

History of STAAD Pro

- STAAD Pro stands for Structural Analysis And Design Programme.
- It is one of the first software in the world to help structural engineers to eliminate long time of manual calculation & automate the whole process so as to minimize error.
- Earlier it was developed in late 1980 by research engineers at Kolkata.
- The earliest version of STAAD was designed for DOS Operating System. In order to operate this version the user had to learn the lengthy procedures of syntax of STAAD language of commands.
- STAAD Pro gained popularity because of its ease of use and data sharing capabilities with major civil related softwares like AutoCAD.
- STAAD Pro was bought by Bentley Systems, Inc., in late 2005 and early 2006.
- The basic steps followed in the software
 - Geometry ->Property -> Boundary Condition -> Loading -> Analysis
- Analysis is divided into two parts i.e Design and Post Processing.
- Design includes Defining Parameters, Commands, and Design using specific code.
- Post Processing includes creating shear force, axial force, flexural moment, bending moment, induced stresses, nodal deflection. Support reaction, structural deflection, energy displacement.

Possibilities of STAAD Pro

- It can handle any complex geometry, any complex boundary condition, typical loading condition. It can do stress analysis, vibration analysis, linear & non-linear analysis, buckling analysis etc.
- Column shortening
- It can design steel, RCC timber, Aluminum sections.
- It can design high rise building, water tank, basement walls, bridges, suspension towers, transmission towers, trusses, swimming pool, retaining wall, industrial structure, machine structure, blast resistance structures etc.

Basic theory to operate STAAD Pro

- ✓ Torsion is when a member rotates about its own axis.
- ✓ Bending and flexure when a member rotates about on axis perpendicular and orthogonal to structural axis.
- ✓ Unit weight of brick is 20KN/m^3

Methods of Analysis

1. Displacement matrix method

2. Force Matrix Method
3. Finite Element Method

- Finite element method is an extension of displacement matrix method
- In STAAD Pro v8i Finite element method is used for analysis.
- It enables the processing of lengthy calculation and in result save a lot of time and give more accurate result.
- The Applied Force $P=k\Delta$ (spacing analysis)
- P is the force matrix. It includes all the forces acting on the model and the reactions at the supports. The loads are always known, which includes Dead Load, Live Load, Seismic Load, Wind Load etc

Where k =Stiffness Factor Matrix

Δ =displacement

$$K = \frac{\alpha EI}{L^\beta}$$

Where $\beta=1$ at most of the cases

α =constant depending on type of material used, type of element used in analysis, configuration or geometry of the structure, assumption in structural engineer, boundary condition of the structure etc

E =Young's modulus of Elasticity

I =Secant modulus or tangential modulus at material non-linear zone

A = Displacement matrix

Degrees of freedom

- Number of direction a structure can move is structural degree of freedom.
- For plane truss it is 3
- For space truss it is 6

Indeterminacy

- It is of two types i)static &ii)dynamic
- Number of equilibrium required to solve a structure.
- Number of dynamic equation or displacement equation required to solve dynamic degrees of freedom.

Types of Support

- i. Fixed Support
- ii. Hinged Support
- iii. Pinned Support
- iv. Partial Fixed Support

- v. Spring Support.

Types of Beams

- i. Fixed Beam
- ii. Simply Supported Beam
- iii. Cantilever Beam
- iv. Propped Cantilever Beam
- v. Overhanging Beam

Types of Column

- i. Long Column
- ii. Short Column
- iii. Braced Column
- iv. Unbraced Column

SLAB

- Supported along two sides is one way slab ($l_y/l_x > 2$)
- Supported along any three or four side is two way slab ($l_y/l_x < 2$)