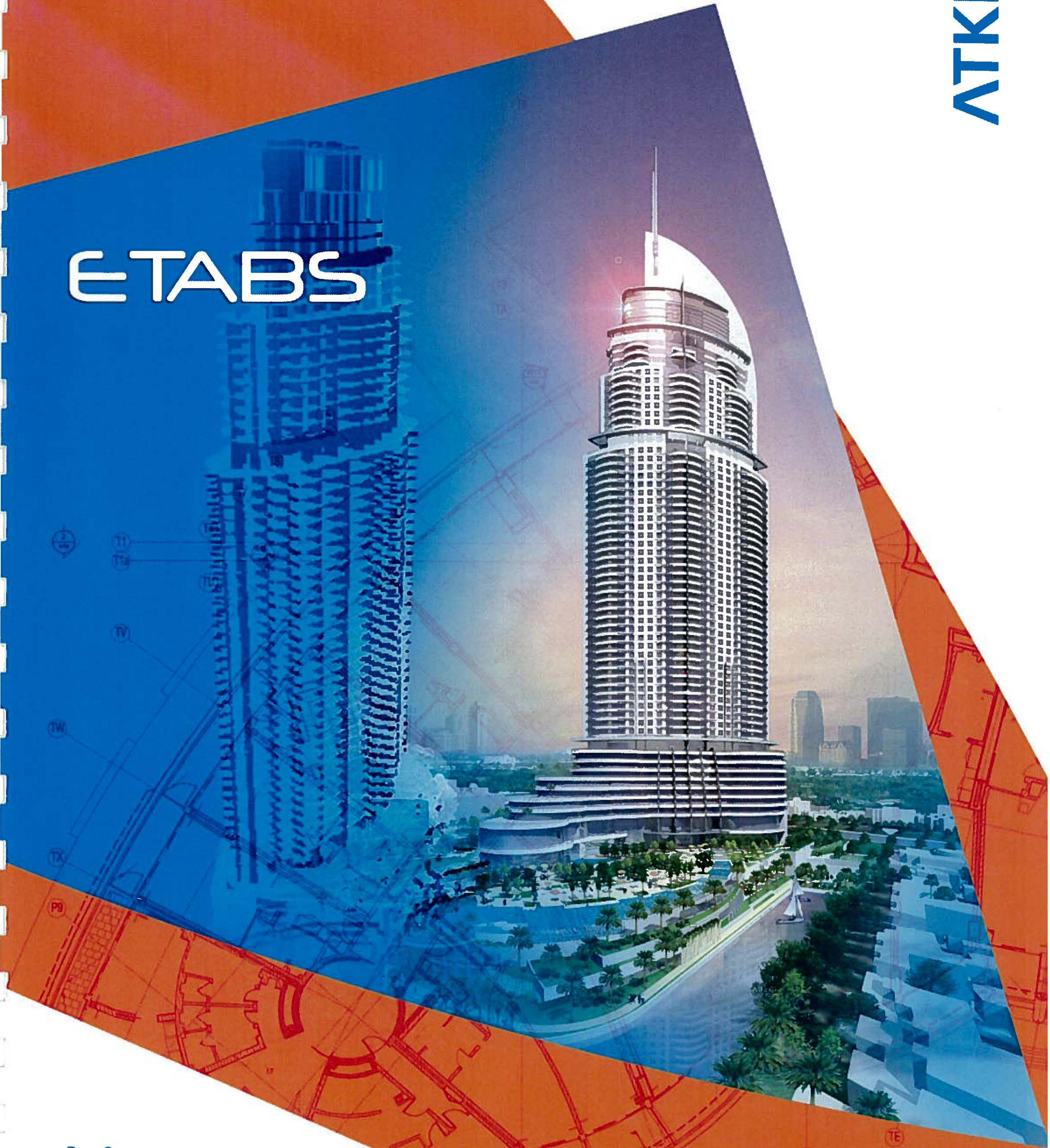
MANUAL FOR ANALYSIS & DESIGN USING ETABS

ATKINS



Rev.0
2007/12/04

Prepared by:
Structural Department, Atkins Dubai

Plan Design Enable

MANUAL FOR ANALYSIS & DESIGN USING ETABS

Objective:

The primary objective of this document is to make sure that ETABS is used consistently by the structural engineers in Atkins office in Dubai in terms of:

- modelling and analysis procedures
- use of applicable built-in international codes
- And complying with local authorities specific requirements.

This document is intended to complement the ETABS manuals and other relevant technical papers published by CSI. It is assumed that the user of this manual has a good command of ETABS and is familiar with the following codes:

- UBC 97 seismic provisions
- ASCE 7 provisions for wind loading
- BS codes of practice

Local Authority specific requirements are covered in Appendices at the end of this document.

The procedures in this document are based on standard practice in Dubai. However, for specific projects, some parameters or procedures need to be revised. This shall be done in accordance with the design statement and in conjunction with the project lead engineer.

Table of Contents:

7. Analysis Options :

7.1 Dynamic analysis options (Ritz vs. Eigenvector)
7.2 P-Delta analysis options
For Local Authorities other than JAFZA
For JAFZA

8. Post-Analysis Checks:

8.1 Analysis log & results
Warnings
Global force balance
8.2 Deformed shape and modal animations
8.3 Modal characteristics (modal amplitude, mass participation ratio)

9. Reinforced Concrete Design Module

9.1 Shear Walls Design Module (BS 8110-97)
9.2 Reinforced Concrete Frame Design (BS 8110-97)Beams

1. *File Menu*
1.1Open a pre-defined Template

To ensure that a consistent procedure is adopted for modelling in ETABS throughout ATKINS Dubai, two templates are prepared and stored in the Structural Models network drive (U-drive):

1) JAFZA.EDB
2) DMTECOM.EDB

These templates are based on the requirements of local authorities; JAZFA and DM/Tecom respectively. These templates incorporate as many of the requirements as possible, however it should be noted that many of the local authorities requirements may only be implemented while a 3D model is developed, therefore a thorough review during modelling is essential to ensure that these provisions are properly taken into account.

NB-The metric unit is used for ATKINS office in Dubai where the force unit is kilo-Newton (kN) and the length is expressed in meters (m). These units are used in the templates.

1.2 Import geometry
1.2.1 Import .DXF file of Architectural grid

To ensure that the architectural grid is appropriately imported in ETABS, make sure that the DXF layer names are consistent with the architectural grid you need to import.

A form appears that has drop-down boxes associated with ETABS elements such as beams, walls, floors and the like. Use the drop-down boxes to select the DXF layer names that contains the lines and insertion points in the DXF file as the ETABS corresponding elements. Select the layer names to be imported by highlighting them. ETABS then imports the lines from any layer in the DXF file as ETABS grid lines and imports the insertion point of any block as an ETABS reference line.

1.2.2 Import .DXF floor Plan

Import the floor plan from a DXF file as follows:

To ensure an accurate geometric modelling in ETABS, it is recommended that the structural floor plan is used as far as possible. Make sure that appropriate layers are selected to be imported.

Note1: ETABS will import 3-d Face and Polyline entities in the DXF drawing as floors or openings and line entities as beams/columns.

Note 2: Use the Story Level Combo box to select the plan location/story level of the entities to be imported from the DXF file into ETABS.

Note 3: The following procedure may be used to create a .DXF file for the model from the Architectural AutoCAD floor plan:

a) Create a layer " ETABS-TYP"
b) Draw lines along the floor extent
c) Draw diagonal lines for columns

d) Draw centerlines for shear walls
e) Draw centerlines of transfer / lateral beams
f) Draw X-Y axis to represent origin in ETABS (geometric center of floor)
g) Save drawings as *.DXF file
h) Import *.DXF file in ETABS as outlines above.

1.2.3 Import .DXF file of 3D model

This option may be used when a 3-D model is available in DXF format. Since 3-D representation is not used for typical floor plan and elevation in Atkins Dubai, this option will not be covered in this manual. The user may refer to ETABS manual for further reference.

2. *Material Properties*

2.1 Concrete

2.1.1 Define Concrete Grade

The following concrete grades are often used in ATKINS Dubai: **C45, C50, C60 & C70**.These grades are already pre-defined in ETABS template files.Use of other grades may be justified based on project's specific requirements. Use the Define menu > Material Properties command to access the Define Materials form. Use that form to add, modify, or delete material properties.

2.1.2 Define Concrete mass and weight per unit volume

The concrete mass and weight per unit volume are taken as **2.54** Ton/m^3 and **25** kN/m^3 respectively unless specifically stated in the project documents otherwise.

2.1.3 Define Concrete modulus of Elasticity

The concrete modulus of elasticity shall be determined based on BS 8110-2 as follows:

$$E_c = 20 + 0.2\, f_{cu,\,28} \qquad (1)$$

Therefore, for the typical concrete grades used in ATKINS Dubai, the corresponding module of elasticity will be as follows:

Concrete Grade	Modulus of Elasticity (GN/m^2)
C45	29
C50	30
C60	32
C70	34

Note: The **Poisson's Ratio** and **Coefficient of thermal expansion** shall be taken as **0.2** and **9.9×10^{-6}** (/°C) respectively unless other specific values are approved in special cases.

2.2 Reinforcement

The reinforcement properties for gravity design shall be based on BS-8110 which is taken as **f_y=460 N/mm2.** This value is the same for **bending** and **shear** reinforcement.
According to local authorities' requirements, the seismic design of reinforced concrete elements shall be based on ACI 318 provisions. As per ACI 318-05 provisions (Section 3.5), the reinforcement yield value of **f_y=420 N/mm2** shall be used. The reinforcement properties that are pre-defined in ETABS templates are consistent with ACI design approach and should be revised for designs based on BS-8110.

3. Definition and Sizing of Elements

3.1 Define Frame Elements

Frame sections may be defined to the desired dimension or be imported from one of the section databases available in ETABS. The user may also import the sections from a user-defined database with ".pro" extension. Complex, unsymmetrical shapes may be modelled using the built-in section designer module. The following general tips may be useful for defining frame sections. The reader is urged to refer to ETABS user manual for further details.

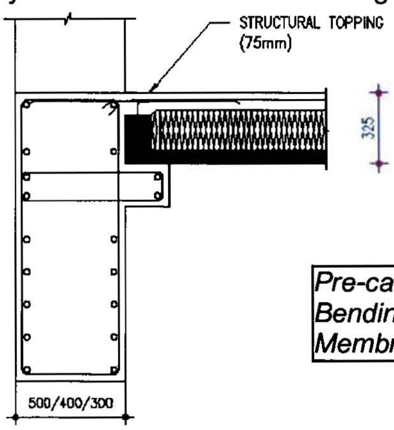
1. It is generally recommended that the material properties are defined first. This assures correct material assignment to the member and allows defining similar sections with different material property. This feature is particularly useful for tall buildings where grade of concrete will change in height.
2. Rectangular and circular sections may be easily modelled from the available drop-down menus, however for irregular shapes the user should use the Section Designer module by selecting Define Frame Properties>Add SD Section option. For further information about this module, refer to Section Designer Manual published by CSI [1].
3. For reinforce concrete rectangular and circular sections, the user may specify one of the design types, e.g., Column or Beam. The column design option allows the provided reinforcement to be checked or designed, whereas the beam design option is limited to just designing the required reinforcement value.
4. Section property modifiers may be assigned to each section at this stage or later. However it should be noted that property modifiers for all frame types may be revised anytime by selecting the appropriate member (beam, column or brace) and there is no need to define them separately for each section. This will be discussed more in this chapter.

3.2 Define Shell Elements

Shell elements are used to define floor, wall and ramp objects as discussed below:

Define Floor and Ramp Objects

There are three options to model floor elements in ETABS; Deck, Plank or Slab. A deck option may be used to model one way joist and slab, one way slab or metal deck systems. Plank and slab options may be used to model one-way or two way slabs with or without one-way special load distribution. Appropriate shell, membrane or plate property shall be assigned to floor members based on their actual behaviour. A membrane element may be used to include only in-plane stiffness properties for the member (e.g. walls) where as plate type behaviour means that only out-of-plane plate bending stiffness is provided for the section. Shell type behaviour considers both in-plane and out-of-plane stiffness properties are considered. This type is generally recommended unless the user is confident about the realistic behaviour of the member. For membrane and shell type elements, different membrane or bending thickness may be defined based on the actual behaviour of the slab system as shown in the following example.



Pre-cast slab with 75mm topping.
Bending thickness – 75 mm
Membrane thickness – 325 mm

For thick shell and membrane element, the program is capable of considering the out-of-plane shear deformation in the analysis. This option is recommended when modelling thick floor such as rafts and transfer slabs.

The section property modifiers may be assigned to each section at this stage or later. However it should be noted that property modifiers for all floor objects may be revised anytime by selecting the appropriate member (floor, ramp or wall) and there is no need to define them separately for each section. This will be discussed more in this chapter.

Define Wall Objects

Walls may be defined as shell or membrane elements. However shell behaviour type is recommended by ETABS manual [2]. Other modelling features are similar to what has been discussed for slabs except for section modifiers which will be discussed more in this chapter.

3.3 *Assign Frame or Shell Section Properties*

Walls and columns may be modelled using either shell or frame sections, however it should be noted that using shell elements provide more flexibility and accuracy for modelling openings and / or variation in member dimension (width, length) along height.

When using a frame element (beam) to model a shear wall spandrel, keep in mind that the analysis results obtained are dependent on the fixity provided by the shell element that the beam connects to. Different sized shell elements provide different fixities and thus, different analysis results.
In general, for models where the spandrels are modelled using frame elements, better analysis results are obtained when a coarser shell element mesh is used; that is, when the shell elements that the beam connects to are larger. If the shell element mesh is refined, consider extending the beam into the wall at least one shell element to model proper fixity.

If the depth of the shell element approaches the depth of the beam, consider either extending the beam into the wall as mentioned above, or modelling the spandrel with shell elements instead of a frame element.

The following criteria may be used for modelling coupling beams:

Length / Depth < 1.0 or Length/thick < 5	Shell Element
Length / Depth > 1.0 or Length/thick > 5	Frame Element

3.4 *Assign Frame Section Modifiers*

Use the Assign menu > Frame/Line > Frame Property Modifiers command to bring up the Analysis Property Modification Factors form to assign modification factors for the following frame analysis section properties in your model.
- Cross-section (axial) area
- Shear Area in 2- direction
- Shear Area in 3-direction
- Torsional Constant
- Moment of Inertia about the 2-axis
- Moment of Inertia about the 3-axis

The modification factors are multiplied by the section properties specified for a frame element to obtain the final analysis section properties used for the frame element. Note that these modification factors only affect the analysis properties. They do not affect the design properties.

The section modifiers for Ultimate limit state analysis for Line Objects are shown in the following table based on UBC 97, clause 1910.11.

Column	Beam
$I_{22}=I_{33}=0.7I_g$	$I_{22}=I_{33}=0.35$
A=1.0 A_g	A=1.0 A_g

*When tensile stress is induced in particular elements under any of the defined ultimate load combinations, the modifier shall be reduced to 0.35 for those elements.

I_g = Moment of Inertia based on Gross Cross sectional Properties assuming rectangular sections for beams. Member design will be based on end-face moments (not centre-point).

This analysis will be used in arriving at the following results;

- Structural design of all elements
- Pile loads
- Building drift / story drift under seismic loads only

The Service limit state analysis shall be carried out with the augmented section modifiers as per ACI 318 clause 10.11.1 and its commentary, R 10.11.1 that allows multiplying the above section modifiers (as per UBC Clause No. 21.3.1) by 1.43. Slabs and beams section modifiers are as per ultimate limit state provisions as mentioned above. This analysis will be used to check:

- Wind Drift (Overall and story drift)
- Acceleration

A detailed finite element analysis shall be performed to check the stresses in columns and walls. If the stress in any member exceeds the allowable tensile stress value, appropriate section modifiers corresponding to the cracked section properties shall be assigned to that member. The drift and accelerations shall be checked accordingly.
To ensure that the stiffness modifiers are assigned to all the elements, it is generally recommended to assign the stiffness modifiers after completion of the model and prior to the analysis using the "Select by Object Type" option in ETABS. This not only relieves the laborious task of defining the stiffness modifiers separately for each frame section, but also provides a quick, yet reliable way to change these modifiers in no time.

3.5 Assign Shell Section Modifiers

Use the Assign menu > Shell/Area > Shell Stiffness Modifiers command to bring up the Analysis Stiffness Modification Factors form. Here you can specify Stiffness Modifiers for the following shell analysis section stiffness in your model.
☐ Membrane f11 Modifier
☐ Membrane f22 Modifier
☐ Membrane f12 Modifier
☐ Bending m11 Modifier
☐ Bending m22 Modifier
☐ bending m12 Modifier

The stiffness for each of the items calculated based on the section properties specified for a shell element are multiplied by the specified modifiers to obtain the final stiffness used for the shell element in the analysis. Note that these modification factors only affect the analysis properties. They do not affect any design properties.

The f11, f22 and f12 modifiers are essentially equivalent to modification factors on the thickness (t) of the shell element. The m11, m22 and m12 modifiers are essentially equivalent to modification factors on the $(t)^3$ of the shell element.

The section modifiers for Ultimate limit state analysis for Area Objects are shown in the following table based on UBC 97, clause 1910.11.

Wall[(1),(2)]	Coupling Beams (Shell)[(3)]	Slab
m_{11}= m_{22}=0.7 (Uncracked)	f_{11}=f_{12}= f_{22}=m_{22}=0.35	m_{11}= m_{12}=m_{22}=0.25
m_{11}= m_{22}=0.35 (Cracked)		

(1)- The correct parameters that need to be modified to reflect cracked section properties for walls are f_{11} & f_{22}, however due to the inevitable axial shortening, JAFZA requires that m-parameters be revised. Refer to the discussion below for further clarification.

(2)-It should be noted that revising stiffness modifiers to cater for cracked sections in shell elements in trivial. The gross section area based on UBC 97 (Clause 1910.11) and ACI 318(Section 10.11) provisions should not be changed. This may be easily accounted for frame elements by just revising the section modifier for moment of inertia. However, the axial and bending stiffness for shell elements can not be de-coupled, i.e., changing the bending stiffness will inevitably affect the axial stiffness. This may cause displacement incompatibility with adjacent frame column which in turn may require revising the axial stiffness for vertical frame elements, as opposed to code explicit provisions.

(3)-Lower stiffness modifier values may be assigned for coupling beams based on the actual state of cracking in the element from the analysis results.

3.6 Assign Pier / Spandrel Label

The pier / spandrel labelling is a convenient way to get the design forces for walls and coupling beams especially when they are modelled as shell elements. Special care shall be taken when defining these labels to ensure realistic values. The reader is urged to refer to CSI's ETABS Manual and Shear Wall Design Manual for further details.

A wall pier can consist of a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall piers, or if you want to design wall piers, you must first define them. Define a wall pier by selecting all of the line and/or area objects that make up the pier and assigning them the same pier label. If a wall pier is made up of both line and area objects, assign the pier label to the line and area objects separately.

A wall spandrel can consist of a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall spandrels, or if you want to design wall spandrels, you must first define them. Define a wall spandrel by selecting all of the line and/or area objects that make up the spandrel and assigning them the same spandrel label. If a wall spandrel is made up of both line and area objects, assign the spandrel label to the line and area objects separately.

3.7 Assign Area Mesh Option

Selected areas can be meshed using the Edit>Mesh Areas command toolbar button. Several options are available in the Mesh Selected Areas form:
□ **Auto Mesh Area (Horiz):** This option meshes the selected area into smaller areas. The smaller areas are three-sided or four-sided and must have beams on all sides.

□ **Cookie Cut at Selected Line Object (Horiz):** This option meshes the selected area at the selected lines. Select one or multiple lines. If the selected line passes through more than one area, all of the areas will be meshed. Note that this and the Auto Mesh Area option only work in plan view.

□ **Cookie Cut at Selected Point at [Specified] Angle:** Use this option to mesh areas at a specified point and angle. The angle will be measured in the counter clockwise direction for the x and y-axis. If the point lies in the overlapping region of two areas, both of the areas will be meshed at the given angle.

□ **Mesh Quads/Triangles into [Specified Number] by [Specified Number] Areas:** This option meshes the selected area in the number of areas specified by the user. For example, specifying a meshing of 2 by 8 means that the selected area will be meshed into 2 areas along the x-axis and 8 areas along the y-axis. The size of the meshed areas will be uniform along a given direction. Only quads and triangles can be meshed using this option.

□ **Mesh Quads/Triangles at Intersections with Visible Grid Lines:** This option meshes each selected area at any location where it intersects a visible grid line, regardless of the coordinate system associated with the grid line.

□ **Selected Point Objects on Edges:** Selecting this option will mesh the area (horizontally and vertically) using the selected point at the edge as reference. One more points can be selected for this type of meshing.

□ **Interactions with Selected Line Objects:** The areas selected are meshed with the line intersecting the area. More than one line can be selected to mesh a desired area.

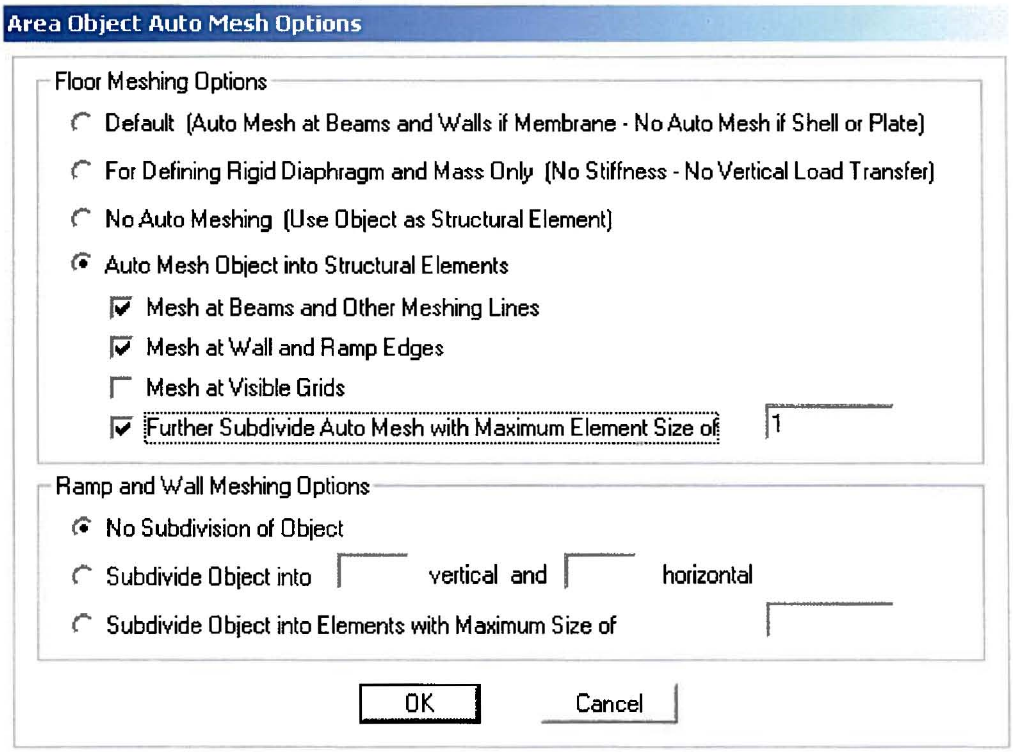
Note the following about Meshing Area Objects:

□The property assignments to meshed area objects are the same as the original area object.

□Load and mass assignments on the original area object are appropriately broken up onto the meshed area objects.

□When this menu item is clicked, all edges of the currently selected area will be split at their mid-points. If clicked again for the same selected area, they will be divided in half again, and so on.

The program does not offer any automatic meshing for walls, however, for slab elements, the automatic meshing option may be done as shown below.

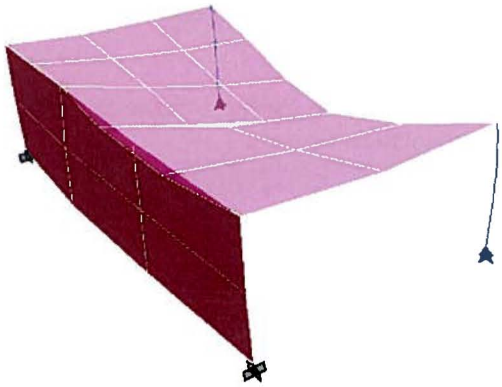


Note-1: In general, slab elements may be drawn manually, but this is time consuming and may lead to unrealistic results if local axes of slabs are different or unsuitable mesh sizes are used. Complex floor systems supporting many walls and columns (e.g. Raft) may be meshed in other finite element programs such as Robot and then imported into ETABS.

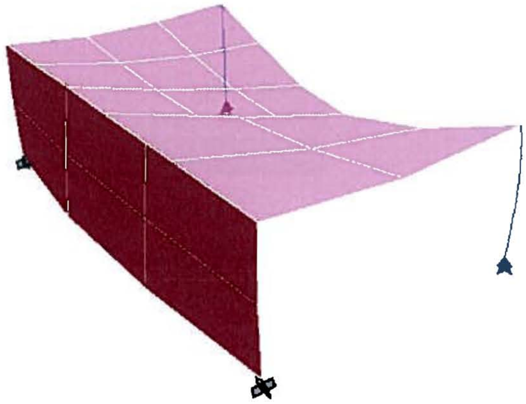
Note-2: In general triangular plate-bending element, with shearing deformations, produces excellent results. However, the triangular membrane element with drilling rotations tends to lock, and great care must be practiced in its application. Because any geometry can be modelled using quadrilateral elements, the use of the triangular element presented can always be avoided.

3.8 Assign Auto-Line Constraint

Auto-line constraint is a technique in ETABS that is very useful in reducing the hassle of fine-tuning meshing of adjacent objects. If the meshes on common edges of adjacent area objects do not match up, automated line constraints are generated along those edges. These Line Constraints enforce displacement compatibility between the mismatched meshes of adjacent objects and eliminate the need for mesh transition elements.
The following figures show the difference in results when applying auto-line constraint to a simple model where slab and wall meshing does not match.
The auto-line constraint is the default option in ETABS and needs to be removed manually if required.



Case1: Without Auto-line constraint



Case2: With Auto-line constraint

4. Supports

4.1 General Support Conditions

- In reinforced concrete structures on single or mat foundation, the support conditions are taken as fixed (all rotational and translational degrees of freedom are locked).

- Where raft (or single) piles are modelled in ETABS, however, the support conditions may be taken as free (no rotational and translation D.O.F is locked) or pinned. The piles for this case need to be modelled with appropriate springs. Some guidelines for this purpose is explained in the following section.

4.2 Modelling Piles as Supports

- Piles are modelled in ETABS as springs where the spring stiffness-corresponding to the pile vertical and horizontal stiffness- is used by ETABS for analysis purpose. The stiffness of these springs may be calculated based on the maximum allowable axial force and settlement of the pile.

- The maximum allowable axial stress on a pile may be limited to $0.25f_{cu}$. On the other hand, the maximum allowable settlement for a pile is generally given by the geotechnical expert. In lieu of these data (and as directed by JAFZA), this value may be taken as 1% of pile diameter (in mm). Therefore the vertical spring stiffness may be expressed as:

$$k_v = \frac{0.25f_{cu}.A_{pile}}{\Delta_{all.}} = \frac{0.25f_{cu}.\pi d^2}{4x0.01d} \cong 20d$$

- The horizontal pile stiffness is taken as 10% of the vertical value.

5. Loading

5.1 Dead Loads

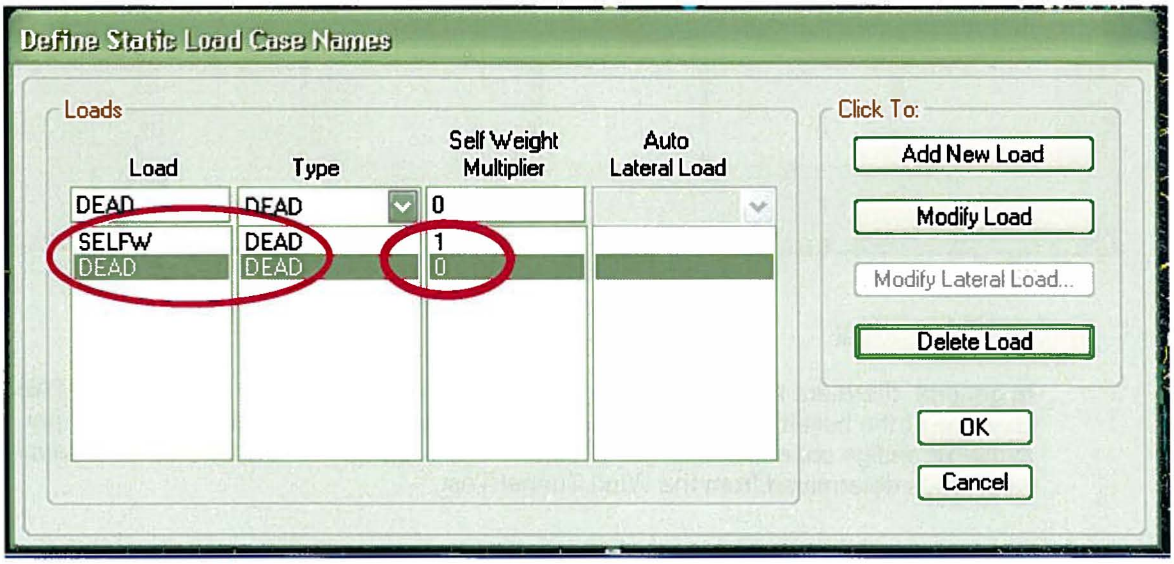
Since ETABS can calculate the self-weight of different elements defined and apply their load in static analysis, it is important to define dead loads appropriately. The self-weight and imposed dead loads shall be defined separately as explained below:

5.1.1 Assign Self Weight

The self-weight multiplier controls what portion of the self-weight is included in a load case. A self-weight multiplier of 1 means that the full self-weight of the structure is included in that load case.

5.1.2 Define Imposed Dead Load

This type of loading shall be used to define any other type of permanent load acting on the structure, excluding the self-weight of structural elements that are modelled in ETABS. Load associated with floor finishes, raised flooring, ceiling, services and permanent partitions are examples of this type of loading.



5.2 Live Loads

5.2.1 General

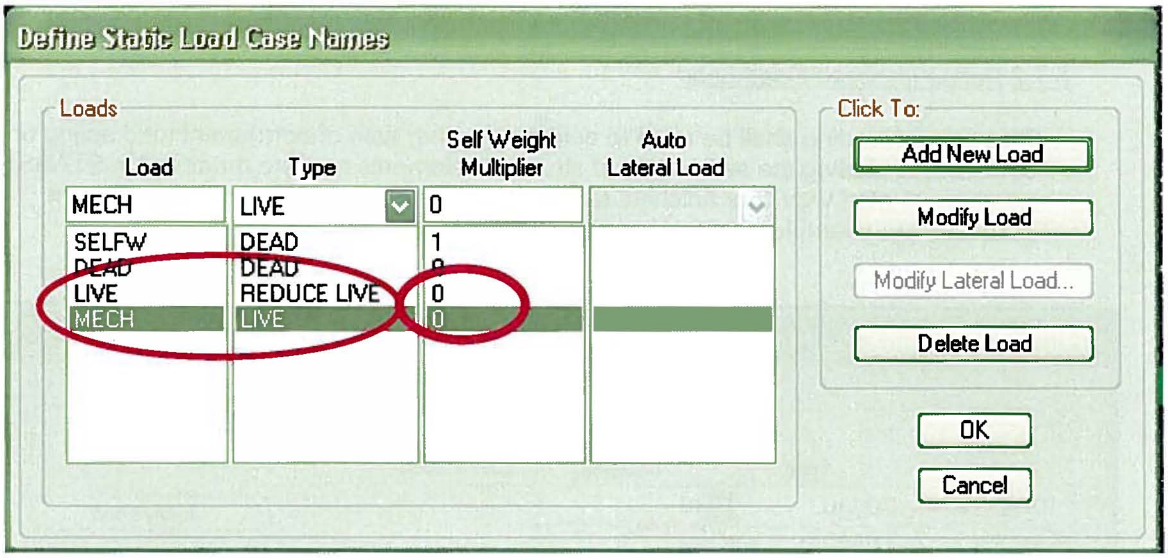
Live loads shall be defined as reducible or irreducible based on their magnitudes. As per ASCE7 provisions (which is also adopted in UBC), lie loads in excess of 4.79 kN/m^2 shall be taken as irreducible.
The live load values shall be assigned in accordance with the values adopted in Design Statement and the specific code requirements.

5.2.2 Reduction of Live Loads

A live load that is specified as reducible is reduced automatically by the program for use in the ***design postprocessors*** (and hence doesn't have any effect in the analysis results). The live load reduction parameters are specified using the **Options menu > Preferences > Live Load Reduction** command.
It is important to ensure that the self-weight multiplier is set to zero (0) for all load cases except self-weight.
It should also be noted that Load Combinations do not include live load reduction unless required specifically. Therefore, this shall be considered when using other supplementary design software (e.g. PROKON).

5.3 Mechanical Loads

Mechanical loads are irreducible live loads that are generally used to represent the effect of areas with special equipment or facility (substations, plant rooms, etc). This definition will help to differentiate between the live loads that are NOT permitted by the code to be reduced. For example, as stated earlier in this chapter, ASCE7 and UBC 97 define any live load exceeding 4.79 kN/m^2 as irreducible live load. Therefore these loads shall be defined as a MECH load to ensure that they are not reduced for member design.



5.4 Wind Loads

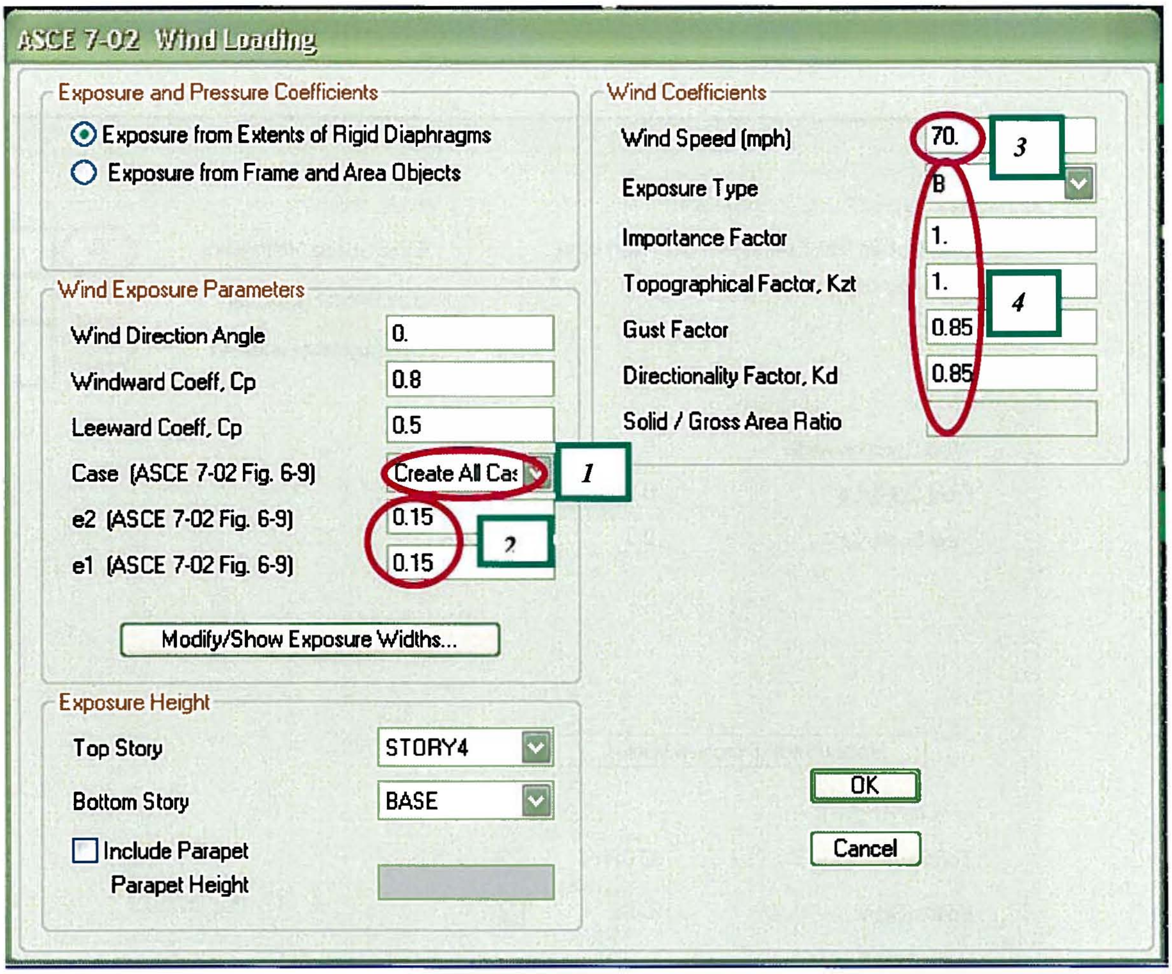
In general, there are two ways to define lateral loads (Wind, earthquake, etc) in ETABS: Use one of the built-in options that will automatically calculate the lateral loads as per available design codes or specify the lateral loads manually. The latter is used to apply the wind loads determined from the Wind Tunnel Test.

5.4.1 Codified Methods

Codified wind loads that are approved by JAFZA are limited to ASCE 7 and AS/NZS 1170.2. However, DM / Tecom currently also accept design wind loads as per BS 6399, Part-2. The procedures to define codified wind loads as per ASCE 7 and BS 6399 Part-2 are described briefly below:

5.4.1.0 Define Wind Load Parameters as per ASCE 7

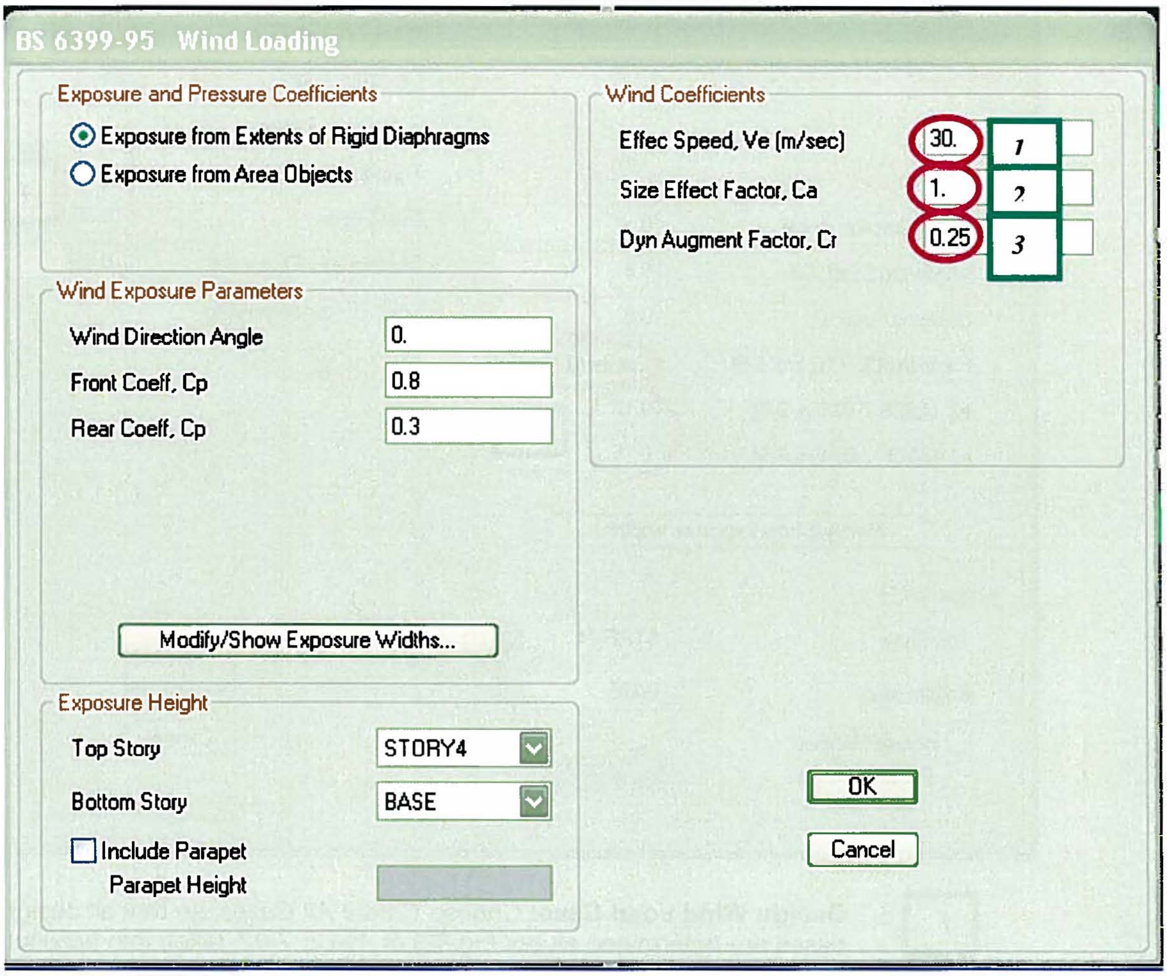
The ASCE 7-02 wind load parameters shall be determined from respective Code sections and input in the ASCE 7-02 Wind Loading Table of ETABS. Then ETABS will automatically calculate the wind loads acting on each story level and use it in the static analysis processor. A sample form of ASCE 7-02 wind parameters is shown below followed by a brief description on key items.



- **1** ○ **Design Wind Load Case:** Choose *Create All Cases*, so that all design wind load cases are determined as per Fig.6-9 of *ASCE 7-02*, taken into account the torsional moment effects.
- **2** ○ **Eccentricity:** Determine the eccentricity values for the structure as per Clause 6.5.12.3 and Fig.6-9 of *ASCE 7-02*. For rigid structures, defined as structures with natural frequency of greater than 1 Hz (T_1<1 sec.), the eccentricity shall be taken as equal to 15% of the building dimension in the perpendicular direction. Otherwise, use Equation (6-21) in Clause 6.5.12.3 of *ASCE 7-02* to calculate this parameter.
- **3** ○ **Wind Speed:** In lieu of reliable wind tunnel studies, the basic wind velocity shall be taken as 45 m/sec (101 mph) as per local authority requirements. Note that the basic wind speed shall be input as <u>mph</u> in ETABS.
- **4** ○ **Other Parameters:** Other parameters shall be determined as per provisions of ASCE 7. The exposure type is generally taken as **Exposure C** for Dubai, but should be verified with the wind specialist accordingly. An approved design spreadsheet may be used to reliably calculate all the parameters of ASCE 7-02 wind load data.

5.4.1.1 *Define Wind Load Parameters as per BS 6399-Part-2*

The BS 6399-95, Part-2, wind load parameters shall be determined from respective Code sections and input in the BS 6399-95 Wind Loading Table of ETABS. ETABS will automatically calculate the wind loads acting on each story level and use it in the static analysis processor. A sample form of BS 6399-95 wind parameters is shown below followed by a brief description on each item.



1 ○ **Effective Speed:** The effective wind speed shall be determined as per Clause 2.2.3 of BS 6399-2. The **Site Wind Speed** (Vs) required for this calculation shall be taken as 26 m/sec for Dubai in lieu of reliable wind tunnel studies.

2 ○ **Size Effect Factor:** The size effect factor shall be determined from Clause 2.1.3.4 of BS 6399-2.

3 ○ **Dynamic Augmentation Factor:** The dynamic augmentation factor shall be determined from Clause 1.6.1 and Fig.3 of BS 6399-2.

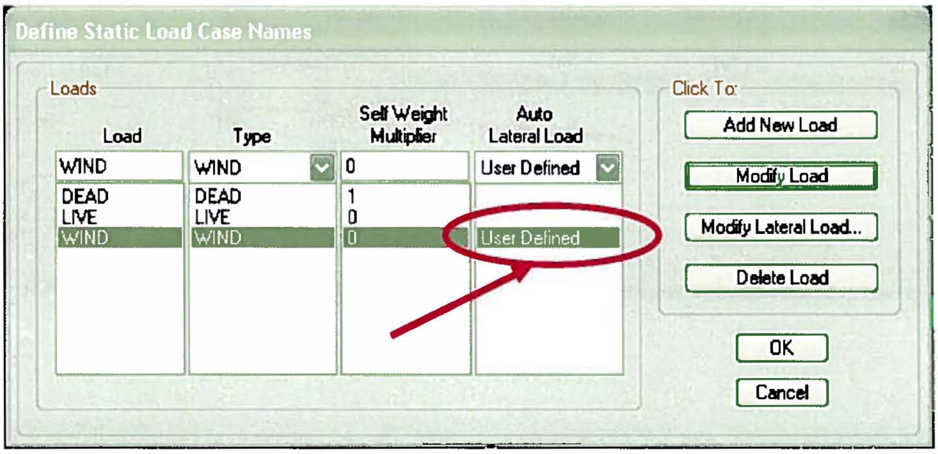
Note 1- An approved design spreadsheet may be used to reliably calculate all the parameters of BS 6399 wind load data.

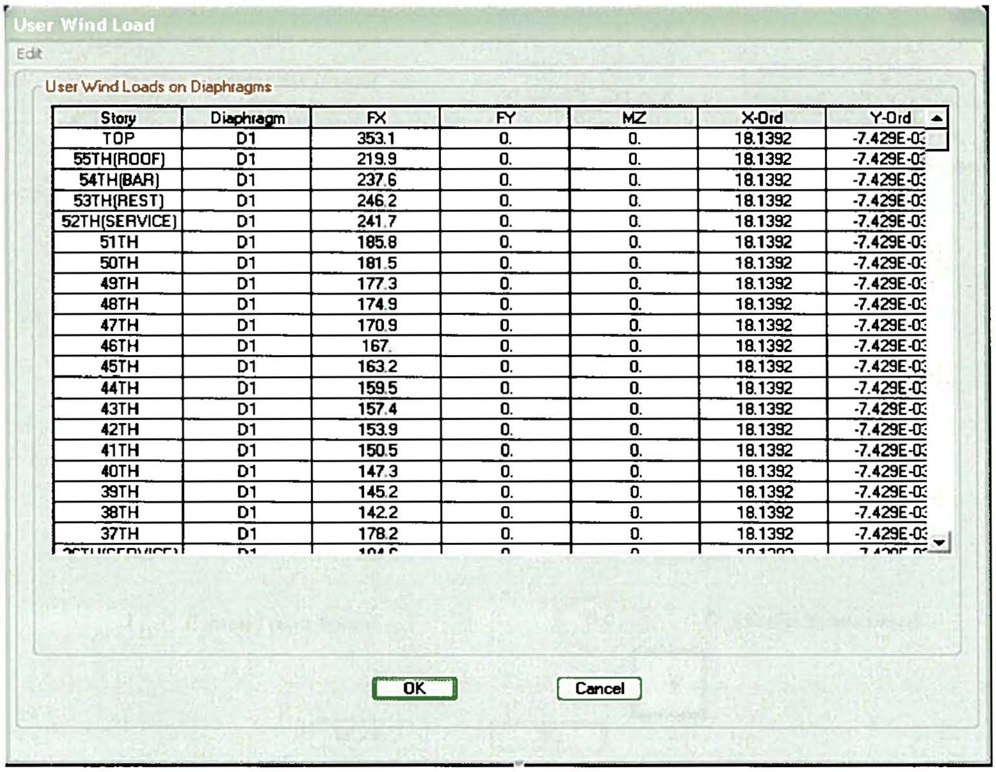
Note 2- As per BS 6399-97, Part-2 provisions of Section 2.1.3.7, accidental torsional effects on the buildings may be accounted for by displacing the wind loads on each face horizontally by 10% of the face width from the centre of the face. This can not be directly taken into account in ETABS and needs to be applied manually. For this purpose, wind loads may be determined as per note-1 and then applied to the building as a User Defined Load in Auto Lateral Load drop-down menu. Refer to Section 7.4.2 for more details.

5.4.2. Extracting Wind Loads from Wind Tunnel Test Results

The results of a reliable wind tunnel test may be used in lieu of the codified values for wind analysis in ETABS. These loads are generally calculated by recognized wind tunnel testing laboratories based on the dynamic properties of the structure as modelled during the preliminary or concept design stages. Wind loads are reported as separate load cases that should be combined through the set of load combinations as reflected in the wind tunnel report. It is important to note that these loads shall be applied to the analytical model at the same reference points that were initially defined for the wind tunnel consultant. Moreover since the Wind consultants generally carry out their calculations at the center of the diaphragm of each floor, it is recommended that these points are taken in locations where are as close to the center of mass of diaphragm as possible.

Wind loads obtained from wind tunnel studies may be defined in ETABS as a *User Defined* Lateral Load. A separate wind load case shall be defined representing the load case as per wind tunnel report. The load values may directly be copied from a spreadsheet. Various load combinations shall also be defined accordingly. The following figures show an example of defining user defined wind load cases.





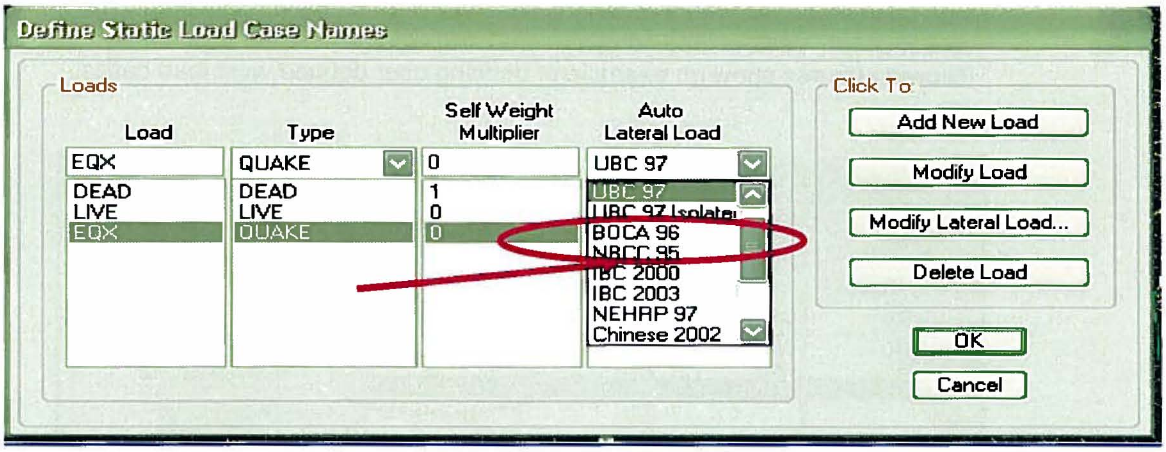
Story	Diaphragm	FX	FY	MZ	X-Ord	Y-Ord
TOP	D1	353.1	0.	0.	18.1392	-7.429E-0
55TH(ROOF)	D1	219.9	0.	0.	18.1392	-7.429E-0
54TH(BAR)	D1	237.6	0.	0.	18.1392	-7.429E-0
53TH(REST)	D1	246.2	0.	0.	18.1392	-7.429E-0
52TH(SERVICE)	D1	241.7	0.	0.	18.1392	-7.429E-0
51TH	D1	185.8	0.	0.	18.1392	-7.429E-0
50TH	D1	181.5	0.	0.	18.1392	-7.429E-0
49TH	D1	177.3	0.	0.	18.1392	-7.429E-0
48TH	D1	174.9	0.	0.	18.1392	-7.429E-0
47TH	D1	170.9	0.	0.	18.1392	-7.429E-0
46TH	D1	167.	0.	0.	18.1392	-7.429E-0
45TH	D1	163.2	0.	0.	18.1392	-7.429E-0
44TH	D1	159.5	0.	0.	18.1392	-7.429E-0
43TH	D1	157.4	0.	0.	18.1392	-7.429E-0
42TH	D1	153.9	0.	0.	18.1392	-7.429E-0
41TH	D1	150.5	0.	0.	18.1392	-7.429E-0
40TH	D1	147.3	0.	0.	18.1392	-7.429E-0
39TH	D1	145.2	0.	0.	18.1392	-7.429E-0
38TH	D1	142.2	0.	0.	18.1392	-7.429E-0
37TH	D1	178.2	0.	0.	18.1392	-7.429E-0

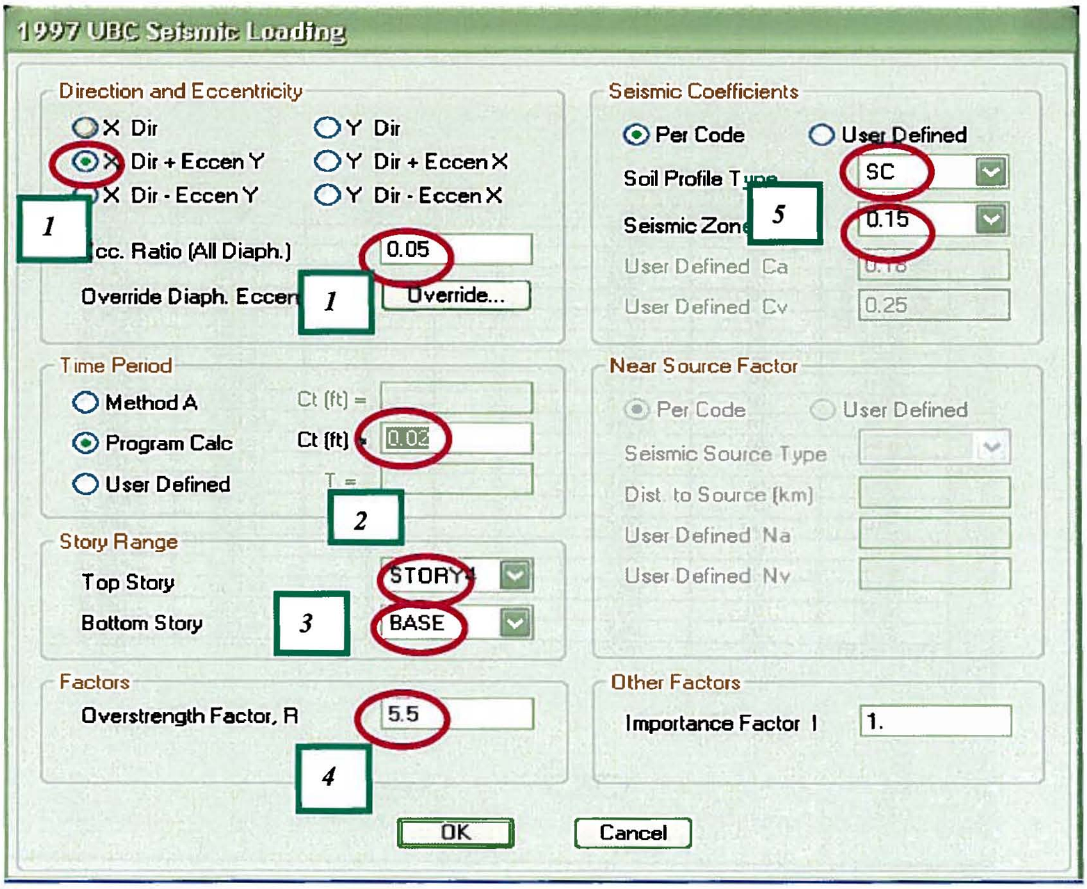
5.5 Earthquake Loads

The earthquake effects shall be determined based on the provisions of UBC 97, Chapter 16. The Response Spectrum Method shall be used for all buildings with more than 12 storeys in height as per JAFZA requirements. However, the results of response spectrum analysis may be scaled to the Equivalent Static Force Method as per Clause 1631.5.4 of UBC 97. Therefore, the Equivalent Static Force Method shall be initially used. The following subsections review the basic parameters required in ETABS.

5.5.1 Equivalent Static Force Method

5.5.1.1 Define Earthquake Parameters as per UBC 97





The parameters that need to be defined in this form are briefly described below:

1
- **Direction and Eccentricity**: Use the ***% Eccentricity*** edit box to specify a value for eccentricity. Five percent is the default and is entered as 0.05. The eccentricity options have meaning only when diaphragms have been assigned to joint/point or shell/area objects. The program ignores eccentricities where diaphragms are not present.

  Note that since the Equivalent Static analysis is often used for scaling the Response Spectrum parameters, the eccentricities need only be calculated for both directions with only one direction of eccentricity (i.e. $X+e_y$ and $Y+e_x$)

2
- **Time Period:** If using the *Program Calculated* option, the C_t coefficient shall be input in Imperial units. This value shall be taken as 0.02 for RC Shear Walls and 0.03 for RC Moment Resisting Frames.

3
- **Story Range:** By default the bottom story is the base of the building and the top story is the uppermost level of the building.
  - In most instances, specify the top story as the upper-most level in the building, typically the roof. However, in some cases, a lower level may be chosen. For example if a penthouse is included in the model, it may be best to calculate the automatic lateral load based on the roof level, excluding the penthouse roof level, as the top story, and then add in additional user-defined load to the load case to account for the penthouse.
  - The bottom level would typically be the base level. However, if, for example, a building has several below-grade levels, and the seismic loads are assumed to be transferred to the ground at ground level, it may be best to specify the bottom story to be above the base of the building.

  Note: No seismic loads are calculated for the bottom story. They are calculated for the first story above the bottom story and for all stories up to and including the top story.

4
- **Overstrength Factor:** Determine the strength reduction factor as per UBC 97, Table16-N based on the structural system used. For Bearing Shear wall system use a value of R=4.5 and for Building frame system with shear walls use R=5.5. Refer to Section 1629.6 of UBC 97 for definition of different structural systems. In general, if the majority of gravity loads are taken by the shear walls, take R=4.5.

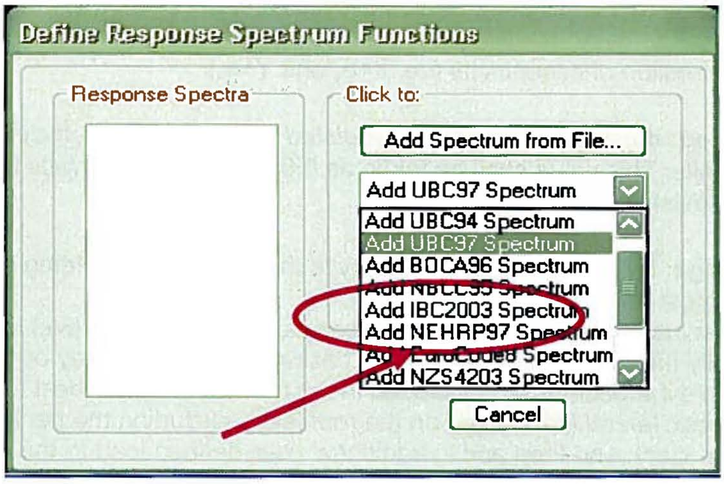
  Note: For structures where more than one system is used throughout all or part of the structure, the provisions of UBC 97 for Dual systems shall be met. Alternatively, one system may be assumed to take all the lateral loads and the other is taken as a building frame system. For such cases, the lateral forces need to be scaled up to ensure that all the lateral loads are carried by respective system.

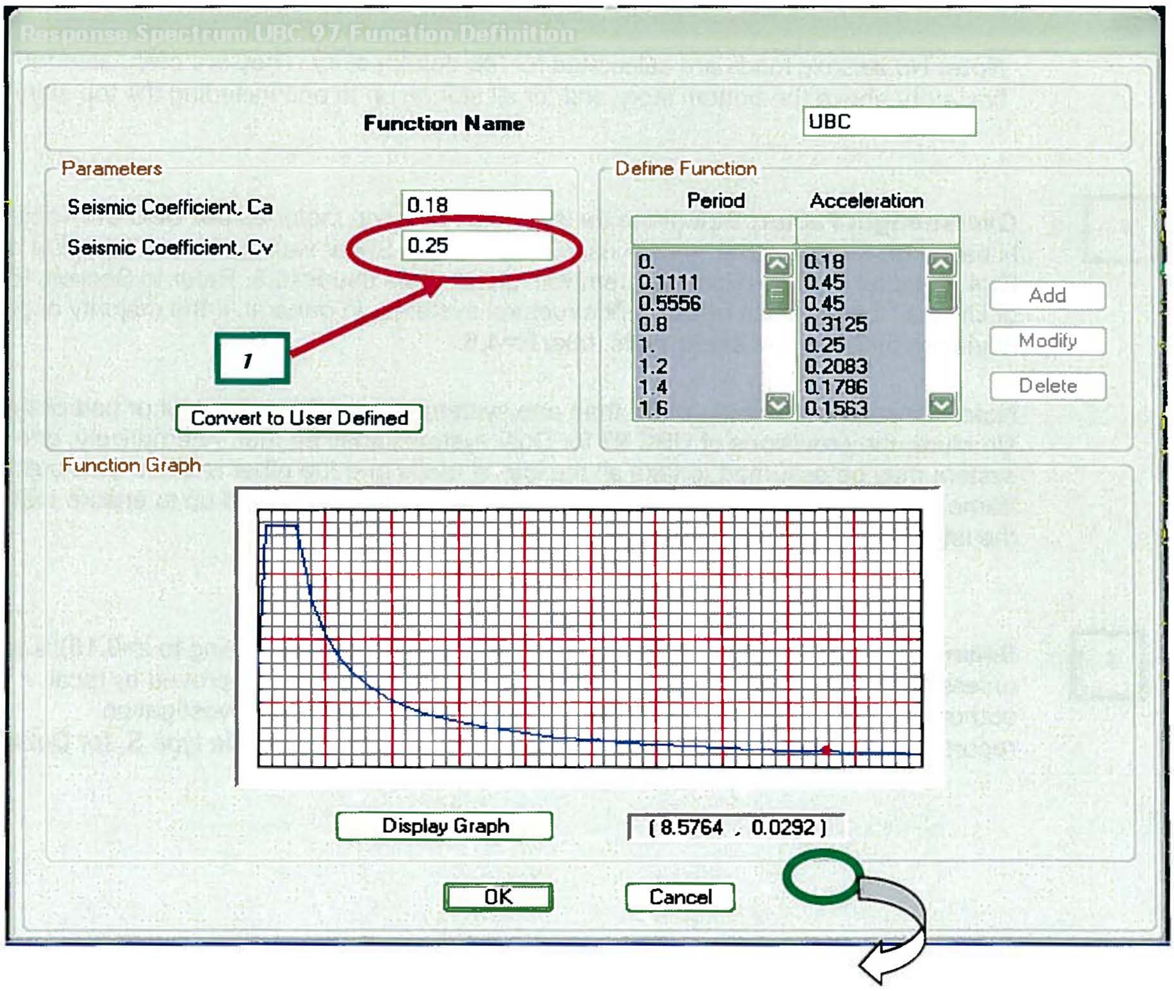
5
- **Seismic Coefficient:** For Dubai, the *seismic zone* 2A (corresponding to z=0.15) is taken unless otherwise stated by a site-specific seismic hazard study approved by local authorities. The *Soil profile* type shall be taken from the site soil investigation report. [Note: Most geotechnical consultants recommend soil profile type S_c for Dubai]

5.5.2 Response Spectrum Method

5.5.2.1 Define Response Spectrum Functions as per UBC 97

Add UBC 97 Spectrum *from the drop-down list to display the Response Spectrum UBC 97 Function Definition form.*

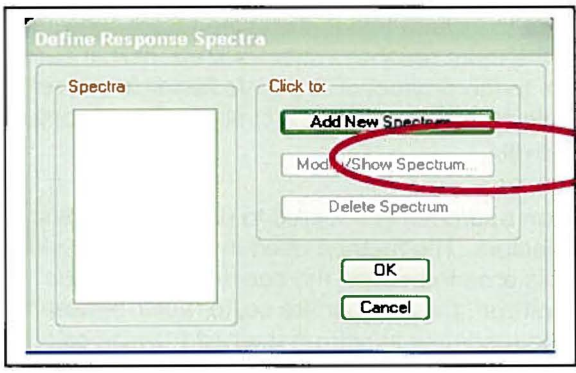


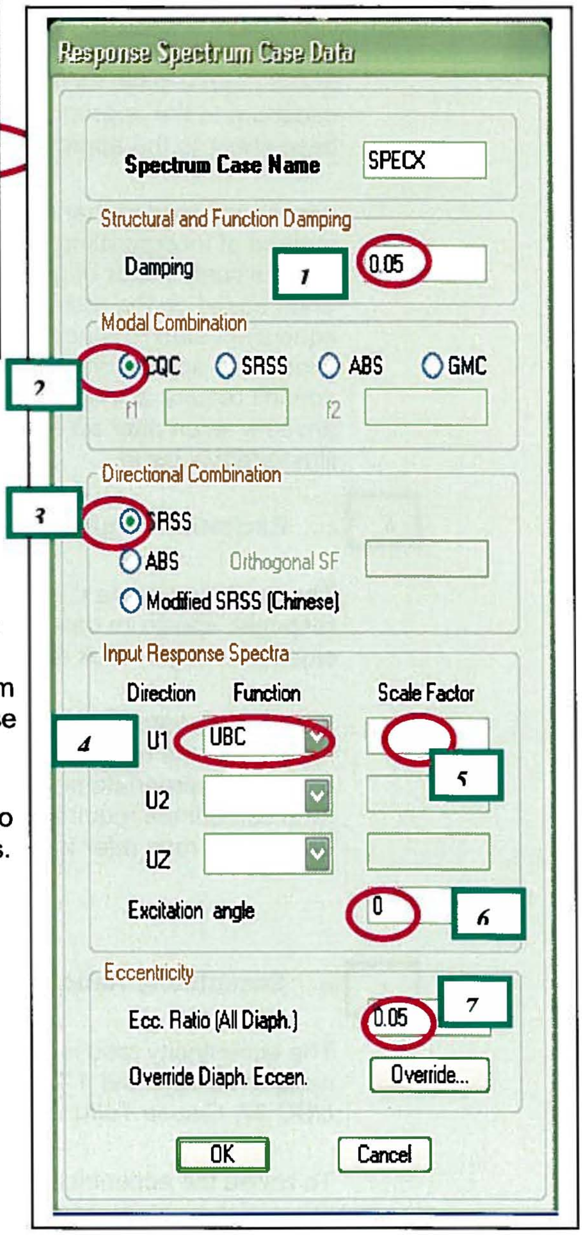


1 • **Parameters**: Specify parameters for *Seismic Coefficient* C_a and *Seismic Coefficient* C_v based on seismic zone and soil profile type (See Tables 16-Q and 16-R in the 1997 UBC). For seismic zone 2A and soil profile type S_c (Typical of Dubai), take C_a =0.18 and C_v =0.25, respectively.

5.5.2.2 Define Response Spectrum Cases

The following topics describe key parameters for the response spectrum case data:





1 ○ **Structural and Function Damping**

This item specifies modal damping that is present for all modes in the response spectrum analysis. Also, ETABS assumes that the response spectrum functions specified for the response spectrum case are all specified for this particular damping ratio.

As per local authorities provisions, a damping ratio of 5% is assumed for response spectrum analysis.

2 ○ **Modal Combination**

Select <u>CQC</u> method since it provides good estimates of the results for most practical cases.

3 ○ **Directional Combination**

Select <u>SRSS</u> method since it provides good estimates of the results for most practical cases and is also consistent with UBC-97 provision for orthogonality effects.

4 ○ **Input Response Spectra**

Specify the defined UBC-97 response spectrum function for each of the three local coordinate system directions as separate response spectrum case.

5 ○ **Scale Factor**

The scale factor has units of Length/seconds^2 and that its value will change as you change the units in your model. Essentially ETABS assumes the response spectrum functions are unitless (normalized) and that the scale factor converts them into the appropriate units (i.e. use a scale factor of 9.81 to convert UBC 97 spectrum into acceleration (m/sec^2))

If you are scaling your response spectrum to match some static analysis results (e.g., static base shear), you may want to include that in the scale factor specified for the response spectrum function in the input response spectra area. In that case you would input a scale factor equal to the product of the scale factor to convert the spectrum to the appropriate units and the scale factor to scale the response spectrum base shear to the appropriate level.

It is recommended that a uniform approach is adopted to provide a consistent method of incorporating scale factors. The method used in Atkins office in Dubai is to use the scale factor of g/R in this area to convert the spectrum into m/sec^2 units. Then based on the results of first-run, the appropriate scale factor between the equivalent static method and the response spectrum method may be calculated and used in an appropriate Load Combination. The advantage of this method is that it doesn't require any additional analysis, since Load Combinations can be modified anytime, even after an analysis. Appendix (---) includes an example that helps to illustrate this issue.

6 ○ **Excitation Angle**

The excitation angle is an angle measured from the positive global X-axis to the response spectrum case positive local 1-axis. A positive angle appears counter clockwise as you look down on the model.

It should be noted that when SRSS method is used for directional combination of responses, the response would be independent of the excitation angle. However, using an appropriate angle –along the principal axes of the building- may be useful (and sometimes required) when uni-directional response parameters are of interest. The reader may refer to Wilson book [Ref.1] for further details.

7 ○ **Eccentricity Ratio**

The eccentricity ratio is initially taken as the accidental eccentricity (5%), but may need to be adjusted if Torsional Irregularity exists in the model based on provisions of *UBC 97, Clause 1630.7* and definitions of *UBC 97, Table 16-M.*

To revise the eccentricity ratio, a preliminary analysis shall be carried out assuming the minimum code mandatory ratio of 5%. The displacement values of four points at the corner edge of the building for each story is then determined. The amplification factor for accidental torsional response, $A_{x,i}$ for each floor is calculated based on the provisions of UBC 97, Clause 1630.7 from the average and maximum displacement results as follows:

$$A_{xi} = \left(\frac{\delta_{\max,i}}{1.2\delta_{avg,i}} \right)^2 \leq 3.0$$

The maximum value of $A_{x,i}$ is used to amplify the eccentricity ratio. It should be noted that this amplification need only be applied once and need not be revised iteratively.

6. Load Combinations

6.1. Define Load Combinations for Serviceability State

SLC1:	1.0 DL + 0.5 LL+ 1.0 MECH
SLCS01:	0.90 DL + 0.714 SRSS
SLCS02:	0.90 DL - 0.714 SRSS
SLCS03:	1.0 DL + 0.375 LL + MECH + 0.536 SRSS
SLCS04:	1.0 DL + 0.375 LL + MECH - 0.536 SRSS
SLCW1:	1.0DL+1.0 Wind(x & y)(1)
SLCW2:	1.0DL-1.0 Wind(x & y)(1)

6.2. Define Load Combinations for Ultimate State

ULC1:	1.4 DL + 1.6 MECH + 1.6 LL	*0.5 C_a. I*
ULCS01	1.29 DL + MECH + 0.5 LL + 1.0 SRSS	*[1.2+0.5x0.18x1=1.29]*
ULCS02	1.29 DL + MECH + 0.5 LL - 1.0 SRSS	
ULCS03	1.11 DL + MECH + 0.5 LL - 1.0 SRSS	
ULCS04	1.11 DL + MECH + 0.5 LL + 1.0 SRSS	
ULCS05	0.81 DL + 1.0 SRSS	*[1.2-0.5x0.18x1=0.81]*
ULCS06	0.81 DL - 1.0 SRSS	
ULCS07	0.99 DL + 1.0 SRSS	
ULCS08	0.99 DL - 1.0 SRSS	
ULCUP01	1.4 DL + 1.6 MECH + 1.6 LL + 1.2 UPLIFT	
ULCUP02	1.0 DL + 1.2 UPLIFT	
ULCUP03	1.4 DL + 1.2 UPLIFT + 1.4 WIND(x & y)(1)	
ULCUP04	1.4 DL + 1.2 UPLIFT - 1.4 WIND(x & y)(1)	
ULCUP05	1.0 DL + 1.2 UPLIFT + 1.4 WIND(x & y) (1)	
ULCUP06	1.0 DL + 1.2 UPLIFT - 1.4 WIND(x & y) (1)	
ULCUP07	1.4 DL + 1.2 UPLIFT	
ULCUP08	1.2DL + 1.2 MECH + 1.2 LL + 1.2 UPLIFT + 1.2 WIND(x & y) (1)	
ULCUP09	1.2DL + 1.2 MECH + 1.2 LL + 1.2 UPLIFT - 1.2 WIND(x & y) (1)	
ULCUP10	1.2DL + 1.2 MECH + 1.2 LL + 1.2 UPLIFT	
ULCW01	1.2 DL + 1.2 MECH + 1.2 LL + 1.2 WIND(x & y) (1)	
ULCW02	1.2 DL + 1.2 MECH + 1.2 LL - 1.2 WIND(x & y) (1)	
ULCW03	1.0 DL + 1.4 WIND(x & y) (1)	
ULCW04	1.0 DL - 1.4 WIND(x & y) (1)	
ULCW05	1.4 DL + 1.4 WIND(x & y) (1)	
ULCW06	1.4 DL - 1.4 WIND(x & y) (1)	

6.3. Define Load Combinations for Pile Design

SLC1:	1.0 DL + 0.5 LL + 1.0 MECH	
SLCS01	0.90 DL + 0.714 SRSS	*[1/1.4=0.714 SRSS]*
SLCS02	0.90 DL - 0.714 SRSS	
SLCS03	1.0 DL + 0.375 LL + MECH + 0.536 SRSS	*[0.75x(1/1.4)=0.536]*
SLCS04	1.0 DL + 0.375 LL + MECH - 0.536 SRSS	
SLCW05	0.8 DL + 0.4LL + 0.8 MECH +0.8 WIND(x & y) (1)	
SLCW06	0.8 DL + 0.4LL + 0.8 MECH -0.8 WIND(x & y) (1)	

(1)- When wind loads are derived from wind tunnel test results, WIND(x & Y) may be taken conservatively as the Envelope of all possible wind load combinations as prescribed by Wind Tunnel specialist.

7. *Analysis Option*

7.1 *Dynamic Analysis Parameters*

- **Number of Modes:** Specify the number of Eigen or Ritz modes that you want ETABS to capture. Generally use a relatively low number of modes for initial analysis to save time. A more refined estimate of the proper number of modes may be determined by interpretation of modal participation factor ratio after the initial run. The total number of modes considered in the analysis shall include at least 90% of the participating mass of the structure for each principal horizontal direction.
- **Type of Analysis** options. Choose **eigenvector** or **ritz-vector** analysis in this area. Input in the balance of the form depends on which option is chosen.

Note: If you are running response spectrum or time history analysis, use ritz-vectors.

Note: For response spectrum analysis, select the ACCELERATION along X and Y directions as the Starting Load Vectors.

7.2 *P-Delta Analysis Options*

General:

- It is recommended [Ref. 1& 2] to use the iterative method in all cases except those where no gravity load is specified in the model. **Iteration Controls:** Generally an iteration value of 3 or 4 will be adequate. Note that the maximum number of iterations specified is the maximum number of additional analyses after the first analysis is run.
- **P-Delta Load Combination:** Specify the single load combination to be used for the initial P-Delta analysis of the structure. The following load cases shall be used based on the adopted Design Code:

Code	P-Delta Load Combination
BS 8110	1.0 D+1.0 Mech+0.5 LL
UBC-97, ACI 318	1.2D+1.2Mech+0.5 LL

7.2.1 *Provisions for Local Authorities Other Than JAFZA*

7.2.2 *Provisions for JAFZA*

JAFZA requires that P-Delta analysis using **Iterative** method shall be carried out for all buildings exceeding 12 storeys in height .However, it should be noted that according to UBC-97 , Clause 1630.1.3, P-Delta analysis is not required when the ratio of the secondary moment to primary moment does not exceed 10. Due to the relative ease of P-Delta analysis in ETABS, it is recommended to initialize the design using this method and use the code exception, whenever applicable, for cases when P-Delta effect results in excessive responses that need to be avoided.

8.0 Post Analysis Checks

After a model is analyzed by ETABS it is very important to check the whether the basic characteristics of the model matches with the expected behaviour or not. In the following sections some key features will be addressed.

8.1 Analysis .LOG and Results

The analysis log file may be reviewed either from the File menu > Last Analysis Run Log... command or it can be opened by a text editor from the directory containing the model. There are two important items that should be checked:

8.1.1 Warnings

A warning is produced during solution of equilibrium equation in ETABS when there is an error in calculation of finite element stiffness matrices, boundary condition or the applied loading. If you come across warning messages for solution along any degrees of freedom, you will need to locate the point(s) and check for any potential error .This may be caused by adjacent points forming a discontinuous mesh, a free-free end support etc. These warnings may be removed by reshaping the objects, defining appropriate boundary conditions / supports or any other suitable action that ensures a sound analytical model.

8.1.2 Global Force Balance

Global force balance relative errors are one of the key measures that can be used to ensure the accuracy in analysis solution. These errors are calculated based on the relative difference in applied external loads and the base reactions for each load case.
The global force balance relative errors should be very small –in order of 10^{-10} -and less. It should be noted that this is a necessary but not sufficient condition to ensure a sound modelling and analysis.

8.2 Deformed Shape and Model Animations

It is recommended to thoroughly investigate the deformed shape of the structure under static loads. This may be done by animating the structure for the required load case. The animated deformation shall match with the anticipated behaviour of the building. Appropriate scale factors may be used to examine the response.

8.3 Modal Characteristic

Mode shapes of the building give a good insight into the dynamic characteristics of the structure. The first few modes of vibration are of particular concern since generally they include most of the dynamic response; however this shall be verified from the modal participation factor. For example if a torsional mode shows a high participation factor, the designed may look for revised structural scheme that may help minimize torsional effects.

9. Reinforced Concrete Design Module

9.1 Shear Wall Design Module (as per BS 8110 – 97)

9.1.1 Overview

In the design of shear walls, the program calculates and reports the required areas of steel for flexure and shear based upon user defined load combinations. The reinforcement requirements are calculated at a user-defined wall labelling (wall pier / spandrel labelling).

The program also performs the following design, check, or analysis procedures in accordance with BS 8110-97 requirements:

- Design and check of concrete wall piers for flexural and axial loads
- Design of concrete wall piers for shear
- Design of concrete shear wall spandrels for flexure
- Design of concrete wall spandrels for shear

9.1.2 Terminology

Analysis Section

These are the objects (area/line) defined in the model to make up the pier or spandrel section.

Wall pier analysis section is the assemblage of wall & column sections & Wall spandrel analysis section is the assemblage of wall & beam sections.

Design Section

The section utilized for design as per pier / spandrel definition. The different types of design sections detailed below;

Pier	Uniform Reinforcing section	Planar / Three dimensional	Designed & Checked
	General Reinforcing section	Planar / Three dimensional	Designed & Checked
	Simplified pier section	Planar	Designed
Spandrel	Spandrel design section	Planar	Designed

Uniform Reinforcing Section

Uniform reinforcing section is applicable to both planar & coupled shear walls.

For flexural design/check, the geometry is picked up automatically by the program from the area objects that define the pier section.

For shear / boundary zone checks, the program automatically (and internally) breaks the analysis section pier up into planar legs and then performs the design on each leg separately and reports the results separately for each leg.

The planar legs are derived from the area objects that defined in the model only.

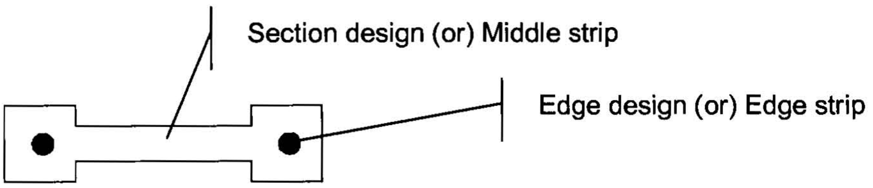
In this process, section will be designed / checked for uniform reinforcing.

General Reinforcing Section

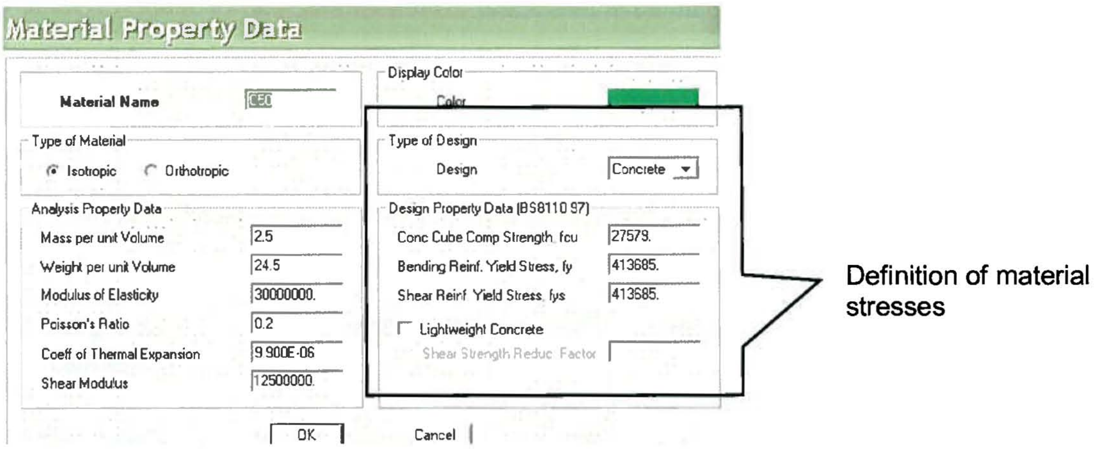
This section is defined using the section designer utility in ETABS module. User can create any geometric shape as in architectural drawings. User can vary vertical rebar diameter and spacing and details.

Simplified Pier Section

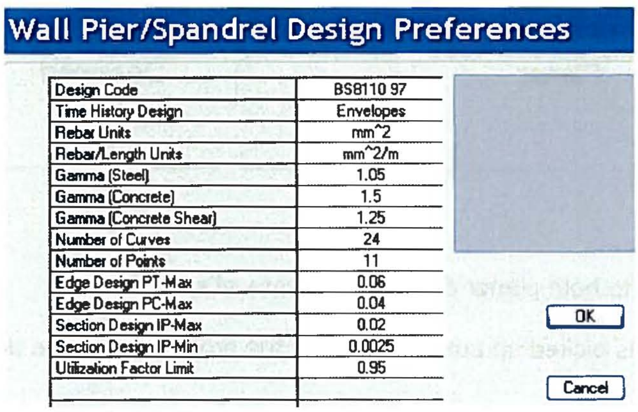
This is applicable to planar walls only. Using this design section, user can only design the walls.



Material design property definition



Preferences:



Design Code	BS8110 97
Time History Design	Envelopes
Rebar Units	mm^2
Rebar/Length Units	mm^2/m
Gamma (Steel)	1.05
Gamma (Concrete)	1.5
Gamma (Concrete Shear)	1.25
Number of Curves	24
Number of Points	11
Edge Design PT-Max	0.06
Edge Design PC-Max	0.04
Section Design IP-Max	0.02
Section Design IP-Min	0.0025
Utilization Factor Limit	0.95

In the Options menu > Preferences > Shear wall design, the design code will be set as BS 8110 97 code with the default design parameters.

Edge Design PT – Max & PC – Max indicates the limiting percentage of rebar requirement at the edge strip if the section is being designed as simplified T & C method.

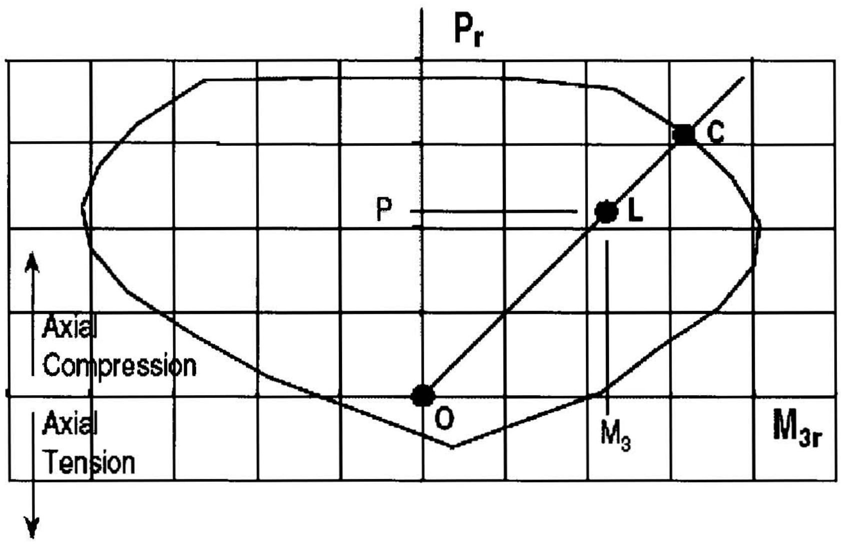
Section Design IP – Max & Min – This indicates the percentage of maximum and minimum rebar requirement at the wall section (applicable to user defined pier section or uniform reinforcing option).

9.1.3 Wall Pier Flexural Design

Checking of a wall design section

When the wall section is specified to check for the given rebar details, the program creates an interaction surface for that pier in three dimensional space at 15 degree intervals (by default) to determine the flexural demand/capacity ratio for that pier.

The required number of degree intervals of interaction surface can be overwritten in the preferences and it is recommended to be 24 or more with number of points required to make an each curve as 11 or more but this needs to be in odd number.



Two- dimensional Wall Pier Demand / Capacity Ratio

Designing of a wall design section

When the pier section is to be designed, the program creates a series of interaction surfaces for the pier based on the following factors;

a) The size of the pier as specified in section designer
b) The location of the reinforcing specified in Section Designer.
c) The size of each reinforcing bar specified in Section Designer relative to the size of the other bars.

The interaction surfaces are developed for 8 different ratios of reinforcing steel area to pier area, which spans between the maximum and minimum ratios as specified in preferences. These limiting ratios can be overwritten in the preferences.

9.1.4 Wall Pier Shear Design

Wall pier shear design is performed at top and bottom of pier section. ETABS follows the procedure as per BS 8110, Cl. No. 3.4.5.

Concrete shear stress is limited to avoid shear cracking prior to the ultimate limit state, hence the shear stress is limited to equation 6b in the code.

In the calculation of concrete shear stress, the area of tension reinforcement is considered as 50% of the total area required for the pier section.

The maximum shear stress is limited to 5 MPa and it doesn't consider any enhancement due to high strength concrete.

The minimum shear steel i.e. horizontal steel is reported for the shear stress is equal to 0.4MPa.

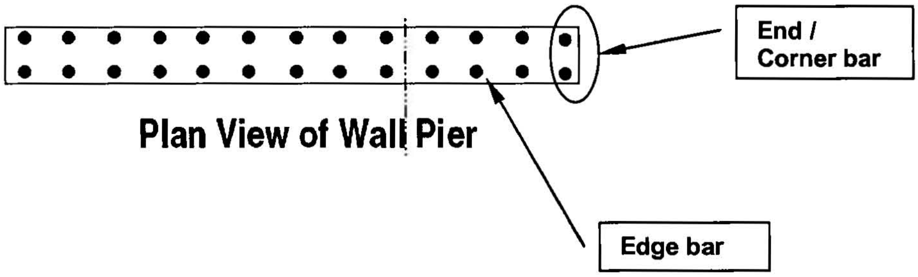
During the shear design process, the reduction in concrete shear capacity will be considered due to axial tension.

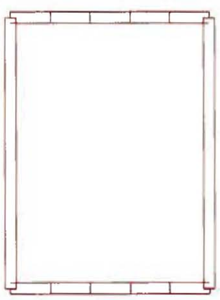
Uniform Reinforcing section

This wall design section will have uniform reinforcing (Edge bar & End bar) throughout the section. This wall section is applicable to two and three dimension shear wall systems. ETABS allows us to check & design for the user defined rebars details.

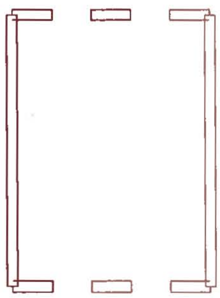
The planar pier sections with uniform reinforcing are designed for major axis moment and the minor axis moment i.e. out of plane moment is always ignored.

The geometry of a uniform reinforcing pier section is picked up automatically by the program from the area objects that define the pier section. If the pier section is made up of line objects only, the line objects are considered by the program when determining the pier geometry.

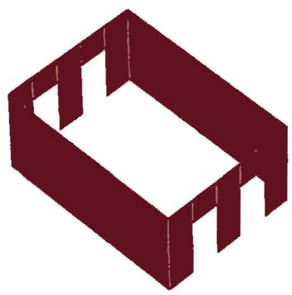




Pier section Top

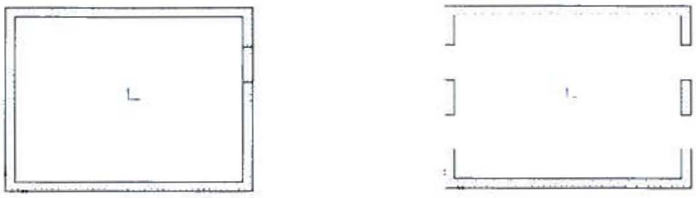


Pier section Bottom



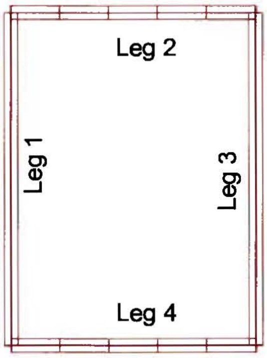
3D - view

View of objects as modelled in ETABS



View of pier section in Section Designer (Flexural Check)

The above picture shows the uniform pier section automatically assumed by the ETABS design module for the flexural design check.



The pier is broken up into four legs labelled Leg 1 through Leg 4. The shear design and boundary zone check are performed separately for each leg based on the forces in the leg. This automatically accounts for torsion in 3D wall piers.

View of pier section for shear design check

User needs to select the "Uniform Reinforcing" option in the Pier section type in the above form.

Uniform Reinforcing Pier Section - Design (BS8110 97)

Story ID: STORY3 Pier ID: P1 X Loc: 9 Y Loc: 12 Units: KN-m

Flexural Design for P-M2-M3 (RLLF = 1.000)

Station Location	Required Reinf Ratio	Current Reinf Ratio	Flexural Combo	P	M2	M3	Pier Ag
Top	0.0025	0.0028	COMB1	15000.000	0.000	0.000	0.500
Bottom	0.0025	0.0028	COMB1	15000.000	0.000	0.000	0.500

Shear Design For V2

Station Location	Rebar mm^2/m	Shear Combo	P	M	V	Capacity Vc	Capacity Vs
Top Leg 1	228.833	COMB1	15000.000	0.000	0.000	1922.744	200.000
Bot Leg 1	228.833	COMB1	15000.000	0.000	0.000	1922.744	200.000

The required flexural reinforcing ratio & horizontal shear reinforcing will reported for the governing design load combination.

Uniform Reinforcing Pier Section - Check (BS8110 97)

Story ID: STORY2 Pier ID: P1 X Loc: 9 Y Loc: 12 Units: KN-m

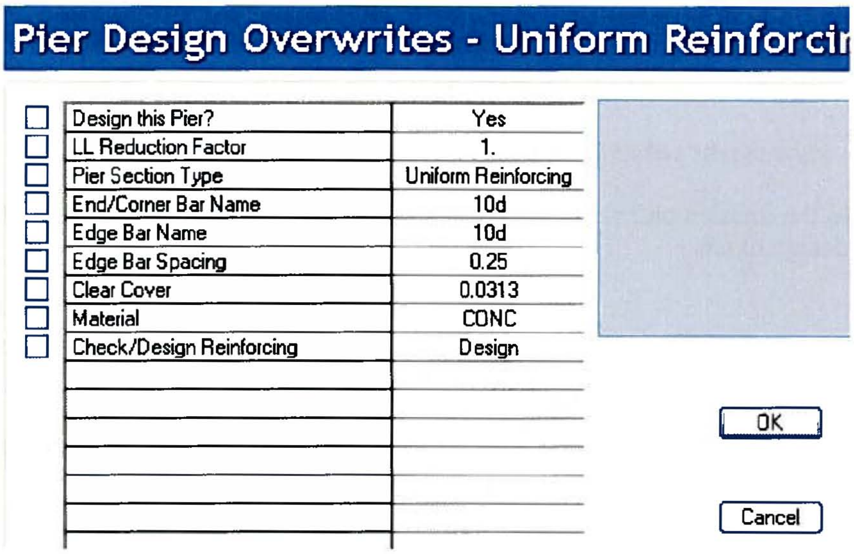
Flexural Check for P-M2-M3 (RLLF = 1.000)

Station Location	D/C Ratio	Flexural Combo	P	M2	M3
Top	0.925	COMB1	15000.000	0.000	0.000
Bottom	0.925	COMB1	15000.000	0.000	0.000

Shear Design For V2

Station Location	Rebar mm^2/m	Shear Combo	P	M	V	Capacity Vc	Capacity Vs
Top Leg 1	228.833	COMB1	15000.000	0.000	0.000	1922.744	200.000
Bot Leg 1	228.833	COMB1	15000.000	0.000	0.000	1922.744	200.000

The same uniform pier section will be checked. The demand / capacity ratio & required shear reinforcing will be reported. Shear strength can't be checked against the provided horizontal shear steel.



In the above pier design overwrite form; user can define rebar diameter and spacing, material and live load reduction factor (Minimum 0.5 as per BS).

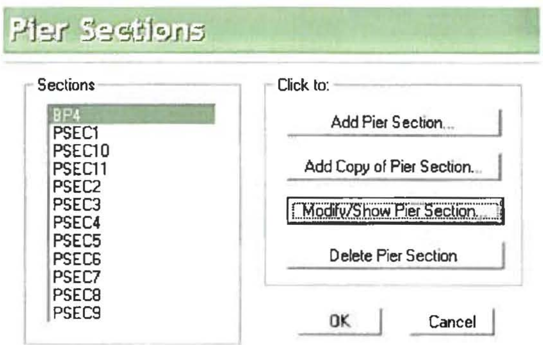
General Reinforcing section

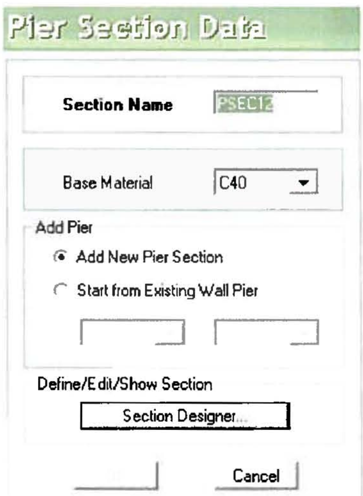
The concept for this design section is same as uniform reinforcing section with user defined reinforcing using section designer in ETABS. Here the user defines the geometry of the pier section and the size and location of vertical rebar.

Shear design will be performed same as uniform reinforcing section.

Define the general reinforcing section using the **Design menu > Shear Wall Design > Define Pier Sections for Checking.**

Base material type needs to be defined at this stage, which can't be revised in the design overwrites form later.

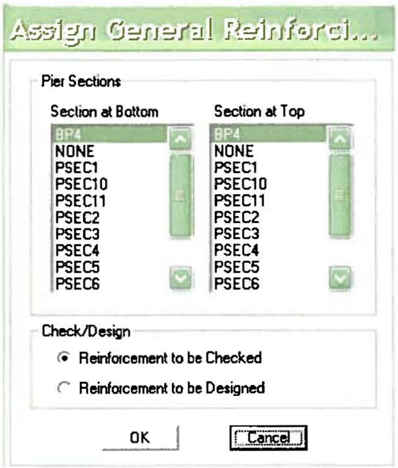




Add the pier section using the **Add Pier Section...** Option, Pier section will be added as a new section or start from existing wall pier.

Assign the general reinforcing pier section to the existing wall pier using the **Design menu > Shear Wall Design > Assign Pier Sections for Checking... >General Reinforcing Pier Section**

User defined pier section will be possible to assign at bottom and top of pier section with the option of design or checking.



User defined section will be checked for the user defined reinforcing pattern. The program will report the demand / capacity ratio and required shear reinforcing for the governing load combination.

If there is a three dimensional wall system, the shear design will be reported to the first inadequate leg or Leg requiring most rebar per unit length.

General Reinforcing Pier Section - Check (BS8110 97)

Story ID: L01 Pier ID: P10 X Loc: 16.475 Y Loc: 11.5 Units: KN-m

Flexural Check for P-M2-M3 (RLLF = 1.000)

Station Location	D/C Ratio	Flexural Combo	P	M2	M3
Top	0.602	ULTENV	137973.600	11957.388	19615.910
Bottom	0.701	ULTENV	136300.534	11502.300	16919.518

Shear Design For V2 - First Inadequate Leg or Leg Requiring Most Rebar per Unit Length

Station Location	Rebar mm^2/m	Shear Combo	P	M	V	Capacity Vc	Capacity Vs
Top Leg 2	274.600	ULTENV	11008.705	3233.422	236.406	4367.618	912.000
Bot Leg 1	274.600	ULTENV	1661.322	22.977	104.497	846.059	133.200

The pier section can also be designed using this option but it reports the required reinforcing ratio assuming uniform reinforcing pattern.

General Reinforcing Pier Section - Design (BS8110 97)

Story ID: L01 Pier ID: P10 X Loc: 16.475 Y Loc: 11.5 Units: KN-m

Flexural Design for P-M2-M3 (RLLF = 1.000)

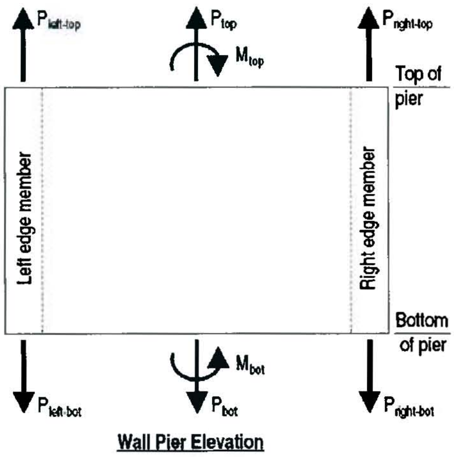
Station Location	Required Reinf Ratio	Current Reinf Ratio	Flexural Combo	P	M2	M3	Pier Ag
Top	0.0059	0.0336	ULTENV	137973.600	11957.388	19615.910	7.890
Bottom	0.0159	0.0346	ULTENV	136300.534	11502.300	16919.518	6.558

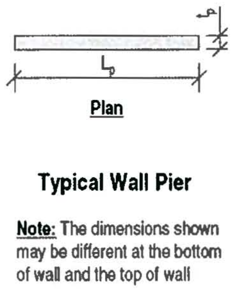
Shear Design For V2 - First Inadequate Leg or Leg Requiring Most Rebar per Unit Length

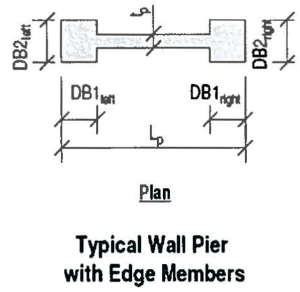
Station Location	Rebar mm^2/m	Shear Combo	P	M	V	Capacity Vc	Capacity Vs
Top Leg 2	274.600	ULTENV	11008.705	3233.422	236.406	4367.618	912.000
Bot Leg 1	274.600	ULTENV	1661.322	22.977	104.497	846.059	133.200

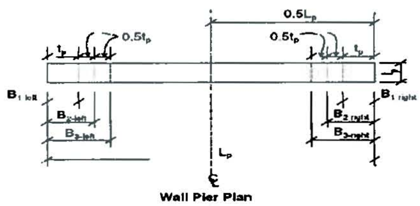
Simplified C & T Pier section

The pier geometry is defined by a length, thickness and size of the edge members at each end of the pier (if any).









Assign the simplified pier section to the existing wall pier using the **Design menu > Shear Wall Design > Assign Pier Sections for Checking... > Simplified C and T Section**

Perform the simplified C & T section design using the **Design menu > Shear Wall Design > Start Design / Check of Structure**

The pier section flexural design will be performed to edge strip of pier section and it ignores the resistance from the middle strip.

Program will report the required width of edge strip to resist the axial and over turning moment and required reinforcing in the governing compression and tension load combination.

Program will perform shear design by considering the full length of the pier section and reports the required shear reinforcing per unit length for the governing load case.

Simplified T and C Pier Section - Design (BS8110 97)

Story ID: STORY4 Pier ID: P1 X Loc: 9 Y Loc: 12 Units: KN-m

Flexural Design for P and M3 (RLLF = 1.000)

Station Location	Edge-Length	Tension Rebar mm^2	Tension Combo	P	M
Left Top	0.875	0.000	COMB1	15000.000	0.000
Right Top	0.875	0.000	COMB1	15000.000	0.000
Left Bottom	0.875	0.000	COMB1	15000.000	0.000
Right Bottom	0.875	0.000	COMB1	15000.000	0.000

Station Location	Edge-Length	Compression Rebar mm^2	Compression Combo	P	M
Left Top	0.875	6232.150	COMB1	15000.000	0.000
Right Top	0.875	6232.150	COMB1	15000.000	0.000
Left Bottom	0.875	6232.150	COMB1	15000.000	0.000
Right Bottom	0.875	6232.150	COMB1	15000.000	0.000

Shear Design For V2

Station Location	Rebar mm^2/m	Shear Combo	P	M	V	Capacity Vc	Capacity Vs
Top	228.833	COMB1	15000.000	0.000	0.000	1727.494	200.000
Bottom	228.833	COMB1	15000.000	0.000	0.000	1727.494	200.000

Limitations:

In planar walls, the minor axis moment effect will be ignored.

Effects resulting from warping stresses i.e. torsion will be ignored. (wall with numerous opening)

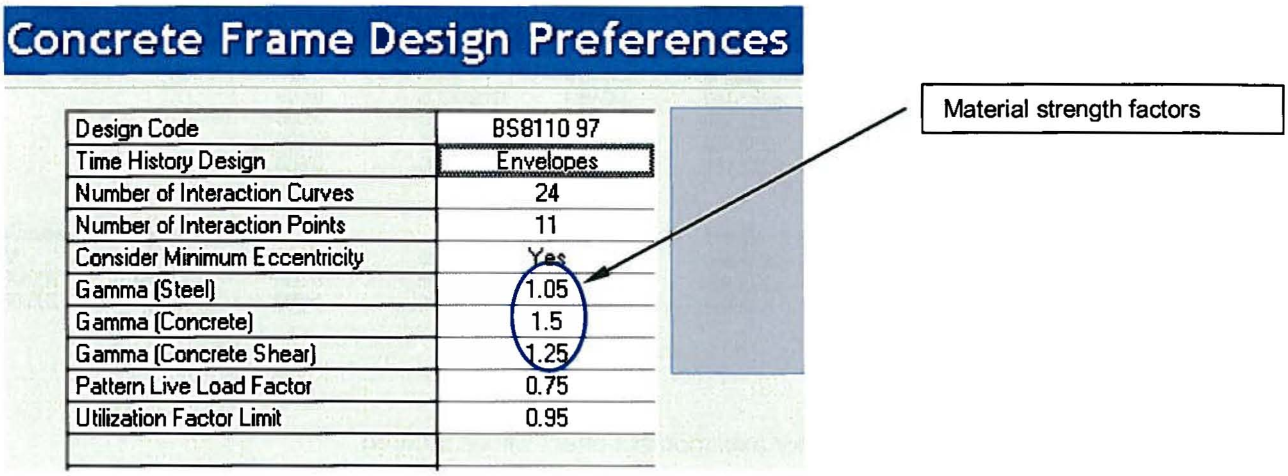
Assumes that the members with shear reinforcement providing a design shear of 0.4 N/mm2.

Shear stress is not limited to the requirement of high strength concrete. (I.e. fcu > 55 MPa)

9.2 Reinforced Concrete Frame Design (BS 8110-97)

Preferences:

To view preferences, select the **Options menu > Preferences > Concrete Frame Design** command



Design Code	BS8110 97
Time History Design	Envelopes
Number of Interaction Curves	24
Number of Interaction Points	11
Consider Minimum Eccentricity	Yes
Gamma (Steel)	1.05
Gamma (Concrete)	1.5
Gamma (Concrete Shear)	1.25
Pattern Live Load Factor	0.75
Utilization Factor Limit	0.95

Overview

In the design of concrete beams, the program calculates and reports the required areas of steel for flexure and shear based upon the factored beam moments and shears. The reinforcement requirements are calculated at a user-defined number of check stations along the beam span.

It is assumed that the design ultimate axial force does not exceed $0.1 f_{cu} A_g$ (BS 3.4.4.1); hence, all the beams are designed for major direction flexure and shear only.

Limitations:

1) Effects resulting from axial forces, minor direction bending and torsion will be ignored.

2) Assumes that the members with shear reinforcement providing a design shear of 0.4 N/mm2. (Minimum shear links)

3) Shear stress is not limited to the requirement of high strength concrete. (i.e. fcu > 55 MPa)

4) This design module helps only in the preliminary sizing of beam members. (major flexure & shear)

5) Limit state of serviceability of beams are not determined (deflection, crack width).