



DESIGN AND ANALYSIS OF TRUSS

K Eswaramma¹, T Radhika²

¹Assistant Professor of Civil Engineering Department, Siddharth Institute of Engineering & Technology, Puttur A.P, India

²Post Graduate Student, Civil Engineering Department, Siddharth Institute of Engineering & Technology, Puttur A.P, India

ABSTRACT

The principle objective of this project is to analyze and design a Truss using STAAD Pro. In order to compete in the ever growing competent market it is very important for a structural engineer to save time. As a sequel to this an attempt is made to analyze and design a Truss by using a software package STAAD pro. The design involves load calculations manually and analyzing the whole structure by STAAD Pro. The structure was subjected to self weight, dead load, live load and wind load 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. 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.

1. INTRODUCTION

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

- 1) easy to use interface,
- 2) conformation with the Indian Standard Codes,

- 3) versatile nature of solving any type of problem,
- 4) 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, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

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 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.

2. STRUCTURES

Structural design is a branch of Engineering that deals with systems comprised from a set of structural members. These members may be characterized as either truss or frame elements, connected by pinned or fixed joints. Common structures include truss bridges, frame buildings, race car and airplane space frames, crane arms, and power line truss towers.

2.1 Structural Size and Shape of Optimization

To better understand optimization of structures and the focus of this work, two definitions must be stated. The first definition is that of the structure, including all implications and capabilities in the static analysis of such systems. The second definition applies to that of structural optimization, more specifically the optimization of size and shape. These two primary definitions will hold true for the entirety of this research and are derived from McGuire (2000). A *structure* is a set of *nodes* (vertices)

that are connected by a set of *elements* (edges). This includes all plane (2D) and space (3D) truss and frame structures. *Loads* may be placed at nodes to exert a force or moment on the structure. *Constraints* may be placed at nodes to restrain the structure from translation or rotation caused by nodal loads. A valid structure must constrain at least all six degrees of freedom as a system, and over constraint will generally produce stiffer structures.

All elements are associated with a material defined by a minimum of two values: modulus of elasticity (E) and Poisson's ratio (ν). These values define the element's behavior under static linear elastic loading conditions. Values used only for the optimization process include the element's yield strength (σ_y), and unit weight (γ) or mass density (ρ). These values are used for stress limit comparison and structural mass, respectively.

2.2. Structural Optimization Methods

Optimization of structures can be broken down to three categories: topology, size, and shape. All three categories generally have the objective of mass minimization with optional stress or displacement constraints.

- 1) **Topology optimization** -- variance of element-node connectivity to find an optimal layout design. Difficulties may arise when a change truss topology causes the structure to become a mechanism.
- 2) **Size optimization** -- variance of element cross sectional properties, which may be continuous or discrete variables.
- 3) **Shape optimization** -- movement of nodes to change the shape of the structure without changing the topology. The element-node connectivity remains intact. This work uses a combination of size and shape optimization techniques with the objective of mass minimization and the inclusion of element stress and nodal displacement constraints. Using only size and shape optimization voids the problems associated with topological optimization while allowing substantial changes to the structure. The size and shape optimization methods are applied in parallel rather than in a series process, meaning size and shape variables are change simultaneously.

2.3. Types of Connections

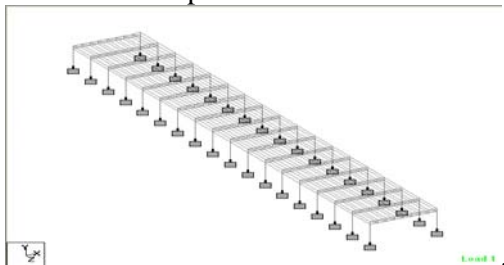
For all the types of member sections, it is possible to design either bolted connections or

welded connections. Generally, bolted connections are preferred on site. Where bolted connections are used with bolts loaded perpendicular to their shank, it is necessary to evaluate the consequences of slack in connections. In order to reduce these consequences (typically, the increase of the deflections), solutions are available such as use of pre-stressed bolts, or limiting the hole size.

3. PROGRAMME ANALYSIS

3.1 A truss:

A truss is a structure made of two force members all pin connected to each other



Design and Analysis of Truss Using Staad Pro

3.1.1 The method of joints

This method uses the free-body-diagram of joints in the structure to determine the forces in each member. For example, in the above structure we have 5 joints each having a free body diagram as follows

3.1.2 Main Steps Of Modeling Truss in STAAD PRO

- a) Entering Job information
- b) Building model geometry
- c) Defining member properties ,sections
- d) Assigning Loads(Load Cases and combination)
- e) Defining Pre-Analysis Print out,analysis type and Post-Analysis print out.
- f) Defining Design Requirements

3.1.3 Design of Footing

A foundation is assumed to act as a rigid body, which is in equilibrium under the action of applied forces from the structure and causes the stress in the soil .This permits the use of theory of bending to determine the stress distribution in soil for the given axial load moment . Just like any structural members a foundation must be designed to sustain the applied loads, moments and the induced reaction and ensure that the safe bearing capacity of the soil is not exceed.

3.1.4 Design of Column

- a) Column dimensions = 300mm x 450²mm
- b) Unsupported length = L = 4.0m
- c) Ends are effectively held in position & direction at both the ends.

3.2 Load:

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.

3.2.1 Joint loads

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint. In which we can define the applied loads, load cases, load combinations.

3.2.2 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.

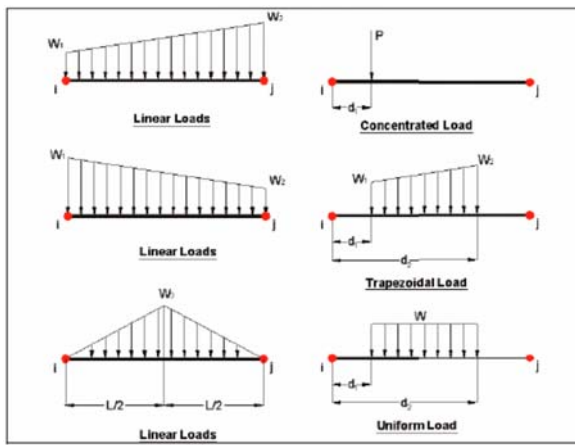


Fig.member load configuration

3.2.3 Area/floor load:

Many times a floor (bound by X-Z plane) is subjected to a uniformly distributed load. It could require a lot of work to calculate the member load for individual members in that floor. However, with the AREA or FLOOR LOAD command, the user can specify the area loads (unit load per unit square area) for members. The program will calculate the tributary area for these members and provide the proper member loads. The Area Load is used for one way distributions and the Floor Load is used for two way distributions

3.2.4 Fixed end member load:

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

3.2.5 Load Generator – Moving load, Wind & Seismic:

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

Moving Load Generator:

This feature enables the user to generate moving loads on members of a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis.

Wind Load Generator:

The STAAD Wind Load generator is capable of calculating wind loads on joints of a structure from user specified wind intensities and exposure factors. Different wind intensities may be specified for different height zones of the structure. Openings in the structure may be modeled using exposure factors. An exposure factor is associated with each joint of the structure and is defined as the fraction of the influence area on which the wind load acts. Built-in algorithms automatically calculate the exposed area based on the areas bounded by members (plates and solids are not considered), then calculates the wind loads from the intensity and exposure input and distributes the loads as lateral joint loads.

4. TEST RESULTS

4.1 Dead Loads:

All permanent constructions of the structure form the dead loads. The dead load comprises of the weights of walls, partitions floor finishes, false ceilings, false floors and the other permanent constructions in the buildings. The dead load loads may be calculated from the dimensions of various members and their unit weights. the unit weights of plain concrete and reinforced concrete made with sand and gravel or crushed natural stone aggregate may be taken as 24 kN/m³ and 25 kN/m³ respectively.

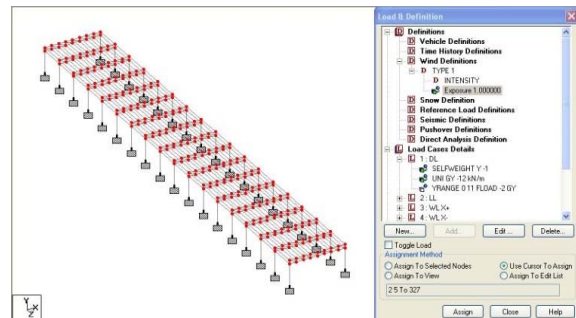
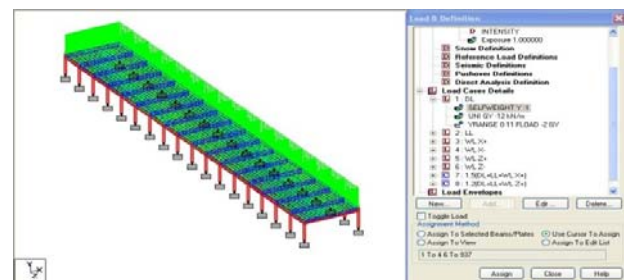


Figure4.1: Toassign the intensity & exposure load



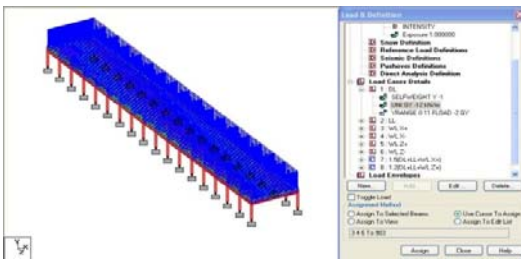


Figure 4.1.2: To assign the unit weight

4.2 Live Loads:

Live load is produced by the intended use or occupancy of a building including the weight of movable partitions, distributed and concentrated loads, load due to impact and vibration and dust loads. Imposed loads do not include loads due to wind, seismic activity, snow, and loads imposed due to temperature changes to which the structure will be subjected to, creep and shrinkage of the structure, the differential settlements to which the structure may undergo.

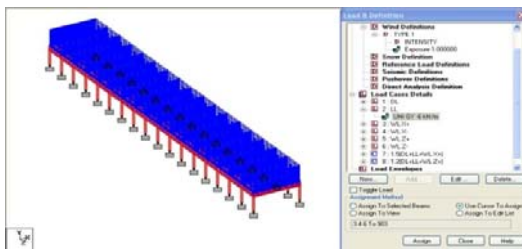


Figure 4.2.1: To assign the live load

4.3 Wind Load:

Wind is air in motion relative to the surface of the earth. The primary cause of wind is traced to earth's rotation and differences in terrestrial radiation. The radiation effects are primarily responsible for convection either upwards or downwards. The wind generally blows horizontal to the ground at high wind speeds. Since vertical components of atmospheric motion are relatively small, the term 'wind' denotes almost exclusively the horizontal wind, vertical winds are always identified as such. The wind speeds are assessed with the aid of anemometers or anemographs which are installed at meteorological observatories at heights generally varying from 10 to 30 metres above ground.

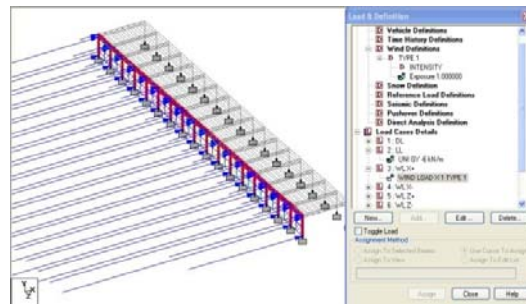


Figure 4.3.1: To assign the wind load x+

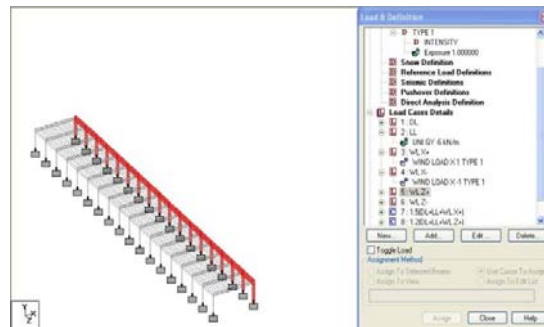


Figure 4.3.2: To assign the wind load z+

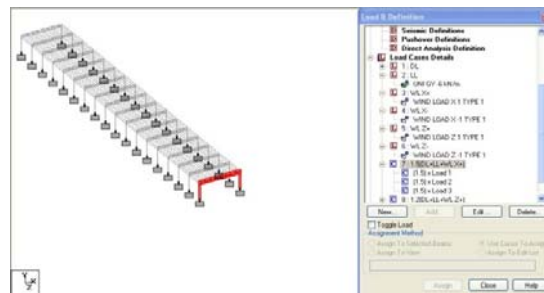


Figure 4.3.3: 1.5(dl+ll+wlx) loads

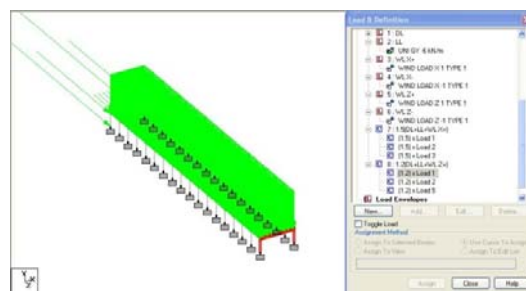
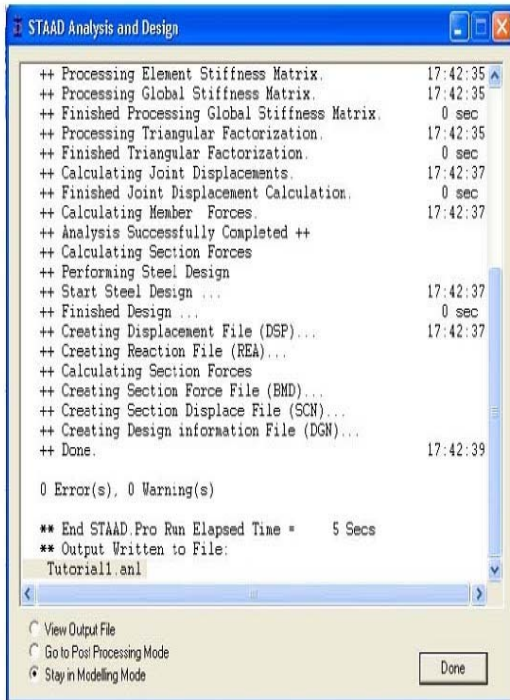
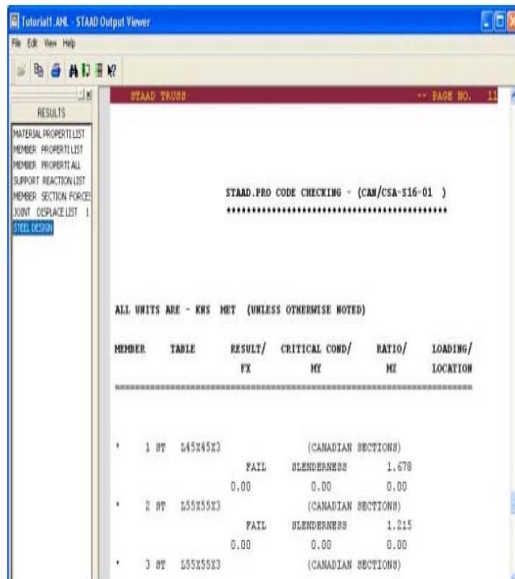


Figure 4.3.4: To assign the 1.5(load1)

4.4. The solving process is shown in pop up screen:



4.5 The Results of the Steel Design Check:



5. CONCLUSION

1) The planning of a truss is developed keeping in view, the building bye laws, environmental conditions prevailing in that area height of a building is restricted as per the municipal authorities of the area. The drawings of the plan is developed using auto cad software.

2) The slabs are designed as per code of practice IS-456-2000 in accordance to LSM.

3) The imposed loads are noted from codes of practice IS-875-1987(part2) the analysis of the frame as carried out in staad pro and design of beams and columns are also carried out in staad pro the footings and stair case and slab are designed manually as per code of practice IS 456-2000 in LSM.

6. REFERENCES

1. Text Books

- 1) N.KrishnaRaju, “Design of Concrete Mixes”, Year 2005
- 2) A.M.Nevile, “Properties of concrete” , ELBS with Longman 1987
- 3) M.S.Shetty, “Concrete Technology”, Year 2008
- 4) M.L.Gambhir, “Concrete Manual”, Dhanpat Rai Publications,
- 5) Planning & Layout of drawings by using N.Kumara Swamy.

2. IS Codes

- 1) IS 456 – 2000 Code of practice for Plain and Reinforced concrete.
- 2) IS 875 - 1987 Part I & II Code of Practice for Design of Loads in Buildings
- 3) SP 16 – 1980 Design Aids for RCC to IS456-2000
- 4) SP 34 – 1987 Hand book Concrete Reinforcement and Detailing
- 5) IS 1893-2002
- 6) IS 875 parts 3 for wind load
- 7) IS 875part 5 for load combination.

Software’s

- STAAD.PRO- V8i for Analysis and Design
- AUTOCAD - 2010 for Drawings
- M.S.OFFICE – 2007 for Documentation