



IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

e-ISSN: 2319-8753 | p-ISSN: 2347-6710



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 13, Issue 1, January 2024

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423



9940 572 462



6381 907 438



ijirset@gmail.com



www.ijirset.com

Design and Analysis Process of Beam using ANSYS

Muralidhara T K, Supriya T K

Senior Scale Lecturer, Department of Mechanical Engineering, DACG Govt. Polytechnic, Chikkamagalur, Karnataka, India

Lecturer, Department of Civil Engineering, DACG Govt. Polytechnic, Chikkamagalur, Karnataka, India

ABSTRACT: Detailing the **Deformation, Stresses & Strain** of the Beam structure is selected, the structural design process starts. Structural design is an art and science of understanding the behavior of structural members subjected to loads and designing them with economy by using ANSYS software the functional parameters relevant to the design of the facility would comprise those related to the user's requirement and guided by the design standard norms, standards by – laws and municipal of different experts such as architect, planner and structural engineer etc.

KEYWORDS: ANSYS, Structural Analysis, Seismic Design, Steel Structures

I. INTRODUCTION

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems

The ANSYS beam design and analysis process involves creating geometry (sketching lines/shapes), defining material properties (steel, aluminum), meshing the model (dividing into elements), applying boundary conditions (supports) and loads (forces), solving, and then post-processing results like deformation, stress, and natural frequencies, often validated against theoretical beam theory. This finite element analysis (FEA) workflow uses tools like ANSYS Workbench for simulation and Space Claim/DesignModeler for geometry, helping engineers understand beam performance.

Beam After getting an architectural plan of the buildings, the structural planning of the building frame is done. This involves determination of the following.

- Position and orientation of columns
- Positioning of beams

Spanning of slabs are widely used structural elements for covering large spans in industrial and commercial buildings due to their high strength-to-weight ratio and economy. In seismically active regions such as Nepal, proper truss design plays a crucial role in ensuring structural safety and performance. Reinforced concrete buildings often face ductility and detailing challenges, while steel trusses provide flexibility, faster construction, and durability. This paper presents an analytical study of trusses with reference to Indian Standards IS 1893-2002, IS 800-2007, and IS 875-1987.

1.1.1 Positioning of Beams

Beams shall normally be provided under the walls or below a heavy concentrated load to avoid these loads directly coming on slabs.

Avoid larger spacing of beams from deflection and cracking criteria. (The deflection varies directly with the cube of the span and inversely with the cube of the depth i.e. L^3/D^3 . Consequently, increase in span L which results in greater deflection for larger span).

1.1.2 Design:

This phase involves a detailed consideration of the alternative solutions defined in the planning phase and results in the determination of the most suitable proportions, dimensions and details of the structural elements and connections for constructing each alternative structural arrangement being considered.

1.2 Structural Design Guidelines

In structural design, there are some guidelines that you have to follow. A lot is going to depend upon the actual structure you want to design. General structural design guidelines are explained briefly in the below section:

- Conditions of the site where you want to build
- The loads that will be acting on your structure
- The design of your structure.

1.3 Various loads acting on the structure

Structure Loads

There will be different loads on your structure. You need to consider these before you build. This has to factor in the building process. Here are some of the loads that you need to consider

Dead Load

The dead load is formed by the self-weight of the actual structural members. These include the beams, columns, finishing, slab loads wall loads, plastering, and so on

Live Load

The load that a structure is subject to when it is occupied. These can be dynamic or static in nature. The load may or not be present during the use of the structure. For example, in industrial structures and buildings, there can be a live load from maintenance tools, people, and so on.

Wind Load

On the windward side of a site, the wind load will act horizontally on the building's surface area. Any site that is under construction or being considered has a wind zone. The maximum wind speed area is calculated based upon the wind zone

Earthquake and Seismic Loads

ASCE 7 or other applicable codes will determine the design for a seismic load. You can use this for the calculation of earthquake forces.

1.4 Structural Design Factors

When designing a structure, you have to look at the following standard codes, which are as follows:

- The maximum allowable settlement of the structure or foundation
- The lateral and vertical deflections of buildings and other structural members
- Overturning of buildings in sliding needs to be checked and prevented during the design
- Environmental factors and conditions should be considered during your design phase
- You need to address the structural members and the construction materials and their related properties that you're going to be using it during the design
- The structure should also be set up so that people can properly escape during an emergency, such as a fire.

1.5 Software Used:

The following software's are used in RULEG Constructions Pvt Ltd

- ANSYS
- MICROSOFT EXCEL SHEET
- AUTOCAD

a. Time Saving Options

The program also includes options that allow you to reduce the time spent creating models. Those options include the following:

- **Similar Stories.** Allows the user to make changes to multiple stories simultaneously.
- **Snap To.** Allows the user to place structural objects with accuracy.
- **Auto Select Sections.** Allows the user to define a list of sections, for Example W18X35, W18X40, W21X44, W21X50 and W24X55, that can be assigned to a frame member. The program can then automatically the most economical, adequate section from the auto selectSection list when it is designing the member.
- **Vertical Load Transfer.** Frees the user from the chore of calculating the load on the members supporting the floor plate, and determines the area tributary to each member for live load reduction.

- **Wall Stacks.** Allows the user to quickly generate complex wall arrangements.
- **Towers.** Allows multiple towers to exist within a single model.
- **Model Explorer.** Allows the user to rapidly create and modify models using a hierarchical tree system with drag-&-drop capability.
- **Design Strips.** Allows for accurate layout of program calculated slab Reinforcement and tendon placement.

ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems.

Working in ANSYS

Step 1: File – clear and start new – do not read file – ok Preferences

Step 2: Ansys Main Menu – Preferences select – STRUCTURAL – h method – ok Preprocessor

Step 3: Element type – Add/Edit/Delete – Add – BEAM – 2 node 188 – ok – close.

Step 4: Material Properties – material models – Structural – Linear – Elastic – Isotropic – EX – 2e5 – PRXY – 0.3 – ok – close.

Step 5: Sections-Beams-common sections- sub type- rectangle (1St element) - enter B=100, H = 50- preview-ok.

Step 6: Modeling – Create – Nodes – In Active CS – Node Number =1 – X, Y, Z Locations = 0,0,0, - Apply (first node is created) – Node Number =2 – X, Y, Z Locations = 2000,0,0, - ok (second node is created)

Step 7: Create – Elements – Auto numbered – Thru Nodes – pick 1 & 2 – ok (elements are created through nodes).

Step 8: Solution – Analysis Type – New Analysis – Static – ok

Step 9: Solution – Define Loads – Apply – Structural – Displacement – On Nodes – Pick 1st node – apply – DOFs to be constrained – ALL DOF – ok.

Step 10: Solution – Define Loads – Apply – Structural – Force/Moment – On Nodes – Pick 2 nd node – apply – direction of force/moment – FY – Real part of force/moment – -10000 – ok.

Step 11: Solve – current LS – ok (Solution is done is displayed) – close

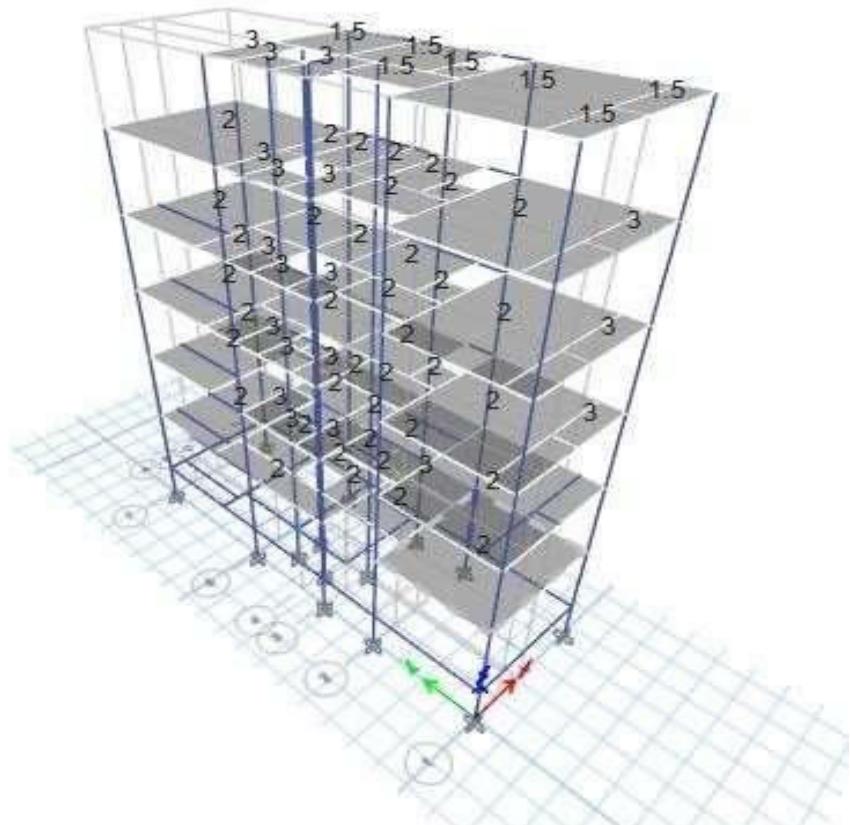
General Post Processor Step 12: Plot Results - Deformed Shape – Select-Def + undef edge' - click 'OK'

II. RESULTS AND DISCUSSION

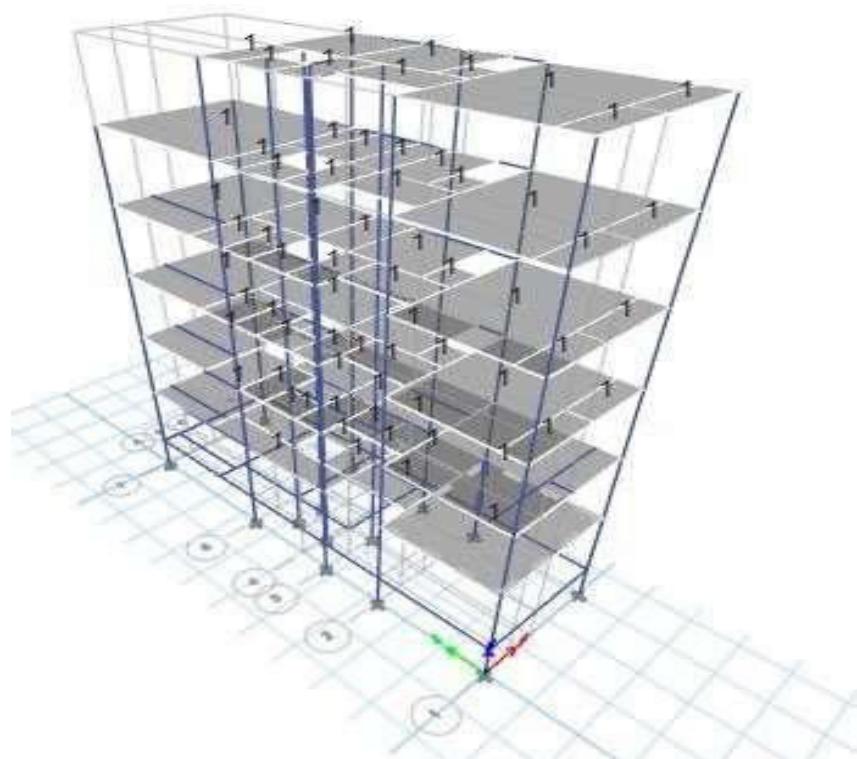
ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering: Structural | Thermal | Fluid (CFD, Acoustics, and other fluid analyses) | Low- and High-Frequency Electromagnetics. The results demonstrated that trusses efficiently distribute applied loads with minimum material usage. The 15m span truss exhibited higher deflection compared to shorter spans, but remained within permissible limits. Steel sections such as ISA 200x200x25 and ISMB500 were found suitable for the analyzed spans. Comparisons with conventional beams showed that trusses provided greater economy and strength-to-weight ratio.

III. MODEL ANALYSIS





Load for Slab



3.1 Beam:

A beam is a structural element that is capable of withstanding load primarily by resisting bending. The bending force induced into the material of the beam as a result of the external loads, own weight, span and external reactions to these loads is called a bending moment. Beams are characterized by their profile (shape of cross-section), their length, and their material.

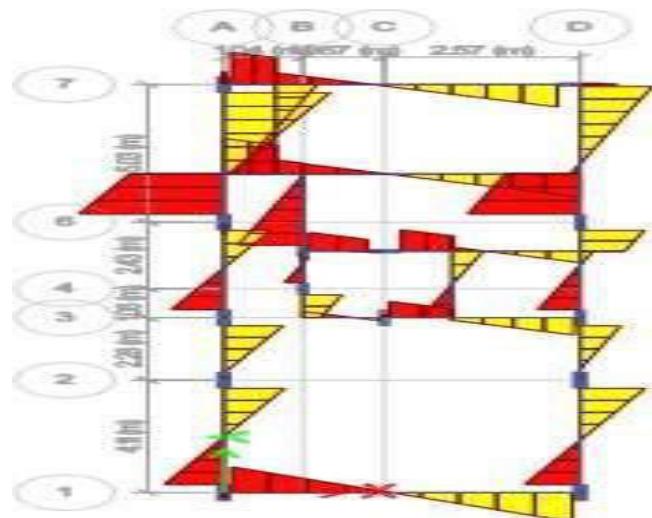


Fig 3.5: SFD of Plinth beam

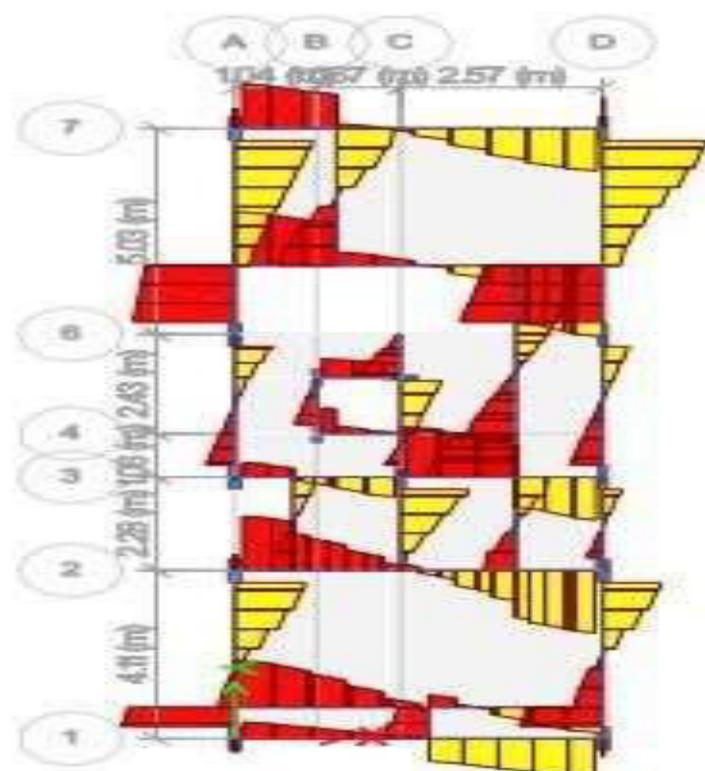


Fig 3.6: SFD of first floor beam

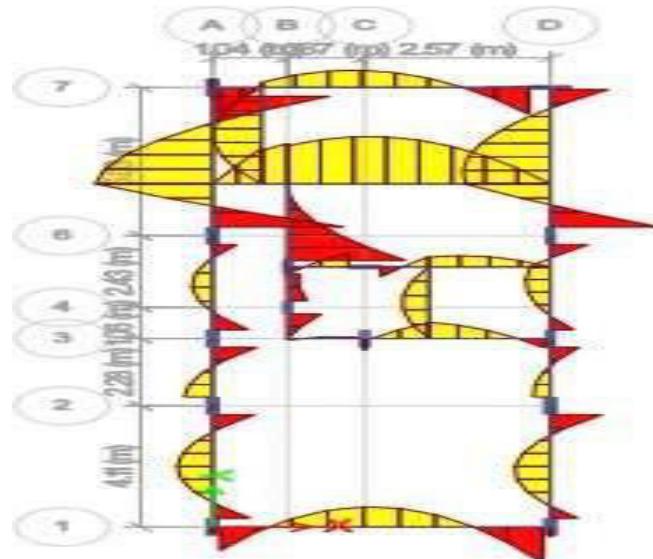


Fig 3.7: BMD of plinth beam

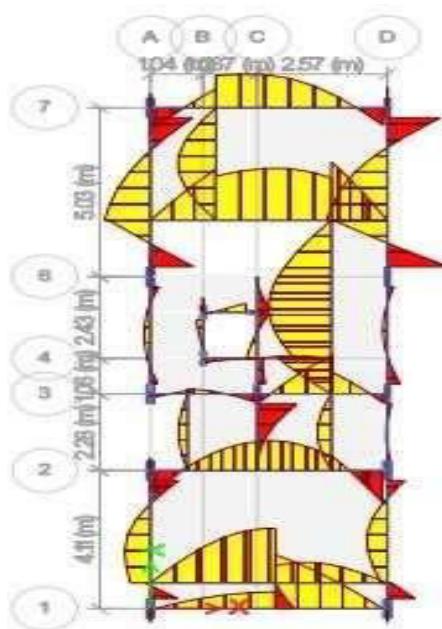


Fig 3.8: BMD of first floor beam

IV. CONCLUSION

- This study confirmed that Identification of Critical Points: ANSYS effectively identifies critical locations of maximum shear force (typically near supports) and maximum bending moment (which dictates the highest tension/compression zones), which is essential for determining where reinforcement is needed and ensuring safe structural design.
- Similarly, the Bending Moment Diagram (BMD) illustrates the distribution of bending moments along the beam, with maximum bending moments occurring at critical sections such as mid-span for simply supported beams or near fixed supports for cantilever beams.
- The locations of zero bending moment and peak values closely match analytical solutions, validating the accuracy of the finite element model.

- Minor deviations, if any, can be attributed to mesh density, element type, and numerical approximations. This study confirms that ANSYS is an effective and reliable tool for beam analysis, enabling accurate prediction of internal forces and moments, which are essential for safe and efficient structural design.

REFERENCES

[1] Finite Element Simulations with ANSYS Workbench by Huei-Huang Lee & Paul J. Schilling.

[2] ANSYS Tutorial Release (latest edition) by Kent L. Lawrence

[3] Strength of Materials / Mechanics of Materials Textbooks

Strength of Materials by R.K. Bansal

Mechanics of Materials by Beer & Johnston

Elements of Strength of Materials by Timoshenko & Young – etc.

[4] Structural Analysis Textbooks

– Books like *Structural Analysis* by Bhavikatti or *Analysis of Structures* explore SFD/BMD in statically

[5] Roark's Formulas for Stress and Strain (Budynas & Sadegh)

CODE BOOKS

- IS 456: 2000, "Indian Standard Code for Plain and Reinforced Concrete", Bureau of Indian Standards, New Delhi.
- IS 875(part 1): 1987, "Indian Standard Code of practice for design loads (other than earthquake for building and structures): Dead loads, Bureau of Indian Standards, New Delhi.
- IS 875(part 2): 1987, "Indian Standard Code of practice for design loads (other than earthquake for building and structures): Imposed loads, Bureau of Indian Standards, New Delhi.
- SP 16: 1980, "Design Aids for Reinforced Concrete to IS: 456- 1978", Bureau of Indian Standards, New Delhi.
- SP 34: 1987, "Hand Book on Concrete Reinforcement and Detailing to IS: 456- 1978", Bureau of Indian Standards, New Delhi.



INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

9940 572 462 6381 907 438 ijirset@gmail.com

www.ijirset.com



Scan to save the contact details