

UNIT 1

INTRODUCTION TO STAAD.Pro

Get familiar with STAAD.Pro environment, workflow and essential navigation tools.



WHAT YOU WILL LEARN

- Introduction to Structural Analysis
- STAAD.Pro Interface
- Workflow Overview
- File Management
- Coordinate System
- Navigation Tools
- Keyboard Shortcuts



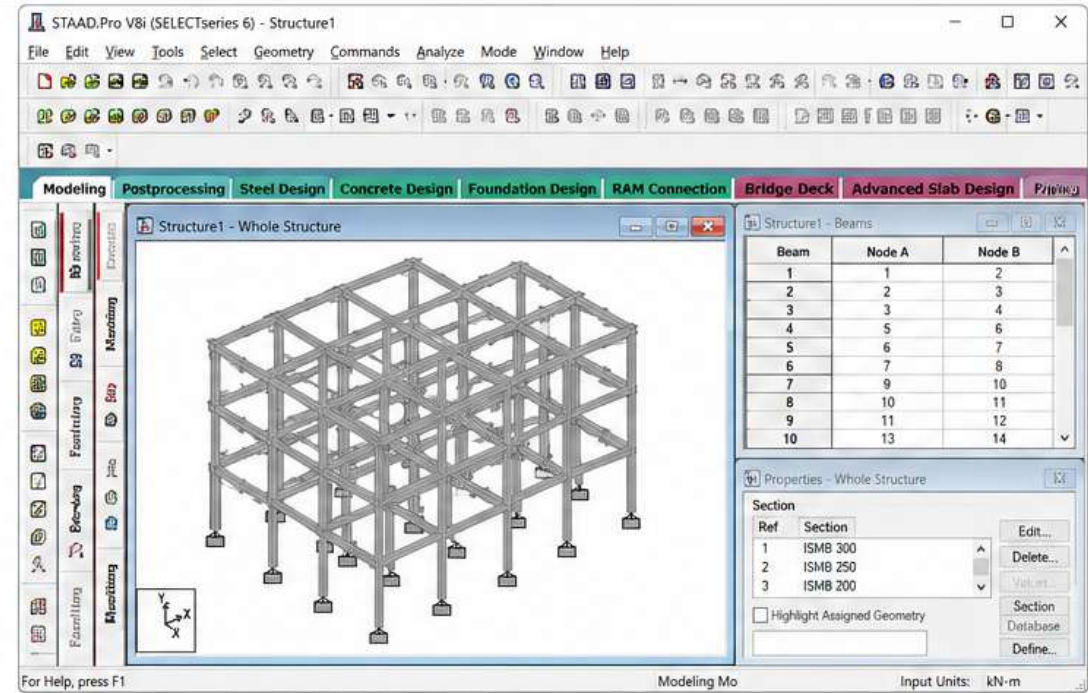
ABOUT THIS UNIT

This unit introduces the basics of structural analysis and familiarizes you with the STAAD.Pro interface, essential tools, and the overall workflow.

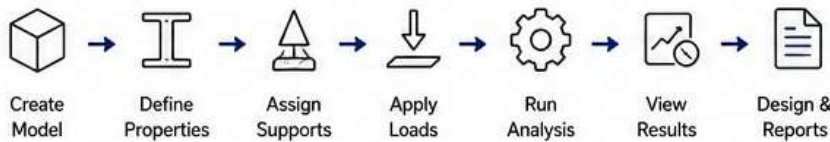


UNIT OUTCOMES

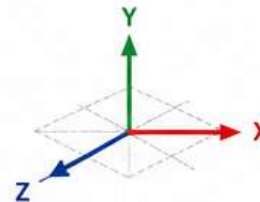
- ✓ Understand the basics of structural analysis
- ✓ Explore STAAD.Pro user interface
- ✓ Learn the complete workflow of a project
- ✓ Manage files and projects efficiently
- ✓ Use navigation tools and shortcuts effectively



WORKFLOW OVERVIEW



COORDINATE SYSTEM



Global Coordinate System
X - Red
Y - Green
Z - Blue

NAVIGATION TOOLS



Rotate



Pan



Zoom



Zoom Window



Dynamic View

KEYBOARD SHORTCUTS (ESSENTIAL)

Ctrl + N	Ctrl + O	Ctrl + S	Ctrl + Z	Ctrl + Y	F3	F4	F5	F6	F7
New File	Open File	Save File	Undo	Redo	Snap Node/Beam	Isometric View	Top View	Front View	Right View

WHY LEARN STAAD.Pro?

- Industry-leading structural analysis and design software
- Supports a wide range of materials and design codes
- Used globally by engineers and consultants
- Efficient, accurate and reliable results



UNIT 2

STRUCTURAL MODELING BASICS

Learn to create, edit and modify structural models using various modeling tools in STAAD.Pro.



WHAT YOU WILL LEARN

- Node Creation
- Beam Elements
- Plate Elements
- Geometry Tools
- Grid System
- Member Incidence
- Model Verification
- Structural Idealization



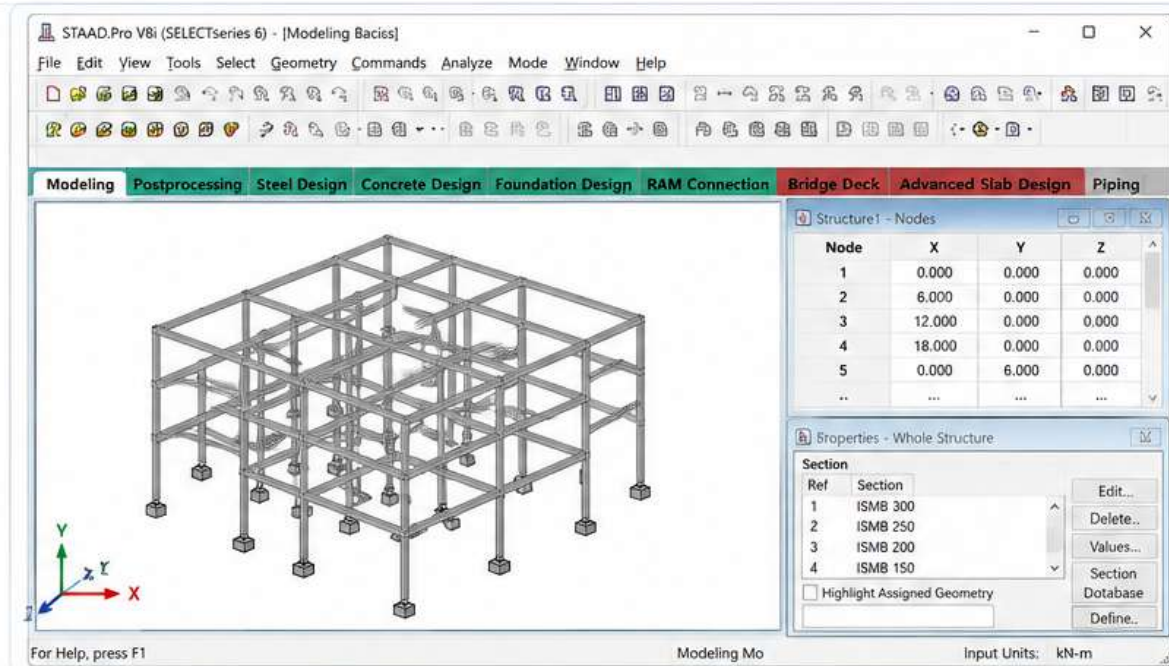
ABOUT THIS UNIT

This unit focuses on building the structural model from scratch. You will learn how to create nodes, beams, plates, use geometry tools, generate grids, assign member incidences and verify your model for accurate analysis.

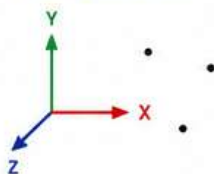


UNIT OUTCOMES

- ✓ Create nodes, beams and plates effectively
- ✓ Use geometry and grid tools for accurate modeling
- ✓ Understand member incidence and connectivity
- ✓ Verify and check the model for errors
- ✓ Idealize real structures for analysis

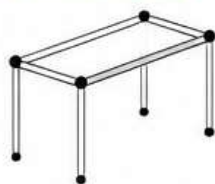


1. NODE CREATION



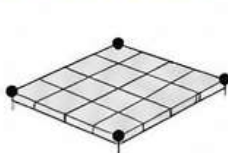
Create nodes manually or using coordinates.

2. BEAM ELEMENTS



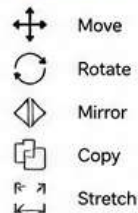
Create beam elements between nodes.

3. PLATE ELEMENTS



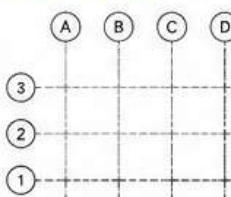
Create plate elements for slabs and walls.

4. GEOMETRY TOOLS



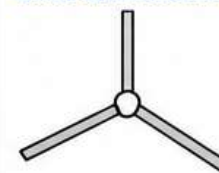
Modify and edit geometry with powerful tools.

5. GRID SYSTEM



Create grids and generate geometry quickly.

6. MEMBER INCIDENCE



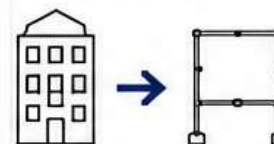
Ensure proper connectivity and member incidence.

7. MODEL VERIFICATION



Check for errors, warnings and model integrity.

8. STRUCTURAL IDEALIZATION



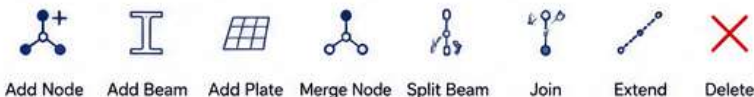
Convert real structures into analysis models.



MODELING TIPS

- Always work in consistent units.
- Use grids and snap for accurate modeling.
- Check member connectivity regularly.
- Verify model before assigning loads.

COMMON MODELING TOOLS



BEST PRACTICES

- Keep the model simple and accurate.
- Use proper sections and alignments.
- Remove duplicate or extra nodes.
- Maintain a clean and organized model.



KEY TAKEAWAY

A well-built model is the foundation for accurate analysis and design.



UNIT 3

MATERIAL PROPERTIES & SECTIONS

Define and assign materials and sections to your structural model for accurate analysis and design.



WHAT YOU WILL LEARN

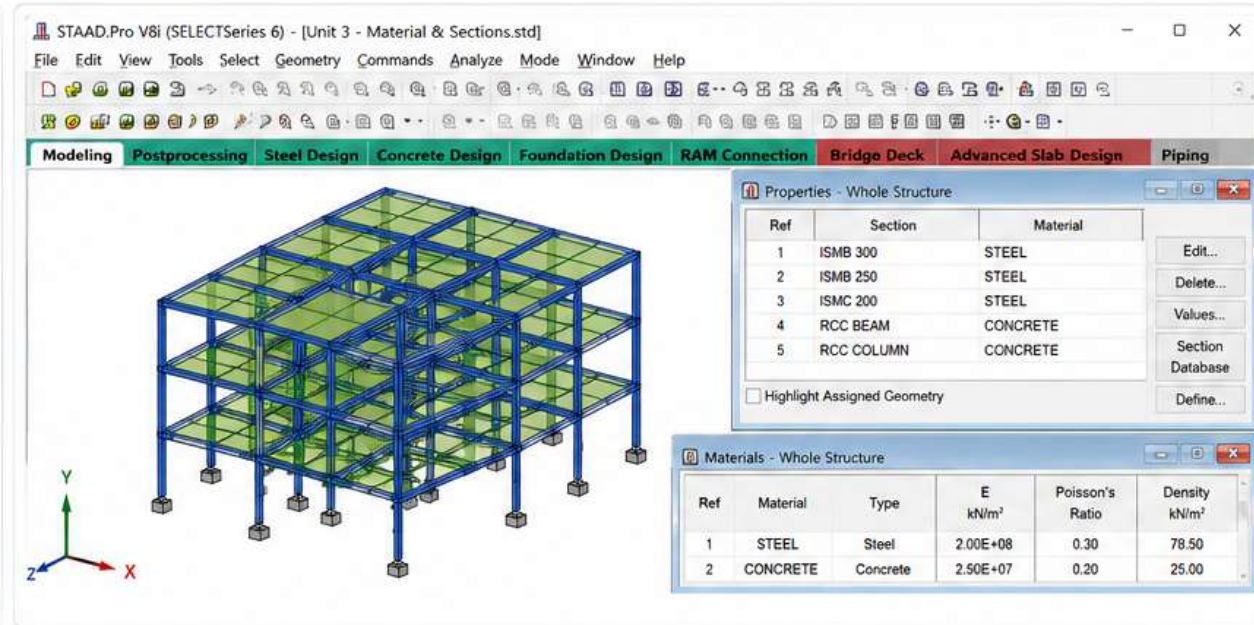
- Material Properties
- Steel Sections
- RCC Sections
- Section Database
- Custom Sections
- Member Properties
- Property Assignment
- Design Parameters

ABOUT THIS UNIT

This unit teaches you how to define different materials and create or select appropriate sections from the database. You will learn to assign properties to members for realistic analysis and design.

UNIT OUTCOMES

- Define and manage material properties
- Use built-in and custom sections
- Assign properties to members correctly
- Understand design parameters and codes
- Improve model accuracy and reliability



1. MATERIAL PROPERTIES

Define materials with accurate physical properties.

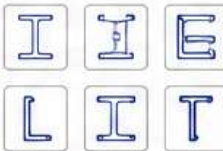
Material	Type	E (kN/m ²)	Poisson's Ratio	Density (kN/m ³)
STEEL	Steel	2.00E+08	0.30	78.50
CONCRETE	Concrete	2.50E+07	0.20	25.00



Define, add, edit and delete materials as per requirement.

2. STEEL SECTIONS

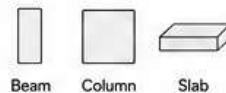
Select steel sections from the built-in database.



Wide range of IS, British, European and other standard steel sections.

3. RCC SECTIONS

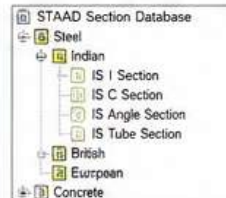
Use standard RCC sections for beams, columns and slabs.



Supports IS code based RCC section definitions.

4. SECTION DATABASE

Access and search sections easily from the database.



Quick search, filter and preview sections before use.

5. CUSTOM SECTIONS

Create and define your own custom sections.



Add user-defined sections as per project needs.

6. MEMBER PROPERTIES

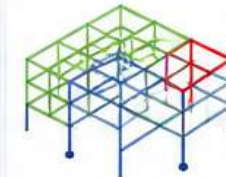
Assign section and material properties to members.

Beam	Section	Material
1	ISMB 300	STEEL
2	ISMB 250	STEEL
3	RCC BEAM	CONCRETE
4	RCC COLUMN	CONCRETE

Manage member properties efficiently.

7. PROPERTY ASSIGNMENT

Assign properties to members or groups of members.



Use cursor or list to assign properties quickly.

8. DESIGN PARAMETERS

Set design parameters and code preferences.

- Design Code (IS 800, IS 456)
- Method of Design
- Safety Factors
- Load Factors
- Design Preferences

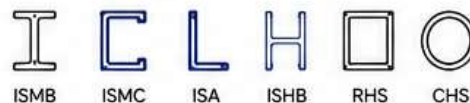
Ensure code-compliant design and accurate results.

TIPS



- Always check units before defining materials.
- Use appropriate sections for better accuracy.
- Verify property assignments before analysis.
- Keep your database organized.

COMMONLY USED STEEL SECTIONS



BEST PRACTICES

- Use standard codes and properties.
- Define materials and sections before modeling.
- Assign properties systematically.
- Review and verify before analysis.



KEY TAKEAWAY

Accurate material properties and section assignment are the foundation of reliable structural analysis and design in STAAD.Pro.



UNIT 4

SUPPORTS & LOADING

Learn how to apply supports and different types of loads to your structural model for accurate analysis.



WHAT YOU WILL LEARN

- Fixed Supports
- Pinned Supports
- Roller Supports
- Member Releases
- Dead Load
- Live Load
- Wind Load
- Seismic Load



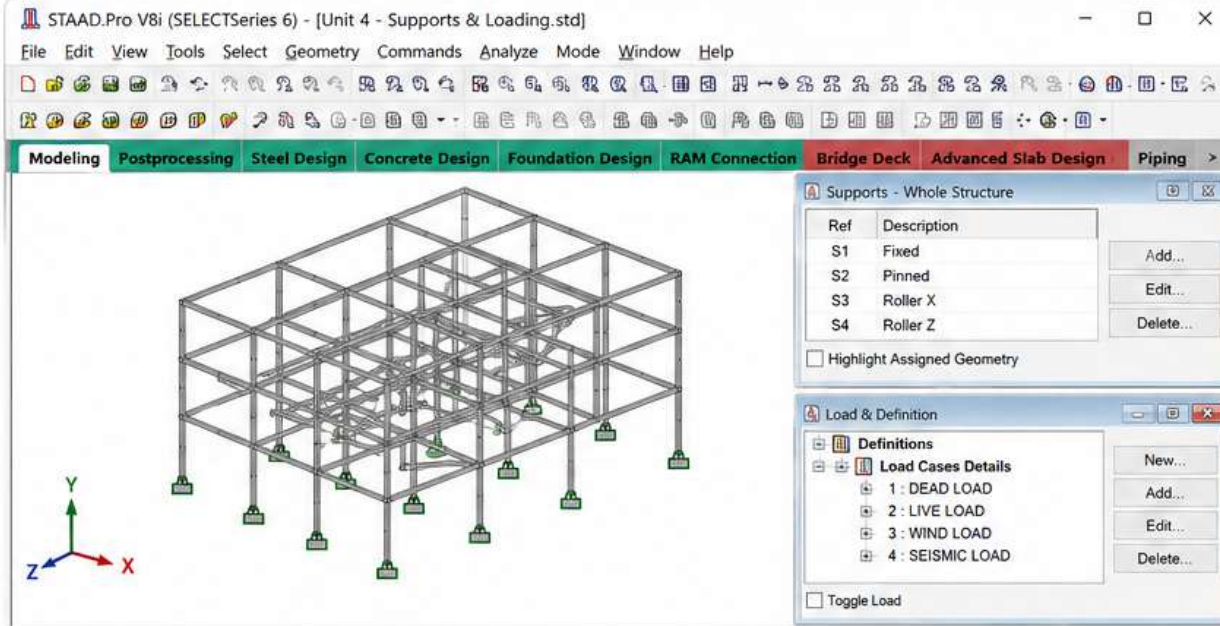
ABOUT THIS UNIT

This unit covers the types of supports and various load cases used in structural analysis. You will learn to apply supports correctly and define realistic loads as per codes.



UNIT OUTCOMES

- ✓ Understand different types of supports
- ✓ Apply supports to the model accurately
- ✓ Learn to define and apply various loads
- ✓ Use member releases for realistic behavior
- ✓ Follow standard load definitions as per codes



1. FIXED SUPPORT



Restraints all 6 degrees of freedom.

Use: Base of columns, retaining walls, etc.

2. PINNED SUPPORT



Restraints translations in all directions. Allows rotation.

Use: Typical column base.

3. ROLLER SUPPORT



Restraints translation in one direction. Allows movement in other direction and rotation.

Use: Bridges, beams, etc.

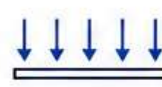
4. MEMBER RELEASES



Releases moments or forces at ends of members.

Use: To model connections and real behavior.

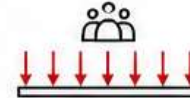
5. DEAD LOAD (DL)



Permanent load of the structure.

Includes self-weight of members, walls, finishes, etc.

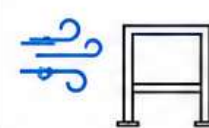
6. LIVE LOAD (LL)



Imposed load due to occupancy and use.

Includes floor load, roof load, etc.

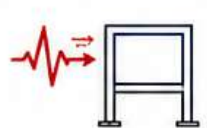
7. WIND LOAD (WL)



Lateral load due to wind pressure or suction.

Define per IS 875 (Part 3).

8. SEISMIC LOAD (EQ)

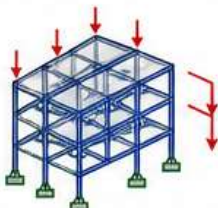


Lateral load due to earthquake forces.

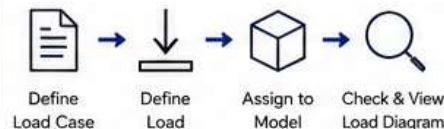
Define per IS 1893 (Part 1).

TYPES OF LOADS IN STAAD.Pro

- Nodal Load
- Member Load (UDL, Point Load, Trapezoidal)
- Member Temperature Load
- Element Load (Plate Loads)
- Self Weight (SW)



LOAD CASES WORKFLOW



BEST PRACTICES

- ✓ Always use correct support conditions.
- ✓ Apply realistic and code-based loads.
- ✓ Check load directions before analysis.
- ✓ Use self weight and load combinations.
- ✓ Verify load diagrams before running analysis.



KEY TAKEAWAY

Correct application of supports and loads is the foundation of accurate structural analysis and design.



UNIT 5

LOAD COMBINATIONS & STRUCTURAL ANALYSIS

Learn how to define load combinations and perform structural analysis to evaluate the behavior of your model.



WHAT YOU WILL LEARN

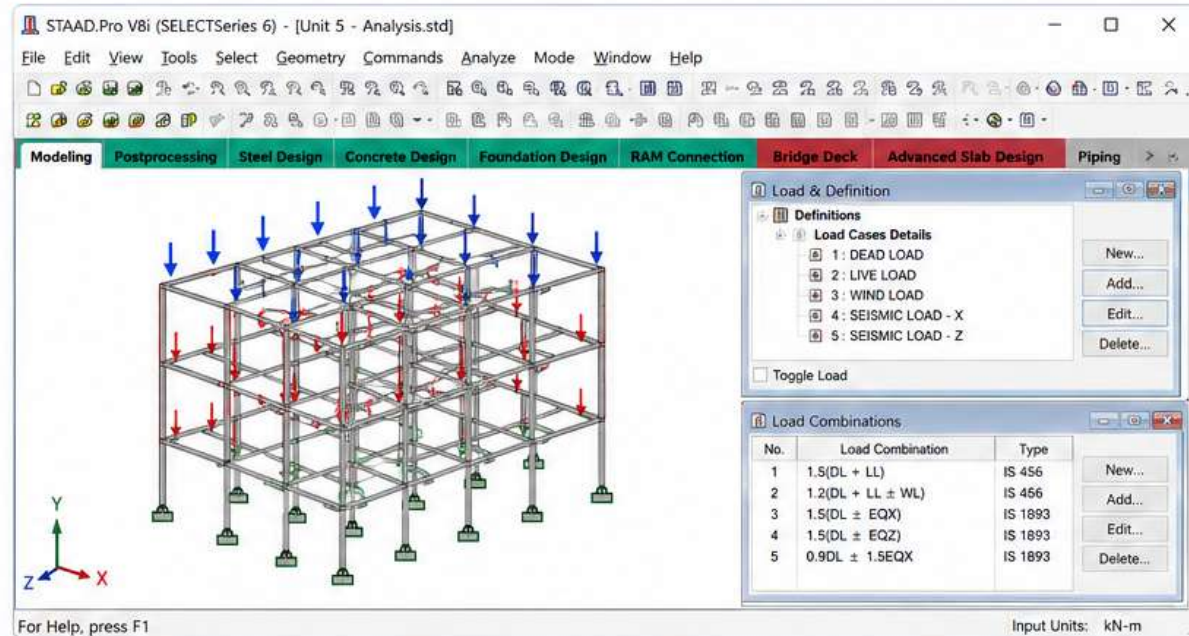
- Load Cases
- Load Combinations
- Analysis Commands
- Perform Analysis
- Stability Check
- Error Checking
- Model Validation
- Analysis Review

ABOUT THIS UNIT

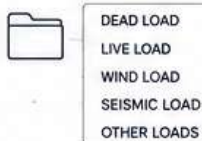
This unit explains how to define different load cases, create code-based load combinations and perform structural analysis. You will learn to check model stability, identify errors and validate results.

UNIT OUTCOMES

- Define and manage load cases effectively
- Create code-compliant load combinations
- Run analysis and interpret results
- Check stability and model errors
- Validate and review analysis results confidently

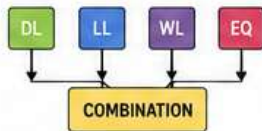


1. LOAD CASES



Define all types of loads acting on the structure as per requirements.

2. LOAD COMBINATIONS



Create code-based load combinations for safe and accurate design.

3. ANALYSIS COMMANDS



Use analysis commands to check and run the structural model.

4. PERFORM ANALYSIS



Run the analysis to solve for joint displacements, reactions and internal forces.

5. STABILITY CHECK



Check for overall stability, stability ratio and structure behavior under loads.

6. ERROR CHECKING



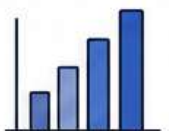
Identify and correct modeling errors and warnings before analysis.

7. MODEL VALIDATION



Validate the model to ensure accuracy and reliability of results.

8. ANALYSIS REVIEW



Review results, reactions, displacements and internal forces for decision making.

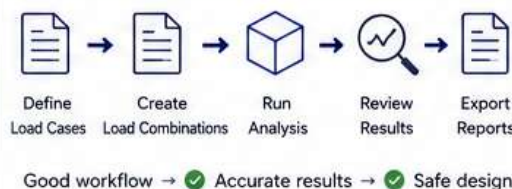
TYPES OF LOADS

- Dead Load (DL)** Self-weight of structure and fixed loads
- Live Load (LL)** Imposed load due to occupancy and use.
- Wind Load (WL)** Wind pressure or suction on structure
- Seismic Load (EQ)** Earthquake forces in defined directions
- Other Loads** Temperature, Snow, Crane, etc.

LOAD COMBINATION EXAMPLES (IS 456 & IS 1893)

No.	Load Combination	Code
1	1.5(DL + LL)	IS 456
2	1.2(DL + LL ± WL)	IS 456
3	1.5(DL ± EQX)	IS 1893
4	1.5(DL ± EQZ)	IS 1893
5	0.9DL ± 1.5EQX	IS 1893
6	0.9DL ± 1.5EQZ	IS 1893

ANALYSIS WORKFLOW

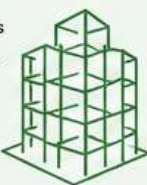


BEST PRACTICES

- Use correct units and directions.
- Define loads as per relevant codes.
- Check model before analysis.
- Verify stability ratios.
- Review results and reactions.
- Validate displacements and drift.
- Document and save reports.

KEY TAKEAWAY

Accurate load combinations and proper analysis ensure a safe, stable and economical structure.



UNIT 6

RCC DESIGN

Learn how to design RCC beams, columns, slabs and footings as per IS 456 using **STAAD.Pro**.



WHAT YOU WILL LEARN

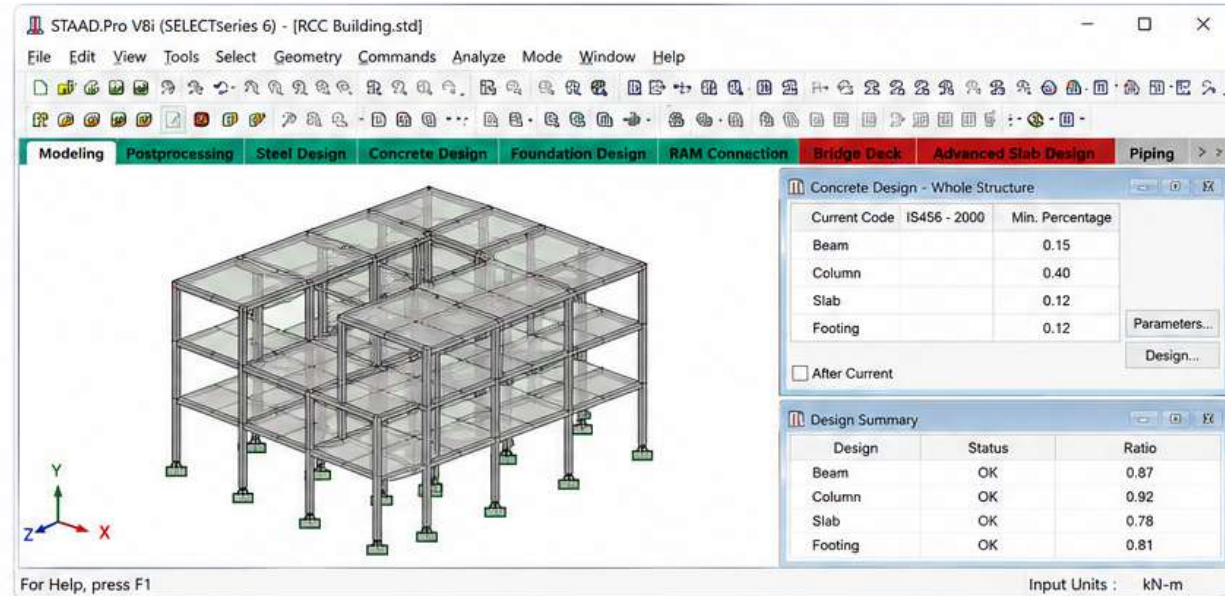
- RCC Beam Design
- RCC Column Design
- Footing Concepts
- Slab Modeling & Design
- Design Parameters
- Reinforcement Results
- Design Reports
- IS Code Design (IS 456)

ABOUT THIS UNIT

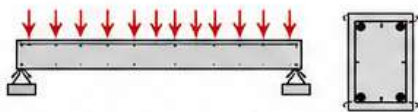
This unit covers the RCC design process in STAAD.Pro using IS 456. You will learn to design beams, columns, slabs and footings, view reinforcement details and generate design reports.

UNIT OUTCOMES

- ✓ Design RCC beams for flexure, shear and torsion
- ✓ Design RCC columns for axial and biaxial loads
- ✓ Model and design slabs for moments and shear
- ✓ Understand footing design concepts and checks
- ✓ Interpret reinforcement results and ratios
- ✓ Generate and review design reports

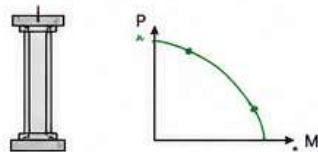


1. RCC BEAM DESIGN



- Checks: Bending, Shear, Torsion
- Design as per IS 456
- Reinforcement detailing
- View required & provided steel

2. RCC COLUMN DESIGN



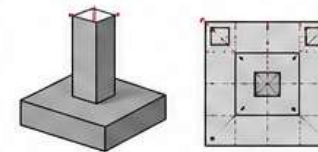
- Axial, Uniaxial & Biaxial design
- Slenderness & minimum steel checks
- Reinforcement detailing
- Column design reports

3. SLAB MODELING & DESIGN



- One way / Two way slab modeling
- Moment & shear design
- Deflection & crack width checks
- Reinforcement results (Top/Bottom)

4. FOOTING CONCEPTS



- Isolated footing design
- Checks: Bearing, Shear, Punching
- Required steel & details
- Footing design report

5. REINFORCEMENT RESULTS

Typical Beam Result			
Beam	Location	Top (mm ²)	Bottom (mm ²)
B1	Mid	804	1210
B1	Support	-	1540
B2	Mid	640	980
B2	Support	-	1350

- Steel area required & provided
- Utilization ratio
- Easy to interpret color coding

IS 456 DESIGN CONSIDERATIONS



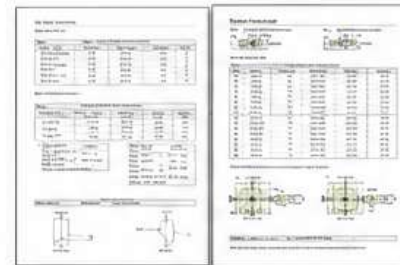
- Limit State Method of Design
- Safety Factors as per IS 456
- Minimum & Maximum reinforcement
- Serviceability checks (Deflection, Crack Width)
- Durability & Spacing requirements

DESIGN PARAMETERS (Typical)

Parameter	Typical Value
Concrete Grade	M25
Steel Grade	Fe 415
Clear Cover	25 mm
Min. Steel (Beam)	0.15 %
Min. Steel (Slab)	0.12 %
Min. Steel (Column)	0.40 %

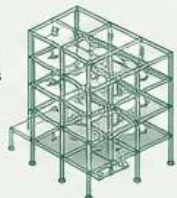
DESIGN REPORTS

- Beam Design Report
- Column Design Report
- Slab Design Report
- Footing Design Report
- Summary Report
- Detailed Reinforcement Report



KEY TAKEAWAY

RCC design in STAAD.Pro ensures safe, code-compliant and optimized structures with accurate reinforcement details and reliable reports.



UNIT 7

STEEL STRUCTURE DESIGN

Learn to design steel beams, columns, trusses and frames as per IS 800:2007 using **STAAD.Pro**.



WHAT YOU WILL LEARN

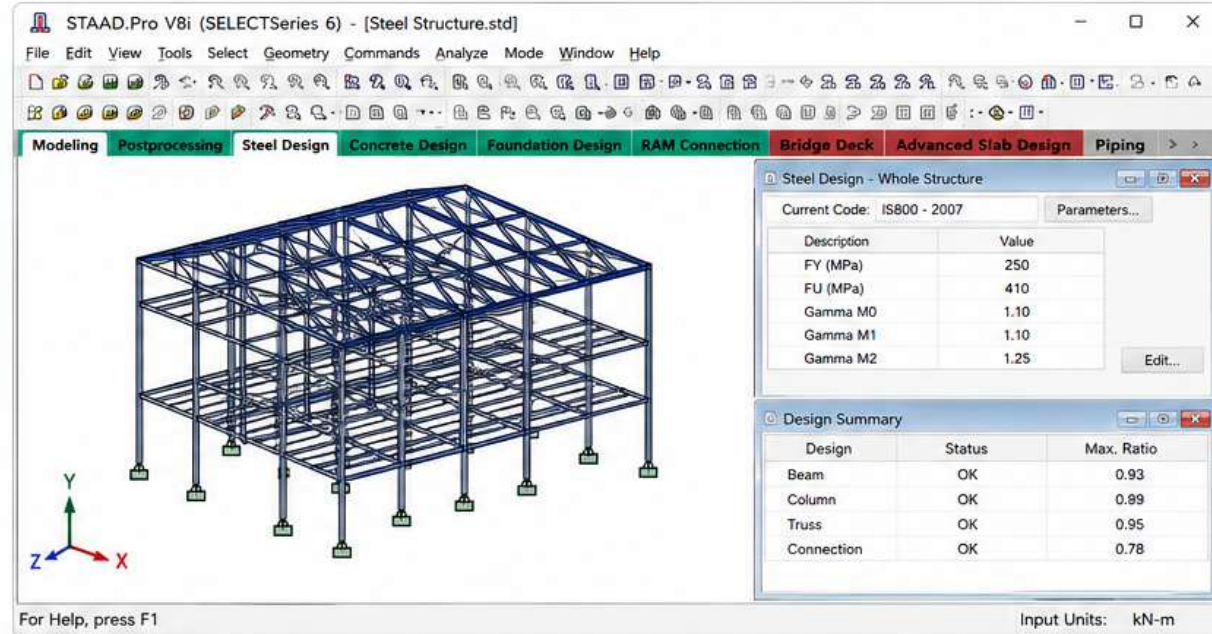
- Steel Beam Design
- Steel Column Design
- Steel Truss Design
- Industrial Frames
- Connection Concepts
- Design Parameters
- Design Optimization
- Steel Design Reports
- IS 800:2007 Design

ABOUT THIS UNIT

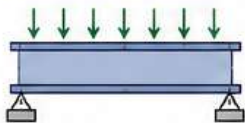
This unit covers the design of steel structures including beams, columns, trusses and industrial frames using STAAD.Pro. You will learn to apply IS 800:2007 design code, check member strength, stability and serviceability, and generate detailed design reports.

UNIT OUTCOMES

- ✓ Design steel beams and columns as per IS 800
- ✓ Design trusses and industrial frames
- ✓ Understand connection concepts and checks
- ✓ Perform design optimization for efficiency
- ✓ Interpret results and utilization ratios
- ✓ Generate and review steel design reports

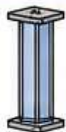


1. STEEL BEAM DESIGN



- Major & Minor bending
- Shear, Bending & Deflection checks
- Lateral Torsional Buckling (LTB)
- Section capacity & utilization

2. STEEL COLUMN DESIGN



- Axial, Biaxial & Uniaxial design
- Buckling & Slenderness checks
- Effective length & K-factors
- Column design report

3. STEEL TRUSS DESIGN



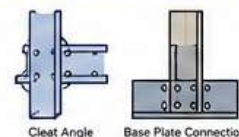
- Truss member design
- Tension & compression checks
- Gusset plate & member checks
- Optimize truss sections

4. INDUSTRIAL FRAMES



- Portal frame modeling
- Purlins, Girts & Bracing
- Wind load on industrial building
- Frame design & stability

5. CONNECTION CONCEPTS



- Types of steel connections
- Bolted & Welded connections
- Base plate & anchorage
- Connection design considerations

6. DESIGN PARAMETERS



- Design code (IS 800:2007)
- Materials & sections
- Design preferences
- Load combinations

7. DESIGN OPTIMIZATION



- Optimize sections automatically
- Reduce steel quantity
- Check different design options
- Achieve economical design

8. STEEL DESIGN REPORTS



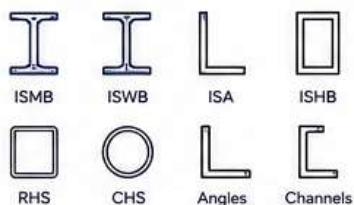
- Member design reports
- Detailing parameters
- Utilization ratio summary
- Export & documentation

IS 800:2007 DESIGN CHECKS

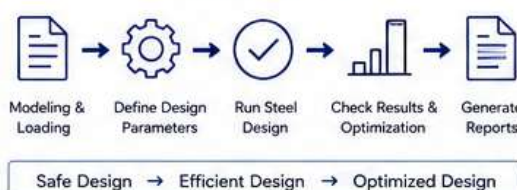


- ✓ Strength of cross section
- ✓ Stability of members
- ✓ Serviceability checks
- ✓ Deflection limits
- ✓ Buckling checks
- ✓ Connection checks

COMMONLY USED SECTIONS



DESIGN WORKFLOW



BEST PRACTICES

- ✓ Use correct design code & parameters
- ✓ Check member stability & slenderness
- ✓ Always check utilization ratios
- ✓ Review design reports thoroughly
- ✓ Optimize for economy & safety
- ✓ Validate the model before final design

KEY TAKEAWAY

Steel design in STAAD.Pro ensures safe, efficient and economical structures as per IS 800:2007 with accurate analysis and reliable design reports.



UNIT 8

RESULT INTERPRETATION

Learn how to read, understand and evaluate analysis and design results in **STAAD.Pro**.



WHAT YOU WILL LEARN

- Bending Moment Diagram
- Shear Force Diagram
- Axial Force Diagram
- Deflection / Displacement
- Stress Contours
- Support Reactions
- Utilization Ratio
- Design Reports



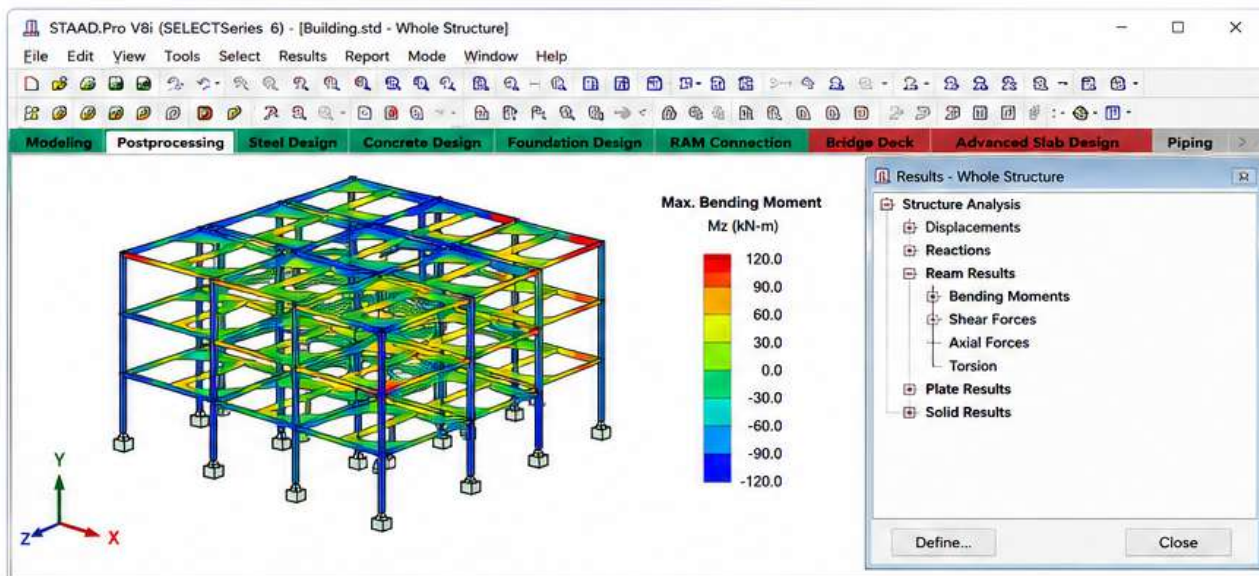
ABOUT THIS UNIT

This unit helps you understand and interpret analysis output, section forces, support reactions and design results to evaluate the performance and safety of a structure.

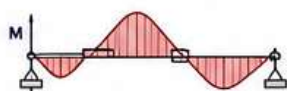


UNIT OUTCOMES

- ✓ Interpret all types of structural analysis results
- ✓ Understand behavior of the structure
- ✓ Evaluate member forces and deflections
- ✓ Review support reactions and load transfer
- ✓ Check utilization ratios and design status
- ✓ Generate and read detailed design reports

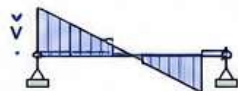


1. BENDING MOMENT DIAGRAM (BMD)



- Shows bending moments along the member
- Positive (sagging) and Negative (hogging)
- Used for design of beams and frames

2. SHEAR FORCE DIAGRAM (SFD)



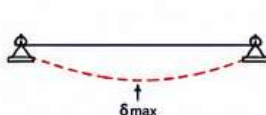
- Shows shear force variation along member
- Important for shear design
- Check for maximum shear at supports

3. AXIAL FORCE DIAGRAM (AFD)



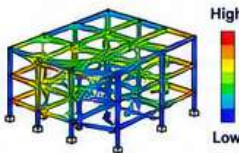
- Shows axial tension (+) or compression (-)
- Critical for columns, trusses and braces
- Helps in checking stability

4. DEFLECTION / DISPLACEMENT



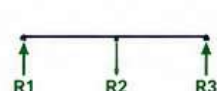
- Shows displacement of joints and members
- Check serviceability criteria (L, L/2, L/250 etc.)
- Identify excessive deflection

5. STRESS CONTOURS



- Shows stress distribution in members
- Identify high stress concentration areas
- Helps in design optimization

6. SUPPORT REACTIONS



- Reactions at supports due to applied loads
- Verify global equilibrium
- Used for foundation design

7. UTILIZATION RATIO (UR)



- Indicates capacity utilization
- UR < 1.0 → Safe
- UR = 1.0 → Limit
- UR > 1.0 → Not Safe

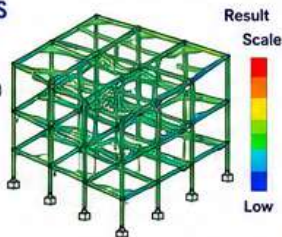
8. DESIGN REPORTS



- Detailed member design results
- Material, section & load details
- Reinforcement / steel summary
- Code compliance check

TYPICAL RESULT OUTPUTS

- Joint Displacements
- Member Forces (Fx, Fy, Fz, Mx, My, Mz)
- Support Reactions
- Element Stresses
- Plate Forces & Stresses
- Design Status (OK / Fail)



CHECKS TO PERFORM

- ✓ Check maximum deflection limits
- ✓ Check max. stress and section capacity
- ✓ Check stability of structure
- ✓ Check support reactions and load path
- ✓ Ensure all members have UR < 1.0
- ✓ Review design reports for critical members



RESULT INTERPRETATION WORKFLOW



Understand → Evaluate → Ensure Safety → Make Decisions

BEST PRACTICES

- ✓ Always run analysis without errors
- ✓ Verify units and load cases
- ✓ Review critical members first
- ✓ Check deflection & stability
- ✓ Validate UR and design results
- ✓ Cross-check with manual calculations
- ✓ Document and communicate results



KEY TAKEAWAY

Proper interpretation of results ensures safe, economical and code-compliant designs. Always rely on analysis results and engineering judgment together.



REMEMBER:

Results are not just numbers and graphs — they tell you how your structure behaves. Interpret them correctly to design safe and efficient structures.

UNIT 9

RESIDENTIAL BUILDING PROJECT

Work on a complete G+3 Residential Building project from modeling to design and reporting in **STAAD.Pro**.



WHAT YOU WILL DO

- 1 Modeling (RCC)
- 2 Load Assignment
- 3 Structural Analysis
- 4 RCC Design
- 5 Result Interpretation
- 6 Report Generation
- 7 Design Optimization
- 8 Drawing Review
- 9 Final Documentation

ABOUT THIS PROJECT

In this project, you will work on a G+3 residential building and follow a complete workflow from creating the model, applying loads, performing analysis, designing RCC members, and generating detailed reports.

PROJECT OUTCOMES

- ✓ Create a complete 3D model of building
- ✓ Apply all loads as per IS Codes
- ✓ Perform analysis and check model behavior
- ✓ Design RCC beams, columns, slabs & footings
- ✓ Interpret results and optimize the design
- ✓ Generate design reports and drawings
- ✓ Prepare final documentation for submission

STAAD.Pro V8i (SELECTSeries 6) - [Residential Building Project.std]

File Edit View Tools Select Geometry Commands Analyze Mode Window Help



Project Information	
Building Type	Residential Building (G+3)
No. of Storeys	4
Storey Height	3.0 m (Typical)
Plan Dimension	18.0 m x 15.0 m (Example)
Concrete Grade	M25
Steel Grade	Fe 415
Code Used	IS 456:2000
Analysis Type	Linear Static

Load Cases	
1. Dead Load (DL)	
2. Live Load (LL)	
3. Wind Load (WLX, WLY)	
4. Seismic Load (EQX, EQZ)	

BUILDING OVERVIEW



TYPICAL FLOOR PLAN (Example)



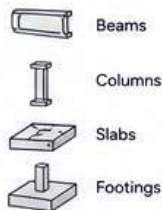
PROJECT WORKFLOW



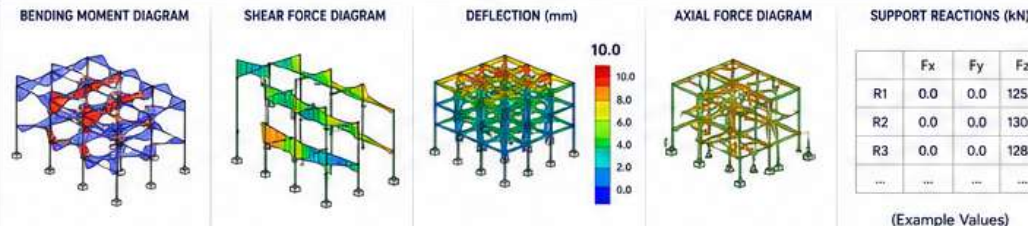
DESIGN CHECKS (IS 456)

- ✓ Flexure
- ✓ Column Axial + Biaxial
- ✓ Shear
- ✓ Slab Punching Shear
- ✓ Torsion
- ✓ Footing Bearing Pressure
- ✓ Deflection
- ✓ Development Length
- ✓ Serviceability
- ✓ Spacing & Detailing

RCC MEMBERS DESIGNED



TYPICAL RESULTS (EXAMPLES)



KEY TAKEAWAY

- ✓ End-to-end workflow on a real residential building.
- ✓ Understand practical application of IS codes.
- ✓ Develop confidence in modeling, analysis and design.
- ✓ Industry-ready skills for professional projects.



★ By the end of this project, you will be able to independently handle residential building projects using STAAD.Pro with confidence.

LEARN ► PRACTICE ► DESIGN ► DELIVER

UNIT 10

INDUSTRIAL STEEL STRUCTURE PROJECT

Work on a complete industrial steel building project from modeling to design, analysis and documentation in STAAD.Pro.



STAAD.Pro
— TRAINING —

WHAT YOU WILL DO

- 1 Structural Modeling
- 2 Load Assignment
- 3 Analysis & Design
- 4 Steel Design (IS 800:2007)
- 5 Connection Design
- 6 Result Interpretation
- 7 Design Optimization
- 8 Drawing & Detailing
- 9 Report Generation
- 10 Final Documentation



ABOUT THIS PROJECT

In this capstone project, you will design an industrial steel building (Pre-Engineered Building) including steel frames, bracing, purlins, girts and connections as per IS 800:2007.



PROJECT OUTCOMES

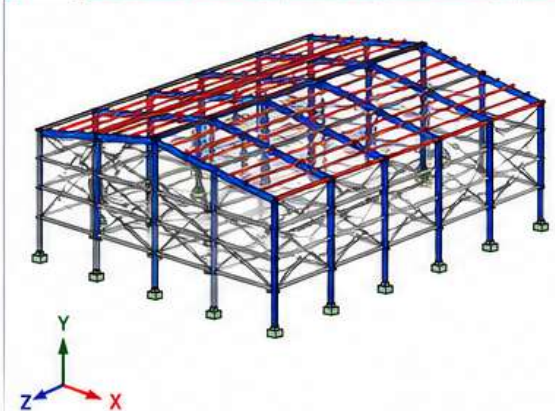
- ✓ Create a complete 3D industrial steel model
- ✓ Apply all relevant loads as per IS Codes
- ✓ Perform analysis and steel design
- ✓ Design connections and check member capacity
- ✓ Interpret results and optimize the design
- ✓ Generate detailed drawings and reports
- ✓ Prepare final documentation for submission

STAAD.Pro V8i (SELECTSeries 6) - [Industrial Steel Structure Project.std]

File Edit View Tools Select Geometry Commands Analyze Mode Window Help



Modeling Postprocessing Steel Design Concrete Design Foundation Design RAM Connection Bridge Deck Advanced Slab Design Piping



Project Information

Building Type	Industrial Steel Building
Building Use	Warehouse
No. of Bays (Along)	6
No. of Bays (Across)	4
Bay Spacing (Along)	8.0 m
Bay Spacing (Across)	7.5 m
Eave Height	8.0 m
Ridge Height	10.5 m
Steel Grade	Fe 250
Code Used	IS 800:2007
Analysis Type	Linear Static

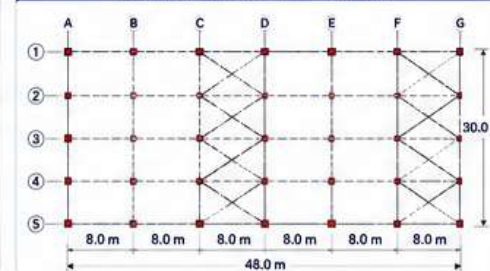
Load Cases

1. Dead Load (DL)
2. Live Load (LL)
3. Wind Load (WLX, WLY, WLZ)
4. Crane Load (CL)
5. Seismic Load (EQX, EQZ)

INDUSTRIAL BUILDING VIEW



TYPICAL FRAMING PLAN (Example)



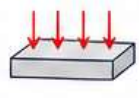
PROJECT WORKFLOW

1. MODELING



Create 3D model with columns, beams, bracing, purlins, girts & supports.

2. LOAD ASSIGNMENT



Apply dead, live, wind, crane & seismic loads as per IS Codes.

3. ANALYSIS



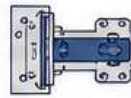
Run analysis and check displacements, reactions and stability.

4. STEEL DESIGN



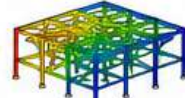
Design all steel members as per IS 800:2007 (LSD Method).

5. CONNECTION DESIGN



Design and check bolted & welded connections for safety and strength.

6. RESULTS



Interpret bending, shear, axial forces, utilization and displacements.

7. OPTIMIZATION



Optimize member sizes to achieve safe and economical design.

8. DRAWINGS & DETAILING



Prepare GA drawings, member drawings and connection details.

9. REPORT GENERATION



Generate design reports, material take-off and summary reports.

10. DOCUMENTATION



Compile all drawings, reports and documents for final submission.

DESIGN CHECKS (IS 800:2007)

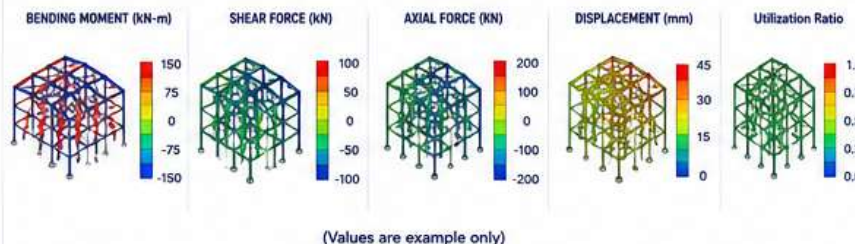
- ✓ Tension & Compression
- ✓ Bending & Shear
- ✓ Buckling (Lateral & Torsional)
- ✓ Deflection Limits
- ✓ Serviceability Checks
- ✓ Connections & Weld Checks
- ✓ Stability & Overall Safety



TYPICAL MEMBERS DESIGNED

- Columns
- Rafters / Beams
- Purlins
- Girts
- Bracing

TYPICAL RESULTS (EXAMPLES)



(Values are example only)

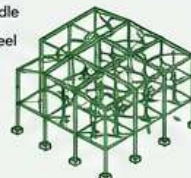
DELIVERABLES

- ✓ 3D Structural Model (.std)
- ✓ Analysis Results & Design
- ✓ Steel Design Reports
- ✓ Connection Design Details
- ✓ GA Drawings & Details
- ✓ Material Take-Off
- ✓ Summary & Calculation Sheets
- ✓ Final Project Documentation



KEY TAKEAWAY

By the end of this project, you will be able to handle real-world industrial steel structure projects confidently using STAAD.Pro.



CAPSTONE PROJECT:

Apply everything you have learned to deliver a safe, efficient and code-compliant industrial steel structure project.

LEARN ► PRACTICE ► DESIGN ► DELIVER

UNIT 11

ADVANCED ANALYSIS & DESIGN

Perform advanced structural analysis and design for complex structures using STAAD.Pro.



STAAD.Pro
— TRAINING —

WHAT YOU WILL LEARN

- 1 Advanced Modeling Techniques
- 2 Pushover / Nonlinear Analysis
- 3 Time History Analysis
- 4 Buckling & Stability Analysis
- 5 Response Spectrum Analysis
- 6 Member Design Optimization
- 7 Cold Formed Steel Design
- 8 Advanced Connection Design
- 9 Performance Based Design
- 10 Code Checks & Compliance
- 11 Advanced Reporting & BIM



ABOUT THIS UNIT

This unit focuses on advanced analysis methods and design techniques for complex structures. You will learn to perform nonlinear, dynamic, and stability analyses, optimize designs, and ensure code compliance as per IS Codes and International Standards.

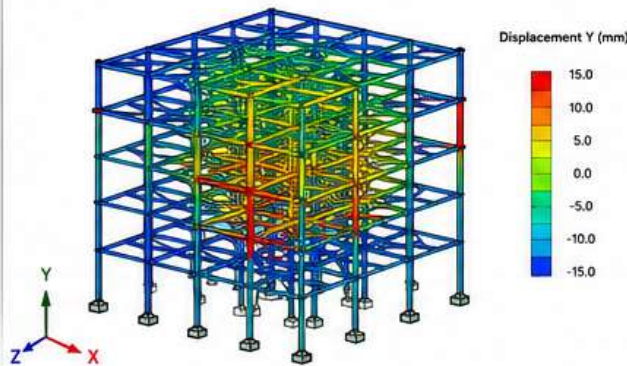


UNIT OUTCOMES

- ✓ Understand advanced analysis concepts
- ✓ Apply dynamic & nonlinear analysis techniques
- ✓ Perform stability and buckling checks
- ✓ Design and optimize structural members
- ✓ Carry out performance based design
- ✓ Ensure code compliance and safety
- ✓ Generate advanced reports and summaries

STAAD.Pro V8i (SELECTSeries 6) - Advanced_Analysis_Design.std

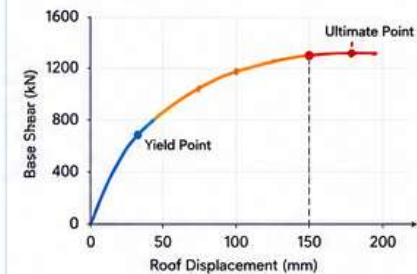
File Edit View Tools Select Geometry Commands Analyze Mode Window Help



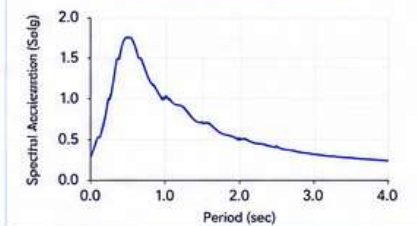
Analysis Information	
Analysis Type	Pushover Analysis
Load Case	PUSH-X
Steps	20
Control Node	Top Node
Target Displacement	150 mm
Base Shear (Max)	1250 kN

Code Information	
Design Code	IS 800:2007 / IS 456:2000
Steel Design Code	IS 800:2007
Concrete Design Code	IS 456:2000

PUSHOVER CURVE (EXAMPLE)

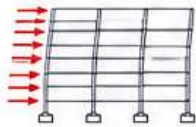


RESPONSE SPECTRUM (EXAMPLE)



ADVANCED ANALYSIS TYPES

Pushover Analysis



Evaluate nonlinear behavior and capacity of structure.

Time History Analysis



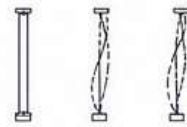
Study dynamic response under seismic ground motions.

Response Spectrum Analysis



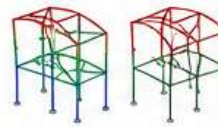
Determine maximum response using design spectrum.

Buckling Analysis



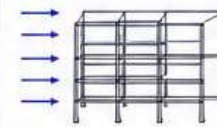
Check critical buckling load and mode shapes.

Modal Analysis



Find natural frequencies, modes and participating mass.

Wind Analysis



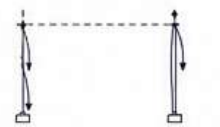
Evaluate wind loads and structural wind response.

Seismic Analysis



Evaluate seismic forces and design for earthquake safety.

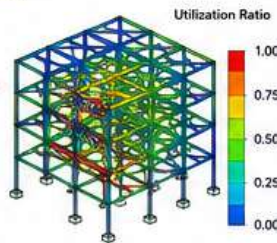
Second Order (P-Delta)



Consider P-Delta effects for stability and accuracy.

DESIGN & OPTIMIZATION

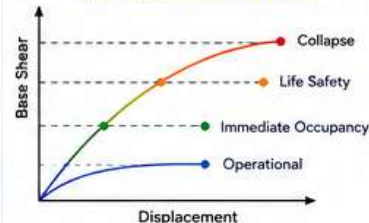
- ✓ Optimize member sizes for economy and safety
- ✓ Check strength, serviceability and deflection limits
- ✓ Design steel, RCC, cold formed & connections
- ✓ Utilization based design improvement



CODE CHECKS

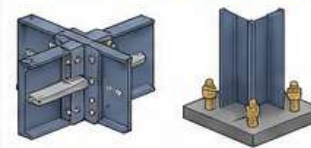
- ✓ IS 800:2007 (Steel)
- ✓ IS 456:2000 (RCC)
- ✓ IS 1893 (Seismic)
- ✓ IS 875 (Loads)
- ✓ IS 801 (Wind)
- ✓ IS 13920 (Ductile Detailing)
- ✓ AISC / Eurocode Support

PERFORMANCE BASED DESIGN



Design structures to meet performance objectives at different limit states.

ADVANCED CONNECTION DESIGN



- Design bolted, welded and base plate connections as per code.
- Check strength, stiffness and detailing.

REPORTS & DOCUMENTATION

- Analysis Results Reports
- Design Summary Reports
- Code Check Reports
- Graphical Output
- Member Take-off
- Detailed Calculation Sheets
- BIM / Model Export



KEY TAKEAWAY

Master advanced analysis and design techniques to handle complex structures confidently and deliver safe, efficient and code-compliant designs using STAAD.Pro.



★ **MASTER ADVANCED DESIGN.** Analyze deeper. Design smarter. Deliver safer structures.

LEARN ► ANALYZE ► DESIGN ► OPTIMIZE ► DELIVER

UNIT 12

PROFESSIONAL WORKFLOW & PROJECT DELIVERY

Learn the end-to-end professional workflow to deliver high-quality structural projects using **STAAD.Pro** with accuracy, clarity and efficiency.



STAAD.Pro
— TRAINING —

WHAT YOU WILL LEARN

- 1 Project Planning
- 2 Standards & Code Compliance
- 3 Modeling Best Practices
- 4 Analysis & Design Workflow
- 5 Document Management
- 6 Quality Check & Review
- 7 Coordination & Communication
- 8 Drawing Production
- 9 Report Generation
- 10 Client Delivery
- 11 Revision & Change Management
- 12 Project Handover



ABOUT THIS UNIT

This unit focuses on the professional workflow required to manage and deliver structural projects successfully. You will learn how to plan, execute, check and deliver projects following industry standards and best practices.

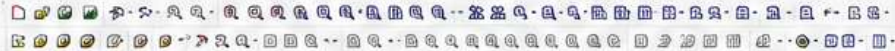


UNIT OUTCOMES

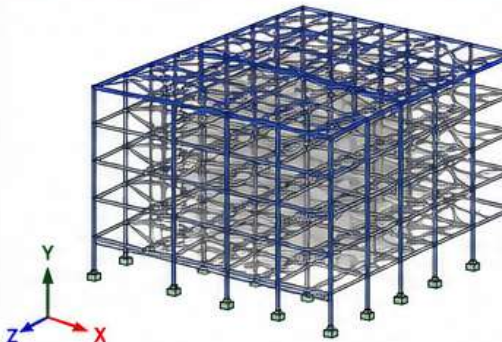
- ✓ Plan and organize structural projects efficiently
- ✓ Follow codes, standards and documentation best practices
- ✓ Implement a systematic analysis & design workflow
- ✓ Ensure quality, accuracy and consistency
- ✓ Communicate and coordinate effectively
- ✓ Deliver complete, professional project outputs
- ✓ Manage revisions and final handover smoothly

STAAD.Pro V8i (SELECTseries 6) - [Professional_Workflow.std]

File Edit View Tools Select Geometry Commands Analyze Mode Window Help



Modeling Postprocessing Steel Design Concrete Design Foundation Design RAM Connection Bridge Deck Advanced Slab Design Piping



Project Summary	
Project Name	Commercial Building
Structure Type	G+5 Office Building
No. of Storeys	6
Analysis Type	Linear Static
Steel Code	IS 800:2007
Concrete Code	IS 456:2000
Unit System	kN - m
Status	Design Completed

Deliverables	
✓	3D Model
✓	Design Reports
✓	GA Drawings
✓	Calculation Sheets
✓	Code Compliance Check

KEY WORKFLOW PRINCIPLES



Plan

Define scope, objectives, standards, schedule and deliverables.



Execute

Model, analyze and design with accuracy and efficiency.



Check

Verify results, perform quality checks and peer review.



Document

Prepare drawings, reports and calculation sheets.



Deliver

Submit final deliverables and ensure client satisfaction.

END-TO-END PROJECT WORKFLOW

1

Project Initiation



Understand requirements, scope, codes and client expectations.

2

Planning



Prepare project plan, timeline, resources and deliverables.

3

Modeling



Create accurate 3D model with proper naming & organization.

4

Load Definition



Define all loads as per codes and assign correctly.

5

Analysis



Run analysis and check stability of the model.

6

Design



Perform member design (steel/RCC) as per applicable codes.

7

Results Review



Interpret results, check utilization and serviceability.

8

Quality Check



Verify model, results, design and code compliance.

9

Documentation



Prepare drawings, reports and calculation sheets.

10

Client Delivery



Share deliverables, explain results and get feedback.

11

Revisions



Incorporate comments and update model, reports & drawings.

12

Final Handover



Submit final files, documents and close the project.

BEST PRACTICES

- ✓ Use templates and naming conventions
- ✓ Keep model clean and organized
- ✓ Assign loads carefully and check units
- ✓ Validate results at every stage
- ✓ Follow code provisions strictly
- ✓ Document everything clearly
- ✓ Communicate proactively with the team
- ✓ Deliver quality, not just quantity



DOCUMENT MANAGEMENT

- Proper file structure
- Version control
- Backup & cloud storage
- Standard templates
- Document register
- Access & permission control



TYPICAL DELIVERABLES



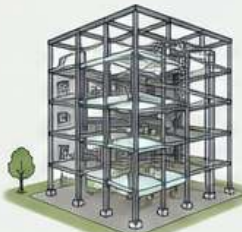
QUALITY CHECKLIST

- ✓ Model accuracy and connectivity
- ✓ Load cases and combinations
- ✓ Analysis warnings/errors resolved
- ✓ Design code and parameters
- ✓ Utilization within limits
- ✓ Serviceability requirements
- ✓ Drawing coordination
- ✓ Reports and documentation complete



KEY TAKEAWAY

A professional workflow ensures efficient project execution, accurate results, clear communication and timely delivery, leading to satisfied clients and successful projects.



Professionalism is not just about building models, it's about delivering complete, accurate and reliable solutions.



PLAN ► EXECUTE ► CHECK ► DOCUMENT ► DELIVER ► EXCEL