Before delving into the definition of STAAD.Pro and its functions in structural analysis and design, it's essential to mention some fundamental concepts in concrete analysis and design that have been previously studied.

Analysis of the singly reinforced rectangular beams

For under reinforcement section
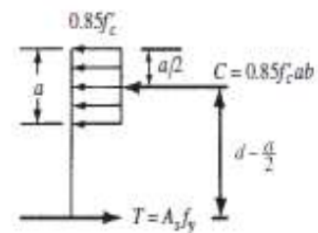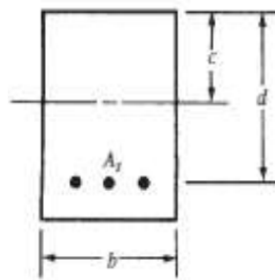
When $\rho < \rho$ b.

C=T

$0.85f'_c \, \alpha b = A_s f_y \quad \rightarrow$

$\alpha = \dfrac{A_s f_y}{0.85 f c' b} \quad$ where $\rho = \dfrac{A_s}{b.d}$

$M_n = T\left(d - \dfrac{\alpha}{2}\right) = A_s f_y (d - \dfrac{1}{2}\dfrac{A_s f_y}{0.85 f c' b})$

$\emptyset M_n = \emptyset b d^2 f_y \rho (1 - 0.59\rho \dfrac{f_y}{f_c})$

$\rho = \dfrac{(1 - \sqrt{1 - \dfrac{2.36 M_u}{\emptyset b d^2 f c'}})}{(1.18\dfrac{f_y}{f c'})}$

Example: Check the section below for flexural requirements? Assume L.L=18 kN/m, D.L(within self weight)= 8 kN/m ,fy=300 Mpa, fc'=35 Mpa

◈ $Mu(ext.)=\dfrac{1}{2}wl^2$

◈ $Wu = 1.6 \times L + 1.2 \times D = 38.4 \ kN/m$

◈ $Mu(ext.)=(38.4\times3^2)/2=172.8 \ kN.m$

◈ $M_u \ (int.) = \emptyset M_n = \emptyset b d^2 f_y \rho \left(1 - 0.59\rho \dfrac{f_y}{f_c}\right)$

check $\emptyset$

$\rho = \dfrac{As}{bd} = \dfrac{3 \times 491}{350 \times 600} = 0.007$

$\rho t = 0.85\beta_1 \dfrac{fc}{fy}\dfrac{\varepsilon u}{\varepsilon u + 0.005}$
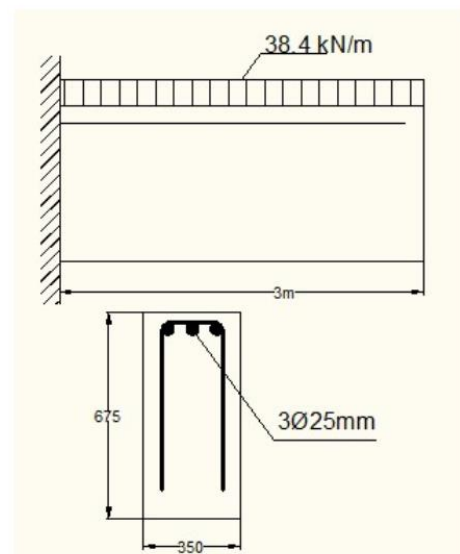
◈ Since fc' >28 Mpa

◈ so $\beta_1 = 0.85 - \dfrac{0.05}{7}(35 - 28) = 0.80$

$\rho t = 0.85\beta_1 \dfrac{fc'}{fy}\dfrac{\varepsilon u}{\varepsilon u + 0.005} = 0.03$, since P<Pt , So ø=0.9

$M_u \ (int.) = 229.7 \ kN.m$

◈ $Mu(ext.) < M_u \ (int.) , O.K.$

Design of singly reinforced Concrete rectangular beam

◆ For unknown dimension.

Find ρ_{max}, ρ_{min} and choice ρ between that values and preferred used $\rho \leq \rho_t$. for used largest reduction factor ($\varnothing=0.9$).

$$\rho_{min\cdot} = \frac{\sqrt{fc}}{4 \times fy} \geq \frac{1.4}{fy}, \quad \rho_{max} = 0.85\beta_1 \frac{fc\prime}{fy} \frac{\varepsilon u}{\varepsilon u + 0.004}, \quad \rho_t = 0.85\beta_1 \frac{fc\prime}{fy} \frac{\varepsilon u}{\varepsilon u + 0.005},$$

$$M_u = \varnothing M_n = \varnothing bd^2 f_y \rho \left(1 - 0.59\rho \frac{fy}{fc\prime}\right)$$

$$bd^2 = \frac{Mu}{\varnothing Fy\rho_t (1 - 0.59\rho_{t,max}\frac{Fy}{Fc\prime})}$$

◆ For known dimension.

◆ $M_u = \varnothing M_n = \varnothing bd^2 f_y \rho \left(1 - 0.59\rho \frac{fy}{fc\prime}\right) = \varnothing bd^2 R$

◆ $R = f_y \rho \left(1 - 0.59\rho \frac{fy}{fc\prime}\right) \frac{2}{2} = f_y \rho \left(\frac{2-\rho m}{2}\right) \quad R = \frac{Mu}{\varnothing bd^2} \quad m = \frac{fy}{0.85 fc\prime} \quad p^2 fym - 2\rho fy + 2R = 0$

◆ $\rho = \frac{1}{m}\left(1 - \sqrt{1 - \frac{2mR}{fy}}\right)$ OR use direct formula $\rho = \frac{(1-\sqrt{1-\frac{2.36M_u}{\varnothing bd^2 fc\prime}})}{(1.18\frac{fy}{fc\prime})}$

Example:A beam has width 300mm and h=700 subjected to dead moment within weight of the beam, Md= 100 kN.m, and live moment=150 kN.m. Calculate the area of steel reinforcement and the number of the bars fc'=20 Mpa, fy=400 Mpa

◆ Mu=1.2*Md+1.6*ML=1.2*100+160*150=360 kN.m

◆ ρ_{min}=0.0035 ρ_{max}=0.0155

◆ d=700-65=635 mm Pmin < P < Pmax

◆ Assume ø=0.9 and check later

◆ $\rho_{requ} = \frac{1-\sqrt{1-\frac{2.36}{\varnothing Fc\prime \times bd^2}}}{1.18\frac{Fy}{Fc}}=0.0096$

◆ As=ρbd=1772 mm2 ρ_t=0.0136 >ρ_{req} =0.0096   so ø=0.9

◆ Try the diameter ø=25 mm   the area of the bar =491 mm2

◆ $N = \frac{1772}{491} = 3.6 \approx 4$ use 4ø25 mm

◆ $S = \frac{300-100-4\times25}{3} = 33 > 25\ mm$

700

4Ø25

300
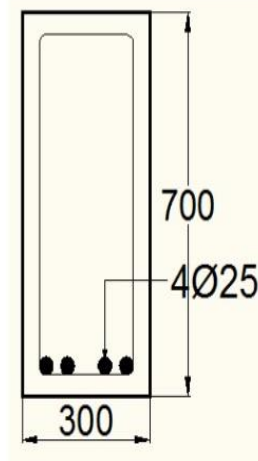
STAAD pro

STAAD stands for Structural Analysis and Design. STAAD.Pro software is widely used in analyzing and designing structures – buildings, bridges, towers, industrial and utility structures. Designs can involve building materials like steel, concrete, timber, aluminum, and cold-formed steel. You can confidently design structures anywhere in the world using over 80 international codes.

The functions performed by this program

1-Reduce project costs and delays: Provide accurate and economical designs to your clients and quickly turnaround change requests.

2-Design and analyze simple or complex structures for a wide range of loading conditions, such as dead and live loads, in combination with lateral loads including wind and seismic.

3- Complete building analysis, design, and drafting for the entire structure accurately and efficiently using our state-of-the-art finite element analysis. Reduce or eliminate the time spent waiting for results using our fast solvers.

4- Quickly model your entire structure, including slabs, and openings, beams, columns, walls, and continuous footings, and pile caps. Efficiently automate many of your time-consuming design and analysis tasks and produce practical system and component designs.

5-Optimize or analyze beams, columns, and walls for lateral loads to quickly obtain safe and economical designs. Confidently produce designs in compliance with I.S requirements and many international design specifications and building codes.

STAADPro is a general purpose structural analysis and design program with applications primarily in the building industry - commercial buildings, bridges and highway structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, culverts and other embedded structures, etc. The program hence consists of the following facilities to enable this task.

1. Graphical model generation utilities as well as text editor based commands for creating the mathematical model. Beam and column members are represented using lines. Walls, slabs and panel type entities are represented using triangular and quadrilateral finite elements. Solid blocks are represented using brick elements. These utilities allow you to create the geometry, assign properties, orient cross sections as desired, assign materials like steel, concrete, timber, aluminum, specify supports, apply loads explicitly as well as have the program generate loads, design parameters etc.

2. Analysis engines for performing linear elastic and pdelta analysis, finite element analysis, frequency extraction, and dynamic response (spectrum, time history, steady state, etc.).

3. Design engines for code checking and optimization of steel, aluminum and timber members. Reinforcement calculations for concrete beams, columns, slabs and shear walls. Design of shear and moment connections for steel members.

4. Result viewing, result verification and report generation tools for examining displacement diagrams, bending moment and shear force diagrams, beam, plate and solid stress contours, etc.

5. Peripheral tools for activities like import and export of data from and to other widely accepted formats, links with other popular soft wares for niche areas like reinforced and pre stressed concrete slab design, footing design, steel connection design, etc.

The analysis and design procedures in STAAD Pro software consist of three main stages

1. Model Creation (Modeling): In this stage, you create the structural model of the system you want to analyze. This involves defining the geometry, material properties, support conditions, and loads applied to the structure. The modeling phase is essential as it sets up the initial representation of the structure for analysis.

2. Model Analysis (Solving): Once the model is created, you perform various types of analyses, such as static analysis, dynamic analysis, or finite element analysis, depending on your project's requirements. During this phase, the software calculates the response of the structure to the applied loads, considering factors like stress, deformation, and stability. The analysis phase helps in understanding how the structure behaves under different conditions.

3. Results Display and Verification: After completing the analysis, you can visualize and interpret the results obtained from the software. STAAD Pro provides various tools for displaying results, including graphical representations, tables, and reports. This stage is crucial for verifying whether the design meets the desired safety and performance criteria. It allows engineers to make informed decisions and make any necessary modifications to the structure's design.

The main components of the start page
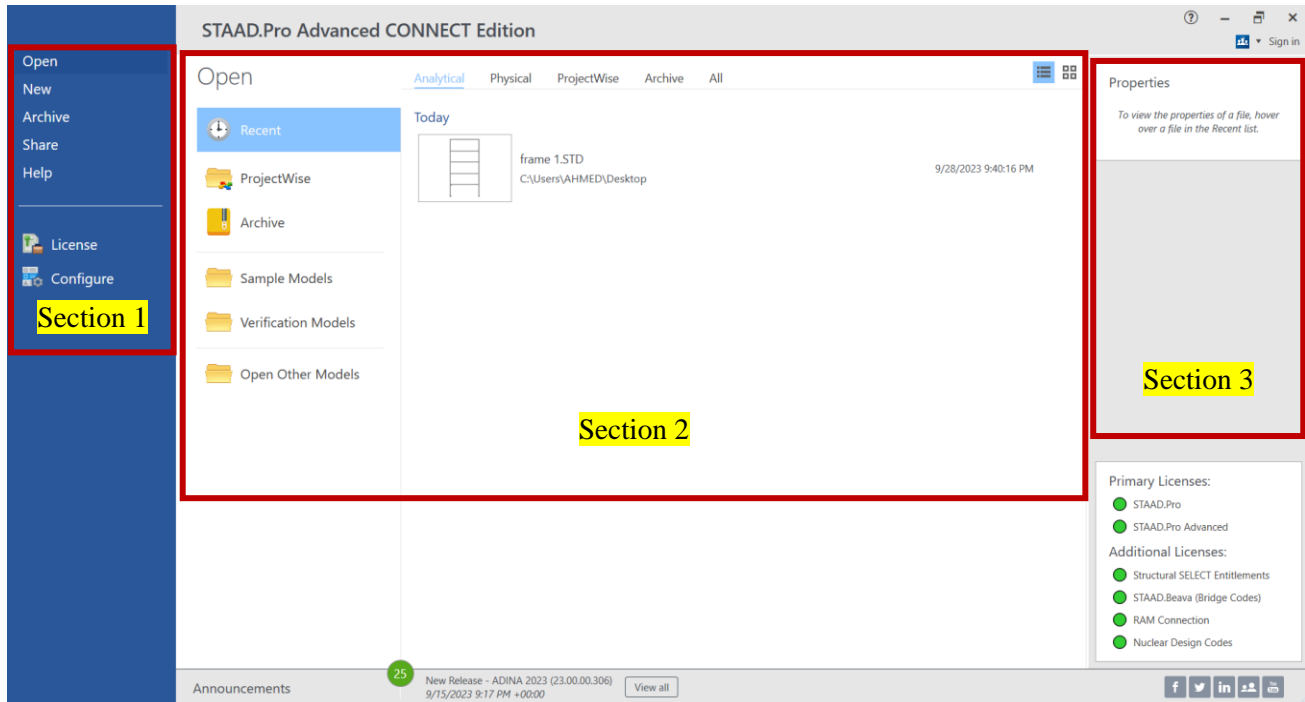
Section1: includes several commands related to files, licensing, and configuring input settings for the program as fallowing

1- Open (To open any previous project that has been worked on, the details of previous projects are displayed in the section 2)
2- New (There are seven step to make a new project)
   1 click on the new item. Then, enter the following information:
   2 .File Name: Storage Shed_1.std
   3 .Location: (navigate to the dataset folder that was supplied with this training)
   4 .Type: Analytical
   5 .Units: Metric
   6 .Job Info: Optional input for job information
   7. Then, click on the Create button.
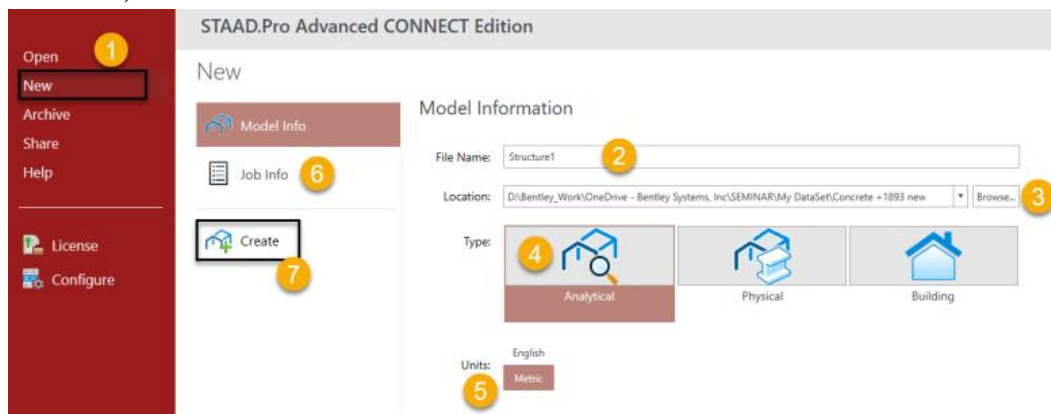
3- Archive (It is used for archiving projects within the program, including creat files from the computer into the program or extract them)
4- Share
5- Help
6- License (This window is dedicated to program licenses)
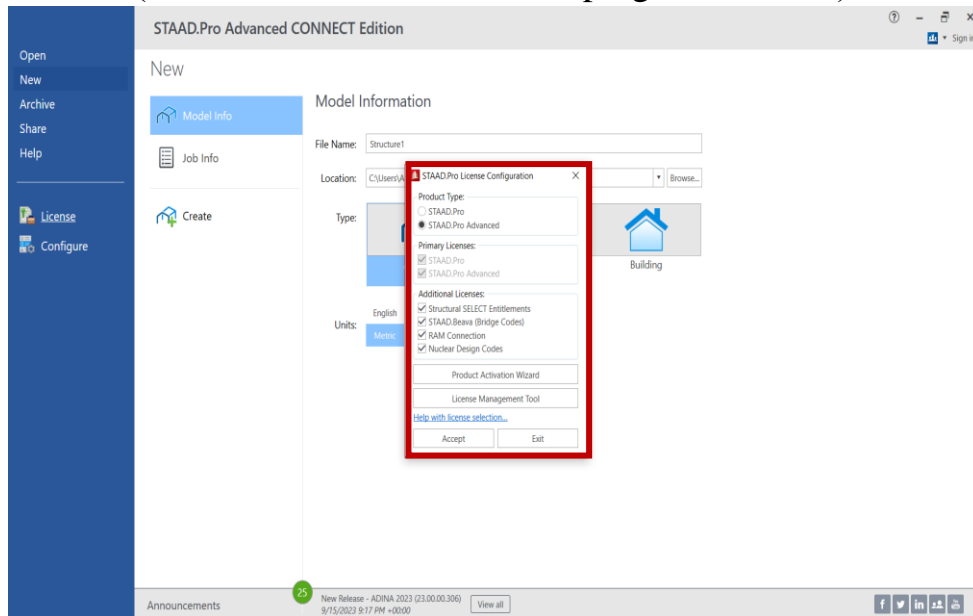
7- Configure: This window includes a set of settings, the most important of which are
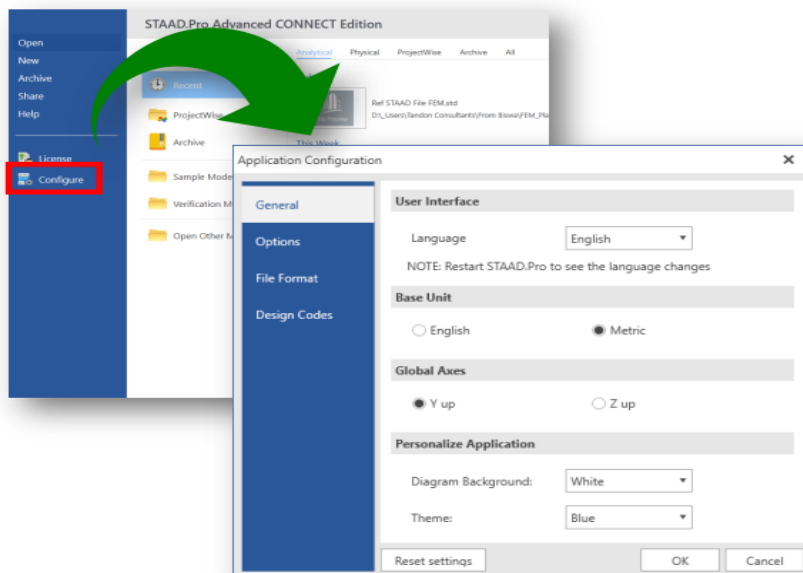
A- Language
B- Base unite (matric or English)

C- Global axis the default orientation of the global coordinate system is that the Y axis points in the vertical direction and a plan view is represented by the XZ plane.

You do have an option to specify the vertical axis as the Z axis, but this option will eliminate the ability to use some other functions in the program, such as the wind load generator)
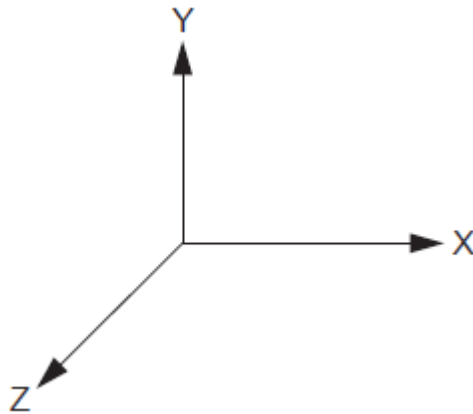
D- Diagram background
E- Theme
F- Default design codes
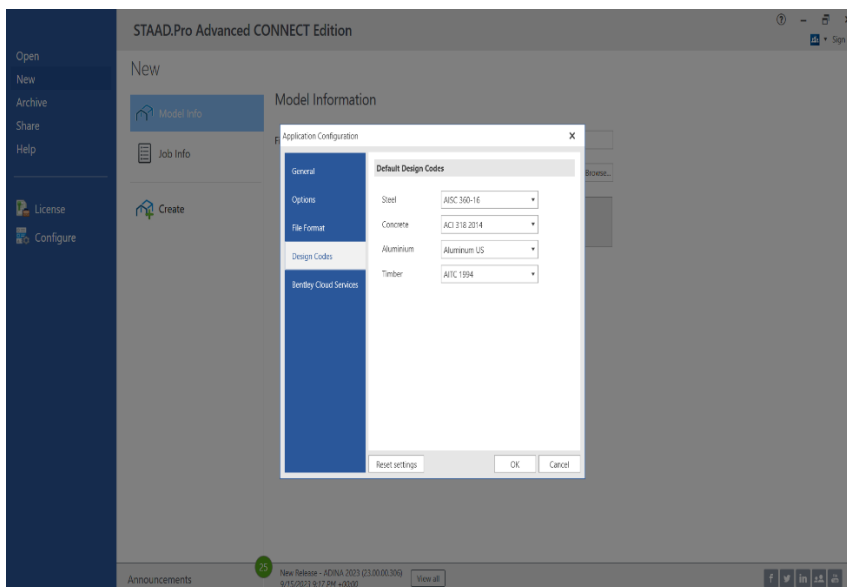
Section 2

The display in this section changes depending on the command selected in the first section. If the command is to **open** a project, it will show recent projects that have been worked on and provide options to import projects from the computer. It also indicates the type of project, whether it's (*analytical, physical, project wise or all*)
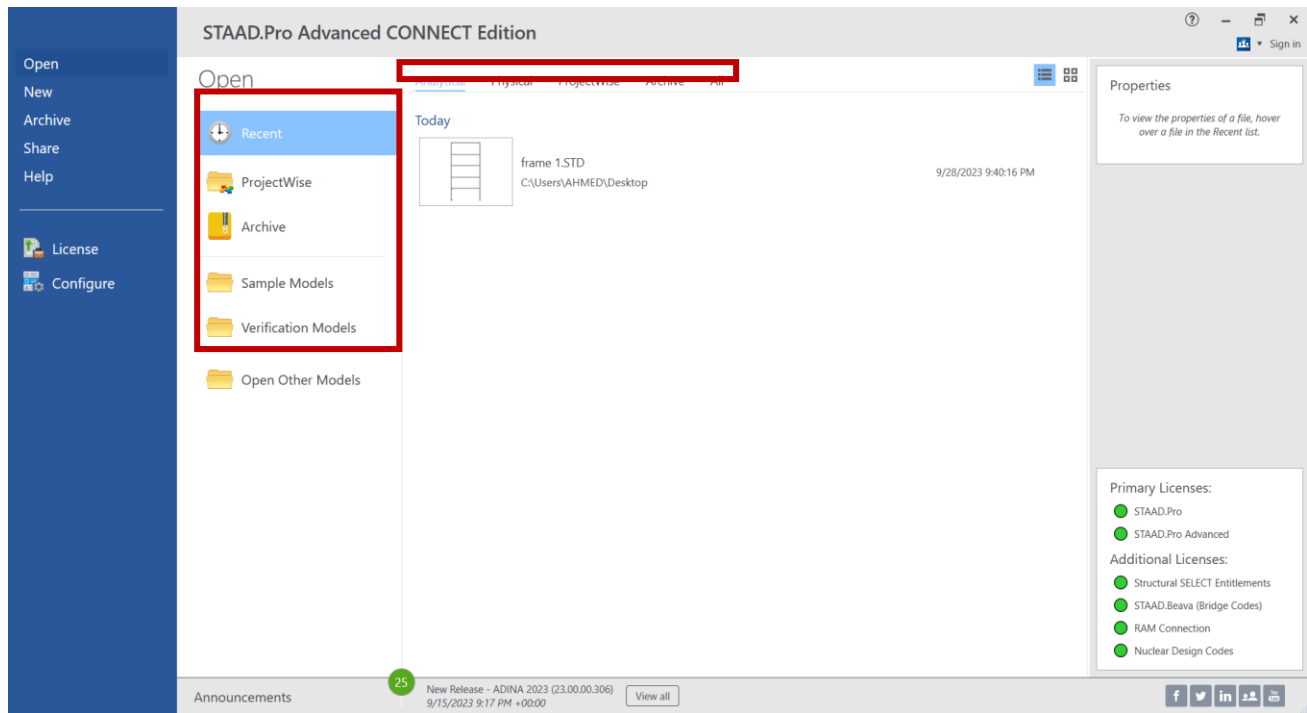
If the command is to create a **new** project from the first section, the display will be different, as explained in the second paragraph of the first section

Section 3: display some details related to the information of previously worked-on projects.

The user interface