# COMPUTER AIDED ANALYSIS AND DESIGN OF MULTI –STOREYED BUILDINGS

**NAMPALLY MAHESH**
14C71D2007, M.Tech (Civil - Structural Engineering), Ellenki College of Engineering & Technology
Email: nampallymahesh2@gmail.com

**N. MONICA MADHURI**
Head & Associate Professor
Ellenki College of Engineering & Technology

## ABSTRACT

*The principle objective of this project is to analyze and design a multi-storeyed building [G + 21 (3D frame)] 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 analyzed and designed a G + 7 storey building [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-storied 2-D and 3-D frames under various load combinations. Our final work was the proper analysis and design of a G + 21 3-D RCC frame under various load combinations.*

*We considered a 3-D RCC frame with the dimensions of 4 bays @5m in x-axis and 3 bays @5m in z-axis. The y-axis consisted of G + 21 floors. The total numbers of beams in each floor were 28 and the numbers of columns were 16. The ground floor height was 4m and rest of the 21 floors had a height of 3.3m.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 practice to be followed were also specified for design purpose with other important details.*

*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.*

## INTRODUCTION

Our paper involves analysis and design of multi-storeyed [G + 21] using a very popular designing software STAAD Pro.

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, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

***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 Aluminum 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 behavior 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.

**WIND PRESSURES AND FORCES ON BUILDINGS/STRUCTURES:**

The wind load on a building shall be calculated for:

a) The building as a whole,
b) Individual structural elements as roofs and walls, and
c) Individual cladding units including glazing and their fixings.

*Pressure Coefficients* - The pressure coefficients are always given for a particular surface or part of the surface of a building. The wind load acting normal to a surface is obtained by multiplying the area of that surface or its appropriate portion by the pressure coefficient (C,) and the design wind pressure at the height of the surface from the ground. The average values of these pressure coefficients for some building shapes Average values of pressure coefficients are given for critical wind directions in one or more quadrants. In order to determine the maximum wind load on the building, the total load should be calculated for each of the critical directions shown from all quadrants. Where considerable variation of pressure occurs over a surface, it has been subdivided and mean pressure coefficients given for each of its several parts.

Then the wind load, F, acting in a direction normal to the individual structural element or Cladding unit is: **F= (Cpe – Cpi) A Pd**

Where,
Cpe = external pressure coefficient,
Cpi = internal pressure- coefficient,
A = surface area of structural or cladding unit, and

Pd = design wind pressure element

**Analysis-**

Dynamic analysis shall be performed to obtain the design seismic force, and its distribution to different levels along the height of the building and to the various lateral load resisting elements, for the following

Buildings:
a) *Regular buildings* -Those greater than 40 m in height in Zones IV and V and those Greater than 90 m in height in Zones II and 111.
b) *Irregular buildings* – All framed buildings higher than 12m in Zones IV and V and those greater than 40m in height in Zones 11 and III.
The analytical model for dynamic analysis of buildings with unusual configuration should be such that it adequately models the types of irregularities present in the building configuration. Buildings with plan irregularities cannot be modelled for dynamic analysis.

For irregular buildings, lesser than 40 m in height in Zones 11and III, dynamic analysis, even though not mandatory, is recommended. Dynamic analysis may be performed either by the Time History Method or by the Response Spectrum Method. However, in either method, the design base shear (VB) shall be compared with abase shear (VB)

**Time History Method-**

Time history method of analysis shall be based on an appropriate ground motion and shall be performed using accepted principles of dynamics.

**Response Spectrum Method-**

Response spectrum method of analysis shall be performed using the design spectrum specified, or by a site-specific design spectrum mentioned.

## WORKING WITH STAAD. Pro:

### 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 who 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.

### Member load:

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads. Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length.

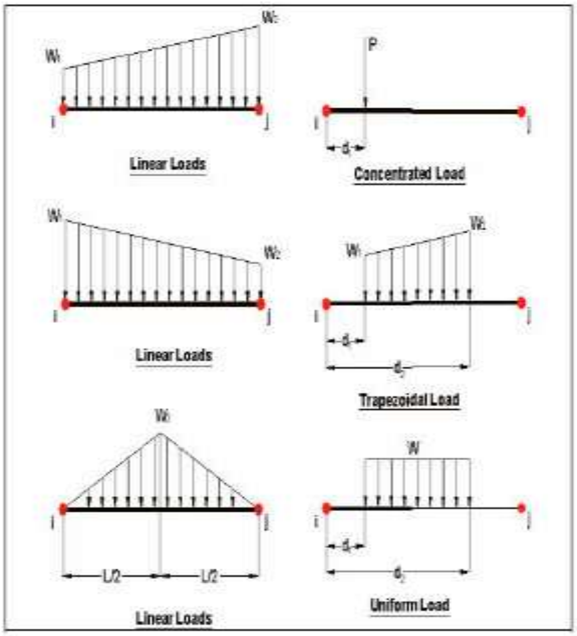



Fig: Member load configuration

### Post Processing Facilities:

All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI.

**Stability Requirements:**

Slenderness ratios are calculated for all members and checked against the appropriate maximum values. IS: 800 summarize the maximum slenderness ratios for different types of members. In STAAD implementation of IS: 800, appropriate maximum slenderness ratio can be provided for each member. If no maximum slenderness ratio is provided, compression members will be checked against a maximum value of 180 and tension members will be checked against a maximum value of 400.

**Deflection Check:**

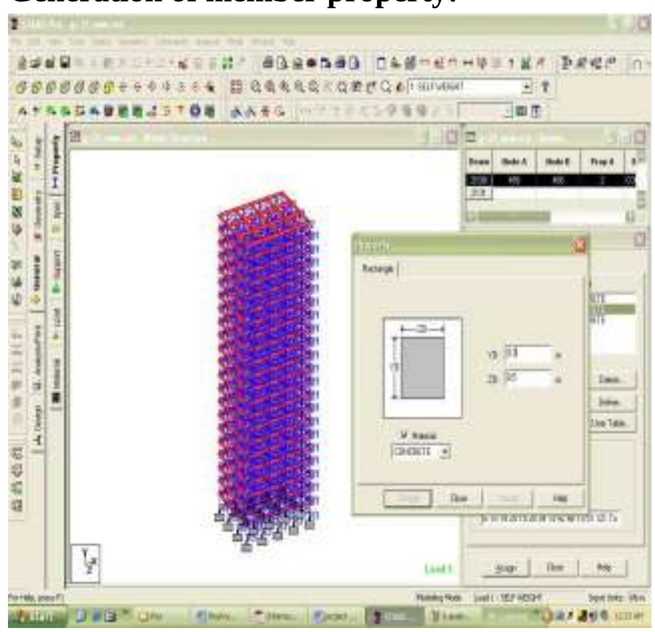
This facility allows the user to consider deflection as criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters. Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

**Code Checking:**

The purpose of code checking is to verify whether the specified section is capable of satisfying applicable design code requirements. The code checking is based on the IS: 800 (1984) requirements. Forces and moments at specified sections of the members are utilized for the code checking calculations. Sections may be specified using the BEAM parameter or the SECTION command. If no sections are specified, the code checking is based on forces and moments at the member ends.

**ANALYSIS OF G + 21 RCC FRAMED BUILDING USING STAAD. Pro**

**Generation of member property:**



**Supports:**

The base supports of the structure were assigned as fixed. The supports were generated using the STAAD. Pro support generator.

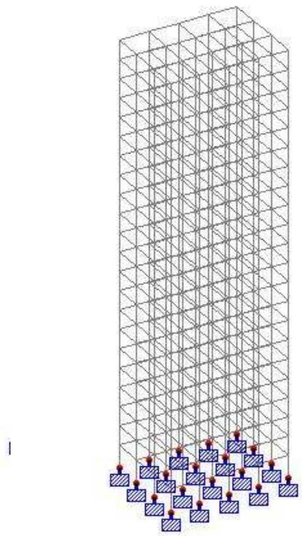


Fig: fixing supports of the structure

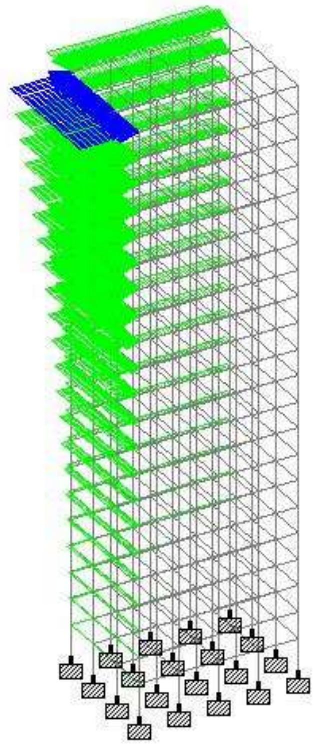


Fig: structure under seismic load 32

**Load combination:**

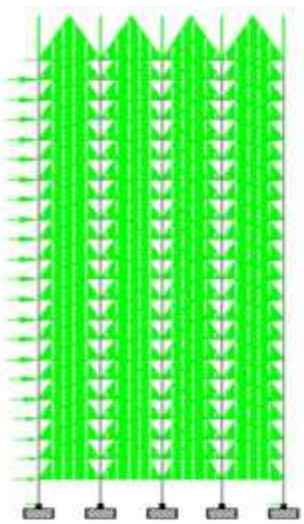


Fig: under combination with wind load

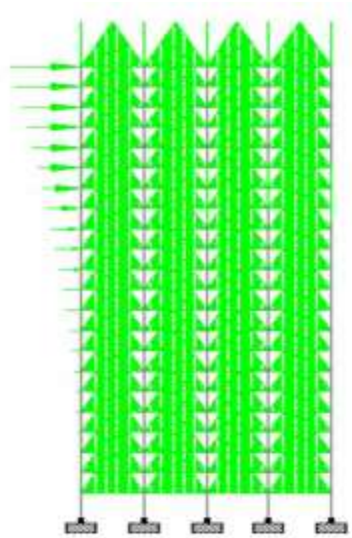
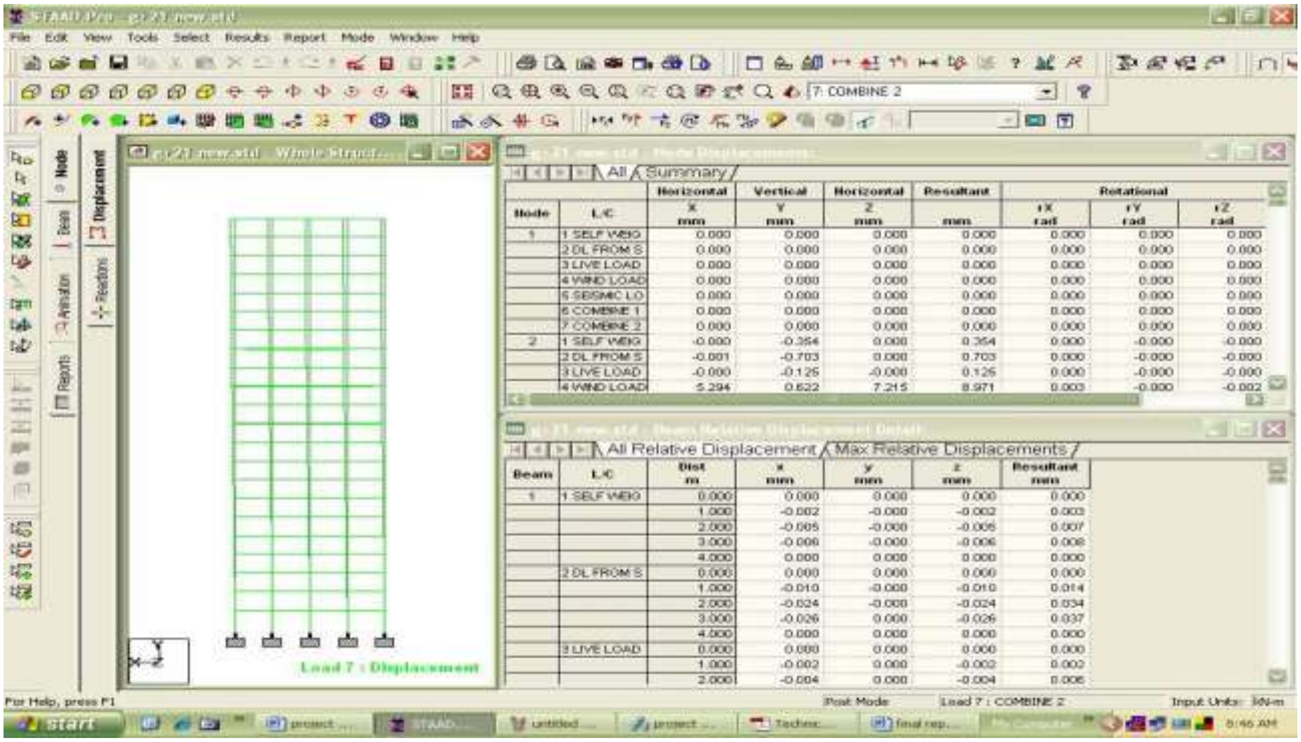


Fig: under combination with seismic load

The structure has been analyzed for load combinations considering all the previous loads in proper ratio. In the first case a combination of self-weight, dead load, live load and wind load was taken in to consideration. In the second combination case instead of wind load seismic load was taken into consideration.

**POST PROCESSING MODE**



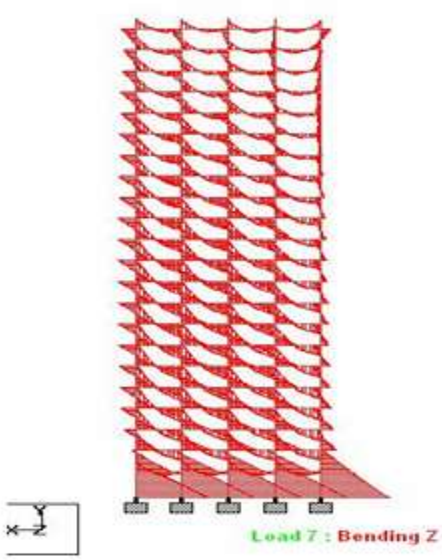
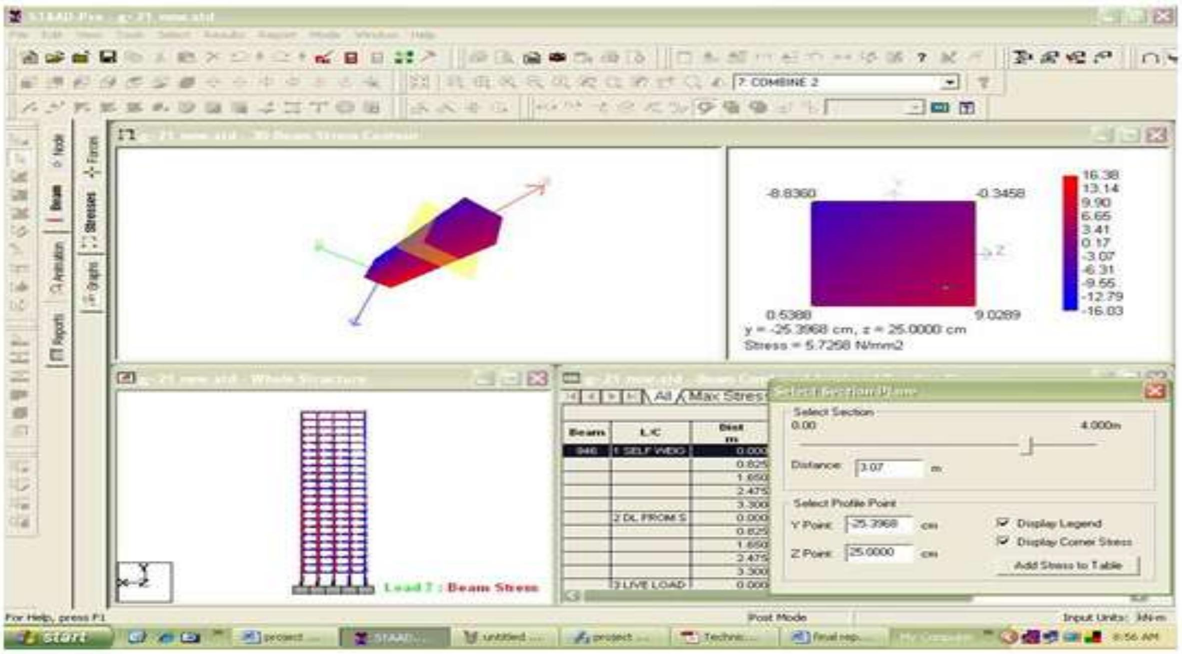




Fig: bending in

The stress at any point of any member can be found out in this mode. The figure below depicts a particular case.



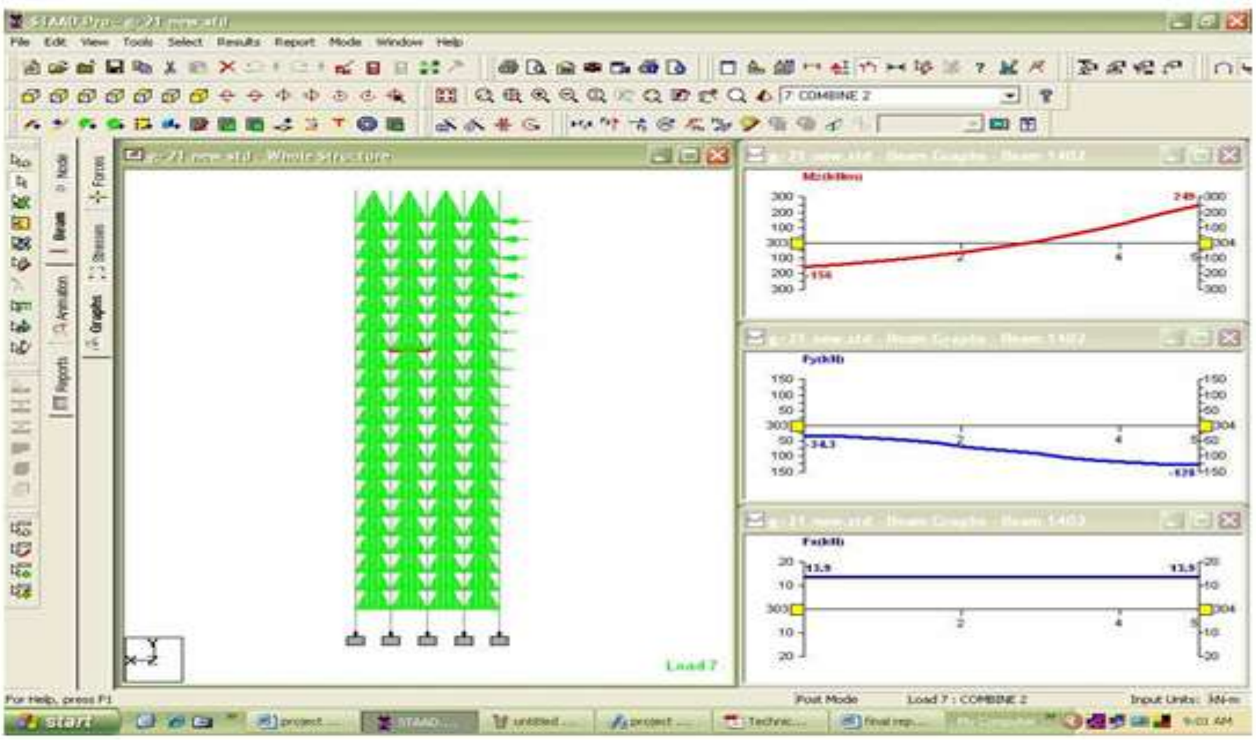




Fig: graph for shear force and bending moment for a beam

The above figure shows that the bending moment and the shear force can be studied from the graphs generated by STAAD. Pro. The whole structure is shown in the screen and we may select any member and at the right side we will get the BMD and SFD for that member. The above figure shows the diagrams for member beam 1402.

## CONCLUSION

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.

## REFERENCE


- Dr. S.R. Karve & Dr. V.L. Shah - "Illustrated design of Reinforced concrete Buildings"
- N. Krishna Raju - "Advanced Reinforced Concrete design"
- "STAAD Pro 2004 – Getting started & tutorials"- Published by: R .E. I.
- "STAAD Pro 2004 – Technical reference manual"- Published by: R.E.I.
- IS 875 - Bureau Of Indian Standards Manak Bhavan, 9 Bahadur Shah Zafar Marg New Delhi 110002
- Is 456 - Bureau Of Indian Standards Manak Bhavan, 9 Bahadur Shah Zafar Marg New Delhi 110002
- Is 1893-2000 - Bureau Of Indian Standards Manak Bhavan, 9 Bahadur Shah Zafar Marg New Delhi 110002
- IS 1893-2002 - Bureau Of Indian Standards Manak Bhavan, 9 Bahadur Shah Zafar Marg New Delhi 110002