ANALYSIS AND DESIGN OF ELEVATED TENNIS COURT BY VIRTUE OF VERTICAL SPACE UTILIZATION CONCEPT BY STAAD.Pro

CH. PRASHANTHI1
M.Tech (Scholar), Structural Engineering, Malla Reddy Engineering College (Autonomous)
Secunderabad, Telangana, India

B. VAMSI KRISHNA2
Assistant Professor, Structural Engineering, Malla Reddy Engineering College (Autonomous)
Secunderabad, Telangana, India

Abstract: The principle objective of this project is to analyse and design a Tennis Court using STAAD Pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The design methods used in STAAD-Pro analysis are Limit State Design conforming to Indian Standard Code of Practice. STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice. Initially we started with the analysis of simple 2 dimensional frames and manually checked the accuracy of the software with our results. The results proved to be very accurate. We analysed and designed Tennis Court [2-D Frame] initially for all possible load combinations [dead, live, wind and seismic loads]. STAAD.Pro has a very interactive user interface which allows the users to draw the frame and input the load values and dimensions. Then according to the specified criteria assigned it analyses the structure and designs the members with reinforcement details for RCC frames.

We continued with our work with some more multi-storeyed 2-D and 3-D frames under various load combinations. Our final work was the proper analysis and design of a tennis Court 3-D RCC frame under various load combinations. We considered a 3-D RCC frame with the dimensions of 28 m dia court. The y-axis consisted of 50 m cylindrical column. The court is at height of 15m from the top. The structure was subjected to self weight, dead load, live load, wind load and seismic loads under the load case details of STAAD.Pro. The wind load values were generated by STAAD.Pro considering the given wind intensities at different heights and strictly abiding by the specifications of IS 875. Seismic load calculations were done following IS 1893-2000. The materials were specified and cross-sections of the beam and column members were assigned. The supports at the base of the structure were also specified as fixed. The codes of practise to be followed were also specified for design purpose with other important details. Then STAAD.Pro was used to analyse the structure and design the members. In the post-processing mode, after completion of the design, we can work on the structure and study the bending moment and shear force values with the generated diagrams. We may also check the deflection of various members under the given loading combinations.. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will ensure the structural safety of the buildings which are being designed. Structure and structural elements were normally designed by Limit State Method.

Complicated and high-rise structures need very time taking and cumbersome calculations using conventional manual methods. STAAD.Pro provides us a fast, efficient, easy to use and accurate platform for analysing and designing structures.

Keywords: STAAD.Pro, Tennis Court, Elevated Structure, Bending, Shear, Compression, Tension, High rise Structure, Wind loads.

1. INTRODUCTION

Our project involves analysis and design of Tennis Court using very popular designing software STAAD Pro. We have chosen STAAD Pro because of its following advantages:

ISSN 2229-5518

- Easy to use interface
- conformation with the Indian Standard Codes,
- versatile nature of solving any type of problem
- Accuracy of the solution

STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is the professional's choice for steel, concrete, timber, aluminium and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

2.STAAD.PRO CONSISTS OF THE FOLLOWING

The STAAD.Pro Graphical User Interface: It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

The STAAD analysis and design engine: It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, Timber and Aluminium design.

To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. In the initial phase of our project we have done calculations regarding loadings on buildings and also considered seismic and wind loads.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behaviour of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.
To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural elements shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to design for durability. Design, including design for durability, construction and use in service should be considered as a whole. The realization of design objectives requires compliance with clearly defined standards for materials, production, workmanship and also maintenance and use of structure in service.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

3. OBJECTIVES

The principle objective of this project is to analyse and design a Tennis Court using STAAD Pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The design methods used in STAAD-Pro analysis are Limit State Design conforming to Indian Standard Code of Practice. STAAD.Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD.Pro is

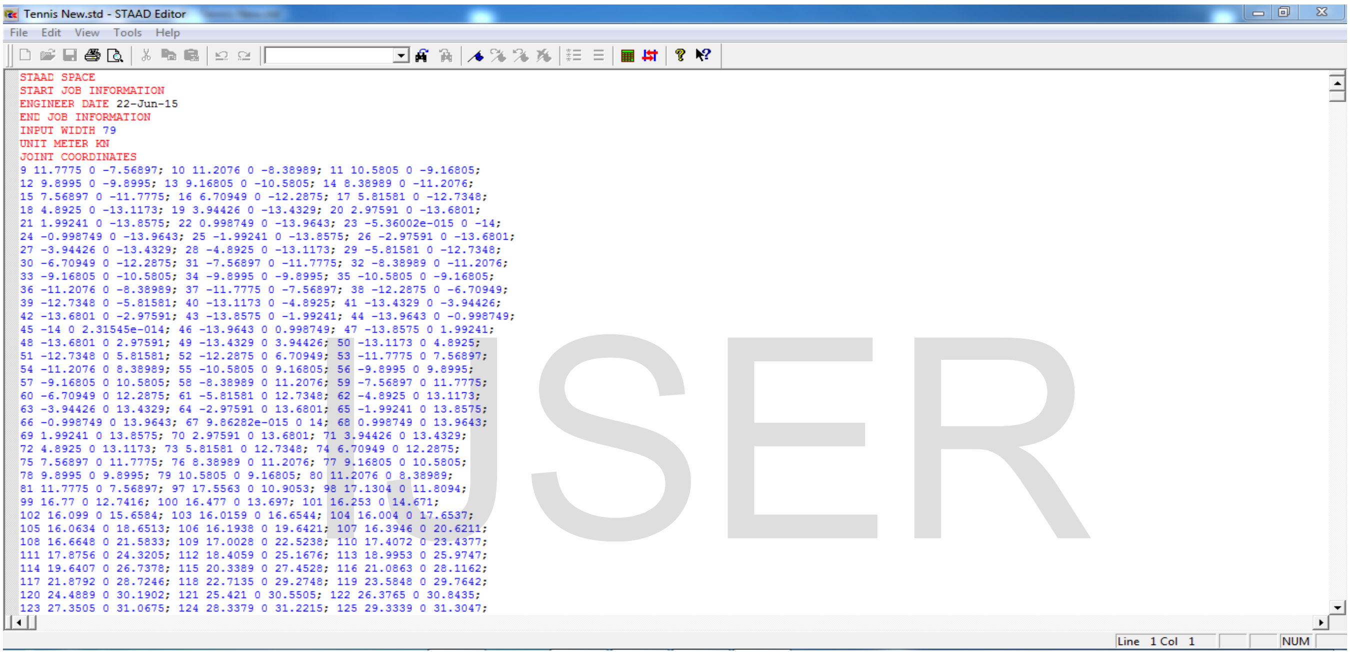
ISSN 2229-5518

the professional's choice. Initially we started with the analysis of simple 2 dimensional frames and manually checked the accuracy of the software with our results. We analysed and designed Tennis Court [2-D Frame] initially for all possible load combinations [dead, live, wind and seismic loads]. STAAD.Pro has a very interactive user interface which allows the users to draw the frame and input the load values and dimensions.

4. WORKING WITH STAAD.Pro

The GUI (or user) communicates with the STAAD analysis engine through the STD input file. That input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The STAAD input file can be created through a text editor or the GUI Modeling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modeling facility creates the input file through an interactive menu-driven graphics oriented procedure.

4.1 INPUT GENERATION




```
STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 22-Jun-15
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
9 11.7775 0 -7.56897; 10 11.2076 0 -8.38989; 11 10.5805 0 -9.16805;
12 9.8995 0 -9.8995; 13 9.16805 0 -10.5805; 14 8.38989 0 -11.2076;
15 7.56897 0 -11.7775; 16 6.70949 0 -12.2875; 17 5.81581 0 -12.7348;
18 4.8925 0 -13.1173; 19 3.94426 0 -13.4329; 20 2.97591 0 -13.6801;
21 1.99241 0 -13.8575; 22 0.998749 0 -13.9643; 23 -5.36002e-015 0 -14;
24 -0.998749 0 -13.9643; 25 -1.99241 0 -13.8575; 26 -2.97591 0 -13.6801;
27 -3.94426 0 -13.4329; 28 -4.8925 0 -13.1173; 29 -5.81581 0 -12.7348;
30 -6.70949 0 -12.2875; 31 -7.56897 0 -11.7775; 32 -8.38989 0 -11.2076;
33 -9.16805 0 -10.5805; 34 -9.8995 0 -9.8995; 35 -10.5805 0 -9.16805;
36 -11.2076 0 -8.38989; 37 -11.7775 0 -7.56897; 38 -12.2875 0 -6.70949;
39 -12.7348 0 -5.81581; 40 -13.1173 0 -4.8925; 41 -13.4329 0 -3.94426;
42 -13.6801 0 -2.97591; 43 -13.8575 0 -1.99241; 44 -13.9643 0 -0.998749;
45 -14 0 2.31545e-014; 46 -13.9643 0 0.998749; 47 -13.8575 0 1.99241;
48 -13.6801 0 2.97591; 49 -13.4329 0 3.94426; 50 -13.1173 0 4.8925;
51 -12.7348 0 5.81581; 52 -12.2875 0 6.70949; 53 -11.7775 0 7.56897;
54 -11.2076 0 8.38989; 55 -10.5805 0 9.16805; 56 -9.8995 0 9.8995;
57 -9.16805 0 10.5805; 58 -8.38989 0 11.2076; 59 -7.56897 0 11.7775;
60 -6.70949 0 12.2875; 61 -5.81581 0 12.7348; 62 -4.8925 0 13.1173;
63 -3.94426 0 13.4329; 64 -2.97591 0 13.6801; 65 -1.99241 0 13.8575;
66 -0.998749 0 13.9643; 67 9.86282e-015 0 14; 68 0.998749 0 13.9643;
69 1.99241 0 13.8575; 70 2.97591 0 13.6801; 71 3.94426 0 13.4329;
72 4.8925 0 13.1173; 73 5.81581 0 12.7348; 74 6.70949 0 12.2875;
75 7.56897 0 11.7775; 76 8.38989 0 11.2076; 77 9.16805 0 10.5805;
78 9.8995 0 9.8995; 79 10.5805 0 9.16805; 80 11.2076 0 8.38989;
81 11.7775 0 7.56897; 97 17.5563 0 10.9053; 98 17.1304 0 11.8094;
99 16.77 0 12.7416; 100 16.477 0 13.697; 101 16.253 0 14.671;
102 16.099 0 15.6584; 103 16.0159 0 16.6544; 104 16.004 0 17.6537;
105 16.0634 0 18.6513; 106 16.1938 0 19.6421; 107 16.3946 0 20.6211;
108 16.6648 0 21.5833; 109 17.0028 0 22.5238; 110 17.4072 0 23.4377;
111 17.8756 0 24.3205; 112 18.4059 0 25.1676; 113 18.9953 0 25.9747;
114 19.6407 0 26.7378; 115 20.3389 0 27.4528; 116 21.0863 0 28.1162;
117 21.8792 0 28.7246; 118 22.7135 0 29.2748; 119 23.5848 0 29.7642;
120 24.4889 0 30.1902; 121 25.421 0 30.5505; 122 26.3765 0 30.8435;
123 27.3505 0 31.0675; 124 28.3379 0 31.2215; 125 29.3339 0 31.3047;
```

Fig 4.1: STAAD input file

4.2 TYPES OF STRUCTURES

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of frame, plate/shell and solid elements. Almost any type of structure can be analyzed by STAAD. A SPACE structure, which is a three dimensional framed structure with loads applied in any plane, is the most general. A PLANE structure is bound by a global X-Y coordinate system with loads in the same plane. A TRUSS structure consists of truss members which can have only axial member forces and no bending in the members. A FLOOR structure is a two or three dimensional structure having no horizontal (global X or Z) movement of the structure [FX, FZ & MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a FLOOR structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure.

4.3 GENERATION OF THE STRUCTURE

The structure may be generated from the input file

or mentioning the co-ordinates in the GUI. The figure below shows the GUI generation method.

ISSN 2229-5518

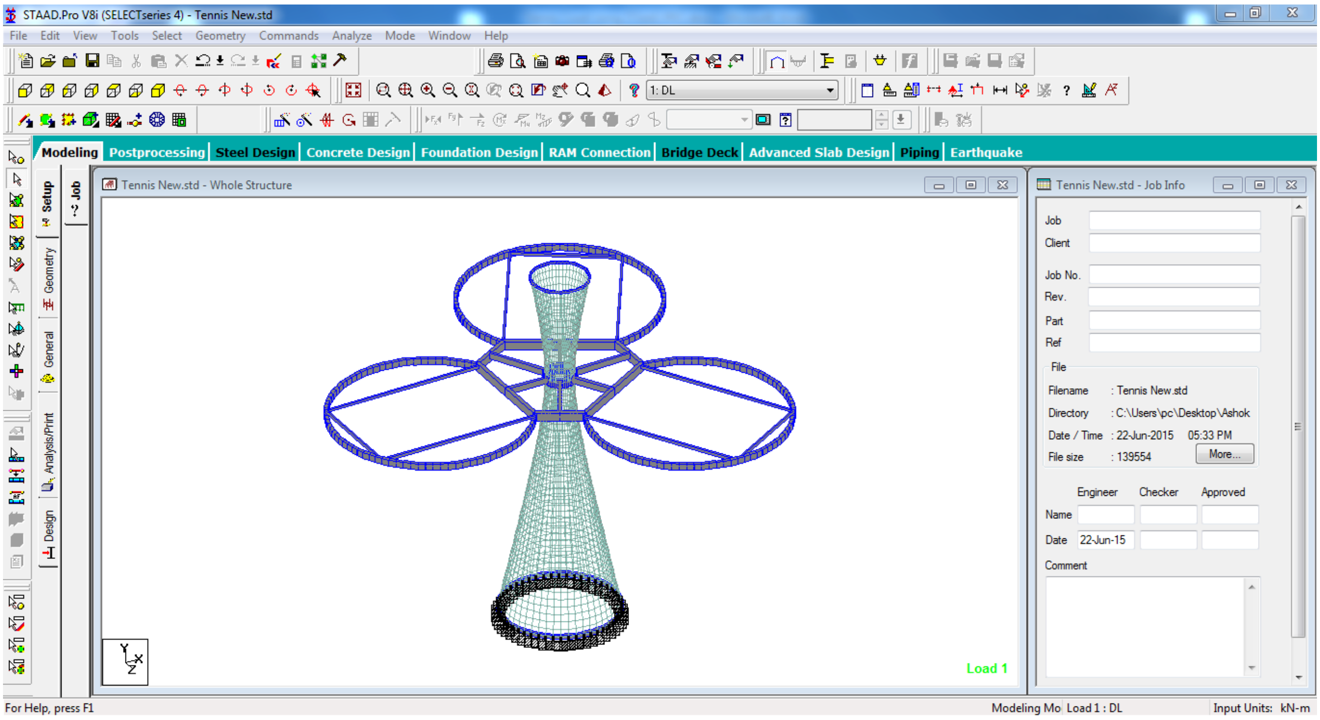


Fig 4.2: Generation of structure through GUI

4.4 MATERIAL CONSTANTS

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and beta angle (BETA) or coordinates for any reference (REF) point. E value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when self weight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula,

$$G = 0.5 \times E / (1 + POISS)$$

If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and ALPHA has to be the same.

4.5 SUPPORTS

Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FIXED BUT). A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement. Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction

4.6 LOADS

Loads in a structure can be specified as joint load, member load, temperature load and fixed-end member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self weight can also be applied in any desired direction.

4.7 SECTION TYPES FOR CONCRETE DESIGN

The following types of cross sections for concrete members can be designed.

- For Beams Prismatic (Rectangular & Square) & T-shape
- For Columns Prismatic (Rectangular, Square and Circular)

4.8 DESIGN PARAMETERS

The program contains a number of parameters that are needed to perform design as per IS 13920. It accepts all parameters that are needed to perform design as per IS: 456. Over and above it has some other parameters that are required only when designed is performed as per IS: 13920. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed by this manual contains a complete list of the available parameters and their default values. It is necessary to declare length and force units as Millimeter and Newton before performing the concrete design.

4.8.1 BEAM DESIGN

Beams are designed for flexure, shear and torsion. If required the effect of the axial force may be taken into consideration. For all these forces, all active beam loadings are prescanned to identify the critical load cases at different sections of the beams. For design to be performed as per IS: 13920 the width of the member shall not be less than 200mm. Also the member shall preferably have a width-to depth ratio of more than 0.3

4.8.1.1 DESIGN FOR FLEXURE

Design procedure is same as that for IS 456. However while designing following criteria are satisfied as per IS-13920:

1. The minimum grade of concrete shall preferably be M20.
2. Steel reinforcements of grade Fe415 or less only shall be used.
3. The minimum tension steel ratio on any face, at any section, is given by:

$$\rho min = 0.24\sqrt{fck}/fy$$

   The maximum steel ratio on any face, at any section, is given by $\rho max = 0.025$

4. The positive steel ratio at a joint face must be at least equal to half the negative steel at that face.
5. The steel provided at each of the top and bottom face, at any section, shall at least be equal to one-fourth of the maximum negative moment steel provided at the face of either joint.

4.8.1.2 DESIGN FOR SHEAR

The shear force to be resisted by vertical hoops is guided by the IS 13920:1993 revision. Elastic sagging and hogging moments of resistance of the beam section at ends are considered while calculating shear force. Plastic sagging and hogging moments of resistance can also be considered for shear design if PLASTIC parameter is mentioned in the input file. Shear reinforcement is calculated to resist both shear forces and torsional moments.

4.8.2 COLUMN DESIGN

Columns are designed for axial forces and biaxial moments per IS 456:2000. Columns are also designed for shear forces. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS: 456 have been taken care of in the column design of STAAD. However following clauses have been satisfied to incorporate provisions of IS 13920:

1. The minimum grade of concrete shall preferably be M20
2. Steel reinforcements of grade Fe415 or less only shall be used.
3. The minimum dimension of column member shall not be less than 200 mm. For columns having unsupported length exceeding 4m, the shortest dimension of column shall not be less than 300 mm.
4. The ratio of the shortest cross-sectional dimension to the perpendicular dimension

shall preferably be not less than 0.

5. The spacing of hoops shall not exceed half the least lateral dimension of the column, except where special confining reinforcement is provided.
6. Special confining reinforcement shall be provided over a length lo from each joint face, towards mid span, and on either side of any section, where flexural yielding may occur. The length lo shall not be less than a) larger lateral dimension of the member at the section where yielding occurs, b) 1/6 of clear span of the member, and c) 450 mm.
7. The spacing of hoops used as special confining reinforcement shall not exceed ¼ of minimum member dimension but need not be less than 75 mm nor more than 100 mm

5. ANALYSIS OF RCC FRAMED TENNIS COURT USING STAAD.Pro

Beam Dimensions:

- Main Beams = 1.2 x 0.6 m
- Court Beams = 0.9 x 0.45 m
- Cross Beams for Court = 0.45 x 0.3 m
- Top Beam = 0.3 x 0.23 m
- Plinth Beam = 0.6 x 0.45 m
- All slabs = 0.20 m thick
- Thickness of Wall = 0.35 m thick avg.
- Parapet = 0.3 m thick RCC

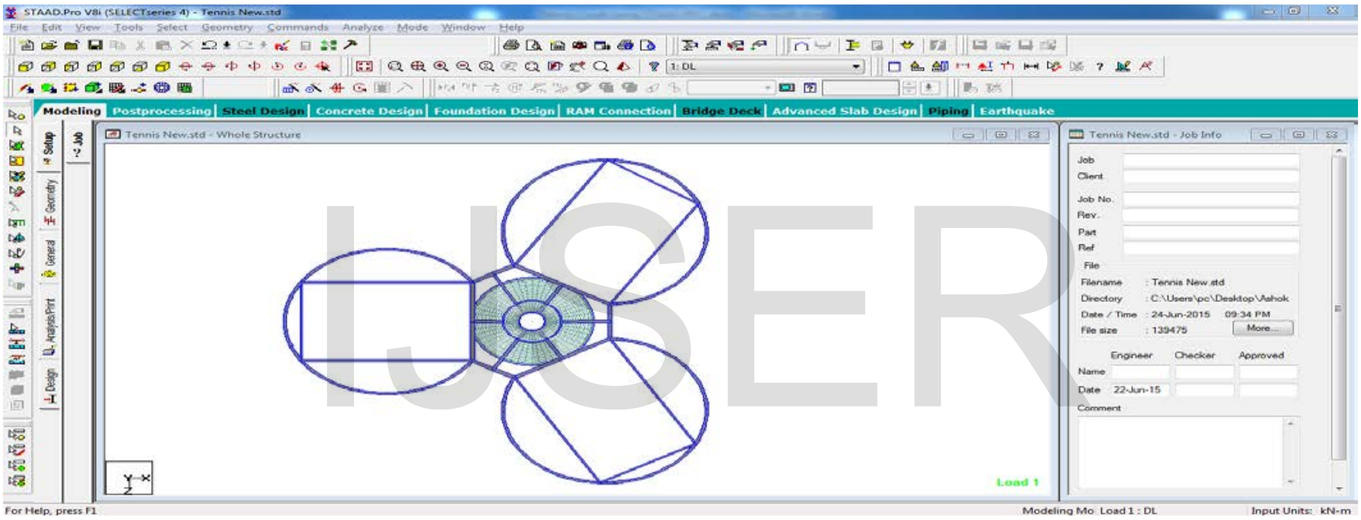
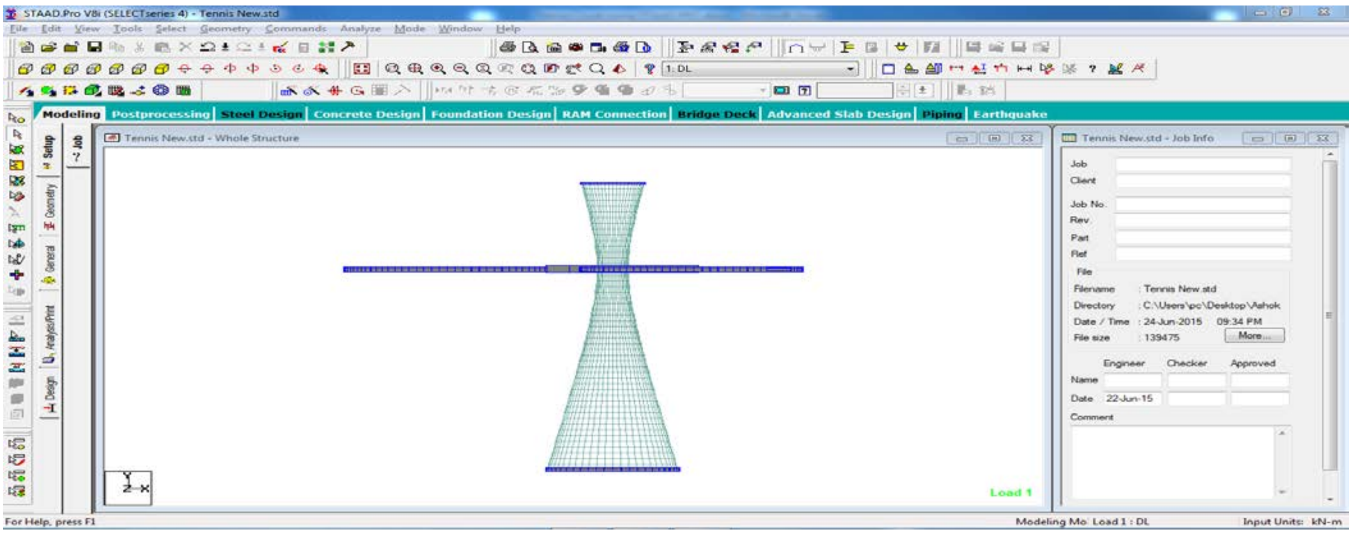


Fig 5.1, 5.2: Plan and Elevation of the Tennis Court



5.1 PHYSICAL PARAMETERS OF BUILDING

- Diameter of tennis court = 28 m
- Height = 50 m

ISSN 2229-5518

(1.0m parapet being non- structural for seismic purposes, is not considered of building frame height)

- Live load on the floors is 4 kN/m^2
- Live load on the roof is 1.5 kN/m^2
- Grade of concrete and steel used:
- Used M35 concrete and Fe 500 steel

5.2 GENERATION OF MEMBER PROPERTY

Generation of member property can be done in **STAAD.Pro** by using the window as shown above.

The member section is selected and the dimensions have been specified. The beams are having a dimension of 1.2 * 0.6 m and the walls are having a dimension of 0.4 m thick at below the court floor and at the other top floors they are having a dimension of 0.3 m thickness.

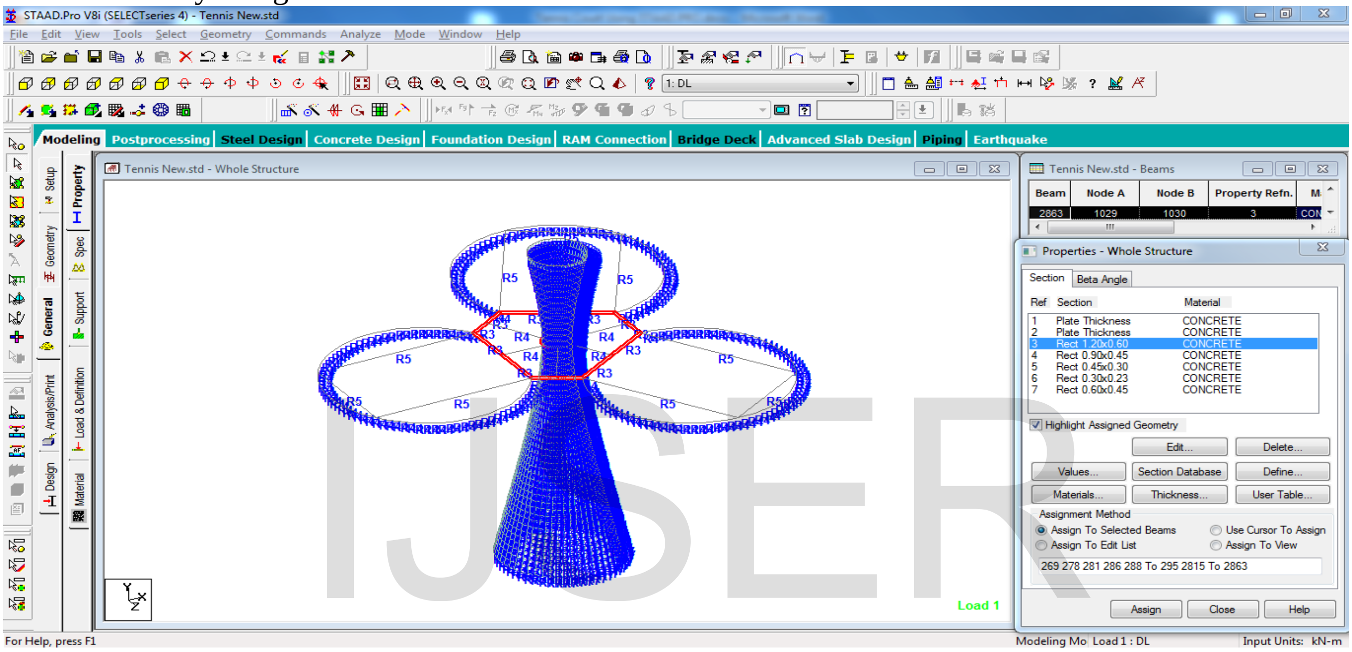
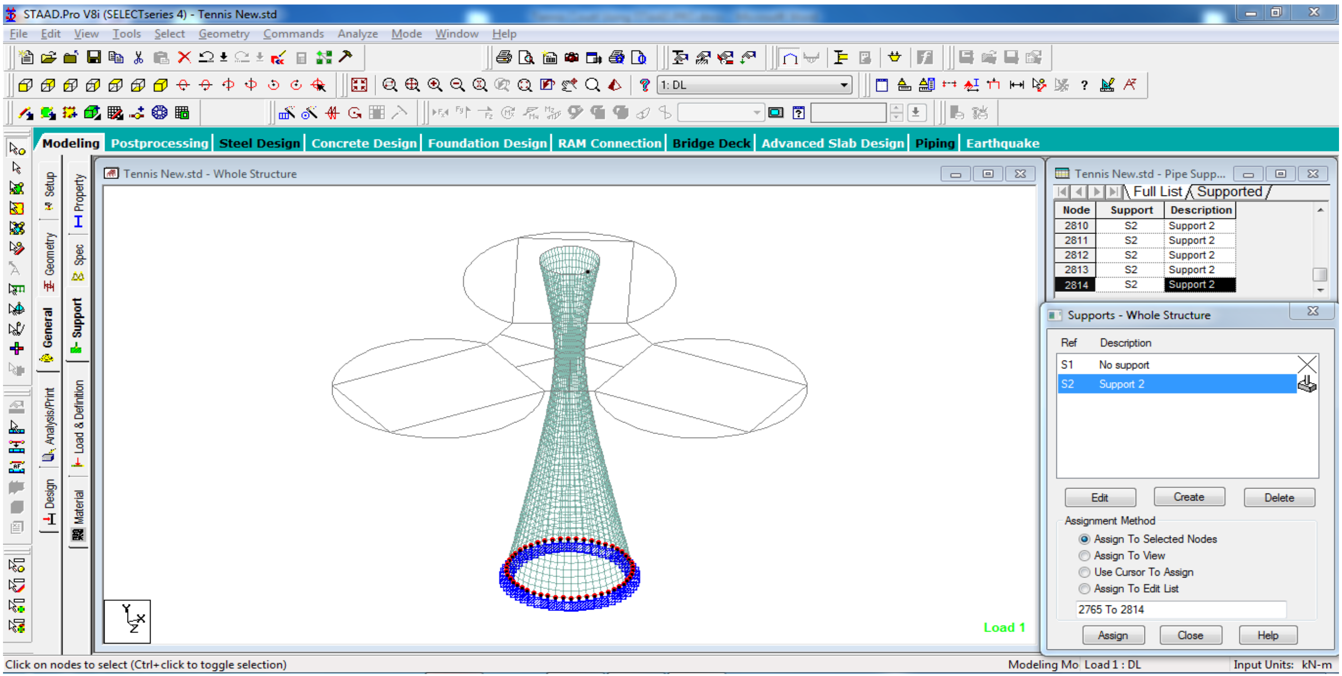


Fig 5.3, 5.4 Generation of member property and fixing supports of the structure



ISSN 2229-5518

5.3 LOADING

The loadings were calculated partially manually and rest was generated using STAAD.Pro load generator. The loading cases were categorized as:

- Self – Weight
- Dead Load from Slab
- Live Load
- Wind Load
- Seismic Load
- Combinations

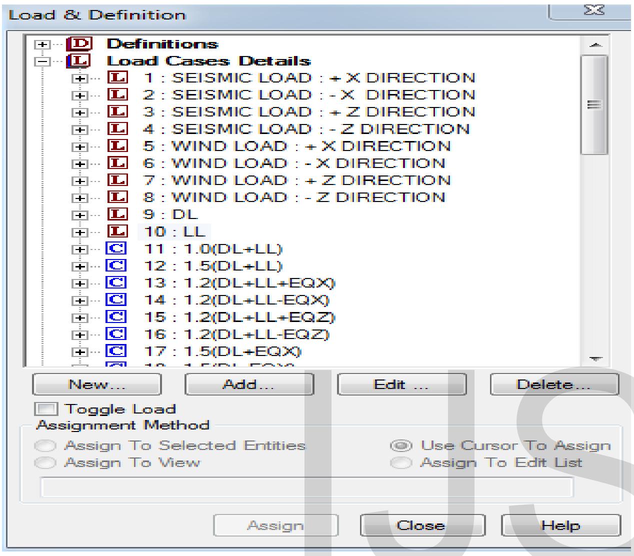


Fig 5.5 primary load cases

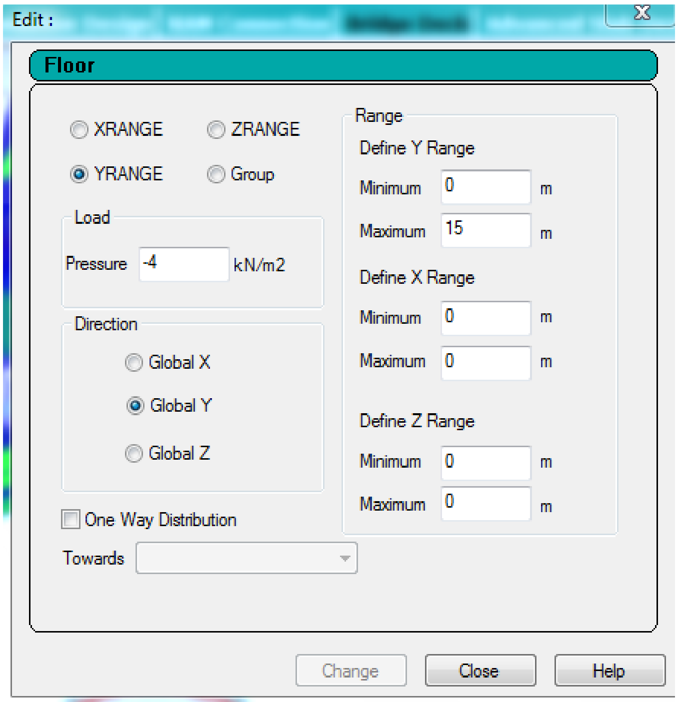


Fig 5.6 Input window of floor load generator

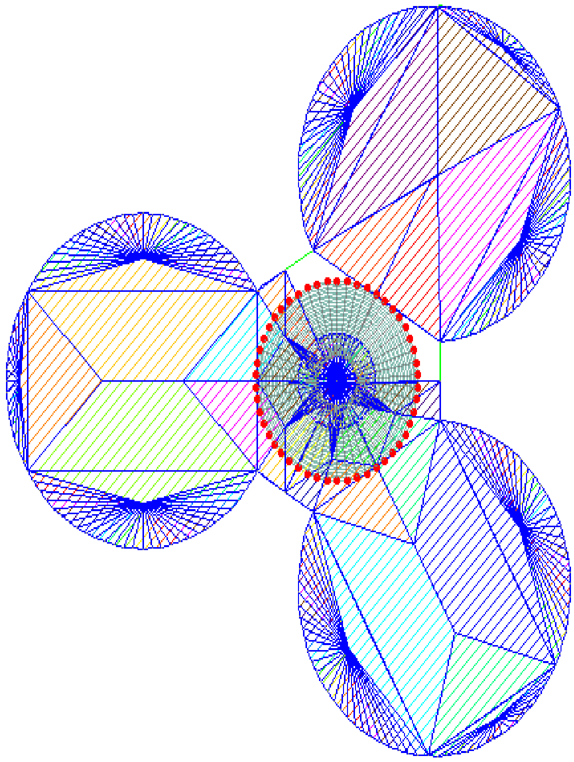


Fig 5.7 load distribution by Triangle & Trapezoidal method

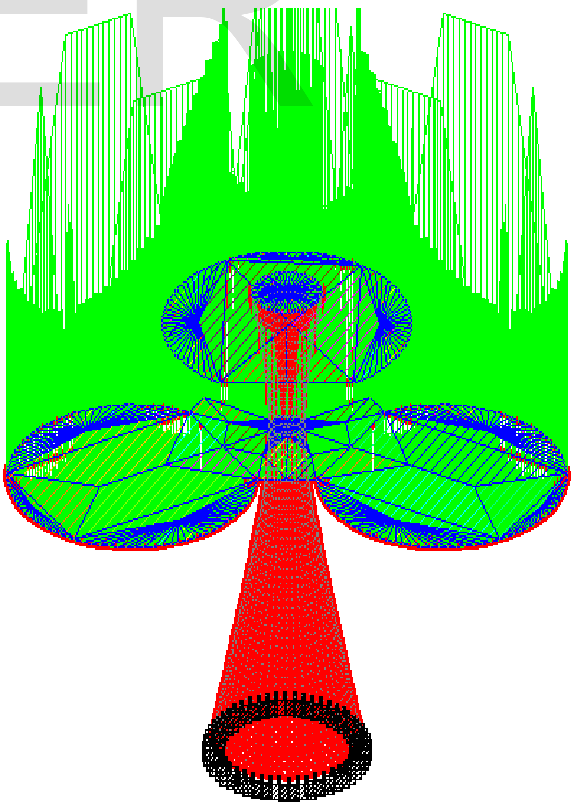


Fig 5.8 The structure under DL from slab

ISSN 2229-5518

5.3.1 LIVE LOAD

The live load considered in each floor was 4 KN/sq m and for the terrace level it was considered to be 1.5 KN/sq m. The live loads were generated in a similar manner as done in the earlier case for dead load in each floor. This may be done from the member load button from the load case column.

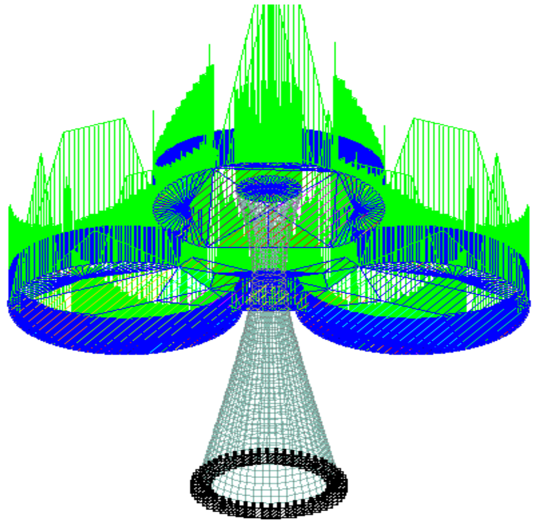


Fig 5.9 The structure under live Load

5.3.1 WIND LOAD

The wind load values were generated by the software itself in accordance with IS 875. Under the define load command section, in the wind load category, the definition of wind load was supplied. The wind intensities at various heights were calculated manually and feed to the software. Based on those values it generates the wind load at different floors.

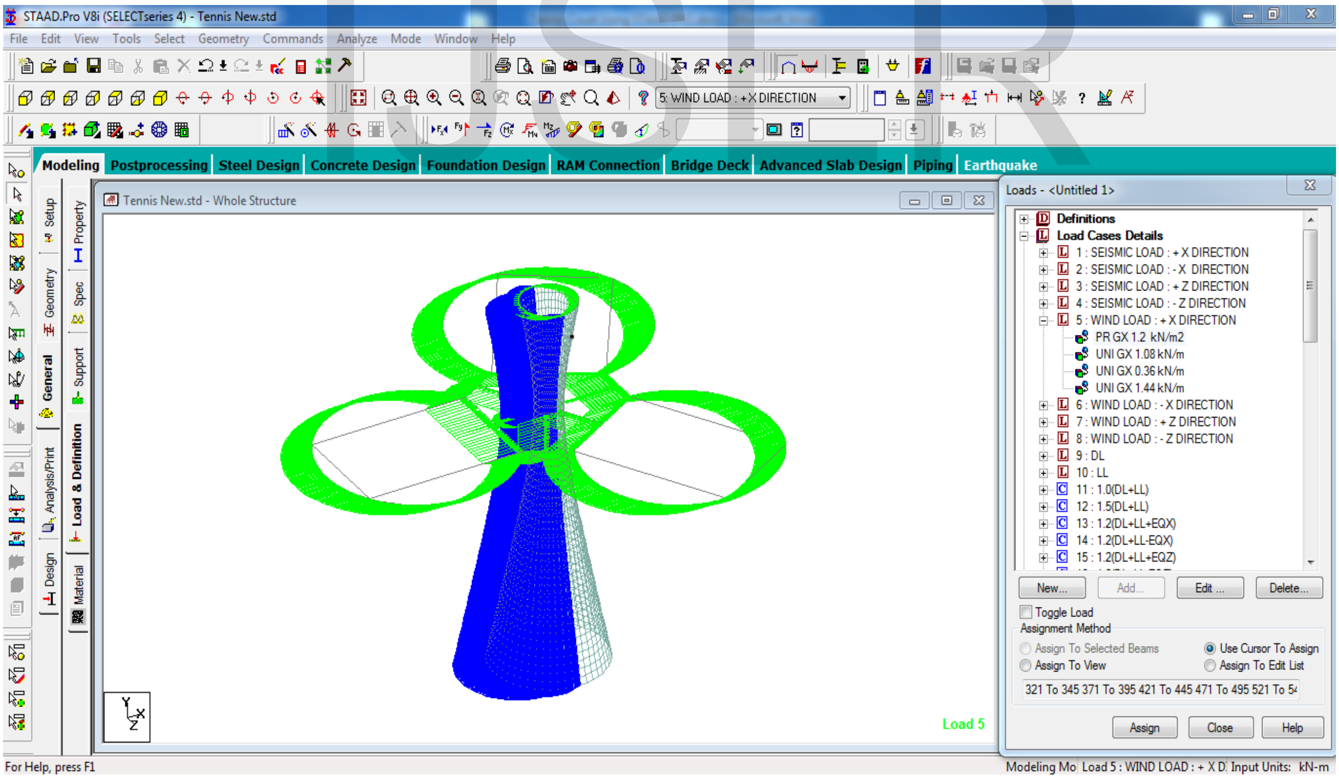


Fig 5.10 wind load effect on structure elevation and plan

6.0 ANALYSIS AND DESIGN RESULTS

Some of the sample analysis and design results have been shown below for beam number 35

SUMMARY OF REINF. AREA (Sq.mm)

SECTION	0.0 mm	249.8 mm	499.7 mm	749.5 mm	999.4 mm
TOP REINF.	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)
BOTTOM REINF.	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)	5010.75 (Sq. mm)

SUMMARY OF PROVIDED REINF. AREA

SECTION	0.0 mm	249.8 mm	499.7 mm	749.5 mm	999.4 mm
TOP REINF.	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)
BOTTOM REINF.	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)	16-20í 1 layer(s)
SHEAR REINF.	2 legged 8í @ 3 mm c/c	2 legged 8í @ 3 mm c/c	2 legged 8í @ 3 mm c/c	2 legged 8í @ 3 mm c/c	2 legged 8í @ 3 mm c/c

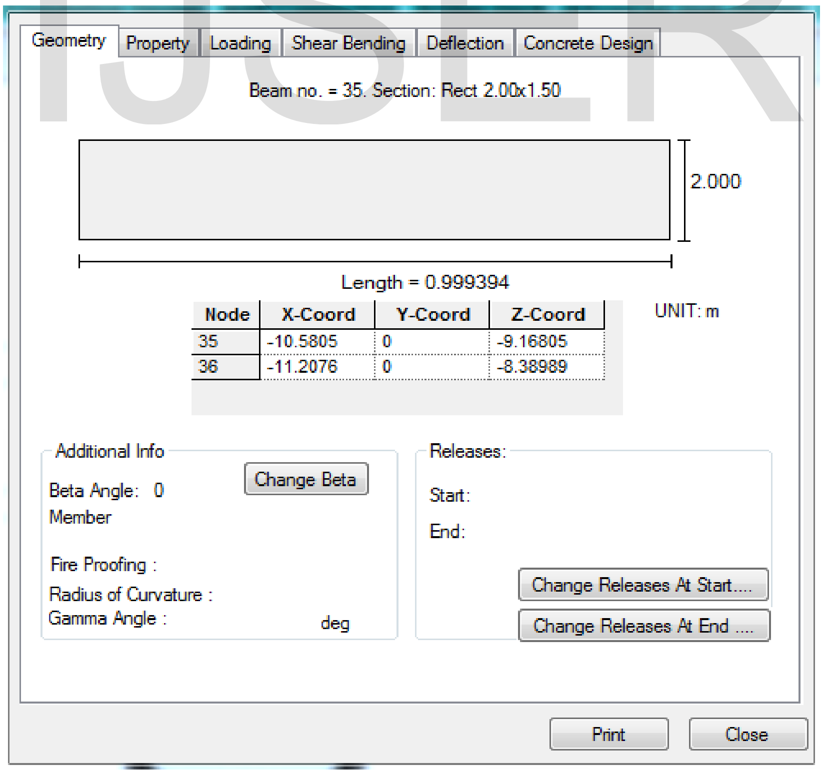


Fig 6.1 Geometry of beam

ISSN 2229-5518

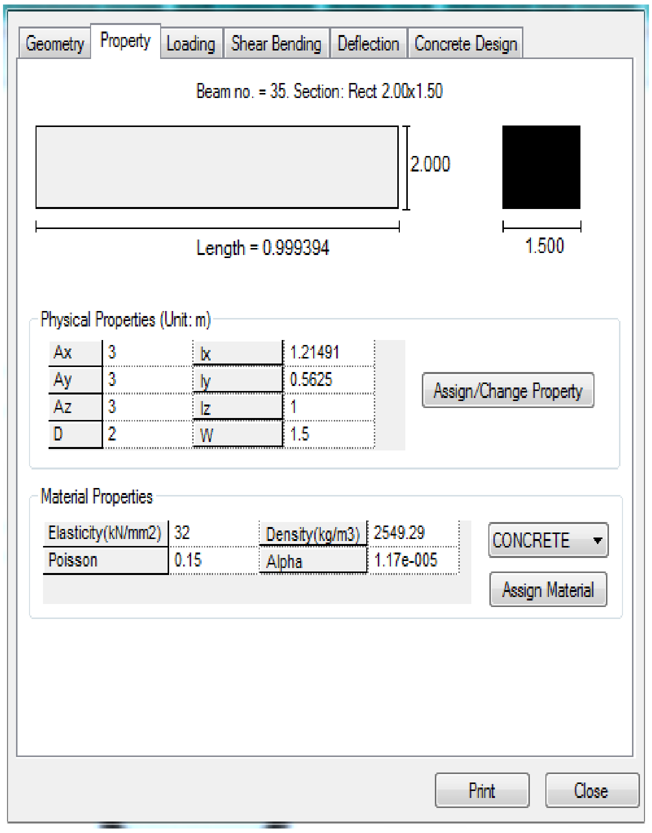


Fig 6.2 Property of beam

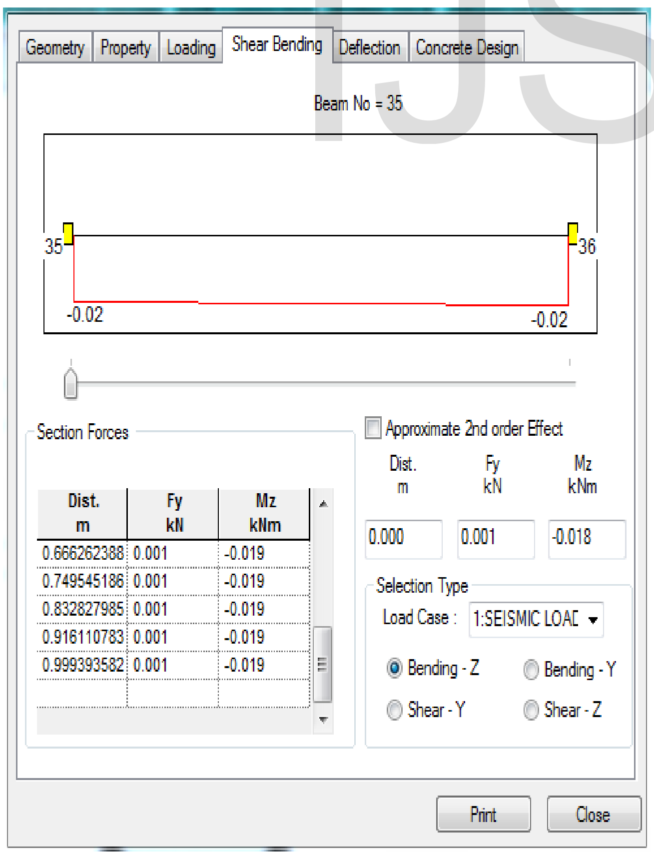


Fig 6.3 Shear bending of beam no. 1

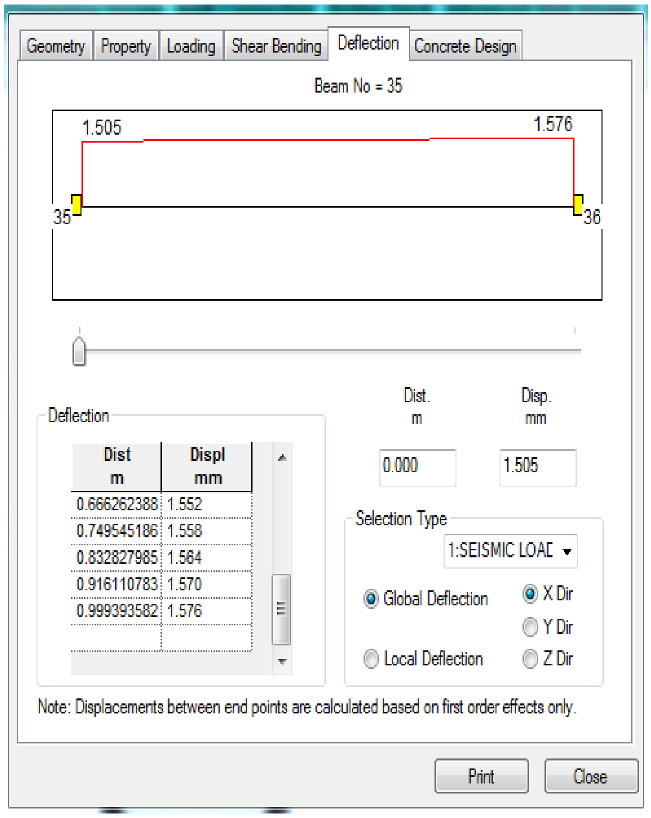


Fig 6.4 Deflection of beam

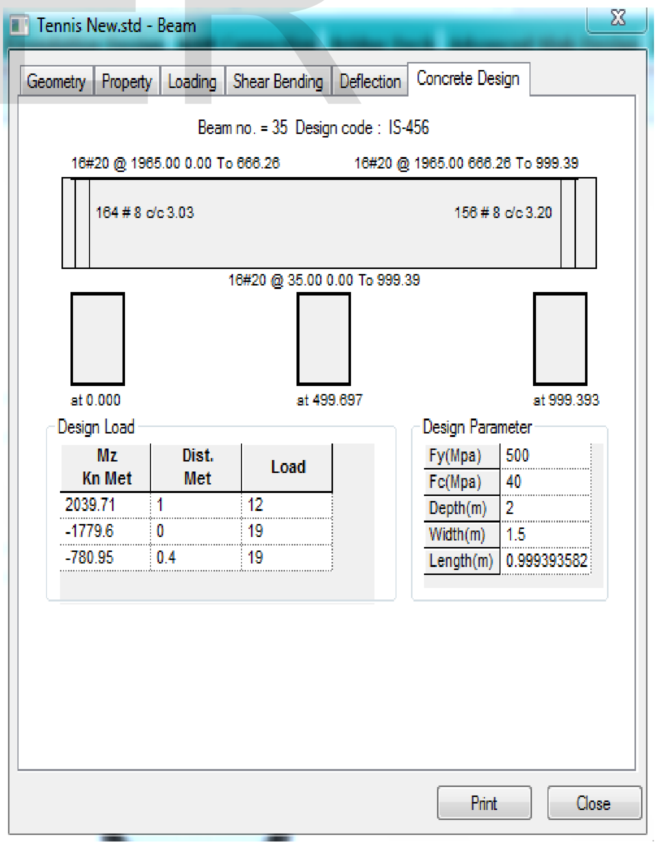


Fig 6.5 Concrete design of beam

7.0 POST PROCESSING MODE

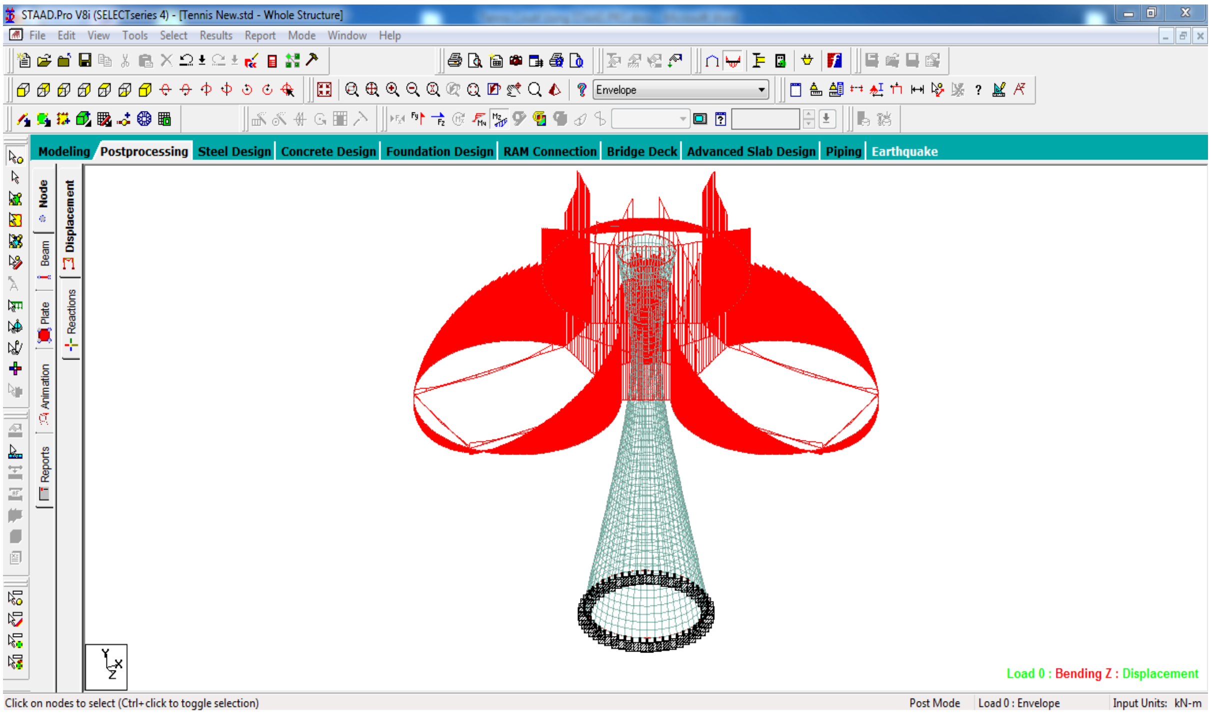


Fig 7.1 Bending in Z

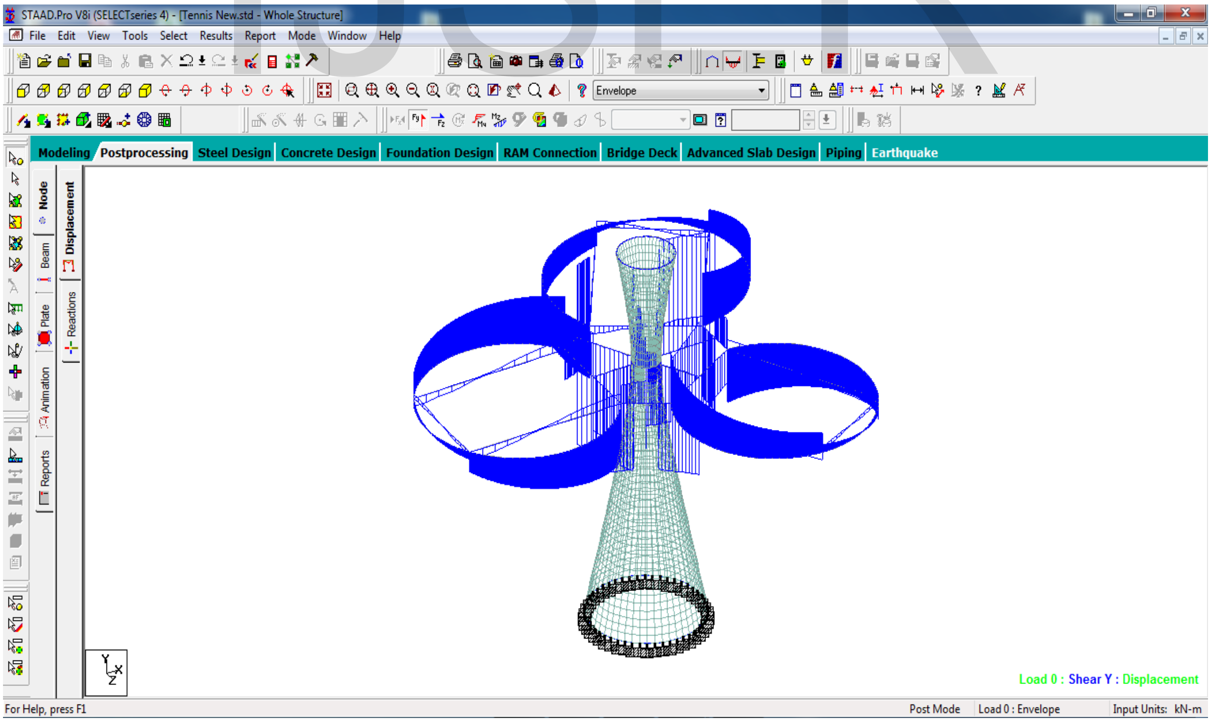


Fig 7.2 Shear force in Y

ISSN 2229-5518

CONCLUSIONS

STAAD PRO has the capability to calculate the reinforcement needed for any concrete section. The program contains a number of parameters which are designed as per IS: 456(2000). Beams are designed for flexure, shear and torsion

Design for Flexure:

Maximum sagging (creating tensile stress at the bottom face of the beam) and hogging (creating tensile stress at the top face) moments are calculated for all active load cases at each of the above mentioned sections. Each of these sections are designed to resist both of these critical sagging and hogging moments. Where ever the rectangular section is inadequate as singly reinforced section, doubly reinforced section is tried.

Design for Shear:

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear capacity calculation at different sections without the shear reinforcement is based on the actual tensile reinforcement provided by STAAD program. Two-legged stirrups are provided to take care of the balance shear forces acting on these sections

Beam Design Output:

The default design output of the beam contains flexural and shear reinforcement provided along the length of the beam.

Column Design:

Columns are designed for axial forces and biaxial moments at the ends. All active load cases are tested to calculate reinforcement. The loading which yield maximum reinforcement is called the critical load. Column design is done for square section. Square columns are designed with reinforcement distributed on each side equally for the sections under biaxial moments and with reinforcement distributed equally in two faces for sections under uni-axial moment. All major criteria for selecting longitudinal and transverse reinforcement as stipulated by IS: 456 have been taken care of in the column design of STAAD.

REFERENCES

1. Dr. S.R. Karve & Dr. V.L. Shah - "Illustrated design of Reinforced concrete Buildings"
2. N. Krishna Raju - "Advanced Reinforced Concrete design"
3. "STAAD Pro 2004 – Getting started & tutorials" Published by: R .E. I.
4. "STAAD Pro 2004 – Technical reference manual" Published by: R.E.I.
5. IS 875 - Bureau Of Indian Standardsmanak Bhavan, 9 Bahadur Shah Zafar Margnew Delhi 110002.
6. IS 456 - Bureau Of Indian Standardsmanak Bhavan, 9 Bahadur Shah Zafar Margnew Delhi 110002.
7. IS 1893-2000 - Bureau Of Indian Standardsmanak Bhavan, 9 Bahadur Shah Zafar Margnew Delhi 110002.
8. IS 1893-2002 - Bureau Of Indian Standardsmanak Bhavan, 9 Bahadur Shah Zafar Margnew Delhi 110002.