all cases of load specified. The unit of length is inch for travel INCH or CM centimeter (depending on the type of system or METRIC units FPS) without considering the drive specified in the UNIT command.

The PRINT command (MEMBER) FORCES prints the member forces (ie, the axial force, shear at the local Y and Z, Torsional Moment, moments about the Y and Z) in a tabular form per member for all specified loading cases.

ANALYSIS RESULTS The PRINT command is equivalent to the last three combined commands. With this command, the movement of the nodes, the support reactions and member forces will be printed in that order.

(MEMBER) SECTION FORCES forces printed in the middle sections give the PRINT command specifies a prior SECTION command. Printing is done in a tabular form per member for all load cases specifies two.

The PRINT (MEMBER) command STRESSES efforts tabula members at the initial node, the end node and all intermediated sections specified days. These efforts include the axial (ie, the axial force on the area), the bending–and (that is, the time–and

on the section modulus in the local axis), the bending-z (i.e., the time-z on the section modulus in the local Z), the shear stress for local addresses and z, and combined efforts (combination absolute axial, bending and flexion-y-z).

For prismatic sections, if not provided AY and / or AZ, the area of the entire section (AX) will be considered in the calculation of shear.

For sections with variable banking, values of AY and AZ are those for the location where stress is printed. So if the effort is printed in place 0.0, the AY in AZ are based on the size of the member in the initial node.

Cant AY = Total * Thickness of the Soul.

AZ = 2/3 area both skates together

The PRINT ELEMENT STRESSES command must be used to print efforts (FX, FY, FXY, QX, QY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) plans for items / shells . Typically, the forces and moments per unit width at the centroid will be printed. Efforts are also printed Von Mises (VONT, VONB) as the angle (ANGLE) defining the orientation of the principal planes.

The variables that appear in the output are as follows. See Fig. 2.13 in section 2 of this reference manual for more information on these variables.

- QX = Shear stress in local face X in the Z direction •

QY = Shear stress in local face And in the Z

direction •   MX = moment per unit width

about local face X •   MY = moment per unit

width about local face And

- MXY = Torsional moment per unit width in local XY plane •   FX =

Axial Stress at the local address X

- FY = Axial Stress at the local Y direction

- FXY = Shear XY locally

- Von Mises VONT = Effort in the upper surface of •   Effort

VONB = Von Mises on the lower surface of the

element

- SMAX = Principal stress maximum flat SMIN = Main Effort minimum flat TMAX = maximum in-plane shear stress

- ANGLE = Angle determines the direction of maximum principal stress with respect to the local axis X.

If the JOINT option is used, the forces and moments at the nodal points are also printed in addition to the centroid of the element.

The AT option can be used to print forces in the elements at any specific point of the element. The AT option must be accompanied by f1 and f2. Note that f1 and f2 are the local coordinates X and Y (in units in use) the point where the time and effort are required. Refer to section 2.6 of this manual for details of the local coordinate system of the elements.

The PRINT ELEMENT (JOINT) SOLID STRESS command allows you to print to the center of effort SOLID elements. The variables that appear in the output are:

- Standard Efforts: SXX, SYY and SZZ •

Efforts Cutters: SXY, SYZ and SZX

- Major efforts: S1, S2 and S3.

- Von Mises Stress: SE

- Cosines Directors   6 are direction cosines DC printed following expression corresponding to the first two directions of effort.

The print option JOINT efforts nodes of solid elements.

The MODE SHAPES PRINT command prints the displacements of the nodes for all the calculated modes.

The PRINT command prints SUPPORT REACTIONS reactions props in tabular form, for support, for all load cases. The LIST option is not available for this command.

The PRINT ENTIRE TABLE command can be used to obtain a print catalog content steel which member properties are being read. This command should be provided following the specification of the Member properties.

Examples

```
PERFORM ANALYSIS
PRINT ELEMENT JOINT FORCES
PRINT ELEMENT FORCES 0.5 0.5 LIST AT 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT TO LIST 1 DISPLACEMENTS 50
PRINT FORCES MEMBER LIST 101 TO 124
```

TECHNICAL MANUAL STAAD.Pro

Notes

1. The output generated by these commands are based on the unit system in use. The user may want to verify the unit system in use and replace if necessary.

2. The results can be printed for all nodes / members / elements or based on a specific list.

5.41 Printing sections displacements

Objective

This command is used to calculate and print displacement sections (intermediate point) of frame members, providing deflection data between nodes.

General Format

```
PRINT SECTION (MAX) DISPLACEMENTS (NSECT i) (SAVE a) {
NOPRINT | ALL | LIST memb-list}
```

Where:

- i = number of sections to be taken. 12 NSECT default if you do not use and also if SAVE (max. = 24, min = 2) is used.

- a = filename where offset values can be stored and used by the graphics program STAADPL. NOPRINT If the command is used in conjunction with the SAVE command, the program writes the data to file only and does not print them out.

Description

This command prints the displacements at intermediate points between two nodes of a member. These displacements are in the directions of the global coordinates (see figure). If the MAX command is used, the program will print only the maximum local travel between all load cases.

Example

```
PRINT PRINT SAVE SECTION
SECTION displ MAX DISP
```
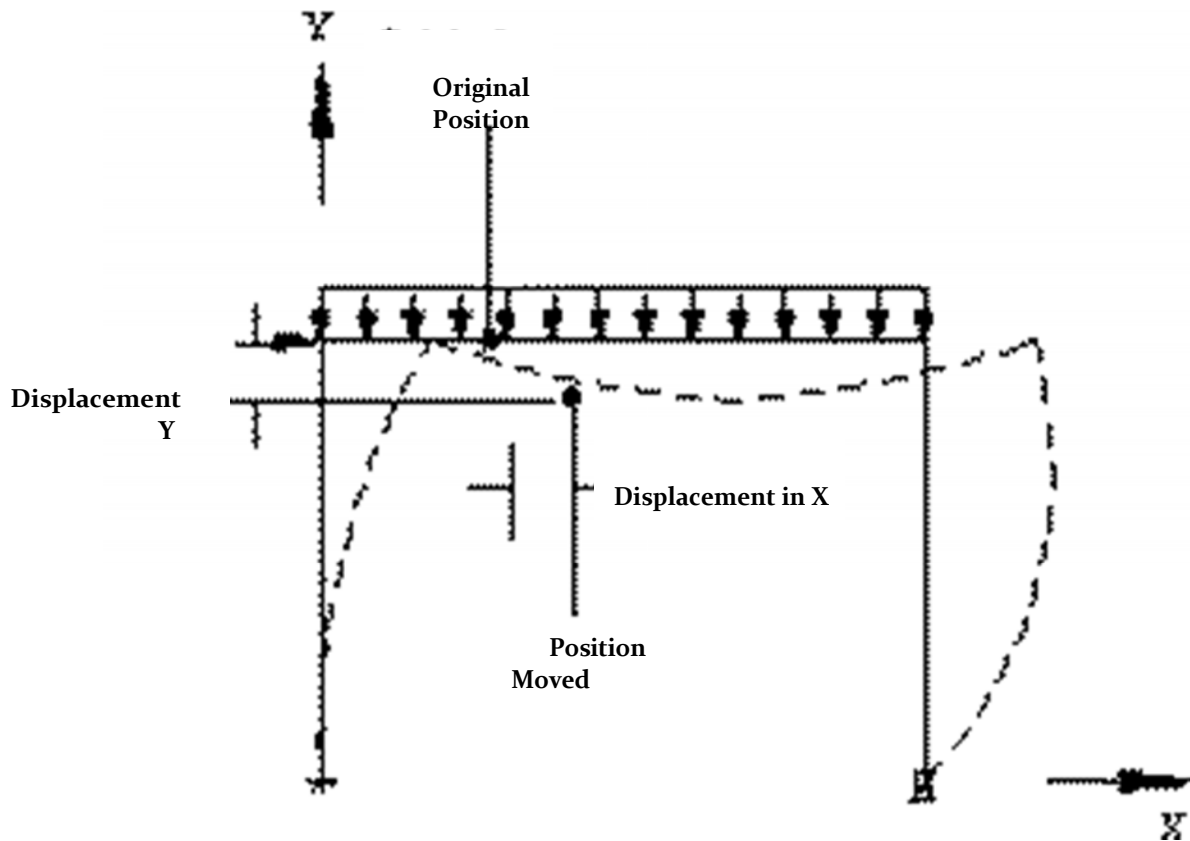
The movements of the sections are measured in global coordinates. The values are measured from the original position (not deflected) to the deflected position. See previous figure.

Also printed is the maximum local displacement. First, the location is determined and then the value is measured from this location, a line connecting the start and end nodes of the deflected member.

Notes

1. The offset values are available in Global section coordinates.
   The undeflected position is used as a baseline for calculating deflections.

2. This command is for secondary analysis. It should run a scan before this command can be used.
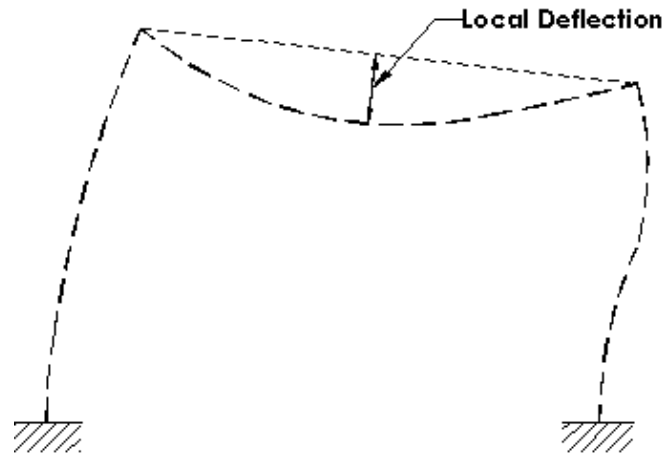
5.42 Specifying printing of the grouping of forces

Objective

This command is used to calculate and print the envelope of force / moment for members of the frame. This command is not available for finite elements.

General Format

```
PRINT {FORCE | MAXFORCE ENV}ELOPE (NSECTION i) list-
spec
spec = {list-LISMemb-list T | (ALL)}
```

Where:

- i is the number of equally spaced sections to be considered in the printing of the envelopes of maximum and minimum forces. If NSECTION i command, the default value for i will 12. MAXFORCE command produces only the values of maximum / minimum force all sections is omitted while the FORCE command prints the values of maximum strength / minimum each section as well as the strength values of max / min of all sections. The components include forces FY, MZ, FZ and MY. Note that the SECTION (as described in Section 5.40) command does not define the number of sections for groups of forces.

Example

```
PRINT  FORCE  ENV  PRINT
MaxF ENV NS 15
DK 4 ENV PRINT FORCE TO LIST March 15
```

Note: This is a secondary analysis of command and should be used after the analysis specification.

5.43 Specification Size

**Objectiv
e**

This command provides an estimate of the required section properties for members of the framework, based on certain results of the analysis and the requirements you define.

General Format

<u>SIZ</u>E * {WIDTH f_1| DEFLECTION f_2| LENGTH f_3| BSTRESS f_4|
<u>SST</u>RESS f_5MEMBER memb} {-list | ALL }

Where:

- f1 = maximum allowable width

- f2 = maximum allowable ratio (length / maximum local deflection)

- f3 = length to calculate the above ratio. The actual length of the member shall be taken by default

- f4 = maximum allowable bending stress

- f5 = Maximum Allowable shear stress

Note that the values should be provided in the system of units being used.

Description

This command is used to calculate the required properties of a section of an individual, based on the analysis results and the criteria the user specifies two. The criteria specified by the user, may include the width of the member, allowable ratio (length / maximum deflection), maximum allowable bending stress and maximum stress allowable cut. Any number of these criteria may be used simultaneously. The output includes: the required section modulus (about the major axis), the required cutting area (cutting parallel to the minor axis), maximum moment capacity (about the major axis), maximum cutting capacity (to cut parallel to minor) axis and the maximum rate (local maximum length deflection).

Example

```
WID SIZE 12 DEFL 300 LEN 240 BSTR 36 ALL
BSTR DEFL SIZE 450 42 16 TO 25 MEMB
```

Note: the sizing is based solely on the criteria specified by the user in the SIZE command.

For example, in the above example, the sizing member is specified based on a user do with a width of 12, a ratio of length / deflection 300 (where length = 240) and a maximum allowable bending stress of 36.

In the second example, the dimensioning will be based on a ratio of length / flex 450 (where the length is equal to the actual length of the limb) and a maximum allowable bending stress of 42.

Note: This is an option for post - analysis. So this command will have to be used after the analysis specification.

5.44 Design Specification for Steel

In this section, all necessary for structural steel design specifications are described.

Section 5.44.1 discusses all the specifications that can be used to control the layout. Sections 5.44.2 and 5.44.3 describe the options and MEMBER SELECTION CODE CHECKING respectively. The selection of members for optimization is discussed in section 5.44.4. STAAD.Pro also provides options for the design of welds which are described in section 5.44.5.

5.44.1 Specifying Parameters

Objective

This group of commands is used to define the parameters required for the design of steel.

General Format

```
PARAMETER
```

```
Design-code CODE
```

```
F1 {parameter-name | PROFILE a1 (a2 a3)} {MEMBER memb-list | ALL|
member-group-name | deck-name}
```

```
design-code = {AAShto, AISC, AUSTRALIAN, BRITISH, CANadian, FRENCH, GERMAN,
                INDIAN, JAPAN LRFD, MEXICAN ...}
```

Description

Parameter-name - refers to the name of the parameter listed in the parameter table contained in section steel design.

For AISC allowable stress design, see Table 2.1. For a list of design parameters for AISC LRFD load resistance, see Table 2.2. For design of steel to other codes, refer to the relevant sections. f1 = Parameter value. The user can control the layout by specifying the appropriate parameters.

The PROFILE parameter is available for code AISC ASD (allowable stress design) only. The user can specify up to three sections (a1, a2 and a3). The profile, as described in Table 3.1, consists of the first three letters of a name in the catalog of AISC steel, for example, W8X, W12, C10, L20, etc. PROFILE Name parameter is used only for the selection of members to these profile names. Also specified under section PROFILE must be the same as specified initially given under MEMBER PROPERTIES. Note that the PROFILE command, you can only use for the catalog of American steel. The CODE parameter selects the type of code will be verified steel do in design. The default code for steel depends on the country of distribution.

Example

```
PARAMETERS
CODE AISC
KY 1.5 MEMB March 7 TO 11
NSF 0.75 ALL
W14 W12 PROFILE MEMB February 1 23
ALL RATIO 0.9
```

Notes

1. All values of the units must be in the system of units in use.

2. For the default values of the parameters, see the appropriate table settings.

3. The PROFILE command is available with the American code AISC ASD only. Not available with AASHTO LRFD or codes.

5.44.2 Specification review under codes

**Objectiv
e**

This command is used to execute the code under verification.

General Format

```
CHECK CODE {MEMBER memb-list | ALL | Group-name member-
| Deck-name | PMEMB pmember-list}
```

Description

This command checks the members specified against the specifications of the desired code. For detailed information, refer to Section 2 of this Manual.

Note: The output of this command can be controlled using the TRACK parameter. Three levels of information are available. See the appropriate section of steel design for more information about the TRACK parameter.

5.44.3 Member Selection Specification

**Objectiv
e**

This command is used to execute the MEMBER SELECTION.

General Format

```
SELECT {MEMBER memb-list | ALL | Member-group-name |
deck- name | PMEMB pmember-list}
```

Description

Using this command, the program selects specific members, based on the constraints on the values of the parameters and the specific code. The selection is made using the latest results and analysis sections iterating until the size of less weight is chosen. See Section 3 of this manual for details.

> **Notes: The output of this command can be controlled using the TRACK parameter. Three levels of detail are available. See the appropriate section Steel Design for more information on the TRACK parameter. Member selection can be made after an analysis has been performed do. Consequently, the command for the analysis has to be specified before the MEMBER SELECT command may be specified.**

5.44.4 Selecting Members for Optimization

**Objectiv
e**

This command executes the member selection optimized using a technique based on multiple iterations of design and analysis.

General Format

<div align="center">

SELECT OPTIMIZED

</div>

Description

Through this command, the program selects all based on a sophisticated optimization technique members. This method requires multiple analysis of the stiffness matrix, as well as the iteration of sizes until the minimum weight of the whole structure is obtained. This command should be used with caution, since it requires more processing time to solve a structure.

Notes

1. The output of this command can be controlled via the TRACK parameter. Three levels of information are available. See appropriate steel design for more information about the TRACK parameter.

2. This command may involve multiple cycles of analysis and design iterations so that could be a very slow process.

5.44.5 Welding Specification Selection

Objective

This command performs the selection of sizes for specified Welding members.

General Format

SELECT WELD (TRUSS) {MEMBER memb-list | ALL }

Description

Through this command, the program selects weld sizes at the beginning and end of the specified members. The selections are tabulated with all necessary information. TRUSS If the command is used, the program designed for angle welds or double members
angle connected to one another to strengthen the splice plates with welding along the length of the member.

> **Note: The calculation of weld size can also be controlled parametrically. For more information on the available parameters, see the appropriate section in Steel Design.**

5.45 Group specifications

Objective

This command is used for grouping members within the analysis and design of steel.

General Format

(FIXED GROUP)

GROUP MEMBER memb prop-spec-list (SAME AS i_1)

prop-spec = {AX | SY | SZ }

Description

This command allows the program to group specific members for analysis according to their most important properties. If the member number in the SAME AS i1 command, the program provides members grouped according to i1. This command is typically used after the selection of members, and where the members can be grouped for further process.

If FIXED GROUP option is used, the specified group will be retained in memory for the program and will be used in selecting operations and subsequent members of the cluster results will not be seen unless a MEMBER SELECT operation is performed.

Example 1

```
SZ MEMB GROUP TO 1 3 7 15 December
MEMB GROUP 17 TO 23 27 SAME AS 30
```

In this example, members 1, 3, 7 to 12 and 15 are assigned with the same properties according to which these members have the largest section modulus.

Members from 17 to 23 and 27 are assigned with the same properties as the member 30, regardless of whether the member 30 has area smaller or larger section. AX is the default property on which grouping is based.

Example 2

```
FIXED GROUP GROUP
1 TO 5 MEMB
SELECT OPTIMIZED
```

In the above example, using the GROUP command FIXED illustrated. OPTIMIZED SELECT command involves the following three processes,

1. Member Selection

2. Grouping members (1-5)

3. Analysis

Until after several iterations convergence properties of members in consecutive cycles is obtained. FIXED GROUP command is necessary for the execution of step 2 in the cycle.

Note: This command is typically used after the selection of members for further analysis and design. This option can be effectively used to develop a design-oriented, where various members need to be the same size.

5.46 Specification of Steel Estimating

Objectiv
e

This command is used to obtain a summary of all the steel sections used along with their lengths and weights.

General format:

```
STEEL (MEMBER) TAKE (OFF)
```

Description

This command provides a complete list of all the different sections of the tables used in the steel structure. The tabulated list include the total length of each section name and its total weight. This can be useful in estimating quantities of steel. The MEMBER option lists the length and weight of each member by number, the type of profile, weight and length.

Note: This option can be effectively used to obtain a quick estimate of the amount of steel in the structure.

5.47 Concrete Design Specification

This command describes the specifications for the particular design. The design procedure implemented STAAD.Pro specifically consists of the following steps.

1. Home Design

2. Specifying parameters

3. Specifying design requirements

4. Requisition estimate

5. Conclusion Design

The design section 5.47.1 describes the design start command. About 5.47.2 section relation to specifying parameters. The specifications of design requirements are described in Sections 5.47.3. The particular estimation command described in section

5.47.4 and finally, the command completion design is described in section 5.47.5.

5.47.1 Home design specifically

Objective

This command is used to start the particular design.

General format:

<u>STAR</u>T   <u>CONC</u>RETE   <u>DES</u>IGN

Description

This command starts the particular design specifications. With this, the design parameters are automatically placed in their corresponding default values (as shown in Table 3.1). Without this command, any of the following commands concrete design will be recognized.

Note: This command must be included before any other command particular design.

5.47.2 Concrete Design Parameters

**Objectiv
e**

This group of commands are used to specify the parameters that control the particular design.
General Format

Design-code CODE

parameter-name f_1{MEMBER member-list | ALL }

Description

Parameter-name refers to particular parameters described in Table 3.1. f1 is the value of the parameter. Note that this value is always given in the units in use. The UNIT command is also accepted during any phase of the particular design.

Note: All values should be provided according to the system of units in use. For the default values of the parameters, refer to Section 3.

5.47.3 Command Concrete Design

**Objectiv
e**

This command is used to specify the type of design required. Members can be designed as beams, columns or elements.

General Format

<u>DES</u>IGN {BEAM | COLUMN | ELEMENT memb-list} {| (ALL)}

Description

Members will be designed must be specified as BEAM, COLUMN or ELEMENT. Note that members, once they have been designed as beams, can not be redesigned again as columns, or vice versa.

> Note: Only the plates can be designed as elements.

5.47.4 Estimation of Concrete

Objectiv e

This command is used to obtain an estimate of the total volume of concrete and rebar that will be used and their respective weights.

General format:

<u>WITH</u>CRETE T AKE OFF

Description

This command can be used to print the total volume of concrete and the number of rods and their respective weights designed for members.

Sample output:

```
            TAKE OFF CONCRETE ************** **************
                 (DESIGNED FOR BEAMS AND COLUMNS
                              ABOVE)


            TOTAL VOLUME OF CONCRETE BAR SIZE =

                87.50 CU.FT          WEIGHT

                  NUMBER             (In lbs)
                 --------            --------

                    4                805.03

                    6                91.60
```

8	1137.60

9	653.84
11	818.67

TOTAL *** =	3506.74

Note: This command is used effectively to quickly estimate quantities.

5.47.5 Completion of Concrete Design

Objective

This command should be used to finalize the design in particular.

General format:

END WITHCRETE D ESIGN

Description

This command specifically design concludes after which normal STAAD.Pro commands continue.

Example

```
START   CONCRETE   DESIGN
CODE ACI
40.0 FYMAIN ALL
FC 3.0 ALL
DESIGN TO BEAM 1 July 4
COLUMN DESIGN TO 16 September 12
DESIGN ELEMENT 20 TO 30 END
```

Note: Without this command, subsequent commands will not be recognized STAAD.Pro.

5.48 Foundations Design Specification

In previous versions of the program STAAD.Pro (For example: STAAD-III). The foundation design specifications was together with the complete package in a single program, such as performing other types of design that just discussed (Design of Steel and Concrete).

The foundation design can be performed using the program separately from STAAD.foundation through user interface design similar to that of STAAD.Pro foundations.

5.49 Design Specification Walls Cutters

STAAD performs the design of reinforced concrete shear walls by the following codes: ACI 318-02 (American), BS 8110 (UK) and is 456: 2000 (Hindi). To design a shear wall, you must first be modeled using the surface element.

The Scope of the design depends on whether the definition of the panel, as explained in Section 5.49.1, precedes the input block design shear walls as explained in Section 5.49.2.

a. No definition of Panel

   Design is made to the specified horizontally full section, located at a distance c from the origin of the local coordinate system.

b. The panels are defined.

   Design is made for all panels of the cross section located at a distance c from the commencement of the panel.

The design of shear walls is not available for dynamic load cases.

5.49.1 Wall Settings Panel for Walls Design Cutters

Due to the presence of openings may be defined three types of structural elements within the limits of a shear wall: wall, column and beam. For each of these entities, a different set of design and details of the rules applies. Types of panels

– Functionally different parts of the shear wall - are assigned manually. Allocation
is based on the geometry of the panel, the overall position and configuration of wall. Wall, column and beam panels are then designed according to the relevant provisions of the Code.

General Format

```
START PANTHE DEFInition
```

<u>SURI</u>**I PAN FACETHE j x1 y1 z1 x2 ptype z2 y2 ... xn yn**

zn PAN ENDTHE DEFInition

Where:

- i: Number of ordinary surface j:

- regular Number of panel

- ptype: panel type. For the ACI code, the available types are Wall, column and beam (for ACI 318 and 456). For the British code, only the type of wall is available.

- x1 y1 z1 (...): coordinates of the corners of the panel

5.49.2 Walls Home Design Cutters

It is used to initiate a design cutting concrete wall.

General Format

```
SHEARWALL START DESIGN
CODE {ACI | BRITISH | INDIAN}
parameter_name f1
DESIGN SHEARWALL (f2 AT) LIST wall_list
TRACK CREINF
i1 i2
SHEARWALL END DESIGN
```

Options are ACI code (ACI 318) or British (BS 8110). Where:

- *parameter-name = refers to the parameter name (s) listed in tabular form (Table 3.4) in section 3.8 of this manual for the ACI code and manual of international codes 456 and BS 8110: 2000.*

- f1 = Parameter value. Not all parameters required values. f2 = optional

- location in which the design is to make

Note: If the AT command is omitted, the design is appropriate for all cross wall panels or sections, as appropriate, as defined by the input values SURFACE SURFACE DIVISION DIVISION X or Y.

- i1 = Parameter column reinforcement. Distribution reinforcing panels within the column is controlled by the CREINF parameter, which can have one of three possible values:

    1. number of bars on all four sides of the column (the default),

    2. equal number of bars perpendicular to the plane of the wall, two sides

    3. equal numbers of two parallel rods on the wall plane faces.

- i2 = TRACK output parameter specifies how detailed should be the output of the

  design:
    1. indicates a core set of outcome data (default)
    2. Full design output is generated.

5.50 Various Special Commands

The following are two special commands, which can be used to control the precision characteristics of the elements of the stiffness matrix and the memory requirements for a particular problem.

5.50.1 Commando Precision

**Objectiv
e**
This command is used to control the characteristics of accuracy in verifying the stiffness matrix of the structure.

General Format

$$\underline{\text{PREC}}\text{ision} \quad \text{f1}$$

Where:

- f1 = number against which the stiffness coefficients be checked (the default is 0.01)

Note: This command must be provided before JOINT COORDINATES specifications.

Description

The global stiffness matrix of a patterned structure right mind is always a symmetric matrix with positive elements on the main diagonal. Negative or positive, but very small components are usually a result of incorrect modeling in a matrix

poorly conditioned stiffness. In STAAD.Pro, all numerically lower stiffness coefficients of 0.01 are identified as potential sources of instability and in such cases warning messages are sent to the user. In certain special situations, these stiffness coefficients may represent correct numbers. So to eliminate the error message in these circumstances, use the PRECISION command re–specifying the value against which the coefficients are double checked. This command should be included at the beginning of the input file.

5.50.2 Command Memory Entry

**Objectiv
e**

This command is used to control the memory requirements in the event that there is an overflow problem.

General Format

<u>INP</u>UT M EMORY f_1

Where:

- $f1$ = memory capacity (the default for the PC version is 16000)

Description

This command can be used in a state of overflow, to reduce the memory requested by the problem. A default value of 16000 is preset in the program. A low number that it can reduce the memory requirements. Problems that require more memory, can be solved using this command. Using this command will result in slower execution. So, we must be cautious with the use of this command. If you have the resources, increasing the number will result in faster execution. This command should be included at the beginning of the input file.

Notes

1. This command must be provided before specifying CCORDINATES JOINT.

2. The value of f1 do can be specified as zero, in which case the command INPUT MEMORY result in a minimum demand for central memory problem. However, the user is warned of its use, since it will reduce the speed of execution. Thus, one should consider the use of this command only when a message has appeared OVERFLOW CORE and all other means of reducing the demand for the main memory have been extensively analyzed.

5.51 Specifying Saving Files

Objective

This command can be used to store execution data and analytical results for later execution of the program.

General Format

$$\underline{SAV}E\ (a_1\ To_2)$$

Description

- a1 = name of the file in which the analytical results and data will be saved.
  Any name up to 12 characters, beginning with alphabetic characters, will be accepted.

- a2 = name of a second file in which the load data will be saved.
  Any name different from the first and up to 12 characters, beginning with an alphabetic character, will be accepted

Note: If the file names are not provided, the data will be saved in files named TEM1 and TEM2 respectively.

5.52 Specifying File Restoration

Objective

This command can be used to restore data from a saved by the SAVE command execution.

General Format

$$\underline{RES}TORE\ (a_1\ To_2)$$

Description

a1 and a2 are the same as on file names are specified in the SAVE command. This command is used to restore a previous run of STAAD.Pro that has been saved with the SAVE command. If a1 and a2 names are omitted, the program will take names as TEM1 and TEM2 respectively. Thus, an updated, without defining any execution file name may be restored in the same way, ie, not specifying the names of the files in the RESTORE command.

**Exampl
e**

```
STAAD SPACE RESTORE
TITLE FILE1 FILE2
```

Note: This command will recognize the names of the files and TEM2 TEM1 default. Operations involving Finite Element can not be performed with the restored file.

5.53 Specifying the Implementation Completion

**Objectiv
e**

This command must be used to complete the execution of STAAD.Pro

General format:

EndISH

Description

This command must be provided as the last input command. With what is left terminating a running STAAD.Pro