Computer Based Structural Analysis and Design

INDEX

Chapter -1
Introduction

STAAD Pro is finite element based software generally used for structural analysis and design. STAAD Pro stands for "**St**ructural **A**nalysis and **D**esign "and is very user friendly and powerful software that provides solution for the required degree of accuracy in wide range of structures from simple residential building to complex industrial structures. STAAD Pro is commonly used in different origination and it is well proved that analysis gives the desired results for the purpose of structural design of RCC and Steel Structures. STAAD Pro was developed by Research Engineers International at Yorba Linda, CA in 1997. In late 2005, Research Engineers International was bought by Bentley Systems. An older version called STAAD-III. STAAD can make use of various forms of 1st order static analysis, 2 nd order P-Delta analysis, geometric non linear analysis, buckling analysis etc. It can also make use of various dynamic analyses like time history analysis and response spectrum.

Founder of Research Engineers and Net Guru, Mr. Amrit Das translated his ideas into reality and created the first version of the STAAD-III. STAAD first version was built for DOS operating system and there was non graphical window. With time STAAD progresses to its own graphical environment.

Chapter -2
Geometry of the Structure

Geometry of the structure is an important part for the structural analysis and design in STAAD Pro.Depending on the variation in the geometry the structural stiffness will also change and inaccurate geometry may not produce correct stiffness matrix and subsequently after analysis design reactions (i.e., shear force, bending moments etc.) may be inaccurate.

Steps to be followed:

I. Opening a New File:

Click on the STAAD-Pro icon in the desk top. After clicking on that icon, this tab will appear

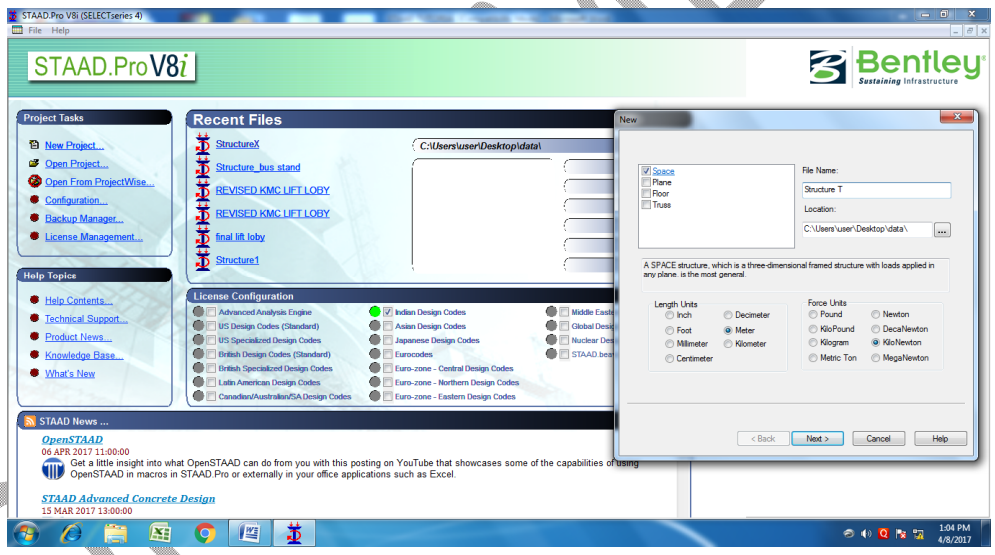

Fig1: Initial Starting Window

Now click on the new project under project task bar. A “new “wizard will seen. Click on the space option and give a file name and create location where you want to save the file. Select the length unit and force unit which you want to give. In the license configuration bar specific design code must be clicked. After completing all the task click on the next and finish option simultaneously.

Whole structure wizard will be appearing. Click on the close of snap node/ beam.

After closing the snap node/beam, in the right side node wizard will appear.

II. Coordinates :

.

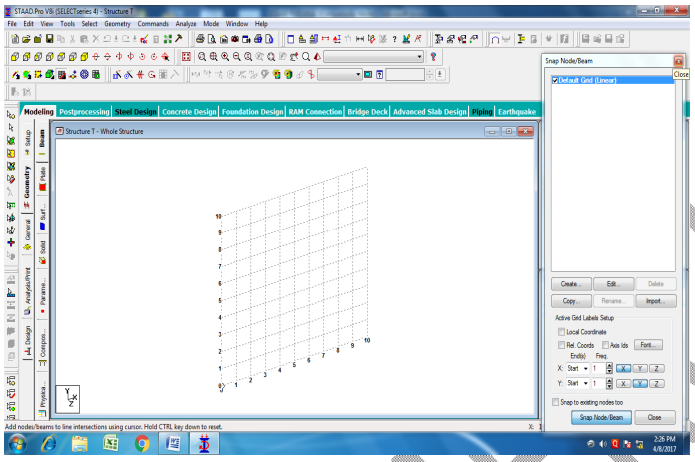

Fig2: Starting of a Geometry preparation

Put the co-ordinates of the foundations supports of the structure or nodes of the plan in the node wizard. After putting the co-ordinates the nodes will visible in the screen. Here two points are created by these method; we can create all the points by the same method. But there are another option is available, that is transitional repeat. Different available cursors are shown below.

Node cursor

Beam cursor

Transilational repeat cursor

Few more cursors are also available in the STAAD window.

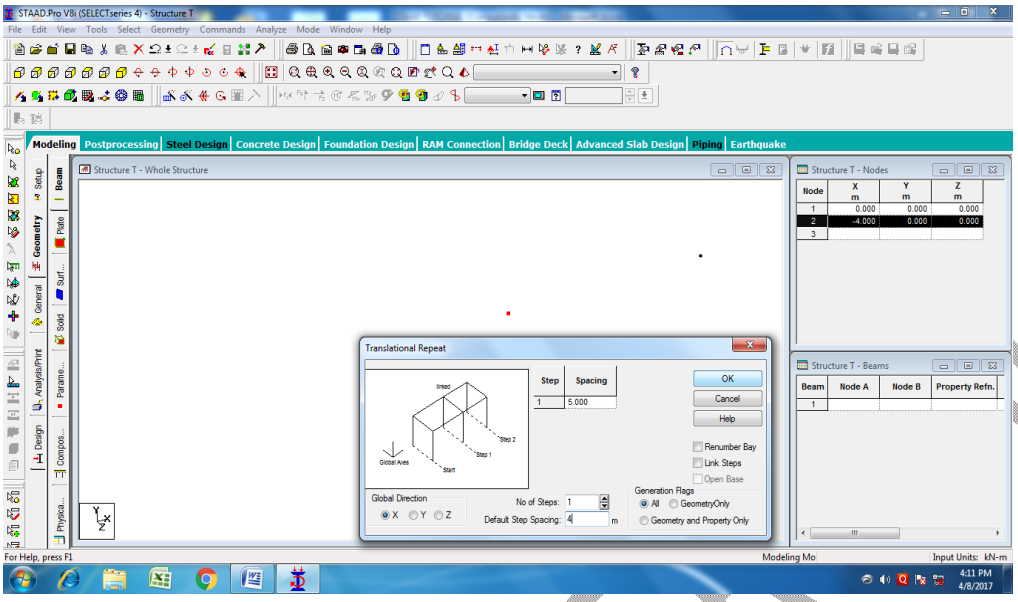

Fig3: Translational Repeat of a Frame

- ✓ Select the node cursor, and select the required node after that select the transitional repeat option. Now choose the x or y or z direction which are required for the desired direction of extension. Side by side select default step stepping, No. of steps and click on the link steps if required.
- ✓ Do the same thing for creating the upper floor but remember here we must click on link steps, for creating the columns.
- ✓ Alternately click on the add beam cursor and join the nodes by this cursor to create the beams of upper floor.
- ✓ Now to add a node in any position of beam, click on the beam by add insert node cursor and put the value of distance from initial point and new created node distance.

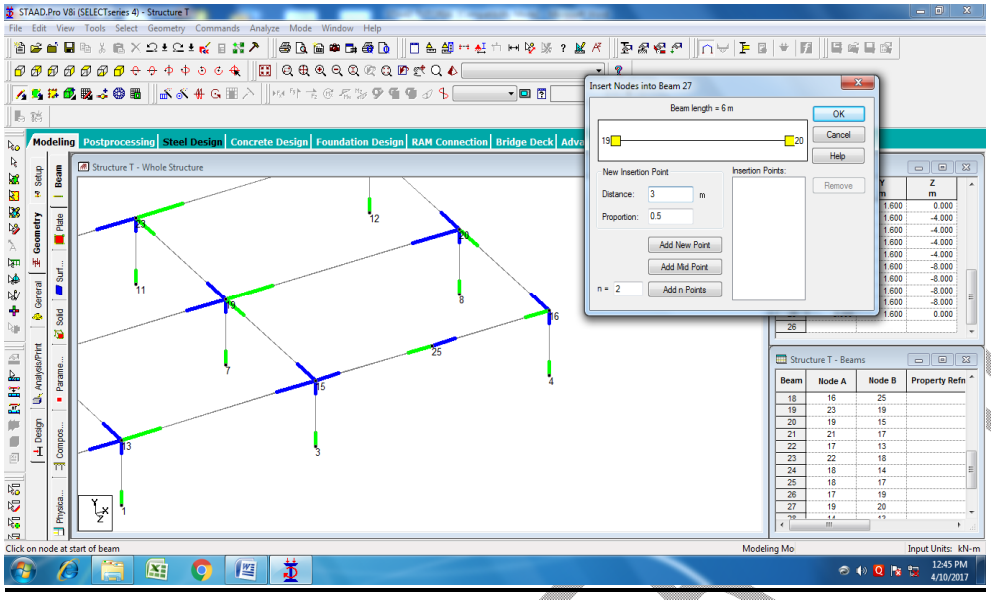

Fig4: Insertion of node in a member

III. **Supports:**

- ✓ Go to the general option in (Left side menu bar) after that click on supports. Support whole structure will appear in the left side of the screen. In this wizard click on create option and select your requirement (e.g. fixed, Pinned, Releases etc.).Then click on add. Now the selected support will be enlisted in the whole structure wizard.

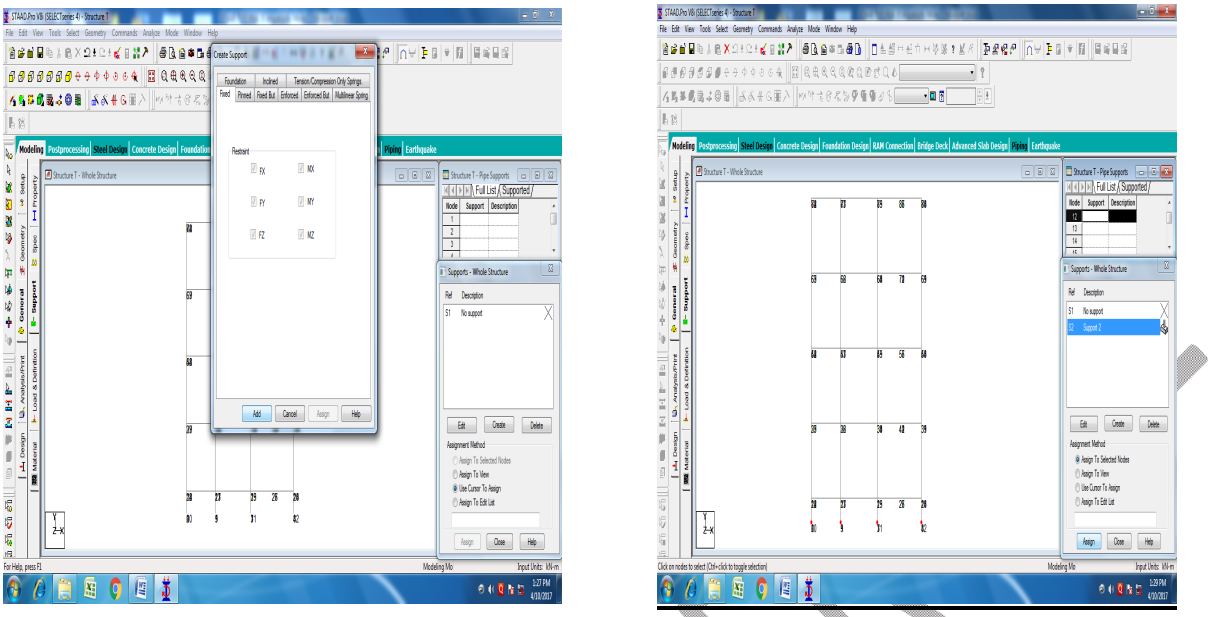

Fig5: Support Specification and Assigning of Support (A)

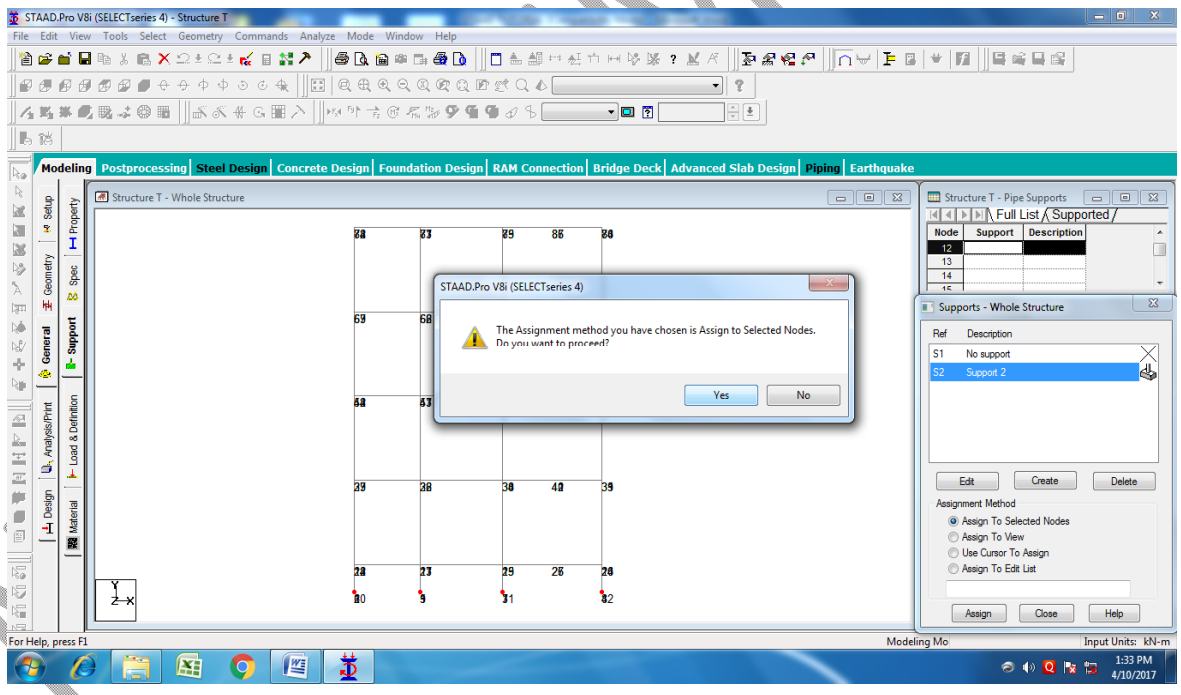

Fig6: Support Specification and Assigning of Support (B)

IV. Material Properties:

After completing the whole frame suitable sizes of all the columns and beams etc. shall be provided.

For beams of a 2-3 story RCC framed structure generally 250x350, 250x400mm or 250x450 mm sizes are adopted. Material shall be selected as concrete.

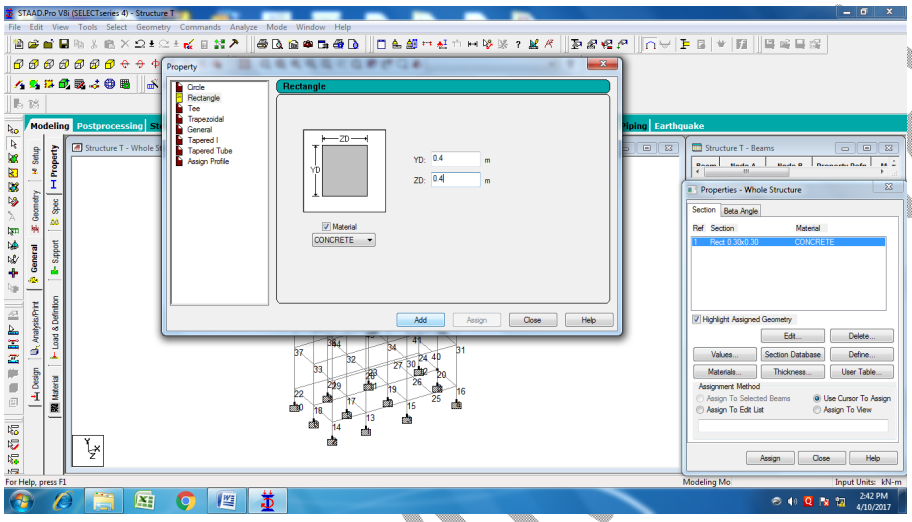

Fig7: Properties of the Members

V. Moment Release:

When it is required to create the hinge type connection suitable moments (My, Mz) shall be released at both the ends. Sometimes torsion to be released at that time Mx shall be released. This situation may occurs when a secondary beam is connected with a main beam and the connection is simply-supported.

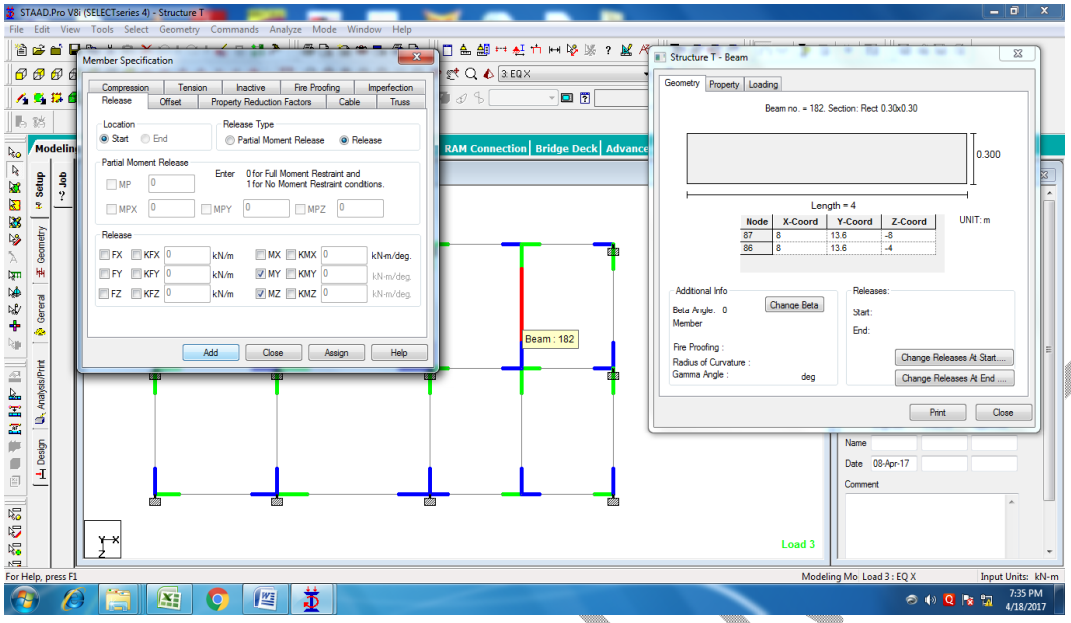

Fig8: Moment Release of a Beam

Chapter -2

Loads on Structures

There are few basic loads to be generated in the STAAD Pro for the analysis and design of the structure

Dead Load (DL)

In the Dead load of the structure the following loads to be applied

i. Self weight of slab

Suppose 120mm thk slab. So self weight of the slab=0.120*25=3.00kN/sqm.

ii. Floor finish weight + Ceiling plaster weight

iii. Weight of outer wall -200/250thk outer wall load shall be calculated as UDL.

iv. Weight of light partition wall - As provided in code or

v. Weight of parapet wall – As per the height and thickness of the parapet wall.

vi. Weight of the roof treatment – As per the requirement of the roof treatment

vii. Weight of any equipment load which are fixed.

(Please refer IS 875(part-1)(1987) for weight of different materials)

Live Load (LL)

- Floor live load - For residential building dowelling house LL is 2.0kn/sqm
- Roof live load – For flat accessible roof it is 1.5kn/sqm . For inaccessible roof it is 0.75kN/sqm And for sloped roof it shall be as per IS: 875(part-II)

(Please refer IS 875(part-II) (1987) for weight of different materials)

Earthquake Load (EQL)

- Earthquake load shall be calculated as per IS: 1893(part-1)-2016.
- Joint weights shall be calculated. (Joint weights are calculated for DL+0.5LL if LL is greater then 3.0kN/sqm otherwise Dl+0.25LL if LL is upto 3.0Kn/sqm.
- Equivalent static load (Vb) shall be calculated and it shall be distributed in each floor.
- Dynamic analysis shall be performed if required by Time history analysis or Response Spectrum method.

- Earthquake shall be applied in both the orthogonal direction and it shall be reversed.

(Please refer IS 1893(part-I) (2016 or 2002) for earthquake load calculation)

Wind Load (WL)

- Wind load shall be calculated as per IS: 875(part-III)-1987.
- Basic wind speed (Vb) of a place is known to us. It is provided in the IS 875 part III.
- k1, k2, k3 factors are to be decided. K1=Probability factor, k2= Height factor,K3= topographic factor.
- Design wind speed to be calculated. (Vz=k1xk2xk3xVb)
- Design wind pressure to be calculated.(Pz=0.6xVz^2)
- External and Internal pressure coefficient shall be calculated from IS 875(part-III):1987
- Wind load on walls and roof shall be calculated.
- Wind shall be applied in both the orthogonal direction and it shall be reversed.

(Please refer IS 875(part-III) (1987) for Wind load calculation)

Factored Load combinations as per IS: 456-2000

1) 1.5(DL+LL)
2) 1.2(DL+LL+EQX)
3) 1.2(DL+LL-EQX)
4) 1.2(DL+LL+EQZ)
5) 1.2(DL+LL-EQZ)
6) 1.5(DL+EQX)
7) 1.5(DL-EQX)
8) 1.5(DL+EQZ)
9) 1.5(DL-EQZ)
10) 0.9DL+1.5EQX
11) 0.9DL-1.5EQX
12) 0.9DL+1.5EQZ

13) 0.9DL-1.5EQZ

Proper Live load reduction shall be done as per IS: 875(part-II) (1987) for the multistoried Buildings.

Unfactored Load combinations (For Foundation base Pressure Checking and Limit State of Serviceability):

1) 1.0(DL+LL)
2) 1.0(DL+LL+EQX)
3) 1.0(DL+LL-EQX)
4) 1.0(DL+LL+EQZ)
5) 1.0(DL+LL-EQZ)
6) 1.0(DL+EQX)
7) 1.0(DL-EQX)
8) 1.0(DL+EQZ)
9) 1.0(DL-EQZ)

Proper Live load reduction shall be done as per IS: 875(part-II) (1987) for the multistoried Buildings

Steps to be followed:

1) Basic Load Case Generation:

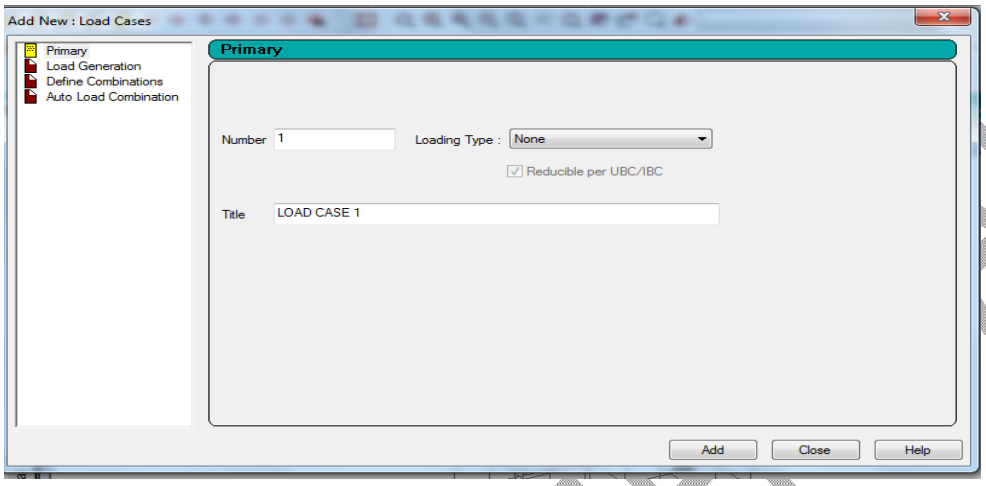

Fig: 9 Basic Load Case Generation

2) Dead and Live Loads:

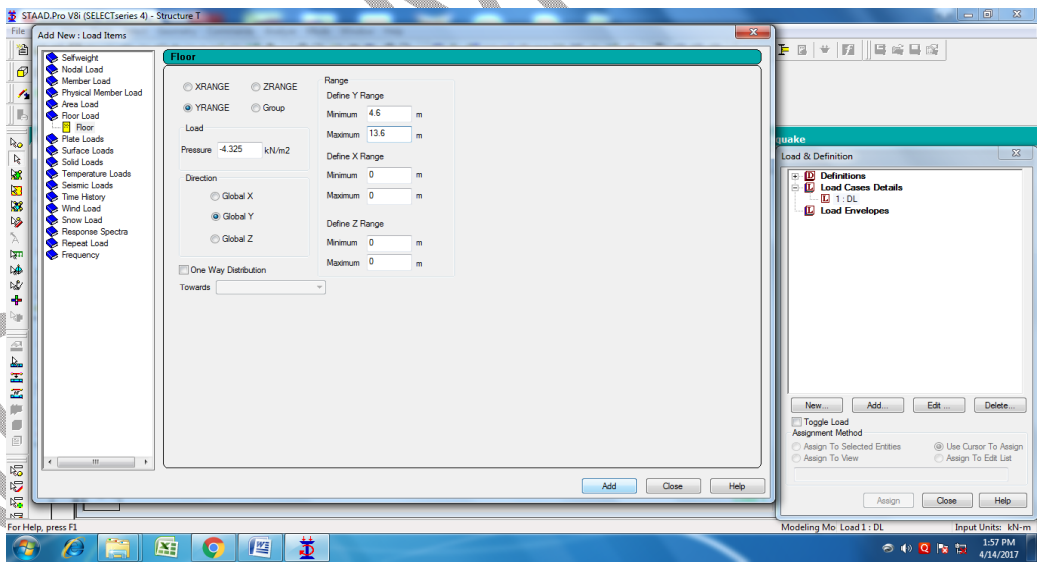

Fig: 10 Floor Load Generations

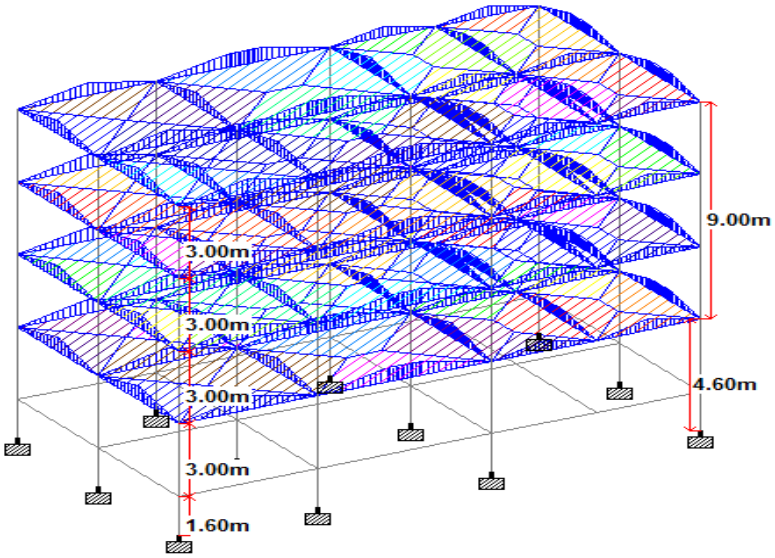

Fig: 11 Floor Dead and Live Load

✓ Continue the process for other dead loads (which may acts as a member load) .Click on member load and choose which is required like uniform force, uniform moment, concentrated force etc. if you want to give the uniform forces then give a look on d1 and d2 value.

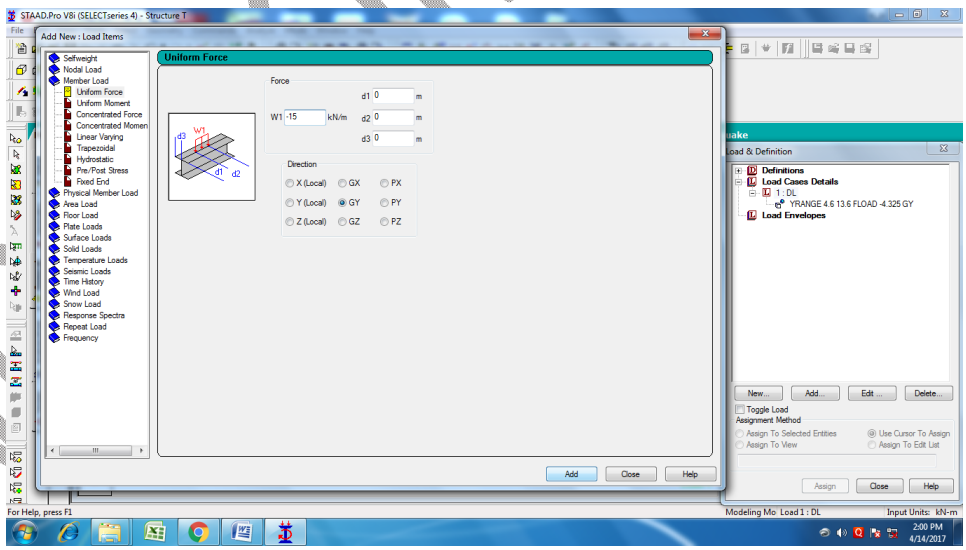

Fig: 12 Application of member load

3) **Earthquake Load:**

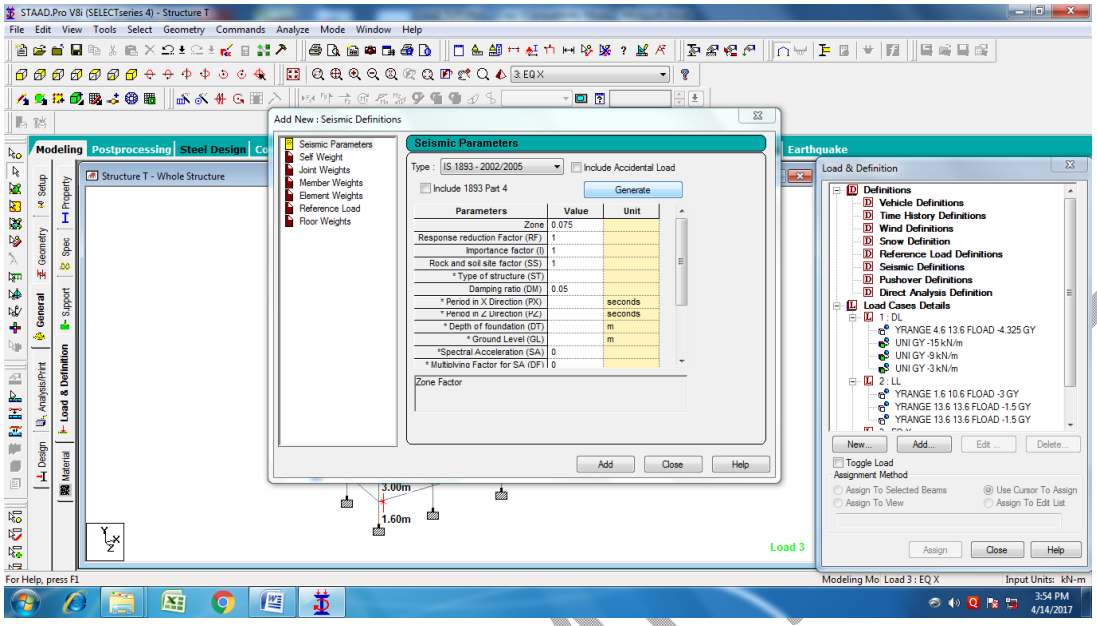

Fig: 13 Earthquake Load Definition (A)

- Click on generate the is 1893-2002 seismic parameter will be appeared, now choose the city put the value of response reduction factor (5 or 3) ,foundation depth, damping ratio ,select importance factor, rock/soil type , structure type .Completing all these steps click on generate again.

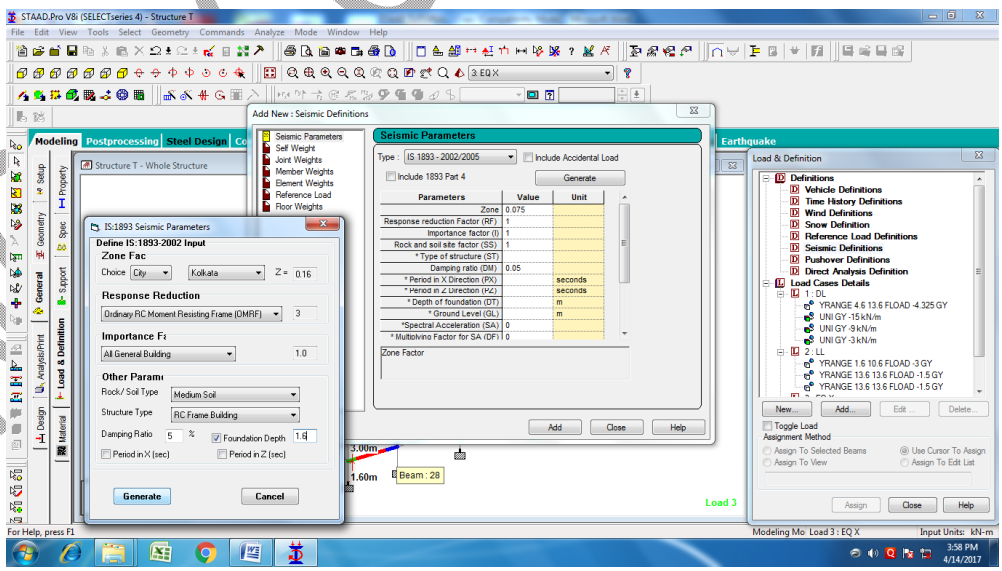

Fig: 14 Earthquake Load Definition (B)

- Calculate the lumped masses for each joint for (DL+0.5LL or DL+0.25LL) as per code in a separate file and the vertical reaction for the proposed combination shall be used as joint weights at different beam column joints.
- Now click on the seismic load named "EQ X" (load case details) and click on add simultaneously. Add new load items wizard will appear in the screen. Click on the seismic loads, choose the direction (X or Z) and put the factor value and close it by click on add & close. Follow the same process for "EQ Z".

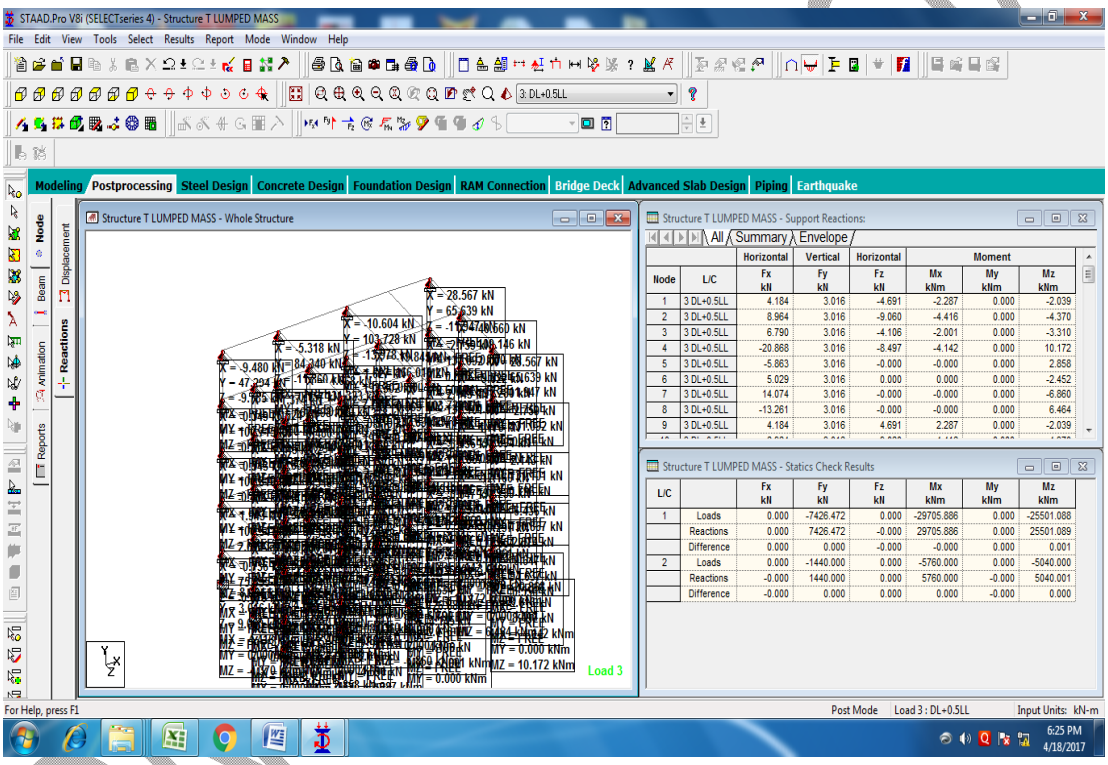

Fig: 15 Lumped Mass Calculation

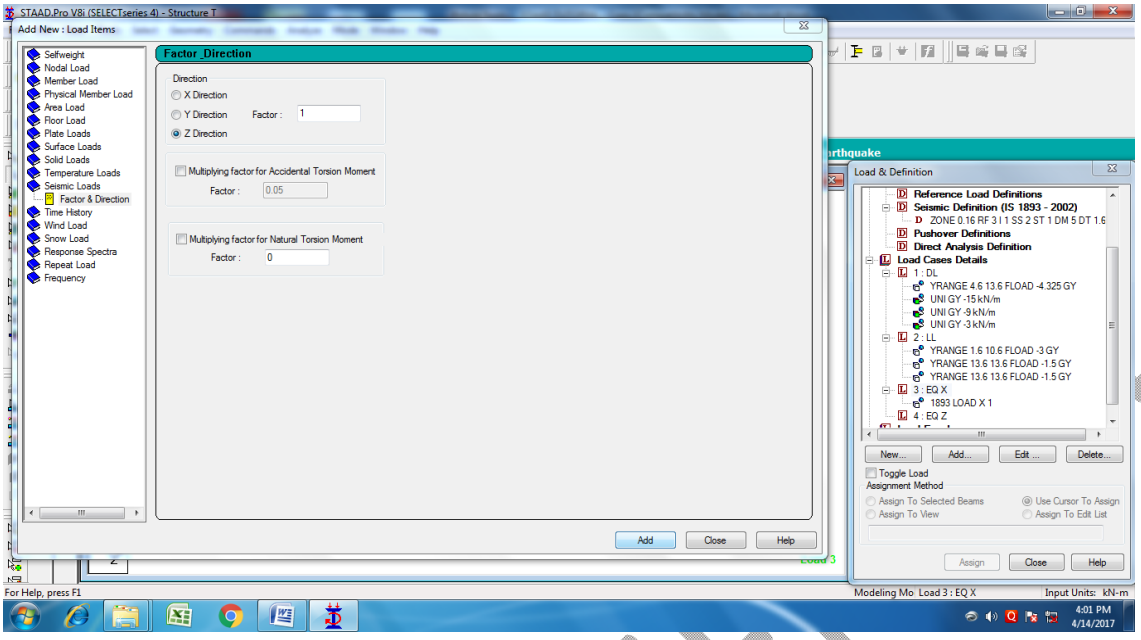

Fig: 16 Earthquake Load Case Creation

4) Generation of Load Combination:

To create the load combination at first go to the load case details and add new load case details wizard will appear change the name as your requirement select the loads from available load case and put the factors as the requirements of load case and click add. Follow the same process for other load combinations.

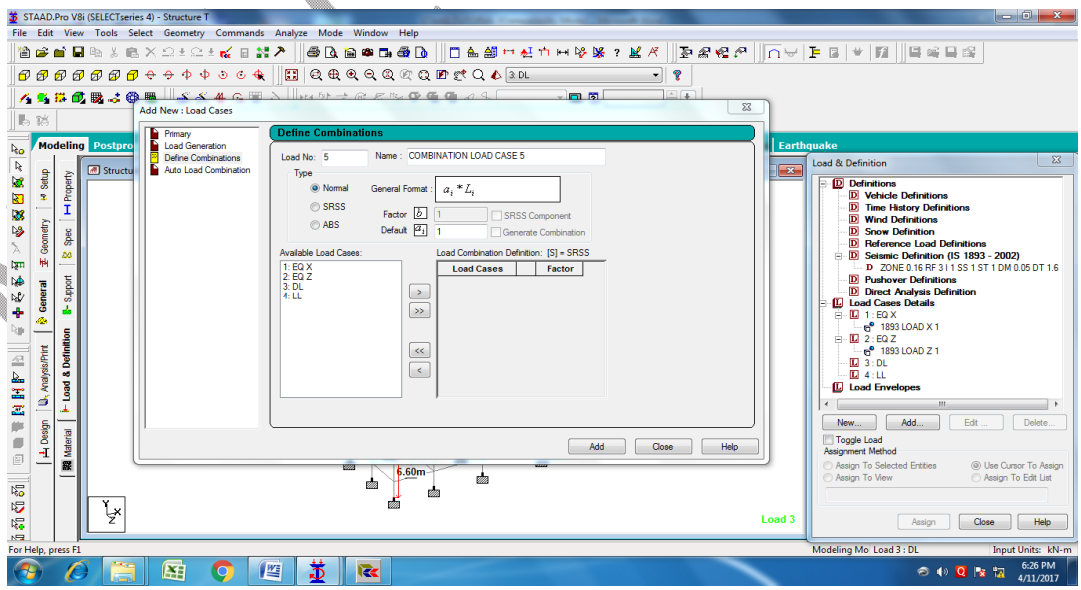

Fig: 17 Load Combinations Generation

Chapter -3

Analysis of Structure

Static Analysis: Linear Static analysis can be performed for the structure.

Main menu> Analysis> Perform Analysis

```
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 0; 2 0 0 4.25; 3 0 0 12.5; 4 4.25 0 0; 5 4.25 0 4.25; 6 4.25 0 12.5;
7 8.5 0 0; 8 8.5 0 4.25; 9 8.5 0 12.5; 10 12.75 0 0; 11 12.75 0 4.25;
12 12.75 0 12.5; 13 17 0 0; 14 17 0 4.25; 15 17 0 12.5; 16 21.25 0 0;
17 21.25 0 4.25; 18 21.25 0 12.5; 19 25.5 0 0; 20 25.5 0 4.25; 21 25.5 0 12.5;
22 29.75 0 0; 23 29.75 0 4.25; 24 29.75 0 12.5; 25 34 0 0; 26 34 0 4.25;
27 34 0 12.5; 28 38.25 0 0; 29 38.25 0 4.25; 30 38.25 0 12.5; 31 0 7.95 0;
32 0 7.95 4.25; 33 0 7.95 12.5; 34 4.25 7.95 0; 35 4.25 7.95 4.25;
36 4.25 7.95 12.5; 37 8.5 7.95 0; 38 8.5 7.95 4.25; 39 8.5 7.95 12.5;
40 12.75 7.95 0; 41 12.75 7.95 4.25; 42 12.75 7.95 12.5; 43 17 7.95 0;
44 17 7.95 4.25; 45 17 7.95 12.5; 46 21.25 7.95 0; 47 21.25 7.95 4.25;
48 21.25 7.95 12.5; 49 25.5 7.95 0; 50 25.5 7.95 4.25; 51 25.5 7.95 12.5;
52 29.75 7.95 0; 53 29.75 7.95 4.25; 54 29.75 7.95 12.5; 55 34 7.95 0;
56 34 7.95 4.25; 57 34 7.95 12.5; 58 38.25 7.95 0; 59 38.25 7.95 4.25;
60 38.25 7.95 12.5; 61 0 -1.2 0; 62 0 -1.2 4.25; 63 0 -1.2 12.5;
64 4.25 -1.2 0; 65 4.25 -1.2 4.25; 66 4.25 -1.2 12.5; 67 8.5 -1.2 0;
68 8.5 -1.2 4.25; 69 8.5 -1.2 12.5; 70 12.75 -1.2 0; 71 12.75 -1.2 4.25;
72 12.75 -1.2 12.5; 73 17 -1.2 0; 74 17 -1.2 4.25; 75 17 -1.2 12.5;
76 21.25 -1.2 0; 77 21.25 -1.2 4.25; 78 21.25 -1.2 12.5; 79 25.5 -1.2 0;
80 25.5 -1.2 4.25; 81 25.5 -1.2 12.5; 82 29.75 -1.2 0; 83 29.75 -1.2 4.25;
84 29.75 -1.2 12.5; 85 34 -1.2 0; 86 34 -1.2 4.25; 87 34 -1.2 12.5;
88 38.25 -1.2 0; 89 38.25 -1.2 4.25; 90 38.25 -1.2 12.5; 91 -4.25 0 4.25;
92 -4.25 0 12.5; 93 -4.25 7.95 4.25; 94 -4.25 7.95 12.5; 95 -4.25 -1.2 4.25;
96 -4.25 -1.2 12.5; 97 -4.25 7.25 4.25; 98 -4.25 7.25 12.5; 99 0 7.25 4.25;
100 0 7.25 12.5; 101 4.25 7.25 4.25; 102 4.25 7.25 12.5; 103 8.5 7.25 4.25;
104 8.5 7.25 12.5; 105 12.75 7.25 4.25; 106 12.75 7.25 12.5; 107 17 7.25 4.25;
108 17 7.25 12.5; 109 21.25 7.25 4.25; 110 21.25 7.25 12.5; 111 25.5 7.25 4.25;
112 25.5 7.25 12.5; 113 29.75 7.25 4.25; 114 29.75 7.25 12.5; 115 34 7.25 4.25;
116 34 7.25 12.5; 117 38.25 7.25 4.25; 118 38.25 7.25 12.5;
119 -4.25 7.25 6.15; 120 -4.25 7.25 10.15; 121 0 7.25 6.15; 122 0 7.25 10.15;
123 4.25 7.25 6.15; 124 4.25 7.25 10.15; 125 8.5 7.25 6.15; 126 8.5 7.25 10.15;
127 12.75 7.25 6.15; 128 12.75 7.25 10.15; 129 17 7.25 6.15; 130 17 7.25 10.15;
131 21.25 7.25 6.15; 132 21.25 7.25 10.15; 133 25.5 7.25 6.15;
134 25.5 7.25 10.15; 135 29.75 7.25 6.15; 136 29.75 7.25 10.15;
137 34 7.25 6.15; 138 34 7.25 10.15; 139 38.25 7.25 6.15; 140 38.25 7.25 10.15;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 1 4; 4 2 5; 5 3 6; 8 4 7; 9 5 8; 10 6 9; 13 7 10; 14 8 11;
15 9 12; 18 10 13; 19 11 14; 20 12 15; 23 13 16; 24 14 17; 25 15 18; 28 16 19;
r Help, press F1
```

Fig: 18 Typical STAAD Pro Input File

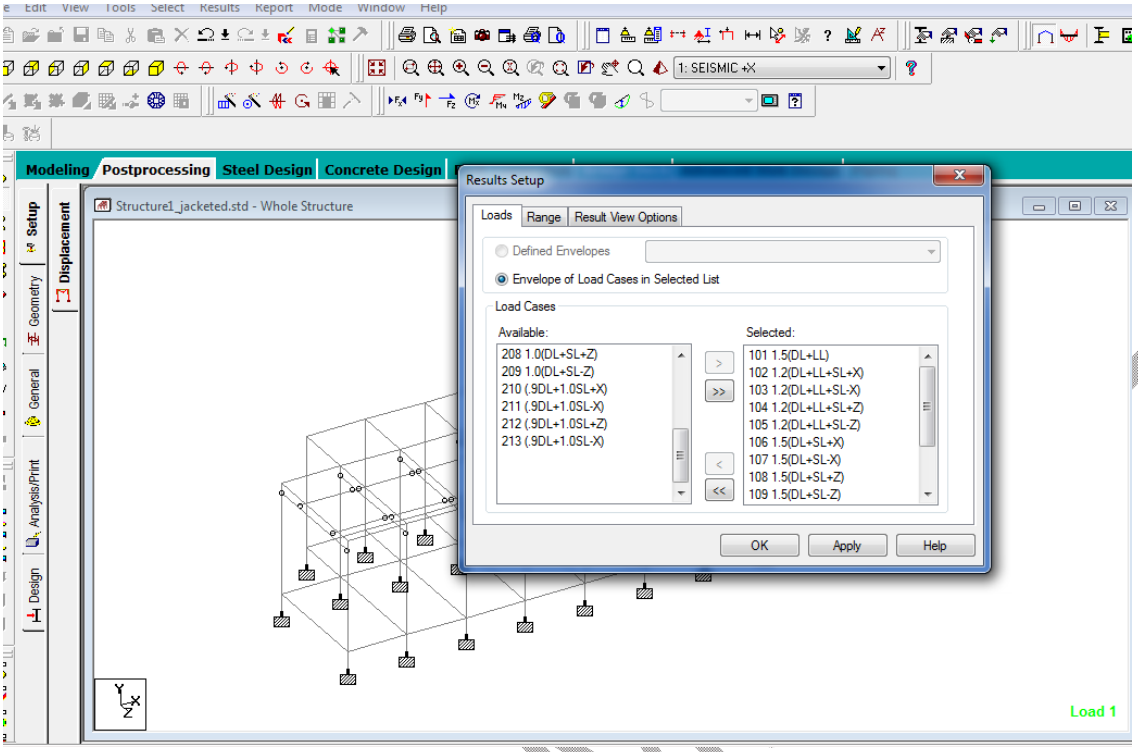

Fig: 19 Typical Selection of load cases for Post processing Data

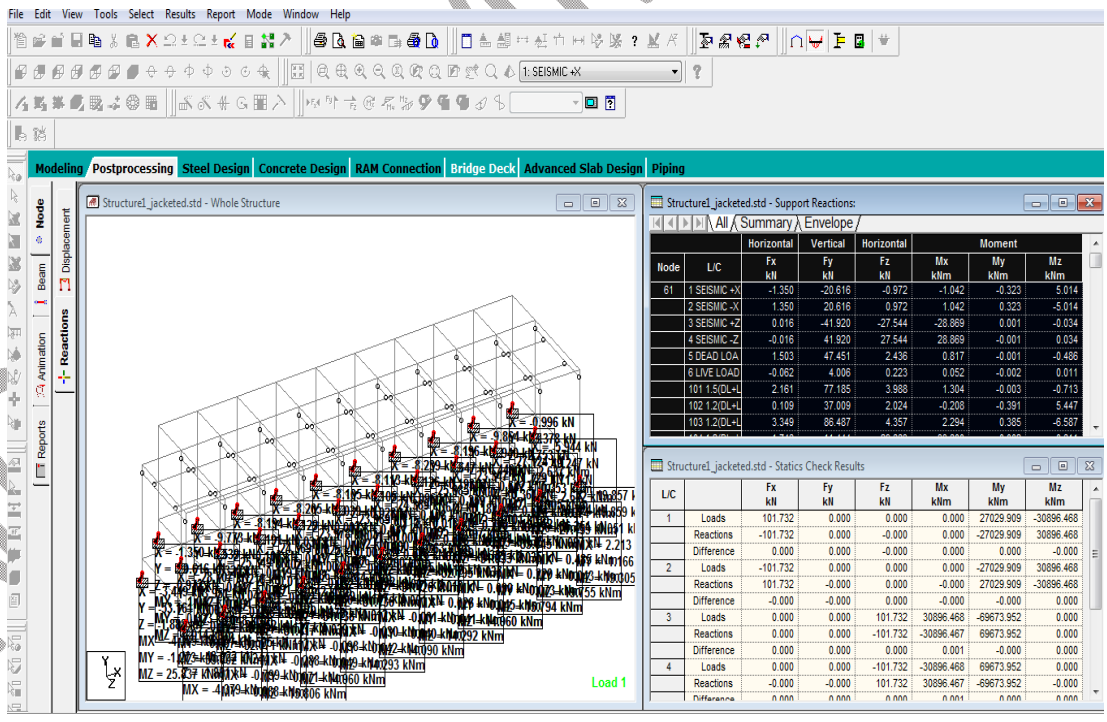

Fig: 20 Typical support reaction for any structure for different load cases

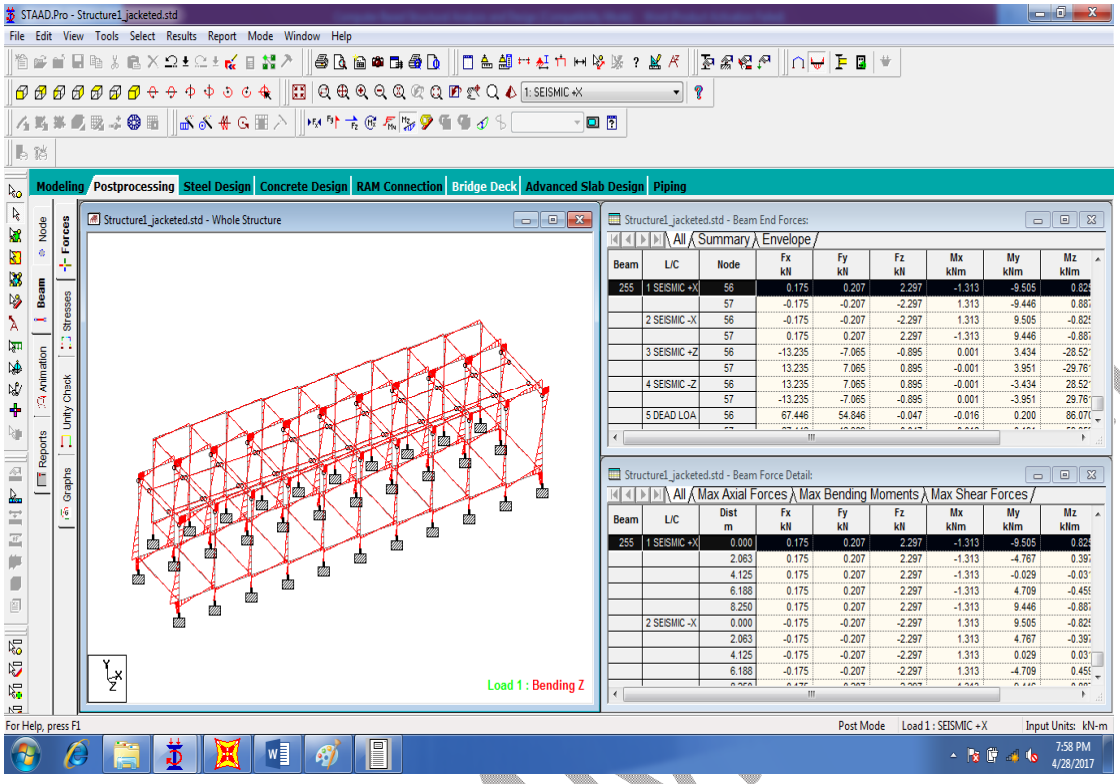

Fig: 21 Typical Member End Reaction for different load cases

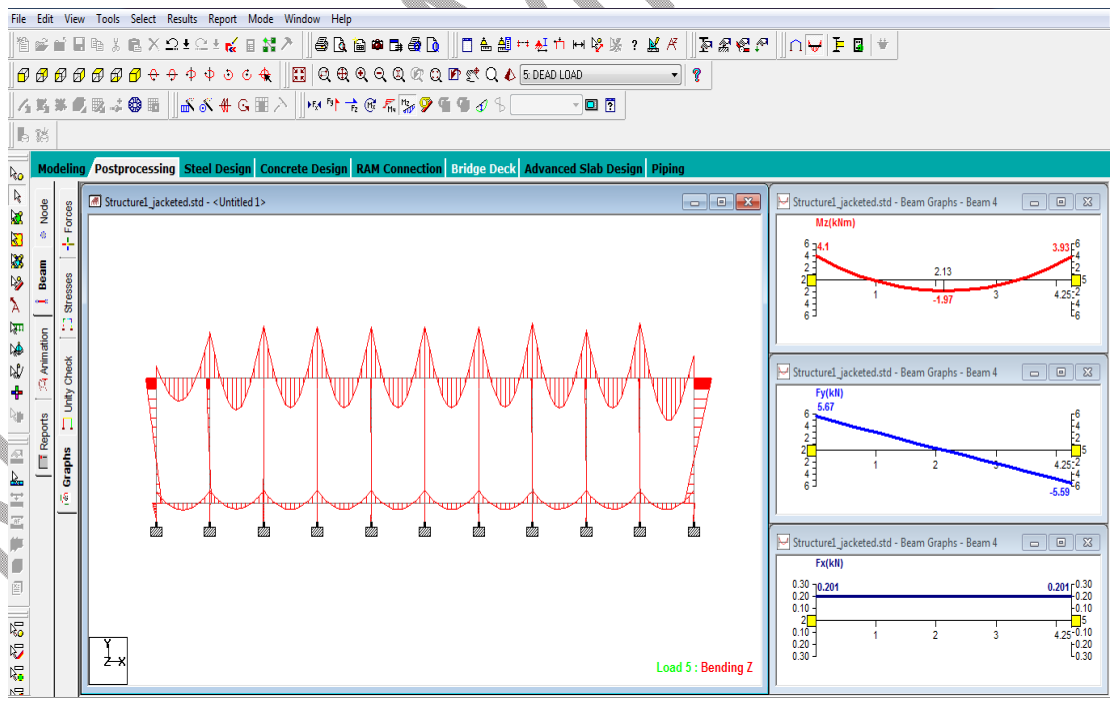

Fig: 22 Typical Bending Moment diagram for Dead Load of a structure

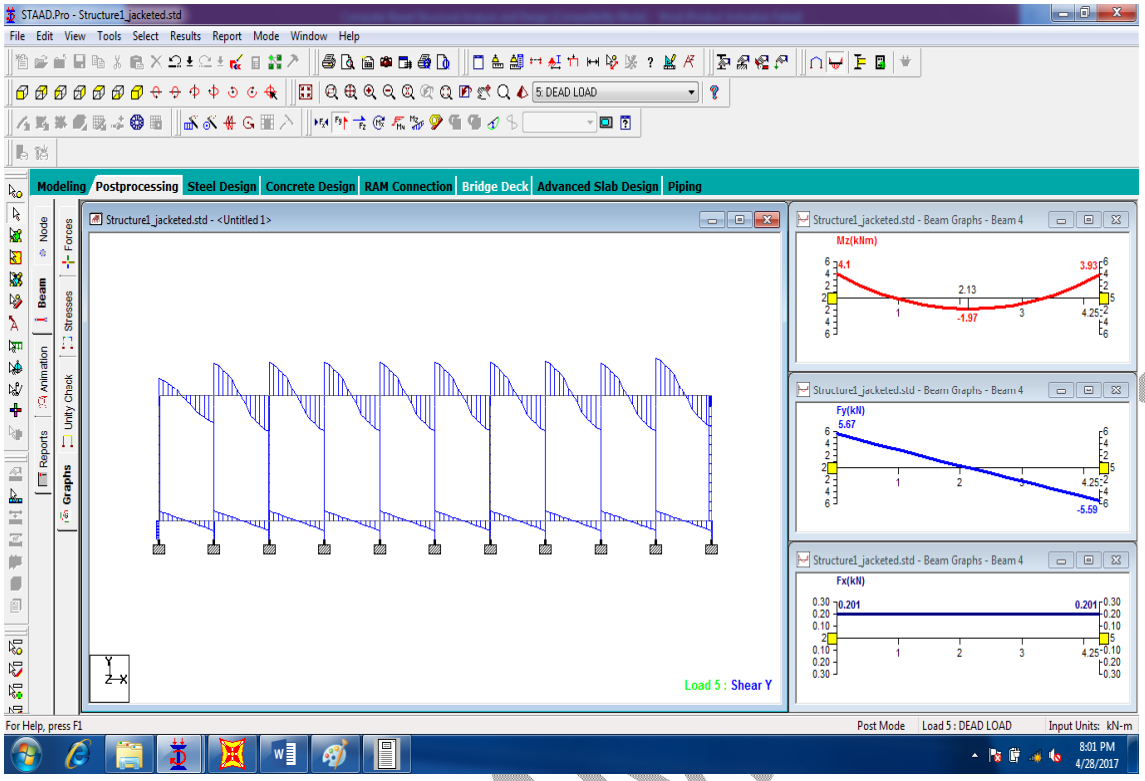

Fig: 23 Typical Shear Force diagram for Dead Load of a structure

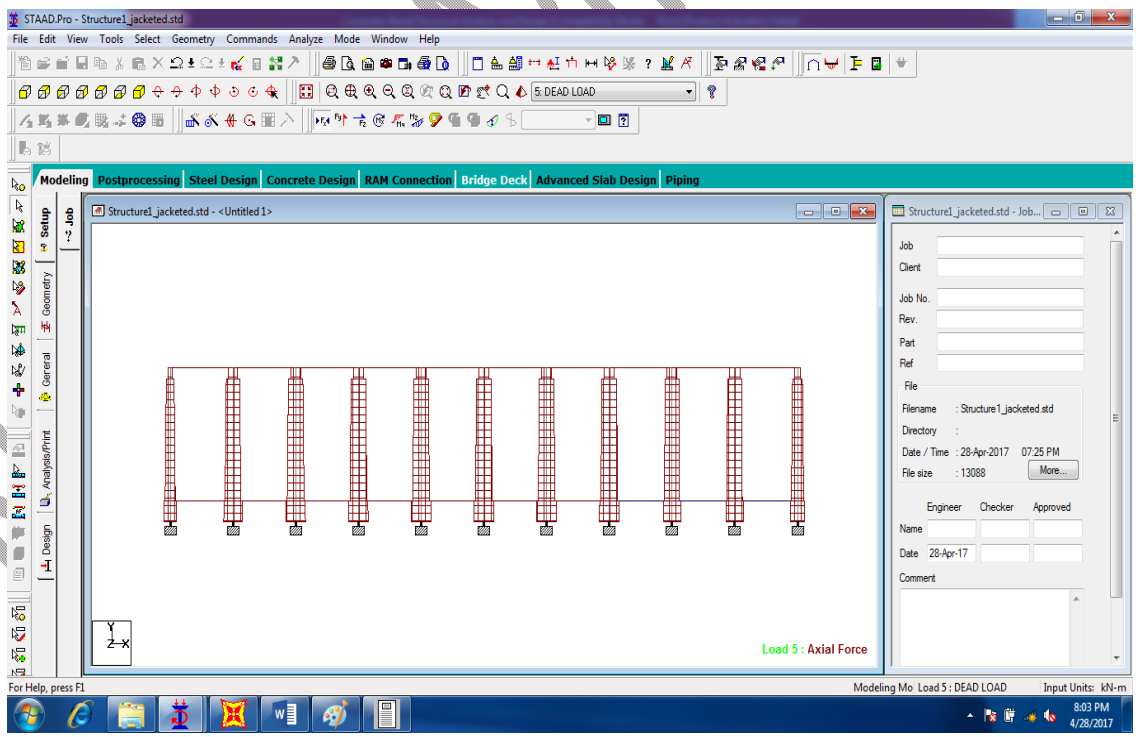

Fig: 24 Typical Axial Force diagram for Dead Load of a structure

---

Dynamic Analysis: Different Dynamic analysis also can be performed by STAAD like Time history analysis and Response spectrum analysis. For the detail dynamic analysis refer separate document.

Chapter -4

Design of RCC Structures

Design of RCC structures shall be performed in STAAD Pro with the help of proper design commands as per IS 456(2000).

Steps to be followed:

- Click on design > concrete and select "IS 456".

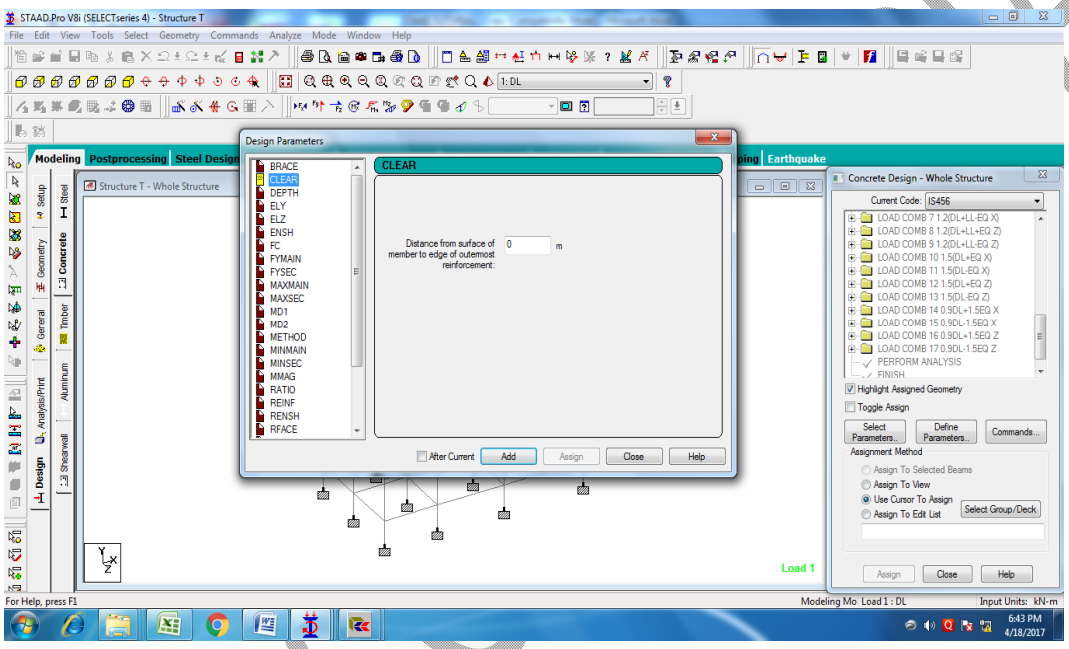

Fig: 25 Different commands for RCC design

- Now put the data for ***CLEAR*** (clear cover), ***FC*** (Grade of Concrete) ***FYMAIN***, ***FYSEC*** (Grade of steel for main steel and secondary reinforcements), ***RATIO, TORSION*** etc.
- The above values shall be provided for both Beams and Columns.

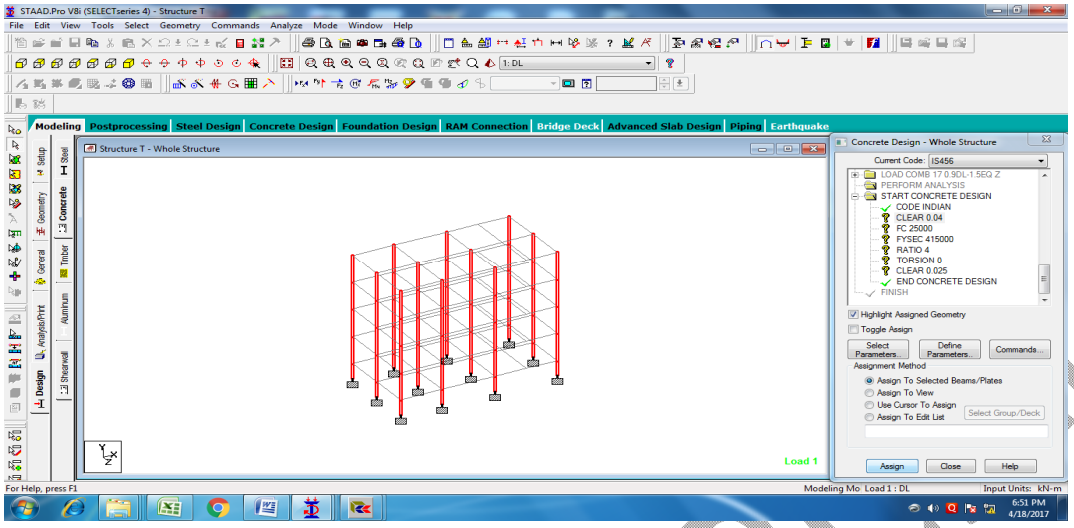

Fig: 26 Few specific commands for RCC column design

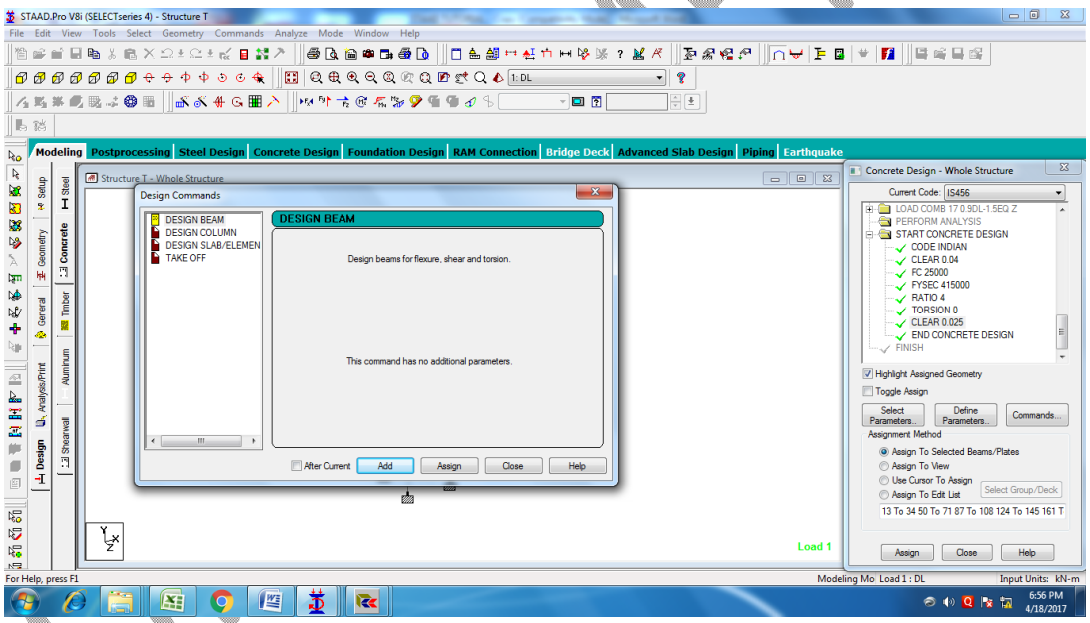

Fig: 27 Commands for distinguishing beam and column design methods

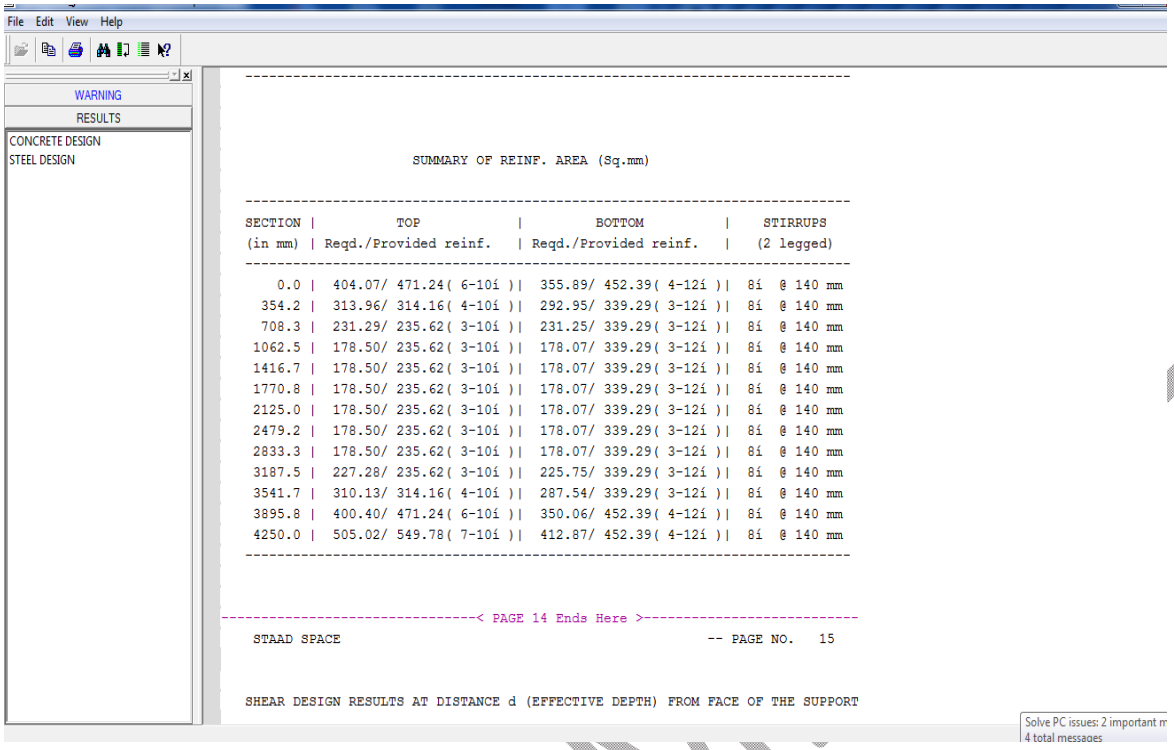

Fig: 28 Typical STAAD Design Output for the Beam design

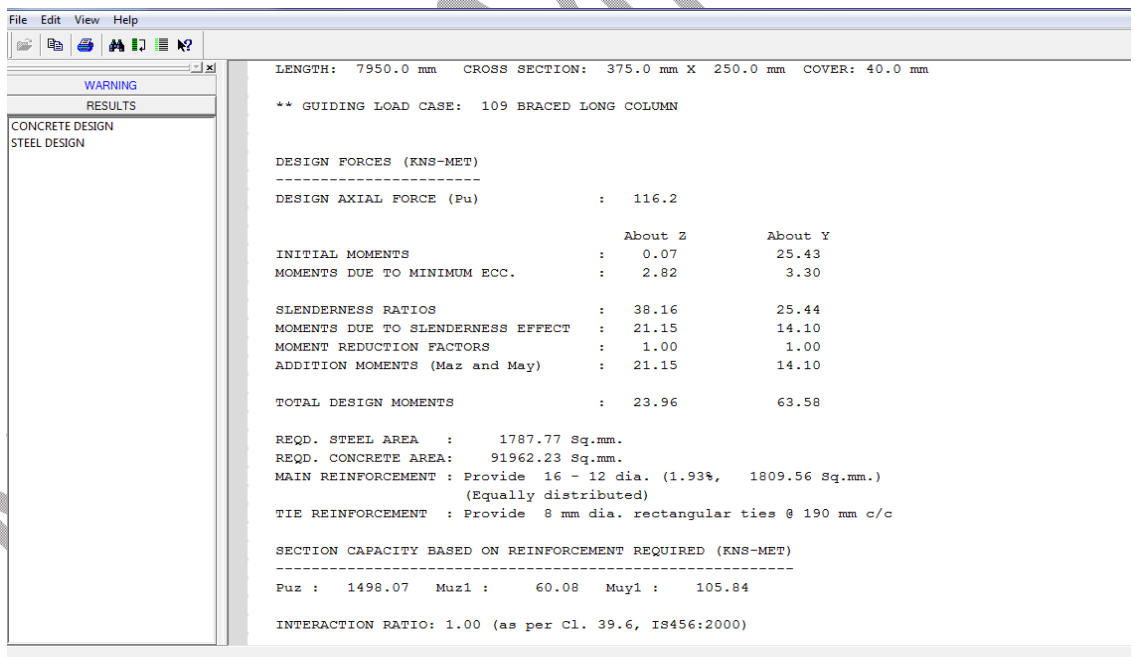

Fig: 29 Typical STAAD Design Output for the column Design

THANK YOU