Structural Design Laboratory

STAAD Pro

STAAD Pro full form stands for Structural Analysis and Designing Program. STAAD Pro is a structural analysis & design computer program that was being developed by Research Engineers International (REL) at Yorba Linda, California in 1997. Today, STAAD Pro is one of the popular and widely used software for structural analysis and design across the globe by Civil engineers. It supports all types of various steel, concrete, and timber design codes.

Using STAAD Pro, civil engineers can design any type of structure, and later share the synchronized model data amongst the entire design team. It ensures on-time and budget-friendly completion of structures and designs related to steel, concrete, timber, aluminium, and cold-formed steel projects, irrelevant to the complexities. STAAD Pro helps structural engineers to automate their tasks by removing the tedious and long procedures of the manual methods. It allows civil engineers to analyze and design various types of structures on virtual platforms. Structural engineering firms, consultancies, various departments of construction companies, and government firms use STAAD pro extensively.

Importance Of STAAD Pro in Civil engineering

- In recent years, STAAD Pro has become an integral part of structural analysis & design solutions mainly using an exposed API called OpenSTAAD to access and drive the program with the help of Visual Basic macrosystem included in the application. Also, OpenSTAAD functionality is used in applications that themselves include suitable programmable macro systems.
- Additionally, STAAD Pro also has added direct links to applications such as RAM Connection and STAAD Foundation. It provides help to engineers working with those applications that are handling design post-processing which is directly not handled by STAAD Pro itself.
- STAAD Pro does schema analysis of the CIMsteel Integration Standard, version 2 commonly known as CIS/2, and used by a number modelling and analysis applications.

Purpose Of STAAD Pro In Civil engineering

1. With the help of STAAD Pro, civil engineers can easily analyze & design civil engineering structures such as buildings, bridges, dams, canals, sewage systems, plane and space trusses.
2. STAAD Pro can generate loads such as wind, or earthquakes as per building codes of selected countries.
3. STAAD Pro can be used to design steel and reinforced concrete buildings as per the codes of selected countries
4. It can carry out linear elastic (static & dynamic), and nonlinear dynamic analysis
5. It has a simple and easy-to-understand interface which makes learning quick for aspirants.

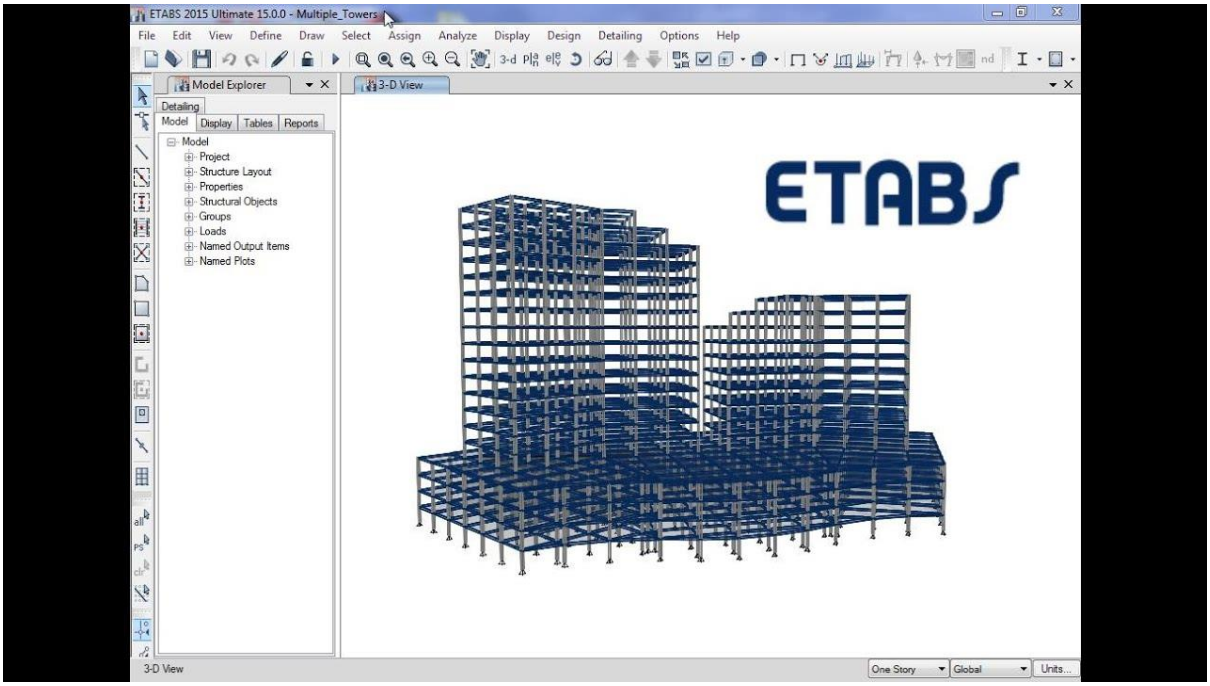
ETABS

ETABS is an engineering software product that caters to multi-story building analysis and design. Modeling tools and templates, code-based load prescriptions, analysis methods and solution techniques, all coordinate with the grid-like geometry unique to this class of structure. Basic or advanced systems under static or dynamic conditions may be evaluated using ETABS. For a sophisticated assessment of seismic performance, modal and direct-integration time-history analyses may couple with P-Delta and Large Displacement effects.

Nonlinear links and concentrated PMM or fiber hinges may capture material nonlinearity under monotonic or hysteretic behavior. Intuitive and integrated features make applications of any complexity practical to implement. Interoperability with a series of design and documentation platforms makes ETABS a coordinated and productive tool for designs which range from simple 2D frames to elaborate modern high-rises.

Fundamental to ETABS modeling is the generalization that multi-story buildings typically consist of identical or similar floor plans that repeat in the vertical direction. Modeling features that streamline analytical-model generation, and simulate advanced seismic systems, are listed as follows:

- Templates for global-system and local-element modeling
- Customized section geometry and constitutive behavior
- Grouping of frame and shell objects
- Link assignment for modeling isolators, dampers, and other advanced seismic systems
- Nonlinear hinge specification
- Automatic meshing with manual options
- Editing and assignment features for plan, elevation, and 3D views



Once modeling is complete, ETABS automatically generates and assigns code-based loading conditions for gravity, seismic, wind, and thermal forces. Users may specify an unlimited number of load cases and combinations. Analysis capabilities then offer advanced nonlinear methods for characterization of static-pushover and dynamic response. Dynamic considerations may include modal, response-spectrum, or time-history analysis. P-delta effect account for geometric nonlinearity. Given enveloping specification, design features will automatically size elements and systems, design reinforcing schemes, and otherwise optimize the structure according to desired performance measures.

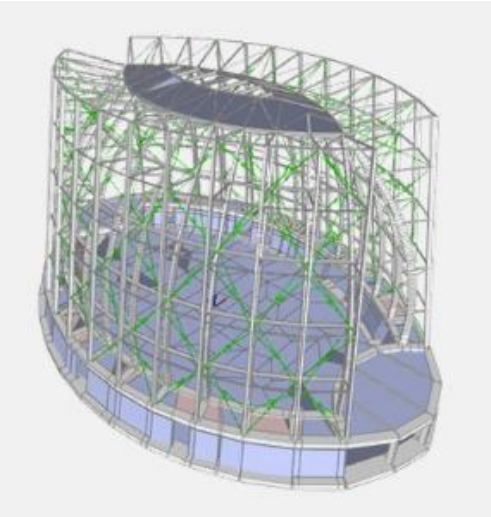
ETABS also features interoperability with related software products, providing for the import of architectural models from various technical drawing software, or export to various platforms and file formats. SAFE, the floor and foundation slab design software with post-tensioning (PT) capability, is one such option for export. CSI coordinated SAFE to be used in conjunction with ETABS such that engineers could more thoroughly detail, analyze, and design the individual levels of an ETABS model.

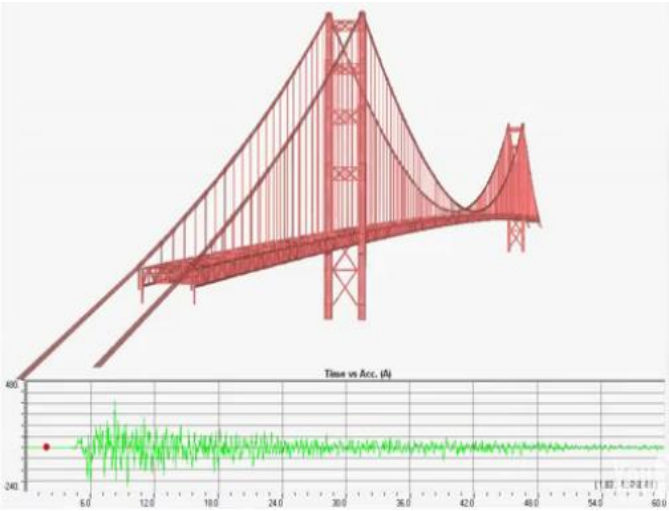
SAP 2000

SAP2000 is general-purpose civil-engineering software ideal for the analysis and design of any type of structural system. Basic and advanced systems, ranging from 2D to 3D, of simple geometry to complex, may be modeled, analyzed, designed, and optimized using a practical and intuitive object-based modeling environment that simplifies and streamlines the engineering process. The SAPFire ® Analysis Engine integral to SAP2000 drives a sophisticated finite-element analysis procedure. An additional suite of advanced analysis features are available to users engaging state-of-the-art practice with nonlinear and dynamic consideration. Created by engineers for effective engineering, SAP2000 is the ideal software tool for users of any experience level, designing any structural system.

Integrated modeling templates, code-based loading assignments, advanced analysis options, design-optimization procedures, and customizable output reports all coordinate across a powerful platform to make SAP2000 especially useful for practicing professionals.

Built-in modeling templates, a versatile and user-friendly interface, intuitive controls and features all combine to simplify and expedite a sophisticated object-based modeling process. A broad range of modeling options provide for methods and technologies at the forefront of structural engineering. Model domain may be component, system, or global-level in scope, while encompassing sub-grade components and soil-structure interaction. Grid line, snap, and replication tools are a few of the many practical features which make the modeling environment and process accessible to beginners, and sophisticated for advanced users.





Linear or curved members, cables and post-tensioned tendons, link elements to model springs, dampers, isolators, and the associated nonlinear and hysteretic behavior, framing, shell or multi-layered shell, solid elements with isoperimetric formulation and nonlinear response are all modeling options for object assembly in SAP2000. When preferred structural members are not provided in the extensive libraries of SAP2000, Section Designer is available for custom cross-section design. Users specify geometry and material composition before Section Designer automatically calculates member properties and generates biaxial-interaction and moment-curvature diagrams.

Nonlinear-fiber-hinge assignment is another advanced modeling technique available. SAP2000 implements code-based or empirical hinging behavior by modeling geometry and materials as discrete points within a cross-section, then correlating these discretized areas with their associated nonlinear behaviors. Limit-state and hysteretic considerations may also be implemented under nonlinear-static and dynamic analyses.

Static and dynamic methods are available for earthquake simulation. Nonlinear-static-pushover analyses may consider modal, uniform, or user-defined lateral load patterns, plastic-hinging behavior of slender elements, inelastic response of shear walls, floor slabs, and steel plates, and then formulate demand-capacity, damping, and performance-point calculations with customizable summary reports.

Dynamic methods include response-spectrum (for likely maximum seismic response given pseudo-spectral acceleration vs. structural period curve), power-spectral-density and steady-state (for fatigue behavior with optional damping and complex-impedance properties), and time-history analyses. Time histories may follow modal or direct-integration methods, and they may be chained together and enveloped with such advanced analyses as P-delta and staged-construction procedures.