

STAAD.Pro Fundamentals

An outline of the STAAD.pro at the new STAAD facility is as follows:

Basic Theoretical Understanding

- Introduction with present form of STAAD.Pro
- Interaction with engineers to identify the proper need
- STAAD.Pro in general - Analysis & Design capabilities
- Clarifications on useful topics

Using the Graphical User Interface to generate the geometry (Pre-processor)

- The general STAAD.Pro environment
- Short discussion on plane frames, space frames, beams, trusses, etc.
- Using the drawing tools for creating nodes and beams. Generation methods such as copying/mirroring, rotating. Using spreadsheets.
- Tools for visualization of the model - panning, zooming, viewing from various angles, using multiple views and windows, renumbering entities.
- Examining the tools for checking structural integrity.
- Understanding the STAAD input file using the STAAD editor.
- Tutorial problem.
- Generating models using the Structure Wizard.
- Hands-on Session

Using the Graphical User Interface to generate a complete STAAD model (Pre-processor)

- Building a truss model using the drawing tools and Structure Wizard
- Creating groups.
- Understanding the various property types, the steel section database.
- Creating and assigning properties from steel tables.
- Material constants.
- Understanding beta angles, local vs. global axis, visualization using 3D diagrams.
- Specifications such as member releases, member offsets, tension only/compression only members, cables, etc.
- Understanding and creating various support types.
- Understanding and specifying various primary load types.
- Creating Load combinations.
- Performing a general elastic analysis.
- Hands-on Session

Understanding the results and post-processing

- Viewing results using the STAAD output file.
- Using the graphical Post-processor for verifying and validating analysis results.
- Using the query analyzer.
- Creating customized reports, plotting result diagrams.
- Exporting results to other programs, printing.
- Tutorial Problem.

Advanced Supports, Loadings and Analysis (if applicable)

- Mat, Foundation, Multilinear Spring, Inclined, Enforced.
- Moving Load, Floor Load, Wind Load (closed and open structure)
- Seismic Load with relevant Seismic Code (primarily IS-1893)

- Time History Analysis
- Response Spectrum Analysis
- Second Order and Non-Linear Analysis techniques

Finite Element analysis (if applicable)

- General rules in using plate elements for modeling surfaces. Modeling plates with beams.
- Creating plate elements in the main GUI.
- Mesh generation in the main GUI and in Structure Wizard.
- Creating a complete beam-column-slab model.
- Using Surface Element
- Assigning plate properties, constants, plate element loading.
- Advanced analysis features such as P-delta analysis and non-linear analysis
- Understanding plate results, stress contours.

Steel Design (Optional)

- Specifying design parameters.
- Code checking, Member Selection.
- Controlling Section Property Database
- Ramifications of member selection on statically indeterminate structures.
- Grouping and physical member design.
- Steel Designer for Member and Connection Design
- Tutorial Problem

Concrete Design (Optional)

- Specifying design parameters.
- Designing beams, columns, elements per IS-456
- Designing beams and columns per IS-13920
- RC Designer for Beam and Column design (physical member concept)
- Tutorial Problem

Open House

- Sample STAAD files
- Discussion on specific issues
- Highlight on Live Projects
- Q&A and Feed Back Sessions