University of Global Village (UGV), Barishal

ETABS II (Building Super Structure)

Content of Laboratory Course

Program: B.Sc. in CE

Prepared By

Oaisul Mostofa Karim
Lecturer
Department of Civil Engineering
University of Global Village (UGV)

BASIC COURSE INFORMATION

Course Title	ETABS II (Building Super Structure)
Course Code	CE 0732-3102
Credits	01
CIE Marks	30
SEE Marks	20
Exam Hours	2 hours (Semester Final Exam)
Level	5^{th} Semester

ASSESSMENT PATTERN

CIE- Continuous Internal Evaluation (30 Marks)

SEE- Semester End Examination (20 Marks)

SEE- Semester End Examination (40 Marks) (should be converted in actual marks (20))

Bloom's Category	**Tests**
Remember	**05**
Understand	**07**
Apply	**08**
Analyze	**07**
Evaluate	**08**
Create	**05**

CIE- Continuous Internal Evaluation (100 Marks) (should be converted in actual marks (30))

Bloom's Category Marks (out of 100)	Lab Final (30)	Lab Report (10)	**Continuous lab performance** (30)	Presentation & Viva (10)	External Participation in Curricular/**Final Project Exhibition** (10)
Remember/**Imitation**	05		05	02	Attendance 10
Understand/**manipulation**	05	05	05	03	
Apply/**Precision**	05		05		
Analyze/**Articulation**	05		05		
Evaluate/**Naturalisation**	05	05	05		
Create	05		05	05	

Course Title: ETABS II (Building Super Structure)

Covered Course: Structural Analysis and Design-II Sessional

COURSE CODE: CE 0732-3102
CREDIT: 01

CIE MARKS: 30

SEE MARKS: 20

CLO1	**Perform Structural Analysis and Checking: Students will be able to analyze and verify structural models for stability, strength, and safety using advanced ETABS features.**
CLO2	**Conduct Serviceability Checks: Students will evaluate structures for deflection, vibration, and other serviceability criteria, ensuring compliance with performance standards.**
CLO3	**Design Cost-Effective Buildings: Students will develop efficient and economical building designs while meeting the requirements of BNBC-2020 or ASCE-7-05 codes.**
CLO4	**Create Reinforcement Details: Students will gain proficiency in generating accurate reinforcement detailing for structural components, aligning designs with professional standards and guidelines.**

Sl.	Course Contents	Hours	CLOs
1	Analysis and Checking: Checking axial force, torsional, shear force, area of reinforcement for each member, fixing the errors.	10	CLO 1, CLO 2
2	Serviceability Check: Torsion, P-delta, Soft-storey and drift.	10	CLO 3
3	Cost-Effective Design of Building, Reinforcement Detailing of Structure: Detailing of beam, slab, shear wall, lift.	15	CLO 4
4	Details Discussion on BNBC-2020/ASCE-7-05: Understanding the design procedure, guidelines and detailing.	10	CLO 4

References:

Bangladesh National Building Code (BNBC) 1993

Finite Element Analysis (2005) by Dr. S.S. Bhavikatti, New Age International (P) Ltd., Publishers

ETABS Version 9.6.2 User's Guide

WEEK	TOPIC	TEACHING-LEARNING STRATEGY	ASSESSMENT STRATEGY	CORRESPONDING CLO_S
01-02	Introduction to ETABS environment	LECTURE, DISCUSSION	Individual Project Evaluation	CLO1
03-07	Beam under vertical loads	LECTURE, DISCUSSION	Individual Project Evaluation	CLO3
08-11	Bridge Structure	LECTURE, DISCUSSION	Individual Project Evaluation	CLO1
12-15	Multi-storied building frame	LECTURE, DISCUSSION	Individual Project Evaluation	CLO2
16	Doubt Solving	Discussion		CLO2
17	Final Assessment	Lab Quiz, Practical exam	Written, Viva	CLO1

Assessment Strategy

CIE- Continuous Internal Evaluation (60 Marks) (Should be converted in 30 marks)

Bloom's Category Marks (out of 60)	Lab Final (30)	Lab Report (10)	Continuous lab performance (10)	Presentation & Viva (10)	External Participation in Curricular/Co-Curricular Activities (10)
Remember	05			02	Attendance 10
Understand	05	05	02	03	
Apply	05		02		
Analyze	05		02		
Evaluate	05	05	02		
Create	05		02	05	

SEE- Semester End Examination (40 Marks) (Should be converted in 20 marks)

Bloom's Category	Tests
Remember	05
Understand	05
Apply	10
Analyze	05
Evaluate	05
Create	10

WEEK 01-02

Introduction to ETAB's Environment

ETABS: ETABS stands for Extended Three Dimensional Analysis of Building System.

Program work flow: Finite element software generally follows three steps as given below-

1. Preprocessing: Object based model generation
   - Define materials
   - Define geometry
   - Define elements
   - Draw (line, area etc)
   - Mesh (convert object based model to element based model)
   - Load application
2. Processing: Analysis / Solution ▪ Static Analysis ▪ Dynamic Analysis
3. Post processing:
   - Result interpretation (SFD, BMD, Displacement, Stress etc)
   - Design

ETABS's key features:

1. Main Title Bar
2. Menu Bar
3. Toolbar
4. Display Title Bar

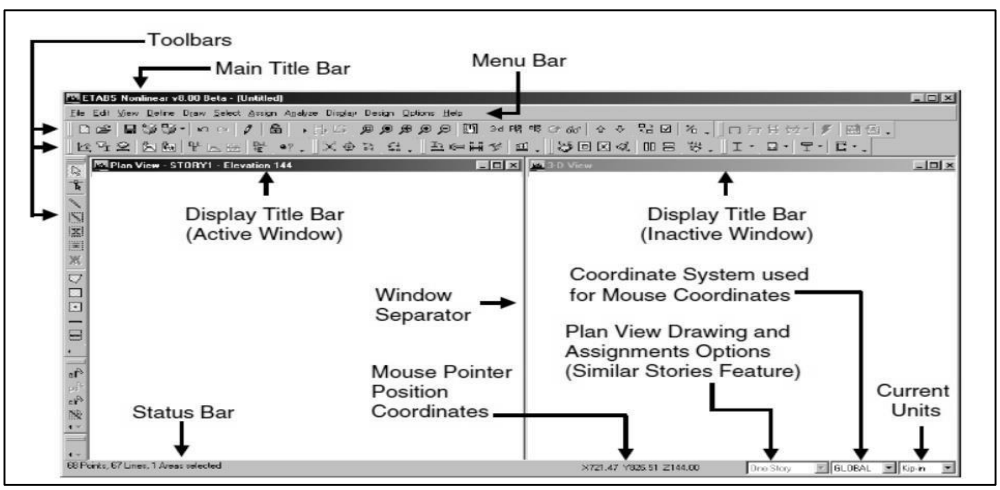


Fig 1.1: Main window of ETABS

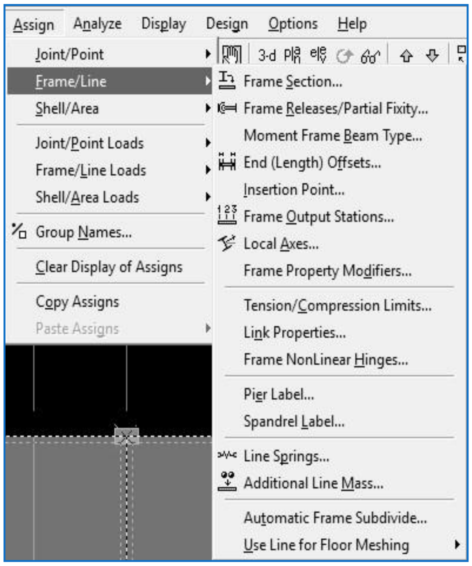


Fig 1.2: AssignToolbar

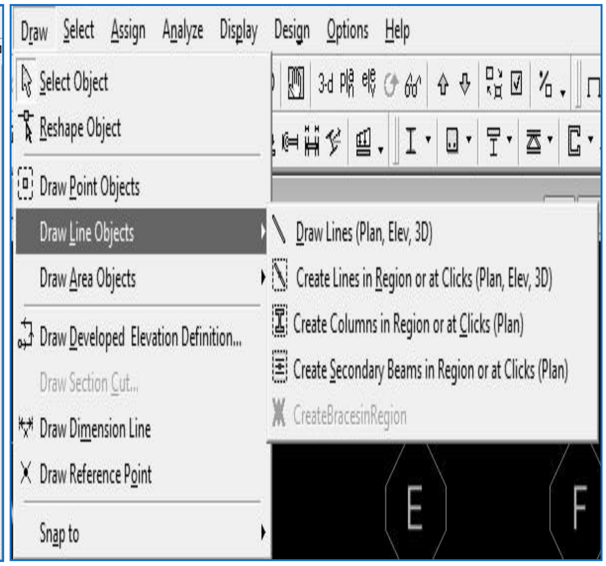


Fig 1.3: Draw Toolbar

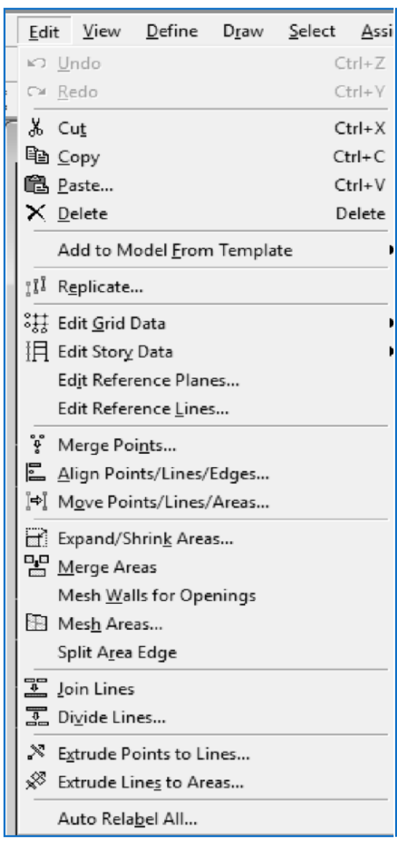


Fig 1.4: Edit Toolbar

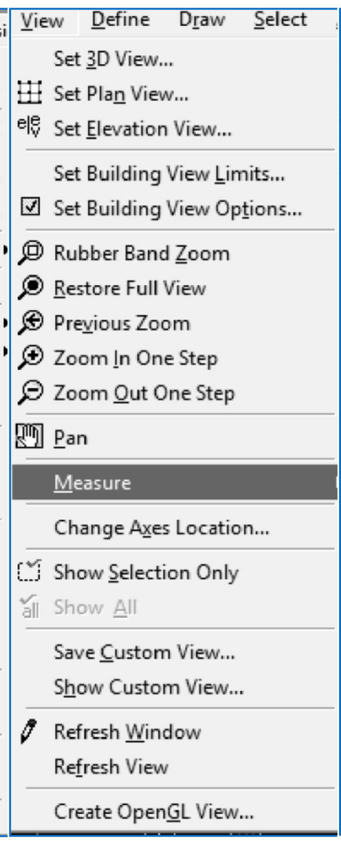


Fig 1.5: View Toolbar

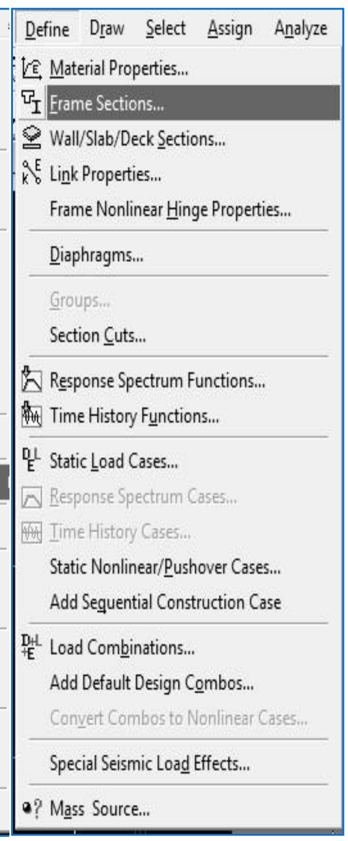


Fig 1.6: Define Toolbar

Work Procedure:

Step-1: Open the ETABS software

Step-2: File→ New Model (Make model Grid as given data)

Step-3: Define

a. Materials Properties
b. Frame Sections
c. Wall/ Slab/Deck Sections
d. Static Load Cases
e. Load Combinations, etc

STEP-4: Draw →Column, Shear wall, Beam, Slab etc.

STEP-5: Assign →Support, Loads (Dead load, Live load etc.)

STEP-6: Analyze →Set Analysis option, Check Model, Run Analysis

UNIVERSITY OF GLOBAL VILLAGE

WEEK 03-04

Beam under vertical loads

Objective: Analyze the following beam under vertical loads and find out the following values;

(Fig: 2.1)

1. Supports Reactions
2. Maximum Shear force on beam AB
3. Maximum Bending moment on beam CD

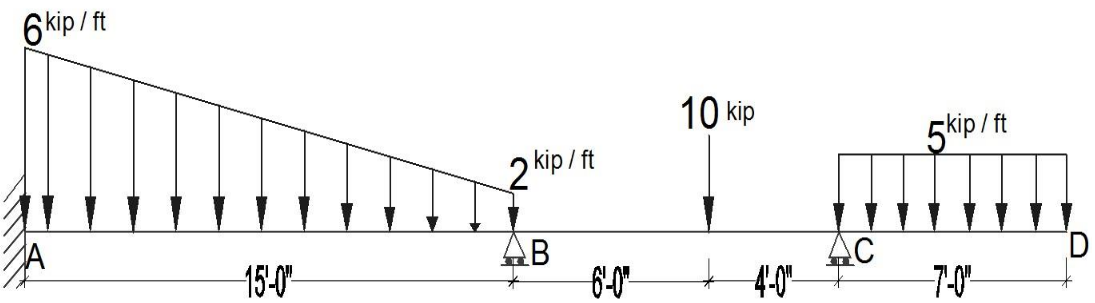


Fig: 2.1

Properties:

Materials= Concrete, f'c = 4000psi

Section Size = 10" X 15"

Procedure:

1. Grid System and Story data definition:

   1.1. Open ETABS software →File → New Model → no →Units = Kip-ft →Number of lines in Xdirection = 4 → Number of lines in Y-direction = 1 → Number of stories = 1 → Click on Custom Grid spacing → Edit Grid → Click on Spacing →Then X-direction Grid spacing A = 15, B=10, C=7, D=0 → OK→Grid only →OK. (Fig: 2.2)

2. Define Material properties & Frame section:

   2.1. Define → Materials Properties → Add New Materials →Material Name = CONC4 → Specified Conc. Comp. Strength = 4 → Modulus of Elasticity = 3600 →OK →OK. (Fig: 2.3)

   2.2. Define → Frame Sections → Add Rectangular →Section Name = B10X15 → Material = CONC4→ Depth=15, Width = 10 → Reinforcement→Beam→OK →OK→ OK. (Fig: 2.4)

3. Draw:

   From menu bar click on Draw → Draw Line Objects →Click on Draw Lines (Plan, Elev, 3D)→Select Property = B10X15 →Then draw the line on elevation view by clicking one point to another point. After finishing press Esc from key board. (Fig: 2.5)

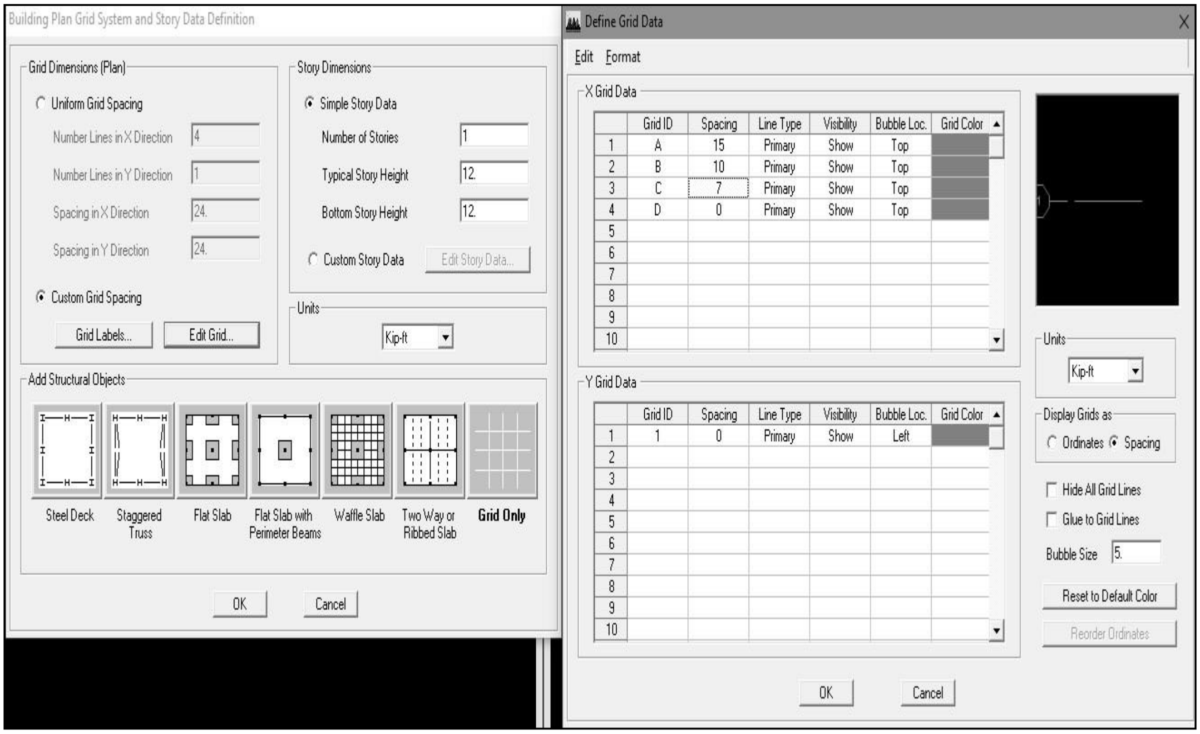

Building Plan Grid System and Story Data Definition
Grid Dimensions (Plan)
Uniform Grid Spacing
Number Lines in X Direction 4
Number Lines in Y Direction 1
Spacing in X Direction 24.
Spacing in Y Direction 24.
Custom Grid Spacing
Grid Labels...
Edit Grid...
Story Dimensions
Simple Story Data
Number of Stories 1
Typical Story Height 12.
Bottom Story Height 12.
Custom Story Data
Edit Story Data...
Units
Kip-ft
Add Structural Objects
Steel Deck
Staggered Truss
Flat Slab
Flat Slab with Perimeter Beams
Waffle Slab
Two Way or Ribbed Slab
Grid Only
OK
Cancel
Define Grid Data
Edit
Format
X Grid Data
Grid ID
Spacing
Line Type
Visibility
Bubble Loc.
Grid Color
1 A 15 Primary Show Top
2 B 10 Primary Show Top
3 C 7 Primary Show Top
4 D 0 Primary Show Top
Y Grid Data
1 1 0 Primary Show Left
Units
Kip-ft
Display Grids as
Ordinates
Spacing
Hide All Grid Lines
Glue to Grid Lines
Bubble Size 5.
Reset to Default Color
Reorder Ordinates
OK
Cancel

Fig : 2.2

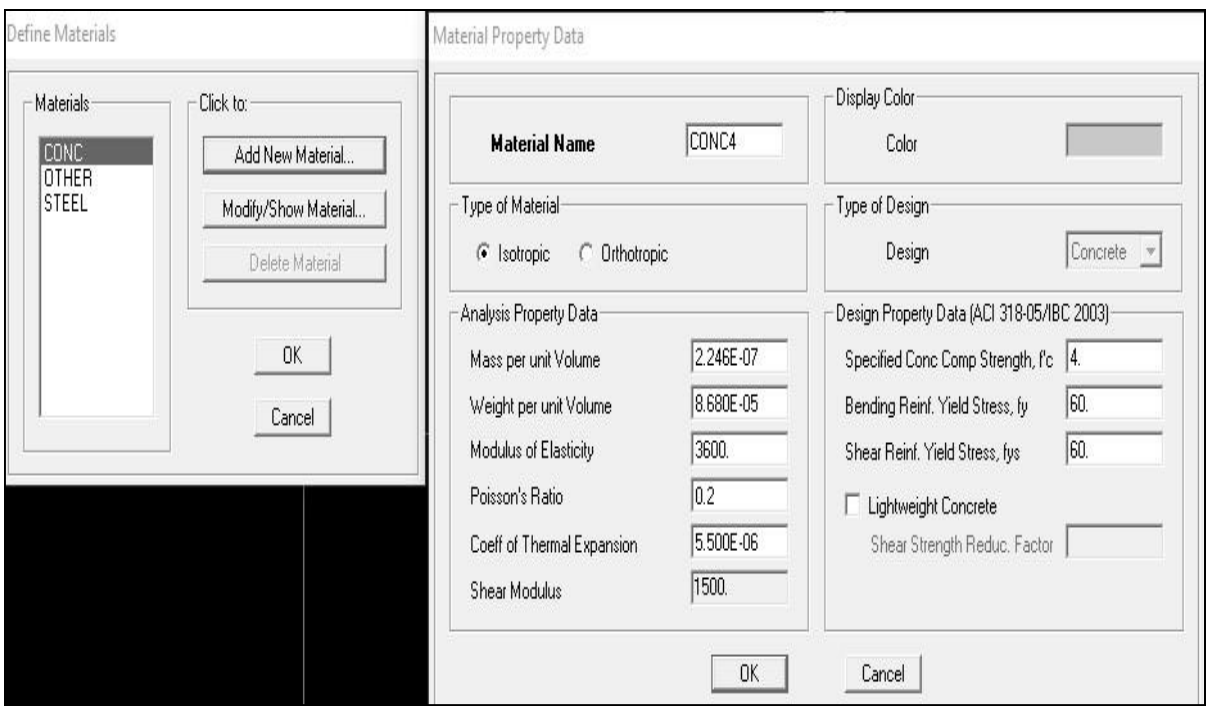

Define Materials
Materials
CONC
OTHER
STEEL
Click to:
Add New Material...
Modify/Show Material...
Delete Material
OK
Cancel
Material Property Data
Material Name CONC4
Display Color
Color
Type of Material
Isotropic
Orthotropic
Type of Design
Design Concrete
Analysis Property Data
Mass per unit Volume 2.246E-07
Weight per unit Volume 8.680E-05
Modulus of Elasticity 3600.
Poisson's Ratio 0.2
Coeff of Thermal Expansion 5.500E-06
Shear Modulus 1500.
Design Property Data (ACI 318-05/IBC 2003)
Specified Conc Comp Strength, f'c 4.
Bending Reinf. Yield Stress, fy 60.
Shear Reinf. Yield Stress, fys 60.
Lightweight Concrete
Shear Strength Reduc. Factor
OK
Cancel

Fig: 2.3

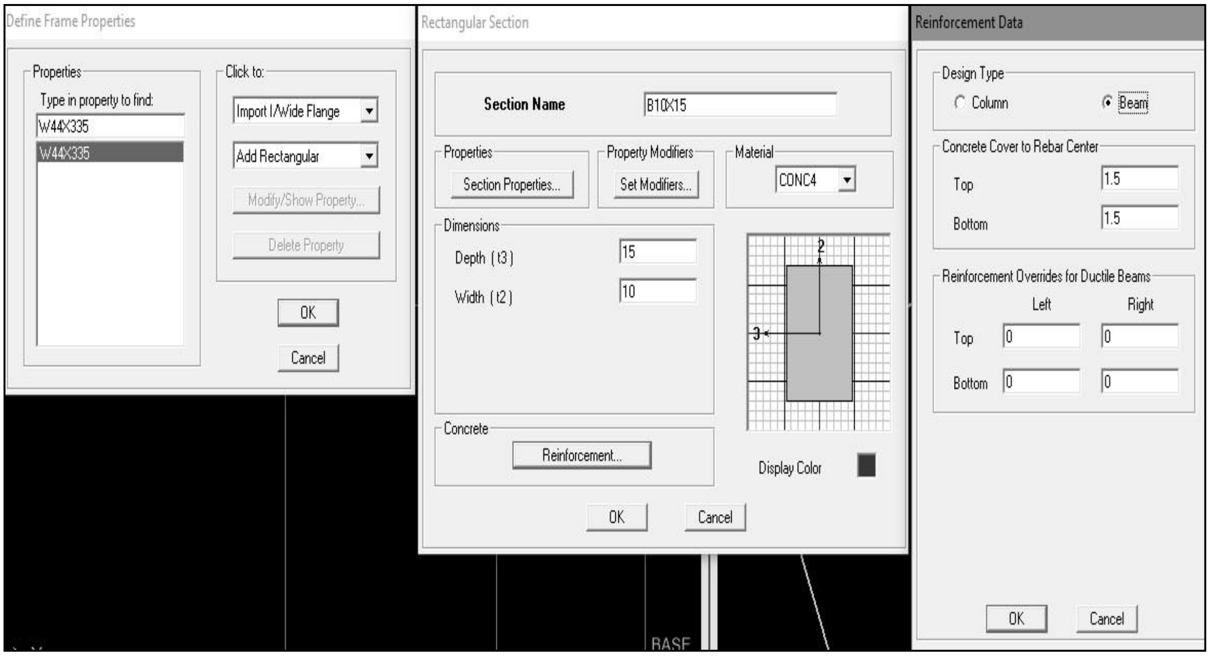

Define Frame Properties
Properties
Type in property to find:
W44X335
W44X335
Click to:
Import I/Wide Flange
Add Rectangular
Modify/Show Property...
Delete Property
OK
Cancel
Rectangular Section
Section Name
B10X15
Properties
Section Properties...
Property Modifiers
Set Modifiers...
Material
CONC4
Dimensions
Depth (t3)
15
Width (t2)
10
Concrete
Reinforcement...
Display Color
OK
Cancel
Reinforcement Data
Design Type
Column
Beam
Concrete Cover to Rebar Center
Top
1.5
Bottom
1.5
Reinforcement Overrides for Ductile Beams
Left
Right
Top
0
0
Bottom
0
0
OK
Cancel

Fig : 2.4

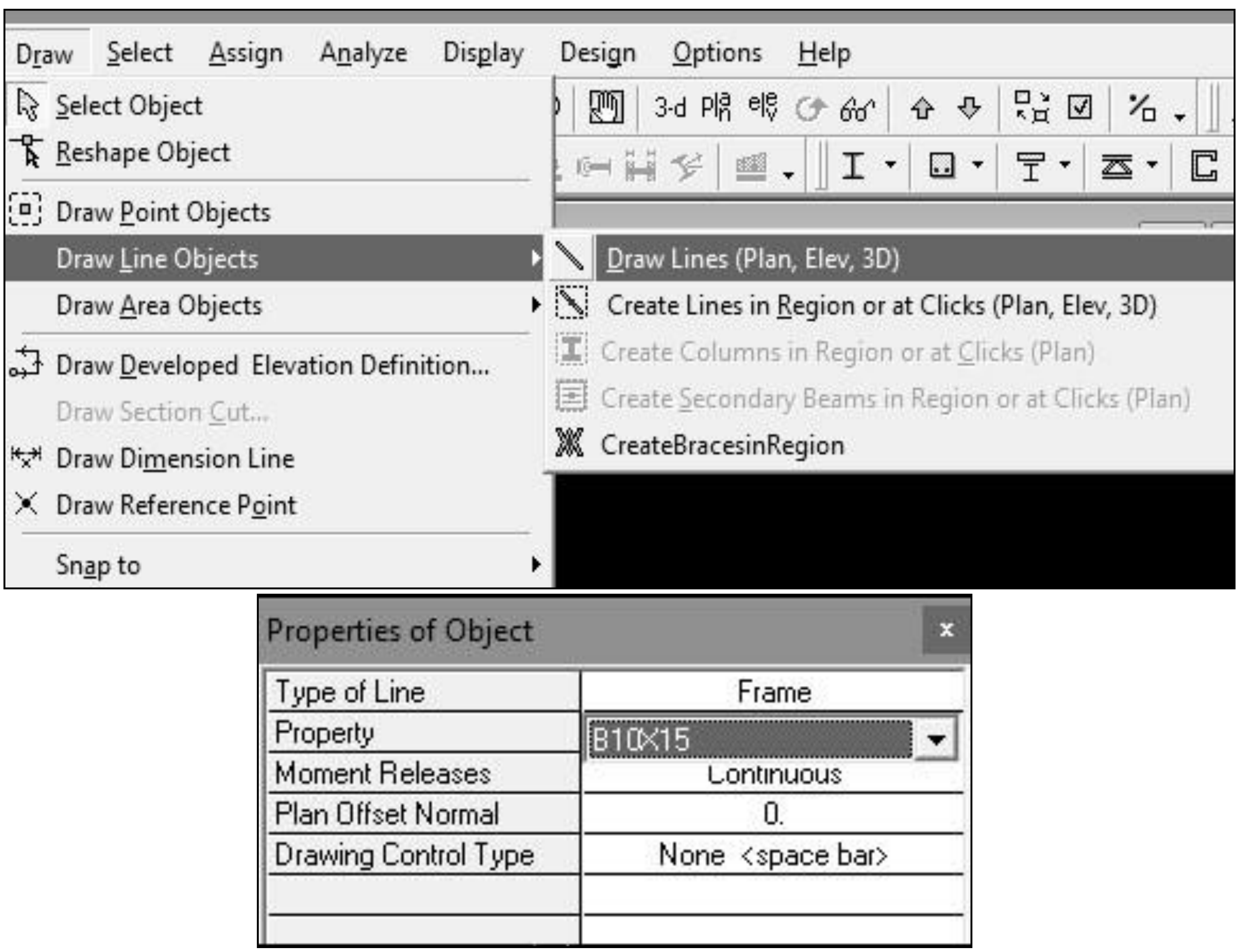

Draw
Select
Assign
Analyze
Display
Design
Options
Help
Select Object
Reshape Object
Draw Point Objects
Draw Line Objects
Draw Area Objects
Draw Developed Elevation Definition...
Draw Section Cut...
Draw Dimension Line
Draw Reference Point
Snap to
Draw Lines (Plan, Elev, 3D)
Create Lines in Region or at Clicks (Plan, Elev, 3D)
Create Columns in Region or at Clicks (Plan)
Create Secondary Beams in Region or at Clicks (Plan)
CreateBracesinRegion
Properties of Object
Type of Line
Frame
Property
B10X15
Moment Releases
Continuous
Plan Offset Normal
0.
Drawing Control Type
None <space bar>

Fig: 2.5

4. Assign:

4.1. Support Assign: Select the support point → Assign →Joint/Point → Restraints (Supports) → then select support type (Fixed, Pin, Roller) by clicking on symbols → OK. (Fig: 2.6)

4.2. Load Assign: Select the Beam → Assign →Frame/Line Loads →Distributed /Point →Load Case Name = Live →Units=Kip-ft →Direction = Gravity →Click on Absolute distance from End-I →Then write the values Distance=0, Load =6 and Distance=15, Load =2 → OK. (Fig: 2.7 to 2.8) Follow the same procedure for other Distributed load and Point load.

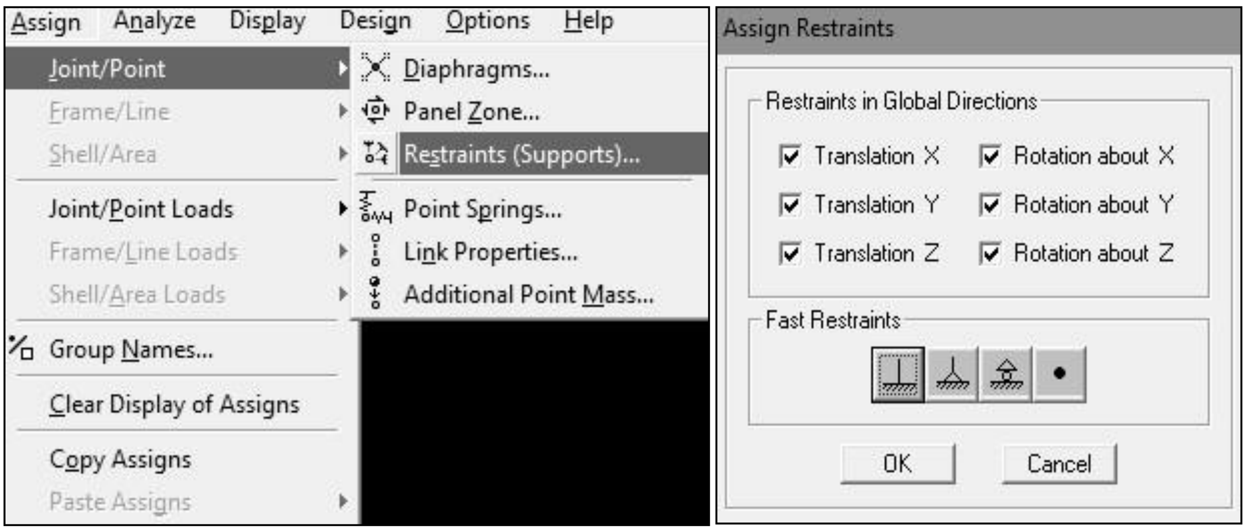


Fig : 2.6

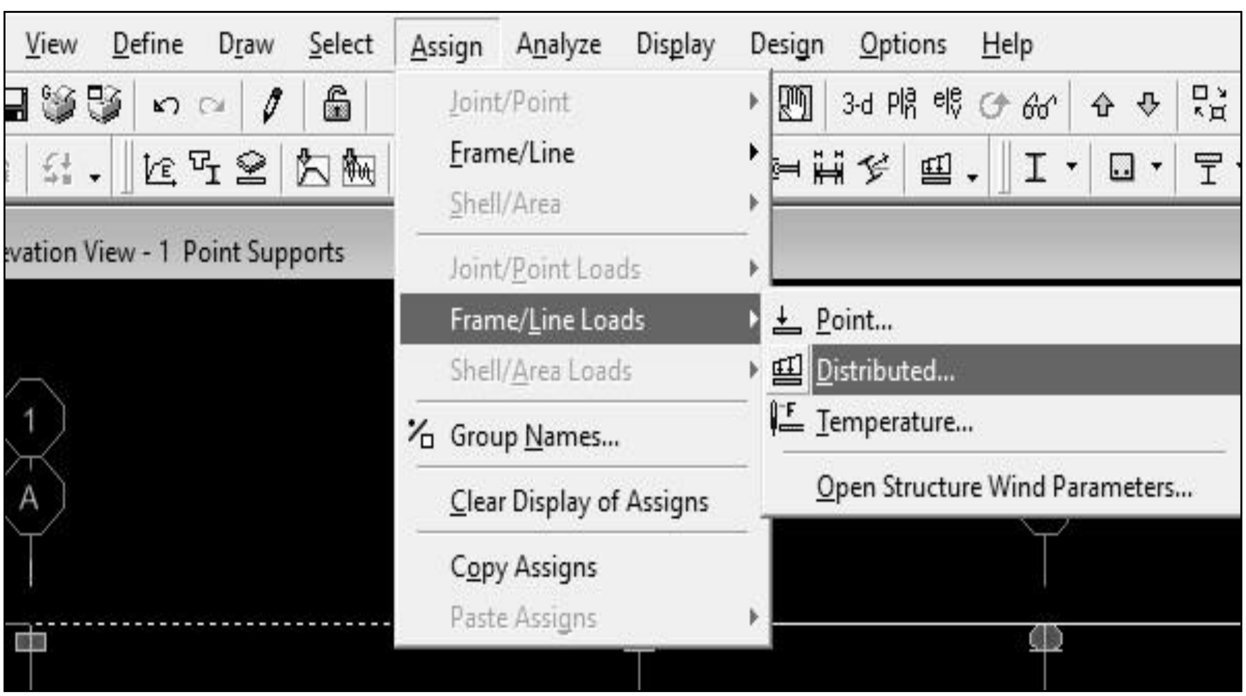


Fig : 2.7

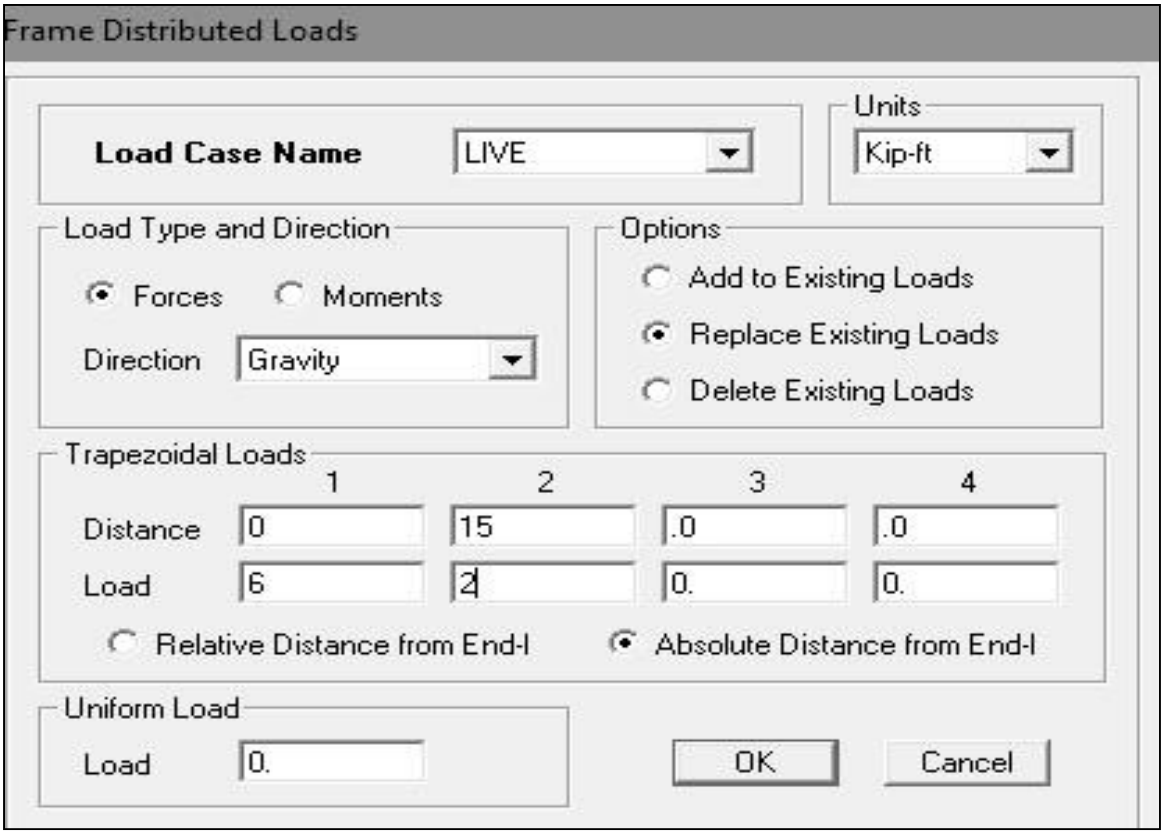


Fig : 2.8

5. Analyze:

Analyze → Model check → OK

Analyze → Run Analysis

6. Results:

6.1. For Support Reactions: Go to display →Show Member Forces/Stress Diagram →Support/Spring Reactions→Select Load (Live/Dead etc.)→ OK→Select the Support point by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 2.9)

6.2. Maximum Shear force on beam AB: Go to display →Show Member Forces/Stress Diagram →Frame/Pier/Spandrel Forces…→Select Load (Live/Dead etc.)→Shear 2-2 →Click on Show Values on Diagram→ OK →Select the Beam by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 2.10 and 2.11)

6.3. Maximum Bending moment on beam CD: Go to display →Show Member Forces/Stress Diagram →Frame/Pier/Spandrel Forces…→Select Load (Live/Dead etc.)→Moment 3-3 →Click on Show Values on Diagram→ OK →Select the Beam by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 2.12)

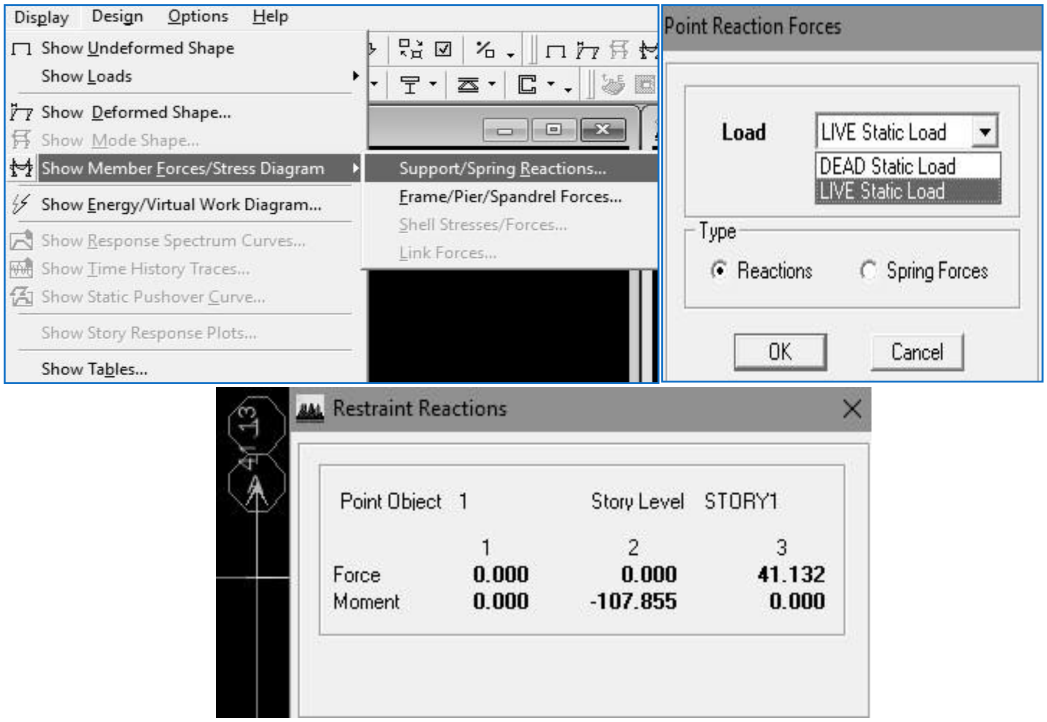


Fig: 2.9

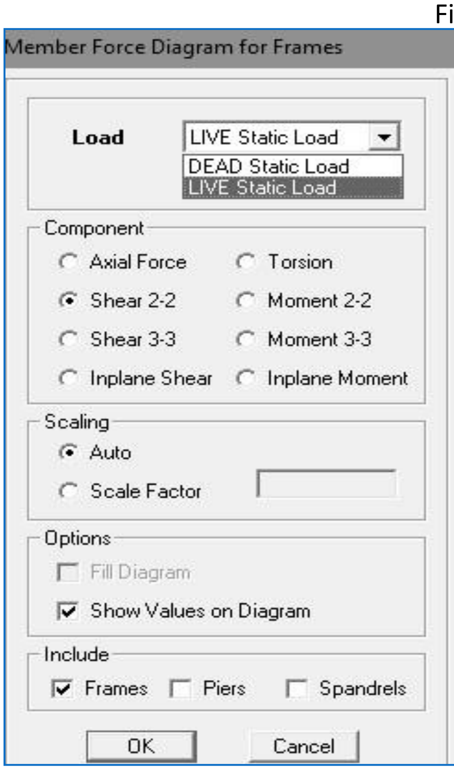


Fig: 2.10

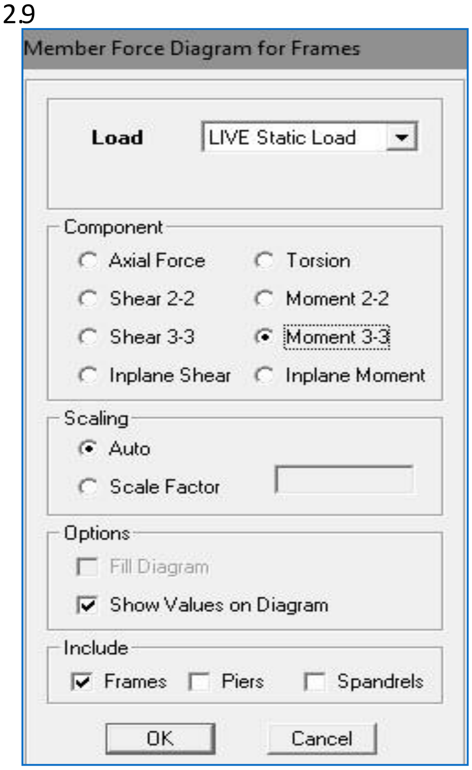


Fig: 2.11

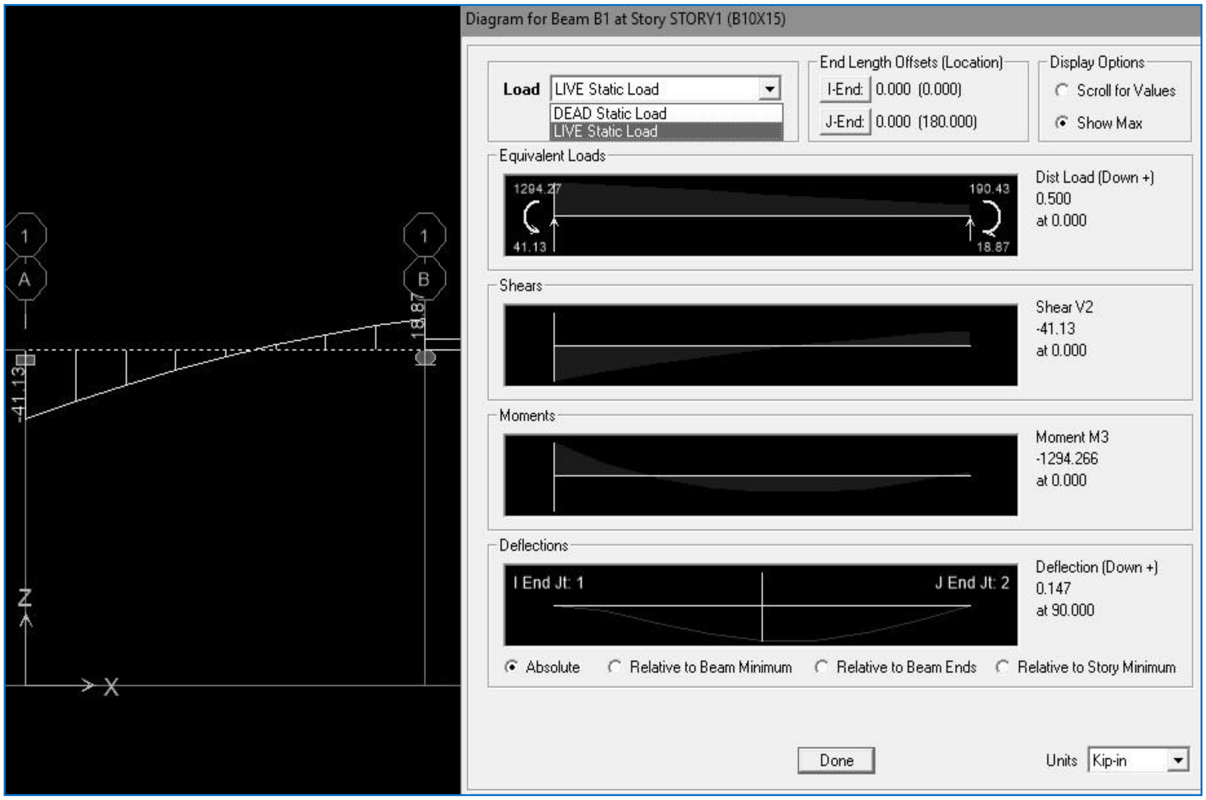


Diagram for Beam B1 at Story STORY1 (B10X15)
Load
LIVE Static Load
DEAD Static Load
LIVE Static Load
End Length Offsets (Location)
I-End: 0.000 (0.000)
J-End: 0.000 (180.000)
Display Options
Scroll for Values
Show Max
Equivalent Loads
1294.27
190.43
41.13
18.87
Dist Load (Down +)
0.500
at 0.000
Shears
Shear V2
-41.13
at 0.000
Moments
Moment M3
-1294.266
at 0.000
Deflections
I End Jt: 1
J End Jt: 2
Deflection (Down +)
0.147
at 90.000
Absolute
Relative to Beam Minimum
Relative to Beam Ends
Relative to Story Minimum
Done
Units
Kip-in
1
A
1
B
-41.13
18.87
Z
X

Fig : 2.12

UNIVERSITY OF GLOBAL VILLAGE

WEEK 05

Beam under vertical loads

Assignment

1) Analyze the following beam under vertical loads and find out the following values; (Fig: 2.13) a) Supports Reactions
   b) Maximum Shear force on beam BC
   c) Maximum Bending moment on beam AB

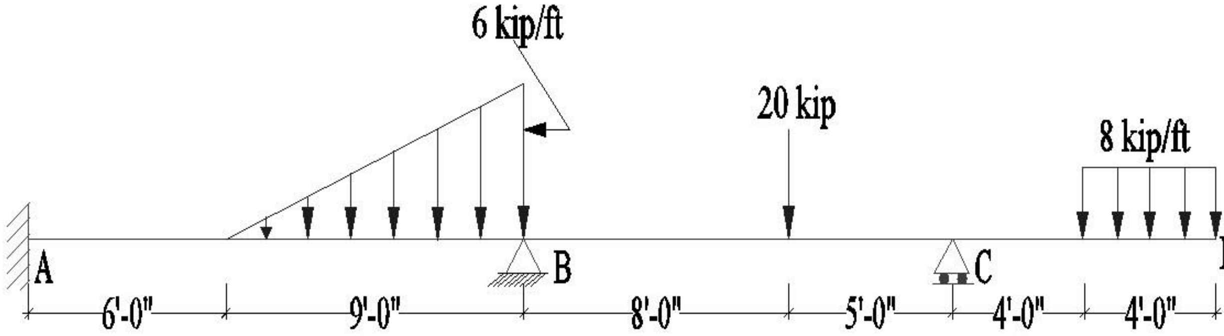


Fig: 2.13

Properties of the Beam:

Materials= Concrete, f'c = 3500 psi

Section Size = 12" X 18"

WEEK 05

Beam under vertical loads

Assignment

2) Analyze the following beam under vertical loads and find out the following values; (Fig: 2.13) a) Supports Reactions
   b) Maximum Shear force on beam CD
   c) Maximum Bending moment on beam AB

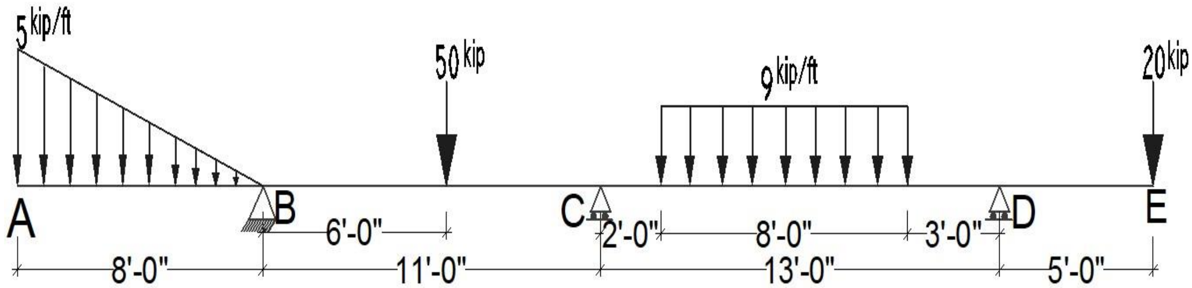


Fig 2.13:

Properties of the Beam:

Materials= Concrete, f'c = 4000 psi

Section Size = 10" X 20"

UNIVERSITY OF GLOBAL VILLAGE

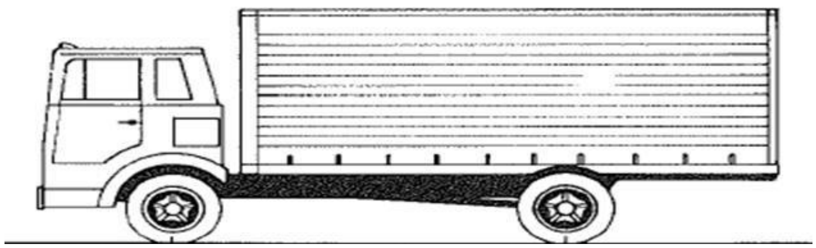
WEEK 06

BRIDGE STRUCTURE

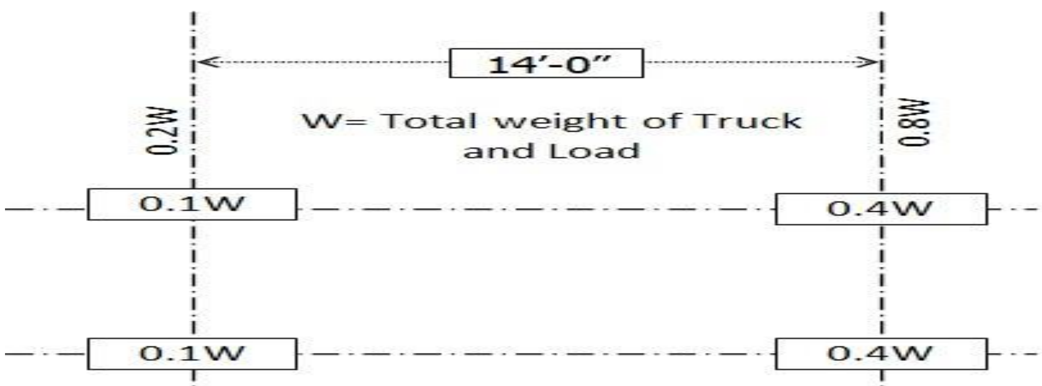
Load distribution on bridge slab according to AASHTO code

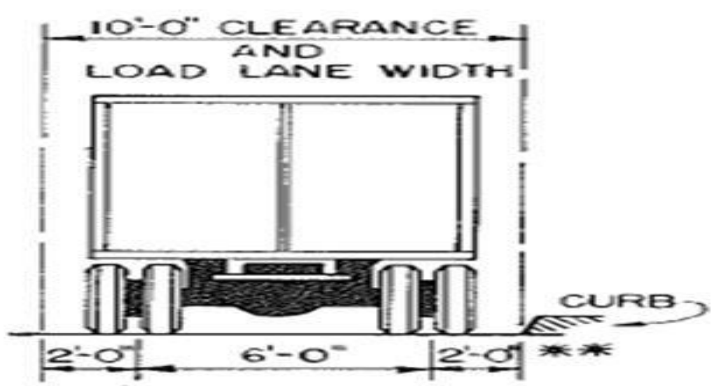
AASHTO (American Association of state Highway and Transportation officials). The highway live loadings on the roadways of bridges or incidental structures shall consist of standard trucks or of lane loads which are equivalent to truck trains. Two systems of loading are provided, the H loadings and the HS loadings, the corresponding HS loadings being heavier than the H loadings.

H Loadings: The H loadings are illustrated in Figures 1.2.5A and l.2.5B. They consist of a two-axle truck or the corresponding lane loading. (Fig: 3.1)



H 20-44	8000 LBS	32000 LBS
H 15-44	6000 LBS	24000 LBS
H 10-44	4000 LBS	16000 LBS





HS Loadings:

The HS loadings are illustrated in Figures 1.2.5B and 1.2.5C. They consist of a tractor truck with semitrailer or of the corresponding lane loading. The HS loadings are designated by the letters HS followed by a number indicating the gross weight in tons of the tractor truck. The variable axle spacing has been in traduced in order that the spacing of axles may approximate more closely the tractor trailers now in use. The variable spacing also provides a more satisfactory loading for continuous spans, in that heavy axle loads may be so placed on adjoining spans as to produce maximum negative moment. (Fig: 3.2)

Loading of either H-20 or HS-20 is based on an axle load of 32 kips. This load is divided into two tires; that is a load at each end of the axle. The tire area contact is defined in AASHTO 3.30. The tire contact area for HS 20-44 shall be assumed as a rectangle with a length in the direction of traffic is 10 inches and a width of tire is 20 inches. The two tires on the axle are spaced six feet apart (center-to-center) transverse to the direction of traffic and the successive axles of an HS truck are 14 feet apart along the direction of traffic.

Surface Pressure Calculation:

Rear axle Load = 32000 Ibs (Two tires)
Load per tire = (32000/2) =16000 Ibs
Contact area for each tire = (20” x 10”) = 200 square inches
Now Surface pressure = (16000/200) = 80 psi (552 KPa) static

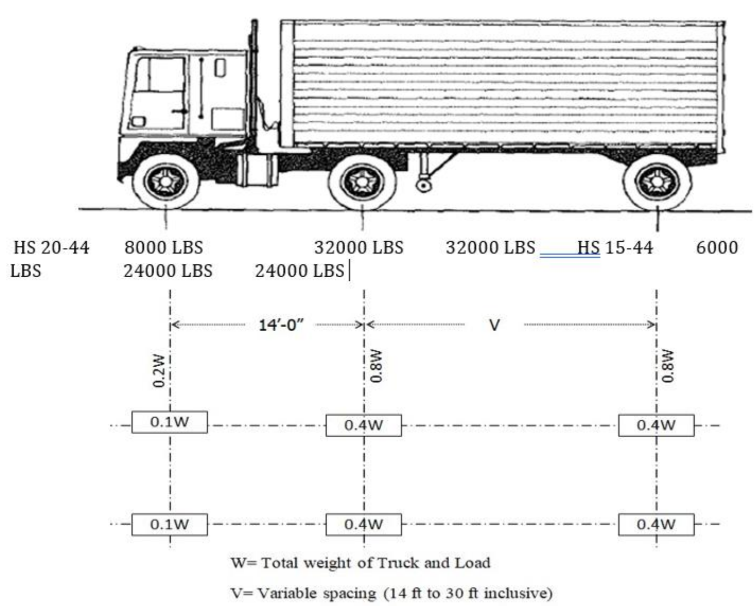


Fig: 3.2

WEEK 07

BRIDGE STRUCTURE

Concrete Bridge Portal Analysis

Objective: Analyze the following Bridge portal frame and slab under HS Loadings and find out the following values; (Fig: 3.3)

1. Support reactions
2. Maximum Stress in Slab
3. Maximum Bending moment in Slab

Properties:

Materials= Concrete, f'c =4000 psi

Bridge pier/Column = 18" diameter

Bridge Girder/Beam = 12" X 24"

Slab Thickness = 12"

Loading:

Dead Load (DL) = Self weight all of given properties.

Live Load (LL) = 80 psi for H/HS Truck.

Load combination = DL+LL

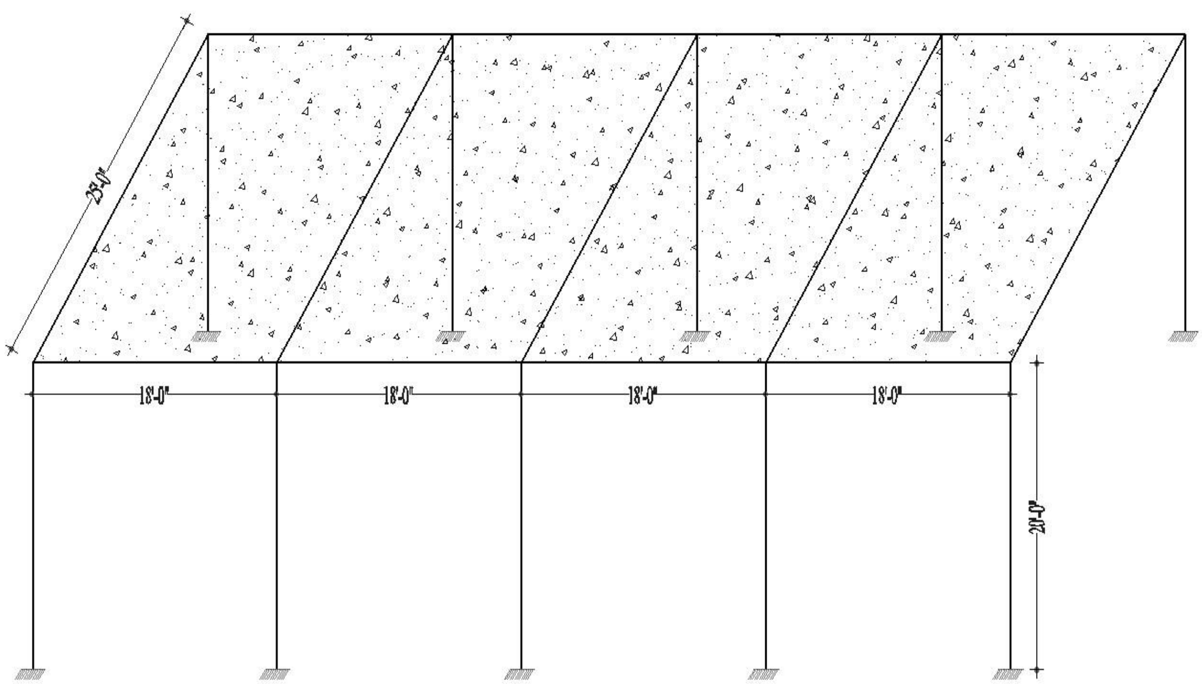


Fig: 3.3

Procedure:

1. Grid System and Story data definition:

Open ETABS software →File → New Model → no →Units = Kip-ft →Number of lines in Xdirection = 5 → Number of lines in Y-direction = 2 → Number of stories = 1 → Spacing in Xdirection =18ft → Spacing in Y-direction = 25ft → Bottom Story Height = 20ft →Grid only →OK. (Fig: 3.4)

2. Define Material properties & Frame section:

2.1. Materials Properties: Define → Materials Properties → Add New Materials →Material Name = CONC4 → Specified Conc. Comp. Strength = 4 → Modulus of Elasticity = 3600 →OK →OK. (Fig: 3.5)

2.2. Sections for Beam: Define → Frame Sections (for Beam, Column)→ Add Rectangular →Section Name: Beam→ Material=CONC4→ Depth=24, Width= 12 → Reinforcement→Beam →OK → OK

For column/Pier:

Define → Frame Sections (for Beam, Column)→ Add Circle →Section Name: Pier→ Material = CONC4→ Diameter = 18"→ Reinforcement→Column →OK → OK

2.3. Wall/Slab/Deck sections: Define → Wall/Slab/Deck sections→SLAB1→Modify/Show Section→Section Name = SLAB12→ Material = CONC4→Thickness: Membrane=12, Bending=12→Type: Shell→Set Modifiers→Bending m11 Modifier=Bending m11 Modifier=Bending m22 Modifier=Bending m12 Modifier= 0.00001→ OK →OK → OK

2.4. Load Combinations: Define → Load Combinations→Add New Combo.. →Load Combination Name=DL+LL→Load Combination Type =ADD→ Case Name=SFW static load, Scale Factor=1→Add→Case Name=LIVE static load, Scale Factor=1→Add→ →OK

3. Draw:

3.1. Beam Draw: From menu bar click on Draw → Draw Line Objects →Click on Draw Lines (Plan, Elev, 3D)→Select Property = Beam or Pier →Then draw the line on Plan view by clicking one point to another point. After finishing press Esc from key board. (Fig: 3.6)

3.2. SLAB Draw**:** Plan view is →Plan ViewStory1or any other without BASE plan and from bottom select →Similar Story then from menu bar click on Draw → Draw Area Objects → Click on Draw Areas (Plan, Elev, 3D) → Select Property = SLAB6→Then draw the Slab by clicking one point to another point at anti clockwise rotations your given Slab Layout Plan.

★To display the slab on screen go to View→Set Building View Options→Click on Object fill→Apply to all Windows→OK. After finishing press Esc from key board

4. Assign:

4.1. Support Assign: Select the support point → Assign →Joint/Point → Restraints (Supports) →Then select support type (Fixed, Pin, Roller) by clicking on symbols → OK. (Fig: 3.7)

4.2. Load Assign: Select Slabs→ Assign →Shell/Area Loads→Uniform→Load Case Name = Live →Units=Ib-inch →Load= 80→Direction = Gravity→ OK

5. Area Mesh:

Select Slabs→Assign→ Shell/Area→Area Object Mesh Option→Further subdivided Auto Mesh with minimum element size of =3 →OK

6. Analyze:

Analyze → Model check → OK

Analyze → Run Analysis

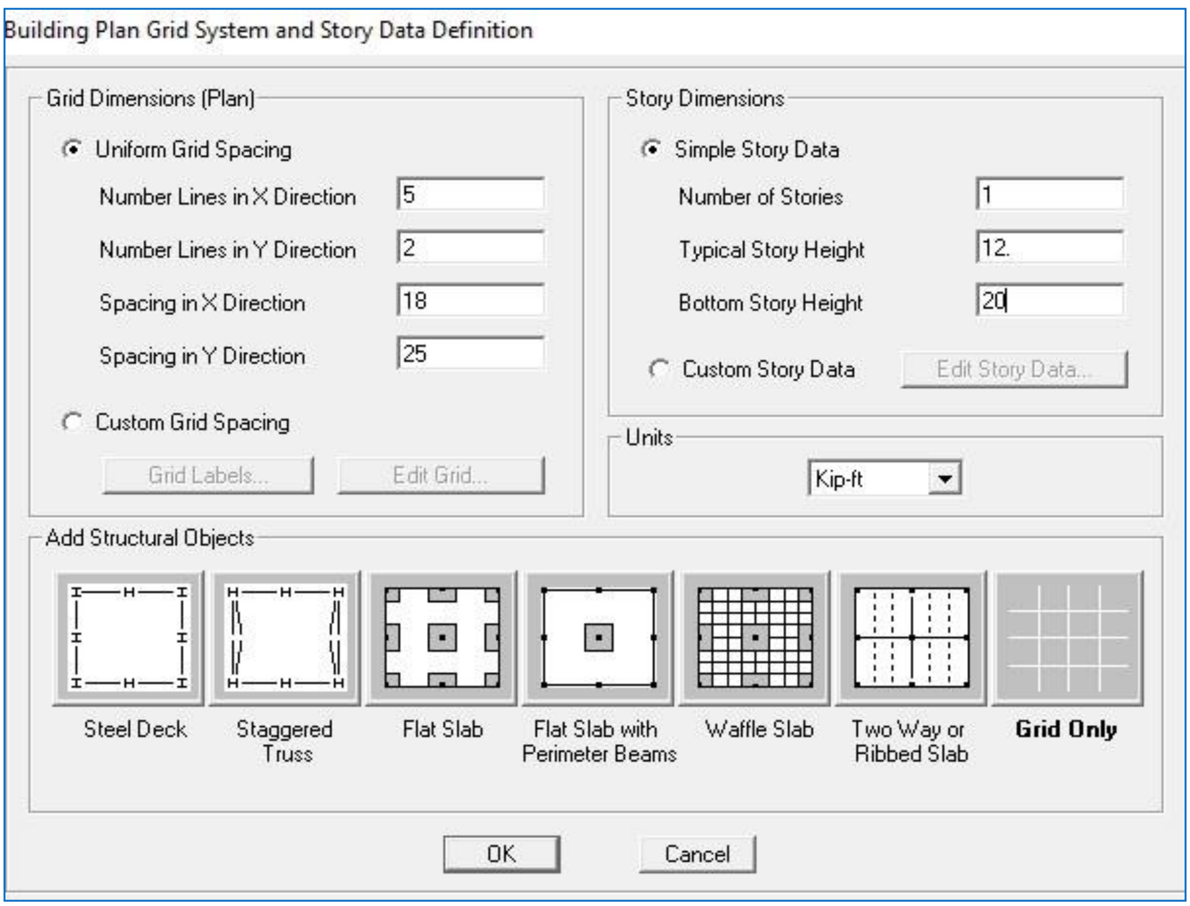


Fig: 3.4

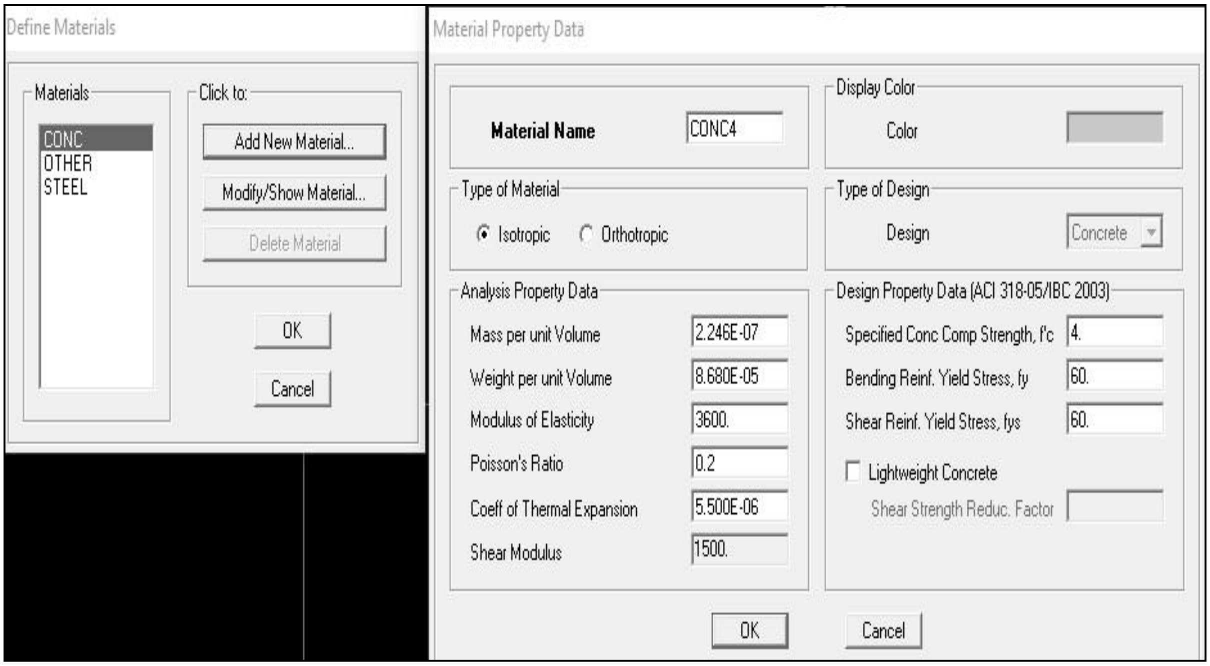

Define Materials
Materials
CONC
OTHER
STEEL
Click to:
Add New Material...
Modify/Show Material...
Delete Material
OK
Cancel
Material Property Data
Material Name
CONC4
Display Color
Color
Type of Material
Isotropic
Orthotropic
Type of Design
Design
Concrete
Analysis Property Data
Mass per unit Volume
2.246E-07
Weight per unit Volume
8.680E-05
Modulus of Elasticity
3600.
Poisson's Ratio
0.2
Coeff of Thermal Expansion
5.500E-06
Shear Modulus
1500.
Design Property Data (ACI 318-05/IBC 2003)
Specified Conc Comp Strength, f'c
4.
Bending Reinf. Yield Stress, fy
60.
Shear Reinf. Yield Stress, fys
60.
Lightweight Concrete
Shear Strength Reduc. Factor
OK
Cancel

Fig: 3.5

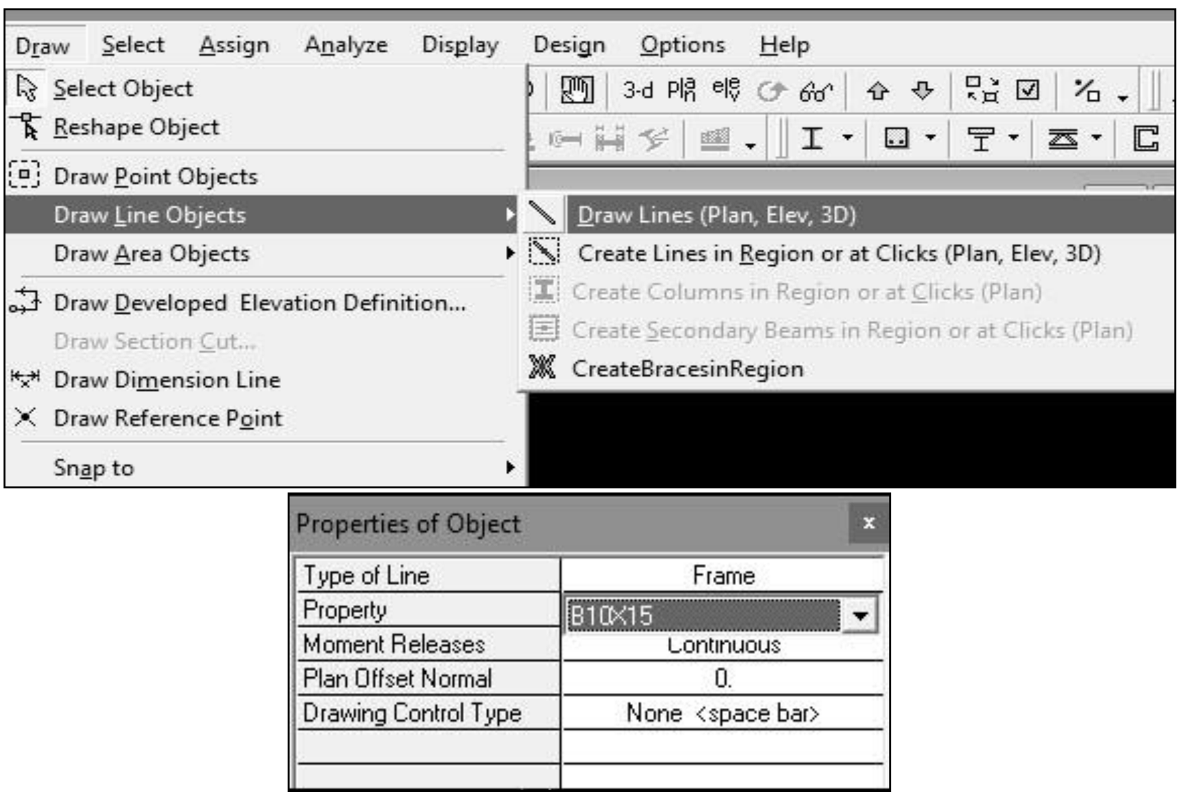

Draw
Select
Assign
Analyze
Display
Design
Options
Help
Select Object
Reshape Object
Draw Point Objects
Draw Line Objects
Draw Area Objects
Draw Developed Elevation Definition...
Draw Section Cut...
Draw Dimension Line
Draw Reference Point
Snap to
Draw Lines (Plan, Elev, 3D)
Create Lines in Region or at Clicks (Plan, Elev, 3D)
Create Columns in Region or at Clicks (Plan)
Create Secondary Beams in Region or at Clicks (Plan)
CreateBracesinRegion
Properties of Object
Type of Line
Frame
Property
B10X15
Moment Releases
Continuous
Plan Offset Normal
0.
Drawing Control Type
None <space bar>

Fig: 3.6

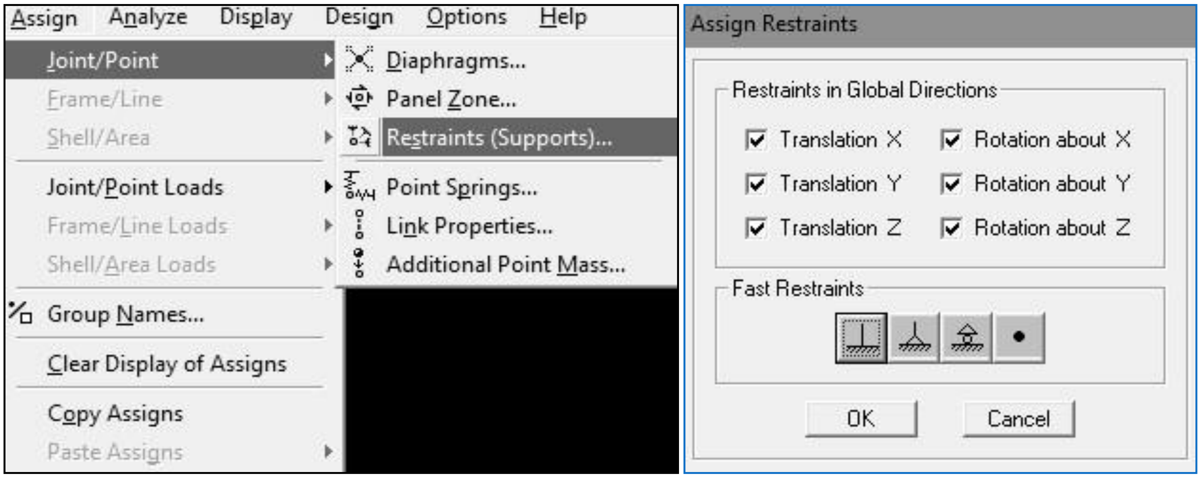


Fig: 3.7

7. Results:

For Support Reactions: Go to display →Show Member Forces/Stress Diagram →Support/Spring Reactions→Select Load (Live/Dead etc.)→ OK→Select the Support point by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment.

WEEK 08-09

MULTI-STORIED BUILDING FRAME UNDER ALL LOADS

Objective: Analyze the following 9-Storied residential building under all loads and find out the following items;

1. Supports Reactions for foundation design
2. Column Axial forces
3. Beams forces

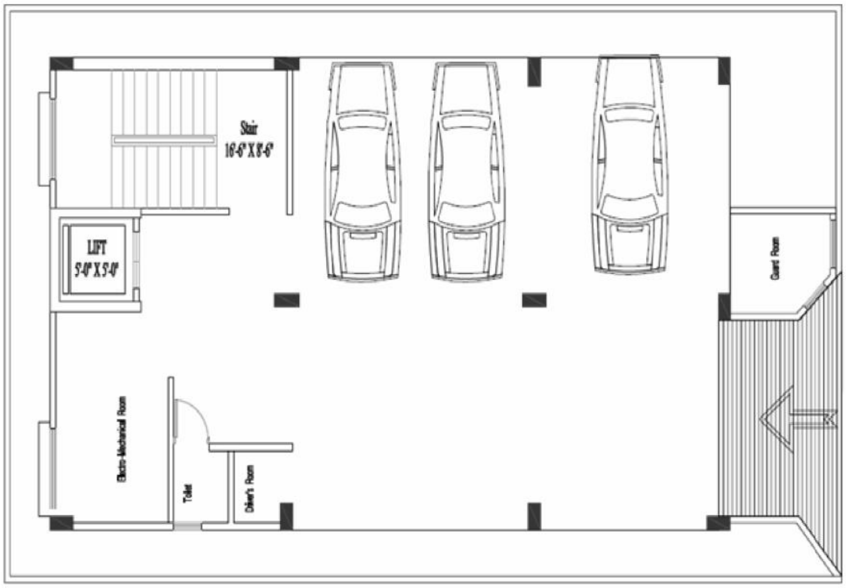


Fig. 4.1 : Ground Floor Plan

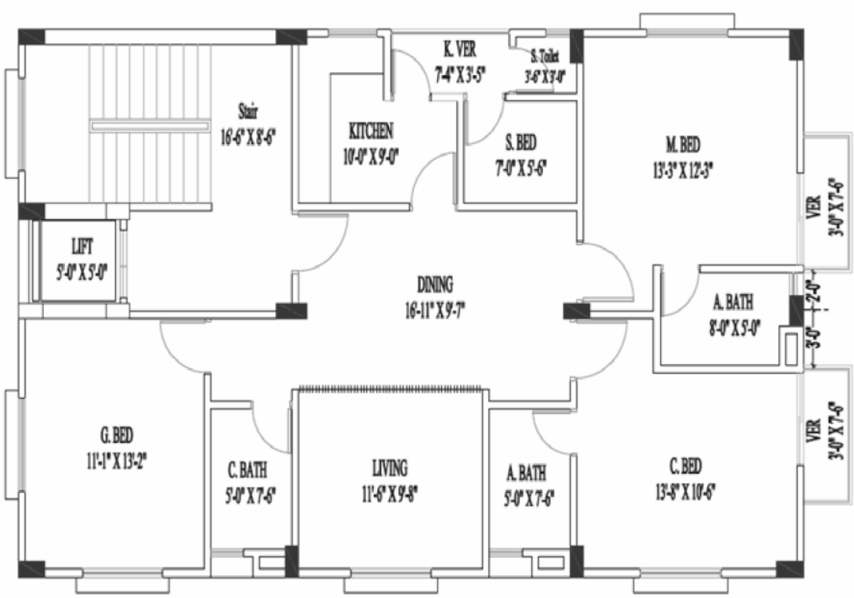


Fig. 4.2:Typical Floor Plan

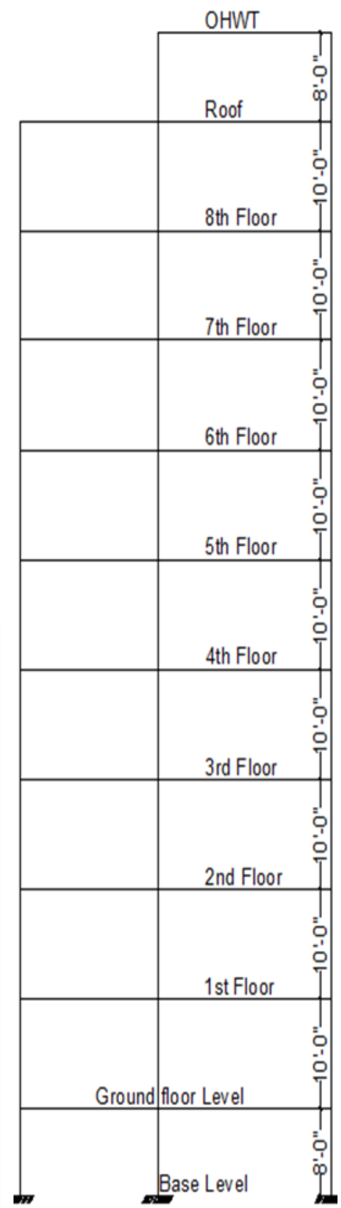


Fig. 4.3:Elevation view

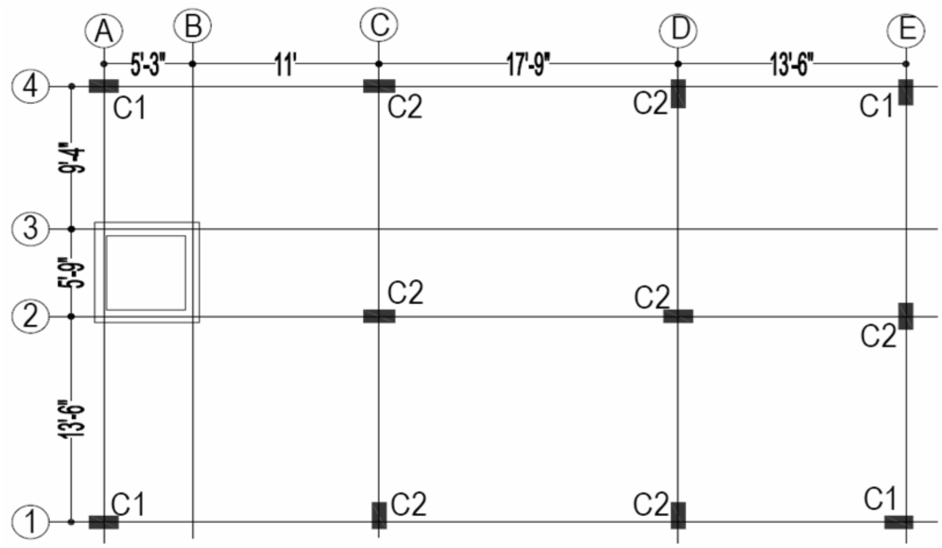


Fig. 4.4: Column Layout Plan

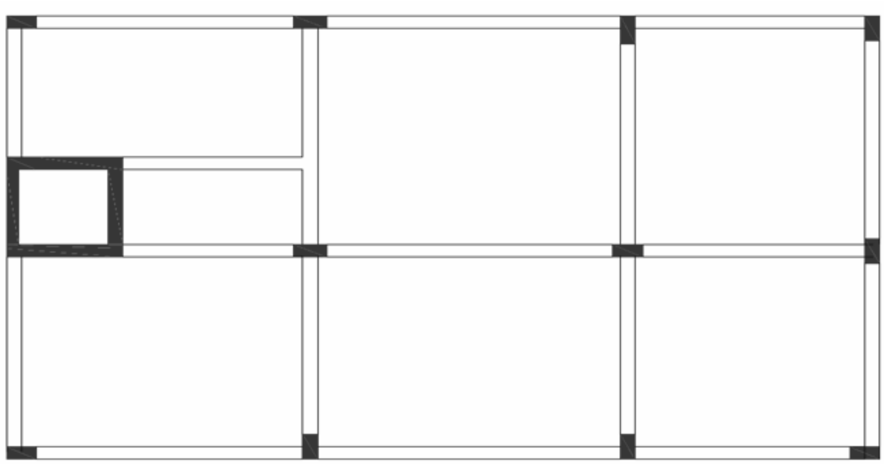


Fig. 4.5: Grade Beam Layout Plan

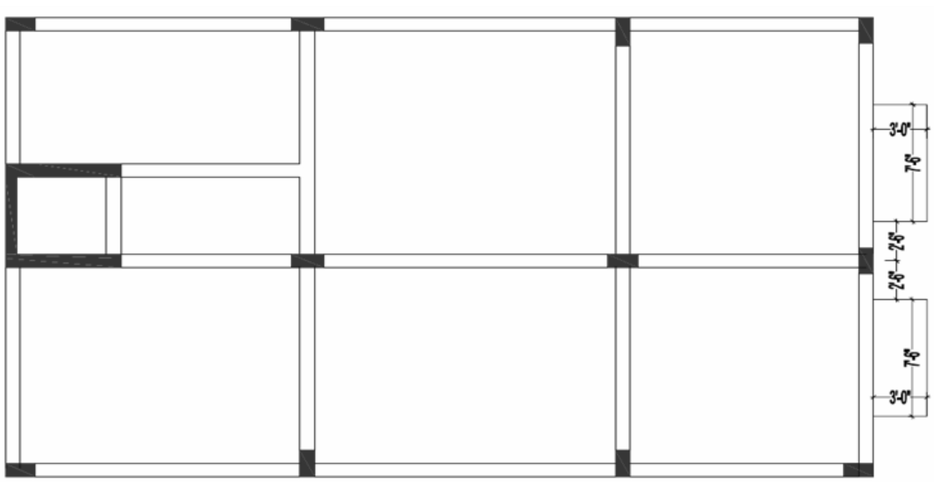


Fig. 4.6: Typical Floor Beam and Slab Layout Plan

Table: 4.1 Geometry and Loads

Properties:	Load Definitions:
Column: C1= 12"X18" C2=12"X20" Materials=Concrete, f'c=4000 psi Beam: GB = 10"X18" FB = 10"X20" Materials=Concrete, f'c=4000 psi Shear wall: Thickness = 8" Materials=Concrete, f'c=4000 psi Slab: Thickness = 6" Materials=Concrete, f'c=3000 psi So, Slab load = (6x150)/12 = 75 psf • All supports are fixed support • Bottom story height = 8'-0" • Typical story height = 10'-0" • Top story for lift & stair = 8'-0"	1. Dead Load: Self weight (Factor=1) Floor Finish (FF) = 30 psf Partition wall load (PW) = 25 psf Wall load on beams (W) = 416 Ib/ft (for 5" brick wall) 2. Live Load: LL = 40 psf 3. Seismic Definition: (Dhaka zone) EQx & EQy 4. Wind Definitions: (for Dhaka) Wx & Wy *Wind speed for Dhaka zone = 210 km/hr = 130 mile/hr **Load Combinations:** UFL = DL+LL FDL = 1.2*DL+1.6*LL FDLEQx = 0.9*DL+1.2*LL+1.32*EQx FDLEQz = 0.9*DL+1.2*LL+1.32*EQy FDLWx =0.9*DL+1.2*LL+1.2*Wx FDLWz = 0.9*DL+1.2*LL+1.2*Wy

Procedure:

1. Grid System and Story data definition: Open ETABS software →File → New Model → no →Units = Kip-ft →Number of lines in X-direction= 5→ Number of lines in Y-direction= 4→ Number of stories=11→Bottom Story height=8→Typical Story height = 10→ Click on Custom Grid spacing → Edit Grid → Click on Spacing →Then X-direction Grid spacing A = 15'3", B=11', C=17'9"→D=13'6"→E=0→ Then Y-direction Grid spacing 1 = 13'6", 2=5'9", 3=9'4"→4=0→ OK → Then click on Custom Story Data→Edit Story Data→Now change the Label as GB,

   STORY1...........STORY8, ROOF,OHWT→Height OHWT =8→Elevation, BASE = -8→Master Story, STORY1 = Yes→Similar to, BASE,ROOF and OHWT = NONE and from STORY2 to STORY8 = STORY1→OK→Grid only→ OK. (Fig: 4.7).

2. Define:

   2.1 Materials Properties: Define → Materials Properties→ Add New Materials →Material Name = CONC3(f'c=3000psi)→ Specified Conc. Comp. Strength = 4 → Modulus of Elasticity = 3122→OK (in the same way define other materials like CONC4 for f'c =4000psi→OK. (Fig: 4.8)

   2.2 Frame Sections (for Beam, Column): Define → Frame Sections→ Add Rectangular →Section Name = C12X18→ Material = CONC4→ Depth=18, Width = 12 → Reinforcement→Column

→Cover to Rect. Center= 1.5→OK →OK → OK. (Fig: 4.9). In the same way define other Columns and Beams

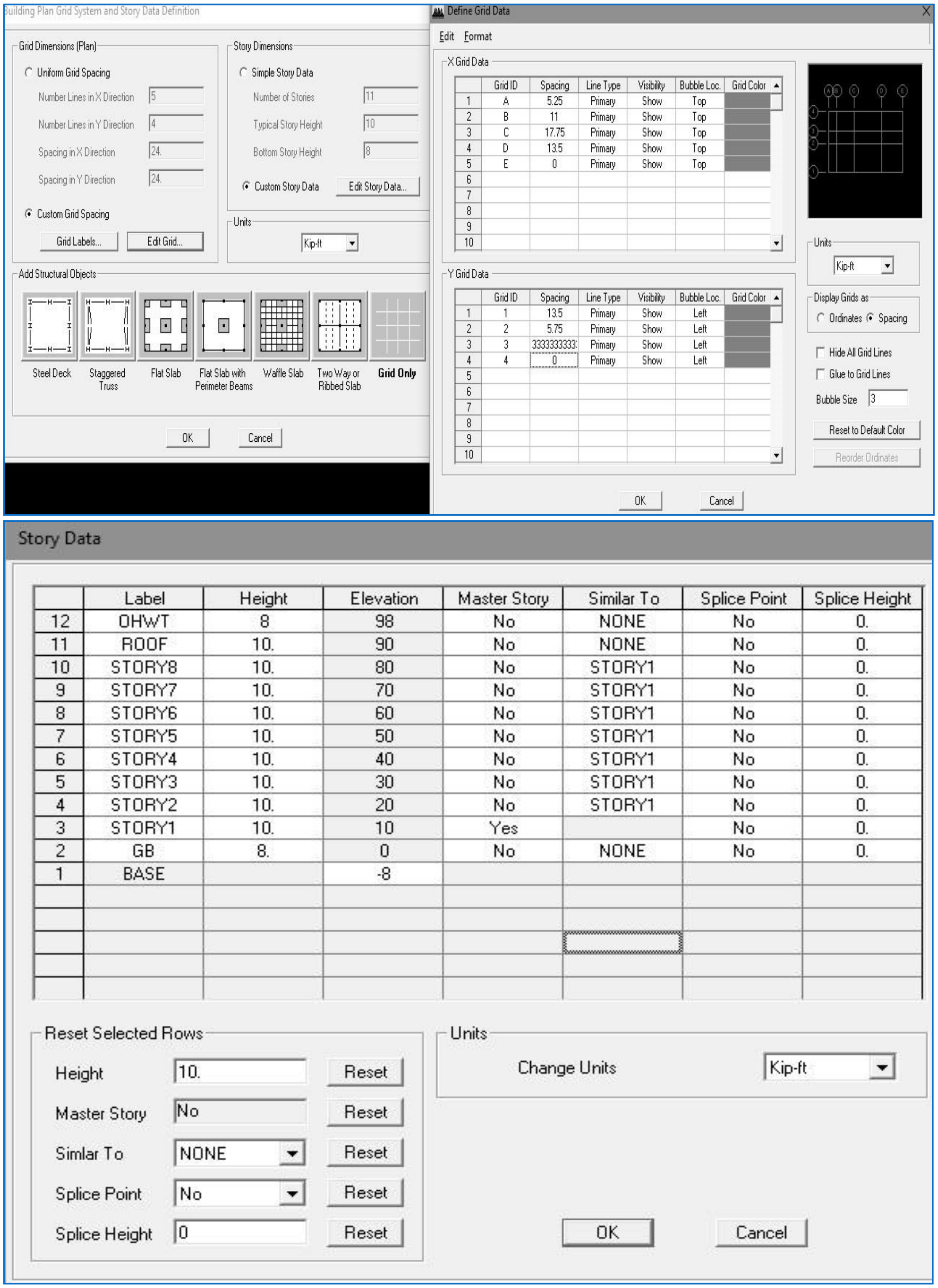


Fig 4.7

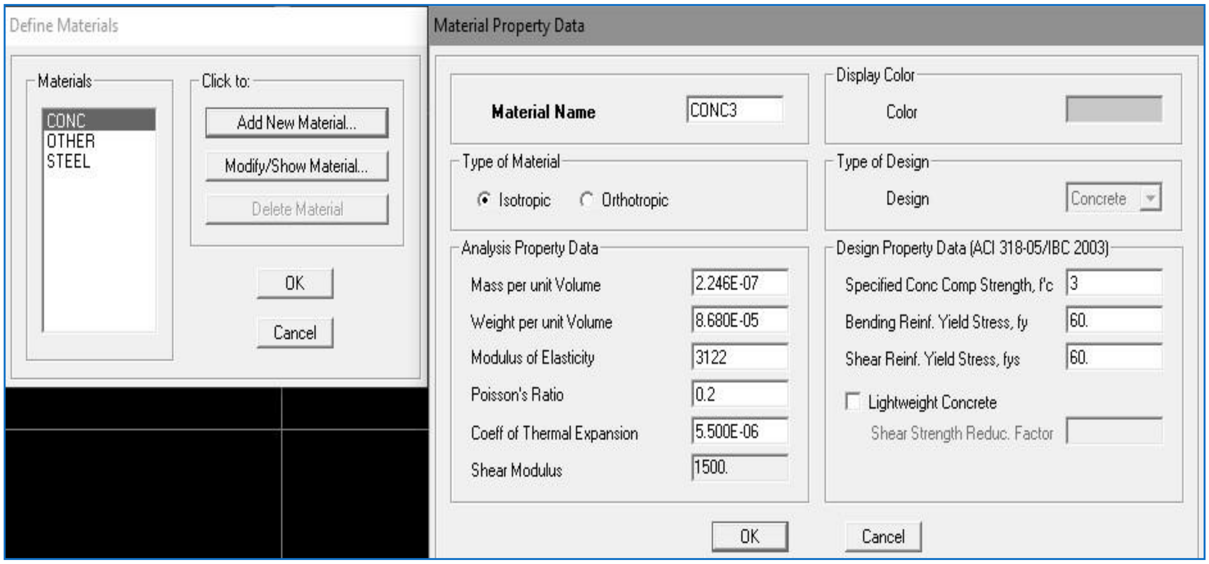


Fig 4.8

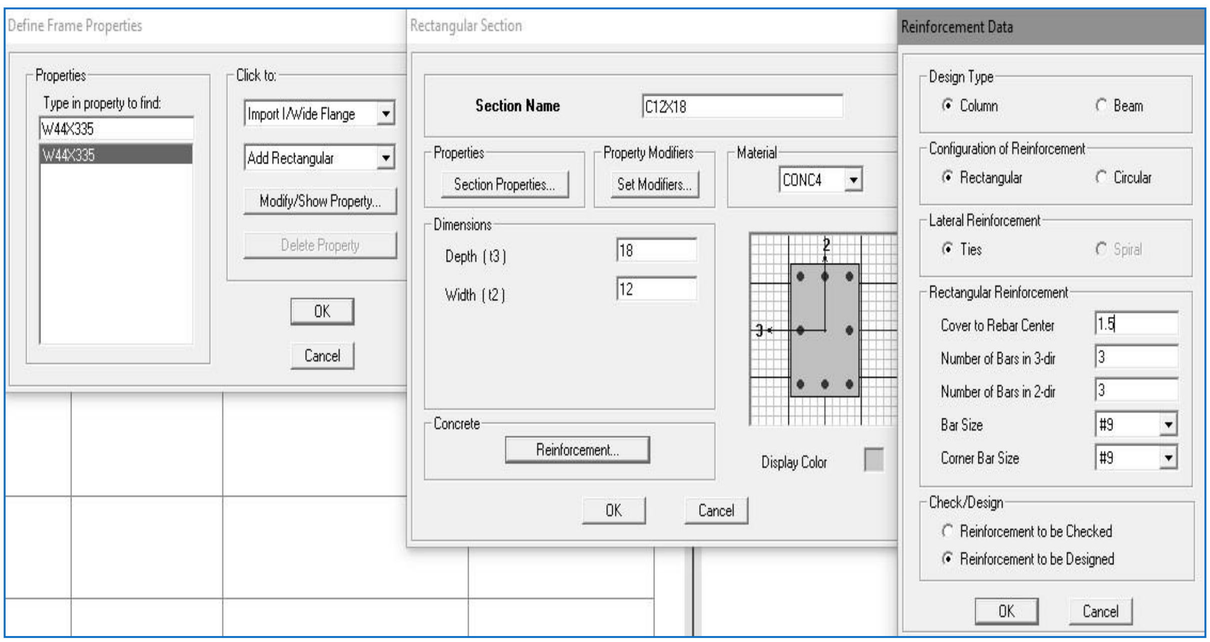


Fig 4.9

2.3 Wall/Slab/Deck Sections: Define→Wall/Slab/Deck sections→SLAB1→Modify/Show Section→ Section Name=SLAB6→Material=CONC3→Thickness: Membrane=6, Bending=6→Type: Shell→Set Modifiers→Bending m11 Modifier=Bending m11 Modifier=Bending m22 Modifier=Bending m12 Modifier= 0.00001→ OK →OK → OK.
(Fig: 4.10).

In the same way define other Slabs and Shear Walls.

2.4 Static Load Cases: Define →Static Load Cases→Load: SFW, Type: DEAD, Self wt Multiplier: 1→Modify Load→Again, Load: FF, Type: DEAD, Self wt Multiplier: 0→Add New Load

Load: PW, Type: DEAD, Self wt Multiplier: 0→Add New Load

Load: WALL, Type: DEAD, Self wt Multiplier: 0→Add New Load

Load: EQX, Type: QUAKE, Self wt Multiplier: 0, Auto Lateral Load= UBC 94→Add New Load →Modify Lateral Load→X Dir, Seismic Zone factor=1.5, Site coefficient =1.2, Importance factor=1, Method A Ct(ft)=0.03, Top story=OHWT, Bottom story= Base, Numerical coefficient

Rw=8→ OK →OK. (Fig: 4.11, 4.12)

In the same process define other seismic and wind loads.

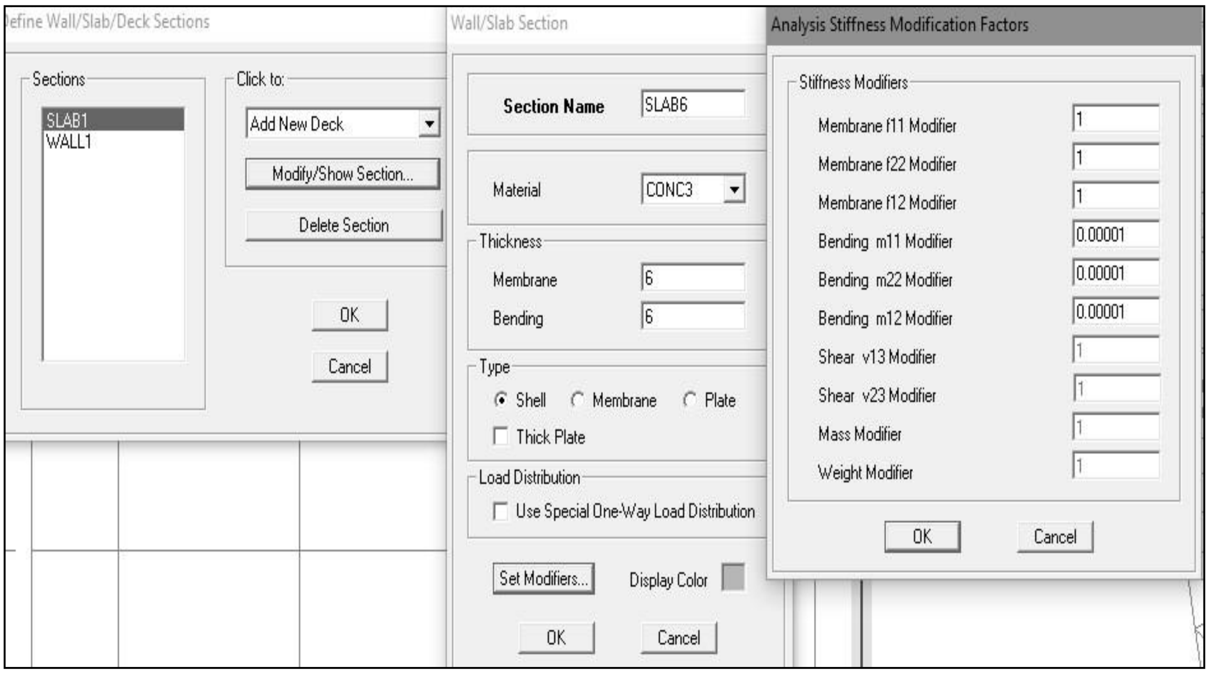


Fig 4.10

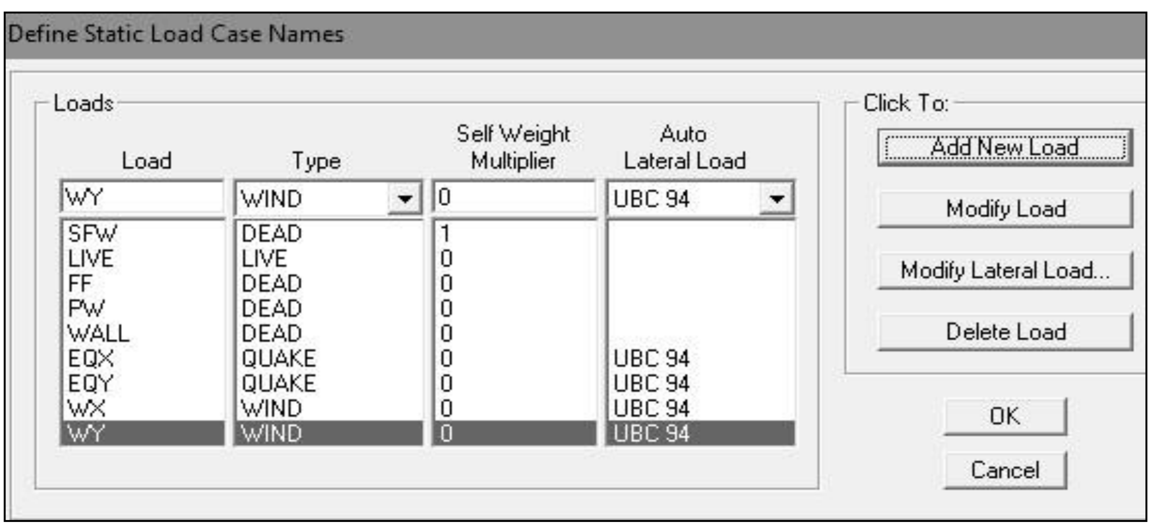


Fig 4.11

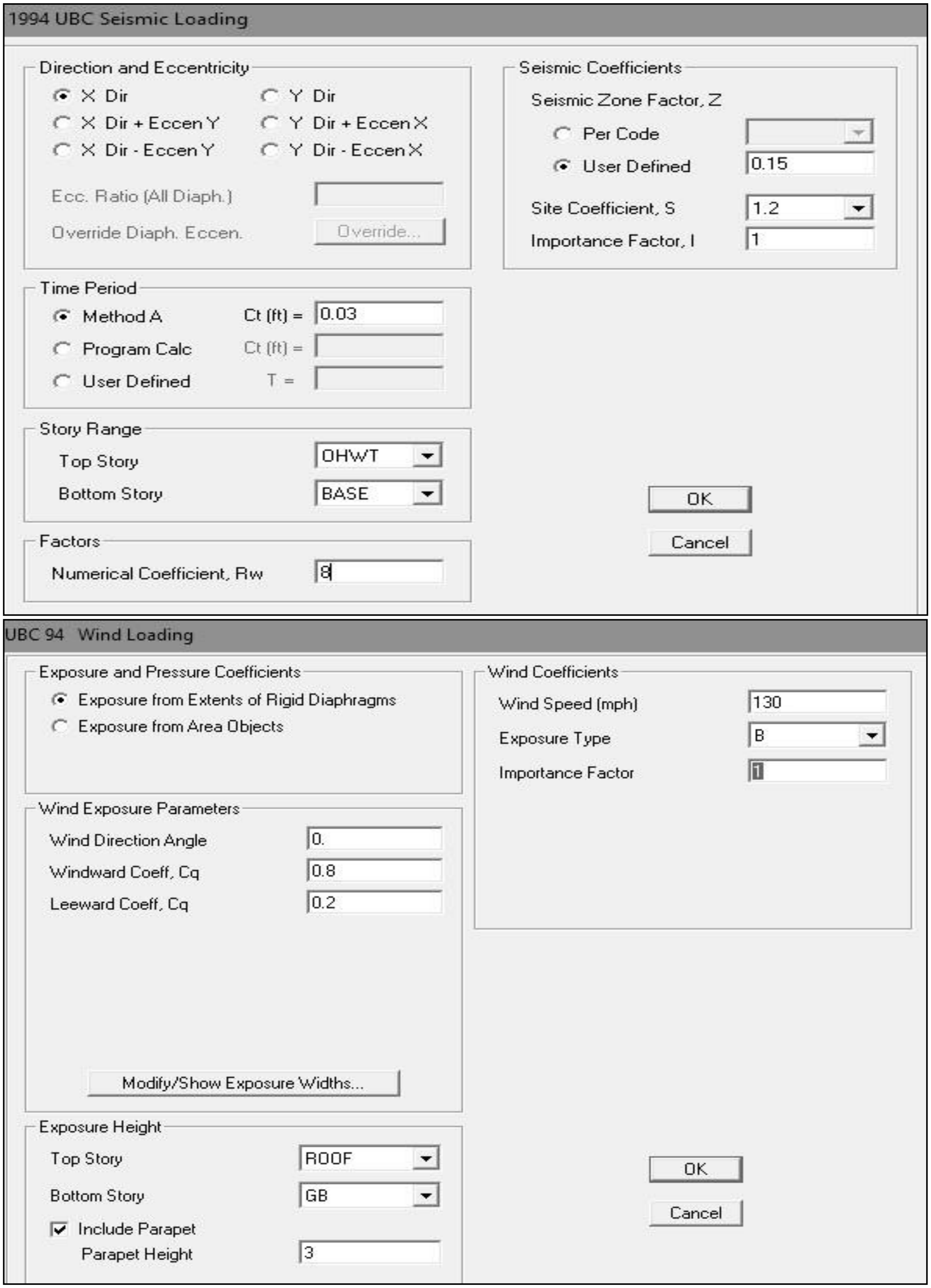

1994 UBC Seismic Loading
Direction and Eccentricity
X Dir
Y Dir
X Dir + Eccen Y
Y Dir + Eccen X
X Dir - Eccen Y
Y Dir - Eccen X
Ecc. Ratio (All Diaph.)
Override Diaph. Eccen.
Override...
Seismic Coefficients
Seismic Zone Factor, Z
Per Code
User Defined
0.15
Site Coefficient, S
1.2
Importance Factor, I
1
Time Period
Method A
Ct (ft) =
0.03
Program Calc
Ct (ft) =
User Defined
T =
Story Range
Top Story
OHWT
Bottom Story
BASE
OK
Cancel
Factors
Numerical Coefficient, Rw
8
UBC 94 Wind Loading
Exposure and Pressure Coefficients
Exposure from Extents of Rigid Diaphragms
Exposure from Area Objects
Wind Coefficients
Wind Speed (mph)
130
Exposure Type
B
Importance Factor
1
Wind Exposure Parameters
Wind Direction Angle
0.
Windward Coeff, Cq
0.8
Leeward Coeff, Cq
0.2
Modify/Show Exposure Widths...
Exposure Height
Top Story
ROOF
Bottom Story
GB
Include Parapet
Parapet Height
3
OK
Cancel

Fig 4.12

2.5 Load Combinations: Define → Load Combinations →Add New Combo.. →Load Combination Name=UDL→Load Combination Type =ADD→ Case Name=SFW static load, Scale Factor=1→Add→Case Name=LIVE static load, Scale Factor=1→Add→Case Name=FF static load, Scale Factor=1→Add→Case Name=PW static load, Scale Factor=1→Add→Case Name=WALL static load, Scale Factor=1→Add→OK

Again→Add New Combo.. →Load Combination Name=FDL→Load Combination Type =ADD→ Case Name=SFW static load, Scale Factor=1.2→Modify→Case Name=LIVE static load, Scale Factor=1.6→Modify →Case Name=FF static load, Scale Factor=1.2→Modify →Case Name=PW static load, Scale Factor=1.2→Modify →Case Name=WALL static load, Scale

Factor=1.2→Modify →OK. (Fig: 4.13)

★Same process follow for other load combination define and finally press OK.

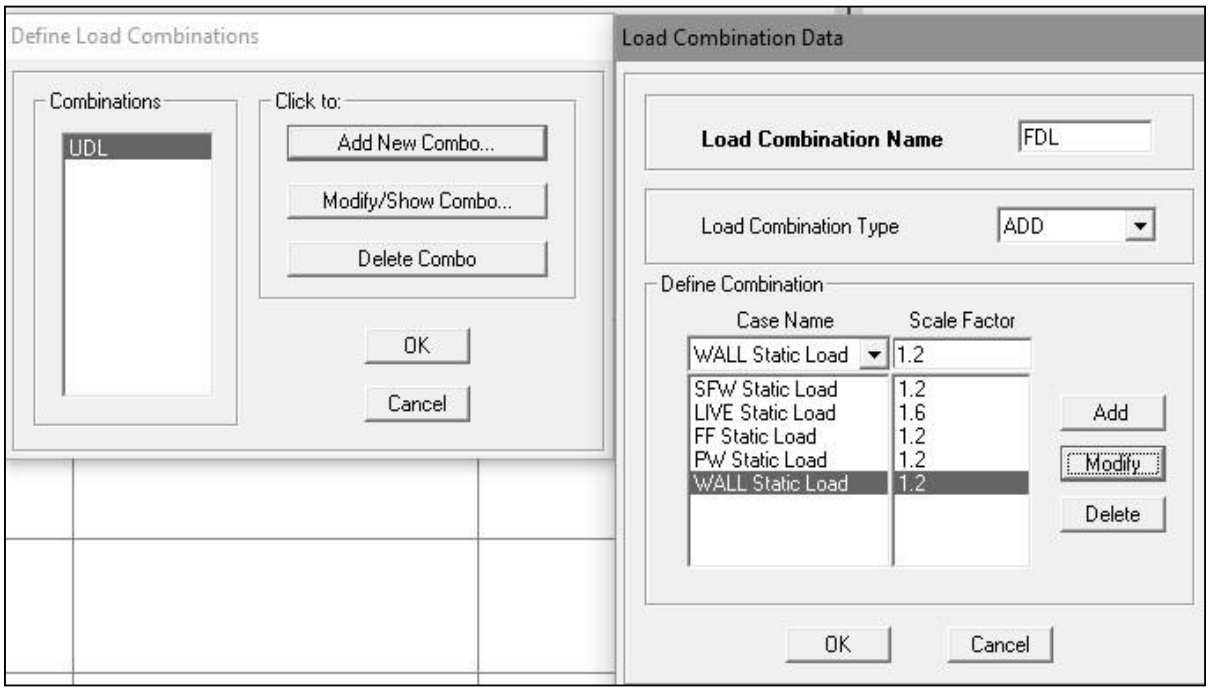


Fig 4.13

WEEK 10-11

MULTI-STORIED BUILDING FRAME UNDER ALL LOADS

3 Draw:

3.1 Column Draw: Plan view is → Story1 or any other without BASE plan and from bottom select →All Story then from menu bar click on Draw → Draw Line Objects → Create Columns in Region or at Clicks (Plan) → Select Property = C12X18 →Then draw the Column on plan view by clicking on every Column points as your Column Layout Plan. (Fig: 4.14)

3.2 Grade/Floor Beam Draw: Plan view is →GB Plan View and for other Story1 or any other without BASE plan and from bottom select →Similar Story then from menu bar click on Draw → Draw Line Objects → Click on Lines (Plan, Elev, 3D) →Select Property = GB10X18→Then draw the line on GB Plan View by clicking one point to another point as your given Grade Beam Layout Plan. (Fig: 4.15, 4.16)

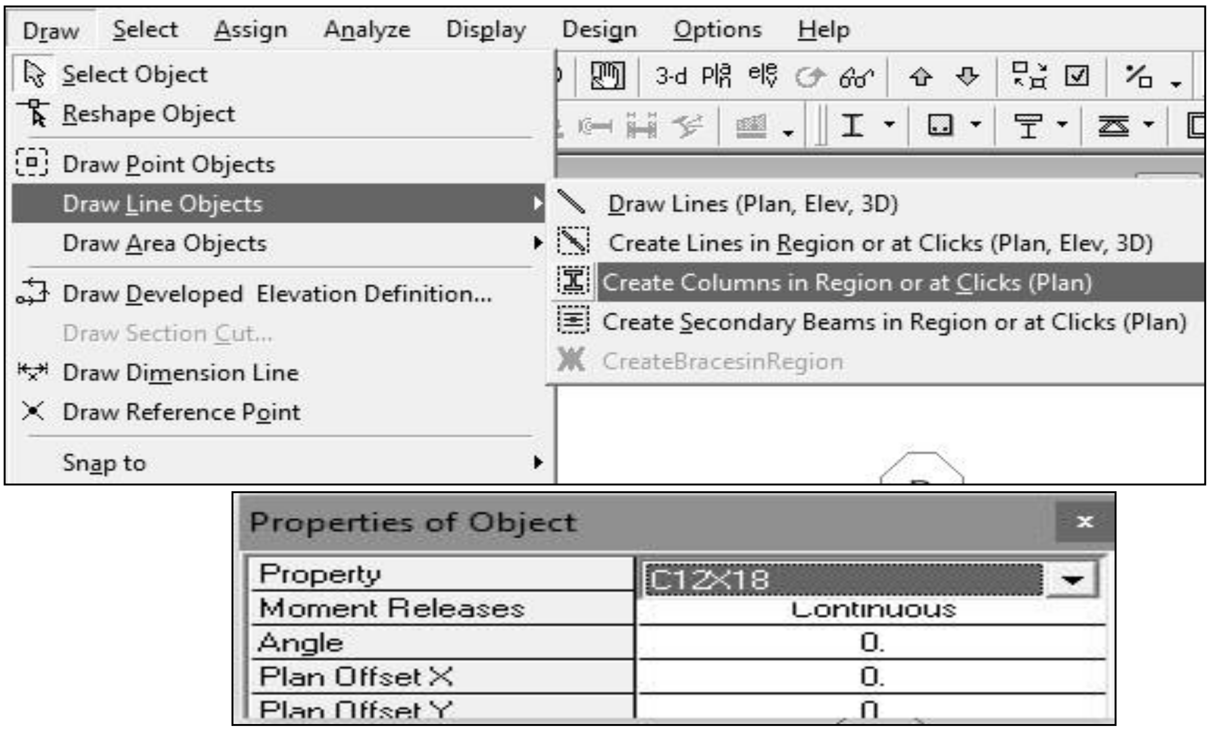


Fig . 4.14

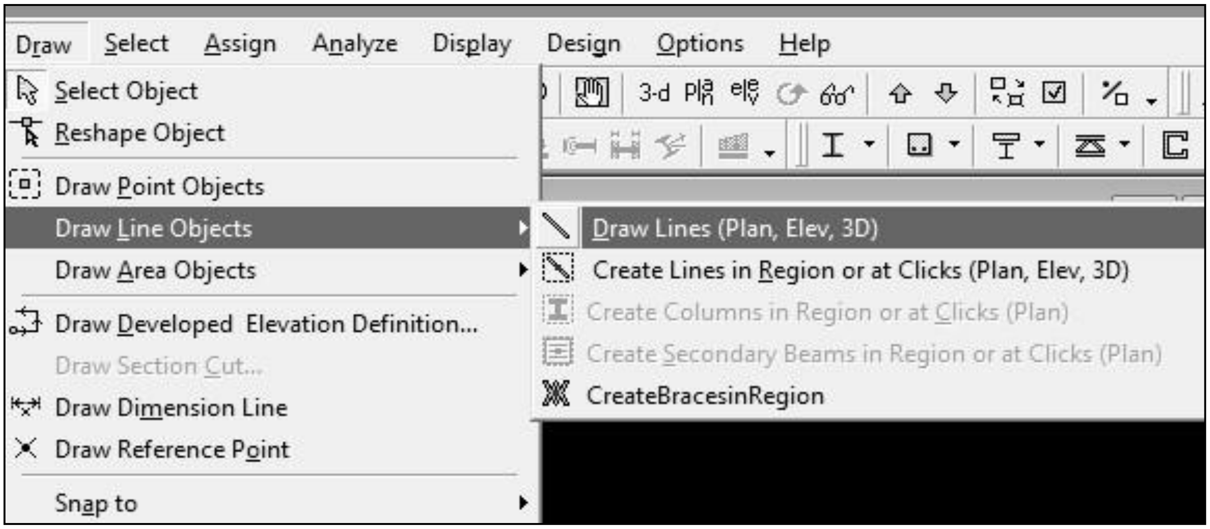


Fig: 4.15

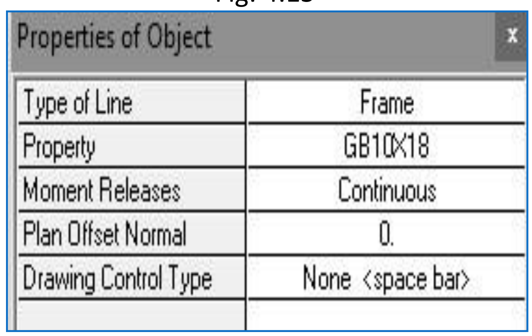


Fig . 4.16

3.3 SLAB Draw: Plan view is →Plan ViewStory1or any other without BASE plan and from bottom select →Similar Story then from menu bar click on Draw → Draw Area Objects → Click on Draw Areas (Plan, Elev, 3D) → Select Property = SLAB6→Then draw the Slab by clicking one point to another point at anti clockwise rotations your given Slab Layout Plan .

- To display the slab on screen go to View→Set Building View Options→Click on Object fill→Apply to all Windows→OK. (Fig: 4.17)

3.4 Varandha Draw: Follow the Fig: 4.18

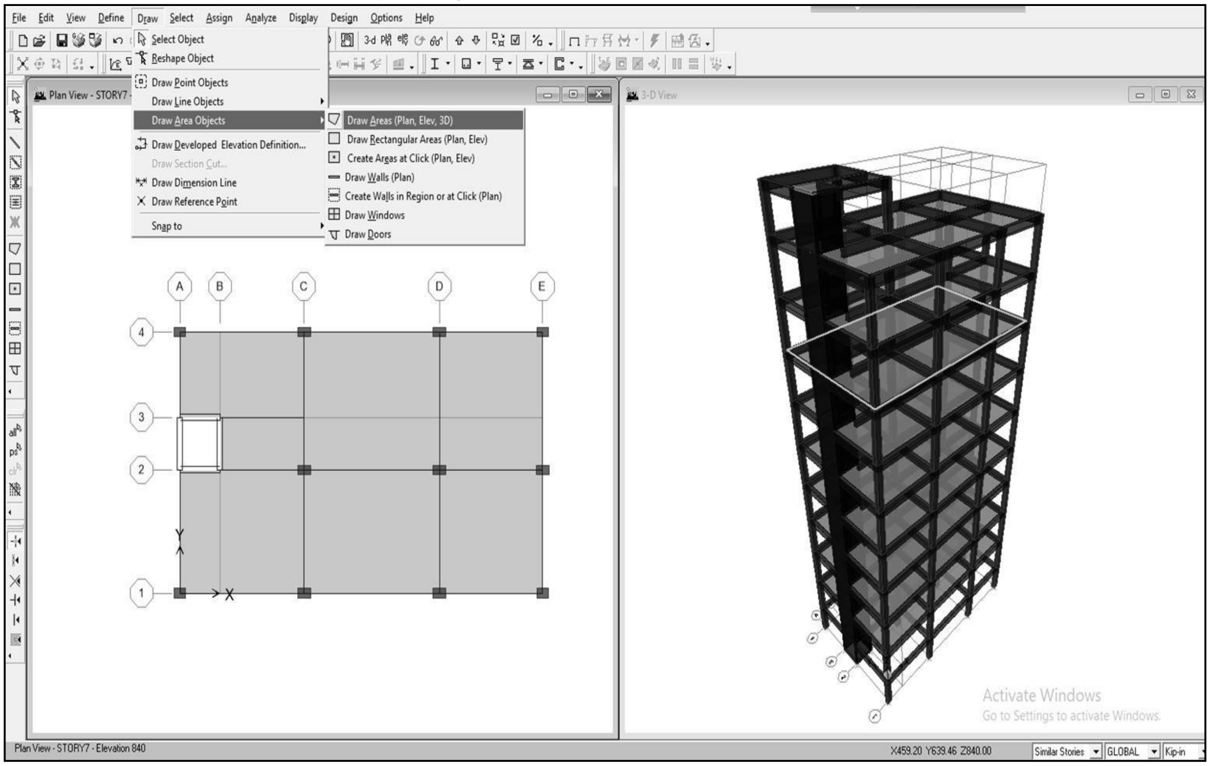


Fig . 4.17

Veranda Draw

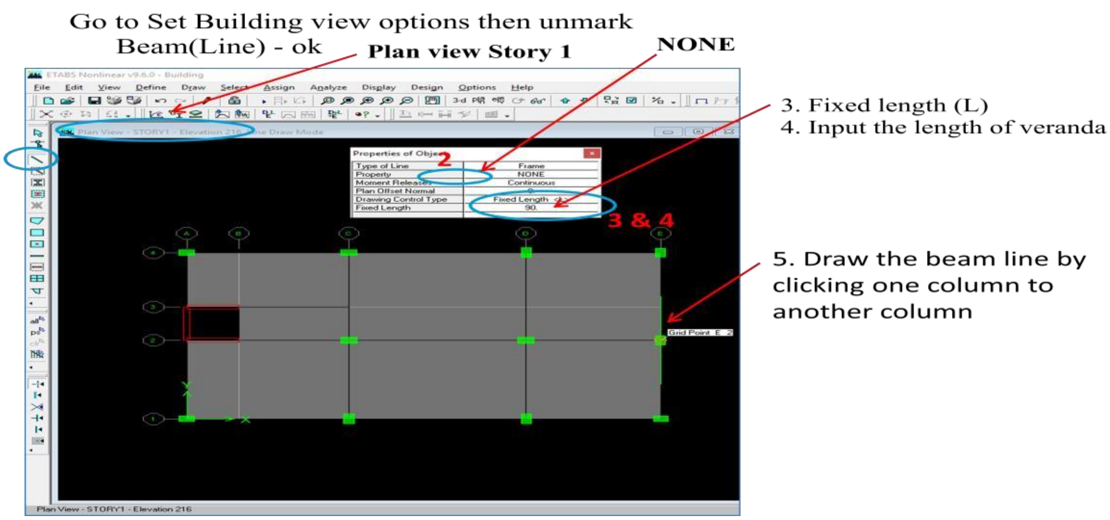
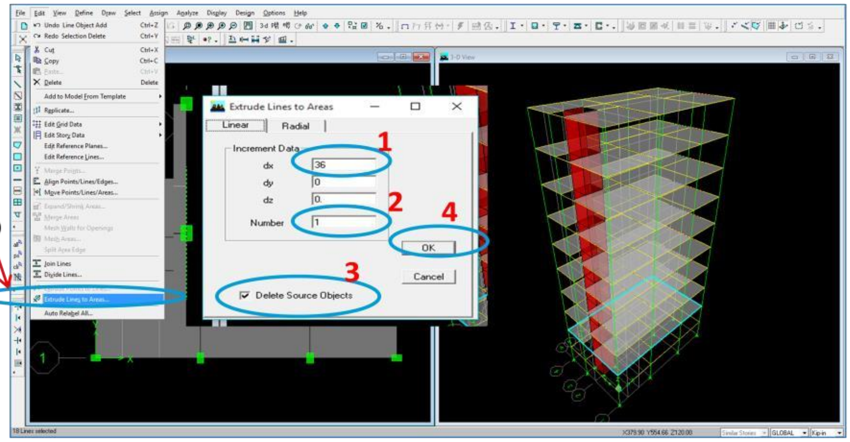


Fig. 4.18a

<u>Veranda Draw</u>

1. **Select line**
2. **Edit**
3. **Extrude lines to Area**
4. **Input Dx/dy/ Value (width of verandah)**



<u>To move</u>

1. **Select Verandah**
2. **Edit**
3. **Move points/lines/Area**
4. **Input distance according to the direction**
5. **ok**

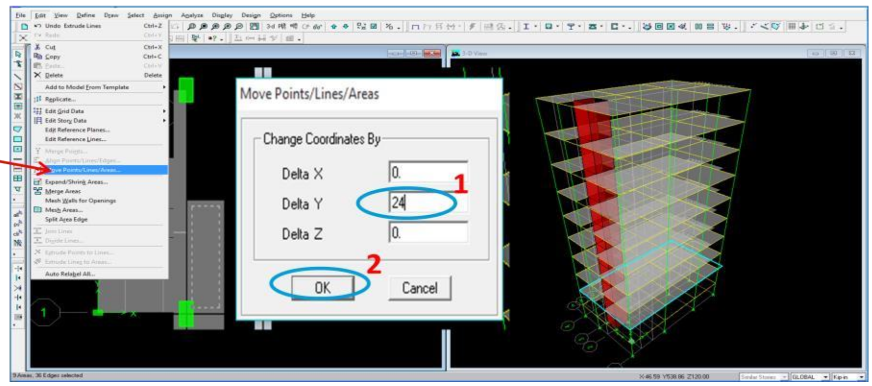


Fig. 4.18b

4 Assign:

4.1 Support Assign:

Select the support point at BASE Plan→ Assign →Joint/Point → Restraints (Supports) →Then select support type (Fixed, Pin, Roller) by clicking on symbols → OK. (Fig: 4.19)

4.2 Load Assign:

4.2.1 Floor Load Assign:

Select Slabs→ Assign →Shell/Area Loads→Uniform→Load Case Name = Live →Units=Ib-ft →Load= 40→Direction = Gravity→ OK. The same procedure follows for other Distributed loads (FF, PW). (Fig: 4.20).

4.2.2 Wall Load Assign:

Select Floor Beams→ Assign →Frame/Line Loads →Distributed→Load Case Name = WALL→Units=Ib-ft →Direction = Local-2→Then write the values of wall Load = 425→ OK. (Fig: 4.21, 4.22)

4.3 Area Mesh and Diaphragm Create:

4.3.1 Area Mesh:

Select Slabs→Assign→ Shell/Area→Area Object Mesh Option→Further subdivided Auto Mesh with minimum element size of =3 →OK. (Fig: 4.23)

4.3.2 Diaphragm:

Select All→Assign→ Shell/Area→Diaphragm →D1→Modify/Show Diaphragm→Rigid →OK→OK. (Fig: 4.24)

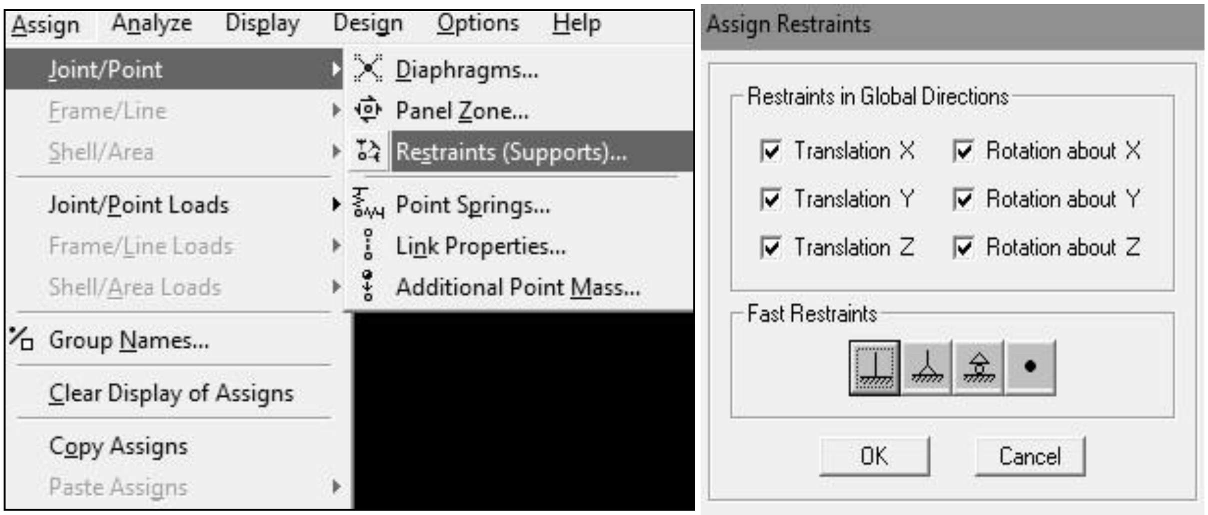


Fig . 4.19

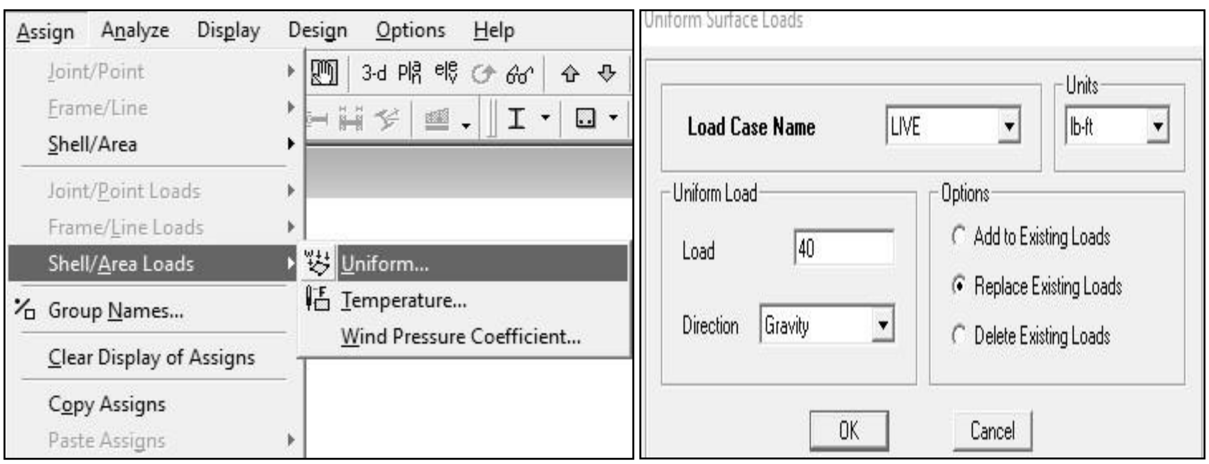


Fig . 4.20

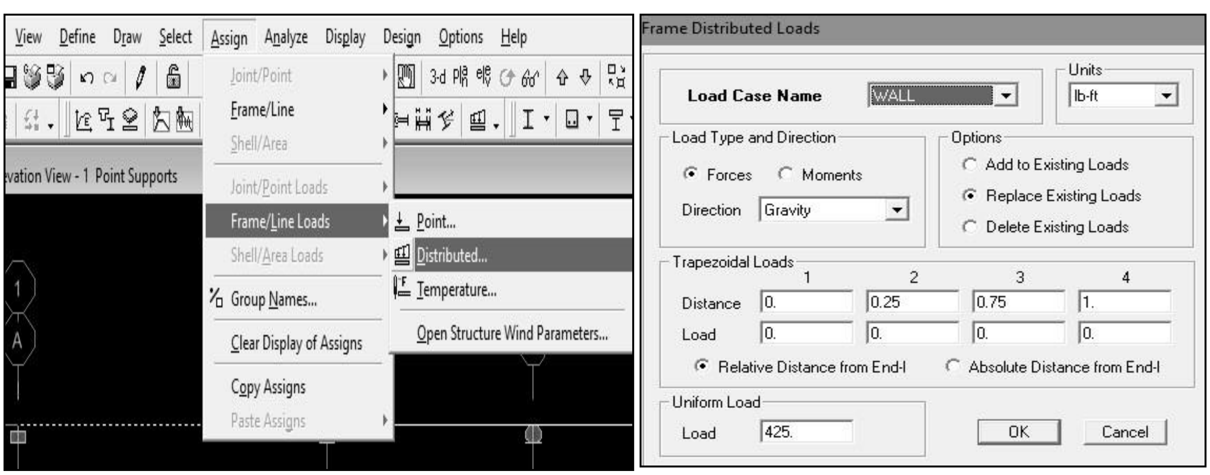


Fig . 4.21

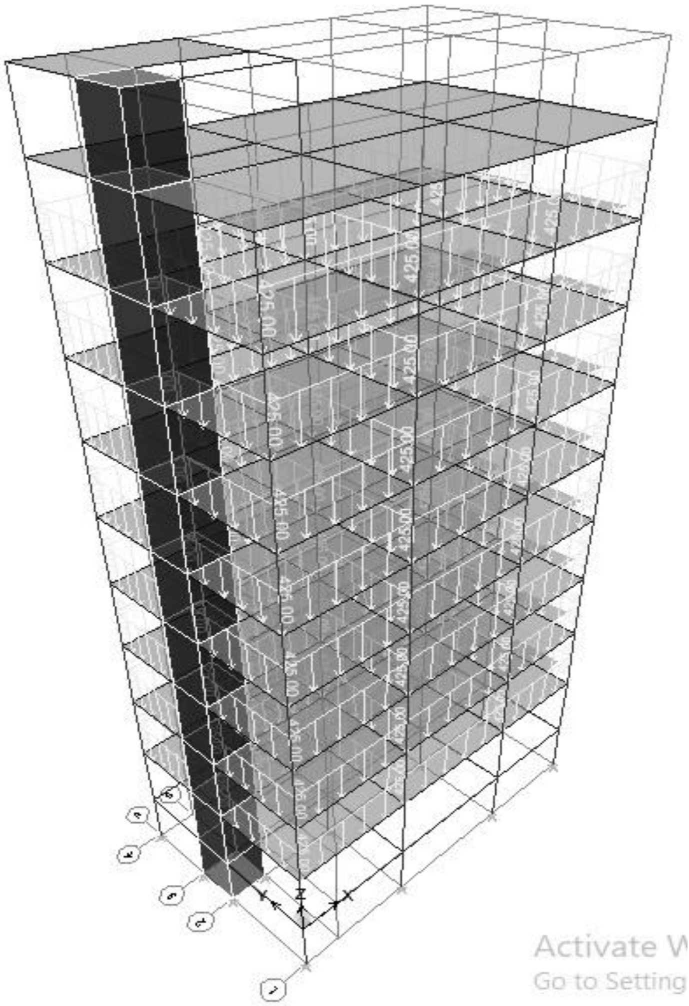


Fig . 4.22

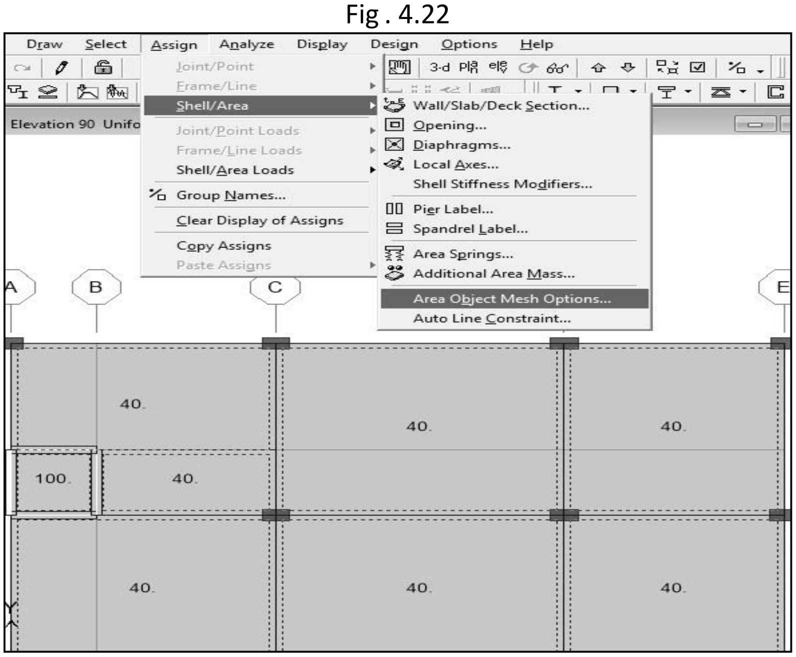


Fig. 4.2 3a

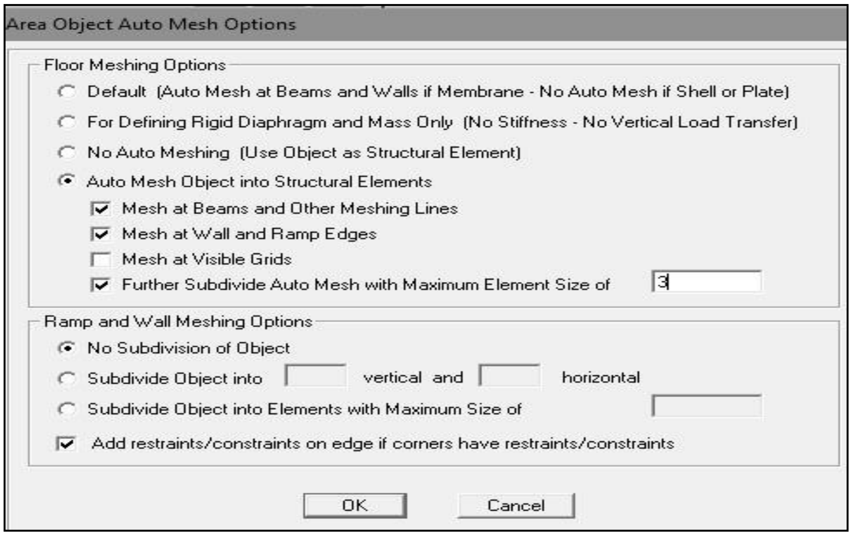


Fig . 4.23b

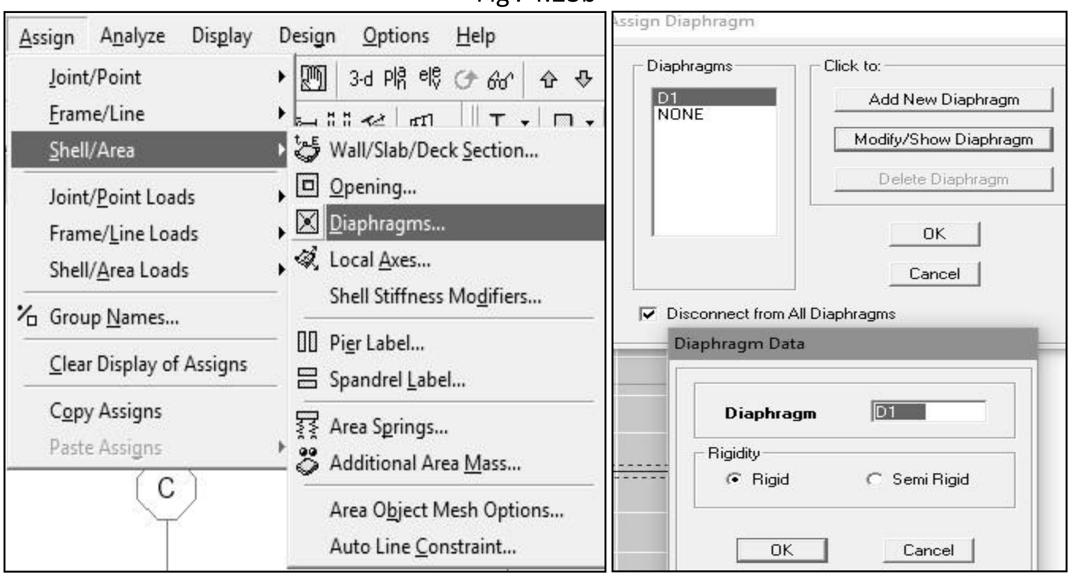


Fig . 4.24

5 Analysis:

Go to Analyze→Check Model (mark all checking options)→OK

Again Go to Analyze→Run Analysis

6 Results:

6.1 Support Reactions:

Go to display →Show Member Forces/Stress Diagram →Support/Spring Reactions→Select Load (Live/Dead etc.)→ OK →Select the Support point by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 4.25, 4.26)

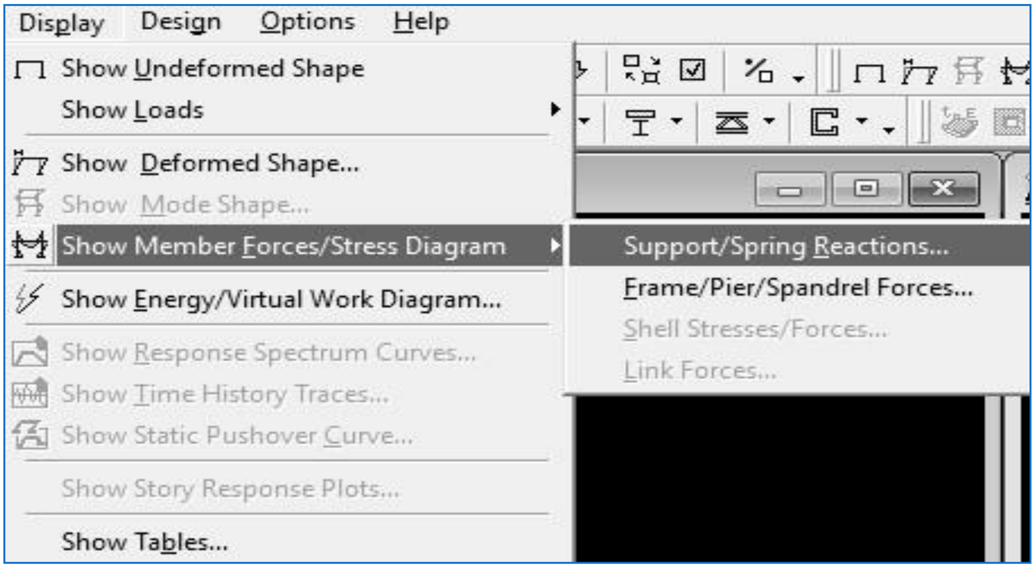


Fig: 4.25

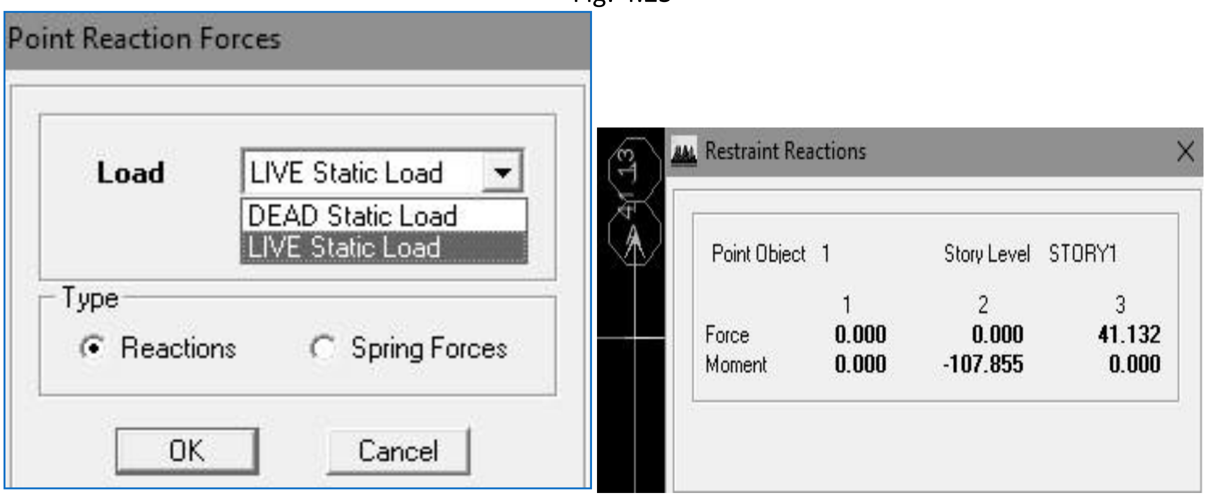


Fig . 4.26

6.2 Maximum Shear force on beam AB: Go to display →Show Member Forces/Stress Diagram →Frame/Pier/Spandrel Forces…→Select Load (Live/Dead etc.)→Shear 2-2 →Click on Show Values on Diagram→ OK →Select the Beam by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 4.27)

6.3 Maximum Bending Moment on beam: Go to display →Show Member Forces/Stress Diagram →Frame/Pier/Spandrel Forces…→Select Load (Live/Dead etc.)→Moment 3-3 →Click on Show Values on Diagram→ OK →Select the Beam by click (from the display elevation view) →press mouse right button and find out your desirable values like vertical force, horizontal force and moment. (Fig: 4.28)

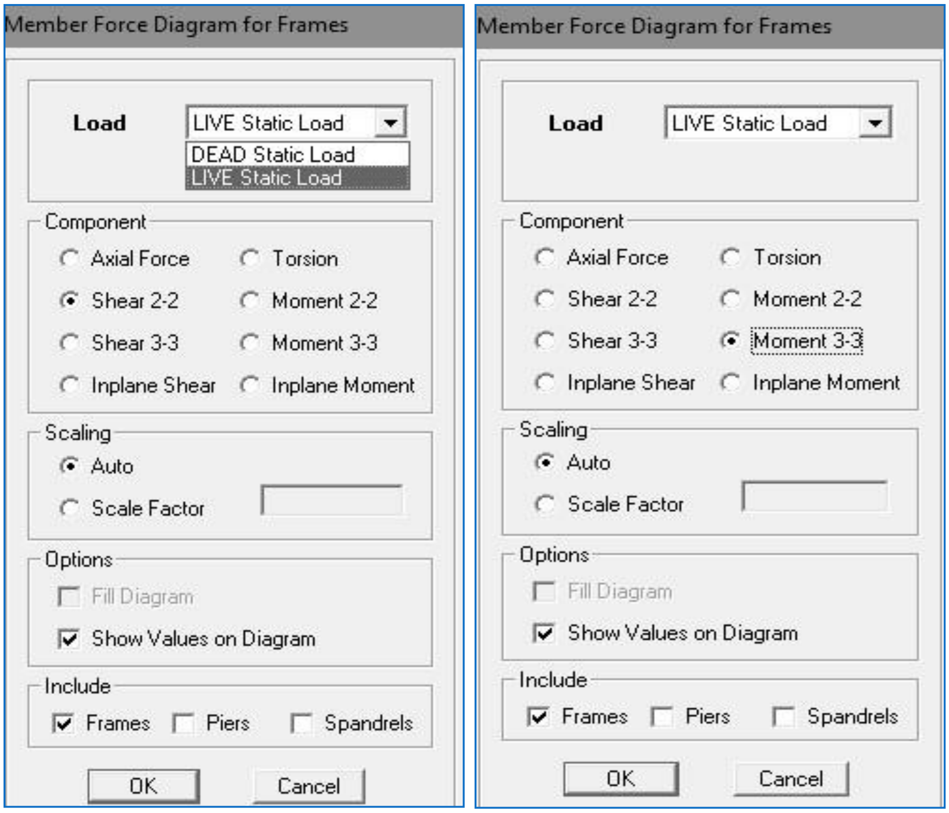


Fig . 4.27

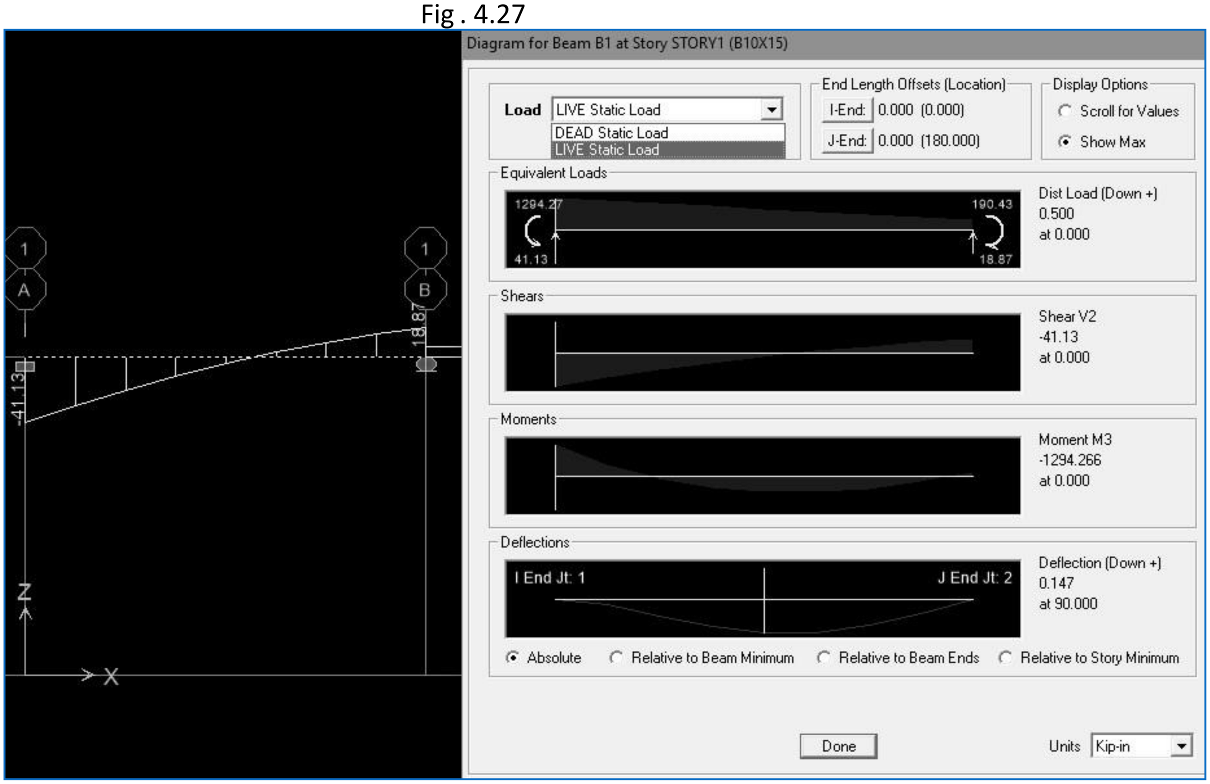


Fig. 4.28

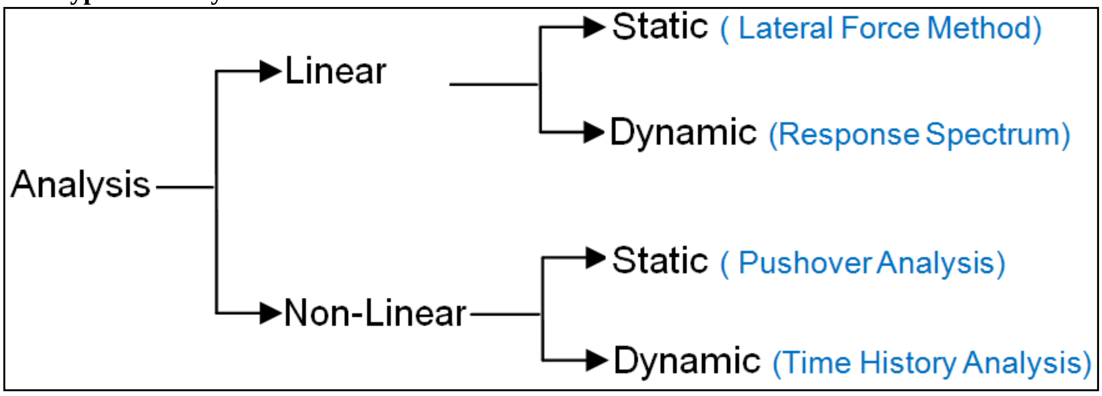
WEEK 12-16

Another Case Study

MULTI-STORIED BUILDING FRAME UNDER ALL LOADS

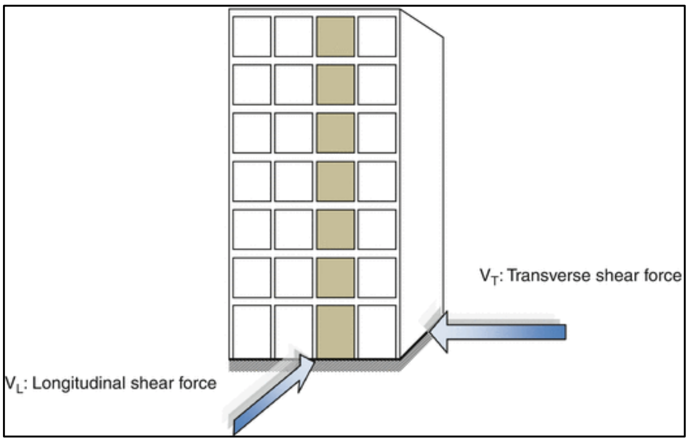
Columns	*Fully exposed to fire*	Siliceous			50	60	70
		Carbonaceous			45	55	65
		Light Weight			40	50	60
	50% exposed to fire	Siliceous			50	55	60
		Carbonaceous			45	50	55
		Light Weight			40	45	45
	One face exposed to fire	Siliceous			40	45	50
		Carbonaceous			40	40	45
		Light Weight			40	40	40
Walls	*Two faces exposed to fire*	Siliceous			50	60	70
		Carbonaceous			45	55	65
		Light Weight			40	50	60
	One face exposed to fire	Siliceous			50	55	60
		Carbonaceous			45	50	55
		Light Weight			40	45	45

5.4 Types of Analysis



5.4.1 Static Lateral Force Method

The similar static lateral force approach gives an easy alternative for constructing against the dynamic loads of an expected earthquake. The whole seismic force, denoted by the symbol V, has been assessed along two horizontal axes that are perpendicular to the primary axis of the structure.



It presupposes lateral response. To eliminate subsurface torsional movement, the structure must be low-rise and symmetrical. Seismic forces must not affect the building in both directions.

5.4.2 Linear Dynamic (Response spectrum) analysis

Construction uses reaction spectrum analysis (RSA). Modal decomposition and the response spectrum simplify response history analysis (RHA). Peak response is quickly determined without response history analysis. This is crucial because response spectrum analysis (RSA) uses fast and simple calculations, unlike time history analysis, which solves the differential equation of motion over time. The response spectrum represents earthquake risk.

The greatest reaction of an SDOF system versus time during an earthquake creates a response spectrum (or frequency). The damping ratio's maximum response locus determines the SDOF system's response spectrum. Response spectra allow peak structural responses in the linear response range to determine earthquake-induced lateral forces. Peak structural responses in linear response range enable this.

SDOF system response is determined by time- or frequency-domain analysis. The study determines the largest reaction over a certain time period. This applies to all SDOF ages. The last figure shows the damping ratio and input ground motion response spectrum 103. Damping ratios determine response spectra.

5.4.3 Non-linear Static (Push-over) Analysis

Pushover analysis—a nonlinear static analysis—loads a structure arbitrarily. Loading exacerbates structural weaknesses and failure mechanisms. A modified monotonic force-deformation criteria and damping approximations characterise cyclic behaviour and load reversals. Structural engineers assess structure strength using static pushover analysis. Performance-based design may use this research.

An evaluation of a pushover based on performance. The ATC-40 states that in order for performance-based design to be possible, demand and capacity must first exist (Applied Technology Council, Seismic Evaluation and Retrofit of Concrete Buildings). The shaking of the ground and buildings that occurs as a result of an earthquake is an analogy for demand. Nonlinear static analysis gives preliminary approximations of structural displacements or deformations in order to take into account demand. Evaluation of earthquake resistance is often done using capacity as the criterion. Key aspects of performance include capacity as well as the ability to satisfy requirements. As a consequence of this, earthquake resilience is an essential component of the building's overall design.

Pushover analysis using the displacement coefficient and capacity spectrum technique.

Pushover analysis—a nonlinear static analysis—loads a structure arbitrarily. Loading exacerbates structural weaknesses and failure mechanisms. A modified monotonic force-deformation criteria and damping approximations characterise cyclic behaviour and load reversals. Static pushover analysis is a method that structural engineers use to evaluate the true strength of a structure. This study may be seen of as a possible strategy for performance-based design.

An evaluation of a pushover based on performance. The ATC-40 states that in order for performance-based design to be possible, demand and capacity must first exist. The shaking of the ground and buildings that occurs as a result of an earthquake is an analogy for demand. Nonlinear static analysis gives preliminary approximations of structural displacements or deformations in order to take into account demand. Evaluation of earthquake resistance is often done using capacity as the criterion. Key aspects of performance include capacity as well as the ability to satisfy requirements. As a consequence of this, earthquake resilience is an essential component of the building's overall design.

5.4.4 Nonlinear Dynamic (Time History) Analysis

Time-history analysis. Nonlinear structural seismic analysis needs it. This investigation needs a structure's representative seismic time history. Time history analysis steps through a structure's dynamic reaction to a variable loading. Time history analysis determines a structure's seismic response under representative earthquake dynamic loads.

VI. ANALYSIS AND DESIGN OF PROPOSED G+3 RESIDENTIAL BUILDING

6.1 Salient Features of Project:

G+3 residential building. It has two tales and a feature piece. Drawings show the site's normal floor arrangement and functioning. The building has a stairwell.

Each level will cover:

6.2 PROPOSED PROJECT

Ground-floor, two-story residential building is proposed. Bangalore's executing.

Table 6.1:- Project details

1.Type of structure	Residential building
2.Layout	As shown in the plan
3.Number of storey	G+3
4.Storey height	3.0m
5.Depth of foundation	2.1
6.Wall	230mm thick
7.Live Load	IS 875(Part-2)1987
8.Material	M20 grade concrete&Fe500 steel
9.Design Philosophy	Limit state method conforming to IS456-2000

6.3 Structural Plan:

Floor-to-floor structural plans vary according to customer needs. Sometimes the architect may propose alternative plans for different levels, considering the aesthetics and space requirements of a structure. The structural plan mostly deals with the column-beam arrangement. Floor plans will vary in this project. The floor plans of the building are shown below.

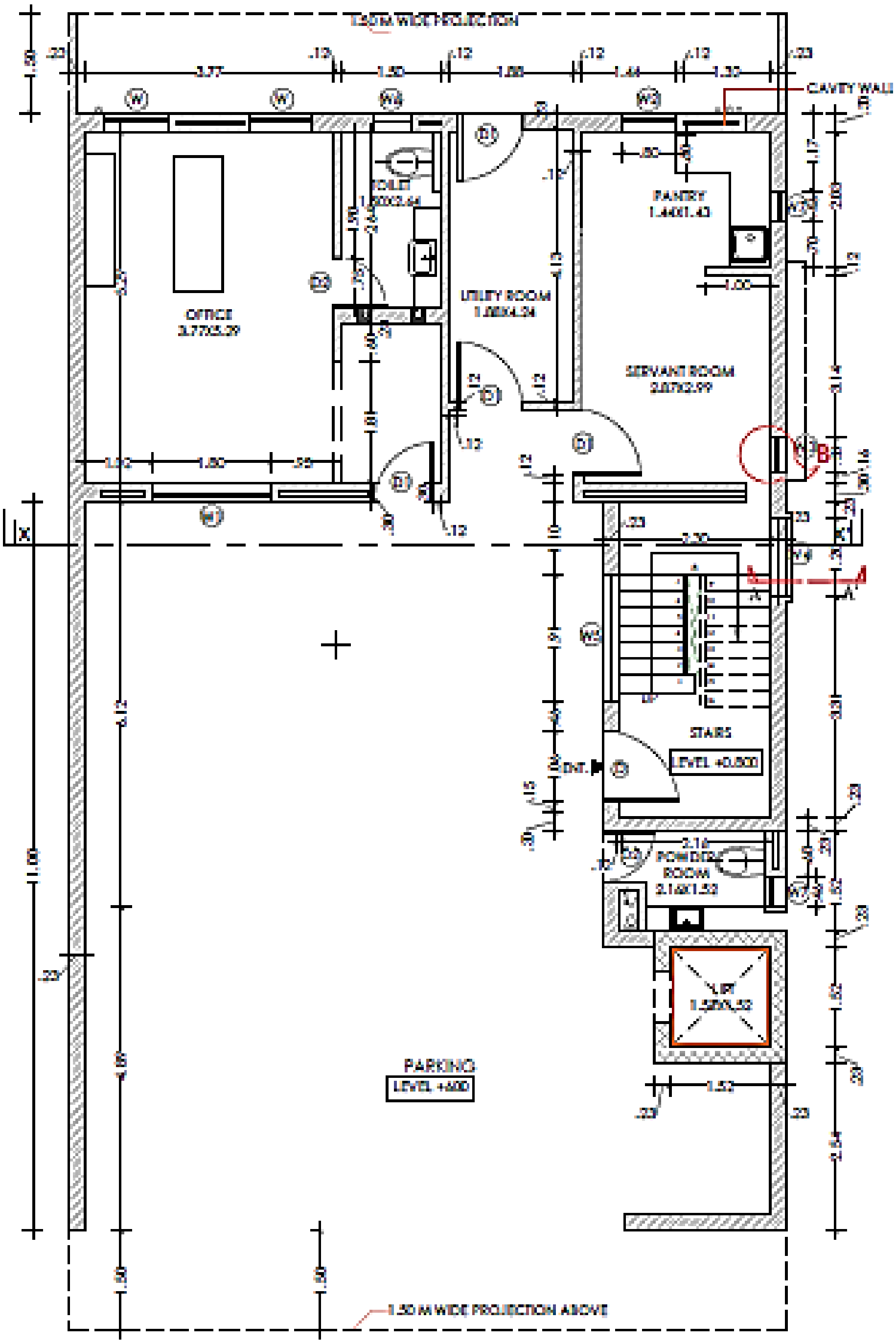


Fig 6.1:- Ground floor

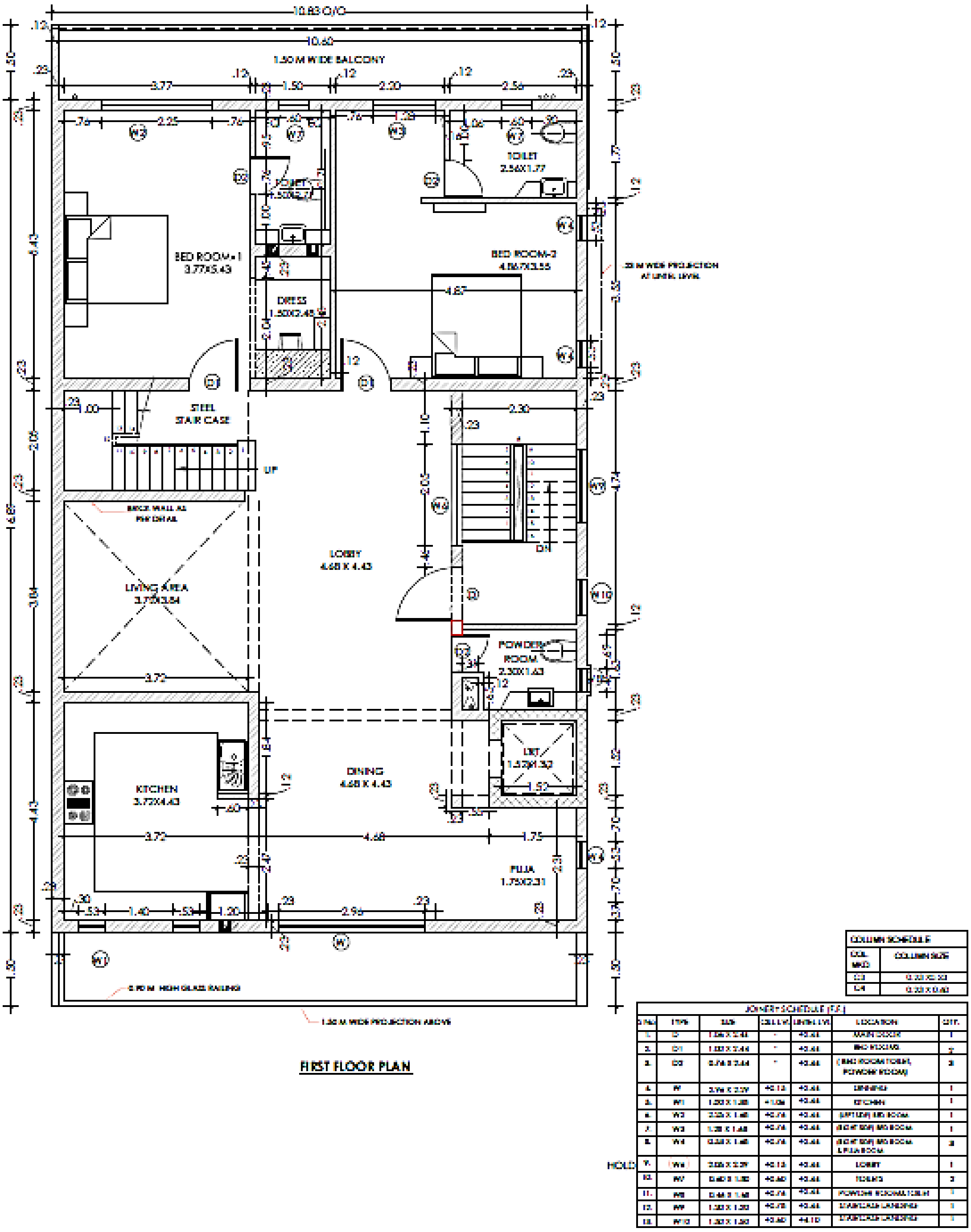
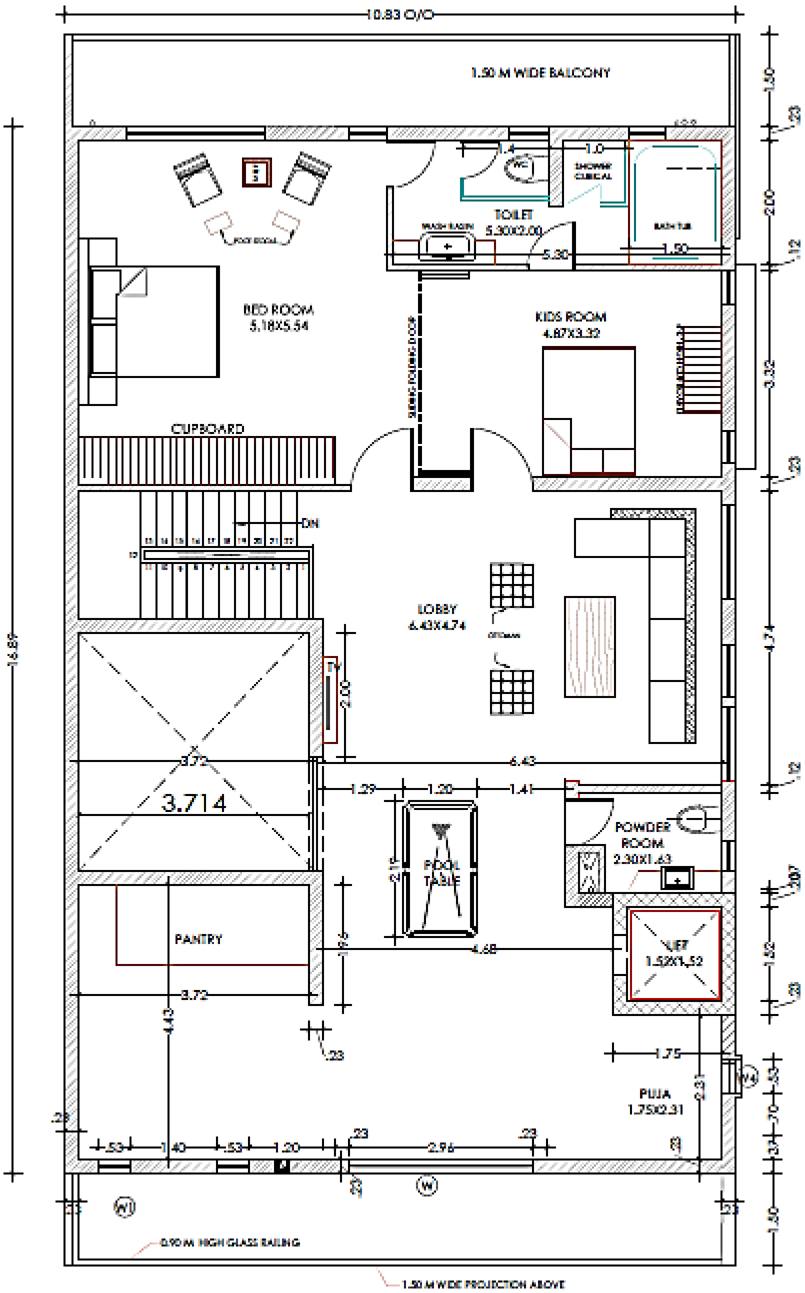
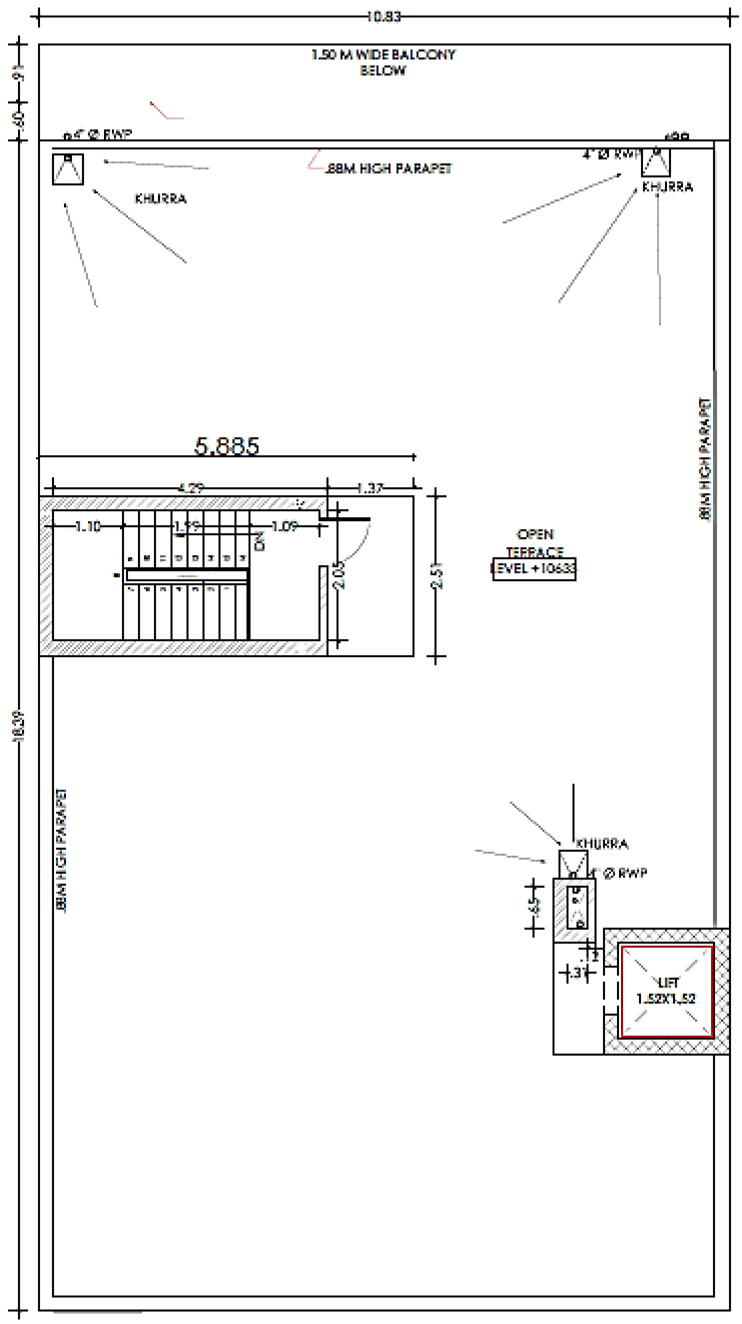


Fig 6.2 :- first floor



SECOND FLOOR PLAN

Fig 6.3:- Second floor



TERRACE PLAN

Fig 6.4:- Terrace floor

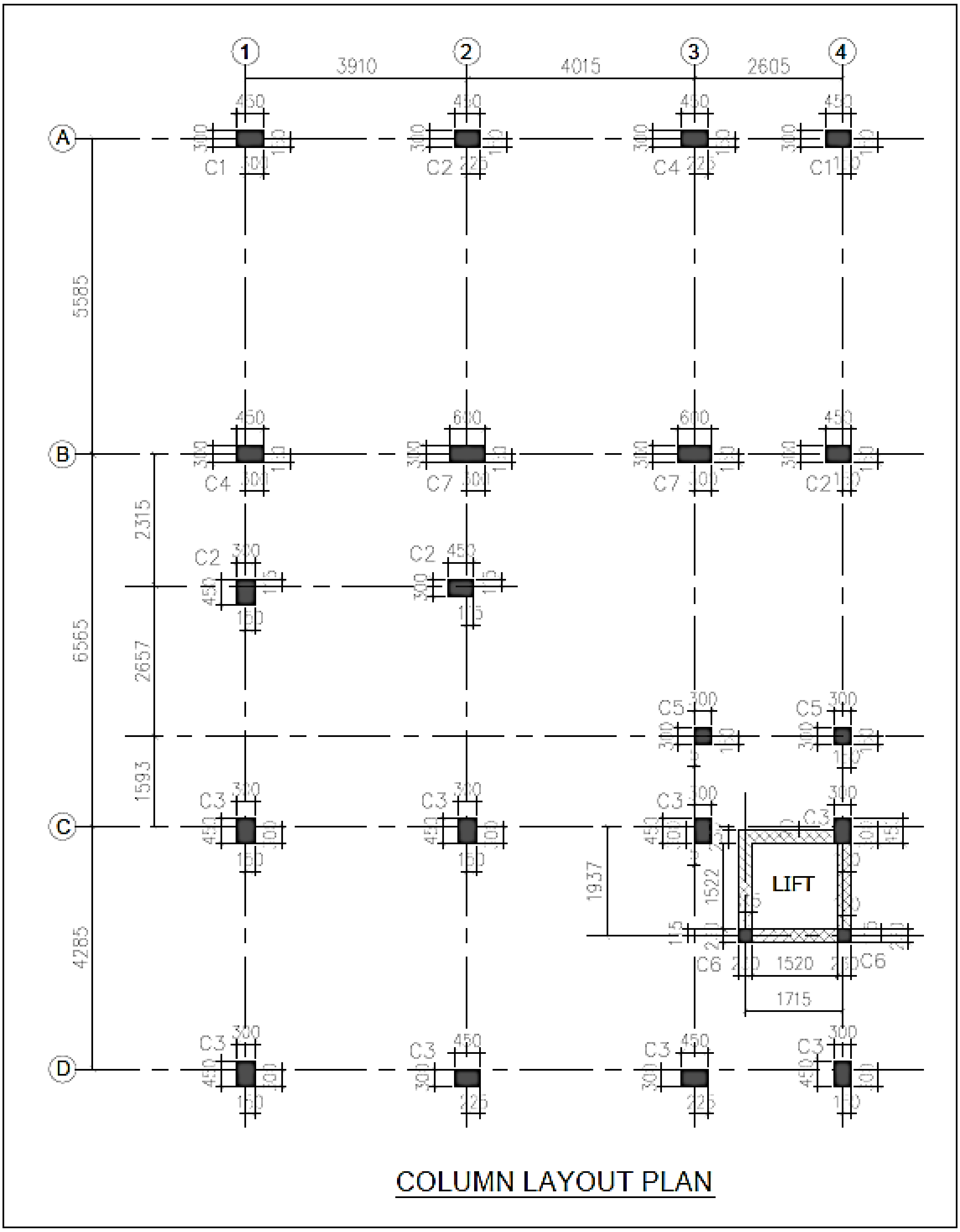
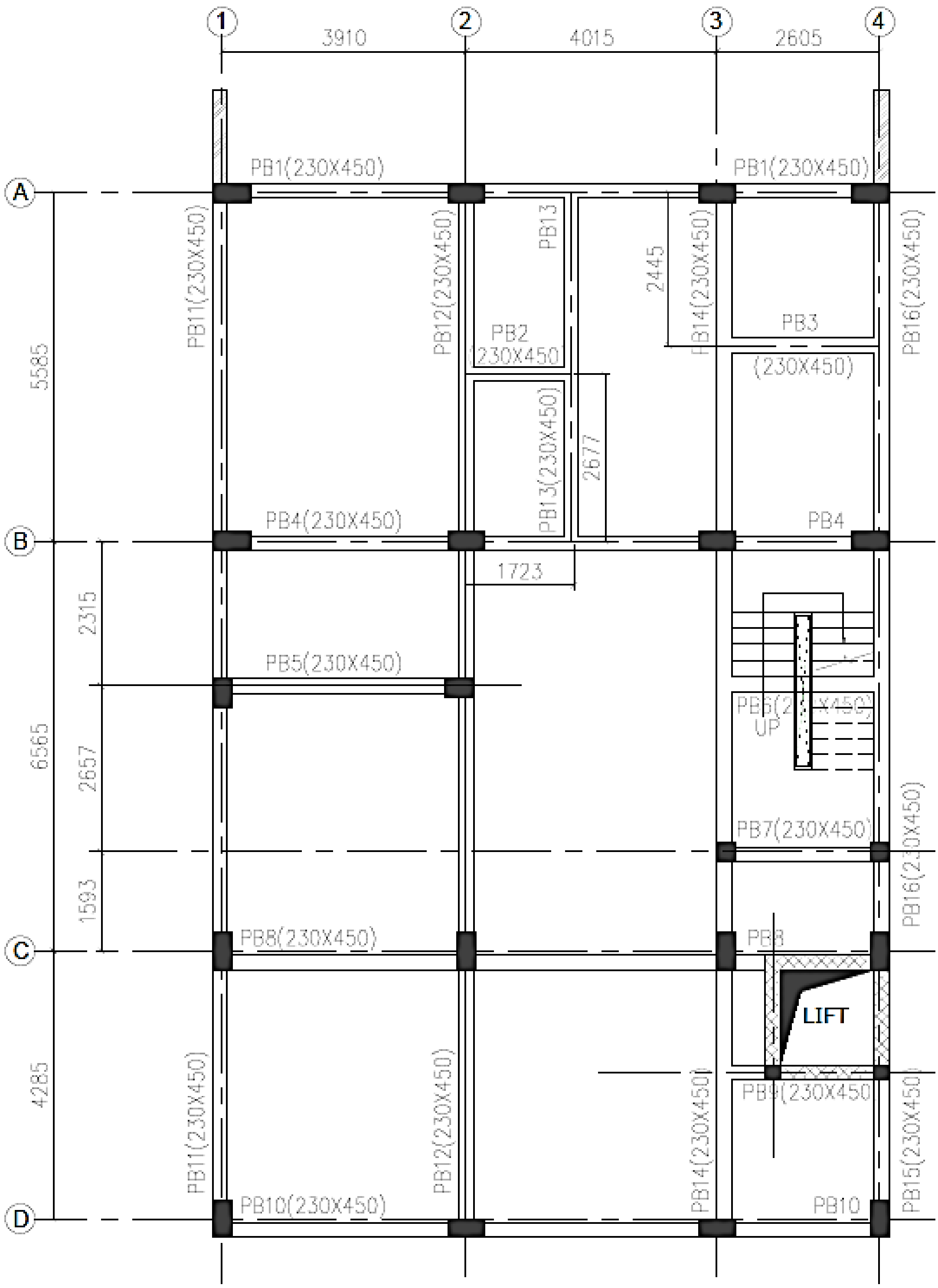
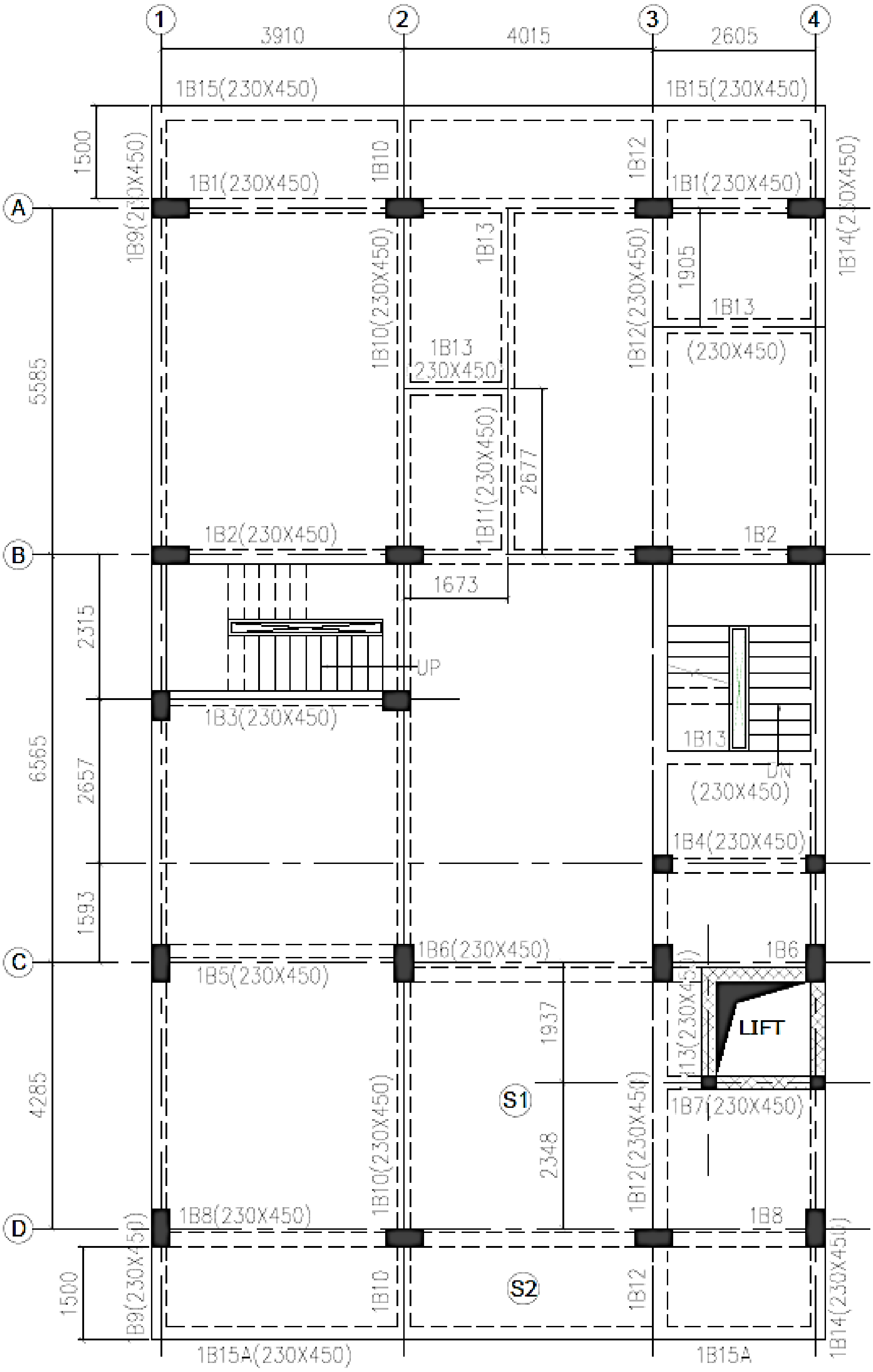


Fig 6.5:- Column layout plan



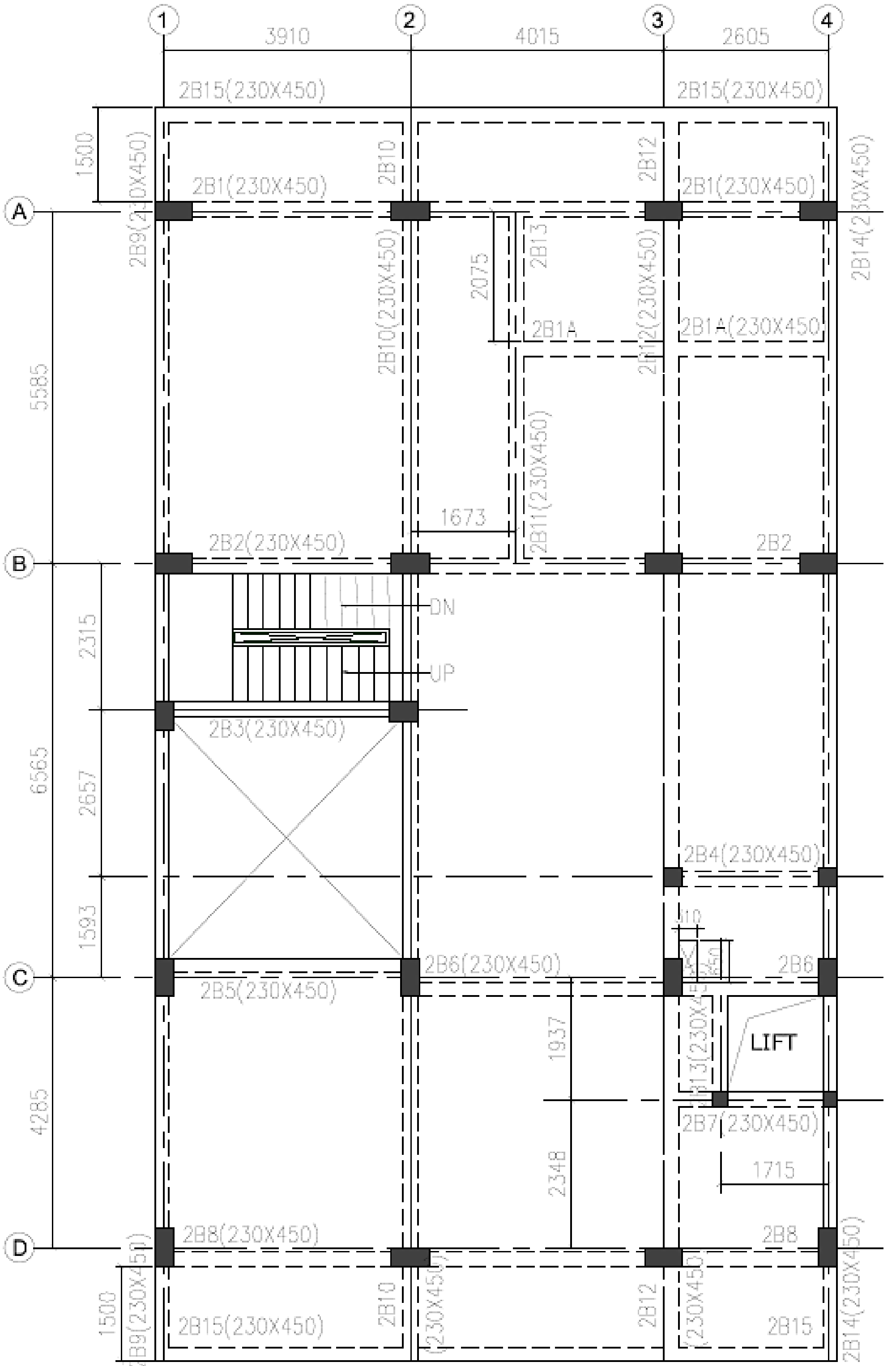
PLINTH LVL. BEAM FRAMING PLAN

Fig 6.6:- Plinth Beam layout plan



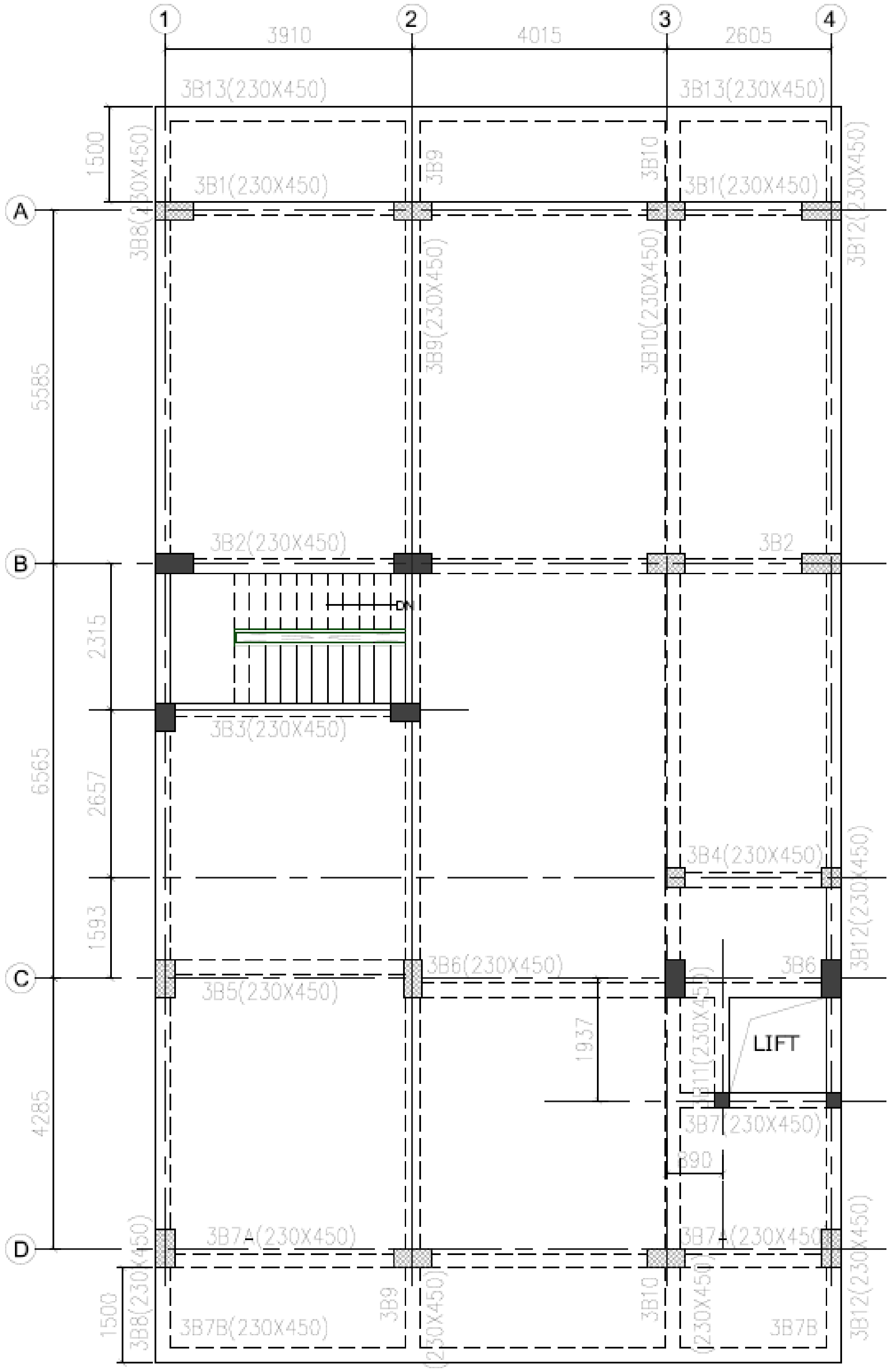
STILT FLOOR ROOF LVL. BEAM FRAMING PLAN

Fig 6.7:- Stil floor Beam layout.



1st FLOOR ROOF LVL. BEAM FRAMING PLAN

Fig 6.8:- 1st Floor roof lvl. beam framing plan



TERRACE FLOOR LVL. BEAM FRAMING PLAN

Fig 6.9:- Terrace Floor lvl. beam framing plan

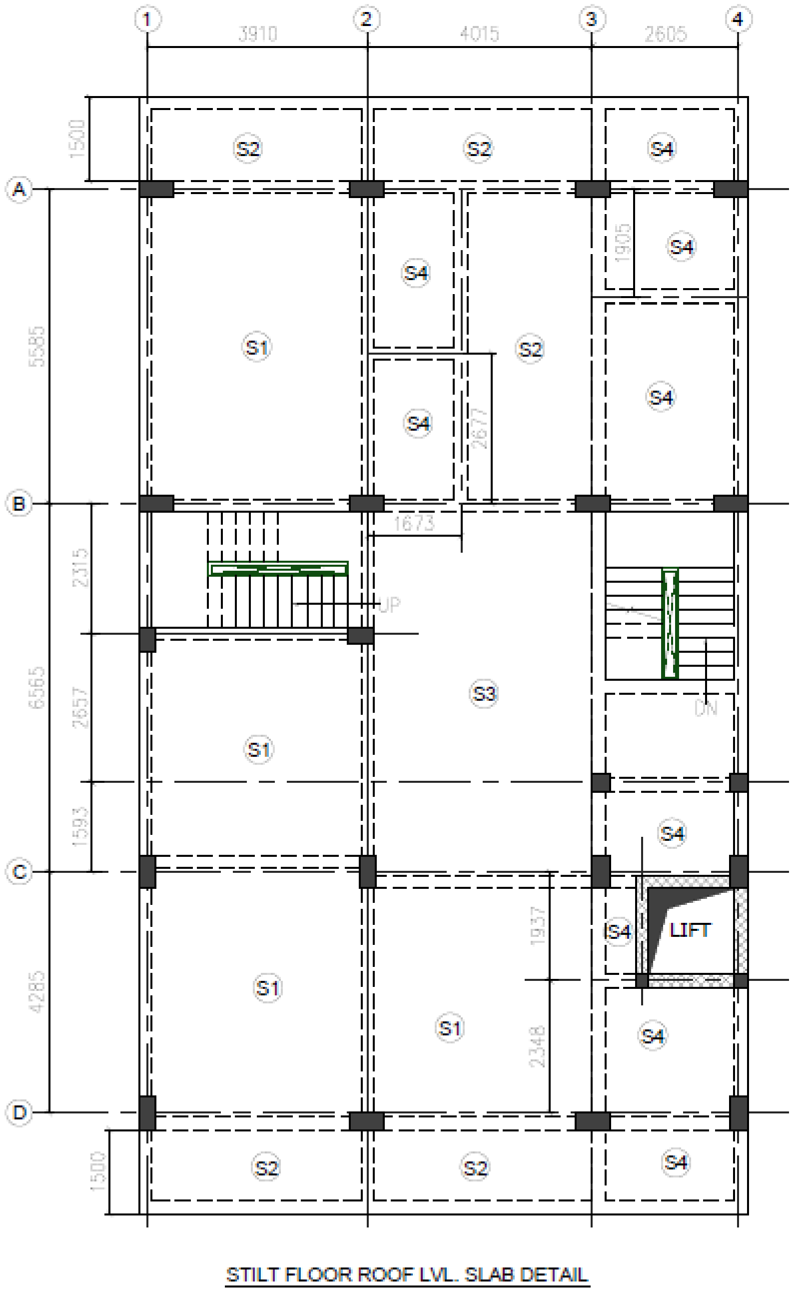
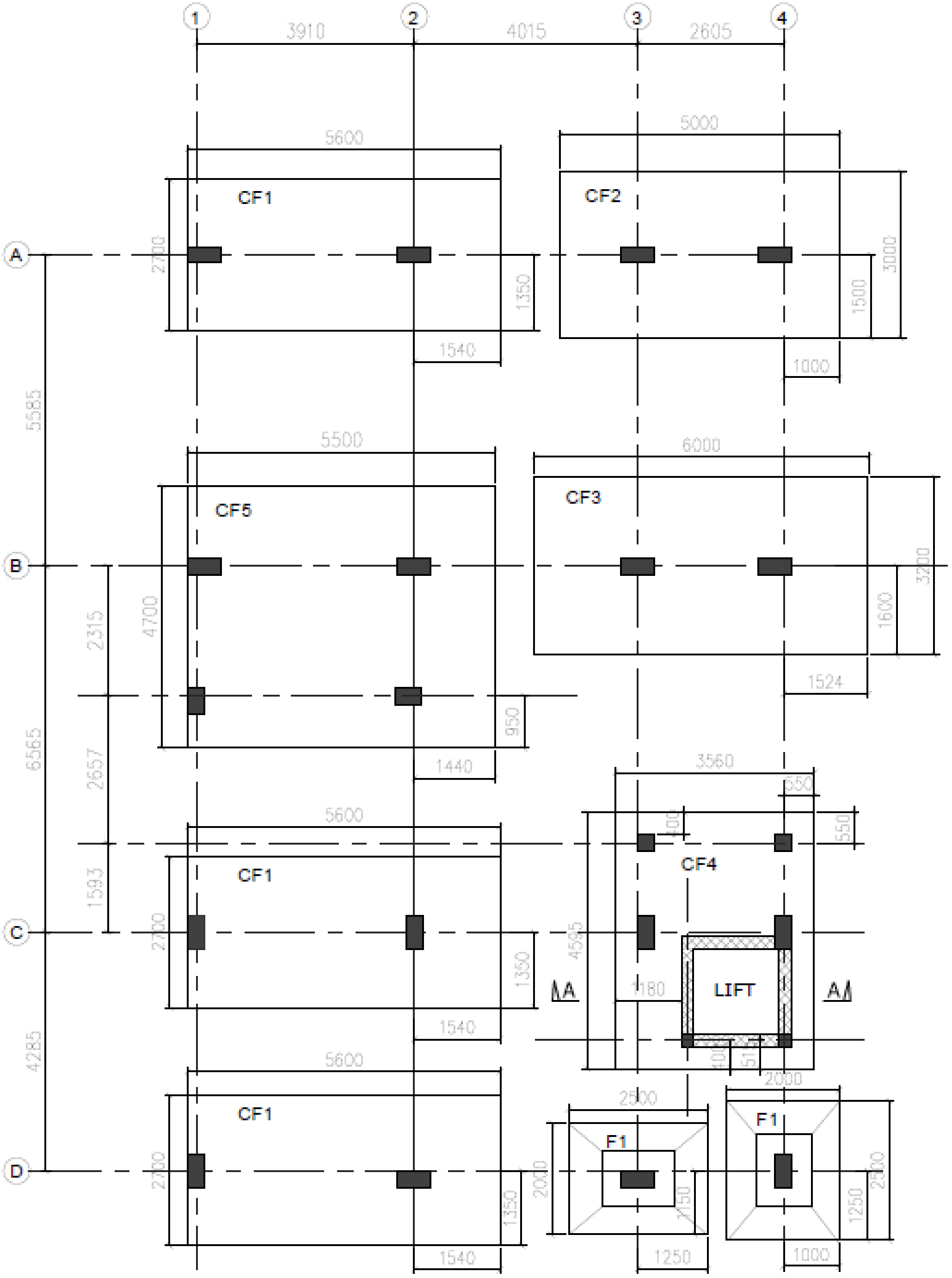


Fig 6.10:- Terrace Floor lvl. Beam framing plan



FOUNDATION LAYOUT PLAN

Fig 6.11:- Foundation layout plan

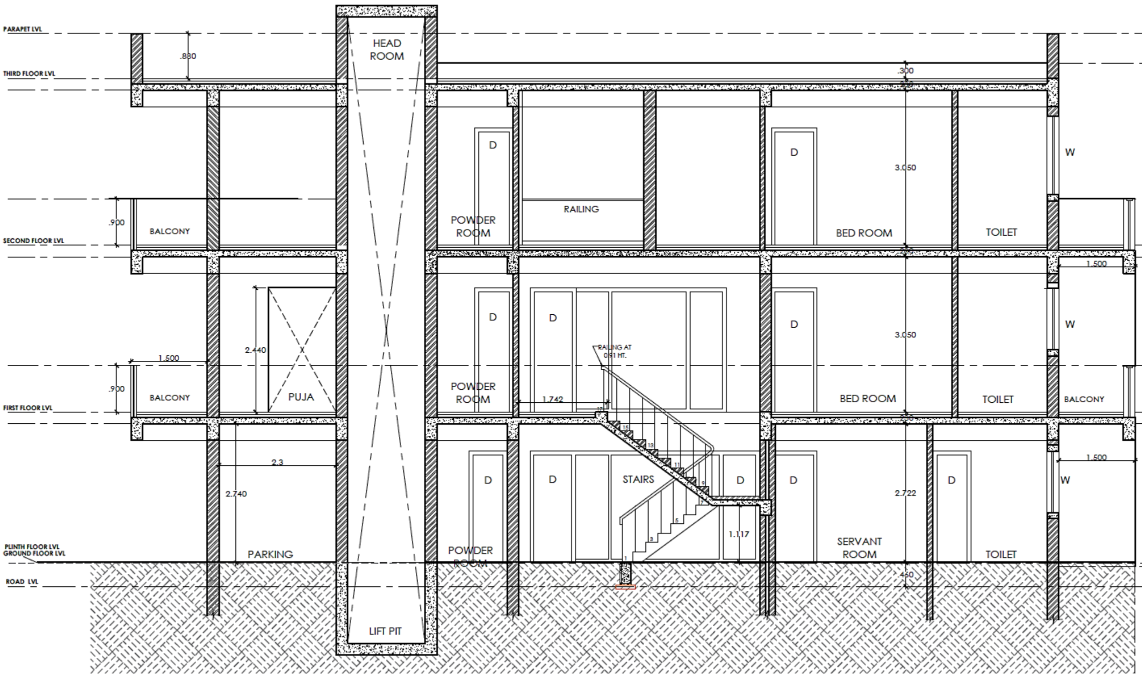


Fig 6.12:- Section elevation

VII. LOADING PARAMETERS

7.1 Design Data:

ETABS software analysed and designed a three-story reinforced concrete asymmetrical frame structure. Residential building. Assuming infinite rigidity in its plane, the building floor moves like a rigid body. Linear static analysis was done. Seismic zone II and medium soil host the building.

7.2 Building details:

Length	= 19.89m
Width	= 10.83m
No. of Storey	= G+3
Storey Height	= 3.0 m
Total No. of Column	= 22
Rectangular Column size	= 230*450mm, 230*230mm, 300*300mm
Beam size	= 230*450
Slab thickness	= 125.00 mm
wall thickness	= 230.00 mm
SBC of soil	= 180 KN/m^2
Grade of concrete	= M20
Steel grade	= Fe500
Density of Concrete	= 25.00 kN/m^3
Density of Brick	= 18.85 kN/m^3

Table 7.1:- load calculation

Si No.	Descriptions	Width/ Thickness	Density	Height	Weight	Remarks
1	**DEAD LOAD - IS 875 (Part-1)**					
A	**Stilt floor lvl. (Plinth Level)**					
i)	**230mm Thick masonry load**					
	230mm thk, upto beam bottom	0.23	18.85	2.4	10.41	KN/m
	Wall Plaster	0.024	20.4	2.4	1.18	KN/m
			Total		11.58	**Say 12 KN/m**
ii)	**115mm Thick masonry load**					
	230mm thk, upto beam bottom	0.115	18.85	2.4	5.20	KN/m
	Wall Plaster	0.024	20.4	2.4	1.18	KN/m
			Total		6.38	**Say 6.5 KN/m**
B	**First floor lvl.**					
i)	**230mm Thick masonry load**					
	230mm thk, upto beam bottom	0.23	18.85	2.83	12.27	KN/m
	Wall Plaster	0.024	20.4	2.83	1.39	KN/m
			Total		13.66	**Say 14 KN/m**
ii)	**115mm Thick masonry load**					
	230mm thk, upto beam bottom	0.115	18.85	2.83	6.13	KN/m
	Wall Plaster	0.024	20.4	2.83	1.39	KN/m
			Total		7.52	**Say 8 KN/m**
iii)	**Parapet wall (balcony)**					
	115mm thk, 900mm height	0.115	18.85	0.9	1.95	KN/m
	Wall Plaster	0.024	20.4	0.9	0.44	KN/m
			Total		2.39	**Say 2.5 KN/m**
iv)	**Floor (Slab) load 125mm thk.**					
	125mm thk slab	0.125	25		3.13	KN/Sqm
	Flooring 105mm thk.					
	cement mortar 65mm thk	0.065	20.4		1.33	KN/Sqm
	40mm thk Kota stone	0.04	23.25		0.93	KN/Sqm
			Total		5.38	**Say 5.5 KN/Sqm**
C	**Second & Third floor lvl.**					
i)	**230mm Thick masonry load**					
	230mm thk, upto beam bottom	0.23	18.85	2.83	12.27	KN/m
	Wall Plaster	0.024	20.4	2.83	1.39	KN/m
			Total		13.66	**Say 14 KN/m**
ii)	**115mm Thick masonry load**					
	230mm thk, upto beam bottom	0.115	18.85	2.83	6.13	KN/m
	Wall Plaster	0.024	20.4	2.83	1.39	KN/m
			Total		7.52	**Say 8 KN/m**
iii)	**Floor (Slab) load 125mm thk.**					
	125mm thk slab	0.125	25		3.13	KN/Sqm
	Flooring 105mm thk.					
	cement mortar 65mm thk	0.065	20.4		1.33	KN/Sqm
	40mm thk Kota stone	0.04	23.25		0.93	KN/Sqm
			Total		5.38	**Say 5.5 KN/Sqm**
D	**Terrace floor lvl.**					
i)	**Mumty room**					
	230mm Thick masonry load					
	230mm thk, upto beam bottom	0.23	18.85	2.3	9.97	KN/m
	Wall Plaster	0.024	20.4	2.3	1.13	KN/m
			Total		11.10	**Say 11.5 KN/m**
ii)	**Parapet wall**					
	230mm Thick masonry load					
	230mm thk, upto beam bottom	0.23	18.85	0.9	3.90	KN/m
	Wall Plaster	0.024	20.4	0.9	0.44	KN/m
			Total		4.34	**Say 4.5 KN/m**
iii)	**Floor (Slab) load 125mm thk.**					
	125mm thk slab	0.125	25		3.13	KN/Sqm

	Waterproofing 150mm thick (brick coba)	0.15	20.4		3.06	KN/Sqm
			Total		6.19	**Say 6.5 KN/Sqm**
E	**Mumty/Machine floor**					
	Floor (Slab) load 125mm thk.					
i)	125mm thk slab	0.125	25		3.13	KN/Sqm
	Waterproofing 150mm thick (brick coba)	0.15	20.4		3.06	KN/Sqm
			Total		6.19	**Say 6.5 KN/Sqm**
2	**LIVE LOAD - IS 875 (Part-2)**					
A	**First floor**					
i)	All rooms, Kitchen, toilet and bathrooms				2	KN/Sqm
ii)	Corridor, passages, staircases including fire escape and store rooms				3	KN/Sqm
B	**Second floor**					
	Floor Slab (Typical floors)					
i)	All rooms, Kitchen, toilet and bathrooms				2	KN/Sqm
			Total		2.0	**Say 2 KN/Sqm**
ii)	Corridor, passages, staircases including fire escape and store rooms				3	KN/Sqm
			Total		3.0	**Say 3 KN/Sqm**
C	**Terrace floor**					
i)	Terrace				1.5	KN/Sqm
			Total		1.5	**Say 1.5 KN/Sqm**
D	**Mumty floor**					
i)	Water tank load		10	1.2	12.00	KN/Sqm
			Total		12.00	**Say 12 KN/Sqm**
E	**Machine room**					
i)	Live load				7.5-10	KN/Sqm
			Total		10.00	**Say 10 KN/Sqm**

7.3 Staircase Loading.
Loading Per Meter Width of Flight

R=150, T=250, Hence Sqrt ((150)^2+(250)^2)) = 291.5 ≅ 292mm
Waist Slab = [0.150 x 292 x 25]/250
= 4.38 KN/Sqm
Steps = (0.150 x 25)/2
= 1.875 KN/Sqm
Finishing = 1 KN/Sqm
Total = 7.3 KN/Sqm ≅ 7.5 KN/Sqm

7.4 Earthquake/Seismic Loads
Assessment of earthquake loading is performed in accordance with the requirements of IS: 1893-2016. You'll find Bangalore in Zone-III. The following considerations are made:
Seismic loads and how they are distributed across the building's height and throughout its numerous lateral load resisting parts as per the design.

Table 7.2 Seismic Load Calculations

BASIC LOAD CALCULATIONS						
Project : Live Project						
Location : Bangalore						
	DESCRIPTIONS	**CALCULATIONS**				**REMARKS**
1	**SEISMIC LOAD - IS 1893 (Part-1) : 2016**					
A	Zone Factor		0.16			Table -3
	Importance factor		1			Table -8
	Response Reduction Factor		5			Table -9
B	**Time Period Calculation**	**T= (0.09xh)/ Sqrt (d)**				**clause 7.6.2 ©**
	Height of the building =		16.902	m		
	Base dimension of building at plinth level (d_x) =		19.89	m		
	Base dimension of building at plinth level (d_z) =		10.83	m		
	Time period in X Direction (T_x) =		0.34	sec		
	Time period in Z Direction (T_z) =		0.46	sec		

Table: 7.2 Covers For Structural Elements

Structural Element	Nominal Cover to all reinforcement (as per IS 456:2000)		Nominal Cover Provided
	For Moderate Exposure condition	For Fire Resistance of 2.0 hrs.	
Beams	25mm	30mm	30mm
Slab	20mm	25mm	25mm
Columns	40mm	40mm	40mm
Footings	50mm		50m

Clause 21 of IS 456-2000 specifies minimum sizes for fire-resistant reinforced concrete members.

Table: 7.3 MINIMUM DIMENSIONS OF REINFORCED CONCRETE

Fire Resistance *h*	Minimum Beam Width *b*	Rib Width of Slabs b_w	Minimum Thickness of Floors *D*	Column Dimension (*b* or *D*): Fully Exposed	Column Dimension: 50% Exposed	Column Dimension: One Face Exposed	Minimum Wall Thickness: $p<0.4\%$	Minimum Wall Thickness: $0.4\%\le p\le 1\%$	Minimum Wall Thickness: $p>1\%$
	mm	mm	mm	mm	mm	mm	mm	mm	mm
0.5	200	125	75	150	125	100	150	100	100
1	200	125	95	200	160	120	150	120	100
1.5	200	125	110	250	200	140	175	140	100
2	200	125	125	300	200	160	-	160	100
3	240	150	150	400	300	200	-	200	150
4	280	175	170	450	350	240	-	240	180

NOTES

1 These minimum dimensions relate specifically to the covers given in Table 16A.

2 *p* is the percentage of steel reinforcement.

According to clause 21 of IS 456 -2000 (table above), minimum beam width for 30 minutes fire resistance is 200 mm and minimum column cover to reinforcement is 40mm. IS 13920-1993 requires 200mm beam width. Clause 7.1.2 requires 200mm columns.

7.5 FOUNDATION SYSTEM

Building footings will be segregated. The foundation will be constructed for 180 KN/m2 Net Safe Bearing Capacity using ETAB Model reaction.

7.6 DETAILING OF STRUCTURES

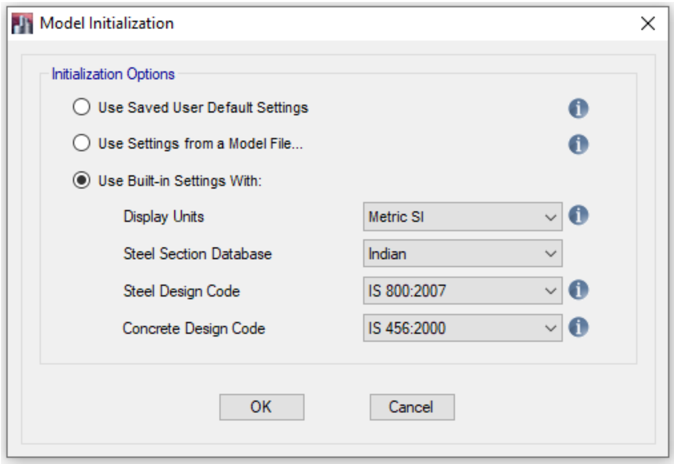
IS: 456 and SP: 34 detail all structural elements. Detailing should fulfil minimum and maximum R/F and bar spacing criteria.

VIII. MODELING USING ETABS

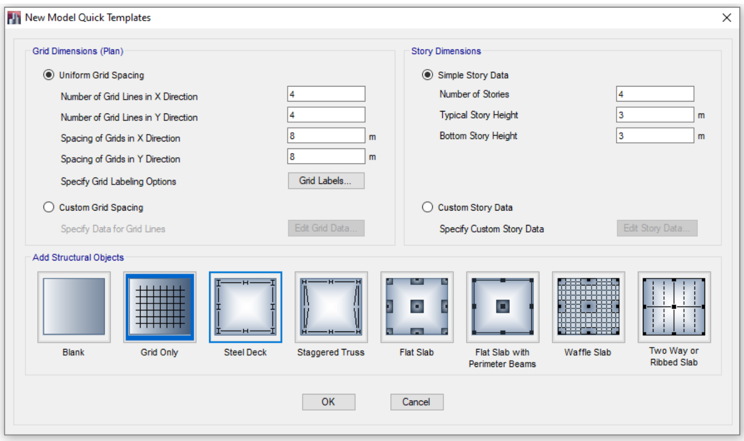
Step 1: Modeling using ETABS

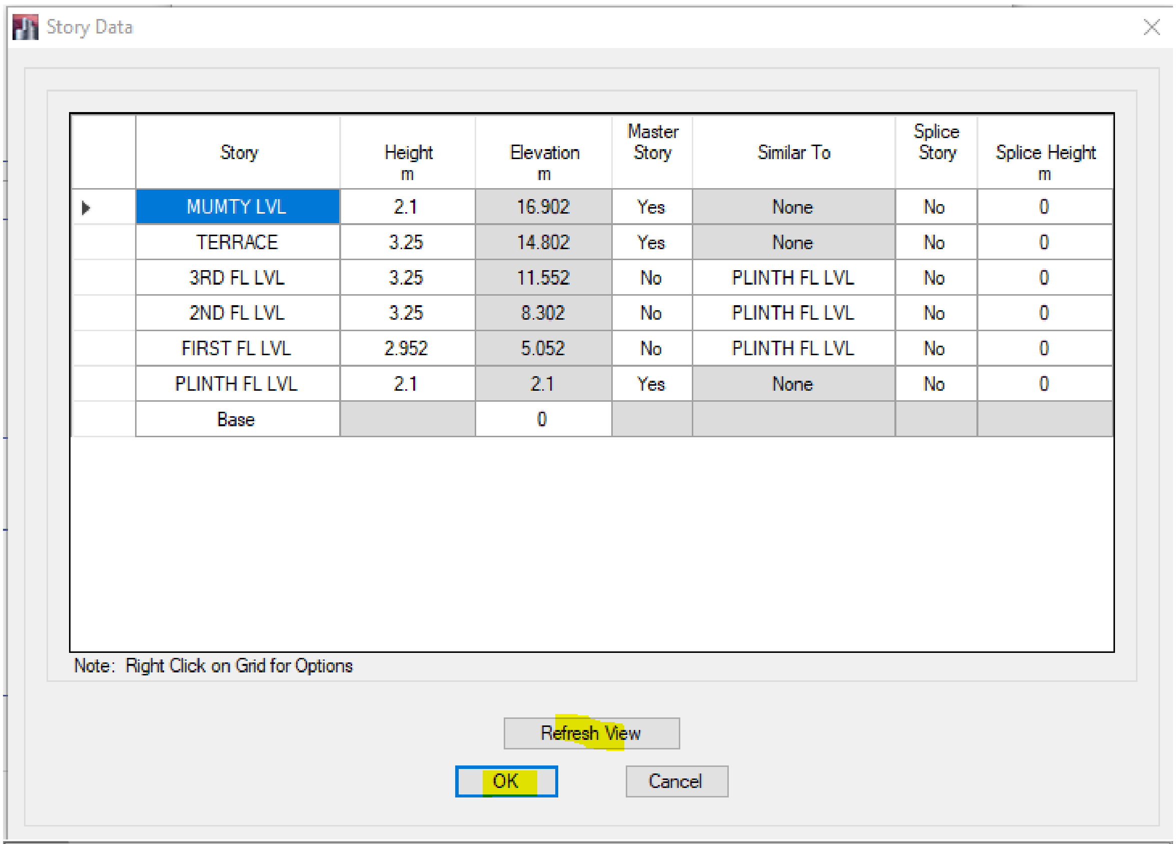
1) Start ETABS
2) In the drop-down box at the very right of ETABS, choose kN-m as the model units.
3) Select File>New model.

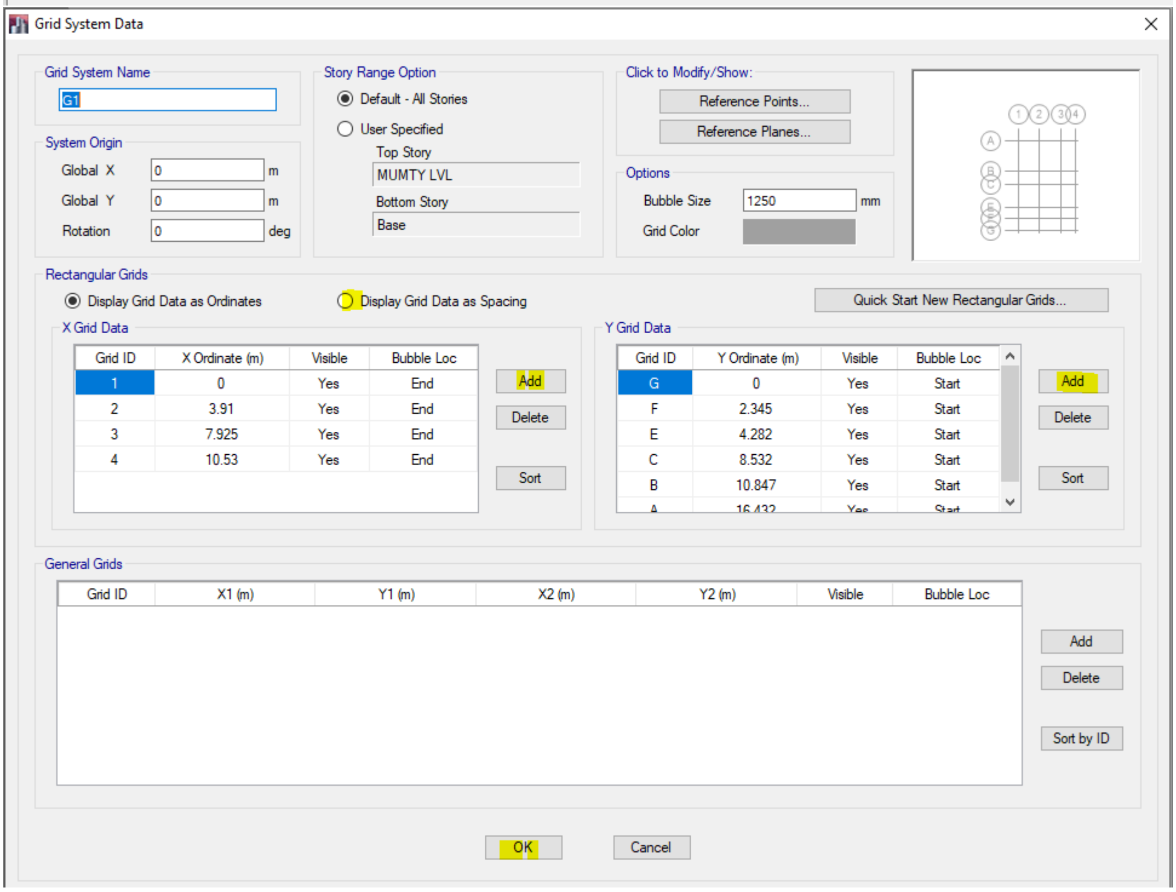
Change units, steel, and concrete codes and click OK.



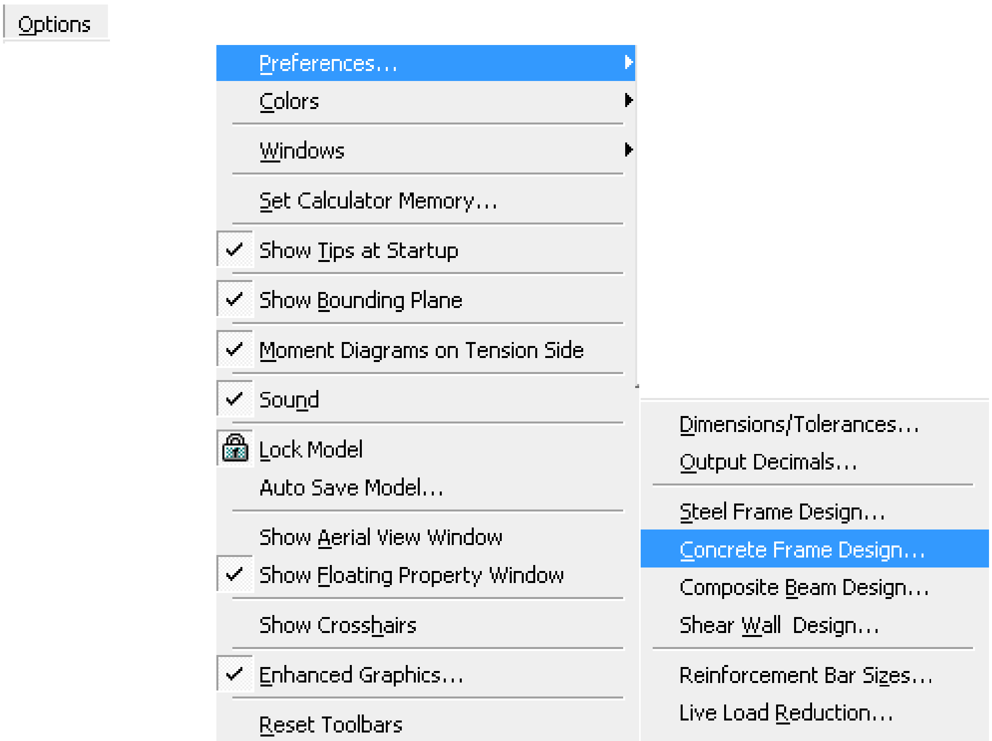
4) The next step is to define the building's data and the building's grid system. Alter the distance between each grid square. Grid spacing and custom storey data are used to determine building height. As seen in drawings.



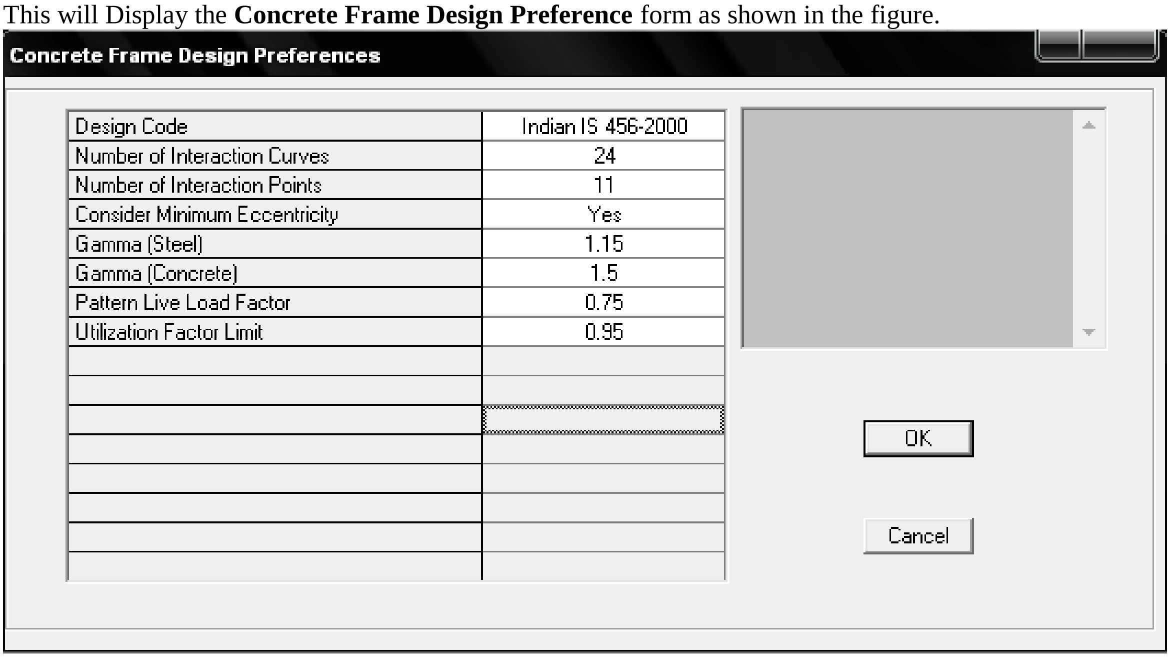

Story Data
Story | Height m | Elevation m | Master Story | Similar To | Splice Story | Splice Height m
MUMTY LVL | 2.1 | 16.902 | Yes | None | No | 0
TERRACE | 3.25 | 14.802 | Yes | None | No | 0
3RD FL LVL | 3.25 | 11.552 | No | PLINTH FL LVL | No | 0
2ND FL LVL | 3.25 | 8.302 | No | PLINTH FL LVL | No | 0
FIRST FL LVL | 2.952 | 5.052 | No | PLINTH FL LVL | No | 0
PLINTH FL LVL | 2.1 | 2.1 | Yes | None | No | 0
Base | | 0
Note: Right Click on Grid for Options
Refresh View
OK
Cancel


Grid System Data
Grid System Name
G1
Story Range Option
Default - All Stories
User Specified
Top Story
MUMTY LVL
Bottom Story
Base
Click to Modify/Show:
Reference Points...
Reference Planes...
Options
Bubble Size 1250 mm
Grid Color
System Origin
Global X 0 m
Global Y 0 m
Rotation 0 deg
Rectangular Grids
Display Grid Data as Ordinates
Display Grid Data as Spacing
Quick Start New Rectangular Grids...
X Grid Data
Grid ID | X Ordinate (m) | Visible | Bubble Loc
1 | 0 | Yes | End
2 | 3.91 | Yes | End
3 | 7.925 | Yes | End
4 | 10.53 | Yes | End
Add
Delete
Sort
Y Grid Data
Grid ID | Y Ordinate (m) | Visible | Bubble Loc
G | 0 | Yes | Start
F | 2.345 | Yes | Start
E | 4.282 | Yes | Start
C | 8.532 | Yes | Start
B | 10.847 | Yes | Start
Add
Delete
Sort
General Grids
Grid ID | X1 (m) | Y1 (m) | X2 (m) | Y2 (m) | Visible | Bubble Loc
Add
Delete
Sort by ID
OK
Cancel

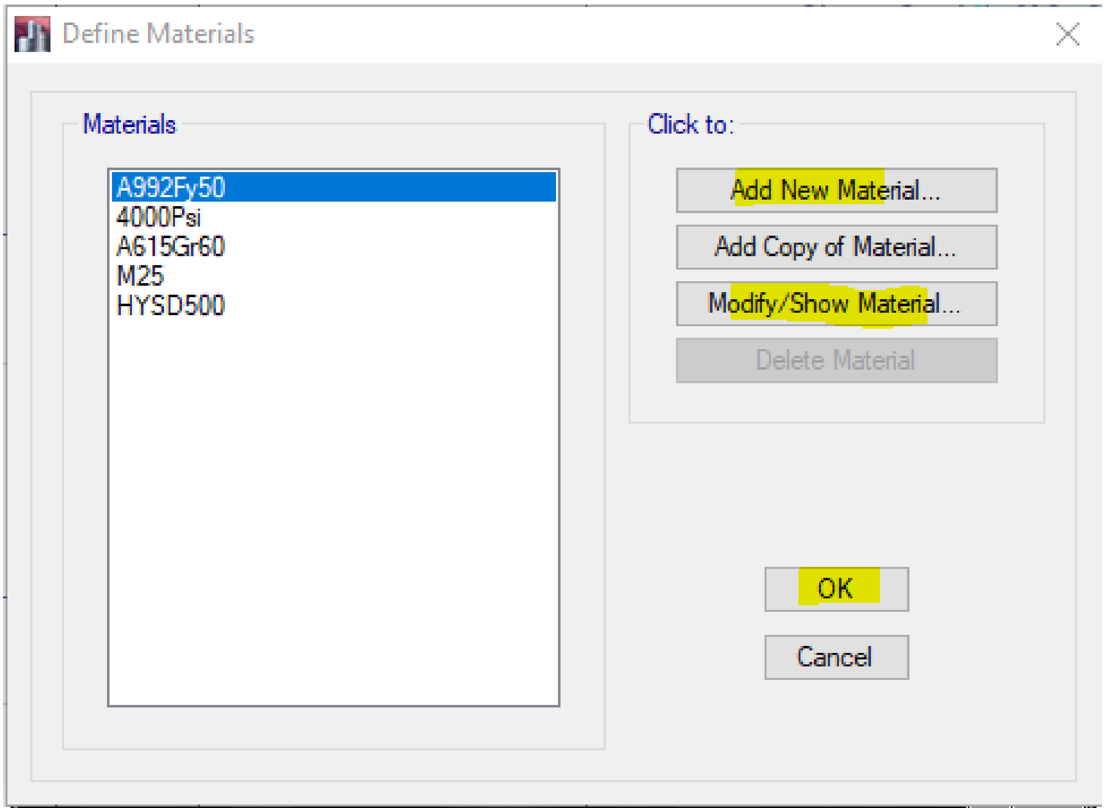
5) Choose Concrete Frame Design from the "Options" menu's "Preferences" to set the design code.
Command

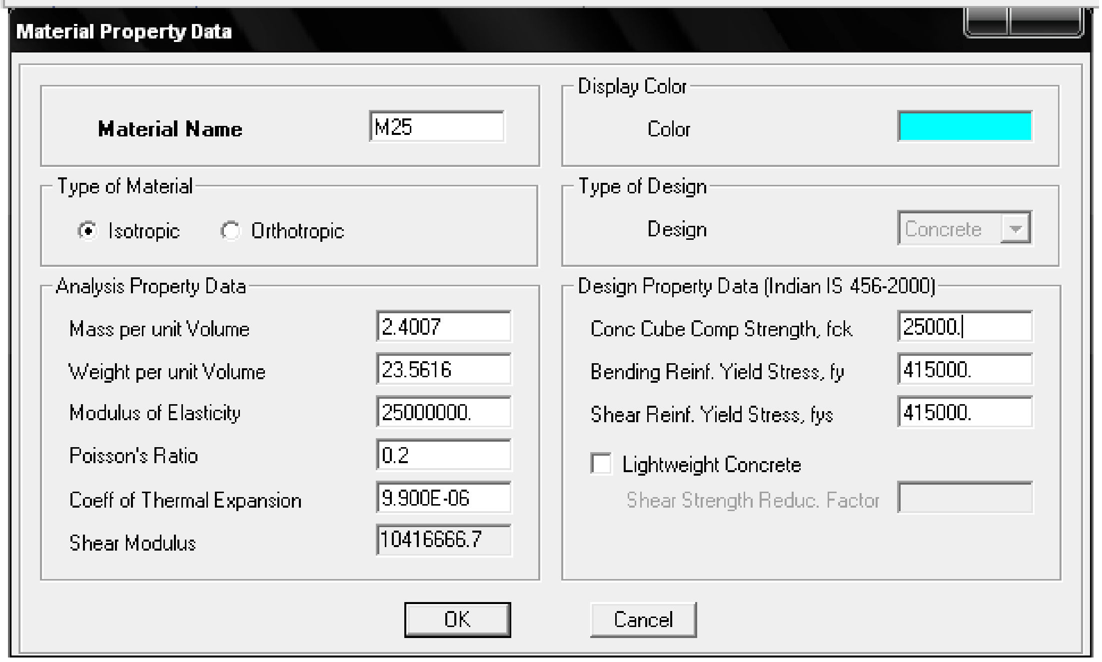


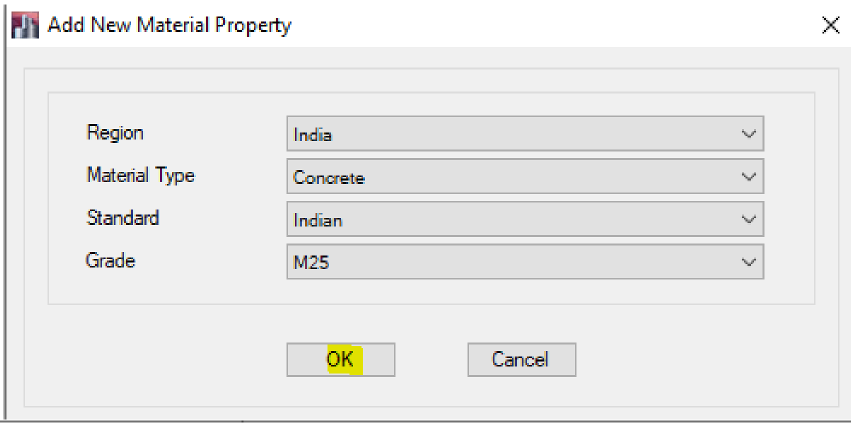
This will Display the **Concrete Frame Design Preference** form as shown in the figure.

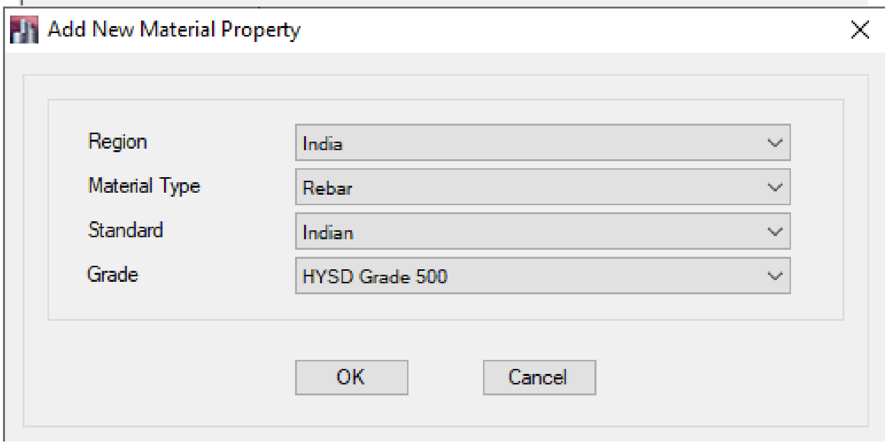


6) Click Define > Material Properties, then click Modify/Show Materials, and then click OK to add materials like concrete grade and steel grade by area. The numbers are below.

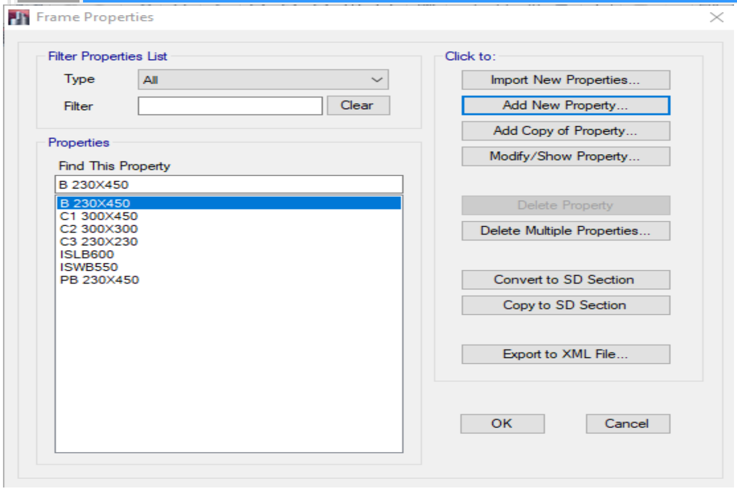

Define Materials
Materials
A992Fy50
4000Psi
A615Gr60
M25
HYSD500
Click to:
Add New Material...
Add Copy of Material...
Modify/Show Material...
Delete Material
OK
Cancel

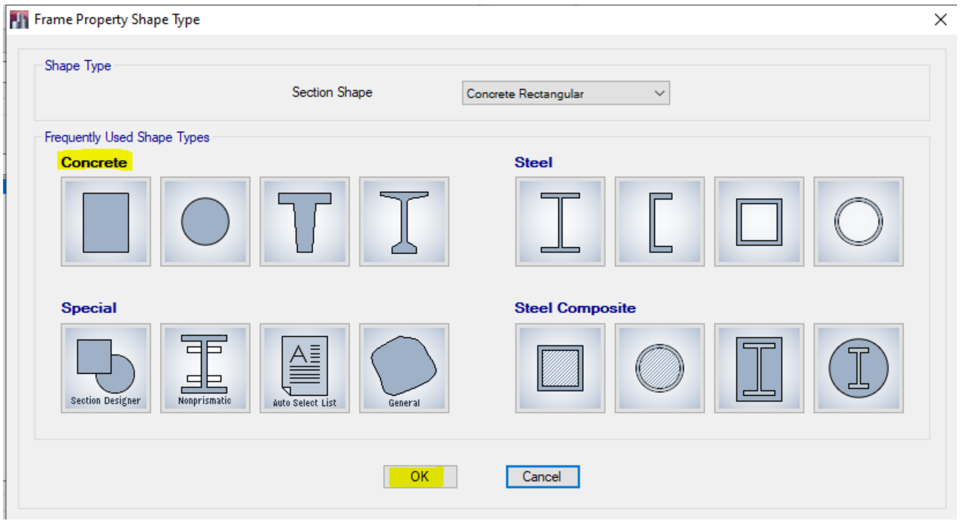

Material Property Data
Material Name
M25
Display Color
Color
Type of Material
Isotropic
Orthotropic
Type of Design
Design
Concrete
Analysis Property Data
Mass per unit Volume
2.4007
Weight per unit Volume
23.5616
Modulus of Elasticity
25000000.
Poisson's Ratio
0.2
Coeff of Thermal Expansion
9.900E-06
Shear Modulus
10416666.7
Design Property Data (Indian IS 456-2000)
Conc Cube Comp Strength, fck
25000.
Bending Reinf. Yield Stress, fy
415000.
Shear Reinf. Yield Stress, fys
415000.
Lightweight Concrete
Shear Strength Reduc. Factor
OK
Cancel



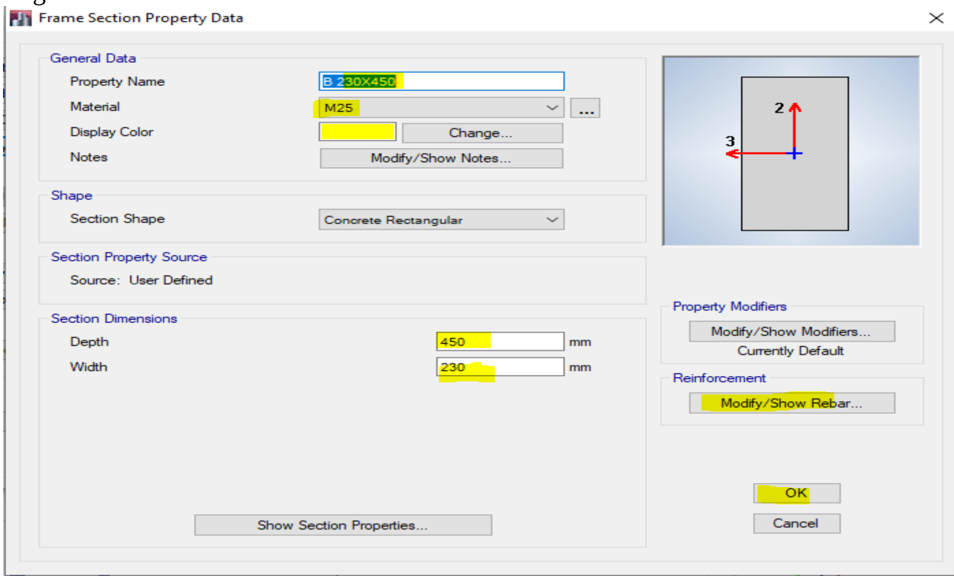


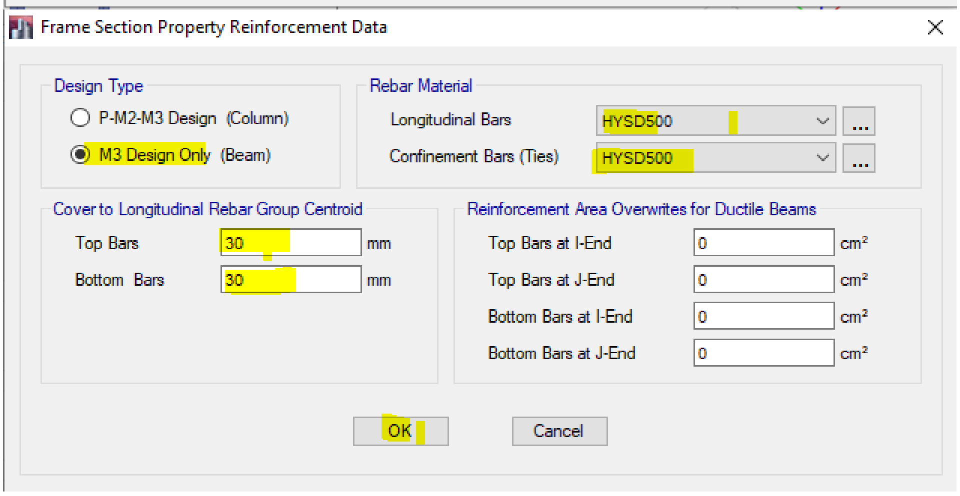
7) Columns and beams may be defined by selecting Define > Frame section.
Click add new property > click rectangle > ok



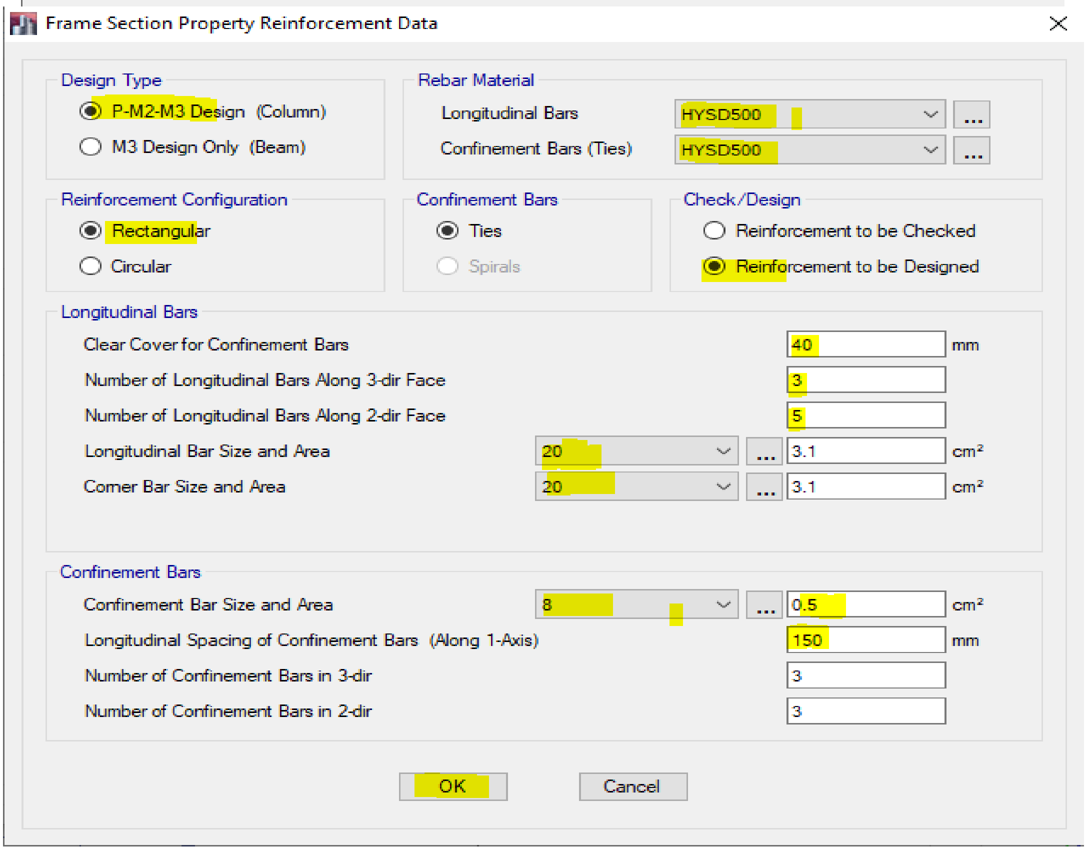


Define beam sizes, click Reinforcement, and cover columns with concrete. Checked or planned reinforcement. Prefer checking reinforcement.





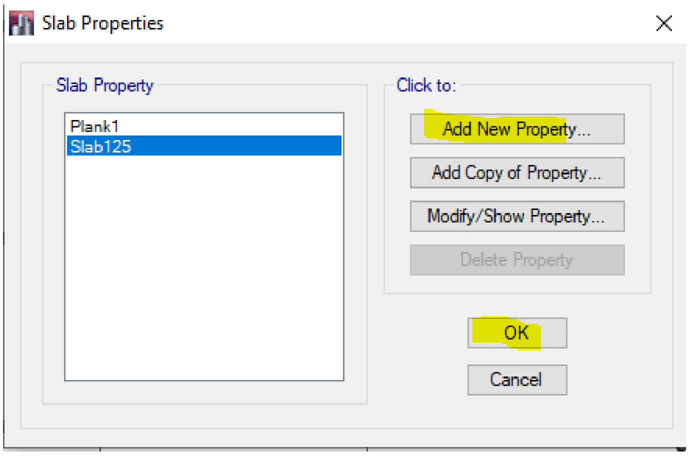

Frame Section Property Data
General Data
Property Name
C1 300X450
Material
M25
Display Color
Change...
Notes
Modify/Show Notes...
Shape
Section Shape
Concrete Rectangular
Section Property Source
Source: User Defined
Section Dimensions
Depth
450
mm
Width
300
mm
Show Section Properties...
2
3
Property Modifiers
Modify/Show Modifiers...
Currently Default
Reinforcement
Modify/Show Rebar...
OK
Cancel

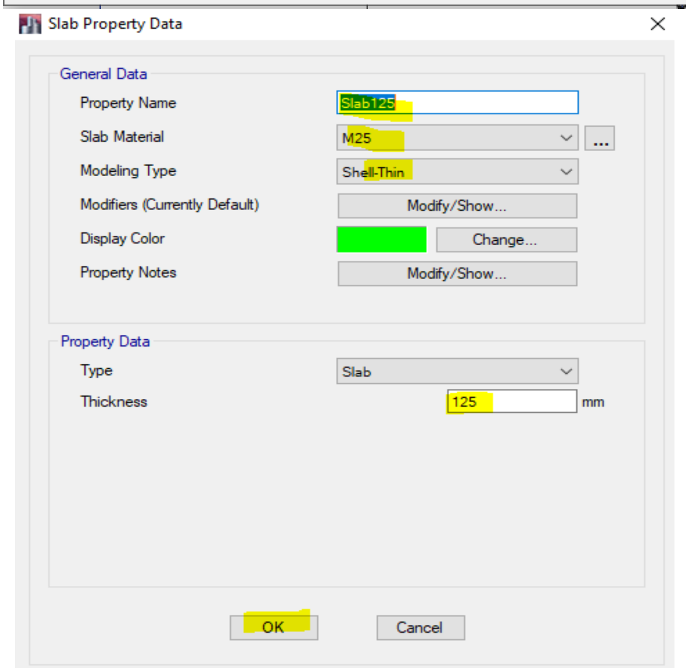

Frame Section Property Reinforcement Data
Design Type
P-M2-M3 Design (Column)
M3 Design Only (Beam)
Rebar Material
Longitudinal Bars
HYSD500
Confinement Bars (Ties)
HYSD500
Reinforcement Configuration
Rectangular
Circular
Confinement Bars
Ties
Spirals
Check/Design
Reinforcement to be Checked
Reinforcement to be Designed
Longitudinal Bars
Clear Cover for Confinement Bars
40
mm
Number of Longitudinal Bars Along 3-dir Face
3
Number of Longitudinal Bars Along 2-dir Face
5
Longitudinal Bar Size and Area
20
3.1
cm²
Corner Bar Size and Area
20
3.1
cm²
Confinement Bars
Confinement Bar Size and Area
8
0.5
cm²
Longitudinal Spacing of Confinement Bars (Along 1-Axis)
150
mm
Number of Confinement Bars in 3-dir
3
Number of Confinement Bars in 2-dir
3
OK
Cancel

8) Define wall/slab/deck.

Special one-way load distribution is used to describe a slab as a membrane or shell thin element and a one-way slab.

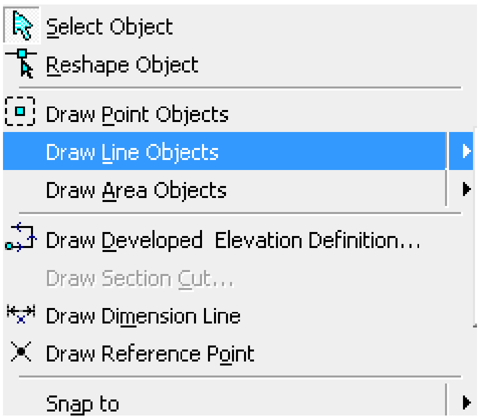
Slab-based features may be added or altered.

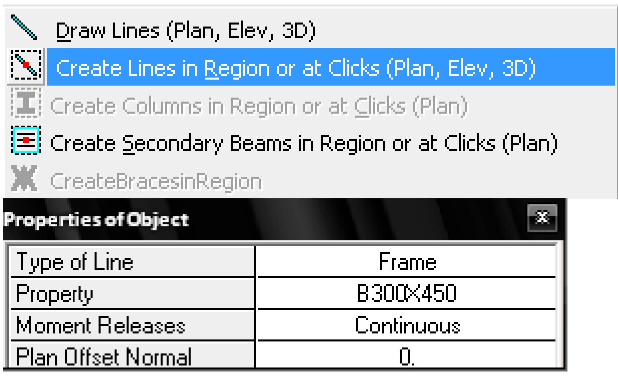




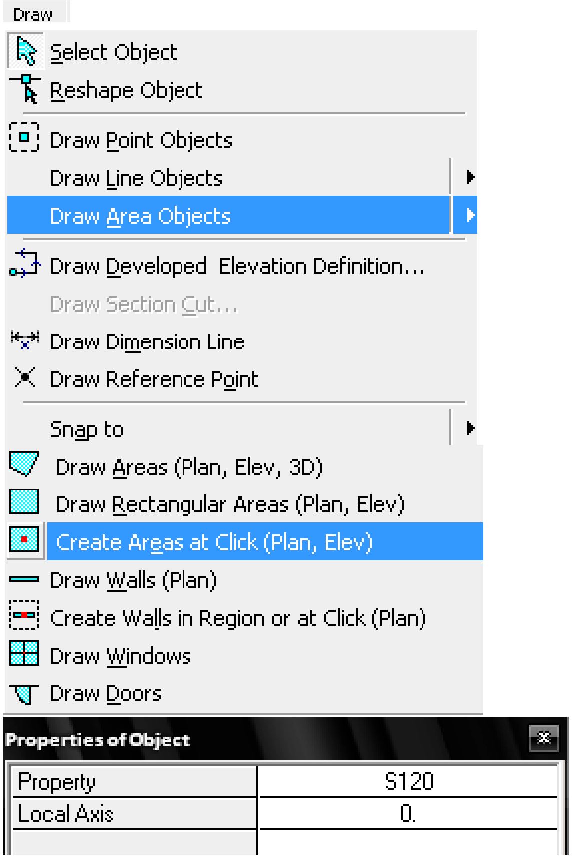
9) Generate the model

Make instructions for a line and a column to sketch a beam and a column.

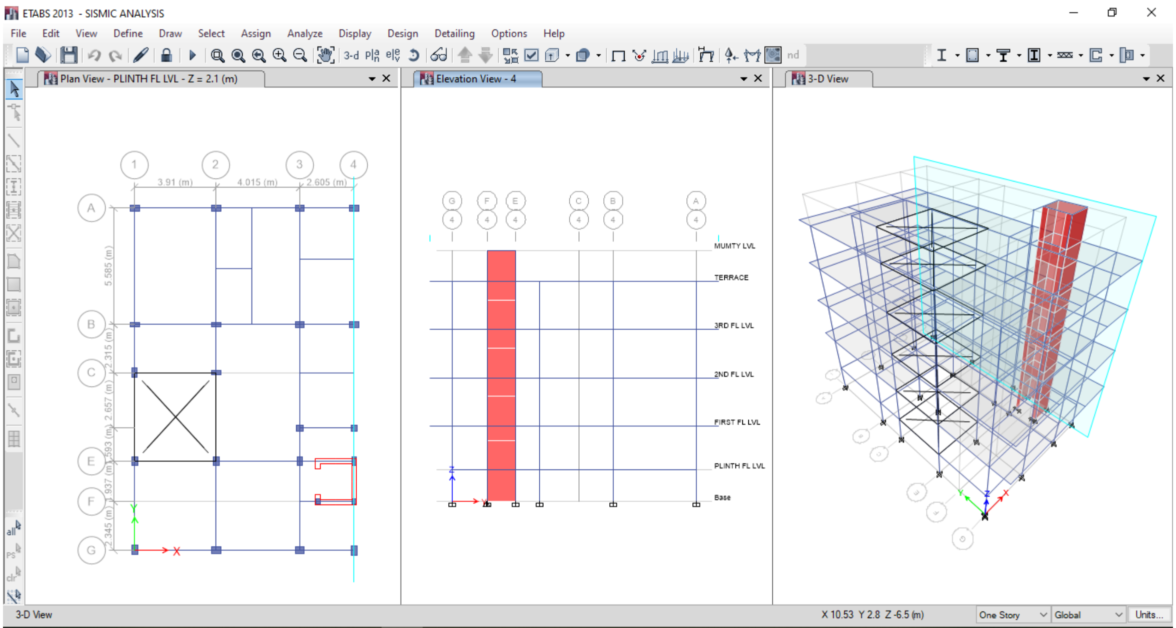




There are three ways to make a slab: (1) draw an area of any form; (2) draw a rectangular area; or (3) make an area in between grid lines.



The above choice created the model shown below.



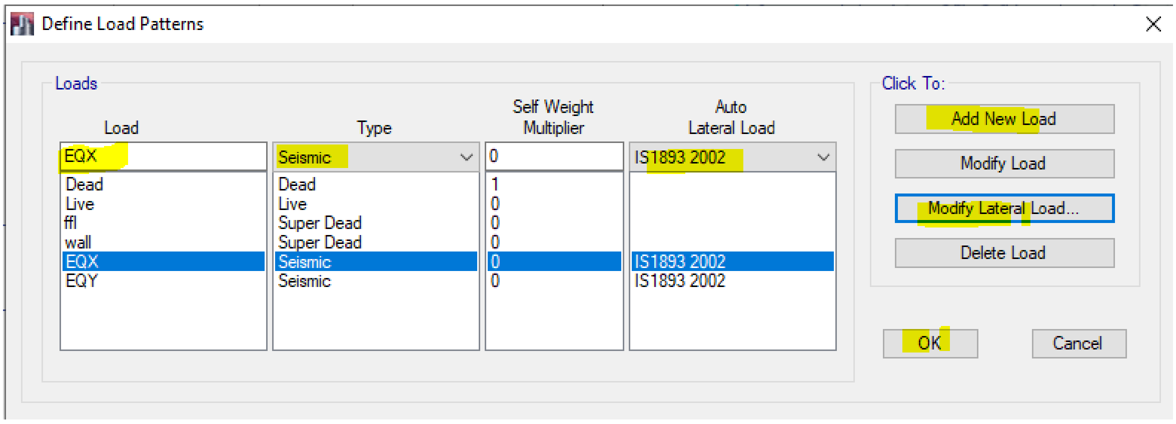
Go to define > load pattern > click

11) Seismic force calculation as per IS: 1893(Part 1) - 2002.

(a) Static Method

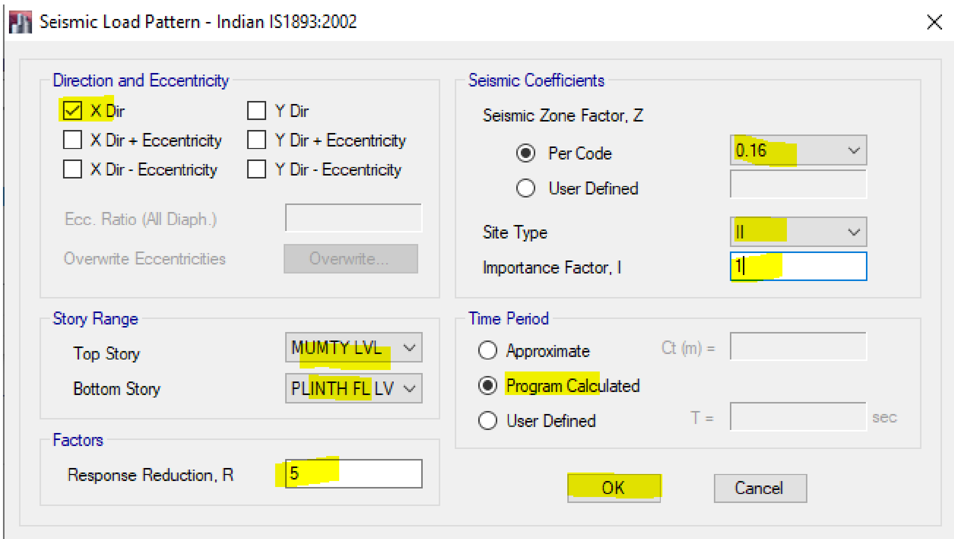
Create the model, analyse and design it (RCC design), and then verify it.

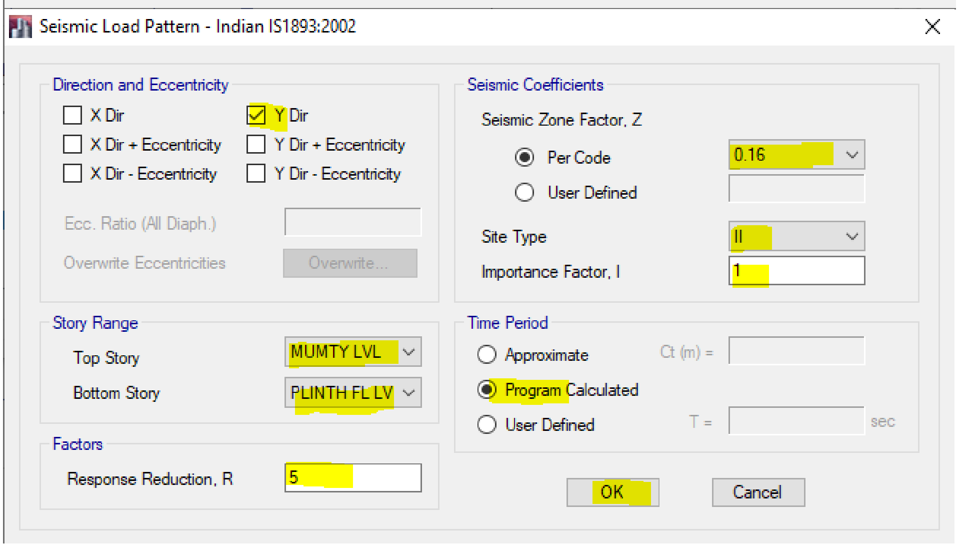
Break the model and define seismic load patterns as indicated below.



Dead Load: default self weight multiplier is 1.
Live or other defined load

Access the definition of static load by selecting Define > Static load.
Please adjust the lateral load to the amount provided below, and assign a different value in accordance with IS 1893-2005.





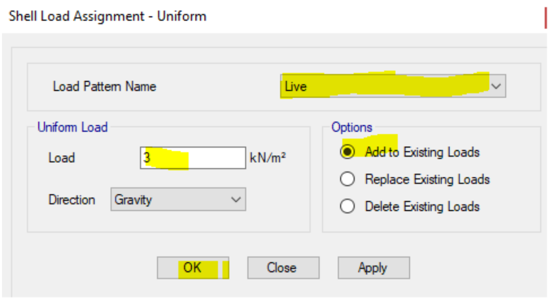
load combination, envelope, and model. Show tables and record these values after analysing the structure:
1. The Story Drift must not be more than 0.004 times the height of the storey.

The most extreme base reaction, FZ (When you're through, compare this number to the results of a straightforward analysis.)

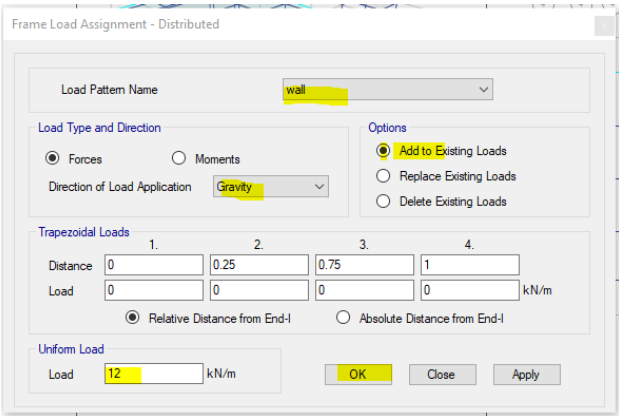
12) Assigning the live , dead, ffl, and wall load

Click the assign button once you have chosen the person to whom this load will be assigned.

Assign > shell area loads > uniform > click ok



Frame load Assignment> uniformly spread Add the load pattern to existing loads > uniform load > ok > member element > ok.



13) Assign support condition

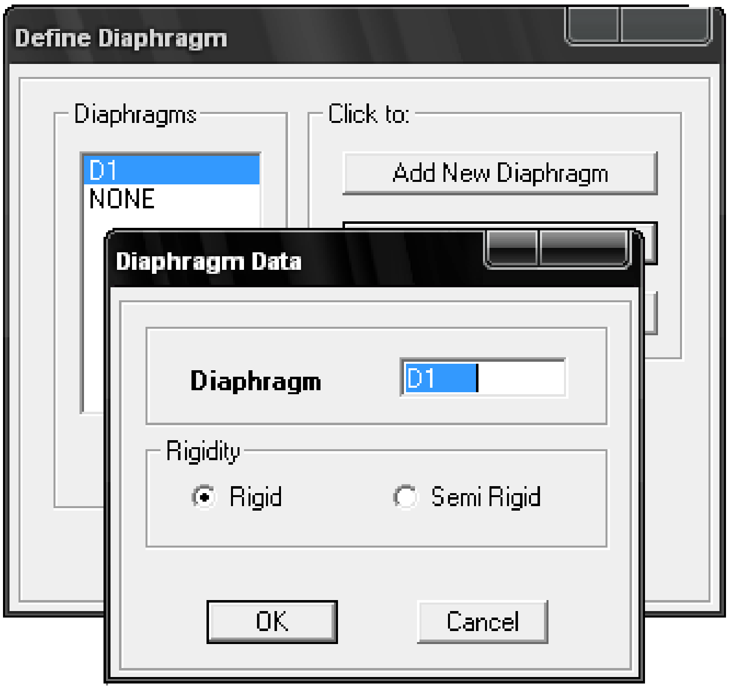
In the ETABS window's lower-right drop-down menu, choose the level you want to give the fixed support for using the "Create One Storey" button.

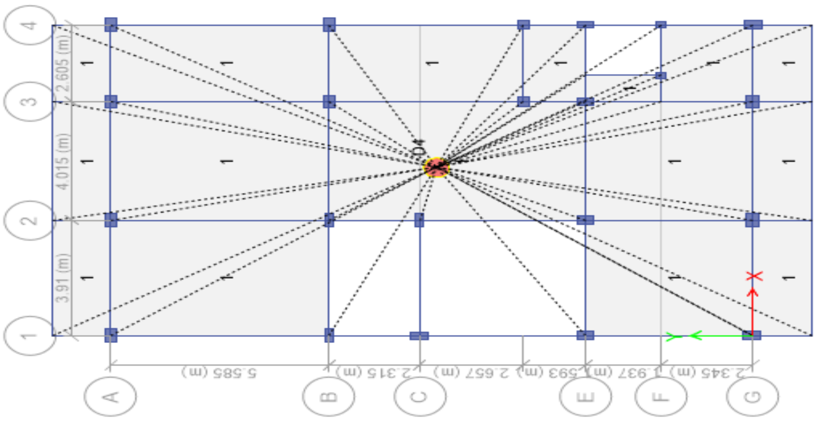
assign > Joint/Point>Restrain (Support) command> fixed

one storey bottom left corner and assign

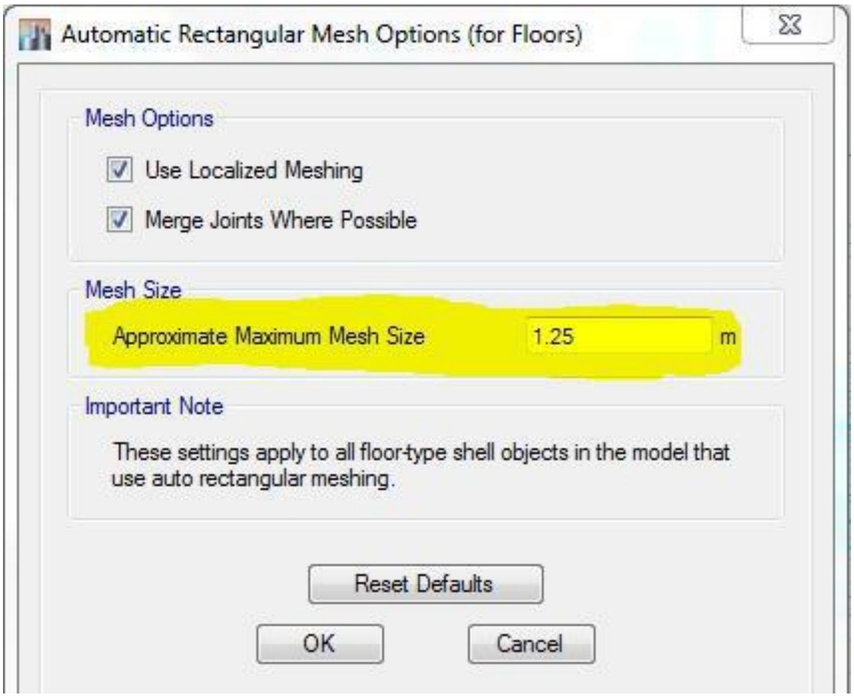
14) Assigning the diaphgram

Earthquake analysis treats slabs as rigid members. First, diaphragm action is applied to all slabs for stiff or semi-rigidity.



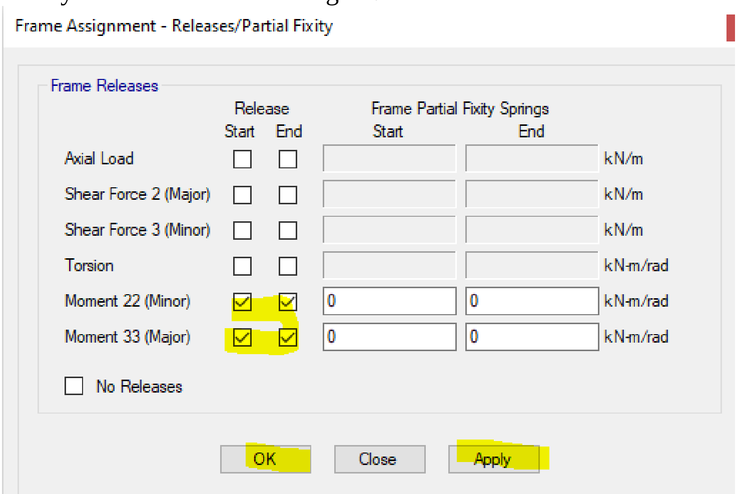


13) Meshing of Slabs
Select all of the slabs that will be used in the analysis (F5), then go to the Analyze menu, select Automatic rectangular mesh settings for floor, and then click on the Mesh customisation panel.
For best results, set the meshing size to 0.6 from the usual 1.25.



14) Assigning end release

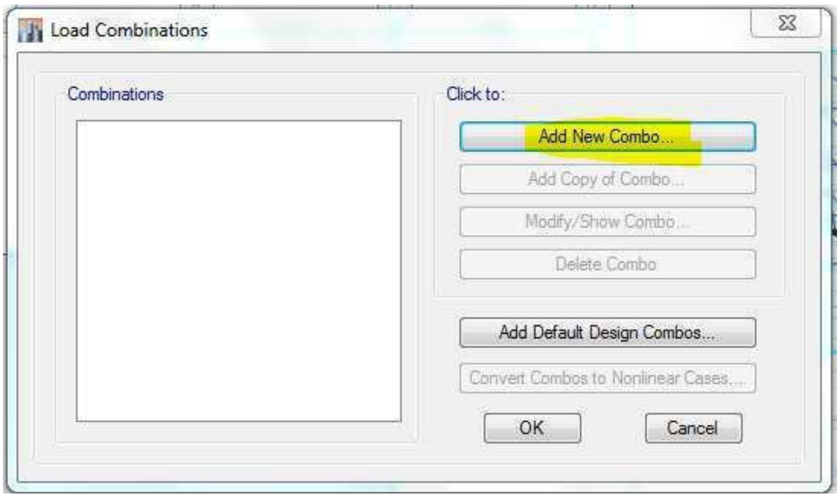
Assign>frame>partial fixity> then as indicated in figure.



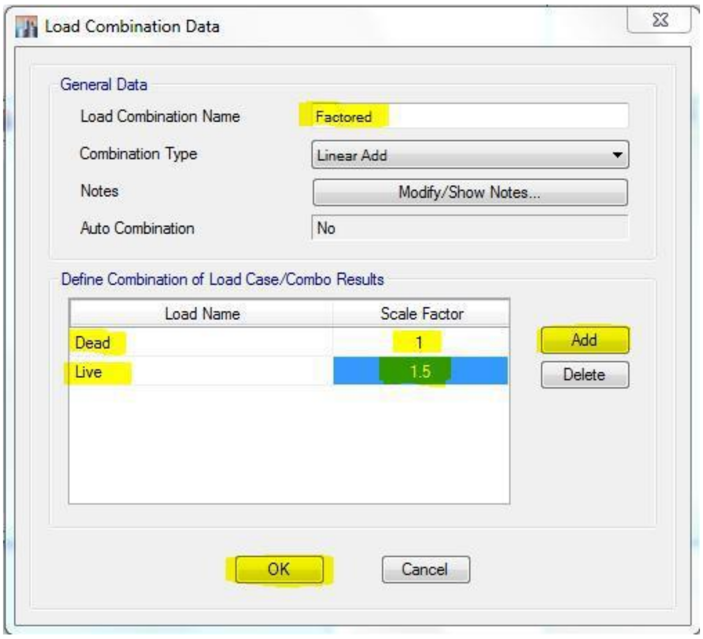
15) Load Combinations

If you want full creative control over your ideas, it's advisable to create your own load combination rather than rely on ETABS's proposed "UDCON2" load combination for dealing with the partial safety problem. Load combinations are created as follows.

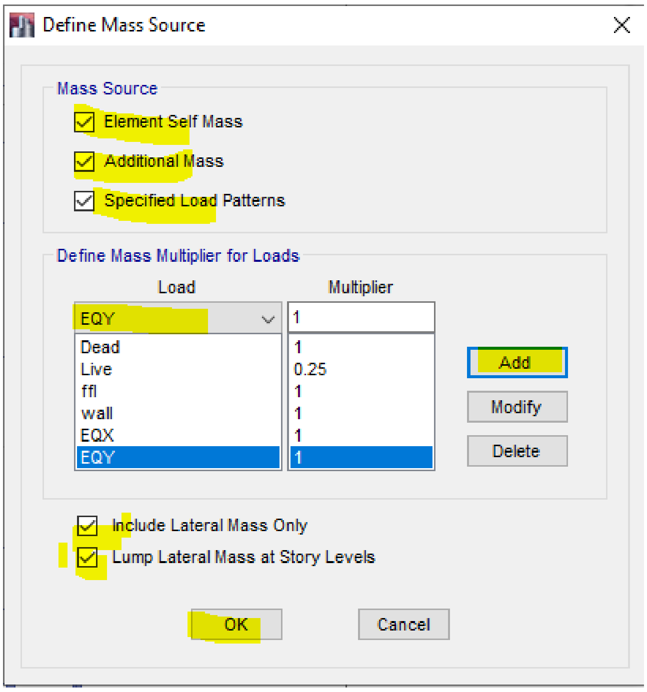
Define——>Load Combinations——> Click add new combination in that window.



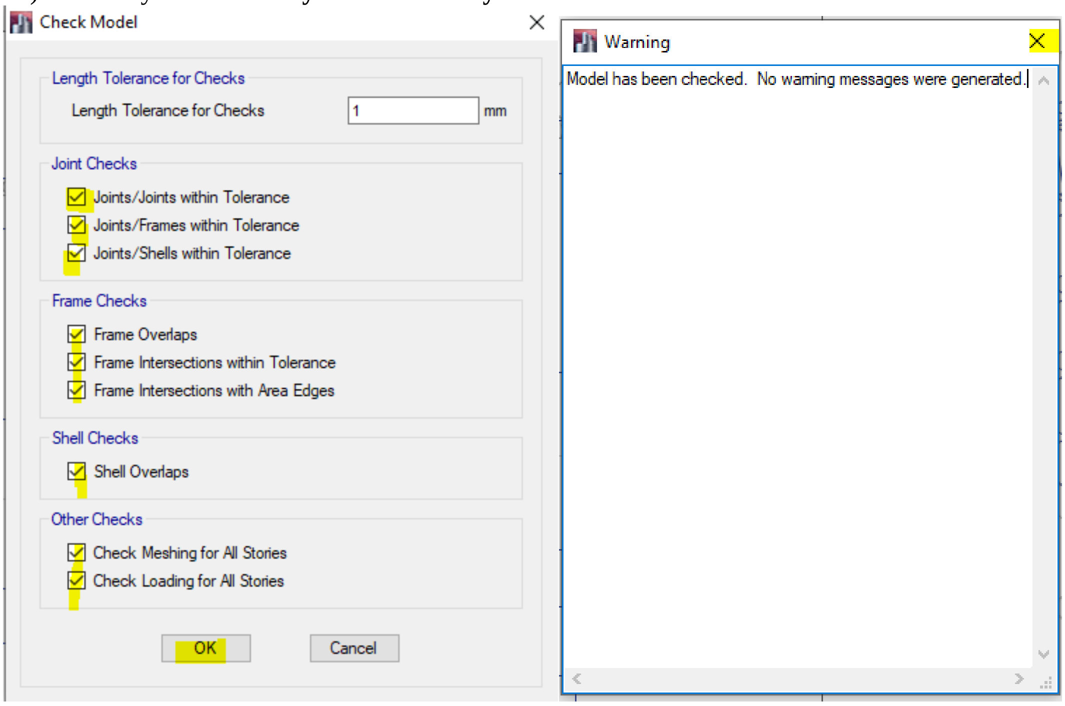
After clicking Add new combination, the safety factor window displays as shown below/next page (1 for DL and 1.5 for LL). The model may be analysed for post-processing outcomes when all load allocations and combinations are defined.



16) Define>mass source defines mass source. IS: 1893-2002 requires 25% live load (of 3 kN/m2) on all floors excluding roof.

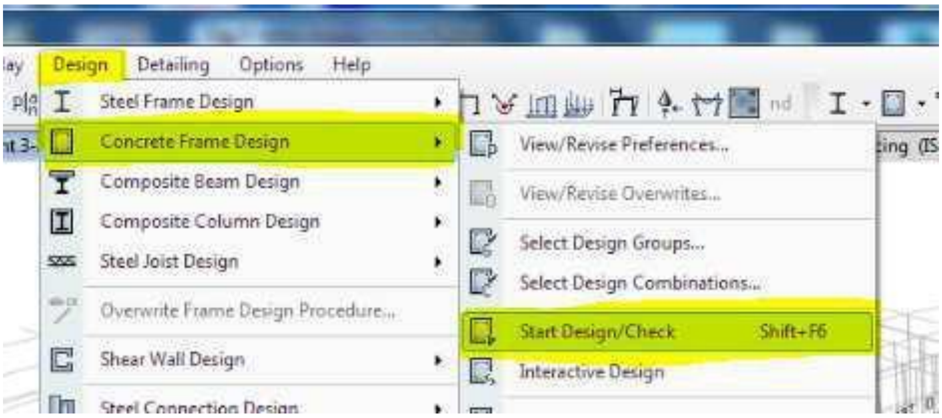


17) Run analysis from **Analysis > Run Analysis command**

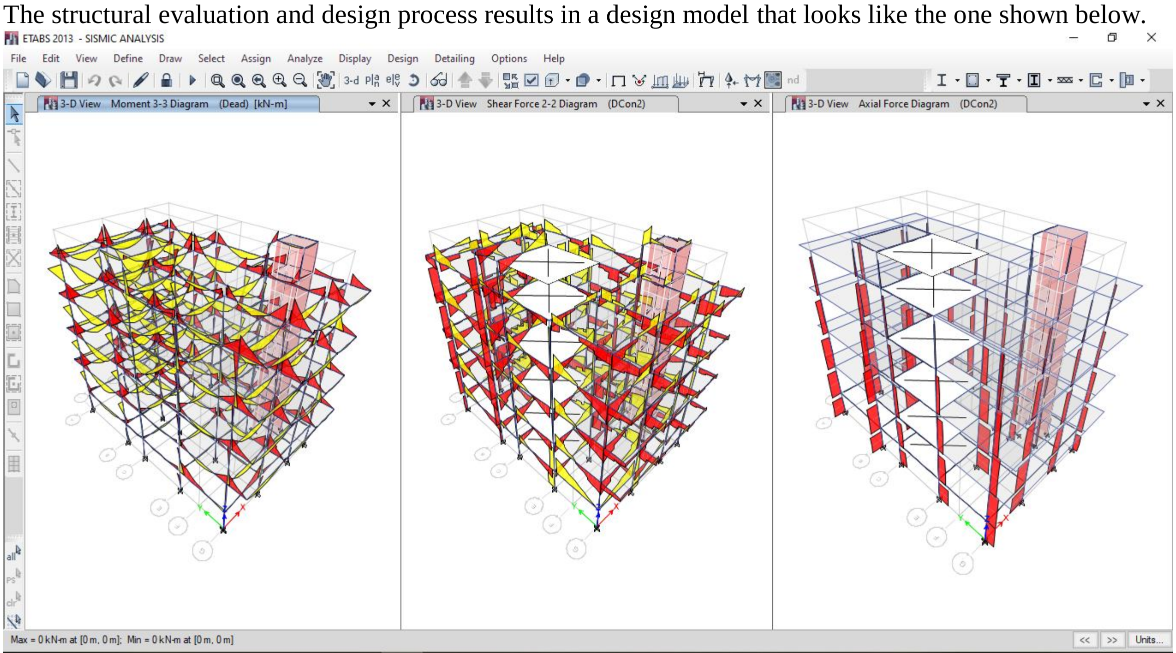


18) Concrete Design

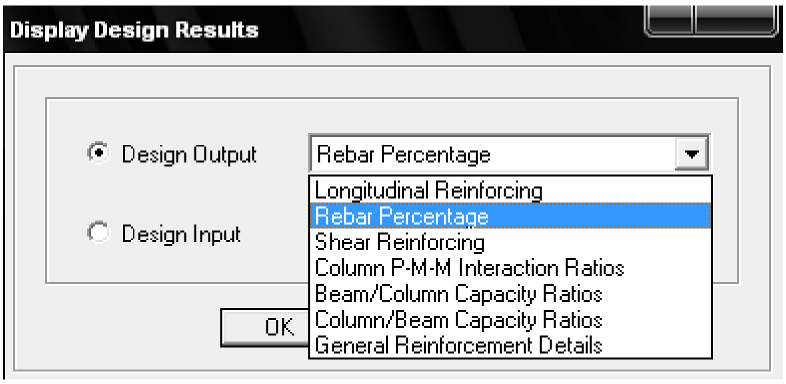
Click Design -> Concrete Frame Design -> Start Design / Check (Shortcut sft + F6) to begin concrete designing once you have finished modelling, allocating loads, and analysing load combinations. This will confirm your post-processing results (SFD & BMD).

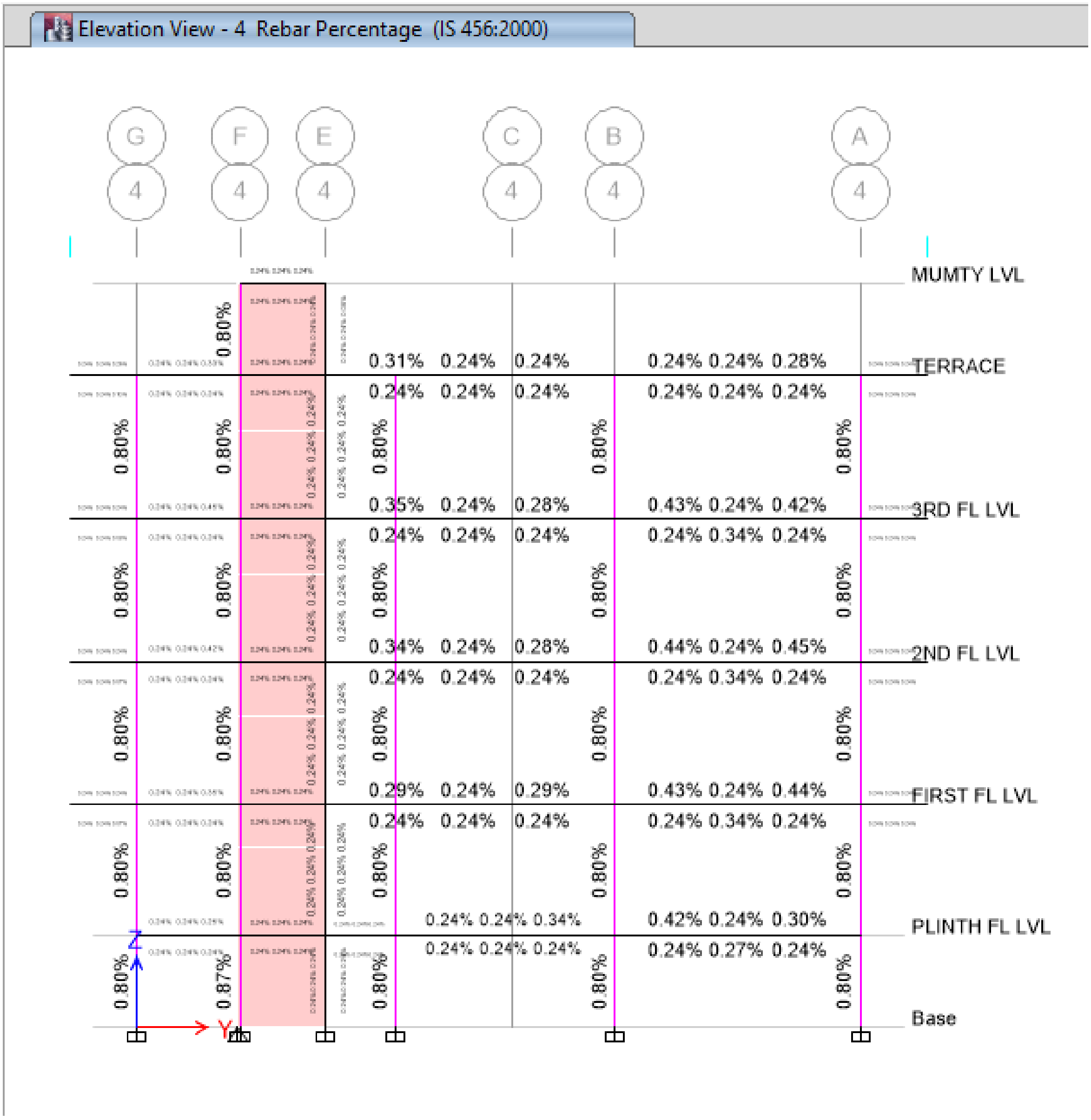


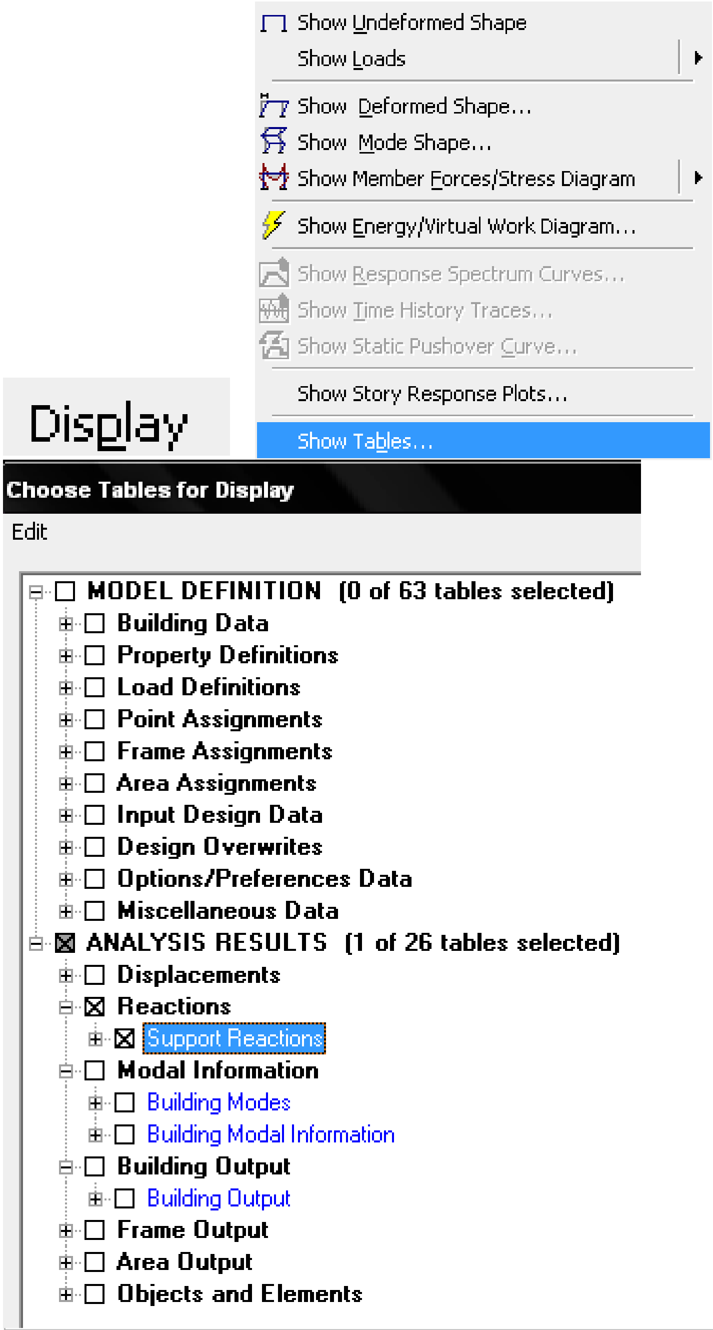
The structural evaluation and design process results in a design model that looks like the one shown below.

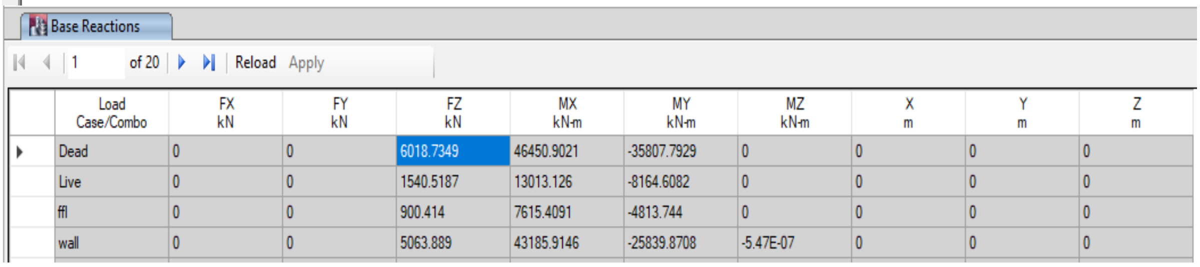


Design > Concrete Frame Design > Display Design Information shows steel percentages and column beam steel areas.




Elevation View - 4 Rebar Percentage (IS 456:2000)
G
F
E
C
B
A
4
MUMTY LVL
TERRACE
3RD FL LVL
2ND FL LVL
FIRST FL LVL
PLINTH FL LVL
Base
Z
Y





Base Reactions

1 of 20 Reload Apply

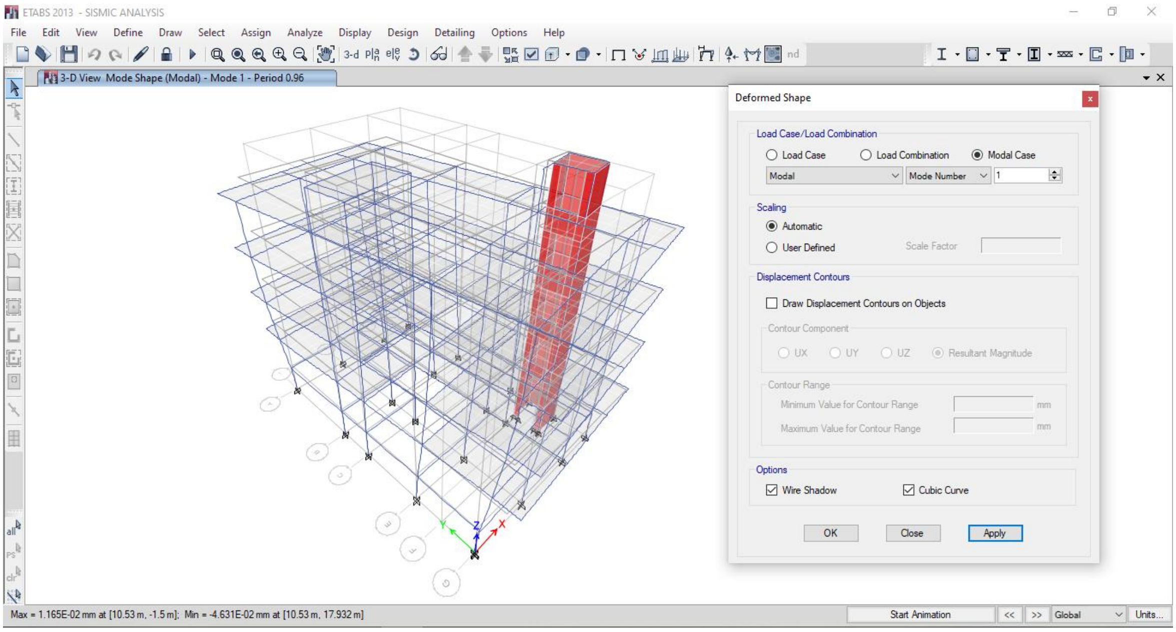
Load Case/Combo	FX kN	FY kN	FZ kN	MX kN-m	MY kN-m	MZ kN-m	X m	Y m	Z m
Dead	0	0	6018.7349	46450.9021	-35807.7929	0	0	0	0
Live	0	0	1540.5187	13013.126	-8164.6082	0	0	0	0
ffl	0	0	900.414	7615.4091	-4813.744	0	0	0	0
wall	0	0	5063.889	43185.9146	-25839.8708	-5.47E-07	0	0	0

The above figure shows total load acting on buling .

19) Dimensions of time and the mode of occupancy in the X and Y axes of the structure.

• According to the IS 1893, the time period is 0.075H0.75, which is equal to 0.34 seconds in the X-direction and 0.46 seconds in the Y-direction.

Select Display > Show Mode Shape to see the time period in ETABS.



Your are able to see the proportion of the total population that is actively participating in the model by choosing Display > Show Table > Model Information > Building Model Information > Model Participating Ratio from the drop-down menu.

Modal Participating Mass Ratios

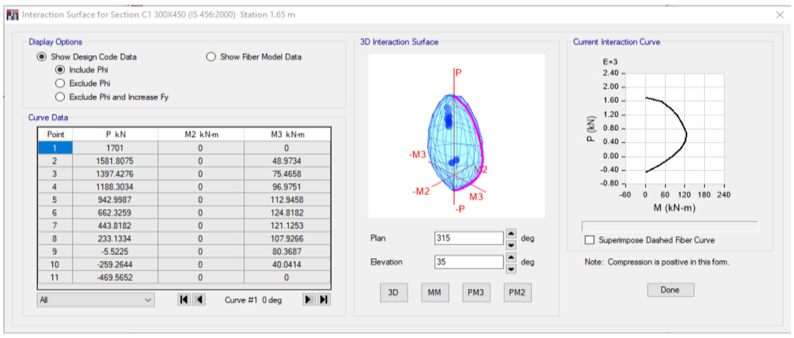
Case	Mode	Period sec	UX	UY	UZ	Sum UX	Sum UY	Sum UZ	RX	RY	RZ	Sum RX
Modal	1	0.96	0.2041	0.2866	0	0.2041	0.2866	0	0.0663	0.0459	0.3389	0.0663
Modal	2	0.67	0.4838	0.2375	0	0.688	0.5241	0	0.0859	0.1991	0.0104	0.1522
Modal	3	0.503	0.0487	0.2272	0	0.7366	0.7512	0	0.1211	0.0436	0.4435	0.2734
Modal	4	0.308	0.0296	0.0441	0	0.7662	0.7953	0	0.1583	0.1158	0.0199	0.4317
Modal	5	0.18	0.0032	0.0074	0	0.7694	0.8027	0	0.0157	0.0045	0.0169	0.4474
Modal	6	0.174	0.0998	0.07	0	0.8693	0.8726	0	0.1987	0.244	3.526E-06	0.6461
Modal	7	0.135	0.015	0.0078	0	0.8843	0.8804	0	0.0274	0.039	0.0028	0.6735
Modal	8	0.127	0.02	0.0587	0	0.9042	0.9391	0	0.135	0.0406	0.0779	0.8084
Modal	9	0.126	0.0005	0	0	0.9047	0.9391	0	0.0001	0.0001	0.0178	0.8085
Modal	10	0.116	0.0081	0.0011	0	0.9128	0.9402	0	0.0041	0.0218	0.0001	0.8126
Modal	11	0.112	0.0053	0.0065	0	0.9182	0.9467	0	0.0192	0.0094	0.0049	0.8318
Modal	12	0.111	0.0019	0.0051	0	0.9201	0.9517	0	0.0201	0.0027	0.0001	0.8519
Modal	13	0.099	0.0019	0.0003	0	0.922	0.952	0	0.0014	0.0029	3.164E-05	0.8533

20) Design check

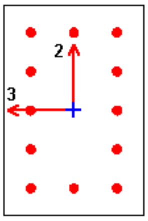
The sixth step is to build a model and load combinations and envelope. Analyze results
Seventh, verify the aforementioned outcomes
The greatest possible drift between floors is 0.004, or 0.004 times the floor height.
You should use the Equivalent Static/Linear Static Method if your Base Reaction FZ is more than the Maximum.
Third, the total mass participation in the x and y modalities must equal at least 90 percent of the lateral loads in order for this condition to be met.
Fourth, verify the mode shapes or distorted forms; the first three should represent x or y changes.
If the following requirements hold true in the response spectrum analysis, the model may be considered stable and unaltered.

Additional Points and Notes :-

1. The limitations of storey drift are specified in IS 1893-2002. (Part-1) Clause 7.11.1
Time periods must be manually determined for determining the load pattern in comparable static analysis; the method for doing so varies depending on the kind of building being analysed.
Page 24 of IS 1893-2002, Part-1, Clauses 7.6.1 and 7.6.2.
Page 25 of IS 1893-2002, Part-1, contains the comparative information of the base reactions. 4. For information on accommodating soft storeys, please see Clause 7.10.1 on Page 27 of IS 1893-2002.
21)Left-click any beam member to see below figure.
The figure illustrates the Pu-Mu interaction curve as well as shear, flexural, and beam/column features.



ETABS 2013 Concrete Frame Design
IS 456:2000 Column Section Design



Column Element Details Type: Ductile Frame (Flexural Details)

Level	Element	Section ID	Combo ID	Station Loc	Length (mm)	LLRF
PLINTH FL LVL	C6	C1 300X450	DCon2	1650	2100	0.534

Section Properties

b (mm)	h (mm)	dc (mm)	Cover (Torsion) (mm)
300	450	58	30

Material Properties

E_c (MPa)	f_{ck} (MPa)	Lt.Wt Factor (Unitless)	f_y (MPa)	f_{ys} (MPa)
25000	25	1	500	500

Axial Force and Biaxial Moment Design For P_u , M_{u2} , M_{u3}

Design P_u kN	Design M_{u2} kN-m	Design M_{u3} kN-m	Minimum M_2 kN-m	Minimum M_3 kN-m	Rebar Area mm²	Rebar % %
1456.7401	1.1705	-29.1348	29.1348	29.1348	1080	0.8

Factored & Minimum Biaxial Moments

	NonSway M_{ns} kN-m	Sway M_s kN-m	Factored M_u kN-m
Major Bending(M_{u3})	-9.4361	0	-9.4361
Minor Bending(M_{u2})	1.1705	0	1.1705

Slenderness Effects (IS 39.7.1) and Minimum Biaxial Moments (IS 39.2, 25.4)

	End Moment M_{u1} (kN-m)	End Moment M_{u2} (kN-m)	Initial Moment (kN-m)	k*M_a Moment (kN-m)	Minimum Moment (kN-m)	Minimum Eccentricity (mm)
Major Bending (M_3)	1.7434	-9.4361	-4.9643	0	29.1348	20

Effective Length Factors (IS 25.2, Annex E)

	K Sway	K Non-Sway	Framing Type	P-Delta Done?	Q Factor	K Used
Major Bend(M_3)	1.603615	0.72758	Ductile Frame	No	0.006397	0.72758
Minor Bend(M_2)	1.529941	0.708686	Ductile Frame	No	0.007429	0.708686

Additional Moment Reduction Factor k (IS 39.7.1.1)

A_g cm²	A_{sc} cm²	P_{uz} kN	P_b kN	P_u kN	k Unitless
1350	10.8	1923.75	599.0777	1456.7401	0.352548

Additional Moment (IS 39.7.1)

	Consider M_a	Length Factor	Section Depth (mm)	KL/Depth Ratio	KL/Depth Limit	KL/Depth Exceeded	M_a Moment (kN-m)
Major Bending (M_3)	Yes	7.857E-07	0.0005	2.668E-06	1.2E-05	No	0
Minor Bending (M_2)	Yes	7.857E-07	0.0003	3.898E-06	1.2E-05	No	0

Column Element Details Type: Ductile Frame (Shear Details)

Level	Element	Section ID	Combo ID	Station Loc	Length (mm)	LLRF
PLINTH FL LVL	C6	C1 300X450	DCon2	1650	2100	0.534

Section Properties

b (mm)	h (mm)	dc (mm)	Cover (Torsion) (mm)
300	450	58	30

Material Properties

E_c (MPa)	f_{ck} (MPa)	Lt.Wt Factor (Unitless)	f_y (MPa)	f_{ys} (MPa)
25000	25	1	500	500

Design Code Parameters

γ_C	γ_S
1.5	1.15

Additional Moment Reduction Factor k (IS 39.7.1.1)

A_g cm²	A_{sc} cm²	P_{uz} kN	P_b kN	P_u kN	k Unitless
1350	10.8	1923.75	599.0777	1456.7401	0.352548

Additional Moment (IS 39.7.1)

	Consider M_a	Length Factor	Section Depth (mm)	KL/Depth Ratio	KL/Depth Limit	KL/Depth Exceeded	M_a Moment (kN-m)
Major Bending (M_3)	Yes	7.857E-07	0.0005	2.668E-06	1.2E-05	No	0
Minor Bending (M_2)	Yes	7.857E-07	0.0003	3.898E-06	1.2E-05	No	0

Shear Design for V_{u2}, V_{u3}

	Rebar A_v /s mm²/m	Design V_u kN	Design P_u kN	Design M_u kN-m	V_c kN	V_s kN	V_p kN
Major Shear(V_2)	0	0	0	0	0	0	0
Minor Shear(V_3)	498.8	1.7395	1456.7401	1.1705	79.69	43.5594	123.2494

Design Forces

	Factored V_u kN	Factored P_u kN	Factored M_u kN-m
Major Shear(V_2)	6.7755	1456.7401	-9.4361
Minor Shear(V_3)	1.7395	1456.7401	1.1705

Design Basis

Shr Reduc Factor Unitless	Strength f_{ys} MPa	Strength f_{ck} MPa	Area A_g cm²
1	415	25	1350

Concrete Shear Capacity

Major Shear(V_2)	Conc.Area A_c cm²	A_{st} %	Allowable Tau_c MPa	CompFactor Delta Unitless	DepthFactr k Unitless	Strengh Factor Unitless
Major Shear(V_2)	0	0	0	0	0	0
Minor Shear(V3)	1089	0.496	0.488	1.5	1	1

Shear Rebar Design

	Design V_u kN	Stress Tau MPa	Conc.Cpcty Tau_{cd} MPa	Allowable $Tau_{c,max}$ MPa	Rebar Area A_{sv} /s mm²/m
Major Shear(V_2)	0	0	0	0	0
Minor Shear(V_3)	1.7395	0.02	0.73	3.1	498.8

ETABS 2013 Concrete Frame Design
IS 456:2000 Beam Section Design

Beam Element Details Type: Ductile Frame (Flexural Details)

Level	Element	Section ID	Combo ID	Station Loc	Length (mm)	LLRF
PLINTH FL LVL	B25	PB 230X450	DCon2	4822	4972	1

Section Properties

b (mm)	h (mm)	b_f (mm)	d_s (mm)	d_{ct} (mm)	d_{cb} (mm)
230	450	230	0	30	30

Material Properties

E_c (MPa)	f_{ck} (MPa)	Lt.Wt Factor (Unitless)	f_y (MPa)	f_{ys} (MPa)
25000	25	1	500	500

Design Code Parameters

γ_C	γ_S
1.5	1.15

Flexural Reinforcement for Moment, M_{u3} & T_u

	Required Rebar mm²	+Moment Rebar mm²	-Moment Rebar mm²	Regular Minimum Rebar mm²	Seismic Minimum Rebar mm²
Top (+2 Axis)	248	0	203	176	248
Bottom (-2 Axis)	101	0	0	0	101

Design Moments, M_{u3} & T_u

Design +Moment kN-m	Design -Moment kN-m	Factored M_{u3} kN-m	Torsion T_u kN-m	Special M_t kN-m
0	-35.4346	-34.6646	0.4427	0.77

Beam Element Details Type: Ductile Frame (Shear Details)

Level	Element	Section ID	Combo ID	Station Loc	Length (mm)	LLRF
PLINTH FL LVL	B25	PB 230X450	DCon2	4822	4972	1

Section Properties

b (mm)	h (mm)	b_f (mm)	d_s (mm)	d_{ct} (mm)	d_{cb} (mm)
230	450	230	0	30	30

Material Properties

E_c (MPa)	f_{ck} (MPa)	Lt.Wt Factor (Unitless)	f_y (MPa)	f_{ys} (MPa)
25000	25	1	500	500

Design Code Parameters

γ_C	γ_S
1.5	1.15

Shear/Torsion Design for V_{u2} & T_u

Rbar A_{sv} /s mm²/m	Rbar A_{svt} /s mm²/m	Design V_{u2} kN	Design T_u kN-m	Design P_u kN
254.94	221.69	43.0429	0.4427	0

Design Forces

Factored V_{u2} kN	Factored T_u kN-m	Equivalent V_e kN
43.0429	0.4427	46.1229

Design Basis

Design V_{u2} kN	Conc.Area A_c cm²	Area A_g cm²	Tensn.Reinf A_{st} mm²	Strength f_{ys} MPa	Strength f_{ck} MPa	LtWt.Reduc Factor Unitless
43.0429	966	1035	263	415	25	1

Concrete Capacity

Conc.Area A_c cm²	Tensn.Rein A_{st} mm²	A_{st} %	Allowable $Tau_{c.max}$ MPa	Strength f_{ys} MPa	CompFactor Delta Unitless	DepthFactr k Unitless	Strengh Factor Unitless
966	263	0.272	0.371	415	1	1	1

Shear Rebar Design

Design V_e kN	Stress Tau MPa	Conc.Cpcty Tau_{cd} MPa	Allowable $Tau_{c.max}$ MPa	Rebar Area A_{sv} /s mm²/m	Shear V_c kN	Shear V_s kN	Shear V_p kN
46.1229	0.48	0.37	3.1	254.94	35.8804	38.64	74.5204

Torsion Capacity

Rebar A_{svt} /s mm²/m	Torsion T_u kN-m	Shear V_u kN	Core b_1 mm	Core d_1 mm
221.69	0.4427	43.0429	190	410

VIII. STRUCTURAL ANALYSIS AND DESIGN

8.1 General

Structural engineering researches, plans, designs, builds, inspects, monitors, maintains, rehabs, and demolishes permanent and temporary structures, structural systems, and their components. Structural design evaluates stability, strength, and stiffness. Structural analysis and design create structures that can sustain all loads over their lifetime. Structures bear loads. If the structure is poorly designed or built, or if the applied loads exceed the design constraints, the device may fail to fulfil its intended purpose, which might be devastating.

8.2 IMPORTANT POINTS FOR BEAMS

- For grades Fe 415 and Fe 500, maximum tension bar spacing is 180 mm and 150 mm, respectively.
- Ast = 0.85bd/fy for beam tension reinforcement. Maximum tension and compression reinforcement in beams is Ast < 0.04bd.
- The clear span between lateral supports of a simply supported or continuous beam should be no more than 60 b or 250 b2/d.
- Cantilevers must have a clear space of 25 b or 100 b2/d from the free end to the lateral constraint.
- For vertical stirrups, shear reinforcement must not exceed 0.75d along the member's axis, where d is the section's effective depth. Never surpass 300mm.
- Asv/b.Sv < 0.4/fy must be given by stirrups.

Where,
Asv = total cross sectional area of stirrup legs effective in shear
Sv = stirrup spacing along the length of the member
b = width of the beam
fy = should not be more than 415 N/mm²

• **Positive moment reinforcement**

i. Simple members require Ld/3 of the positive moment reinforcement along the same face into the support, whereas continuous members need Ld/4.
ii. The positive reinforcement that must be extended into the support in accordance with (a) must be anchored such that it produces its design stress in tension at the support face when the flexural member is part of the principal system for withstanding lateral loads.
iii. At inflection points and simple supports, reinforcing positive moment tension must be less than M1/V + Lo.
Where, M1 = moment of resistance if all reinforcement is strained to fd
fd = 0.87 fy in the case of limit state design and the permissible stress in the case of working stress design σst
V = shear force at the section due to design loads
Lo = sum of the anchoring beyond the centre of the support and the support provided by any hook or mechanical attachment. At inflection, Lo cannot exceed 12 or the effective depth of the members, whichever is larger. Crushing the reinforcing ends increases the bar's diameter by 30% to the M1/V ratio.

• **Negative moment reinforcement**

i. At the point of inflection, one-third of the negative moment reinforcement at the support must extend outward for a distance equal to or more than the effective depth of the member by 12, which is one sixteenth of the clear span.

• **Curtailment of tension reinforcement in flexural members**

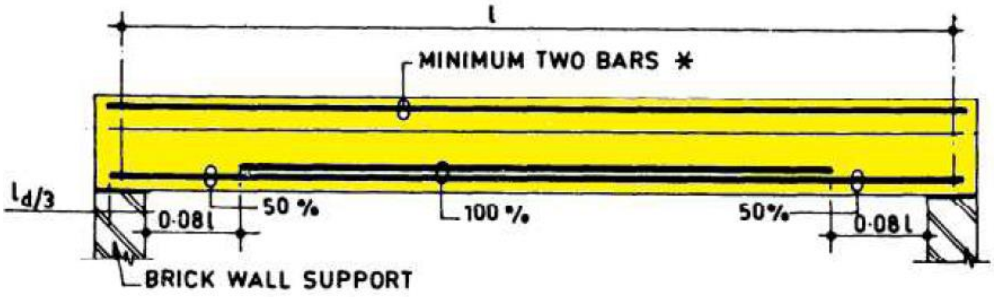
i. Except for the simple support or cantilever end, reinforcement must continue beyond the point at which it no longer resists flexure for the effective depth of the member or 12 times the bar diameter, whichever is larger. This criteria does not applicable when the reinforcement cannot withstand flexure.
ii. Any of the following in a stress zone requires flexural reinforcement removal.
a. At the cutoff point, shear strength is two-thirds of the maximum allowed (this includes the web reinforcement). Web reinforcement limits shear at the cutoff point.
b. Each terminated bar has stirrup area greater than shear and torsion for three fourths the member's effective depth from the cutoff point. The extra stirrup area must be at least 0.4bs/fy, where b is beam width, s is spacing, and fy is reinforcing strength in N/mm2. The spacing must not exceed d/8βb, where βb is the ratio of bars chopped off to total bars at the section and d is the effective depth.
c. For bars with a diameter of 36 millimetres or less, continuing bars offer twice as much room as needed for flexure at the point where they are cut off, and the shear does not exceed three quarters of what is allowed.

• **Lap splices**

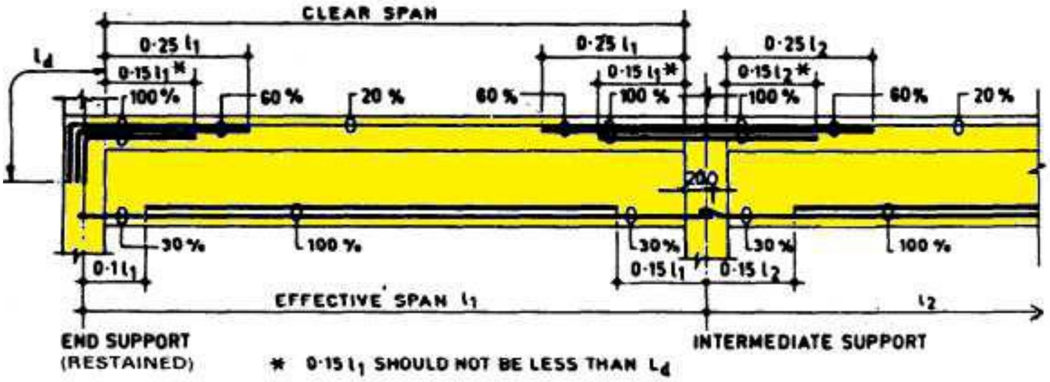
i. In order to join bars larger than 32mm, welding or mechanical splicing is required.

ii. For bars in flexure tension, the lap length is Ld or 30, whichever is bigger; for bars in direct tension, the lap length is 2Ld or 30, whichever is greater. The minimum length of the lap is 15 inches (or 200mm).
iii. Compression lap length should equal development lap length (as calculated in 26.2.1), but not be less than 24.
iv. Joining 2-inch-gap bars. The shorter bar's diameter determines lap length.

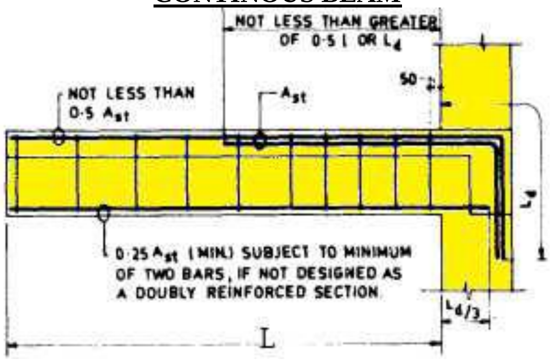
• Temperature, creep, and shrinkage after drywall and finishes are completed shouldn't be more than span/350, or 20 mm.
• Most of the time, the ultimate deflection owing to all loads (temperature, creep, and shrinkage) should not exceed effective span/250, measured from the floor supports' as cast level.



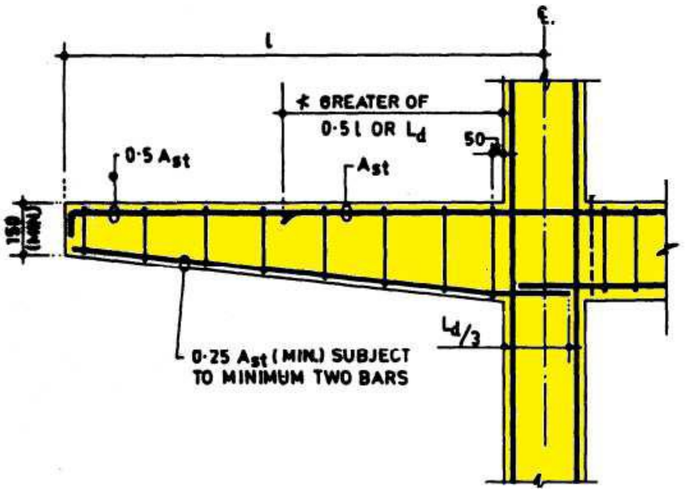
SIMPLY SUPPORTED BEAM



CONTINOUS BEAM



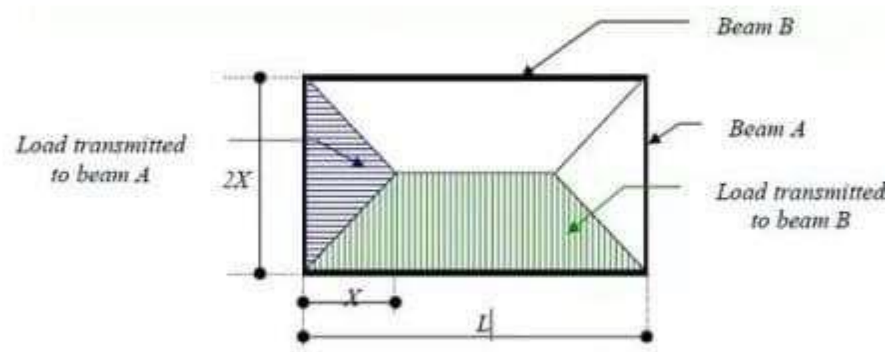
CANTILEVER BEAM PROJECTION FROM A COLUMN



CANTILEVER BEAM PROJECTION FROM A BEAM OVER COLUMN

8.2 IMPORTANT POINTS FOR SLABS

• If a slab extends in many directions, choose the shortest route to calculate its span-to-effective depth ratio.
• The span-to-depth ratio below should meet vertical deflection requirements for loads up to 3 kN/m2 in two-way slabs reinforced with mild steel and shorter spans (up to 3.5 m).Continuity 35, Structures That Are Supported By Slabs Simply multiplying by 0.8 will provide you the high strength Fe 415 warped bars you need.
The figure takes into account loads that are equally distributed when depicting beams that support solid slabs that span in two different directions at right angles. below.



• Major reinforcing bar spacing on the horizontal plane must not exceed 300 mm or three times the effective depth of the solid slab.
• Shrinkage and temperature impacts need minimum strengthening of slabs. Mild steel reinforcement at the rate of 0.15 percent in both directions is required for slabs. When using high-strength bent bars, that percentage drops to 0.12%.
• Reinforcing bars cannot exceed 1/8 of the slab's thickness.
• Slabs have maximum bending moments per unit width.
equations:
$Mx = \alpha x.\ w.\ lx^2$
$My = \alpha y.\ w.\ lx^2$
Where, αx and αy are coefficient given in table 26.
w = total design load per unit area
Mx and My = moments on strips of unit width spanning lx and ly respectively.
lx and ly = • In accordance with figure 25 of IS code 456, slabs are divided into central strips and edge strips in both directions. The central strip is three quarters the slab's width, while the outer strip is one eighth.

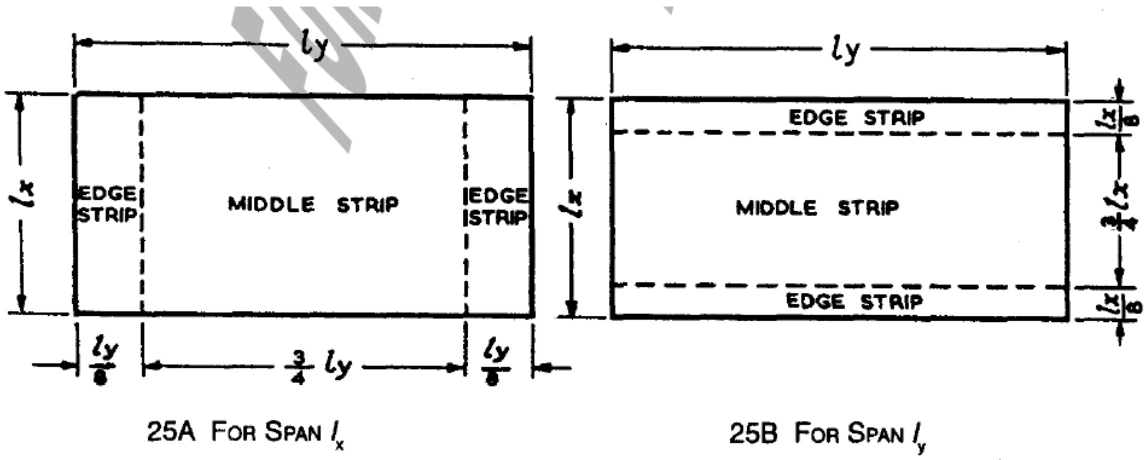


FIG. 25 DIVISION OF SLAB INTO MIDDLE AND EDGE STRIPS

D-1.1 does not disperse centre strip maximum moments (IS 456).

• At least half of the tension reinforcement must reach 0.3l from the support in the top section of the slab over the continuous borders of a middle strip.

Edges that don't quite connect might have negative moments. Tension reinforcement equal to 50% of the mid span and 0.1 times l into the span is usually adequate.

The minimum requirements of Section 3 and the torsion criteria of D-1.8 through D-1.10 must be met by the reinforcing strips installed parallel to the edge.

When a slab is supported on two sides, it is essential that torsional reinforcement be provided to each corner. Bars running parallel to the slab's edges and outward for at least a quarter of the shorter span will be used for top and bottom reinforcement. Three-quarters of the slab's mid-span moment reinforcement will be distributed between these four layers.

• In a corner with a single continuous slab edge, half the D-1.8 torque reinforcement is required.

Torsion reinforcements are unnecessary in the corners of a slab with a continuous edge at the corner.

If torsion ly/lx is more than 2, slabs should span only in one direction.

• For slabs that are simply supported, the maximum moment per unit width is given by the following equation, assuming no special measures are taken to prevent twisting at corners or lifting.

$Mx = \alpha x.\ w.\ lx^2$

$My = \alpha y.\ w.\ lx^2$

Where, αx and αy are coefficient given in table 27.

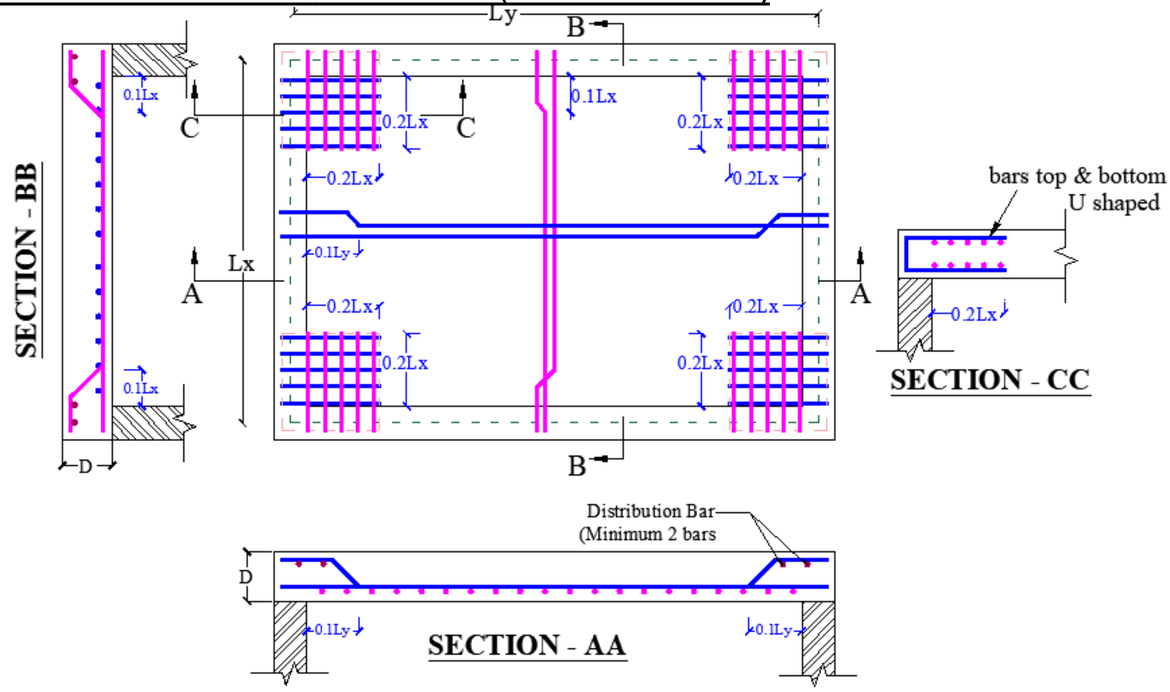
w = total design load per unit area

Mx and My = moments on strips of unit width spanning lx and ly respectively.

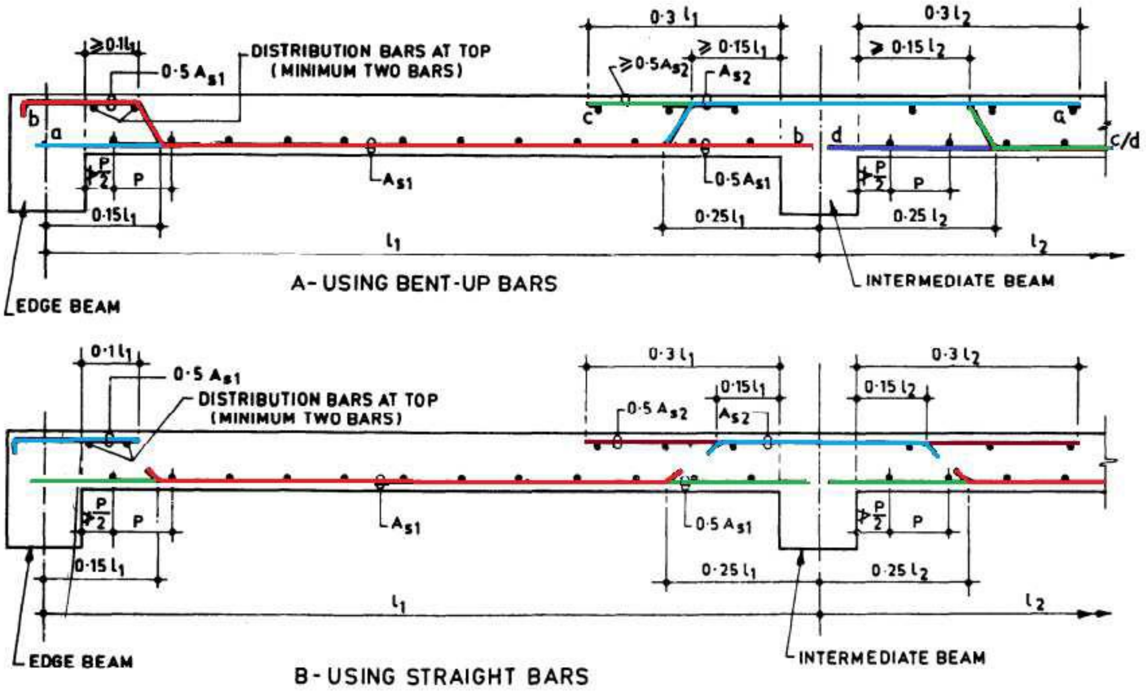
lx and ly = lengths of the shorter span and longer span respectively.

• A minimum of fifty percent of the tension reinforcement that is given at the midpoint of the span should extend to the supports. The remaining fifty percent need to stretch to within 0.1lx or 0.1ly of the support, depending on the circumstance.

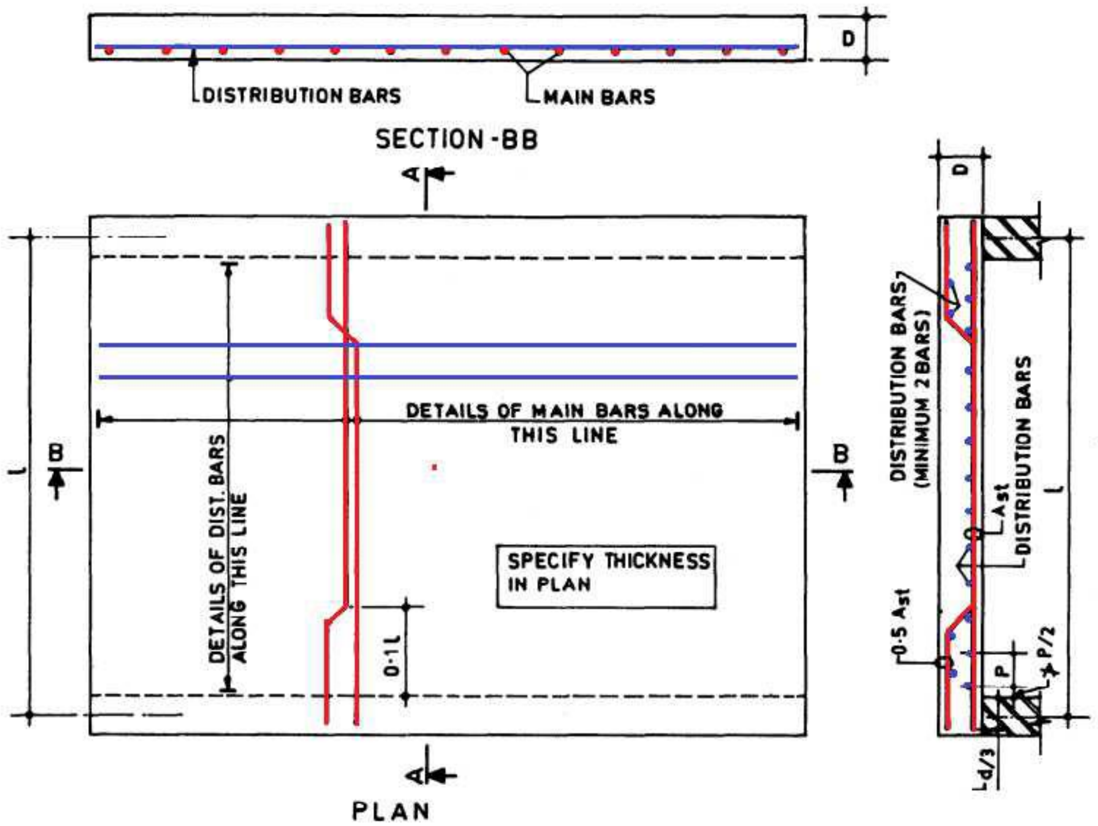
TYPICAL DETAILS OF TWO WAY SLAB (RESTRAINED SLAB)



DETAILING OF ONEWAY CONTINUOUS SLAB AS PER SP 34 (PG:127)



TYPICAL DEAILS OF ONE WAY SLAB AS PER SP34:1987 (#CL.9.3.1)



8.3 IMPORTANT POINTS FOR COLUMNS

• Columns and struts have effective lengths over three times the least lateral dimension. Pedestal if less than three.

• The lex/D and lex/b ratios of a compression member must be less than 12 for it to be considered short.

• The apparent gap between the compression members and the restraints is referred to as the unsupported length, which is denoted by the letter l.

The distance between end restraints must be no more than 60 times the smallest lateral dimension of the column.

• The unsupported length, l, of a column must be less than or equal to 100b2/D.

There can be no quirkiness in the columns.

Minimum allowed dimension (emin) = unsupported length (lun) 500 + side length (dx) 30 or 20 mm.

The gross column cross section must be between 0.8 and 6% of the longitudinal reinforcement cross section. Use less reinforcing as 6 percent may hinder concrete laying and compacting. Lapping bars from lower columns with those in the column under consideration should employ no more than 4% steel. The minimal steel percentage should be computed on the concrete area that must sustain direct stress in columns with a larger cross sectional area than needed. Instead of basing the proportion on area, do this. The minimum diameter for a bar is 12mm; four longitudinal bars are needed for rectangular columns and six for circular ones.

8.4 IMPORTANT POINTS FOR FOOTINGS

• Reinforced and plain concrete footings on soils need 150mm edge thickness.

• The greatest bending moment at a section must be determined for a freestanding concrete base supporting a column, pedestal, or wall.

i.e., foundations for a single column or wall.

Footings for masonry walls should be placed midway between the centre line and the wall's edge.

iii) For footings beneath gusseted bases, midway between the column or pedestal and the base.

• Footings are vulnerable to shear failures in both directions (punching shear)

The effective depth of a footing, denoted by the symbol d, is what establishes the minimum distance that a critical section must be from the face of a column or wall in the case of one-way shear.

There is a critical region that is d/2 distant from the face of the column in two-way shear. where d refers to the depth of the foundation that may be used.

A bearing transfers concrete compressive stress from the base of a column or pedestal to the top of the footing or pedestal. The loaded zone bearing pressure must not exceed the bearing stress limit in direct compression times (A1/A2) up to 2.