

Example: 1 Five storey building with wind Load

DATA:

Plan of Building : 12m x 10m

Height of Building :18.5m ; Typical Floor height=3.0m; Ground Floor height=4.0m

Depth of Foundation: 2.50m

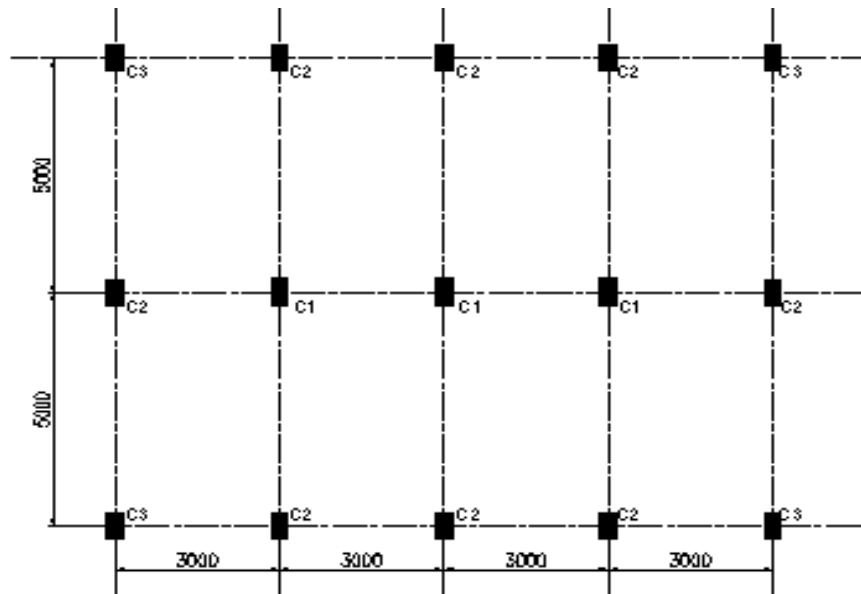
Size of RCC members:

COLUMNS C1,C2, C3 :300 x 600mm

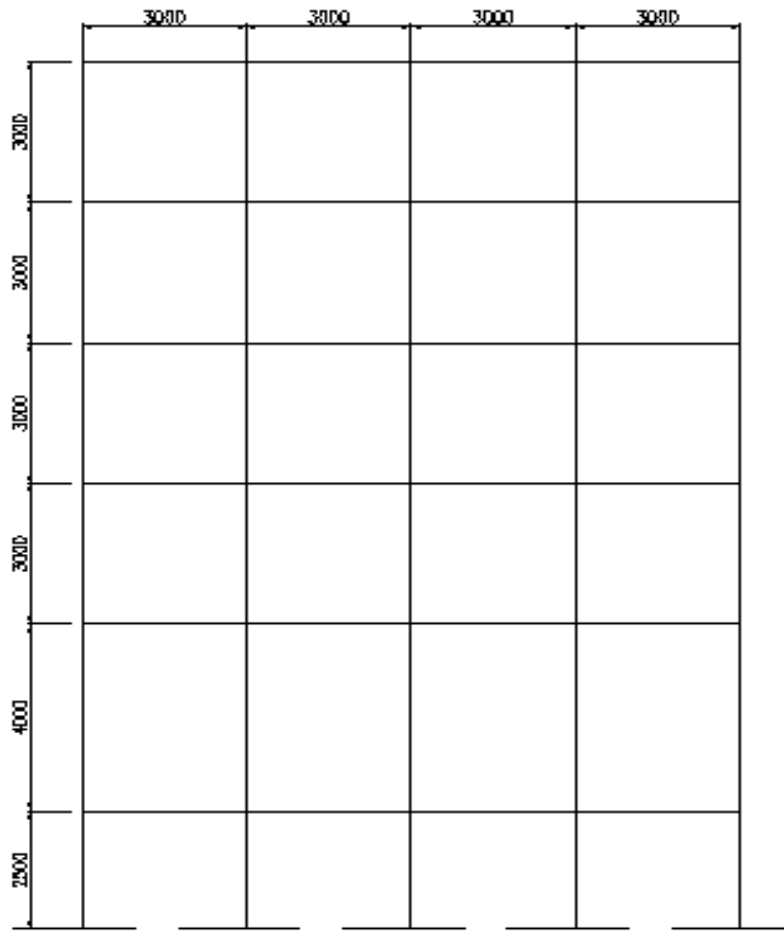
BEAMS :230 x 300 mm (End beam)

 : 230 x 450mm (Main Beam)

SLAB Thickness :120mm




PLAN



ELEVATION

1. Geometry Creation :

Open STAAD.Pro V8i. New Project → Select  Space → Length =meters;
Force=KN;


File Name= Five storied Building → Next

Select  Open Structure Wizard → Finish.

Change to Frame Models from Truss models → Select Bay Frame and double click on it.


Length (x)= 12.0m No. of bays =4  Click Bay No.1=3.0

Bay No.2=3.0; Bay No.3=3.0; Bay No.4=3.0

Height (y)= 18.5m No. of bays =6  Click Bay No.1 =2.50

Bay No.2 =4.00; Bay No.3 =3.00; Bay No.4 =3.00;

Bay No.5 =3.00; Bay No.6 =3.00

Width (z)= 10.0m No. of bays =2  Click Bay No.1= 5.0

Bay No.2 =5.0

Apply→ Transfer model→ click yes→ o.k.

Go to Front view icon (first view)

1. Member Properties :

Front view → Select beam along X axis for top 5 levels by windowing → Go to Isometric view and confirm the selection of beams in X direction of top 5 levels of beams → From main menu → Commands → Member property→

Prismatic→ Rectangular → YD=0.30 ZD=0.23→ Assign→

De select the member. Go to side view → Select beam along Z direction for top 5 levels by windowing → Go to Isometric view and confirm the

selection of beams in z direction of top 5 levels of beams → From main menu → Commands → Member property→ Prismatic→ Rectangular →

YD=0.45 ZD=0.23→ Assign→ De select the member and select the beam at lower level (Plinth beam along x and z direction) → From Main menu→

Commands→ member property →Prismatic→Rectangular YD=0.4

ZD=0.23→ Assign→ Close→ De select the member and select column member (Go to top view). Window the Corner column on four ends (type 1 column 0.6m x 0.3m) → From Main menu → Commands→Member Property→Prismatic YD=0.60 ZD=0.30 → Assign→ Confirm by viewing the column in Isometric view. Go to top view→ Window the intermediate column of end rows (type 2 column 0.6m x 0.3m) → From Main menu→Commands→Member Property→Prismatic→Rectangular YD=0.6m ZD=0.30→ Assign→ Confirm by viewing the column in Isometric view. Window the internal column (type 3 column YD=0.60 ZD=0.3) → From Main menu → Commands→Member Property→Prismatic YD=0.60 ZD=0.30 → Assign→ Confirm by viewing the column in Isometric view.

To confirm whether any missing property

Main menu →Select→Missing attributes →Missing Property→ No entity with missing property is found→o.k.

2. Supports :

Change to Node cursor (joint) and select the nodes (joints).

(If more than one node use CTRL key and select the nodes). Go to front view → Select the bottom most nodes for supports → From Main menu → Commands → Support specification → Fixed → Assign → Close → De select the nodes and change to beam cursor.

4. Loading :

*DEFINE WIND LOAD: X DIRECTION

- (1) From Main menu →Commands→ Loading → Definitions → Wind Load→ Type 1→ Comments: Wind → Add. Click Type 1 →Add. Wind Definition screen will appear. Intensity → Select type1.

Intensity (KN/m²)	Height(m)
1.8	10
2.0	15
2.10	20

→ Add.

Exposure → Factor = 1.0 → Add → Close.

Click the Exposure 1.0 and using node cursor select entire structure → Assign.

Note:

STAAD TEXT MODE

DEFINE WIND LOAD

TYPE 1(WIND INTENSITY IN X DIRECTION)

INTENSITY 1.8 2.0 2.1 HEIGHT 10.0 15.0 20.0

EXPOSURE 1.0 YRANGE 15.0 20.0

Z DIRECTION

_ From Main menu → Commands → Loading → Definitions → Wind Load → Type 2 → Comments: Wind → Add. Click Type 2 → Add.

Wind Definition screen will appear. Intensity → Select type 2.

Intensity	Height
1.8	10
2.0	15
2.10	20

→ Add.

Exposure → Factor = 1.0 → Add → Close.

Click the exposure and Select node cursor & entire structure → Assign.

STAAD TEXT MODE

TYPE 2 (WIND INTENSITY IN Z DIRECTION)

INTENSITY 1.8 2.0 2.1 HEIGHT 10 15 20

EXPOSURE 1.0 YRANGE 15.0 20.0

- **DEAD LOAD**

From Main menu → Commands → Loading → Primary Load →

Load case 1=Dead Load → Add → Close .

Select Load case1→Add

Self weight (dead load) Factor=--1 Direction ⊙ Y (Beams and columns) →

Select entire structure and self weight →Assign to view → Assign.

Floor Load (Dead Load of slab) Pressure=--4.00 (Self wt =3.00 KN/m² +
Floor finish=1.0 KN/m²)

Define Y range Mini.=6.50(2.50+4.0). Max.=18.50

Define X range Mini.=0. Max.=12.0

Define Z range Mini.=0 Max.=10.0 → Add

Member Load (Wall Loads) →Uniform Force w1=--12 KN/m (9" wall load)

→Add→ Member load → w1=--6 KN/m(4½" wall Load) → Add→ Close→

Assign 9" wall load to external beams and 4½" wall load to internal beams
in Top view by selecting each floor separately

LIVE LOAD

Select load case 2 (live load) → Add→ Floor load (Live load on slab)

Pressure=--3.00(LIVE LOAD FOR OFFICE BUILDING)

Define Y range Mini.=6.50(2.50+4.0). Max.=18.50

Define X range Mini.=0. Max.=12.0

Define Z range Mini.=0 Max.=10.0 → Add → Close→

WIND LOAD:

*APPLYING THE ABOVE DEFINED WIND LOAD

Click the Load case details in data area → Add.

Number 3 Loading type: wind

Title: **Wind X +ve**→ Add.

Click load case 3 → Add

Wind load→ ⊙X direction Factor=1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 4 loading type: wind

Title: **Wind X -ve** → Add

Click Load case 4 wind X -ve → Add.

Wind load ⊙X direction Factor=-1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 5 Loading type: wind

Title: **Wind Z +ve** → Add

Click Load case 5 wind Z +ve → Add.

Wind load ⊙Z direction Factor=1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 6 loading type: wind

Title: **Wind Z -ve** → Add

Click Load case 6 wind Z -ve → Add.

Wind load ⊙Z direction Factor=-1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

AUTO LOAD COMBINATION

Main menu → Commands → Loading → Edit Auto Loads → Select Indian Code

Select : New category say Wind

No. of rule :4

Rule No.	Dead	Live	Wind
1	1.5	1.5	--
2	1.5	--	1.5
3	1.2	1.2	1.2
4	0.9	---	1.5

Click Update Table → Close.

Main menu → Commands → Loading → Auto Load Comb (OR)

Click Load case details → Add → Auto Load Combination.

Select : Indian code

Category: Wind → Generate loads→Add→ Close.

Main menu→Commands→Loading→ Load List (selecting Design loads) →

Load List

Available Load		Load list
1.DL	>	Load Comb 7 to 19
2.LL	>>	
3.WIND X+ve	<	
4.WIND X-ve	<<	
5.WIND Z+ve		
6.WIND Z-ve		
7. 1.5(DL+LL)		
8. 1.5(DL+W LX+)		
9. 1.5(DL-W LX-)		
10. 1.5(DL+W LZ+)		
11. 1.5(DL-W LZ-)		
12. 1.2(DL+LL+W LX+)		
13. 1.2(DL+LL+W LX-)		
14. 1.2(DL+LL+W LZ+)		
15. 1.2(DL+LL+W LZ-)		
16. 0.9DL+1.5W LX+		
17. 0.9DL+1.5W LX-)		
18. 0.9DL+1.5W LZ+)		
19. 0.9DL+1.5W LZ-)		

→O.k. → Close.

3. Analysis :

From Main menu → Commands – Analysis → Perform Analysis → ☉ No print → OK.

4. **Post Analysis Print:**

From Main Menu → Commands → Post Analysis Print → Support Reactions → ☉ To view → OK.

7. **Design :**

From Main menu → Commands → Design → Concrete Design → Current code=IS 456 → From Main menu → Tools → Set current input unit

Length =mm; Force = N → OK.

Define parameters → FC =20 → Add

FYMAIN= 415 → Add

FYSEC =415 → Add.

Ratio=3% → Add.

Highlight Fc=20 and select the entire structure →☉ Assign to selected members →Assign→Yes.

Highlight Fymain=415 and select the entire structure →☉ Assign to selected members →Assign→Yes.

Highlight Fysec=415 and select the entire structure →☉ Assign to selected members →Assign→Yes.

Highlight Ratio=3% and select the entire structure →☉ Assign to selected members →Assign→Yes →Close.

Click commands →Click Design Beam →Add. Click Design column →Add.

Click concrete take off →Add.

Highlight Design beam →Main menu→Select→Beam parallel to→X and Select→Beam parallel to Z →☉ Assign to selected members →Assign→Yes →Close.

MAXSEC =8 → Assign

MINMAIN =12 → Assign

MINSEC =8 - Assign → Close.

De select all members and select beams only (Main menu →Select →Beams parallel to X and Beam parallel to Z axis) → Commands (in data area) (Concrete Design) → Design beam → Assign → Deselect all beams and select columns only(Main menu→Select→ Beam parallel to Y axis) → Design column → Assign→ Take off → Assign → Close.

Note: Save the File and Run the Program.

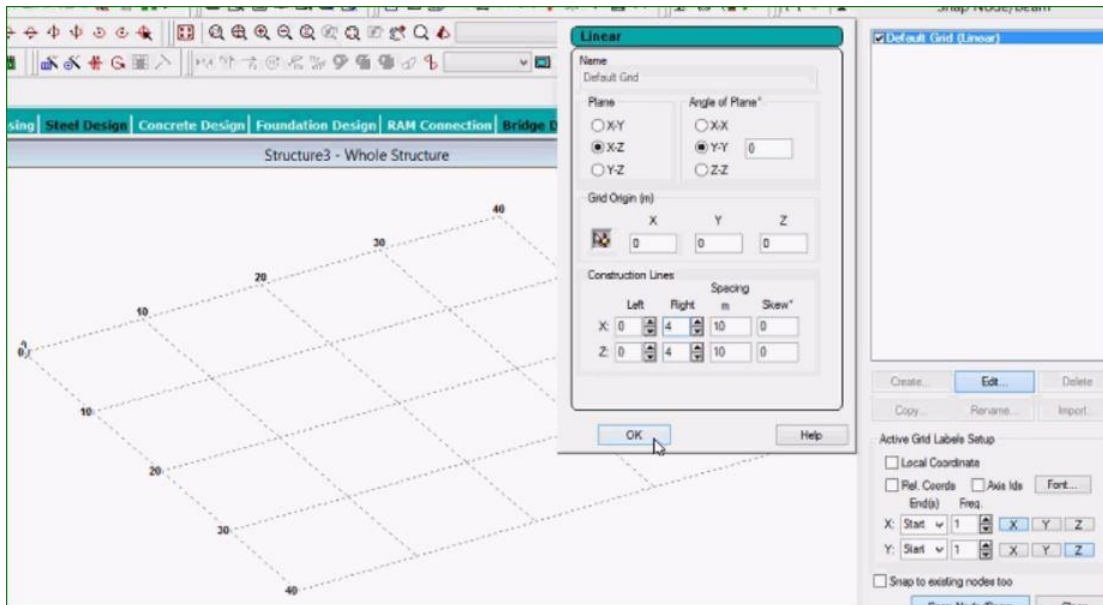
5. Analysis :

From Main Menu → ` Analysis → Run Analysis → Run Analysis → Done.

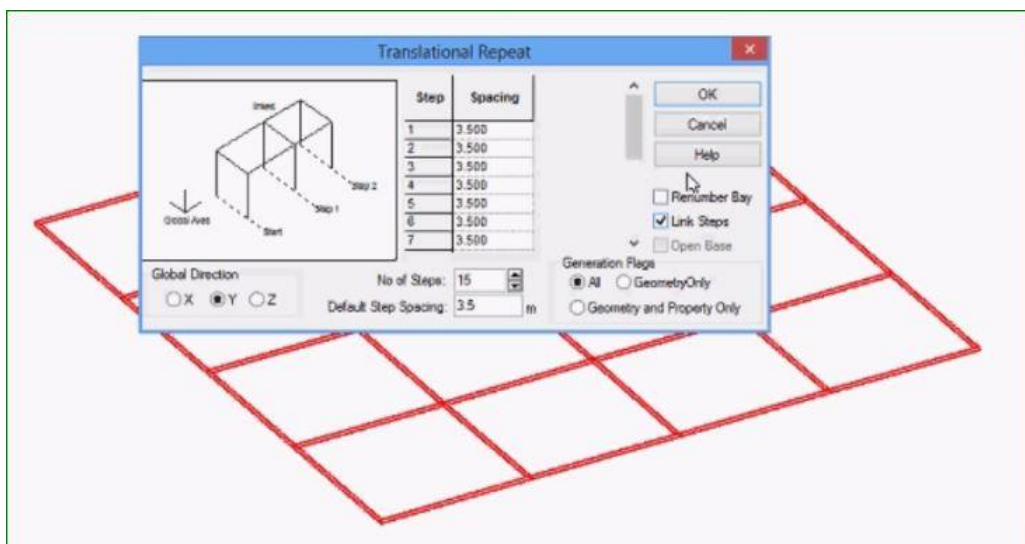
Select the member and double click on it → Shear bending → Close.

Design Procedure To Calculate Wind Load Intensity:

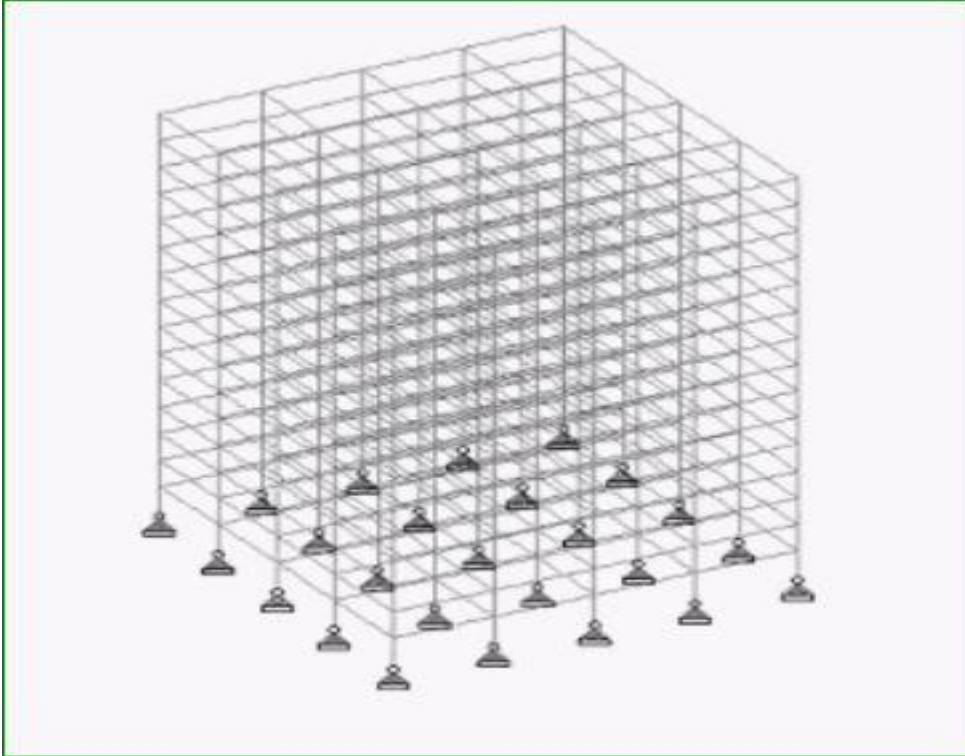
1. Open STAAD.Pro V8i.
2. Click **New Project** and set the units as **Kilo Newton & Meter**.
3. Open the Grid and Form the following grid.



4. By using the Snap/Node Beam Add the members and select the members.
5. Now use the [Translational Repeat](#) option to build the structure.



6. Assign the respective support for the structure.

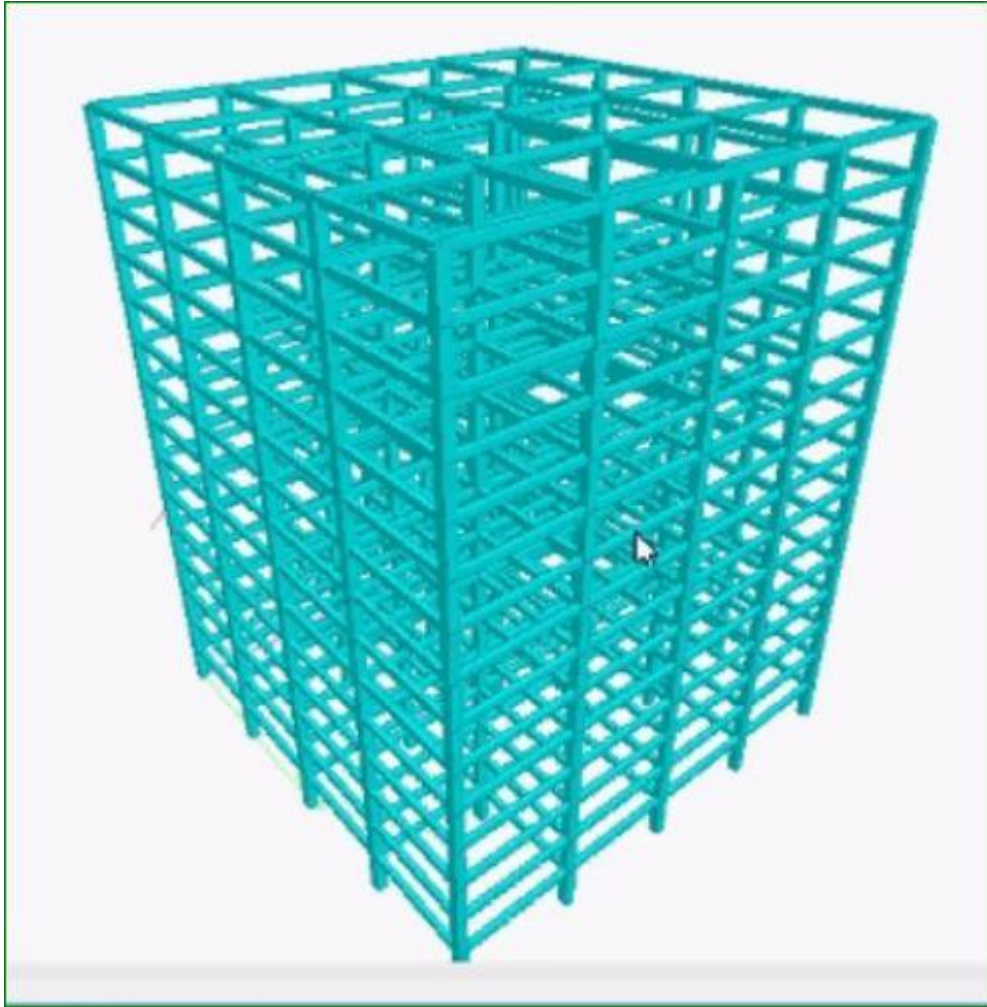


7. Assign the suitable member property for the model.

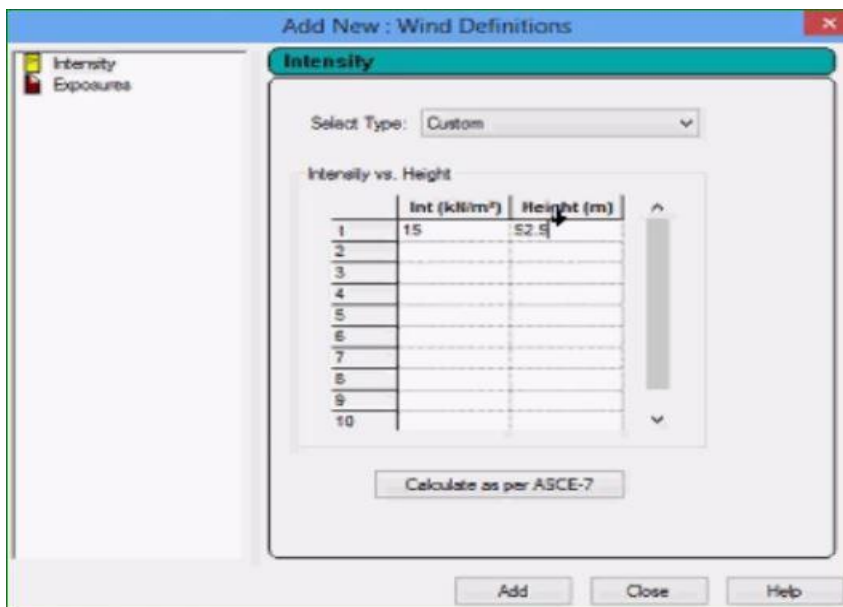
For Column = 0.75×0.75 m

For Beam: YD = 0.60; ZD = 0.40

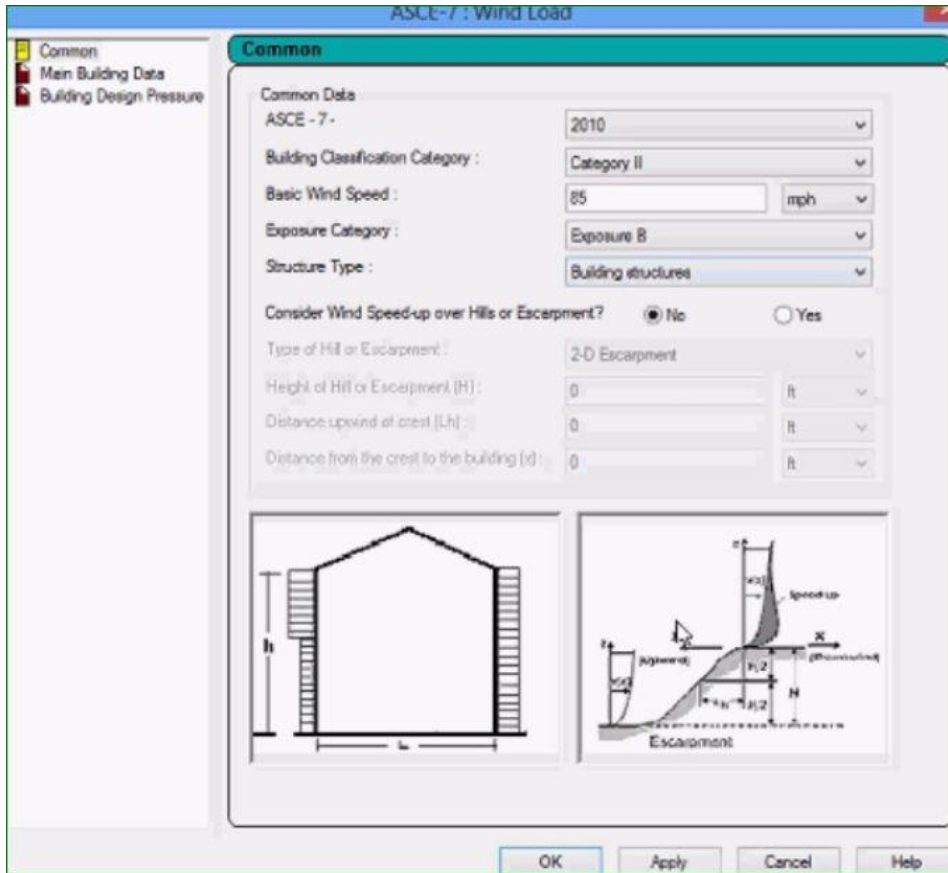
8. Now open the structure in 3D rendering.



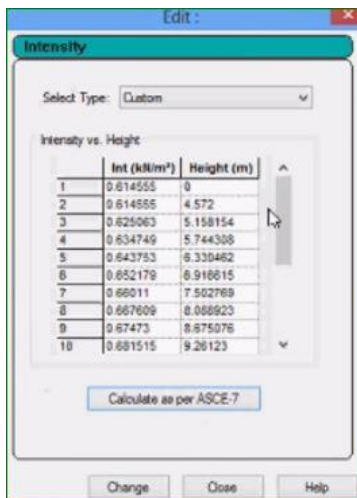
9. Then Wind Definition in Load Case.



10. Now click Calculate as per ASCE-7 button. Now the select respective code and type of building. Click OK



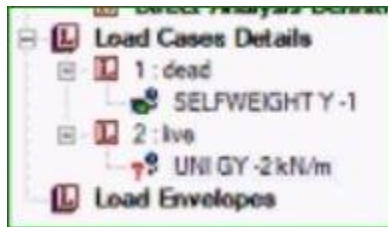
11. Wind load is calculated according the varying height of the structure. Click Close button.



12. Add the suitable factor of exposure.

13. Assign the Exposure against the structure.

14. Assign the following Load Cases.

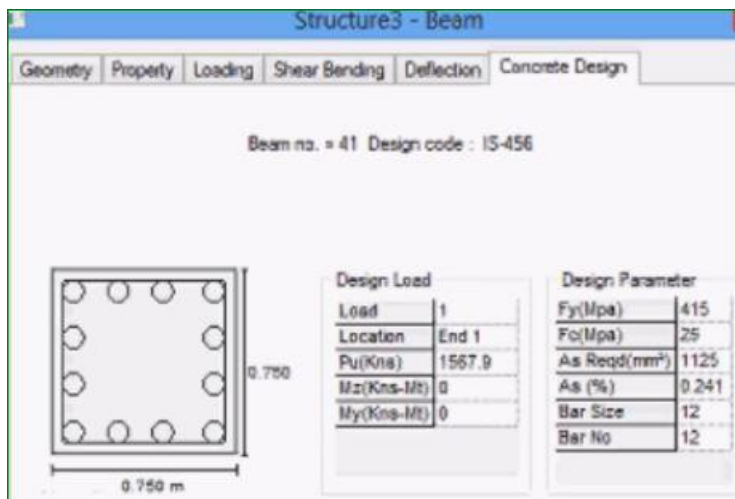


15. Perform Analysis and Run Analysis.

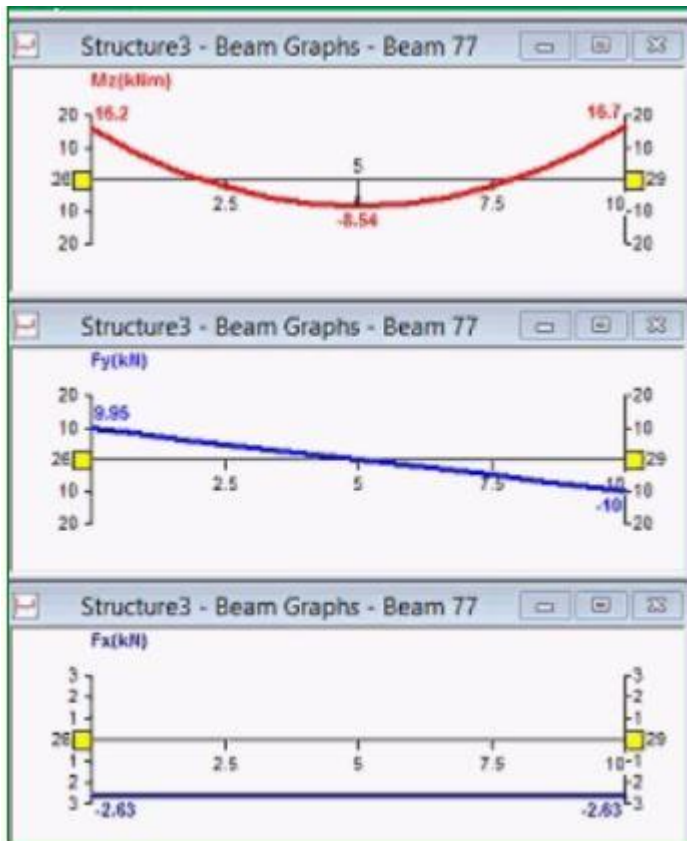
16. According to the result values provide the suitable [concrete design](#) for the structure.

17. Again Run the Analysis.

18. Now you get the concrete design of the elements.



19. In Post Processing, You can get the Bending Moment and Shear Force values.



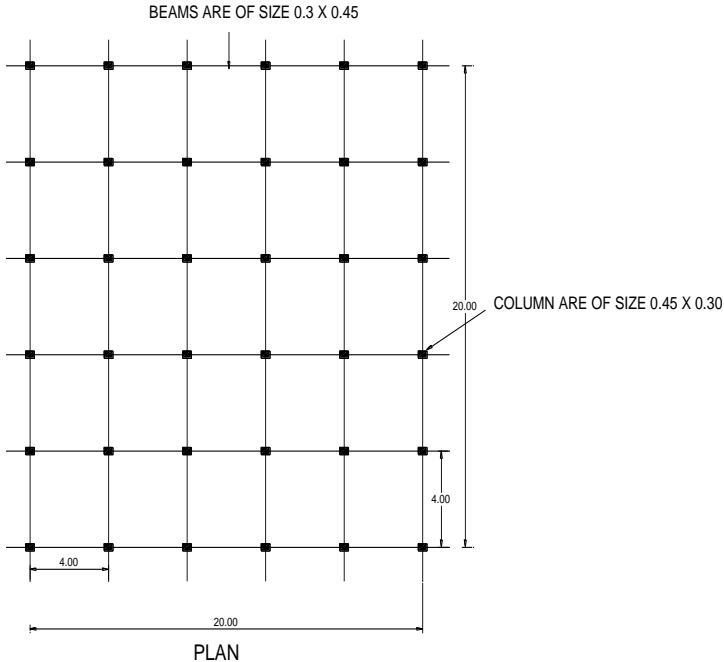
Example 2

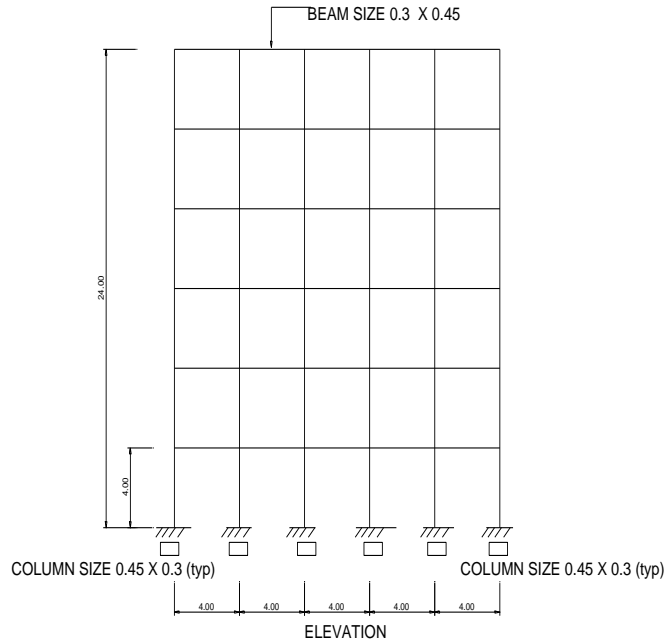
Data:

Building plan: Length 20m (5 bays each 4m centre to centre)

Width:20m (5 bays each 4m centre to centre)

Height 24m (6 bays each 4m)





1. Geometry Creation :

Open STAAD.Pro V8i. New Project → Select Space → Length =meters;

Force=KN; File Name= Five storied Building → Next

Select Open Structure Wizard → Finish.

Change to Frame Models from Truss models → Select Bay Frame and double click on it.

Length (x)= 12.0m No. of bays =4 Each Bay =3.0

Height (y)= 18.5m No. of bays =6 Click Bay No.1 =2.50

Bay No.2 =4.0m ; Bay No.3 =3.0m ; BayNo. 4=3.0m;

Bay No.5 =3.0m; Bay No.6=3.0m

Width (z)= 10.0m No. of bays =2 Each Bay = 5.0

Transfer model→ click yes→ o.k.

Go to Front view icon (first view)

2.Member Properties:

Front view → Select beam along X axis for top 5 levels by windowing → Go to Isometric view and confirm the selection of beams in x direction of top

5 levels of beams → From main menu → Commands → Member property → Prismatic → Rectangular → YD=0.30 ZD=0.23 → Assign → De select the member. Go to side view → Select beam along Z direction for top 5 levels by windowing → Go to Isometric view and confirm the selection of beams in z direction of top 5 levels of beams → From main menu → Commands → Member property → Prismatic → Rectangular → YD=0.45 ZD=0.23 → Assign → De select the member and select the beam at lower level (Plinth beam along x and z direction) → From Main menu → Commands → member property → Prismatic → Rectangular YD=0.4 ZD=0.23 → Assign → Close → De select the member and select column member (Go to top view) window the Corner column on four ends (type 1 column 0.6m x 0.3m) → From Main menu → Commands → Member Property → Prismatic YD=0.60 ZD=0.30 → Assign → Confirm by viewing the column in Isometric view. Go to top view → Window the intermediate column of end rows (type 2 column) → From Main menu → Commands → Member Property → Prismatic → Rectangular YD=0.6m ZD=0.30 → Assign → Confirm by viewing the column in Isometric view. Window the internal column (type 3 column YD=0.60 ZD=0.3) → From Main menu → Commands → Member Property → Prismatic YD=0.60 ZD=0.30 → Assign → Confirm by viewing the column in Isometric view.

To confirm whether any missing property

Main menu → Select → Missing attributes → Missing Property → No entity with missing property is found → o.k.

3.Supports:

Change to Node cursor (joint) and select the nodes (joints).

(If more than one node use CTRL key and select the nodes). Go to front view → Select the bottom most nodes for supports → From Main menu → Commands → Support specification → Fixed → Assign → Close → De select the nodes and change to beam cursor.

4.Loading :

*DEFINE WIND LOAD: X DIRECTION

- (2) From Main menu →Commands→ Loading → Definitions → Wind Load→ Type 1→ Comments: Wind → Add. Click Type 1 →Add.
Wind Definition screen will appear. Intensity → Select type1.

Intensity	Height
1.8	10
2.0	15
2.10	20

→ Add.

Exposure →Factor = 1.0 → Add → Close.

Click the exposure and select node cursor & entire structure → Assign.

Note:

STAAD TEXT MODE

DEFINE WIND LOAD

TYPE 1(WIND INTENSITY IN X DIRECTION)

INTENSITY 1.8 2.0 2.1 HEIGHT 10.0 15.0 20.0

EXPOSURE 1.0 YRANGE 15.0 20.0

Z DIRECTION

_ From Main menu →Commands→ Loading → Definitions→ Wind Load→

Type 2→ Comments: Wind → Add. Click Type 2 →Add.

Wind Definition screen will appear. Intensity → Select type 2.

Intensity	Height
1.8	10
2.0	15
2.10	20

→ Add.

Exposure → Factor = 1.0 → Add → Close.

Click the exposure and Select node cursor & entire structure → Assign.

STAAD TEXT MODE

TYPE 2 (WIND INTENSITY IN Z DIRECTION)

INTENSITY 1.8 2.0 2.1 HEIGHT 10 15 20

EXPOSURE 1.0 YRANGE 15.0 20.0

• **DEAD LOAD**

From Main menu → Commands → Loading → Primary Load →

Load case 1=Dead Load → Add → Close .

Select Load case1→Add

Self weight (dead load) Factor=--1 Direction ⊙ Y (Beams and columns) →

Select entire structure and self weight →Assign.

Floor Load (Dead Load of slab) Pressure=--4.00 (Self wt =3.00 KN/m² +
Floor finish=1.0 KN/m²)

Define Y range Mini.=6.50(2.50+4.0). Max.=18.50

Define X range Mini.=0. Max.=12.0

Define Z range Mini.=0 Max.=10.0 → Add

Member Load (Wall Loads) →Uniform Force W1=--12 KN/m (9" wall load)

→Add→ Member load → Uniform force w1=--6 KN/m(4½" wall Load) →

Add→ Close→ Assign 9" wall load to external beams and 4½" wall load to
internal beams in Top view by selecting each floor separately

LIVE LOAD

Select load case 2 (live load) → Add→ Floor load (Live load on slab)

Pressure=--3.00(LIVE LOAD FOR OFFICE BUILDING)

Define Y range Mini.=6.50(2.50+4.0). Max.=18.50

Define X range Mini.=0. Max.=12.0

Define Z range Mini.=0 Max.=10.0 → Add → Close→

WIND LOAD:

*APPLYING THE ABOVE DEFINED WIND LOAD

Click the Load case details in data area → Add.

Number 3 Loading type: wind

Title: **Wind X +ve** → Add.

Click load case 3 → Add

Wind load → ⊙X direction Factor=1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 4 loading type: wind

Title: **Wind X -ve** → Add

Click Load case 4 wind X -ve → Add.

Wind load ⊙X direction Factor=-1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 5 Loading type: wind

Title: **Wind Z +ve** → Add

Click Load case 5 wind Z +ve → Add.

Wind load ⊙Z direction Factor=1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

Click the Load case details → Add.

Number 6 loading type: wind

Title: **Wind Z -ve** → Add

Click Load case 6 wind Z -ve → Add.

Wind load ⊙Z direction Factor=-1.

Y range Minimum: 2.50 maximum:18.5 → Add. → Close.

AUTO LOAD COMBINATION

Main menu → Commands → Loading → Edit Auto Loads → Select Indian Code

Select : New category say Wind

No. of rule :4

Rule No.	Dead	Live	Wind
1	1.5	1.5	--
2	1.5	--	1.5
3	1.2	1.2	1.2
4	0.9	---	1.5

Click Update Table →Close.

Main menu→Commands → Loading →Auto Load Comb (OR)

Click Load case details → Add → Auto Load Combination.

Select : Indian code

Category: Wind → generate loads→Add→ Close.

Main menu→Commands→Loading→ Load List (selecting Design loads) →

Load List

Available Load		Load list
1.DL	>	Load Comb 7 to 19
2.LL	>>	
3.WIND X+ve	<	
4.WIND X-ve	<<	
5.WIND Z+ve		
6.WIND Z-ve		
7. 1.5(DL+LL)		
8. 1.5(DL+W LX+)		
9. 1.5(DL-W LX-)		
10. 1.5(DL+W LZ+)		
11. 1.5(DL-W LZ-)		
12. 1.2(DL+LL+W LX+)		
13. 1.2(DL+LL+W LX-)		

14. 1.2(DL+LL+WLZ+)		
15. 1.2(DL+LL+WLZ-)		
16. 0.9DL+1.5WLX+		
17. 0.9DL+1.5WLX-)		
18. 0.9DL+1.5WLZ+)		
19. 0.9DL+1.5WLZ-)		

→O.k. → Close.

5. Analysis :

From Main menu → Commands - Analysis → Perform Analysis → No print → OK.

6.Post Analysis Print:

From Main Menu → Commands → Post Analysis Print → Support Reactions → To view → OK.

7. Design :

From Main menu → Commands → Design → Concrete Design → Current code=IS 456 → From Main menu → Tools → Set current input unit Length =mm; Force = N → OK.

→ Select all the member → Define parameters

ELY =0.85 → Assign

ELZ =0.85 → Assign

FC =20 → Assign

FYMAIN= 415 → Assign

FYSEC =415 → Assign

MAXMAIN =16 → Assign

MAXSEC =8 → Assign

MINMAIN =12 → Assign

MINSEC =8 - Assign → Close.

Deselect all members and select beams only (Main menu → Select → Beams parallel to X and Beam parallel to Z axis) → Commands (in data area) (Concrete Design) → Design beam → Assign → Deselect all beams and select columns only (Main menu → Select → Beam parallel to Y axis) → Design column → Assign → Take off → Assign → Close.

Note: Save the File and Run the Program.

8. Analysis:

From Main Menu → ` Analysis → Run Analysis → Run Analysis → Done.

Select the member and double click on it → Shear bending → Close.