

HELLO ETABS

ETABS stands for extended three-dimensional Analysis of Building Systems. It is a very powerful programme that can perform variety of analysis and design. It is used worldwide as it is simple to use and offers large number of options with variety of features. It is developed to cater the need of design of building systems specifically. It can handle tall and sophisticated building system's model easily. Latest version of this programme offers the non linear analysis and performance based designs thus making it a favorite for structural engineers community. It has evolved over the span of 40 years from TABS to ETABS.

The algorithms used in TABS were no doubt different from ETABS but so are the computers. Faster and higher processing power computers have given privilege to developers to add more complex analytical options and completely integrate it that would not had been possible few decades back.

Housing several international codes in its core, it helps the user to analyze any kind of building system with various country codes and compare the results. Both 2-D and 3-D models can be analyzed depending upon the user requirements.

THINK AND IT WILL DO IT

The structural members modelled in the programme can be of various shapes. Fixidity can be partial or complete or none depending upon the end conditions.

Loads can be uniform , triangular or trapezoidal ,point depending upon the loading conditions. If the structure under consideration has to bear temperature difference or extreme temperatures, load for temperature can be applied.

Name the analysis and this programme can do it. Response spectrum, linear and nonlinear time history, pushover, performance based design and analysis.

Various types of building structures can be analyzed through this programme. They can be

- Multi story residential buildings
- Multi story commercial complexes
- Multi story parking garages.
- Steel PEB sheds
- Steel Equipment supporting structures
- Pipe Racks
- Steel and RCC girders
- Very tall buildings with complex shear wall arrangement
- Circular or curved buildings
- Flat slab buildings
- Buildings having waffle slab arrangement
- Steel building connection designs
- Much required performance based design
- Non linear static or dynamic analysis of building ssystems
- P-Delta analysis , both static and dynamic
- Eigen vector or rtiz vector analysis
- Linear and non linear time history aanlysis Building ssystems
- Foundation settlement analysis
- Non linear static push over analysis
- Yes , Base isolators and dampers also!!
- Base plate design of columns

The list never ends

PROCESS OF MODELLING

ETABS offers object based programming of structures. By object based programming, It means that beams and columns becomes frames, floors , ramp and walls becomes shell , springs becomes links.

Objects represents the structural members modelled by the user as an input. Elements are the finite elements developed by the programme algorithms internally to generate the much required stiffness matrix. For correct analysis, objects need to as close to the real structure to be analyzed and designed. Small discrepancy can also lead to the result being different from the real life.

STORY DEFINITION

ETABS offers user to define story levels in a user-friendly manner.

Building system in this programme can consists of combination of shell, frame, joint, tendon or links. Each of these represents a structural type.

Shell – Structural Slab or Diaphragm (Can be semi rigid to consider in-plane deformation)

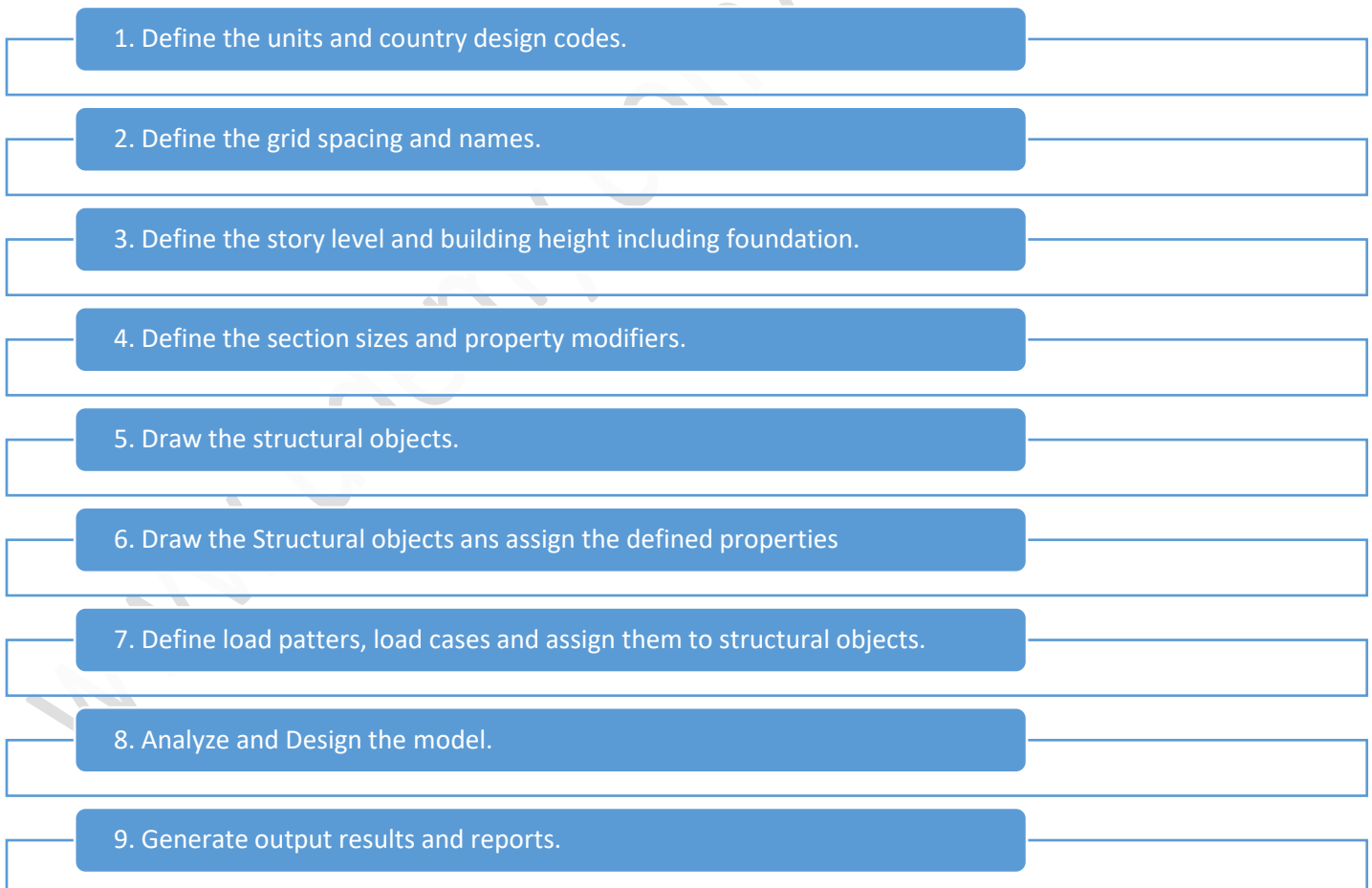
Frame – Structural beam or girder

Link – Isolators or special supporting arrangement

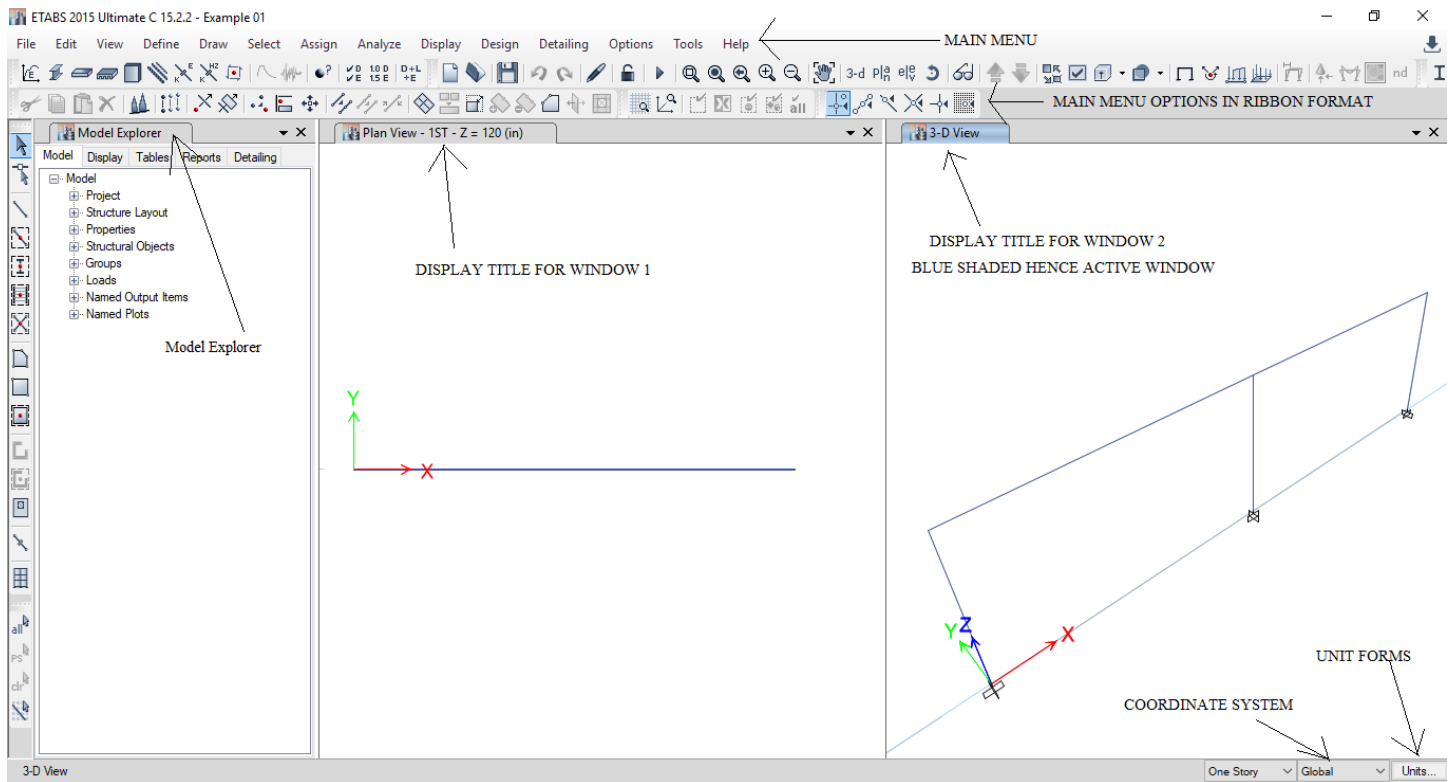
Tendon – Strands

Basic methodology of using this programme includes drawing grid lines, placing structural objects, assigning loads, analyzing, designing and detailing the structure. Results are generated in graphical or tabulator form.

Basic tabular chart of working on this software is as follows.

FLOW CHART OF WORK FLOW

TIP: At any stage, for help press F1 function key.

UNDERSTANDING ENVIRONMENT OR GRAPHICAL USER INTERFACE

1. **Coordinate System** – ETABS offers user to switch between Global Coordinate System, which acts as default, and User Defined Grid system. User defined grid system is activated through this form. Model orientation and position coordinates are different for both coordinate systems.
2. **Unit Forms** – This is the best feature of this programme that provides much required comfort to the user and save him from applying conversion factors while working in different unit. User can change the units at any time of model creation process.
3. **Model Explorer** – It allows user to have one click access to all types of parameters involved in the model. It is simple hierarchical structure showing all types of model information. It shows model definition data, properties, load definitions, object forms, analysis, design and detailing results. Clicking on (+) sign in front of required entity expands Hierarchical tree. For Example, clicking on properties will open all types of properties defined in the model. Like frame sections, material properties, diaphragms, hinge properties etc. The beautiful feature of the model explorer is the pick and drag feature for sections. Sections may be assigned to the model by dragging the section name and dropping it on the appropriate object to be assigned. User can drop the slab section property on slab only. It is also the case with beams or any other section property user think of to assign. This feature helps user to save the time for revisions and fast-forward the modelling process.
4. **Title Bar** – This bar defines the program and model name under use. It is used to maximize, minimize and close the programme. To work on this programme in parallel window with other programme, user can use “Restore down” option and snap the ends of the window to adjust its size. In the below figure, ETABS is used with excel to expedite the design process.

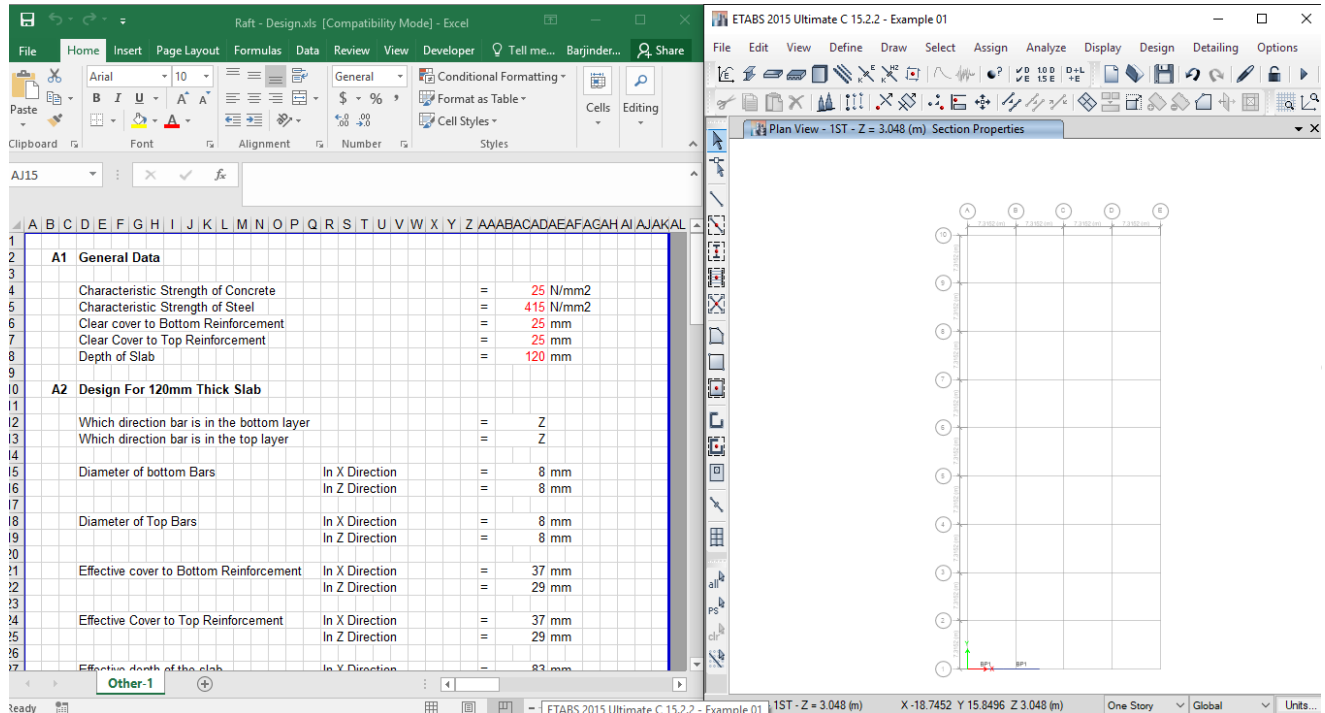
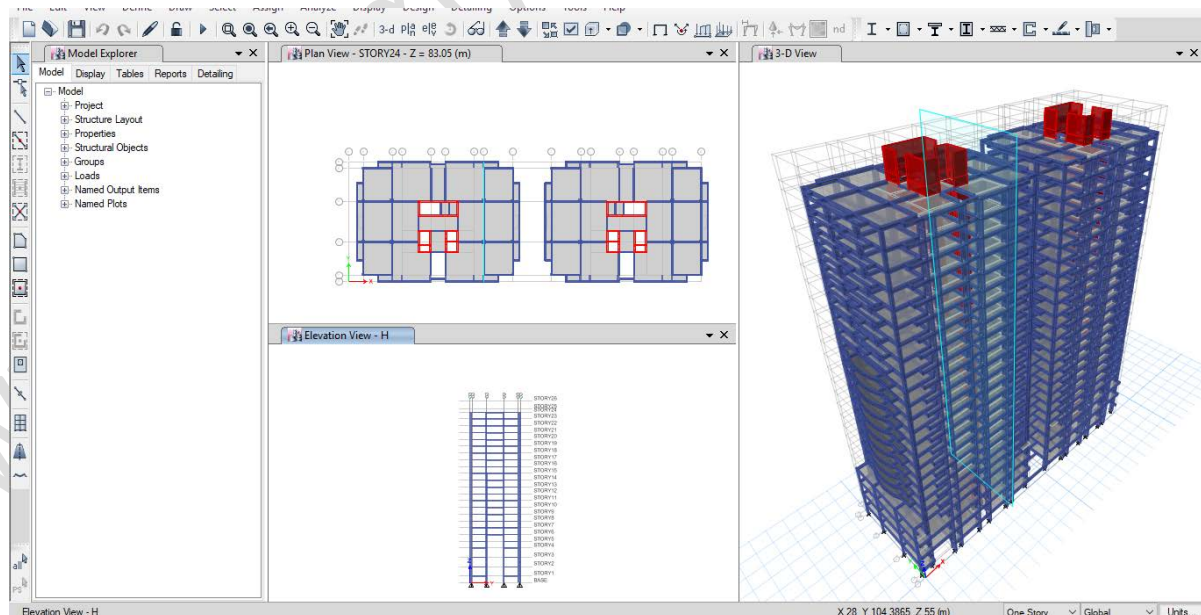
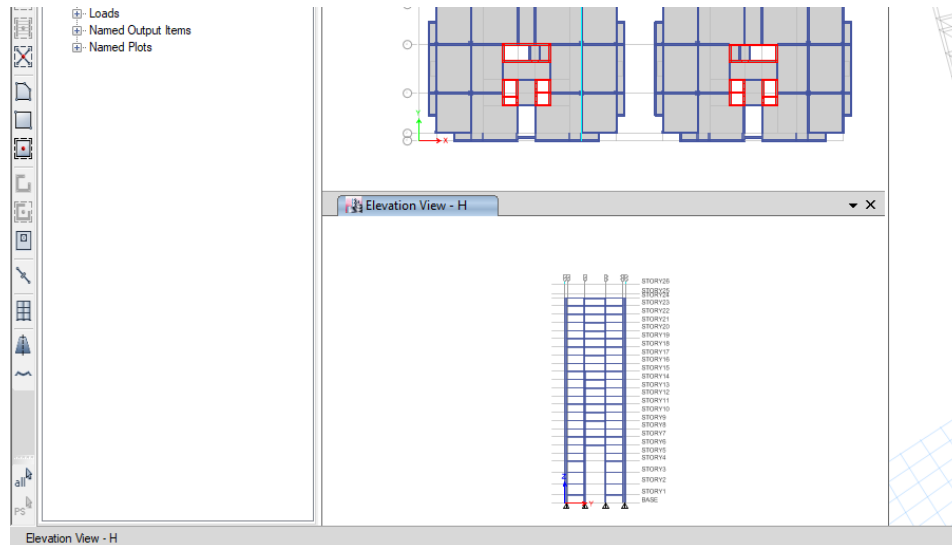


Fig- Parallel Windows to increase productivity

- Display Windows** – It is an interface between the programme and the user. It is an output of user commands and actions. User can see what exactly he is modelling through the display windows. User can see plan, elevation, 3-d views, properties, loading values and type in one window. It is very important for user to observe his actions in 3-d view. Any mistake if done by the user in plan view or any other view reflects instantly in 3-D view. The number of windows can be as much as user required. There is no limit. After analysis and design, user can display properties, analysis results, design output results and reinforcement detailing, connection design etc.



6. **Status Bar** – The status bar shows the status of the program. It is located on the bottom left. Example in this window, it shows the current view as Elevation-H i.e. it shows the view along elevation H. Similarly, when user clicks on the 3-D view, it shows 3-D view. Always check the status bar before doing any modification.



7. **Position Coordinates** – Mouse pointer coordinates are displayed when the user hovers over the joints. Joints may be intersection of beams or wall stacks etc. User does not need to click on any option to activate coordinates. It is always available by default. User may see the coordinates for better understanding of the model. It is also very helpful when the user is copying or replicating any frame/joint or shell.




8. **Drawing Selection Drop Down** – This option is used to select the stories on which the user wants to work upon. It shows user three options. It is located on the right hand side of the status bar at the bottom of the window.

(A) **One Story** – User works upon only one story level, which is visible on the working window. Any object drawn in the plan only occurs at the story level where it is drawn. No other story level is modified. Similarly, selection made in the plan only occurs only at the story level selected. User can refer 3-D view to see what exactly is selected, drawn or modified.



(B) **All Stories** – User draws or modifies one story level and this modification is done at all story levels on the same plan location except the base level. When the user clicks on any object in a story level with this option activated, the programme selects all the objects at the same plan location except the base level. Usually, stories are similar in normal structural buildings. Therefore, this option helps the user to speed up the modelling process. Please note that while working with this option, the base level is not modified.

(C) **Similar Stories** - Similar stories option is effectively used when the user had designated master story and the slave story i.e. story dependent on master story. Selection or modification of any object results in the objects on the same location on similar stories to be selected or modified excluding base or support level.

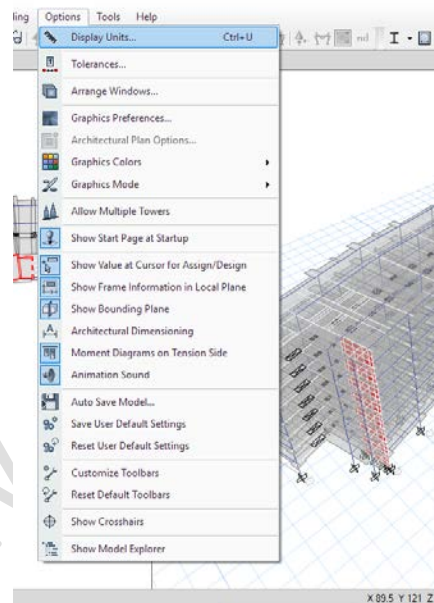
 **Tip:** - User can switch between these three options whenever required in the plan view. New users should practice using one story only first and then switch to all stories to understand the difference.

9. **Units Form** – Units form is another step to make this program user friendly. Units can be selected from the Model Initialization form. This form helps the user to change the units at any time while working on the model. User may define the grids in meter and size of column in mm. User should try to select the units initially only as the software may round off the when the user changes the units in the working stage and there might be some round off error variations in that case.

User should take units seriously. If the user is working on SI units, selecting US default units in starting or in between is not appreciated. In addition, vice versa.

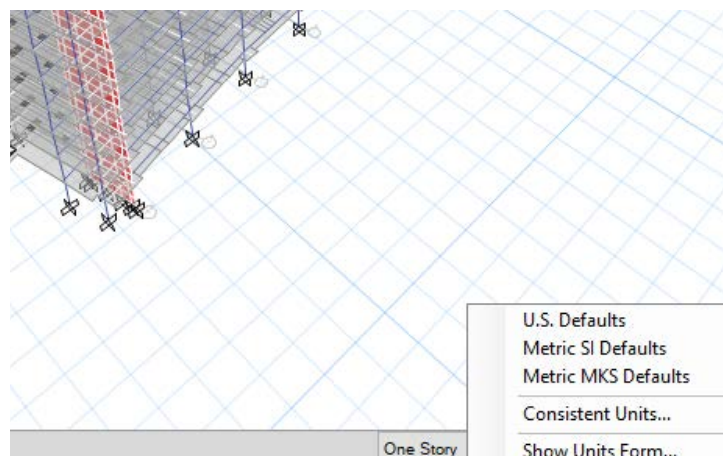
The units form can be accessed from the following options

(A) Go to Options Menu> Display Units command to display the units form.



(B) Press Ctrl+U key on keyboard to display the units form.

(C) Click on units form and select the option “Show units form”.



Units form is shown as below

Item	Length Unit	Force Unit	Temperature Unit	Units Label	Decimal Places	Minimum Significant Figures	Zero Tolerance
Structure Dimensions							
Absolute Distance	m			m	4	1	5E-07
Relative Distance					4	1	5E-07
Structure Area	m			m ²	2	1	5E-05
Angles				deg	3	1	5E-06
Section Dimensions							
Length	mm			mm	1	1	0.0005
Area	cm			cm ²	1	1	0.0005
Length3	cm			cm ³	1	1	0.0005
Length4	cm			cm ⁴	1	1	0.0005
Length6	cm			cm ⁶	1	1	0.0005
Rebar Area	mm			mm ²	0	1	0.005
Rebar Area/Length	mm ² /m			mm ² /m	2	1	5E-05
Displacements							
Translational Displ	mm			mm	3	1	1E-12
Rotational Displ				rad	6	1	1E-12
Drift					6	1	5E-09
Gen Displ L/Rad	mm			mm/rad	3	1	5E-06
Gen Displ Rad/L	mm			rad/mm	3	1	5E-06
Forces							
Force		kN		kN	4	1	5E-07
Force/Length	m	kN		kN/m	3	1	5E-06
Force/Area	m	kN		kN/m ²	3	1	5E-06
Moment	m	kN		kN-m	4	1	5E-07
Moment/Length	m	kN		kN-m/m	3	1	5E-06
Temperature			C	C	3	1	5E-06
Temperature Change			C	C	3	1	5E-06

Expand Form OK Cancel

Change the units by clicking on the white box. Drop down list open up showing the possible alternative units. Considering example of length, user can change units to inch, feet, mm, centimeter, or meter. No other unit of length is possible in the program except the list shown in the units form.

ETABS 2016 Ultimate 16.2.0 - CURVED TOWER

File Edit View Define Draw Select Assign Analyze Display Design Detailing

Plan View - 7TH FLOOR - Z = 9.9

Display Units

Item	Length Unit
Structure Dimensions	
Absolute Distance	m
Relative Distance	
Structure Area	m
Angles	
Section Dimensions	
Length	mm
Area	in
Length3	ft
Length4	mm
Length6	cm
Rebar Area	m

One Story

U.S. Defaults
Metric SI Defaults
Metric MKS Defaults
Consistent Units...
Show Units Form...

PREDEFINED UNITS SELECTION

User can change the units to

1. U.S defaults

In this units system, user can change to U.S defaults. This option converts the unit system to feet, inches and pound. It additionally offers the option to expand form and use architectural dimensioning with precision. By clicking on

the “Use architectural dimension with precision”, user can change the option to increase precision to 1/256” precision.

Item	Length Unit	Force Unit	Temperature Unit	Units Label	Decimal Places	Minimum Significant Figures	Zero Tolerance
Structure Dimensions							
Absolute Distance	ft			ft	4	1	5E-07
Relative Distance					4	1	5E-07
Structure Area	ft			ft ²	2	1	5E-05
Angles				deg	3	1	5E-06
Section Dimensions							
Length	in			in	4	1	5E-07
Area	in			in ²	2	1	5E-05
Length3	in			in ³	2	1	5E-05
Length4	in			in ⁴	2	1	5E-05
Length6	in			in ⁶	2	1	5E-05
Rebar Area	in			in ²	4	1	5E-07
Rebar Area/Length	in ² /ft			in ² /ft	4	1	5E-07
Displacements							
Translational Displ	in			in	6	1	1E-12
Rotational Displ				rad	6	1	1E-12
Drift					6	1	5E-09
Gen Displ L/Rad	in			in/rad	4	1	5E-07
Gen Displ Rad/L	in			rad/in	4	1	5E-07
Forces							
Force		kip		kip	3	1	5E-06
Force/Length	ft	kip		kip/ft	3	1	5E-06
Force/Area	ft	lb		lb/ft ²	2	1	5E-05
Moment	ft	kip		kip-ft	4	1	5E-07
Moment/Length	ft	kip		kip-ft/ft	3	1	5E-06
Temperature			F	F	3	1	5E-06
Temperature Change			F	F	3	1	5E-06
Temperature Gradient	ft		F	F/ft	2	1	5E-05
Stresses							
Modulus	in	lb		lb/in ²	2	1	5E-05
Stress Input	in	lb		lb/in ²	2	1	5E-05
Stress Output	in	lb		lb/in ²	2	1	5E-05
Strain	in			in/in	6	1	5E-09
Stiffness							

☒ Expand Form
 ☒ Use Architectural Dimensioning with Precision
 1/16"

☒ Use Architectural Dimensioning with Precision
 1/16"

- 1"
- 1/2"
- 1/4"
- 1/8"
- 1/16"
- 1/32"
- 1/64"
- 1/128"
- 1/256"

2. Metric S.I defaults

In this units system, user can opt Metric S.I unit's defaults. This option converts the unit system to mm, centimeter and Kilo Newton.

Display Units

Item	Length Unit	Force Unit	Temperature Unit	Units Label	Decimal Places	Minimum Significant Figures	Zero Tolerance
Structure Dimensions							
Absolute Distance	m			m	4	1	5E-07
Relative Distance					4	1	5E-07
Structure Area	m			m ²	2	1	5E-05
Angles				deg	3	1	5E-06
Section Dimensions							
Length	mm			mm	1	1	0.0005
Area	cm			cm ²	1	1	0.0005
Length3	cm			cm ³	1	1	0.0005
Length4	cm			cm ⁴	1	1	0.0005
Length6	cm			cm ⁶	1	1	0.0005
Rebar Area	mm			mm ²	0	1	0.005
Rebar Area/Length	mm ² /m			mm ² /m	2	1	5E-05
Displacements							
Translational Displ	mm			mm	3	1	1E-12
Rotational Displ				rad	6	1	1E-12
Drift					6	1	5E-09
Gen Displ L/Rad	mm			mm/rad	3	1	5E-06
Gen Displ Rad/L	mm			rad/mm	3	1	5E-06
Forces							
Force		kN		kN	4	1	5E-07
Force/Length	m	kN		kN/m	3	1	5E-06
Force/Area	m	kN		kN/m ²	3	1	5E-06
Moment	m	kN		kN-m	4	1	5E-07
Moment/Length	m	kN		kN-m/m	3	1	5E-06
Temperature			C	C	3	1	5E-06
Temperature Change			C	C	3	1	5E-06
Temperature Gradient	m		C	C/m	2	1	5E-05
Stresses							
Modulus	mm	N		MPa	2	1	5E-05
Stress Input	mm	N		MPa	2	1	5E-05
Stress Output	mm	N		MPa	2	1	5E-05
Strain	mm			mm/mm	6	1	5E-09
Stiffness							

☒ Expand Form

OK Cancel

3. Metric M.K.S defaults

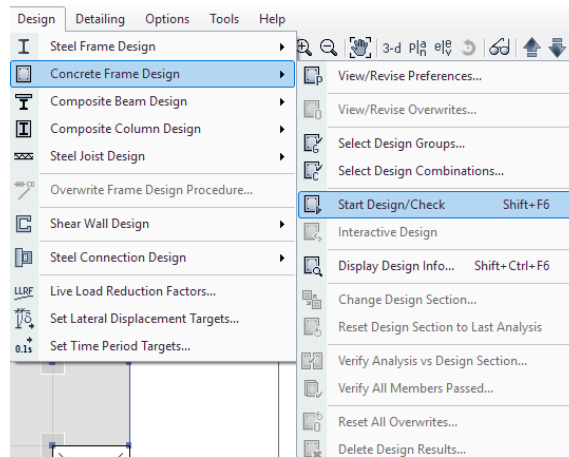
In this units system, user can opt Metric M.K.S unit's defaults. This option converts the unit system to mm, Kilogram and second.

Display Units

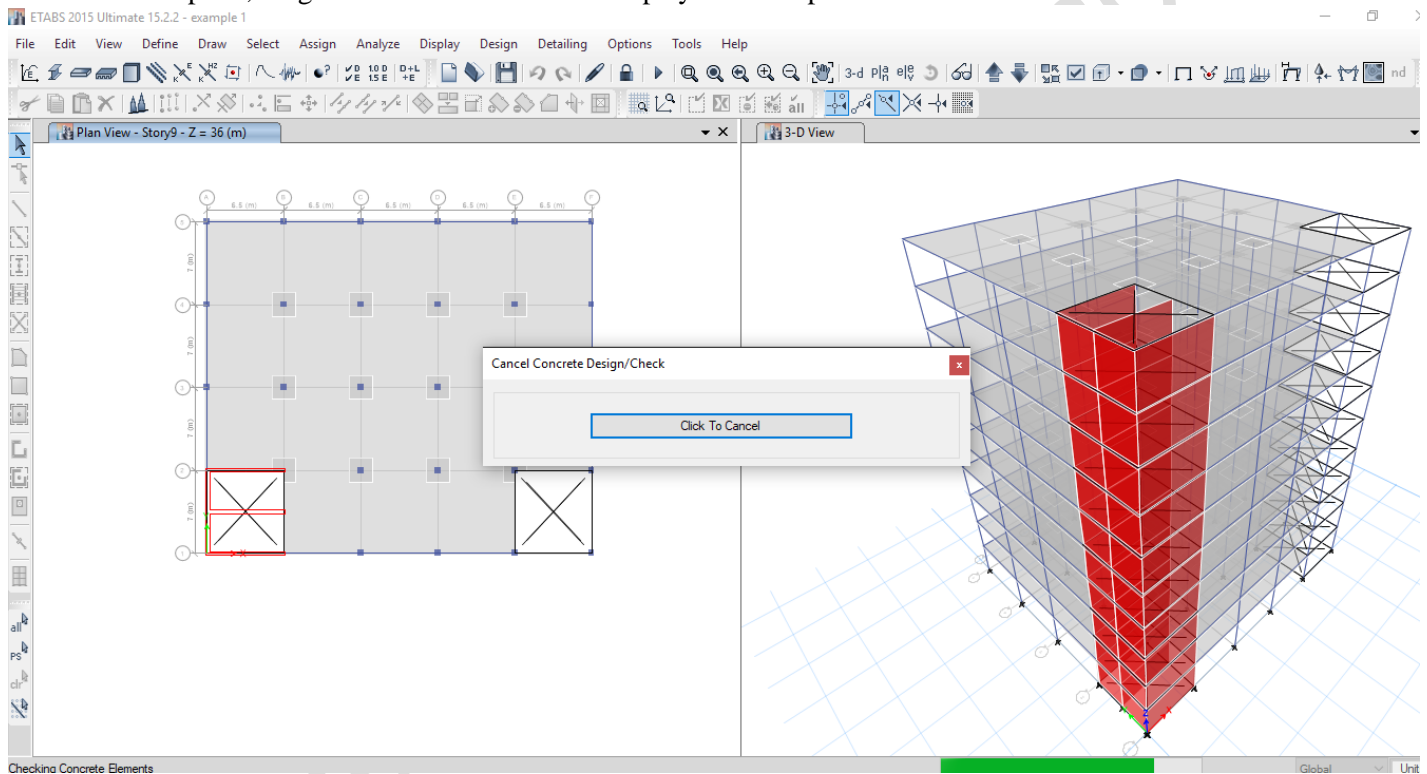
Item	Length Unit	Force Unit	Temperature Unit	Units Label	Decimal Places	Minimum Significant Figures	Zero Tolerance
Structure Dimensions							
Absolute Distance	m			m	4	1	5E-07
Relative Distance					4	1	5E-07
Structure Area	m			m ²	2	1	5E-05
Angles				deg	3	1	5E-06
Section Dimensions							
Length	mm			mm	1	1	0.0005
Area	mm			mm ²	1	1	0.0005
Length3	mm			mm ³	1	1	0.0005
Length4	mm			mm ⁴	1	1	0.0005
Length6	mm			mm ⁶	1	1	0.0005
Rebar Area	mm			mm ²	0	1	0.005
Rebar Area/Length	mm ² /m			mm ² /m	2	1	5E-05
Displacements							
Translational Displ	mm			mm	3	1	1E-12
Rotational Displ				rad	6	1	1E-12
Drift					6	1	5E-09
Gen Displ L/Rad	mm			mm/rad	3	1	5E-06
Gen Displ Rad/L	mm			rad/mm	3	1	5E-06
Forces							
Force		tonf		tonf	4	1	5E-07
Force/Length	m	tonf		tonf/m	3	1	5E-06
Force/Area	m	kgf		kgf/m ²	2	1	5E-05
Moment	m	tonf		tonf-m	4	1	5E-07
Moment/Length	m	tonf		tonf-m/m	4	1	5E-07
Temperature			C	C	3	1	5E-06
Temperature Change			C	C	3	1	5E-06

☐ Expand Form

OK Cancel



(A) A new window will appear indicating programme is performing the concrete design. When the design is complete, longitudinal reinforcement is displayed on the plan view.



(B) Right Click on any perimeter beam to display the beam design information. Check different options.