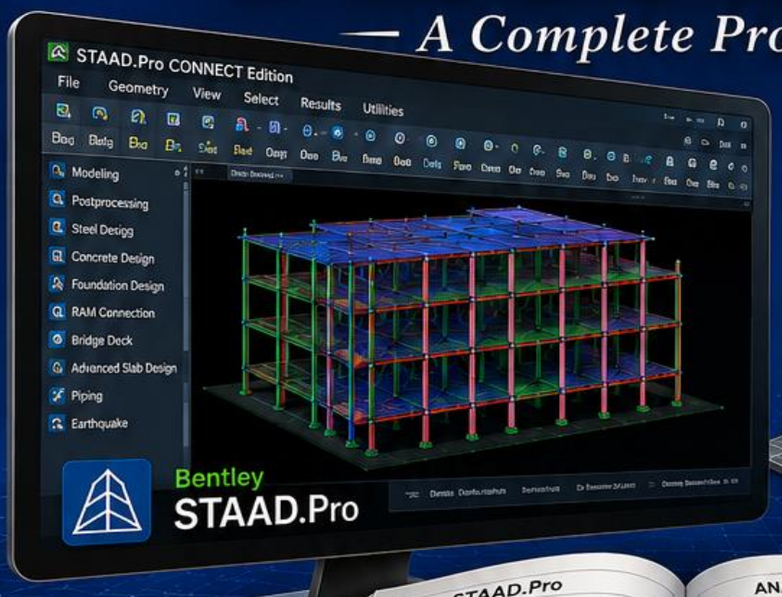
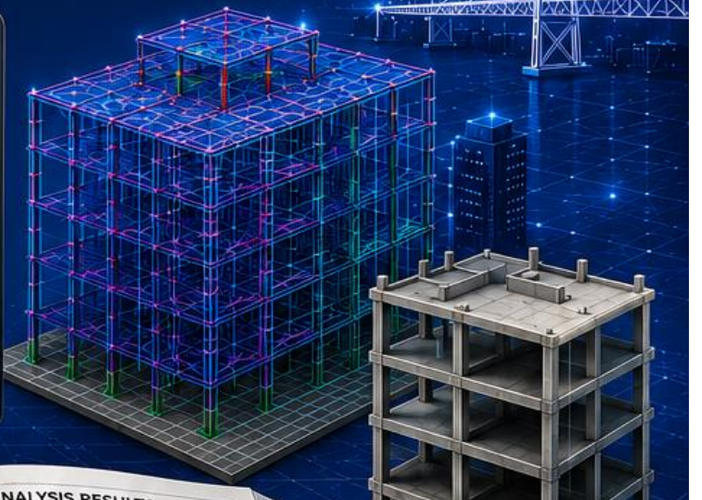
NRCM
Your roots to success...

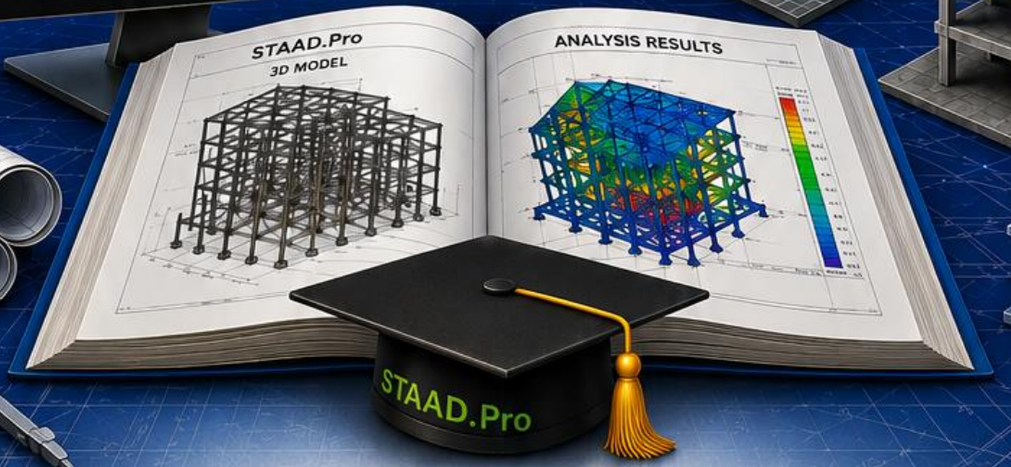
NARSIMHA REDDY ENGINEERING COLLEGE
(AUTONOMOUS)

STAAD.Pro PROJECTS

— A Complete Project Guide —







Prof. D. Venkatesh
HOD
Department of Civil Engineering

www.nrcmec.org | principal@nrcmec.org | Kompally, Hyderabad, Telangana - 500100

STAAD.Pro. Software Projects

A Collection of STAAD. Pro Design Projects

1. Analysis and Design of G+10 Residential Building Plan Using STAAD.Pro Software

Abstract

A **G+10 residential building** consists of a ground floor and ten upper floors designed to accommodate residential apartments. The structural analysis and design of such buildings are essential to ensure safety, stability, and durability under different loading conditions. This project presents the analysis and design of a G+10 reinforced concrete residential building using **STAAD.Pro** software. The building is modeled in three dimensions by defining beam, column, slab, and support elements. Dead load, live load, self-weight, and seismic load are applied according to Indian Standard (IS) codes. STAAD.Pro performs structural analysis and provides bending moments, shear forces, axial forces, support reactions, and deflections. Based on these results, the structural members are designed to satisfy safety and serviceability requirements. The study demonstrates that STAAD.Pro is an efficient software for the analysis and design of high-rise RCC residential buildings.

Introduction

The demand for high-rise residential buildings has increased significantly due to rapid urbanization and limited land availability. A **G+10 residential building** provides efficient utilization of land while accommodating more families. Such structures must safely resist dead loads, live loads, wind loads, and earthquake forces throughout their service life.

Modern structural design relies on software to perform accurate and efficient analysis. **STAAD.Pro** is one of the most widely used structural engineering software packages for modeling, analyzing, and designing reinforced concrete and steel structures. In this project, a G+10 residential building is modeled and analyzed using STAAD.Pro. The analysis results are used to design beams and columns, ensuring the building meets the requirements of relevant Indian Standard codes.

Theory of G+10 Residential Building

Theory

A **G+10 RCC building** is a framed structure consisting of beams, columns, slabs, and foundations connected in three dimensions. The loads are transferred from slabs to beams, beams to columns, and columns to the foundation.

Main Components

- RCC Beams
- RCC Columns
- RCC Slabs
- Staircase
- Isolated Footings
- Fixed Supports

Advantages

- Efficient land utilization
- High structural stability
- Better load distribution
- Suitable for urban residential projects

Methodology & Software Used

Methodology

1. Prepare the building plan.
2. Create the 3D model in STAAD.Pro.
3. Assign beam, column, and slab properties.
4. Define material properties.
5. Assign support conditions.
6. Apply dead load, live load, and seismic load.
7. Perform structural analysis.

8. Design structural members.
9. Check structural safety.
10. Prepare the final report.

Software Used

Software	Purpose
STAAD.Pro	Structural Analysis & Design
AutoCAD	Plan & Structural Drawings
MS Excel	Load Calculations
MS Word	Project Documentation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Elastic Modulus	27 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Floors	G+10
Total Height	33 m
Bay Size	5 m × 5 m
Beam Size	300 × 450 mm
Column Size	300 × 600 mm
Slab Thickness	150 mm
Support Type	Fixed

Load Calculations

Dead Load (DL)

- Self-weight = Calculated automatically in STAAD.Pro
- Floor Finish = **1.0 kN/m²**
- Wall Load = **12 kN/m**

Live Load (LL)

As per **IS 875 (Part 2)**

- Residential Live Load = **2.0 kN/m²**

Seismic Load (EQ)

As per **IS 1893 (Part 1):2016**

- Zone factor and other seismic parameters are assigned based on the project location.

Load Combination

- 1.5 (DL + LL)
- 1.2 (DL + LL + EQ)
- 1.5 (DL + EQ)

Analysis & Design Results

Analysis Results

The G+10 building was analyzed using STAAD.Pro. The software generated:

- Joint Displacements
- Support Reactions
- Axial Forces
- Shear Force Diagram (SFD)

- Bending Moment Diagram (BMD)
- Storey Drift
- Deflected Shape

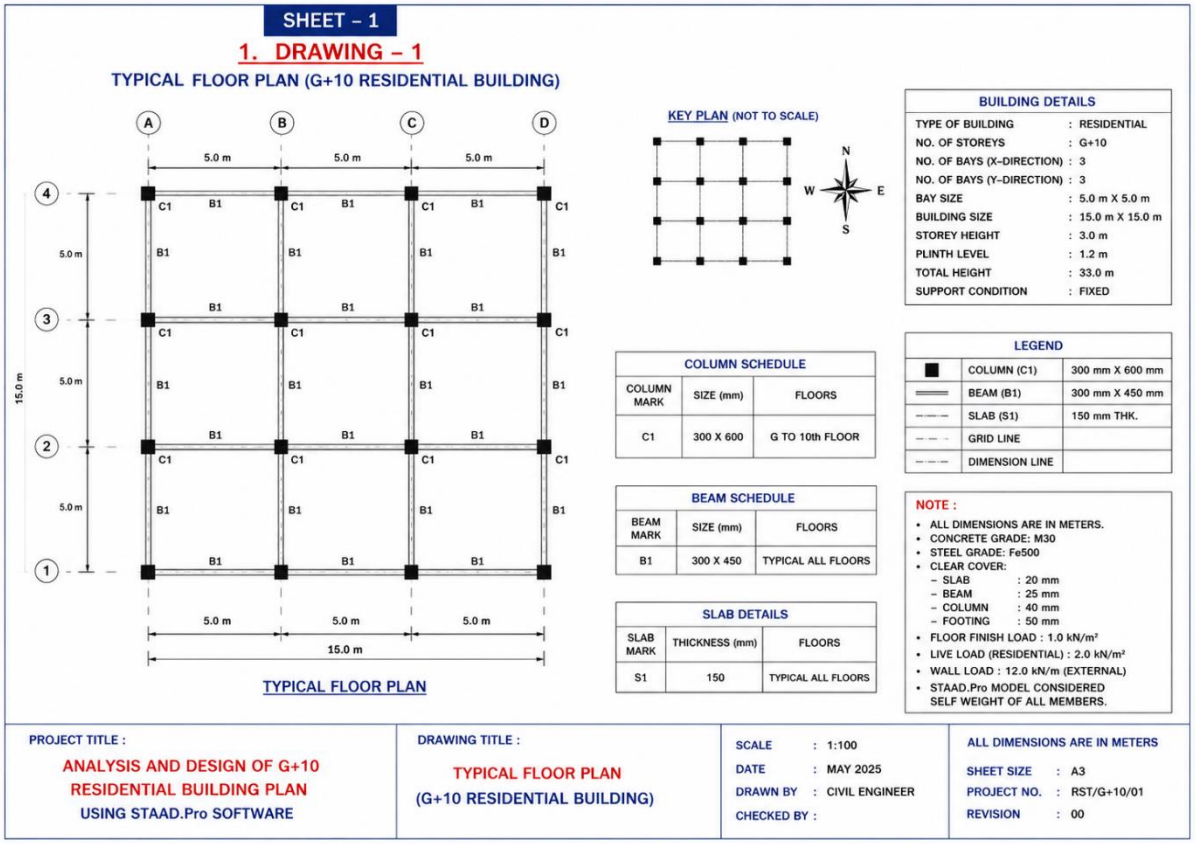
Design Results

- Beams satisfy bending and shear requirements.
- Columns are safe under combined axial and bending loads.
- Storey drift is within permissible limits.
- The overall structure is stable under the applied load combinations.

Typical Floor Plan

Contents:

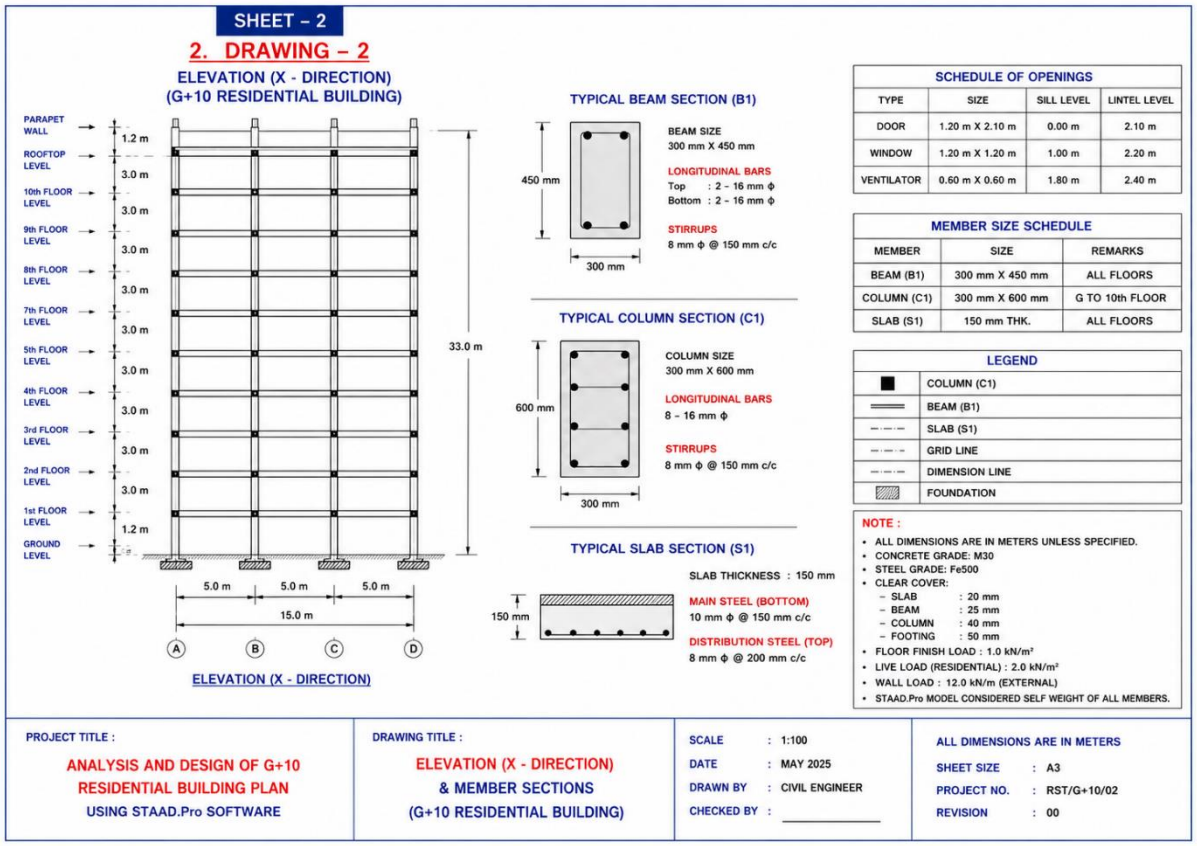
- Column grid (4 × 4)
- Beam layout
- Grid labels (A–D and 1–4)
- Bay dimensions: **5.0 m × 5.0 m**
- Overall building size: **15 m × 15 m**
- Legend and notes



Elevation (X-Direction)

Contents:

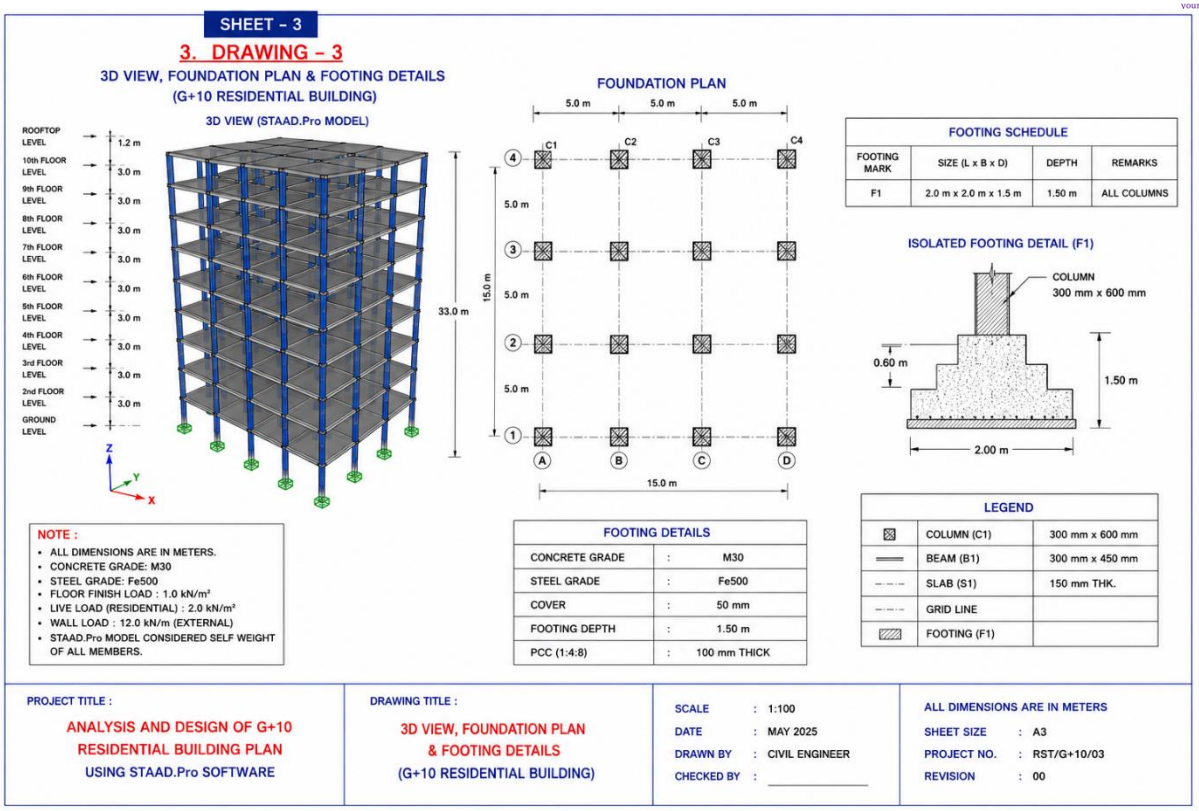
- G+10 elevation
- Storey height: **3.0 m**
- Total height: **33.0 m**
- Beam section: **300 × 450 mm**
- Column section: **300 × 600 mm**
- Foundation level
- Dimensions and notes



3D STAAD.Pro View & Foundation Plan

Contents:

- 3D STAAD.Pro structural model
- Isolated footing layout
- Footing size: **2.0 m × 2.0 m × 1.5 m**
- Column locations
- Foundation details



Applications

- Residential Apartment Buildings
- High-Rise Housing Projects
- Staff Quarters
- Urban Residential Complexes
- Mixed Residential Developments

Conclusion

The analysis and design of the **G+10 residential building** were successfully completed using STAAD.Pro software. The structure was analyzed under dead, live, and seismic loads, and the analysis confirmed that the beams, columns, and foundations were structurally safe. The software accurately calculated member forces, reactions, and deflections while reducing design time and improving accuracy. STAAD.Pro proved to be an effective tool for the structural analysis and design of multi-storey RCC residential buildings.

2. Analysis and Design of G+5 3D Plan Using STAAD.Pro Software"

Abstract

The **Analysis and Design of a G+5 Residential Building** is an important aspect of structural engineering that ensures the safety, stability, and durability of multi-storey buildings. This project presents the structural analysis and design of a G+5 reinforced concrete (RCC) building using **STAAD.Pro** software. The building is modeled in three dimensions by defining beams, columns, slabs, and support conditions. Dead load, live load, self-weight, and seismic load are applied according to relevant Indian Standard (IS) codes. STAAD.Pro performs the structural analysis and provides results such as bending moments, shear forces, axial forces, support reactions, and deflections. Based on these results, the structural members are designed to satisfy safety and serviceability requirements. The project demonstrates the effectiveness of STAAD.Pro in producing accurate, economical, and reliable structural designs for multi-storey residential buildings.

Introduction

The rapid growth of urban populations has increased the demand for multi-storey residential buildings, making efficient structural design essential. A **G+5 residential building** provides an effective solution for accommodating more families while making optimum use of available land. The structural system must safely withstand various loads, including dead load, live load, wind load, and earthquake load, throughout its service life.

Modern structural engineering relies on advanced software to improve the accuracy and efficiency of analysis and design. **STAAD.Pro** is one of the most widely used structural analysis software applications for modeling, analyzing, and designing reinforced concrete and steel structures. It enables engineers to create three-dimensional structural models, apply different loading conditions, and obtain accurate results such as bending moments, shear forces, axial forces, support reactions, and deflections. In this project, a **G+5 RCC residential building** is modeled and analyzed using STAAD.Pro according to relevant Indian Standard (IS) codes. The project helps in understanding structural behavior and

demonstrates the practical application of computer-aided design in developing safe, stable, and economical residential buildings.

Theory of G+5 3D Building

- Definition
- Structural Components
- Advantages
- Load Transfer Mechanism

Methodology

1. Building planning
2. 3D modeling in STAAD.Pro
3. Material definition
4. Member property assignment
5. Support conditions
6. Load application
7. Structural analysis
8. RCC member design
9. Design verification
10. Report preparation

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Structural Drawings
MS Excel	Load Calculations
MS Word	Report Documentation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m³
Elastic Modulus	27 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Floors	G+5
Total Height	18 m
Bay Size	5 m × 5 m
Beam Size	300 × 450 mm
Column Size	300 × 600 mm
Slab Thickness	150 mm
Support Type	Fixed

Load Calculations

Dead Load

- Self-weight (STAAD.Pro)
- Floor finish = 1.0 kN/m²
- Wall load = 12 kN/m

Live Load

- 3.0 kN/m² (IS 875 Part 2)

Seismic Load

- As per IS 1893 (Part 1):2016

Load Combinations

- 1.5(DL + LL)
- 1.2(DL + LL + EQ)
- 1.5(DL + EQ)

Analysis Results

- Joint Displacements
- Support Reactions
- Axial Forces
- Shear Force Diagram
- Bending Moment Diagram
- Storey Drift
- Deflected Shape

Design Results

- Beam design satisfies bending and shear requirements.
- Columns are safe under axial and bending loads.
- Deflection and drift are within permissible limits.
- Overall structure is stable.

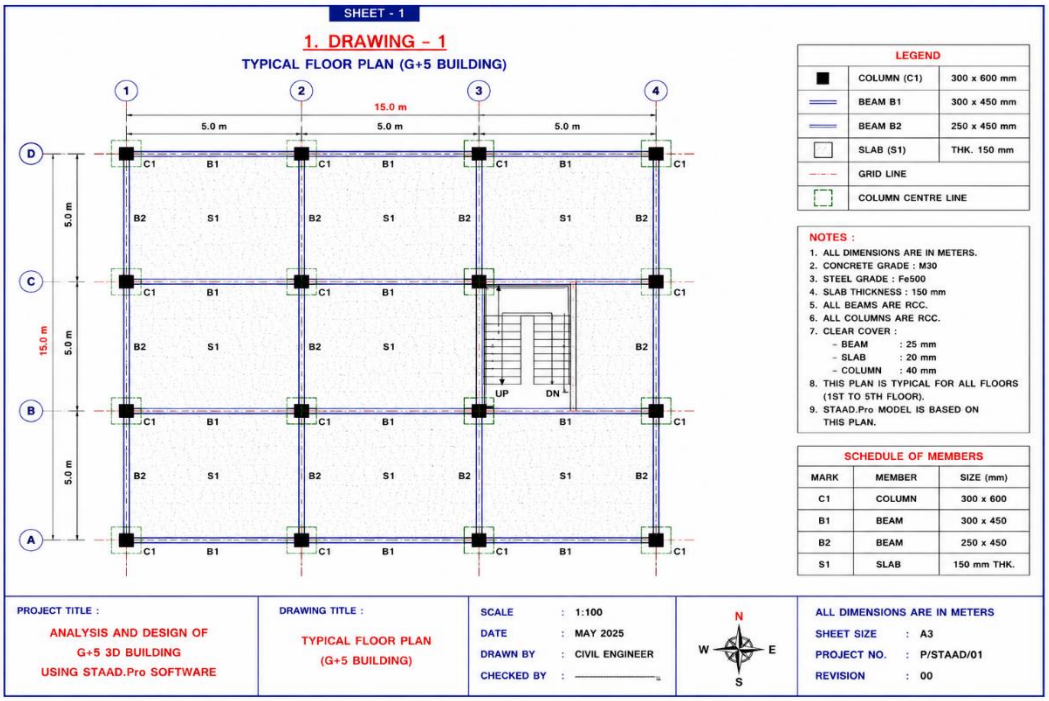
Drawing Sheet

Typical Floor Plan

Contents:

- Column Grid (4 × 4)
- Beam Layout

- Grid Labels (A–D and 1–4)
- Bay Size = 5 m × 5 m
- Overall Plan = 15 m × 15 m
- Dimension Lines
- Notes
- Legend

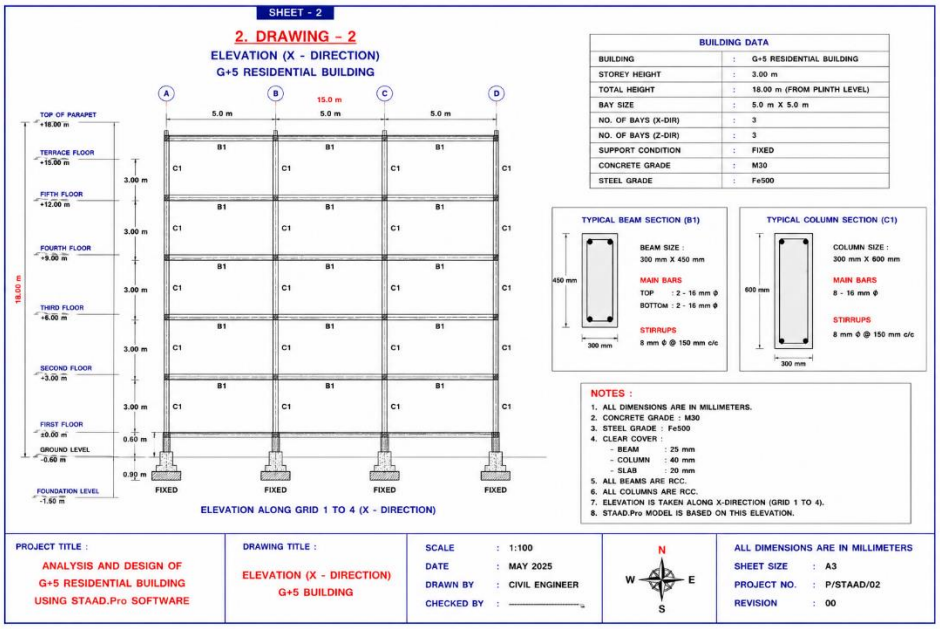


Drawing Sheet

Elevation (X-Direction)

Contents:

- G+5 Elevation
- Storey Height = 3.0 m
- Total Height = 18 m
- Beam Size = 300 × 450 mm
- Column Size = 300 × 600 mm
- Foundation Level
- Structural Dimensions
- Section Marks

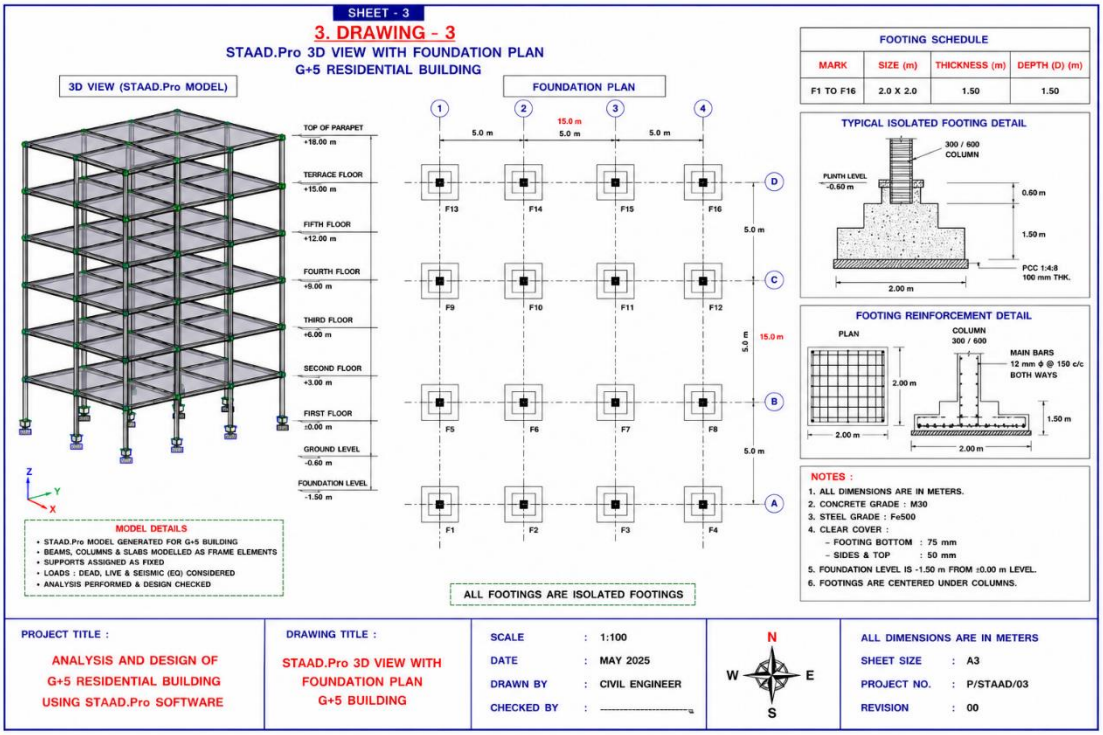


Drawing Sheet

STAAD.Pro 3D View & Foundation Plan

Contents:

- 3D Structural Model
- Isolated Footing Layout
- Footing Size = 2.0 m × 2.0 m × 1.5 m
- Column Locations
- Foundation Details
- Footing Schedule
- Notes & Legend



Applications

- Residential Apartments
- Commercial Buildings
- Educational Institutions
- Office Buildings
- Hotels

Conclusion

The analysis and design of the **G+5 3D building** were successfully completed using STAAD.Pro software. The structure was analyzed under dead, live, and seismic loads. The results showed that the beams, columns, and foundations satisfy the design requirements as per the relevant Indian Standards. STAAD.Pro provided accurate analysis and design results while reducing calculation time, making it an effective software for the structural design of multi-storey RCC buildings.

3. Analysis of 2D Portal Frame Using STAAD.Pro Software

Abstract

The **Analysis of a 2D Portal Frame** is an important study in structural engineering for understanding the behavior of framed structures under different loading conditions. This project presents the structural analysis of a reinforced concrete 2D portal frame using **STAAD.Pro** software. The portal frame is modeled by defining beam and column members with fixed support conditions. Dead load, live load, and wind load are applied according to relevant Indian Standard (IS) codes. STAAD.Pro performs the structural analysis and generates results such as bending moments, shear forces, axial forces, support reactions, and joint displacements. Based on the analysis results, the structural members are verified for safety and stability. The project demonstrates the accuracy and efficiency of STAAD.Pro in analyzing portal frame structures and helps understand their structural behavior under different loading conditions.

Introduction

A **2D portal frame** is one of the most commonly used structural systems in civil engineering for industrial buildings, warehouses, workshops, garages, and commercial structures. It consists of two vertical columns connected by a horizontal beam through rigid joints, enabling the frame to resist both vertical and horizontal loads efficiently. The rigid connection between the beam and columns allows the structure to transfer bending moments, shear forces, and axial loads safely to the foundation.

Structural analysis is necessary to determine the strength, stability, and serviceability of portal frames under various loading conditions. Manual analysis of portal frames becomes time-consuming and complex when multiple load combinations are considered. Therefore, engineers use structural analysis software such as **STAAD.Pro** to obtain accurate and reliable results. In this project, a reinforced concrete 2D portal frame is modeled and analyzed using STAAD.Pro. The frame is subjected to dead load, live load, and wind load as per Indian Standard (IS) codes. The analysis results help evaluate the structural performance of the frame and ensure that it meets the required safety and design criteria.

Theory of 2D Portal Frame

Theory

A **2D portal frame** is a rigid framed structure consisting of beams and columns connected through moment-resisting joints. The frame resists vertical loads such as dead and live loads and horizontal loads such as wind loads. Due to the rigid beam-column connections, the frame develops bending moments at the joints, increasing the overall stability of the structure.

Main Components

- RCC Beam
- RCC Columns
- Fixed Supports
- Beam–Column Rigid Joints
- Foundation

Advantages

- High structural stability
- Economical construction
- Large column-free space
- Simple analysis and design
- Suitable for industrial structures

Methodology & Software Used

Methodology

1. Define portal frame geometry.
2. Model the frame in STAAD.Pro.
3. Assign material properties.
4. Define beam and column sections.
5. Assign fixed supports.

6. Apply dead, live, and wind loads.
7. Perform structural analysis.
8. Review analysis results.
9. Check member safety.
10. Prepare the project report.

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Engineering Drawings
MS Excel	Load Calculations
MS Word	Report Preparation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Elastic Modulus	27 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Frame Width	10.0 m
Frame Height	5.0 m
Beam Size	300 × 450 mm
Column Size	300 × 500 mm
Support Type	Fixed

Load Calculations

Dead Load (DL)

- Self-weight (Calculated automatically by STAAD.Pro)
- Roof finish = **1.0 kN/m²**
- Wall load (if applicable) = **10 kN/m**

Live Load (LL)

As per **IS 875 (Part 2)**

- Roof Live Load = **1.5 kN/m²**

Wind Load (WL)

As per **IS 875 (Part 3)**

Wind load parameters are assigned based on the selected wind zone and building dimensions.

Load Combinations

- 1.5 (DL + LL)
- 1.2 (DL + LL + WL)
- 1.5 (DL + WL)

Analysis & Design Results

Analysis Results

The 2D portal frame was analyzed using STAAD.Pro. The software generated:

- Joint Displacements
- Support Reactions
- Axial Forces

- Shear Force Diagram (SFD)
- Bending Moment Diagram (BMD)
- Deflected Shape

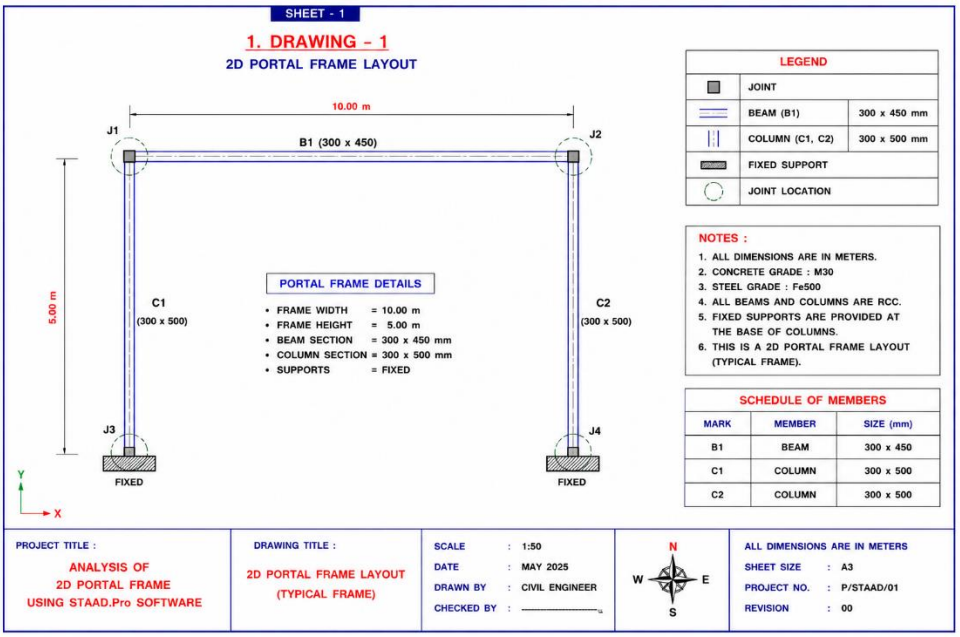
Design Results

- Beam section is adequate for bending and shear.
- Columns safely resist axial load and bending moment.
- Maximum deflection is within permissible limits.
- The frame is structurally stable under all load combinations.

2D Portal Frame Layout

Contents

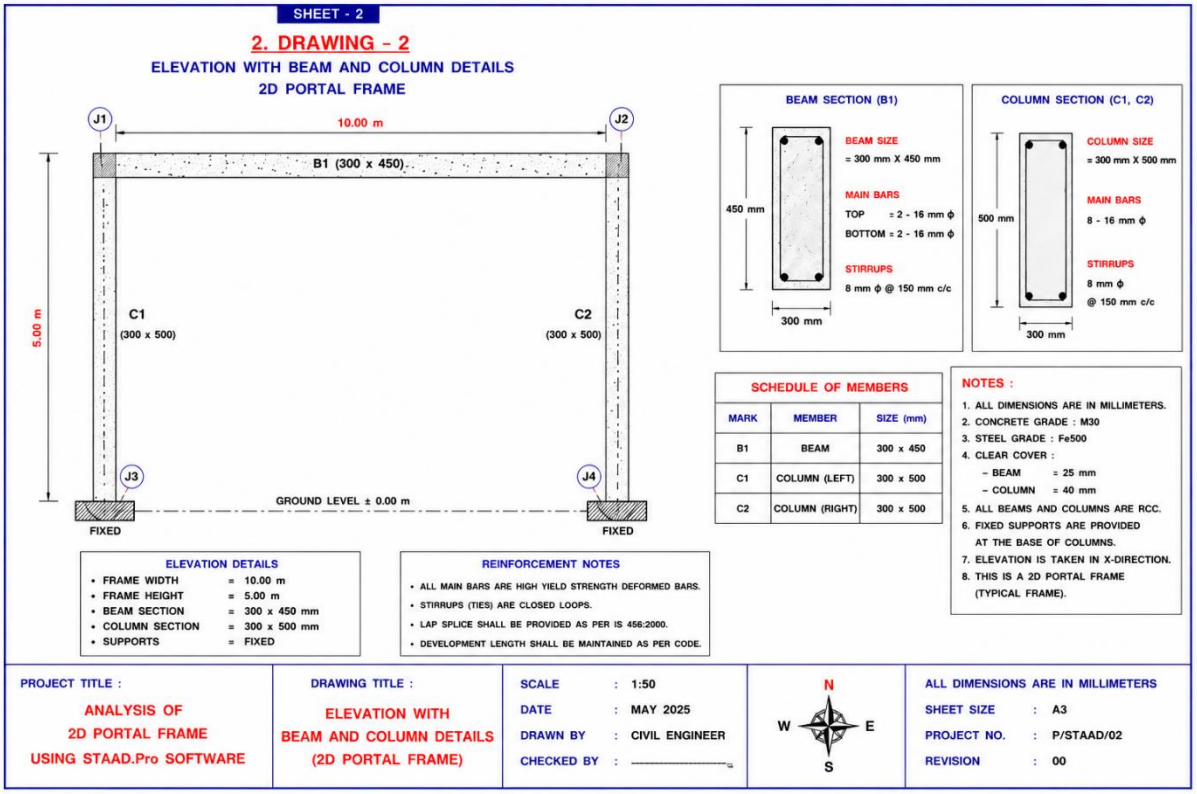
- Portal frame geometry
- Beam and column layout
- Overall dimensions (10 m × 5 m)
- Member labels
- Fixed support symbols
- Notes and legend



Elevation & Member Details

Contents

- Portal frame elevation
- Beam section (300 × 450 mm)
- Column section (300 × 500 mm)
- Overall dimensions
- Foundation level
- Reinforcement notes
- Member schedule

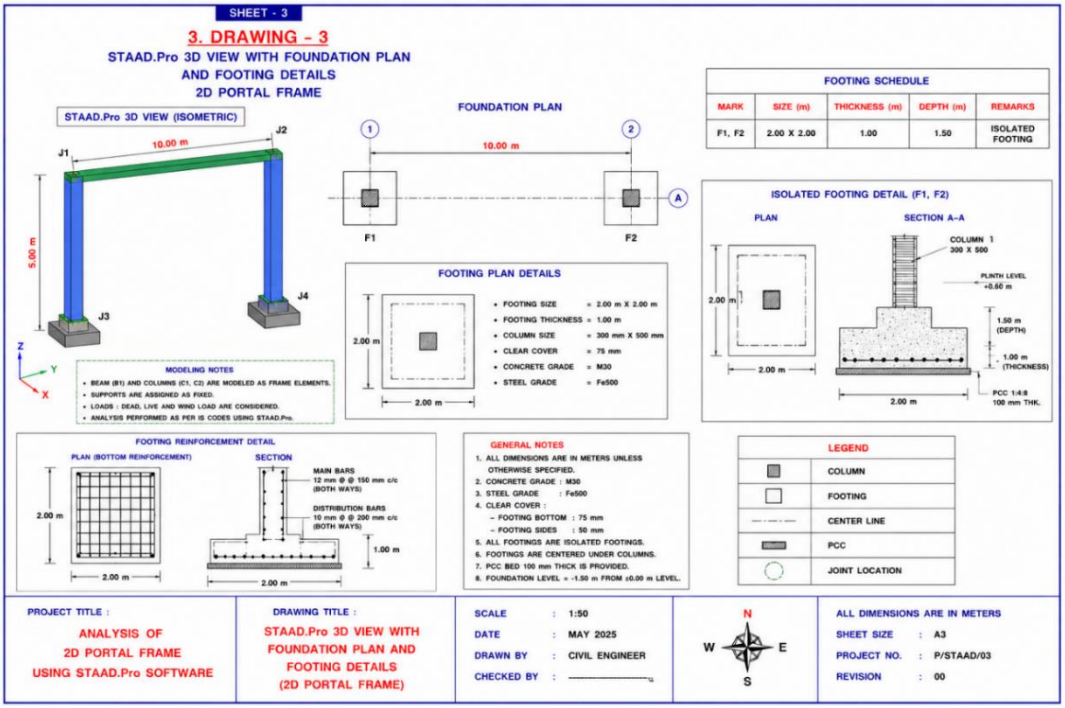


STAAD.Pro 3D View & Foundation Plan

Contents

- STAAD.Pro isometric model (illustrative view)
- Foundation plan
- Isolated footing details

- Footing size: **2.0 m × 2.0 m × 1.0 m**
- Column locations
- Notes and footing schedule



Applications

- Industrial Buildings
- Factory Sheds
- Warehouses
- Workshops
- Aircraft Hangars

Conclusion

The analysis of the **2D portal frame** was successfully completed using STAAD.Pro software. The structure was analyzed under dead, live, and wind loads, and the generated results confirmed that the beam and columns satisfy the required strength and stability criteria. The analysis showed acceptable bending moments, shear forces, axial forces, and deflections. STAAD.Pro simplified the structural analysis process and provided accurate results, making it an effective tool for the analysis and design of portal frame structures.

4. Analysis of 3 Bay – 5 Storey 2D Frame Using STAAD.Pro Software

Abstract

The **Analysis of a 3 Bay – 5 Storey 2D Frame** is an important structural engineering study that evaluates the behavior of a multi-storey reinforced concrete frame under various loading conditions. This project presents the modeling and analysis of a five-storey, three-bay RCC frame using **STAAD.Pro** software. The frame consists of beams and columns connected through rigid joints and is analyzed by considering dead load, live load, self-weight, and seismic load according to relevant Indian Standard (IS) codes. STAAD.Pro is used to determine structural responses such as bending moments, shear forces, axial forces, support reactions, and joint displacements. Based on the analysis results, the structural members are checked to ensure safety, stability, and serviceability. This project demonstrates the effectiveness of STAAD.Pro in providing accurate structural analysis while reducing manual calculations, making it a reliable tool for the design and evaluation of multi-storey RCC frame structures.

Introduction

The increasing demand for residential and commercial buildings has led to the widespread use of multi-storey reinforced concrete framed structures. Among these, the **3 Bay – 5 Storey 2D Frame** is one of the most commonly adopted structural systems due to its simplicity, strength, and economical design. A framed structure consists of beams and columns connected by rigid joints, which effectively transfer loads from the slabs to the foundation. Proper structural analysis is essential to ensure that the building safely withstands dead loads, live loads, wind loads, and earthquake forces throughout its service life.

With advancements in structural engineering, software-based analysis has become an essential part of the design process. **STAAD.Pro** is one of the most widely used structural analysis and design software packages, capable of modeling complex structures and

producing accurate analysis results. In this project, a **3 Bay – 5 Storey 2D RCC frame** is modeled using STAAD.Pro, and various load cases are applied according to Indian Standard (IS) codes. The software calculates member forces, bending moments, shear forces, axial forces, support reactions, and deflections, enabling engineers to verify the structural safety and performance. This project provides practical knowledge of structural modeling and demonstrates the importance of computer-aided analysis in designing safe, stable, and economical multi-storey framed structures.

Theory of 3 Bay – 5 Storey 2D Frame

- Definition
- Structural Components
- Structural Behaviour
- Load Transfer Mechanism
- Advantages
- Typical Applications

Methodology & Software Used

Methodology

1. Collection of structural data
2. Preparation of 2D frame geometry
3. Modeling in STAAD.Pro
4. Material property assignment
5. Member property definition
6. Support assignment
7. Load definition
8. Load combinations
9. Structural analysis
10. Design verification

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Structural Drawings
MS Excel	Load Calculations
MS Word	Report Documentation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Young's Modulus	27 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Number of Bays	3
Number of Storeys	5
Bay Width	5.0 m
Storey Height	3.0 m
Total Width	15.0 m
Total Height	15.0 m
Beam Size	300 × 450 mm
Column Size	300 × 600 mm
Support Type	Fixed

Load Calculations

Dead Load (DL)

- Self-weight = Automatically calculated by STAAD.Pro
- Floor finish = 1.0 kN/m²
- Wall load = 12 kN/m

Live Load (LL)

As per **IS 875 (Part 2)**

- Floor live load = 3.0 kN/m²

Seismic Load (EQ)

As per **IS 1893 (Part 1):2016**

- Zone factor
- Importance factor
- Response reduction factor
- Soil type

Load Combinations

- 1.5(DL + LL)
- 1.2(DL + LL + EQ)
- 1.5(DL + EQ)

Analysis & Design Results

Analysis Results

The structural analysis was performed using STAAD.Pro under all specified load combinations. The software generated:

- Joint Displacements
- Support Reactions
- Axial Forces
- Shear Force Diagram (SFD)
- Bending Moment Diagram (BMD)
- Deflected Shape
- Storey Drift

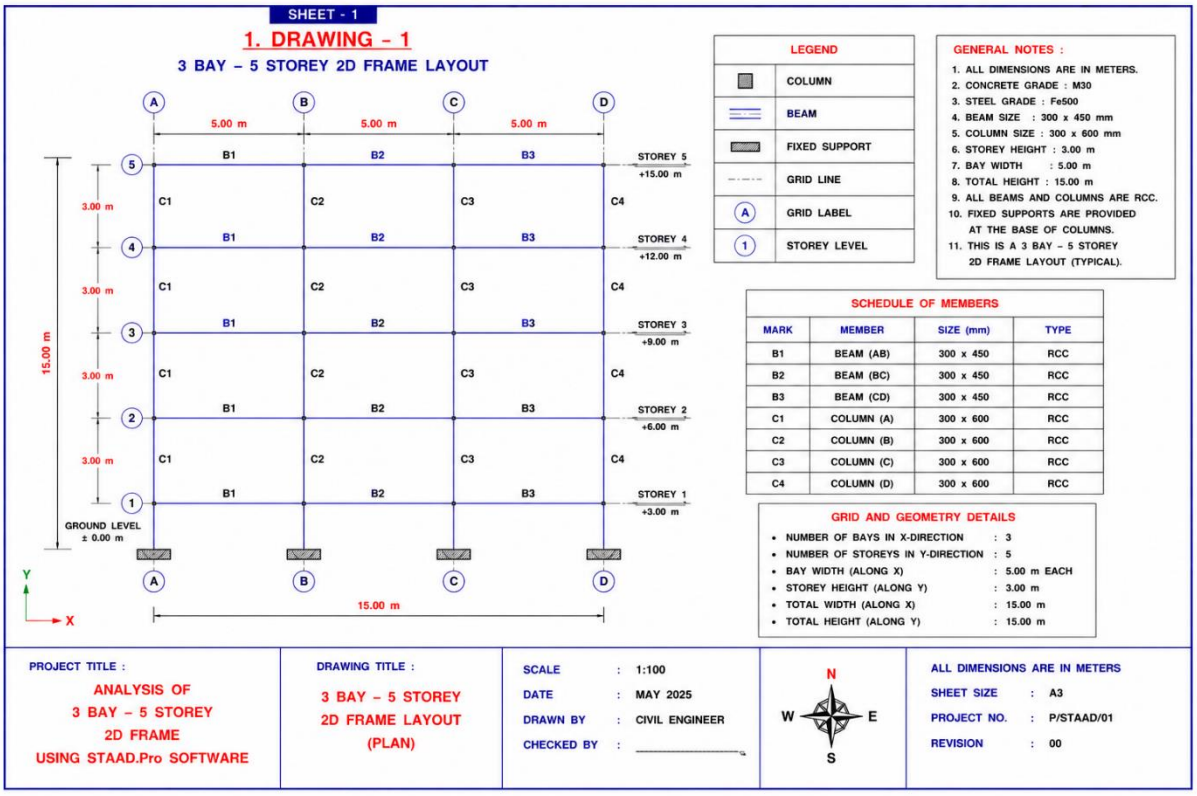
Design Results

- Beams satisfy bending and shear requirements.
- Columns are safe under combined axial and bending loads.
- Deflections are within permissible limits.
- Storey drift satisfies IS code requirements.
- Overall structure is stable and safe.

Bay – 5 Storey 2D Frame Layout

Contents

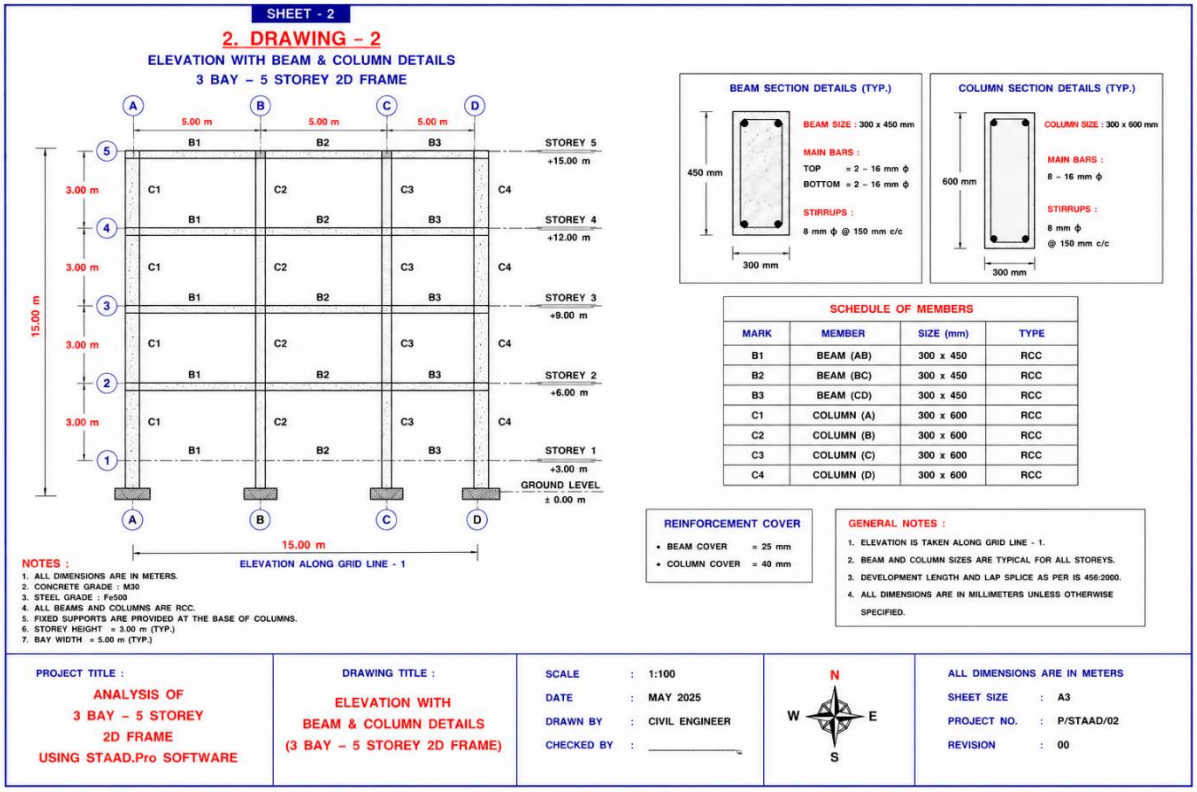
- Grid Layout
- Three Bays @ 5.0 m
- Five Storeys
- Overall Dimensions
- Beam Layout
- Column Layout
- Member Labels
- Grid Labels
- Notes
- Legend



Elevation with Beam & Column Details

Contents

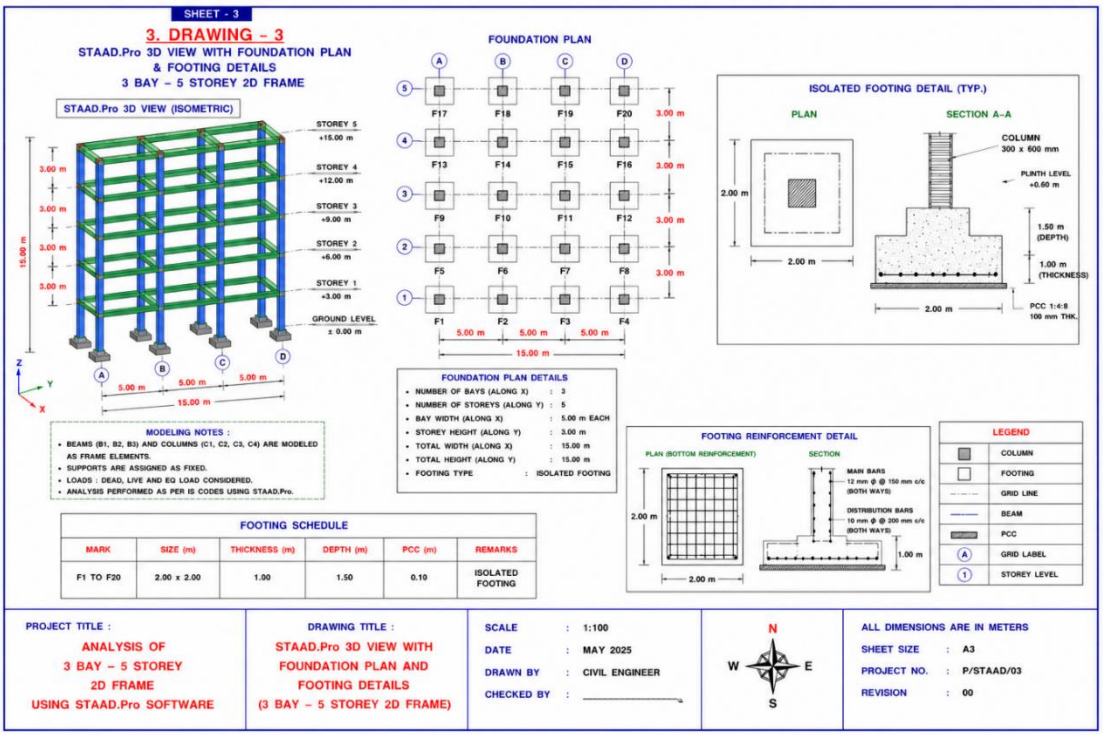
- Complete Elevation
- Storey Heights
- Beam Section
- Column Section
- Reinforcement Details
- Member Schedule
- Dimensions
- Foundation Level
- Notes



STAAD.Pro 3D View with Foundation Plan & Footing Detail

Contents

- STAAD.Pro 3D Model
- Foundation Plan
- Isolated Footing Layout
- Footing Detail (Section)
- Footing Schedule
- Column Locations
- Notes
- Legend



Applications

- Residential Buildings
- Commercial Buildings
- Office Structures
- Educational Buildings
- RCC Framed Structures

Conclusion

The **3 Bay – 5 Storey 2D frame** was successfully analyzed and designed using STAAD.Pro software. The structure was evaluated under dead, live, and seismic load combinations, and the analysis results showed that the beams, columns, and supports satisfy the strength and serviceability requirements of the relevant Indian Standard codes. The generated bending moments, shear forces, axial forces, support reactions, and deflections remained within permissible limits. The study demonstrates that STAAD.Pro provides accurate structural analysis, reduces manual calculation effort, and supports the efficient design of safe and economical multi-storey reinforced concrete framed structures.

5. Analysis of Simply Supported Beam Using STAAD.Pro Software

Abstract

A simply supported beam is one of the most commonly used structural elements in civil engineering. It transfers loads safely to the supporting members while resisting bending and shear forces. This project presents the structural analysis of a reinforced concrete simply supported beam using **STAAD.Pro** software. The beam model is developed by defining its geometry, material properties, support conditions, and loading. Dead load, self-weight, and concentrated live load are applied according to relevant Indian Standard codes. The analysis provides bending moment, shear force, support reactions, and deflection values. The results confirm the structural safety of the beam and demonstrate the efficiency and accuracy of STAAD.Pro for beam analysis.

Introduction

A simply supported beam is a horizontal structural member supported by a pin support at one end and a roller support at the other. It is widely used in buildings, bridges, industrial structures, and roof systems because of its simple construction and efficient load-carrying capacity. When external loads act on the beam, bending moments, shear forces, and deflections develop along its length. Structural analysis is required to determine these forces and ensure that the beam remains safe under service loads.

Manual calculations are suitable for simple cases but become time-consuming when multiple loading conditions are involved. STAAD.Pro is a widely used structural engineering software that provides accurate analysis and design of structural members. In this project, a simply supported RCC beam is modeled and analyzed using STAAD.Pro to study its structural behaviour, support reactions, bending moments, shear forces, and deflections under standard loading conditions.

Theory of Simply Supported Beam

Theory

A simply supported beam is supported at one end by a **pin support** and at the other end by a **roller support**. These supports allow rotation but prevent vertical movement. The beam primarily resists bending and shear due to applied loads.

Components

- Beam
- Pin Support
- Roller Support
- Point Load / Uniformly Distributed Load (UDL)

Structural Behaviour

- Maximum bending moment occurs near the centre under symmetrical loading.
- Maximum shear force occurs at the supports.
- Deflection is maximum at mid-span.

Advantages

- Simple and economical design
- Easy to construct
- Suitable for buildings and bridges
- Efficient load transfer

Methodology & Software Used

Methodology

1. Define beam geometry.
2. Model the beam in STAAD.Pro.
3. Assign material properties.
4. Define pin and roller supports.

5. Apply self-weight, dead load, and live load.
6. Run structural analysis.
7. Study SFD, BMD, reactions, and deflection.
8. Verify structural safety.

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis
AutoCAD	Beam Layout Drawing
MS Excel	Load Calculations
MS Word	Report Preparation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M25
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Young's Modulus	25 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Beam Length	6.0 m
Beam Size	300 × 450 mm
Support Type	Pin & Roller
Concrete Cover	25 mm

Load Calculations

Dead Load (DL)

Self-weight of beam (calculated automatically by STAAD.Pro)

Beam self-weight:

0.30 × 0.45 × 25 = **3.375 kN/m**

Live Load (LL)

Concentrated Load = **50 kN** at mid-span

Load Combination

- 1.5 (DL + LL)

Analysis & Design Results

Analysis Results

The beam was analyzed using STAAD.Pro.

The following results were obtained:

- Support Reactions
- Shear Force Diagram (SFD)
- Bending Moment Diagram (BMD)
- Beam End Forces
- Maximum Deflection

Design Results

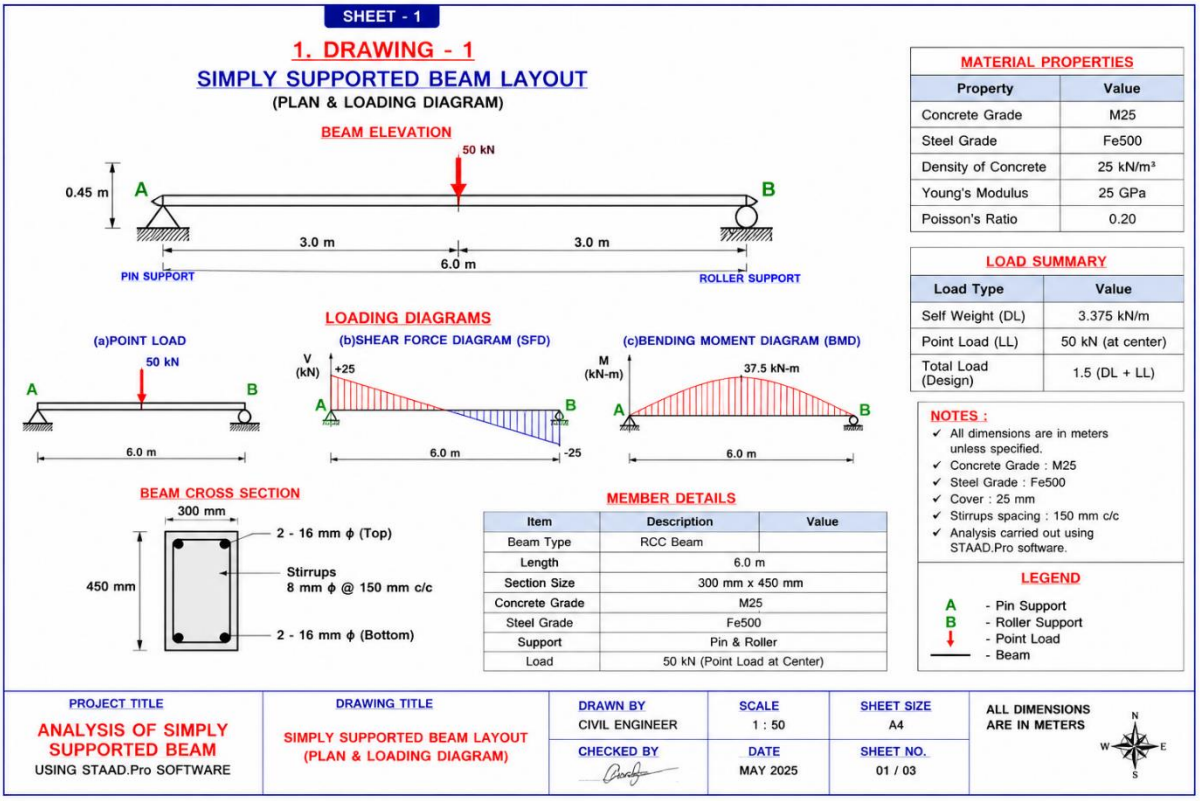
- Beam is safe in bending.
- Beam is safe in shear.

- Maximum bending moment occurs at the beam centre.
- Maximum deflection is within permissible limits.

Beam Layout Drawing

Contents:

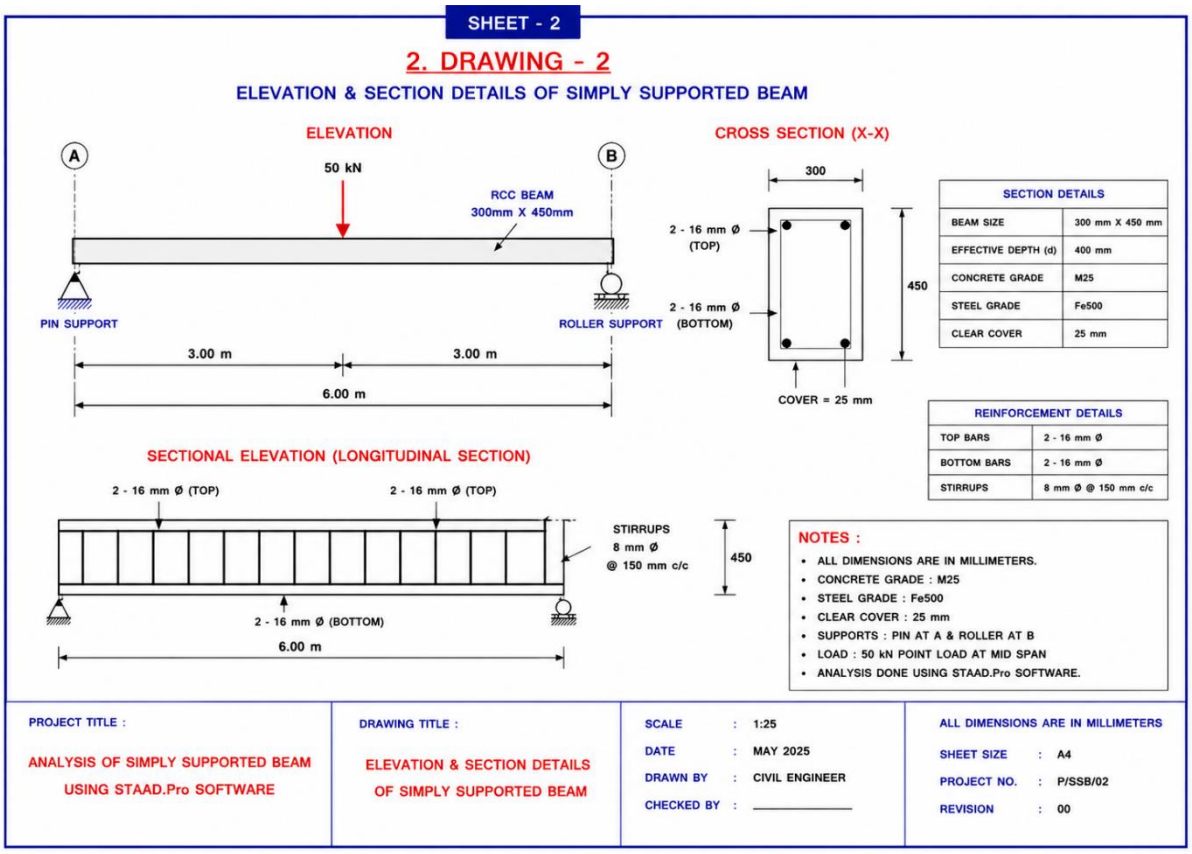
- Beam length = **6.0 m**
- Pin support at left
- Roller support at right
- 50 kN point load at centre
- Dimensions
- Labels
- Notes



Beam Section & Elevation

Contents:

- Beam Elevation
- Cross Section (300 × 450 mm)
- Concrete Cover
- Member Dimensions
- Reinforcement indication (schematic)
- Notes



Contents:

- 3D Beam Model
- Pin & Roller Supports
- Applied Load
- Member Labels

Applications

- Residential Buildings
- RCC Floor Beams
- Roof Beams
- Bridges
- Industrial Structures

Conclusion

The analysis of the simply supported beam was successfully carried out using STAAD.Pro software. The beam was modeled with appropriate material properties, support conditions, and loading. The analysis results showed that the beam safely resists the applied loads, with bending moments, shear forces, and deflections remaining within permissible limits. STAAD.Pro proved to be an efficient and reliable tool for structural analysis, reducing manual calculations while providing accurate and dependable results.

6. Analysis and Design of G+5 Building Plan with Wind Loads Using STAAD.Pro Software

Abstract

The analysis and design of multi-storey buildings subjected to wind loads are essential for ensuring structural safety and serviceability. This project presents the **analysis and design of a G+5 reinforced concrete (RCC) building** using **STAAD.Pro** software by considering the effects of dead load, live load, self-weight, and wind load in accordance with relevant Indian Standard (IS) codes. A three-dimensional structural model is developed by defining beams, columns, slabs, and supports. The software performs structural analysis to determine bending moments, shear forces, axial forces, support reactions, joint displacements, and storey drift. Based on these results, the structural members are designed to satisfy strength and serviceability requirements. The project demonstrates the importance of wind load analysis in multi-storey buildings and highlights the effectiveness of STAAD.Pro in producing accurate, safe, and economical structural designs.

Introduction

The rapid growth of urban development has increased the construction of multi-storey residential and commercial buildings. As the height of a building increases, the influence of **wind loads** becomes more significant and must be considered during structural design. Wind forces can produce lateral displacements, overturning moments, and additional stresses in structural members, making proper analysis essential for the safety and stability of the building.

Modern structural engineering relies on computer-aided software to perform accurate and efficient analysis. **STAAD.Pro** is one of the most widely used structural analysis and design software applications for reinforced concrete and steel structures. It enables engineers to create three-dimensional models, define material properties, assign support conditions, apply different load cases, and evaluate the structural response under various loading conditions. In this project, a **G+5 RCC building** is modeled and analyzed using STAAD.Pro by considering dead load, live load, self-weight, and wind load as per relevant Indian

Standard codes. The analysis results are used to verify the strength and stability of beams, columns, and foundations, ensuring that the structure is safe, economical, and suitable for practical construction.

Theory of G+5 Building with Wind Loads

Theory

A **G+5 RCC building** is a framed structure consisting of a ground floor and five upper floors, where beams and columns are rigidly connected to transfer structural loads safely to the foundation. The building must resist both **vertical loads** (dead load and live load) and **horizontal loads** (wind load). Wind forces increase with building height and can cause lateral displacement, storey drift, and overturning effects. Proper structural analysis ensures that the building remains stable and serviceable under all design load combinations.

Main Components

- RCC Beams
- RCC Columns
- RCC Slabs
- Staircase
- Isolated Footings
- Fixed Supports

Advantages

- Efficient land utilization
- Improved structural stability
- Better resistance to lateral wind forces
- Economical and durable design
- Suitable for residential and commercial buildings

Methodology & Software Used

Methodology

1. Prepare the building plan.
2. Develop the 3D model in STAAD.Pro.
3. Define beam, column, and slab properties.
4. Assign material properties.
5. Define support conditions.
6. Apply dead load, live load, self-weight, and wind load.
7. Generate load combinations.
8. Perform structural analysis.
9. Design structural members.
10. Verify structural safety and prepare the report.

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Structural Drawings
MS Excel	Load Calculations
MS Word	Project Documentation

Material Properties & Structural Geometry

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Elastic Modulus	27 GPa

Property	Value
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Floors	G+5
Total Height	18.0 m
Bay Size	5.0 m × 5.0 m
Building Size	15.0 m × 15.0 m
Beam Size	300 × 450 mm
Column Size	300 × 600 mm
Slab Thickness	150 mm
Support Type	Fixed

Load Calculations

Dead Load (DL)

- Self-weight (Automatically calculated by STAAD.Pro)
- Floor finish = **1.0 kN/m²**
- Wall load = **12 kN/m**

Live Load (LL)

As per **IS 875 (Part 2)**

- Residential Live Load = **2.0 kN/m²**

Wind Load (WL)

As per **IS 875 (Part 3):2015**

Wind load is calculated based on:

- Basic wind speed
- Terrain category
- Building height
- Risk coefficient
- Topography factor

Load Combinations

- 1.5(DL + LL)
- 1.2(DL + LL + WL)
- 1.5(DL + WL)

Analysis & Design Results

Analysis Results

The G+5 building was analyzed using STAAD.Pro, and the following outputs were obtained:

- Joint Displacements
- Support Reactions
- Axial Forces
- Shear Force Diagram (SFD)
- Bending Moment Diagram (BMD)
- Storey Drift
- Deflected Shape

Design Results

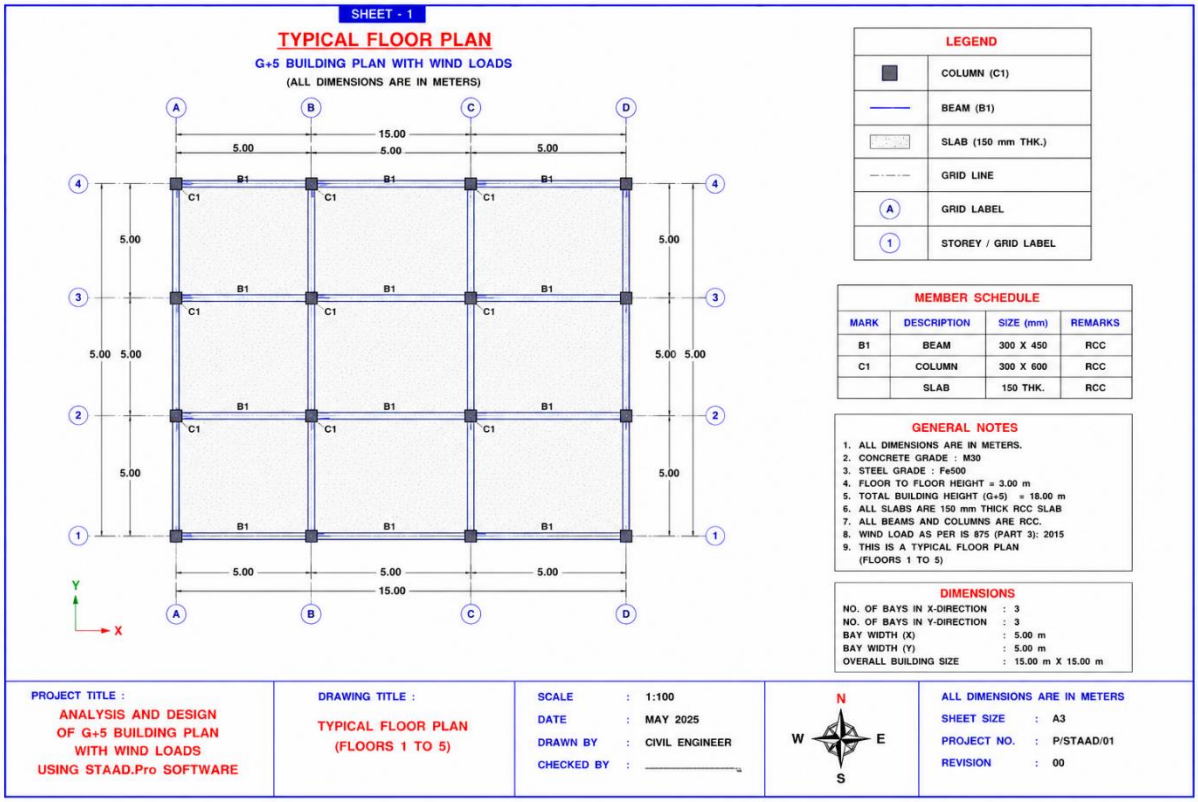
- Beams satisfy bending and shear requirements.
- Columns safely resist axial and lateral loads.

- Storey drift is within permissible limits.
- Wind load effects are within acceptable design limits.
- The overall structure is safe and stable.

Typical Floor Plan

Contents:

- 4 × 4 Column Grid
- Beam Layout
- Grid Labels (A–D and 1–4)
- Bay Dimensions: 5.0 m × 5.0 m
- Overall Building Size: 15.0 m × 15.0 m
- Dimension Lines
- Legend
- General Notes



Drawing Sheet 2 (A3)

Elevation (X-Direction)

Contents:

- G+5 Building Elevation
- Floor Height = 3.0 m
- Total Height = 18.0 m
- Beam Section: 300 × 450 mm
- Column Section: 300 × 600 mm
- Foundation Level
- Dimensions
- Beam & Column Details
- Notes

Drawing Sheet 3 (A3)

STAAD.Pro 3D View with Foundation Plan & Footing Detail

Contents:

- 3D STAAD.Pro Structural Model (illustrative view)
- Foundation Plan
- Isolated Footing Layout
- Footing Size: 2.0 m × 2.0 m × 1.5 m
- Column Locations
- Footing Detail (Section)
- Footing Schedule
- Legend and Notes

Applications

- Residential Apartment Buildings
- Office Buildings

- Commercial Complexes
- Educational Institutions
- Multi-storey RCC Structures

Conclusion

The **G+5 RCC building** was successfully analyzed and designed using STAAD.Pro by considering wind loads along with dead and live loads. The analysis showed that the beams, columns, and foundations satisfy the required strength, stability, and serviceability criteria. The wind load analysis confirmed that lateral displacement and storey drift remain within permissible limits specified by the relevant Indian Standard codes. The study demonstrates that STAAD.Pro is an efficient and reliable software for the structural analysis and design of multi-storey buildings subjected to wind loading.

7. Analysis and Design of G+5 Building Plan with Seismic Loads Using STAAD.Pro Software

Abstract

The analysis and design of multi-storey buildings subjected to seismic loads are essential for ensuring structural safety, stability, and serviceability in earthquake-prone regions. This project presents the structural analysis and design of a **Ground Plus Five (G+5) Reinforced Cement Concrete (RCC) building** using **STAAD.Pro CONNECT Edition** by considering dead load, live load, self-weight, and earthquake loads in accordance with **IS 456:2000**, **IS 875 (Part 1 & Part 2)**, and **IS 1893 (Part 1):2016**. A three-dimensional structural model is developed by defining beams, columns, slabs, supports, and material properties. The software performs structural analysis to determine bending moments, shear forces, axial forces, support reactions, joint displacements, base shear, storey drift, and deflected shapes under various load combinations. Based on the analytical results, the structural members are designed to satisfy strength, stability, and serviceability requirements. The study demonstrates the significance of seismic analysis in improving the earthquake resistance of multi-storey RCC buildings and highlights the effectiveness of STAAD.Pro in producing safe, reliable, and economical structural designs for practical engineering applications.

Introduction

The increasing demand for multi-storey buildings due to rapid urbanization has made structural safety a major concern, especially in earthquake-prone regions. Earthquakes generate lateral forces that can cause significant damage to buildings if they are not properly designed. Therefore, seismic analysis has become an essential part of structural engineering to ensure the stability, strength, and serviceability of reinforced concrete (RCC) structures. Modern structural analysis software such as **STAAD.Pro CONNECT Edition** enables engineers to model, analyze, and design buildings accurately while complying with relevant Indian Standard codes. In this project, a **G+5 RCC building** is analyzed and designed by considering **dead load, live load, self-weight, and seismic loads** as per **IS 456:2000**, **IS 875 (Part 1 & Part 2)**, and **IS 1893 (Part 1):2016**. The software evaluates the structural response under various load combinations and generates design results for beams, columns, and foundations. The project aims to demonstrate the importance of earthquake-resistant design and the effective application of STAAD.Pro in developing safe, stable, and economical multi-storey buildings.

Theory of G+5 Building with Seismic Loads

Including:

- Introduction to RCC framed structures
- Seismic loading theory
- Structural behavior during earthquakes
- Importance of ductility
- Load transfer mechanism
- Building components:
  - RCC Beams
  - RCC Columns
  - RCC Slabs
  - Staircase
  - Foundation
  - Fixed Supports
- Advantages of seismic-resistant buildings

Methodology

- Planning the building layout
- Creating the AutoCAD plan
- Developing the STAAD.Pro model
- Assigning material properties
- Defining supports
- Applying loads
- Seismic load generation
- Creating load combinations
- Running analysis
- Designing structural members
- Checking results
- Preparing the report

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Structural Drawings
MS Excel	Load Calculations
MS Word	Report Preparation

Material Properties

Property	Value
Concrete Grade	M30
Steel Grade	Fe500
Density of Concrete	25 kN/m^3
Elastic Modulus	27 GPa
Poisson's Ratio	0.20

Structural Geometry

Parameter	Value
Floors	G+5
Floor Height	3.0 m
Total Height	18 m
Building Size	15 m × 15 m
Bay Size	5 m × 5 m
Beam Size	300 × 450 mm
Column Size	300 × 600 mm
Slab Thickness	150 mm
Support Type	Fixed

Load Calculations

Dead Load (DL)

- Self-weight
- Floor Finish
- Wall Load

Live Load (LL)

As per **IS 875 (Part 2)**

Seismic Load (EQ)

As per **IS 1893 (Part 1):2016**

Parameters:

- Seismic Zone
- Zone Factor
- Importance Factor
- Response Reduction Factor
- Soil Type
- Damping Ratio

Load Combinations

- 1.5(DL + LL)
- 1.2(DL + LL + EQX)
- 1.2(DL + LL − EQX)
- 1.2(DL + LL + EQZ)
- 1.2(DL + LL − EQZ)
- 1.5(DL + EQ)
- 0.9DL ± 1.5EQ

Analysis & Design Results

Results obtained from STAAD.Pro:

- Joint Displacements
- Support Reactions
- Axial Forces
- Shear Force Diagram
- Bending Moment Diagram
- Storey Drift
- Base Shear
- Deflected Shape

Design Results

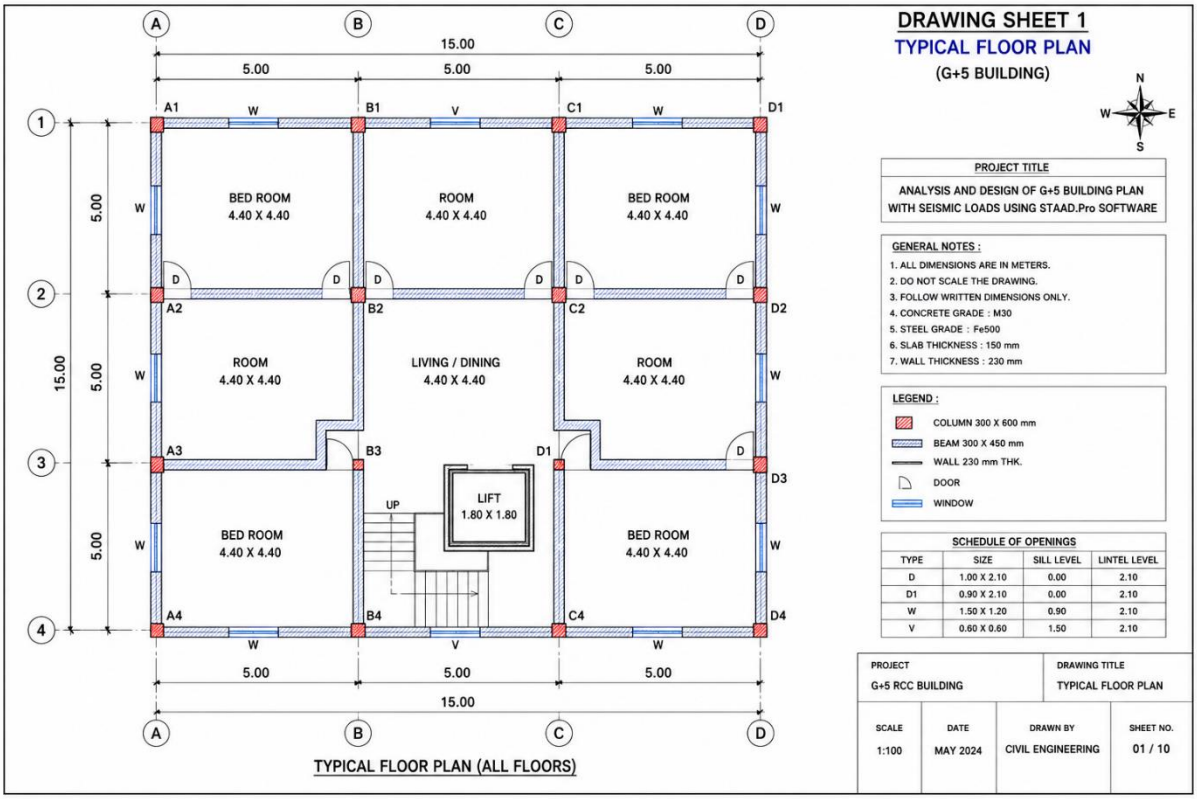
- Beam Design
- Column Design
- Foundation Design
- Drift Check
- Stability Check

Typical Floor Plan

Including:

- 4 × 4 Grid

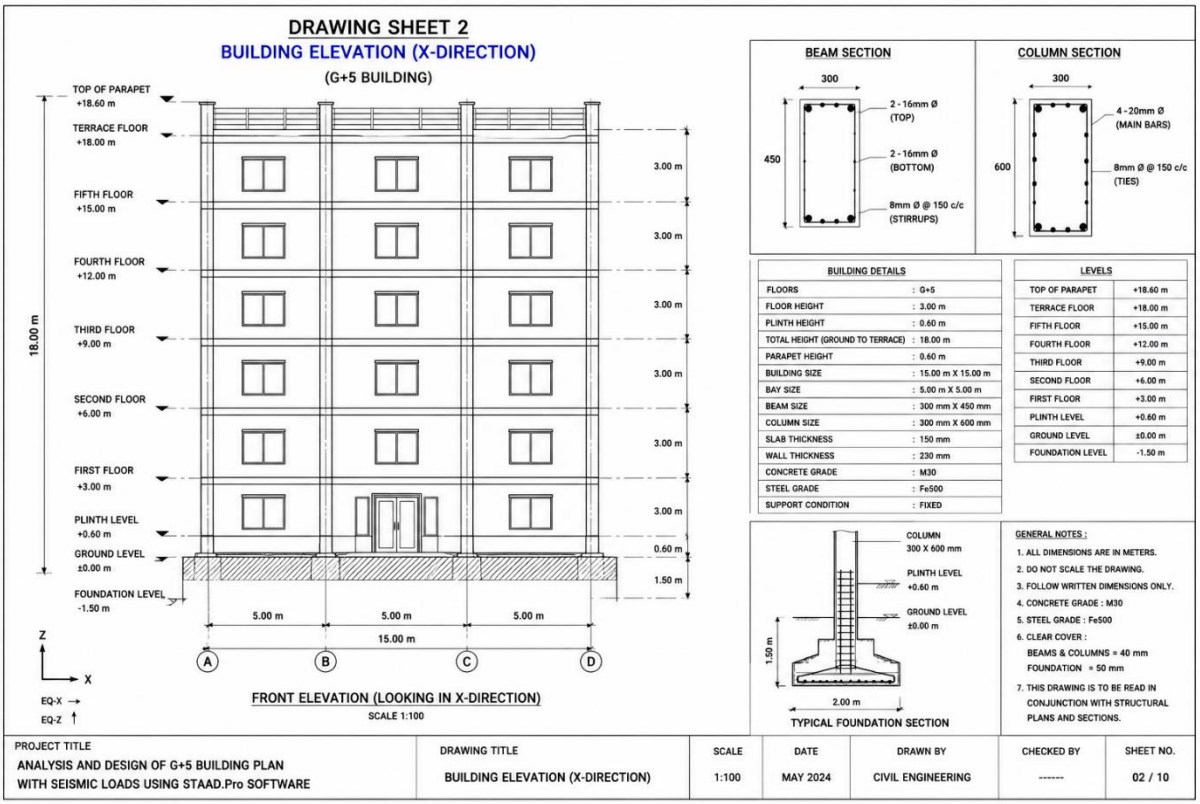
- Beam Layout
- Column Layout
- Room Layout
- Dimensions
- Notes



Building Elevation

Including:

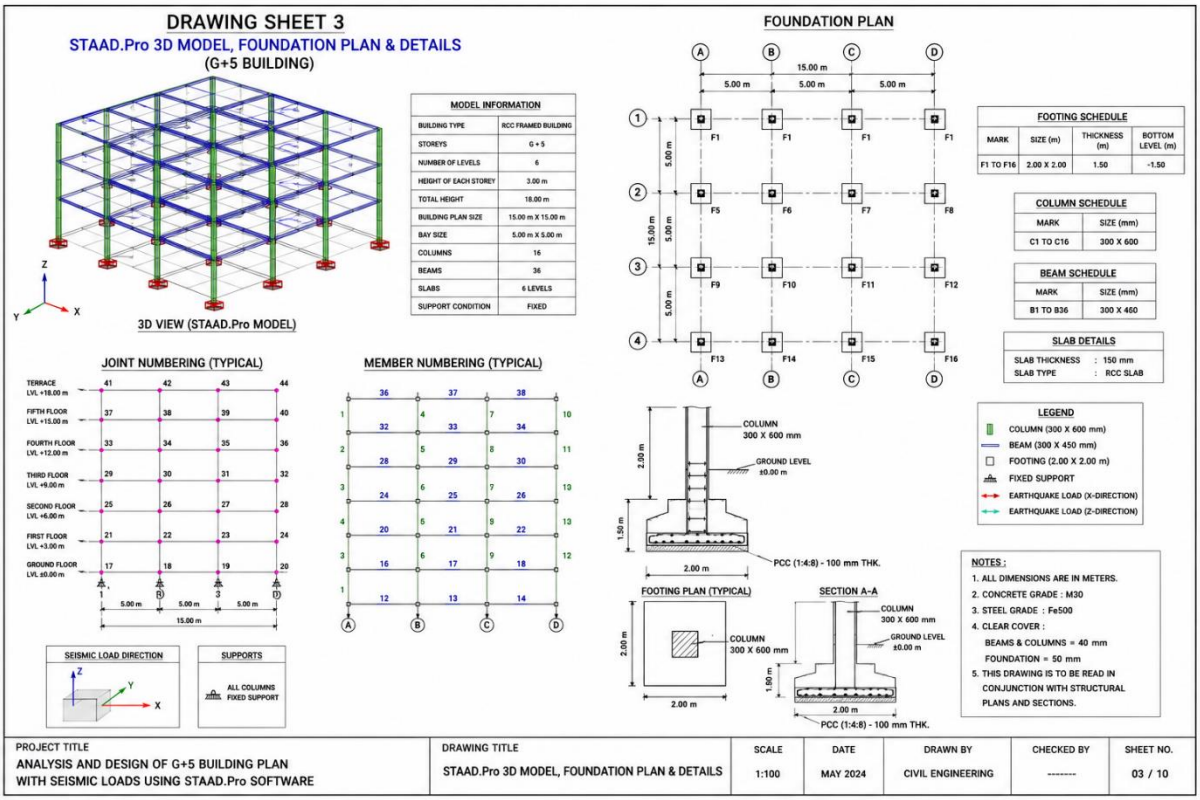
- G+5 Elevation
- Beam Details
- Column Details
- Floor Heights
- Overall Height
- Dimensions
- Notes



Drawing Sheet 3 (A3)

Including:

- STAAD.Pro 3D Model
- Foundation Plan
- Isolated Footing Layout
- Footing Section
- Column Schedule
- Legend



Applications

- Residential Apartments
- Commercial Buildings
- Office Buildings
- Hospitals
- Educational Institutions
- Earthquake-Prone Areas
- Multi-storey RCC Structures

Conclusion

A professional conclusion explaining that the G+5 RCC building was successfully analyzed and designed using STAAD.Pro under seismic loading. It will summarize the structural performance, seismic safety, code compliance, and the effectiveness of STAAD.Pro in achieving an economical and reliable design.

8. Analysis of 2D Steel Truss Using STAAD.Pro Software

Abstract

The **Analysis of 2D Steel Truss Using STAAD.Pro Software** is an important study in structural engineering for evaluating the behavior of steel truss systems under various loading conditions. This project presents the modeling, analysis, and design of a two-dimensional steel truss using **STAAD.Pro CONNECT Edition** by considering dead load, live load, and self-weight in accordance with relevant Indian Standard codes, including **IS 800:2007** and **IS 875 (Part 1 & Part 2)**. The software is used to determine member forces, joint displacements, support reactions, and deflected shapes. Based on the analysis results, the structural members are verified for strength, stability, and serviceability. The study demonstrates that STAAD.Pro provides an accurate, efficient, and reliable method for analyzing steel truss structures used in industrial, commercial, and infrastructure applications.

Introduction

Steel trusses are widely used in modern construction because of their high strength, lightweight nature, and ability to span long distances economically. They are commonly employed in industrial buildings, warehouses, factories, bridges, railway stations, aircraft hangars, and roof structures where large column-free spaces are required. A **2D steel truss** consists of straight members connected at joints, with loads applied at the joints so that members primarily experience axial tension or compression. Proper structural analysis is essential to ensure the safety, stability, and serviceability of the truss under different loading conditions.

In this project, a **2D steel truss** is modeled and analyzed using **STAAD.Pro CONNECT Edition** by considering **self-weight, dead load, and live load** in accordance with **IS 800:2007** and **IS 875 (Part 1 & Part 2)**. The software evaluates member axial forces, joint displacements, support reactions, and the overall structural behavior under various load combinations. The analysis results are used to verify the strength and stability of the truss members, ensuring a safe and economical design. This project demonstrates the effectiveness of STAAD.Pro in accurately analyzing steel truss structures and its importance as a reliable tool for practical structural engineering applications.

Theory of 2D Steel Truss

This chapter will contain detailed explanations of:

Introduction to Steel Truss

- Definition and purpose
- Historical development
- Common applications

2D Steel Truss

- What is a 2D truss?
- Difference between 2D and 3D trusses
- Structural behaviour

Load Transfer Mechanism

- Tension members
- Compression members
- Force transfer to supports

Components of a Steel Truss

- Top chord
- Bottom chord
- Vertical members
- Diagonal members
- Gusset plates
- Supports

Types of Steel Trusses

- Pratt Truss
- Howe Truss
- Warren Truss
- Fink Truss
- King Post Truss

Advantages of Steel Trusses

- High strength-to-weight ratio
- Long span capability
- Easy fabrication and erection
- Durability
- Cost effectiveness

Methodology

Each step will be explained:

- Planning the truss geometry
- Creating the truss layout
- Developing the STAAD.Pro model
- Assigning material properties
- Defining supports
- Applying loads
- Creating load combinations
- Running analysis
- Designing members
- Verifying safety
- Preparing the report

Software Used

Software	Purpose
STAAD.Pro CONNECT Edition	Structural Analysis & Design
AutoCAD	Truss Drawings
MS Excel	Load Calculations
MS Word	Report Preparation

Material Properties

Property	Value
Steel Grade	Fe410 (or Fe250, depending on design assumptions)
Yield Strength	250 MPa
Density	78.5 kN/m^3
Elastic Modulus	200 GPa
Poisson's Ratio	0.30

Each property will include a brief engineering explanation.

Structural Geometry

Parameter	Value
Truss Type	Pratt Truss (or another selected type)
Span	20 m
Height	4 m
Panel Length	2.5 m
Number of Panels	8
Support Type	Pin and Roller

Dead Load (DL)

- Self-weight of truss members
- Roofing sheet load (if applicable)
- Purlin load

Live Load (LL)

- Roof maintenance load
- Imposed load as per **IS 875 (Part 2)**

Wind Load (if applicable)

- Basic wind speed
- Pressure coefficients
- Wind load distribution

Load Combinations

Typical combinations based on the relevant design code.

Analysis & Design Results

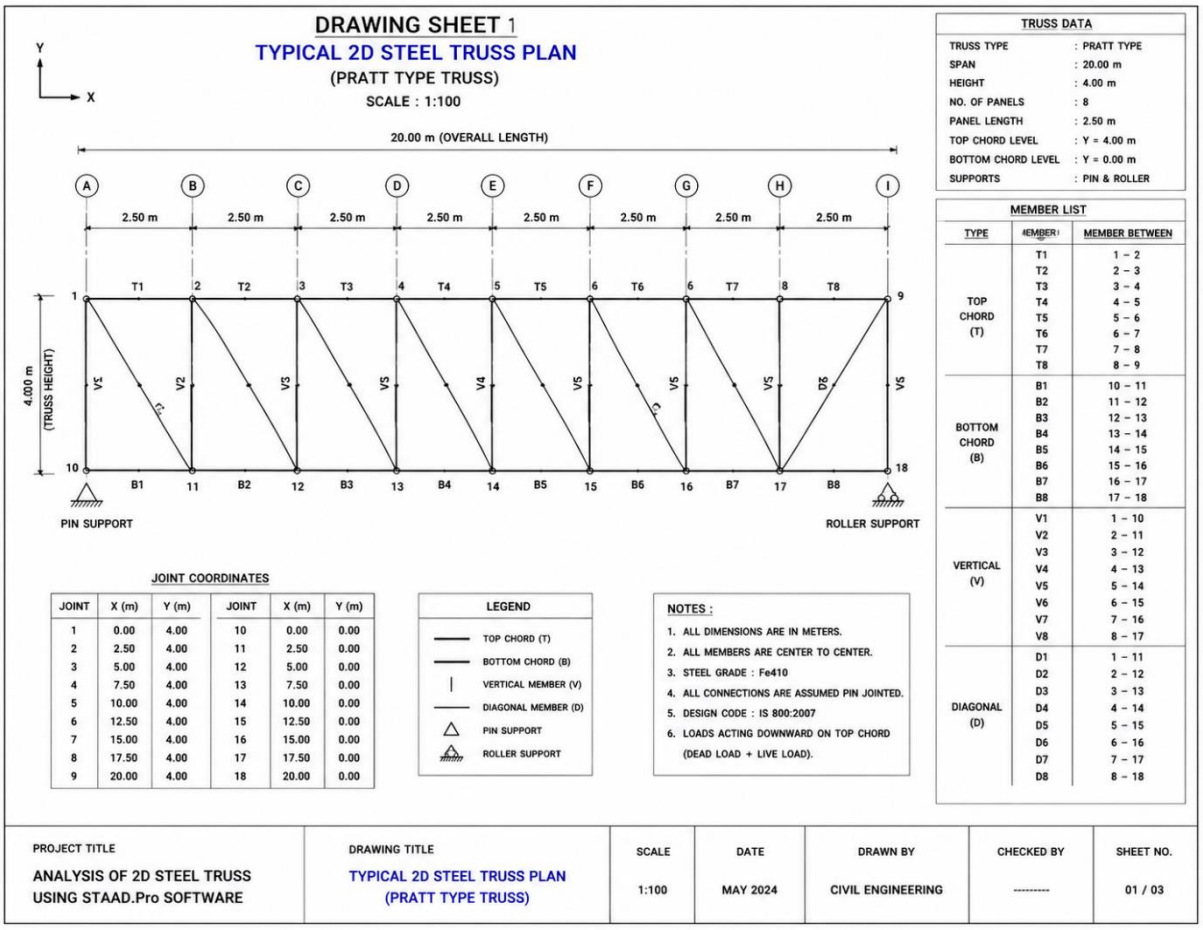
STAAD.Pro results will be discussed, including:

- Joint Displacements
- Support Reactions
- Member Axial Forces
- Tension and Compression Forces
- Deflected Shape
- Member Utilization
- Structural Stability

Typical 2D Steel Truss Plan

Includes:

- Truss layout
- Joint numbering
- Member numbering
- Overall dimensions
- Span and panel lengths
- Notes

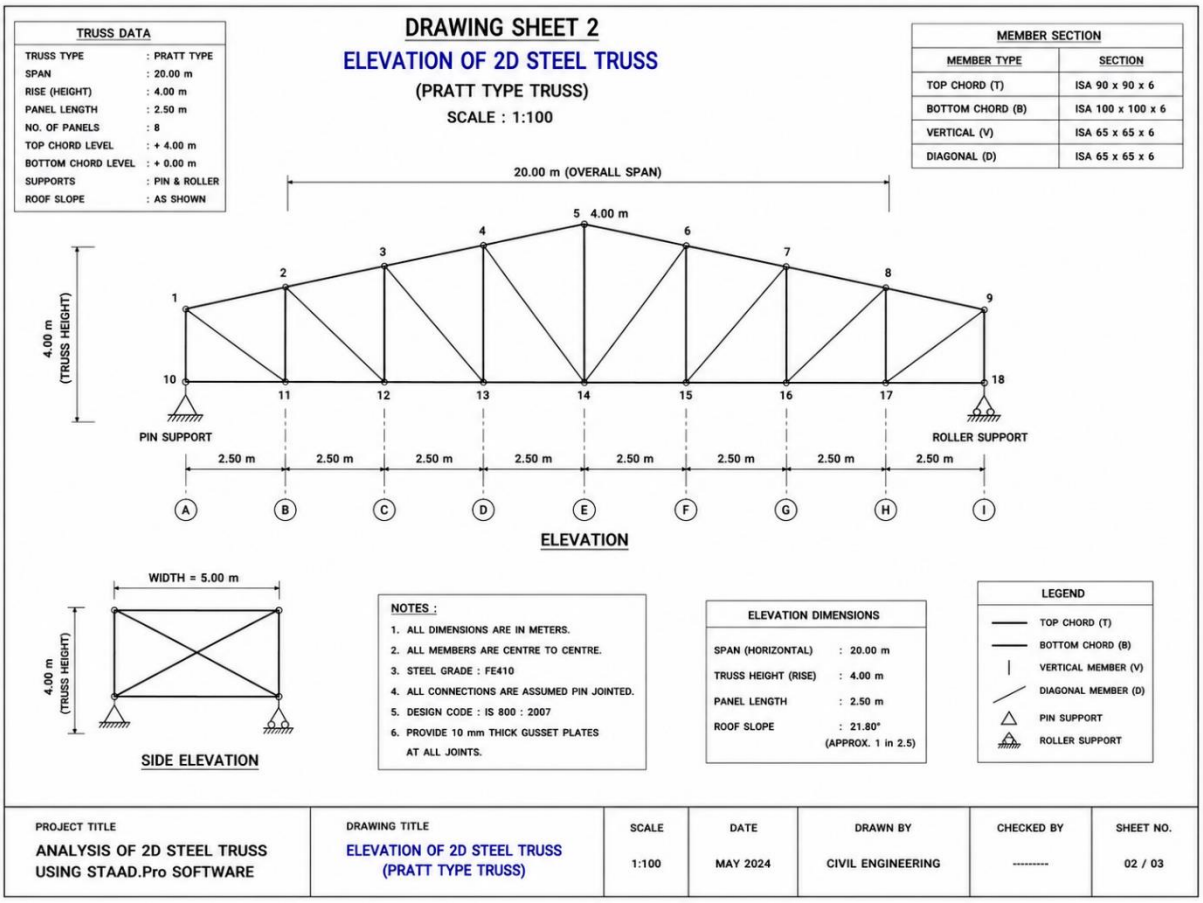


Elevation of Steel Truss

Includes:

- Front elevation
- Roof slope
- Truss height
- Member arrangement
- Support details
- Dimensions

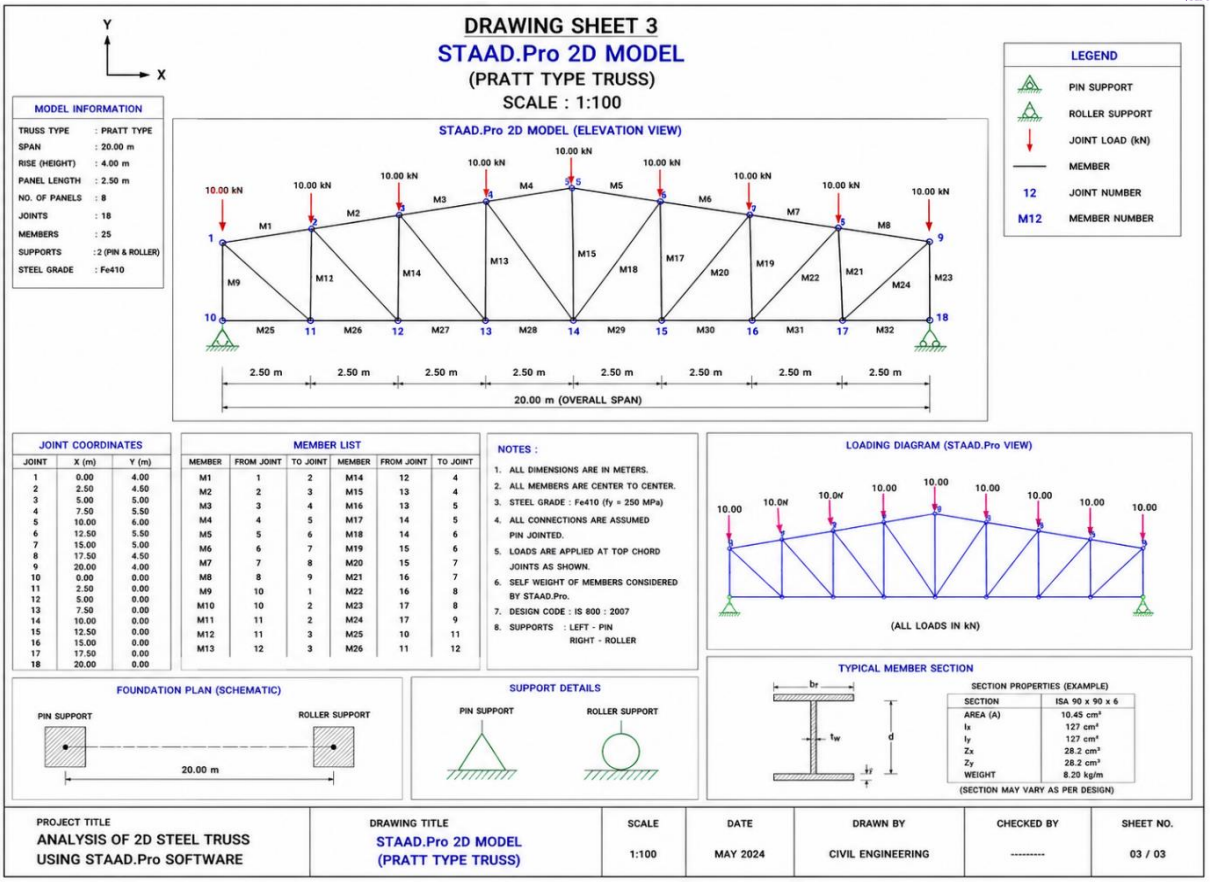
- Notes



STAAD.Pro 2D Model

Includes:

- STAAD.Pro wireframe model
- Node numbers
- Member labels
- Pin and roller supports
- Applied load directions
- Legend



Applications

This section will explain where 2D steel trusses are commonly used, such as:

- Industrial sheds
- Warehouses
- Factory buildings
- Aircraft hangars
- Railway station roofs
- Auditorium roofs
- Agricultural storage structures

Conclusion

A detailed conclusion summarizing the structural performance of the 2D steel truss, the accuracy and usefulness of STAAD.Pro analysis, and the suitability of steel trusses for long-span roofing systems.

9. Project Title: Analysis of 3D Steel Truss Using STAAD.Pro

ABSTRACT

A steel truss is an important structural system widely used in industrial buildings, railway stations, warehouses, bridges, and large-span structures due to its high strength and lightweight characteristics. This project focuses on the analysis of a three-dimensional steel truss structure using STAAD.Pro software. The structural model is created by defining nodes, members, material properties, supports, and loading conditions. Dead load, live load, and wind load are applied according to relevant Indian Standard codes. The structure is analysed to determine member forces, support reactions, and displacements. The obtained results help in understanding the structural behaviour and safety of the steel truss system.

INTRODUCTION

Steel trusses are widely used in modern construction because they can cover large spans with minimum material consumption. A truss is a structural framework consisting of interconnected members arranged in triangular patterns. The members mainly resist axial forces such as tension and compression, which makes the structure economical and efficient.

The analysis of a steel truss is necessary to ensure stability, strength, and serviceability under different loading conditions. Manual analysis of complex three-dimensional structures is time-consuming; therefore, structural analysis software such as STAAD.Pro is used for accurate and efficient design.

In this project, a 3D steel truss model is developed and analysed using STAAD.Pro. Different loads acting on the structure are considered, and the behaviour of structural members is studied. The analysis results help engineers select suitable member sizes and ensure that the structure satisfies safety requirements according to design codes.

OBJECTIVES OF THE PROJECT

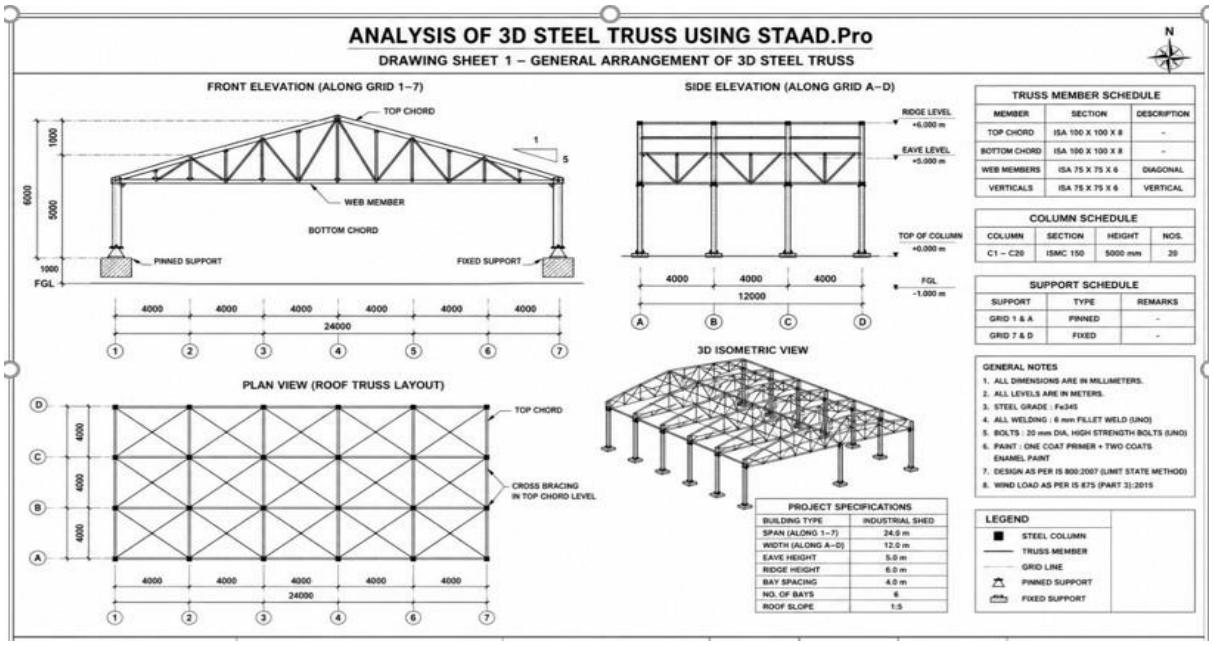
The main objectives of this project are:

- To create a 3D steel truss model using STAAD.Pro.
- To understand the structural behaviour of steel trusses.
- To apply different loading conditions on the structure.
- To determine member forces, bending effects, and displacements.
- To check the adequacy of steel members according to design codes.
- To study the importance of computer-based structural analysis.

DRAWINGS

Drawing 1: 3D Steel Truss Geometry Drawing

This drawing represents the overall arrangement of the steel truss structure. It shows the position of nodes, main truss members, diagonal members, and supporting elements. The triangular arrangement provides stability and allows the structure to transfer loads efficiently.



Purpose:

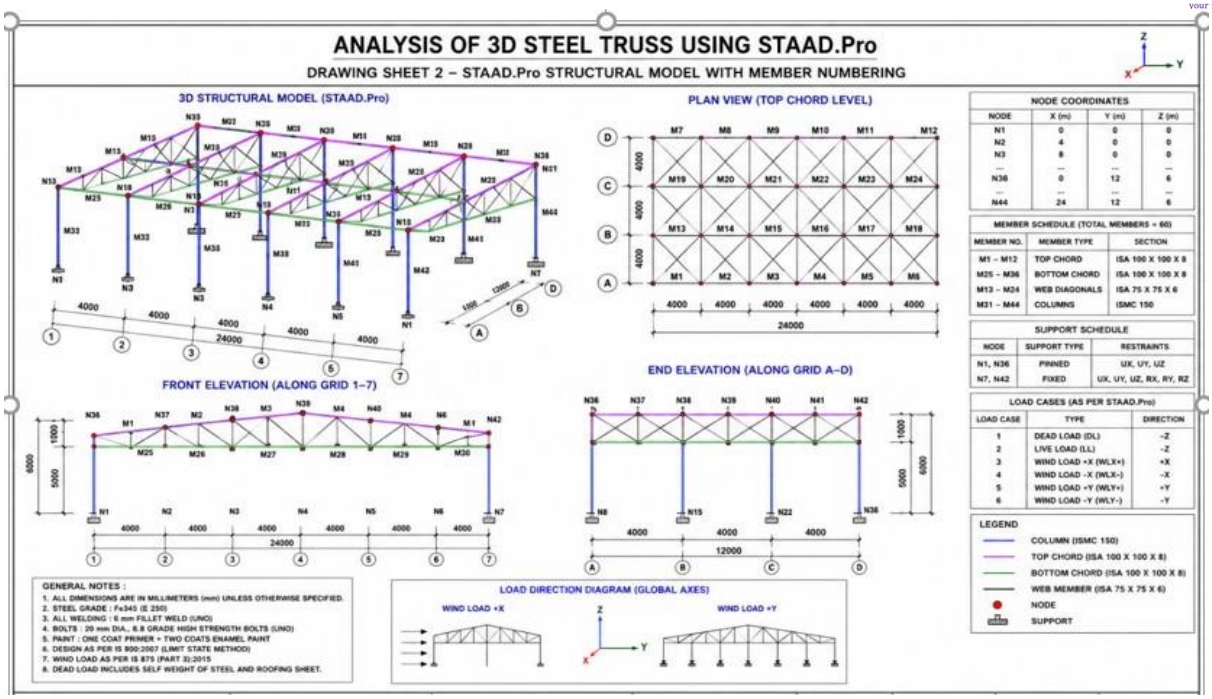
- Shows the structural configuration.
- Helps in understanding member arrangement.
- Provides dimensions and span details.

Drawing 2: STAAD.Pro 3D Model

The STAAD.Pro model represents the digital simulation of the actual steel truss structure. The model consists of joints connected by structural members. Supports are assigned at required locations, and material properties are defined before analysis.

Purpose:

- Represents actual structural behaviour.
- Helps in applying loads.
- Used for analysis and design calculations.



Drawing 3: Loading Diagram

The loading diagram shows different forces acting on the truss structure, such as dead load, live load, and wind load.

Purpose:

- Shows load distribution.
- Helps understand load transfer.
- Used for structural analysis.

METHODOLOGY

The methodology followed for this project includes the following steps:

Collection of Structural Data

The required dimensions, span length, height, and loading conditions are selected based on design requirements.

Structural Modelling

The 3D steel truss is modelled in STAAD.Pro by creating nodes and connecting them with steel members.

Defining Material Properties

Steel properties such as yield strength, modulus of elasticity, density, and Poisson's ratio are assigned.

Applying Loads

Different loads acting on the structure are applied according to IS codes.

Structural Analysis

STAAD.Pro performs analysis to calculate:

- Member forces
- Support reactions
- Displacement
- Stress values

Design Verification

The results are checked according to steel design standards.

SOFTWARE USED

STAAD.Pro

STAAD.Pro is a structural analysis and design software developed for analysing different types of structures. It is widely used by civil and structural engineers for designing buildings, bridges, towers, and industrial structures.

Features of STAAD.Pro:

- 3D structural modelling
- Static and dynamic analysis
- Steel and RCC design

Load application `

- Code-based design checking
- Generation of analysis reports

AutoCAD

AutoCAD is used for preparing structural drawings and detailing of the truss arrangement.

Microsoft Excel

Excel is used for performing preliminary calculations and organizing design data.

THEORY OF 3D STEEL TRUSS

A steel truss is a framework made up of straight steel members connected at joints. The members form triangular shapes to provide stability.

The main components of a truss are:

Top Chord

The upper horizontal members mainly resist compression forces.

Bottom Chord

The lower members resist tension forces.

Vertical Members

These transfer forces between top and bottom chords.

Diagonal Members

These provide stability and resist tensile and compressive forces.

Structural Behaviour

When loads are applied, the forces are transferred through the truss members to the supports. Ideally, truss members carry only axial forces.

Advantages of steel trusses:

- High strength-to-weight ratio
- Suitable for large spans
- Easy fabrication
- Faster construction

MATERIAL PROPERTIES

The properties of structural steel used in analysis are:

Property	Value
Material	Structural Steel
Yield Strength	250 MPa
Ultimate Strength	410 MPa

Property	Value
Density	7850 kg/m³
Modulus of Elasticity	2 × 10⁵ MPa
Poisson's Ratio	0.3

Steel is preferred because of its high strength, durability, and flexibility.

STRUCTURAL GEOMETRY

Structural geometry defines the dimensions and arrangement of the steel truss.

Example assumptions:

- Type of structure: Roof Steel Truss
- Span length: 20 m
- Height of truss: 3 m
- Number of panels: 8
- Support condition: Hinged and Roller support
- Member type: Steel angles / tubular sections

The geometry is selected to achieve proper load distribution and structural stability.

LOAD CALCULATIONS

Dead Load

Dead load includes the self-weight of structural members and permanent components.

Examples:

- Weight of steel members
- Roofing materials
- Purlins

STAAD.Pro automatically calculates member self-weight after assigning material properties.

Live Load

Live load includes temporary loads acting on the structure.

Live loads are applied according to IS 875 Part 2.

Wind Load

Wind load is the pressure exerted by wind on the structure.

It depends on:

- Wind speed
- Height of structure
- Location
- Shape of structure

Wind load is calculated as per IS 875 Part 3.

Load Combinations

Different load combinations are considered for safe design:

Examples:

- 1.5(DL + LL)
- 1.5(DL + WL)
- 1.2(DL + LL + WL)

DESIGN RESULTS

After analysis in STAAD.Pro, the following results are obtained:

Support Reactions

These show the forces transferred from the structure to the foundation.

Member Forces

The software provides:

- Axial tension forces
- Compression forces
- Shear forces

Displacement Results

Displacement indicates the movement of the structure under applied loads.

Design Check

Steel members are checked for:

- Strength
- Stability
- Buckling resistance
- Allowable stresses

APPLICATIONS

Steel trusses are used in:

- Industrial buildings
- Warehouses
- Aircraft hangars
- Railway stations
- Sports stadiums
- Bridges
- Transmission structures
- Large-span roofs

CONCLUSION

The analysis of a 3D steel truss using STAAD.Pro helps in understanding the behaviour of steel structures under different loading conditions. The software provides accurate results regarding member forces, reactions, and displacements. Steel trusses are economical and efficient structural systems for large-span construction due to their high strength and lightweight properties.

The project demonstrates the importance of computer-aided structural analysis in modern civil engineering. STAAD.Pro reduces calculation time and helps engineers design safe and reliable structures according to standard design codes.

10. Analysis and Design of G+3 RCC Building using STAAD.Pro

ABSTRACT

This project presents the analysis and structural design of a G+3 reinforced cement concrete residential building using STAAD.Pro software. The building is modeled as a three-dimensional frame consisting of beams, columns, slabs, and isolated footings. The structure is analyzed under dead load, live load, wind load, and seismic load according to Indian Standard codes. STAAD.Pro is used to calculate bending moments, shear forces, axial forces, support reactions, and nodal displacements. Structural members are designed based on strength and serviceability criteria. AutoCAD is used for preparing structural drawings, including the floor plan and reinforcement layout. The project demonstrates how structural engineering software helps produce safe, economical, and efficient building designs while reducing manual calculations and improving design accuracy.

INTRODUCTION

Reinforced Cement Concrete (RCC) buildings are widely used in residential, commercial, and institutional construction due to their strength, durability, and economy. Modern structural design requires accurate analysis to ensure the building safely resists gravity and lateral loads.

STAAD.Pro is a structural analysis and design software that performs comprehensive analysis for multi-storey buildings. It enables engineers to model the structure in three dimensions, assign material properties, define support conditions, apply loads, and obtain design results in accordance with Indian Standards.

This project involves the analysis and design of a G+3 RCC building. The structural model includes columns, beams, slabs, and foundations. Load calculations are carried out according to IS codes, and the structure is analyzed to ensure safety, stability, and serviceability.

OBJECTIVES

- To model a G+3 RCC building using STAAD.Pro.
- To calculate dead, live, wind, and seismic loads.
- To analyze the structure under various loading conditions.
- To design beams and columns according to IS 456:2000.
- To obtain bending moment, shear force, and deflection diagrams.
- To prepare professional structural drawings using AutoCAD.
- To develop economical and safe structural members.

METHODOLOGY

1. Study architectural plan.
2. Fix column locations.
3. Model beams and columns in STAAD.Pro.
4. Define concrete material properties.
5. Assign support conditions.
6. Apply dead loads.
7. Apply live loads.
8. Apply wind loads.
9. Apply earthquake loads.
10. Create load combinations.
11. Perform structural analysis.
12. Design RCC members.
13. Check beam and column adequacy.
14. Prepare AutoCAD drawings.
15. Finalize structural report.

SOFTWARE USED

Primary Software

- STAAD.Pro CONNECT Edition

Supporting Software

- AutoCAD 2024

Features of STAAD.Pro

- 3D Structural Modeling
- Load Definition
- Load Combinations
- RCC Design
- Steel Design
- Foundation Design
- Seismic Analysis
- Wind Analysis
- Design Reports
- Quantity Estimation

BUILDING DETAILS

Item	Value
Building Type	Residential
Floors	G+3
Floor Height	3.2 m
Total Height	12.8 m
Concrete Grade	M25
Steel Grade	Fe500
Slab Thickness	150 mm
Beam Size	230 × 450 mm
Column Size	300 × 450 mm
Footing	Isolated Footing

LOAD CALCULATIONS

Dead Load

Self-weight of beams, columns, slabs, and walls.

Example:

- RCC Density = 25 kN/m³
- Brick Wall = 20 kN/m³

Wall Load

= 0.23 × 3 × 20

= **13.8 kN/m**

Slab Load

150 mm Slab

= 0.15 × 25

= **3.75 kN/m²**

Floor Finish

= **1.0 kN/m²**

Live Load

Residential Building

= **3.0 kN/m²**

Wind Load

Calculated according to

IS 875 Part-3

Seismic Load

Calculated according to

IS 1893 (Part-1):2016

Zone III

Importance Factor = 1.0

Response Reduction Factor = 5

ANALYSIS RESULTS

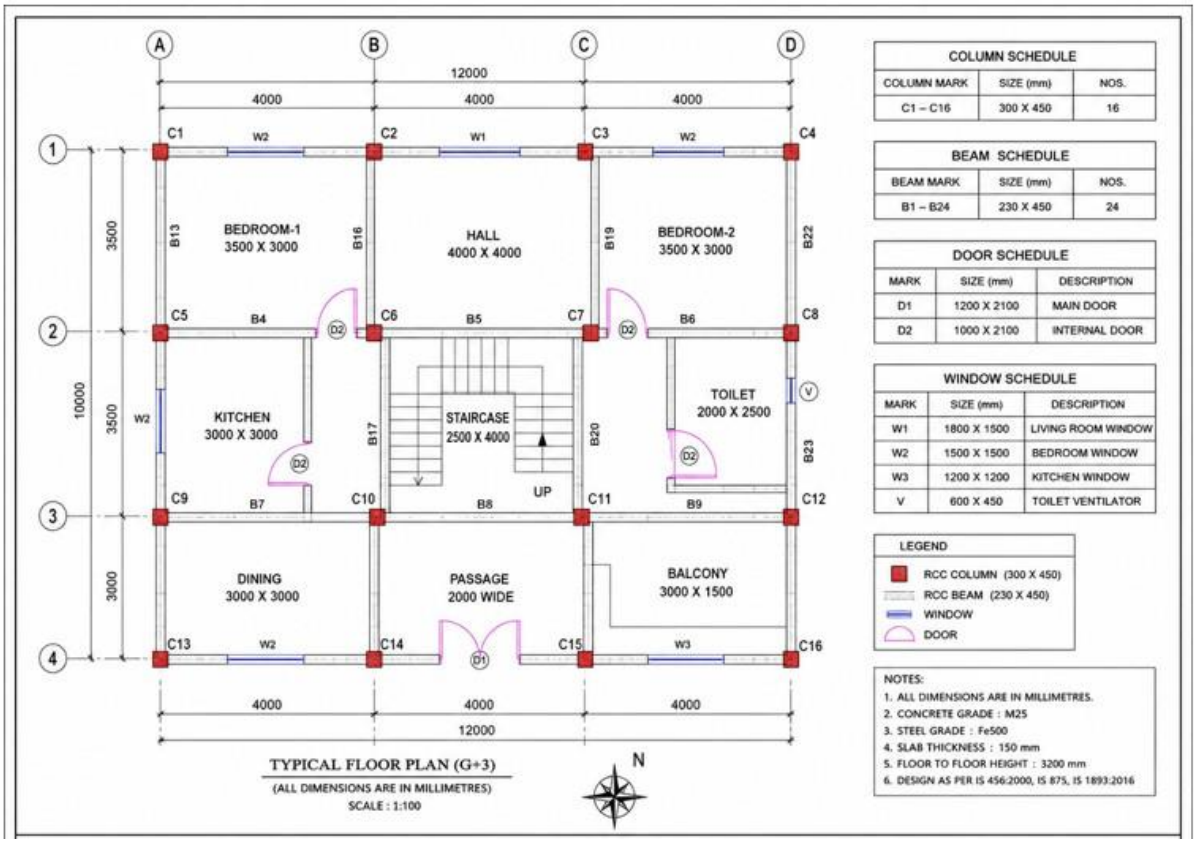
STAAD.Pro provides:

- Bending Moment Diagram
- Shear Force Diagram
- Axial Force Diagram
- Node Displacement
- Beam Deflection
- Column Forces
- Support Reactions
- Reinforcement Details
- Design Summary

DRAWING SHEETS

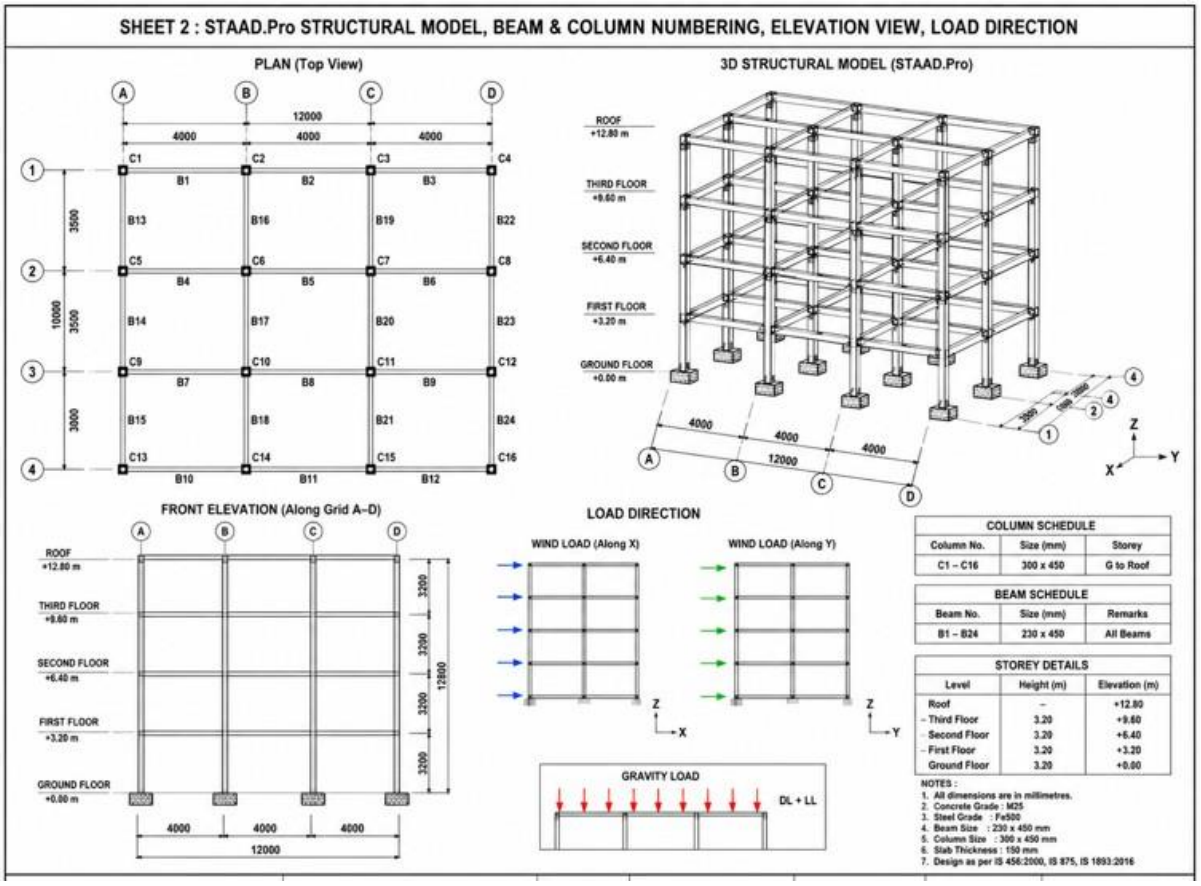
Sheet 1 (AutoCAD)

- Typical Floor Plan
- Beam Layout
- Column Layout
- Dimensions
- Grid Lines
- Room Labels



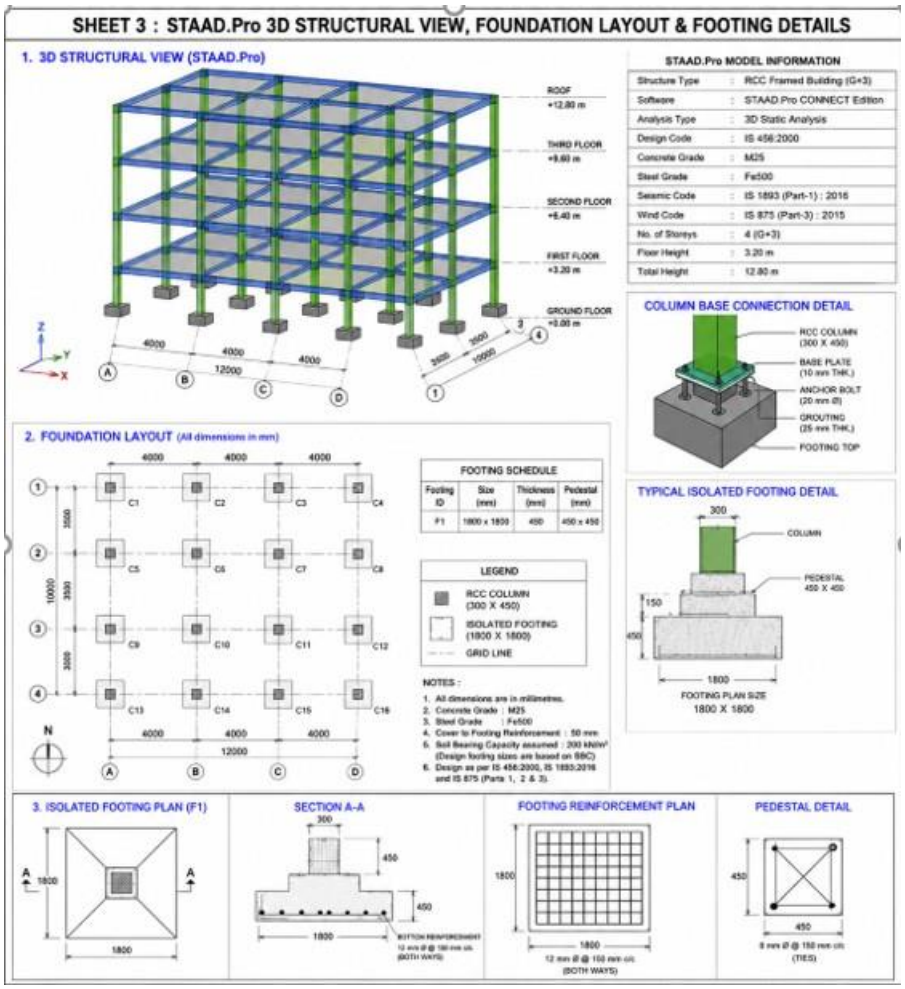
Sheet 2 (STAAD.Pro)

- Structural Model
- Beam Numbering
- Column Numbering
- Elevation View
- Load Direction



Sheet 3 (STAAD.Pro)

- 3D Structural View
- Foundation Layout
- Isolated Footing Details
- Column Base Connections
- Professional Title Block



ADVANTAGES

- Faster structural analysis
- High design accuracy
- Automatic IS code checking
- Easy load combinations
- Economical member sizing
- Professional structural drawings
- Time-saving design process

CONCLUSION

The G+3 RCC building was successfully modeled, analyzed, and designed using STAAD.Pro. The structure satisfies the requirements of Indian Standard codes under dead, live, wind, and seismic loading. The software effectively determined member forces, deflections, and reinforcement requirements, enabling a safe and economical structural design. AutoCAD drawings complement the analysis by providing clear construction details, making the project suitable as a complete B.Tech structural engineering submission.