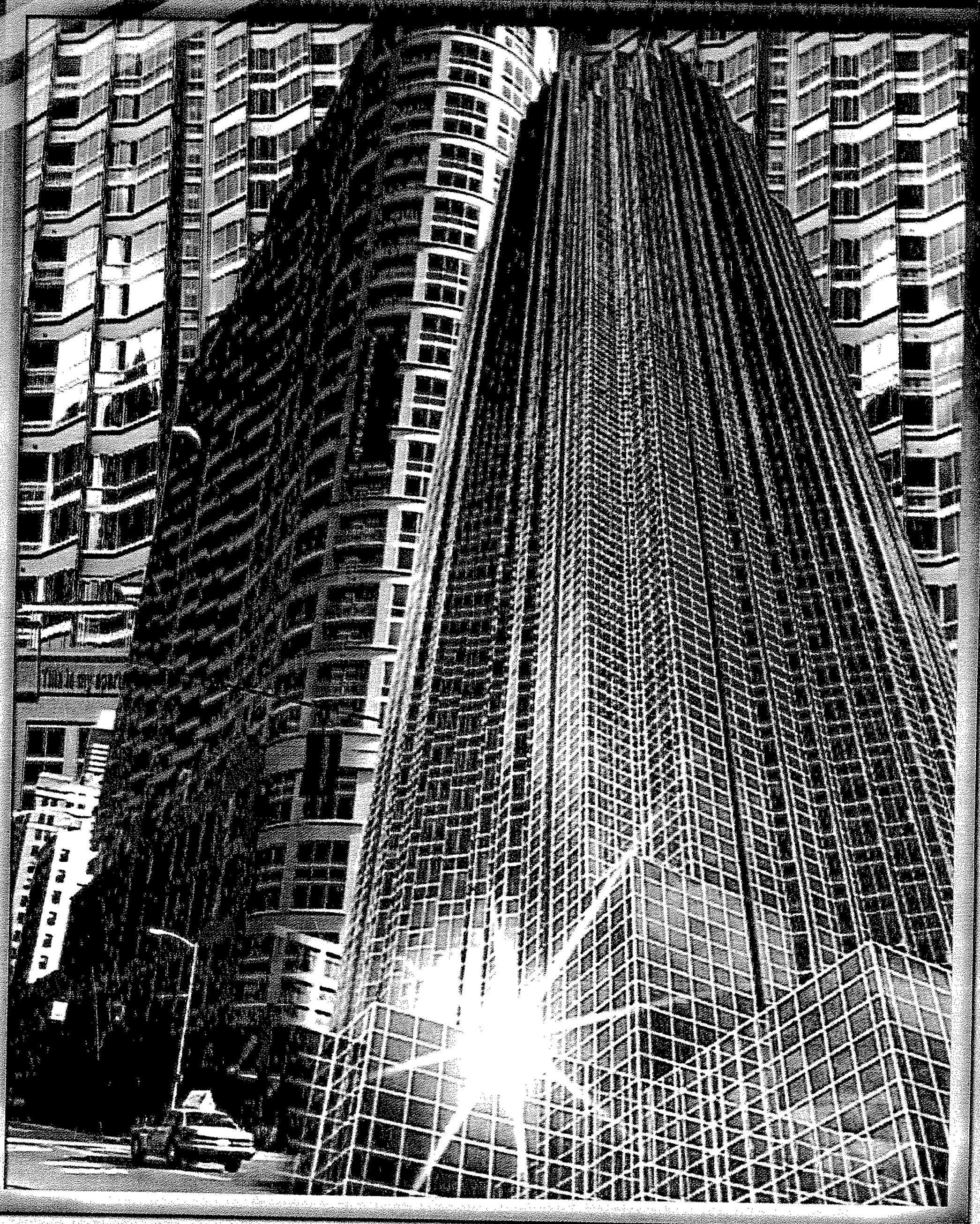
HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program



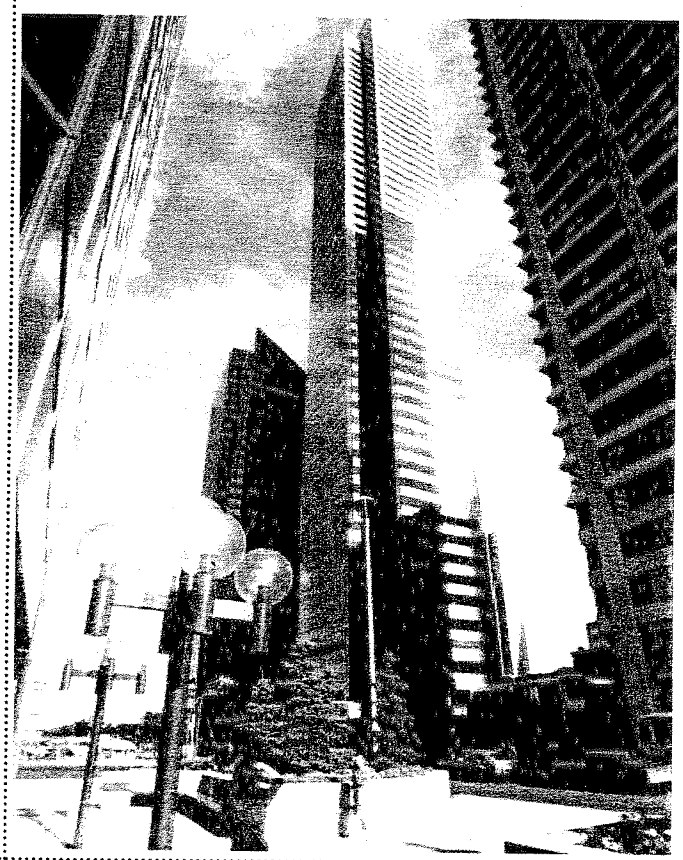
Prepared by

Eng. Makar Nageh

SBH

HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING

ETABS Program



Prepared by

Eng. Makar Nageh

Dep. No. 2196 / 2196
ISBN : 977-287-691-4

Design By : Gamal A. Khlifa

Scientific Book House
For Publishing & Distributing

50, El-Sheikhh Rehan Str,
Aabdeen, Cairo
Tel &Fax 7954229 - 7948619

لمزيد من المعلومات يرجى زيارة موقعنا على الأنترنت

www.sbheg.com
e-mail: sbh@link. net

Acknowledgments

In particular, thanks are for my father and my mother for their continuous support and encouragement

Foreword

Most present –day design is carried out using computer program. This book offers a lucid and coherent presentation of the most famous program of high-rise building (ETABS v9). This book is based on ETABS v9 and applicable to Etabs v8. The book contains detailed example of 40 story tower and detailed explain for the most important utility of the program (Like design, Dynamic analysis) without descanting due to the limitations on the size of the book without any effect of the purport.

I didn't take about the codes details because this out of the scope of this book but I mention any data required from the codes through the book chapters.

I attempted to not repeat explanation of some point many times but I refer to the explanation of this point when it is repeated.

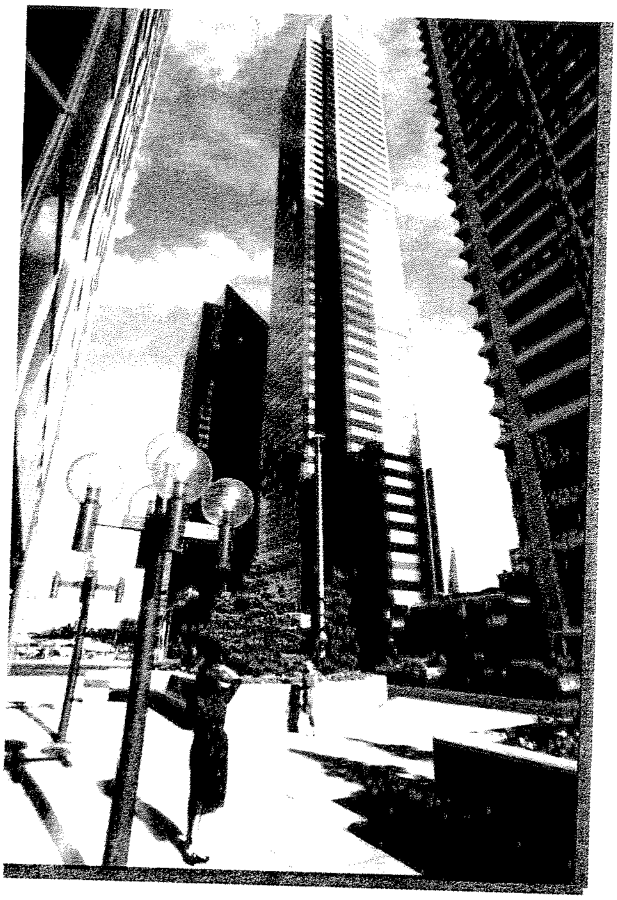
I used the step-by-step instruction guide though development of ETABS model to show how quickly and easily a model can be created using this program.

The book contain the very important nonlinear analysis called sequential construction and explanation of the difference between the normal analysis and Sequential construction analysis, and also contain explanation of the ability of the program to work with any different sections, and how to define this sections to the program .

I explained through this book the program ability to transfer information from the ETABS database for use with other software package.

Makar Nageh

Index



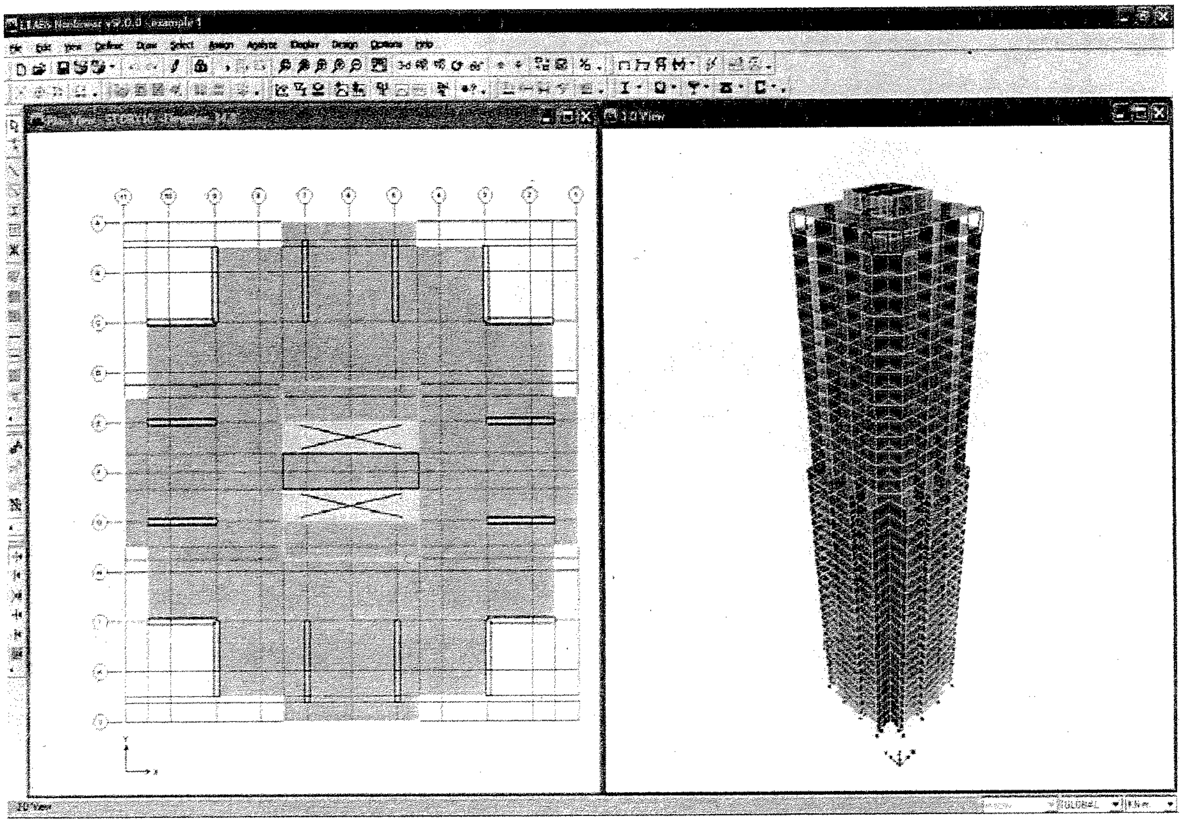
HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

Modeling & Analysis

Chapter 1

An Example Model

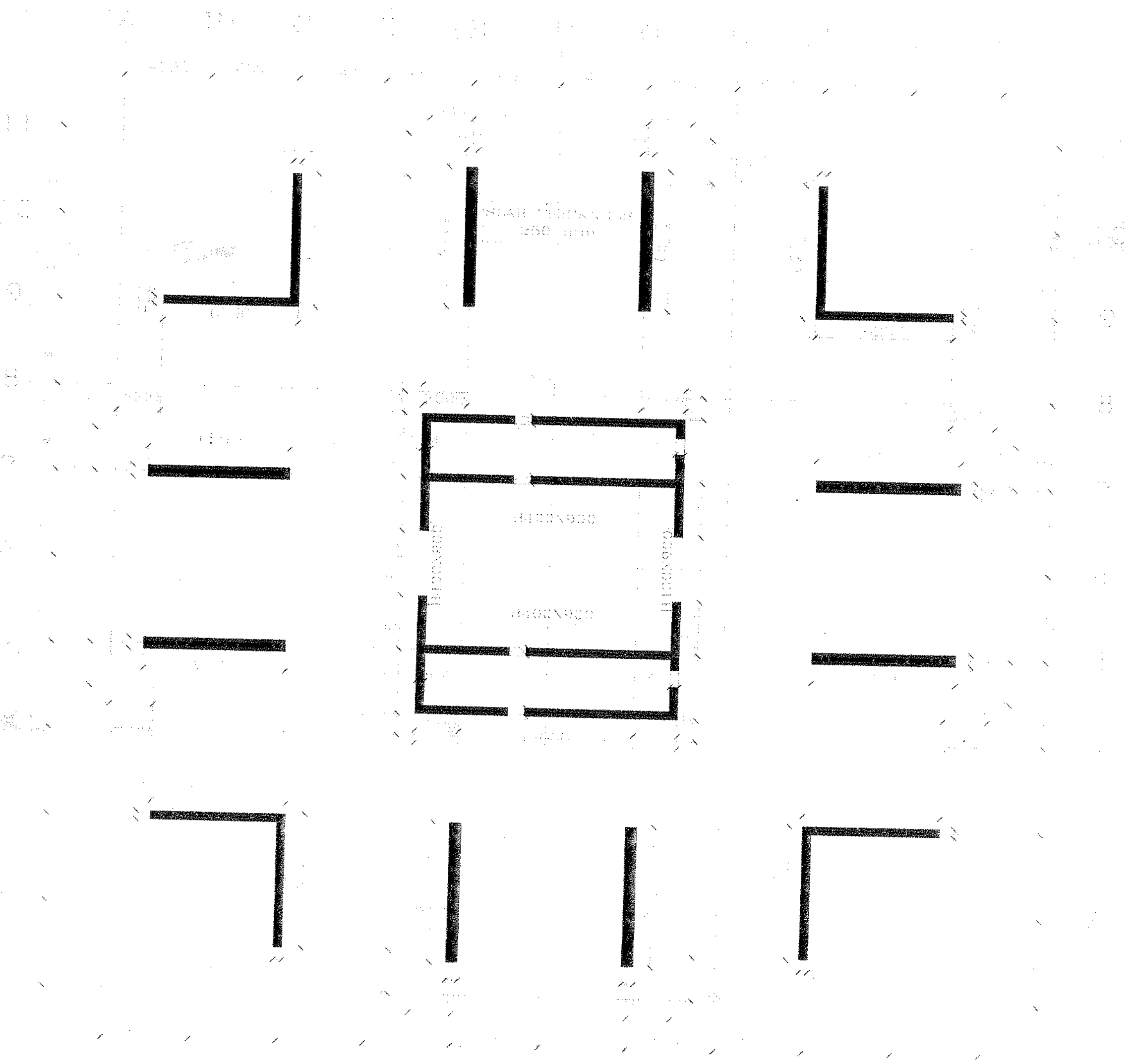
- In this Example we will built the model shown in the figure. Each step of the model creation process is identified, and various modal construction techniques are introduced. At the completion of this Example, You will have found that, it is so easy to built Etabs model in a few hours.



- The step by step instruction will guide you through development of your first model

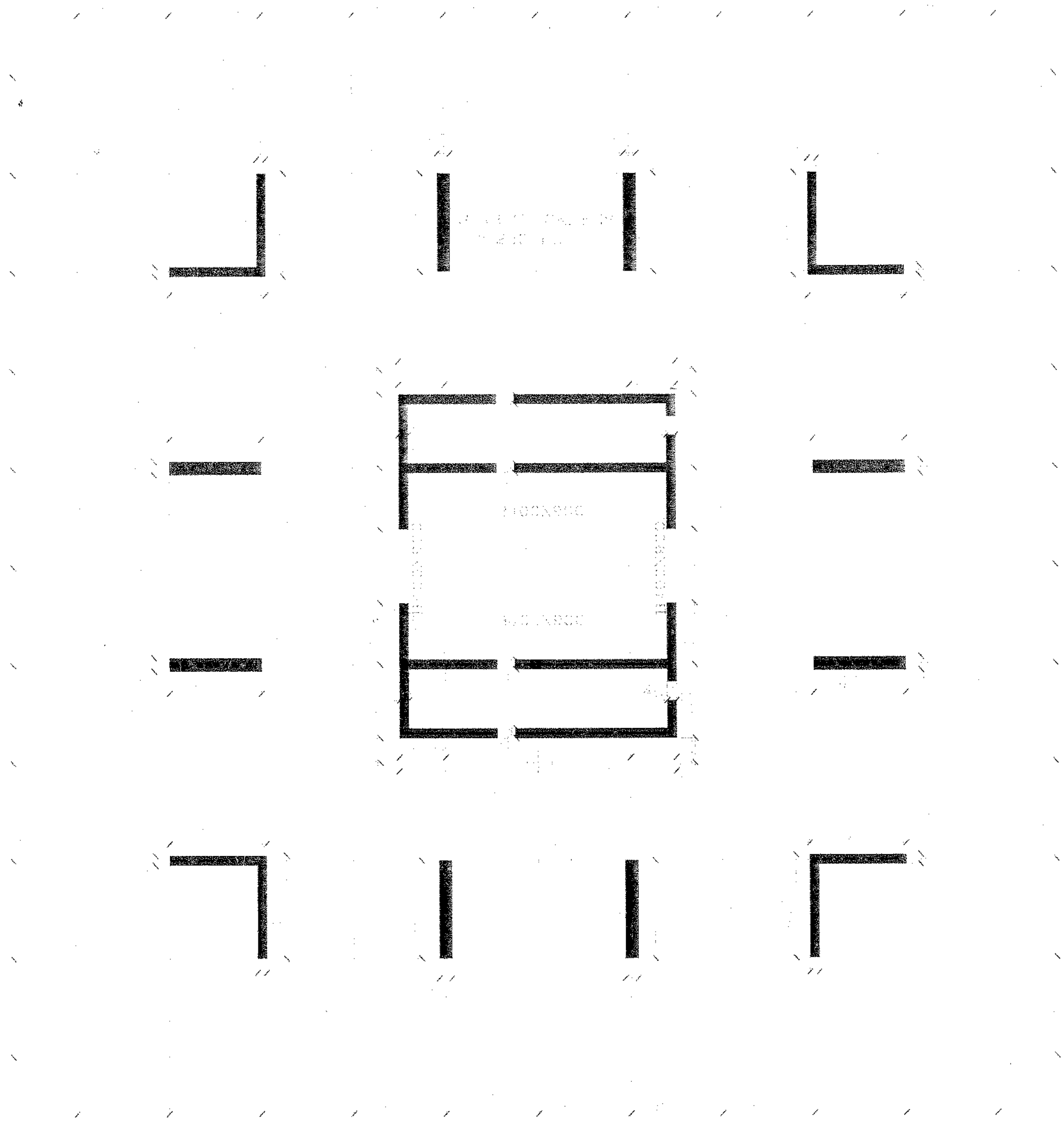
The Example Project

- The tower of our example is 40 story building.
- The first story height is 4.2m and the typical stories height is 3.4m.
- The lateral force resisting system consists of 8 Shear walls,4 L shear walls, and 1 core wall
- The plans of the tower floors as shown blow

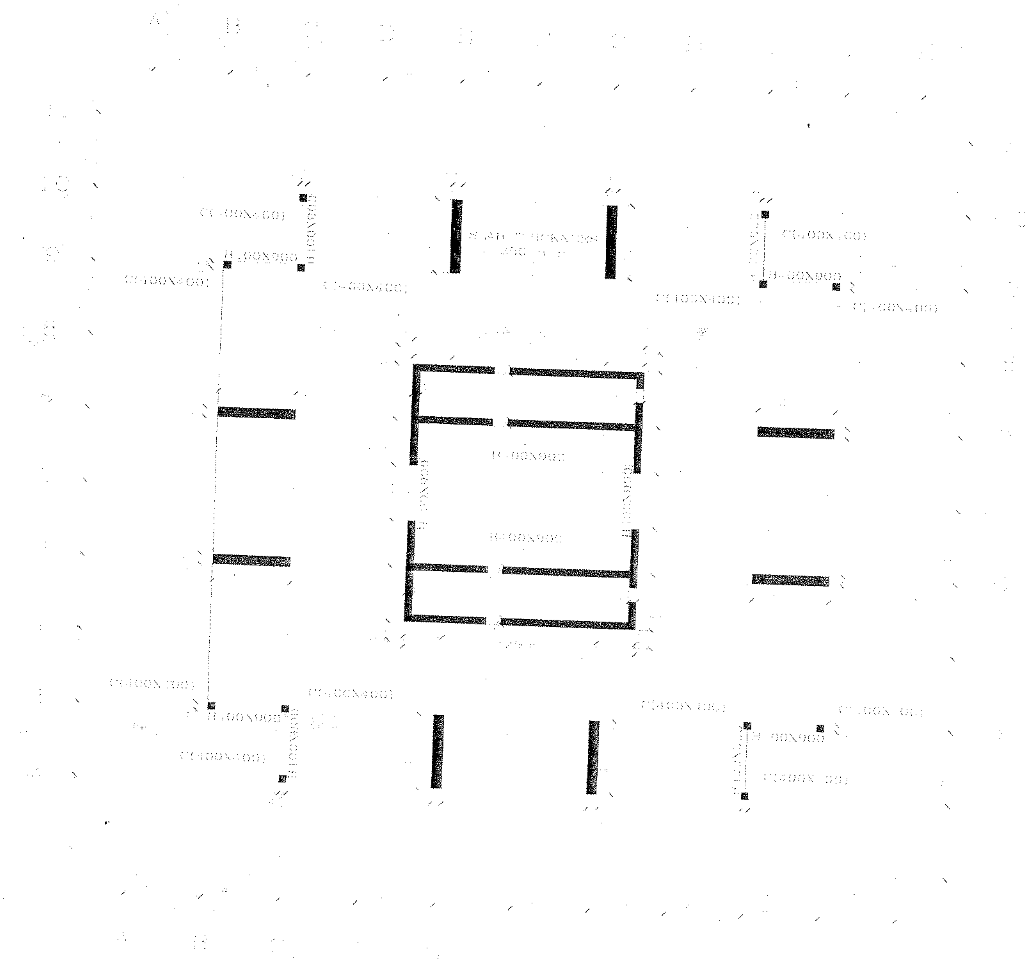


From level 1 to level 23

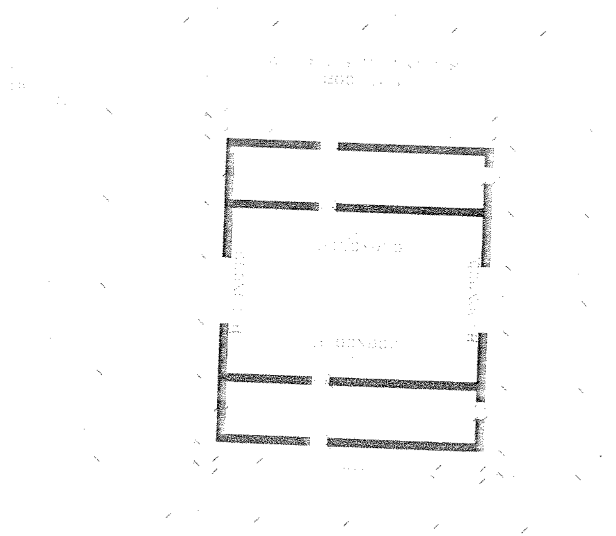
Note: the slab thickness of level 23 is 300 mm



From level 24 to level 38



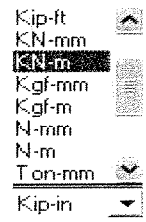
Level 39



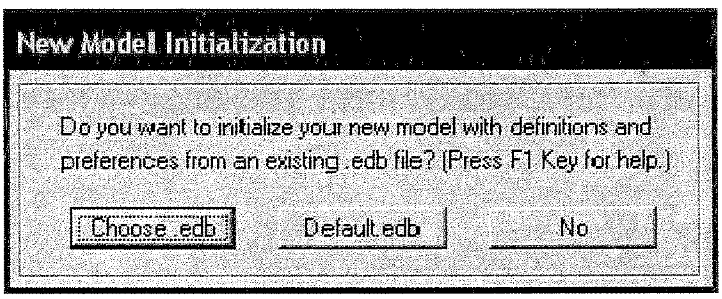
Level 40 (ROOF FLOOR)

Step 1: Begin anew model

- In this step, the dimensions and story height are set.
  1. open the program
  2. Check the units of the model in the drop-down box in the lower right-hand corner of the Etabs window , click the drop-down box to set the units to **KN-m**

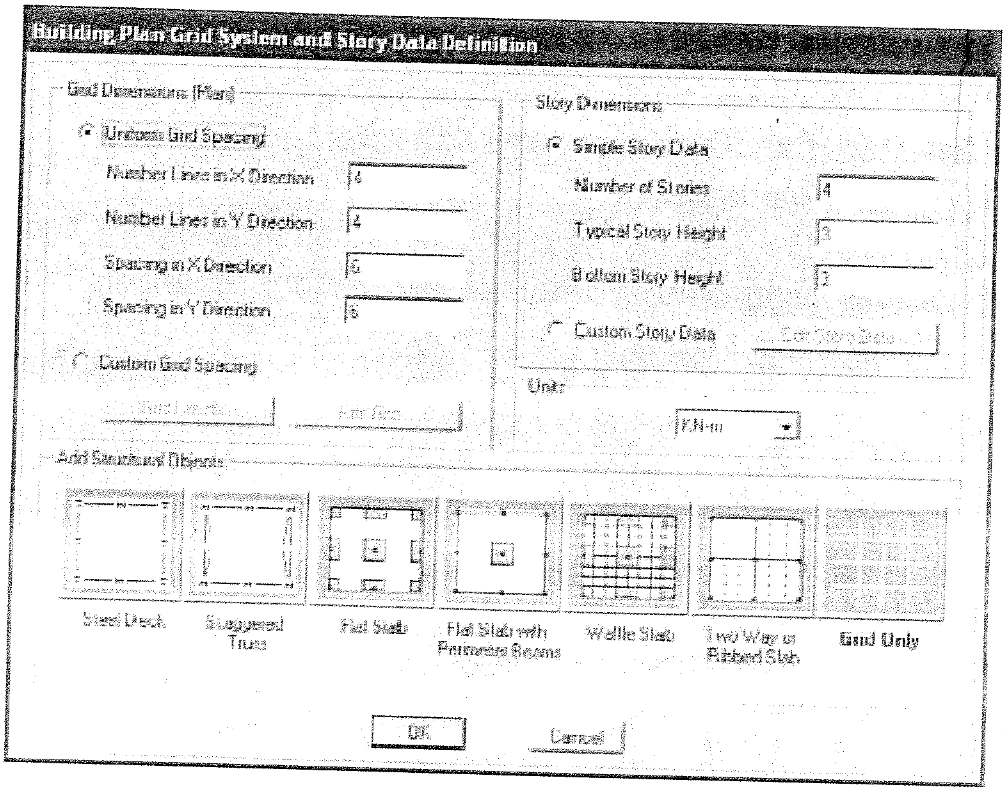


  3. Click the **File menu** ⟶ **New model command** Or the New Model button the form shown in figure will be displayed



Note: we select No because this first model you will built, and we discuss in the another chapter this form again because by this form you can safe more than 40%of the time in building the model

  4. the next form of **Building Plan Grid System and Story Data Definition** will be displayed after You select click **NO** button



- In the form of **Plan Grid System and Story Data Definition** you can:

1. **Set the Grid Dimensions(plan)**

   - Set the Number of Grid lines in X direction in the Number of lines in X direction edit box =11 (Horizontal Grid)

   - Set the Number of Grid lines in Y direction in the Number of lines in Y direction edit box =11(Vertical Grid)

   - Set the Spacing between Grid lines in X direction in the Spacing in X direction edit box =4 m

   - Set the Spacing between Grid lines in Y direction in the Spacing in Y direction edit box =4 m

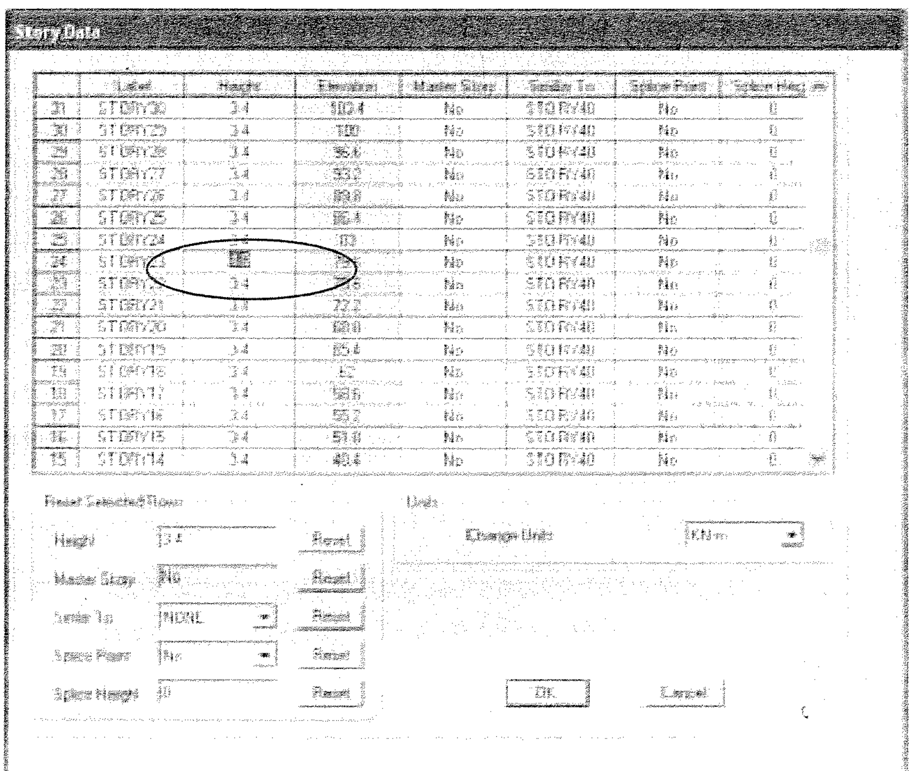
2. Set the Story Data

- **Set the number of stories**, in our example we have 40 stories building so that we will set the number of stories in the number of stories edit box equal to **40**.

- **Set the story height** Set the Typical Story Height in the **Typical story height** edit box to **3.4 m**

- Set the Bottom Story Height in the **Bottom story height** edit box to **4.2 m**

- You can change any story height by choose **custom story data** , then click on **Edit Story Data**, the program will open the form of all stories level, we will change the height of 23rd floor (mechanical floor)to 4m

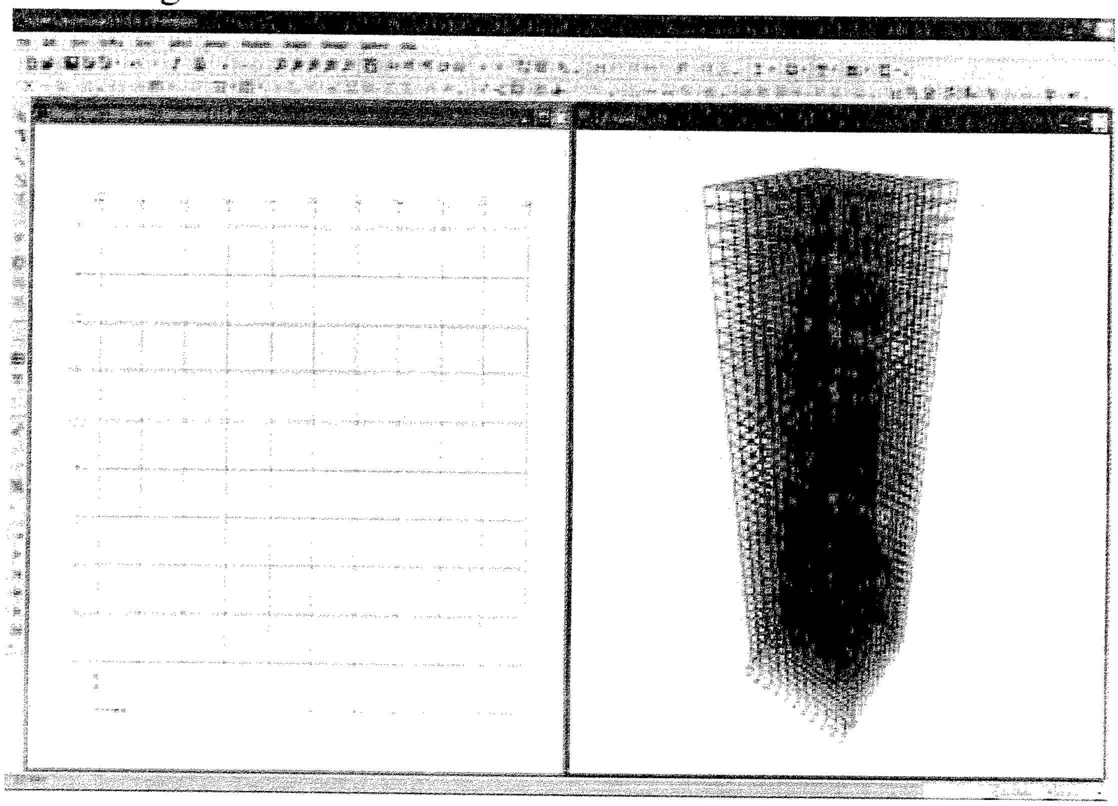


And also in this form we can adjust similar story, our master Stories is level1, level 23, level 39, level40

5. Select the **Grid Only** button

***Note**: it is highly recommended that to start your models using templates,*

6. Click the **OK** button to accept your changes. Your Model appears on screen in the main ETABS window with two view window, plan view and 3-D view as shown in the Figure.

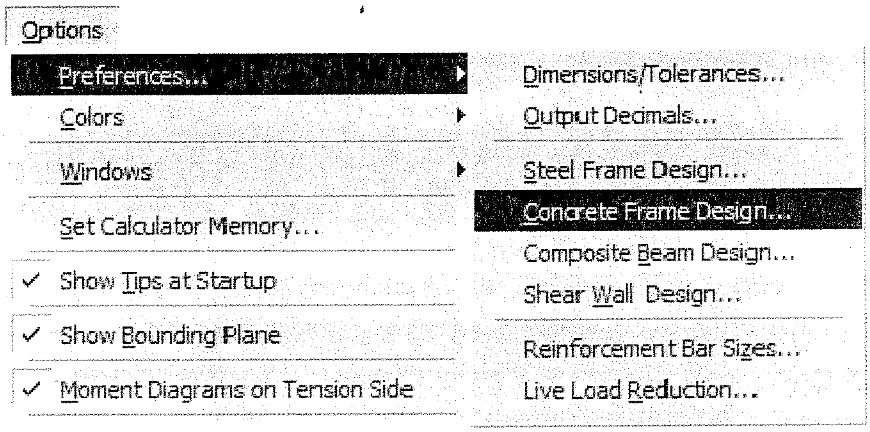


Define the Codes Used in the design of the Building:

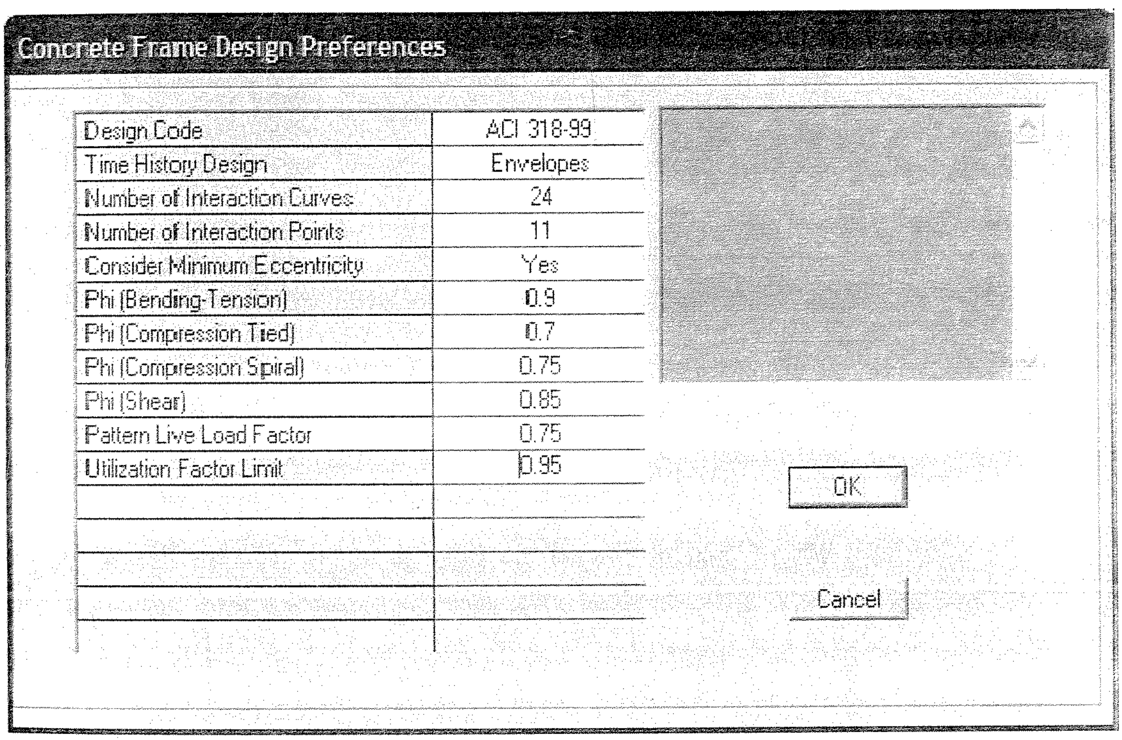
In this step, we will define the codes used in the project, to define the properties of material according to this code, and make the program create the load of combination automatically according to this code, and use this code for design

1. Define the code for design The Frame Elements

- Click the **Option menu** ⟶ **Preferences** ⟶ **Concrete Frame Design**
-

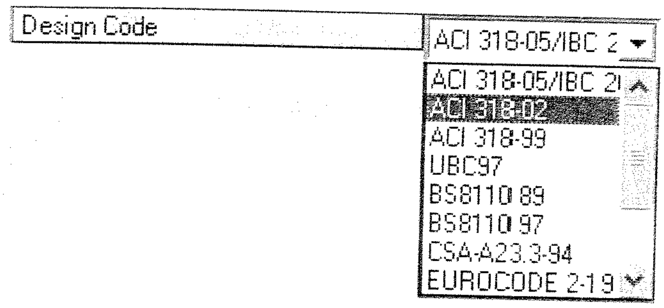


This will Display the **Concrete Frame Design Preference** Form as shown in the figure.

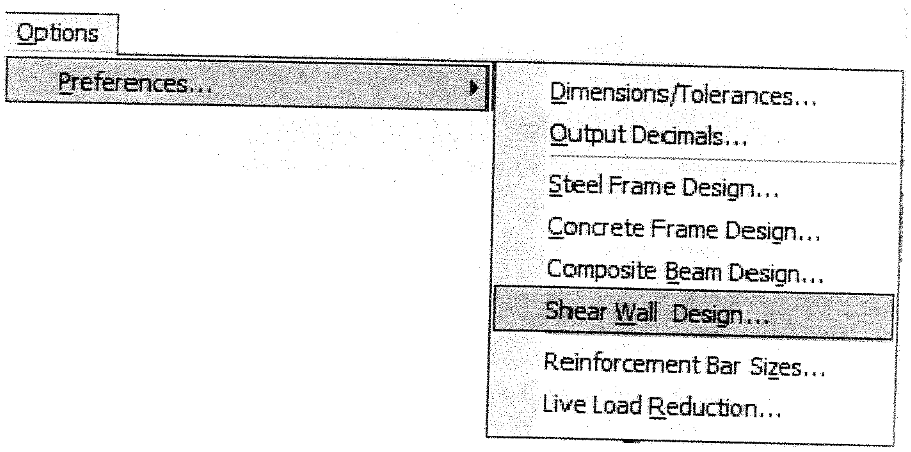


From this form, you can adjust the code and the another data of design according to the requirement of design, and in our example we will choose the **ACI 318-02** as a code of our design

- Click the drop-down box of the design code to set the code (**ACI 318-02**)



- Repeat the same steps for assigning the code of design for shear wall design from the same menu



- **Step 2: Define the Materials & Sections**

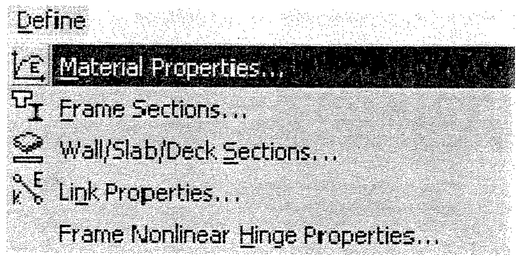
In this step, we will define the materials, and the sections of Walls, Slabs, and Beams of the structure

1. *Define of materials*

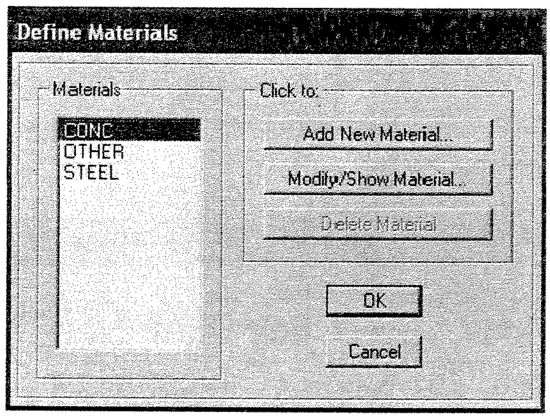
Note: we will use in our model 2 different types of concrete

- Concrete grade 60000 KN/m^2 for wall columns
- Concrete grade 40000 KN/m2 for Slabs Beams

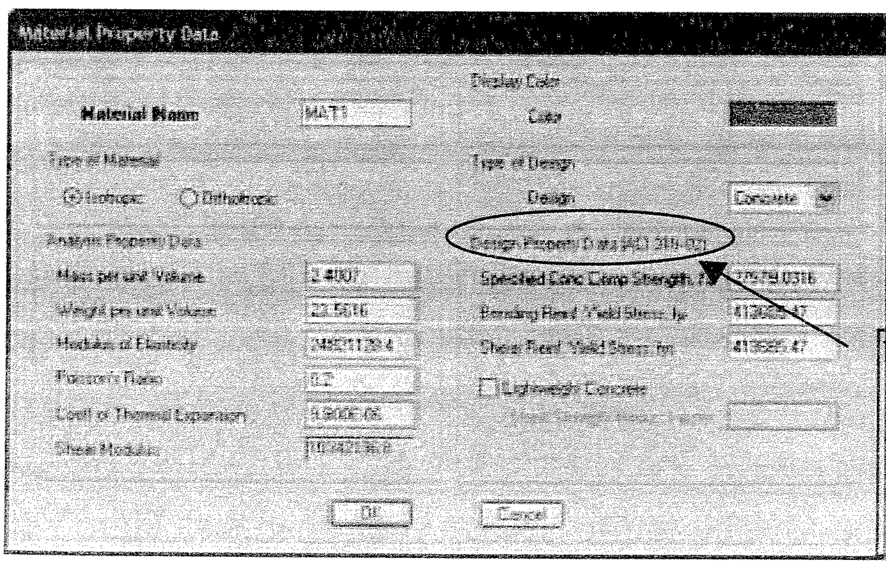
1. Click the **Define menu** ——► **Material Properties**



Or Material Properties button which will Display the define Material Properties form as shown in the figure.

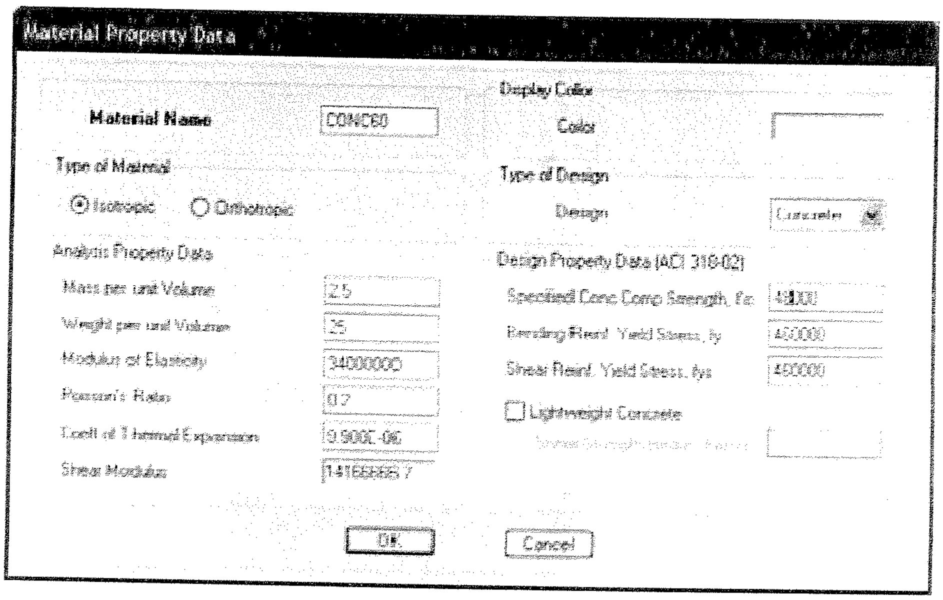


Click **Add New Material** button to Display the Material Property Data form which is shown in the figure.

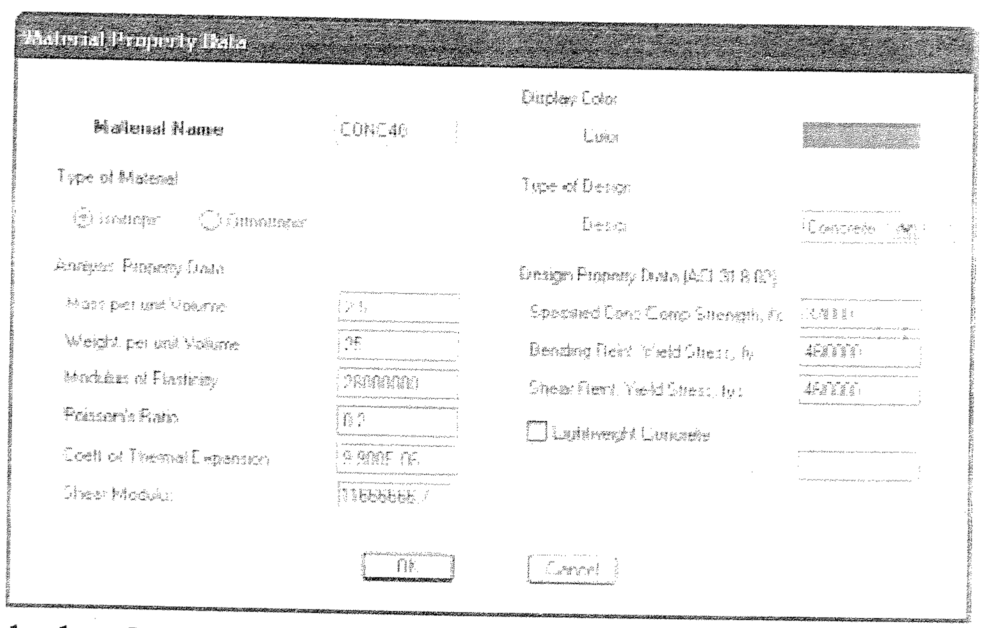


The definition will be according to ACI 318-02

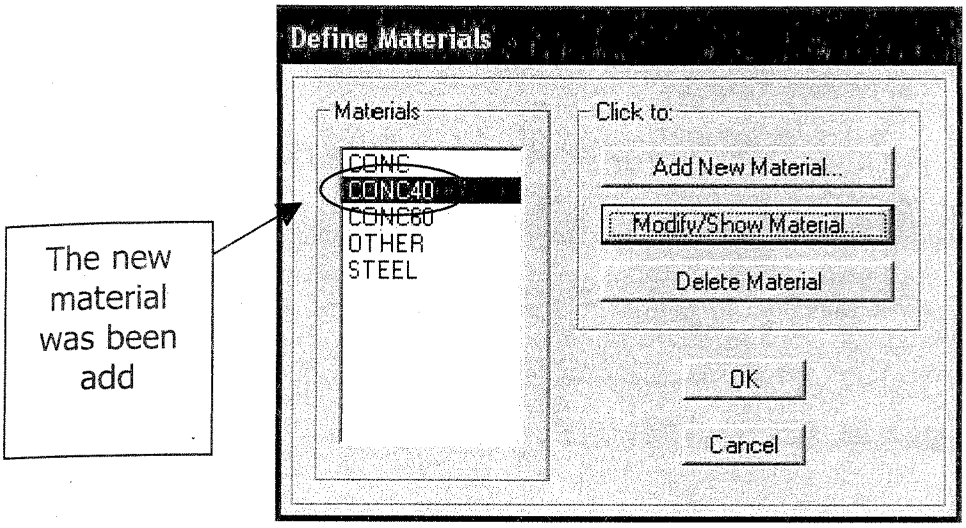
- Define the material properties in that form to be according the design code.
    - Concrete grade 60000 KN/m^2 for wall



- Repeat the previous step, Click **Add New Material** button to define the concrete for slabs (Concrete grade 40000 KN/m^2 for Slabs Beams) and fill the form to be as in the figure



- Click the **OK** button to accept your changes. Thin You will find the new materials were Add



- Click the **OK** button to accept your changes.

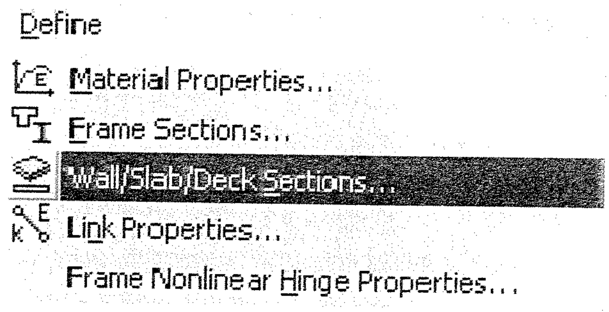
2. *Define of sections(Walls,Columns,Slabs,Beams)*

Note: for simplification of our first model we will use

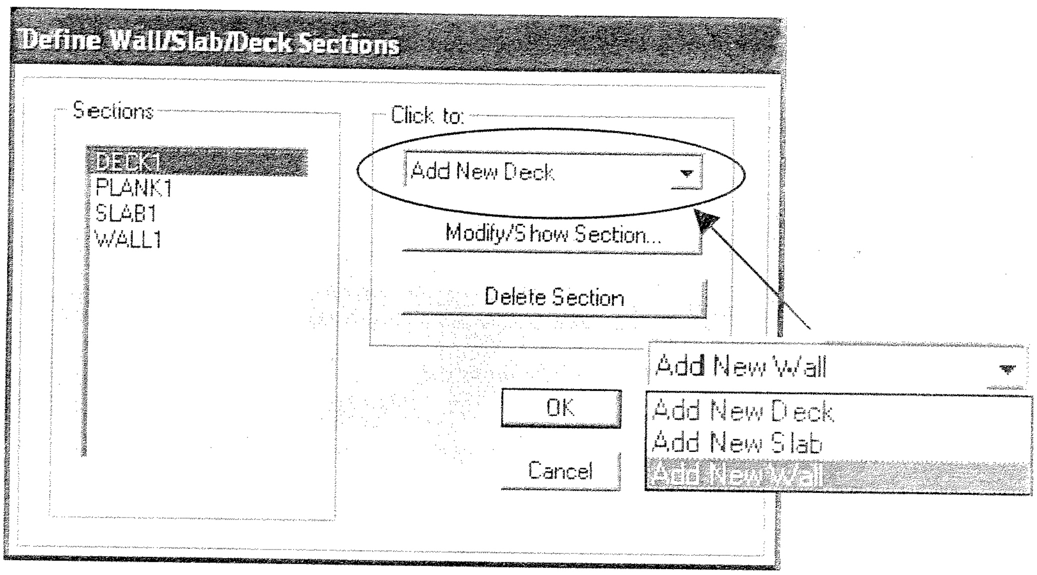
- 2 Wall Thickness
    1. Wall 400=40 cm
    2. Wall 500=50 cm
- 2 slab Thickness
    1. Slab 250=25 cm
    2. Slab 200=20 cm
- 1 Beam Size (For Beams and Spandrel beams)
    1. B (400X900)
- 1 Column Size
    3. C400x400

1. Define of the Walls section :

1. Click the **Define menu** ——▸ **Wall/Slab/Deck Sections**



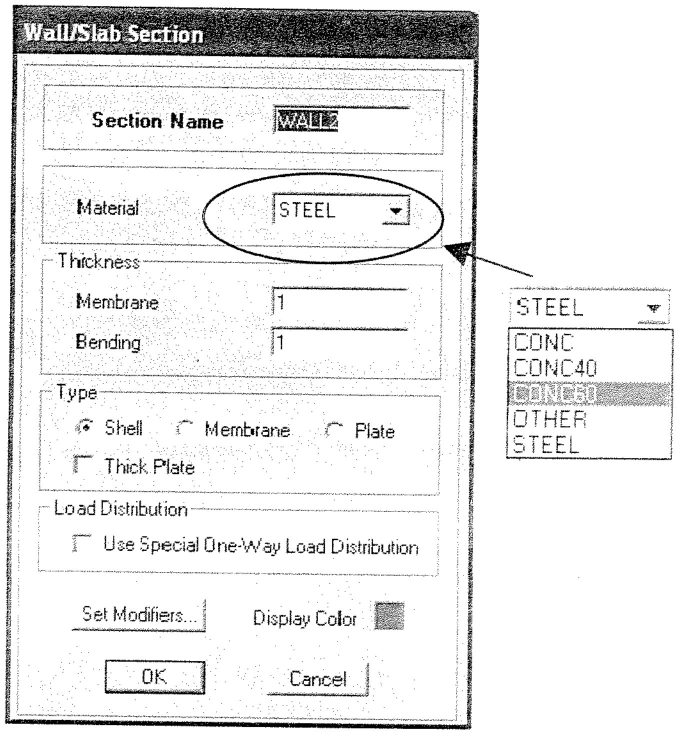
Or Define Wall/Slab/Deck Sections button which will Display the define Wall/Slab/Deck Sections form



Note: From this form we can define Walls and Slabs

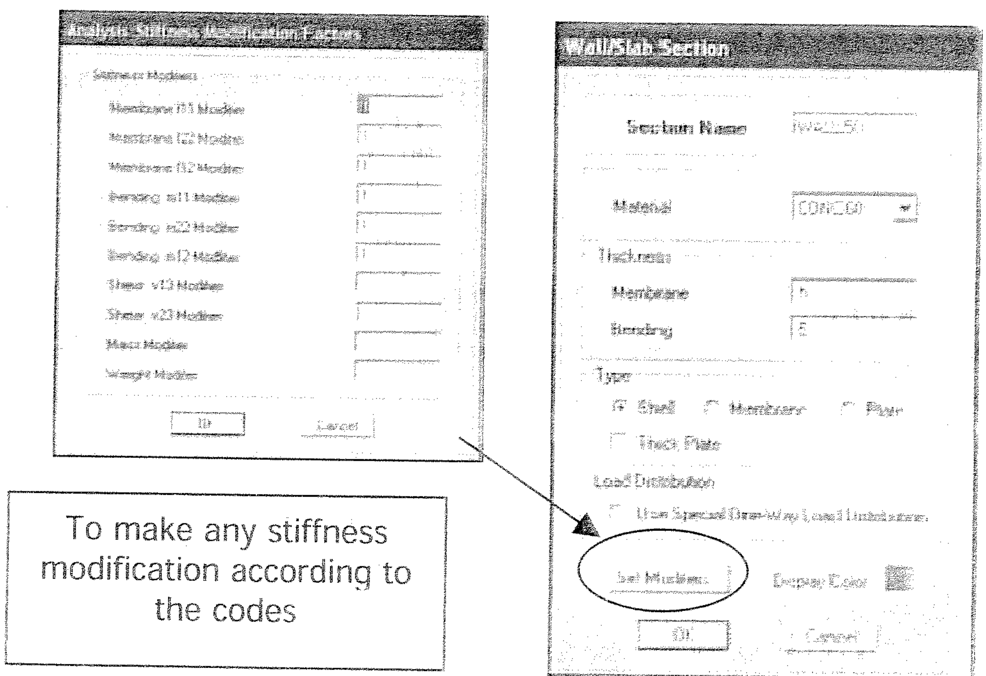
2. Click the drop-down box that reads "**Add New Deck**"

   Choose Add New Wall from the list, thin the form of Wall define will be opened



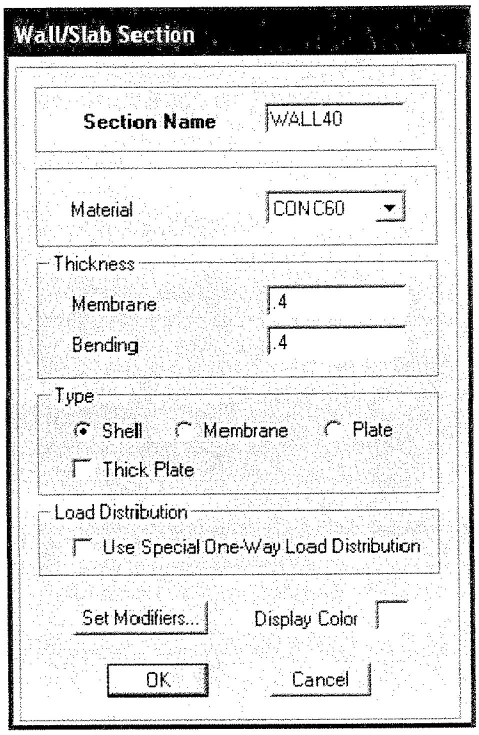
Note: we will define the Wall 50 cm thickness

- In this from change the section name to wall 50
- Click the drop-down box of material and choose conc. 60
- Define the thickness of wall =.5 m
- The form will be as shown

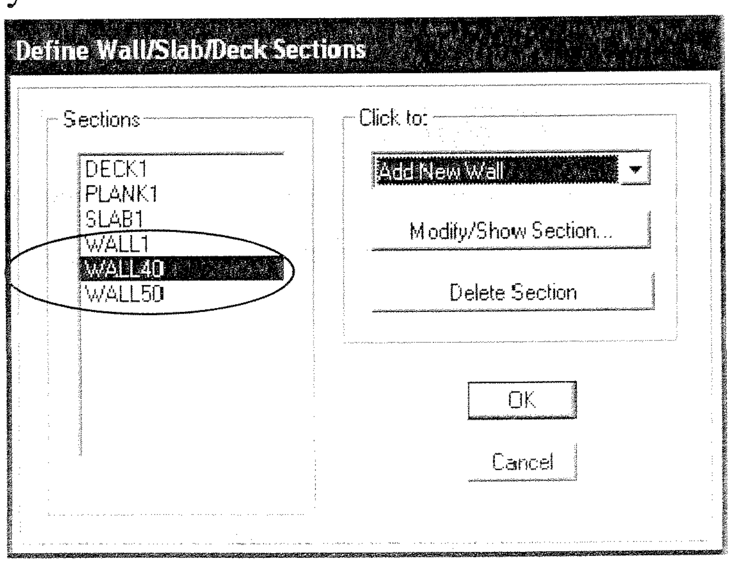


Note: for simplification in our example we will not change the stiffness modifiers but in any another modal you can changed according to the design code

- Click the **OK** button to accept your changes.
- Repeat the same steps for the wall 40 cm thickness and the form will be as shown in the figure

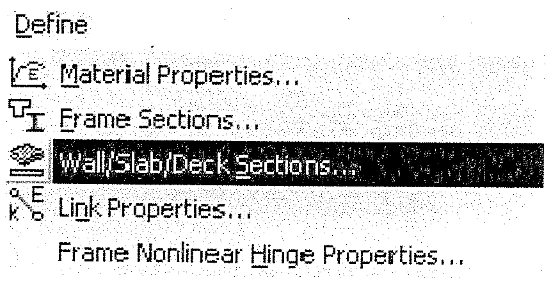


- Click the **OK** button to accept the changes. and the program automatically will add the new sections to the list of the sections

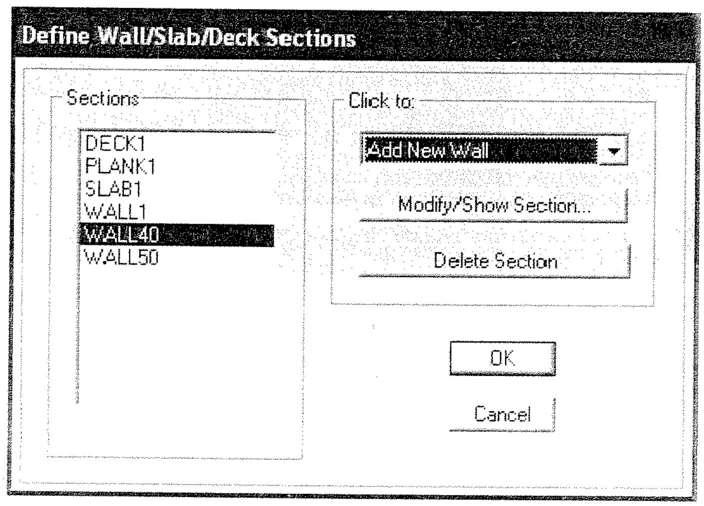


2. Define of the Slabs

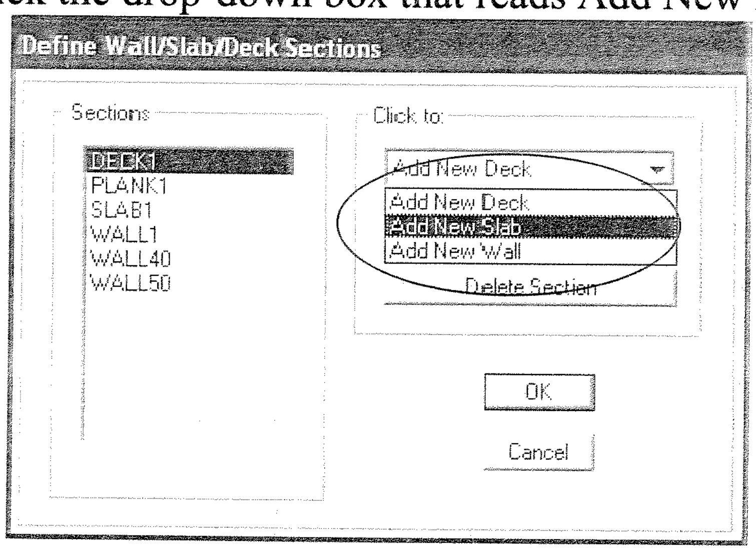
1. Click the **Define menu** ——▶**Wall/Slab/Deck Sections**



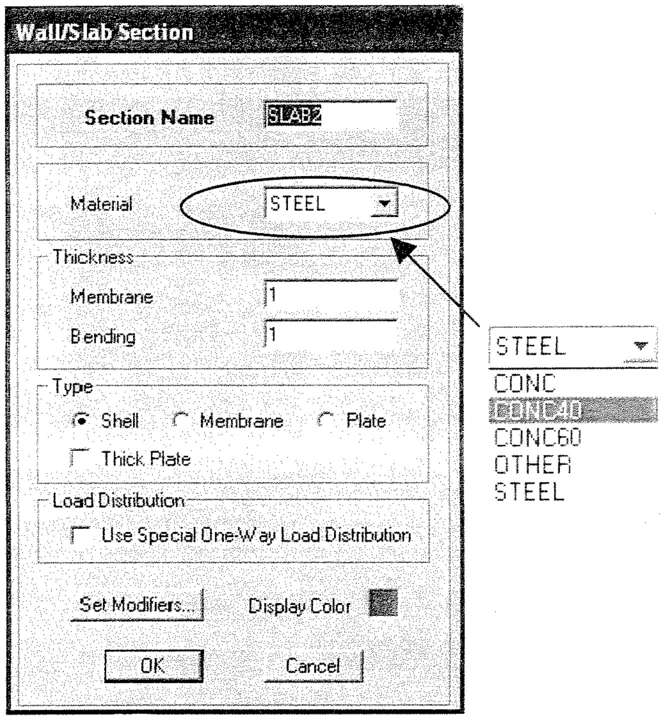
Or Define Wall/Slab/Deck Sections button which will Display the define Wall/Slab/Deck Sections form



2. Click the drop-down box that reads Add New Deck

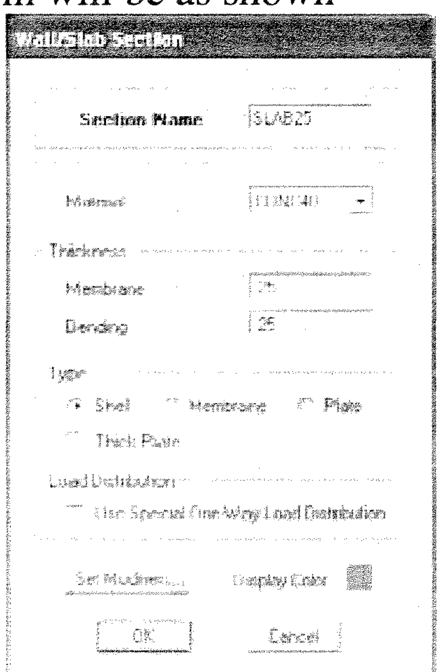


Choose Add New Slab from list thin the form of Slab define will be opened

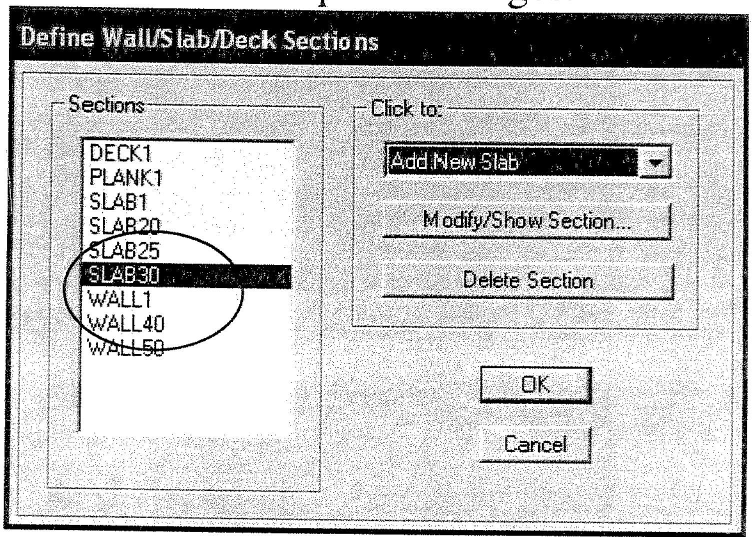


Note: we will define the Slab 25 cm thickness

- In this from change the section name to Slab 25
- Click the drop-down box of material and choose conc. 40
- Define the thickness of the Slab =.25 m
- Thin the form will be as shown

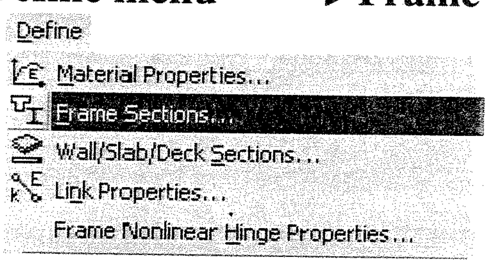


- Click the **OK** button to accept your changes.
- Repeat the same steps for the Slab 30cm thickness and for the Slab 20cm thickness thin the sections will be add to the section list
- Click the **OK** button to accept the changes.



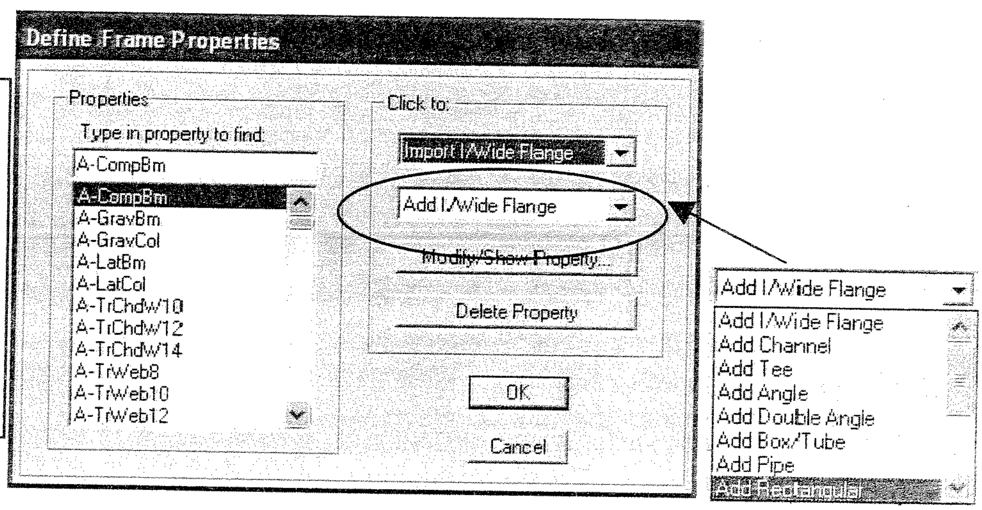
3. Define of the Beams

3. Click the **Define menu** ——► **Frame Sections ….**

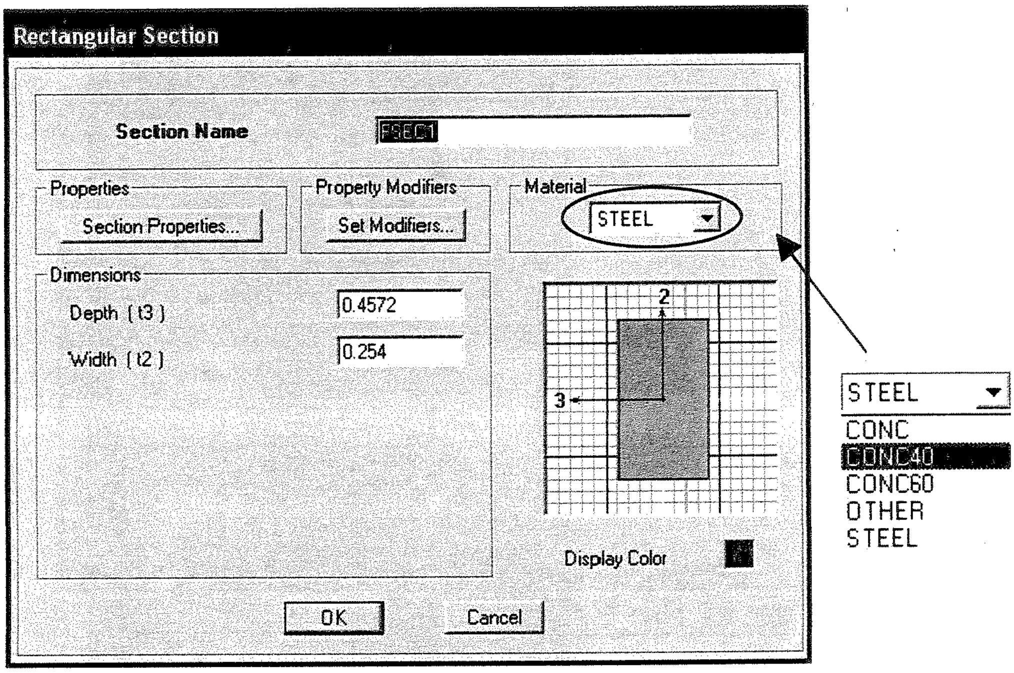


Or Define Frame Sections button which will Display the define Frame Sections form

You can remove the template section by click on the Delete Property

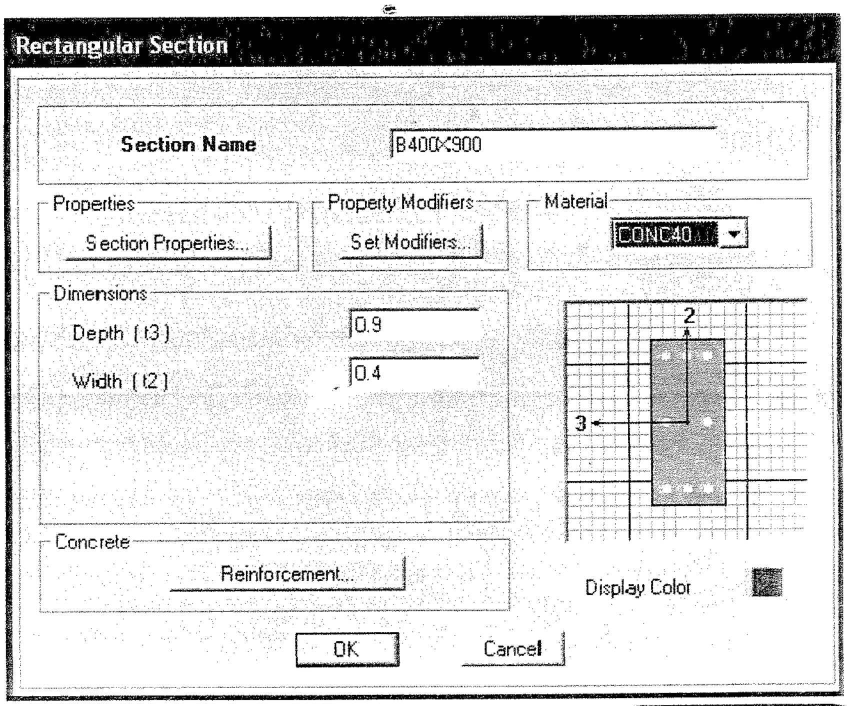


3. Click the Second drop-down box that reads Add I/Wide Flange ,Choose Add Rectangular from list thin the form of Beam define will be opened

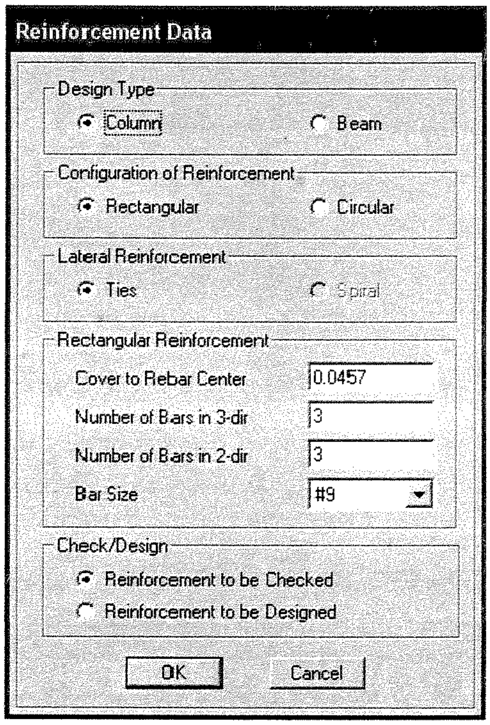


Note: we will define the Beam Dimension (0.4x0.9)

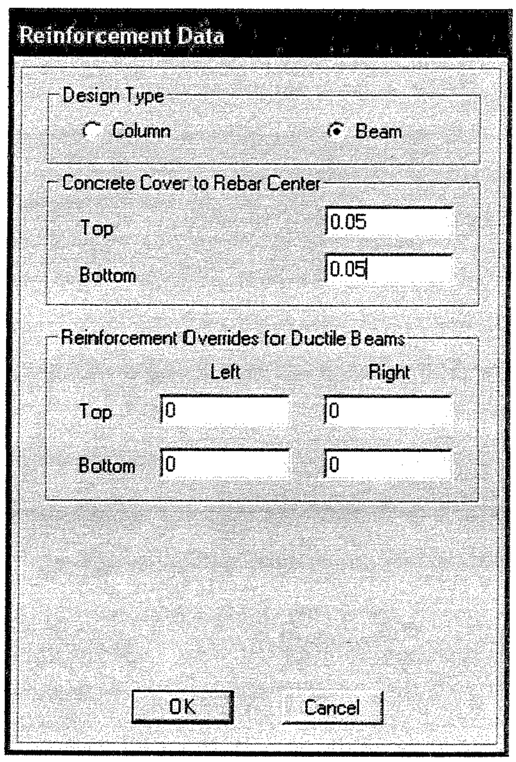
- In this from change the section name to be B40x90
- Click the drop-down box of material and choose conc. 40
- Define the Dimensions (0.4x0.9),thin the form will be as shown



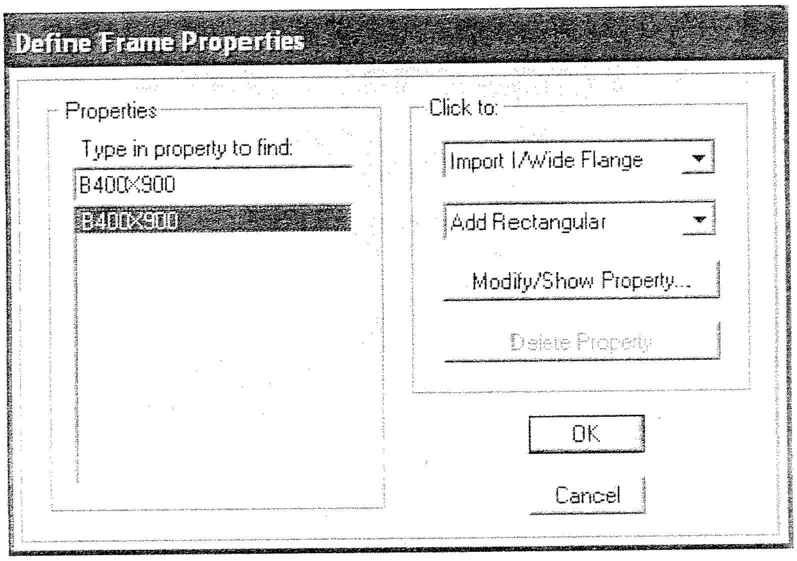
- Click the Reinforcement button to Define the Design of this Section



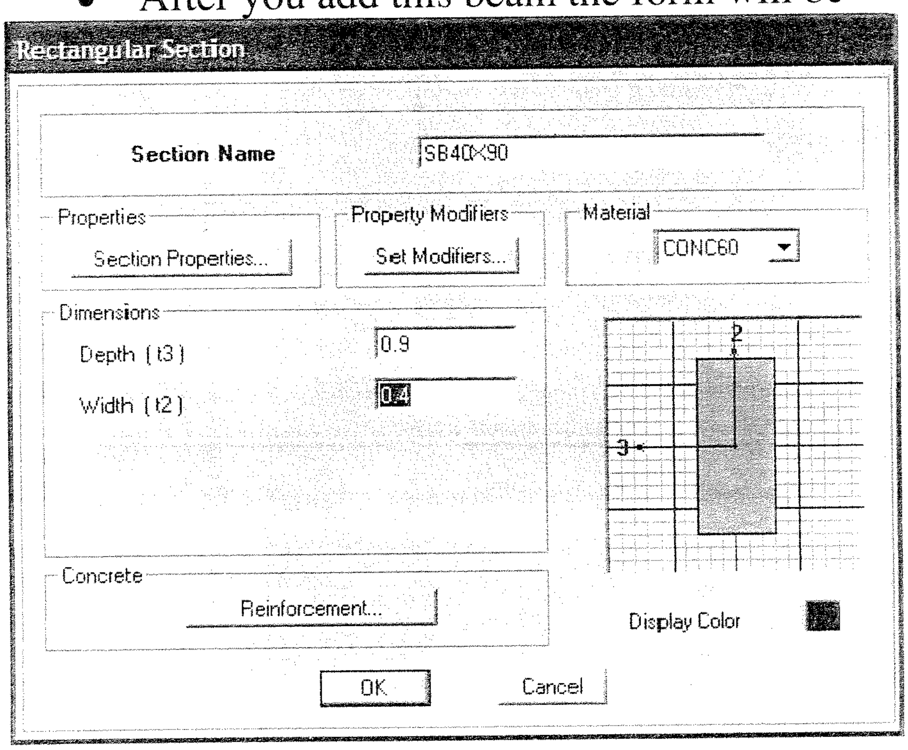
- In this form click Beam and define the cover for reinforcement to make the form like the next form



- Click the OK button to accept your changes. and Ok for main form ,you will find the Section already Defined



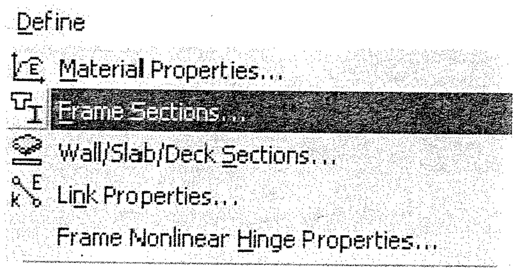
- Repeat the same steps to define the spandrel beam (the beam connecting the core walls because it will be casted with walls of the core so that the concrete grade of this beam will be conc. 60 and the reinforcement of this beam be different than the normal beam
- After you add this beam the form will be



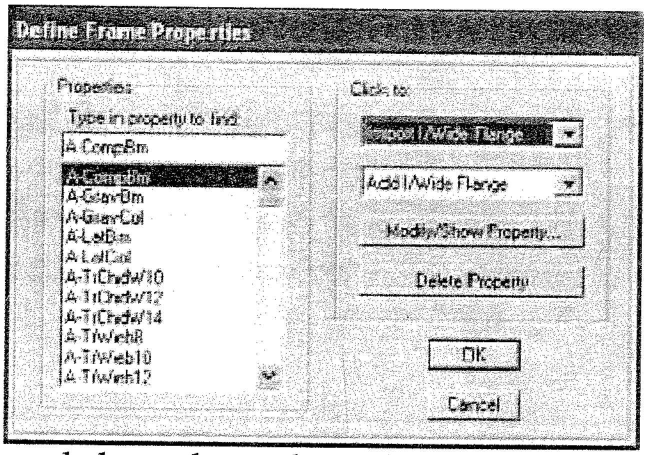
4. <u>Define of The columns</u>

To define the columns follow the same steps of defining the beams

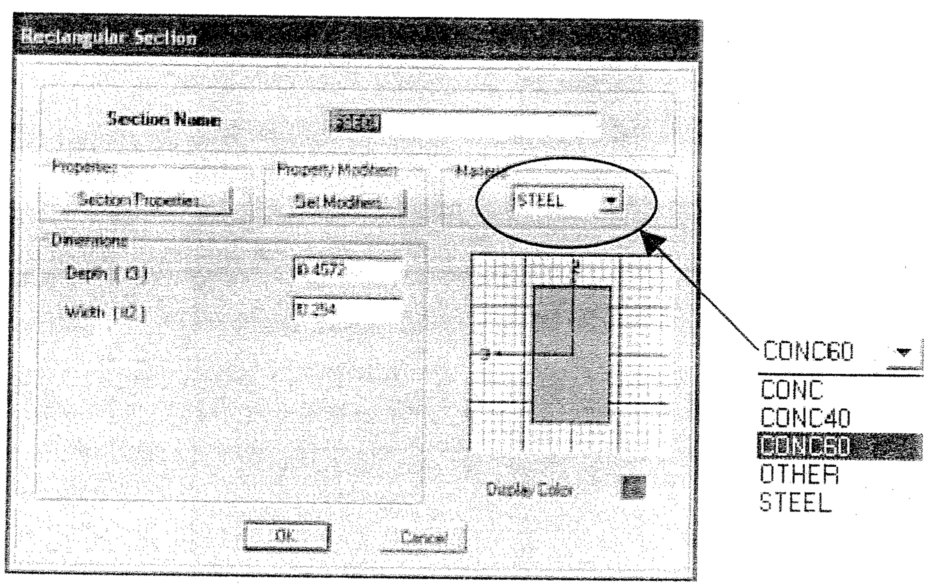
1. Click the **Define menu** ——▶ **Frame Sections ….**



Or Define Frame Sections button which will Display the define Frame Sections form

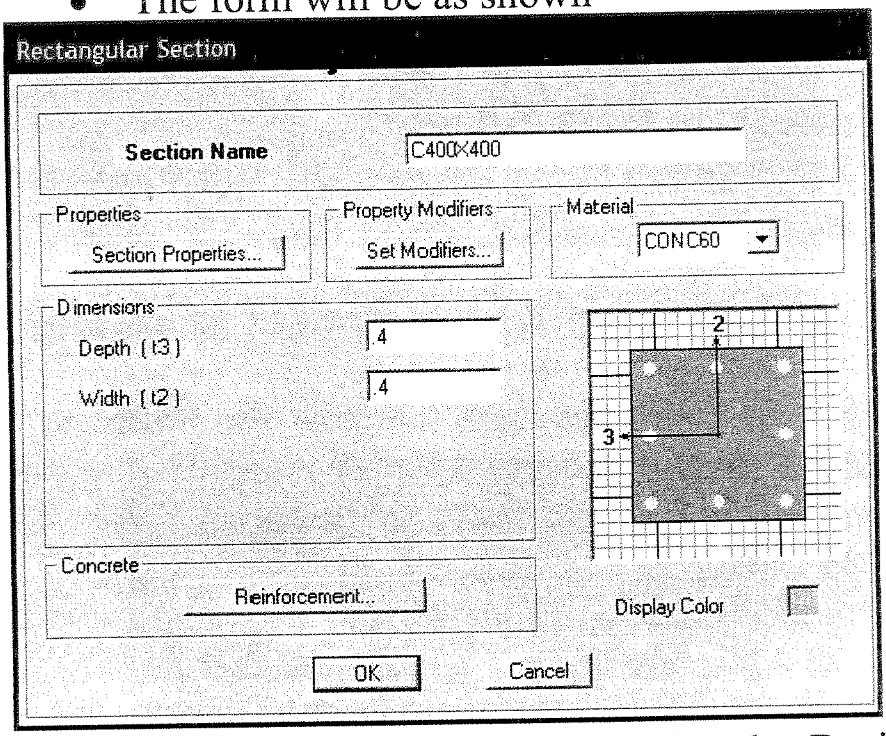


4. Click the Second drop-down box that reads "Add I/Wide Flange" ,Choose Add Rectangular from list thin the form of columns define will be opened

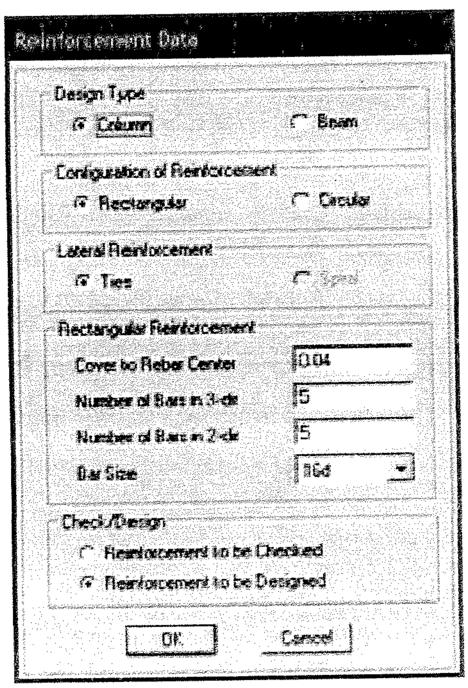


Note: we will define the columns Dimension (0.4x0.4)

- In this from change the section name to be C400x400
- Click the drop-down box of material and choose conc. 60
- Define the Dimensions (0.4x0.4)
- The form will be as shown



- Click the Reinforcement button to Define the Design of this Section



- In this form click column and define the cover for reinforcement, number of bars in each face, and bar size to make the form like the above form
- Click the OK button to accept your changes. and Ok for the main form
- **Step 3: Add the Geometry of the Tower**

In this step, we will draw the Structure elements

1. Add Slabs, Walls Beams:

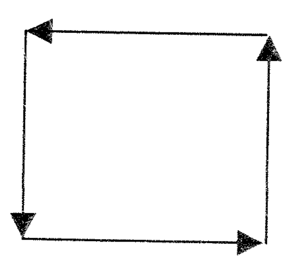
You have 2 methods to add the structure elements to the program

a. By using the program drawing tools
b. By using DXF file which is creating by AutoCAD Program

NOTE: I Will Explain How to Draw the Structure Element Using AutoCAD Program because most of engineers are familiar with this method. And I will take about the program utility in the chapter of important notes

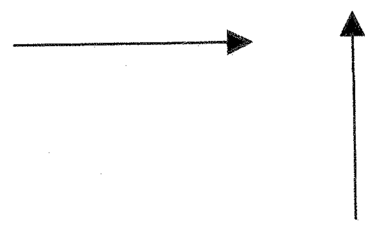
1. Open **AutoCAD** Program
2. Open the File of the plane for the first floor or open new file in AutoCAD Program and make 4 deferent layers
   i. Beams : for beams and walls
   ii. Floor :for slabs
   iii. Opening :for the openings
   iv. Columns : for the columns
3. Draw **PLINE** on the edge of the slab in the layer **Floor**, you dont need to draw 3D Face a nd mesh it yourself ,we will make the program make automatic mesh for the elements

Direction of Drawing
The PLINE of the slab

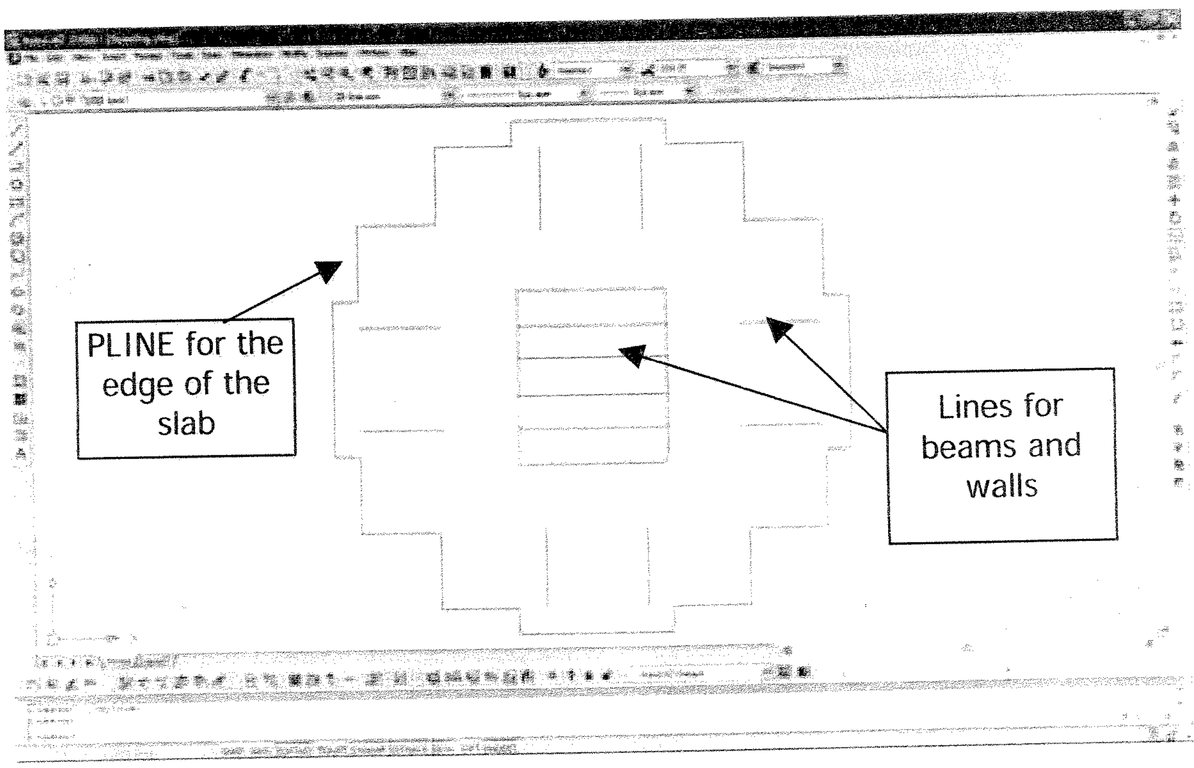


4. Draw the beams and walls as **lines** in the layer called **beams,** and if you have columns draw it as a line in the 3D

Direction of Drawing
The beams and the
Walls lines



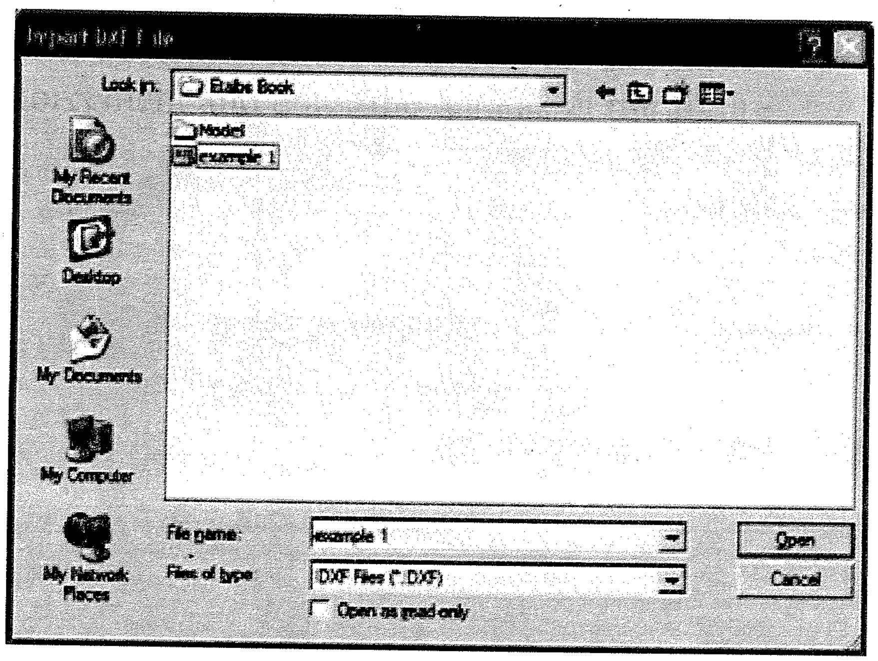
5. Draw the **openings** in the layer called opening as **PLINE** on the edges of the openings



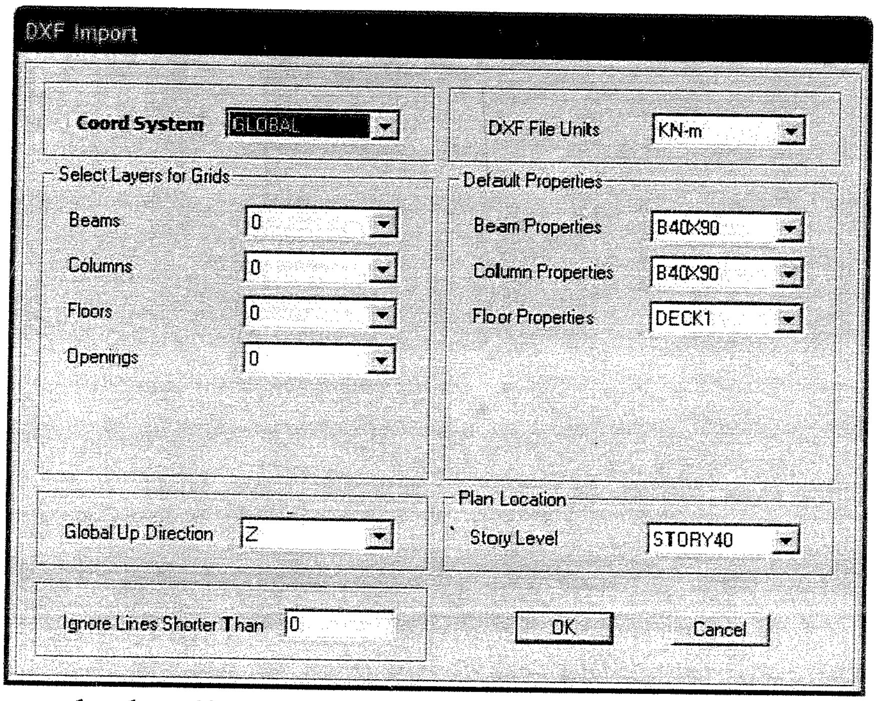
6. save the file as DXF file named Example1

7. Click **File menu** ⟶ **Import** ⟶ **DXF floor plan ,** which will Display the Import DXF File form

NOTE: Before Exporting the DXF to ETABS Program , make sure that the referance point of (0,0) is the Same as the model referans point (0.0), if not move all the Drawing from the reference point (intersecting the grid (A,1)) to (0,0)

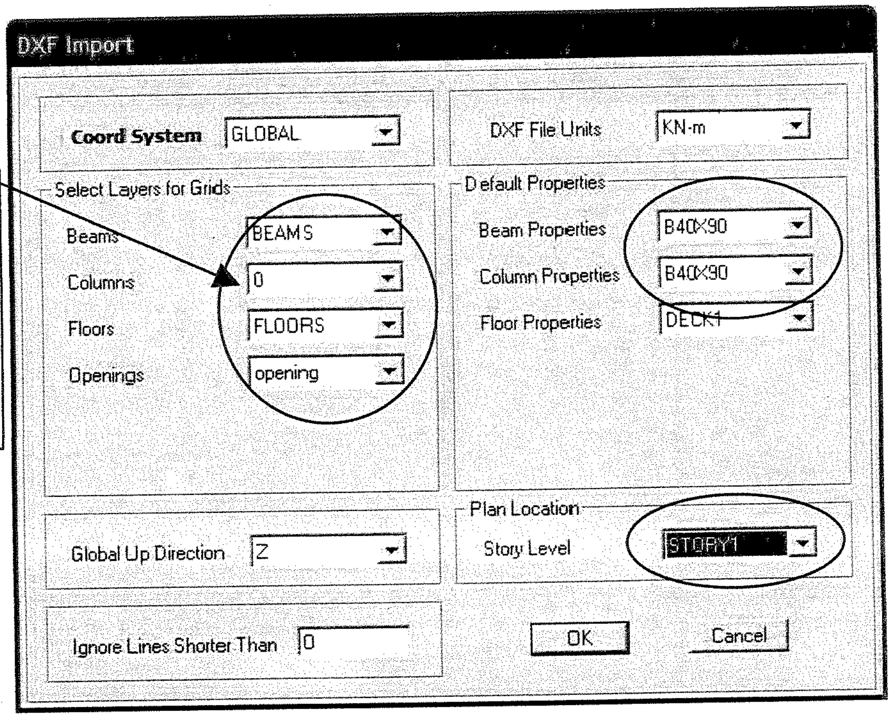


Choose the file of DXF and Click **Open;** thin DXF Import Form will be displayed

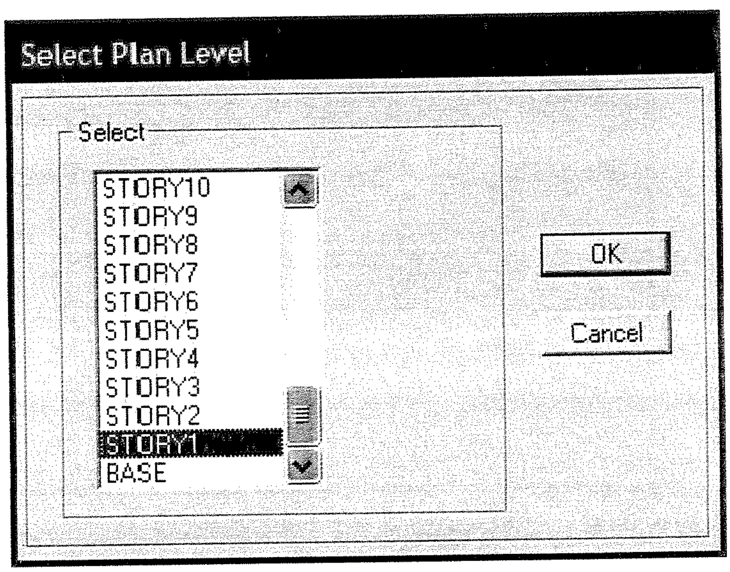


Adjust the marked pull down box as in the next fig.

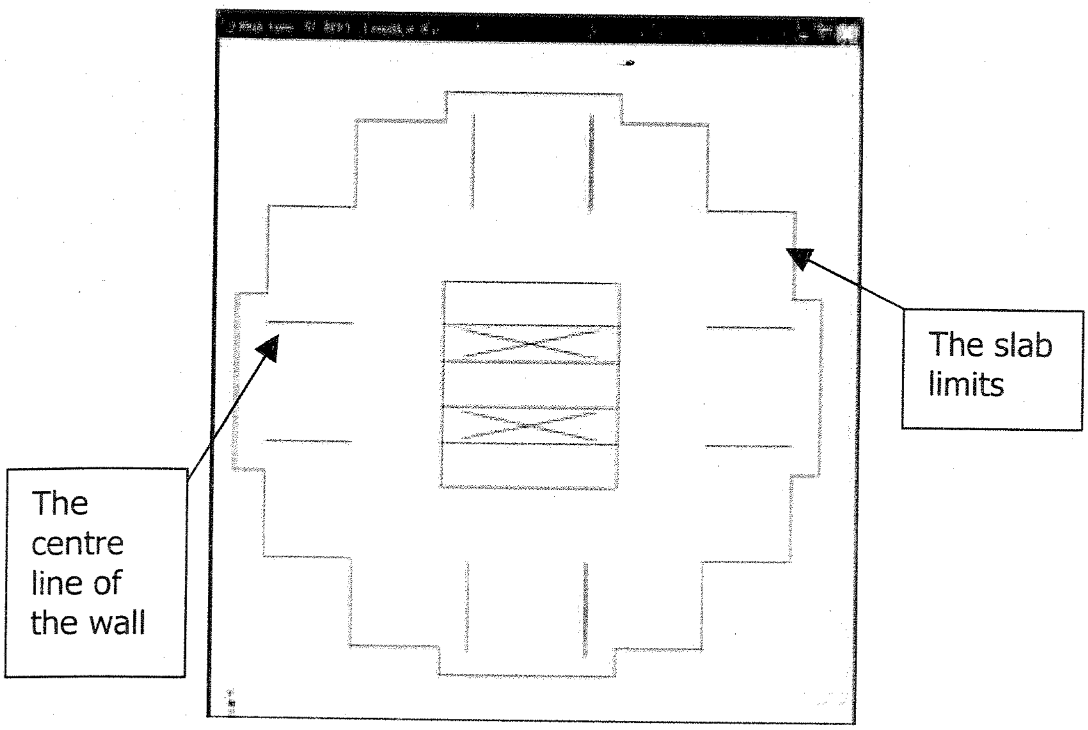
We don't have columns in this floor



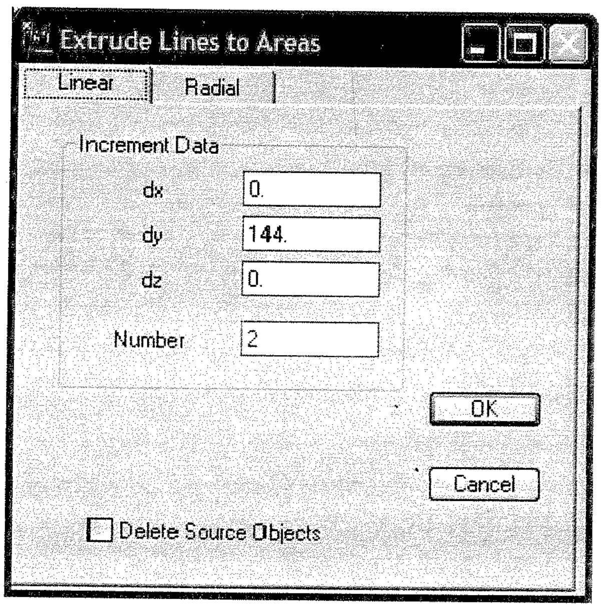
8. Make sure that the plan view is the view of story 1,if not Click set plan view button thin the next form will be displayed, from this form choose story 1,thin click **OK**



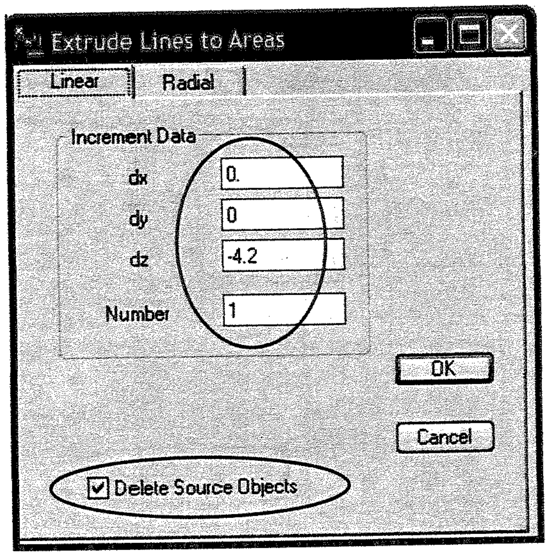
Thin the floor will be displayed in the plan as follow

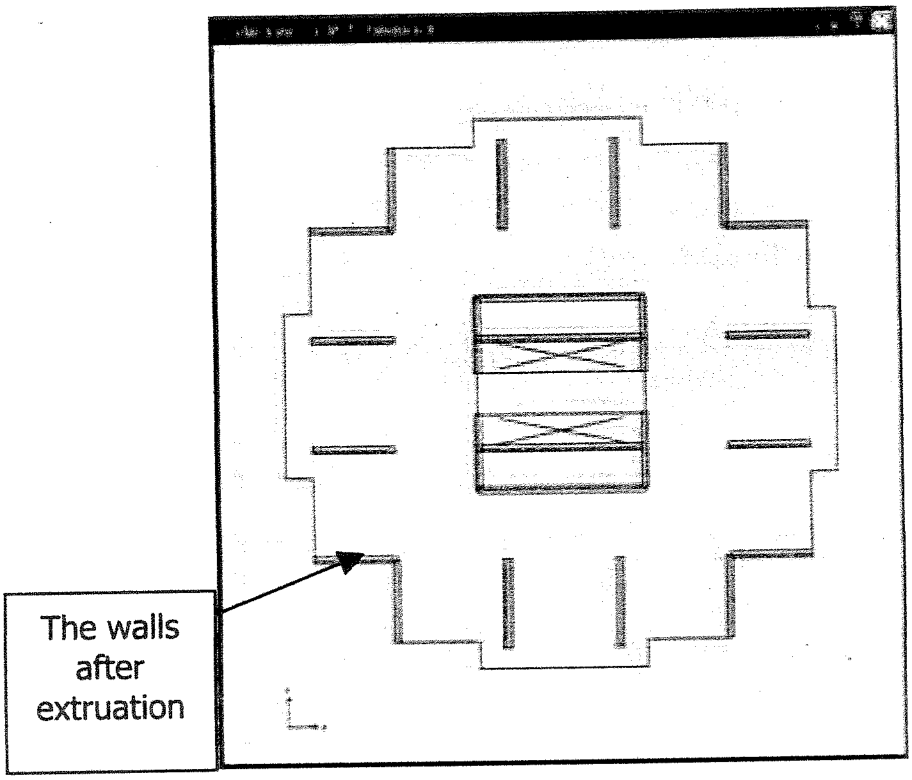


9. Choose the wall centre line thin click Edit menu ⟶ . Extrude Lines to Areas thin the next form will be displayed



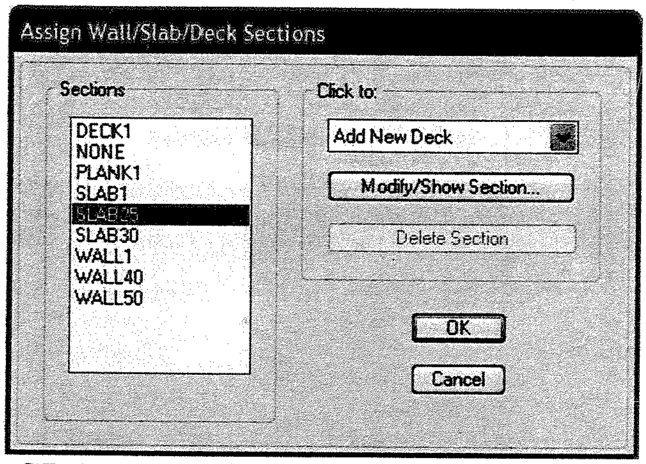
From this form change the dz equal to the height of the floor by (-) sign and highlight the Delete Source Objects check box and the number of divide the walls make 1 thin click **OK**



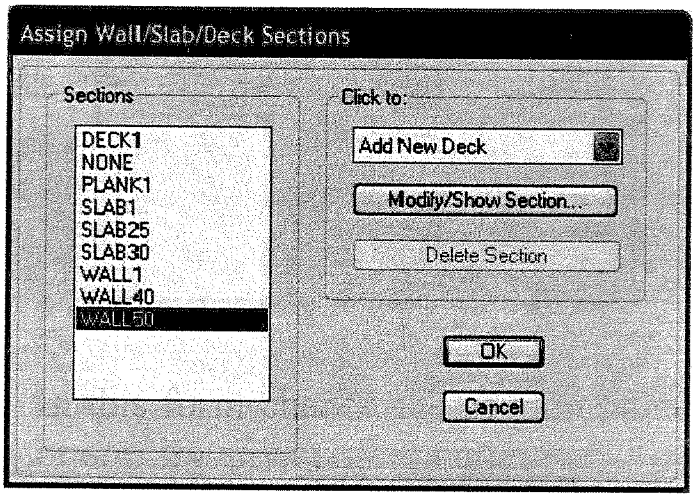


10. Assign of the sections

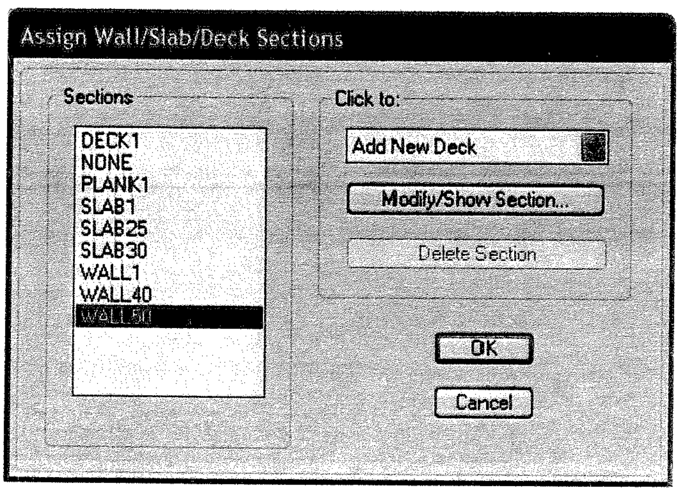
- Choose the slab and click **Assign menu** ⟶ **Shell/Area** ⟶ **Wall/Slab/Deck Sections**



- choose SLAB 25 thin click **OK button**
- **to assign the walls thickness**
  - Choose the outer Walls and click **Assign menu** → **Shell/Area** → **Wall/Slab/Deck Sections**

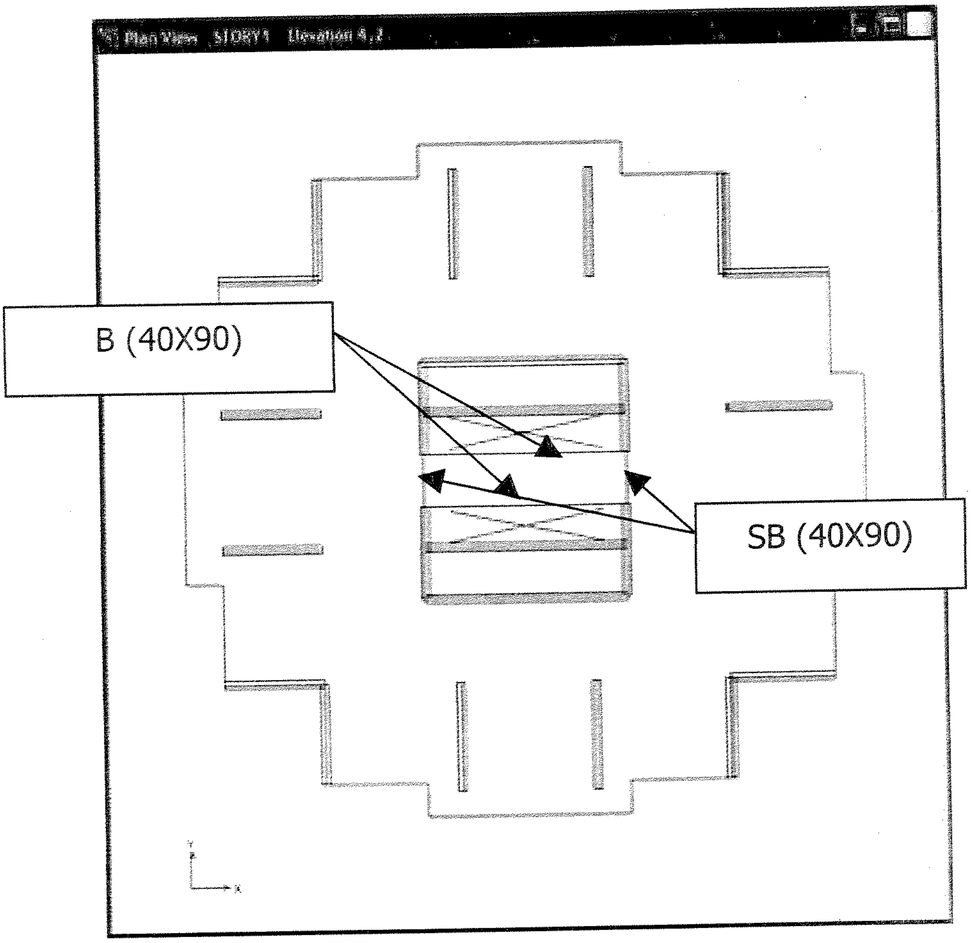


  - Choose Wall 50 thin click **OK**
  - Choose the Core Walls and click **Assign menu** → **Shell/Area** → **Wall/Slab/Deck Sections**

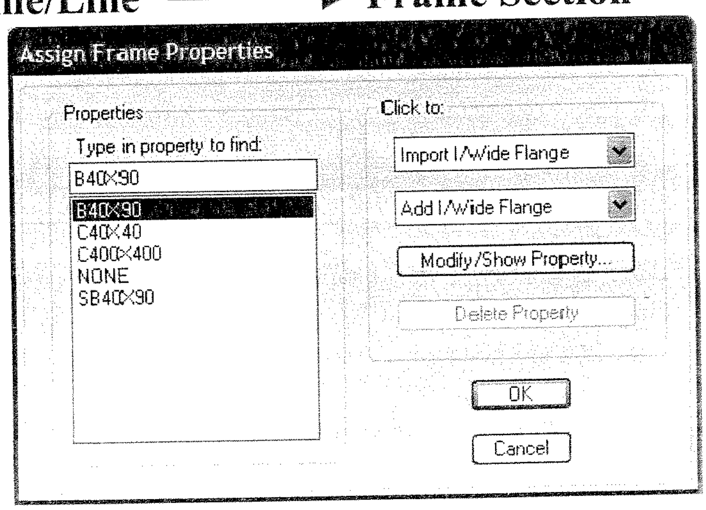


- Choose Wall 40 thin click **OK**

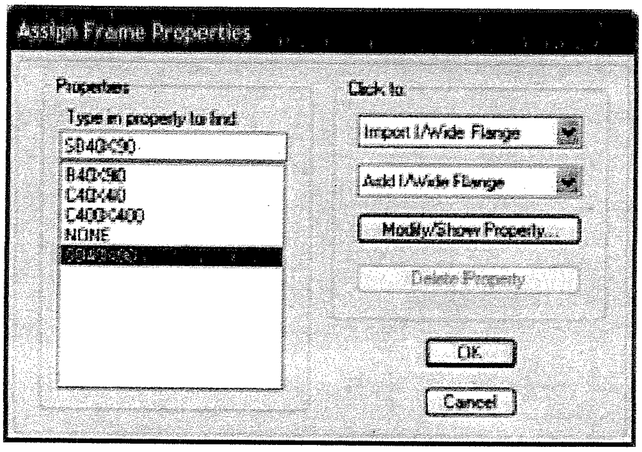
- **To assign the Beams Dimensions**



- Choose the Long beams in the core and click **Assign menu** → **Frame/Line** → **Frame Section**



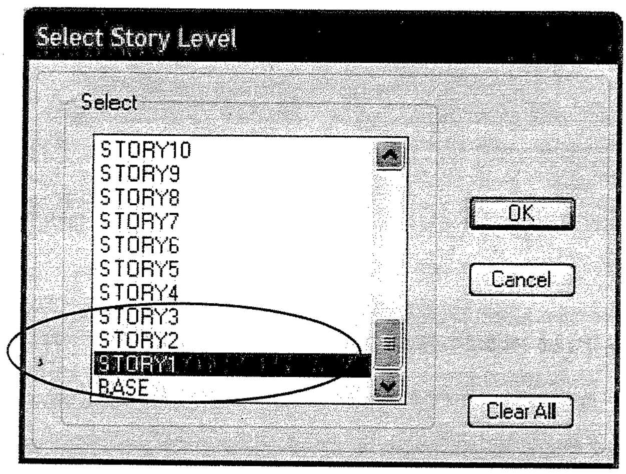
- Choose B 40X90 thin click **OK**
- Choose the short beams in the core and click **Assign menu** ⟶ **Frame/Line** ⟶ **Frame Section**



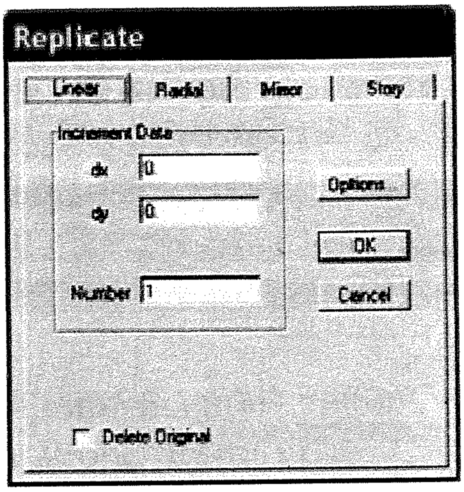
- Choose SB 40X90 thin click **OK**

11. Replicate of the story

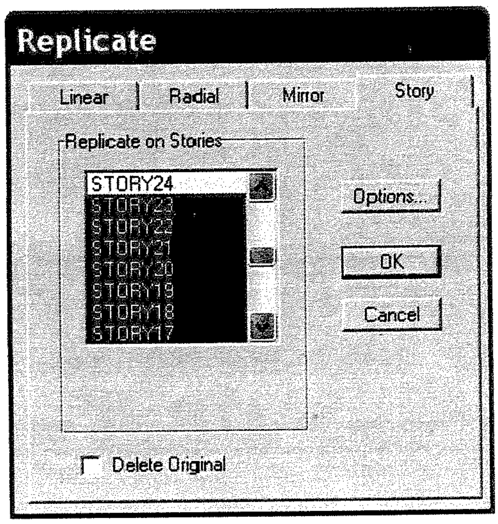
- Choose this floor and make replicate to other floors. Choose the floor by window or click **Select menu** ⟶ **select by story level thin** choose the story from the form thin click **OK**



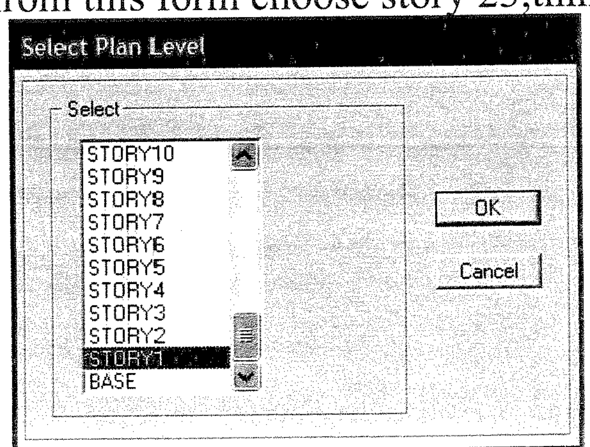
- Thin click **Edit** ⟶ **replicate**. the next form will be displayed



Choose story and the story you want to replicate this floor in it (from story 1 to story 23), thin click **OK**

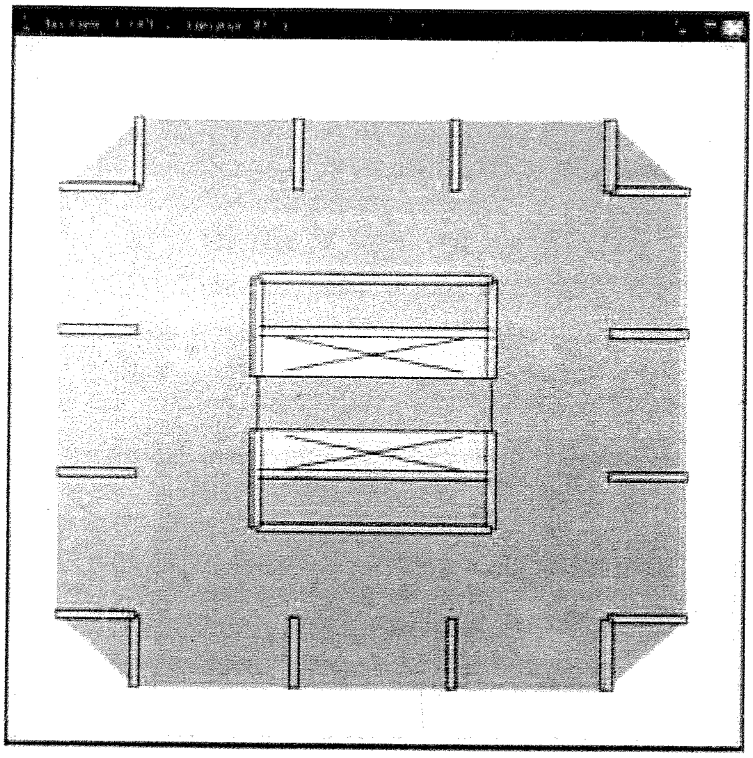


12. As it is mentioned in the begin of the example the story 23 is mechanical floor so that it is needed to reassign the slab thickness again by choose the plane of the floor in plane view , Click set plan view button thin the next form will be displayed, from this form choose story 23,thin click **OK**

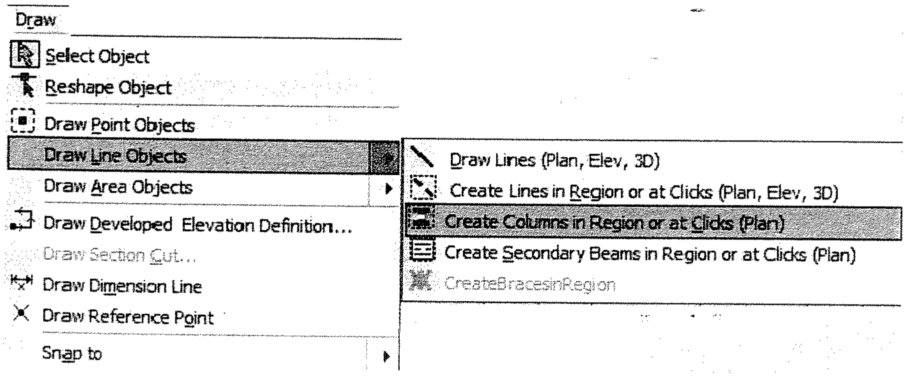


- Choose the slab by click on the slab in plan; thin repeat the same steps for assign the slab and assign the slab to be slab 30.

13. Draw other floors as we explained in the previous steps as DXF files and repeat the same steps for import and replicate.



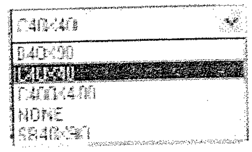
- In the story 39 which have columns ,you have 2 choice to add this floor
    1. replicate the floor of the story 38 to story39 thin erase the edge walls and click **Draw menu** ——→ **Draw Line Objects** ——→**Create Column in Region or at Click (plan)**



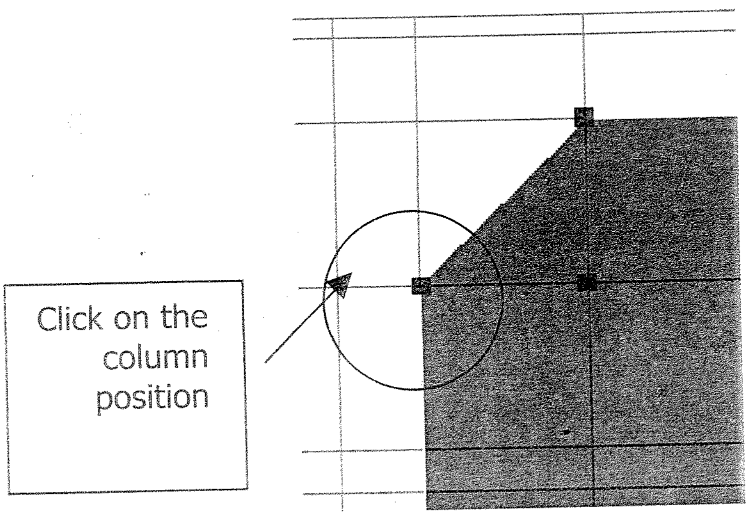
Or click **Create Column in Region or at Click (plan)** button, thin the carser of drawing will be displayed and also the form of the properties of the column, from the drop down box of Property choose the column section C40x40

Properties of Object

Property	B40×90
Moment Releases	Continuous
Angle	0.
Plan Offset X	0.
Plan Offset Y	0.

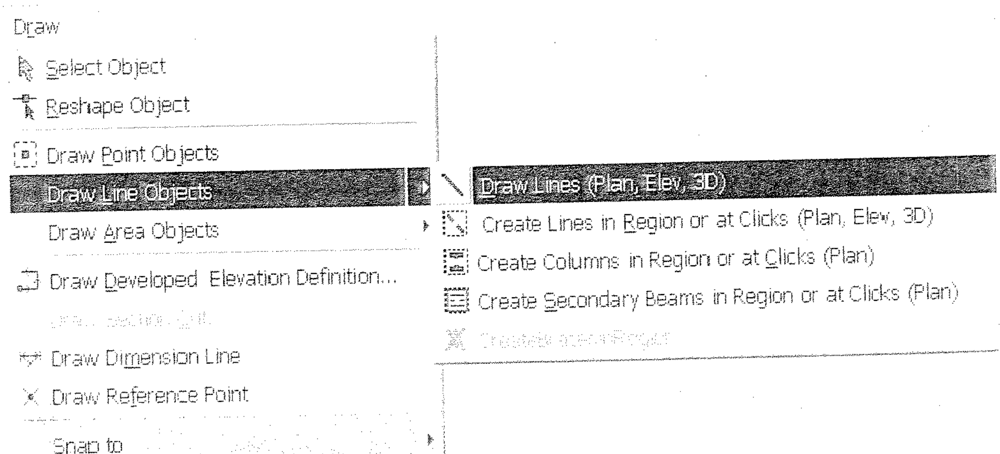


Thin click on the position of the column



- **Add the Beams**

  1. Click the **Draw menu** ⟶ **Draw Line Objects** ⟶ **Draw Lines**



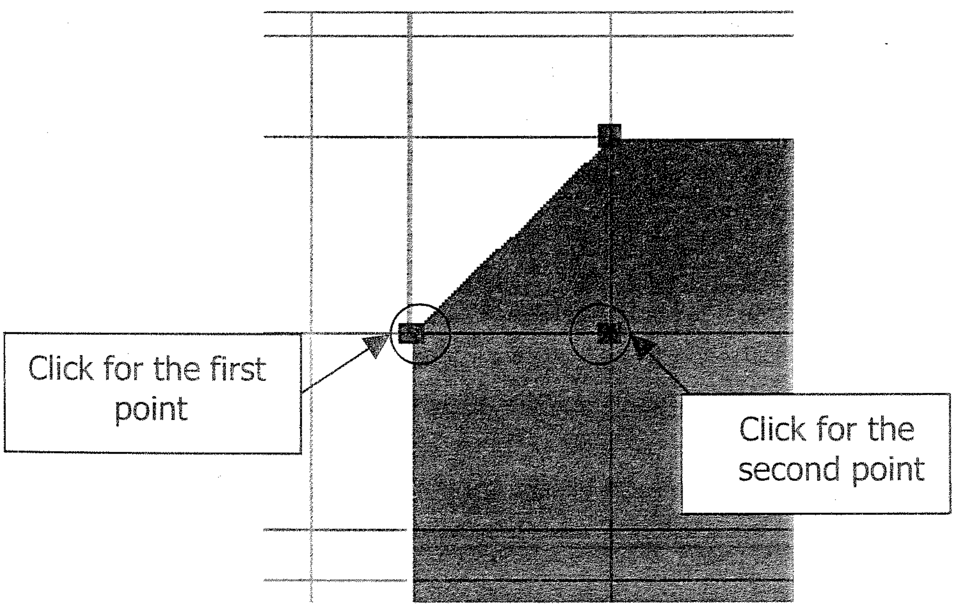
Or Draw Area button which will Display the Properties of Object form as shown in fig. and crasser for drawing Area

Properties of Object

Type of Line	Frame
Property	A-LatBm
Moment Releases	Continuous
Plan Offset Normal	0.
Drawing Control Type	None <space bar>

- In previous form you can
    1. Choose Type of Line (Frame)
    2. Choose the Property of the Line (B40x90)

Thin click on the first point of the beam thin click on the second point thin you will find the beam created



Second method for the add the columns to draw the columns in the DXF file in 3D thin imported to the Tabs Program as explained before **For the story40 replicate the core walls and draw slab on the edge of the core or repeat the same step or create DXF thin imported to the Etabs program..**

Step 4: Meshing &assign id the Diaphragm

Meshing

There are 2 types of meshing

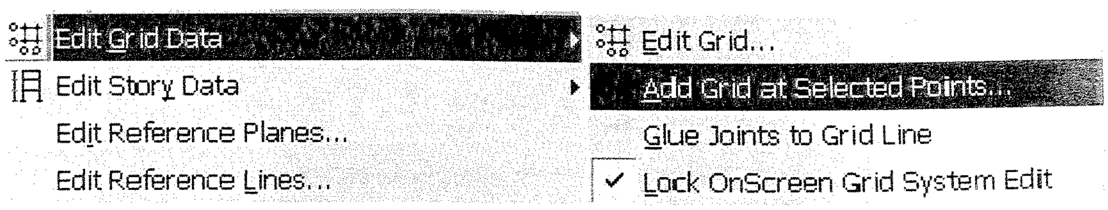
1. manual meshing
2. Automatic meshing

   - for walls, it is better to make manual meshing
   - for slabs, it is better to make automatic meshing

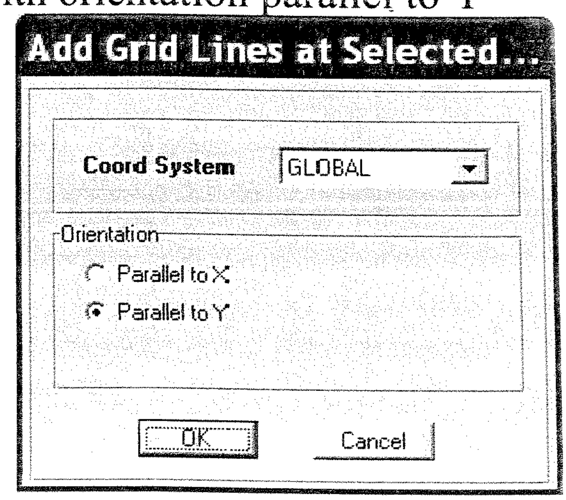
Note: the important thing you must do it before the meshing to create secondary Grids for the structure to be as a guide for the meshing

To create the secondary grids

- Select the whole structure or you can select only the point you want to it to be guide for the meshing
- Click the **Edit menu** ⟶ **Edit Grid Data** ⟶ **Add Grid at Selected Points**

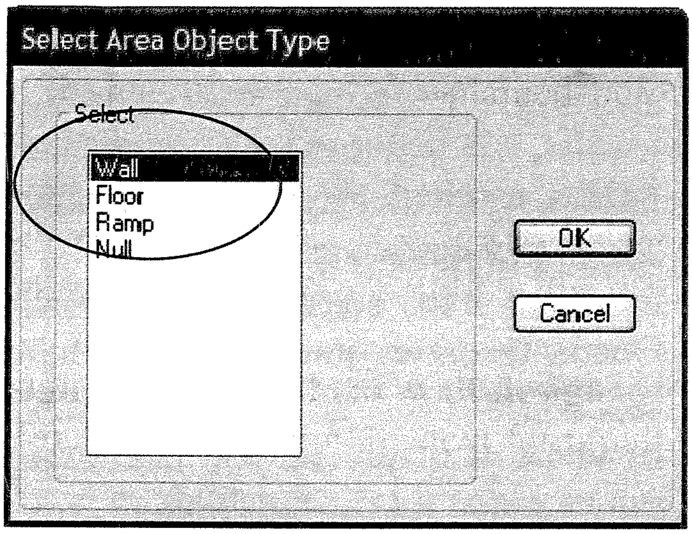


- Thin from next form choose Orientation Parallel to X, thin repeat the same step with orientation parallel to Y



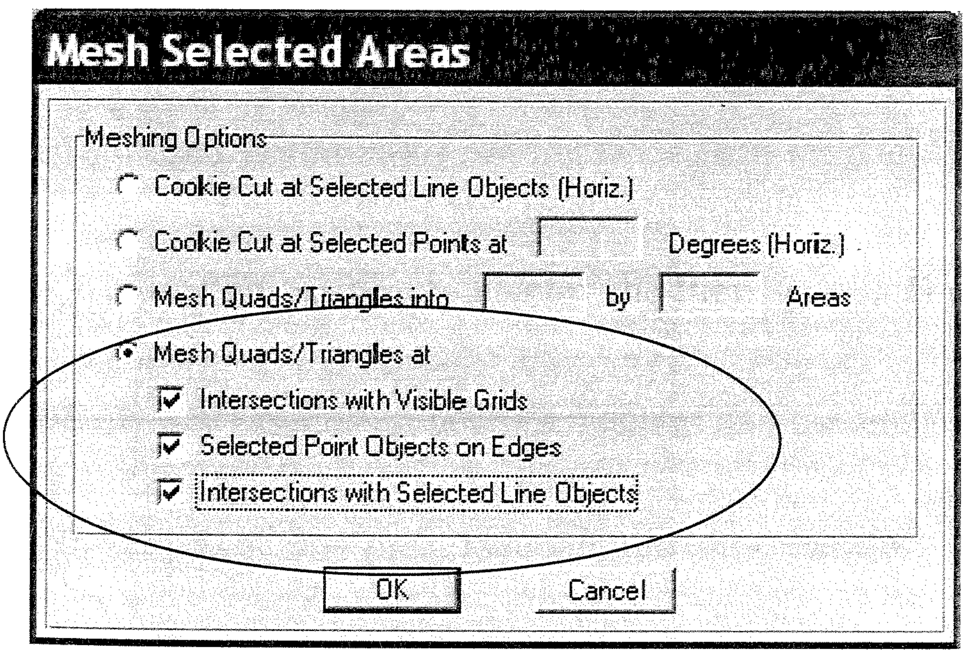
1. Meshing of the walls

- Choose the whole walls ,click **select** ⟶ **by area object type**, thin the next form will be displayed



- Choose **wall** thin click **OK**

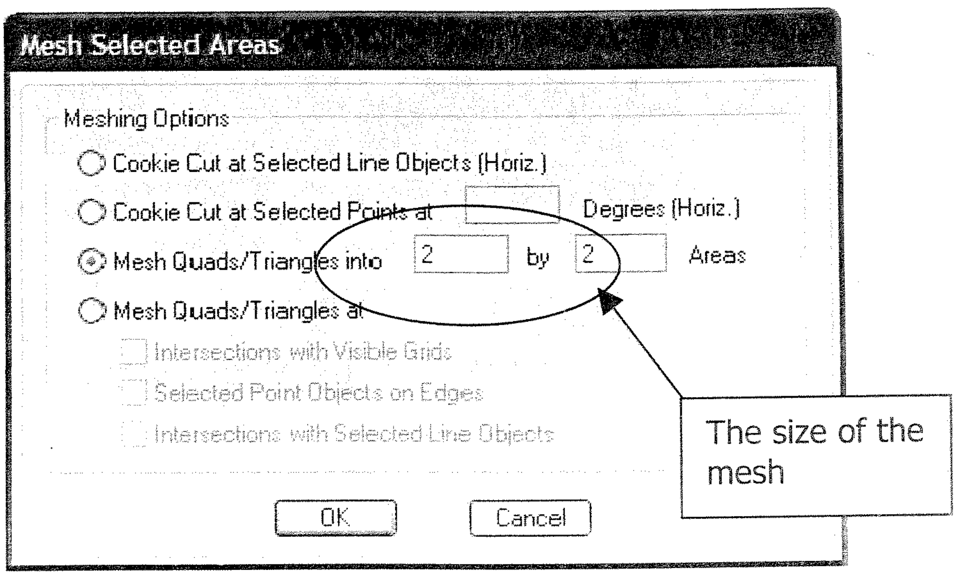
- Click the **Edit menu** ⟶ **Mesh Area** the next form will be displayed



- From this form choose Mesh Quads/triangles at
  - Intersection with Visible Grids
  - Selected Point Objects on Edges
  - Intersection with Selected Line Object

  - Thin click OK and display any plan to see the meshing of the walls

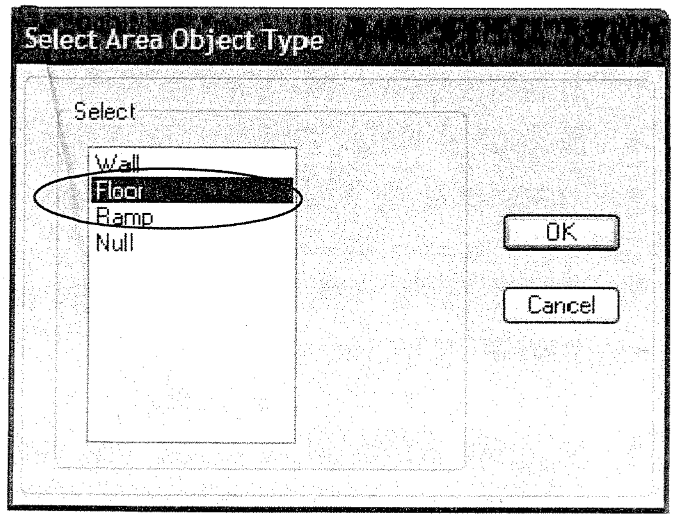
  - If you want to add farther subdivide to the walls select the walls again and display the form of the meshing thin select the third choice of Mesh Quads/Triangles into, thin give to the program the size of the mesh which you want to mesh the walls by this size



2. Meshing of the Slabs

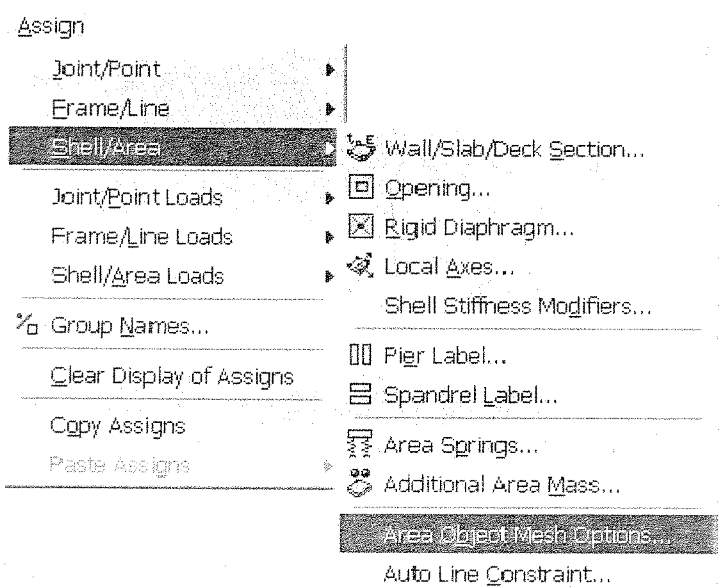
Note: for slabs we will make automatic meshing

- Choose the whole Slabs walls ,click **select** ——► **by area object type**, thin the next form will be displayed

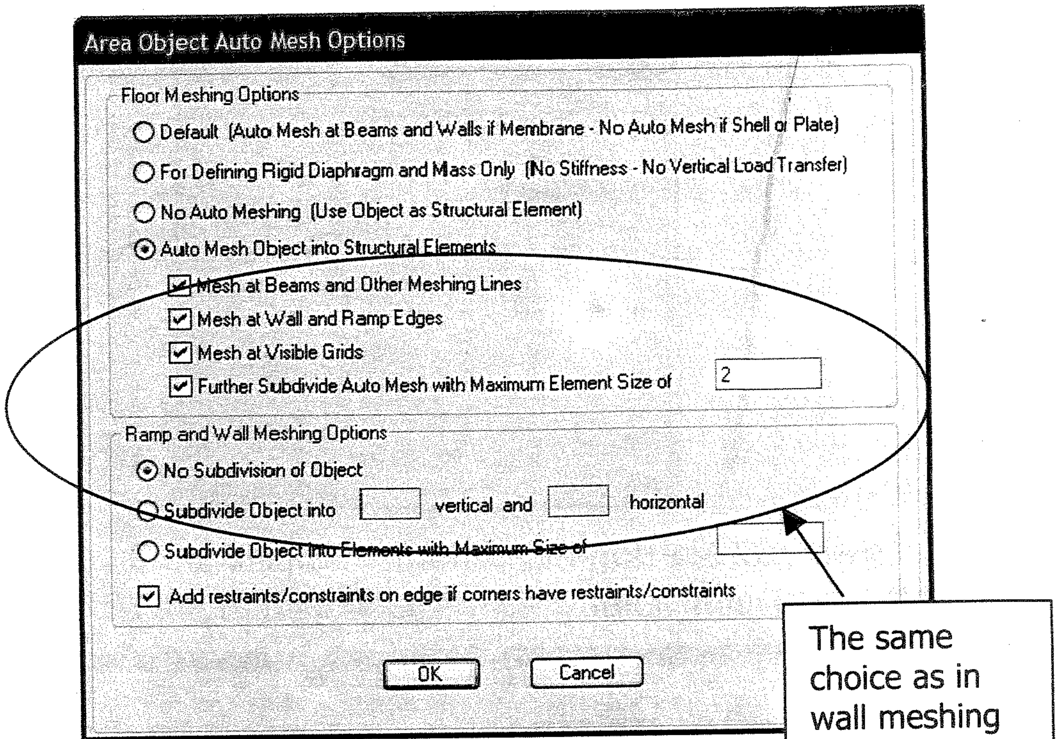


- Choose **Floor** thin click **OK**

- Click the **Assign menu** ——► **Shell/Area** ——► **Area Object Mesh Option**



The next form of **Area Object Auto Mesh Option** will display Choose the marked option for mesh thin click **OK**



Assign of the diaphragm:

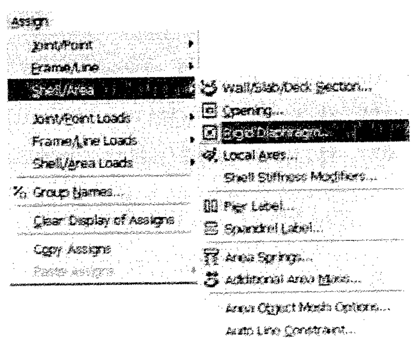
We have 2 types of Diaphragm

i. Rigid Diaphragm
ii. Semi Rigid diaphragm

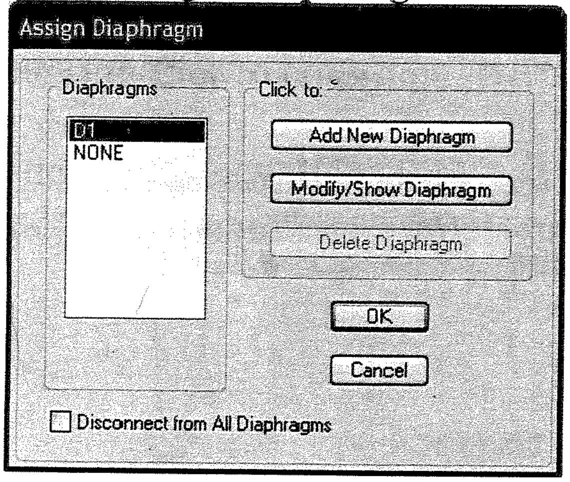
Rigid Diaphragm Constraint causes all of its constrained joints to move together as planar diaphragm that is rigid against membrane deformation. And used for define the wind load

Semi Diaphragm used mainly for defines the wind load and give the building the ability to behave as its actual behavior (Recommended) (not available in Etabs Ver. 8)

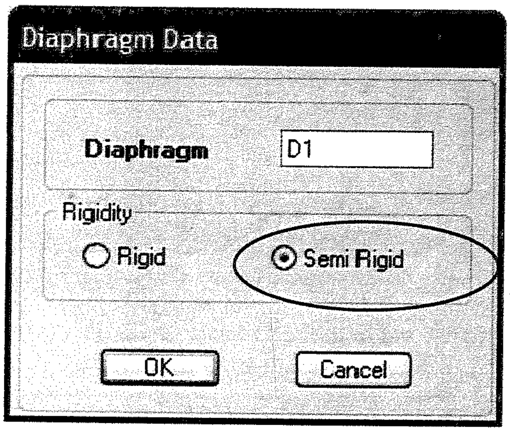
- Choose the whole building by click in all button
- Click the **Assign menu** ⟶ **Shell/Area** ⟶ **Diaphragms…**



- The next form of **Assign Diaphragm** Will be displayed



- Choose D1 thin Click **Modify /show Diaphragm,** highlight Semi Rigid thin click **OK**



- Thin click **OK** for the main form

Note: you don't need to assign each floor by different Diaphragm name because the program automatically assigns the Diaphragm with reference to the floor name and the choice of semi Digphram will make the floor beheave with its actual beheaver

- **Step 5: Define Static Load Cases and assign the loads**

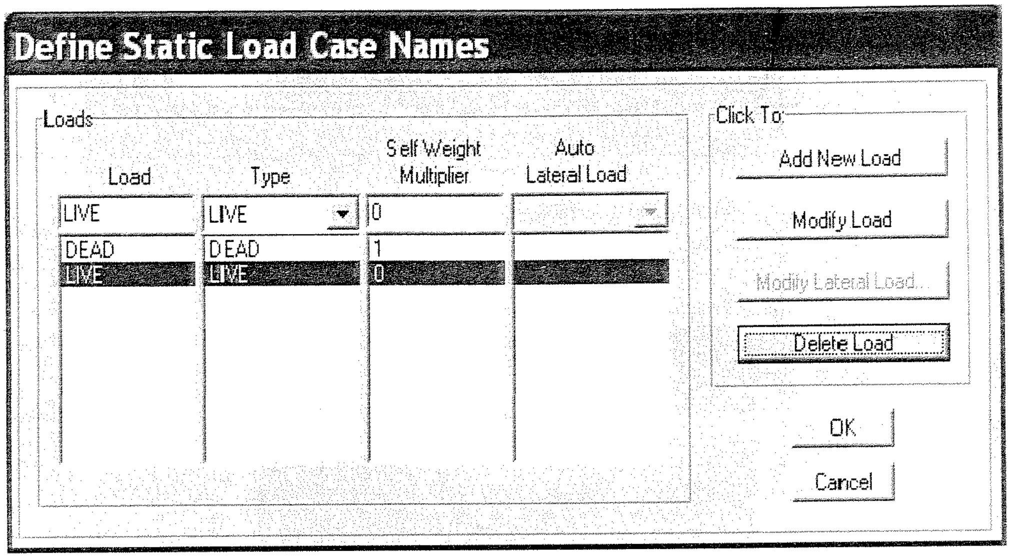
The static loads acting on the building used in this example consist of

- Dead Loads
- Live Loads
- Earthquake Loads
- Wind Loads

- **Define of Earthquake Load**

1. Click the **Define menu** ——► **Define Static Load Cases**

   Or Define Static Load Cases button which will Display **Define Static Load Case Names** Form

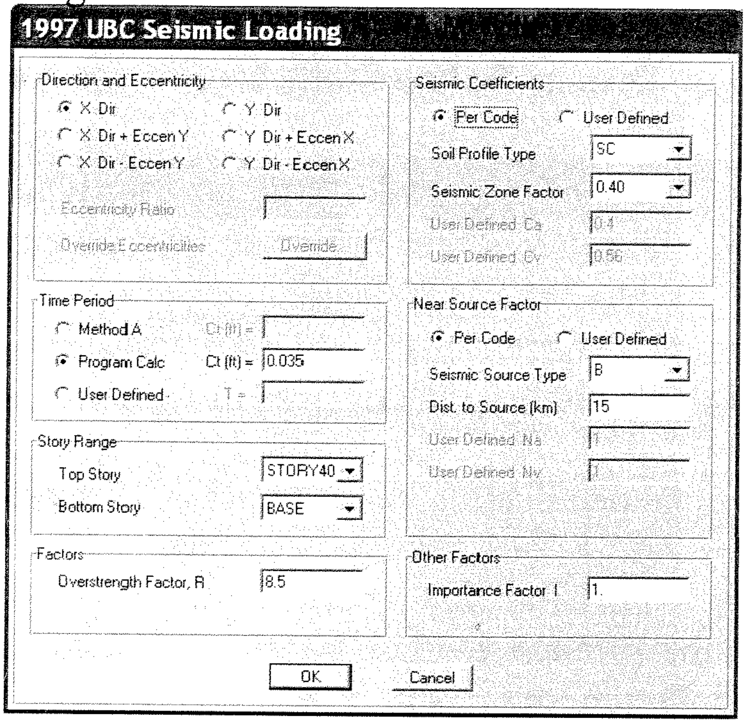


Note: the self weight multiplier is set to 1 for DEAD case only. This indicates that this load case will automatically include 1.0 times the self weight of all members.

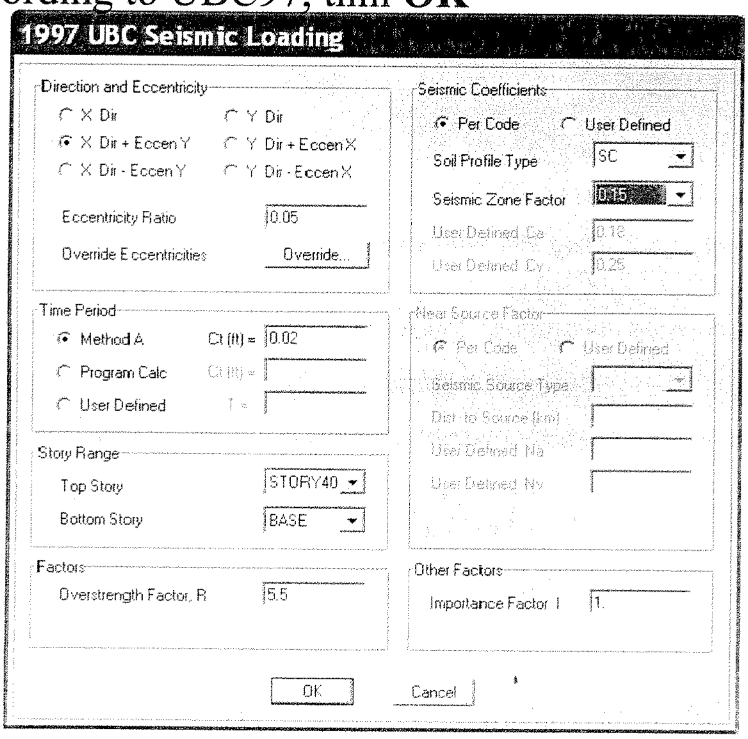
2. to define the earthquake case of loading
   - Click in the edit box for the **Load** column.
   - Type the name of the new load, in this case type **EQx**.
   - Select from the **Type** pull down menu, in this case, select quack.
   - Make sure the self **weight Multiplier** is zero,
   - Use the **Auto lateral load** pull down menu to select **UBC97**, with this option selected, ETABS will automatically apply static earthquake load based on the 1997 UBC code requirement.

- Click **Add New Load** button

3. with the EQx load highlighted, click the Modify Lateral Load button ,this will bring up the 1997 UBC Seismic Loading form



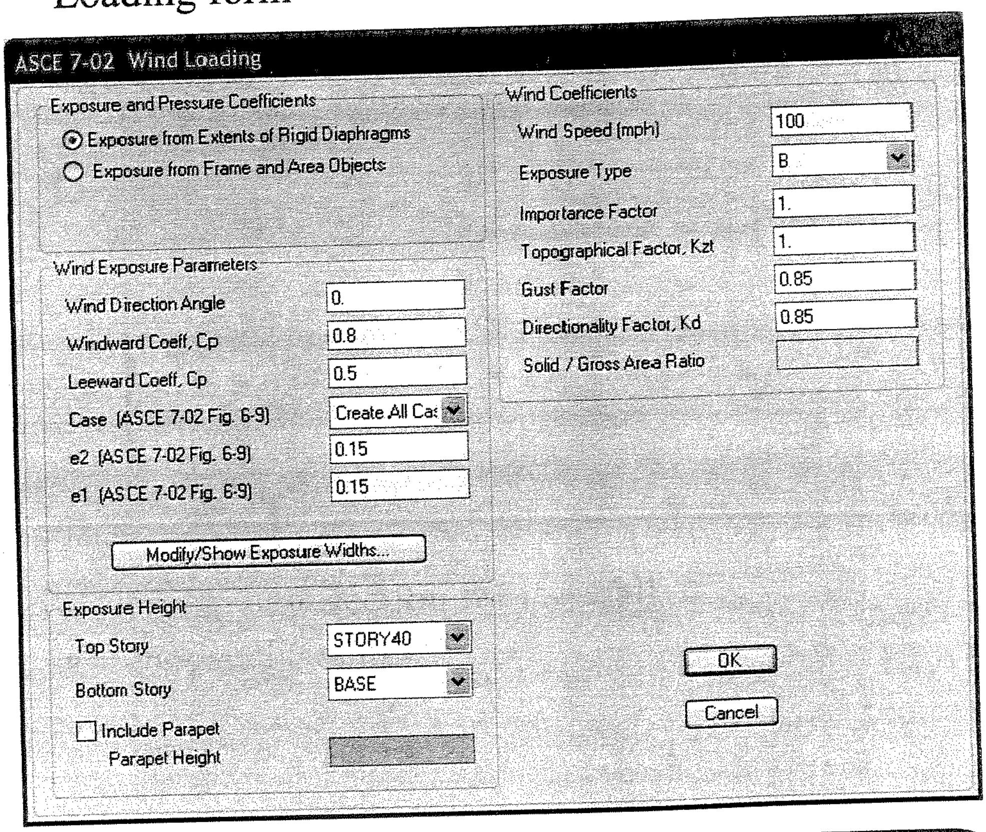
4. from this form you can adjust the data of the code for example you can change it as in the next form in zone 2A according to UBC97, thin **OK**



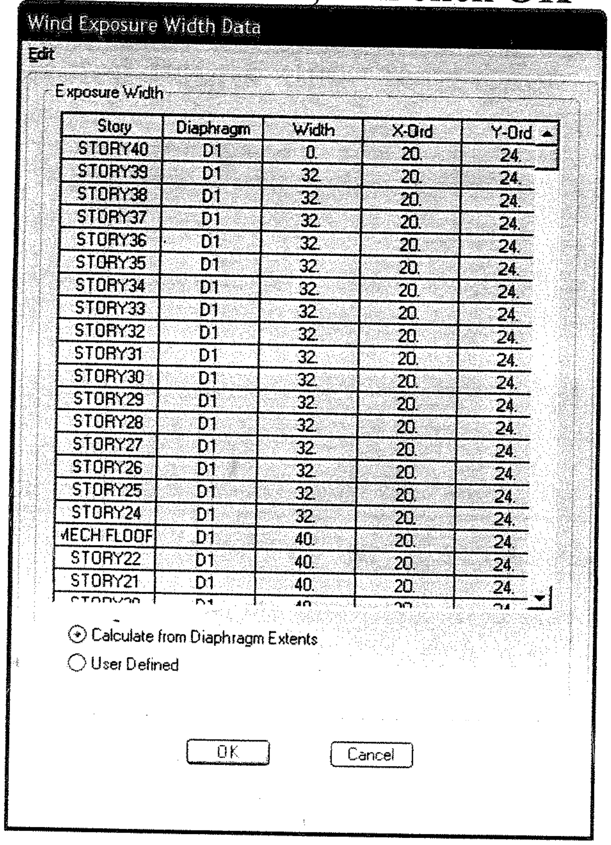
5. for earthquake in the next direction repeat the same steps but in 1997 UBC Seismic Loading form modify the direction to Y direction + Eccen X

- **Define of the Wind Load**

1. To define the earthquake case of loading
   - Click in the edit box for the **Load** column. Type the name of the new load, in this case type WINDX.
   - Select from the **Type** pull down menu, in this case, select **WIND**.
   - Make sure the self weight Multiplier is zero,
   - Use the **lateral load** pull down menu to select **ASCE 7-02** with this option selected, ETABS will automatically apply static wind load based on the **ASCE 7-02** code requirement.
   - Click Add New Load button
2. with the WINDX load highlighted, click the Modify Lateral Load button ,this will bring up the 1997 UBC Seismic Loading form

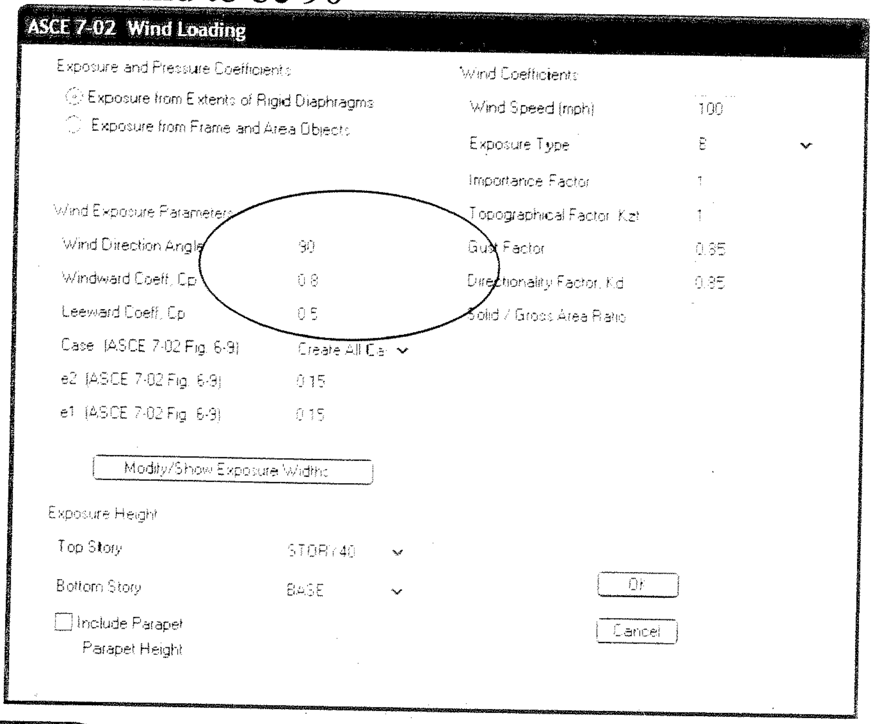


3. from this form you can adjust the data of the code according to your area ,to show the diaphragm Data Click **Modify /Show Exposure Width**, thin click **OK**

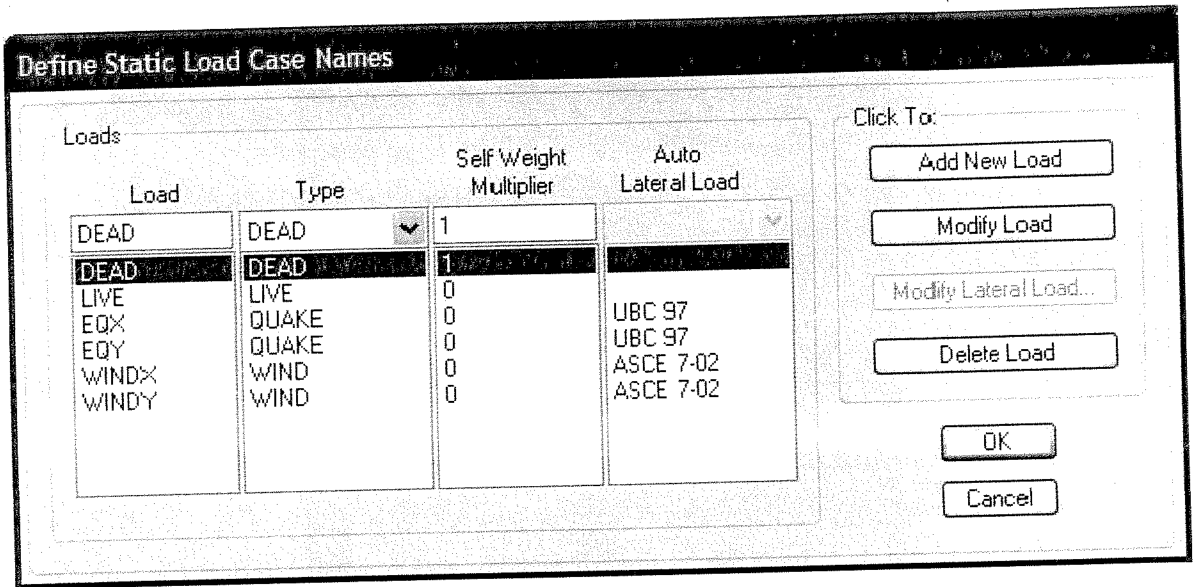


Story	Diaphragm	Width	X-Ord	Y-Ord
STORY40	D1	0.	20.	24.
STORY39	D1	32.	20.	24.
STORY38	D1	32.	20.	24.
STORY37	D1	32.	20.	24.
STORY36	D1	32.	20.	24.
STORY35	D1	32.	20.	24.
STORY34	D1	32.	20.	24.
STORY33	D1	32.	20.	24.
STORY32	D1	32.	20.	24.
STORY31	D1	32.	20.	24.
STORY30	D1	32.	20.	24.
STORY29	D1	32.	20.	24.
STORY28	D1	32.	20.	24.
STORY27	D1	32.	20.	24.
STORY26	D1	32.	20.	24.
STORY25	D1	32.	20.	24.
STORY24	D1	32.	20.	24.
MECH FLOOR	D1	40.	20.	24.
STORY22	D1	40.	20.	24.
STORY21	D1	40.	20.	24.

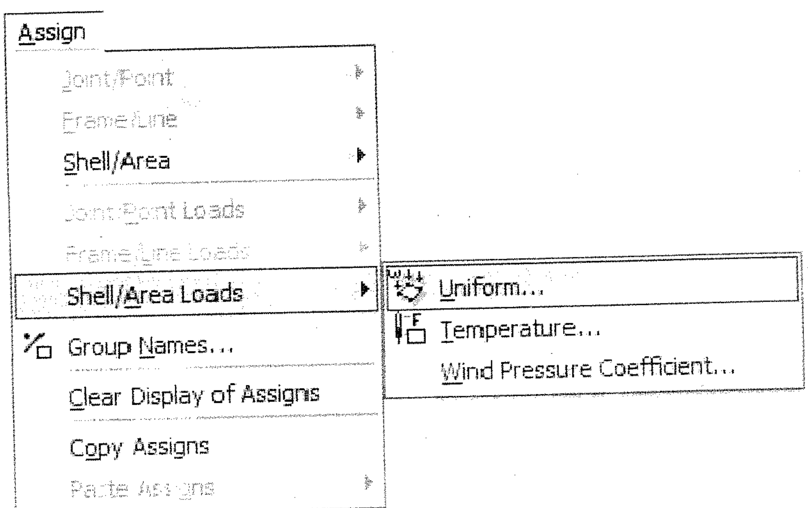
Repeat the same steps for wind Y and don't forget to change the direction of the wind to be 90



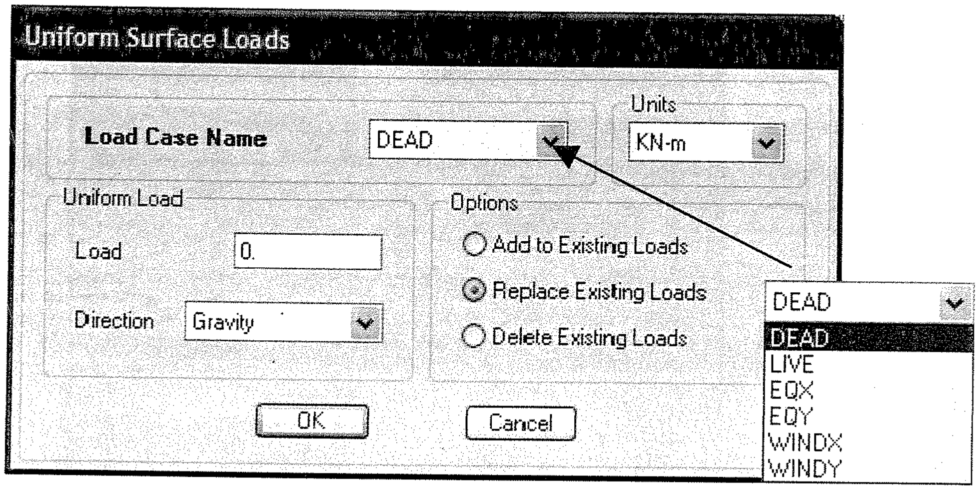
The Static Load case Names Should now Appear as in the next form



- <u>**Assign Dead &Live Load to the model**</u>
  - Choose the slab by click by mouse on the slab in plan view
  - Click the **Assign menu** ⟶ **Shell/Area Loads** ⟶ . **Uniform…..**



- The next form of **Uniform Surface Loads** Will be displayed

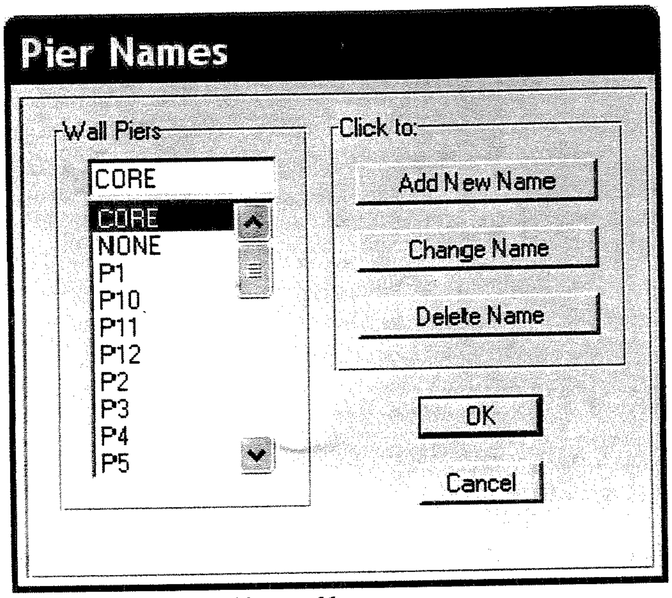


- From the Load Case Name pull down box choose Dead Load
- for typical floors the Uniform Load Value =5.5 KN/m'
- for Mechanical floors the Uniform Load Value=1 KN/m'
- click **OK**
- choose the slab again from the plan and repeat the assigning to the Live Load
- From the Load Case Name pull down box choose Live Load
- for typical floors the Uniform Load Value =2.0 KN/m'
- for Mechanical floors the Uniform Load Value=10 KN/m'

- **Step 6: Assign of Pier and Spandrel Labels**

- Choose the wall which you want to assign the pier label for it

- Click the **Assign menu** ⟶ **Shell** ⟶ **Pier Label** ,thin the next form will be displayed, from this form add pier name thin click Add New Name Thin **OK**



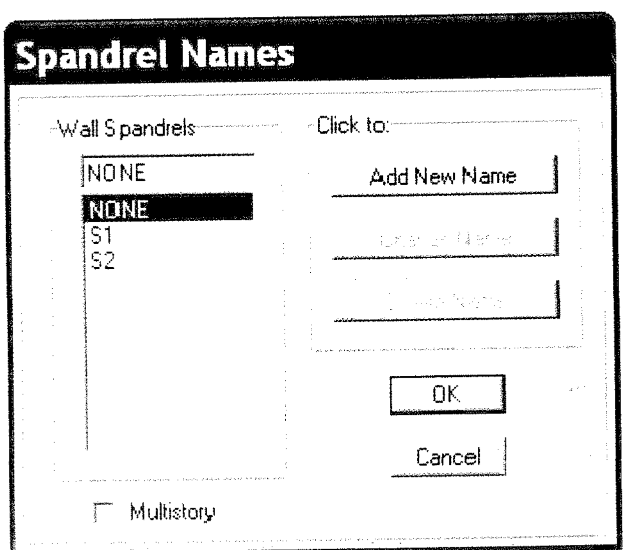
- Repeat the same steps for all walls

Note: you can not make design for any wall not have pier label

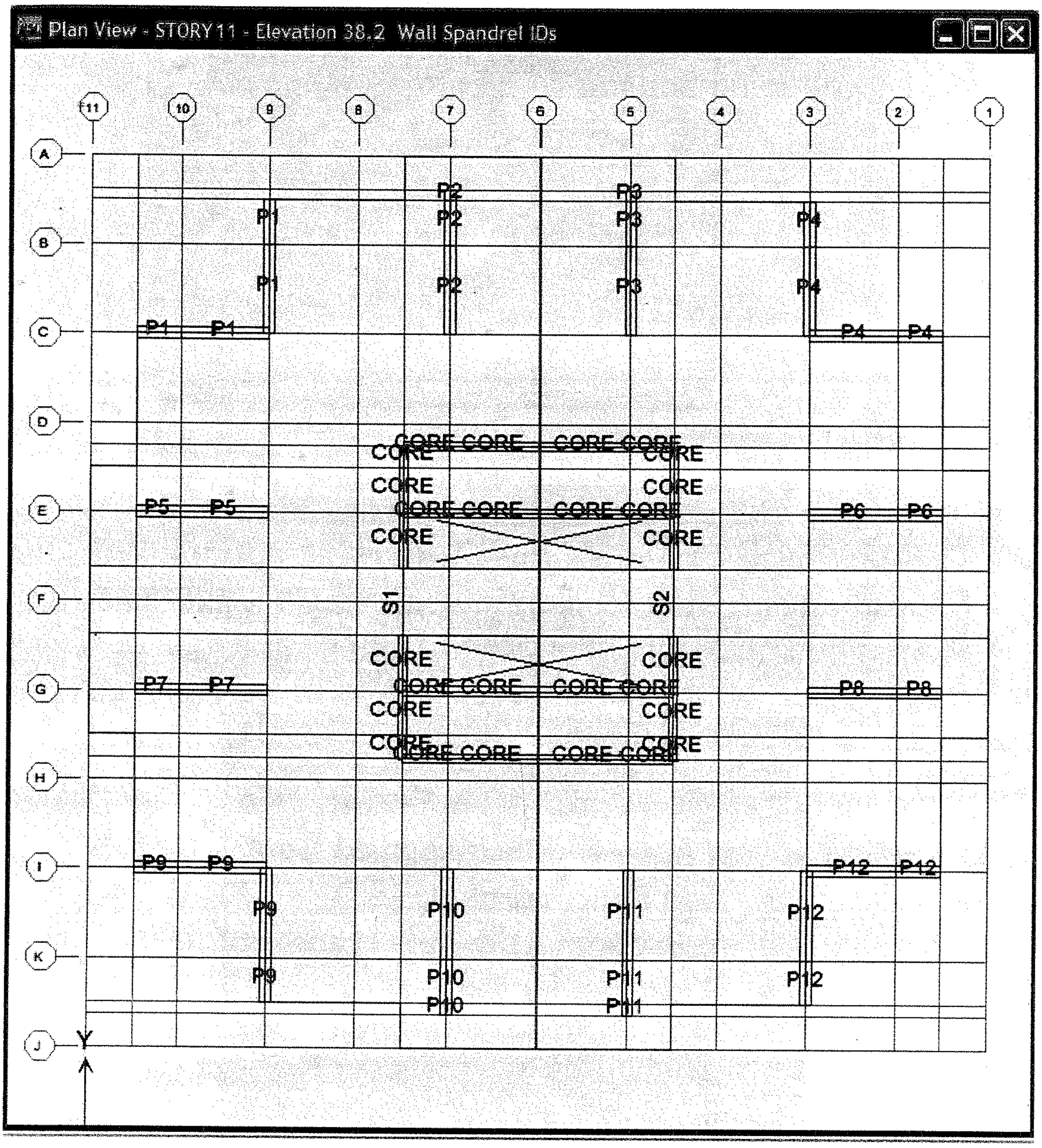
All the walls must have different pier name

- Choose the Spandrel beam which you want to assign the Spandrel label for it

- Click the **Assign menu** ⟶ **Frame/ line** ⟶ **Spandrel Label** the next form will be displayed from this form
  - add Spandrel name (S1)
  - thin click **Add New Name** button
  - Thin **OK** button



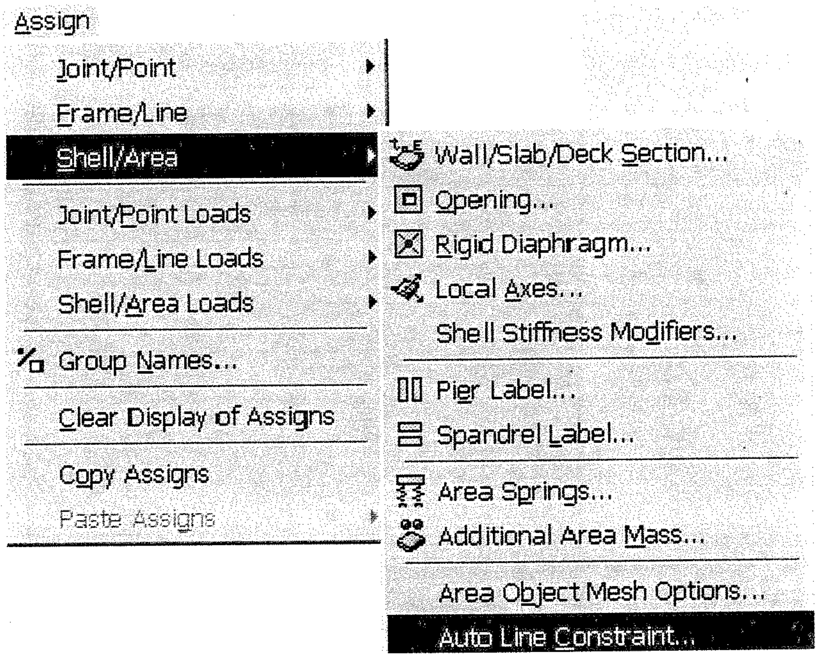
After you finish assigning the walls label and spandrel beams labels, the labels will display on the plan as in the figure



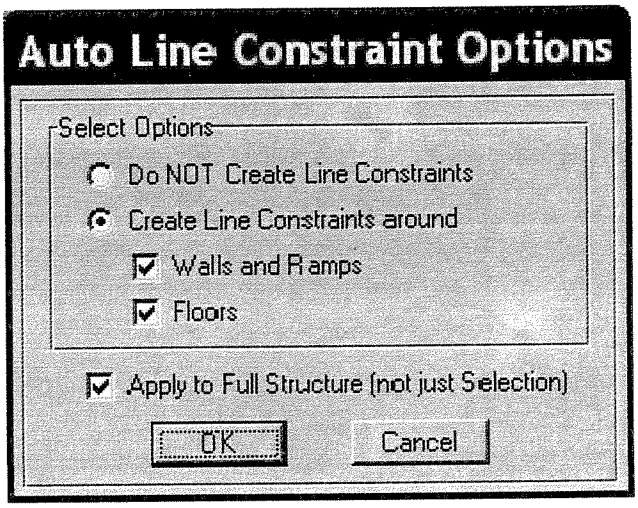
- **Step 7: Define the mass source & AutoLine constraints &Assign the supports**

Assign of the Auto Line Constrain

- Choose the whole building
- Click the **Assign menu** ⟶ **Shell/Area** ⟶ **Auto Line Constraint...**

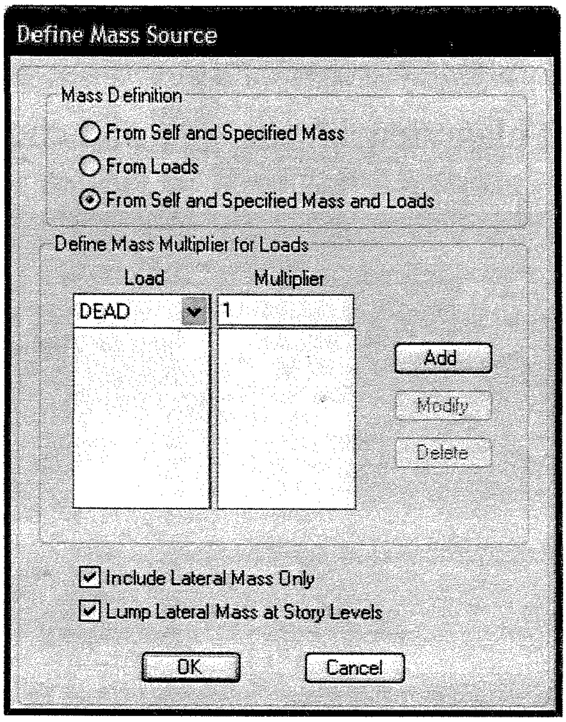


- The next form of **Auto Line Constraint options** Will display Choose create line constraints around (Walls and Ramps, Floors) thin click Ok

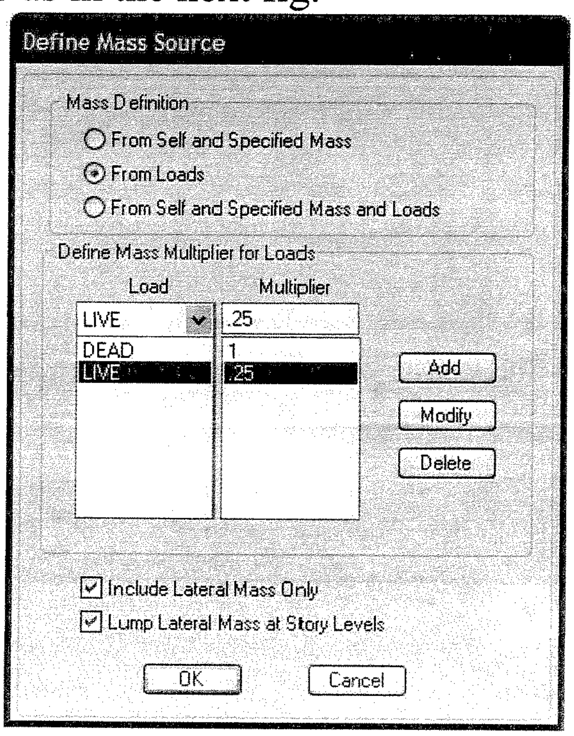


Define the mass source

- Click the **Define menu** → **Mass Source** ,thin the next form will be displayed

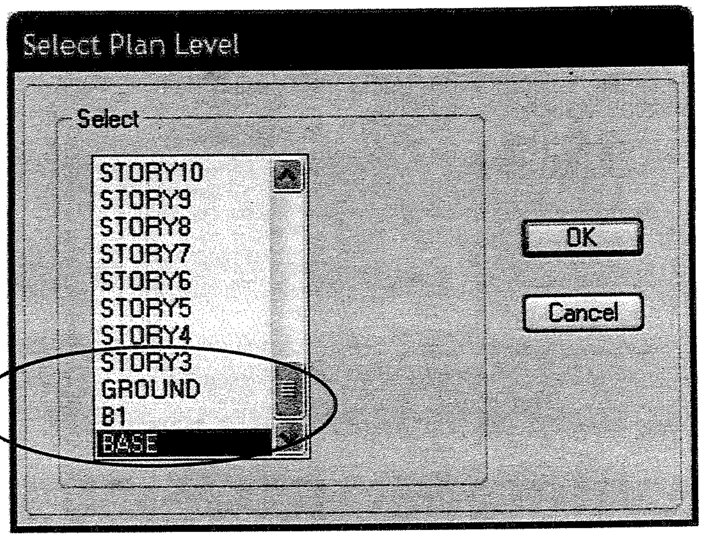


- From the previous form highlight **From Loads** from Mass Definition Box
- Thin adjust the value of dead load multiplier to be equal 1
- And the live load you have the choice to make it equal to .25 or less (according to UBC code if the live load more than 5 KN/m' take the live load multiplier =0.25), thin the form will be as in the next fig.



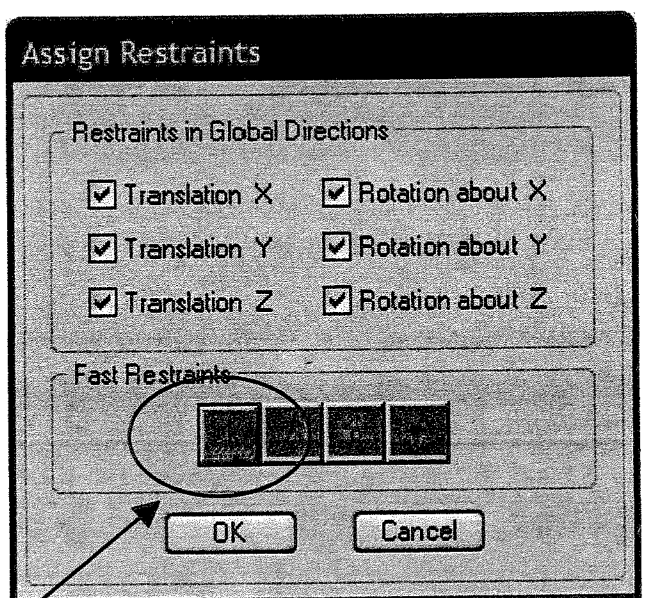
Assign the supports.

- click set plan view button and choose base plan view



- Select all the point in the Base plan by window.

- Click the **Assign menu** → **Joint /Point Restraints** ,thin the next form will be displayed

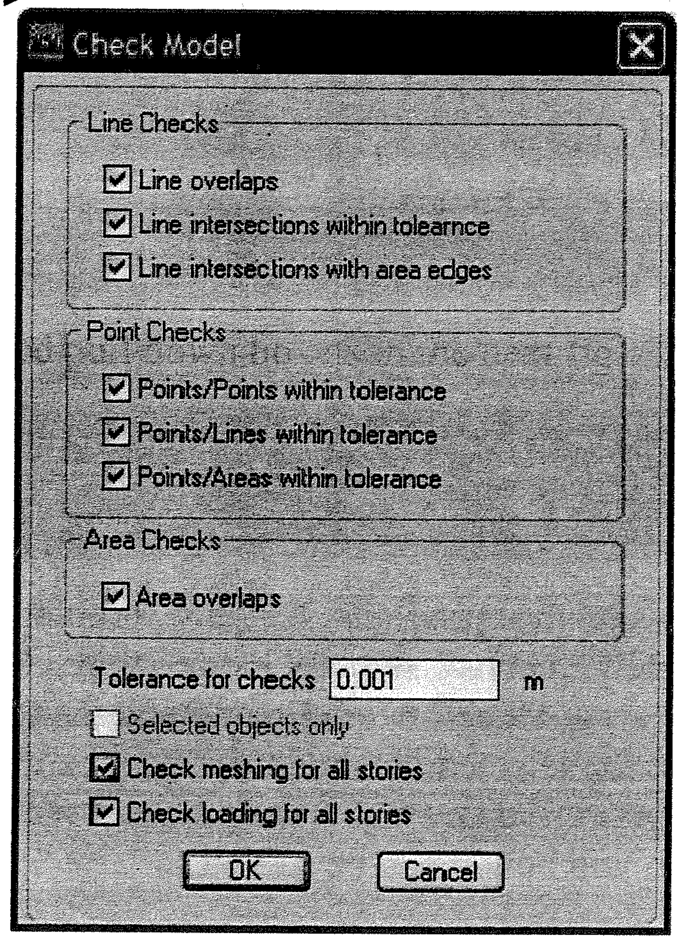


Click here to assign fixed support

- Click Fixed support Icon
- Click **OK**

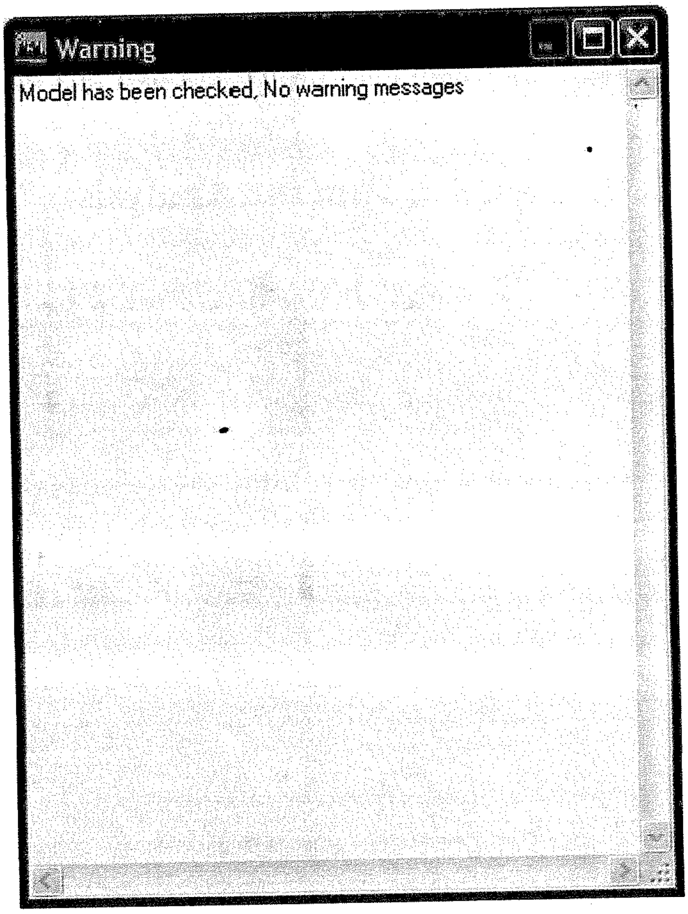
- **Step8: Check the model & Run the Analysis**

- To check you model before run the analysis, Click the **Analyze menu** ⟶ **Check Model** command

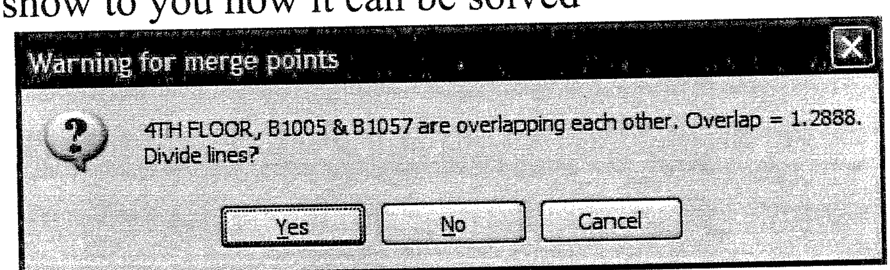


- Check the all check boxes to allow the program to check all the checks of your model

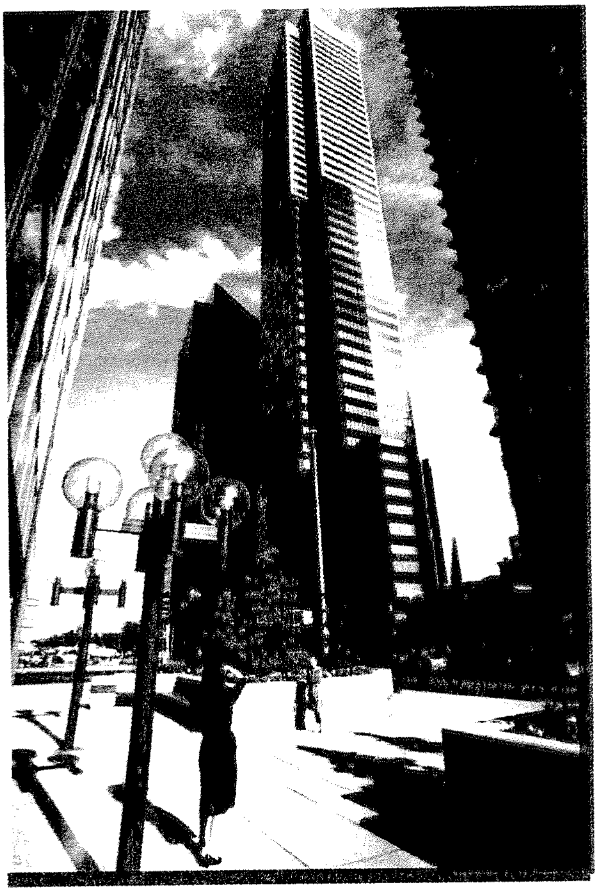
- After the program check your mode ,if there are no warning the program display no warning if any problem in your model the program will display warning massage to adjust your model



- if you want the program fix any problem in the model automatically ,choose the all model thin click the **Analyze menu** ⟶ **Check Model** command ,and you will find the check box for selected object only is active , check this box and if the program find any errors the program will display massage to show to you how it can be solved



- After you check your model Click the **Analyze menu** ⟶ **RUN Analysis** command or the Run Analysis button
- The program will create the analysis model from your object-based ETABS model after the program finish the run the model will be locked and display the deformed shape of the building



HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

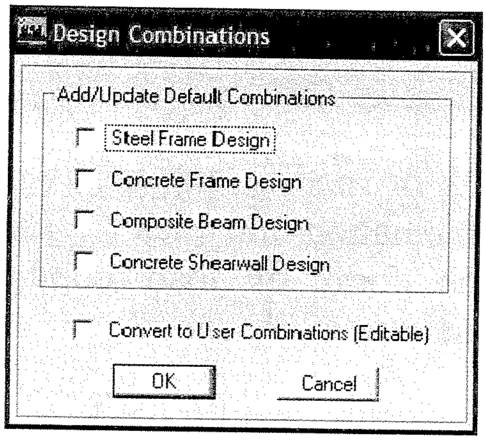
Display the Results

Chapter 2

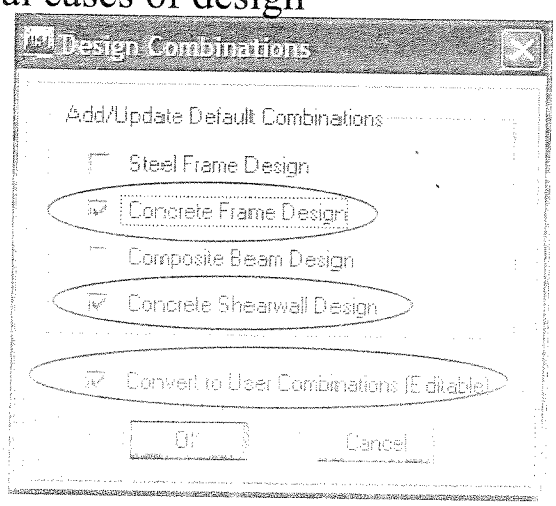
Step 1: automatic case of loading creation

Note :Etabs program has the ability to create the case of loading automatically according to the code which you will use in design ,and the program give you the ability to make any modification for this case of loading according to the special requirement of the codes

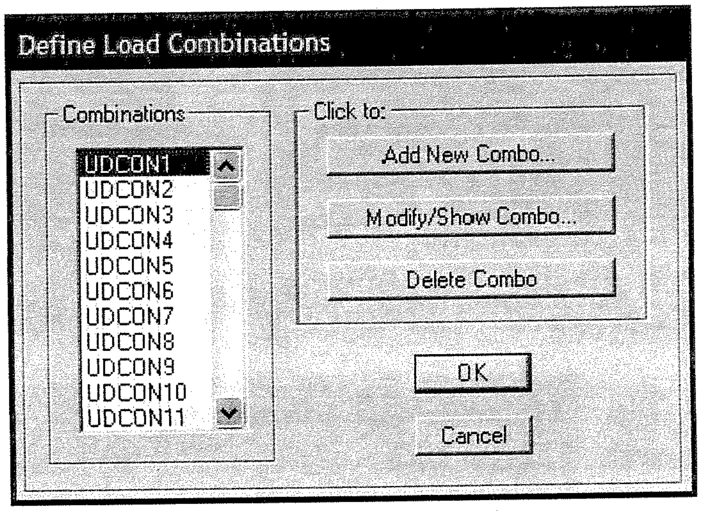
- Click the **Define** ⟶ **Add Default Load Combos... thin** Design Combination form as shown in fig will be displayed.



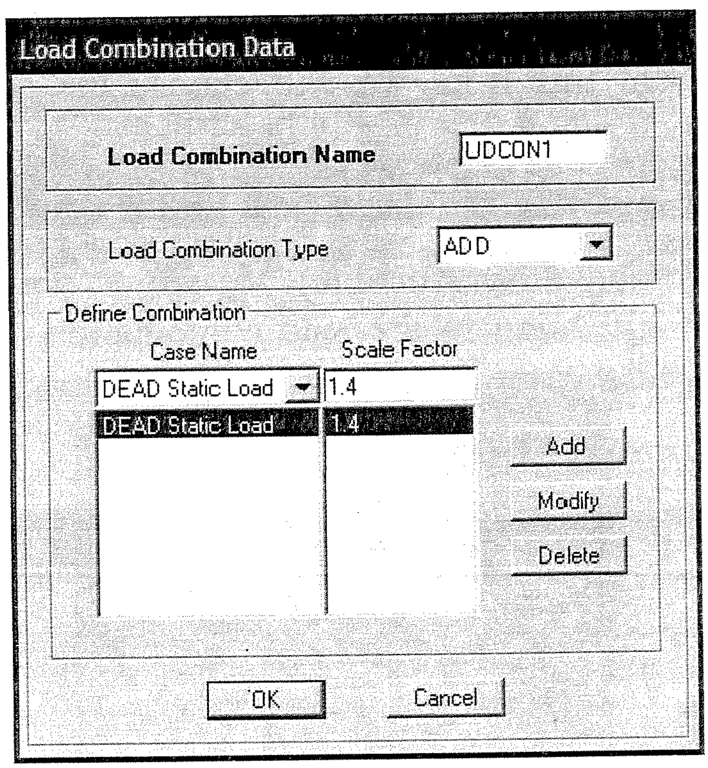
- From this from we will choose the load combination code which we was defined before in chapter 1 step 2 (ACI – 318 02), highlight concrete frame design and concrete shear wall design and convert to User Combinations(Editable) to allow you to make any modification in the load combination according to any special cases of design

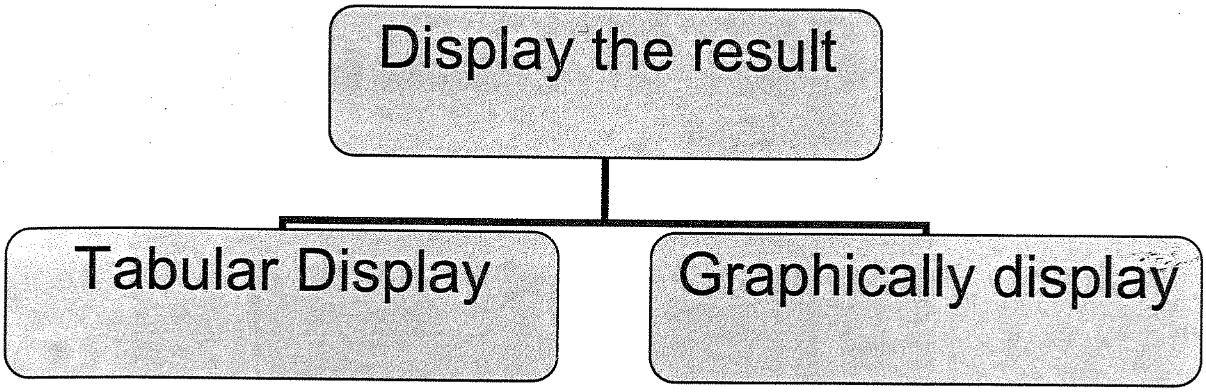


- Click the **Define** ⟶ **Load Combinations;** thin Define Load Combinations form as shown in fig will be displayed.



- To check or modify any load combination ,choose the load combination thin click Modify/Show Combo... Thin the form of load combination data will be displayed





- **Step 2: tabular Display of the analysis result**

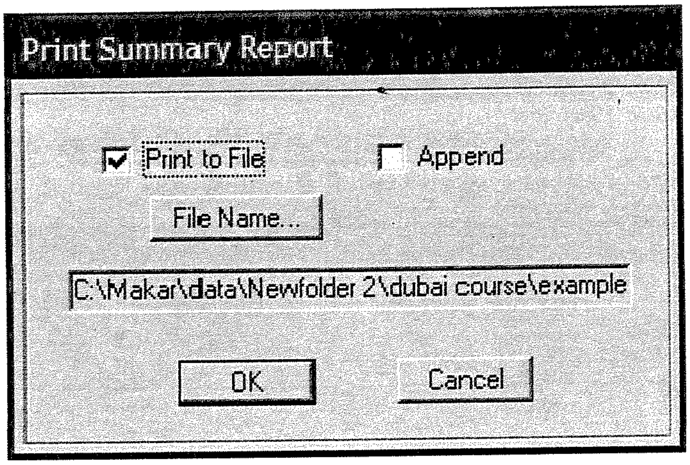
  In this step, we will review the most important data coming from the model like,

  1. Base Shear of the building
  2. inter story drift
  3. straining actions of the frame elements in tabular form
  4. straining actions of the walls elements in tabular form

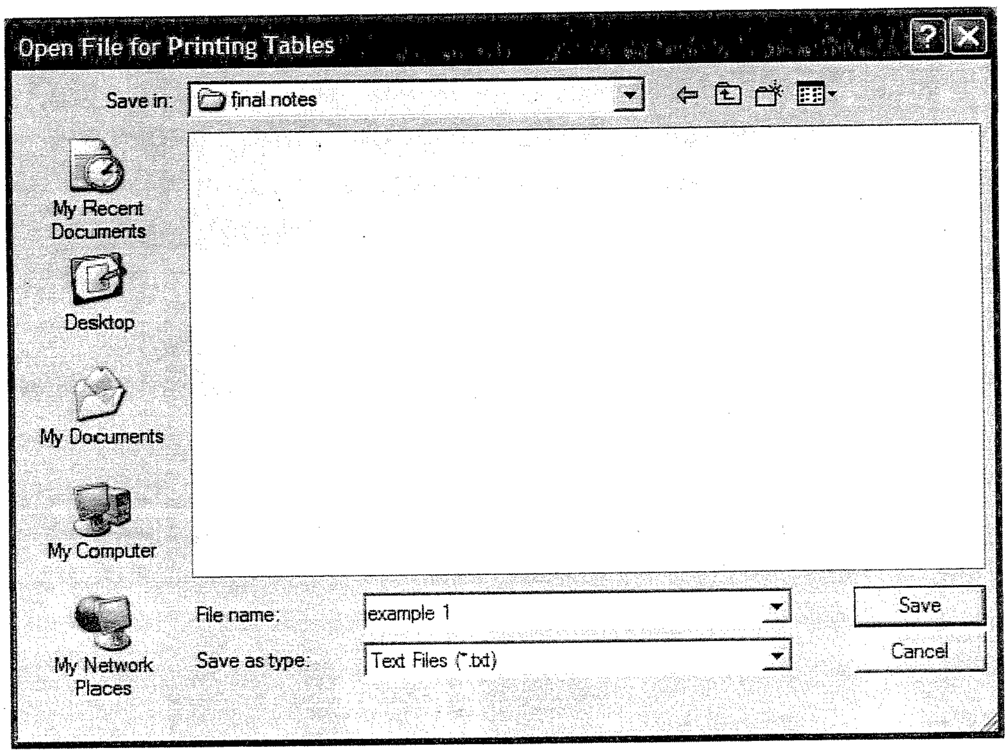
The most important tabular review for the result is the summery report which gives to you the most important data for the model and the calculation details for wind, and earthquake loads, and the summary for the result

1. **Summary Report**

   - Click the **File** ⟶ **Print Table** ⟶ **Summary Report,** thin the form as shown in fig will be Displayed.



- Click File Name... to choose the location where the file will be saved using the next form



- The next form will show to you extract of some information in summary report (Earthquake calculation) example 1 - Notepad

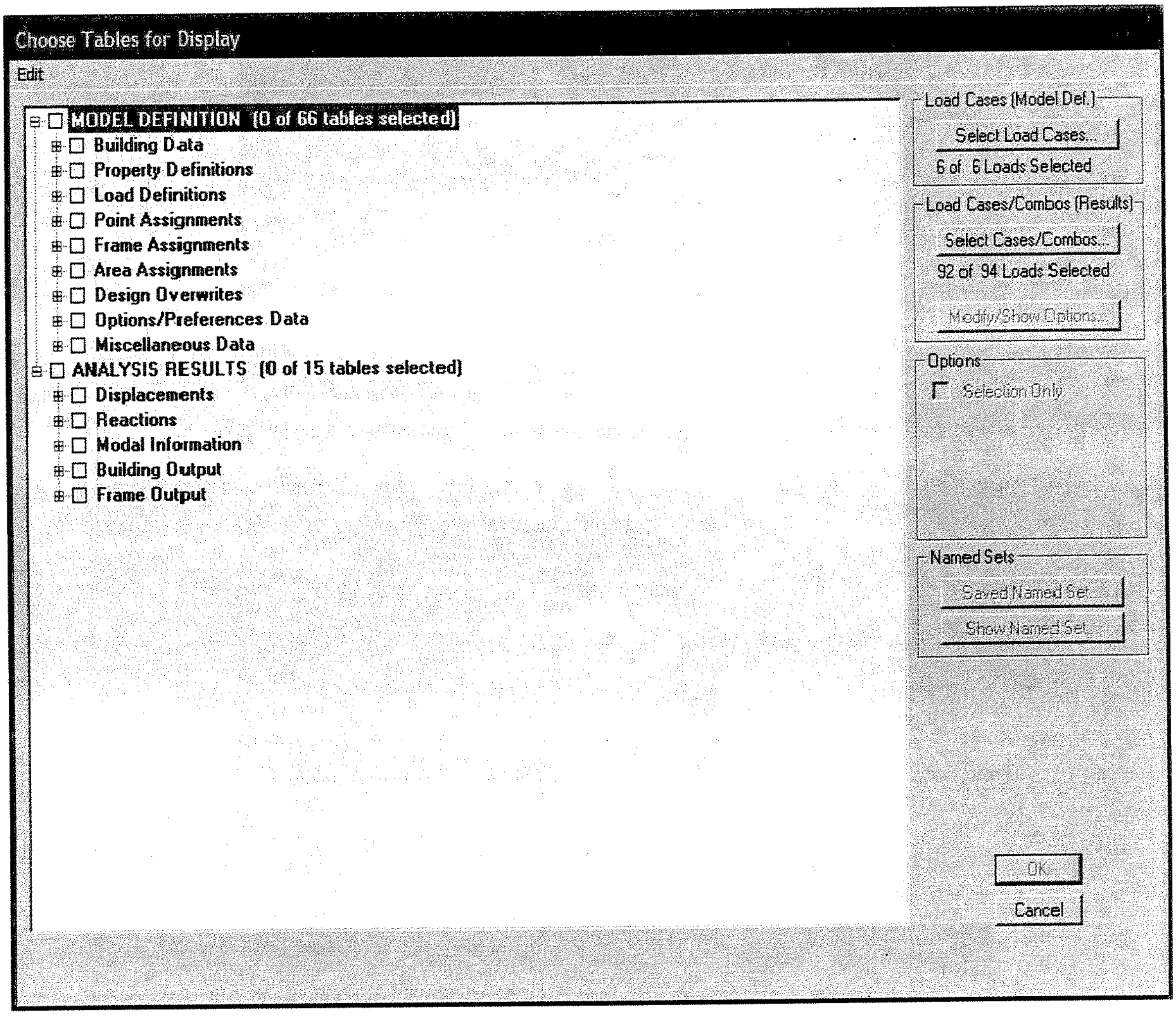
File Edit Format View Help

```
AUTO SEISMIC INPUT DATA

  Direction: X + EccY
  Typical Eccentricity = 5%
  Eccentricity Overrides: No

  Period Calculation: Program Calculated
  Ct = 0.02 (in feet units)

  Top Story: STORY40
  Bottom Story: BASE

  R = 5.5
  I = 1
  hn = 137.400 (Building Height)

  Soil Profile Type = SC
  Z = 0.15
  Ca = 0.1800
  Cv = 0.2500


AUTO SEISMIC CALCULATION FORMULAS

  Ta = Ct (hn^(3/4))

  If Z >= 0.35 (Zone 4) then:        If Tetabs <= 1.30 Ta then T = Tetabs, else T = Ta
  If Z < 0.35 (Zone 1, 2 or 3) then: If Tetabs <= 1.40 Ta then T = Tetabs, else T = Ta

  V = (Cv I W) / (R T)                                (Eqn. 1)
  V <= 2.5 Ca I W / R                                 (Eqn. 2)
  V >= 0.11 Ca I W                                    (Eqn. 3)

  If T <= 0.7 sec, then Ft = 0
  If T > 0.7 sec, then Ft = 0.07 T V <= 0.25 V


AUTO SEISMIC CALCULATION RESULTS

  Ta = 1.9566 sec
  T Used = 2.7393 sec
  W Used = 499320.21

  V (Eqn 1) = 0.0166W
  V (Eqn 2) = 0.0818W
  V (Eqn 3) = 0.0198W
  V (Eqn 4) = 0.0349W

  V Used = 0.0198W = 9886.54

  Ft Used = 1895.74


AUTO SEISMIC STORY FORCES

  STORY          FX        FY       FZ       MX       MY        MZ

  STORY40     1966.33     0.00     0.00    0.000    0.000  -1376.430
  STORY39      306.18     0.00     0.00    0.000    0.000   -489.882
```

You can see from the previous form the whole steps the program do to calculate the seismic force and the values of the force acting in each floor in details ,also the program give to you the details for the wind calculation , summary of the input data .and the results

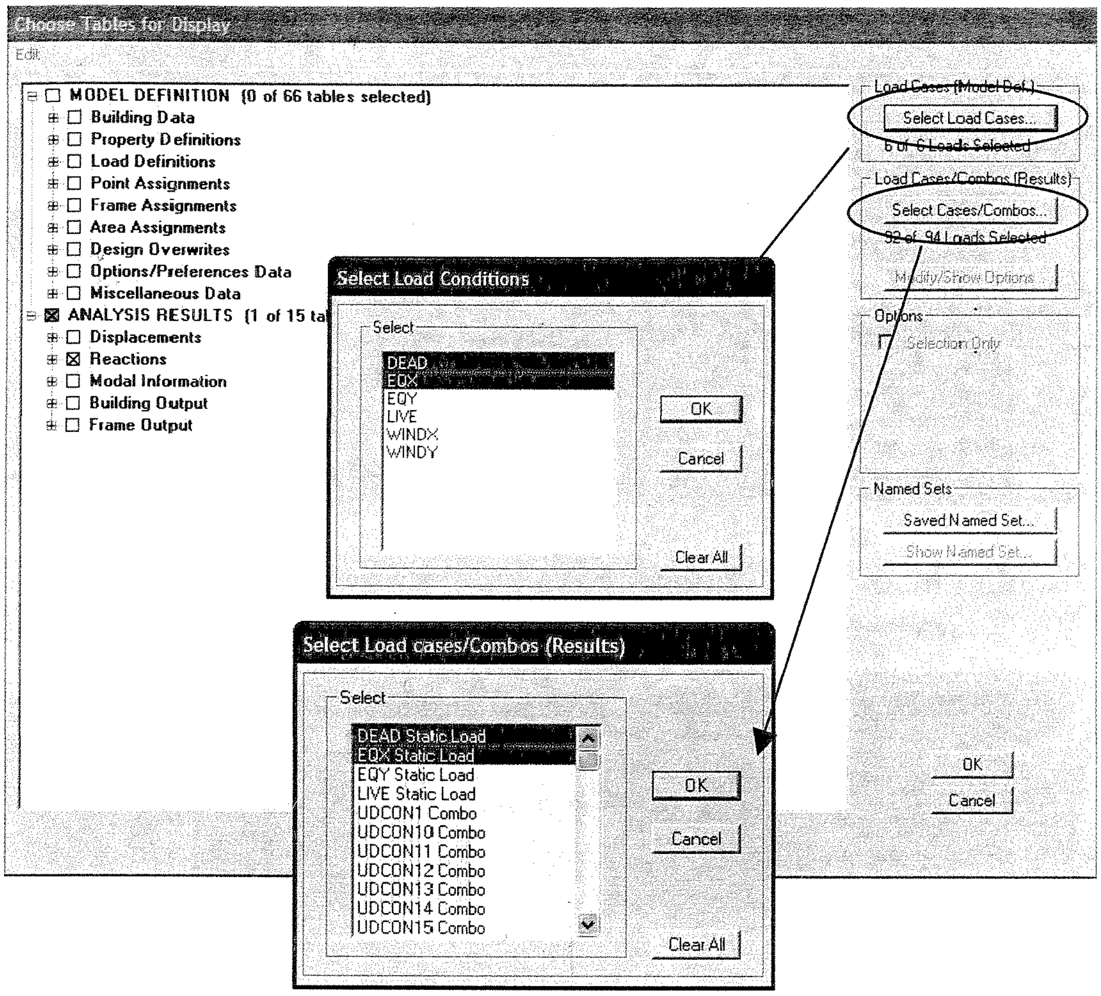
2. Tables Display

- Click the **Display** ⟶ **Show Tabels**
- thin the form as shown in fig will be displayed.

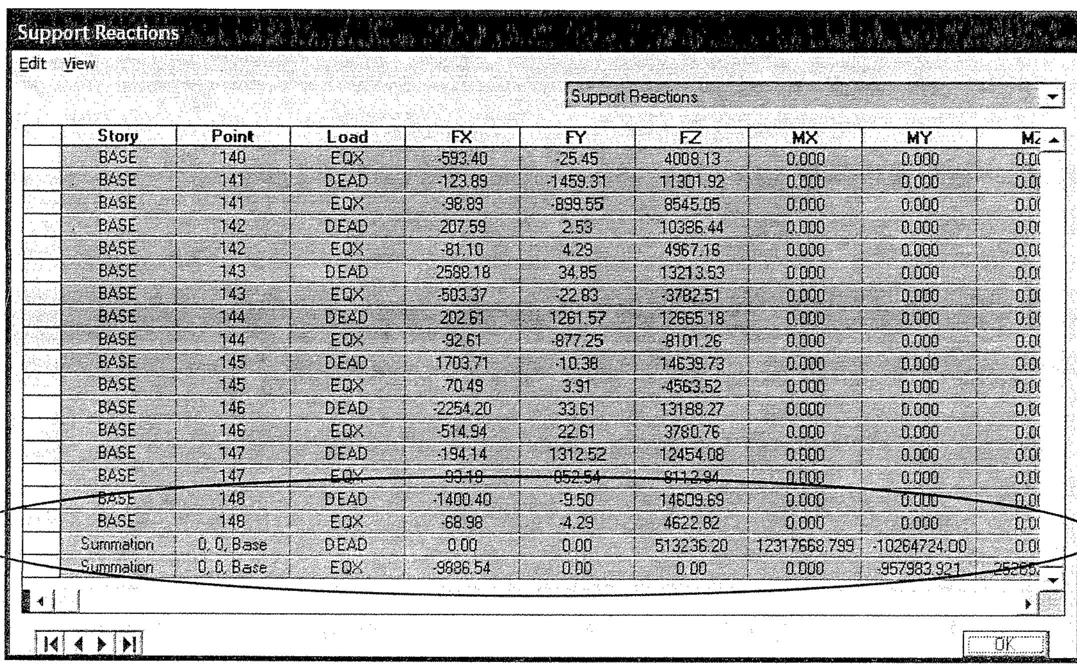


- From previous form choose and mark the data which you want to display ,and the case of loading thin click OK

- As an example , mark Reactions ,and select Dead, and EQX case of loading, thin click Ok



- The next form of the reaction of the supports will be displayed



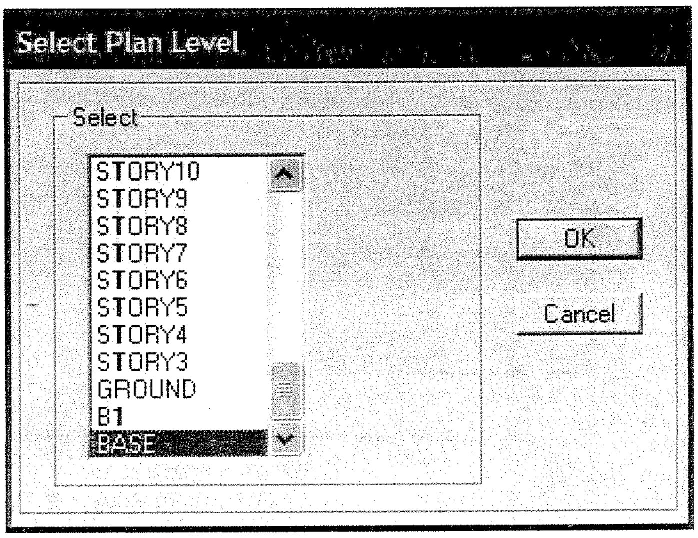
Story	Point	Load	FX	FY	FZ	MX	MY	MZ
BASE	140	EQX	-593.40	-25.45	4008.13	0.000	0.000	0.0
BASE	141	DEAD	-123.89	-1459.31	11301.92	0.000	0.000	0.0
BASE	141	EQX	-98.89	-899.55	8545.05	0.000	0.000	0.0
BASE	142	DEAD	207.59	2.53	10386.44	0.000	0.000	0.0
BASE	142	EQX	-81.10	4.29	4967.16	0.000	0.000	0.0
BASE	143	DEAD	2588.18	34.85	13213.53	0.000	0.000	0.0
BASE	143	EQX	-503.37	-22.83	-3782.51	0.000	0.000	0.0
BASE	144	DEAD	202.61	1261.57	12665.18	0.000	0.000	0.0
BASE	144	EQX	-92.61	-877.25	-8101.26	0.000	0.000	0.0
BASE	145	DEAD	1703.71	-10.38	14639.73	0.000	0.000	0.0
BASE	145	EQX	-70.49	3.91	-4563.52	0.000	0.000	0.0
BASE	146	DEAD	-2254.20	33.61	13188.27	0.000	0.000	0.0
BASE	146	EQX	-514.94	22.61	3780.76	0.000	0.000	0.0
BASE	147	DEAD	-194.14	1312.52	12454.08	0.000	0.000	0.0
BASE	147	EQX	-93.19	852.54	8112.94	0.000	0.000	0.0
BASE	148	DEAD	-1400.40	-9.50	14609.69	0.000	0.000	0.0
BASE	148	EQX	-68.98	-4.29	4622.82	0.000	0.000	0.0
Summation	0, 0, Base	DEAD	0.00	0.00	513236.20	12317668.799	-10264724.00	0.0
Summation	0, 0, Base	EQX	-9886.54	0.00	0.00	0.000	-957983.921	25265.

You will find in the end of the form the summation of the reactions

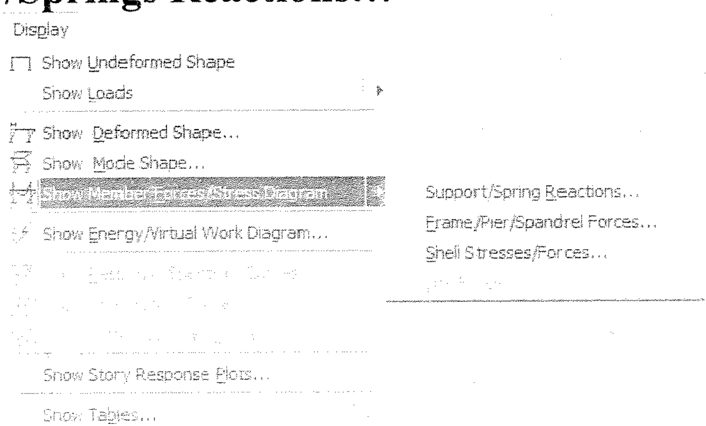
- **Step 3: Graphically Display of the analysis result**
  In this step, we will review the result coming from the model in Graphical Display, this result like

  1. Reactions.
  2. Beams and columns straining actions.
  3. Walls &Spandrel straining actions.

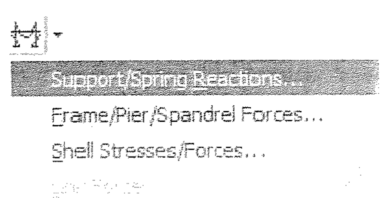
1. **Reactions**: to display reaction click on the plan view window to activate it ,and click on set plan view button Pla Pln ,thin from plan level form choose base level thin click OK



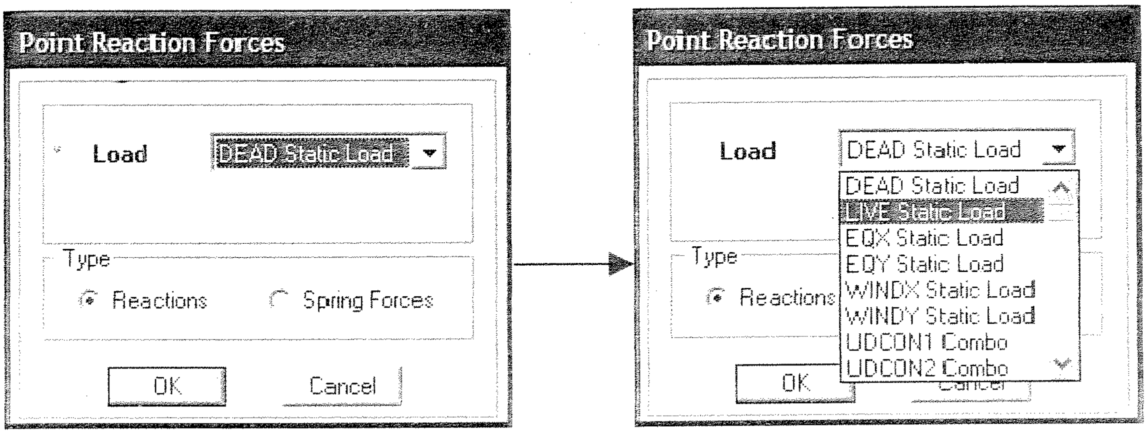
Thin the plan view of the base level will be displayed, click **Display** → **Show Member Forces/Stress Diagram** → **Support /Springs Reactions…**



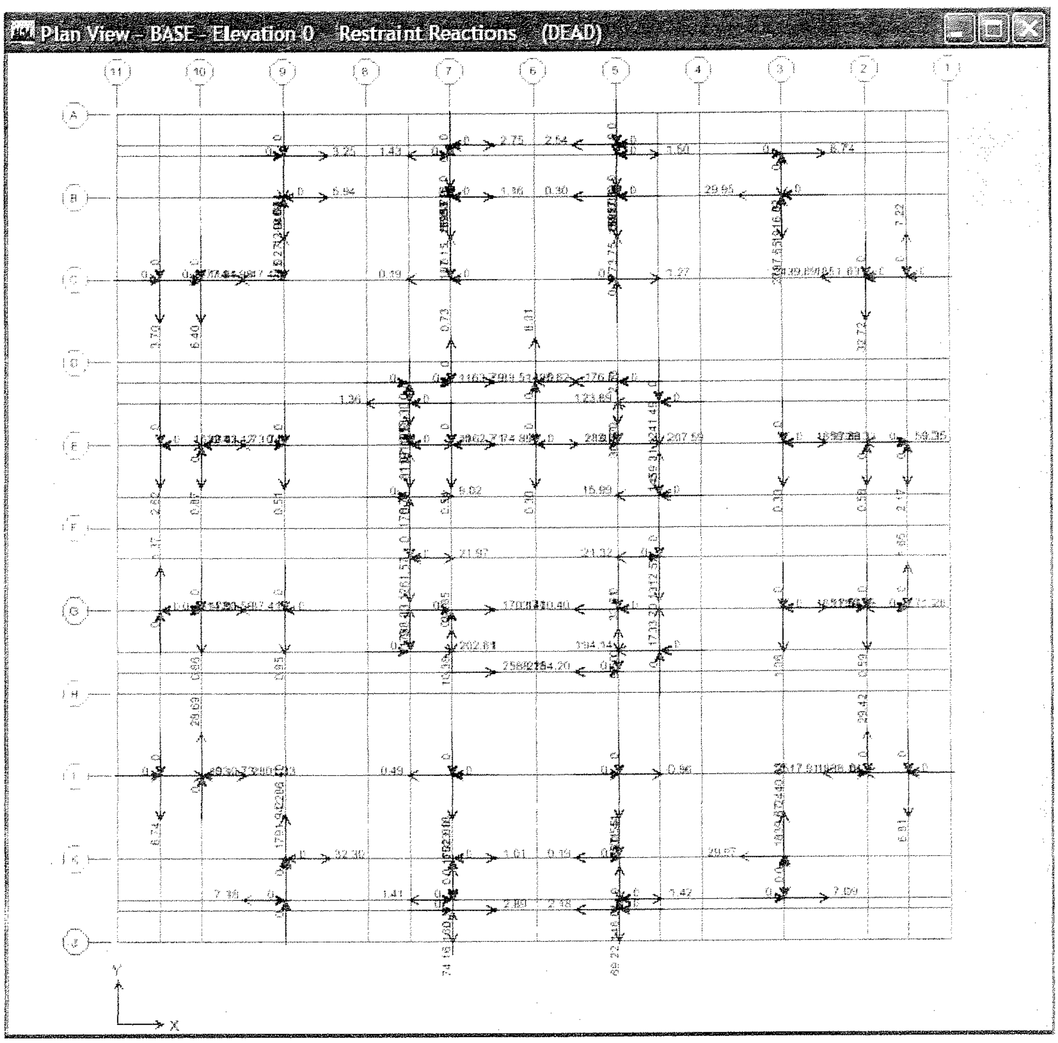
Or



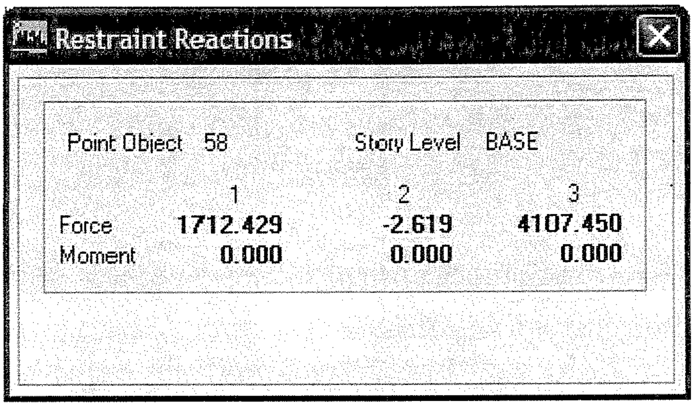
The Point Reaction Force form will displayed, thin choose the case of loading



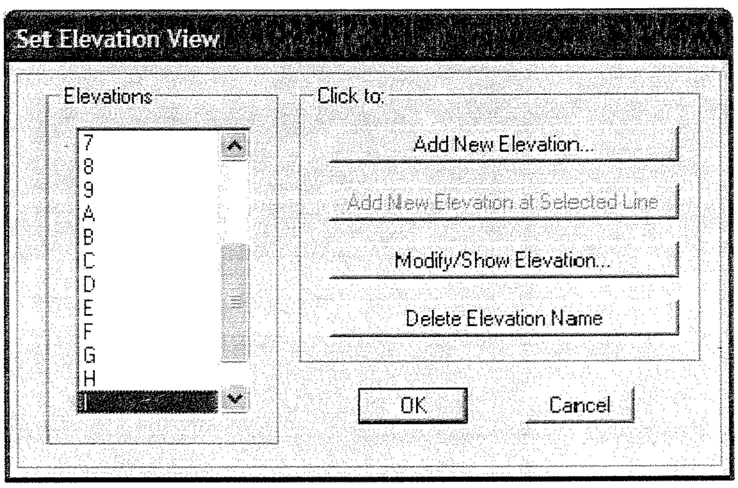
Click the **OK** button to generate the reaction force diagram



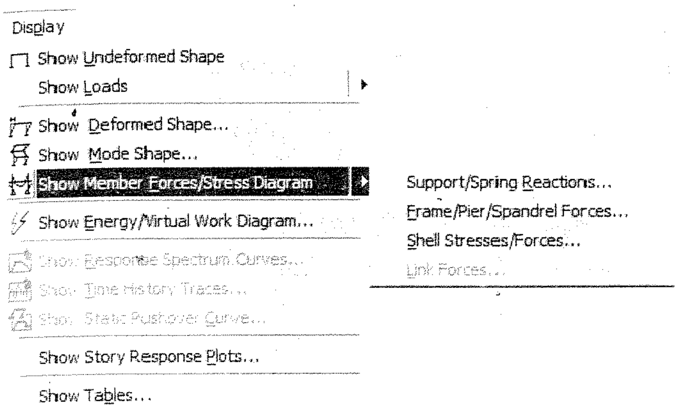
To display the value of any point reaction, choose this point by click on the left button of the mouse and thin click on the right button of the mouse, thin the restraints reaction values will be displayed as shown in the fig.



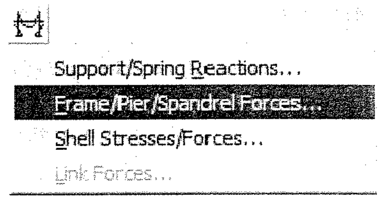
2. **Beams & Columns Straining actions**: to display Straining actions(normal force , shear force ,moments ,and torsion) click on the 3D view window to activate it and click on set Elevation view button thin from Elevation level form choose (I) thin click **OK** (you can choose any another elevation ,this elevation as example only)



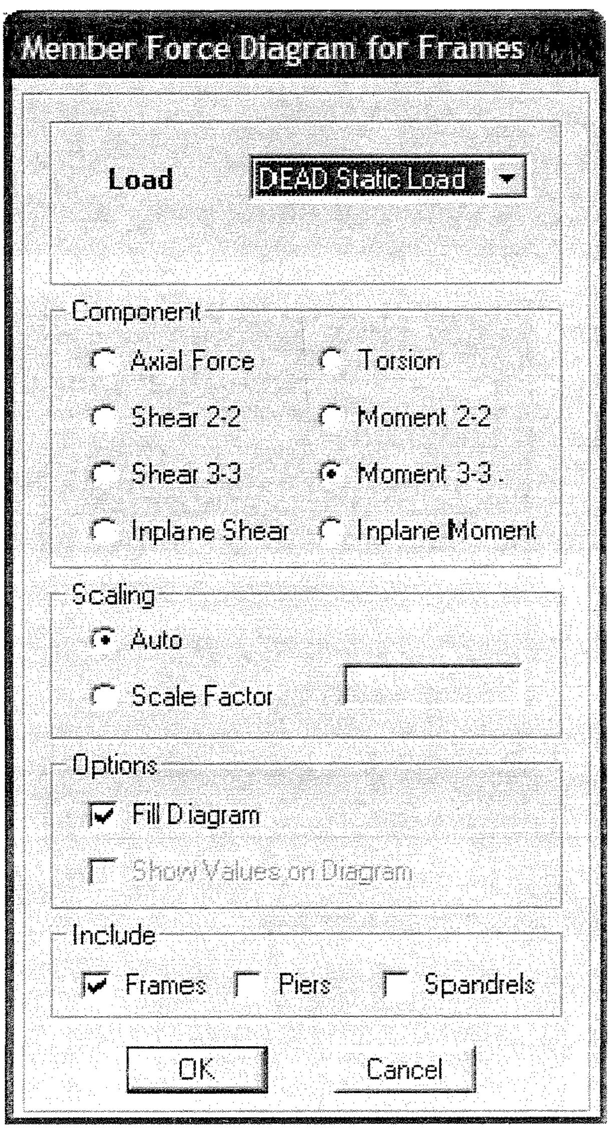
Thin the Elevation view will be displayed,
Click **Display** ⟶ **Show Member Forces/Stress Diagram** ⟶ **Frame/Pier/spandrel Forces**

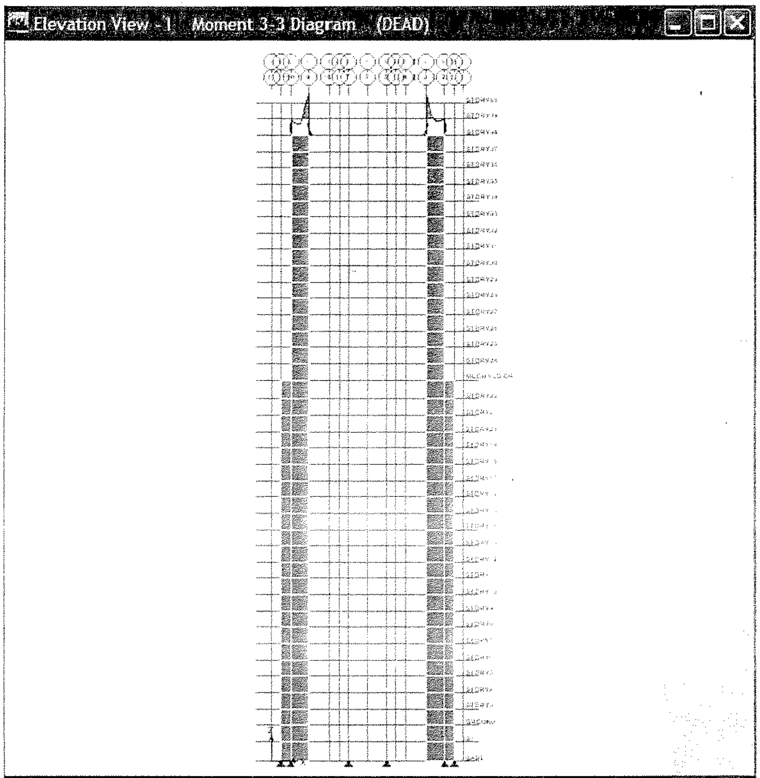


Or



The Member Force Diagram for Frames form will displayed, thin choose the case of loading

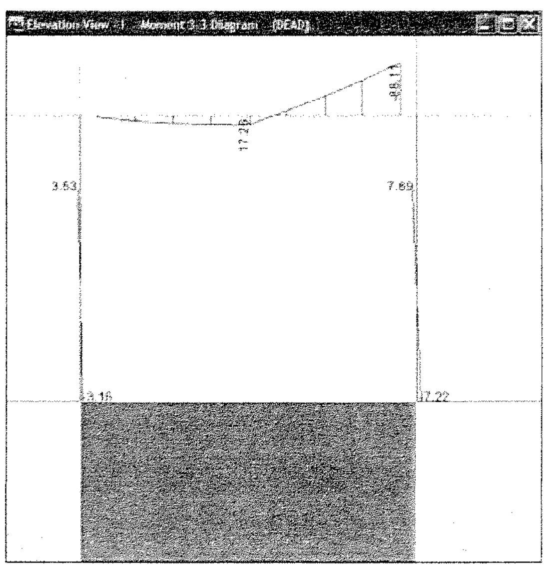




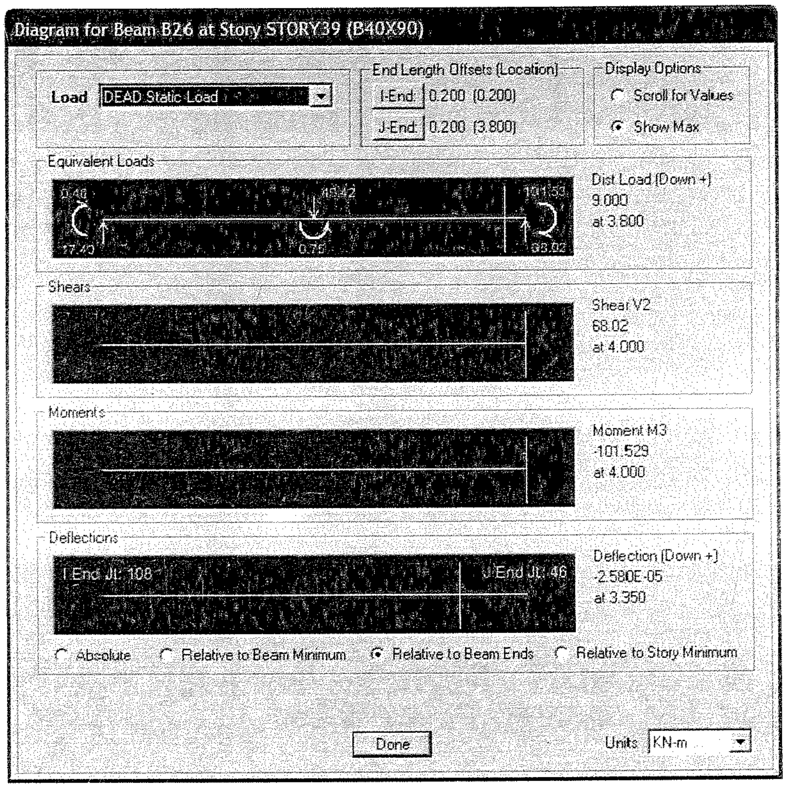
Click the OK button to generate the Member Force diagram for frames

To see the moment values click **Display** ——▶ **Show Member Forces/Stress Diagram** ——▶ **Frame/Pier/spandrel Forces**

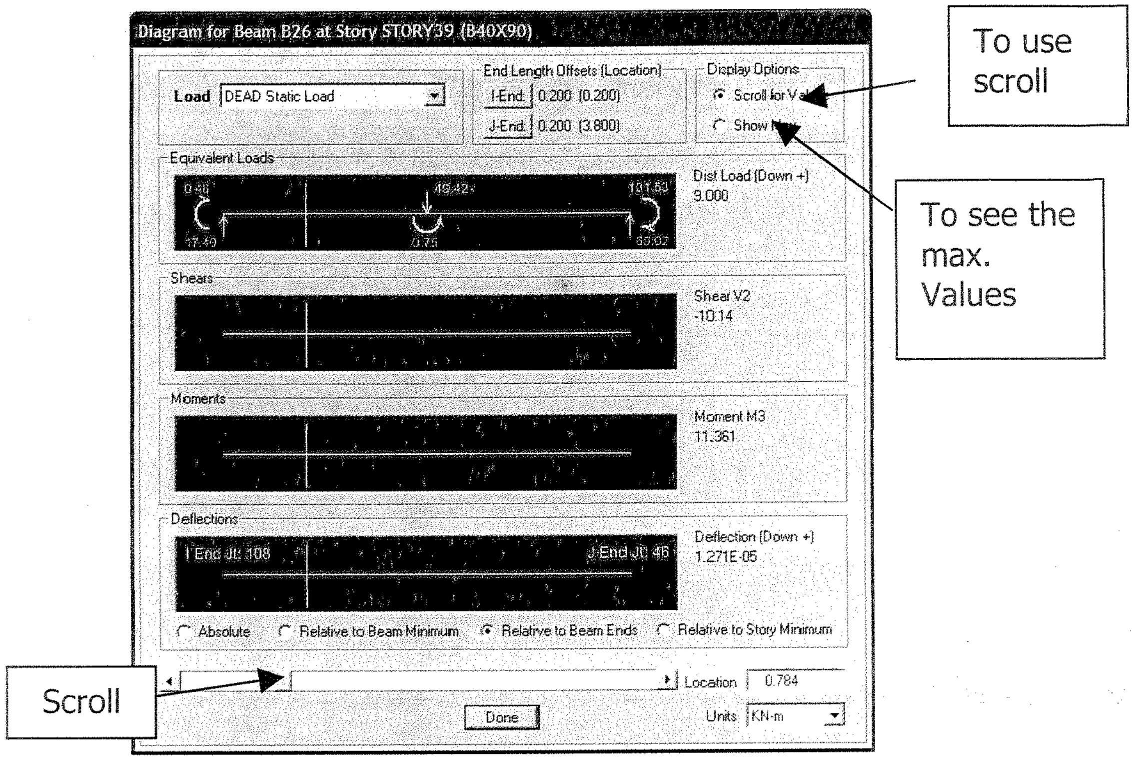
- Uncheck the fill Diagram
- Check the show Values on Diagram check box
- Click Ok thin the diagram will be shown as in the next fig



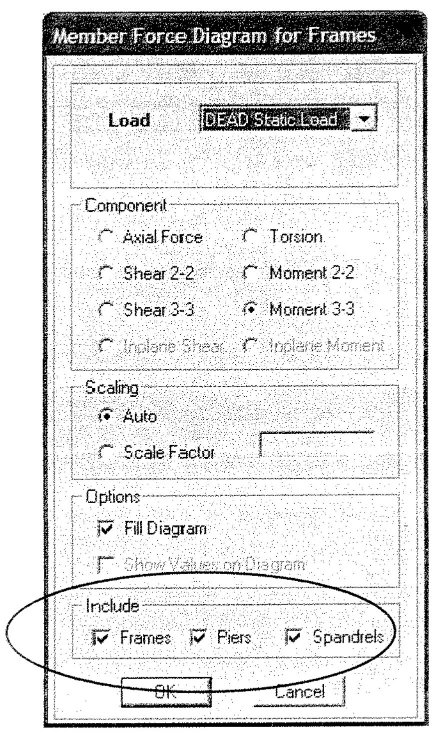
Left click on the beam by the mouse thin right click to display the form to bring up the Diagram for Beam form



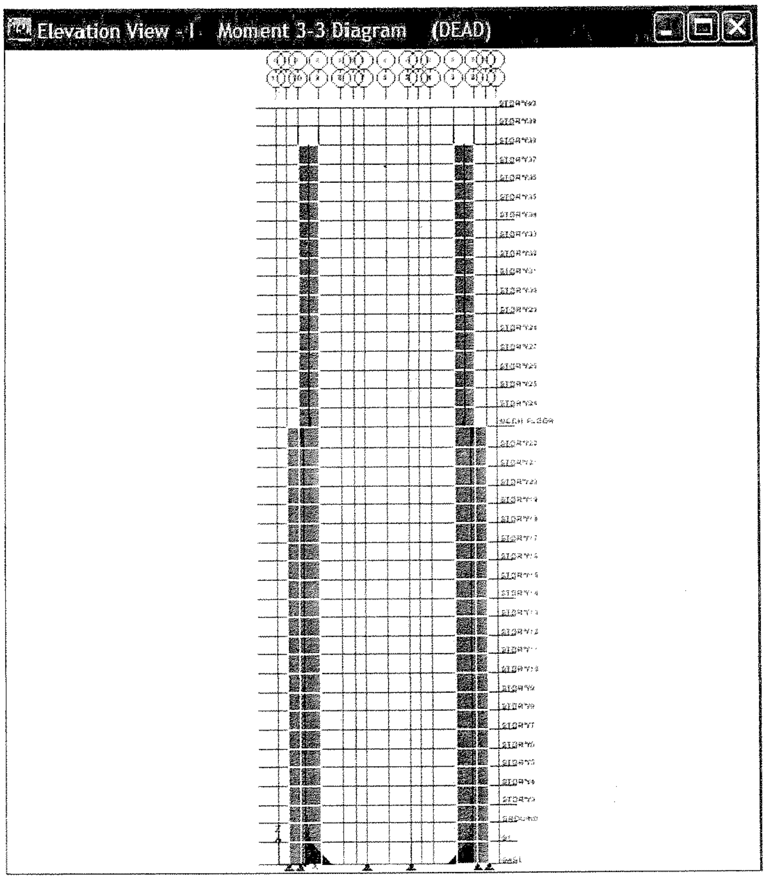
The maximum values for applied load, shear, moment, and deflections are identified on the diagram, if you want to know the value at any station of the beam click the Scroll for Values option and a scroll bar appears at the bottom of the form, you can drag the scrollbar by the mouse to see the different values at different locations



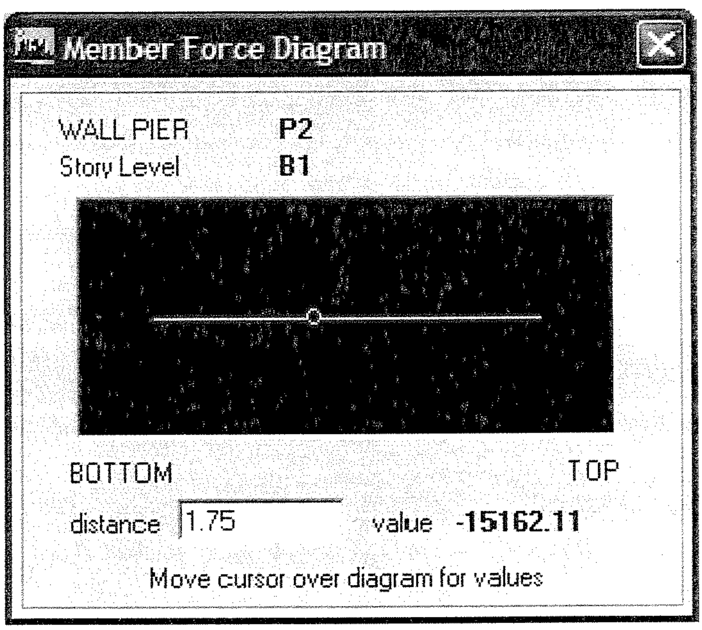
3. **Walls & Spandrel Straining actions**: to display Straining actions(normal force , shear force ,moments ,and torsion) for the walls and spandrel beams
    - repeat the same steps as in point 2
    - click on the check box of piers& Spandrels

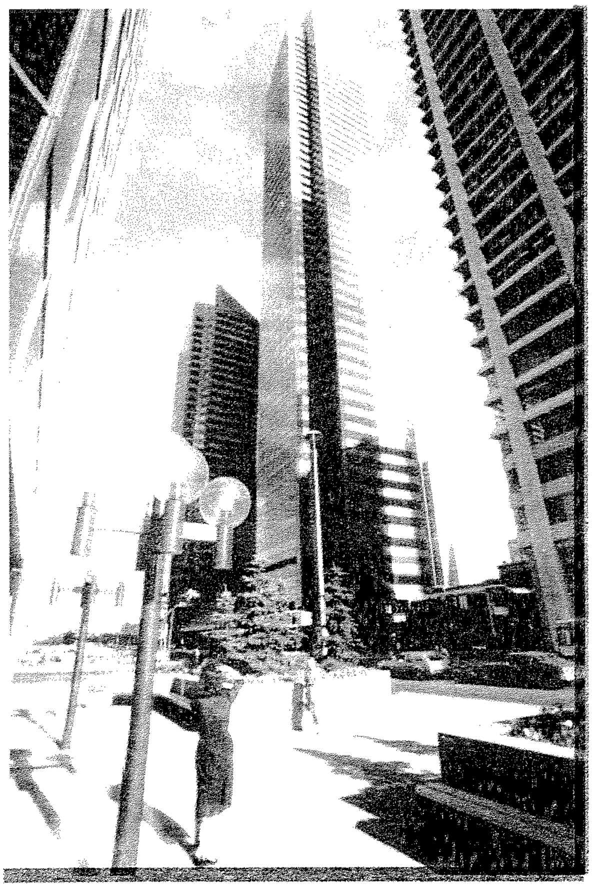


- click Ok



Left click on the beam by the mouse thin right click to display the form to bring up the Diagram for wall form





HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

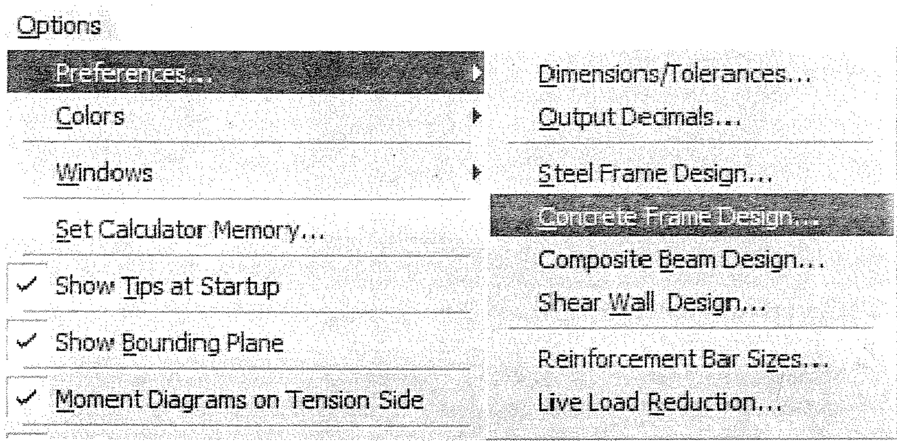
Concrete Design

Chapter 3

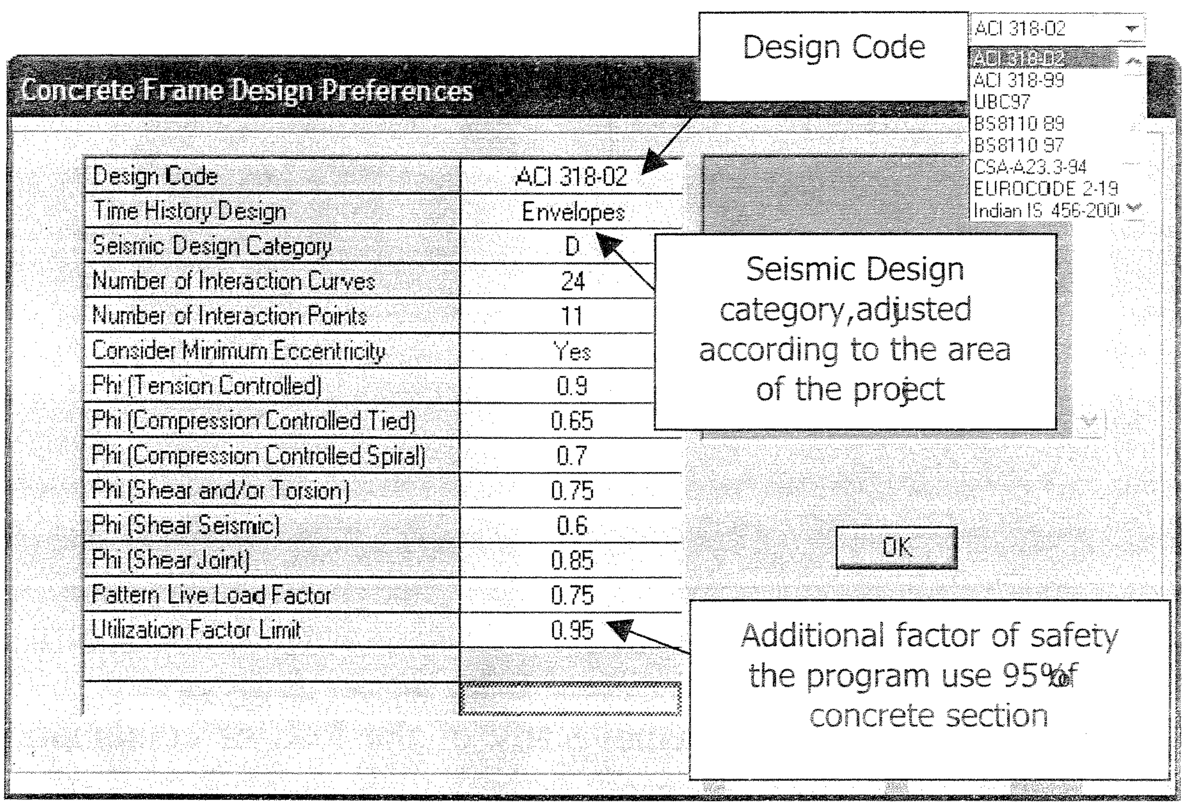
Concrete Frame Design(beams &columns)

1. Adjust the design code and Data of Design

- Click the **Option menu** ——▶ **Preferences** ——▶ **Concrete Frame Design**



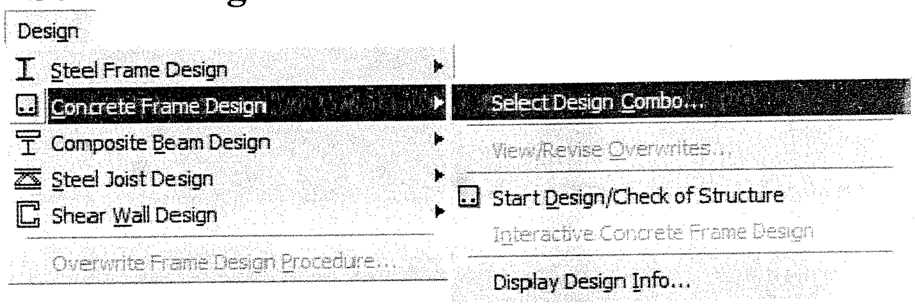
This will Display the Concrete Frame Design Preference form as shown in fig.



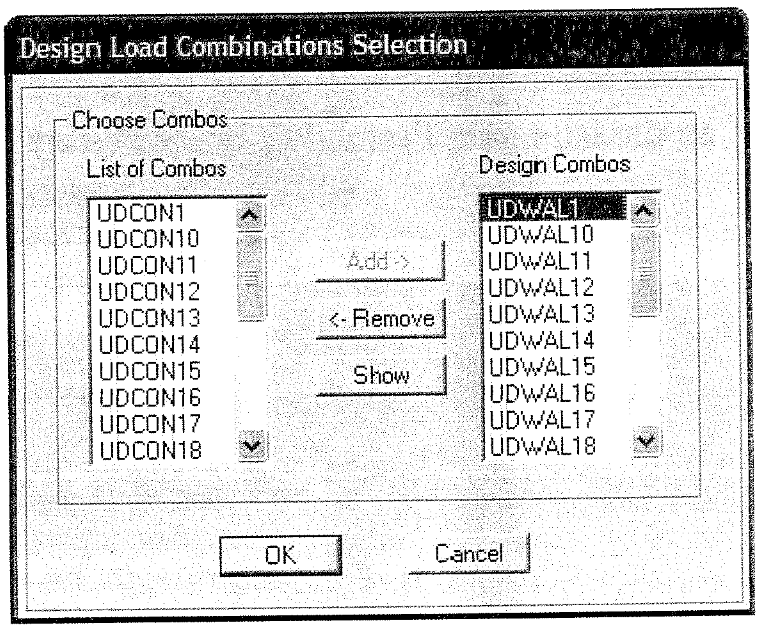
From this form, you can adjust the code and another data of design as shown in pervious figure.

2. **select of the Load of Combinations**

- Click the **Design menu** ——► **Concrete Frame Design** —► **Select Design Combo**



- Thin the next form will be displayed

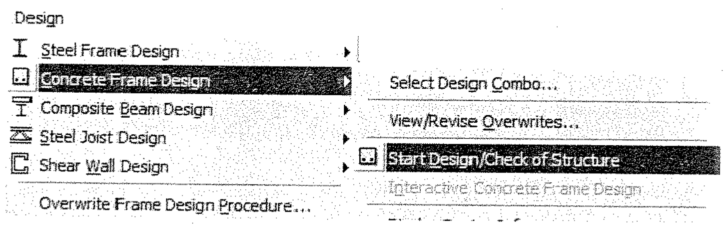


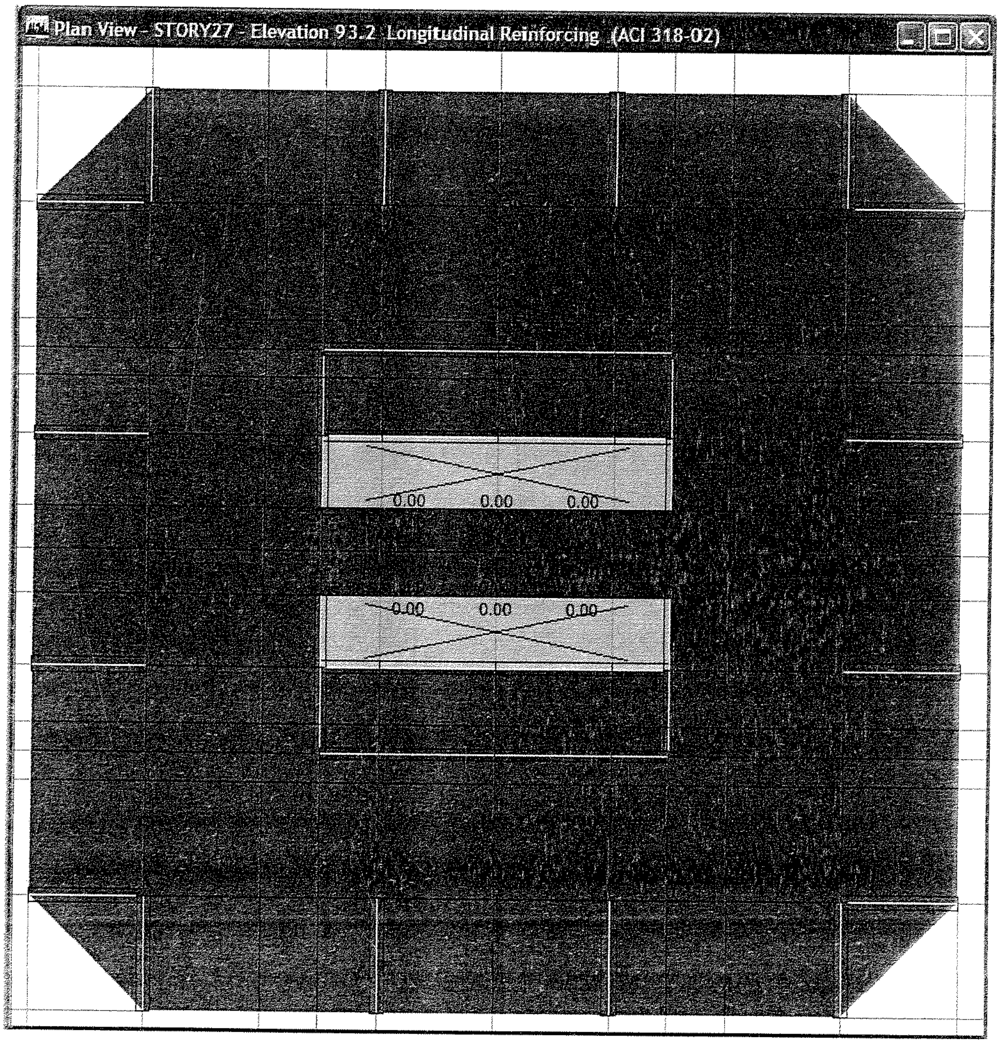
Note: As we mentioned before the program create automatically the load of combinations according to the design code ,and allow you from this from to remove or add any case of loading according to the design process, in this example we will use the load of combination which the program create

- After you add or remove any of the load of combination click OK (in our example we will use the default of the program)

3. Start Design

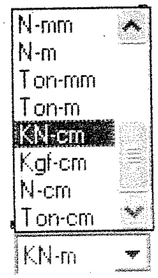
- Click the **Design menu** ⟶ **Concrete Frame Design** ⟶ **Start Design/Check of Structure**

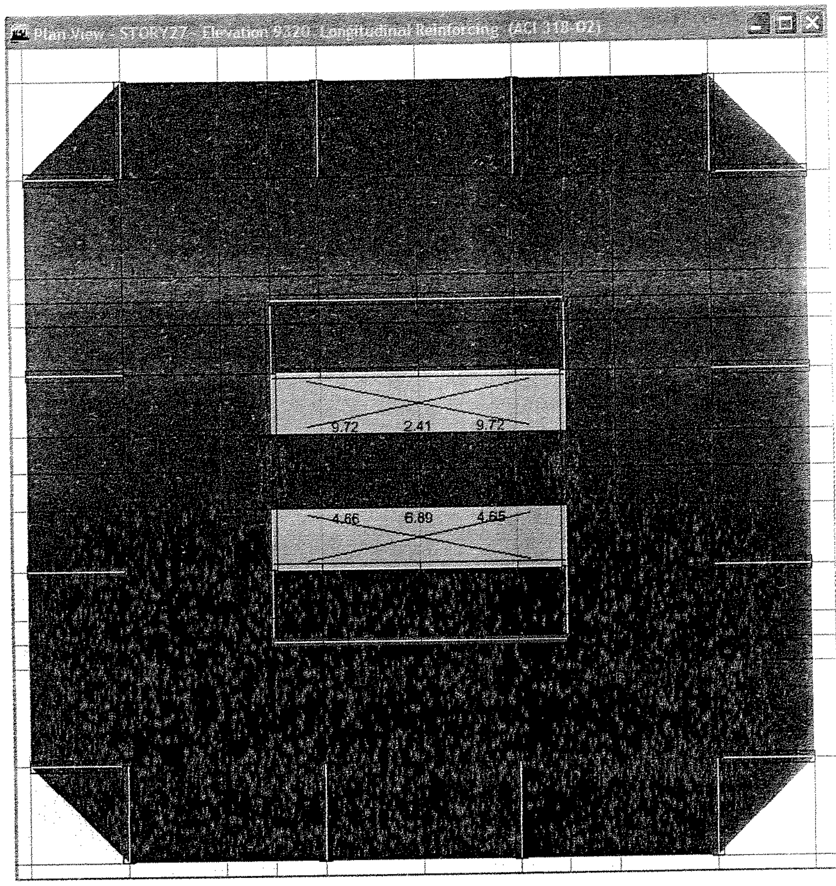




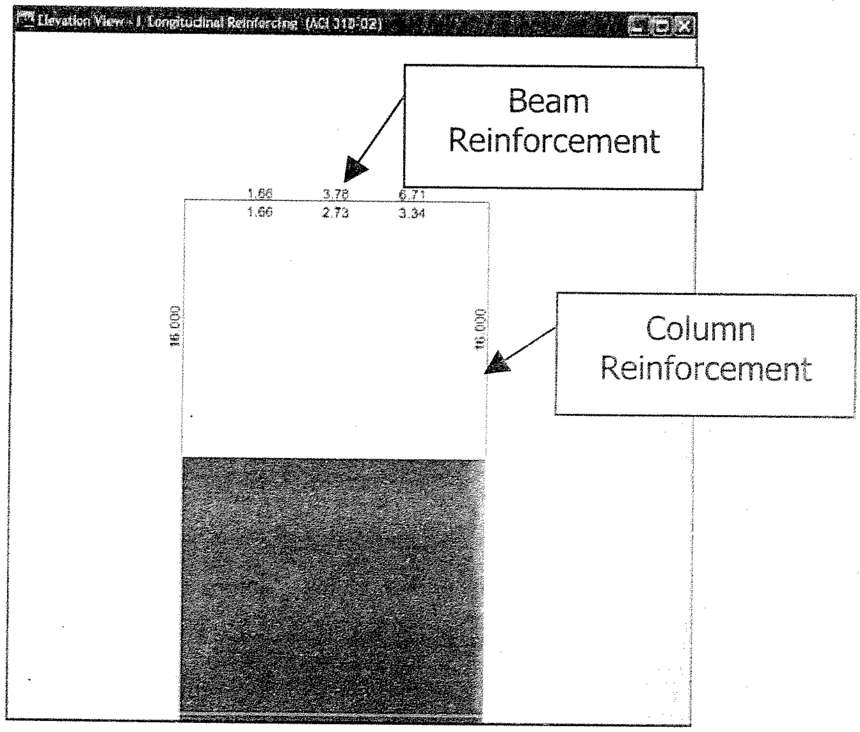
- All values are 0.00 because the unit which is used in program is m, Check the units of the model in the drop-down box in the

lower right-hand corner of the Etabs window , click the drop-down box to set the units
To KN-cm, thin the value of reinforcement will be Displayed in cm

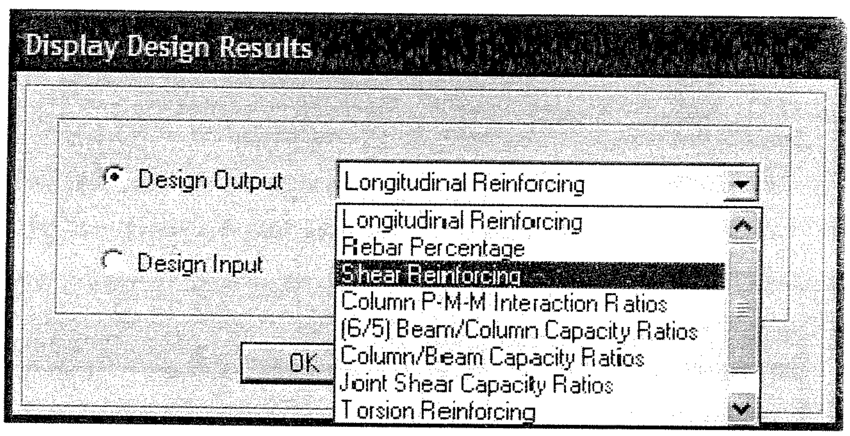




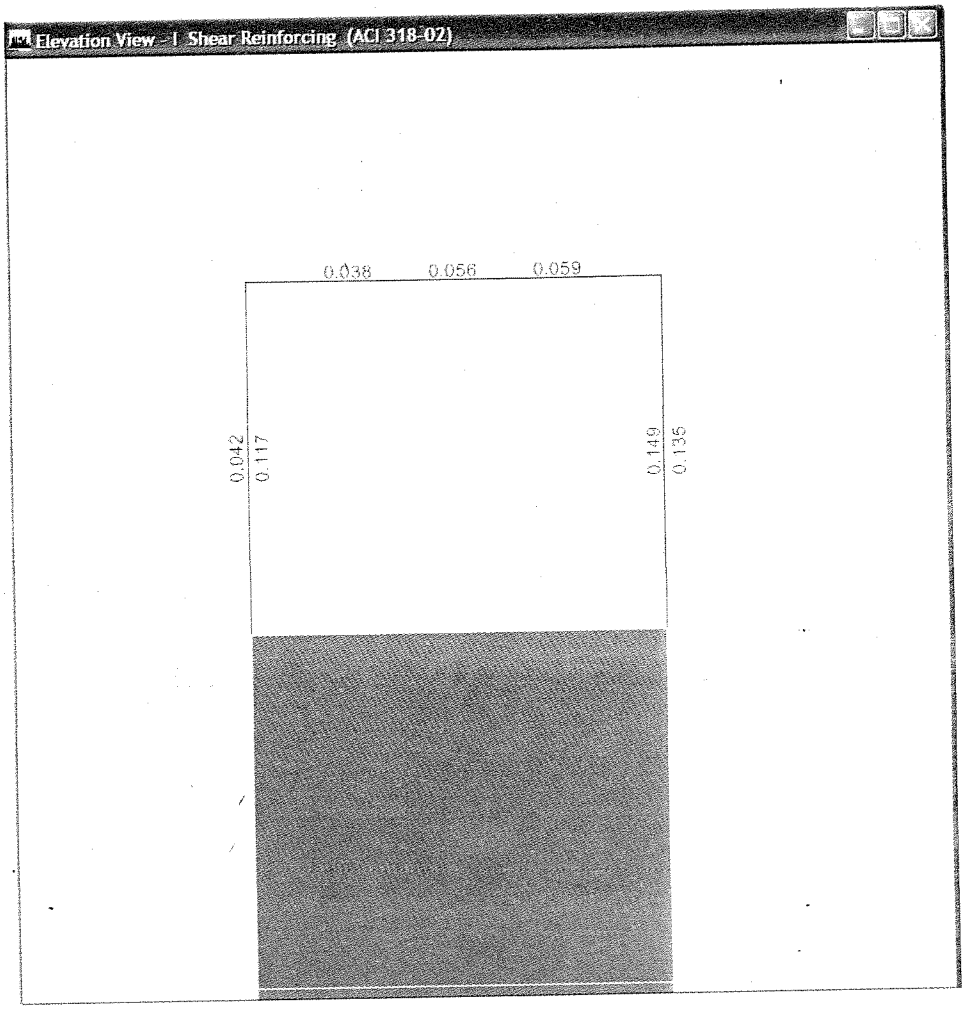
1. For beams the program give to you the reinforcement at 3 stations of the beam(end right,middle, and end left),but for the columns the program give to you one value of reinforcement



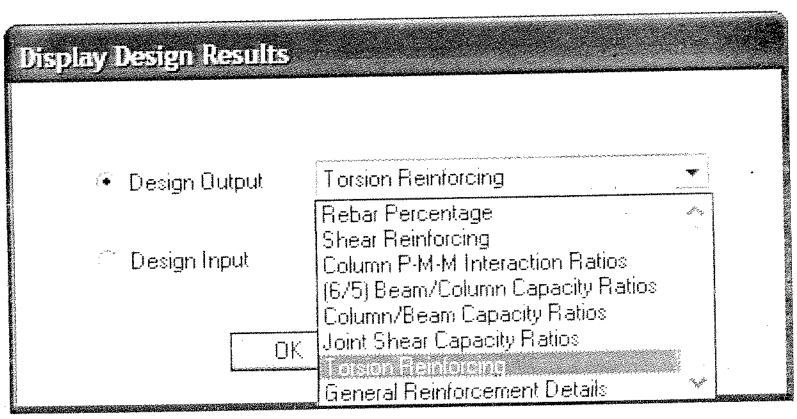
2. To display the shear reinforcement ratio for the shear ,Click the **Design menu** → **Concrete Frame Design** → **Display Design Result, thin** from the next form choose Shear Reinforcing



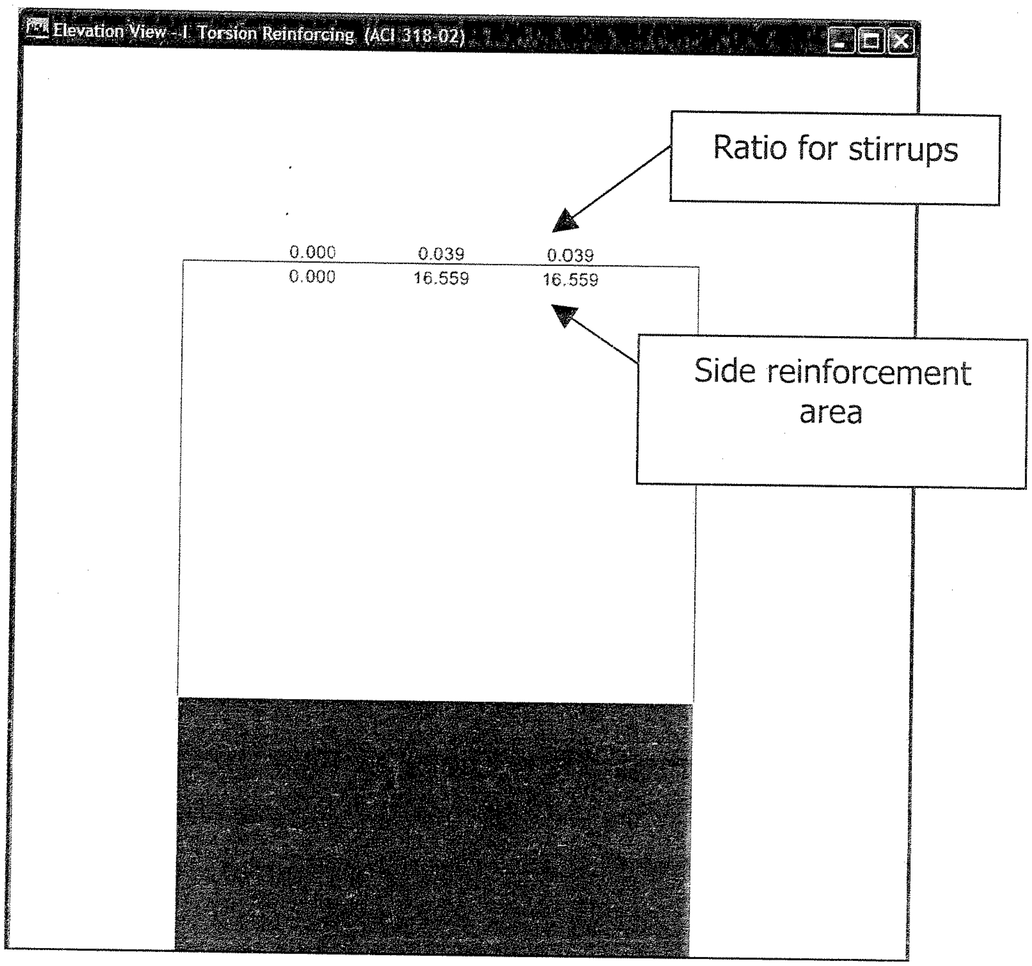
3. Thin the ratio for shear reinforcement will be displayed as shown in the fig.



4. To display the Torsion reinforcement ratio for the beam ,Click the **Design menu** ⟶ **Concrete Frame Design** ⟶ **Display Design Result, thin** from the next form choose Torsion Reinforcing

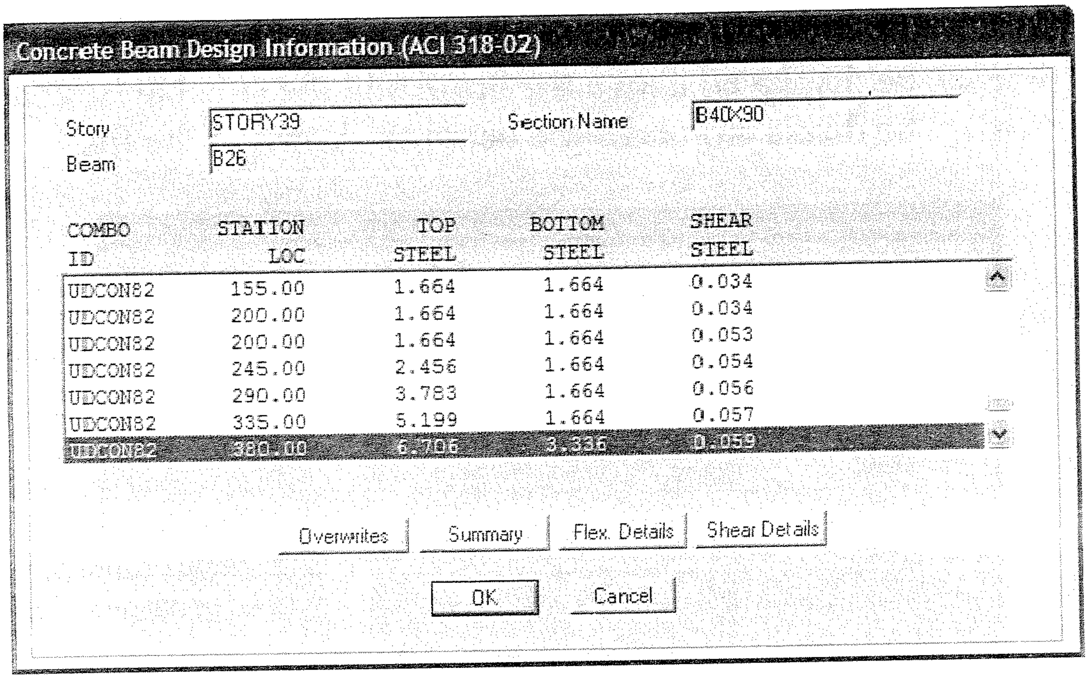


5. Thin the ratio for stirrups of torsion reinforcement will be displayed above the beam and the side bares area will be displayed under the beam as shown in the fig.



6. To display the Concrete Beam Design Information

   1. Choose the beam member by click the left button of the mouse
   2. Click the right button of the mouse
   3. The next form of the Concrete Beam Design Information will be displayed as in the fig.


Concrete Beam Design Information (ACI 318-02)

Story STORY39 Section Name B40X90
Beam B26

COMBO ID	STATION LOC	TOP STEEL	BOTTOM STEEL	SHEAR STEEL
UDCON82	155.00	1.664	1.664	0.034
UDCON82	200.00	1.664	1.664	0.034
UDCON82	200.00	1.664	1.664	0.053
UDCON82	245.00	2.456	1.664	0.054
UDCON82	290.00	3.783	1.664	0.056
UDCON82	335.00	5.199	1.664	0.057
UDCON82	380.00	6.706	3.336	0.059

Overwrites Summary Flex. Details Shear Details

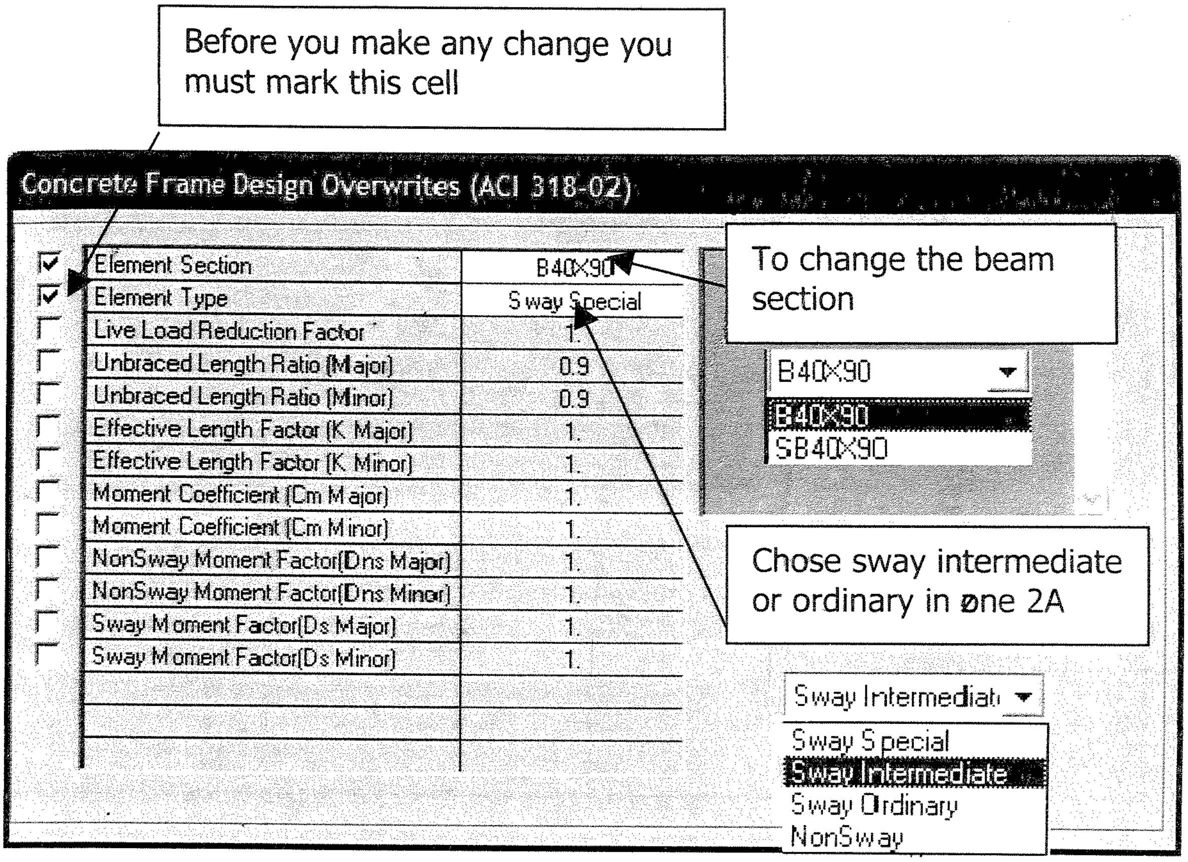
OK Cancel

4. from this form the program display all the case of loading and the corresponding area of steel which is required for the section according to the design data

7. The Concrete Beam Design Information has 4 important icons

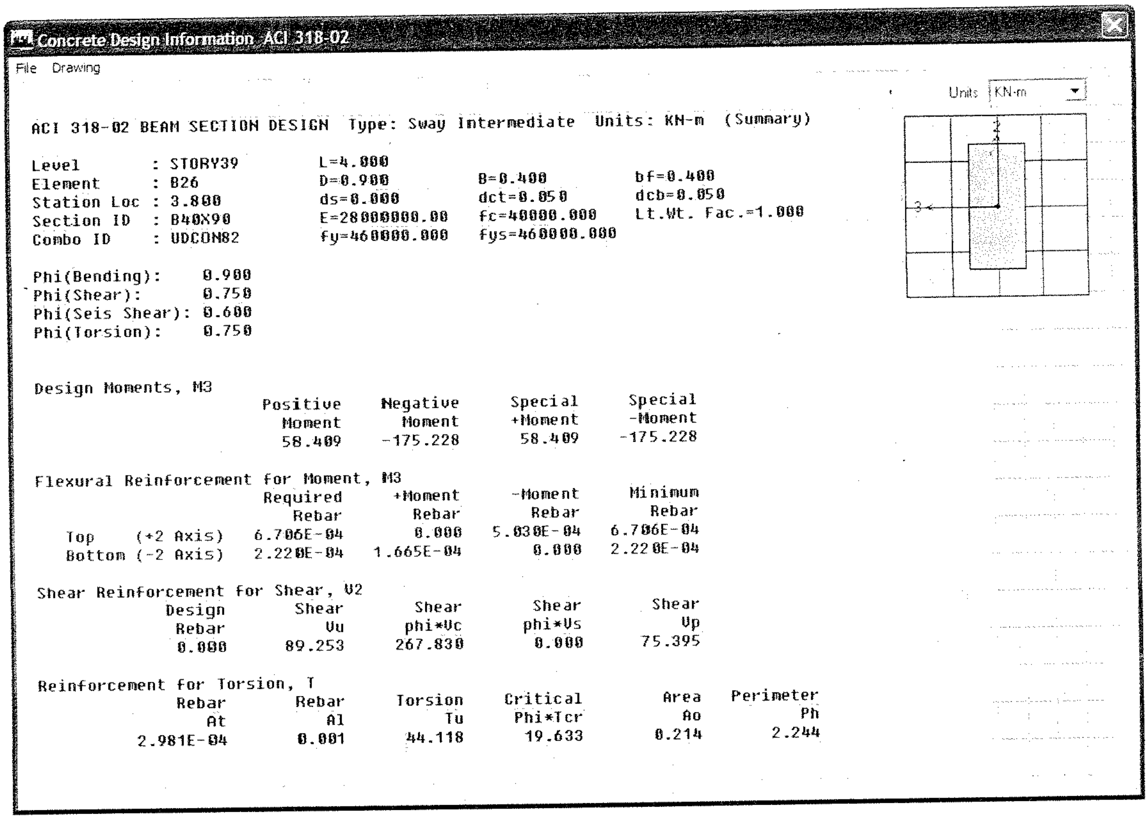
1. Overwrites :this icon to display the concrete Frame Design overwrites and allow you to

- change the element section
- change the element type
- change the information for the beam properties

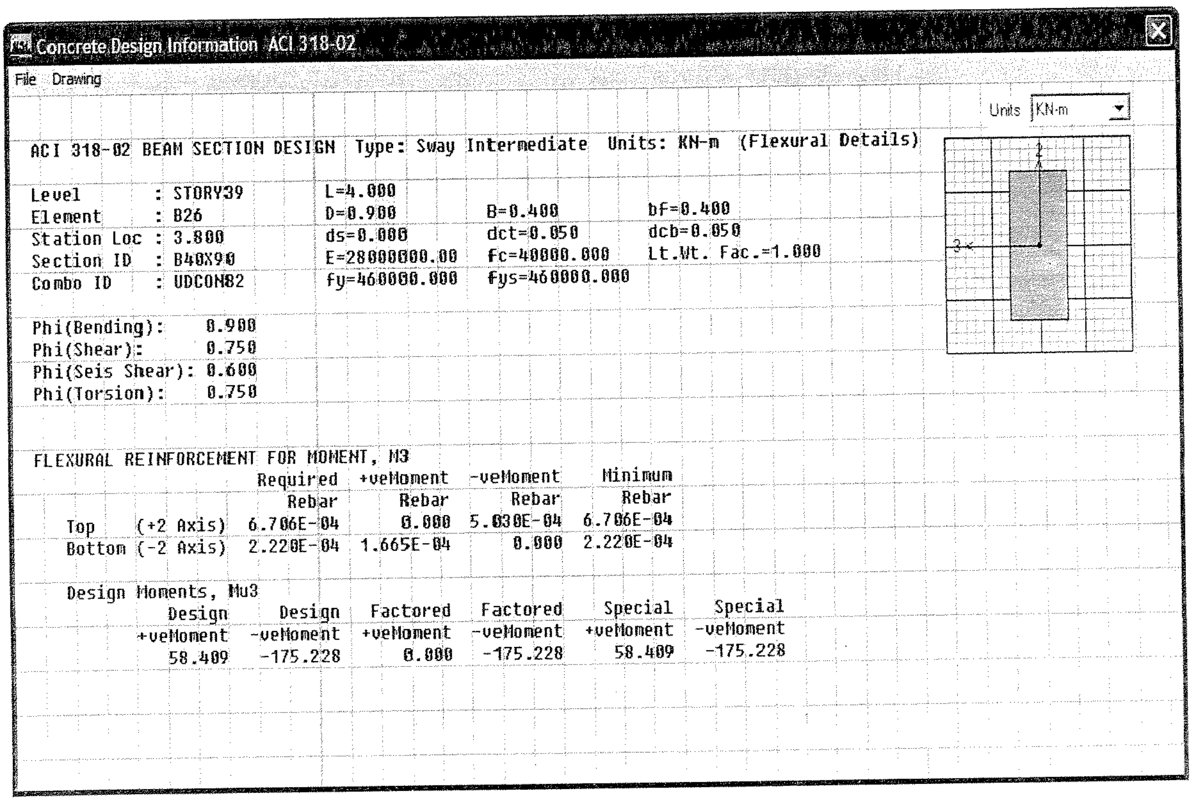


2. Summary :this icon to display the summary of the design of the beam

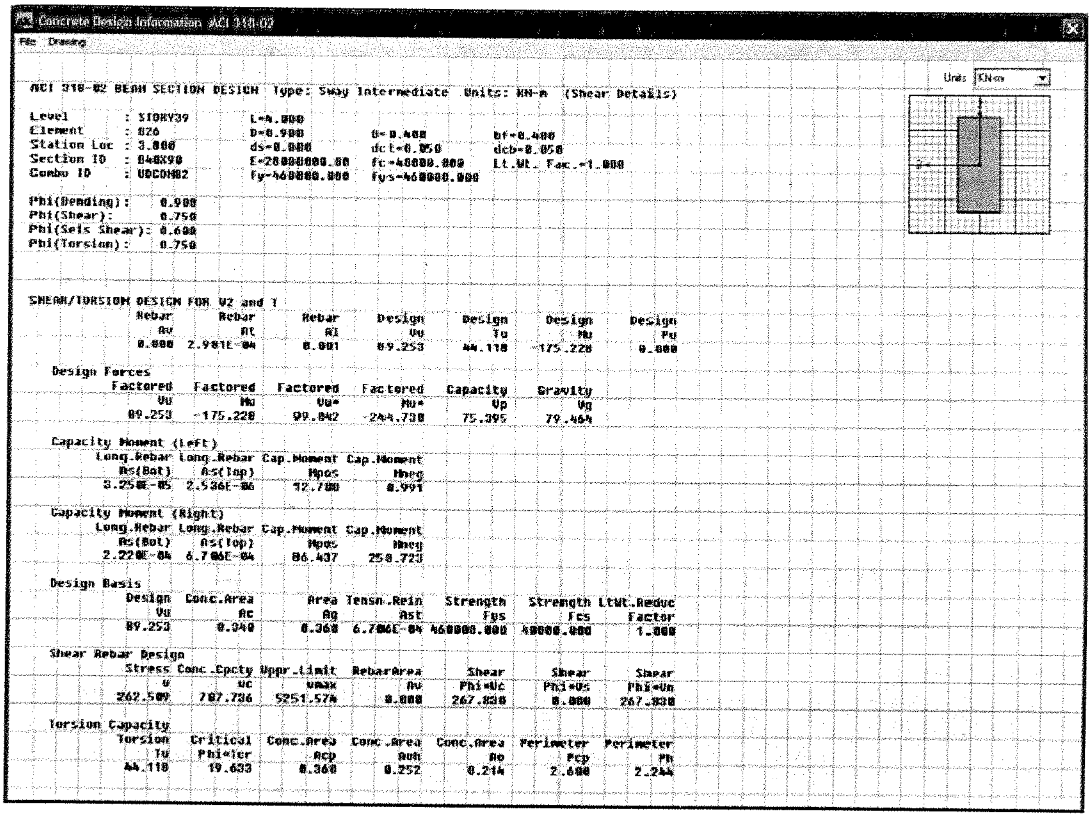
1. information of the beam
2. flexure Reinforcement
3. Shear Reinforcement
4. Torsion Reinforcement
5. Click Summary the next form will be displayed



3. Flex. Details :this icon to display the information of the design of the beam for flexure

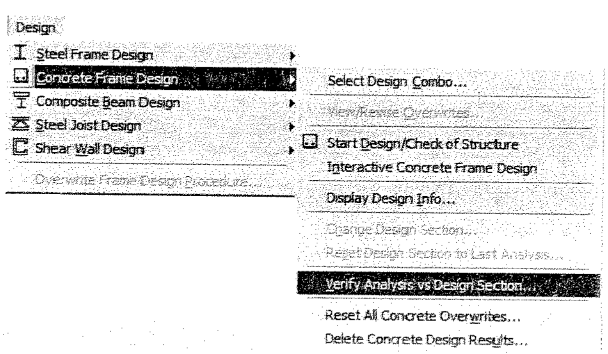


4. Shear Details :this icon to display the information of the design of the beam for Shear & Torsion



- **Important Note** : after you finish the process of the design ,sometimes you change some sections of the beams and columns to make the program reassign the frame element again according to the designed sections ,

8. Click the **Design menu** ⟶ **Concrete Frame Design** ⟶ **Verify Analysis vs. Design Section……** ,Automatically the program reassign all the frame sections in the next run according to the new sections of design



- **Shear Wall Design**

- There are there method for design shear walls:

    1. Simplified C and T Section
    2. Uniform Reinforcing Pier Section
    3. General Reinforcing Pier Section

- **Simplified C and T Section :**
    - this method is very simple method for design ,the simplified method can only be used for design ,not for checking.(not recommended method)

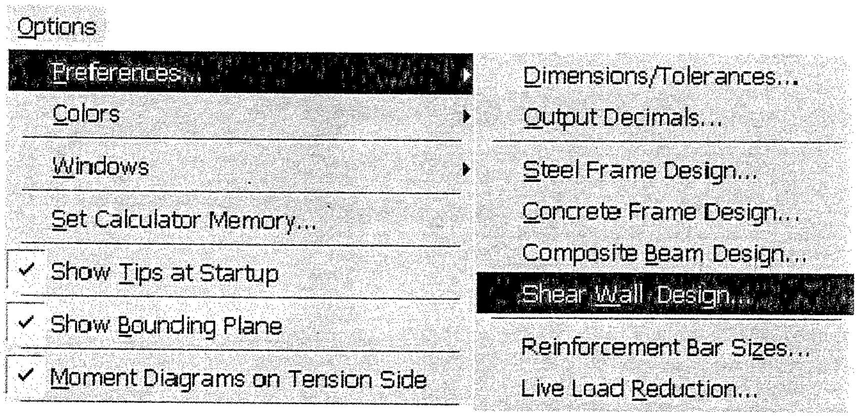
- **Uniform Reinforcing Pier Section:**
    - this method is the most famous method for design; this method can be used for both design and checking. (recommended method)

- **General Reinforcing Pier Section :**
    - this method is the most accurate method for design but it is need more efforts from the user because if you use this method you must draw all the walls section and its reinforcement of the all walls

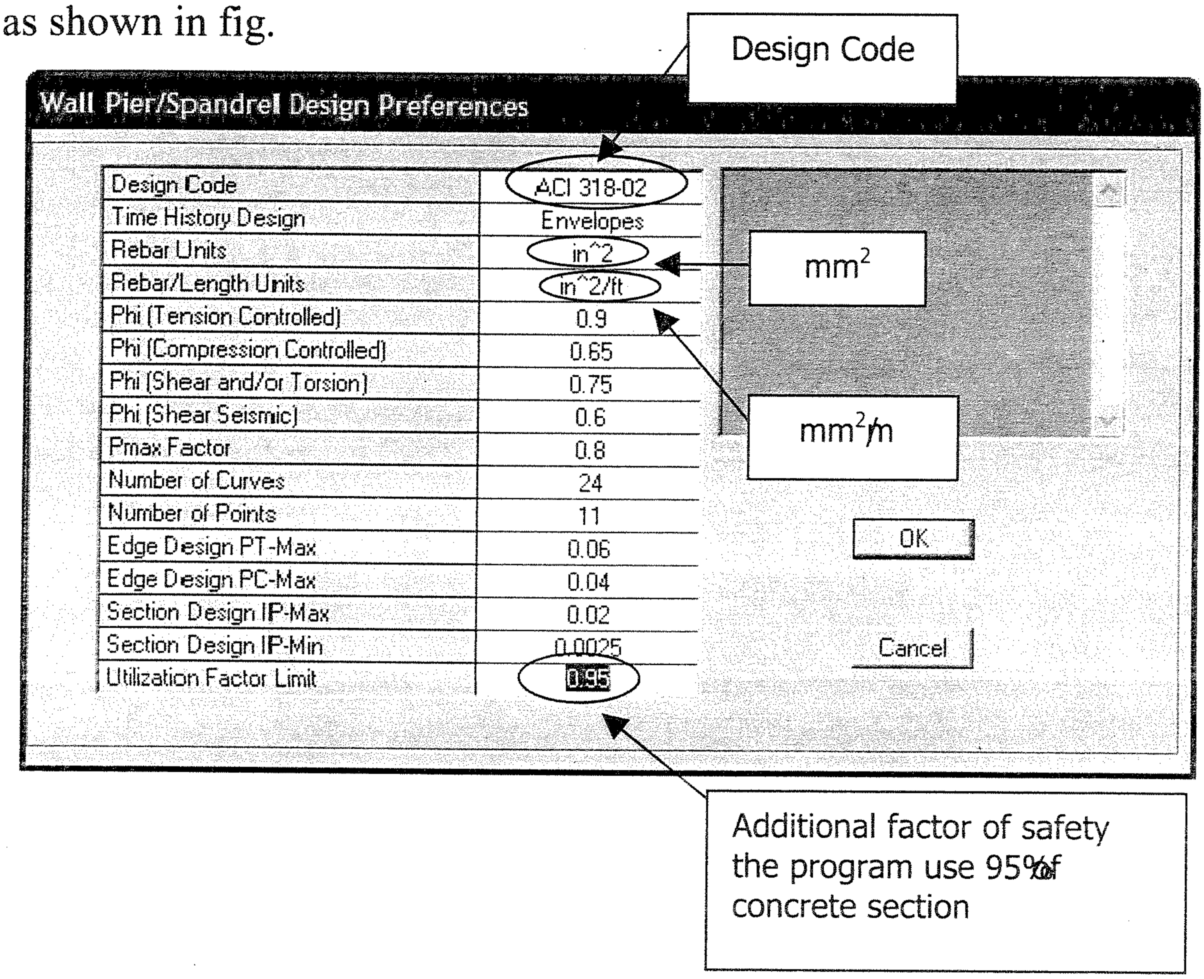
Note: in our example we will not discuss the **Simplified method**, we will explain how to use the **Uniform Reinforcing Pier Section**, and in the section of section designer we will explain how to use the **General Reinforcing Pier Section**

1. Adjust the design code and Data of Design

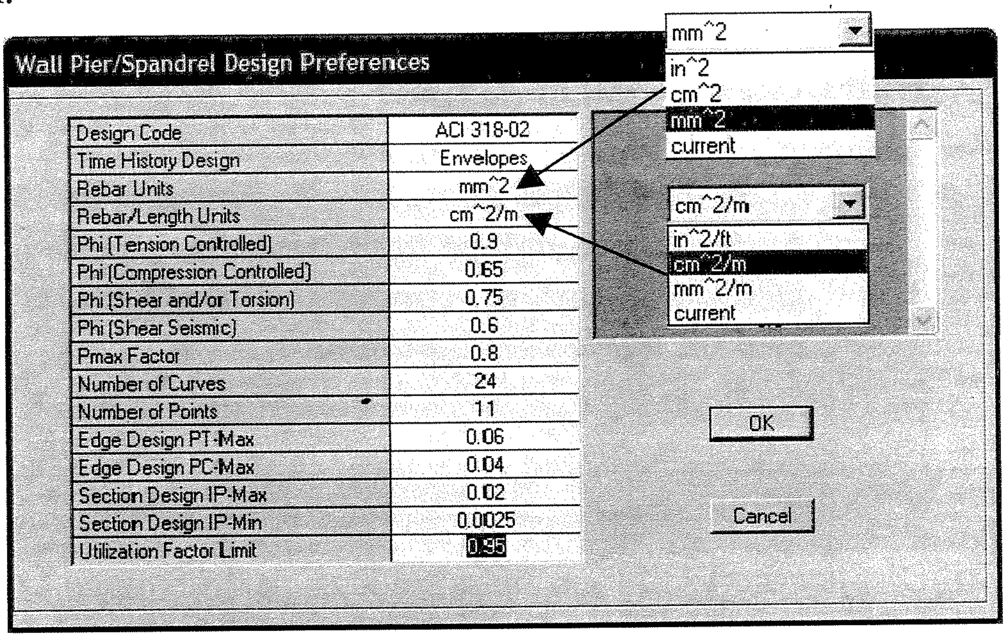
- Click the **Option menu** → **Preferences** → **Shear Wall Design**



This will Display the Wall Pier/Spandrel Design Preference form as shown in fig.

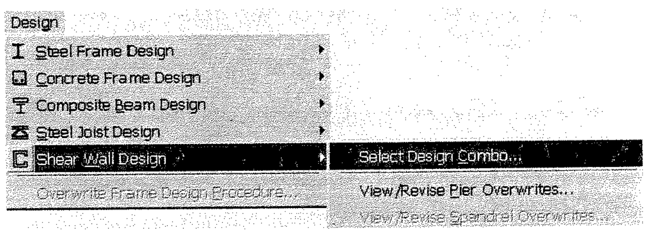


From this form, you can adjust the code and another data of design as shown.

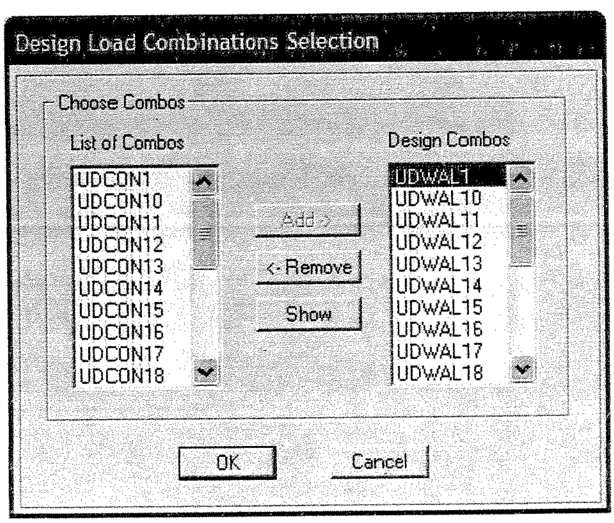


2. select of the Load of Combinations

- Click the **Design menu** ⟶ **Shear Wall Design** ⟶ **Select Design Combo**



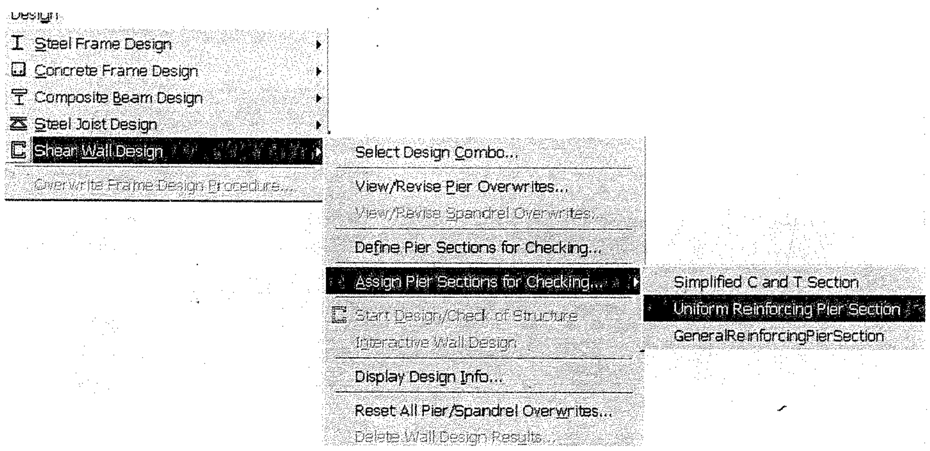
- Thin the next form will be displayed



- After you add or remove any of the load of combinations for design click OK

4. **Choose Method of design**

- Click the **Design menu** → **Shear Wall Design** → **Assign Pier Sections for Checking** → **Choose the method of design; (choose Uniform Reinforcing Pier Section as we mention before.)**

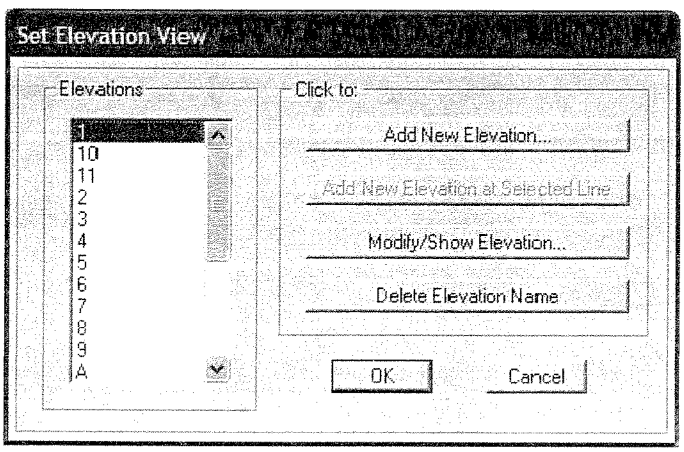


5. **set the Elevation view of the wall**

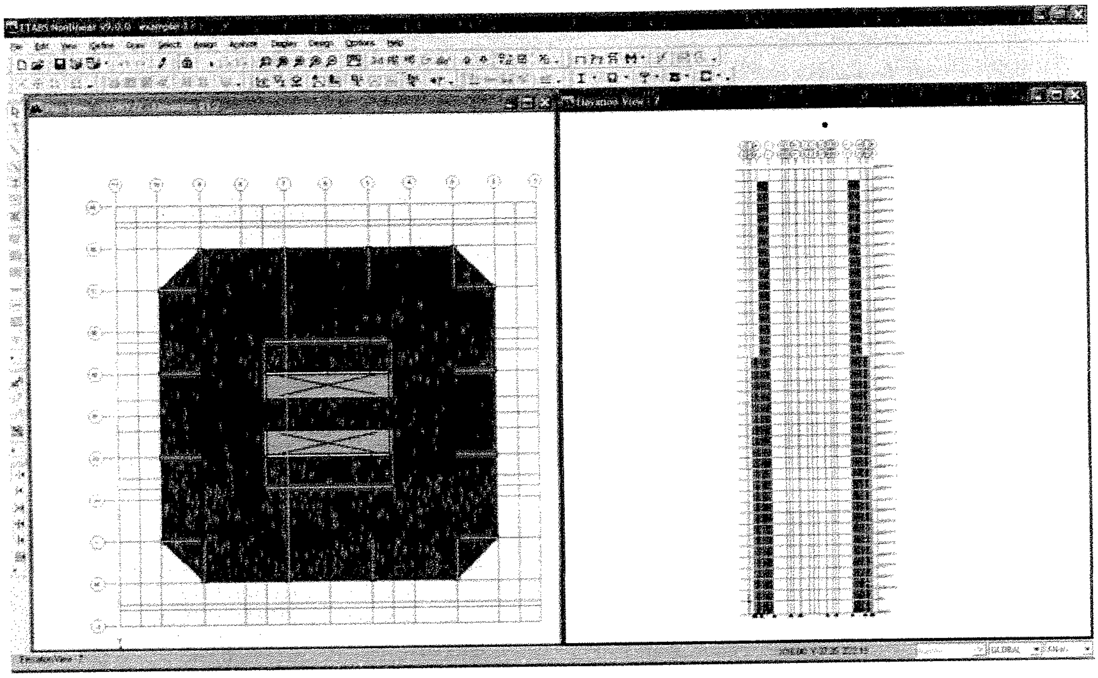
Note: the program display the result of design for the wall in the elevation views so that we will display the elevation view of wall to check the result of the design

- activate the window of 3D View
- Click the menu of **View** → **Set Elevation View**

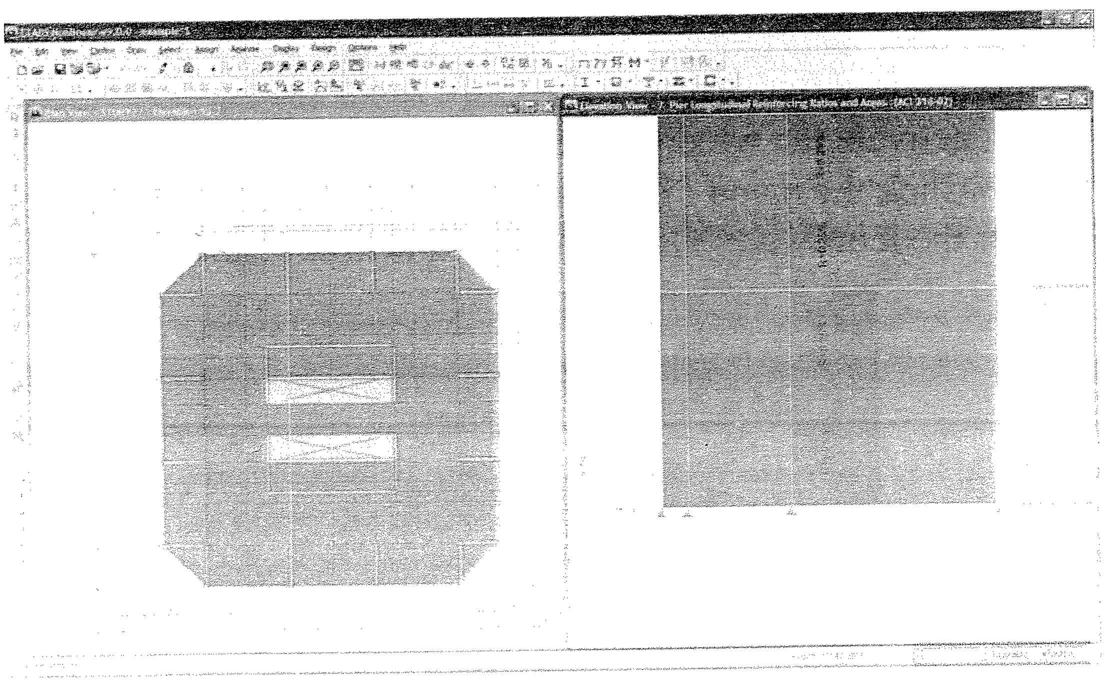
Or the icon ele, Thin the next form will be displayed



- From this form choose elevation 7 (you can choose any elevation view, but this for our example),thin click ok
- The screen of the program will be as in the next fig.



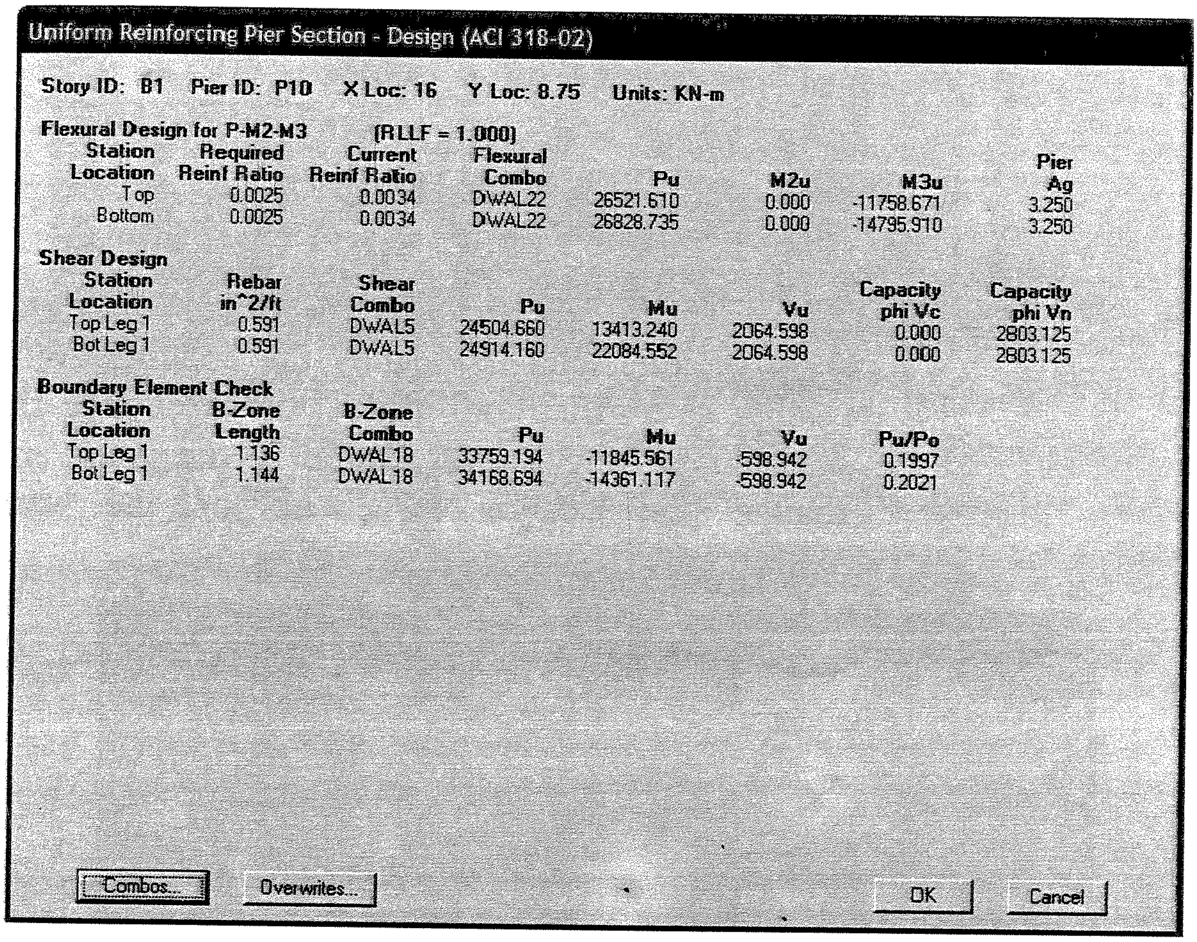
6. **Start Design**
   - Click the **Design menu** ——► **Shear wall Design** ——► **Start Design/Check of Structure**



- From previous figure we see that the program display the ratio of reinforcement required for this wall

7. To display the Shear Wall Design Information

    1. Choose the Wall by click the left button of the mouse
    2. Click the right button of the mouse
    3. The next form of the Wall Design Information will be displayed as in the fig.



Uniform Reinforcing Pier Section - Design (ACI 318-02)

Story ID: B1 Pier ID: P10 X Loc: 16 Y Loc: 8.75 Units: KN-m

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

Station Location	Required Reinf Ratio	Current Reinf Ratio	Flexural Combo	Pu	M2u	M3u	Pier Ag
Top	0.0025	0.0034	DWAL22	26521.610	0.000	-11758.671	3.250
Bottom	0.0025	0.0034	DWAL22	26828.735	0.000	-14795.910	3.250

Shear Design

Station Location	Rebar in^2/ft	Shear Combo	Pu	Mu	Vu	Capacity phi Vc	Capacity phi Vn
Top Leg 1	0.591	DWAL5	24504.660	13413.240	2064.598	0.000	2803.125
Bot Leg 1	0.591	DWAL5	24914.160	22084.552	2064.598	0.000	2803.125

Boundary Element Check

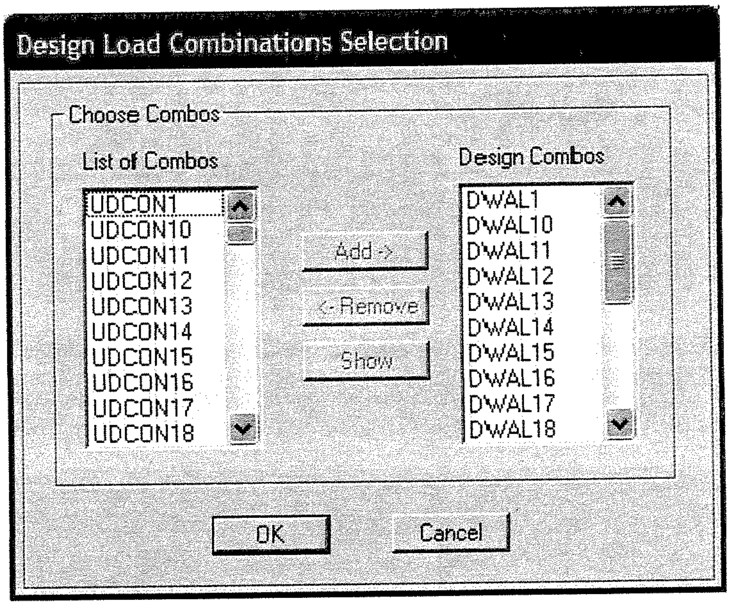
Station Location	B-Zone Length	B-Zone Combo	Pu	Mu	Vu	Pu/Po
Top Leg 1	1.136	DWAL18	33759.194	-11845.561	-598.942	0.1997
Bot Leg 1	1.144	DWAL18	34168.694	-14361.117	-598.942	0.2021

Combos... Overwrites... OK Cancel

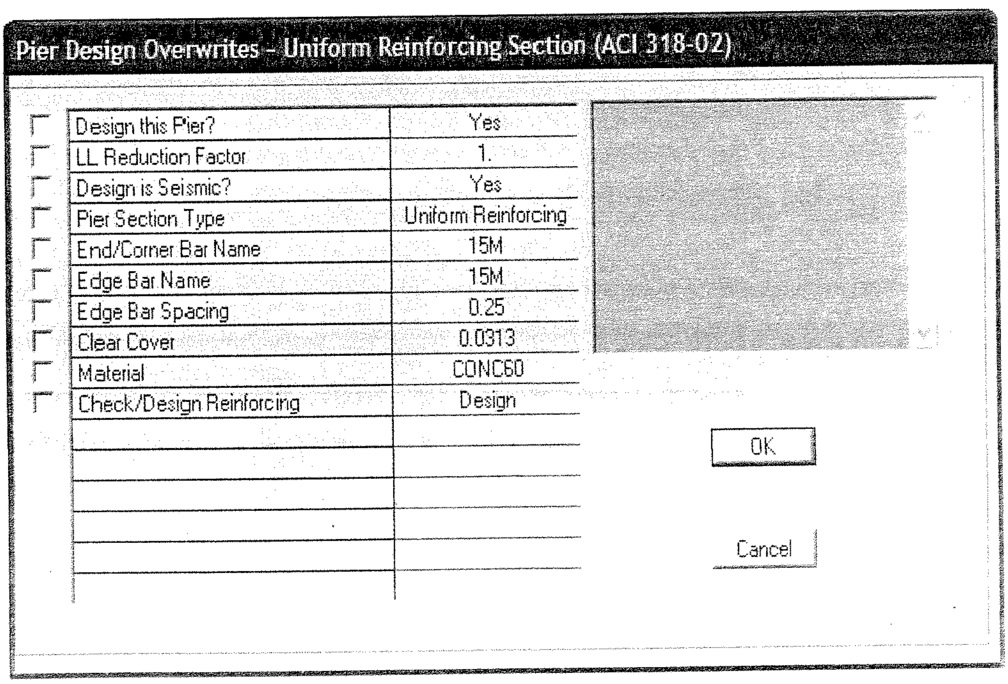
4. From this form the program display the design result for flexural design, shear design, and boundary element check (boundary element check is neglected in some codes.)

8. The Wall Design Information has 2 icons

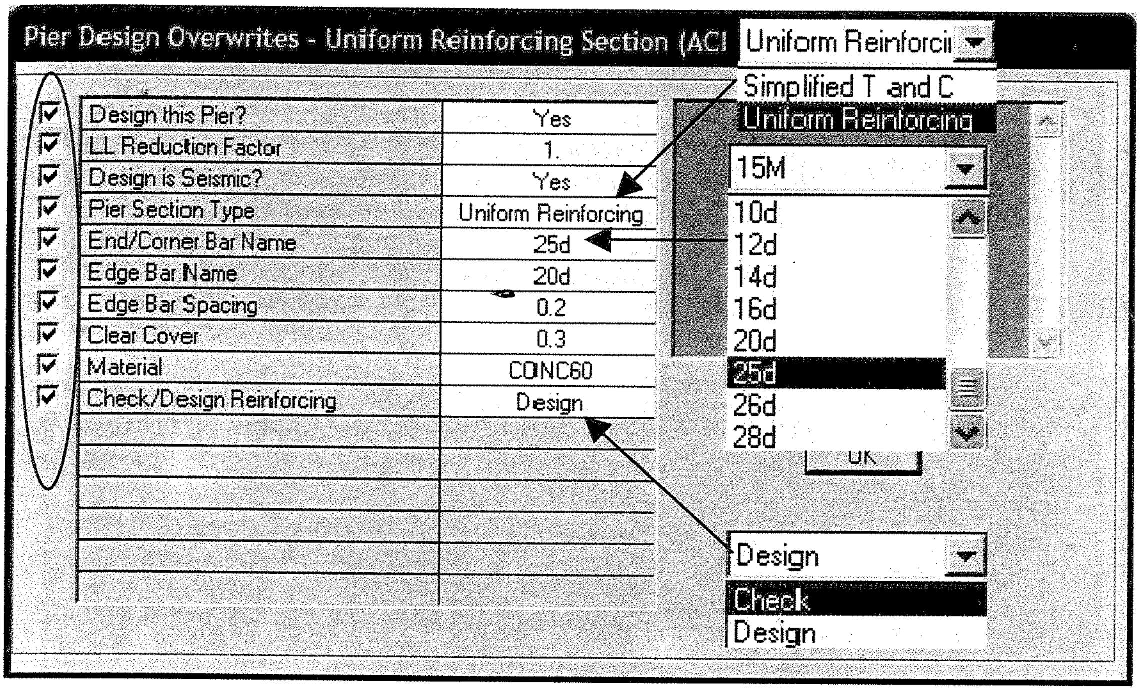
1. Combos... :this icon allow you to change the load of combination for the design



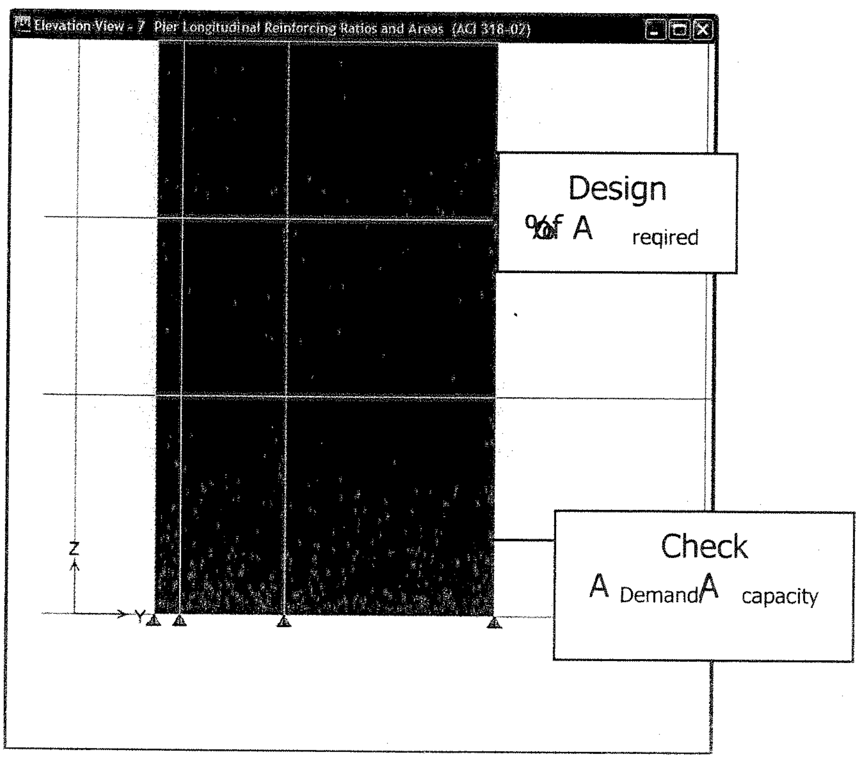
2. Overwrites :this icon to display the concrete Wall Design overwrites and allow you to change the design type, the reinforcement in the wall, and change the design to check of the wall reinforcement



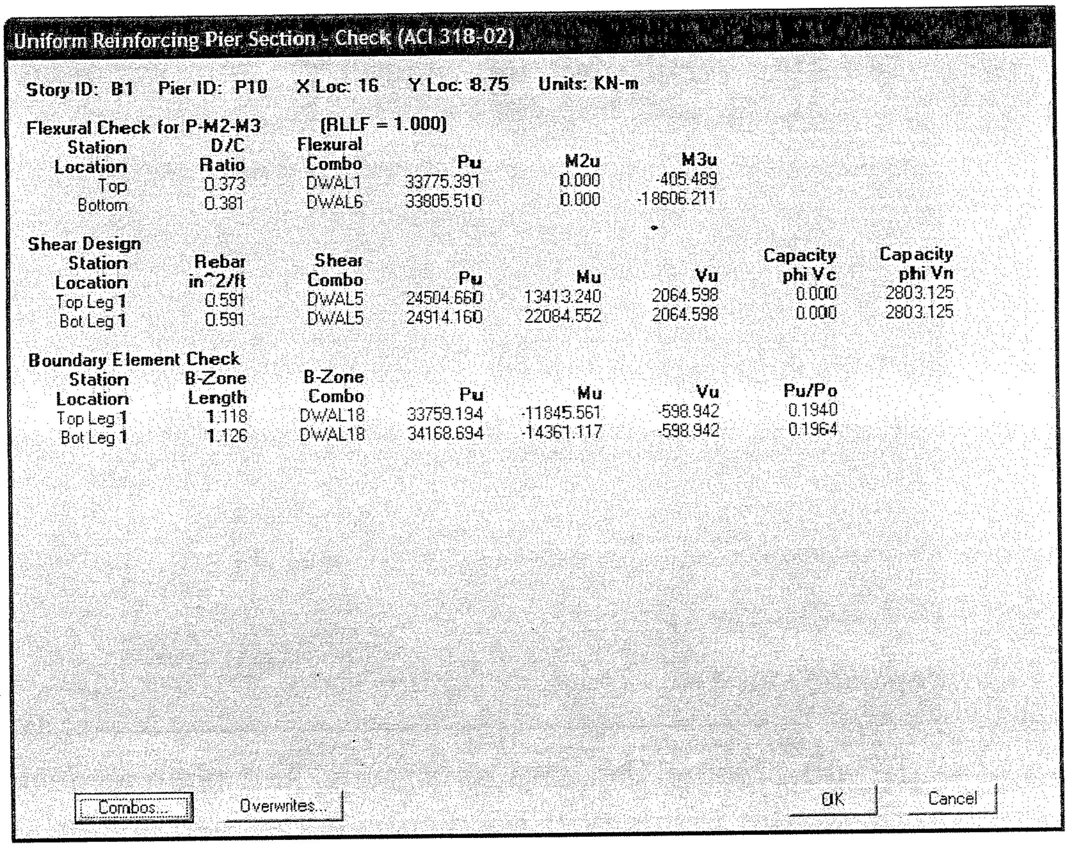
- To make any change in the previous form you must check the cell before the change thin begin the change to the checked cells



- If you change the data as shown ,and change design to check the result will be displayed as percentage of the safety for the wall(steel Demand/capacity of the section)

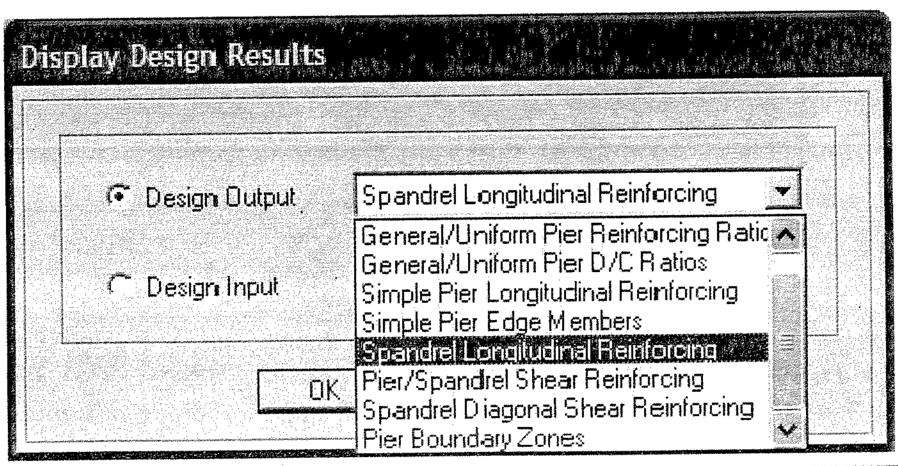


- After you make check repeat step 7 again to see check the result form



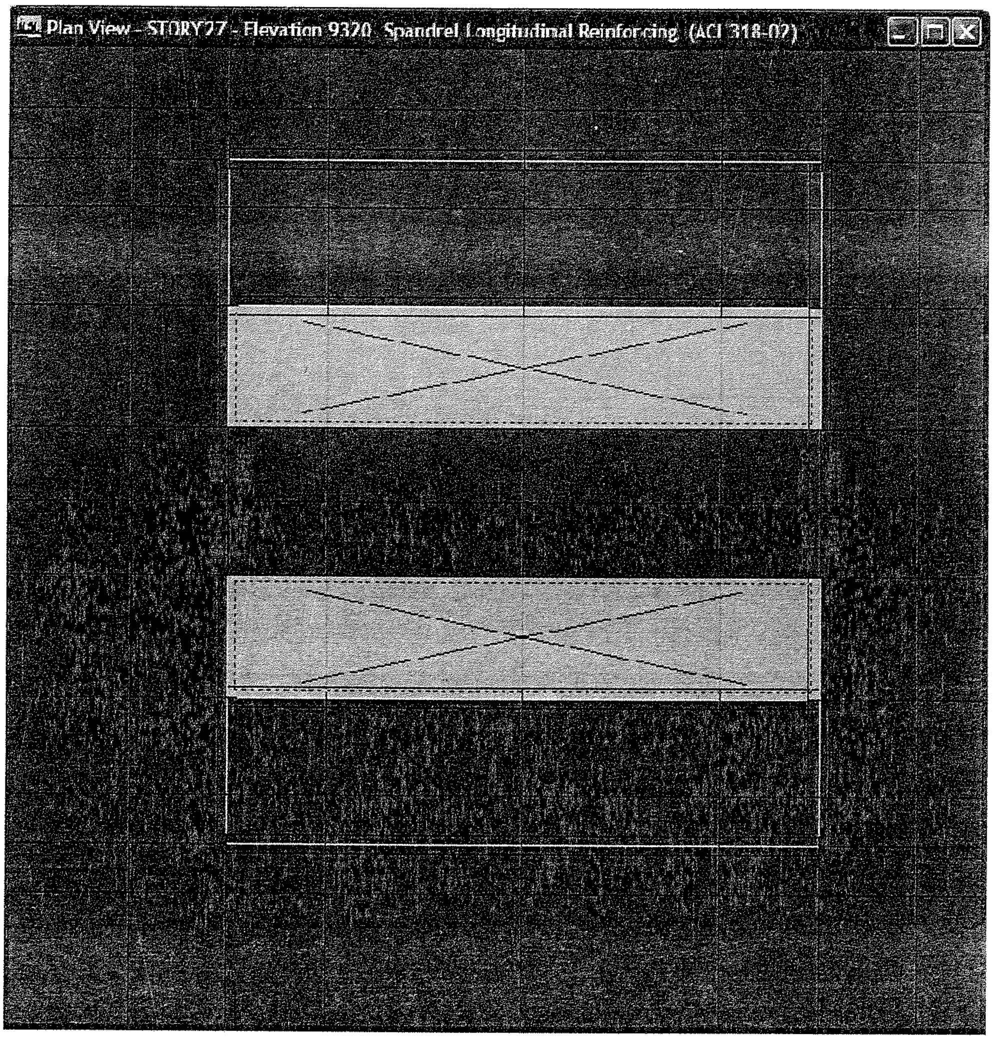
- We can notice the difference in flexure check the program display D/C Ratio not the steel required

9. To display the Spandrel beam reinforcement ,Click the **Design menu** ——► **Shear Wall Design** ——► **Display Design Info...,** thin from the next form choose **Spandrel Longitudinal Reinforcing**



Note: don't forget to change the unit of the model to KN/cm as we mentioned before.

- The result of the spandrel reinforcement will be displayed

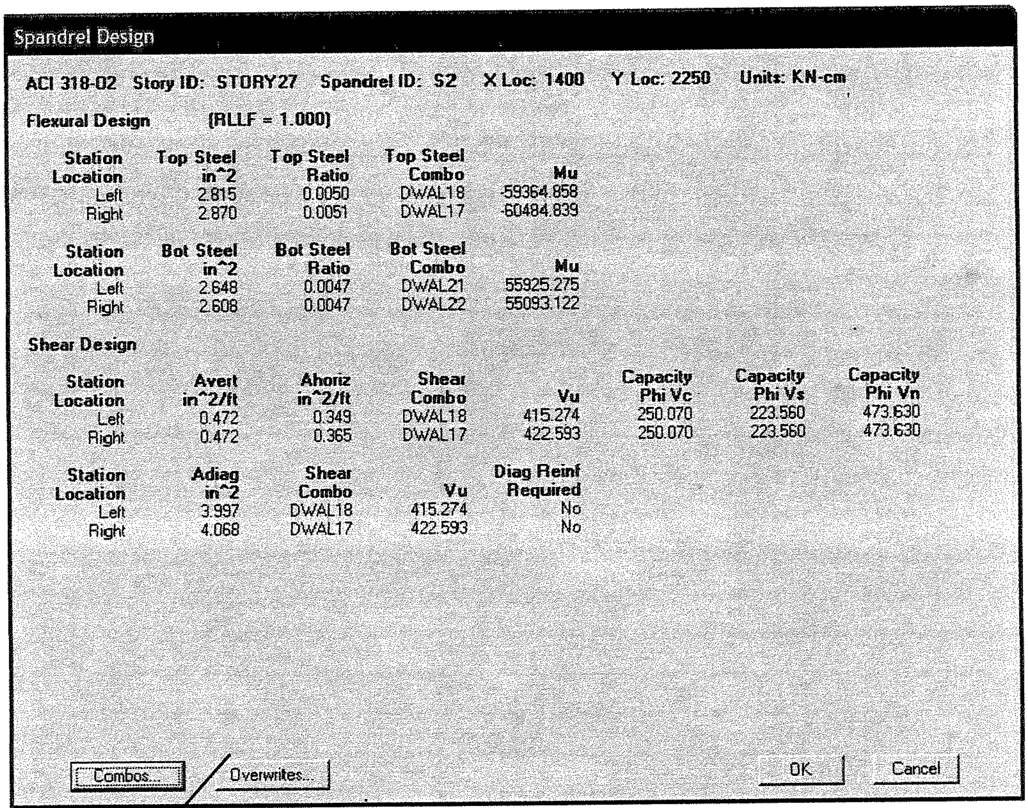


10. To display the Spandrel Beam Design Information

    1. Choose the beam member by click the left button of the mouse

    2. Click the right button of the mouse

    3. The next form of the Concrete S. Beam Design Information will be displayed as in the fig.



Spandrel Design

ACI 318-02 Story ID: STORY27 Spandrel ID: S2 X Loc: 1400 Y Loc: 2250 Units: KN-cm

Flexural Design (RLLF = 1.000)

Station Location	Top Steel in^2	Top Steel Ratio	Top Steel Combo	Mu
Left	2.815	0.0050	DWAL18	-59364.858
Right	2.870	0.0051	DWAL17	-60484.839

Station Location	Bot Steel in^2	Bot Steel Ratio	Bot Steel Combo	Mu
Left	2.648	0.0047	DWAL21	55925.275
Right	2.608	0.0047	DWAL22	55093.122

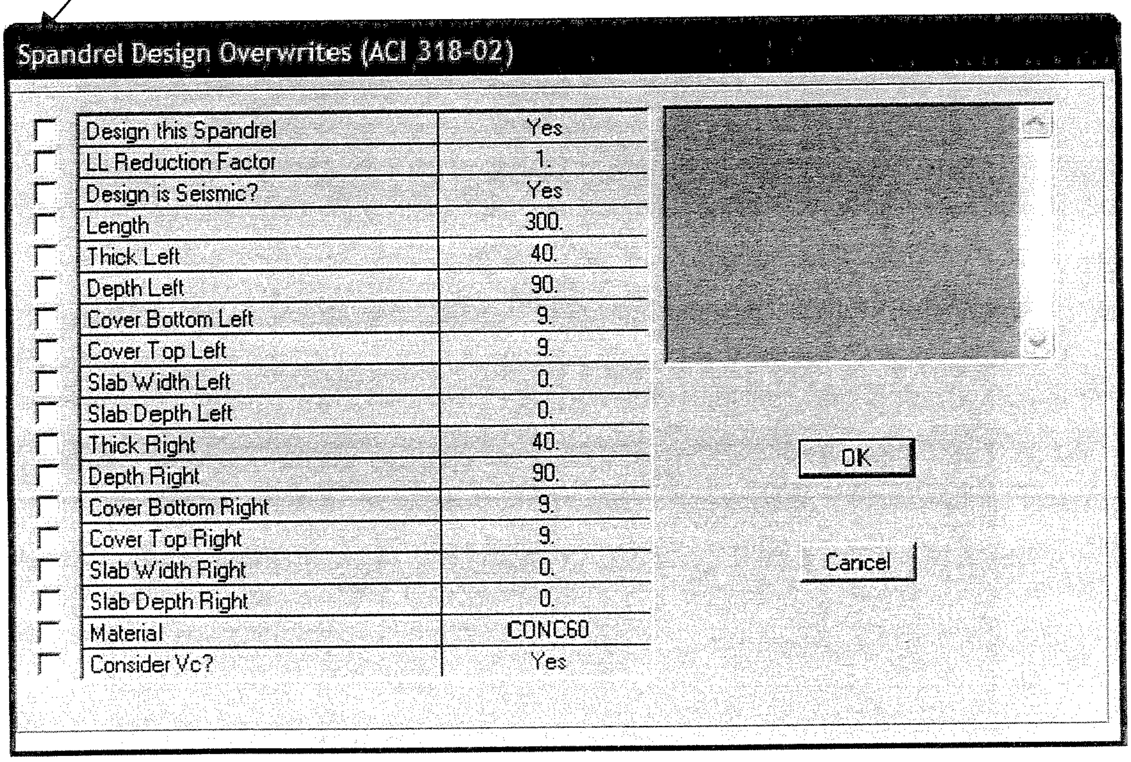
Shear Design

Station Location	Avert in^2/ft	Ahoriz in^2/ft	Shear Combo	Vu	Capacity Phi Vc	Capacity Phi Vs	Capacity Phi Vn
Left	0.472	0.349	DWAL18	415.274	250.070	223.560	473.630
Right	0.472	0.365	DWAL17	422.593	250.070	223.560	473.630

Station Location	Adiag in^2	Shear Combo	Vu	Diag Reinf Required
Left	3.997	DWAL18	415.274	No
Right	4.068	DWAL17	422.593	No

Combos... Overwrites... OK Cancel

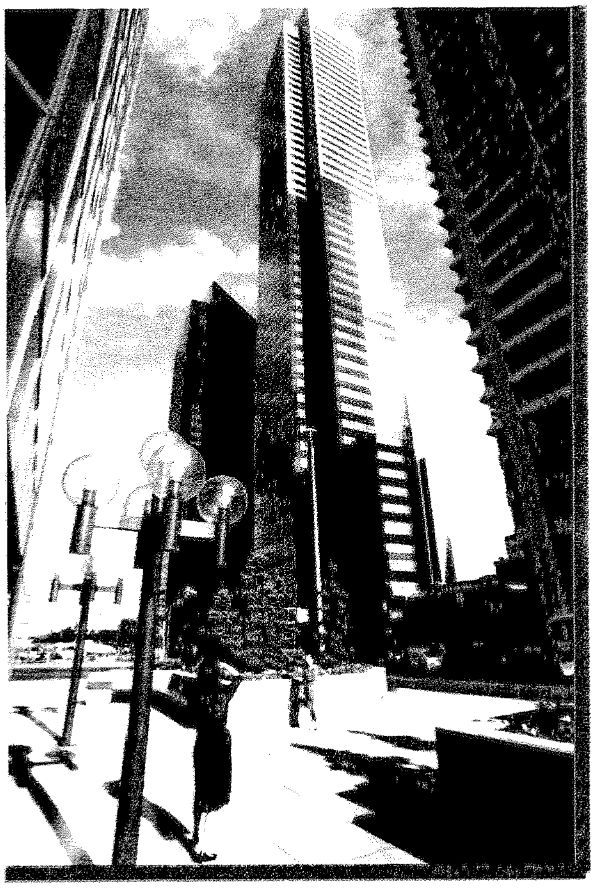
- As we explained before you can change the data for the design using Overwrites icon which is displayed in the spandrel Design Overwrites form



Spandrel Design Overwrites (ACI 318-02)

Design this Spandrel	Yes
LL Reduction Factor	1.
Design is Seismic?	Yes
Length	300.
Thick Left	40.
Depth Left	90.
Cover Bottom Left	9.
Cover Top Left	9.
Slab Width Left	0.
Slab Depth Left	0.
Thick Right	40.
Depth Right	90.
Cover Bottom Right	9.
Cover Top Right	9.
Slab Width Right	0.
Slab Depth Right	0.
Material	CONC60
Consider Vc?	Yes

OK Cancel



HOW TO MODEL AND DESIGN
HIGH RISE BUILDING USING
ETABS Program

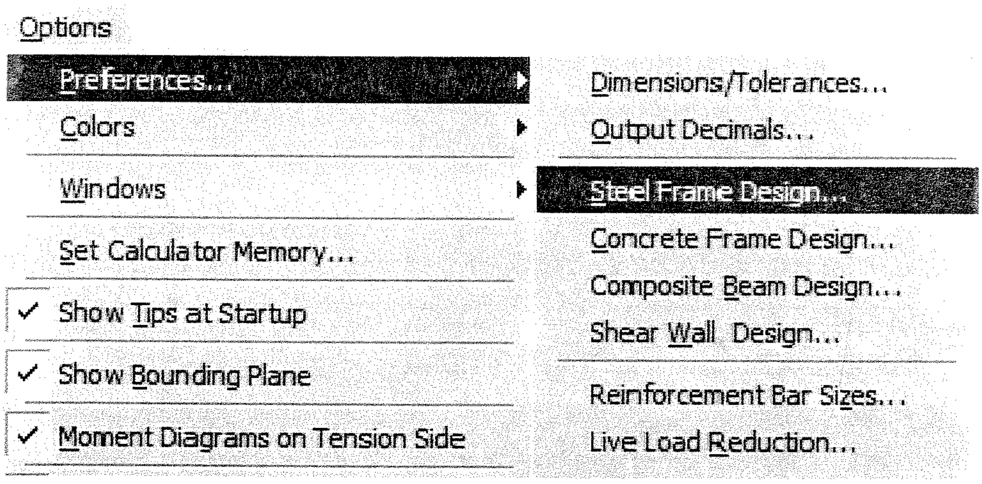
Steel Design

Chapter 4

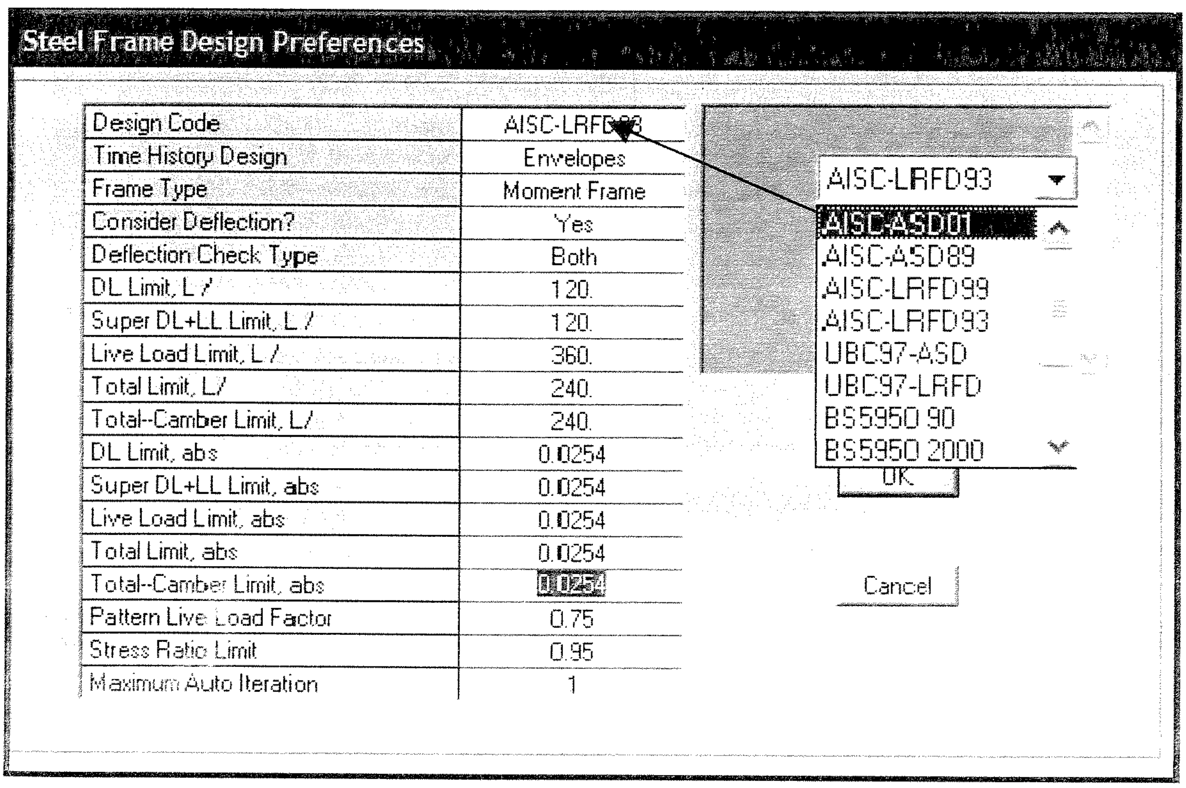
Steel Design:

1. Adjust the design code and Data of Design

- Click the **Option menu** ──► **Preferences** ──► **Steel Frame Design**



This will Display the Steel Frame Design Preference form as shown in fig.

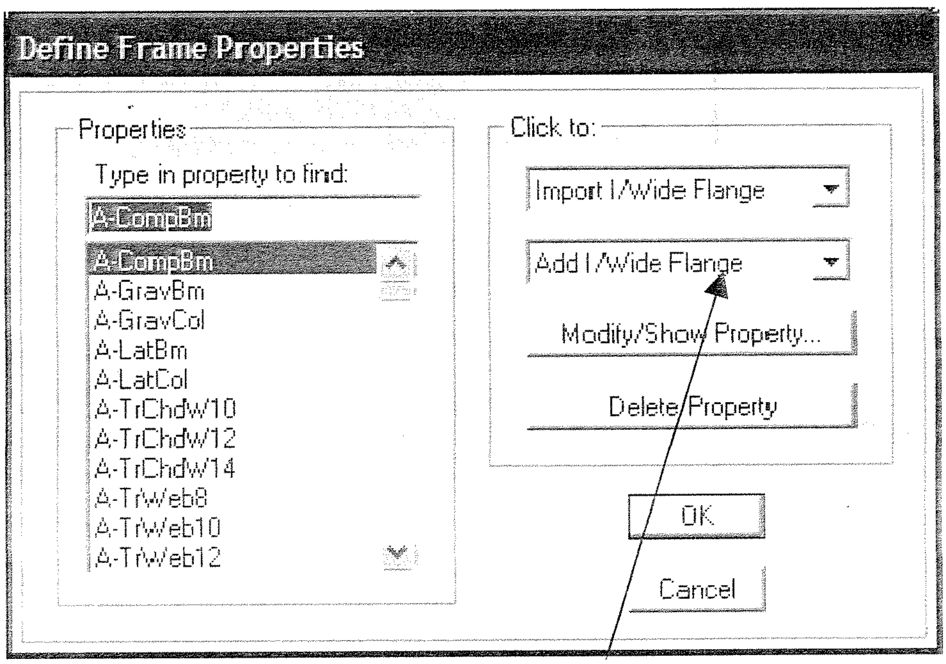


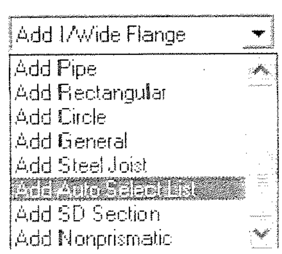
1. From this form, you can adjust the code and another data of design as shown in pervious figure.
2. From the Design Code drop-down list choose AISC-ASD01 Code or any another Code you want your design based in.

2. Define an Auto Select Section List

- The program has several built-in auto select section lists, and you can add in built up section to create an **auto select section**.

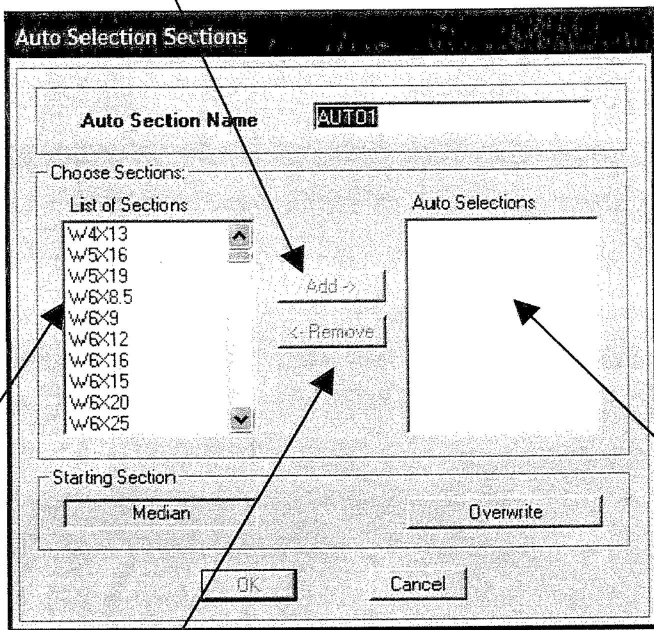
- Click the **Define menu** ———► **Frame Sections** command, which will display the Define Frame Properties form shown in the next Figure.





3. From the Shown drop-down list choose **Add Auto Select List** thin the next figure of **Auto Select Section** Form.

Click this button to add the sections from the list of the sections



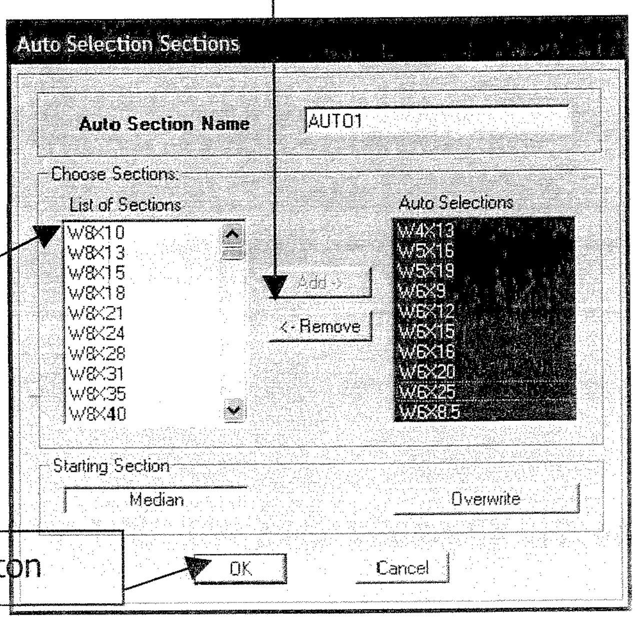
The all available section of the program (built in section and the section you defined to the program

Add here the list of the sections which the program will use in design

Click this button to remove any section from Auto Selections List

2-Click Add button

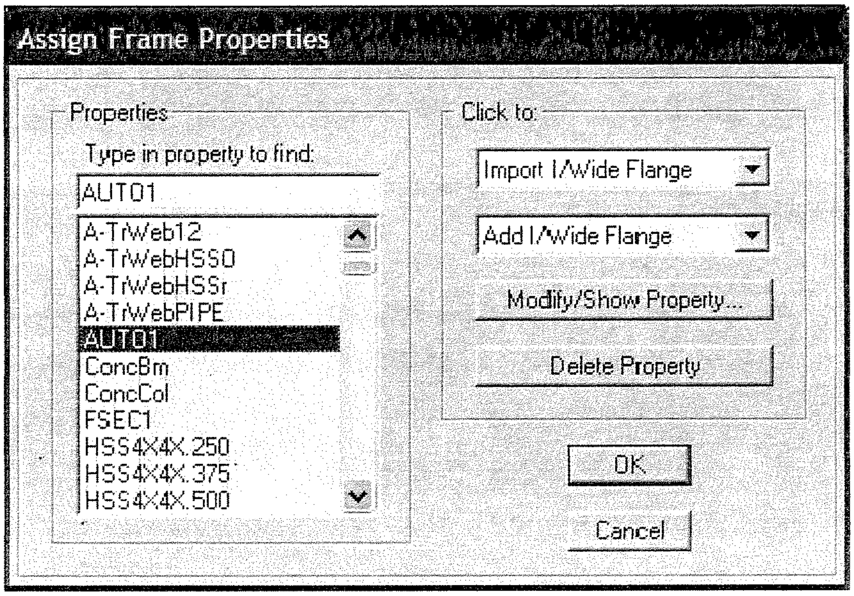
1-select



3-Click OK button

3. **Assign the Auto Frame section to the modal**

- Select the frame elements using the window selection
- Click the **Assign menu** ⟶ **Frame /Line** ⟶ **Frame Section,** thin the Assign Frame Properties Form will be displayed as shown



- Select Auto1 section thin click **OK**

4. **complete of the modal creation**

- Complete the creation of the modal as we explain before thin Run the analysis.
- In the initial analysis ,the program used the medium section by weight from the **Auto Select Section** List

5. **select of the Load of Combinations**

- Click the **Design menu** ⟶ **Steel Frame Design** ⟶ **Select Design Combo**

Note: As we mentioned before the program create automatically the load of combinations according to the design code ,and allow you from this from to remove or add any case of loading according to the design process, in this example we will use the load of combination which the program create .

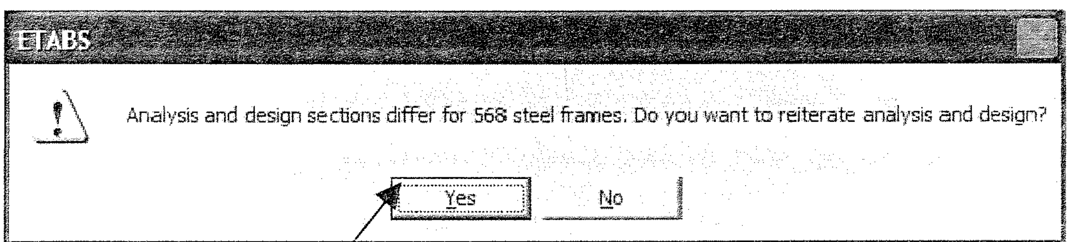
- After you add or remove any of the load of combination click OK

6. Start Design

Note: When an auto select selection list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

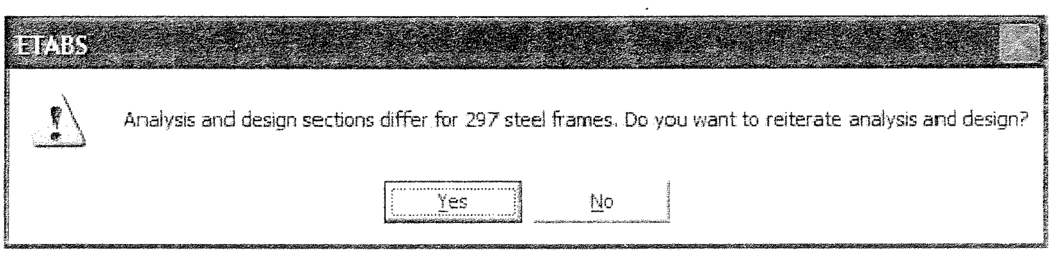
- click the **Design menu** ⟶ **Steel Frame Design** ⟶ **Start Design/Check of Structure**

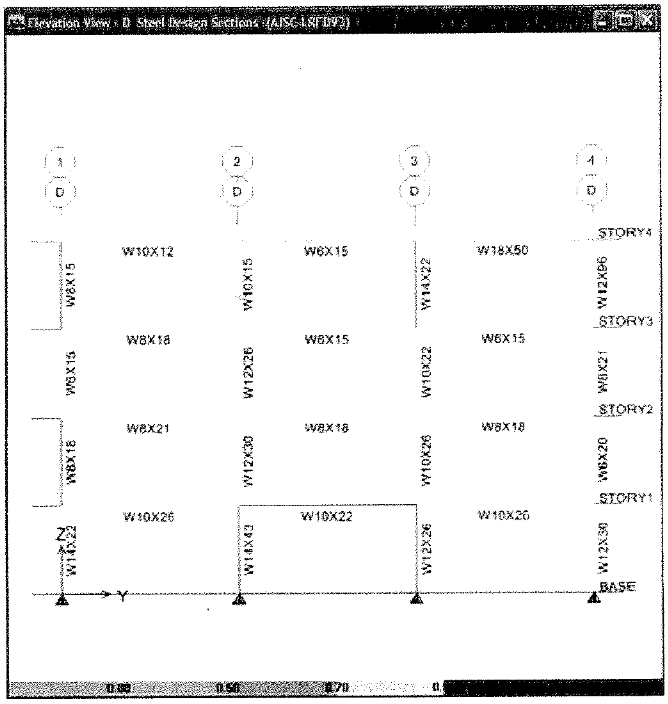
- After the program finish the run the next form will displayed



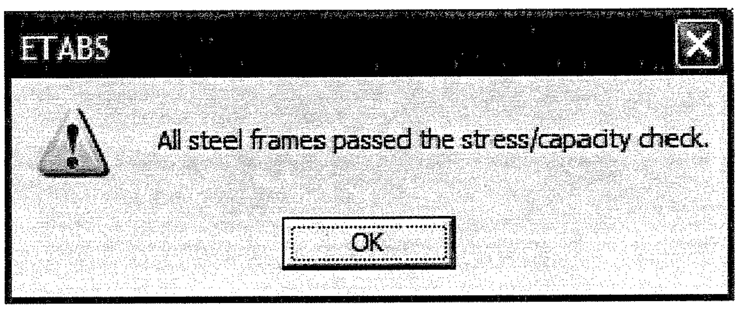
Click Yes to reanalysis again with the new sections which coming from the design

- After the program redesign the all sections according to the design sections the previous massage will be displayed again mentioning the no. of difference between design sections and analysis sections





- repeat the previous steps until the analysis and design sections are the same and the next message will be displayed

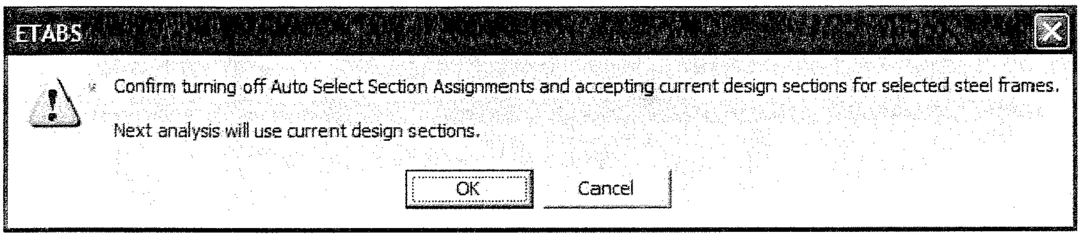


Important Note:

1- Some times it is take too much time to make the analysis and design sections the same and it is up to you to take last design sections or to complete until the analysis and design section be the same

2- If some sections not pass the stress capacity check, this mean you need to add larger section to the auto select list

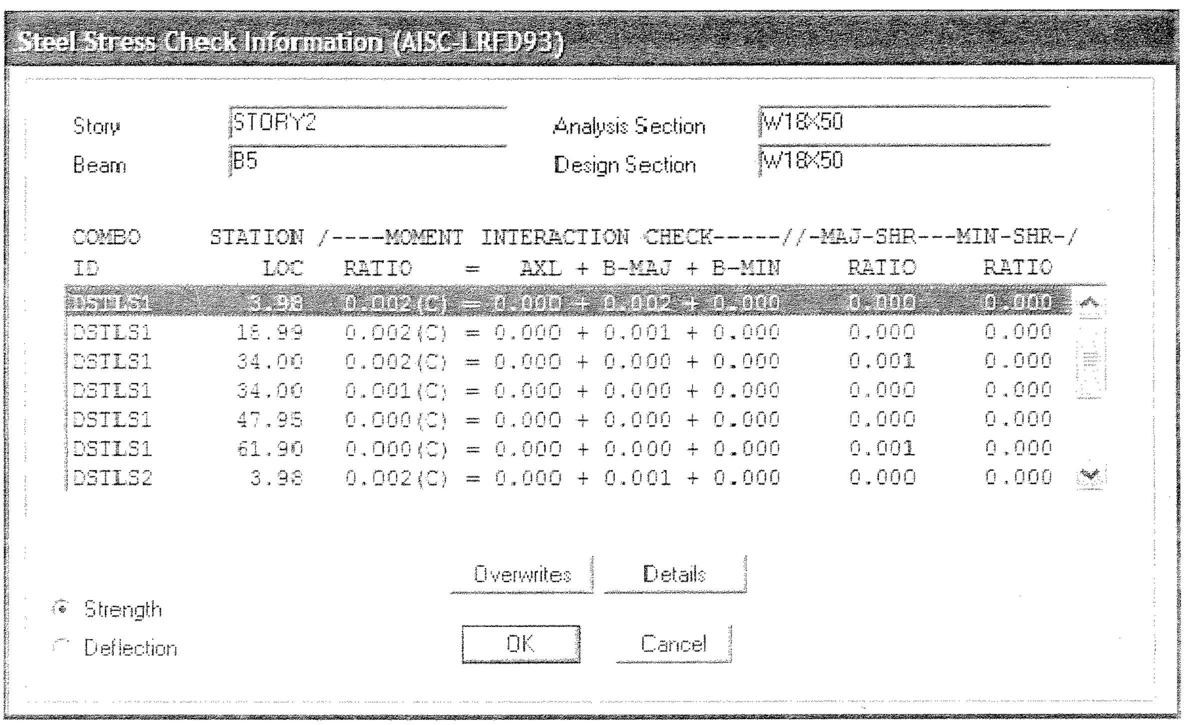
- select all the members thin click **Design** ⟶ **Steel Frame Design** ⟶ **Make Auto Selection Null** ,thin the next form will be displayed

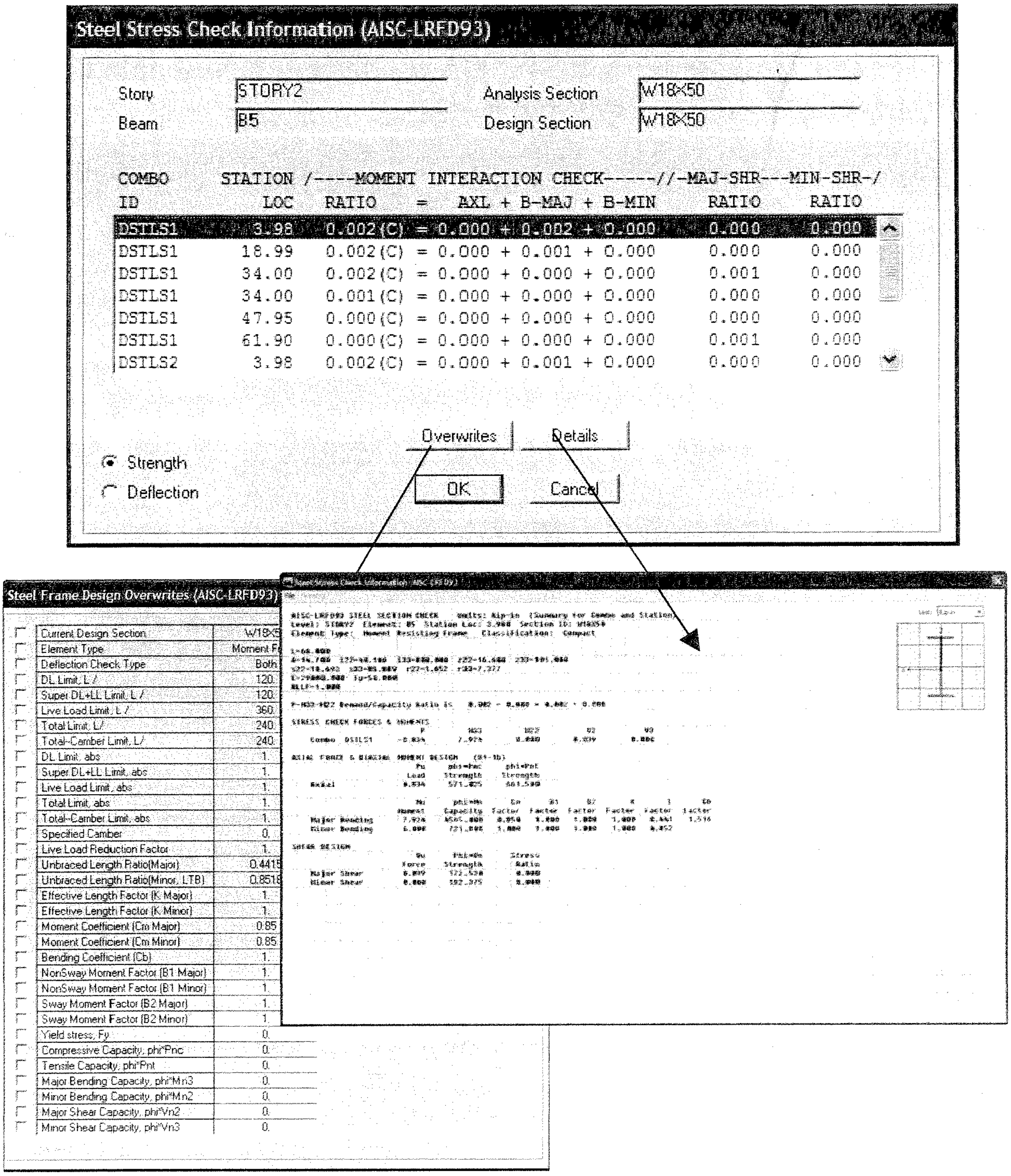


- Click **OK** button to remove the Auto Select Section List and Replace it with the Design sections which is mean every element reassign again according to the design Sections.

7. **Review the Result of Design**

1. Right click on one of the Steel Elements the program will display the Review form, you can click of any of the icon of this form to make any change in design data or to check any of the results data




Steel Stress Check Information (AISC-LRFD93)
Story STORY2
Beam B5
Analysis Section W18X50
Design Section W18X50
COMBO ID STATION LOC /----MOMENT INTERACTION CHECK-----//-MAJ-SHR---MIN-SHR-/
RATIO = AXL + B-MAJ + B-MIN RATIO RATIO
DSTLS1 3.98 0.002(C) = 0.000 + 0.002 + 0.000 0.000 0.000
DSTLS1 18.99 0.002(C) = 0.000 + 0.001 + 0.000 0.000 0.000
DSTLS1 34.00 0.002(C) = 0.000 + 0.000 + 0.000 0.001 0.000
DSTLS1 34.00 0.001(C) = 0.000 + 0.000 + 0.000 0.000 0.000
DSTLS1 47.95 0.000(C) = 0.000 + 0.000 + 0.000 0.000 0.000
DSTLS1 61.90 0.000(C) = 0.000 + 0.000 + 0.000 0.001 0.000
DSTLS2 3.98 0.002(C) = 0.000 + 0.001 + 0.000 0.000 0.000
Overwrites
Details
Strength
Deflection
OK
Cancel
Steel Frame Design Overwrites (AISC-LRFD93)
Current Design Section
Element Type
Deflection Check Type
DL Limit, L /
Super DL+LL Limit, L /
Live Load Limit, L /
Total Limit, L/
Total-Camber Limit, L/
DL Limit, abs
Super DL+LL Limit, abs
Live Load Limit, abs
Total Limit, abs
Total-Camber Limit, abs
Specified Camber
Live Load Reduction Factor
Unbraced Length Ratio(Major)
Unbraced Length Ratio(Minor, LTB)
Effective Length Factor (K Major)
Effective Length Factor (K Minor)
Moment Coefficient (Cm Major)
Moment Coefficient (Cm Minor)
Bending Coefficient (Cb)
NonSway Moment Factor (B1 Major)
NonSway Moment Factor (B1 Minor)
Sway Moment Factor (B2 Major)
Sway Moment Factor (B2 Minor)
Yield stress, Fy
Compressive Capacity, phi*Pnc
Tensile Capacity, phi*Pnt
Major Bending Capacity, phi*Mn3
Minor Bending Capacity, phi*Mn2
Major Shear Capacity, phi*Vn2
Minor Shear Capacity, phi*Vn3
W18X5
Moment F
Both
120.
120.
360.
240.
240.
1.
1.
1.
1.
1.
0.
1.
0.4415
0.8518
1.
1.
0.85
0.85
1.
1.
1.
1.
1.
0.
0.
0.
0.
0.
0.
0.

HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

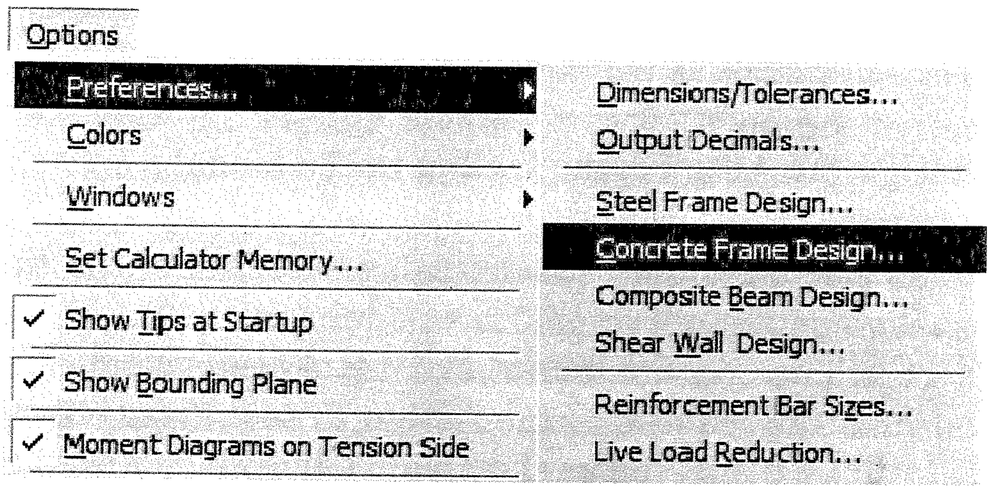
Composite Beams Design

Chapter 5

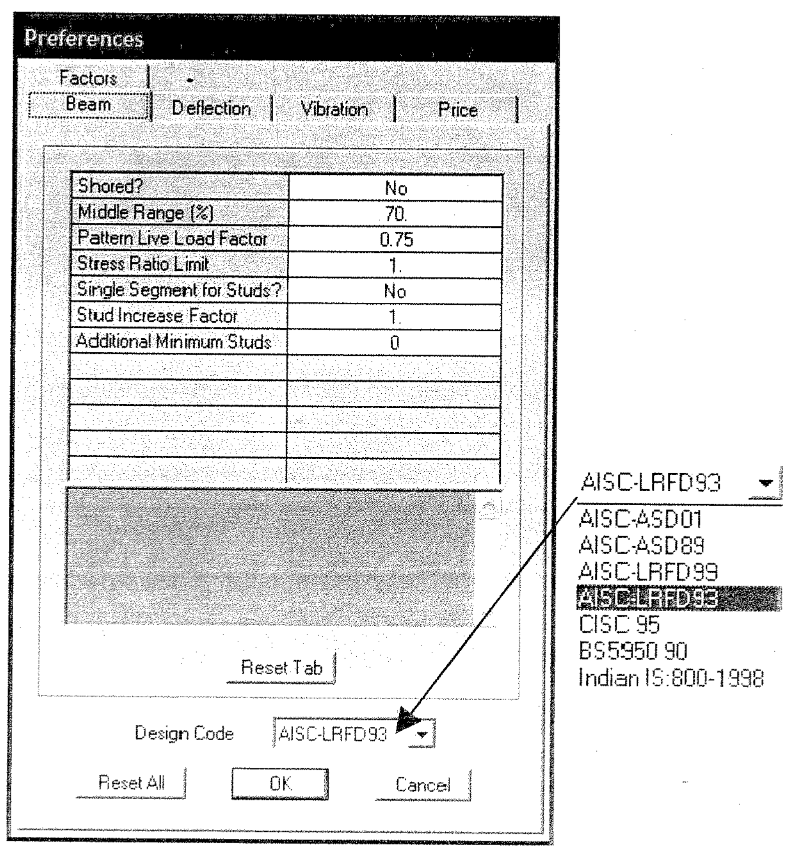
Composite Beams Design:

1. Adjust the design code and Data of Design

- Click the **Option menu** ⟶ **Preferences** ⟶ **Composite Beam Design**



This will Display the Composite Beam Design Preference form as shown in fig.



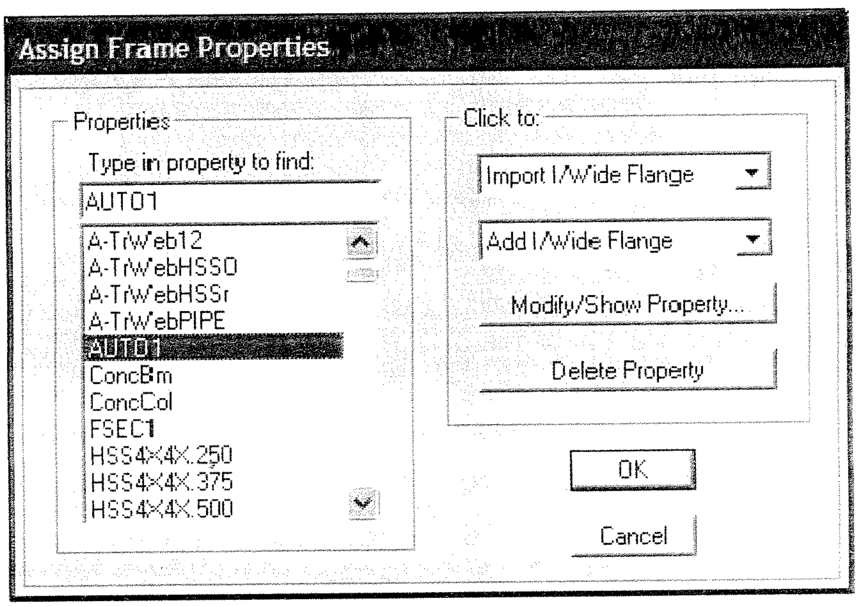
- From this form, you can adjust the code and another data of design as shown in pervious figure.
- From the Design Code drop-down list choose AISC-ASD01 Code or any another Code you want your design based in.

2. Define an Auto Select Section List

- The same steps as we explain in steel frame design

3. Assign the Auto Frame section to the modal

- Select the frame elements using the window selection
- Click the **Assign menu** ⟶ **Frame /Line** ⟶ **Frame Section thin the Assign Frame Properties Form will be displayed as shown**



- Select Auto1 section thin click **OK**

4. complete of the modal creation

- Complete the creation of the modal as we explain before thin Run the analysis.

- In the initial analysis ,the program used the medium section by weight from the **Auto Select Section** List

Note: the default design procedure to secondary beams is composite beam design

5. **select of the Load of Combinations**

- Click the **Design menu** ⟶ **Composite Beam Design** ⟶ **Select Design Combo**

Note: As we mentioned before the program create automatically the load of combinations according to the design code ,and allow you from this from to remove or add any case of loading according to the design process, in this example we will use the load of combination which the program create .

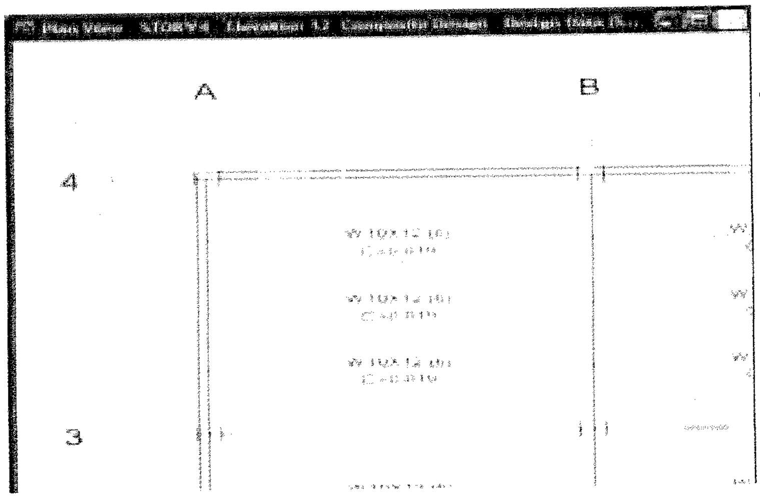
- After you add or remove any of the load of combination click OK

6. **Start Design**

Note: When an auto select selection list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

- For Design of **Composite beams** you have 2 choices :

1. Design Using similarity : this method make the program design all section and select the large section for all

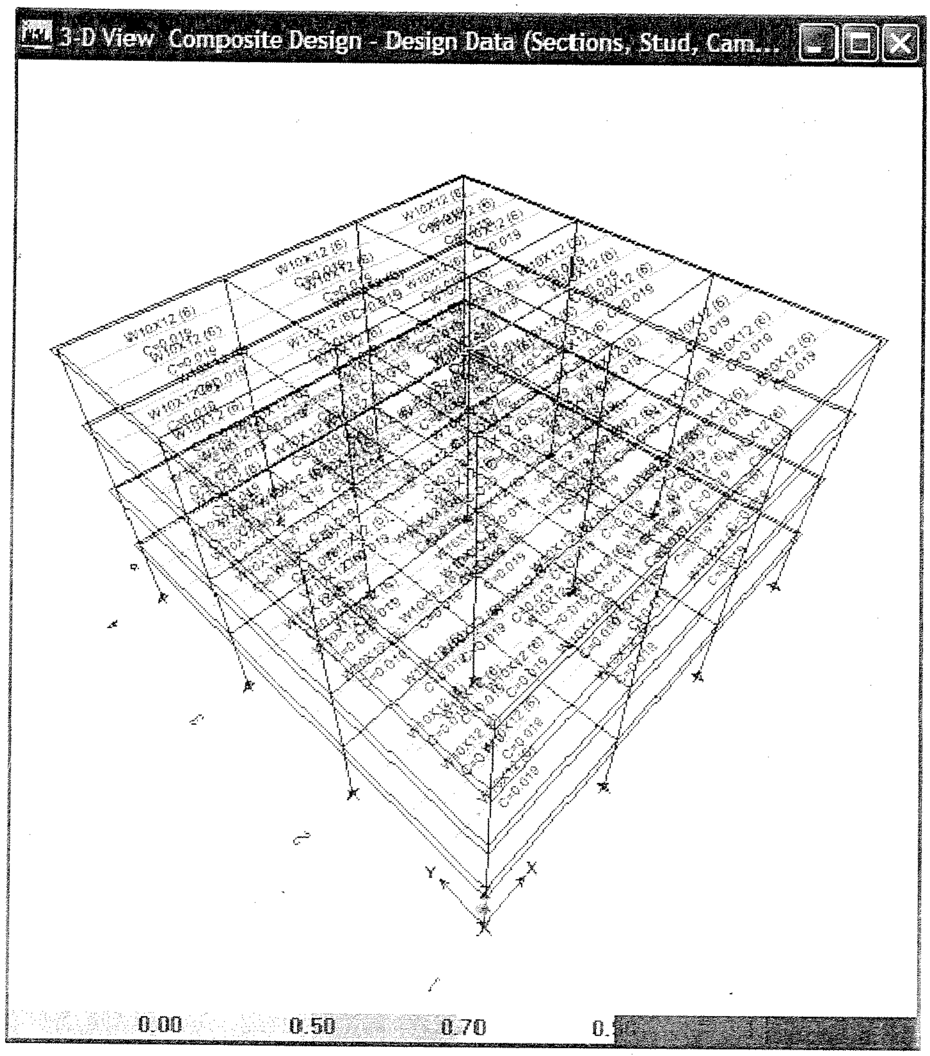


2. Design without similarity : this method make the program design all section and select the optimum beam size from the auto select list

Note: for the both method we can do the same step of design, we will explain **Design without similarity** method as example for both methods.

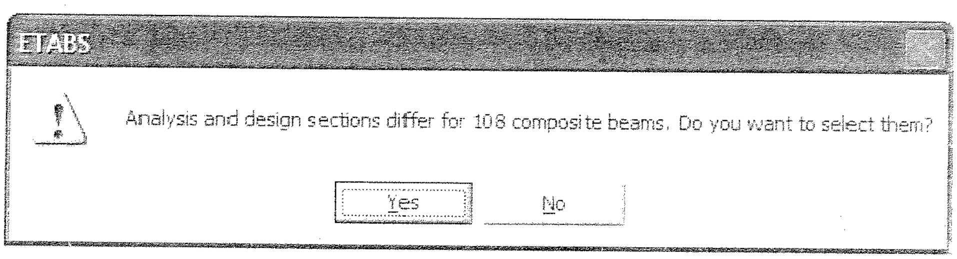
- Click the **Design menu** ⟶ **Composite Beam Design** ⟶ **Start Design/Check of Structure**

- After the program finish the run the Result of the design of the section will be displayed in the 3D view as Shown in the next Figure.



- As we mention before in steel frame design the design sections differs from the analysis sections. we will do the same steps as we do in steel frame design to be sure of the design sections is the same ad the analysis sections

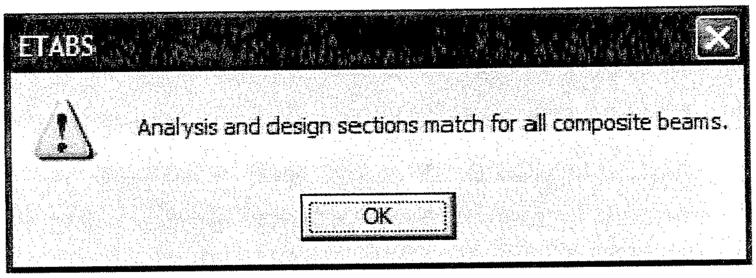
- Click **Design menu** → **Composite Beam Design** →. **Verify Analysis Design Sections**, thin the next massage will be displayed mentioning the no. of difference between design sections and analysis sections



- Click Yes

- Rerun the analysis again and the program will take the design section as analysis sections ,Click Run Button

- Repeat the previous step of design unit the program display the next form

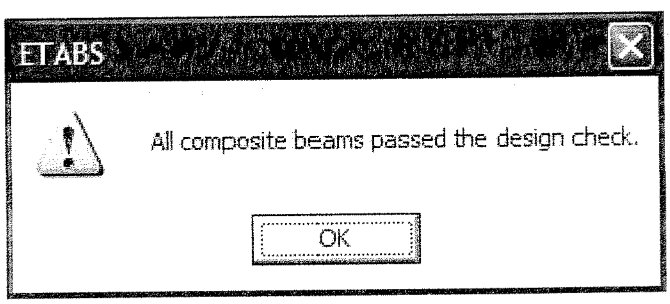


Important Note: we mentioned before in steel design

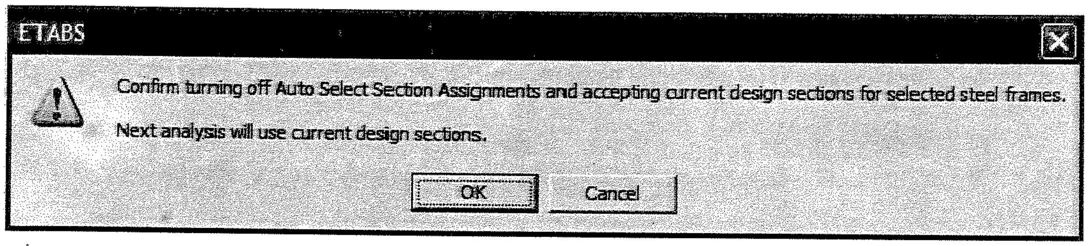
1- Some times it is take too much time to make the analysis and design sections are the same and it is up to you to take last design sections or to complete until the analysis and design section be the same

2- If some sections not pass the stress capacity check, this mean you need to add larger section to the auto select list

- To check the sections passed in design process or not click **Design** ——▶ **Composite Beam Design** ——▶ **Verify All Members Passed** thin the next form will be displayed indicate if the members pass or not



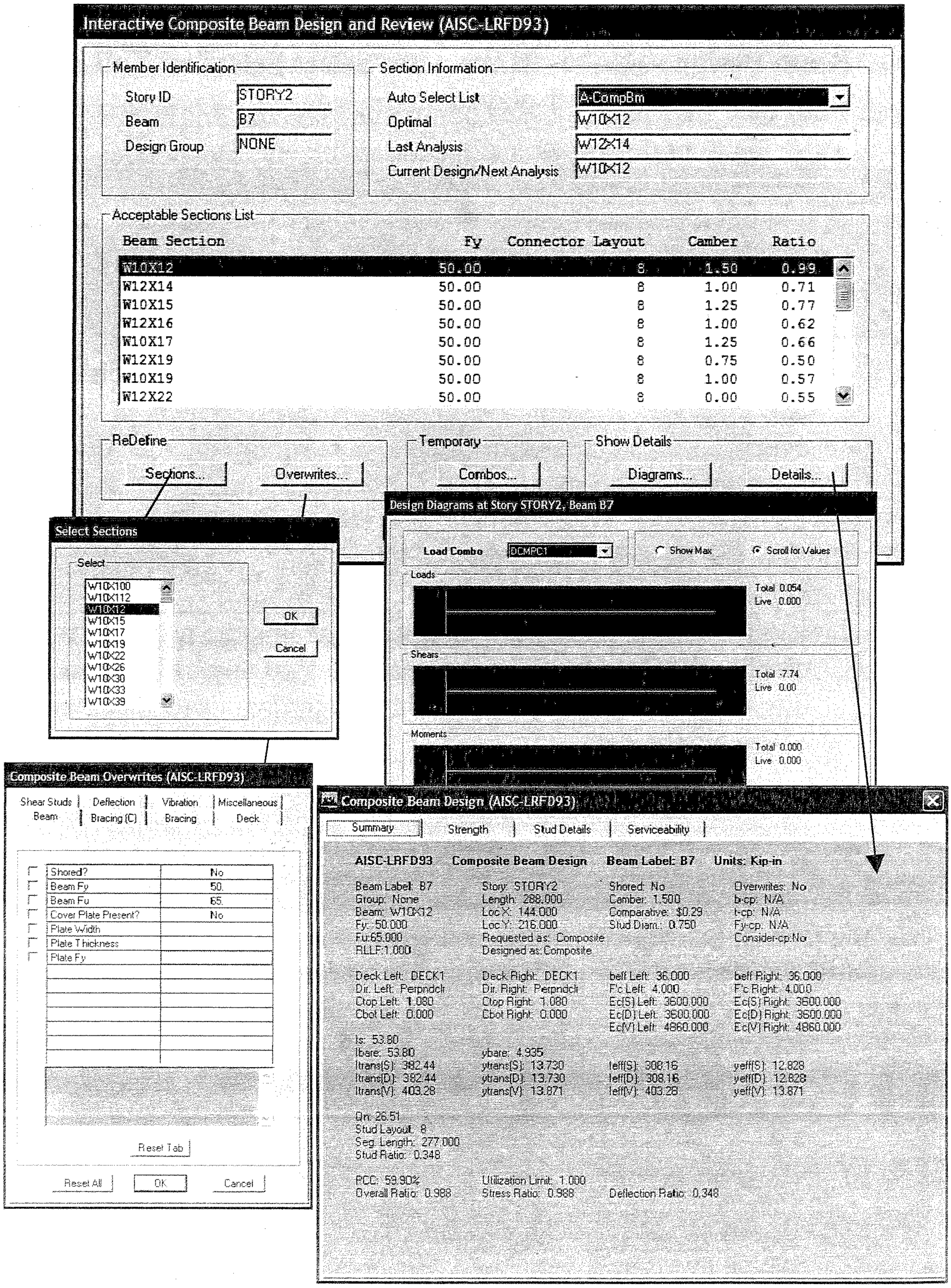
1. Select all the members thin click **Design** ——▶ **Composite Beam Design** ——▶ **Make Auto Select Selection Null** ,thin the next form will be displayed

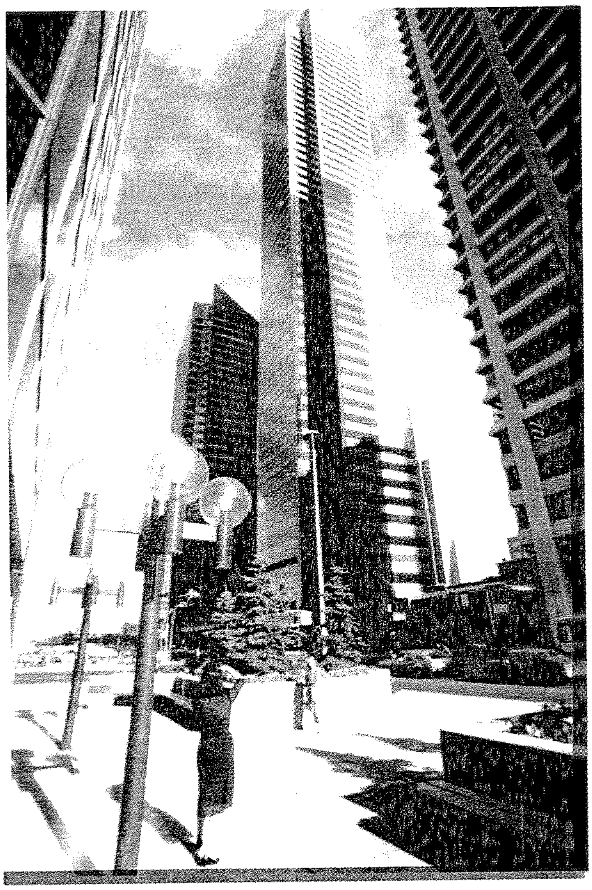


1. Click **OK** button to remove the Auto Select Section List and Replace it with the Design sections which is mean every element reassign again according to the design Sections.

7. Review the Result of Design

2. Right click on one of the composite beams the program will display the Review form, you can click of any of the icon of this form to make any change in design data or to check any of the results data


Interactive Composite Beam Design and Review (AISC-LRFD93)
Member Identification
Story ID STORY2
Beam B7
Design Group NONE
Section Information
Auto Select List A-CompBm
Optimal W10X12
Last Analysis W12X14
Current Design/Next Analysis W10X12
Acceptable Sections List
Beam Section Fy Connector Layout Camber Ratio
W10X12 50.00 8 1.50 0.99
W12X14 50.00 8 1.00 0.71
W10X15 50.00 8 1.25 0.77
W12X16 50.00 8 1.00 0.62
W10X17 50.00 8 1.25 0.66
W12X19 50.00 8 0.75 0.50
W10X19 50.00 8 1.00 0.57
W12X22 50.00 8 0.00 0.55
ReDefine
Sections...
Overwrites...
Temporary
Combos...
Show Details
Diagrams...
Details...
Select Sections
Select
W10X100
W10X112
W10X12
W10X15
W10X17
W10X19
W10X22
W10X26
W10X30
W10X33
W10X39
OK
Cancel
Design Diagrams at Story STORY2, Beam B7
Load Combo DCMPC1
Show Max
Scroll for Values
Loads
Total 0.054
Live 0.000
Shears
Total -7.74
Live 0.00
Moments
Total 0.000
Live 0.000
Composite Beam Overwrites (AISC-LRFD93)
Shear Studs
Beam
Deflection
Bracing (C)
Vibration
Bracing
Miscellaneous
Deck
Shored? No
Beam Fy 50.
Beam Fu 65.
Cover Plate Present? No
Plate Width
Plate Thickness
Plate Fy
Reset Tab
Reset All
OK
Cancel
Composite Beam Design (AISC-LRFD93)
Summary
Strength
Stud Details
Serviceability
AISC-LRFD93 Composite Beam Design Beam Label: B7 Units: Kip-in
Beam Label: B7
Group: None
Beam: W10X12
Fy: 50.000
Fu:65.000
RLLF:1.000
Story: STORY2
Length: 288.000
Loc X: 144.000
Loc Y: 216.000
Requested as: Composite
Designed as: Composite
Shored: No
Camber: 1.500
Comparative: $0.29
Stud Diam.: 0.750
Overwrites: No
b-cp: N/A
t-cp: N/A
Fy-cp: N/A
Consider-cp:No
Deck Left: DECK1
Dir. Left: Perpndclr
Ctop Left: 1.080
Cbot Left: 0.000
Deck Right: DECK1
Dir. Right: Perpndclr
Ctop Right: 1.080
Cbot Right: 0.000
beff Left: 36.000
F'c Left: 4.000
Ec(S) Left: 3600.000
Ec(D) Left: 3600.000
Ec(V) Left: 4860.000
beff Right: 36.000
F'c Right: 4.000
Ec(S) Right: 3600.000
Ec(D) Right: 3600.000
Ec(V) Right: 4860.000
Is: 53.80
Ibare: 53.80
Itrans(S): 382.44
Itrans(D): 382.44
Itrans(V): 403.28
ybare: 4.935
ytrans(S): 13.730
ytrans(D): 13.730
ytrans(V): 13.871
Ieff(S): 308.16
Ieff(D): 308.16
Ieff(V): 403.28
yeff(S): 12.828
yeff(D): 12.828
yeff(V): 13.871
Qn: 26.51
Stud Layout: 8
Seg. Length: 277.000
Stud Ratio: 0.348
PCC: 59.90%
Overall Ratio: 0.988
Utilization Limit: 1.000
Stress Ratio: 0.988
Deflection Ratio: 0.348



HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

P-Δ Analysis

Chapter 6

P-Δ Analysis :

- This option is particularly useful for considering the effect of gravity loads upon the lateral stiffness of building structures, as required by certain design codes

- **P-Δ Force :**

  - The resulting member forces and moments and the story drifts introduced by PΔ effects shall be considered in the evaluation of overall structural frame stability and shall be evaluated using the forces producing the displacement of Δs

Note:

- When you request initial P-Delta analysis, it is performed before all linear-static, modal, response-spectrum, and time-history analyses in the same analysis run.

- The initial P-Delta analysis essentially modifies the characteristics of the structure, affecting the results of all subsequent analyses performed. As an important exception,

- Initial P-Delta analysis does NOT affect nonlinear-static analysis. Nonlinear static analyses consider the P-Delta effect separately, if requested

- **ETABS PROGRAM** has the ability to calculate the PΔ effects by tow different methods

  1. Non-Iterative -- Based on Mass

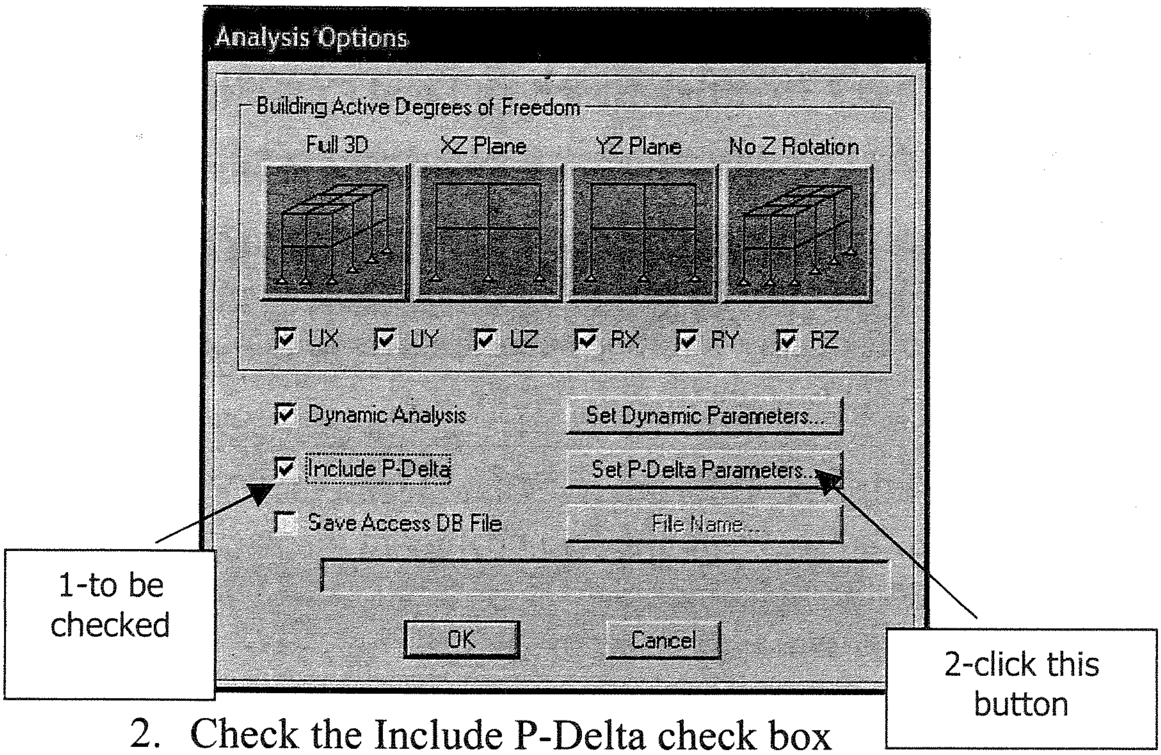
  2. Iterative -- Based on Load Cases

- The next table show the difference between the tow methods

	Non-Iterative	Iterative
The load computation	automatically from the mass at each level	from a specified combination of static load cases
Accuracy	approximate	accurate
iterative solution	not require an iterative solution	require an iterative solution
Time taken in analysis	much faster	Take more time than Non-Iterative
local buckling	It does not capture local buckling	It captures local buckling
Gravity Load Definition	It is not needed to define any gravity loads	It is needed to define gravity loads
Recommendation	Not Recommended	Recommended

- In the next steps we will explain how to use the **Iterative** method in analysis to take the PΔ effect.

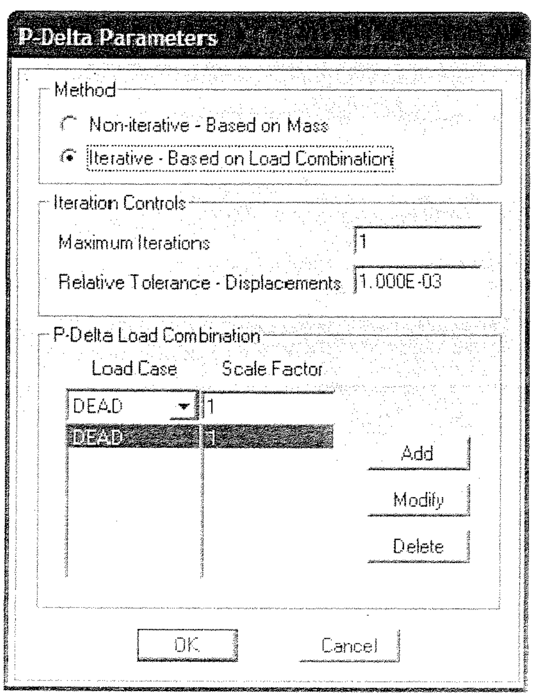
1. Click the **Analyze menu** ⟶ **Set Analysis Options** command to bring up the Analysis Options form.



2. Check the Include P-Delta check box

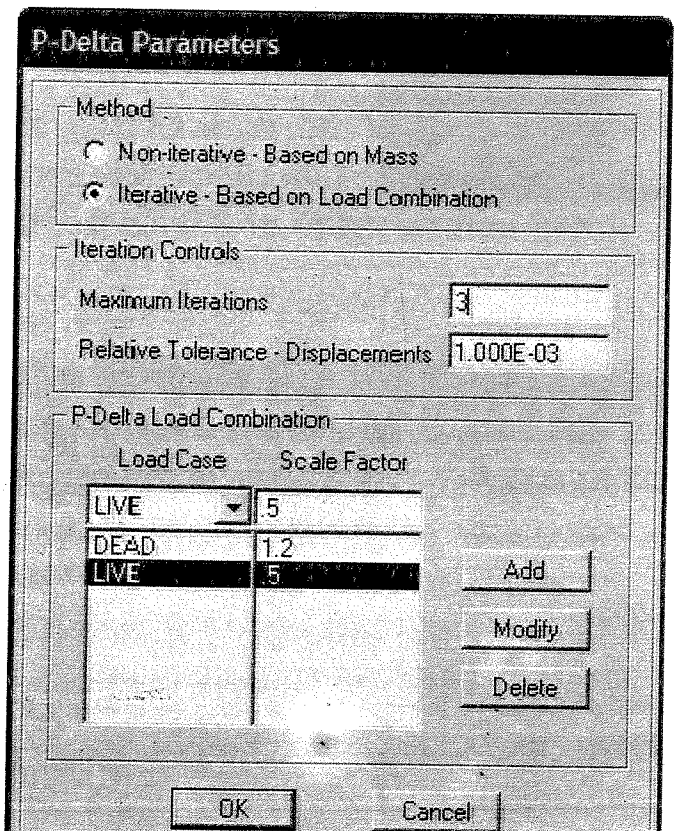
3. click the Set P-Delta Parameters button

4. the P-Delta Parameters form will be displayed

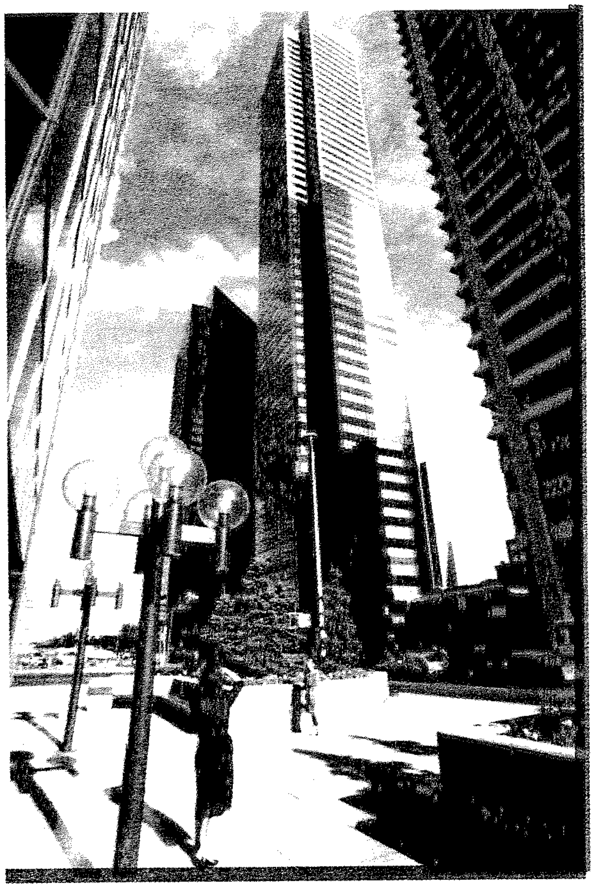


- From previous form we will define the next Data:
- **Method**:
  - Check **Iterative-Based on Load Combination.**
- **Iteration control:**
  - **Maximum Iteration :** A reasonable number is usually 2 to 5 or any number according to your building (we will make our maximum iteration 3)
  - **Relative Tolerance – Displacement:** The default value is 0.001. If the relative change in displacement from one iteration to the next is less than the tolerance, and we will keep it as it is.
- **P-Delta Load Combination**:
  - some of the engineers are confused of this item because they suggest this item represent the all load of combinations which the program will use in analysis and this completely wrong the load of combination which we will put it in this cell is the initial value which the program begin its iteration, we already have in our example according to ACI 318-02 code 22 case of loading, the program will use it all in analysis but you must Specify the single load combination to be used for the **initial P-Delta analysis** of the structure .in our example we will specify the P-Delta load combination to be 1.2 times dead load plus 0.5 times live load, and this value is conservative. Because we take the first part of the load of combination of earthquake (**1.2 D + 0.5 LL** + Ex) and we have another combinations less in value like (0.9 D + EX)
  - To Add the load of combination definition

1. Select the load case name from the Case Name drop-down box.(Dead)
2. Type in an appropriate scale factor in the Scale Factor edit box. (1.2)
3. Click the Modify button.
4. Select a new load case from the drop-down list (Live) change the scale factor by typing directly in the Scale Factor edit box (0.5).
5. Click the Add button.



5. then click OK button
6. click Ok again in analysis option form
7. After the last step , run model and the program will take automatically the P-Delta effect



HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

Dynamic Analysis

Chapter 7

1) Modal analysis

- **Introduction :**

  - All Structure behave dynamically when subjected to load or displacement

  - Additional inertia force (from Dynamic behavior)= Mass X acceleration

  - If the load applied very slowly the acceleration value will be small hence the additional inertia force will be small and can be neglected, and static load analysis can be justified.

  - The acceleration of earthquakes can not be neglected hence the additional inertia force can not be neglected too (especially in irregular buildings)

  - The conclusion is, it is better to make dynamic analysis for all the buildings not only for the buildings which is mentioned in the codes, because it is very easy to make dynamic analysis as we will see in the next lines ,it is not need more efforts to do it, and this is the actual behavior of the building.

The first step in dynamic analysis is modal analysis

Modal analysis: are initially conducted to determine the elastic periods and the modes of vibration. This simple analysis is also useful as an initial validation tool of the analytical models. And modal analysis is step one for response spectrum analysis and time history analysis.

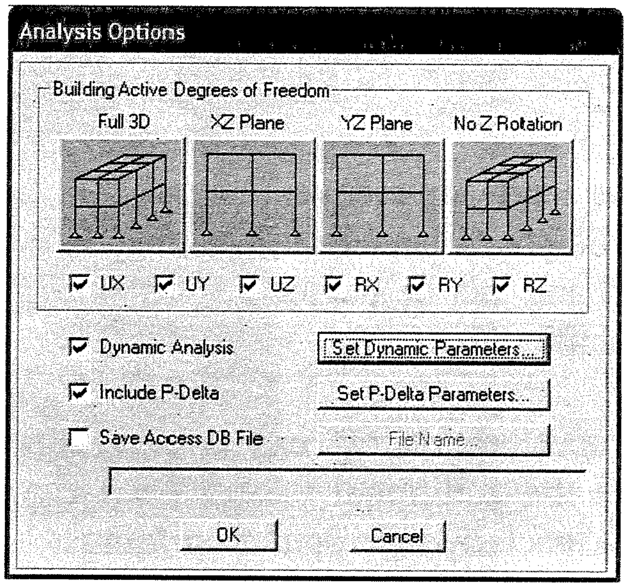
Type of analysis:

1-Eigenevectors 2-Ritz vectors

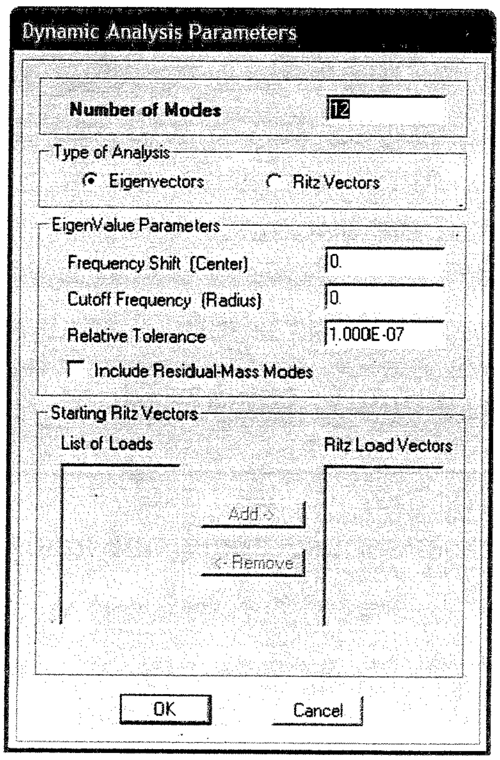
Note: the program recommend the Ritz vectors analysis because dynamic analyses based on load-dependent Ritz vectors yield more accurate results than the use of Eigenvectors

Steps for dynamics analysis

1. Click the Analyze **menu** ⟶ **Set Analysis Options** command to display the Analysis Options form.

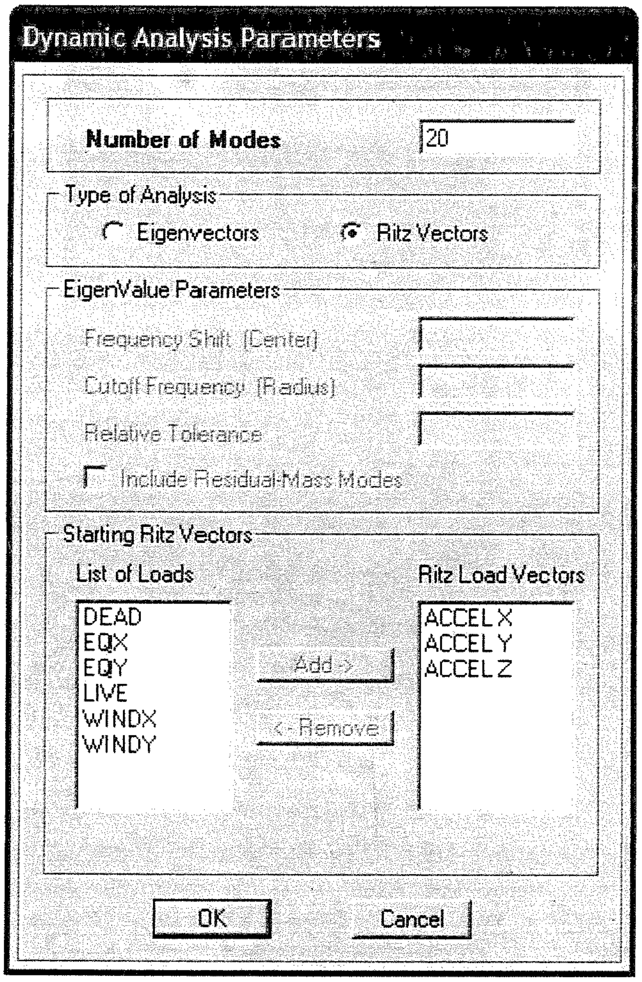


2. Click the Set Dynamic Parameters button.



3. Set the number of Eigen or Ritz modes to 20 in Number of

4. Check **Ritz vector** check box, thin the form will be as in the next fig.

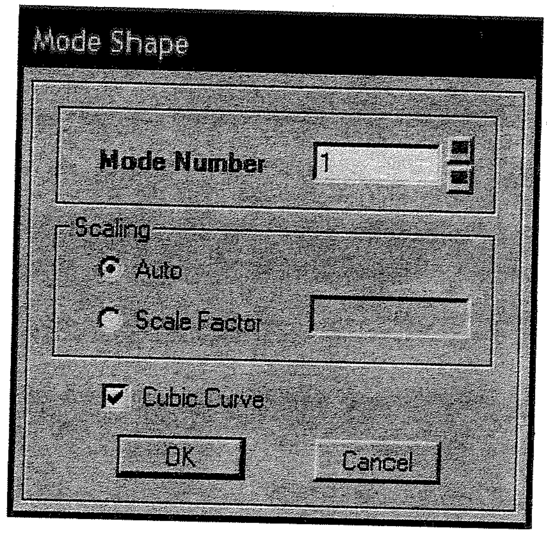


Note: For response-spectrum analysis, only the acceleration loads are needed.

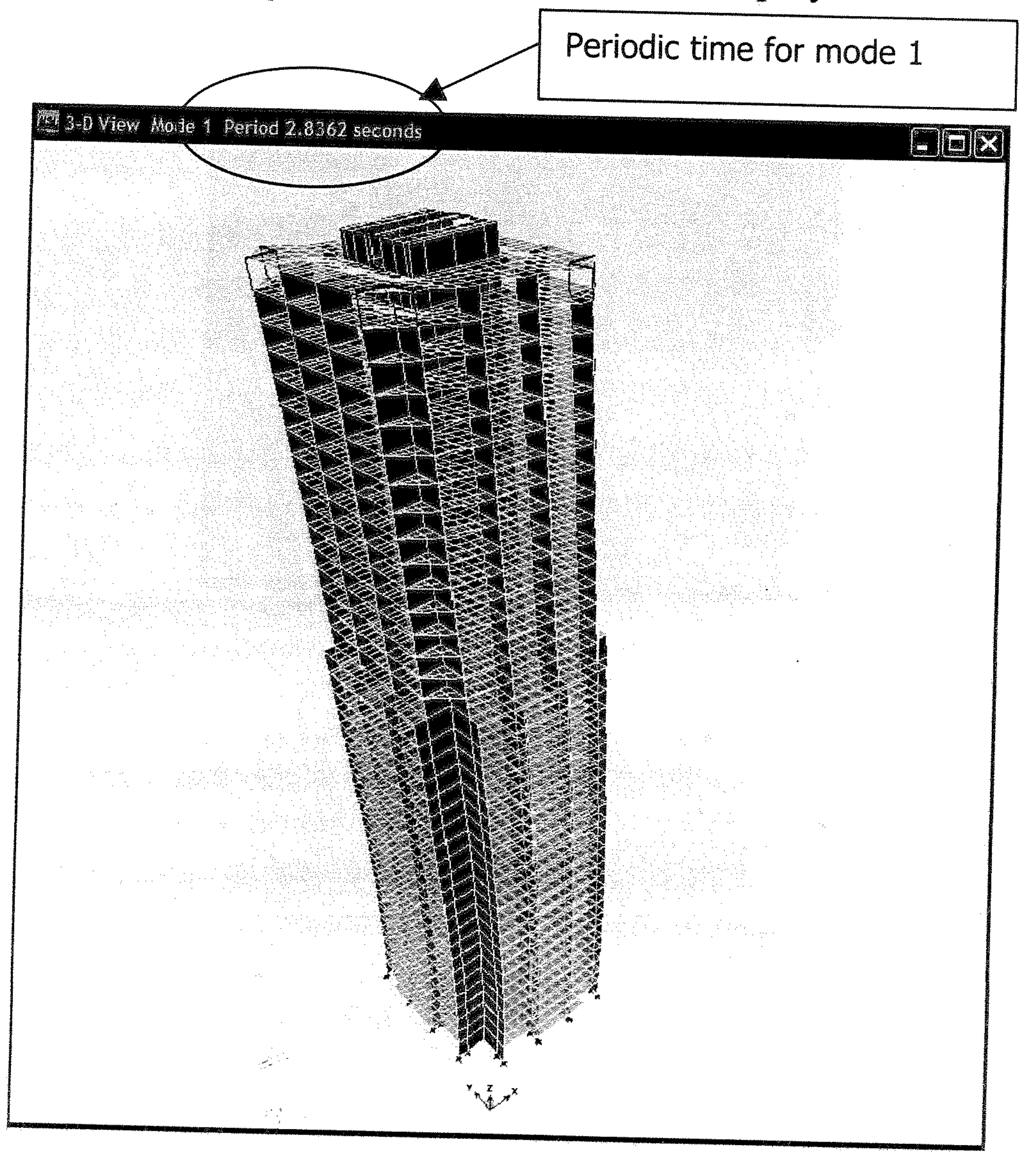
5. Click OK button in **Dynamic Analysis Parameters** form ,and **Analysis Option** form

6. Run the modal

7. once the analysis is completed ,click **Display menu** → **Display Mode Shapes** ,thin the **Mode Shape** form will be displayed

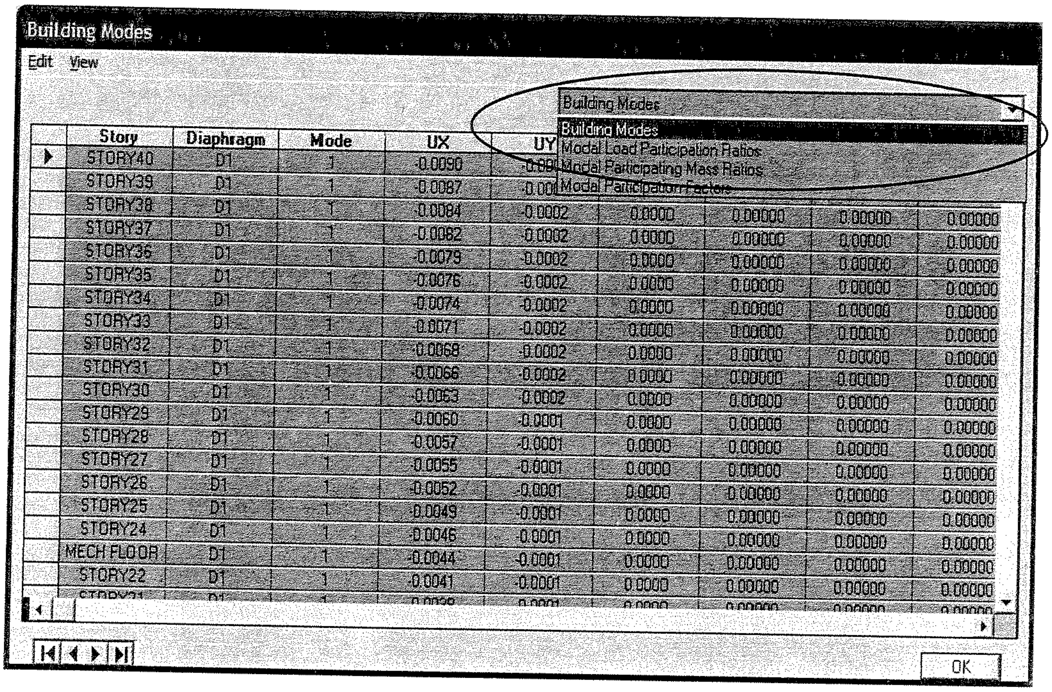


8. click OK to make the first mode shape display



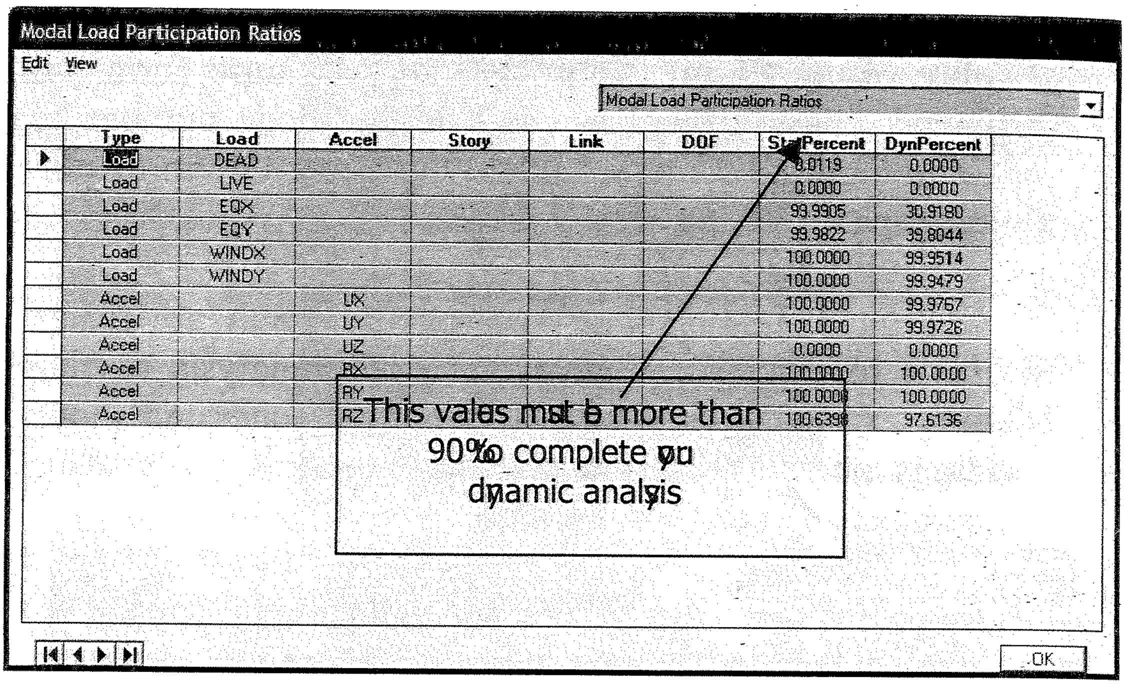
9. To display another mode shapes click **Display menu** ⟶ **Display Mode Shapes** ,thin the **Mode Shape** form will be displayed ,click in the arrow beside the mode number box to make the number 2 instead of 1, thin Click OK to display the mode shape 2

10. In addition to graphical display tabular result are ready and available to display ,to show the Modal analysis result click **Display** ⟶ **Show Tables** , The table generated shows the buildings mode



Story	Diaphragm	Mode	UX	UY				
STORY40	D1	1	-0.0090	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]
STORY39	D1	1	-0.0087	[illegible]	[illegible]	[illegible]	[illegible]	[illegible]
STORY38	D1	1	-0.0084	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY37	D1	1	-0.0082	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY36	D1	1	-0.0079	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY35	D1	1	-0.0076	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY34	D1	1	-0.0074	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY33	D1	1	-0.0071	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY32	D1	1	-0.0068	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY31	D1	1	-0.0066	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY30	D1	1	-0.0063	-0.0002	0.0000	0.00000	0.00000	0.00000
STORY29	D1	1	-0.0060	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY28	D1	1	-0.0057	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY27	D1	1	-0.0055	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY26	D1	1	-0.0052	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY25	D1	1	-0.0049	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY24	D1	1	-0.0046	-0.0001	0.0000	0.00000	0.00000	0.00000
MECH FLOOR	D1	1	-0.0044	-0.0001	0.0000	0.00000	0.00000	0.00000
STORY22	D1	1	-0.0041	-0.0001	0.0000	0.00000	0.00000	0.00000

11. To display the another result for the modal analysis choose the result you want to display from the down pull box which is marked



Type	Load	Accel	Story	Link	DOF	StatPercent	DynPercent
Load	DEAD					0.0119	0.0000
Load	LIVE					0.0000	0.0000
Load	EQX					99.9905	30.9180
Load	EQY					99.9822	39.8044
Load	WINDX					100.0000	99.9514
Load	WINDY					100.0000	99.9479
Accel		UX				100.0000	99.9767
Accel		UY				100.0000	99.9726
Accel		UZ				0.0000	0.0000
Accel		RX				100.0000	100.0000
Accel		RY				100.0000	100.0000
Accel		RZ				100.6398	97.6136

2) Response Spectrum Analysis

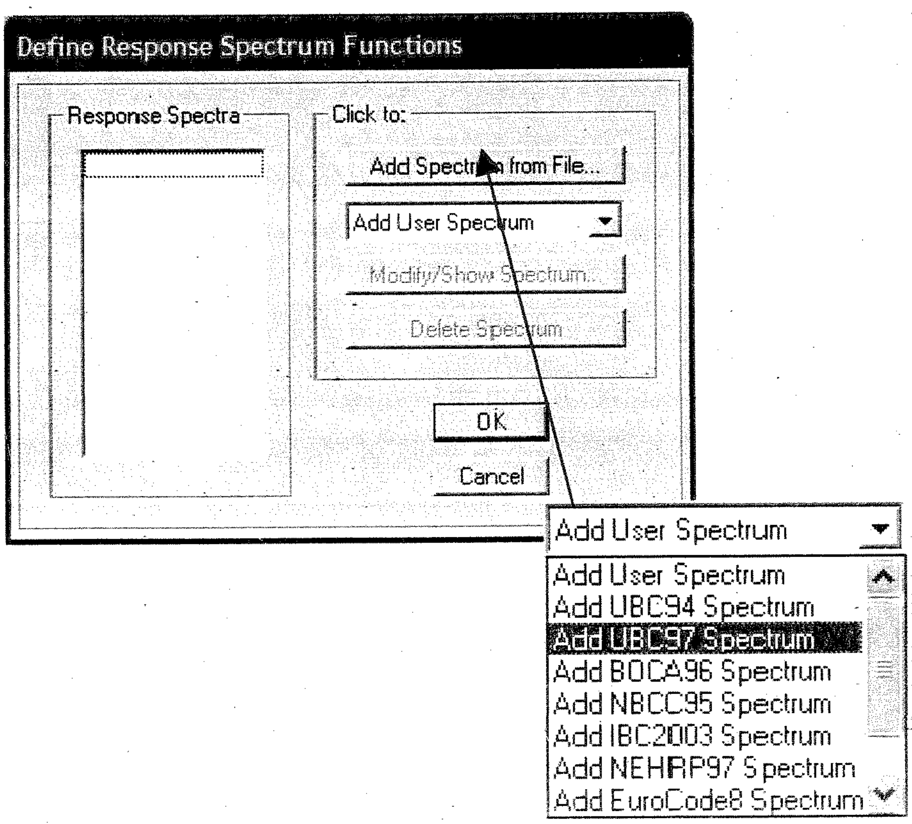
- **Modal analysis :** is the first step to begin Dynamic analysis
- **Types of Dynamic Analysis:**

  1. Response Spectrum analysis
  2. Time history analysis

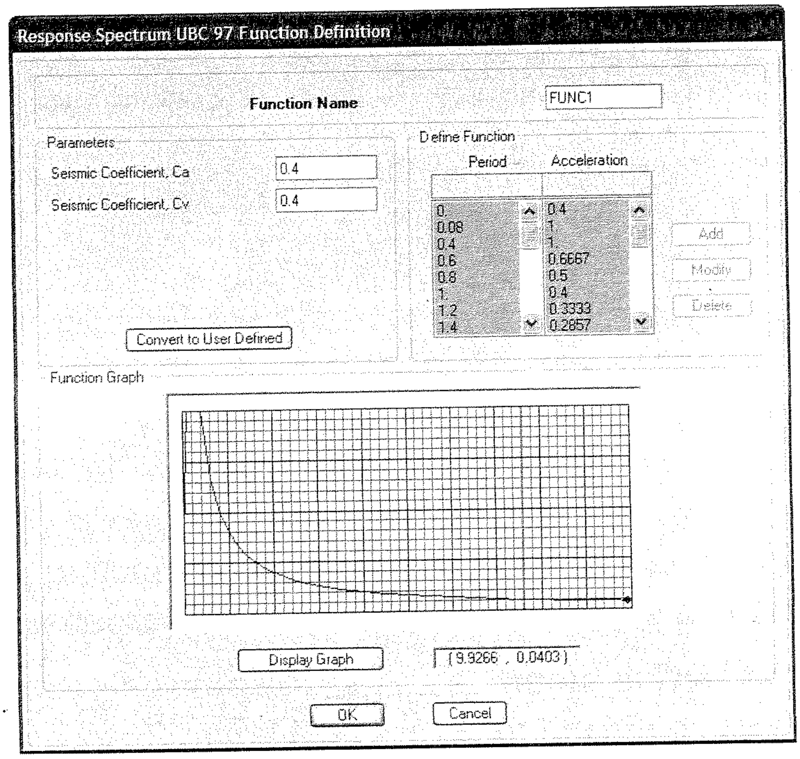
Note: Response Spectrum analysis is more common analysis and time history mainly used for research .

Response spectrum analysis Are calculated using the ordinates of the appropriate response spectrum curve which correspond to the modal periods. Maximum modal contributions are combined in a statistical manner to obtain an approximate total structural response.

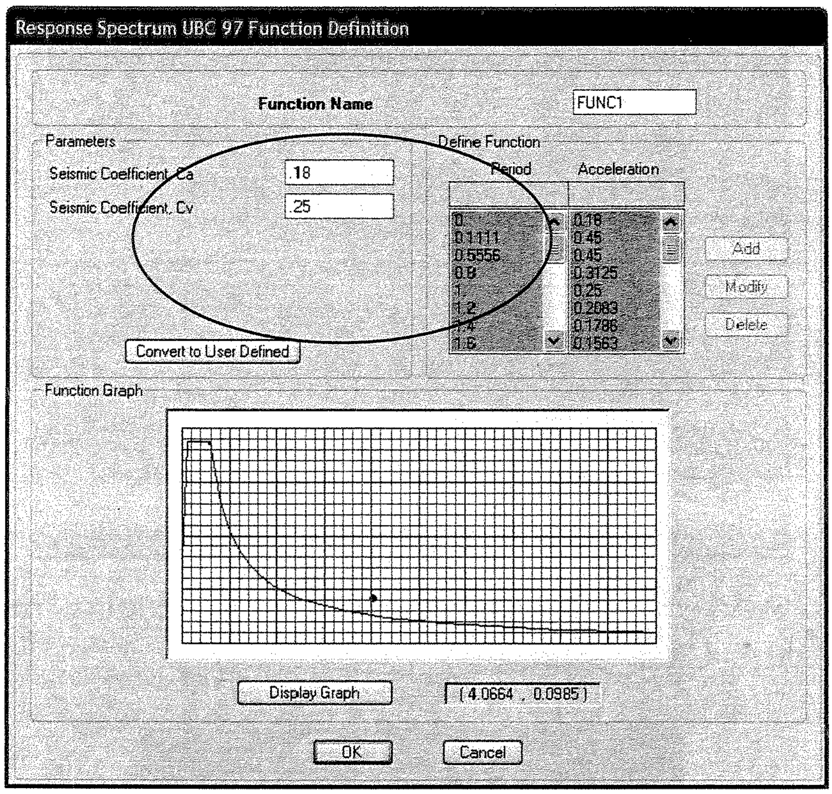
1. Click the **Define menu** ⟶ **Response Spectrum Functions** command to access the Define Response Spectrum Functions form.



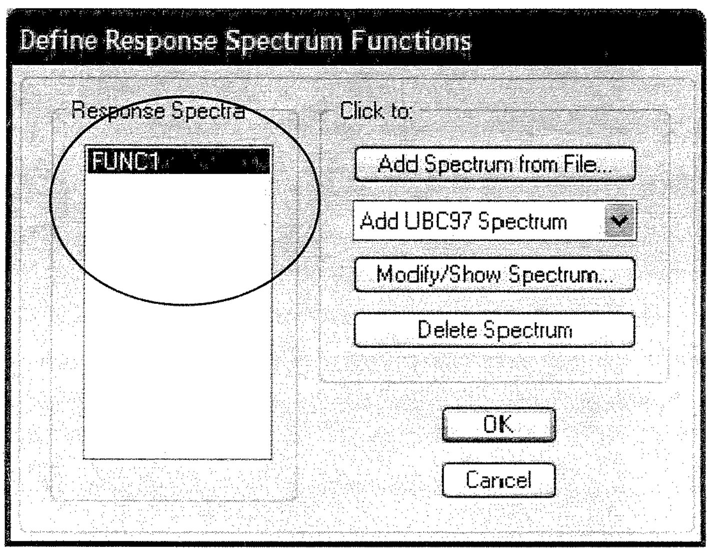
2. from the drop down box choose **Add UBC97 Spectrum** thin the next for will be displayed



From the previous form adjust the parameters of seismic Coefficient C_a and C_V to be according to your area for zone 2A make C_a =0.18 and C_V =0.25

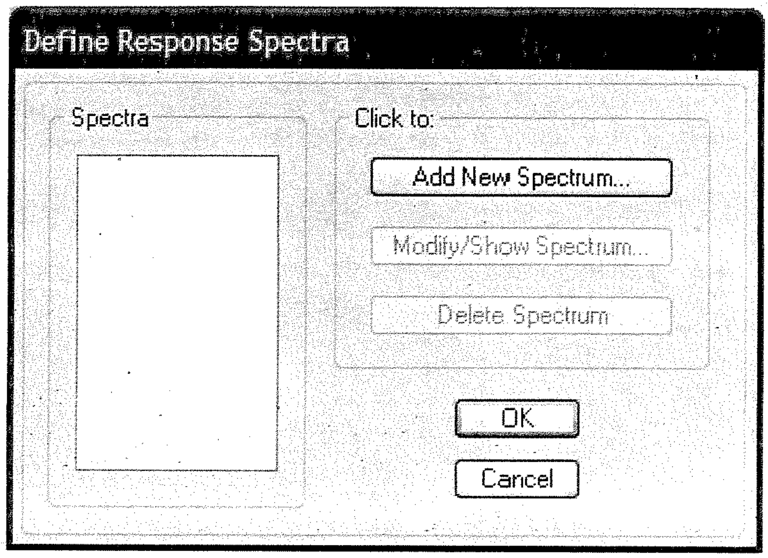


1. Click OK the unction will be add automatically

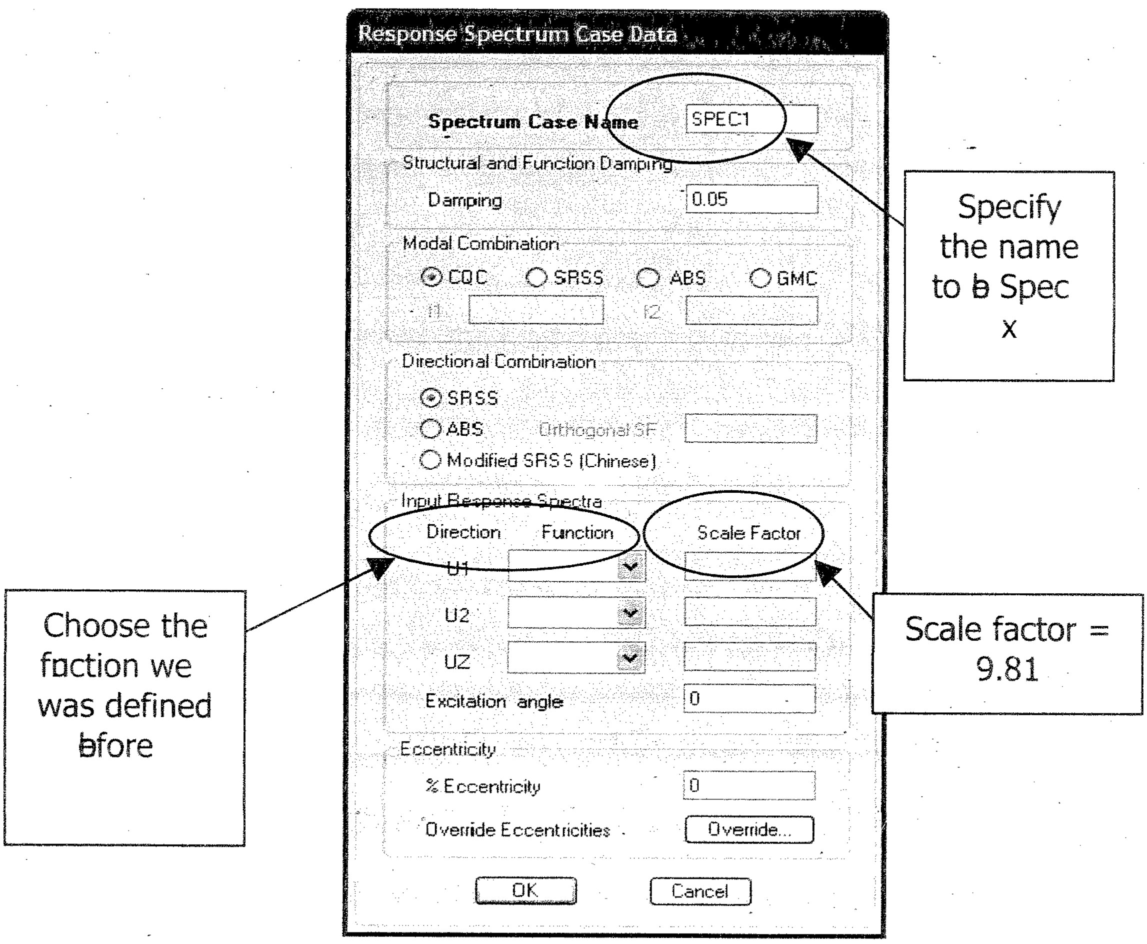


2. click **OK** for the main form of Define Response Spectrum Function

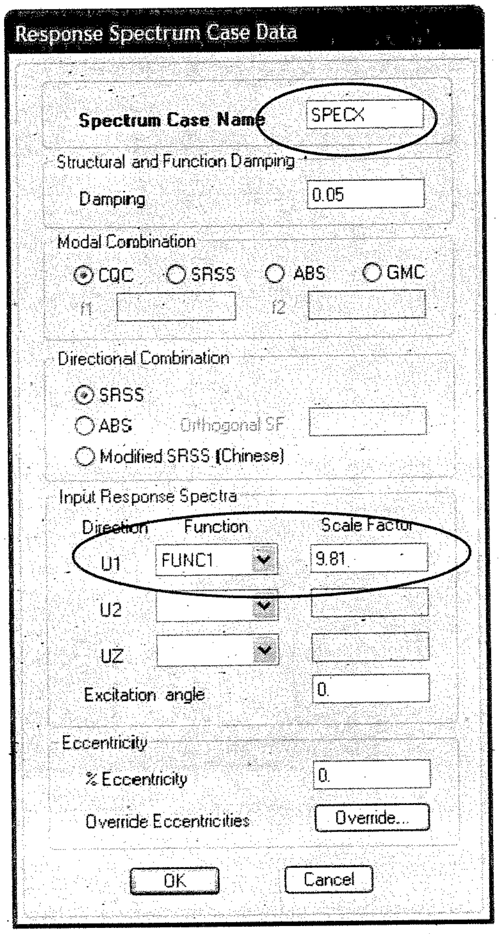
3. Click the **Define menu** ⟶ **Response Spectrum Cases** command to access the Define Response Spectrum cases form.



4. Click **Add New Spectrum** button, to display the **Response spectrum Case Data**



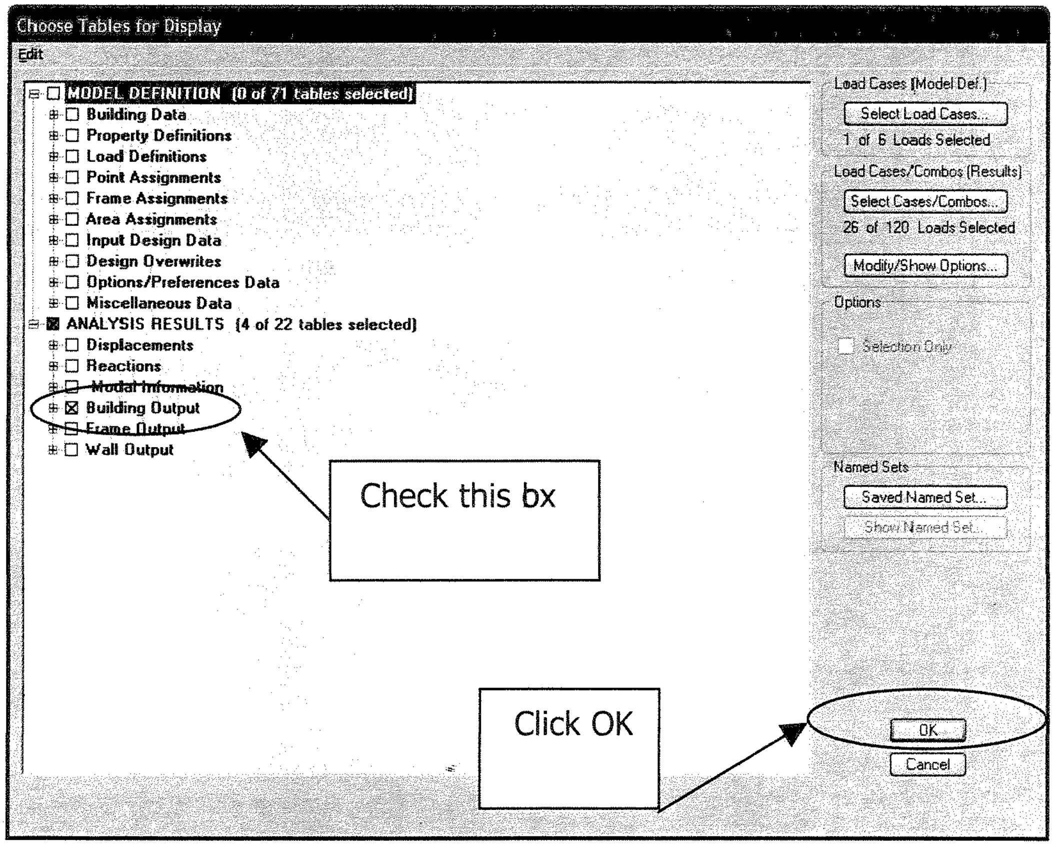
Thin the form will be as in the next figure



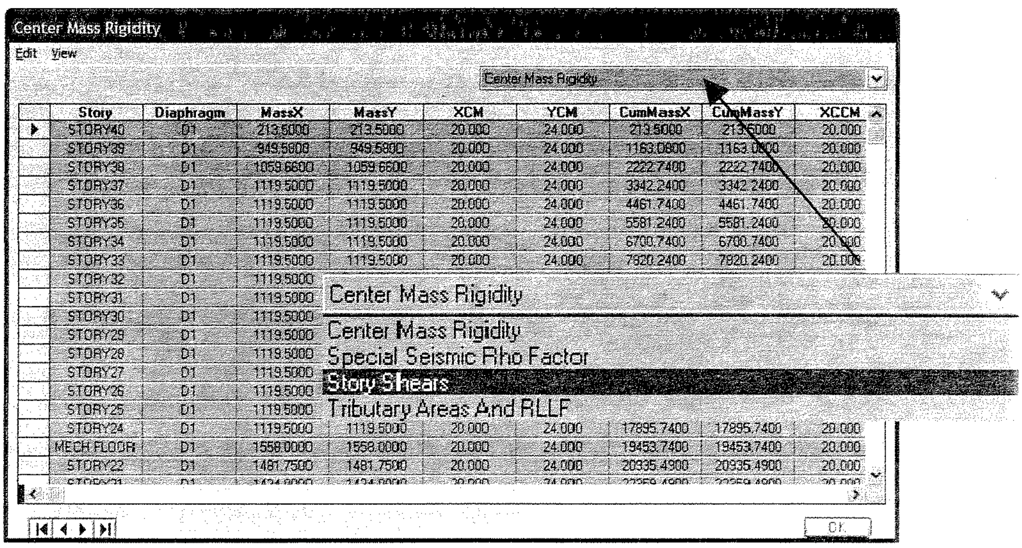
Thin click **OK** for this form and **OK** for the main form

5. Run the model

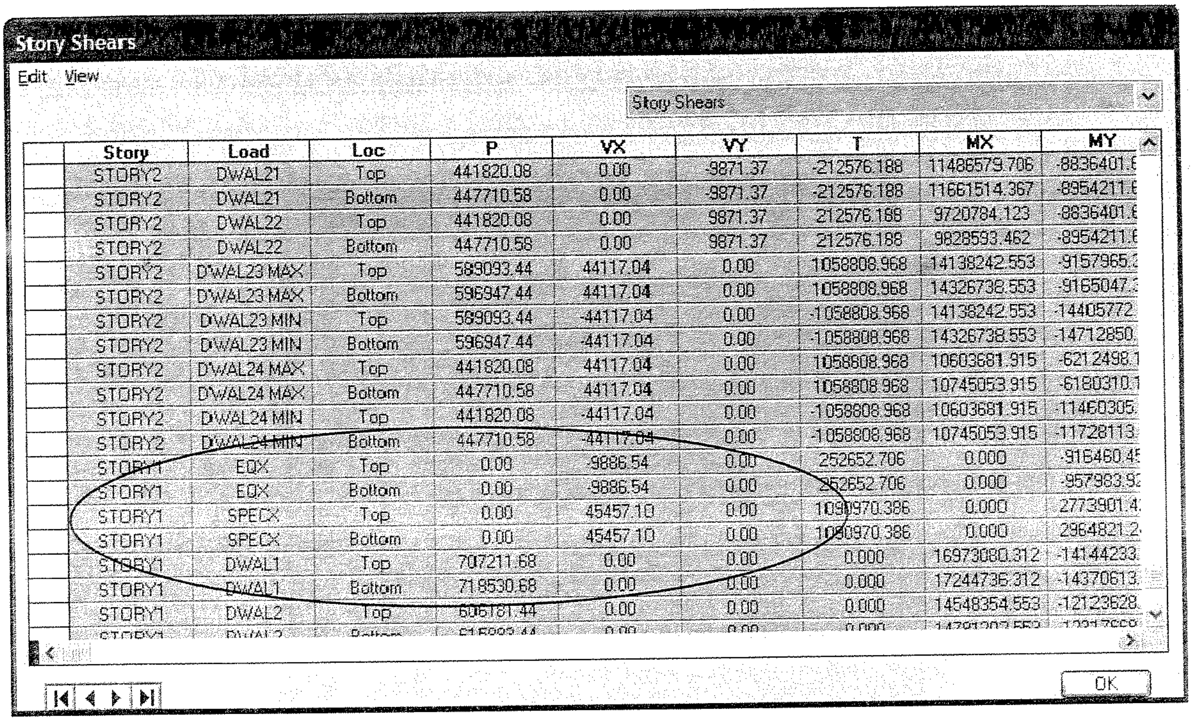
6. After the analysis run is completed, click **display menu** . ———▶ **Show Tables** to display the result of the run and make scaling for the dynamic force



Thin the next form will be displayed, from the pull down box of displayed data choose Story shears



Thin check the first story result and divide the base shear of EQX/base shear or SPECX to get the scale factor



Story Shears

Edit View

Story Shears

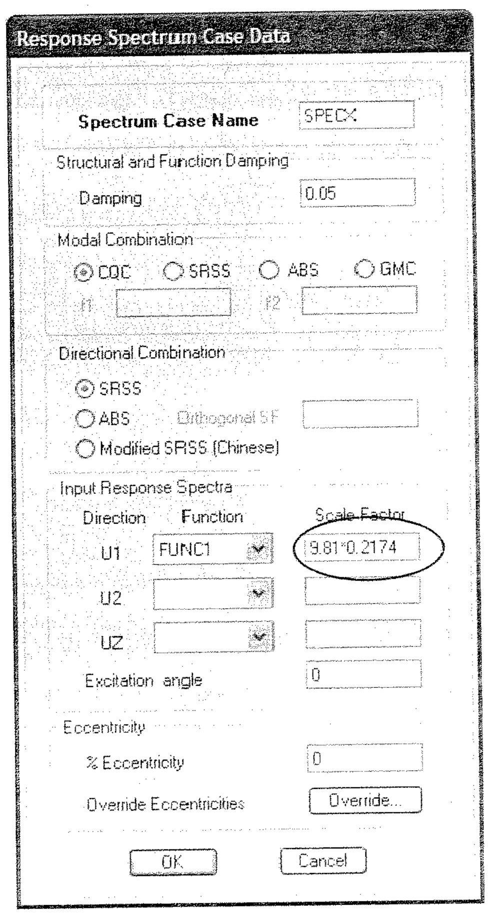
Story	Load	Loc	P	VX	VY	T	MX	MY
STORY2	DWAL21	Top	441820.08	0.00	-9871.37	-212576.188	11486579.706	-8836401.€
STORY2	DWAL21	Bottom	447710.58	0.00	-9871.37	-212576.188	11661514.367	-8954211.€
STORY2	DWAL22	Top	441820.08	0.00	9871.37	212576.188	9720784.123	-8836401.€
STORY2	DWAL22	Bottom	447710.58	0.00	9871.37	212576.188	9828593.462	-8954211.€
STORY2	DWAL23 MAX	Top	589093.44	44117.04	0.00	1058808.968	14138242.553	-9157965.:
STORY2	DWAL23 MAX	Bottom	596947.44	44117.04	0.00	1058808.968	14326738.553	-9165047.:
STORY2	DWAL23 MIN	Top	589093.44	-44117.04	0.00	-1058808.968	14138242.553	-14405772.
STORY2	DWAL23 MIN	Bottom	596947.44	-44117.04	0.00	-1058808.968	14326738.553	-14712850.
STORY2	DWAL24 MAX	Top	441820.08	44117.04	0.00	1058808.968	10603681.915	-6212498.1
STORY2	DWAL24 MAX	Bottom	447710.58	44117.04	0.00	1058808.968	10745053.915	-6180310.1
STORY2	DWAL24 MIN	Top	441820.08	-44117.04	0.00	-1058808.968	10603681.915	-11460305.
STORY2	DWAL24 MIN	Bottom	447710.58	-44117.04	0.00	-1058808.968	10745053.915	-11728113.
STORY1	EQX	Top	0.00	-9886.54	0.00	252652.706	0.000	-916460.4
STORY1	EQX	Bottom	0.00	-9886.54	0.00	252652.706	0.000	-957983.9
STORY1	SPECX	Top	0.00	45457.10	0.00	1090970.386	0.000	2773901.4
STORY1	SPECX	Bottom	0.00	45457.10	0.00	1090970.386	0.000	2964821.2
STORY1	DWAL1	Top	707211.68	0.00	0.00	0.000	16973080.312	-14144233.
STORY1	DWAL1	Bottom	718530.68	0.00	0.00	0.000	17244736.312	-14370613.
STORY1	DWAL2	Top	606181.44	0.00	0.00	0.000	14548354.553	-12123628.

OK

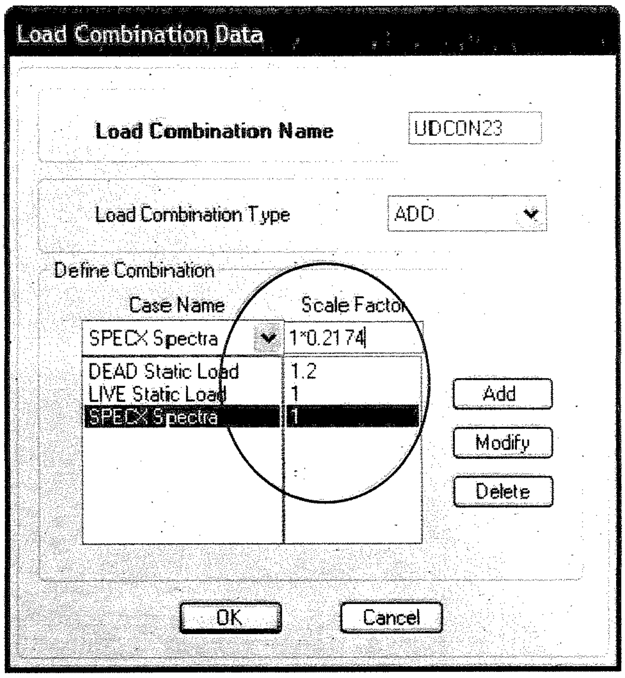
- You have 2 choice for the scaling

1. Multiply the scale factor x9.81 and put this value in the Response spectrum case Data. but this method need to run the modal again

- Thin create the load of combination and complete your design steps

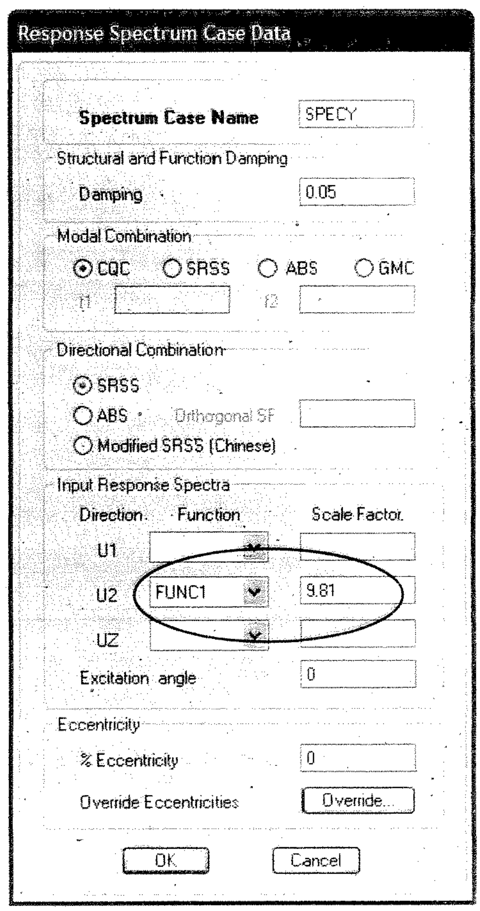


2. create the load of combination and multiply the coefficient of the spectrum by the scale factor .and this method not need from you to run the model again

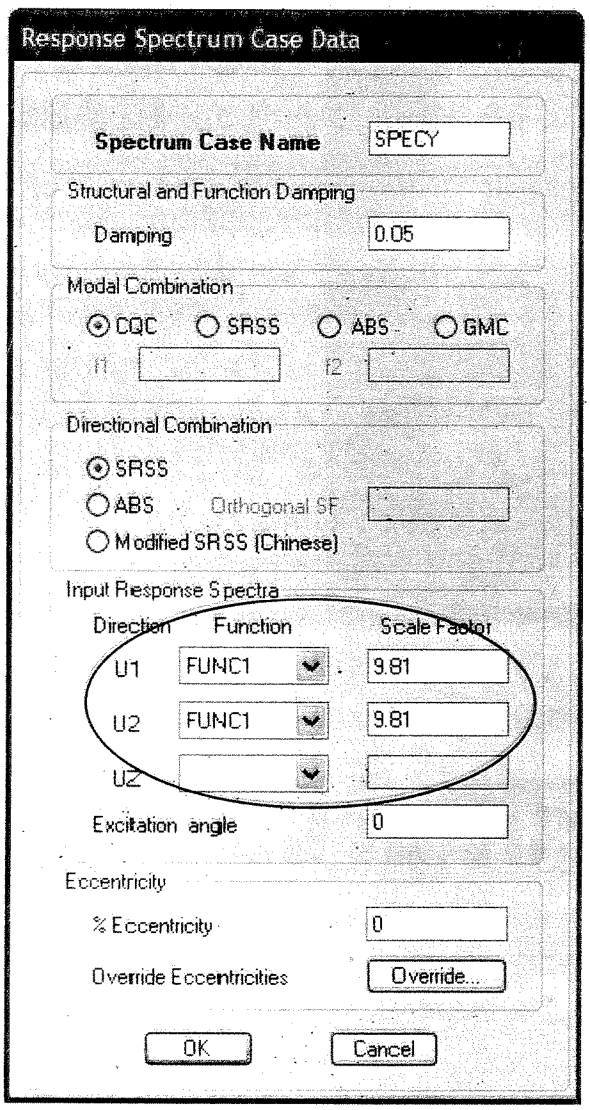


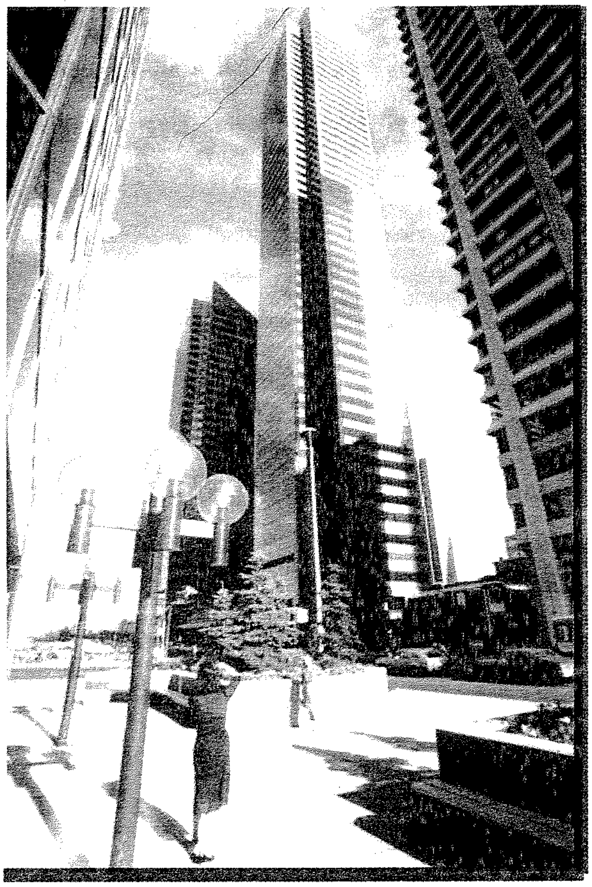
- after you change the scale in all load of combination begin you design steps

3. repeat the previous steps in Y direction and the same method for scaling with reference to EQY



There are some conservative opinion for the combination of the spectrum cases and the most conservative one to take the response in the both direction in the same case and scale the both. But till now this opinion not used in the wide range





HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

Sequential Construction

Chapter 8

Why we need to use Sequential Construction:

1. In cast-in-place reinforced concrete structures, the amount of support shortening before slab installation is of no importance, since the forms are usually leveled at the time the concrete for each floor slab is placed.

2. To analyze incremental construction, the structure is modeled as a series of stages. In the most common case of construction sequencing, each subsequent active group will include the previous active group plus some additional objects.

3. the straining actions which is created in each structure elements are effected directly by the sequence of construction ,and it is varies from sequence of construction to another sequence because the values of displacement for the points effected by the method of construction where forms are usually leveled at the time the concrete for each floor slab is placed.

4. The big difference in result for the members supporting planted elements like columns and walls.

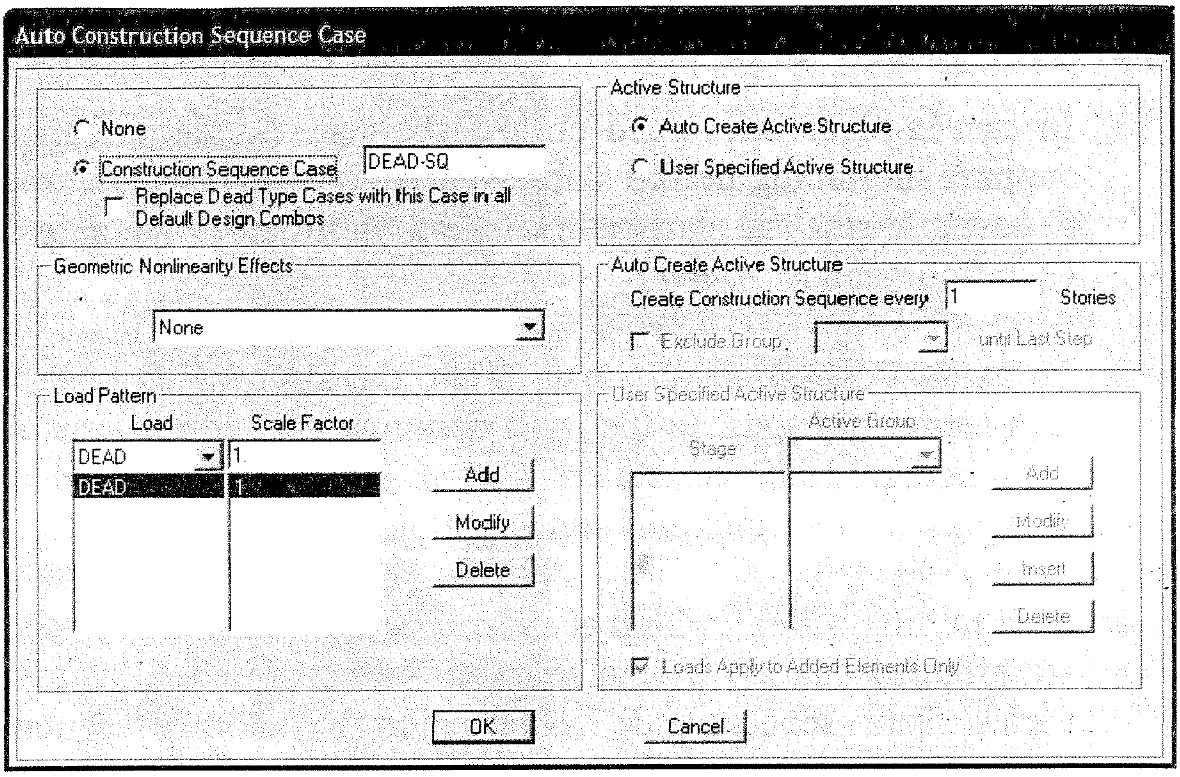
Assumption of the example:

1. the sequence for construction is floor by floor

2. we will use the full dead load during construction .(In reality we can use OW only plus the construction loads)

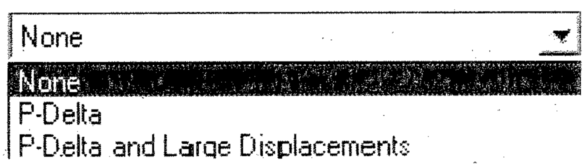
Steps for Construction sequence:

1. Click the **Define menu** ——► **Set Analysis Options** command to display the Analysis Options form.



2. Select **Construction Sequence Case** option and keep the name of the case as it is (DEAD-SQ), if you want to use the result of **CONSTRUCTION SEQUENCE** in design instead the dead load case check the box of replace **Dead Type Cases with this Case in All Default Design Combos.**

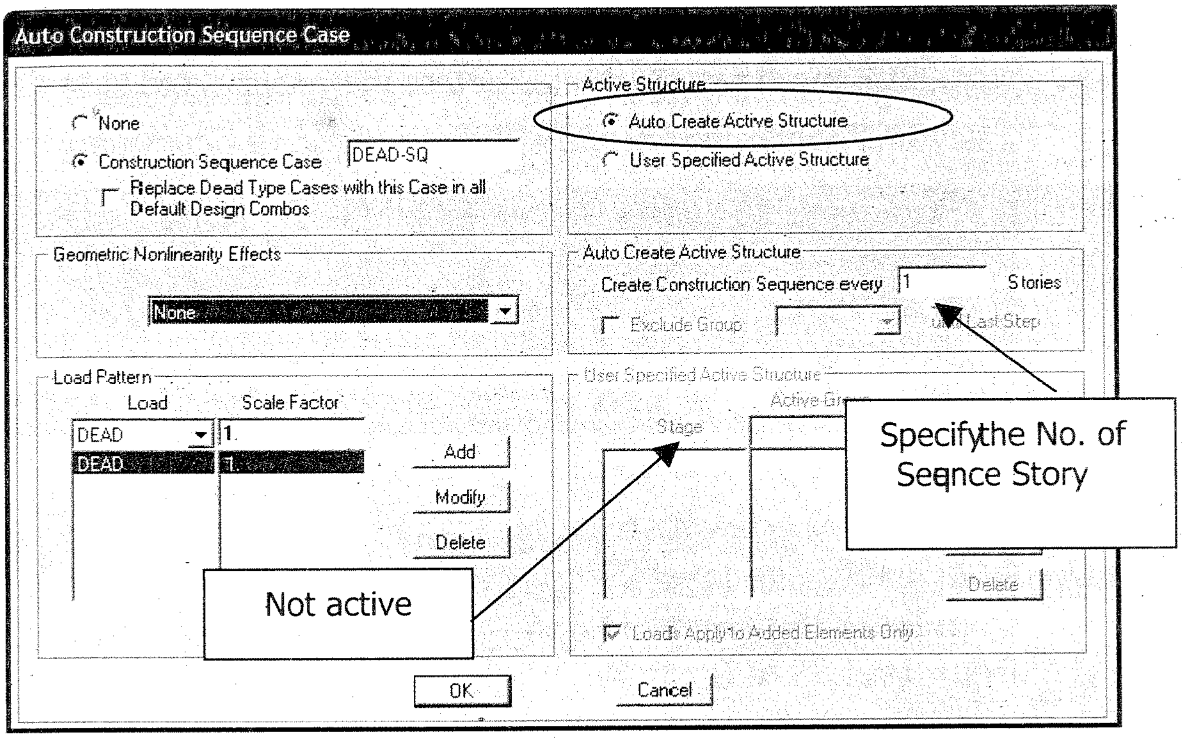
3. From **Geometric Nonlinearity Effects** drop-down list, choose **None** (is recommended that the analysis be performed first without P-delta)



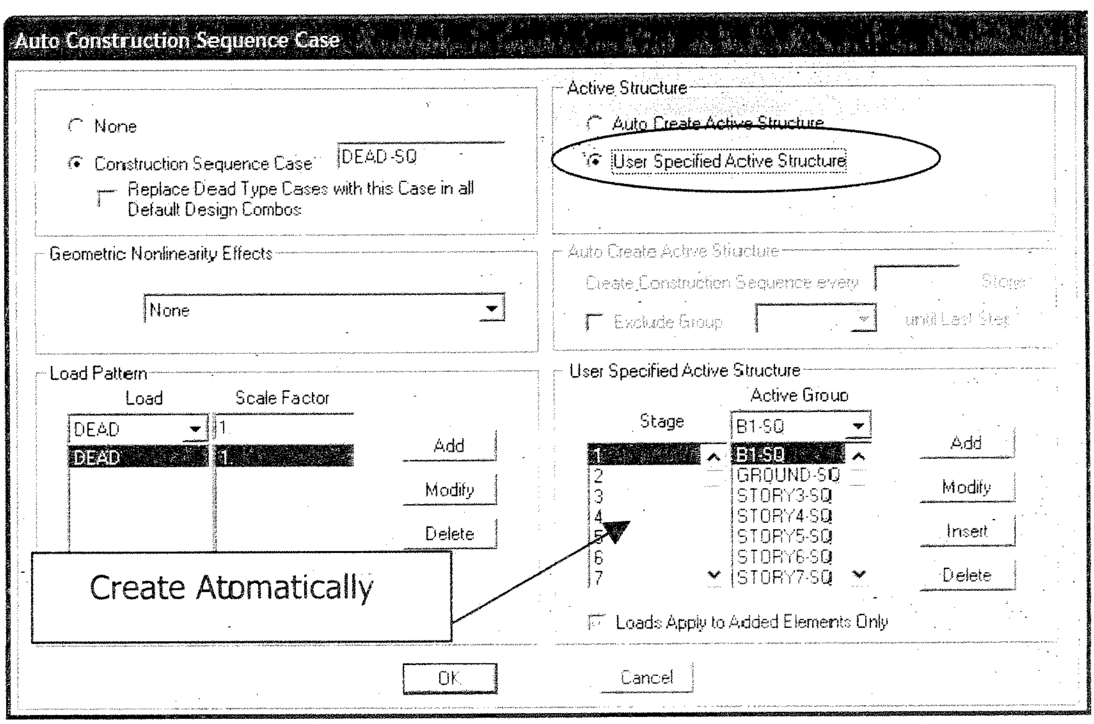
4. In Load Pattern area. Use the edit boxes, buttons, and display lists in this area to specify the Load Pattern we will use the default of the program Dead load case with scale factor 1.

5. **From Active Structures** options. We will Choose how the structure will be defined:

- **Auto Create Active Structure** option. Select this option to have the program automatically create the active structure.



- **User Specified Active Structure** option. Select this option to define the construction sequence manually.

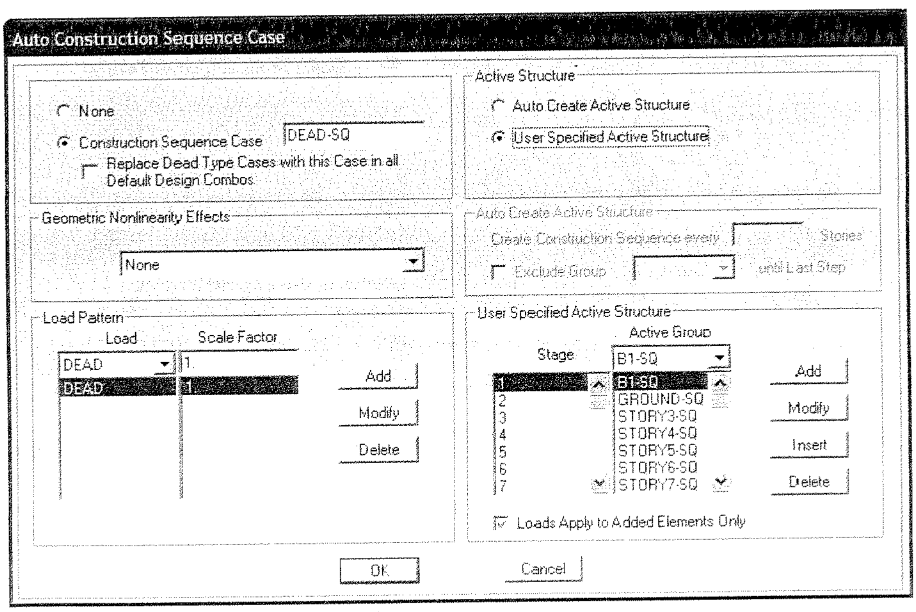


- User Specified Active Structure area. By default, the program creates an initial sequence on the basis of Story levels (e.g., Story1-SQ, Story2-SQ and so on). Use the buttons, drop-down list and display area in this area to add, modify, insert, and delete the groups and stories in each construction sequence.

Note: if the structure has different sequence of construction make each as a group, thin add in active Group cell for example you can divide the sequence of construction for the core to 5 stage and assign each 8 stories as group if you use slip form in constructions to be as follow

Core1	from level 1 to level 8
Core2	from level 9 to level 16
Core3	from level 17 to level 25
Core4	from level 26 to level 34
Core5	from level 35 to level 40

- **Loads Apply to Add Elements only** check box. When this check box is checked the specified Load Pattern will be applied to the added elements only. This box usually should be checked if more than one stage exists. When the check box is unchecked, the specified Load Pattern is applied to the entire structure.



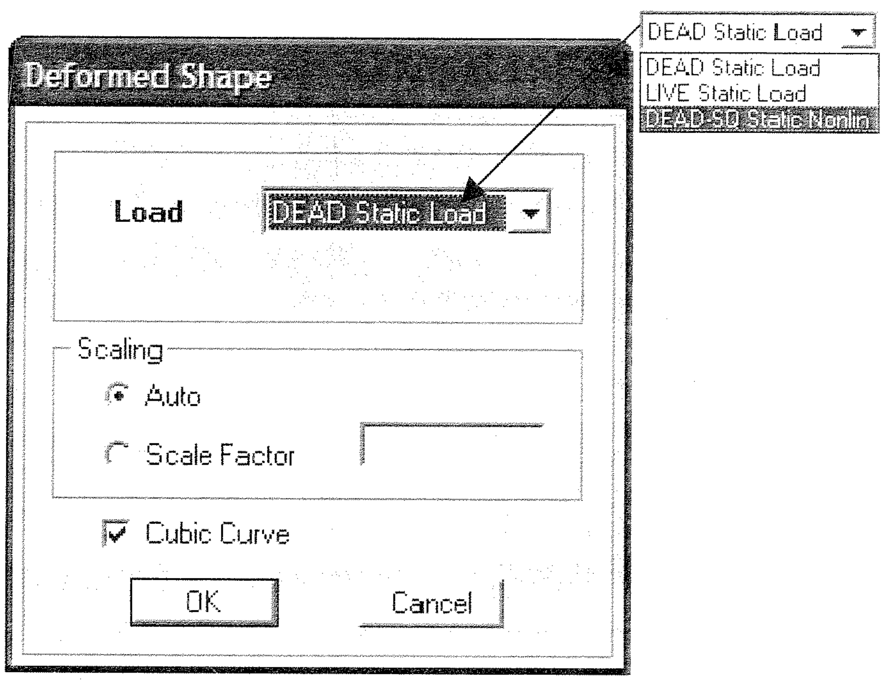
6. Keep the form as shown in the fig. thin click OK
7. Click **Run Analysis** Button

8. After the analysis completed Click
   Analyze menu ──► **Run Construction Sequence Analysis** command to begin the construction sequence analysis.

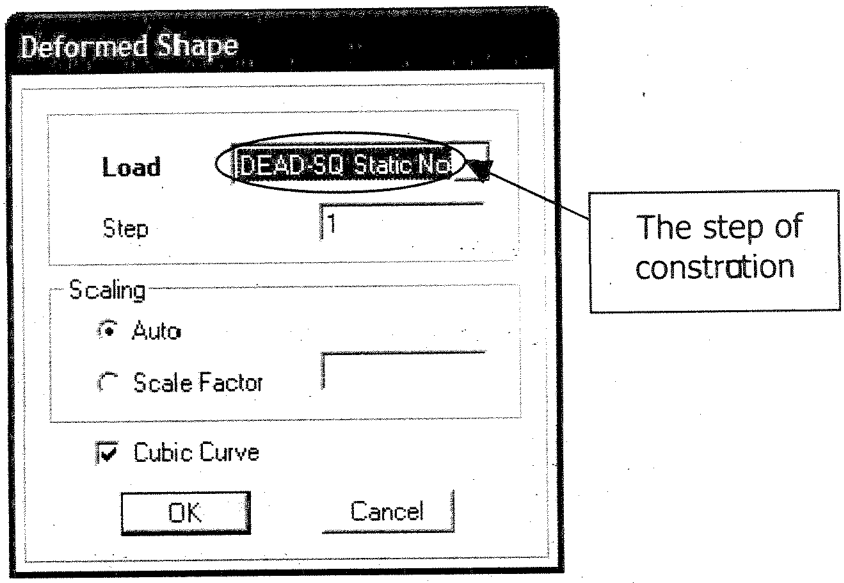
- Example

  - we ill see together in the next example the difference between normal analysis and construction sequence analysis

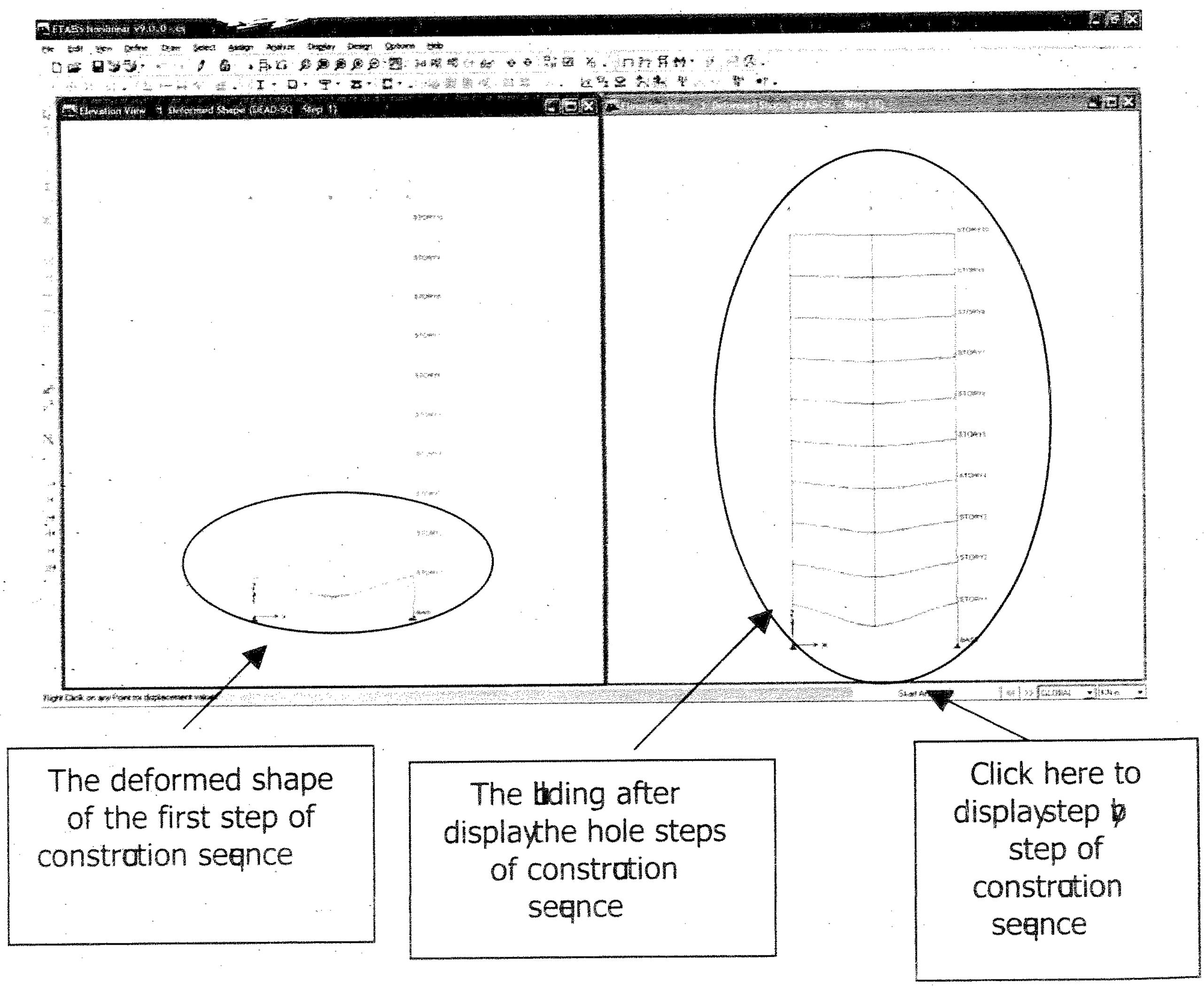
  - The next form show the display of construction sequence result. Click **Display** ──► **Show deformed shape** ,thin the next form will be displayed



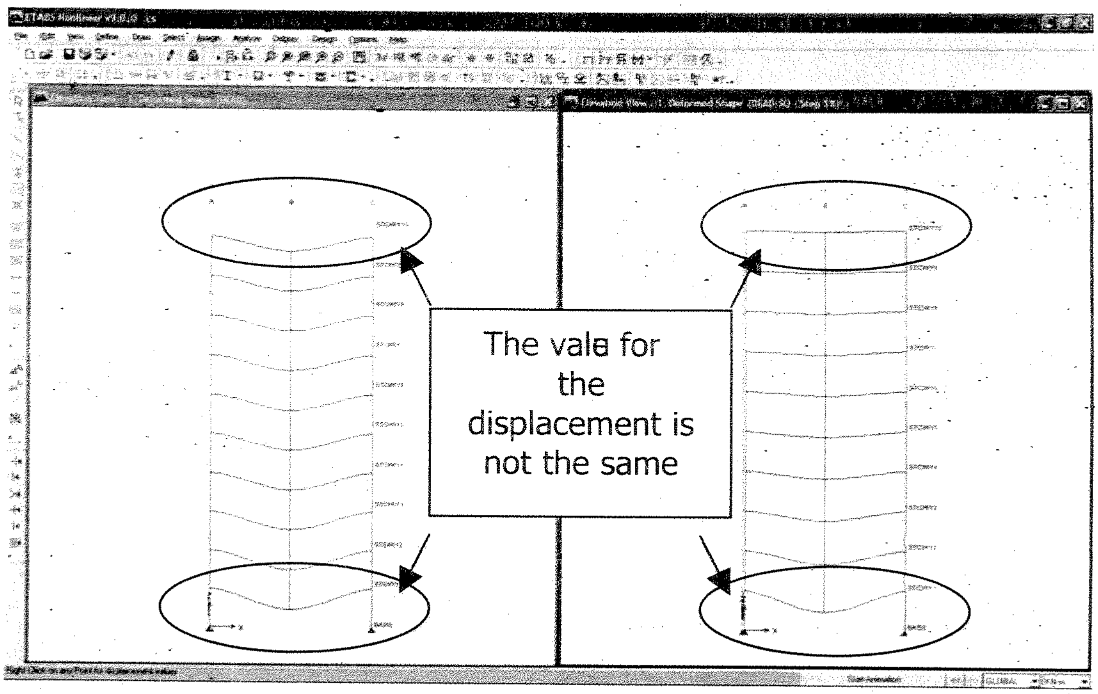
  - Choose the sequential load from Load drop-down list thin the form will be as shown

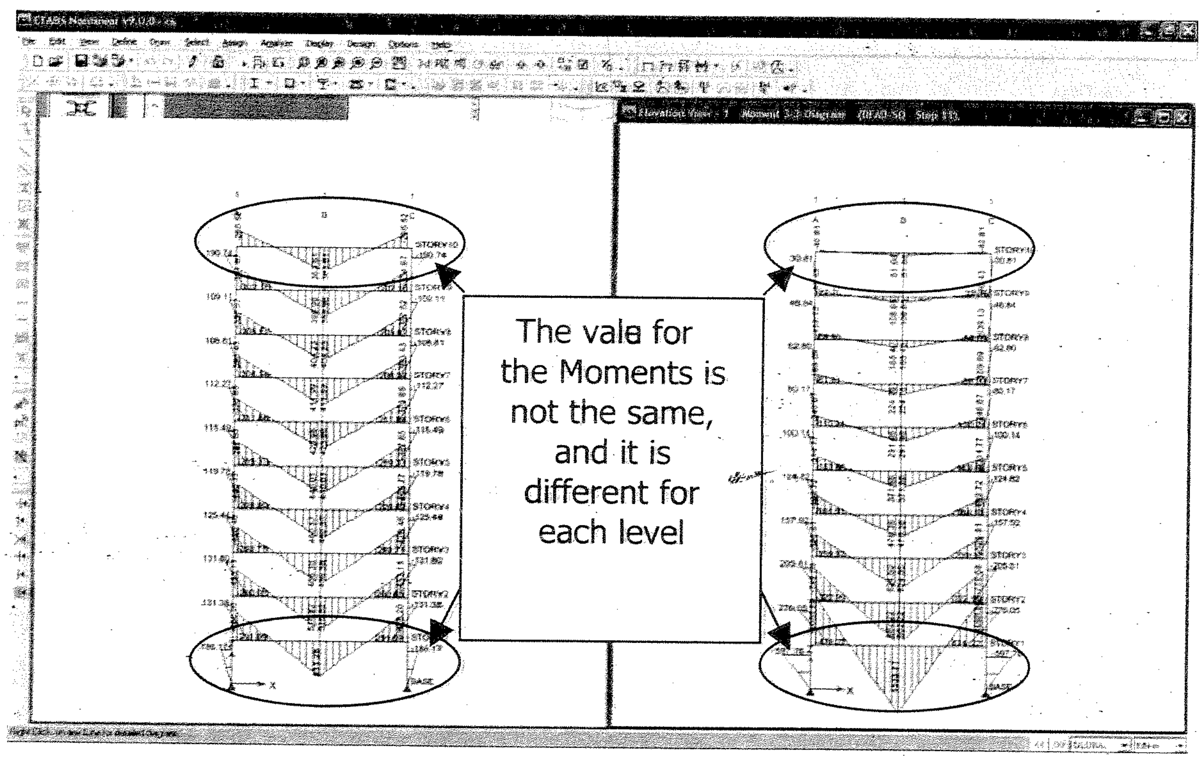


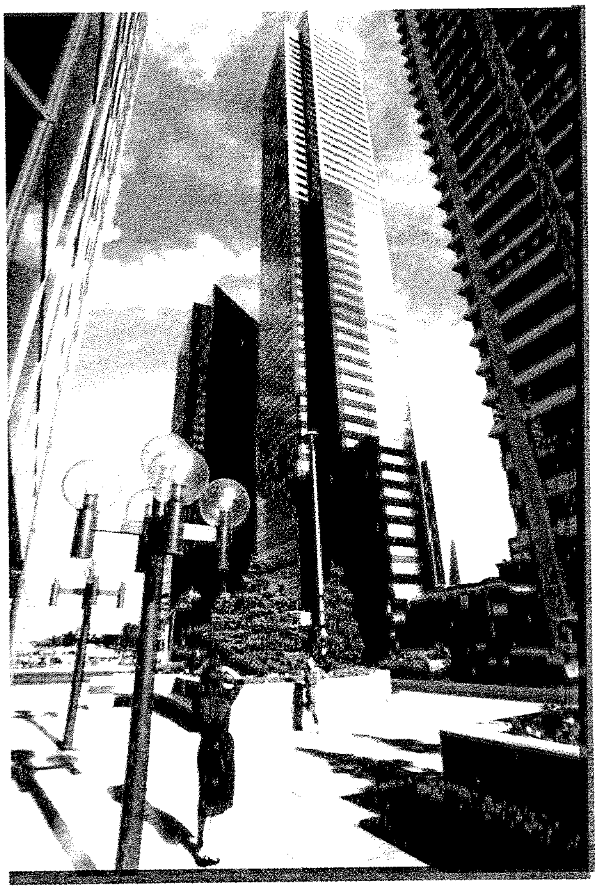
- Click OK to see the first step thin click for the arrow in the down of screen to see all the steps



- There are big difference in result between normal analysis and construction sequence analysis as shown in the next 2 figure.







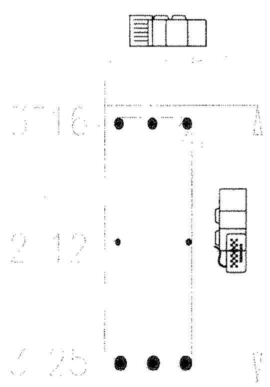
HOW TO MODEL AND DESIGN
HIGH RISE BUILDING USING
ETABS Program

Section Designer

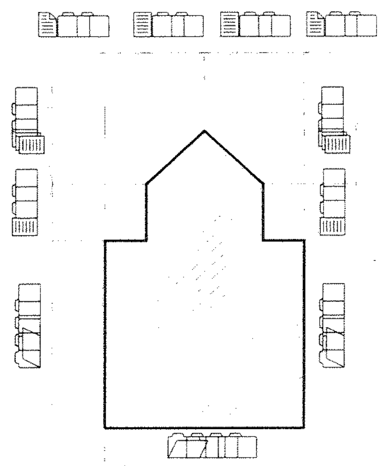
Chapter 9

Etabs Program has the ability to define unusual frame element sections like (beams and columns) and unusual walls sections, and also calculate the section properties for those sections.

1. **Section Designer for Frame Elements:** We will define two different shapes to the program one for beam and another one for column as an example to show to you how you can use **Section Designer**.



Sec (1)

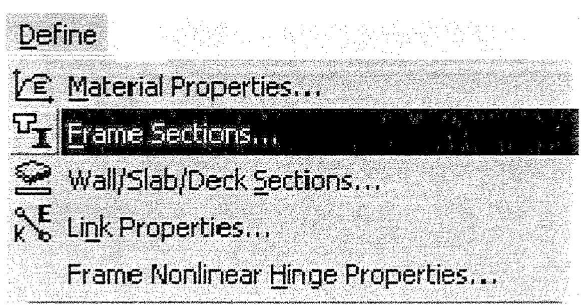


Sec (2)

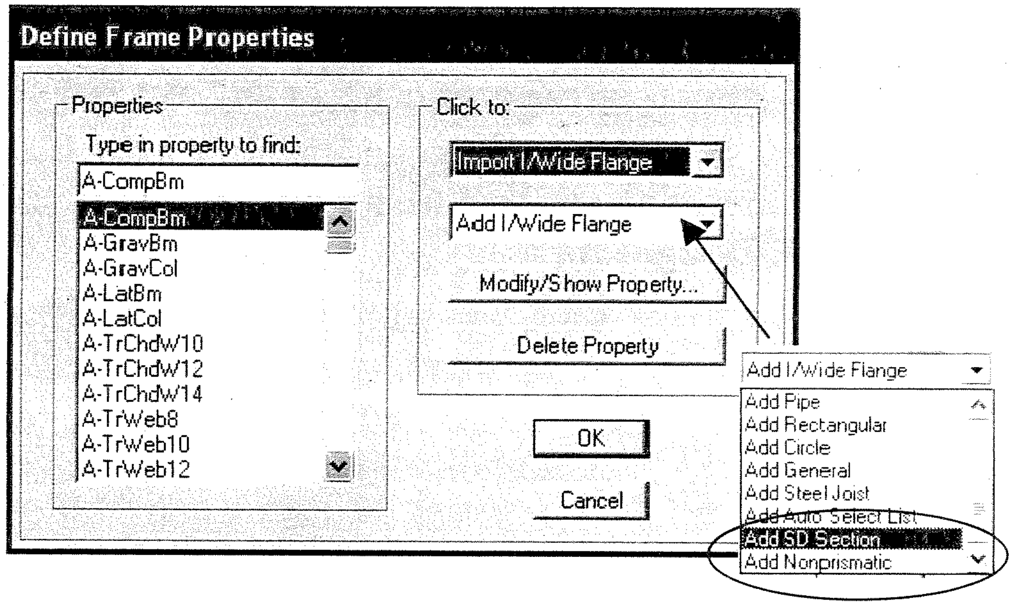
All dimensions are in mm

Section (1)

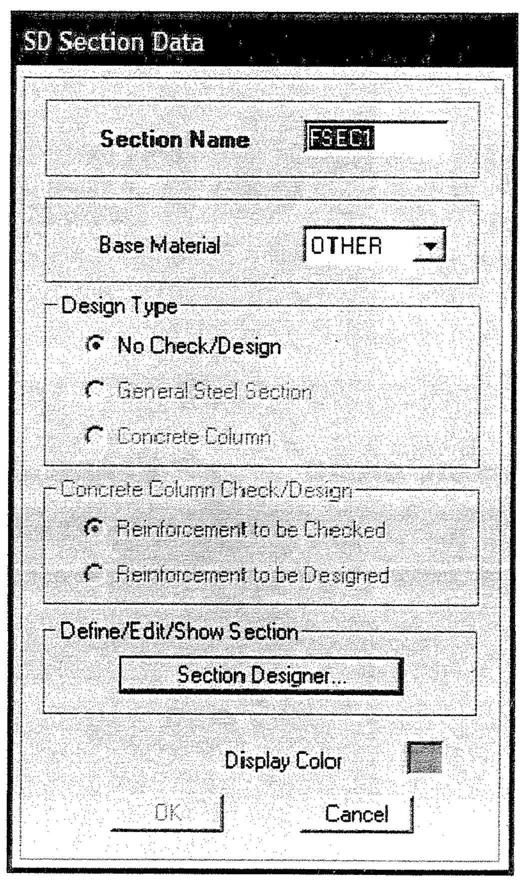
1. Click the **Define menu** ⟶ **Frame Sections ….**



Or click Define Frame Sections button which will Display the define Frame Properties form

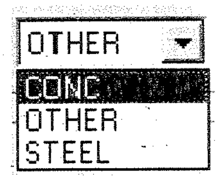


2. Click the Second drop-down box that reads **"Add I/Wide Flange"** and Choose Add SD Section from the list, thin the form of SD Section Data will be Displayed



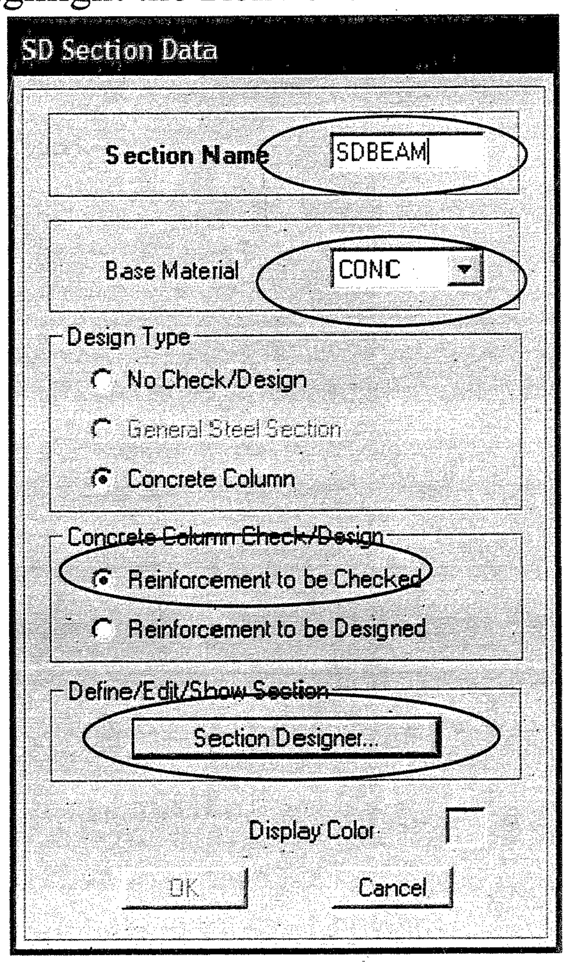
- In this form change the section name to SD BEAM or specify any another name

- Click the drop-down box of Base material and choose conc. or any material you was defined before

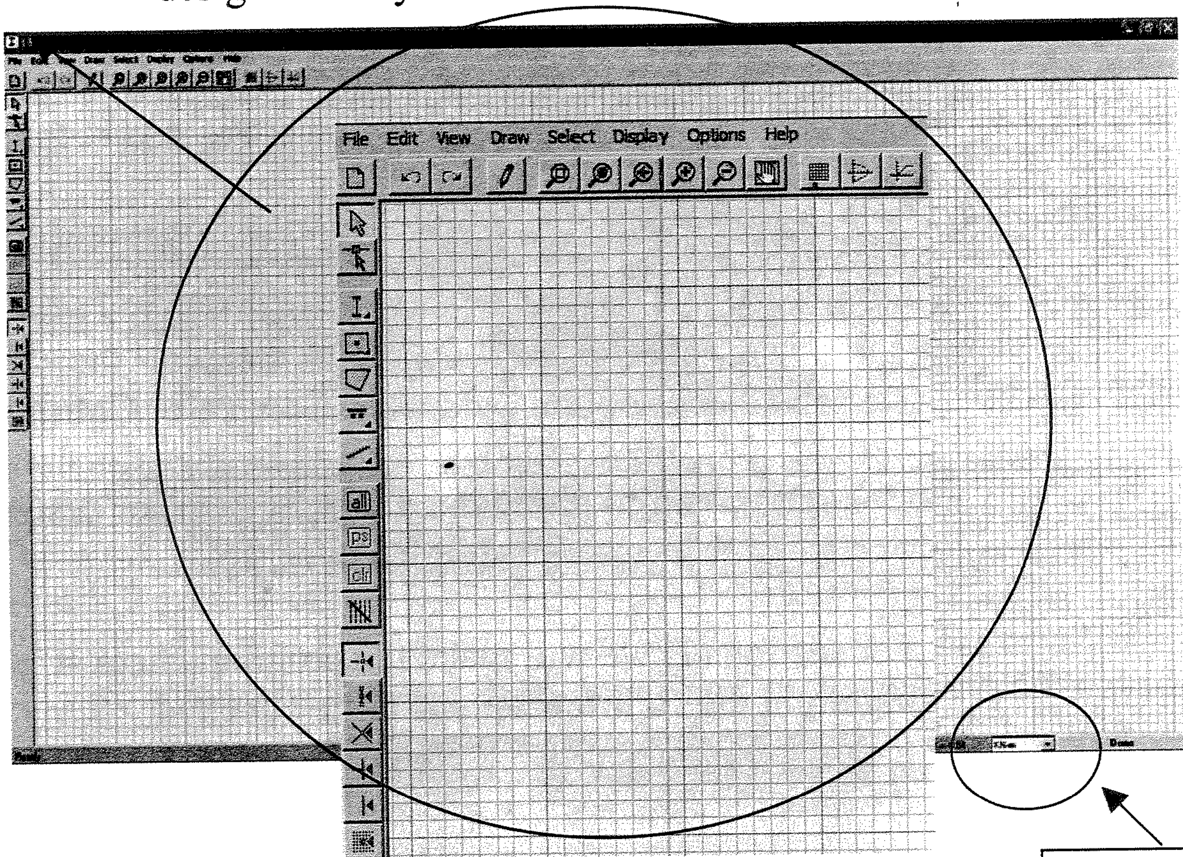


- After you choose any concrete material the No Design/Check option and the **Concrete Column** option will be available, highlight the Concrete Column option to allow the program to make design or check for the section after analysis

- For section (1) we will make the program check it ,so highlight the **Reinforcement to be checked**

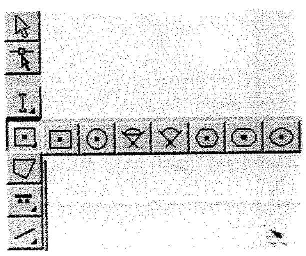


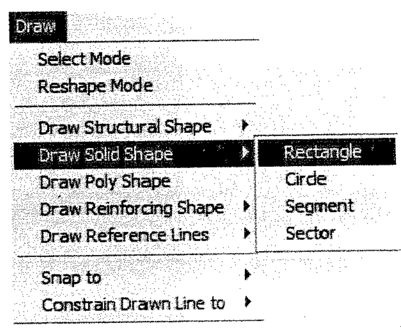
3. Click the **Section Designer** button to go to the section designer utility and draw the section on the next screen



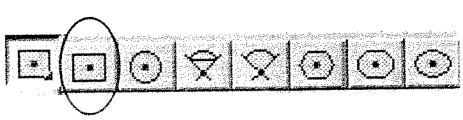
The nits will b the same as the model nits

- Click **Draw menu** ——▶ **Draw Solid Shape** ——▶ **Rectangle**

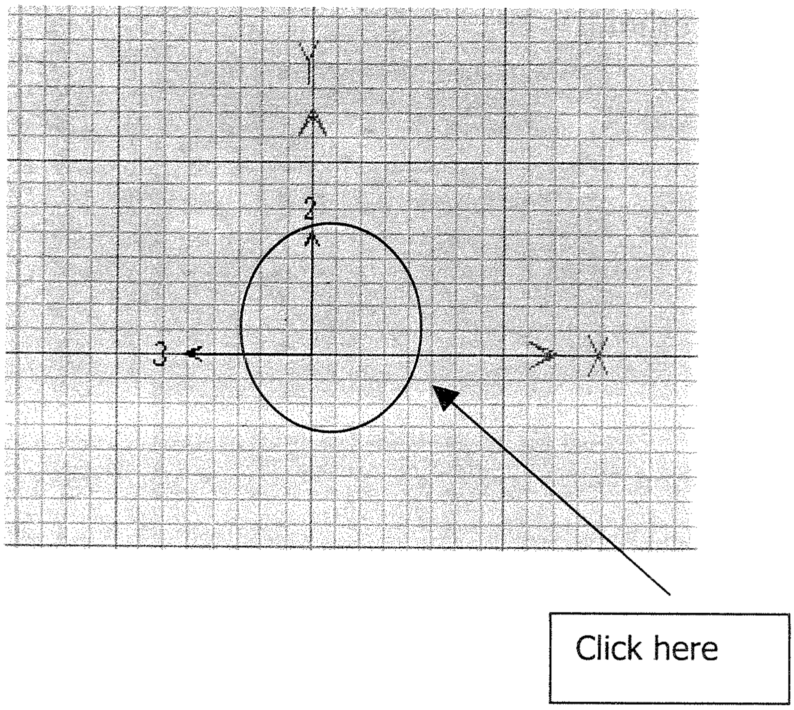




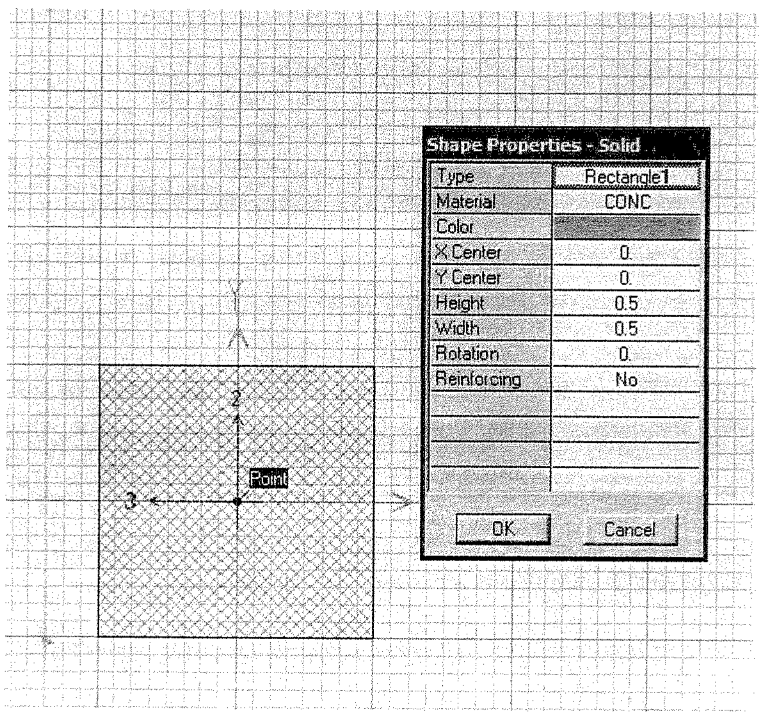
- Or Click Draw Solid Shape button thin Rectangular

 Thin the crasser for drawing

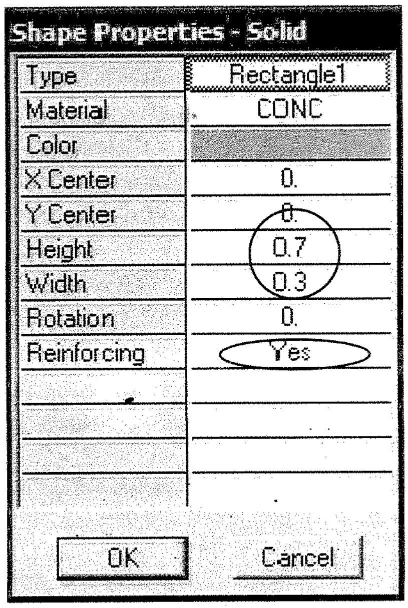
Rectangular Shape will be displayed, click on the Guide intersection



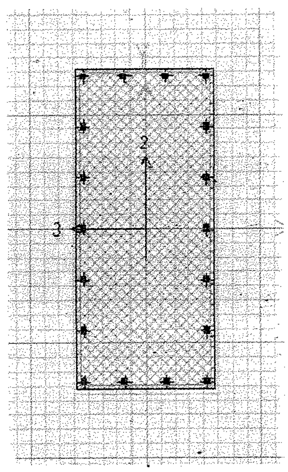
- Thin the Rectangular shape will appear ,click escape and left click on the shape by mouse thin right click to display the Shape Properties -Solid



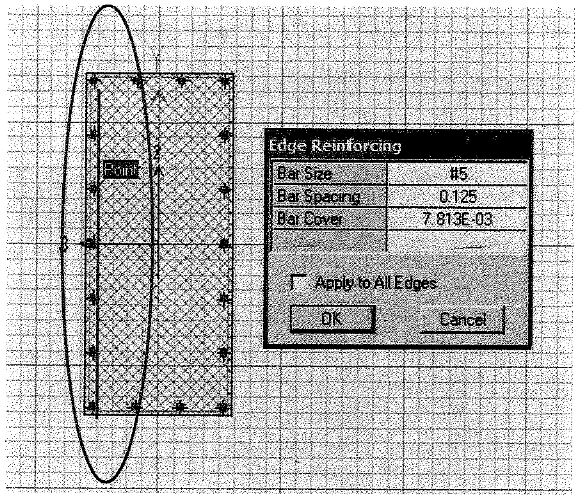
- Chang the size of the section to be 30x70 and from Reinforcement drop-down box highlight **Yes** to add Reinforcement to the section



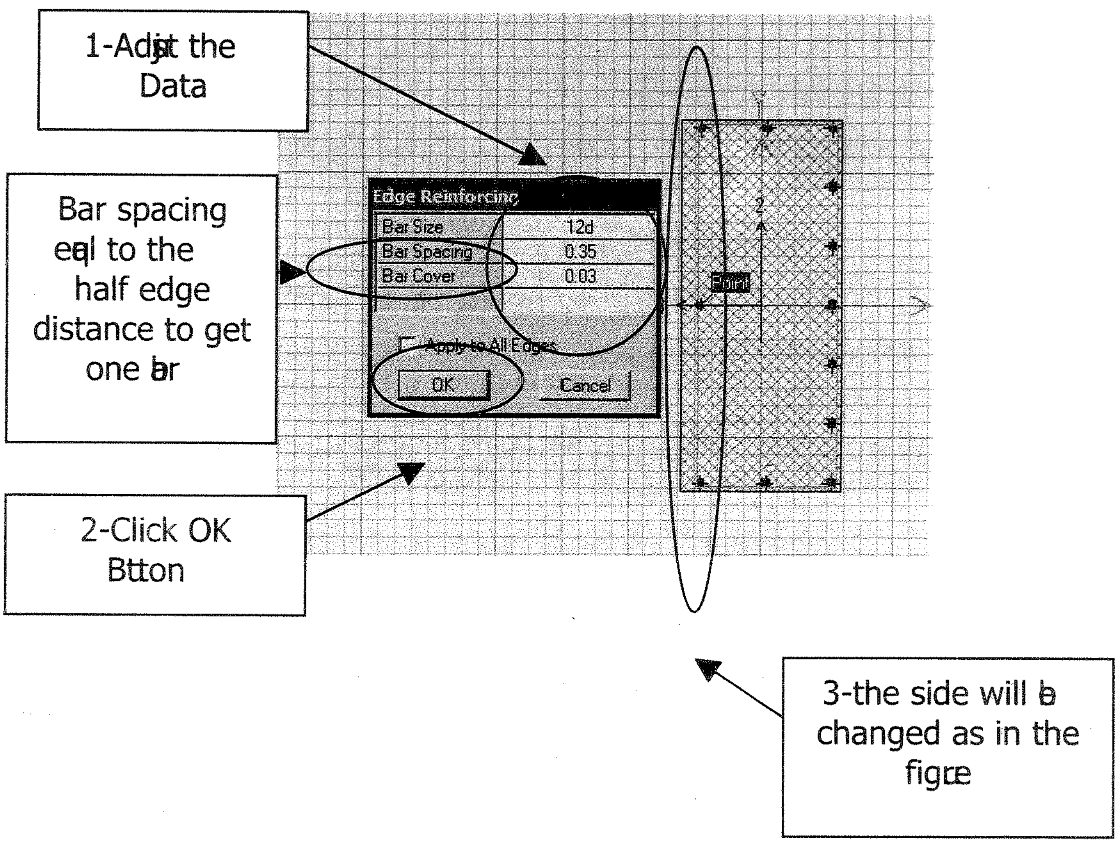
- Thin the section will be as in the next Figure



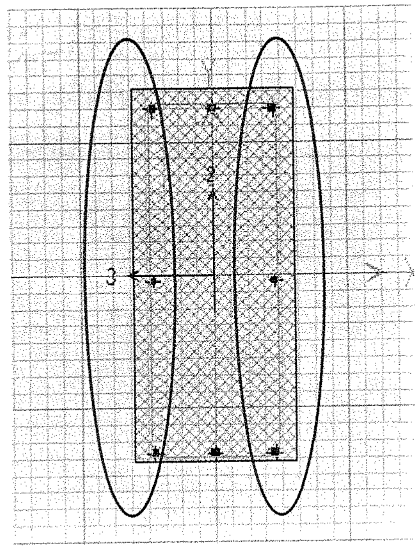
- To adjust the reinforcement click on the left edge by the mouse thin click the right button of the mouse to display the Edge Reinforcement Form



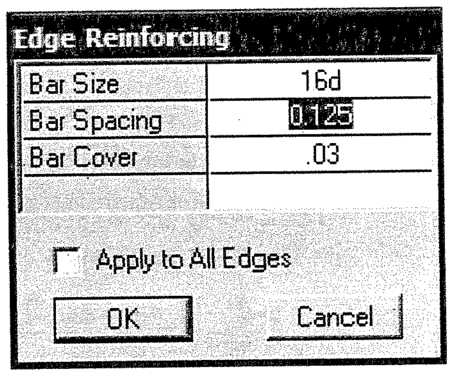
- From the Edge Reinforcement form adjust **the side Bar size** to be d12 and **The Bar Spacing** to be .35 and **The Bar Cover** to be .03



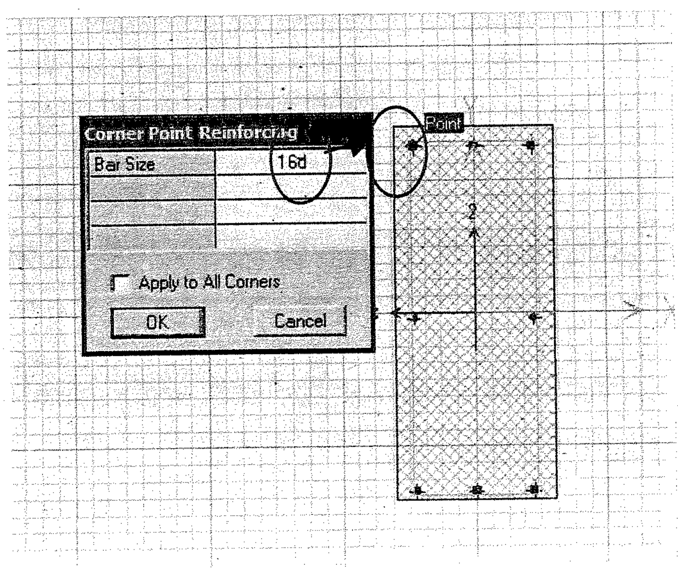
- Repeat the previous step for the left side ,thin the shape will be as in the figure



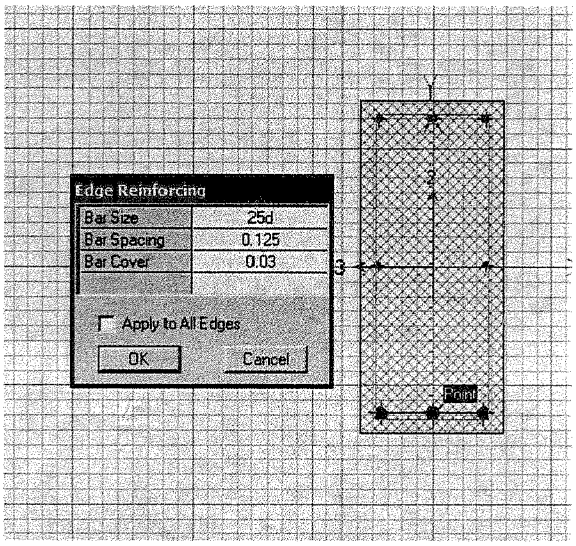
- For the top reinforcement repeat the same steps and enter the following data



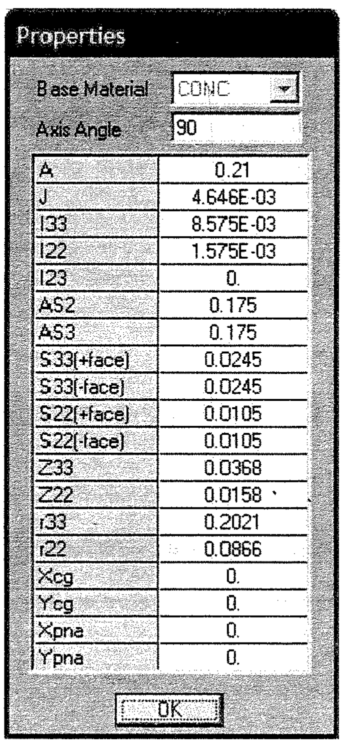
- For the corner bars click on the Bar by the mouse thin click the right button of the mouse to display the corner Point Reinforcement Form modify bar size to be 16d



- For the bottom reinforcement repeat the same steps for the top reinforcement and enter the data as in the figure



- To display the properties of the section click **Display menu** . ——► **Show Section Properties**, thin the next form of the section properties will be displayed



Properties

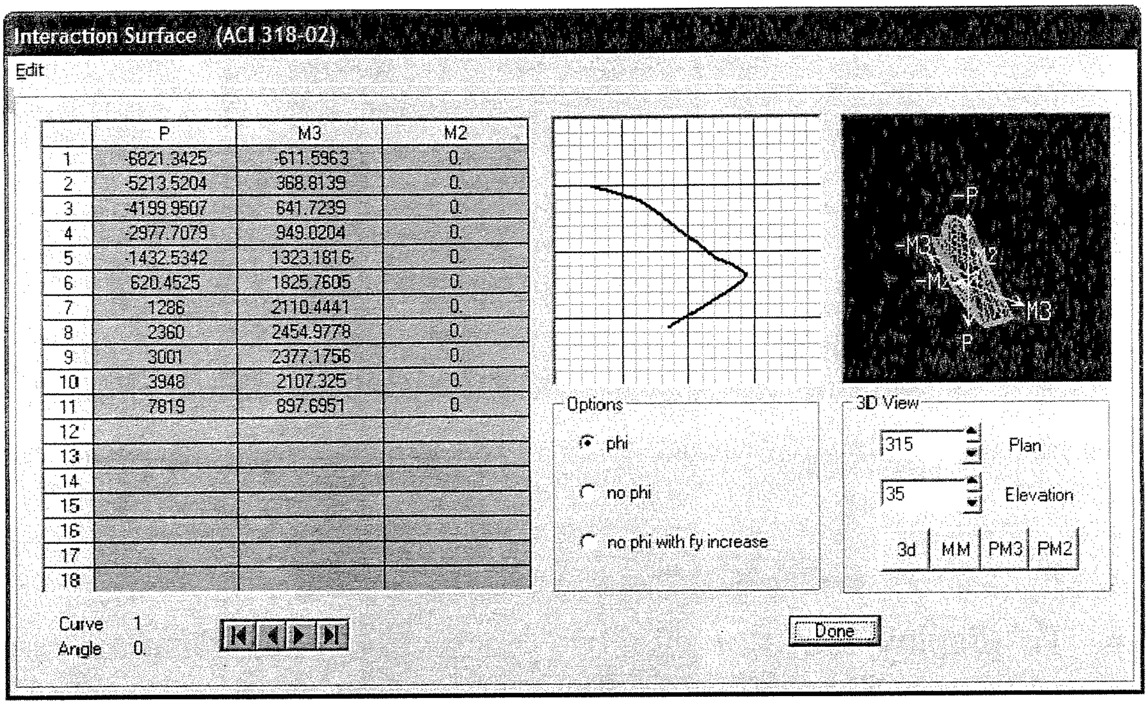
Base Material: CONC

Axis Angle: 90

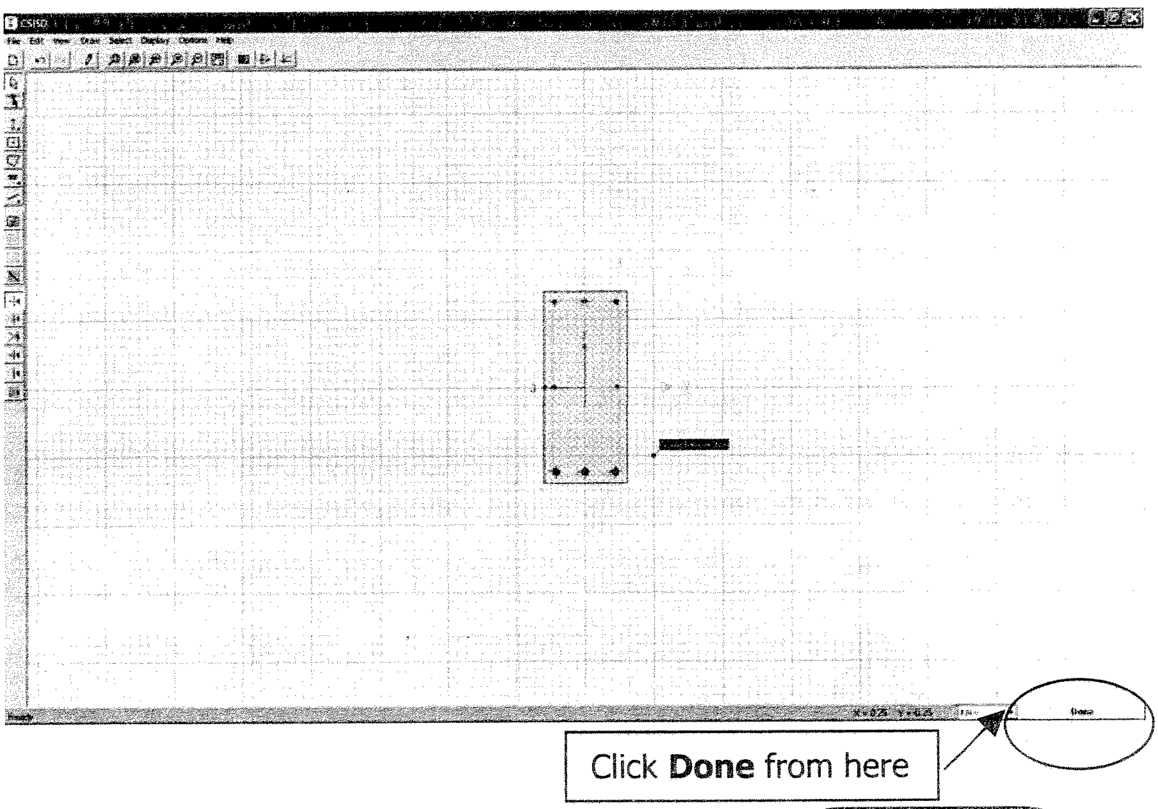
A	0.21
J	4.646E-03
I33	8.575E-03
I22	1.575E-03
I23	0.
AS2	0.175
AS3	0.175
S33(+face)	0.0245
S33(-face)	0.0245
S22(+face)	0.0105
S22(-face)	0.0105
Z33	0.0368
Z22	0.0158
r33	0.2021
r22	0.0866
Xcg	0.
Ycg	0.
Xpna	0.
Ypna	0.

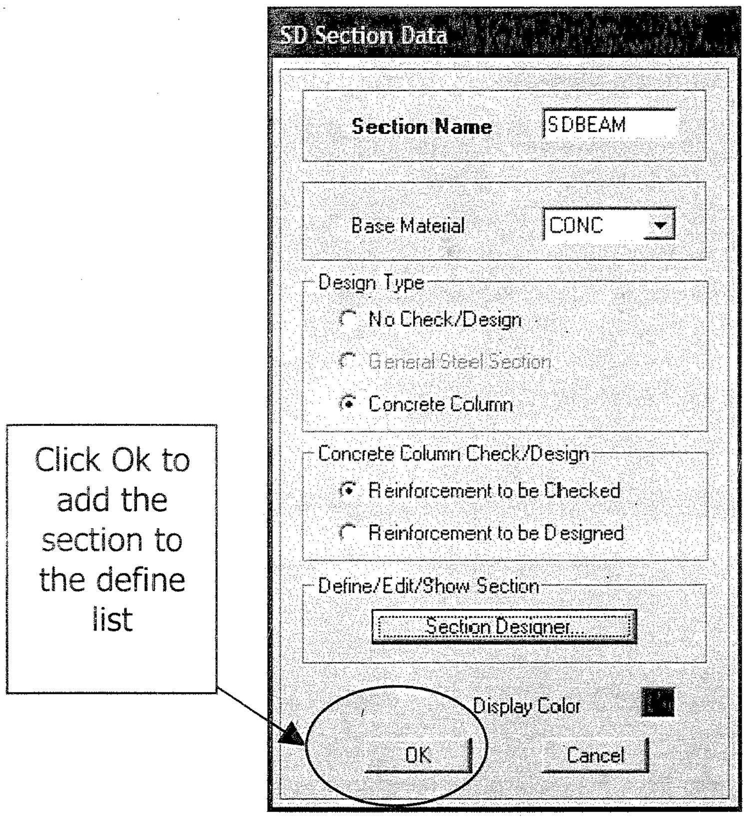
OK

- To display the interaction diagram for the section click **Display menu** ——► **Show Interaction Surface**, thin the next form of the Interaction diagram for the section according to your design code will be displayed

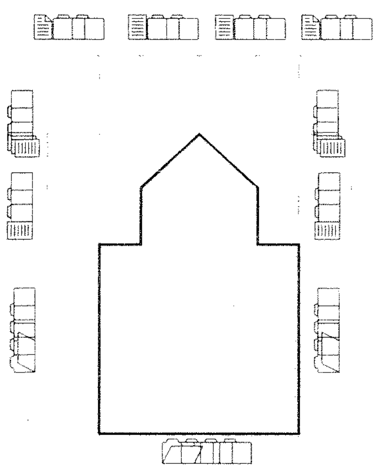


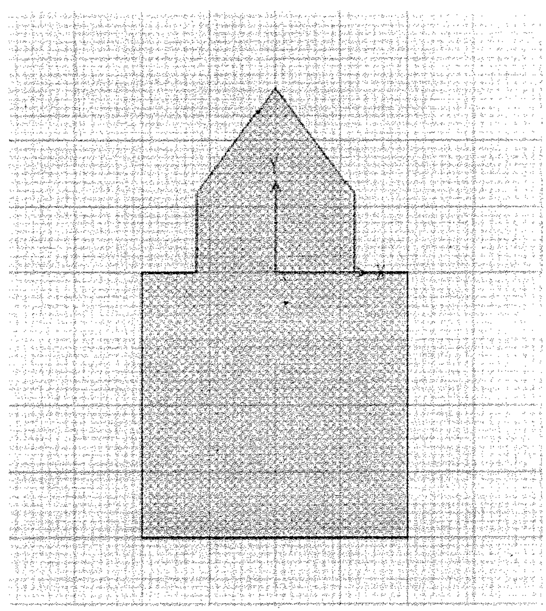
- In the end the beam section will be as in the next figure and ready to be checked , click **Done** from the form and thin **Ok** from the main form



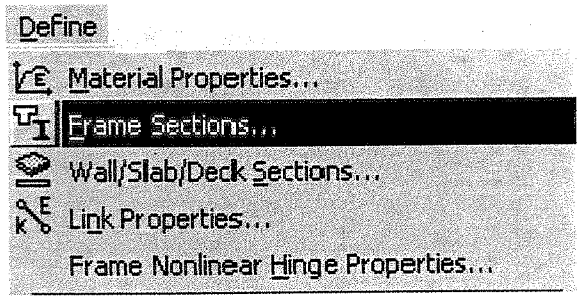


Section (2)

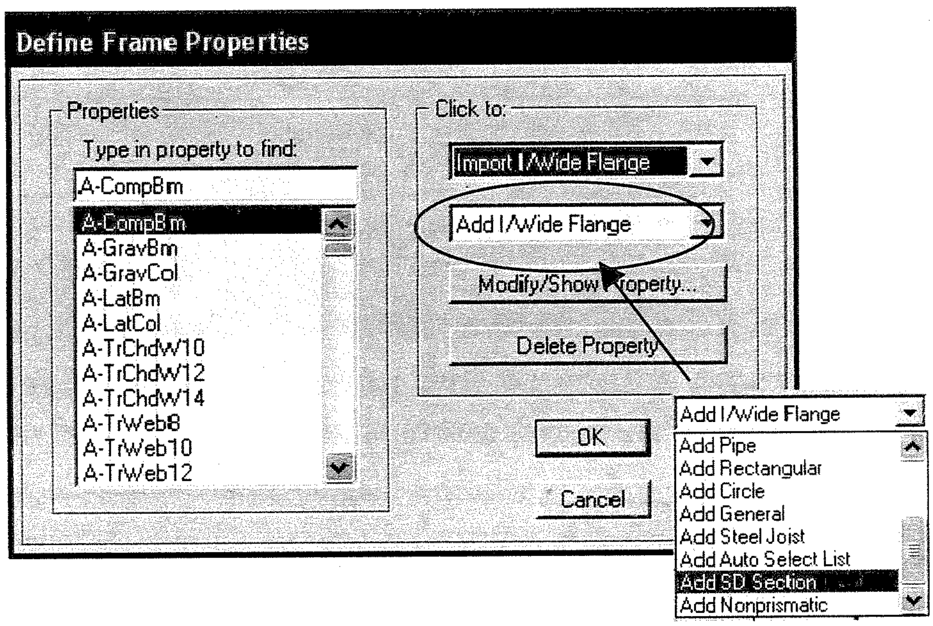




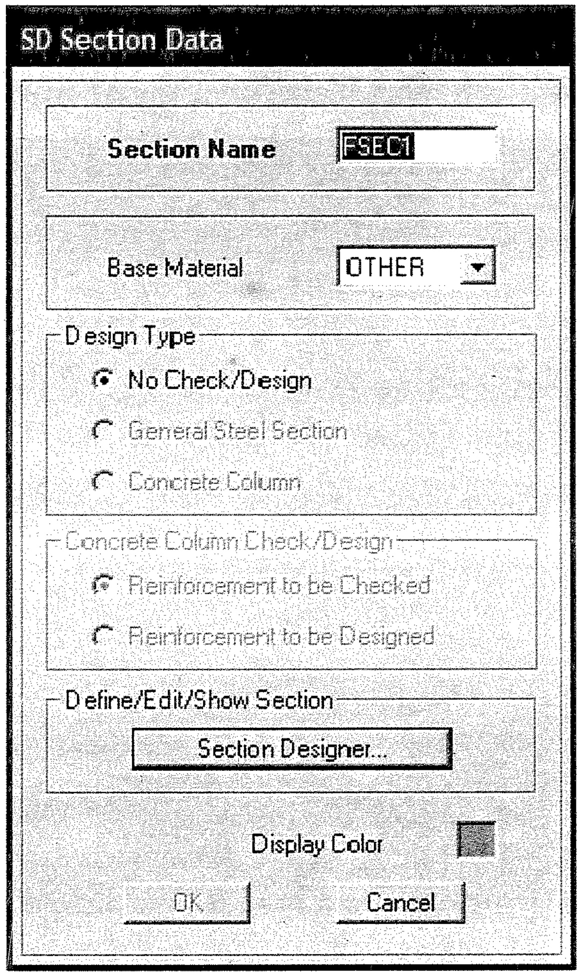
1. Click the **Define menu** ⟶ **Frame Sections ....**



Or click Define Frame Sections button which will Display the define Frame Properties form

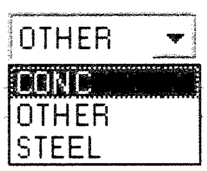


2. Click the Second drop-down box that reads **"Add I/Wide Flange"** and Choose Add SD Section from the list, thin the form of SD Section Data will be Displayed



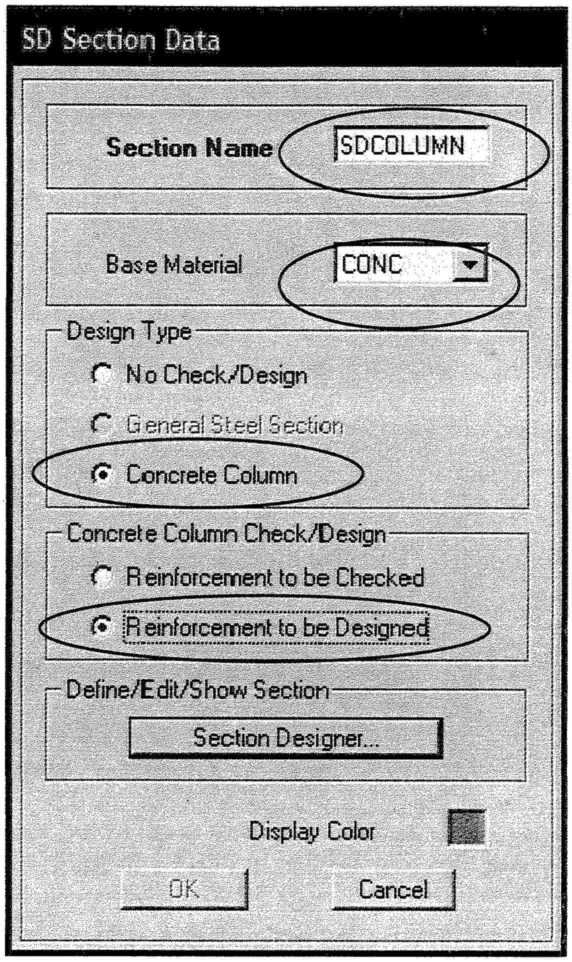
- In this form change the section name to SD Column or specify any another name

- Click the drop-down box of Base material and choose conc. or any material you was defined before

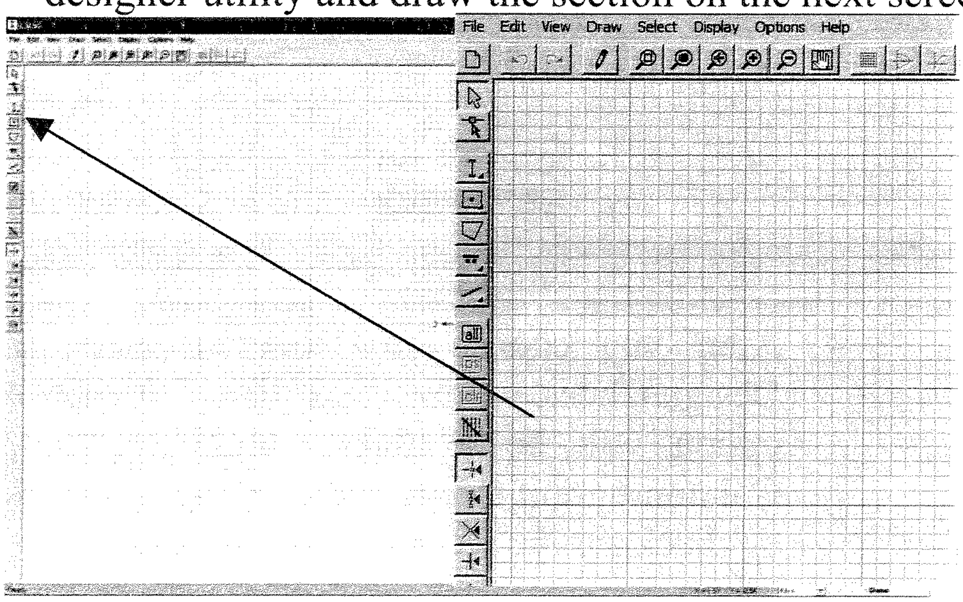


- After you choose any concrete material the No Design/Check option and the **Concrete Column** option will be available, highlight the Concrete Column option to allow the program to make design or check for the section after analysis

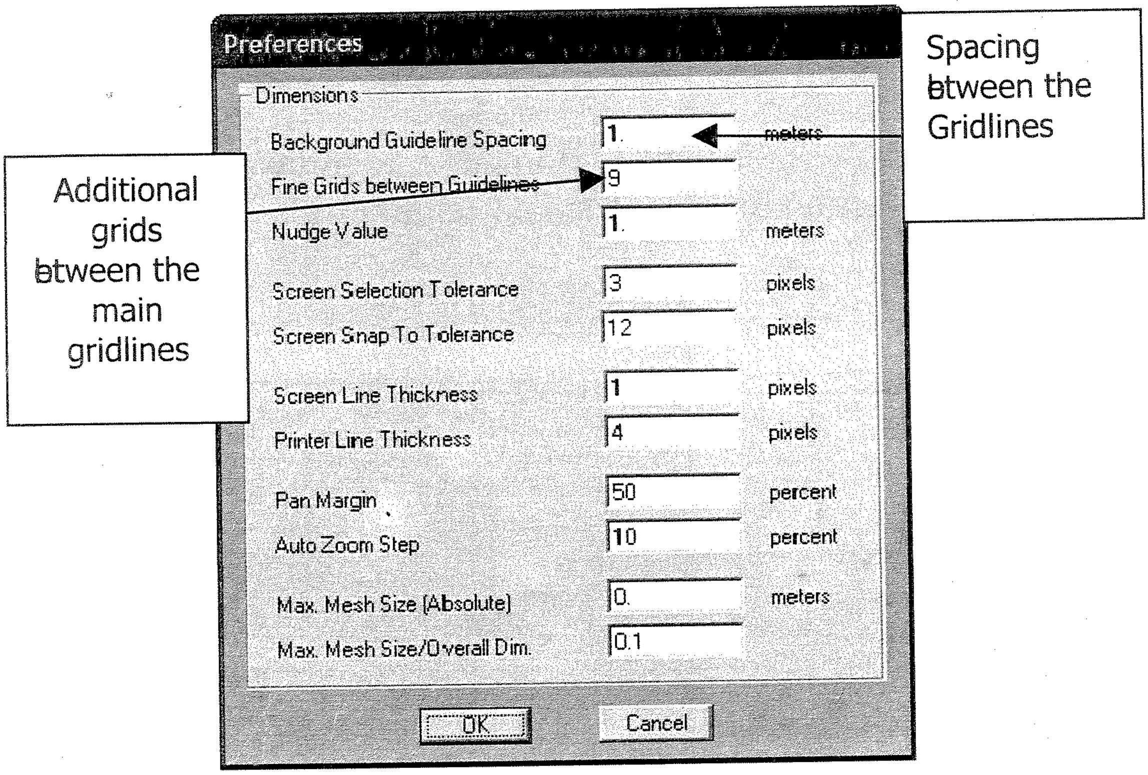
- For section (1) we will make the program check it so highlight the **Reinforcement to be designed.**



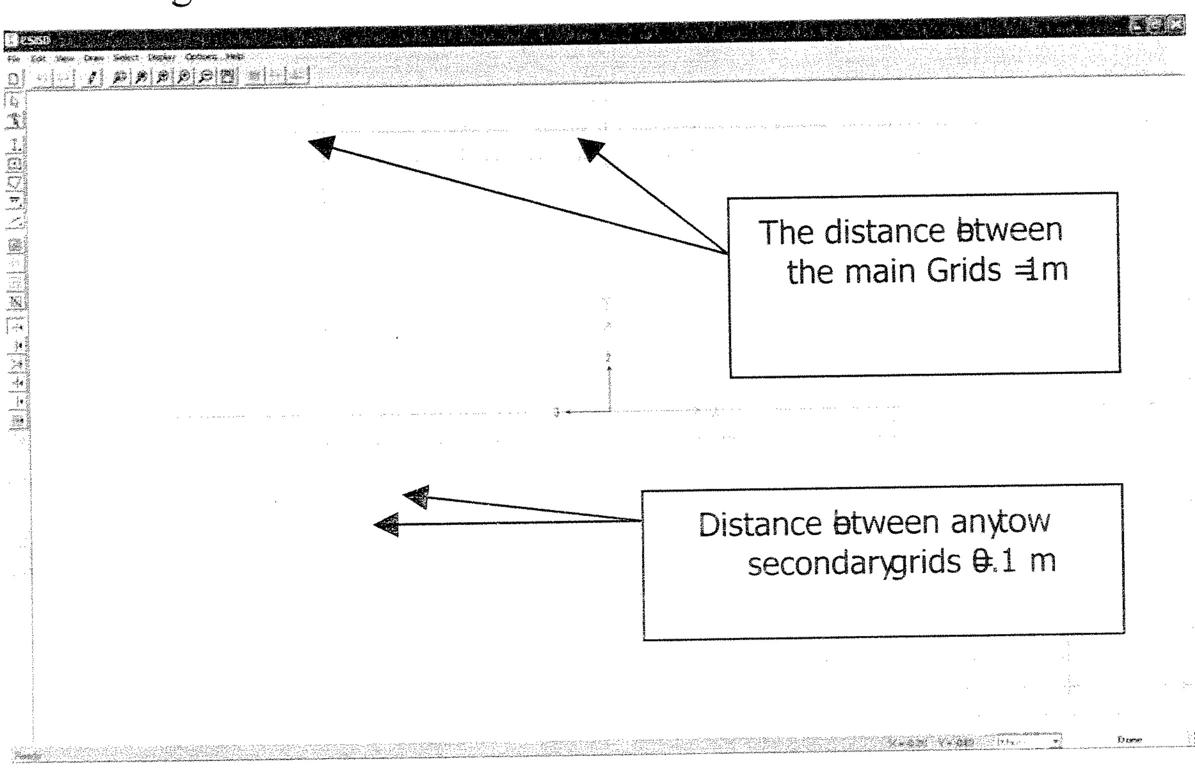
3. Click the **Section Designer** button to go to the section designer utility and draw the section on the next screen



- Click **Options menu** ——► **Preferences** to adjust the spacing between the grid lines, adjust the values to be as in the next form, thin click **OK**

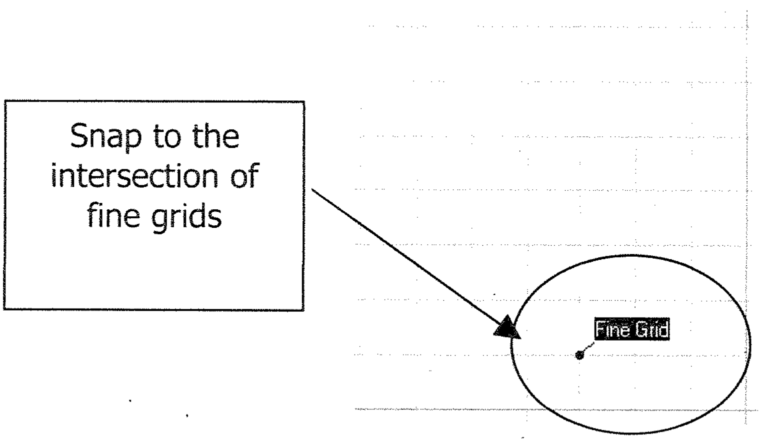


- The grid in the screen will be as follow

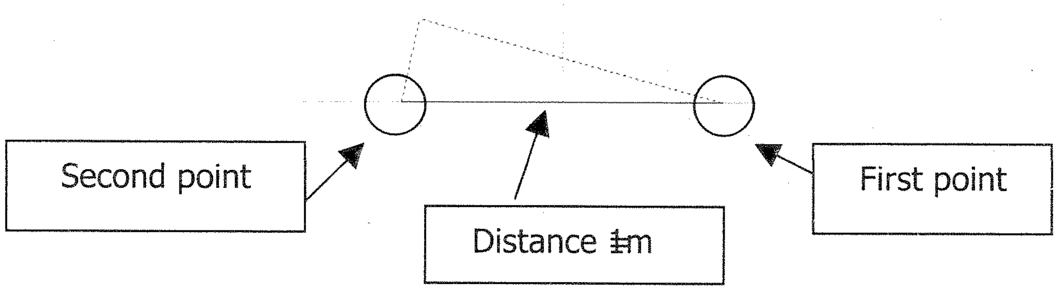


- Click **Draw menu** ⟶ **Draw Poly Shape** Or Click Draw Poly Shape button ,thin the crasser for drawing Poly Shape will be displayed, click on the Guide intersection

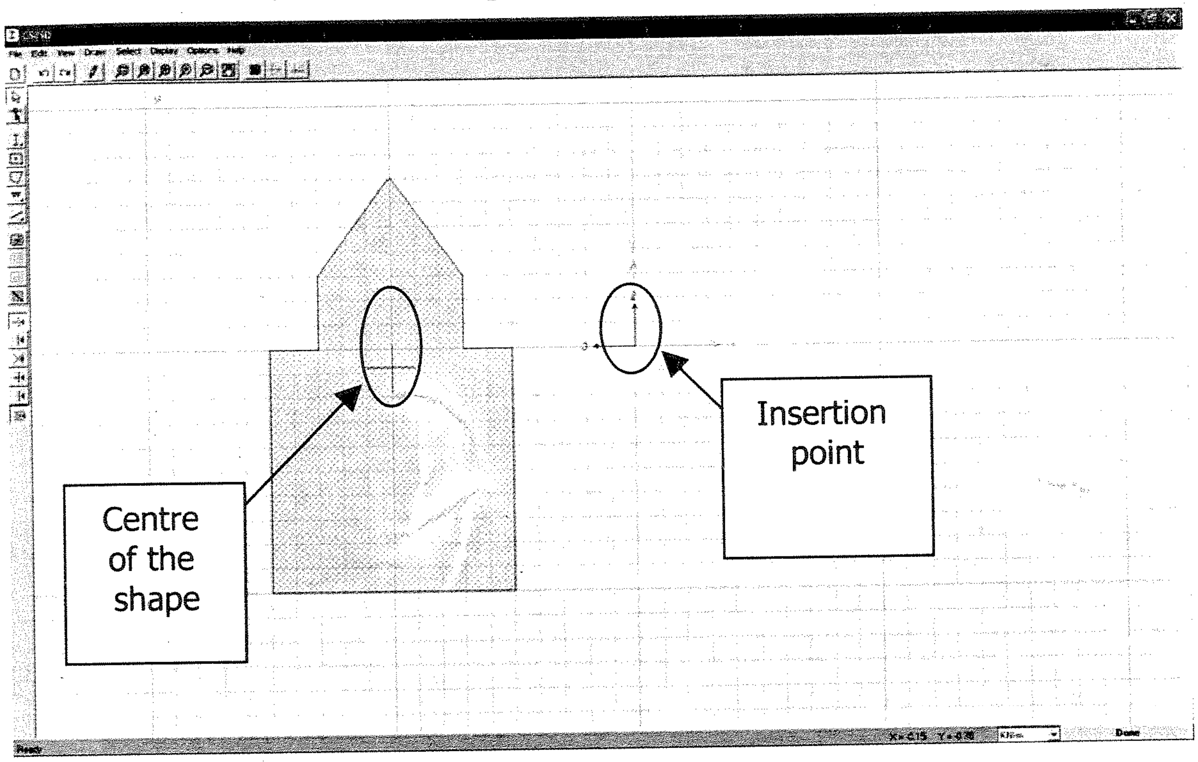
- Click Snap to Fine Grid button to allow you to snap to the intersection of the secondary (fine) Grids in the drawing



- Click at any intersection of the grids to get the first point of the shape ,thin click at a distance of 10 grids to draw the second point



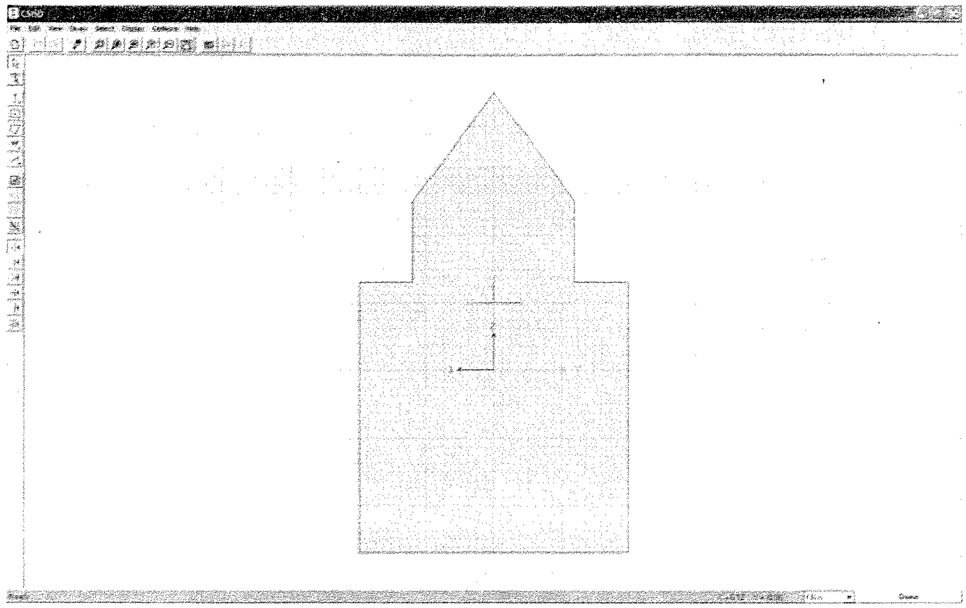
With the help of the grid line complete the drawing of the shape by click of the point of intersection, after you finish the shape drawing click ESC button, and the shape will be as in next figure



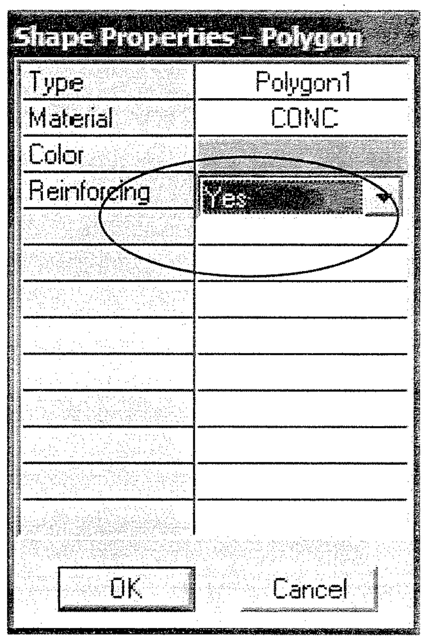
The centre of the shape not match with the insertion point

a. f you have eccentricity of the colum n you can move the centre of the shape with the same value of eccentricity from the insertion point
b. f you don't have eccentricity m ake the centre of the shape at the intersection point

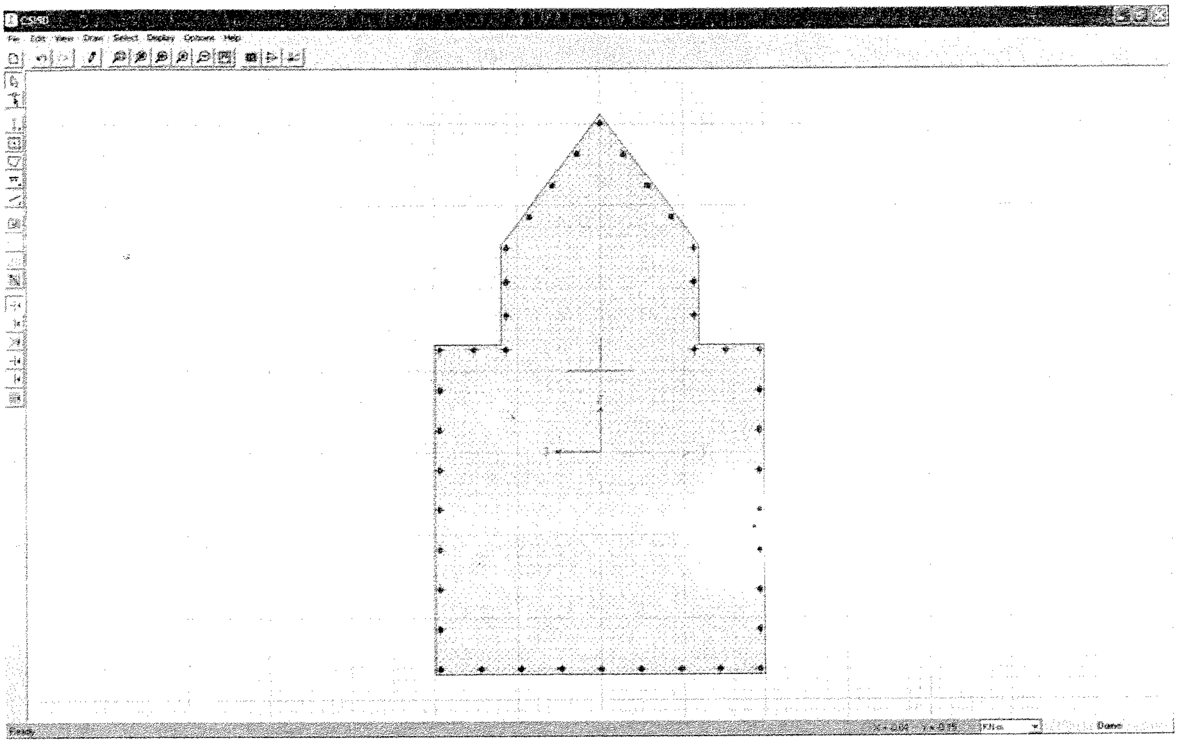
- To move the centre of the shape to the insertion point click Reshape button ,choose the shape and click on the point of the centre of the shape and drag it to the insertion point



- Right click on the shape by mouse thin left click to display the Shape Properties –Polygon, end from the drop down box of reinforcement choose yes, this to make the program know the shape of reinforcement



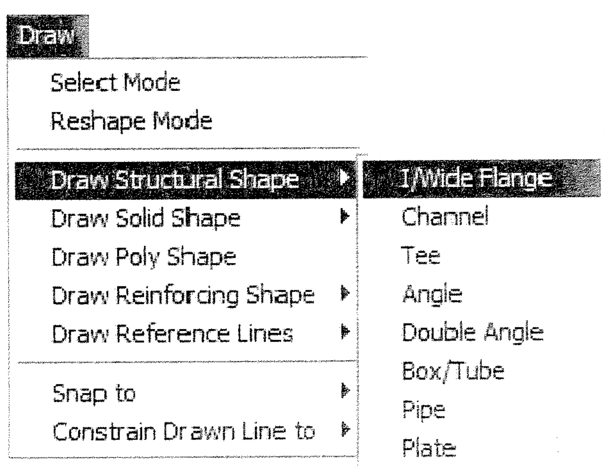
- Thin the section will be as in the next Figure

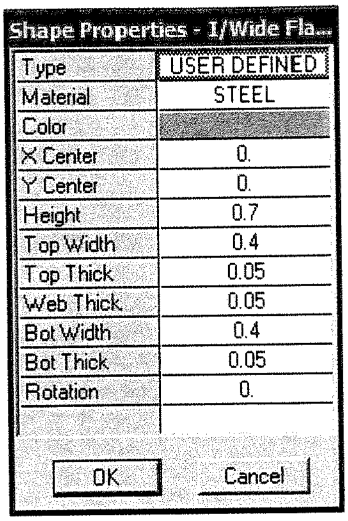


- In the end the column section will be as in the previous figure and ready to be designed, click **Done** from the form and thin **Ok** from the main form

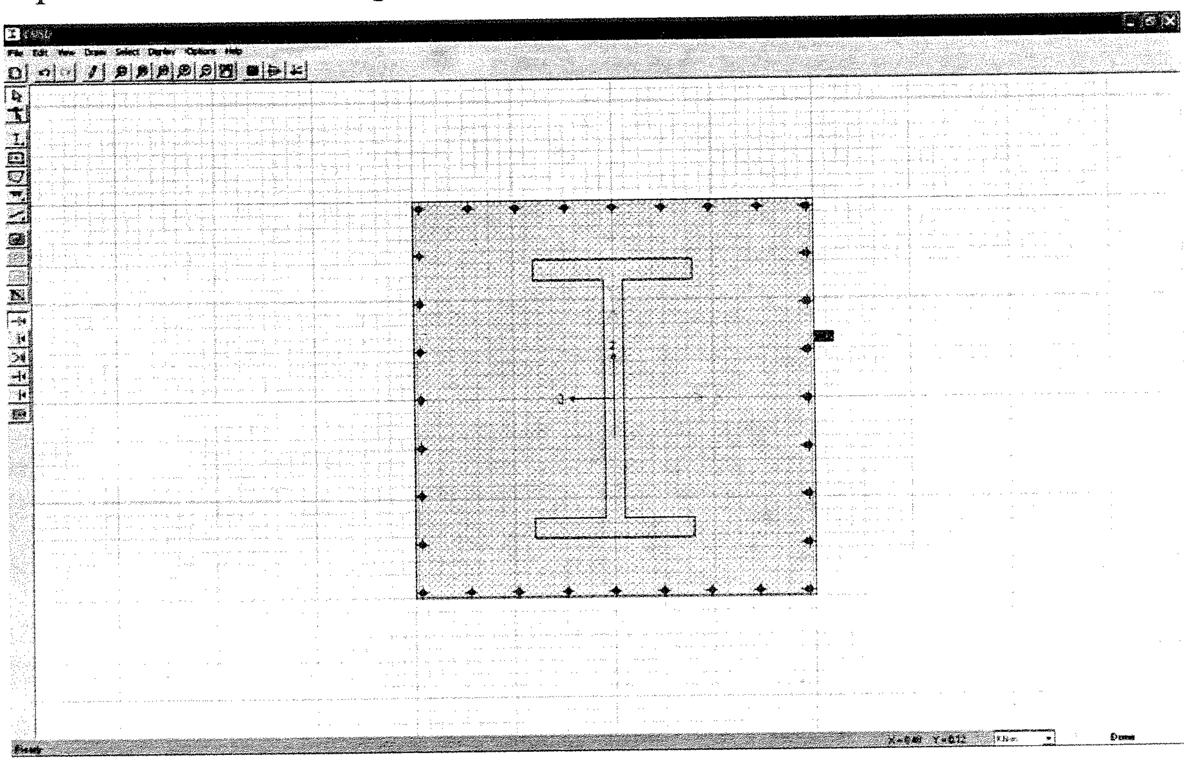
You can also draw composite section with the same steps, the only new step in draw this section is how to draw steel section

- To draw any steel section click **Draw menu** → **Draw Structural Shape** → **I/Wide Flange**, thin click on the center of the shape





Right click on the shape by mouse thin left click to display the Shape Properties I/Wide Flange to adjust the dimensions of the shape



2. <u>**Section Designer for Pier Walls**</u>:

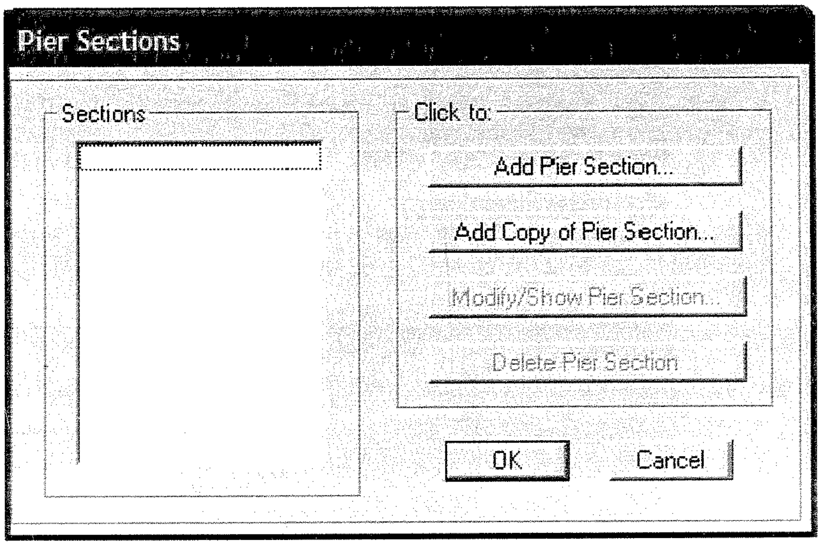
The main difference between the section designer of the frame element and the section designer for the walls is the section designer for the walls must be done after the analysis run.

- Steps for **Section Designer for the Walls.**

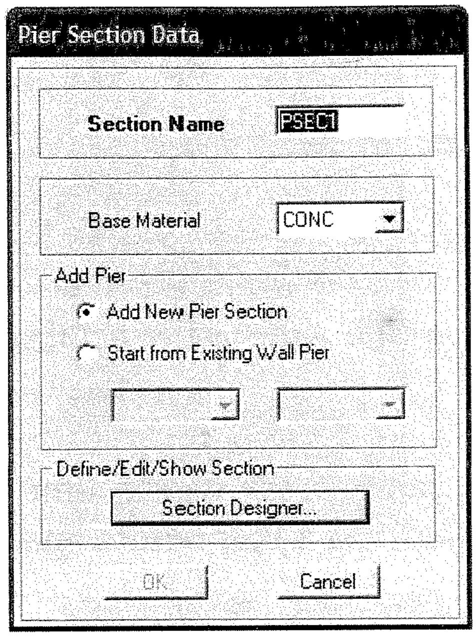
1. Built the structure model
2. Run the analysis
3. Begin the design steps as we explained before in shear wall design.

Important note: you can not use the section designer for the walls unless you named the walls as a Pier before analysis run

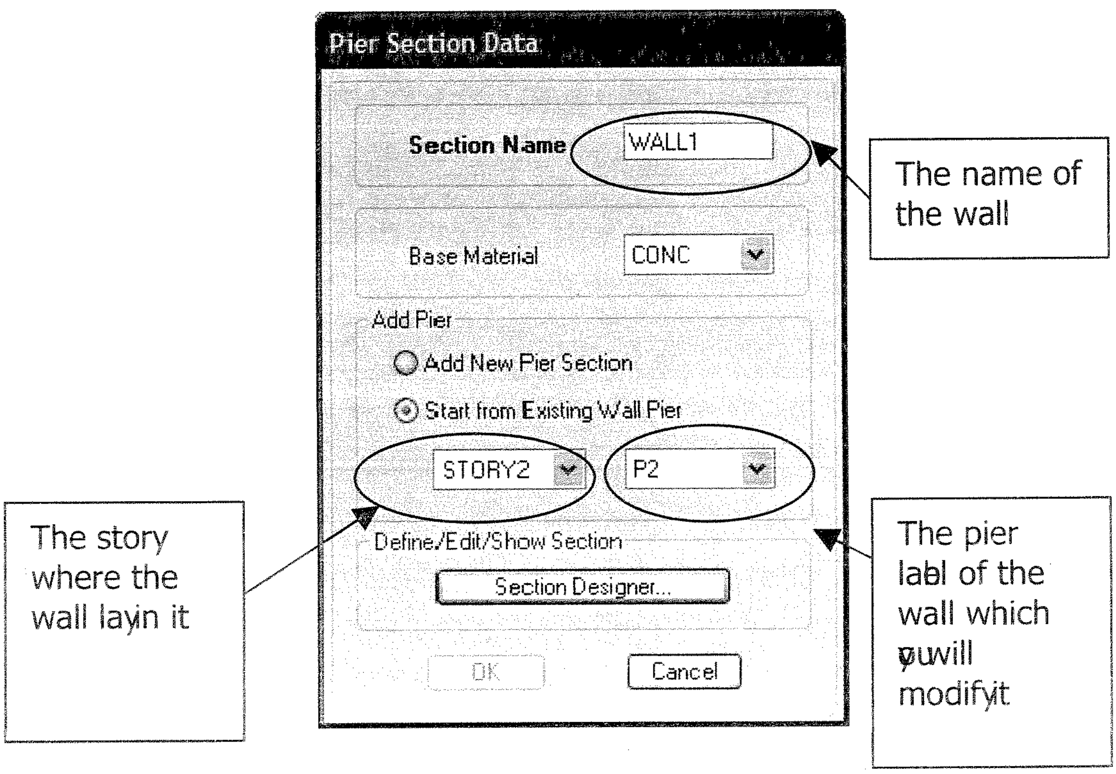
4. Click **Design menu** → **Shear wall design** → **Define Pier Sections for Checking**, thin the next form will be displayed.



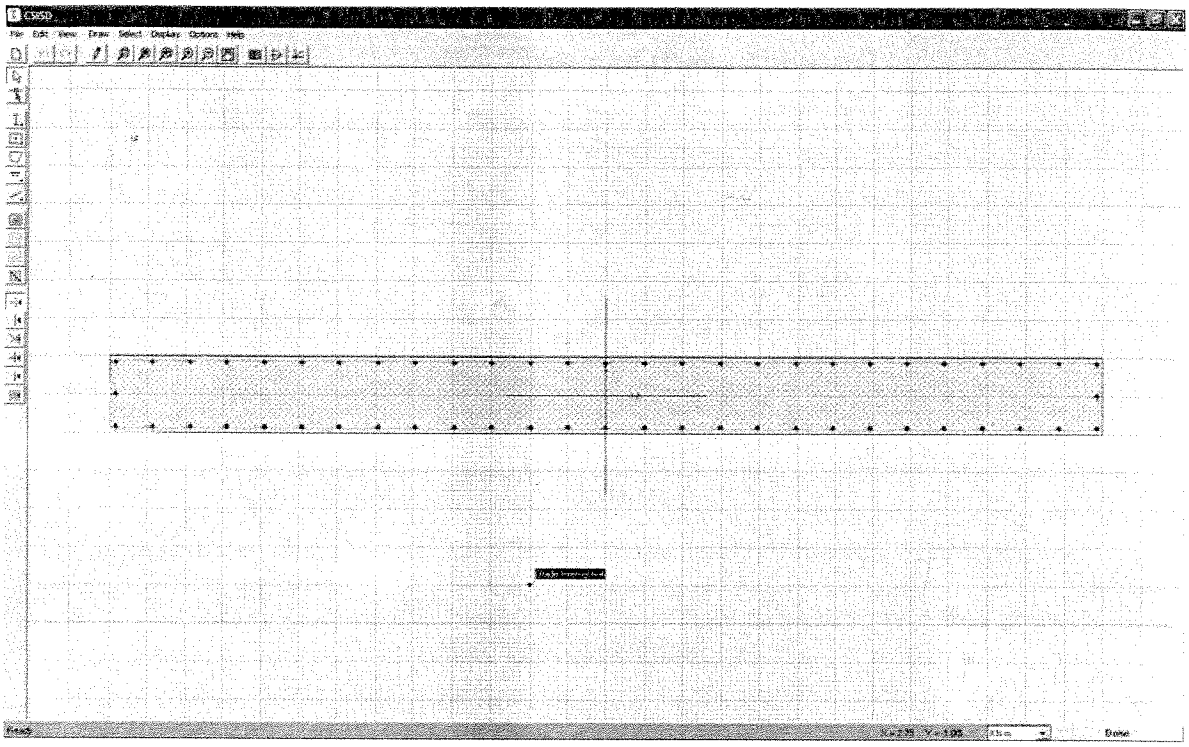
5. Click **Add Pier Section** to display the **Pier Section Data form**



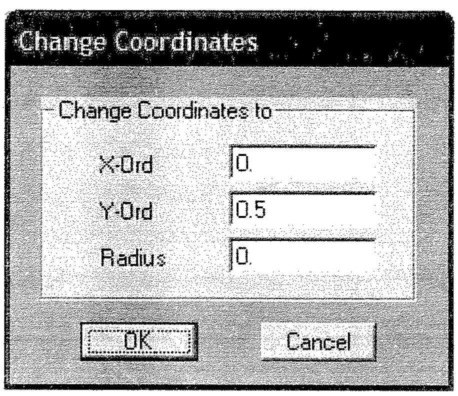
6. you have tow choice to begin the creation of the wall shape
   - To Add anew Shape (the same steps as the frame element creation)
   - To begin with any existing wall and make modification for this wall (Dimension or Reinforcement)
7. choose the wall you want to change its size or reinforcement

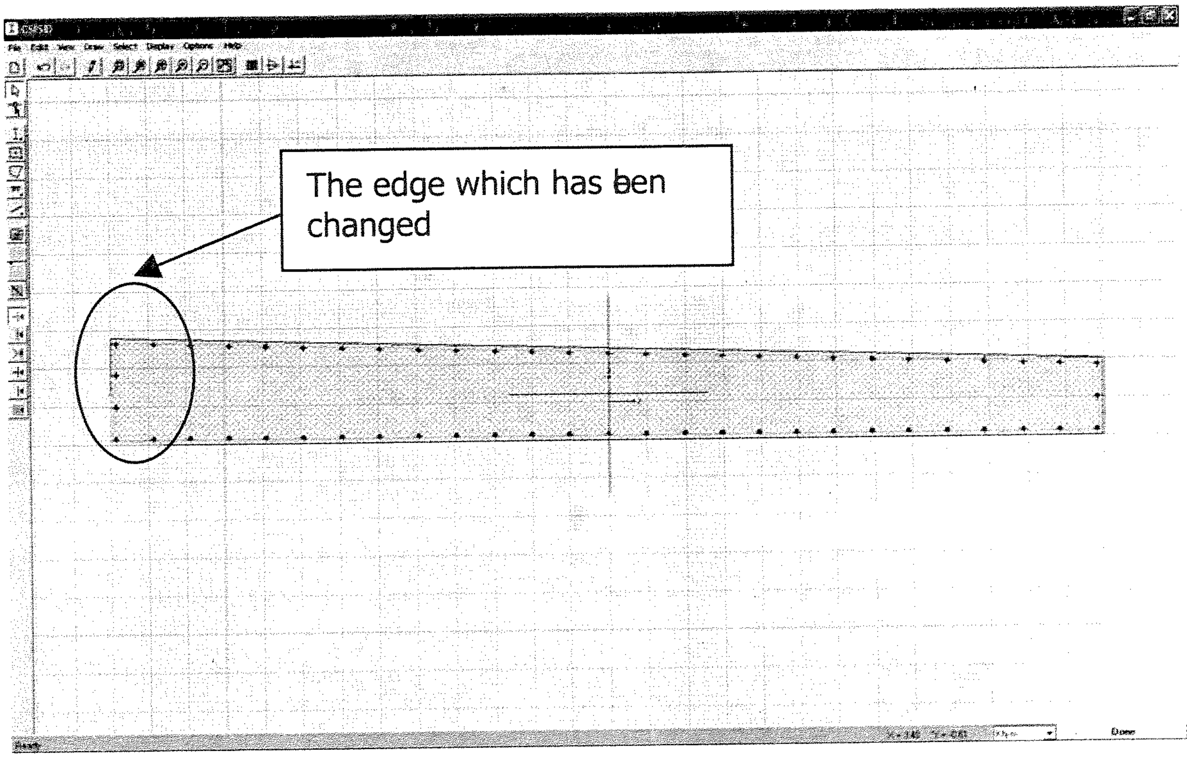


8 Click the **Section Designer** button to go to the section designer utility and draw or modify the section on the next screen

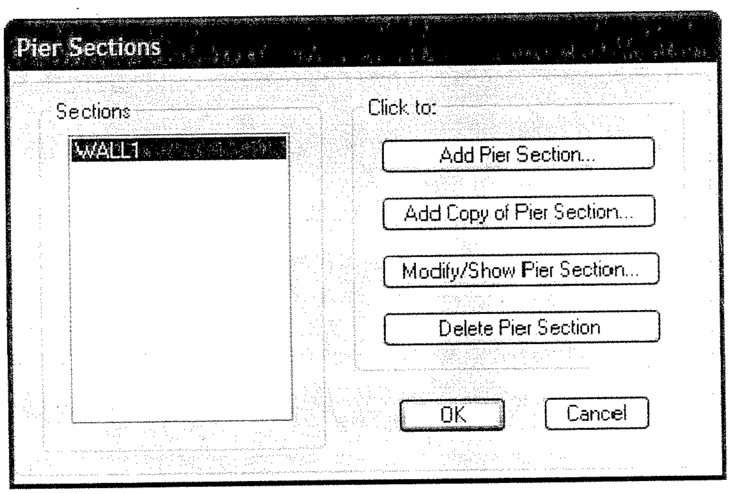


9. To change the dimension of the wall click **Draw menu** ⟶ **reshape Mode** ,thin the shape of the crasser will changed click on the edge point by left button of the mouse thin right click ,the **change coordinates** form will be displayed, change the coordinates according to the new size thin click **OK**





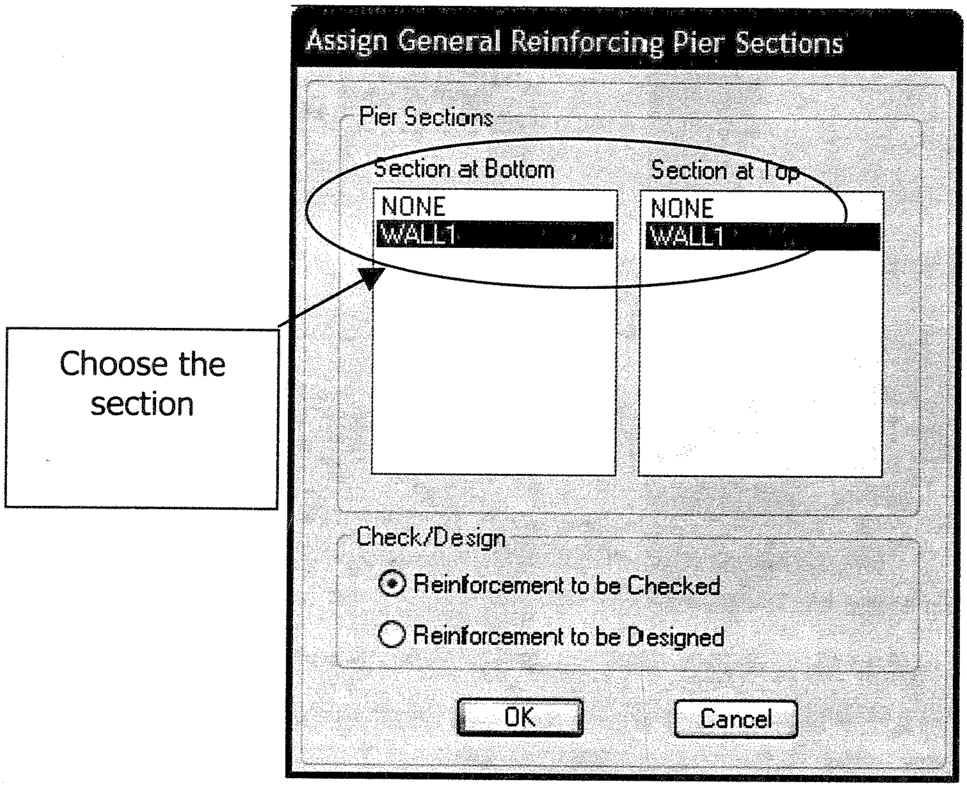
- And for change the reinforcement follow the same steps as in section (1)
- After you change the dimension or reinforcements click **OK,** thin you will find the new section will be added



10. To assign the new section for the wall.
    - choose the wall

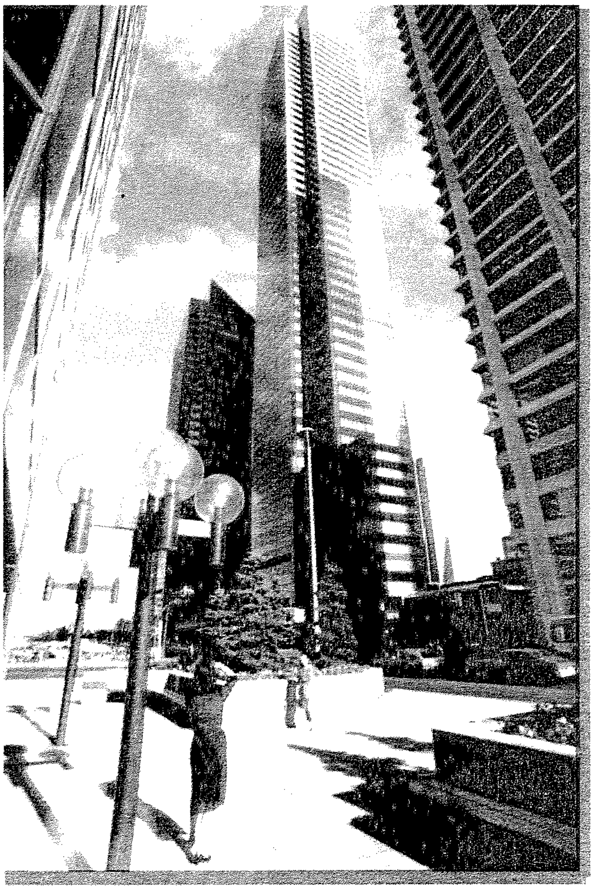
- Click **Design menu** → **Shear wall design** →. **Assign pier sections for checking** → . **General Reinforcing Pier Section.**

- Thin the next form will be displayed



- From this form choose the wall section in the top and bottom of the wall thin click **OK**

- Follow the same steps of design as it was explained before in shear wall design



HOW TO MODEL AND DESIGN HIGH RISE BUILDING USING ETABS Program

Meshing

Chapter 10

ETABS is finite-element program based in the meshing and divide of the structure elements. Also the Meshing helps distribute loads realistically

- There are 3 method of meshing

  1. Manual meshing
  2. Automatic meshing
  3. Meshing done by special program for meshing

1. **Manual meshing**: we use Manual meshing for the next cases

   1. for the complicated structure shapes
   2. for the structure have curved edges or combination between rectangular and circular shapes

- The most common use for manual meshing is done using DXF files thin exported to Etabs program. the manual meshing is the most famous method done by the most of Engineers

2. **Automatic meshing**: we use Automatic meshing for the next cases

   1. rectangular and circular structure
   2. Deck slabs
   3. Slab properties with membrane behavior only

- The Automatic meshing is powerful tools from the program save your time and give you the ability to change the size of meshing at different runs according to the accuracy you wanted from this run

- To use the **Area Object Auto Mesh Options** form to control meshing of area objects

1. Select the area object to be meshed

2. Click the **Assign menu** ⟶ **Shell/Area** ⟶ **Area Object Mesh Options** command to display the **Area Object Auto Mesh** Options form.

3. On the Area Object Auto Mesh Options form, select the desired options from the following choices:

❖ **Floor Meshing Options**

➢ **Default (Auto Mesh at Beams and Walls if Membrane - No Auto Mesh if Shell or Plate)** option. This option meshes the objects at beams and walls, no meshing for the object which is **defined as a shell or plate,**

➢ **For Defining Rigid Diaphragm and Mass Only (No Stiffness - No Vertical Load Transfer)** option.

➢ **No Auto Meshing (Use Objects as Structural Elements)** option. No méshing for the elements

➢ **Auto Mesh Object into Structural Elements** option.

- **Mesh at Beams and Other Meshing Lines** option. Similar to the default option.

- **Mesh at Wall and Ramp Edges** option. Meshes the selected object at wall and ramp edges

- **Mesh at Visible Grids** option. Meshes the selected objects at grid lines in the floors level.

The importance of the visible grid:

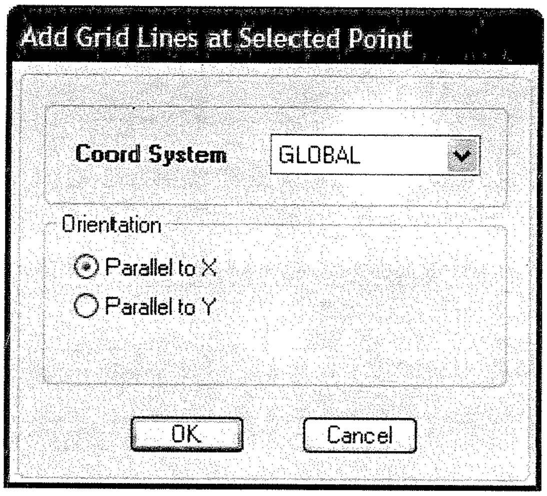
1. guide for the meshing

2. take the effect for the columns and another points which the program don't taken in consideration in meshing

3. to make the secondary grids for the meshing

a. select the points of the columns, end of the walls, and point load

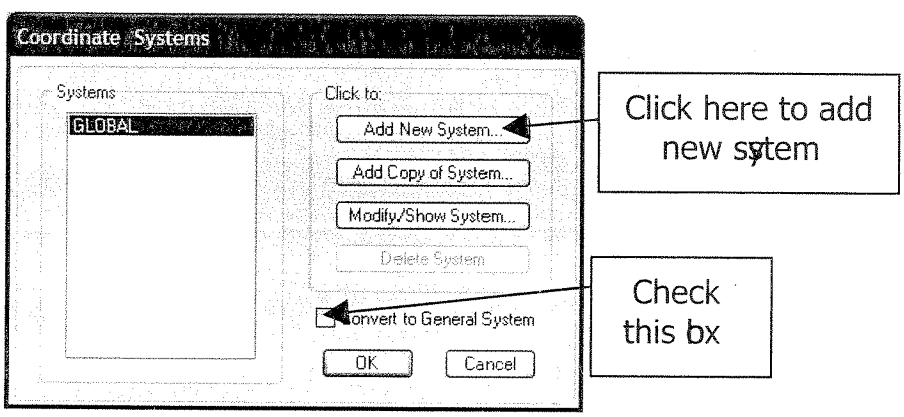
b. click **Edit menu** → **Edit Grid Data** → **Add Grid at Selected Points**, thin the next form will be displayed



c. Click Ok for the Orientation Parallel to X ,thin repeat it for the Orientation parallel to Y

- Also You can make meshing for one floor different than other floors to reduce the effect of the meshing of this floor to other floors ,

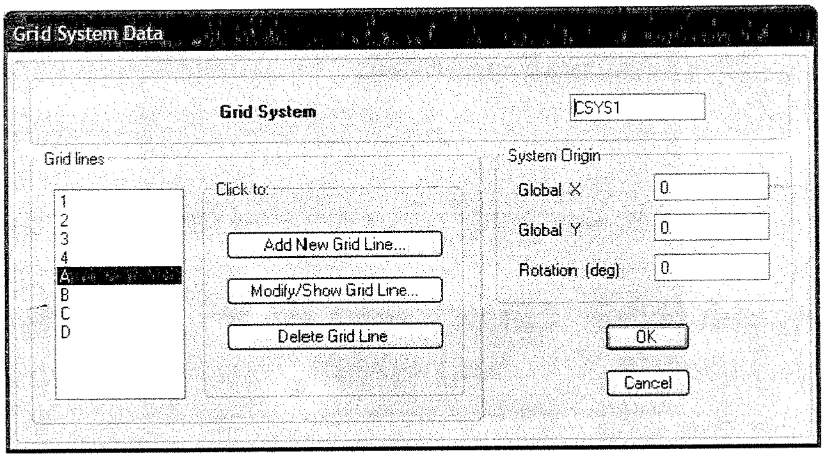
a. click **Edit menu** → **Edit Grid Data** → **Edit Grid**, thin the next form will be displayed



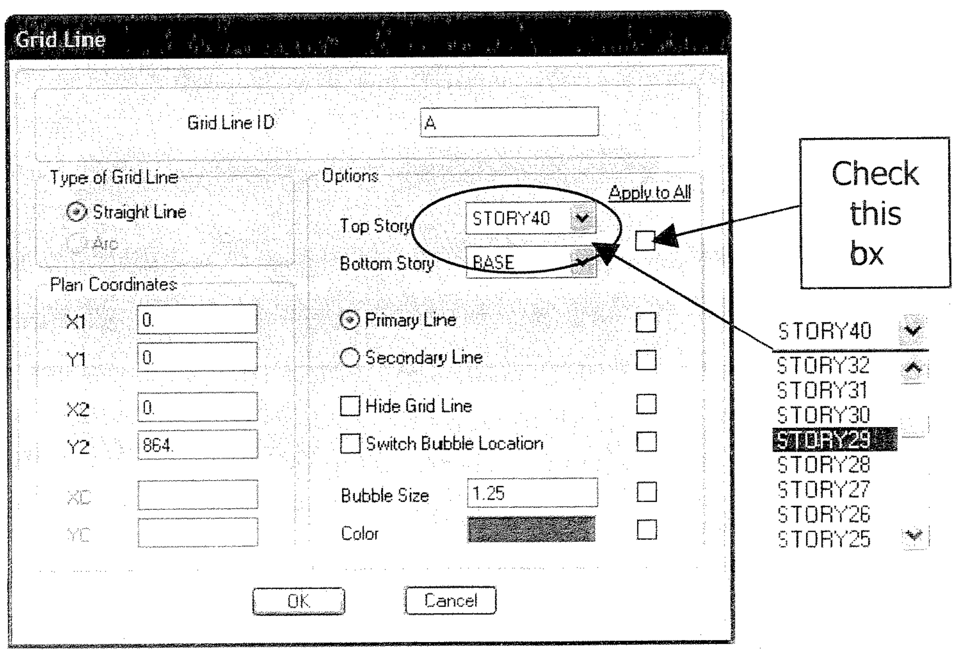
b. Click **Add New System** button to add new coordinate system

c. check the box reads Convert to General System

d. Click modify /Show System Button to display the **Grid System Data** Form



e. **Click Modify /Show Grid Line** button to display the **Grid Line** form , from this form choose the floors you want your system of Grid to be applied from From Top & Bottom Story drop dpwn box and check the box of applay to All thin click **OK**



- **Further Subdivide Auto Mesh with Maximum Element Size of {specify value}** option. Meshes the selected object(s) using the specified maximum element size.

❖ **Ramp and Wall Meshing Options**

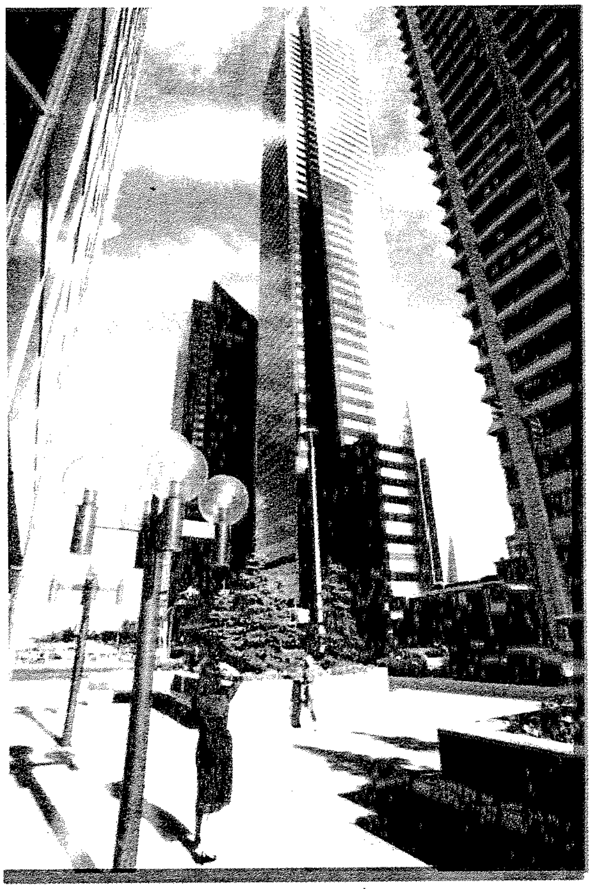
➢ **No Subdivision of Object** option.

➢ **Subdivide at Wall Openings** option. Meshes the selected objects at wall openings.

➢ **Subdivide Object into {Specify Number} vertical and {Specify Number} horizontal** option.

➢ **Subdivide Object into Elements with Maximum Size of {Specify Number}** option.

3. **Meshing done by special program:** now there are many program are specialized in meshing of the floors .you can use any one of this program to get DXF for the floor ,thin Export the DXF to Etabs Program

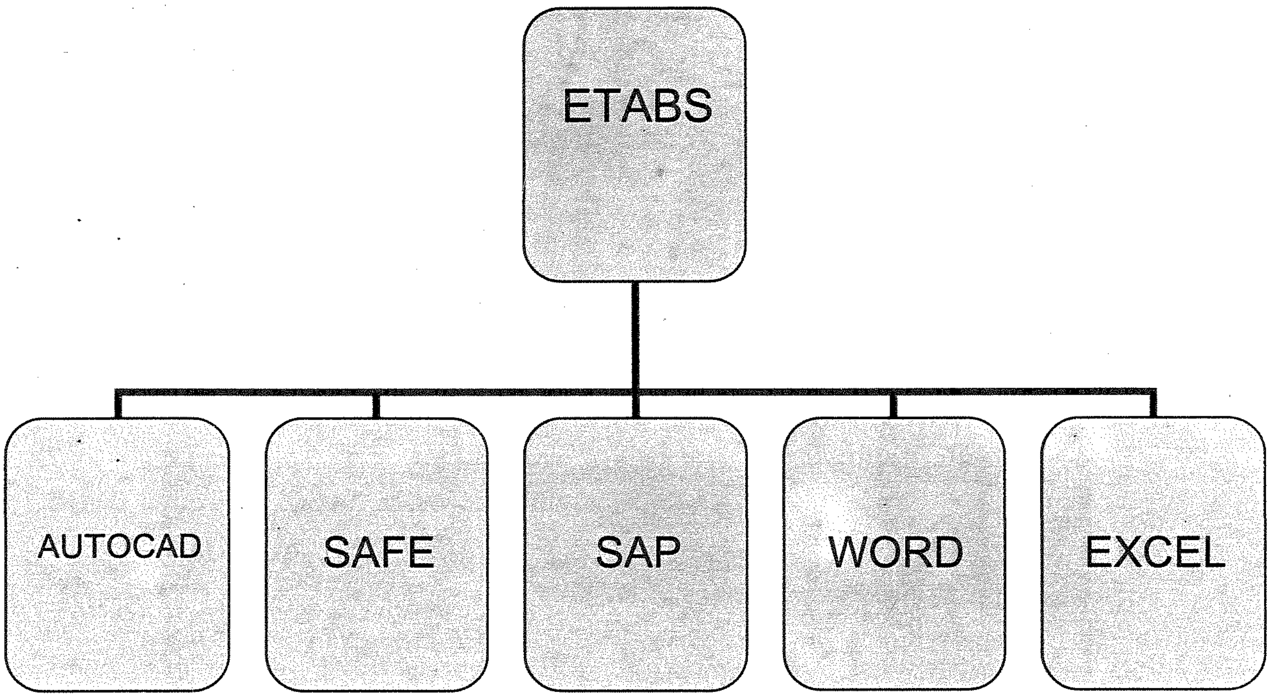


HOW TO MODEL AND DESIGN
HIGH RISE BUILDING USING
ETABS Program

Etabs & other programs

Chapter 11

- In this chapter we will talk about the relation between Etabs Program and 5 of the most important program
- ETABS program allow you from importing and exporting to share the data between the program and other programs

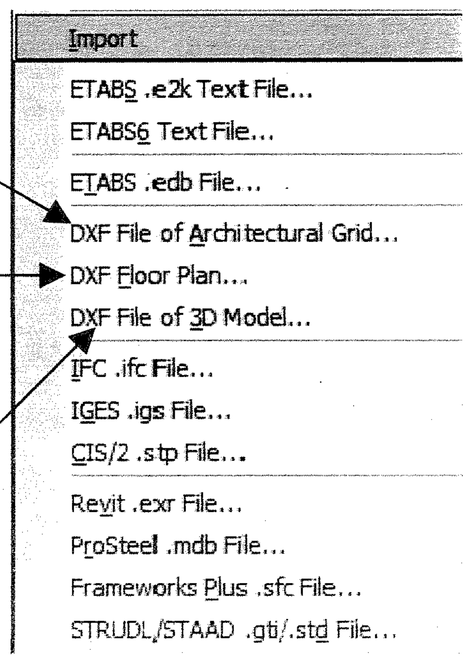


Etabs & AutoCAD Program

1- To import the grid lines

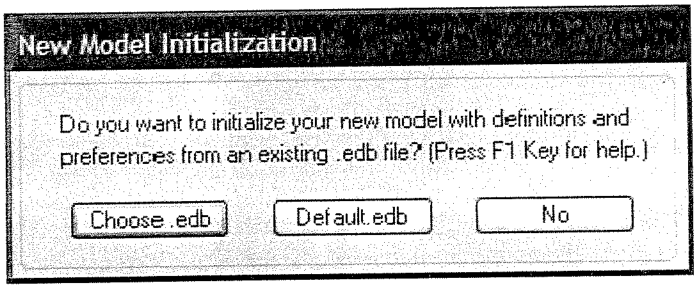
2-to import the floor plan (beams, columns, and slabs)

3-to import the full structure

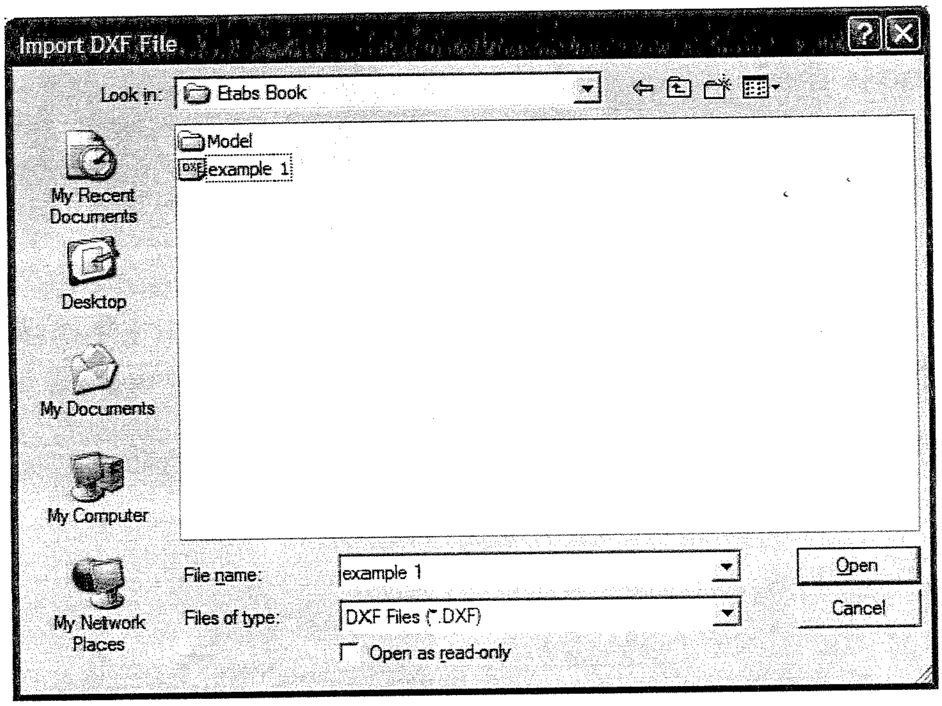


1. Import the grid lines

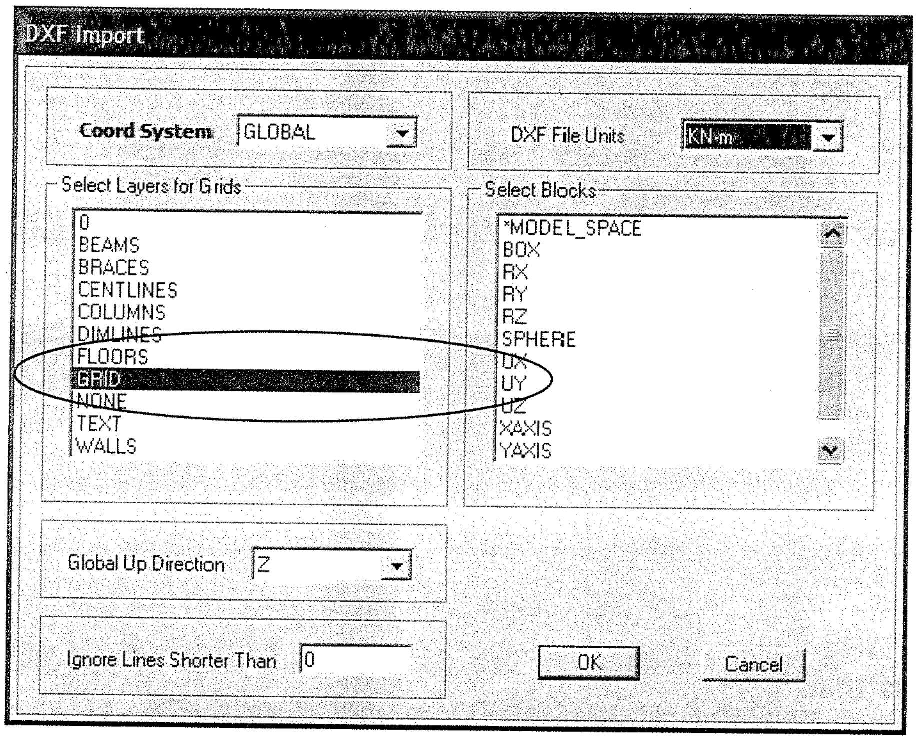
1. Save the AutoCAD which contain the Grid lines as DXF file
2. Click **File menu** ⟶ **Import** ⟶ **DXF File of Architectural Grid…**
3. if this file is anew file the form of **New Model Initialization** will be displayed, from this form choose Default.edb



4. thin the **Import DXF File** form will be displayed



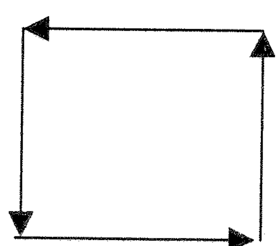
5. Choose the file of DXF and Click **Open;** thin DXF Import Form will be displayed



Choose the layer contain the Grids thin click **OK**

2. **Import the Floor Plan**
   1. Open AutoCAD Program
   2. Open the File of the plane for the first floor or open new file in AutoCAD Program and make 4 deferent layers
      i. Beams : for beams and walls
      ii. Floor :for slabs
      iii. Opening :for the openings
      iv. Columns: for the columns
   3. Draw the PLINE on the edge of the slab in the layer Floor, you don't need to draw 3D Face and mesh it yourself ,we will make the program make automatic mesh for the elements

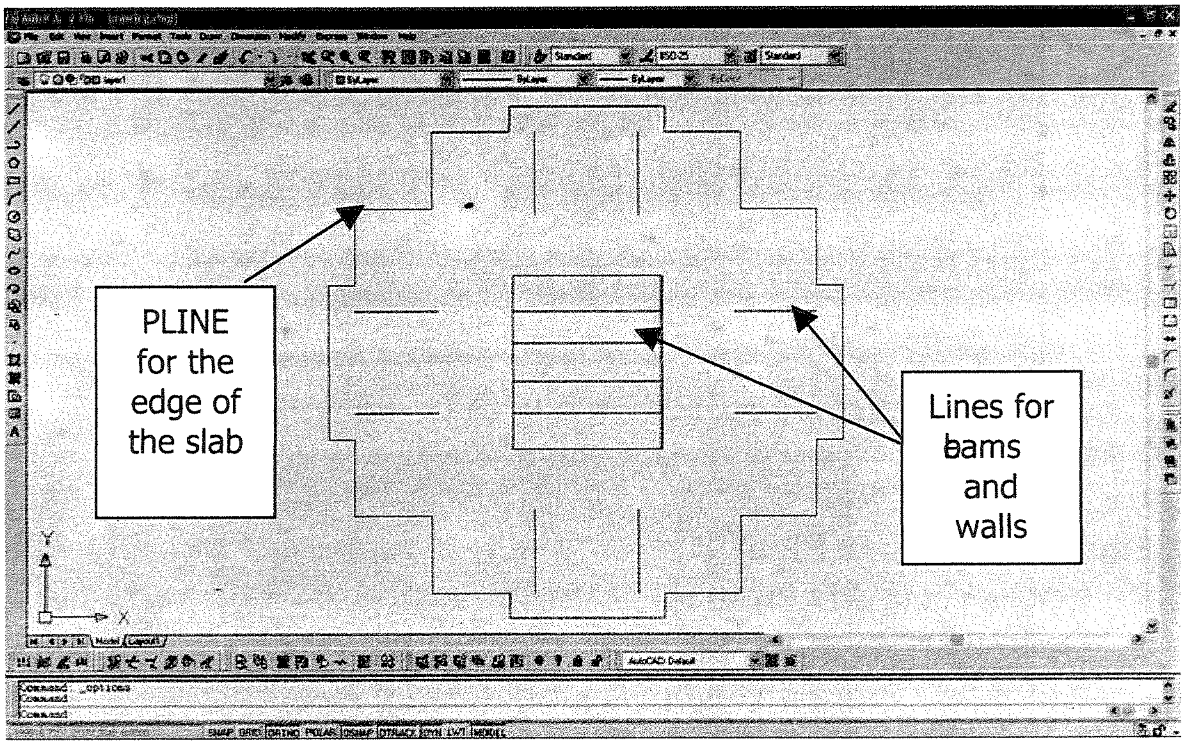
Direction of Drawing
The PLINE of the slab



Draw the beams and walls as lines in the layer called beams

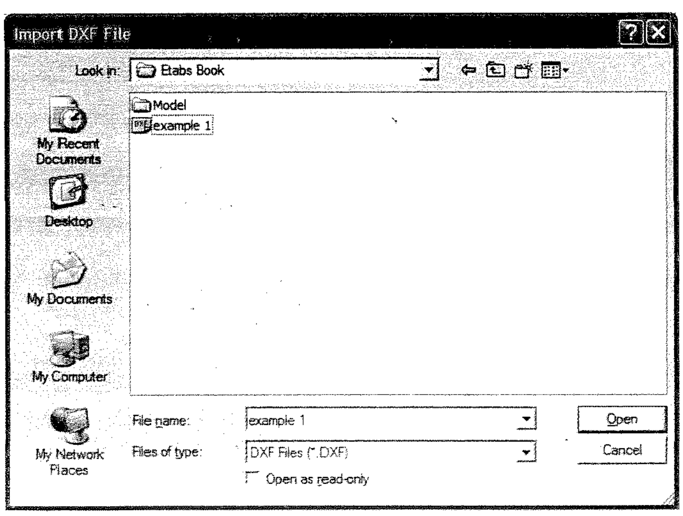
Direction of Drawing
The beams and the
Walls lines

4. Draw the openings in the layer called opening as PLINE on the edges of the openings

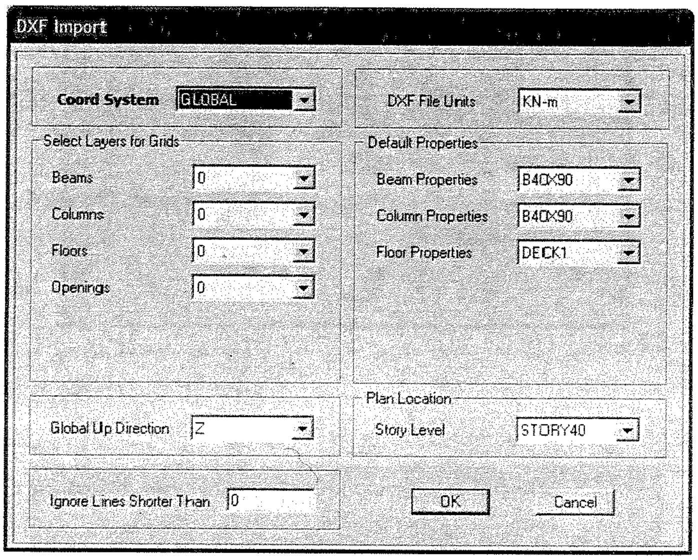


5. save the file as DXF file named Example1

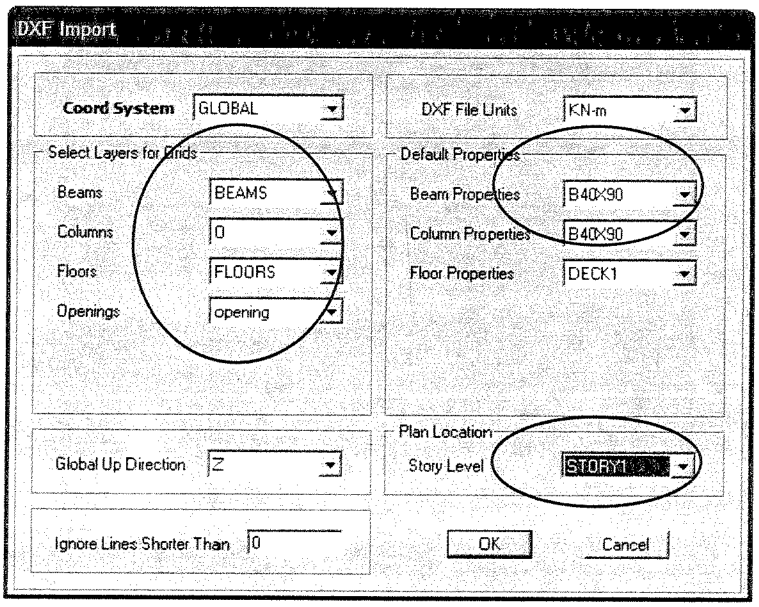
6. Click **File menu** → **Import** → **DXF floor plan** , which will Display the Import DXF File form



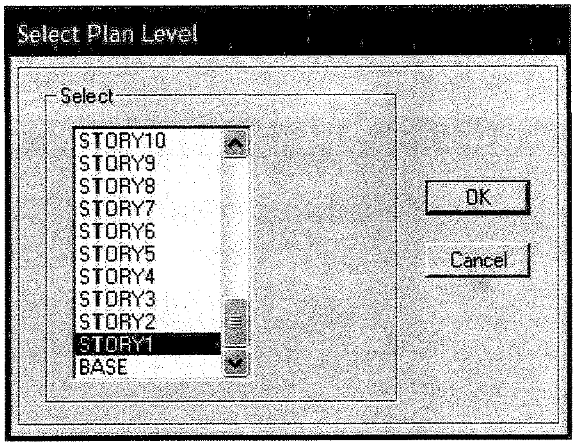
Choose the file of DXF and Click **Open;** thin DXF Import Form will be displayed



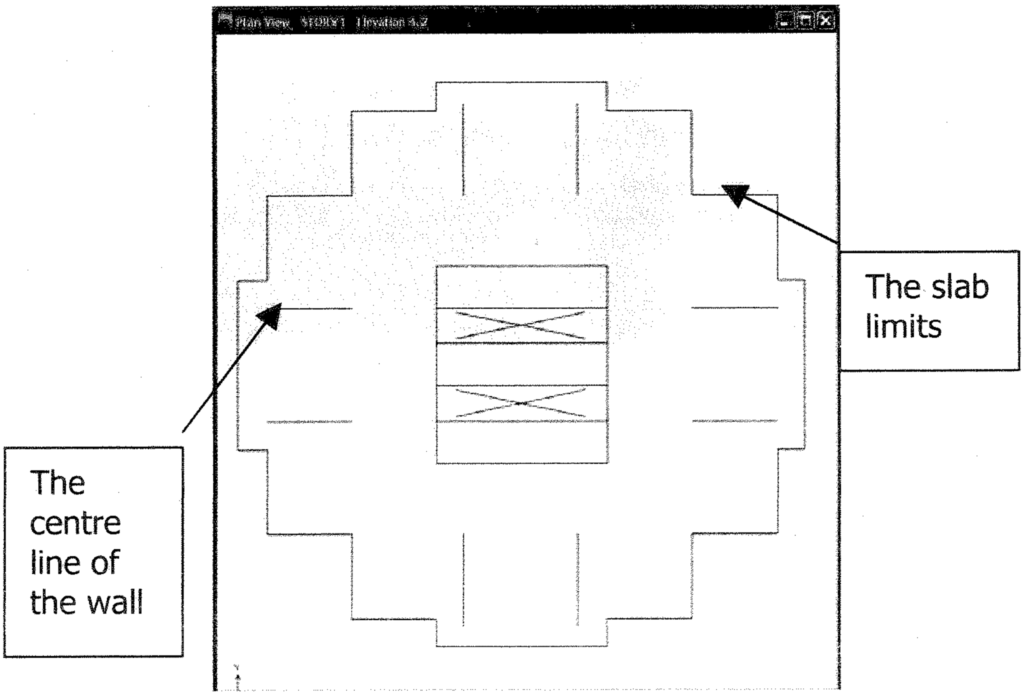
Adjust the marked pull down box as in the next fig.



7. Make sure that the plan view is the view of story 1,if not Click set plan view button thin the next form will be displayed, from this form choose story 1,thin click **OK**



Thin the floor will be displayed in the plan as follow



8. Choose the wall centre line thin click **Edit menu** ⟶ . **Extrude Lines to Areas** ,and follow the same steps of chapter (1)

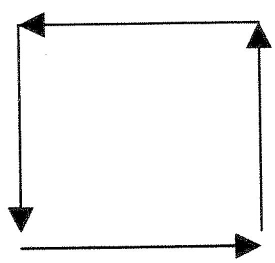
Note: for the columns draw line in the 3D in auto cad to Import as column

3. **Import 3D Model**
   1. Open AutoCAD Program
   2. Open the File of 3D or open new file in AutoCAD Program and make 7 deferent layers

i. Beams : for beams and walls
ii. Floor :for slabs
iii. Opening :for the openings
iv. Columns: for the columns
v. Braces :for the bracing
vi. Ramps :for the ramps
vii. Grids :for the grid lines

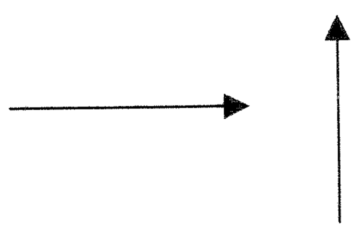
3. Draw the PLINE on the edge of the slab and ramps and walls in the corresponding layers,

Direction of Drawing
The PLINE of the slab



Draw the beams and columns as lines in the corresponding layer

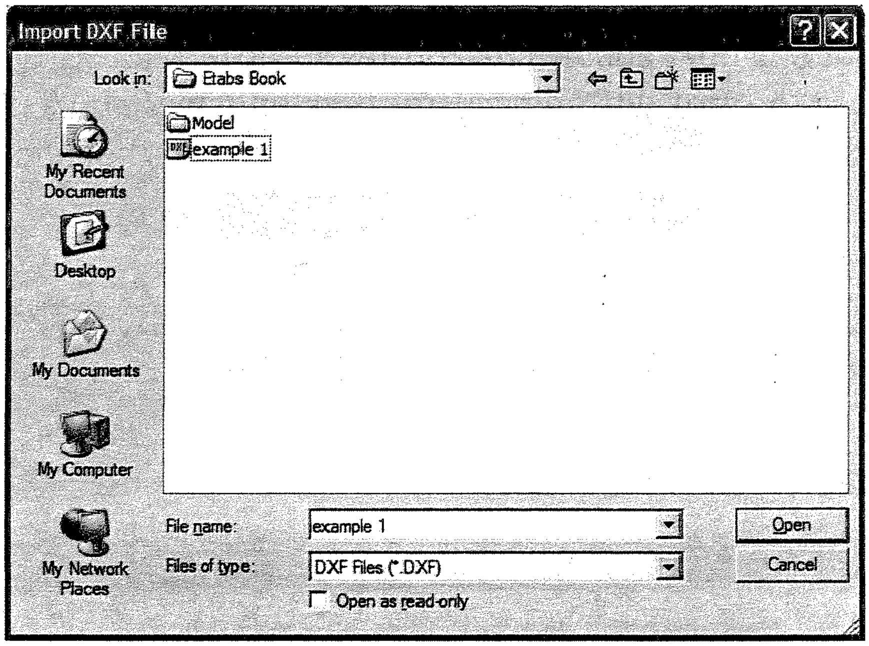
Direction of Drawing
The beams and the
Walls lines



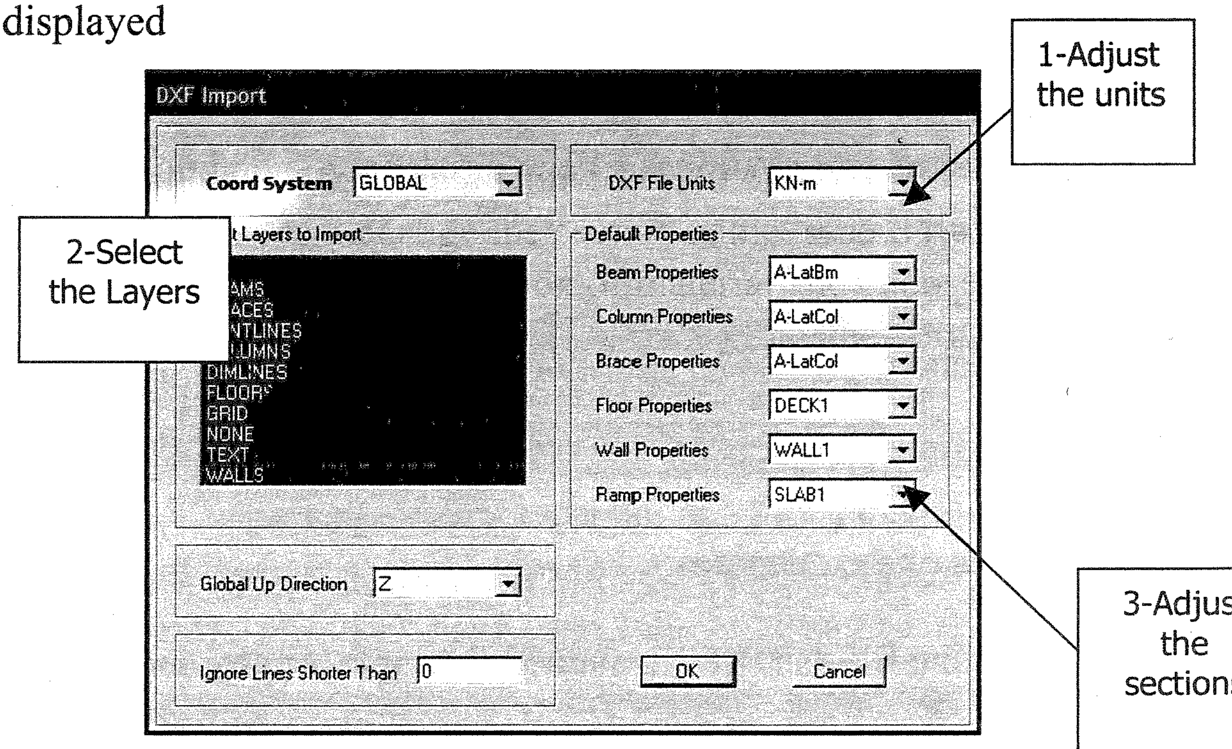
4. Draw the openings in the layer called opening as PLINE on the edges of the openings

5. save the file as DXF file named Example1or any another name

6. Click **File menu** → **Import** → **DXF File for 3D Model ,** which will Display the Import DXF File form



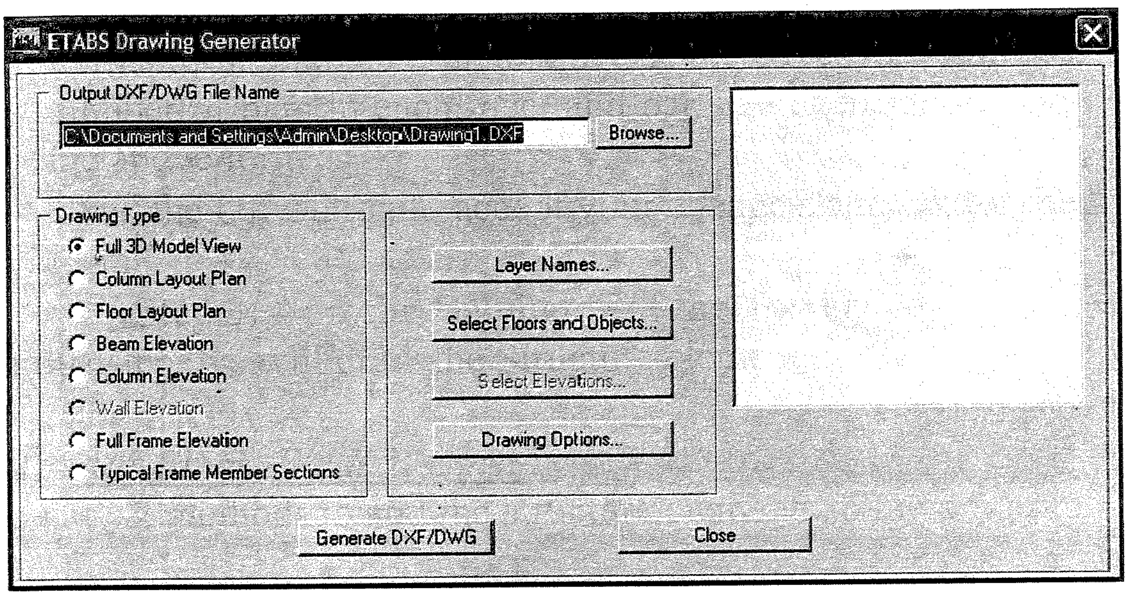
Choose the file of DXF and Click **Open;** thin DXF Import Form will be displayed



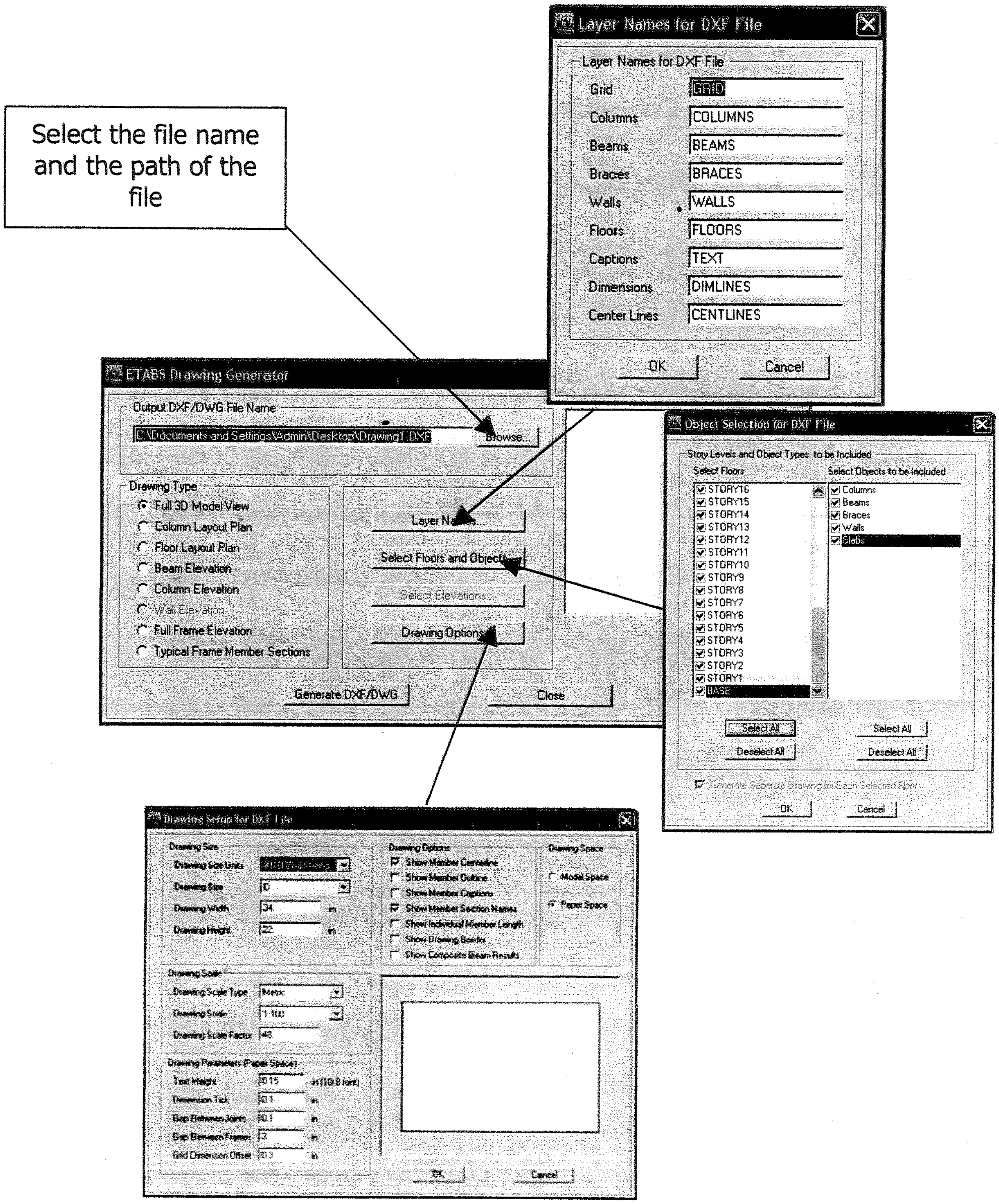
Adjust the DXF File Units, thin choose the layers to import and adjust the default Properties sections for the elements thin click **OK**

4. Export of DXF File from Etabs

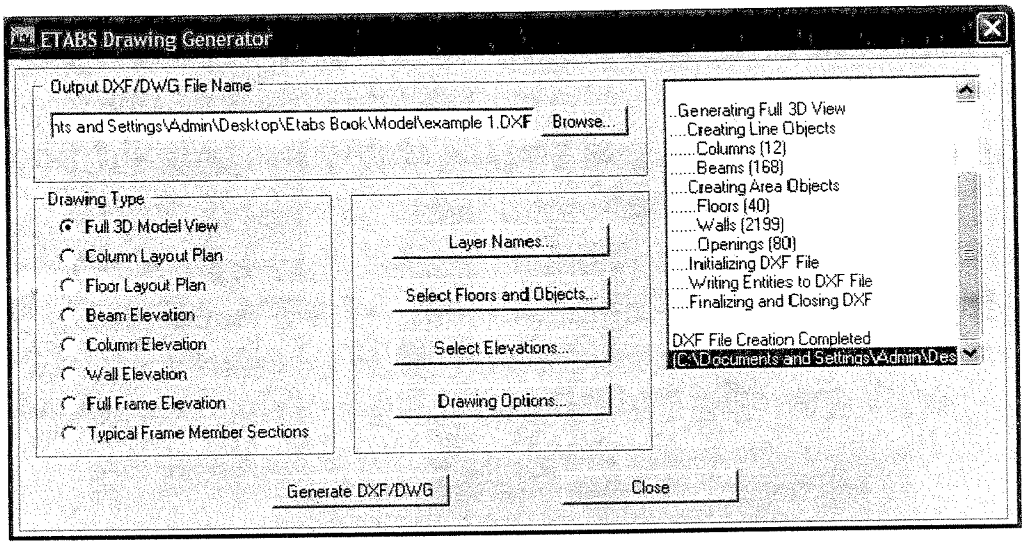
1. Click **File menu** ⟶ **Export** ⟶ **Save as DXF File,** which will Display the Export DXF File form



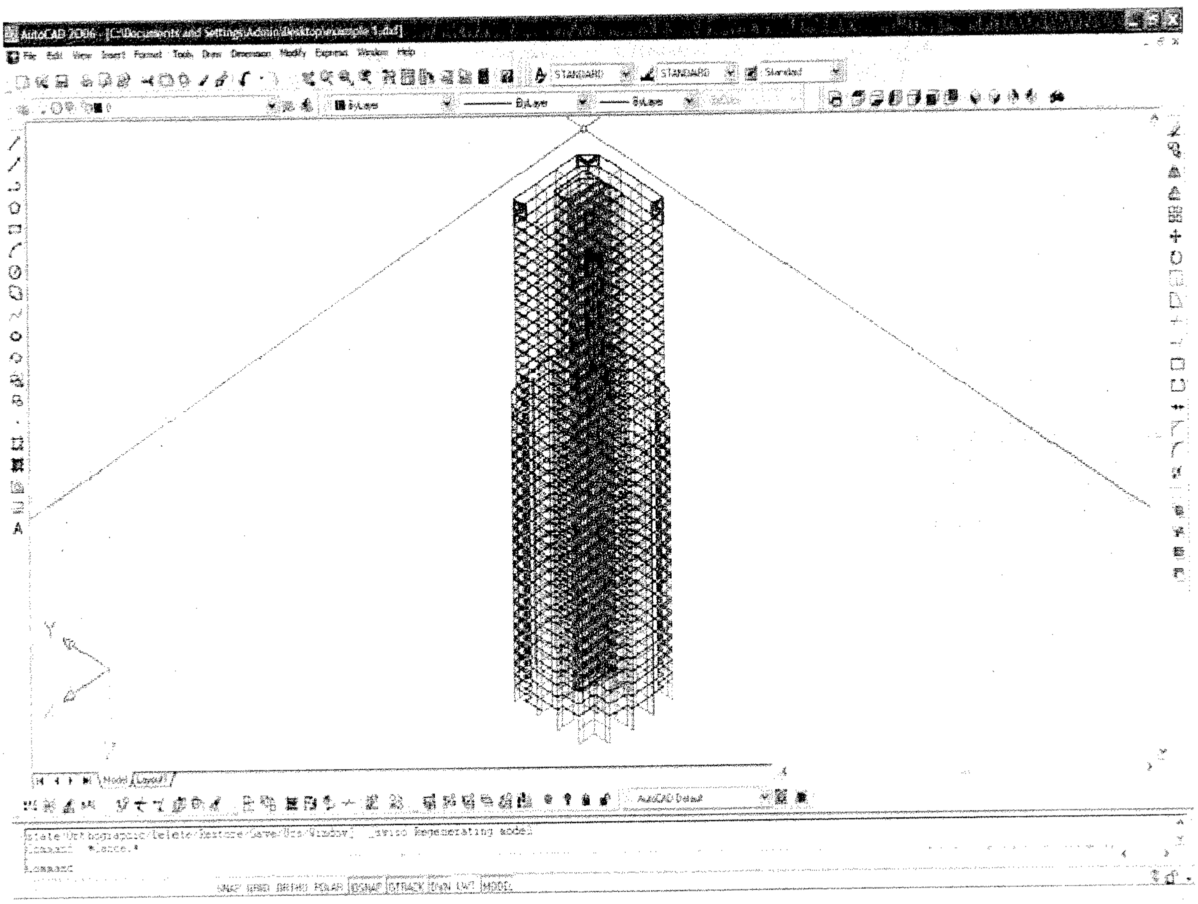
2. choose the type of DXF File which you want to export
    i. Full 3D Model View
    ii. Column Layout Plan4Flor Layout Plan
    iii. Beam Elevation
    iv. Columns Elevation
    v. Wall Elevation
    vi. Full Frame Elevation
    vii. Typical Frame Member Sections
3. Select the file name and the path of the file



4. adjust the Layers Name
5. Select Floors and Objects you want to exports
6. Adjust the Drawing Options
7. Click **Generate DXF/DWG** ,thin the generation process will be displayed



8. after the program finish the generation Click **Close**
9. open the DXF file by AutoCAD program



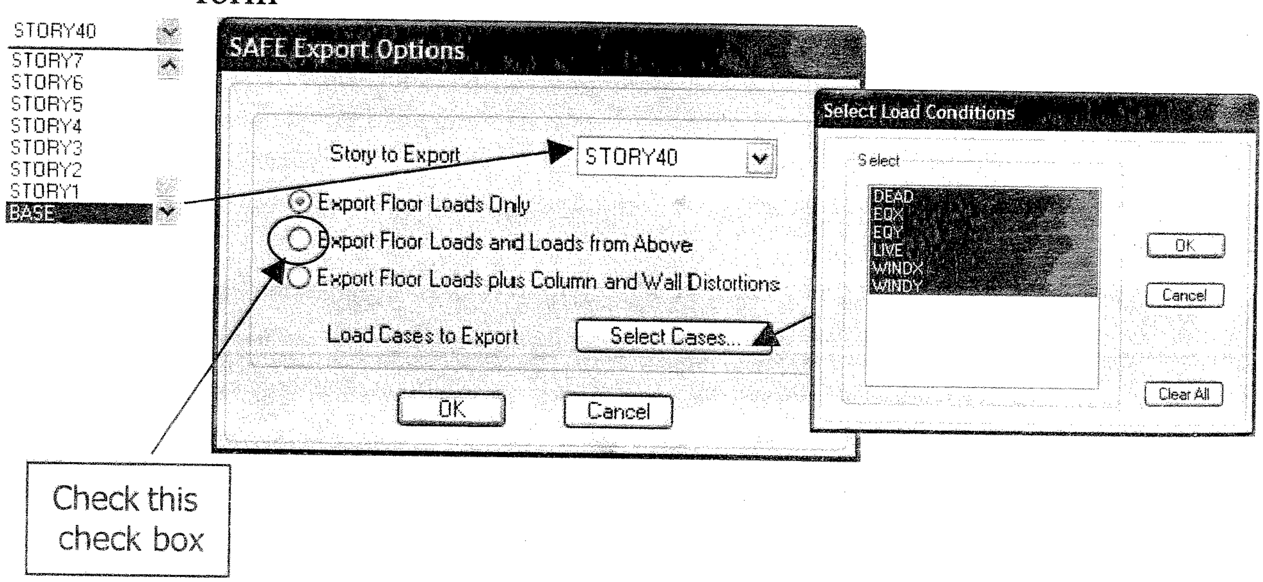
- **Etabs & Safe Program**

  - The program give to you the ability to export the floor plans to the safe program
  - You don't need to remodel the slabs again in the safe program, you can use the same floor plan of Etabs program with the effect of all static case of loading (Dead, Live, Earthquake X, Earthquake Y, Wind X, Wind Y)
  - Also the program give to you the ability to export the reactions of the walls and the columns to Safe Program to Solve the raft Foundation

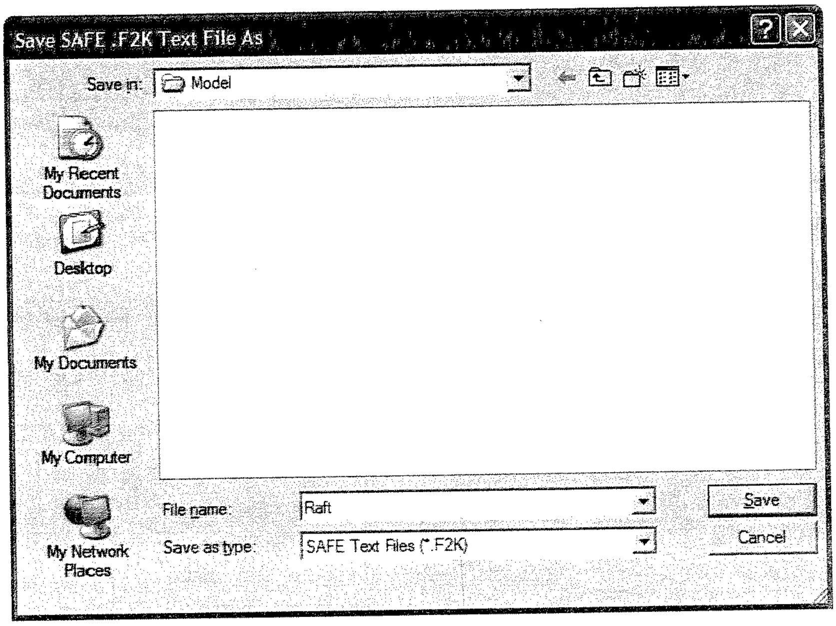
Note: to take the effect of the earthquake and wind load, you must run you model before exporting the reaction of the columns and walls or the floor plan

1. **Export of Reactions of the columns and Walls to Safe Program**

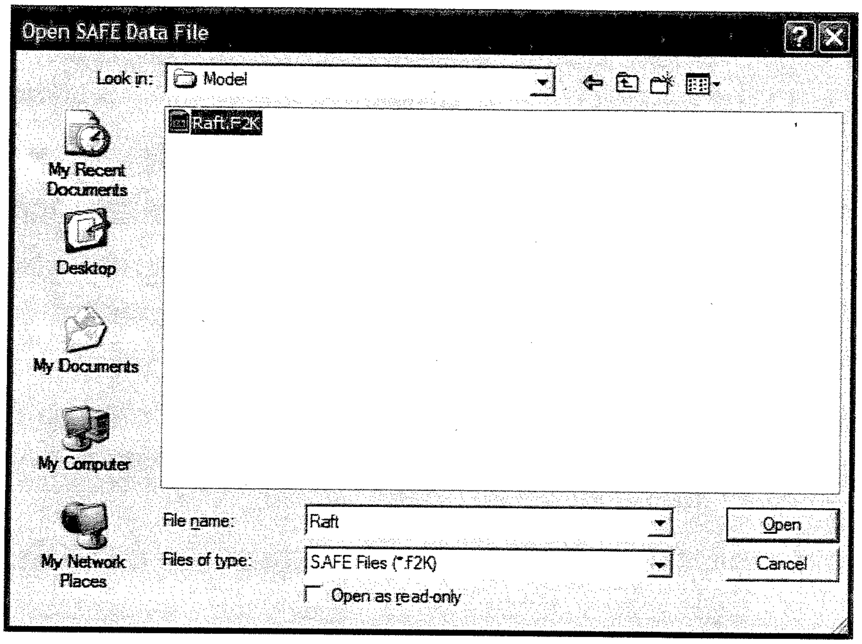
   - Click **File menu** ⟶ **Export** ⟶ **Safe Story as SAFE f2k Text file ,** which will Display the SAFE Export Options form

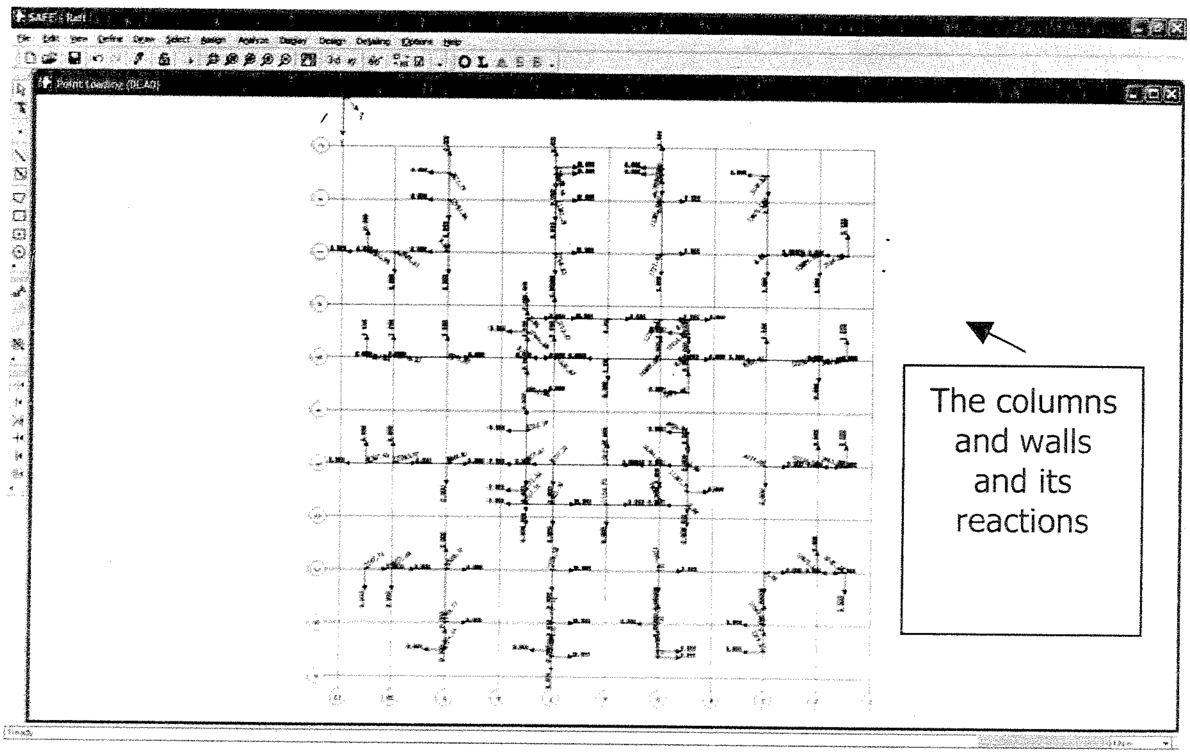


- Select the story level to export from the drop-down list of **Story to Export** to be Base.
- Check the box **of Export Floor Loads and Loads from Above** to take the effect of the loads of all floors
- Click Select Cases... button to display Select Load Conditions Form ,thin highlight all case of loading thin click **OK** ,thin click **OK** for the main form to display the Save of the file form, Name the file thin click **Save**



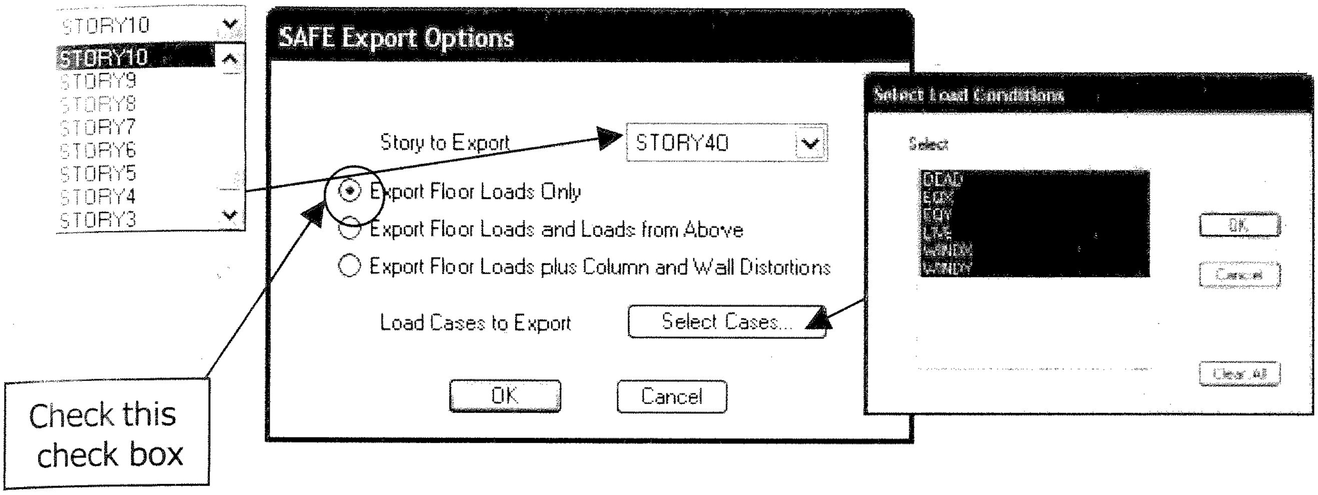
- Open safe program and click **File menu** → **Import** → **Safe v6/V7 F2K File** ,thin the next form will be displayed ,choose the file thin click **Open**



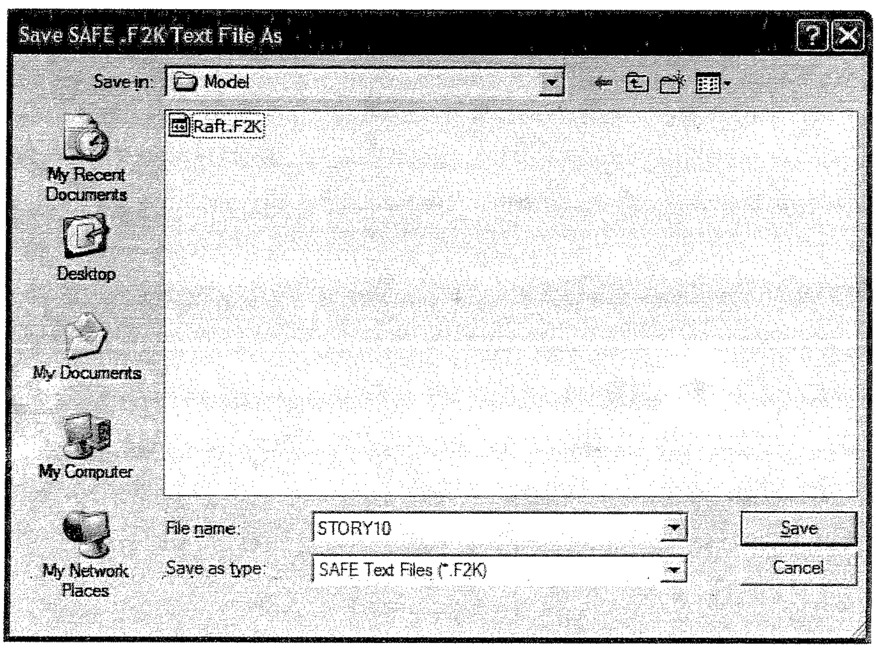


2. **Export of the floor plan to Safe Program**

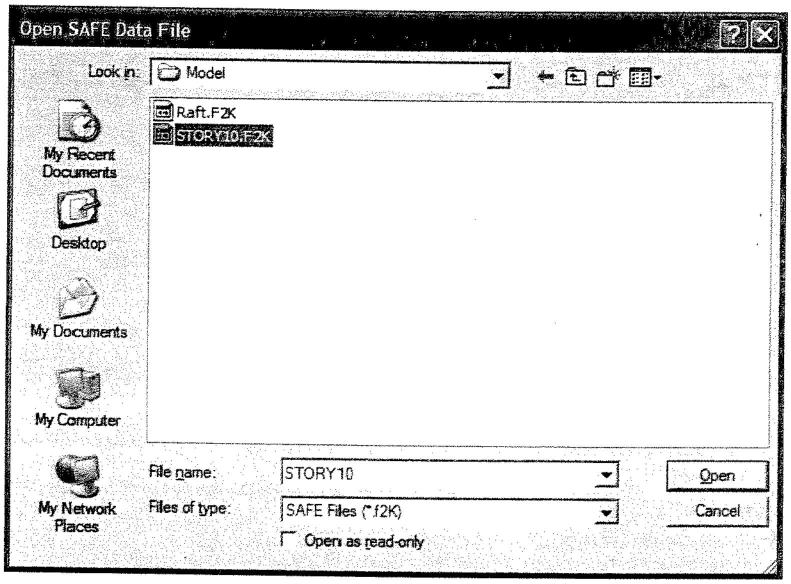
- Click **File menu** ——► **Export** ——► **Safe Story as SAFE f2k Text file ,** which will Display the SAFE Export Options form

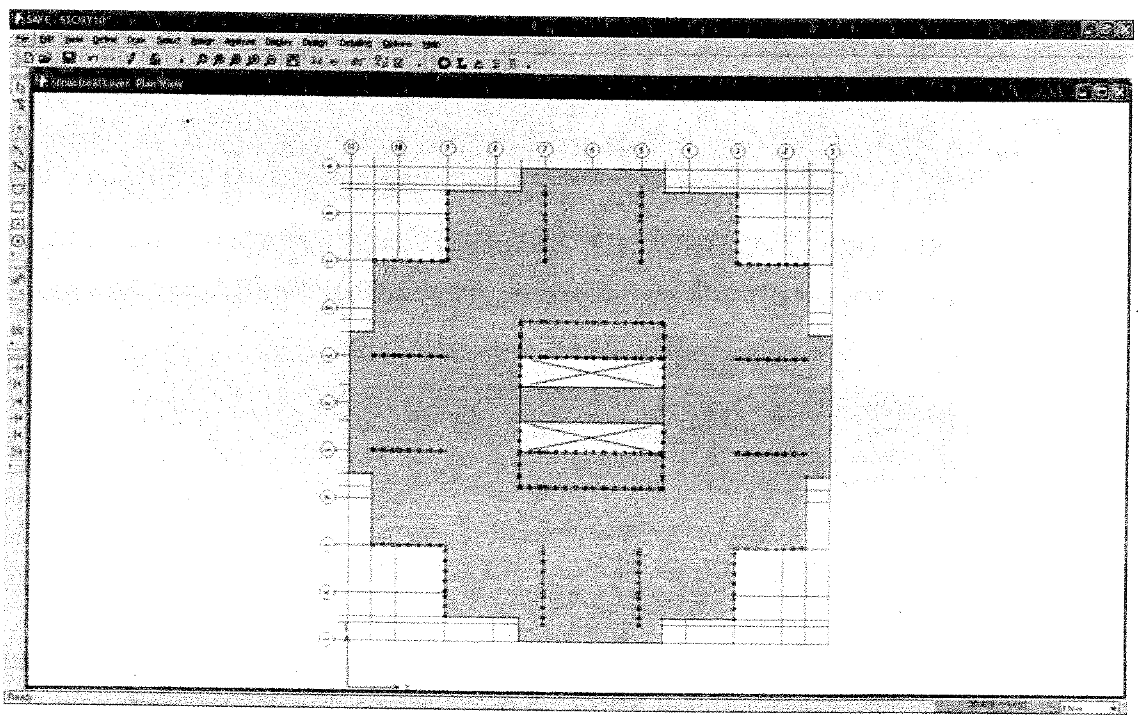


- Select the story level to export from the drop-down list of **Story to Export** to be Story 10 (as an example).
- Check the box **of Export Floor Loads Only** to take the effect of the loads of the floor and it is not recommended to check **Export Floor Loads plus Column and Wall Distortions** because as we explained in the chapter of construction sequence this deformation not have true value
- Click Select Cases... button to display Select Load Conditions Form ,thin highlight all case of loading thin click **OK** ,thin click **OK** for the main form to display the Save of the file form, name the file thin click **Save**



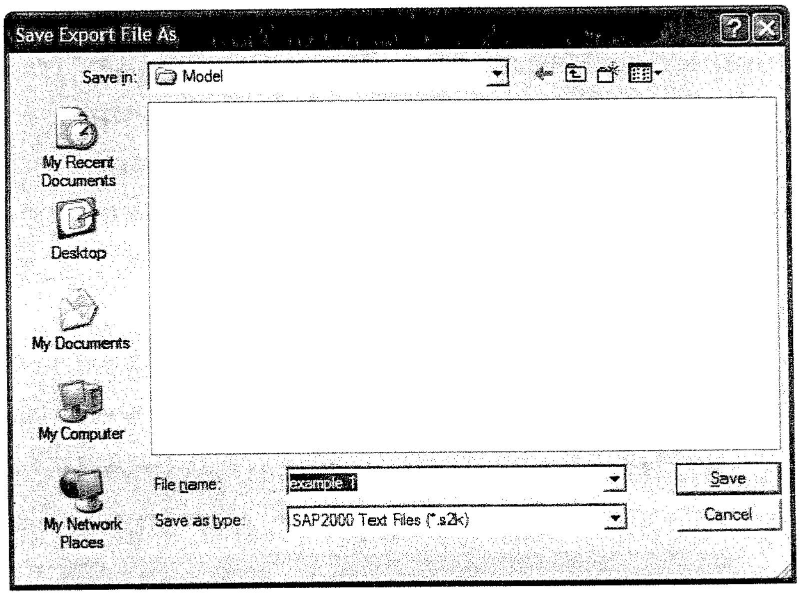
- Open safe program and click **File menu** ⟶ **Import** ⟶. **Safe v6/V7 F2K File** ,thin the next form will be displayed ,choose the file thin click **Open**



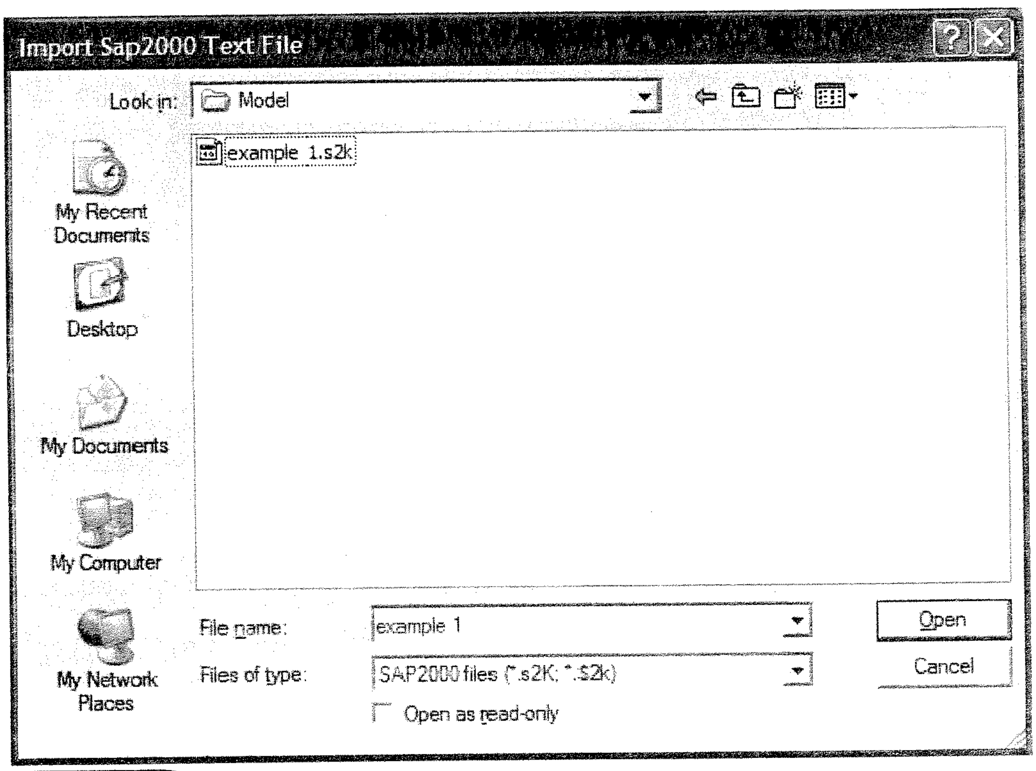


- **Etabs & SAP Program**
    - if you want to add certain other special purpose elements to your model which are not available in ETABS, such as Solid elements ,you can export your model to SAP program

- after you export your ETABS model to SAP2000, you can not import that SAP2000 model back into ETABS
- Click **File menu** ⟶ **Export** ⟶ **Safe Story as SAP2000 s2k Text file** , which will Display the Save Export File As form ,write the name of the model thin click **Save**



- Open SAP program and click **File menu** ⟶ **Import** ⟶ . **SAP V8/V9/V10 S2K Text File** ,thin the next form will be displayed ,choose the file thin click **Open**

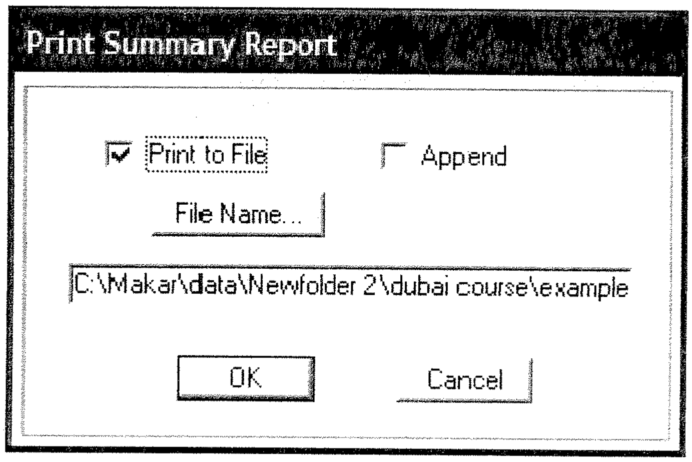


- **Etabs & Word Program**
- You can display any input or output result in tabular form which we can open it by word program ,thin you can adjust this result format according to your company format and use it in the calculation sheet of the building

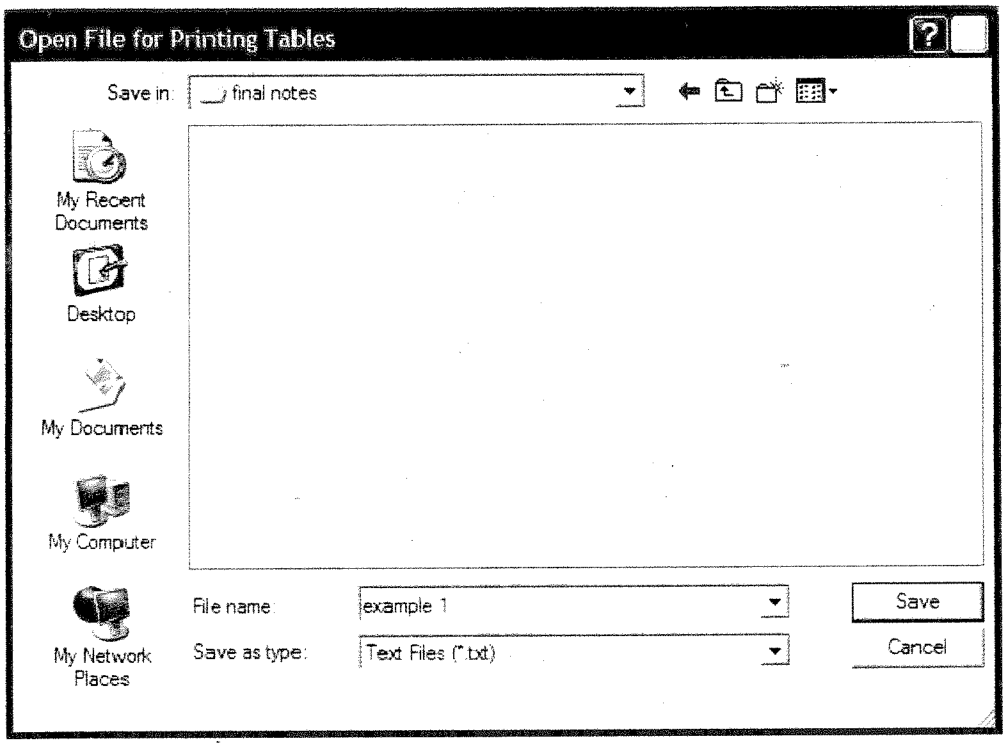
- As we explained before in the chapter of Display result The most important tabular review for the result is the summery report which gives to you the most important data for the model and the calculation details for wind, and earthquake loads, and the summary for the result

1. Summary Report

- Click the **File** ⟶ **Print Table** ⟶ **Summary Report,** thin the form as shown in fig will Display.



- Click File Name... to choose the location where the file will be saved using the next form



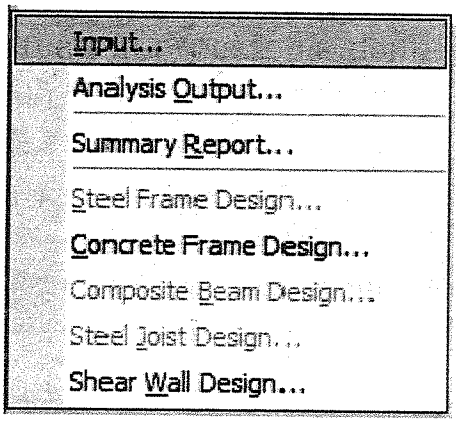
- The next form will show to you extract of some information in summary report (Earthquake calculation) example 1 - Notepad

File Edit Format View Help

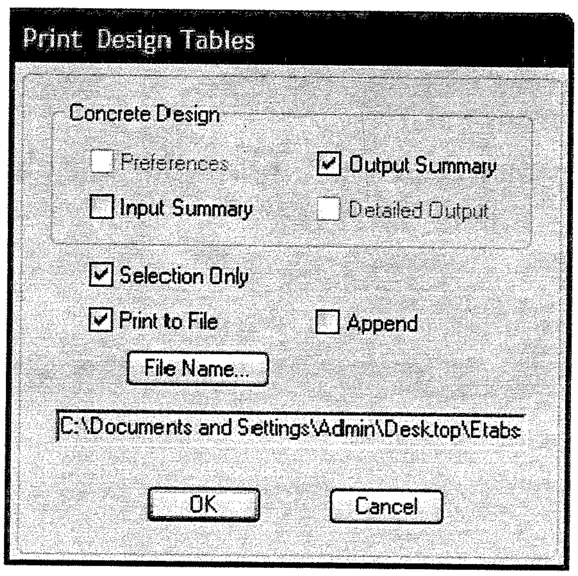
```
AUTO SEISMIC INPUT DATA

  Direction: X + EccY
  Typical Eccentricity = 5%
  Eccentricity Overrides: No

  Period Calculation: Program Calculated
  Ct = 0.02 (in feet units)

  Top Story: STORY40
  Bottom Story: BASE

  R = 5.5
  I = 1
  hn = 137.400 (Building Height)

  Soil Profile Type = SC
  Z = 0.15
  Ca = 0.1800
  Cv = 0.2500


AUTO SEISMIC CALCULATION FORMULAS

  Ta = Ct (hn^(3/4))

  If Z >= 0.35 (Zone 4) then:         If Tetabs <= 1.30 Ta then T = Tetabs, else T = Ta
  If Z < 0.35 (Zone 1, 2 or 3) then: If Tetabs <= 1.40 Ta then T = Tetabs, else T = Ta

  V = (Cv I W) / (R T)                                        (Eqn. 1)
  V <= 2.5 Ca I W / R                                         (Eqn. 2)
  V >= 0.11 Ca I W                                            (Eqn. 3)

  If T <= 0.7 sec, then Ft = 0
  If T > 0.7 sec, then Ft = 0.07 T V <= 0.25 V


AUTO SEISMIC CALCULATION RESULTS

  Ta = 1.9566 sec
  T Used = 2.7393 sec
  W Used = 499320.21

  V (Eqn 1) = 0.0166W
  V (Eqn 2) = 0.0818W
  V (Eqn 3) = 0.0198W
  V (Eqn 4) = 0.0349W

  V Used = 0.0198W = 9886.54

  Ft Used = 1895.74


AUTO SEISMIC STORY FORCES

  STORY             FX        FY        FZ        MX        MY          MZ

  STORY40      1966.33      0.00      0.00     0.000     0.000   -1376.430
  STORY39       306.18      0.00      0.00     0.000     0.000    -489.882
```

You can see from previous form the whole steps the program do to calculate the seismic force and the values of the force acting in each floor in details ,also the program give to you the details for the wind calculation , summary of the input data .and the results

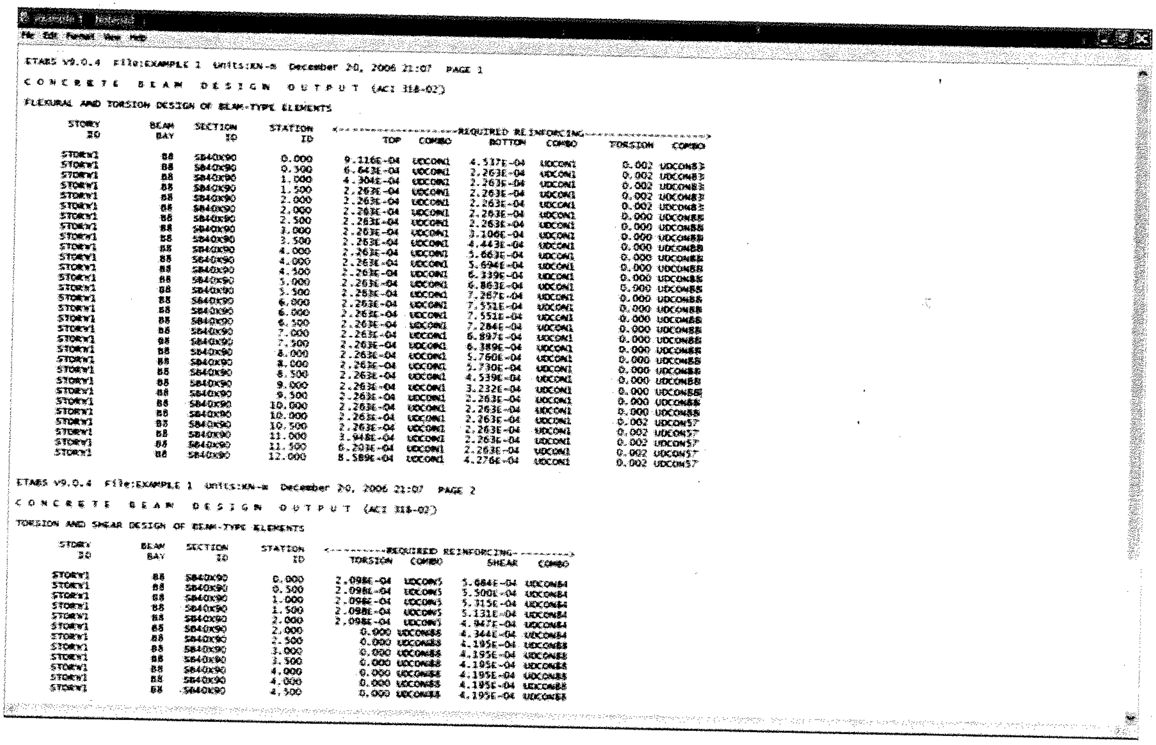
- Also you can display the result mentioned in the next form and as example of this result we will display the concrete frame design for one of the beams of the tower



- Select the element you want to display the result for it
- Click the **File** ⟶ **Print Table** ⟶ **Concrete Frame Design;** thin the form as shown in fig will be Displayed.



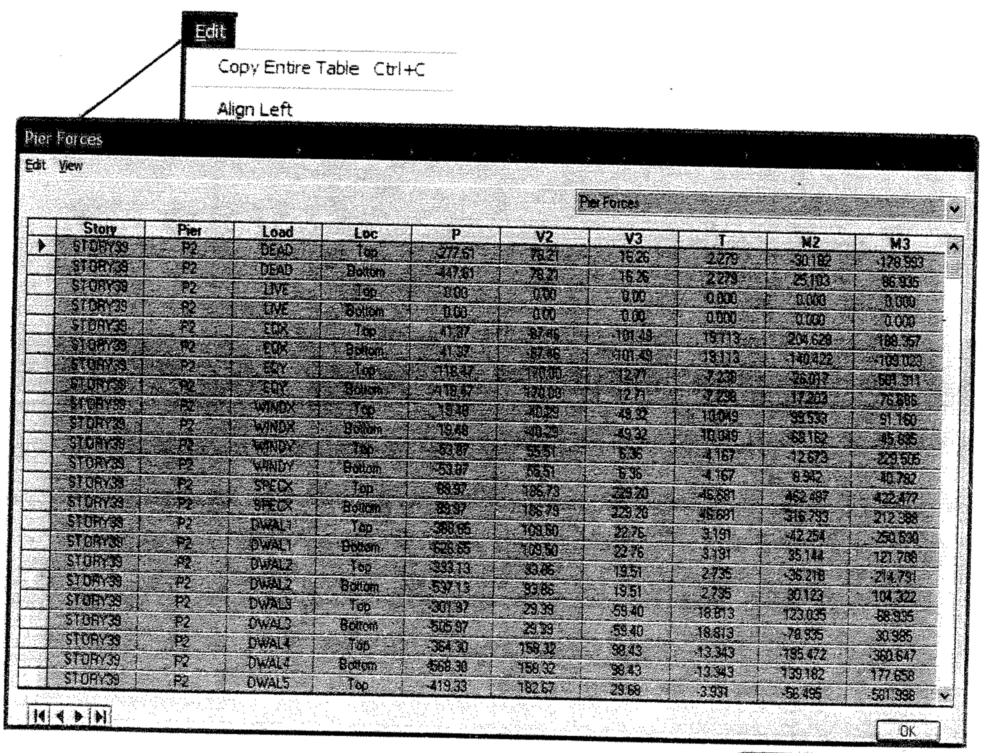
- Highlight **output Summary** and **print to File** thin **OK** ,thin open the result file in word



```
ETABS v9.0.4  File:EXAMPLE 1  Units:KN-m  December 20, 2006 21:07  PAGE 1

C O N C R E T E   B E A M   D E S I G N   O U T P U T  (ACI 318-02)

FLEXURAL AND TORSION DESIGN OF BEAM-TYPE ELEMENTS

   STORY      BEAM  SECTION   STATION   <------------------REQUIRED REINFORCING------------------>
      ID       BAY       ID        ID        TOP    COMBO      BOTTOM   COMBO     TORSION  COMBO

  STORY1       B8   S840x90     0.000  9.116E-04  UDCON1  4.537E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     0.500  6.643E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     1.000  4.304E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     1.500  2.263E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     2.000  2.263E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     2.000  2.263E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON83
  STORY1       B8   S840x90     2.500  2.263E-04  UDCON1  2.263E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     3.000  2.263E-04  UDCON1  3.106E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     3.500  2.263E-04  UDCON1  4.443E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     4.000  2.263E-04  UDCON1  5.663E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     4.000  2.263E-04  UDCON1  5.694E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     4.500  2.263E-04  UDCON1  6.339E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     5.000  2.263E-04  UDCON1  6.863E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     5.500  2.263E-04  UDCON1  7.287E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     6.000  2.263E-04  UDCON1  7.551E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     6.000  2.263E-04  UDCON1  7.551E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     6.500  2.263E-04  UDCON1  7.284E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     7.000  2.263E-04  UDCON1  6.897E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     7.500  2.263E-04  UDCON1  6.389E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     8.000  2.263E-04  UDCON1  5.760E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     8.000  2.263E-04  UDCON1  5.730E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     8.500  2.263E-04  UDCON1  4.539E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     9.000  2.263E-04  UDCON1  3.232E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90     9.500  2.263E-04  UDCON1  2.263E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90    10.000  2.263E-04  UDCON1  2.263E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90    10.000  2.263E-04  UDCON1  2.263E-04  UDCON1     0.000  UDCON88
  STORY1       B8   S840x90    10.500  2.263E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON57
  STORY1       B8   S840x90    11.000  3.948E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON57
  STORY1       B8   S840x90    11.500  6.203E-04  UDCON1  2.263E-04  UDCON1     0.002  UDCON57
  STORY1       B8   S840x90    12.000  8.589E-04  UDCON1  4.276E-04  UDCON1     0.002  UDCON57

ETABS v9.0.4  File:EXAMPLE 1  Units:KN-m  December 20, 2006 21:07  PAGE 2

C O N C R E T E   B E A M   D E S I G N   O U T P U T  (ACI 318-02)

TORSION AND SHEAR DESIGN OF BEAM-TYPE ELEMENTS

   STORY      BEAM  SECTION   STATION  <---------REQUIRED REINFORCING--------->
      ID       BAY       ID        ID     TORSION   COMBO       SHEAR   COMBO

  STORY1       B8   S840x90     0.000  2.098E-04  UDCON5   5.684E-04  UDCON84
  STORY1       B8   S840x90     0.500  2.098E-04  UDCON5   5.500E-04  UDCON84
  STORY1       B8   S840x90     1.000  2.098E-04  UDCON5   5.315E-04  UDCON84
  STORY1       B8   S840x90     1.500  2.098E-04  UDCON5   5.131E-04  UDCON84
  STORY1       B8   S840x90     2.000  2.098E-04  UDCON5   4.947E-04  UDCON84
  STORY1       B8   S840x90     2.000      0.000  UDCON88  4.344E-04  UDCON84
  STORY1       B8   S840x90     2.500      0.000  UDCON88  4.195E-04  UDCON88
  STORY1       B8   S840x90     3.000      0.000  UDCON88  4.195E-04  UDCON88
  STORY1       B8   S840x90     3.500      0.000  UDCON88  4.195E-04  UDCON88
  STORY1       B8   S840x90     4.000      0.000  UDCON88  4.195E-04  UDCON88
  STORY1       B8   S840x90     4.000      0.000  UDCON88  4.195E-04  UDCON88
  STORY1       B8   S840x90     4.500      0.000  UDCON88  4.195E-04  UDCON88
```

Also you can Export the Etabs file to text file (E2K) which is less in size and you can imported again to the program without any effect to the model

- **Etabs & Excel program**
  - After you display the result in tabular for click **Edit** ⟶ . **Copy Entire Table Ctrl+C**

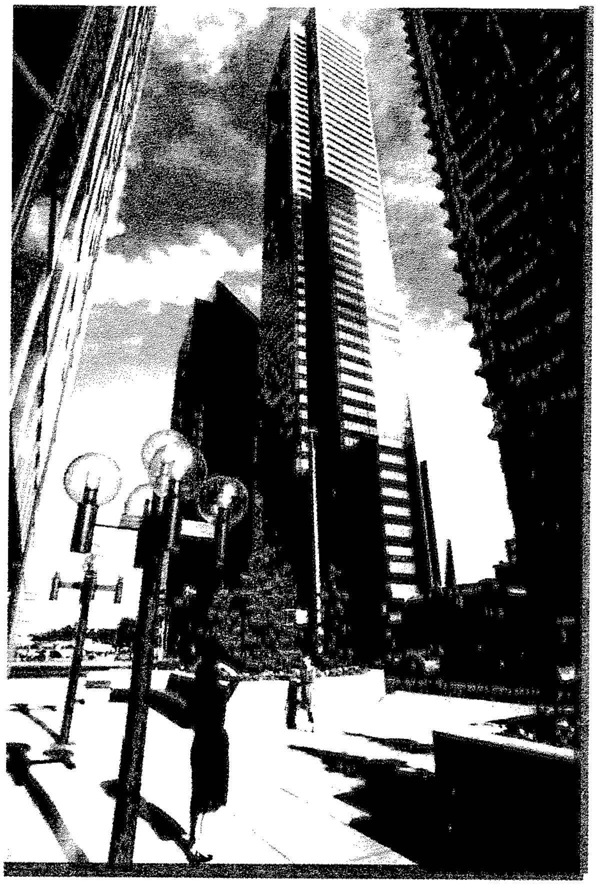


Story	Pier	Load	Loc	P	V2	V3	T	M2	M3
STORY39	P2	DEAD	Top	-277.61	78.21	16.26	2.279	-30.182	-178.593
STORY39	P2	DEAD	Bottom	-447.61	78.21	16.26	2.279	25.103	86.935
STORY39	P2	LIVE	Top	0.00	0.00	0.00	0.000	0.000	0.000
STORY39	P2	LIVE	Bottom	0.00	0.00	0.00	0.000	0.000	0.000
STORY39	P2	EQX	Top	41.37	-87.46	-101.49	19.113	204.629	188.357
STORY39	P2	EQX	Bottom	41.37	-87.46	-101.49	19.113	-140.422	-109.023
STORY39	P2	EQY	Top	-118.47	170.00	12.71	-7.238	-26.017	581.311
STORY39	P2	EQY	Bottom	-118.47	170.00	12.71	-7.238	17.203	76.886
STORY39	P2	WINDX	Top	19.48	-40.29	-49.32	10.049	99.539	91.160
STORY39	P2	WINDX	Bottom	19.48	-40.29	-49.32	10.049	-68.162	45.695
STORY39	P2	WINDY	Top	-53.87	55.51	6.36	-4.167	12.673	-229.505
STORY39	P2	WINDY	Bottom	-53.87	55.51	6.36	-4.167	8.942	40.782
STORY39	P2	SPECX	Top	88.97	186.73	229.20	45.681	462.497	422.477
STORY39	P2	SPECX	Bottom	88.97	186.73	229.20	45.681	316.793	212.388
STORY39	P2	DWAL1	Top	-388.65	109.50	22.76	3.191	-42.254	-250.630
STORY39	P2	DWAL1	Bottom	-526.65	109.50	22.76	3.191	35.144	121.708
STORY39	P2	DWAL2	Top	-333.13	93.86	19.51	2.735	-35.218	-214.791
STORY39	P2	DWAL2	Bottom	-537.13	93.86	19.51	2.735	30.123	104.322
STORY39	P2	DWAL3	Top	-301.97	29.39	-59.40	18.813	123.035	-88.935
STORY39	P2	DWAL3	Bottom	-505.97	29.39	-59.40	18.813	-78.935	30.985
STORY39	P2	DWAL4	Top	-364.30	158.32	98.43	-13.343	-195.472	-360.647
STORY39	P2	DWAL4	Bottom	-568.30	158.32	98.43	-13.343	139.182	177.658
STORY39	P2	DWAL5	Top	-419.33	182.67	29.68	-3.931	-56.495	-581.998

- Thin open Excel program and Paste the data and you can work with file with all the utility of Excel program

1	Story	Pier	Load	Loc	P	V2	V3	T	M2	M3
2	STORY39	P2	DEAD	Top	-277.61	78.21	16.26	2.279	-30.182	-178.993
3	STORY39	P2	DEAD	Bottom	-447.61	78.21	16.26	2.279	25.103	86.935
4	STORY39	P2	EQX	Top	41.37	-87.46	-101.49	19.113	204.628	188.357
5	STORY39	P2	EQX	Bottom	41.37	-87.46	-101.49	19.113	-140.422	-109.023
6	STORY39	P2	EQY	Top	-118.47	170	12.71	-7.238	-26.017	-501.311
7	STORY39	P2	EQY	Bottom	-118.47	170	12.71	-7.238	17.203	76.686
8	STORY39	P2	WINDX	Top	19.48	-40.29	-49.32	10.049	99.533	91.16
9	STORY39	P2	WINDX	Bottom	19.48	-40.29	-49.32	10.049	-68.162	-45.835
10	STORY39	P2	WINDY	Top	-53.87	55.51	6.36	-4.167	-12.673	-229.505
11	STORY39	P2	WINDY	Bottom	-53.87	55.51	6.36	-4.167	8.942	-40.782
12	STORY39	P2	SPECX	Top	89.97	186.73	229.2	46.681	462.487	422.477
13	STORY39	P2	SPECX	Bottom	89.97	186.73	229.2	46.681	316.793	212.388
14	STORY39	P2	DWAL1	Top	-388.65	109.5	22.76	3.191	-42.254	-250.59
15	STORY39	P2	DWAL1	Bottom	-626.65	109.5	22.76	3.191	35.144	121.708
16	STORY39	P2	DWAL2	Top	-333.13	93.86	19.51	2.735	-36.218	-214.791
17	STORY39	P2	DWAL2	Bottom	-537.13	93.86	19.51	2.735	30.123	104.322
18	STORY39	P2	DWAL3	Top	-301.97	29.39	-59.4	18.813	123.035	-68.935
19	STORY39	P2	DWAL3	Bottom	-505.97	29.39	-59.4	18.813	-78.935	30.985
20	STORY39	P2	DWAL4	Top	-364.3	158.32	98.43	-13.343	-195.472	-360.647
21	STORY39	P2	DWAL4	Bottom	-568.3	158.32	98.43	-13.343	139.182	177.658
22	STORY39	P2	DWAL5	Top	-419.33	182.67	29.68	-3.931	-56.495	-581.998
23	STORY39	P2	DWAL5	Bottom	-623.33	182.67	29.68	-3.931	44.43	39.071
24	STORY39	P2	DWAL6	Top	-246.94	5.05	9.34	9.402	-15.941	152.416
25	STORY39	P2	DWAL6	Bottom	-450.94	5.05	9.34	9.402	15.817	169.572
26	STORY39	P2	DWAL7	Top	-317.55	61.62	-19.95	10.774	43.409	-141.863
27	STORY39	P2	DWAL7	Bottom	-521.55	61.62	-19.95	10.774	-24.406	67.653
28	STORY39	P2	DWAL8	Top	-348.71	126.09	58.97	-5.304	-115.845	-287.719
29	STORY39	P2	DWAL8	Bottom	-552.71	126.09	58.97	-5.304	84.653	140.99
30	STORY39	P2	DWAL9	Top	-376.23	138.26	24.6	-0.598	-46.357	-398.395
31	STORY39	P2	DWAL9	Bottom	-580.23	138.26	24.6	-0.598	37.277	71.696
32	STORY39	P2	DWAL10	Top	-290.03	49.45	14.43	6.069	-26.08	-31.168
33	STORY39	P2	DWAL10	Bottom	-494.03	49.45	14.43	6.069	22.97	138.947
34	STORY39	P2	DWAL11	Top	-218.68	5.92	-84.28	18.13	132.09	-15.238

Sheet1 / Sheet2 / Sheet3



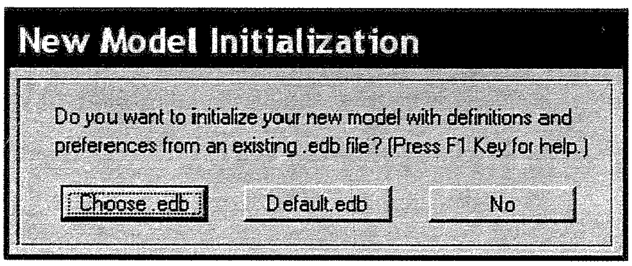
HOW TO MODEL AND DESIGN
HIGH RISE BUILDING USING
ETABS Program

Important Notes

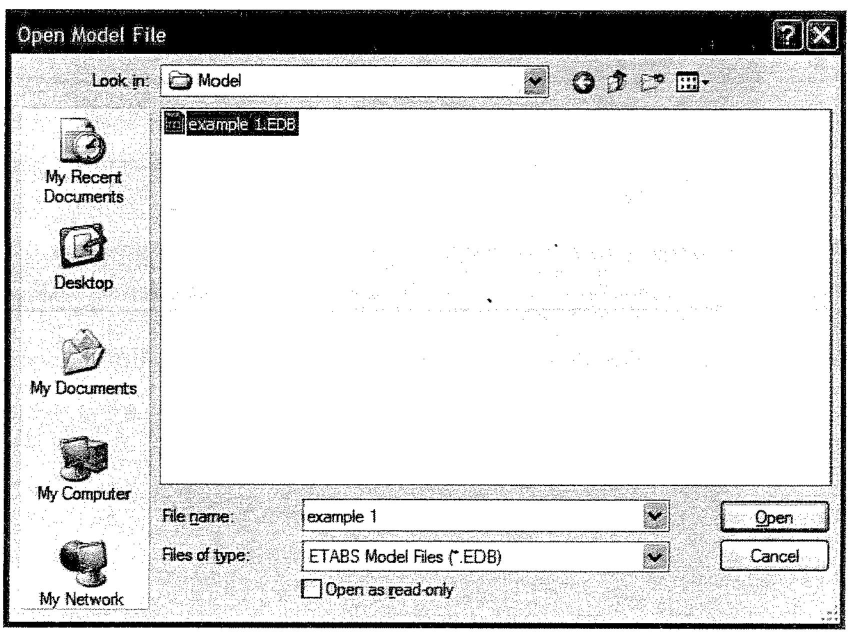
Chapter 12

New Model Initialization

- When You Begin the model you can safe more than 40 % of the modeling time this by using the Definitions from existing model
- The program will import all the data from the existing file except
    - Grid Lines
    - Story Data
    - Objects
    - Assignments to objects
    - Information on the number of windows
- Click the **File menu** ⟶ **New Model** the next form will display

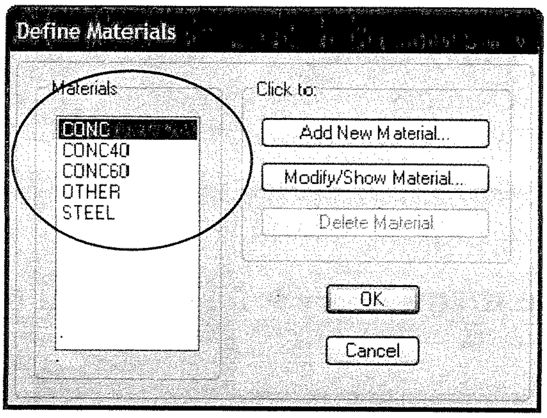


- From the previous form choose Choose.edb the program will open menu of **Open Model File**

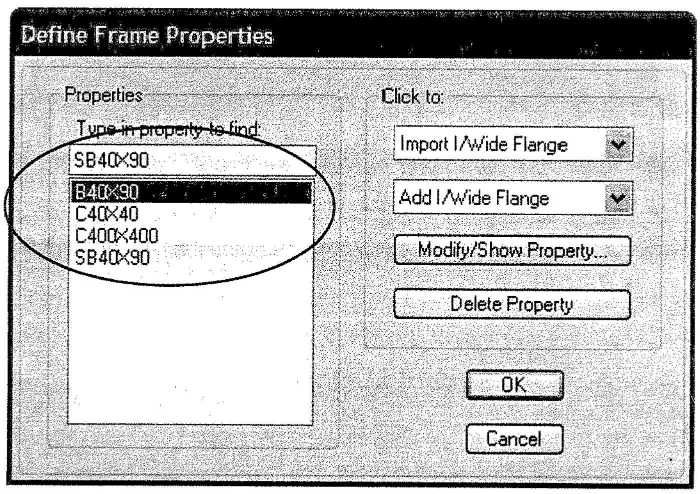


- From this menu choose the model which we created together thin click **OK**
- The first thing you will notice the units of the model changed to be KN.m

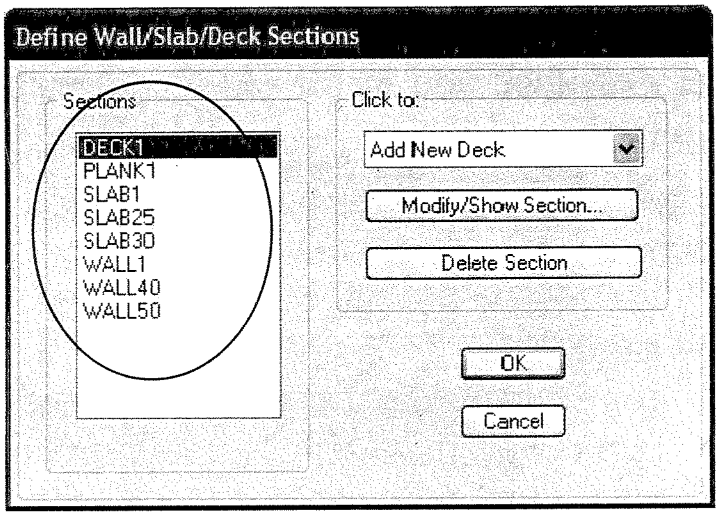
- All material definition will be directly imported to the new model



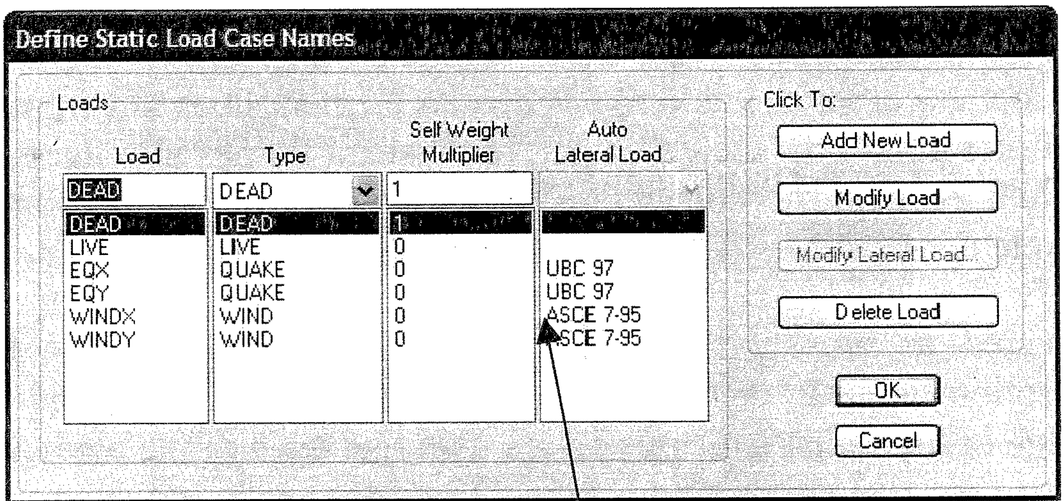
- All the definition of the frame sections will be imported to new model

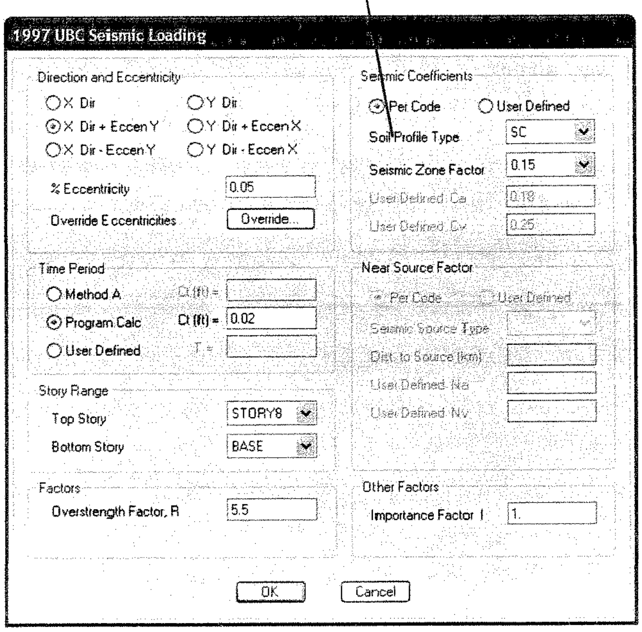


- All the definition of the walls and slabs sections will be imported to new model



- All the definition of static and dynamic case of loading will be imported to new model





- And also the load of combinations and the design codes and any other definition add in the old model will be imported to the new model ….

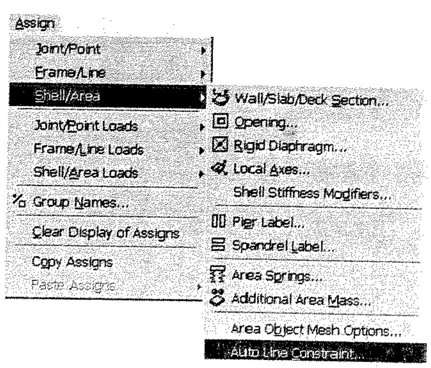
- You can adjust the file of etabs and make it as default for the program for any new model this by

    1. name the etab file default .edb
    2. copy this file to the folder of etab program (c/program files /computer and structure/etabs)
    3. when you begin a new model choose default .edb and the program will imported all this data directly

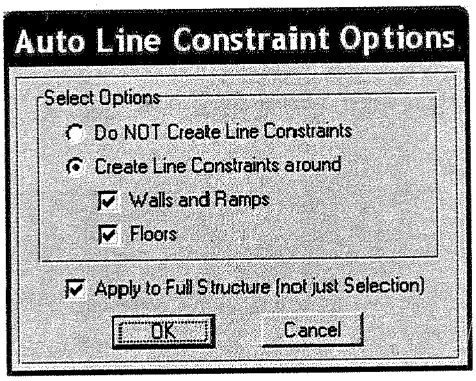
Auto Line Constraints

In finite element analysis, shell elements are connected to other elements at corner points only. When an element does not frame into the corner point of a shell element, but instead frames into the edge of the shell element, no connection exists between the element and the shell element. The ETABS auto line constraints feature allows you to specify that elements framing into the edge of a shell element be connected to the shell element. ETABS internally takes care of connection between the elements by constraining points lying along an edge of the shell element to move with that edge of the element.

- Before you make Run to any Model you must make Auto Line Constraints for the model.
  - Choose the whole building
  - Click the **Assign menu** ──► **Shell/Area** ──► **Auto Line Constraint...**

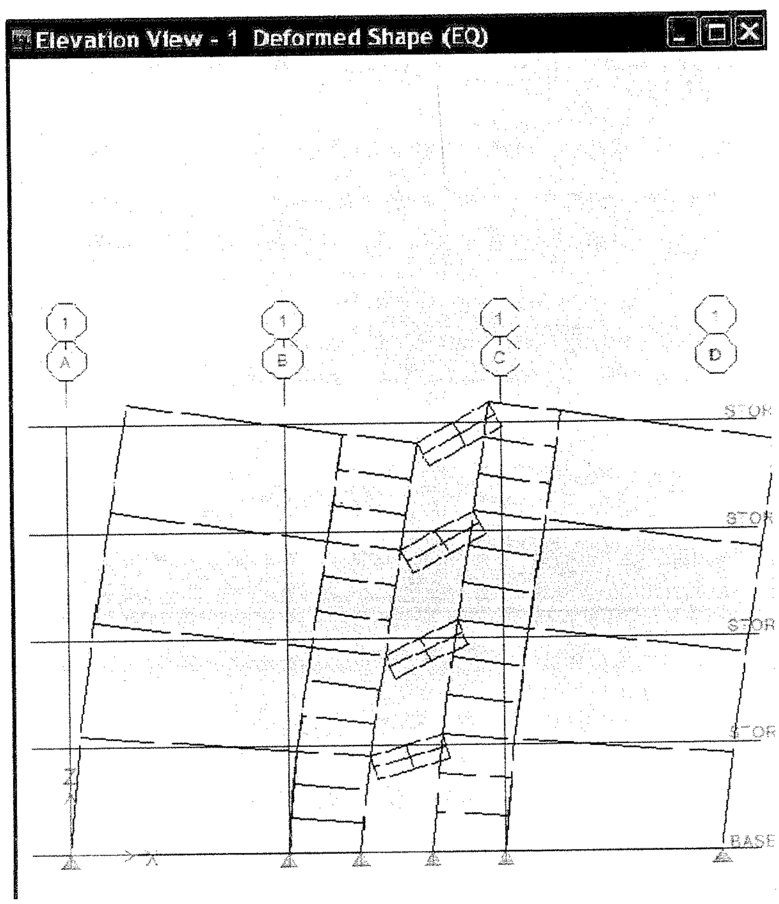


- The next form of **Auto Line Constraint options** Will display Choose create line constraints around (Walls and Ramps, Floors) thin click Ok

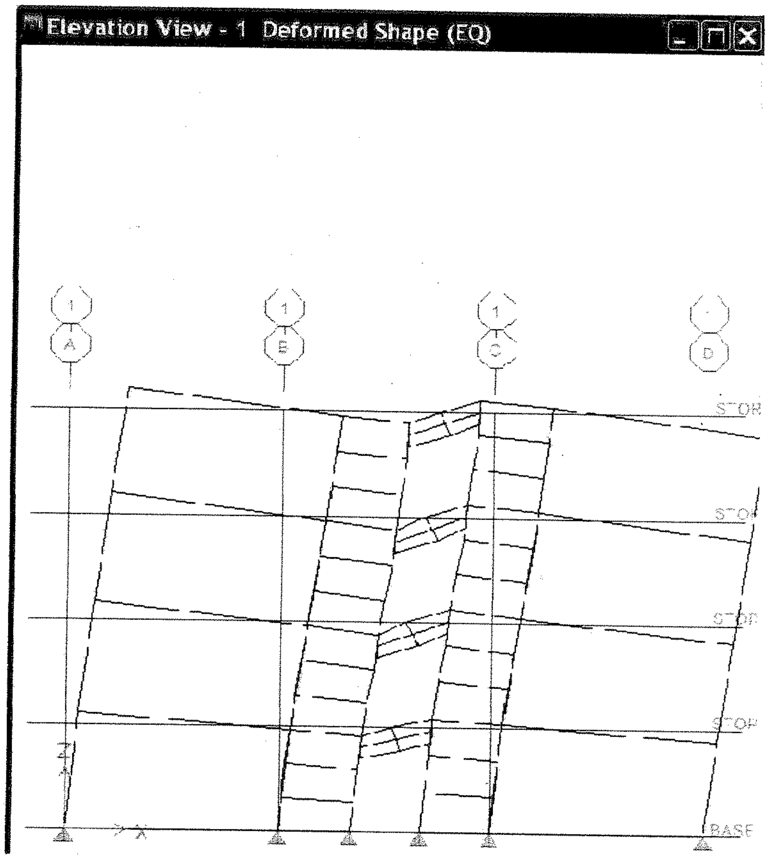


- When you use Auto Line Constraints Options always check the box of **Apply to Full Structure**

- We have example for model run with Auto Line Constraint and another one run with Auto Line Constraints
    - o Model without Auto Line Constraints

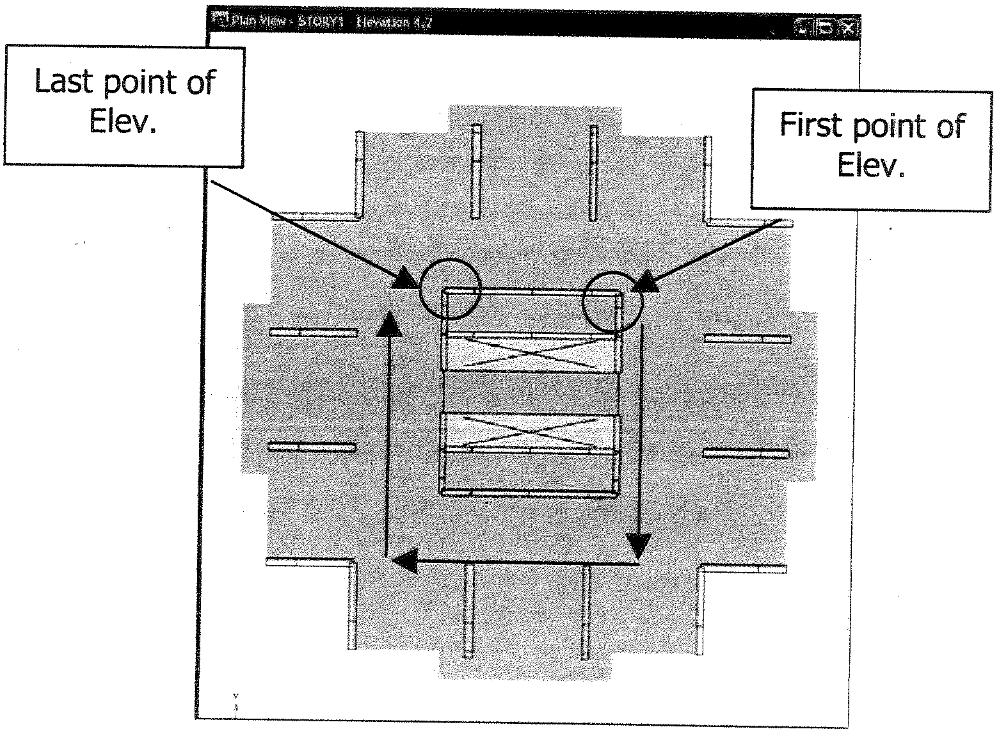


- The same Model with Auto Line Constraints



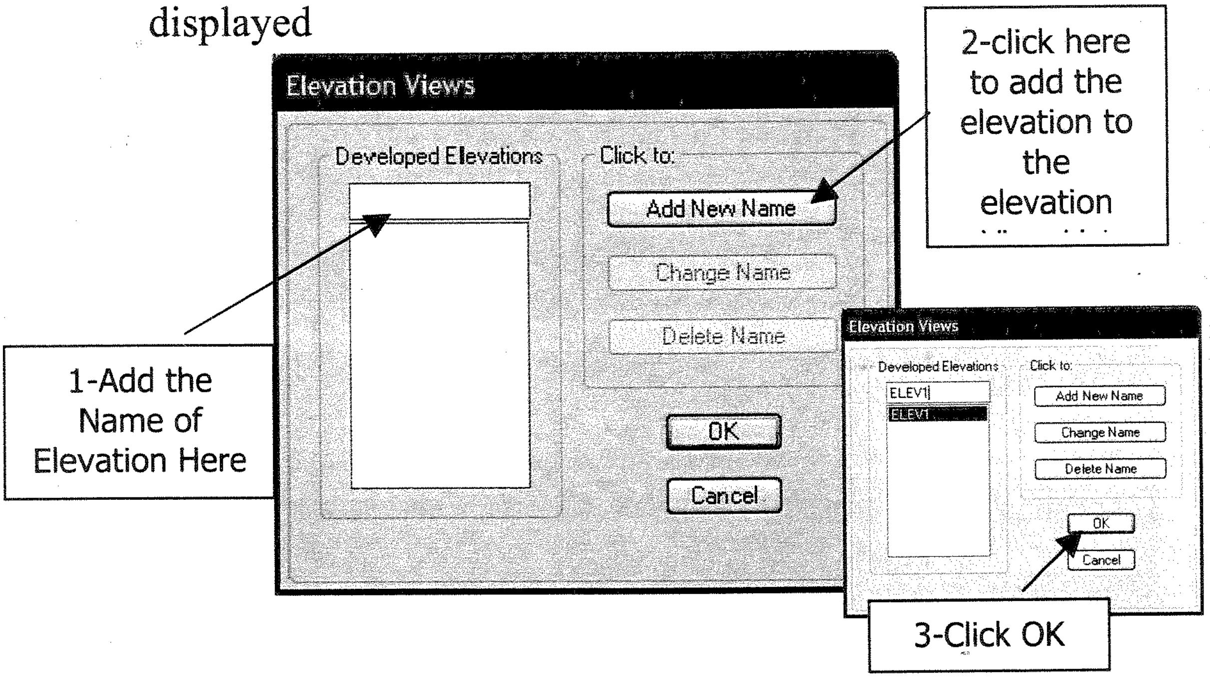
Developed Elevation:

You can develop elevation foe any part of the building as follow

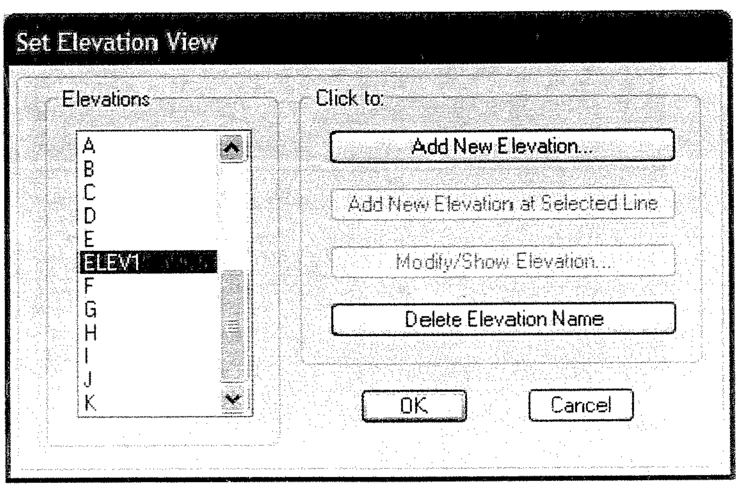


To make elevation view for this part of the core

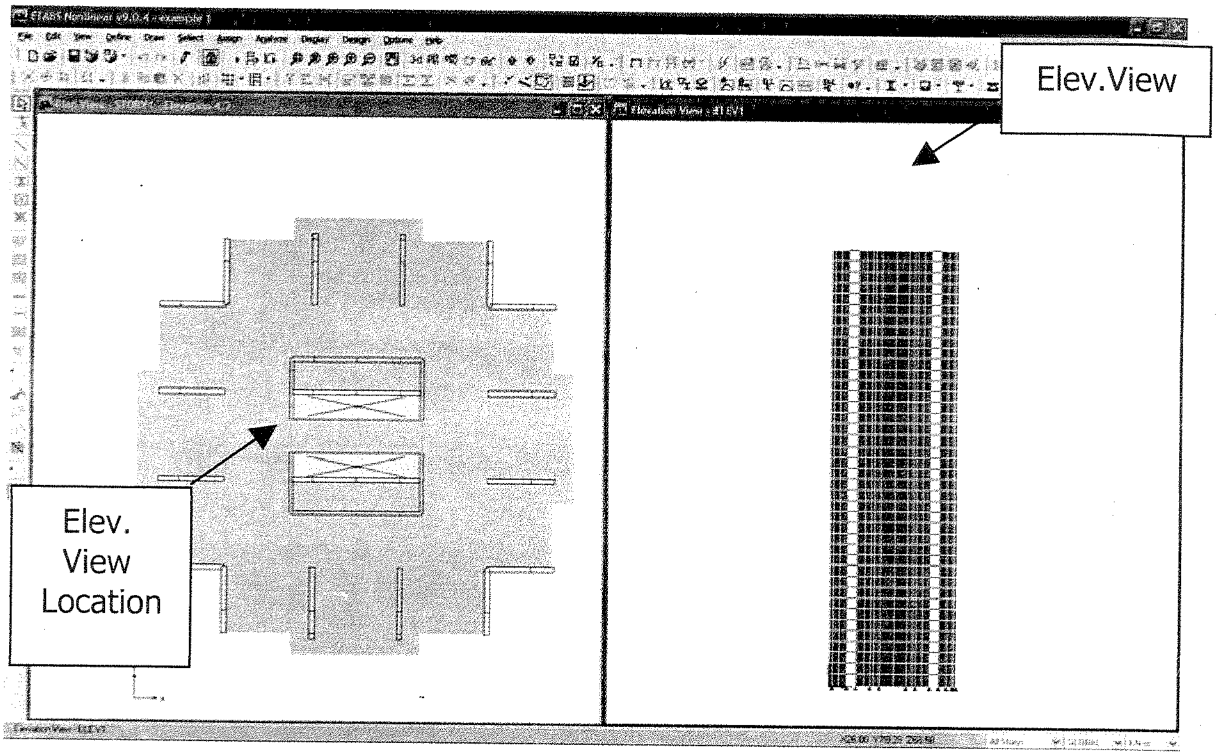
1. Click **Draw menu** ——▶ **Draw Developed Elevation Definition** ,thin the next form of Elevation View will be displayed



2. After you add the name of the view Click **OK** button, thin the crasser for Selection the elevation will be displayed in the plan view. click on the first point to the end point passing be the intermediate points like you draw it thin click **escape** button
3. click **Set Elevation view** Button , the next form will be displayed



4. select the elevation view name (ELEV1),thin click **OK** button,and the elevation view will be displayed

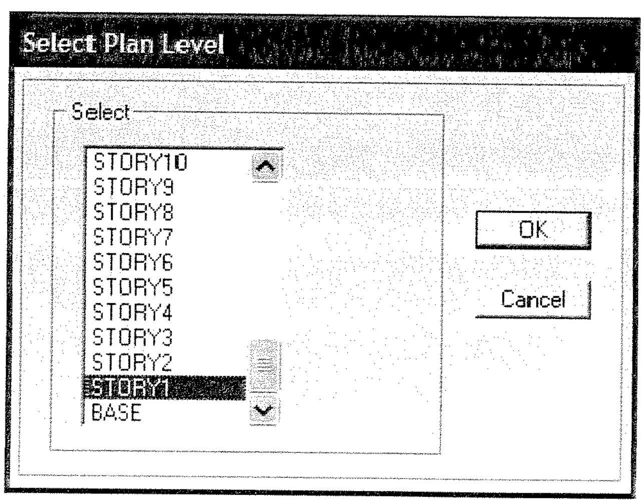


Add the Geometry of the Buildings

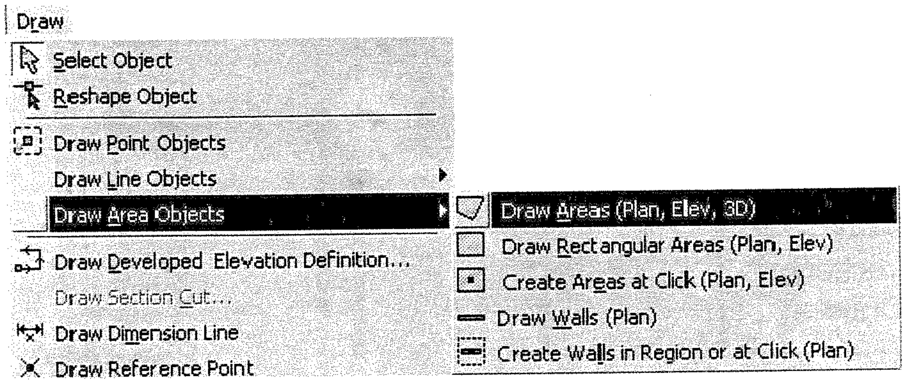
In this step, we will draw the Structure elements using the program utility, and we will add the same plan of our example using the program utility

1. Add Slabs:

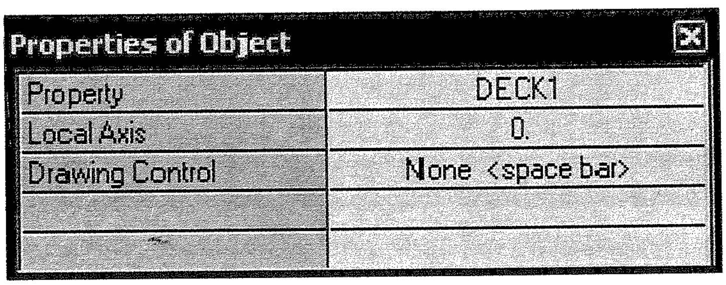
1. Make sure that the plan view is the view of story level ,if not ,Click set plan view button the next form will be displayed, from this form choose story level



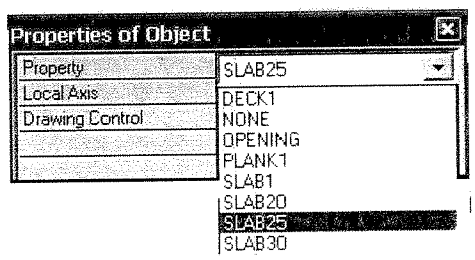
2 Click the **Draw menu** → **Draw Area Objects** → **Draw Area**

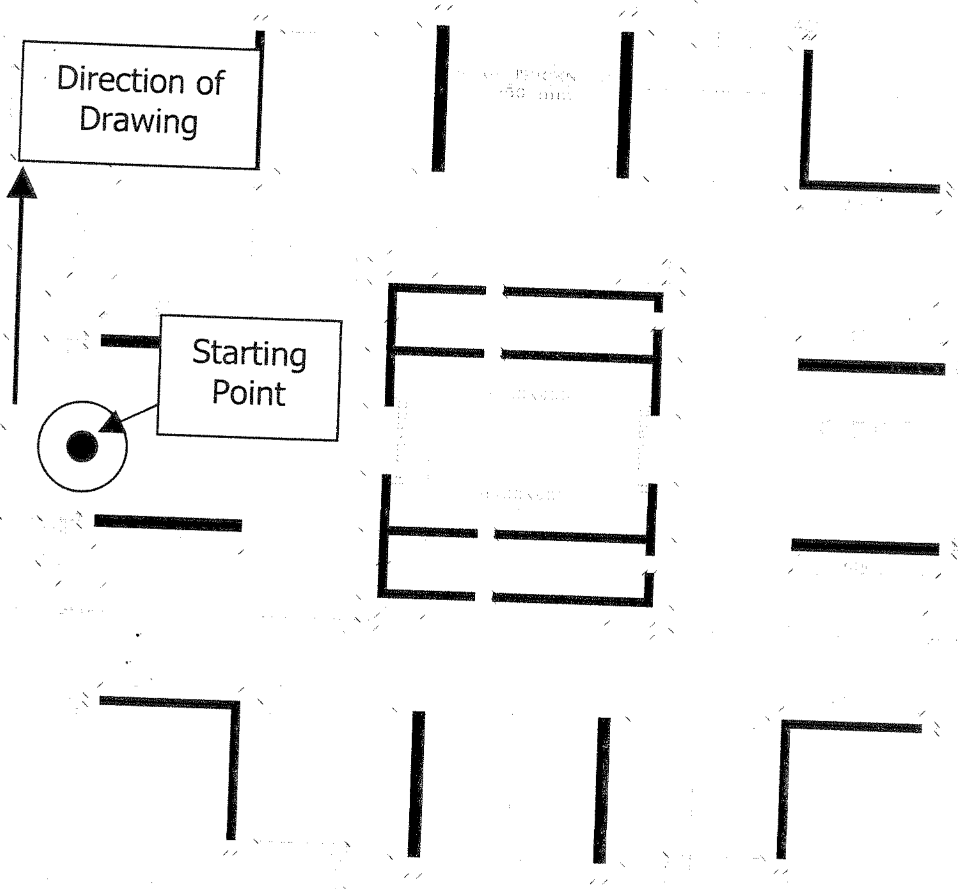


Or Draw Area button thin the Properties of Object form will display as shown in fig. and crasser for drawing Area



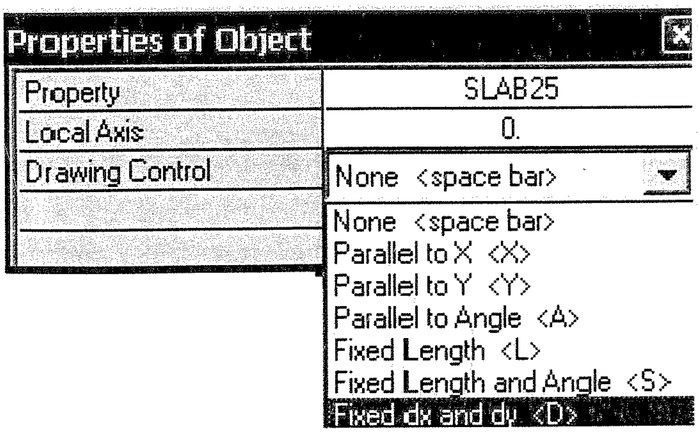
- In previous form you can
  - Choose area thickness for the Slab you was defined before from **Property**
  - Change the local axis of area
  - Using Drawing control for drawings
  - Set the Property item in the box Slab 25



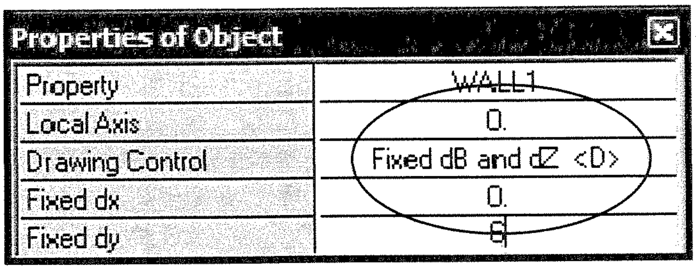


3. Click on the first point for drawing as it is mentioned in the previous drawing

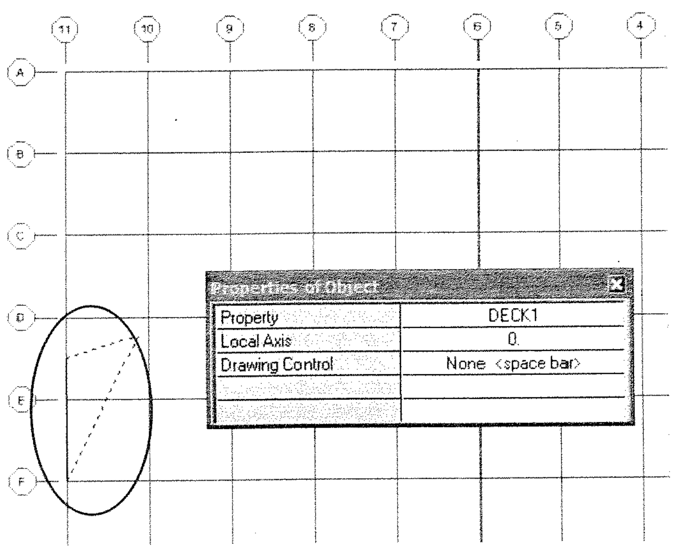
- Click Drawing Control pull down menu and choose Fixed dx and dy



- Write the coordinate value with reference to the starting point

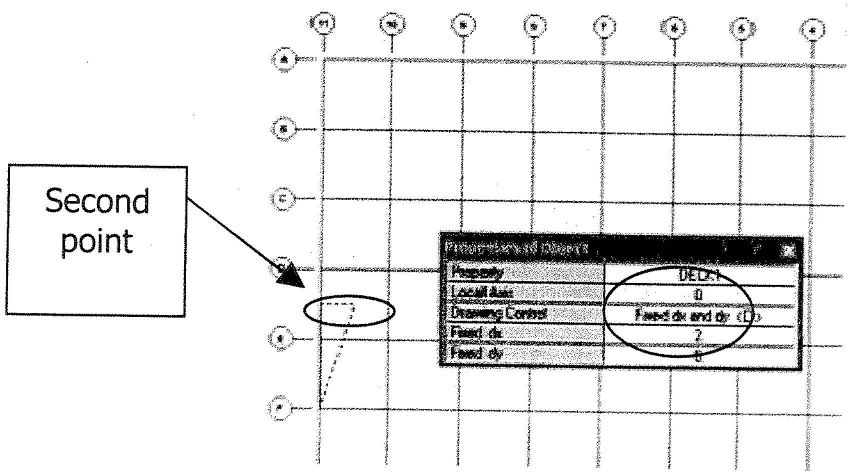


- Thin click any where the program will select the second point directly

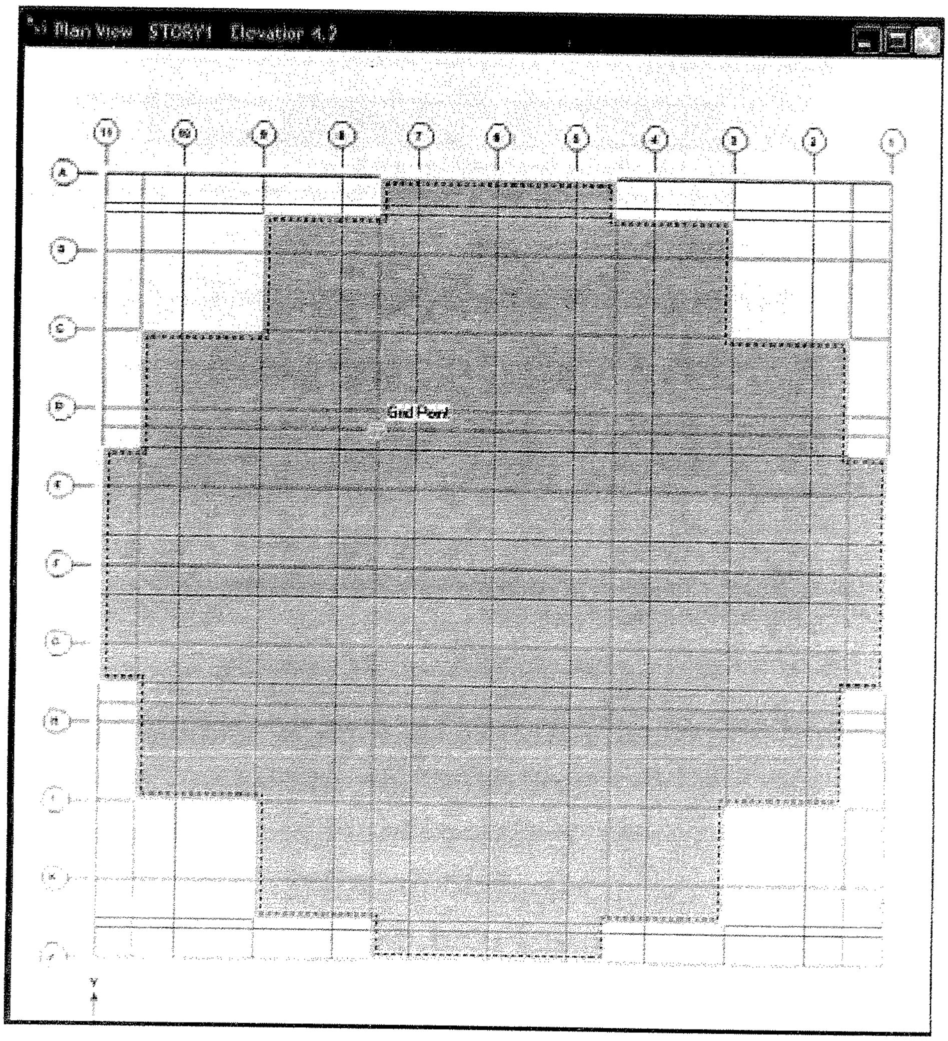


- Thin for the next point Click Drawing Control pull down menu and choose Fixed dx and dy again thin Write the coordinate value for the next point with reference to the previous point

- Thin click any where the program will select the second point directly

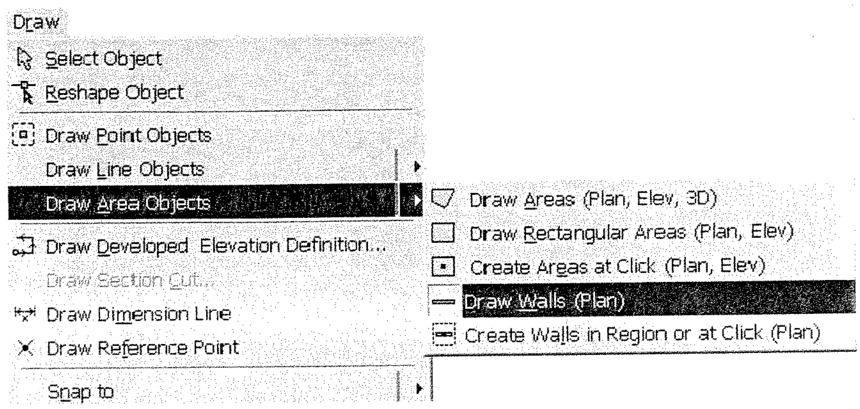


Repeate the previous step for the another points ,thin after you finish Draw the plan it will be as in the figure.

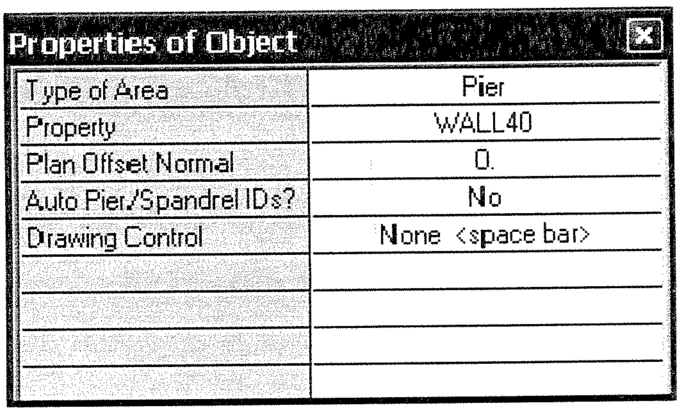


2. *Add Walls*

4. Click the **Draw menu** ⟶ **Draw Area Objects** ⟶ **Draw Walls**



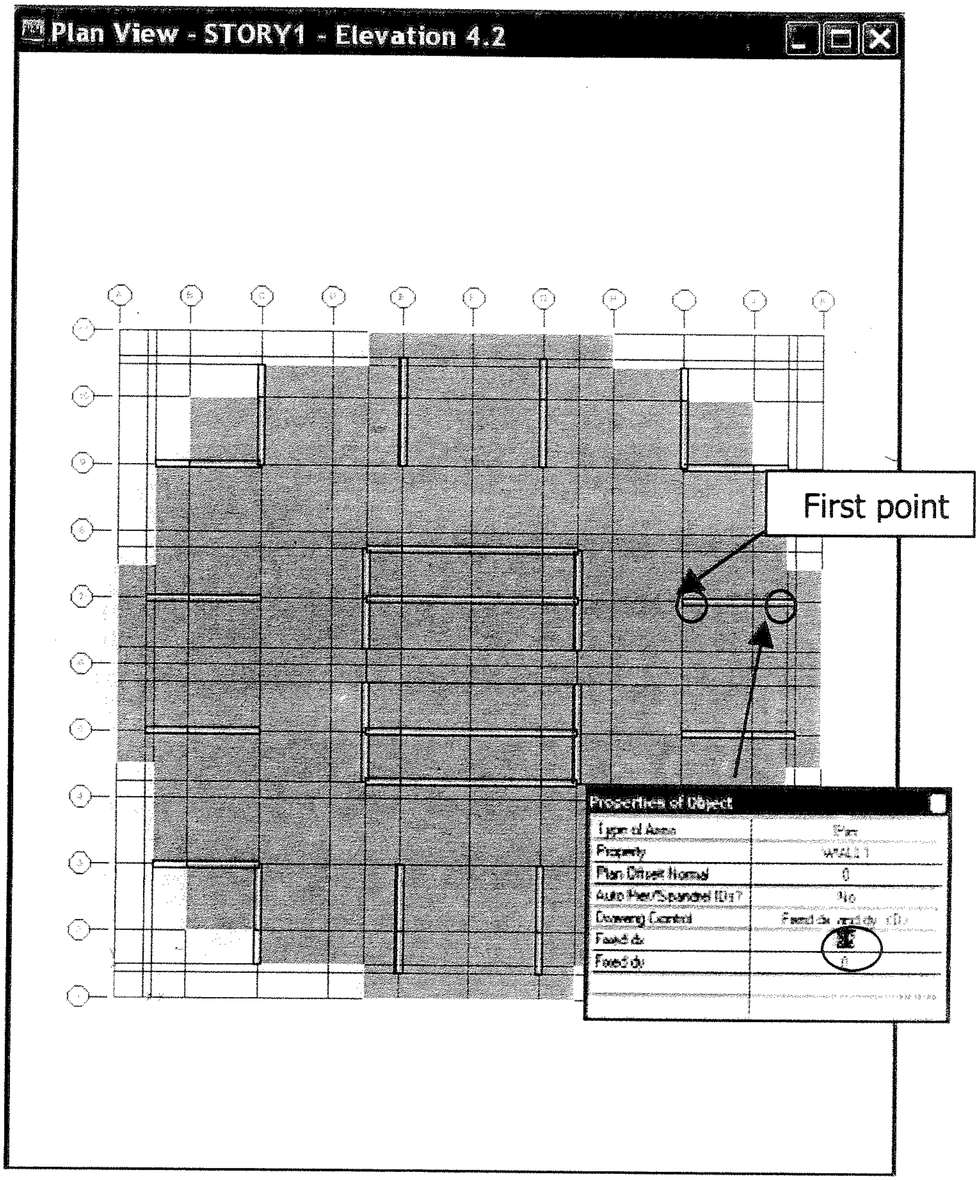
Or Draw Area button which will Display the Properties of Object form as shown in fig. and crasser for drawing Area



- In previous form you can
  1. Choose Type of area (Pier , Spandrel)
  2. Choose the Property of the wall
  3. Make offset while drawing this wall
  4. let the program make label for piers and spandrel automatically
  5. and use Drawing control option as we use it in the slab drawing

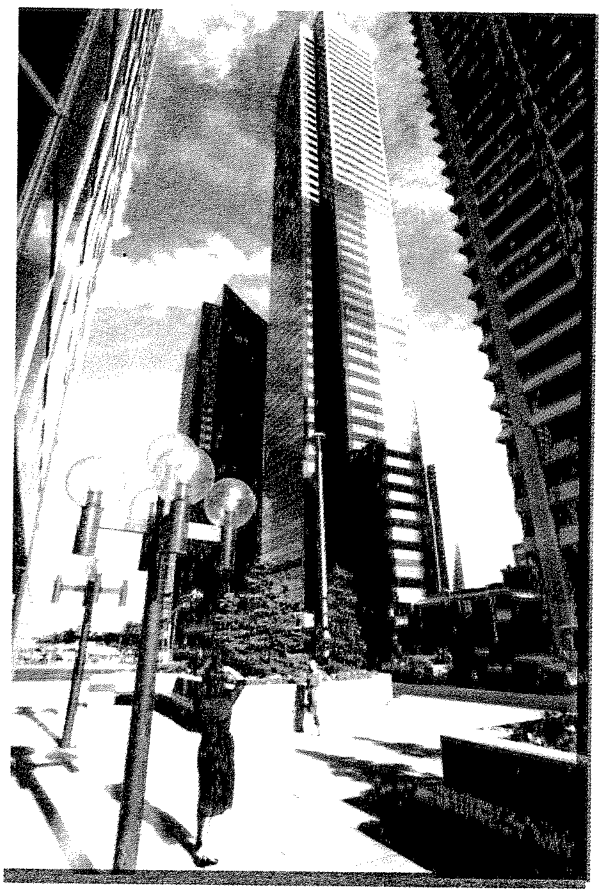
- click for the first point of the wall .thin for the next point Click Drawing Control pull down menu and choose Fixed dx and dy again thin Write the coordinate value for the next point with reference to the first point

After you draw the walls it will be as in the fig



- **ETABS Version 9.0.0 is a new version, and is a direct upgrade from Version 8.5.6. New features include the following.**

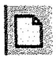
  - Added Semi-rigid diaphragm option
  - Added Design output to Database
  - Added Story vertical load, shear and overturning plots
  - Improved plan display of most design quantities
  - Improved analysis model creation time
  - Enhanced Model Building of Walls with openings
  - Added IBC 2003 seismic and wind loads
  - Added auto-permutation of Wind directions and eccentricities
  - Added Open-structure wind loads
  - Added Export to SAFE V8 with poly areas
  - Updated Concrete Frame Design to ACI 2005
  - Updated Concrete Shear wall Design to ACI 2005
  - Updated Steel design to AISC-ASD 2001 (Seismic Provisions 2002)
  - Updated Steel design to AISC-LRFD 2001 (Seismic Provisions 2002)
  - Updated Steel design to AISC 2005 (Not in initial release)
  - Added Import/Export from Autodesk Rivet Structural
  - Added Import/Export from ProSteel
  - Added Import/Export from IFC
  - Updated CIS/2 Import/Export
  - Added Import from STRUDL
  - Added Import from STAAD



HOW TO MODEL AND DESIGN
HIGH RISE BUILDING USING
ETABS Program

Appendix

Main Menu

 **New Model**

 **Open**

 **Save**

 **Print Graphics**

 **Refresh Window**

 **Lock/Unlock button**

Selection Menu

 **Pointer button**

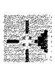
Get Previous Selection

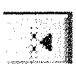
Clear Selection

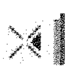
Select All

 **Select using Intersecting Line**

Snap Menu

 **Snap to Grid Intersections and Points**

 **Snap to Line Ends and Midpoint**

 **Snap to Intersections**

Snap to Perpendicular Projections

Snap to Lines and Edges

Snap to Fine Grid

Define menu

Define Material Properties

Define Static Load

Define Load Combinations

Define Frame Sections

Define Wall/Slab/Deck Sections

Define Mass Source

Define Response Spectrum Functions

Define Response Spectrum

Assign menu

Assign Frame Section

Assign joint Rigid Diaphragm

Assign Shell/Area Rigid Diaphragm

Assign Joints Restraints

 **Assign Point Springs**

 **Assign Frame Releases/Partial Fixity.**

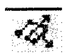
 **Assign End (Length) Offsets**

 **Assign Frame Output Stations**

 **Assign Local Axes**

 **Assign Line Springs**

 **Assign Wall/Slab/Deck Section**

 **Assign Opening**

 **Assign Shell/Area Local Axes**

 **Assign Pier Label**

 **Assign menu Spandrel Label**

 **Assign Additional Area Mass**

 **Assign Joint/Point Loads Force**

 **Assign Joint/Point Loads Temperature**

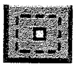
 **Assign Frame/Line Point Loads**

 **Assign Frame/Line Distributed Loads**

 **Assign Frame/Line Temperature Loads**

 **Assign Group Names**

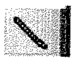
Draw Objects

 Draw Point Objects

Draw Line Objects

 Pointer

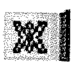
 Reshape Object

 Draw Lines (Plan, Elev, 3D)

 Create Lines at Regions or at Clicks (Plan, Elev, 3D)

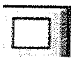
 Create Columns in Regions or at Clicks (Plan)

 Create Secondary Beams in Regions or at Clicks (Plan)

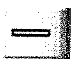
 Create Braces in Regions (Elev)

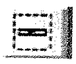
Draw Area Objects

 Draw Areas (Plan, Elev, 3D).

 Draw Rectangular Areas (Plan, Elev)

 Create Areas at Click (Plan, Elev)

 Draw Walls (Plan)

 Create Walls in Regions or at Clicks (Plan)

 Draw Developed Elevation Definition

Edit Menu

Undo

Redo

Delete

Replicate

Edit Grid

Edit Story

Merge Points

Align Points/Lines/ Edges

Move Points/Lines/ Areas

Expand/Shrink Areas

Merge Areas

Mesh Areas

Join Lines

Divide Lines

Extrude Lines to Areas

Analysis Menu

Run Analysis

Lock/Unlock Model

Display menu

Show Unreformed Shape

Show Deformed Shape

Show Mode Shape

Display Member Force Diagram

Show Joint Loads

Show Frame Loads

Display Input Table Mode

Display Output Table Mode

Show the Response Spectrum Curves

View Command

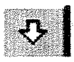
Set Building View Options

3-d 3D view

 **Rotate 3D View**

 **Plan View**

 **Move Up in List**

 **Move Down in List**

 **Elevation View**

 **Perspective Toggle**

Zoom Commands

 **Rubber Band Zoom**

 **Restore Full View**

 **Previous Zoom**

 **Zoom In One Step**

 **Zoom Out One Step**

 **Pan**

Design Commands

 **Steel Frame Design Command**

 **Concrete Frame Design Command**

 **Composite Beam Design Command**

 **Shear Wall Design Command**

Shortcut

Open	Ctrl + O
New Model	Ctrl + N
Save	Ctrl + S
Print Graphics	Ctrl + P
Undo	Ctrl + Z
Redo	Ctrl + Y
Cut	Ctrl + X
Copy	Ctrl + C
Paste	Ctrl + V
Select All	Ctrl + A
Help	F1
Run Analysis	F5
Show/Hide Grid	F7
File Menu	Alt + F
Edit Menu	Alt + E
View Menu	Alt + V
Define Menu	Alt + D
Draw Menu	Alt + R

Select Menu	Alt + S

Assign Menu	Alt + A
Analyze Menu	Alt + N
Design Menu	Alt + P
Display Menu	Alt + G
Options Menu	Alt + O

PREFERENCES

Etabs Manuals (Integrated Building Design Software)

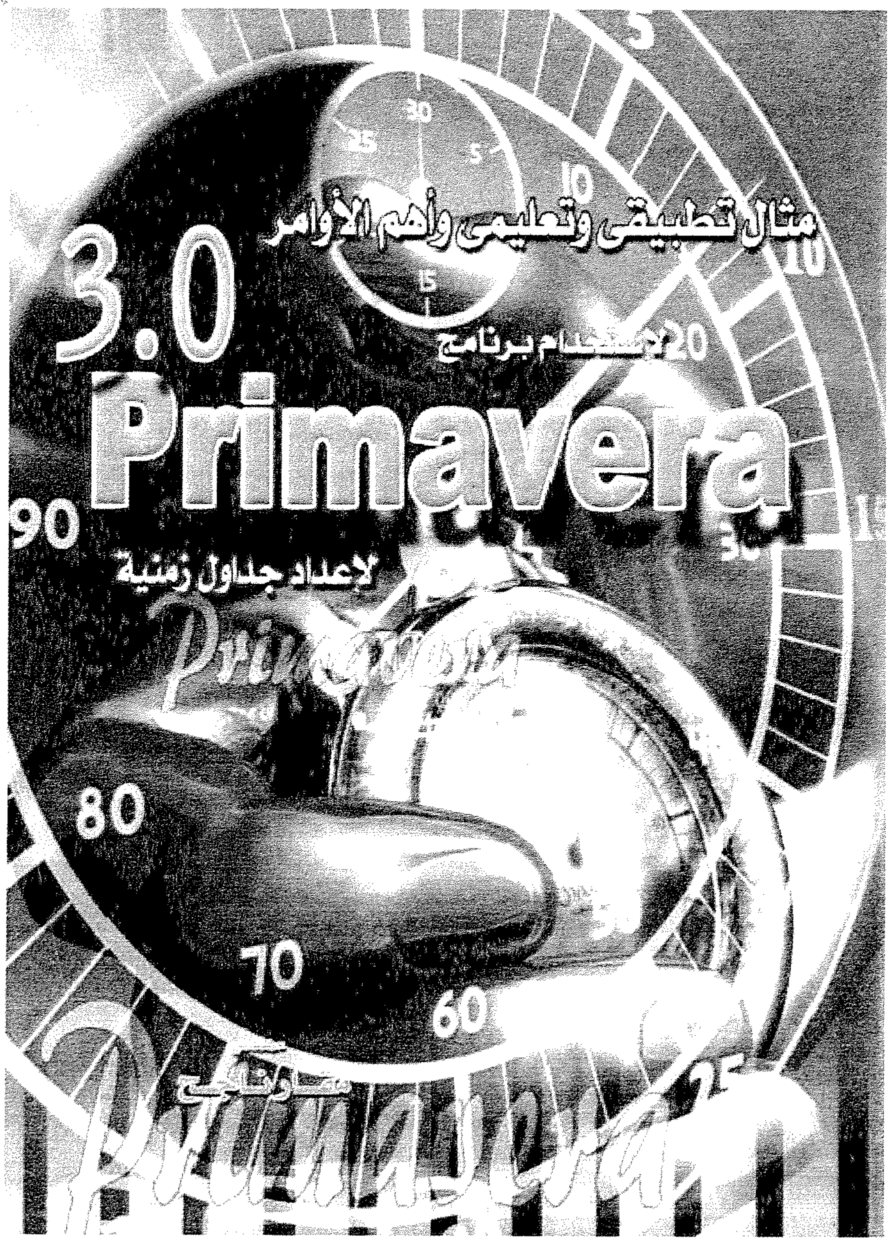
ACI 318-02, ACI318-05"Building code requirements for structural concrete, American Concrete Institute

Uniform building code, 1997

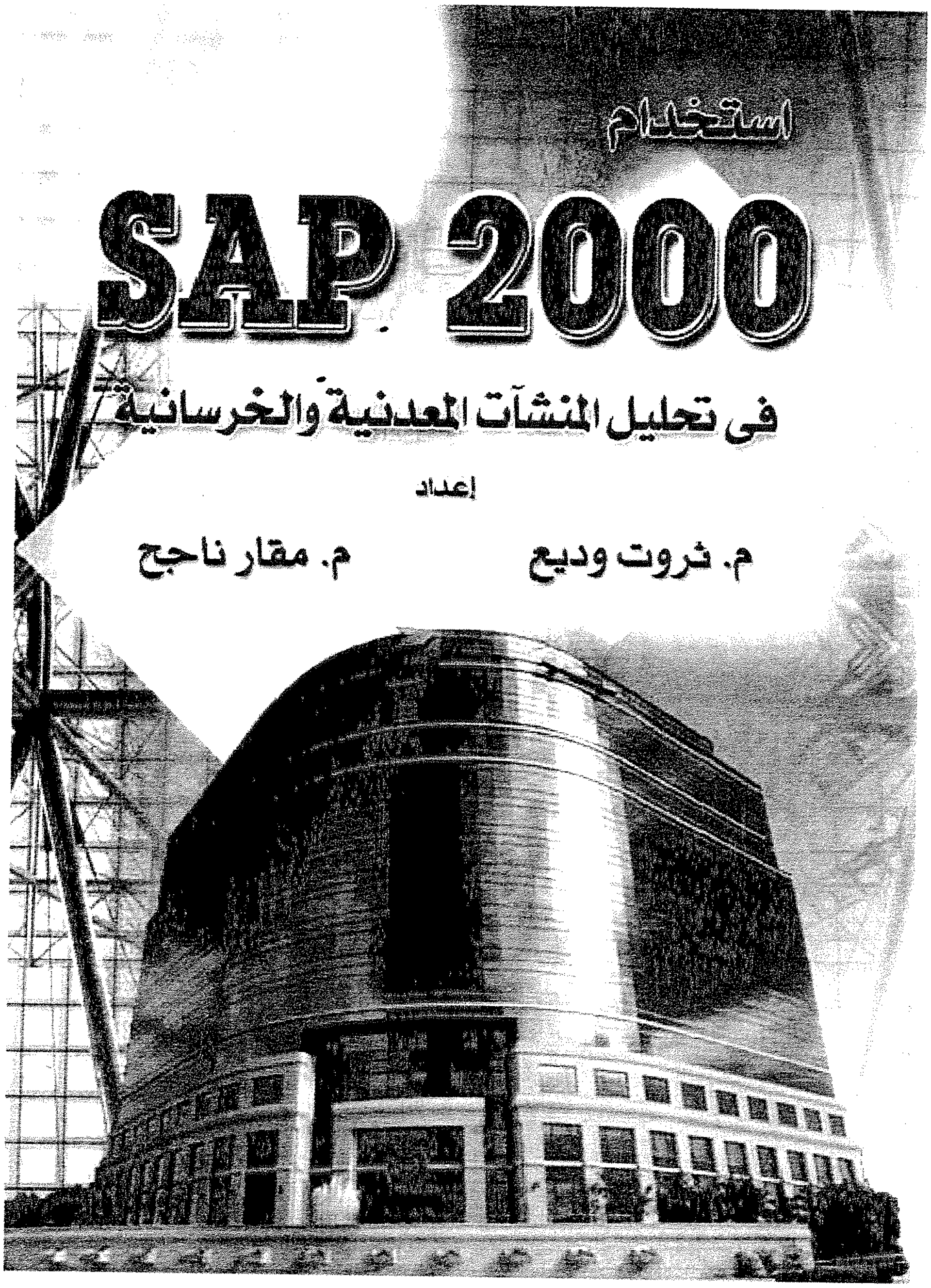
Structural Dynamics (Earthquake Engineering for Practicing Structural Engineers) Aone-Day Course –Instructor: Ashraf Habibullah

Design of concrete structures Arthur H. Nilson, David Darwan, and Chrles W. Dolan

- **صدر للمؤلف**


مثال تطبيقى وتعليمى وأهم الأوامر
3.0
لاستخدام برنامج
Primavera
لإعداد جداول زمنية

- **صدر للمؤلف**


استخدام
SAP 2000
فى تحليل المنشآت المعدنية والخرسانية
إعداد
م. ثروت وديع
م. مقار ناجح

ETABS
Program


جمعية المهندسين
SOCIETY of ENGINEERS - UAE

شهادة شكر وتقدير

بمناسبة مرور 25 عام على تأسيس الجمعية
11 فبراير 2006

تحت رعاية معالي الشيخ
حمدان بن مبارك آل نهيان
وزير الأشغال العامة

تتقدم جمعية المهندسين بدولة الامارات العربية المتحدة
بخالص الشكر والتقدير الى
المهندس / مقار ناجح سوريال
لجهوده المتميزة ودعمه المستمر فى تطوير وتنمية القطاع الهندسي بالدولة

المهندس / مطر محمد الطاير
رئيس جمعية المهندسين

Scientific Book House
For Publishing & Distributing
50, El-Sheikhh Rehan Str,
Aabdeen, Cairo
Tel &Fax 7954229 - 7948619

ISBN 977-287-691-4
9 789772 876914