Introduction To STAAD Pro Training

Description

Course Description

This course provides an overall look over STAAD Pro 2005. It demonstrates the steps to be followed to produce the structural analysis of many types of buildings.

Also the course concentrates over the different results generated from the program, and how to read them, view them, and finally generate the necessary reports from them.

At the completion of this course, the trainee will be able to:

- Understand how the multipurpose finite element programs conduct the structural analysis.
- Understand STAAD Pro way of doing the job
- Understand STAAD Pro way of pre processing the job
- Creating the geometry using different methods
- Use more advanced technique in creating geometry
- Understand STAAD Pro element library
- Defining the Cross-Sections of Beams, Columns, Plates
- Defining the Constants, Specifications, and Supports
- Defining the Load Systems
- Analyzing your Model using the appropriate Analysis method
- Reviewing the Analysis Results

Course Objective

This course is intended to overview modern procedures for the structural analysis using different software. The course may be attended by civil engineers involved in design. Each participant may draw on the elements of the course that most complement his area of interest and practice.

For those engineers with limited design experience, the course will provide ample illustration of real structures that may assist the designer to understand STAAD Pro and apply it on different types of buildings. Different types of editing data to STAAD Pro will be discussed, and the easiest way to deal

with it will be reached.

The wide range of issues to be discussed, revolve around the use of STAAD Pro in structural Analysis of different types of structures. The examples to be used would vary from the typical two-dimensional beams and frames to the three-dimensional multistory frames to special systems such as space frames and three-dimensional slabs and raft foundations. Throughout the course, the instructor shall start from the basics to allow to ensure the full participation and comprehension of all attendant, bearing in mind variations in background from education to practice.

Course Outline

- The stiffness method as an introduction to finite element method.
- The finite element method used by most of the used software.
- Presentation of some available multi-purpose finite element computer packages used in design market.
- Introduction to STAAD Pro 2005, computer requirements and size.
- Different methods of analysis.
- Creating a new file with STAAD Pro, 2005.
- Recognizing STAAD Pro, screen (Menu bar, tool bar, title bar, modes …etc.
- Dealing with an existing file.
- Geometry
- This module contains:
- Understanding STAAD Pro way
- Defining Nodes, Beams, and Plates.
- Arrangement of input file.
- Using Structure Wizard to create Geometry
- using drafting to create Geometry
- Using Copy/Cut with Paste to create Geometry
- Using Spreadsheet to create Geometry
- Using DXF importing to create Geometry
- Analysis of two-dimensional continuous beams.
- Loads affecting beam elements.
- Analysis of two-dimensional frames and trusses.
- Loads affecting frames.
- Analysis of three-dimensional frames and trusses.
- Loads affecting frames and trusses.
- Analysis of plate elements (slabs, flat slabs)
- Loads affecting plate elements.
- Dealing with output data