DIGITAL EDUCATION
MARTCOST

CIVIL ENGINEERING

Introduction of E-tabs

E-TABS & STAAD FILE

Page Length : 18

INTRODUCTION ON ETABS

- Various Civil softwares available in the market are **STAAD Pro**, STRUTS, STRAP, STRUDS, RISA, GTS, **ETABS**, NISA, **SAP 2000**, ANSYS, ABAQUS, Msc Natran, PLAXIS and **SAFE**.
- Among them STAAD Pro and ETABS are leading design software in the market.
- ETABS latest version available is ETABS-2018

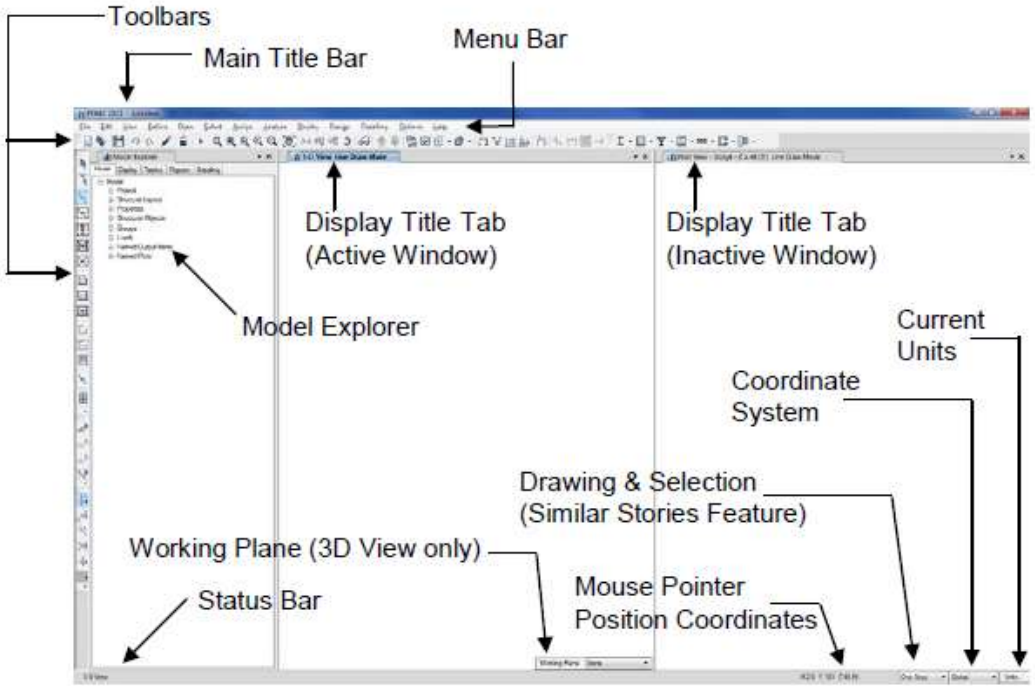
- ETABS is special purpose analysis and design program developed specially for Building Systems.
- **ETABS** is short form for **E**xtended **T**hree-dimensional **A**nalysis of **B**uilding **S**ystem.
- ETABS uses Stiffness Matrix method of structural analysis for members and FEA for Plates.
- 3D Rendering is available in ETABS.
- ETABS has time saving option (One storey, All storey, Similar storey).
- ETABS has built in grid No. A,B,C,D. 1,2,3
- It has two screen in default. Left side shows plan view and Right side shows 3D Isometric view or elevation. We can create 3 or more screen as desired using option available in ETABS.
- In ETABS Beams and Columns are known as line objects. Slabs and Shear walls are known as Area objects.
- It has built in template.
- It has built in code books.
- It will calculate loads automatically on beams and columns.
- It is easy to give floor load for irregular panels.
- It performs wind load and earthquake analysis.
- Automatic lumping of masses for earthquake.

- Diaphragm action.
- Choice of Eigen or Ritz vector for Response Spectrum Analysis.
- Ductile detailing as per IS 13920.
- Auto calculation of beam reinforcements based on moments at column face, rather than at column centre line and column reinforcements based on moments at beam soffit, rather than at beam centre line.
- Design of columns of cross section shape including T, L, +.
- It design shear walls.
- It displays reinforcement areas or percentage on each beam and column.
- Construction sequence analysis and Pushover analysis can be done by ETABS.
- It has in built Live load reduction for upper floors as per codal provision.
- Additional software for component design specially foundation design using SAFE software(Mat foundaion, PT slab design).

Disadvantages:

- It doe not support for sloping roofs.
- It cannot generate floor loads. You have to model slabs. You have to model slabs as plate element & mesh it and apply floor loads on it.
- You canot know what is happening inside the analysis engine of the software as like STAAD Pro.
- The report documentation is not so good when compare to STAAD Pro's report.

ETABS Screen:



Title Bar: It is on the top of the screen. When the programme is in use, this title bar is highlighted. It includes program and model name.

Menu Bar : It contains program's menus from which various commands can be selected to perform specific actions.

Tool Bars and Buttons: The tool bars are made up of buttons placed at top and side of the screen. We can access commonly used commands by single click on the button.

Model Explorer: It allows easy access to model definition data including Property forms, load definitions and object forms as well as Analysis, Design, Detailing results in graphical.

Display windows: It shows geometry of the model. The display includes property, loading, analysis or design results and detailing .

Display title tab: It is located at top of display window. It is highlighted when the associated display window is active.

Status bar: It is located at bottom of main window. Text describes current status of the program.

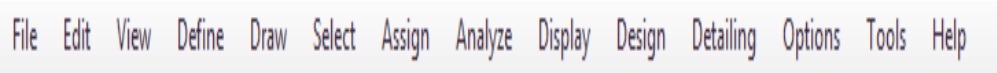
Mouse pointer position coordinates : it displays on the right side of status bar.

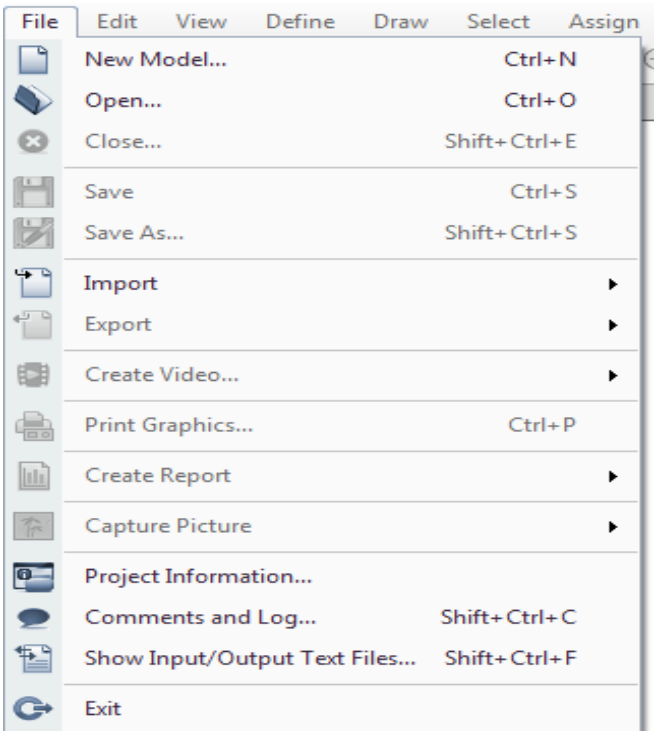
Drawing & selection Drop down list: It is on the right side of status bar. One Storey, All Stories, Similar Stories are the three options in the drop down menu. With One Storey option, an object is created only at the storey level on which it is drawn. All Stories option enabled to create objects at all storey levels in the model at the same plan location. Similar storey option enabled to create objects at all similar storey levels in the model.

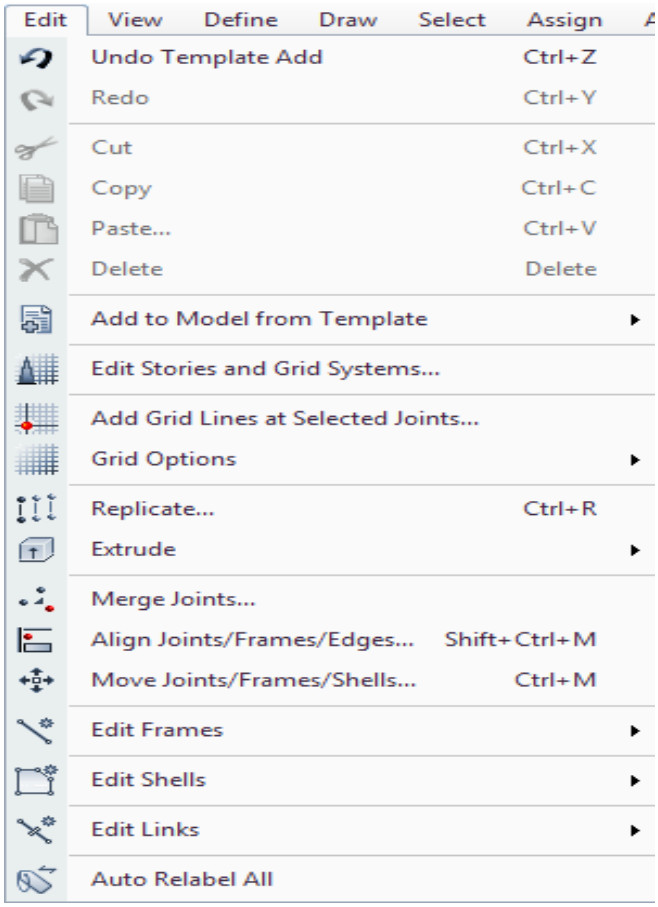
Current units: It is located on the far right side of the status bar. This can be changed at any time during model creation process.

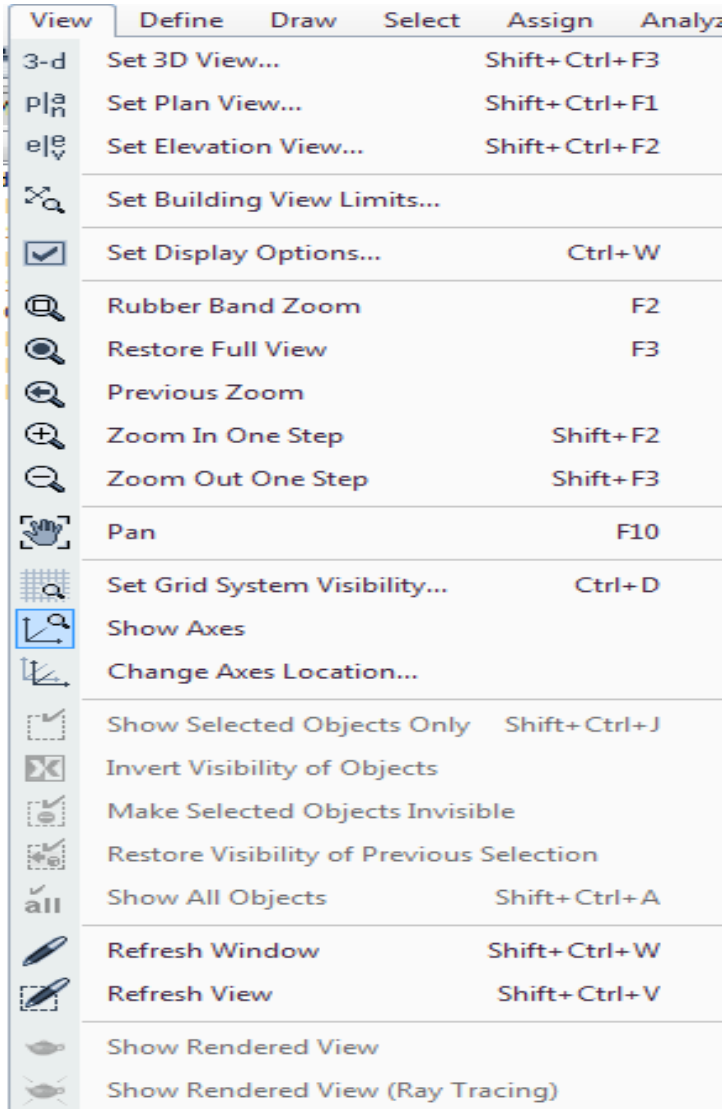
The various **Main menu** and sub menus are as Follows:

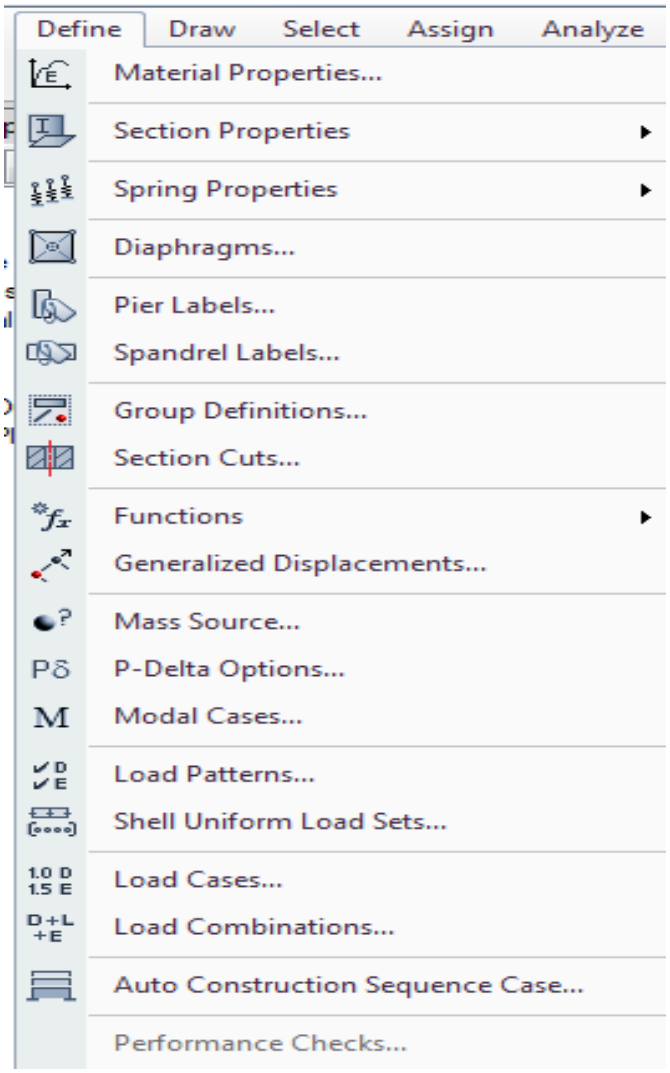
Main Menus

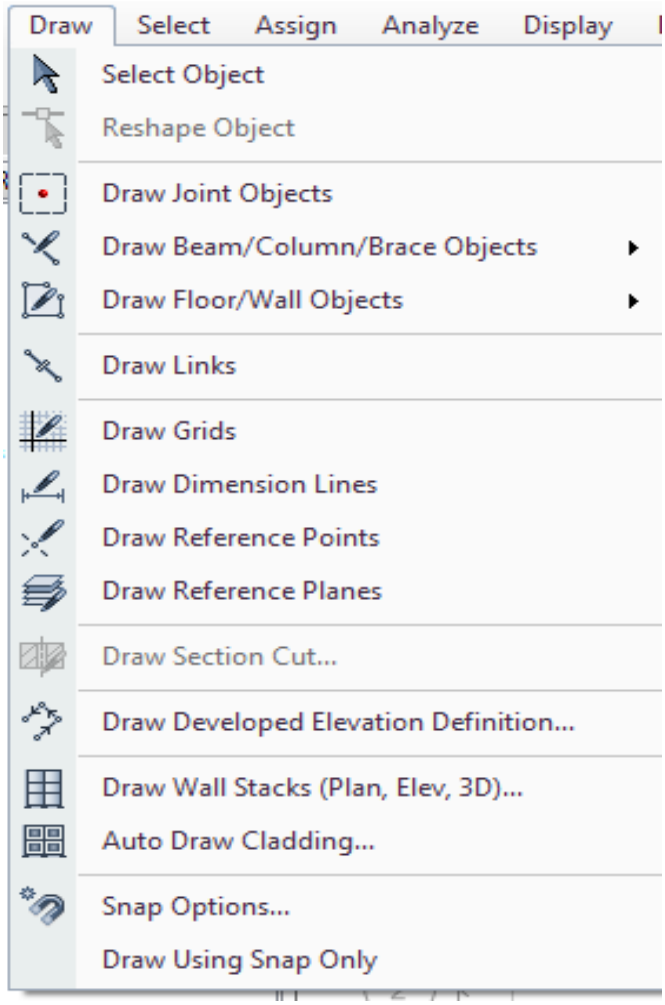


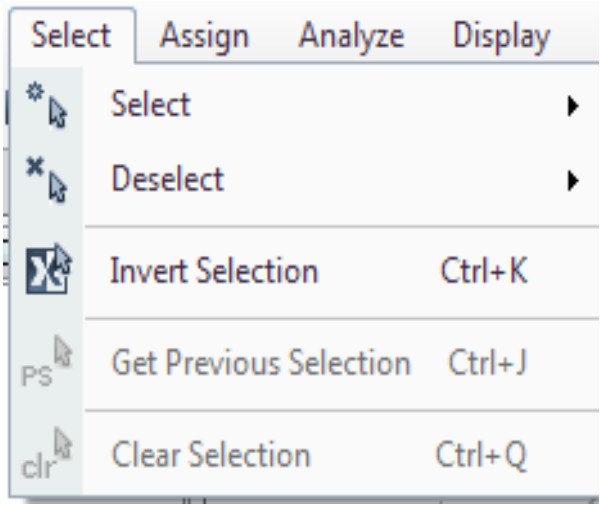


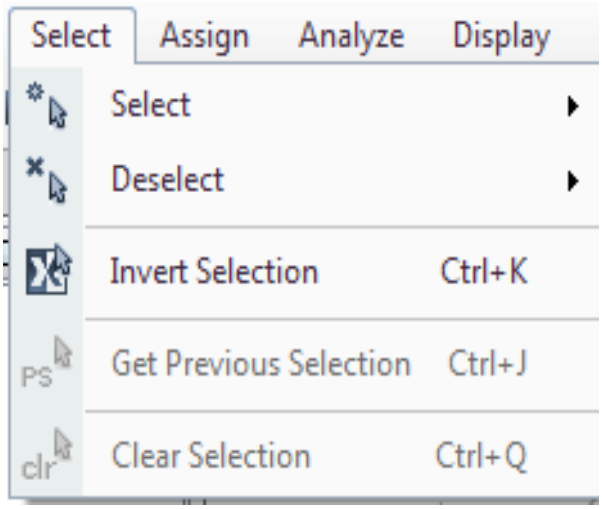

Edit
View
Define
Draw
Select
Assign
Undo Template Add Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Paste... Ctrl+V
Delete Delete
Add to Model from Template
Edit Stories and Grid Systems...
Add Grid Lines at Selected Joints...
Grid Options
Replicate... Ctrl+R
Extrude
Merge Joints...
Align Joints/Frames/Edges... Shift+Ctrl+M
Move Joints/Frames/Shells... Ctrl+M
Edit Frames
Edit Shells
Edit Links
Auto Relabel All

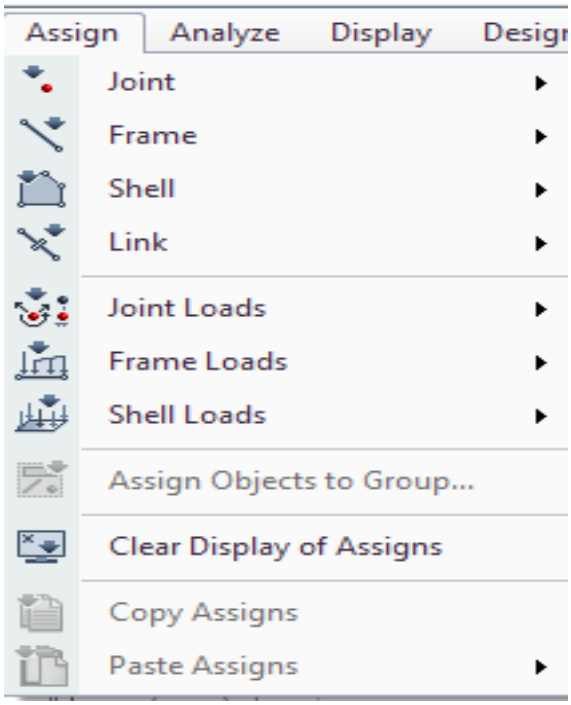

View
Define
Draw
Select
Assign
Analyz
Set 3D View... Shift+Ctrl+F3
Set Plan View... Shift+Ctrl+F1
Set Elevation View... Shift+Ctrl+F2
Set Building View Limits...
Set Display Options... Ctrl+W
Rubber Band Zoom F2
Restore Full View F3
Previous Zoom
Zoom In One Step Shift+F2
Zoom Out One Step Shift+F3
Pan F10
Set Grid System Visibility... Ctrl+D
Show Axes
Change Axes Location...
Show Selected Objects Only Shift+Ctrl+J
Invert Visibility of Objects
Make Selected Objects Invisible
Restore Visibility of Previous Selection
Show All Objects Shift+Ctrl+A
Refresh Window Shift+Ctrl+W
Refresh View Shift+Ctrl+V
Show Rendered View
Show Rendered View (Ray Tracing)

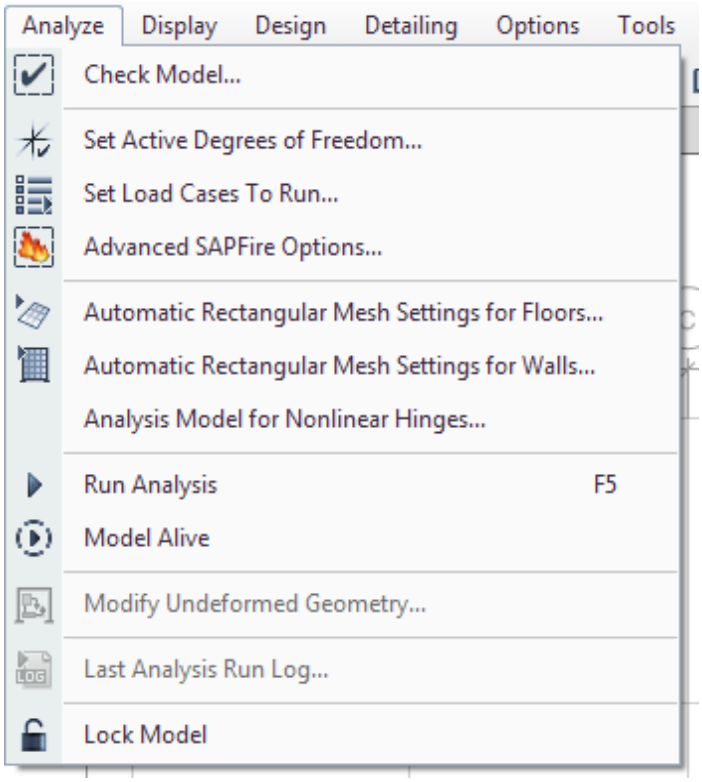

Define
Draw
Select
Assign
Analyze
Material Properties...
Section Properties
Spring Properties
Diaphragms...
Pier Labels...
Spandrel Labels...
Group Definitions...
Section Cuts...
Functions
Generalized Displacements...
Mass Source...
P-Delta Options...
Modal Cases...
Load Patterns...
Shell Uniform Load Sets...
Load Cases...
Load Combinations...
Auto Construction Sequence Case...
Performance Checks...

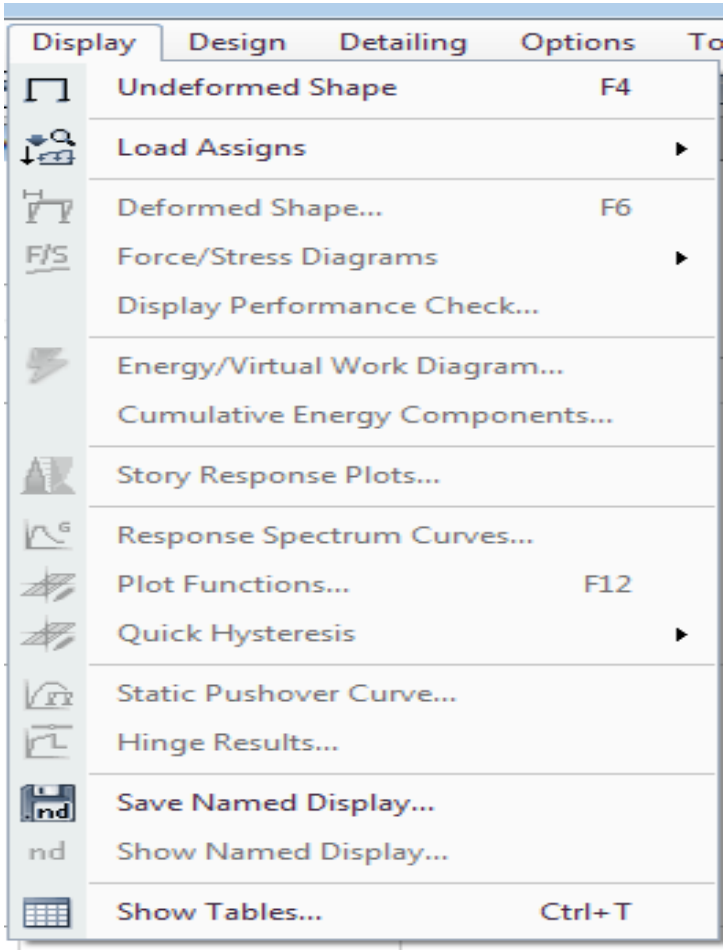

Draw
Select
Assign
Analyze
Display
Select Object
Reshape Object
Draw Joint Objects
Draw Beam/Column/Brace Objects
Draw Floor/Wall Objects
Draw Links
Draw Grids
Draw Dimension Lines
Draw Reference Points
Draw Reference Planes
Draw Section Cut...
Draw Developed Elevation Definition...
Draw Wall Stacks (Plan, Elev, 3D)...
Auto Draw Cladding...
Snap Options...
Draw Using Snap Only

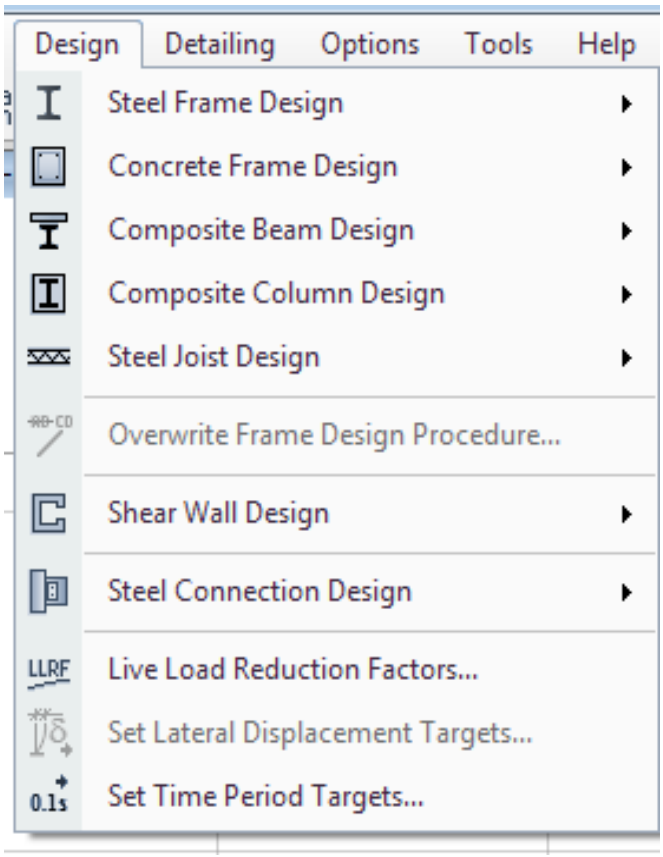

Select
Assign
Analyze
Display
Select
Deselect
Invert Selection Ctrl+K
Get Previous Selection Ctrl+J
Clear Selection Ctrl+Q

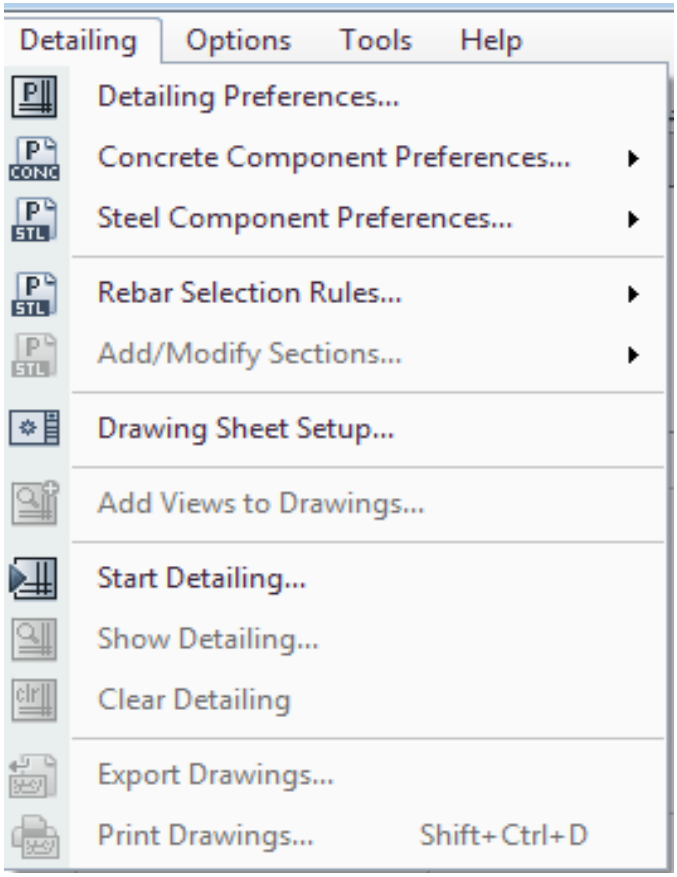

Select
Assign
Analyze
Display
Select
Deselect
Invert Selection Ctrl+K
Get Previous Selection Ctrl+J
Clear Selection Ctrl+Q

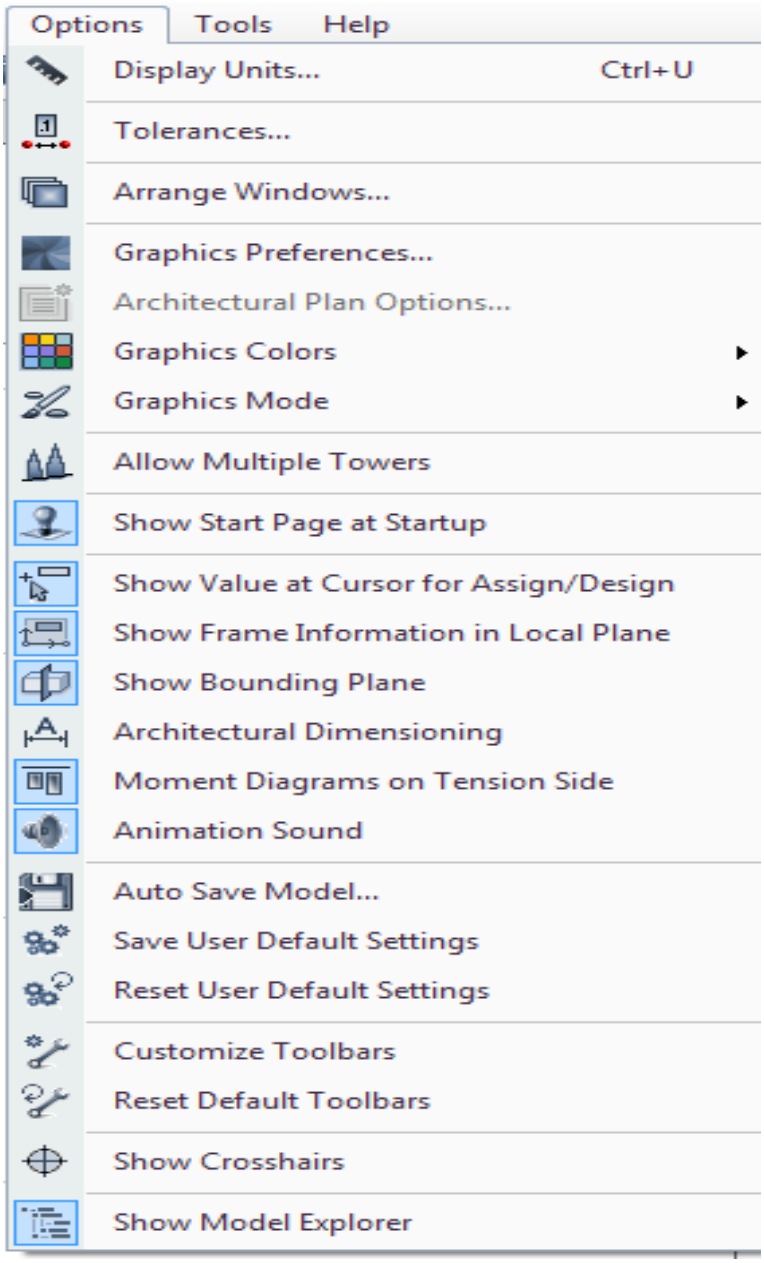

Assign
Analyze
Display
Desig
Joint
Frame
Shell
Link
Joint Loads
Frame Loads
Shell Loads
Assign Objects to Group...
Clear Display of Assigns
Copy Assigns
Paste Assigns

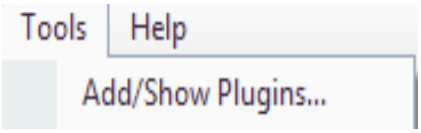

Analyze
Display
Design
Detailing
Options
Tools
Check Model...
Set Active Degrees of Freedom...
Set Load Cases To Run...
Advanced SAPFire Options...
Automatic Rectangular Mesh Settings for Floors...
Automatic Rectangular Mesh Settings for Walls...
Analysis Model for Nonlinear Hinges...
Run Analysis F5
Model Alive
Modify Undeformed Geometry...
Last Analysis Run Log...
Lock Model


Display
Design
Detailing
Options
To
Undeformed Shape F4
Load Assigns
Deformed Shape... F6
Force/Stress Diagrams
Display Performance Check...
Energy/Virtual Work Diagram...
Cumulative Energy Components...
Story Response Plots...
Response Spectrum Curves...
Plot Functions... F12
Quick Hysteresis
Static Pushover Curve...
Hinge Results...
Save Named Display...
Show Named Display...
Show Tables... Ctrl+T


Design
Detailing
Options
Tools
Help
Steel Frame Design
Concrete Frame Design
Composite Beam Design
Composite Column Design
Steel Joist Design
Overwrite Frame Design Procedure...
Shear Wall Design
Steel Connection Design
Live Load Reduction Factors...
Set Lateral Displacement Targets...
Set Time Period Targets...

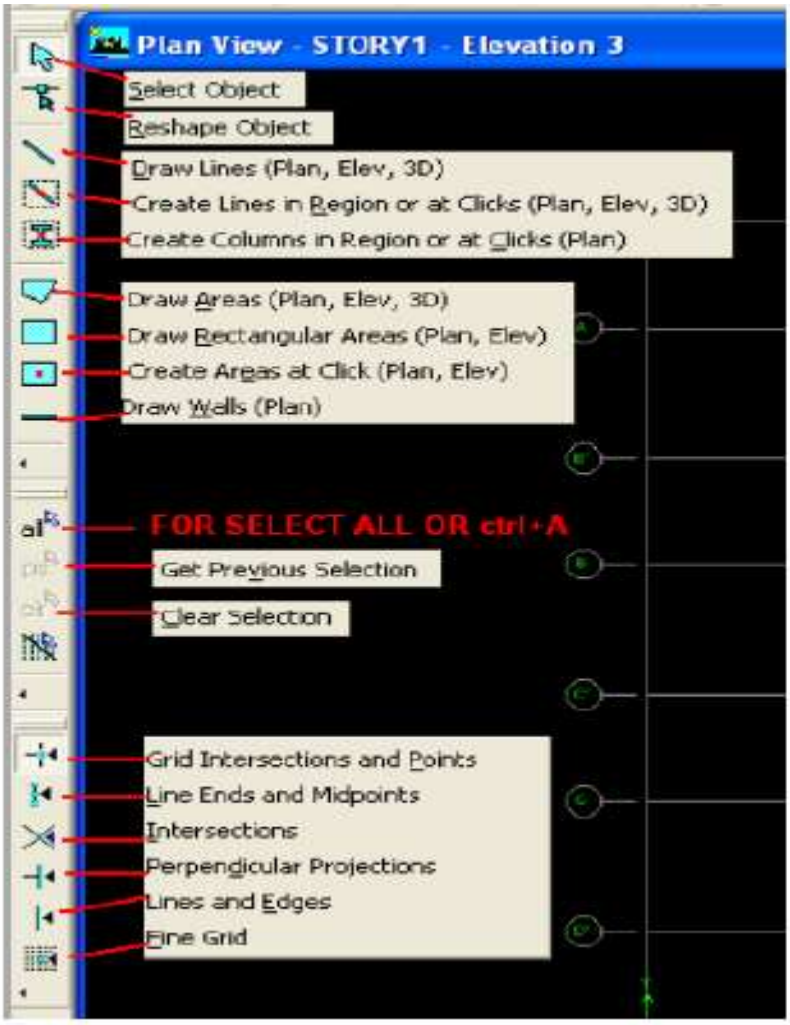

Detailing
Options
Tools
Help
Detailing Preferences...
Concrete Component Preferences...
Steel Component Preferences...
Rebar Selection Rules...
Add/Modify Sections...
Drawing Sheet Setup...
Add Views to Drawings...
Start Detailing...
Show Detailing...
Clear Detailing
Export Drawings...
Print Drawings... Shift+Ctrl+D


Options
Tools
Help
Display Units... Ctrl+U
Tolerances...
Arrange Windows...
Graphics Preferences...
Architectural Plan Options...
Graphics Colors
Graphics Mode
Allow Multiple Towers
Show Start Page at Startup
Show Value at Cursor for Assign/Design
Show Frame Information in Local Plane
Show Bounding Plane
Architectural Dimensioning
Moment Diagrams on Tension Side
Animation Sound
Auto Save Model...
Save User Default Settings
Reset User Default Settings
Customize Toolbars
Reset Default Toolbars
Show Crosshairs
Show Model Explorer


Tools
Help
Add/Show Plugins...

Tool bars :

Left Side Tool bars



Top Tool bars:

New model, Open, Save, Undo, Redo, Refresh Window, Run analysis (F5), Rubber band zoom, Restore Full view, Restore previous zoom, Zoom in one step, Zoom out one step, Pan, Set Default 3D view, Set Plan view, Set Elevation view, Rotate 3D view, Perspective Toggle, Above Up, Move down, Object shrink Toggle, Set display Options, Extruded view Toggle, transparency Toggle, Show Un deformed shape(F4), Display Load joint Assign, Display Framed load Assign, Display Shell load Assign, Show Deformed shape(F6), Display Support/Spring Reaction(F7), display frame/Pier/ Spandrel/link force(F8), Display Shell Stresses. Forces (F9), Show named display, Steel Frame Design, Concrete Frame Design Composite Beam Design,, Composite Column Design, Steel Joist Design, Shear Wall Design, Steel Connection Design.

:

Comparison of STAAD Pro & ETABS

S.No.	Description	STAAD Pro	ETABS
1	Model Creation	Any type of structure can be analysed & design with its flexible modeling environment	It has inbuilt library for modeling building structures only.
2	3D rendering view	It shows 3D rendering view to some extent only.	It shows 3D rendering view with real good effect on the structure.
3	Structural analysis	Stiffness Matrix method for structures and FEM for plate elements.	It uses Stiffness Matrix for structural analysis For members & FEA for plates.
4	Bending moment & Shear forces	It gives BMD & Shear forces to the centre line of columns	BMD & Shear forces are calculated from face of column accurately.
5	Floor Loading	Difficult to apply floor load for odd shape.	Easy to apply floor load even for irregular panels.
6	Floor loads	It can generate Floor Loads easily.	It cannot generate Floor Loads. You have to model slabs as plate element & mesh it and apply floor loads on it.
7	Live load reduction	It does not have Live Load Reduction facility. We have to define command prompt to do the same.	Live load Reduction for upper floors is available.
8	Earthquake analysis	It cannot generate Earthquake lumping of masses.	Auto lumping of masses for earthquake. Earthquake force can be applied on any angle (Not just X or Y alone.
9	Time saving options	Member property, loading can be given by cut section with range by selecting new view.	It has time saving options One storey, Similar storey and All storey.

S.No.	Description	STAAD Pro	ETABS
10	Column design	Column design with cross section of T,L, + shape are not possible.	Design columns with cross section of any arbitrary shape(including T,L, +)
11	Schematic representation of Reinforcement details	It does not have schematic representation of reinforcement on columns & beams	It display reinforcements area, % of steel on column & beam frames.
12	Diaphragm action	It does not have diaphragm facility for wind & earthquake automatically	It have diaphragm action(with earthquake loads & wind loads automatically applied on the diaphragm centre)
13	Report	It gives analysis & design report documentation in a good manner.	It does not give such documentation report.