JUPITER PRIME
Consulting • Training • Advisory
جوبيتر برايم للاستشارات والتدريب

COURSE BROCHURE

Advanced Structural Analysis with STAAD Pro Training Course

13 – 24 June 2027
Manama

jupiterprime.uk
info@jupiterprime.uk
Registered in the United Kingdom and Saudi Arabia

Advanced Structural Analysis with STAAD Pro Training Course

Ref: 167_4792 **Date:** 13 – 24 June 2027 **Location:** Manama **Fees:** 8000 GBP

Advanced Structural Analysis with STAAD Pro is a 2-week training course for structural engineers and design modellers, built around one work product: a verified STAAD Pro analysis model and calculation package for a real structure. Tighter design programmes, heavier reliance on software output and closer scrutiny from checking engineers make disciplined modelling and verification essential. Jupiter Prime Training delivers this course on structural analysis with STAAD Pro.

Why Structural Analysis with STAAD Pro Matters Now

Three pressures shape structural analysis with STAAD Pro today: compressed design schedules that push engineers to model faster, independent checkers and clients who expect transparent load paths and traceable calculations, and taller, more slender or industrial structures where second-order and dynamic effects govern. Practices that model supports, releases and load combinations deliberately produce designs that pass checking first time; those that trust default settings risk unconservative members, wasted steel or costly redesign. The course rests on matrix stiffness analysis, response spectrum methods and the design modules of STAAD Pro, with concrete member checks referenced to the American Concrete Institute ACI code ACI 318.

Who Gains Most from Structural Analysis with STAAD Pro

- Structural engineers who build analysis models and sign off member designs
- Design modellers responsible for geometry, properties and load input
- Checking engineers who verify reactions, deflections and design ratios
- Industrial structures engineers who analyse pipe racks, platforms and equipment supports
- Bridge and building engineers who assess wind, seismic and moving load effects

Structural consultancies and contractor design offices gain most when modellers and checking engineers attend together and agree one modelling and verification standard.

Structural Analysis with STAAD Pro Capabilities

Participants move from producing output to defending it, linking each modelling assumption to member forces and design ratios.

- Build parametric geometry with supports, releases and offsets that reflect real behaviour — design checkers
- Specify primary load cases and code-based combinations — lead structural engineers
- Apply wind, seismic and P-delta analysis to sway-sensitive frames — senior design reviewers
- Calculate dynamic response with modal and response spectrum methods — seismic design decisions
-

Test steel and concrete members through design modules — drawing and fabrication teams
- Assess results with hand checks and equilibrium tests — clients and independent checkers

Key Structural Analysis with STAAD Pro Terms

- Member Release — a setting that frees selected end forces or moments so a connection behaves as pinned or partially fixed.
- Load Combination — a factored sum of primary load cases used to find the governing design forces for each member.
- P-Delta Analysis — a second-order analysis that adds the effect of axial load acting through lateral displacement.
- Response Spectrum Analysis — a dynamic method that combines modal responses to estimate peak seismic forces and displacements.
- Modal Mass Participation — the share of total mass captured by the analysed vibration modes, used to judge whether enough modes are included.
- Utilisation Ratio — the ratio of member demand to capacity, where values above one indicate a failing member.

Structural Analysis with STAAD Pro Day by Day

Day 1: Analysis Strategy and Modelling Philosophy

- Matrix Stiffness Method Refresher — Interpreting What the Solver Actually Computes
- Structural Idealisation Choices — Deciding Between Frame, Plate and Solid Elements
- Building, Bridge and Industrial Case Structures — Matching Approach to Structure Type
- Units, Coordinate Systems and Conventions — Preventing Sign and Scale Errors
- Modelling Brief and Assumptions Register — Recording Decisions for the Checker

Day 2: Geometry Generation and Properties

- Structure Wizard and Parametric Generation — Building Repetitive Frames Quickly
- Section Databases and Custom Profiles — Assigning Correct Member Properties
- Material Definitions for Steel and Concrete — Setting Stiffness and Density
- Member Offsets and Beta Angles — Orienting Sections Correctly
- Plate Meshing for Slabs and Walls — Controlling Mesh Quality

Day 3: Supports, Releases and Connectivity

- Fixed, Pinned and Spring Supports — Representing Foundations and Bearings
- Member End Releases — Modelling Simple and Moment Connections
- Master-Slave and Rigid Links — Simulating Diaphragms and Eccentric Connections
- Instability and Mechanism Detection — Fixing Under-Restrained Models
- Connectivity and Duplicate Node Checks — Cleaning Geometry Before Loading

Day 4: Loads and Load Combinations

- Dead, Imposed and Equipment Load Cases — Assigning Gravity Loads Accurately
- Area and Floor Load Generation — Distributing Slab Loads to Members
- Temperature and Settlement Loads — Capturing Restraint Effects
- Automatic Load Combination Tables — Generating Factored Design Combinations
- Moving Load Generation for Bridges and Crane Girders — Finding Governing Positions

Day 5: Wind and Seismic Loading

- Wind Load Definition on Open and Clad Frames — Applying Pressure Profiles
- Notional and Lateral Load Patterns — Testing Frame Sway Sensitivity
- Equivalent Static Seismic Method — Deriving Base Shear and Distribution
- Mass Model Definition — Converting Loads to Seismic Weight
- Accidental Torsion and Directional Combinations — Covering Seismic Eccentricity

Day 6: Second-Order and Dynamic Analysis

- P-Delta Analysis Settings — Capturing Second-Order Moments in Slender Frames
- Modal Analysis and Frequency Extraction — Identifying Dominant Vibration Modes
- Response Spectrum Analysis — Combining Modes With SRSS and CQC Rules
- Time History Analysis Overview — Assessing Machine and Blast Loading
- Buckling Analysis — Estimating Critical Load Factors for Frames

Day 7: Steel Design Checks

- Steel Code Settings and Effective Lengths — Setting Unbraced Lengths Correctly
- Member Selection and Optimisation — Reducing Tonnage Within Capacity
- Utilisation Ratio Reports — Identifying Governing Members and Clauses
- Deflection and Drift Limits — Checking Serviceability of Steel Frames
- Connection Force Extraction — Passing Design Forces to Detailers

Day 8: Concrete Design Checks

- Concrete Beam and Column Design to ACI 318 — Sizing Reinforcement From Analysis
- Slab and Wall Design From Plate Results — Reading Moment Contours
- Cracked Section Stiffness Modifiers — Adjusting Stiffness for Realistic Deflection
- Foundation Reactions for Footing Design — Exporting Support Forces
- Reinforcement Takeoff Reports — Estimating Steel Quantities for Costing

Day 9: Results Interpretation and Verification

- Equilibrium and Reaction Sum Checks — Confirming Load Paths Balance
- Hand Calculation Benchmarks — Validating Critical Member Forces
- Deflected Shape and Mode Shape Review — Spotting Modelling Errors Visually
- Sensitivity Studies on Supports and Releases — Bounding Uncertain Assumptions
- Calculation Report Structure — Presenting Output for Independent Checking

Day 10: Practising the Verified Analysis Package

- Case Structure Geometry and Supports — Applying Modelling Choices From Earlier Days
- Load Combination and Seismic Set-Up — Rehearsing Loading Workflows
- Second-Order and Dynamic Run — Testing Frame Stability and Response
- Steel and Concrete Design Pass — Confirming Member Adequacy
- Verified STAAD Pro Analysis Model and Calculation Package — Completing the Checker-Ready Work Product

Applying Structural Analysis with STAAD Pro After the Course

- Run equilibrium, reaction and hand-check benchmarks on a current STAAD Pro model and record the results in its assumptions register.
- Add P-delta and response spectrum runs to one live frame design and present the change in governing utilisation ratios to the lead engineer.

Structural Analysis with STAAD Pro Questions FAQ

What is structural analysis with STAAD Pro and why does it matter now?

Structural analysis with STAAD Pro is the use of the STAAD Pro finite element program to model, load, analyse and design building, bridge and industrial structures. It matters now because faster design cycles and stricter checking demand models whose assumptions are explicit and verifiable.

Which roles gain most from structural analysis with STAAD Pro training?

Structural engineers, design modellers and checking engineers gain most, along with industrial and bridge engineers. Each decides on supports, loads, analysis type or design ratios that determine whether a structure is safe, economical and approved without rework.

How does structural analysis with STAAD Pro training differ from a reinforced concrete design course?

Structural analysis with STAAD Pro training centres on building and verifying analysis models across materials. A reinforced concrete design course focuses on hand design of members to code; this training covers geometry, releases, load combinations, P-delta, response spectrum, design modules and verification.

What do you bring back from structural analysis with STAAD Pro training?

You bring back a verified STAAD Pro analysis model and calculation package for a real structure, with an assumptions register, load combination tables, dynamic results and hand-check benchmarks your design office can reuse.

Structural Analysis with STAAD Pro: The Way Forward

The verified STAAD Pro analysis model and calculation package first informs the member sizing decision on the next design submission. Modellers, lead engineers and checkers use it as one shared reference for assumptions and results. After its first use, compare checker comments and site observations with the model and refine support stiffness, releases and combination tables accordingly.