DEPARTMENT OF CIVIL ENGINEERING

LABORATORY MANUAL

FOR

CADD LAB& DESIGN &DETAILING PRACTICE LAB, 6TH SEMESTER

C. V. RAMAN POLYTECHNIC

(Affiliated to SCTE & VT and Approved by Govt. Odisha)
BIDYANAGAR, MAHURA, JANLA, BHUBANESWAR-752054, ODISHA

Vision:

Civil engineering department is committed to impart knowledge and excellence in civil Engineering to the students and to produce civil engineers of high calibre, technical skills and ethical values to meet current and future challenges.

Mission:

M1: To produce civil engineers with quality technical skills aligned with industry needs to solve real life problems of the society.

M2: To create teaching learning environment for students to acquire knowledge as per need and to motivate towards entrepreneurship and to pursue higher studies.

M3: To serve construction industries, civil engineering profession and the community at large through dissemination of knowledge and technical services to improve quality of life and enhance employability.

M4: To inculcate self-learning attitude and professionalism.

Program Educational Objectives (PEOs)

PEO1- To analyze in civil engineering profession or Higher education by acquiring thorough knowledge and concepts in fundamentals of engineering.

PEO2-To Apply knowledge and skills to real life problems and there by rendering safe and economical structures against natural calamities and also environmentally sustainable and useful to society.

PEO3- To understand entrepreneurial endeavors and to develop effective communication skill and passion for learning.

Program Specific outcomes (PSO)

PSO1- Able to meet the needs of public in the design and execution of quality construction work considering health, safety, cultural and environmental factors.

PSO2- Analyze and design regular and complex structures applying knowledge of building analysis software package.

PSO3- Able to work effectively as an individual or in a team having acquired leadership skills and manage projects in multidisciplinary environment.

DESIGN OF FRAMED STRACTURE USING STAAD Pro

Step 1

Open the staad pro

New → New file → Space

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

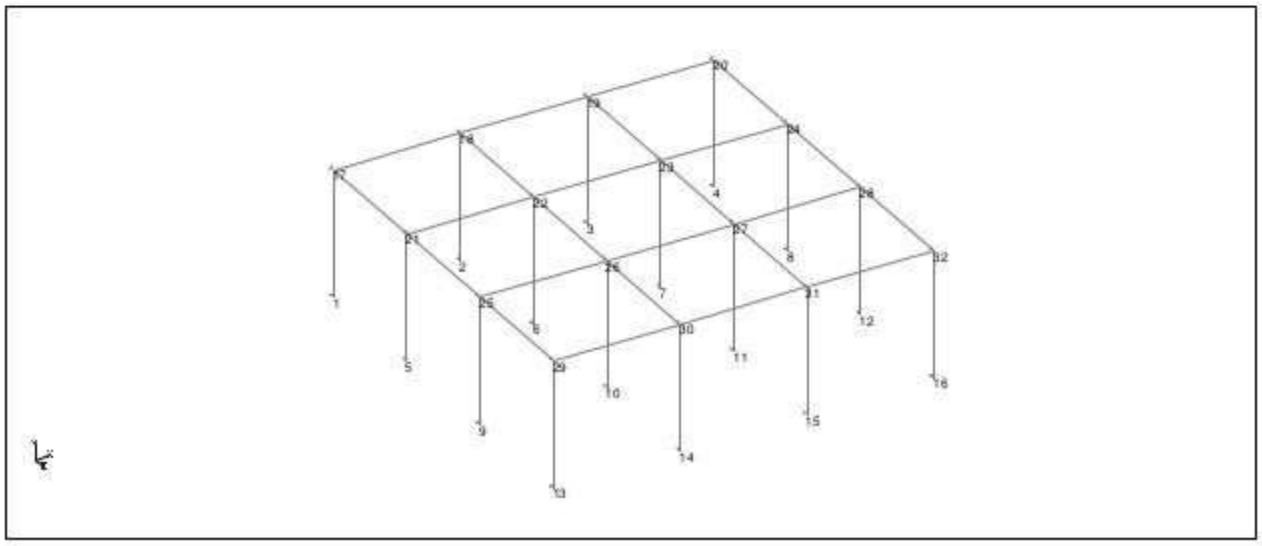
Add beam → **Finish**

Step 3

Click → geometry → **Nodes**

Node	x	y	z
1	m	m	m

Diagram



Nodes	x	y	z
1	0	0	0
2	3	0	0
3	6	0	0
4	9	0	0
5	0	0	3
6	3	0	3
7	6	0	3
8	9	0	3
9	0	0	6
10	3	0	6
11	6	0	6
12	9	0	6
13	0	0	9
14	3	0	9
15	6	0	9
16	9	0	9

Then close the node (**x**) → You see the node point on the screen → How to find the node numbers → Click mouse left side → **Click labels** → Diagrams → Structure → Node number & Node point → **Apply** → **ok**

Step 4

Select Node Crusher→ Fist selects all Node point

Click → Geometry → Translational repeat → 3D repeat → Global direction → **y** → No of steps **1** → Default step spacing **3m** → **ok**

to see the display of node.

Step 5

To join the node points

Click → Geometry → Add beam → Add beam from point to point

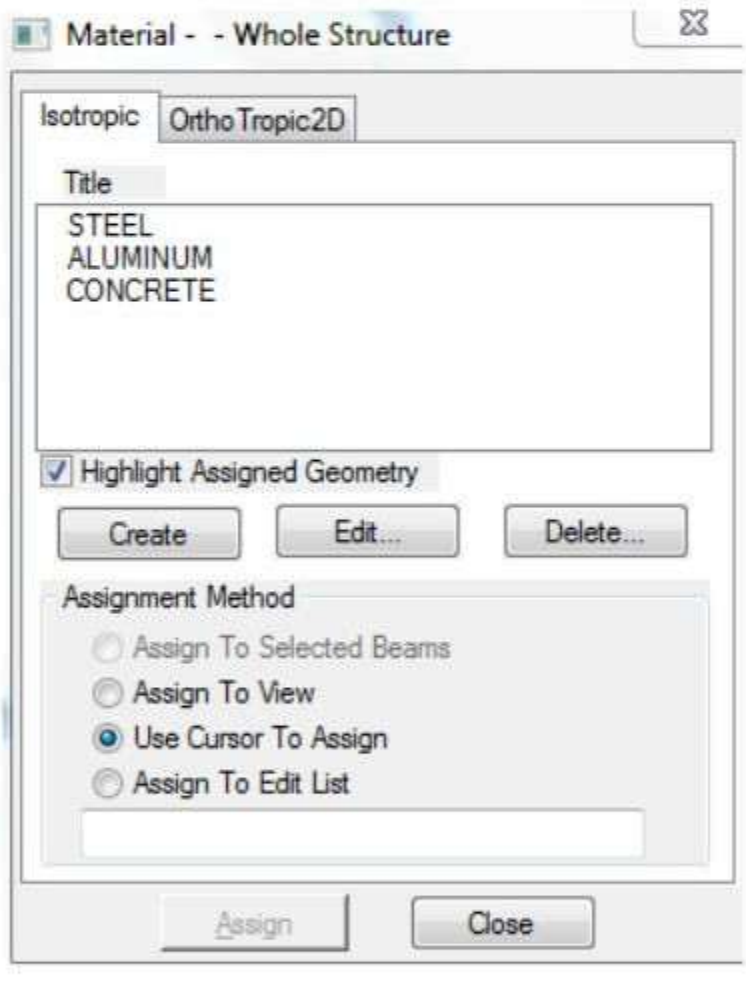
Select the node point and joint all node point like framed structure.

Step 6

- Material
- Property
- Support
- Load
- Analysis

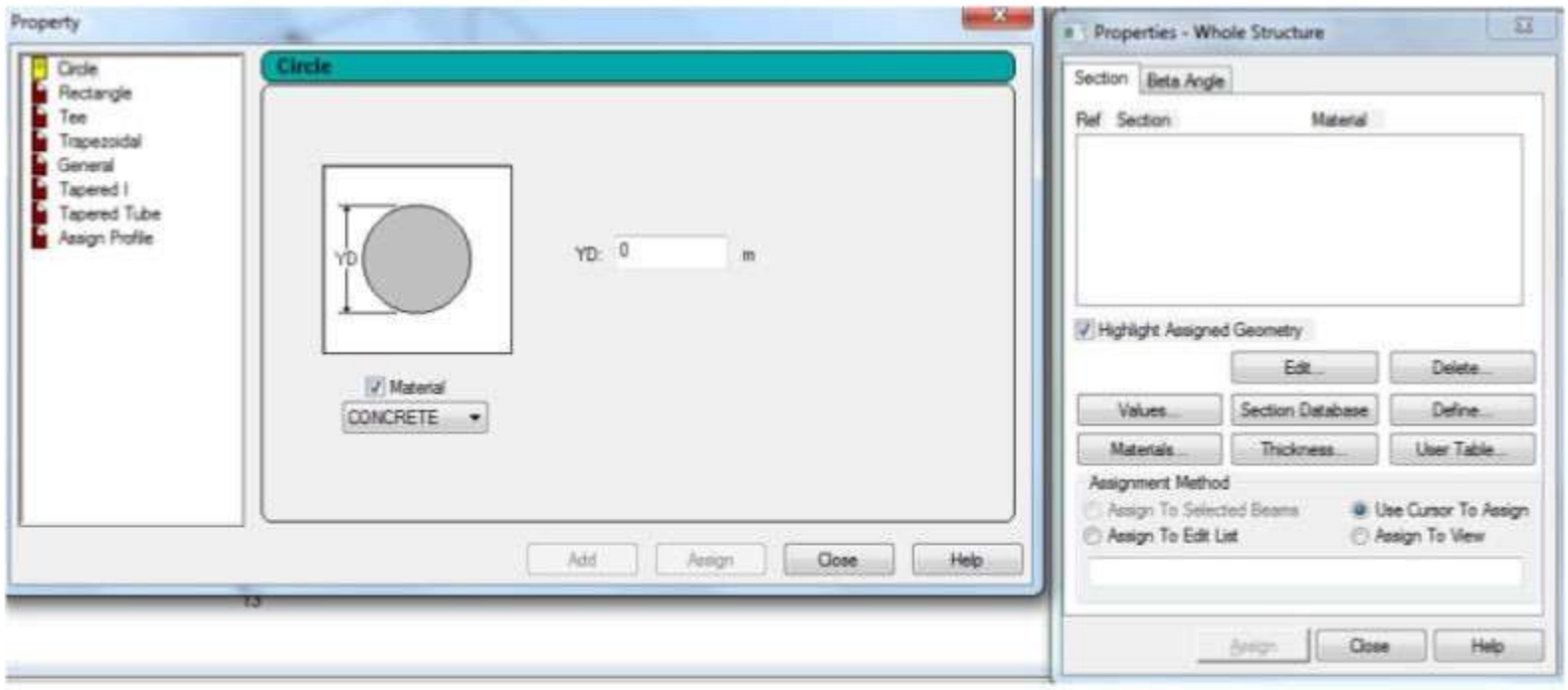
Material

Click→ Modeling → General→ Material → Material Whole structure → **click Concrete** →Assign to view → **Assign** → **Yes**



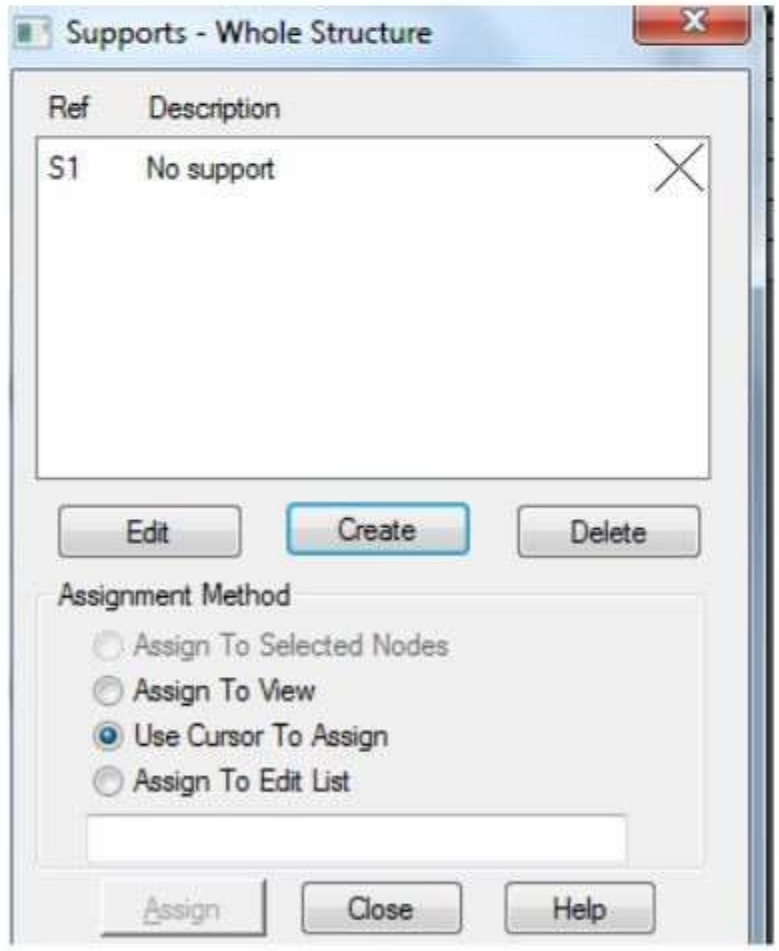
Property

Click → Modeling → General → property → Property Whole structure → Define → Property → Rectangle YD **.23m ZD .23m** → Add → Close → Assign to view → **Assign** → **Yes**



Support

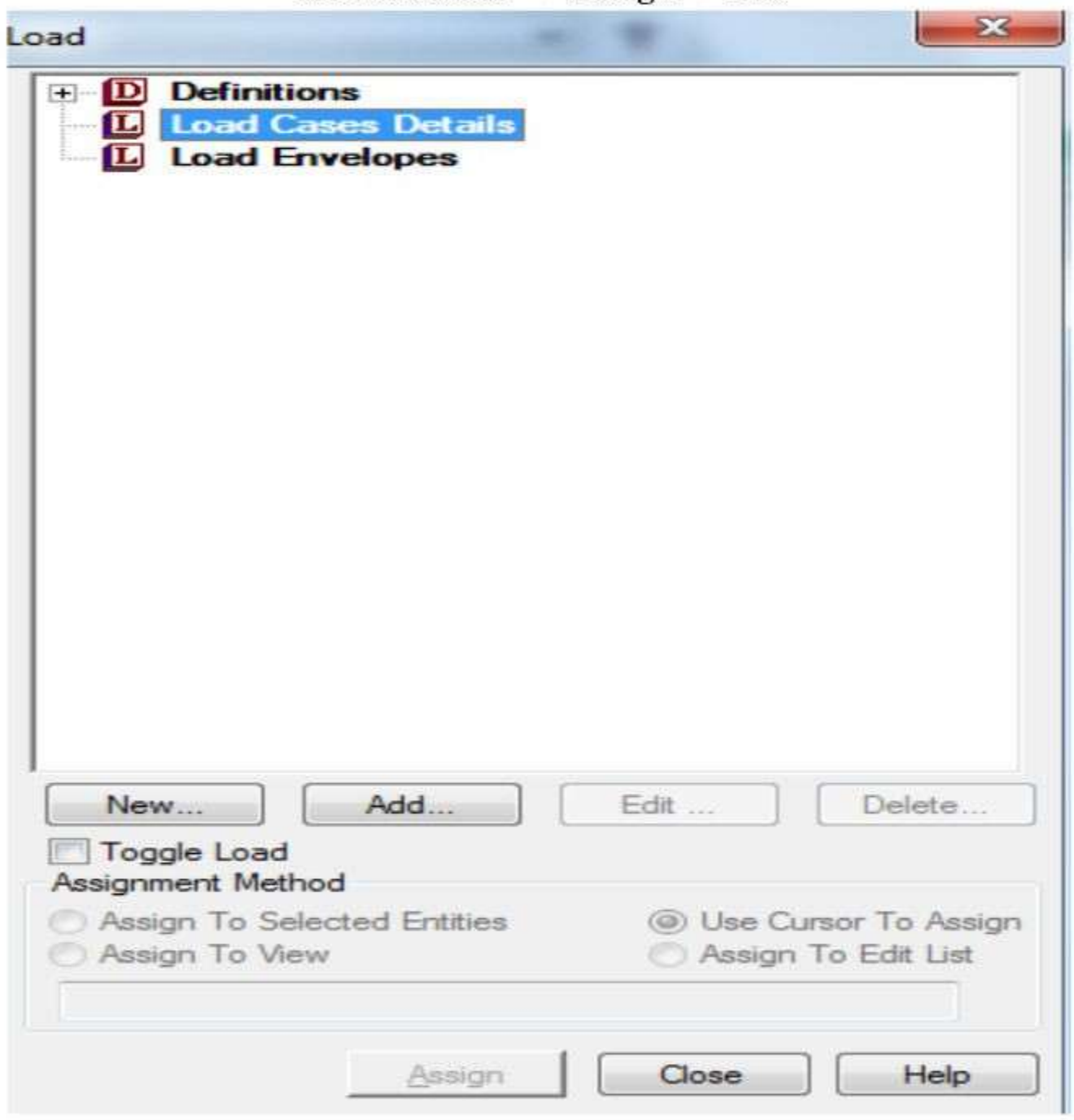
Click → Modeling → General → Support → Support Whole structure → Create → **pinned** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**

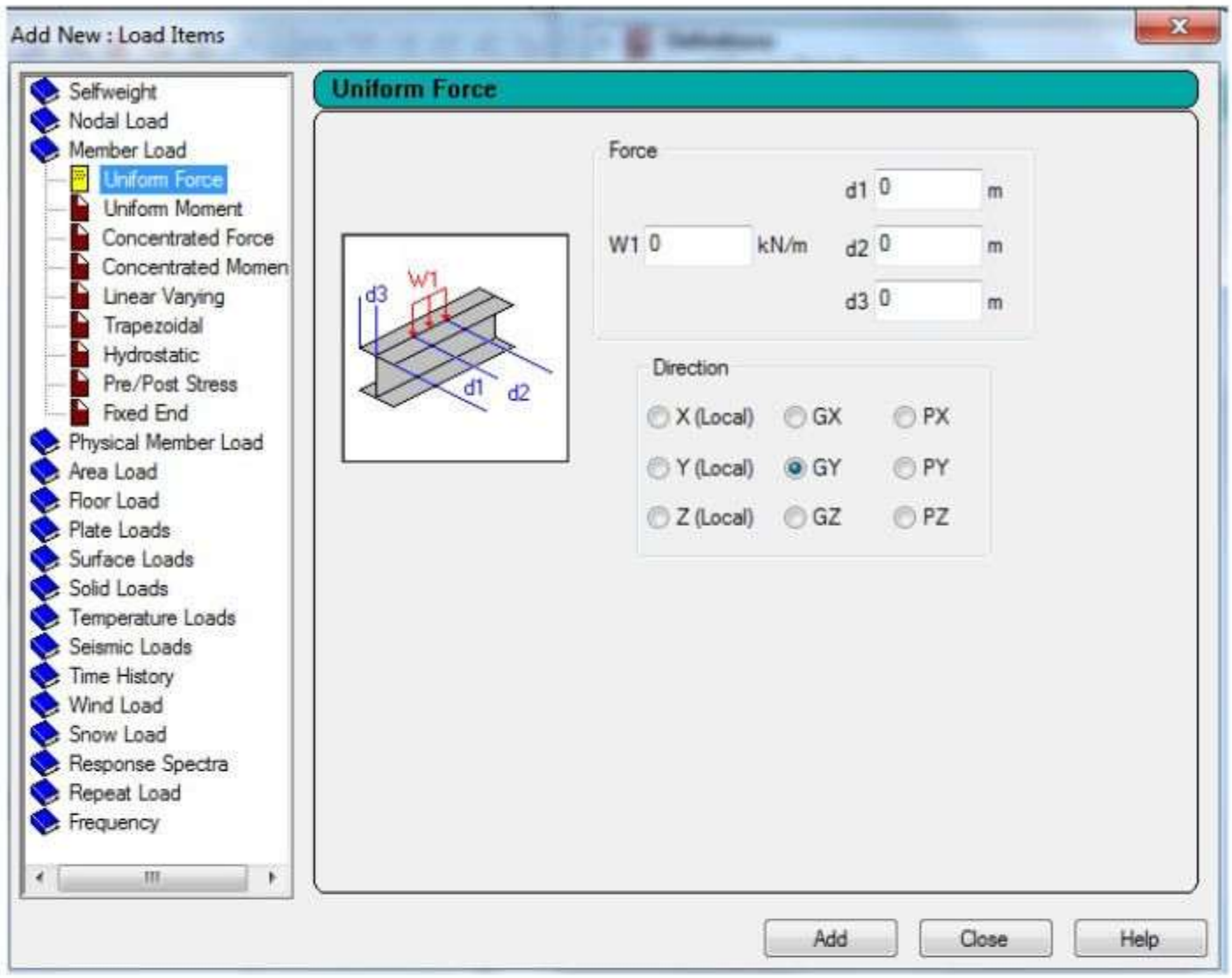


Loads

Click → Modeling → General → Load → Click load case details → Add → Add new load cases → **Add – Load case 1** → **Add- Load case 2** → Click Load case 1 Add **self-weight** → Add → Close → Click Load case 2 → Add →Member → load → **Uniform force** wl = -2 kN/m →**Add** → **Close**

Select load→ UGL GY -2 kN/m →Select →**Beam Parell to** → **X Z** → Assign selected beam → **Assign**→ **Yes**





ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

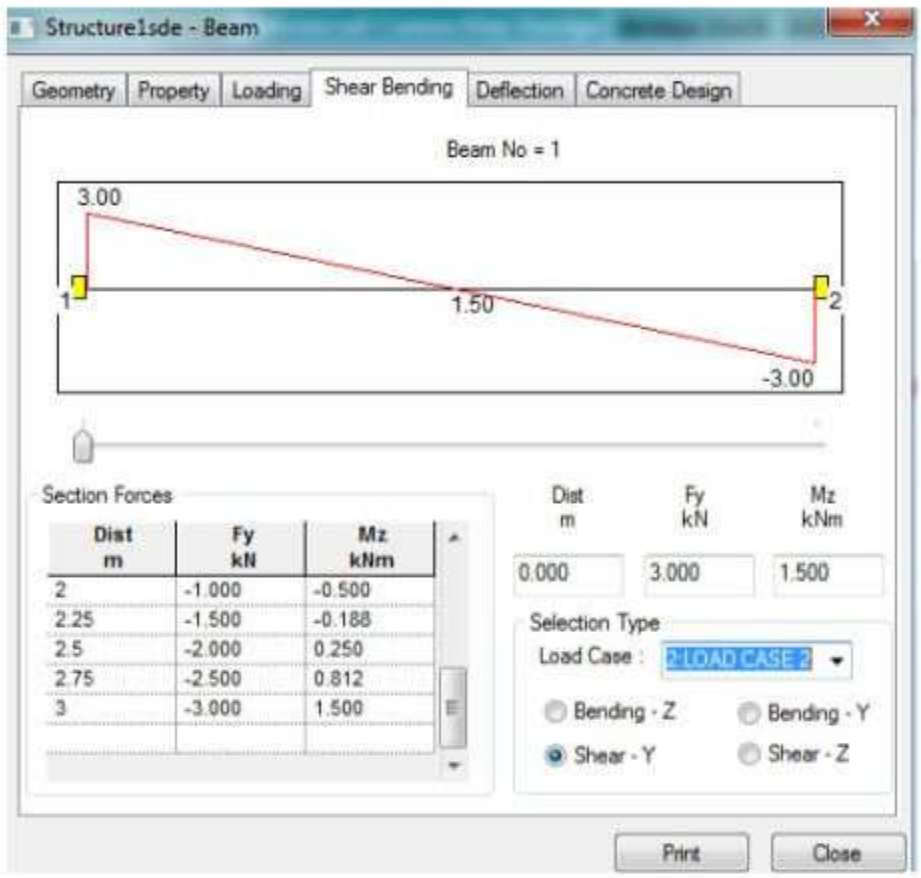
Click the beam see the result on beam

SHEAR FORCE DIAGRAM

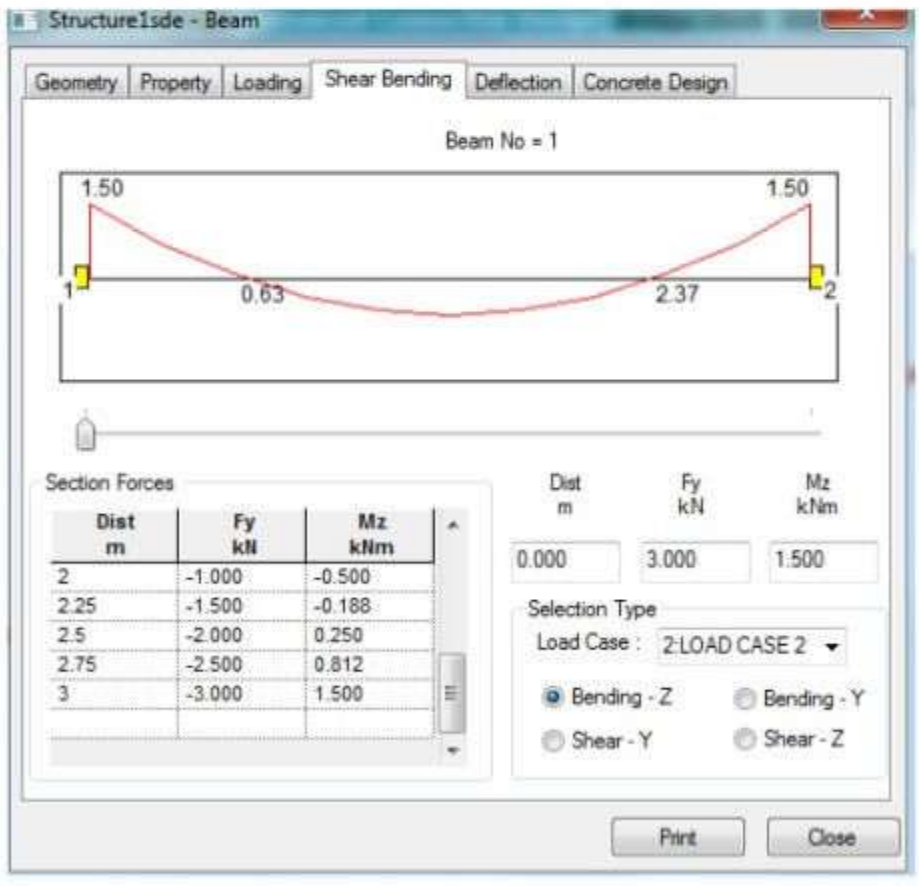
BENDING MOMEMT DIAGRAM

DEFLECTION DIAGRAM

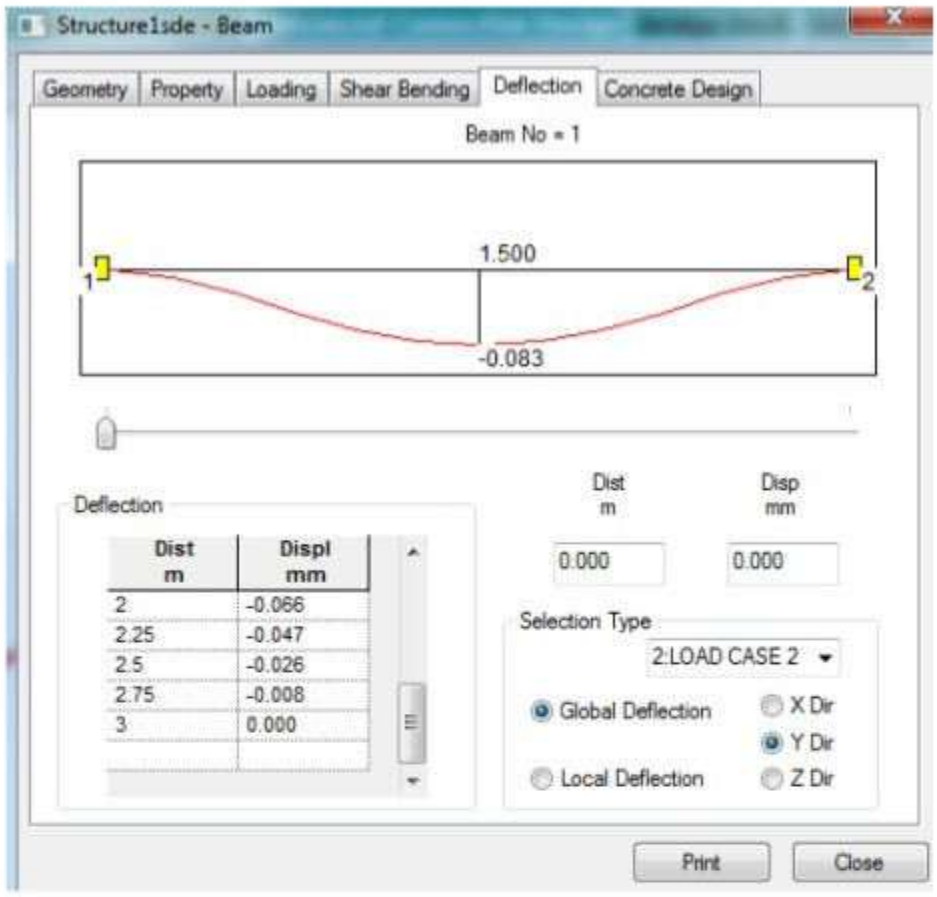
Shear force



Bending moment



Deflection



Step 6

Post processing

Click → Post processing → Result setup → Select load case → **Ok**

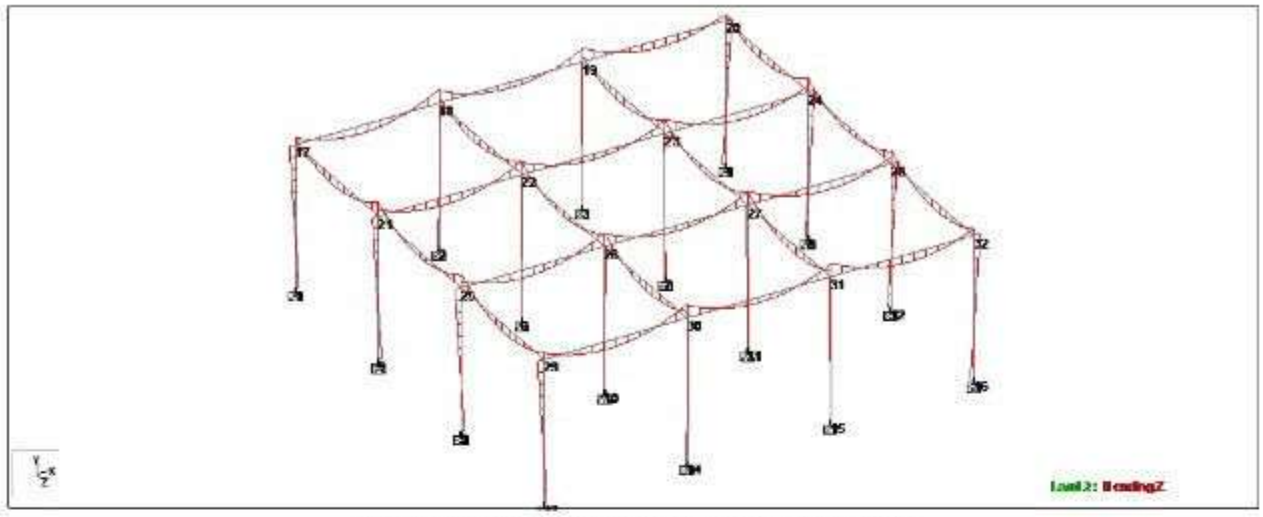
New screen will be displayed →Click → Result → Animation →**Deflection** → **Ok** → F12 to see full screen of deflection

Click→ Result→ **Bending moment** →Scroll the mouse → to see the whole structure bending moment diagram → Value to be noted

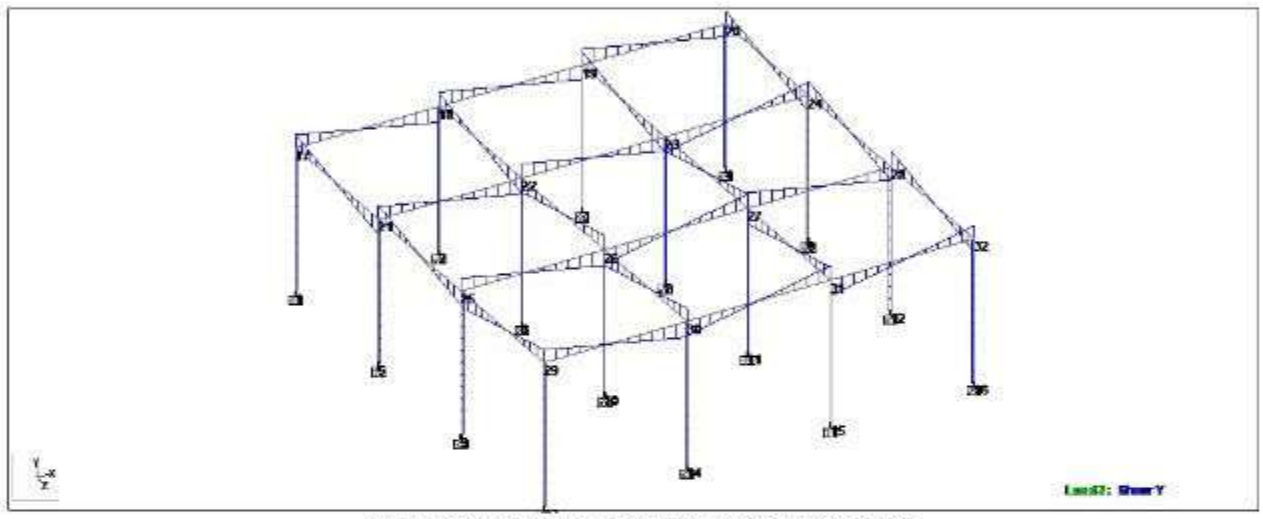
Click→ Result→ **Section displacement** → Scroll the mouse → to see the whole structure section displacement diagram →Value to be noted

Click → Result → **Beam stress** → Click→ Beam stresses → to click any one beam → Open 3d beam stress coutour displayed → Distance to be provided the beam → Add to stress table → Values to be noted

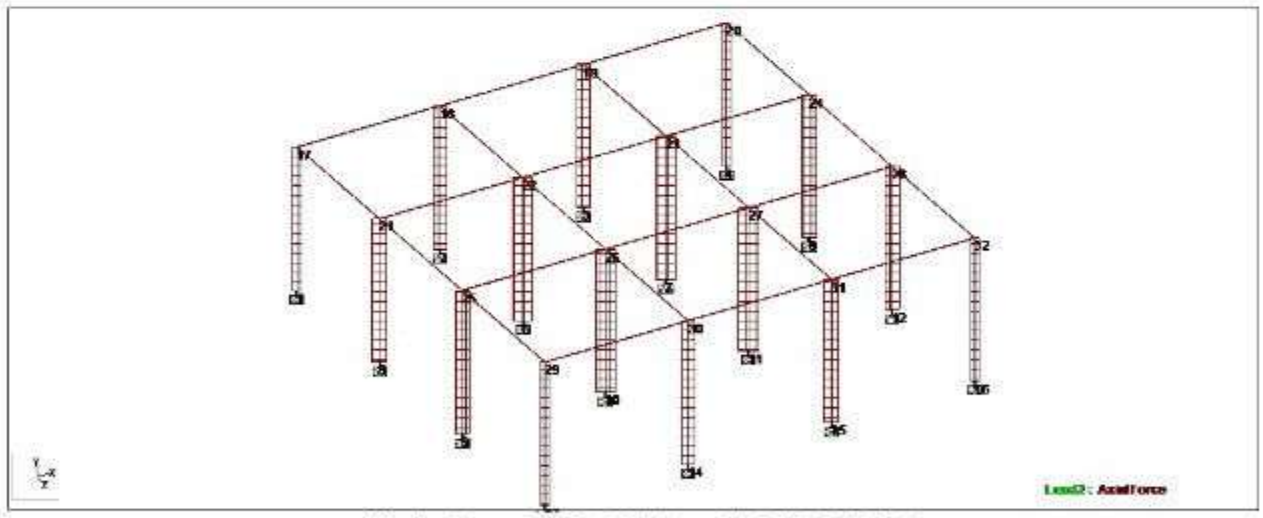
Beam →Graphs → **Beam graphs**→ diagram Show



Whole Structure Mz 6kNm:1m 2 LOAD CASE 2



Whole Structure Fy 10kN:1m 2 LOAD CASE 2



Whole Structure Fx 70kN:1m 2 LOAD CASE 2

STEP 7

DESIGN OF CONCRETE ELEMENTS BEAM & COLUMN

BEAM DESIGN

Click →Modeling → Click → Design→ Click → **Concrete** → Concrete design whole structure → **Current code IS 456** →Click → Selected parameters

- fc = compressive strength of concrete
- fy = yield strength of concrete
- Maximum Main reinforcement
- Minimum secondary reinforcement
- Track

Ok → Then change the unit **kN/m to N/mm^2** → Click → Tools → Set current input units → Change unite

Click → **Define parameter**

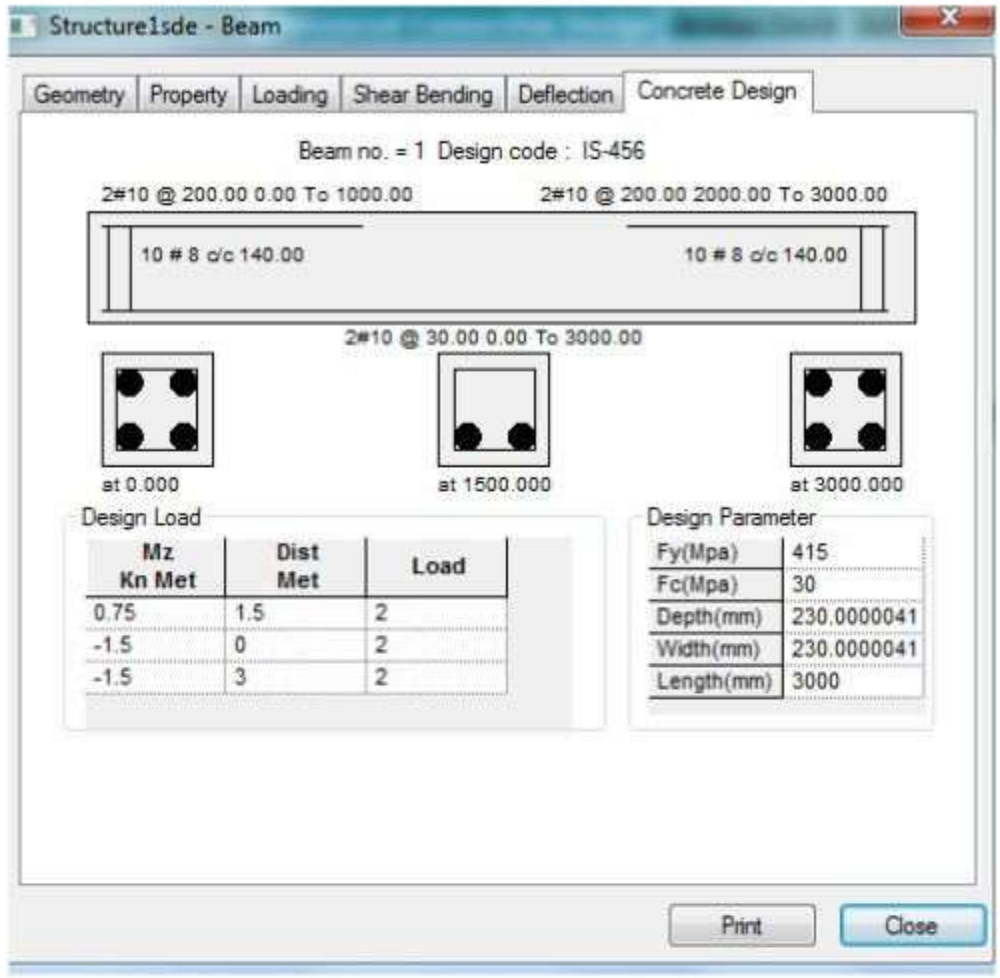
- fc = compressive strength of concrete = 30 N/mm^2 → Add
- fy = yield strength of concrete = 415 N/mm^2 →Add
- Maximum Main reinforcement = 32 mm dia→ Add
- Minimum secondary reinforcement = 12 mm dia → Add
- Track = 3→ Add

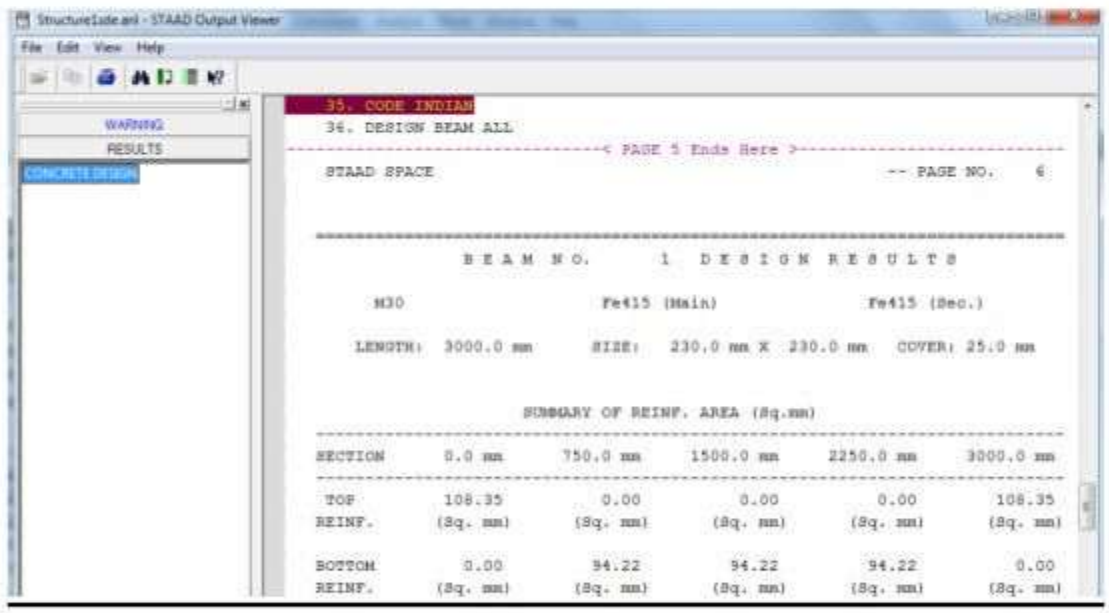
Assign → close

First select beam member's → Click→ Select beam parallel to → x **z**

Click → **commands** → Click → Beam design →Add → Assign → Close → **Analysis** → Run analysis → save → Close →View output file → Done →To see the result concrete design result

Click the beam member on mouse to see the → **Beam design**





DESIGN OF COLUMN

Click →Modeling → Click → Design→ Click → **Concrete** → Concrete design whole structure → **Current code IS 456** →Click → Selected parameters

- fc = compressive strength of concrete

- fy = yield strength of concrete
- Maximum Main reinforcement
- Minimum secondary reinforcement
- Track

Ok → Then change the unit **kN/m to N/mm²** → Click → Tools → Set current input units → Change unite

Click → **Define parameter**

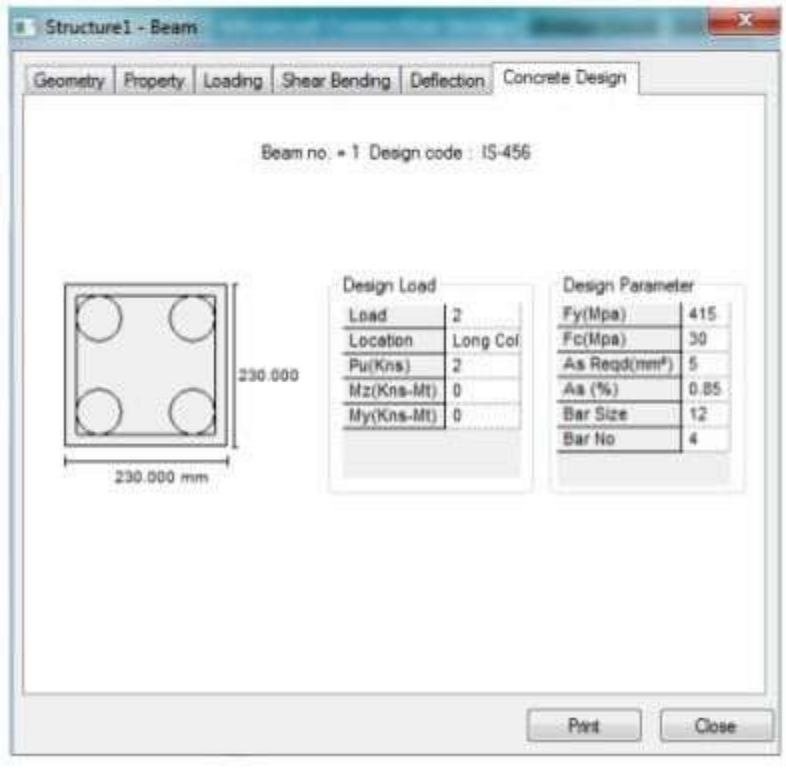
- fc = compressive strength of concrete = 30 N/mm² → Add
- fy = yield strength of concrete = 415 N/mm² →Add
- Maximum Main reinforcement = 32 mm dia→ Add
- Minimum secondary reinforcement = 12 mm dia → Add
- Track = 3→ Add

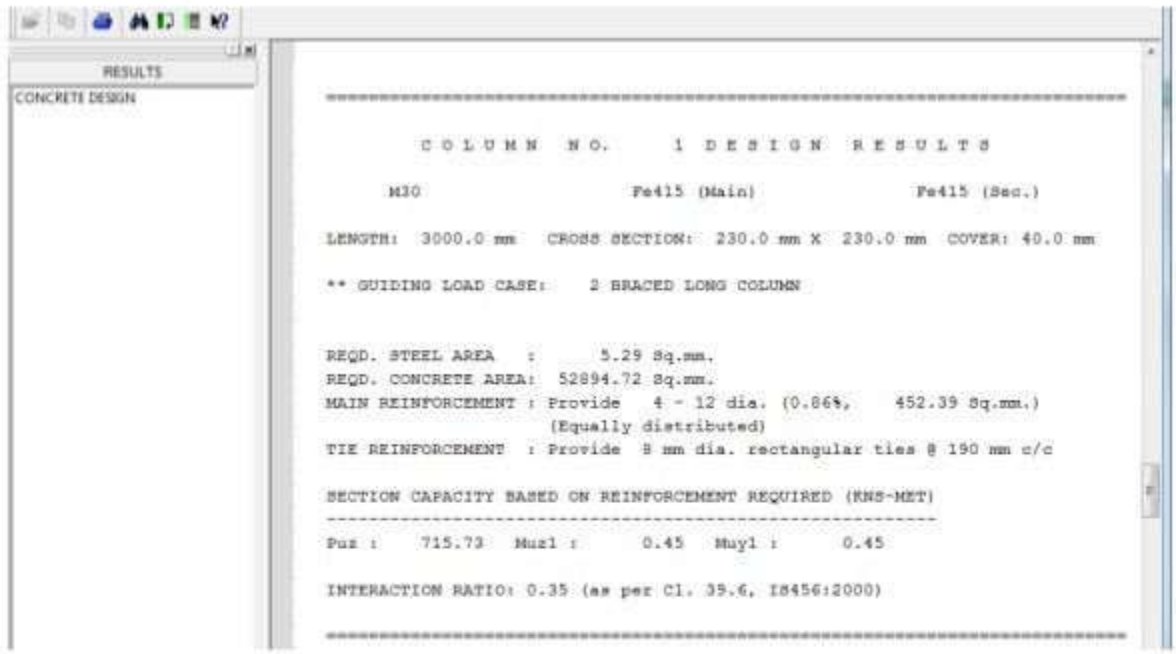
Assign → close

First select Column member's → Click→ Select beam parallel to → Y

Click → **commands** → Click → Column design →Add → Assign → Close → **Analysis** → Run analysis → save → Close →View output file → Done →To see the result concrete design result

Click the beam member on mouse to see the → **column design**





STEP 8

DESIGN OF SHEAR & MAIN LAYOUT FOR CONCRETE

- **E**nvelops
- Groups & Briefs
- Members
- Design

Envelops

Click → **Envelops** → Click → New Envelops → E1: name → Ok → Define Enevolops → un tick box → Load transfer from left side to right side → **Ok**

Groups & Briefs

Click → **Groups & Briefs** → Click → **New briefs** → B1:name → Design code **IS 456** → Ok → Click → Edit briefs general main rft shear rft to changes values → **Ok**

Click → **New design groups** → G: design group 1 → Design beiefs you created name → **Ok**

Members

Click → Members → Select the beam → Click → Top box members → Click → Auto form members → Beam created name M1 → Click → The top bar → Mode member → Design → To shown the dialogue box → Click the beam on mouse left → Click → Add → Members to design group → Design group 1 → Add → Ok

Design

Click → Design → Design option selected M1 → Ok → Design → Result noted

Result Are

- Main rft
- Main layout
- Shear rft
- Shear layout
- Summary

Structure1 - Design Group 1 - Summary

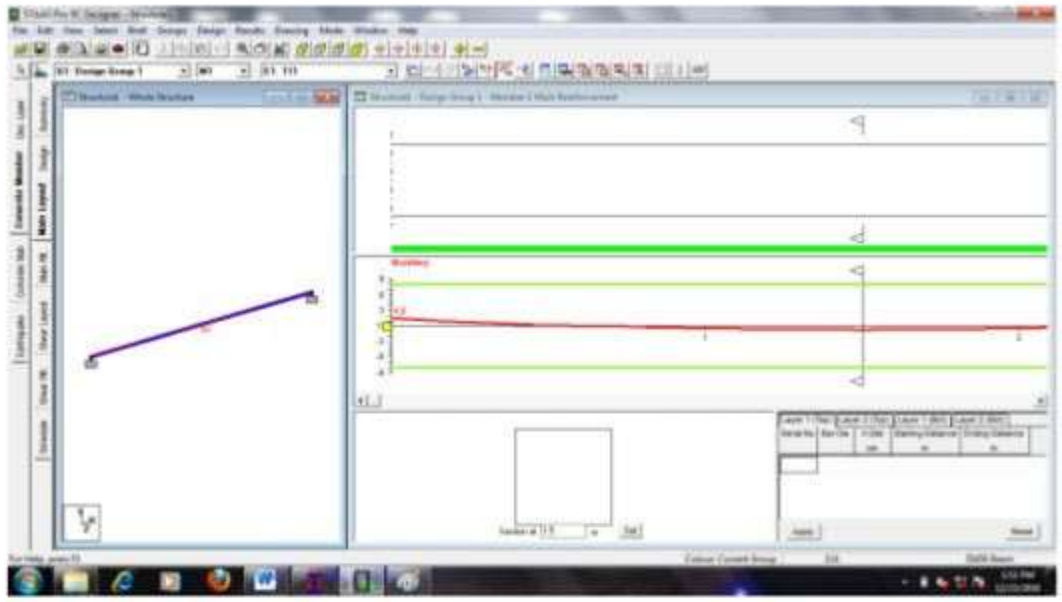
Mem	Design	Span	Type	Main Bars		Shear Bars	Span Depth
				Hog	Sag		
M1	None	1	Beam	Ok	Ok	Ok	Ok

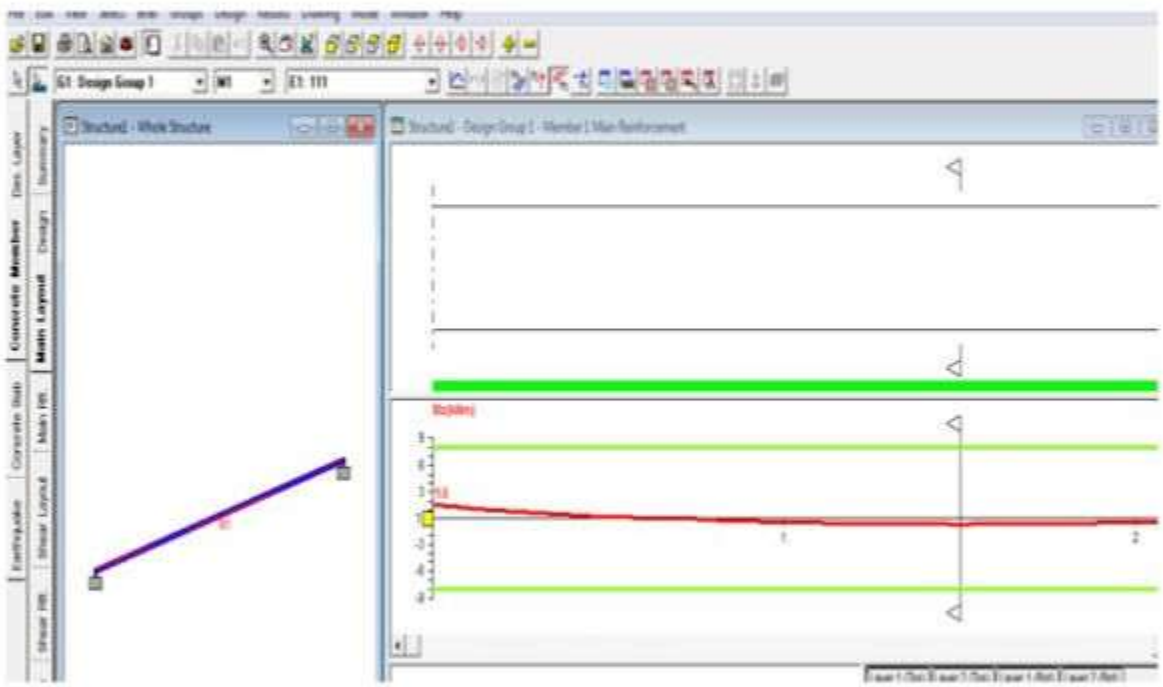
Structure1 - Design Group 1 - Beam Spans

Spans | Supports

Mem	Span	Type	Length m	Covers			Link Size
				Hog cm	Sag cm	Side cm	
M1	1	Beam	3.000	2.5	2.5	2.5	8

Summary





DESIGN OF STEEL STRUCTURE USING STAAD Pro

Step 1

Open the staad pro

New → New file → **Truss**

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

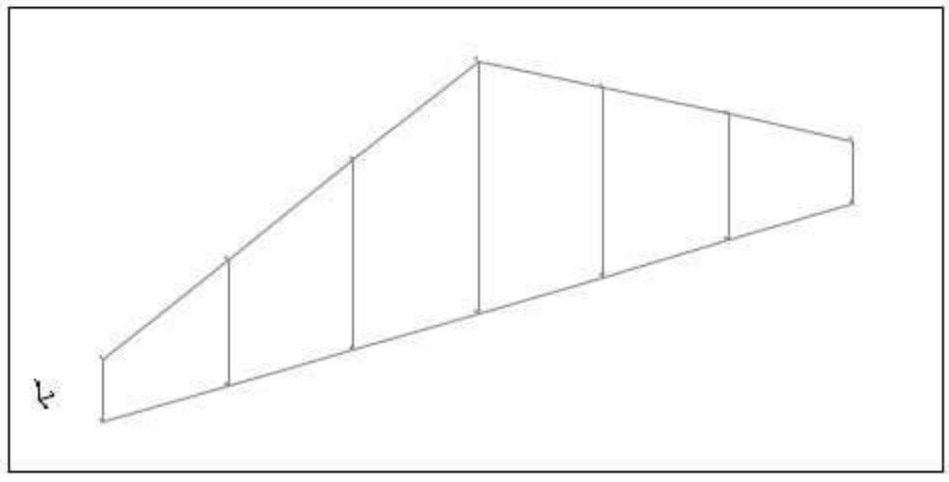
Add beam → **Finish**

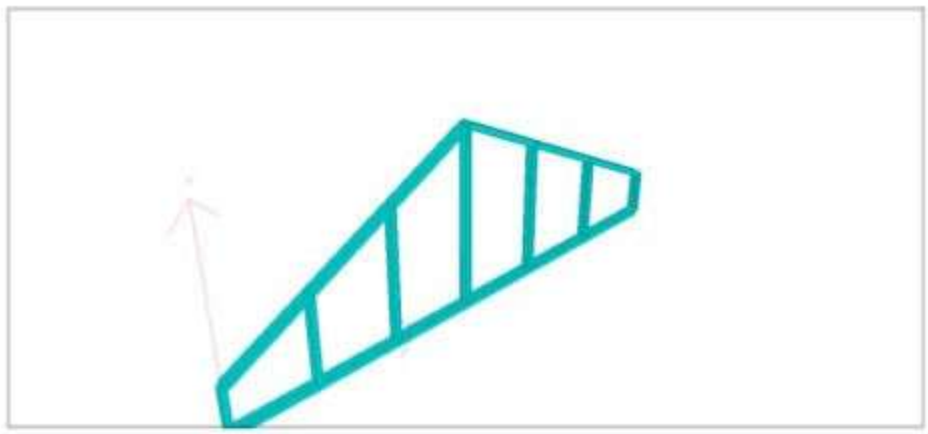
Step 3

Click → geometry → **Nodes**

Node	x	y	z
1	m	m	m

Diagram





Nodes	x	y	z
1	0	0	0
2	2	0	0
3	4	0	0
4	6	0	0
5	8	0	0
6	10	0	0
7	12	0	0
8	0	1	0
9	2	2	0
10	4	3	0
11	6	4	0
12	8	3	0
13	10	2	0
14	12	1	0

Then close the node (**x)** → You see the node point on the screen → How to find the node numbers → Click mouse left side → **Click labels** → Diagrams → Structure → Node number & Node point → **Apply** → **ok**

Step 4

To join the node points

Click → Geometry → Add beam → Add beam from point to point

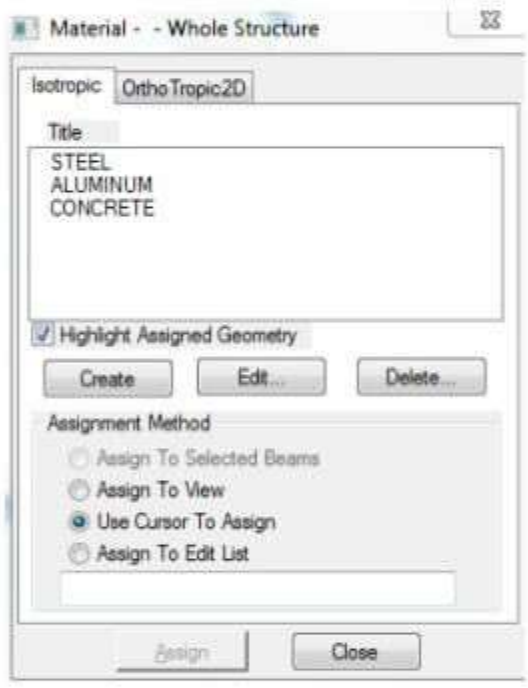
Select the node point and joint all node point like framed structure.

Step 5

- Material
- Property
- Support
- Load
- Analysis

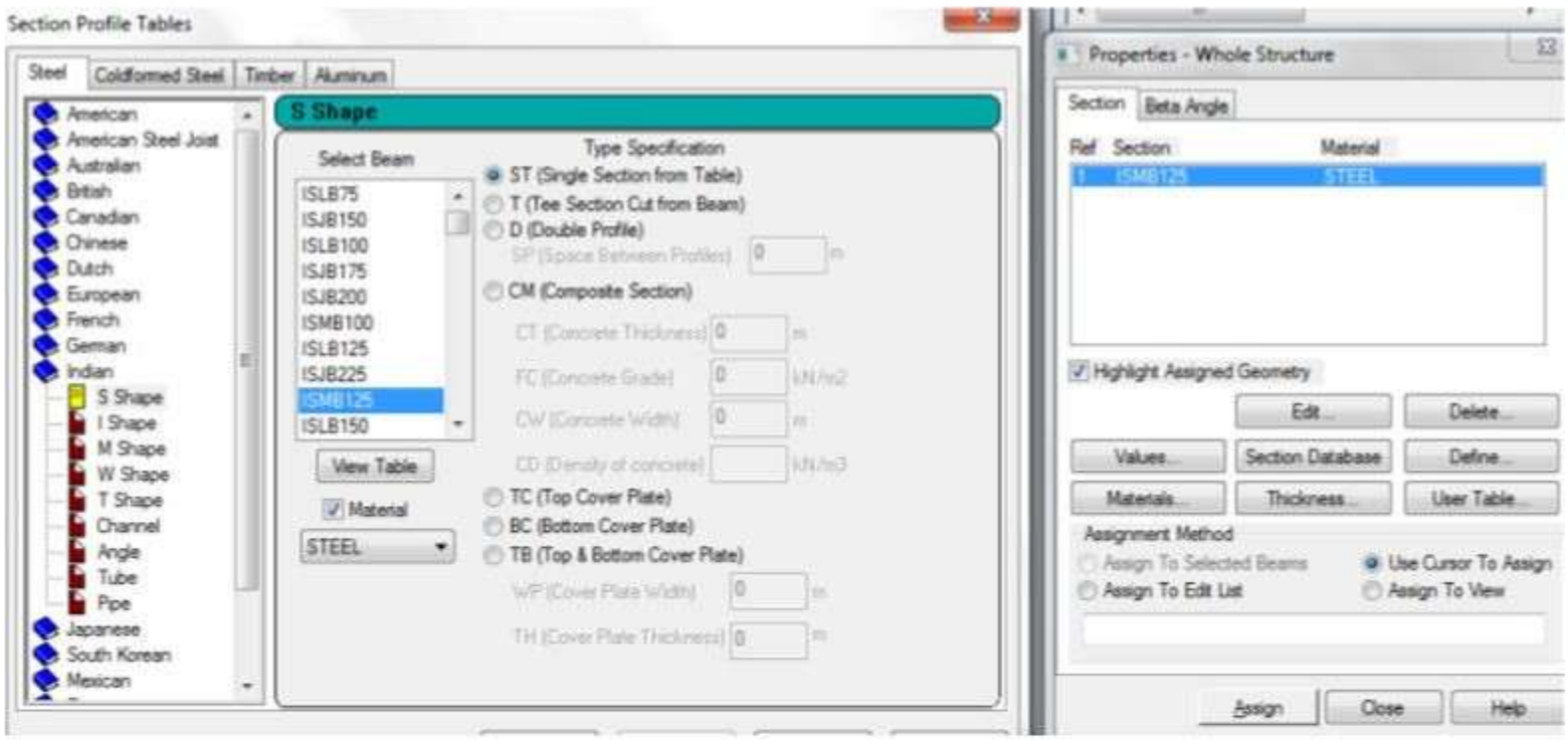
Material

Click→ Modeling → General→ Material → Material Whole structure → **click Steel** →Assign to view → **Assign** → **Yes**



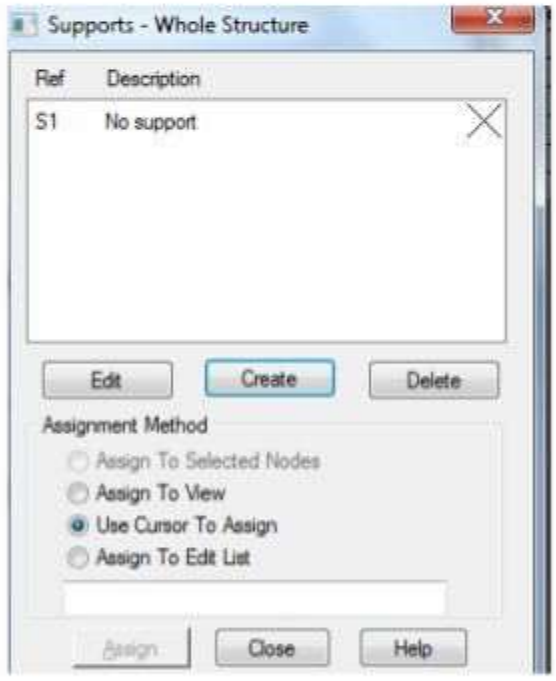
Property

Click → Modeling → General → property → Property Whole structure → **Section database** → Section profile table→ click → Indian →s →shape → **ISMB 125** Double channel → Add → Close → Assign to view → **Assign** → **Yes**



Support

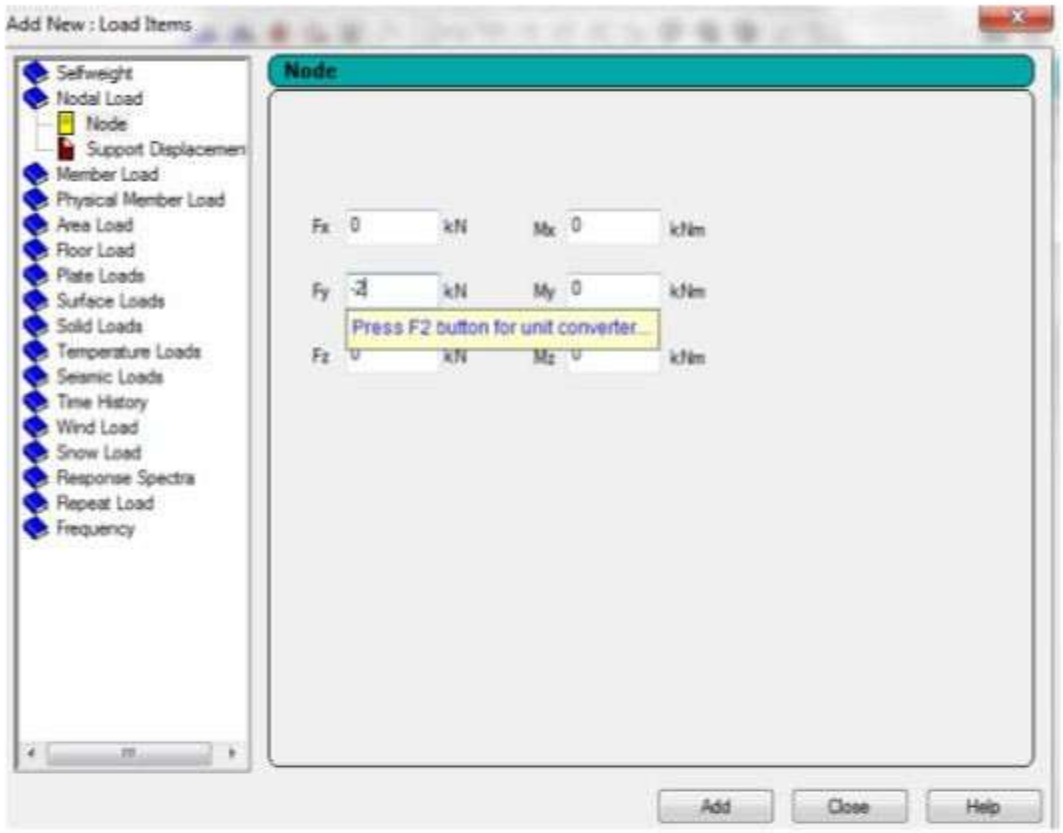
Click → Modeling → General → Support → Support Whole structure → Create → **pinned** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**



Loads

Click → Modeling → General → Load → Click load case details → Add → Add new load cases → **Add – Load case 1** → **Add- Load case 2** → Click Load case 1 Add **self-weight** → Add → Close → Click Load case 2 → Add →Member → load → **Nodel load** fy = -2 kN →**Add** → **Close**

Select load→ fy -2 kN → Assign selected node → **Assign→ Yes**



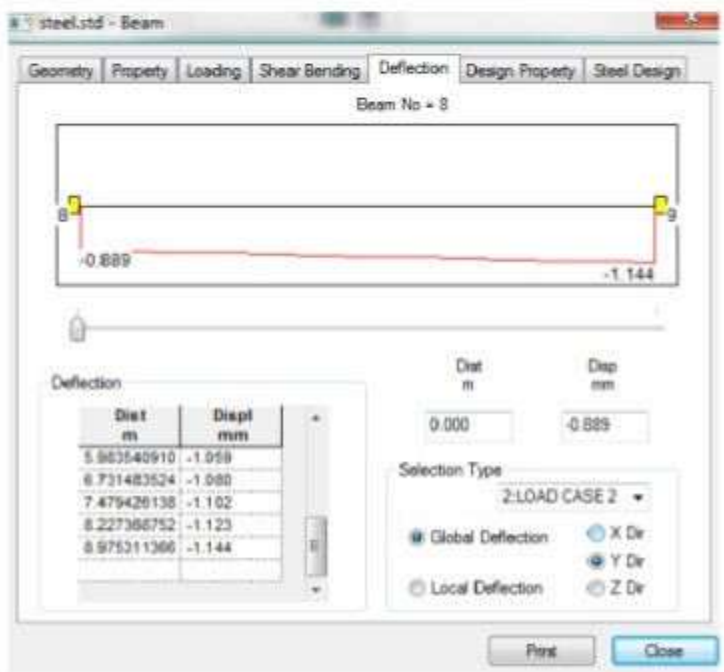
ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

Click the beam see the result on beam

DEFLECTION DIAGRAM



Step 5

POST PROCESSING

Click → Post processing → Result setup → Select load case → **Ok**

Click→ **Reactions** → see the results

Click→ Instability joints → see the results

Click →**Beam→ Unity check → Stresses → Forces → Graphs**

New screen will be displayed → Click → Result → Animation →**Deflection → Ok** → F12 to see full screen of deflection

STEP 6

DESIGN OF STEEL ELEMENTS

Click → Modeling → Click → **Design** → Click → **Steel** → Steel design whole structure → **Current code IS 800** → Click → Selected parameters

- Fvb = Allowable shear stress in rivet
- Fyld = Yield strength of steel
- Ly = Length in local y axis for slenderness ratio
- Lz = Length in local z axis for slenderness ratio
- Track = Track parameter
- Weld = Design weld

Click → **Define parameter**

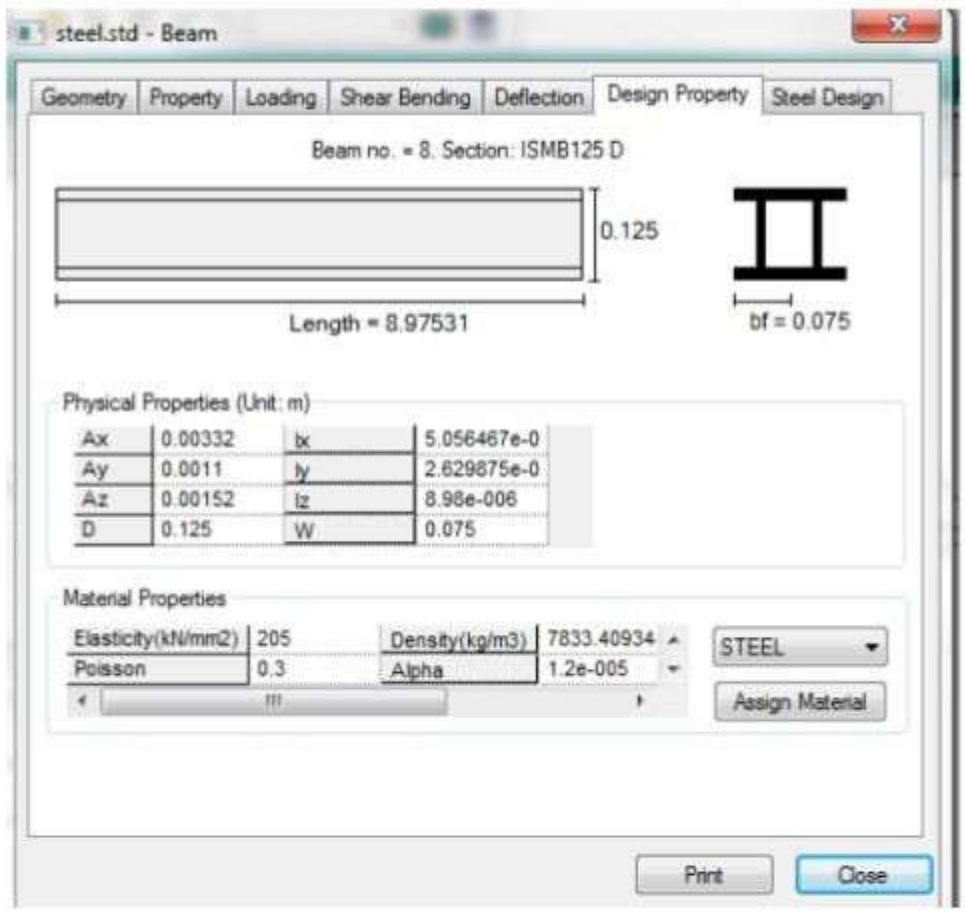
- Fvb = Allowable shear stress in rivet =100 N/mm^2 → Add
- Fyld = Yield strength of steel = 250 N/mm^2 → Add
- Ly = Length in local y axis for slenderness ratio = 100 mm → Add
- Lz = Length in local z axis for slenderness ratio = 100 mm → Add
- Track = Track parameter = max detail level → Add
- Weld = Design weld = closed weld → Add

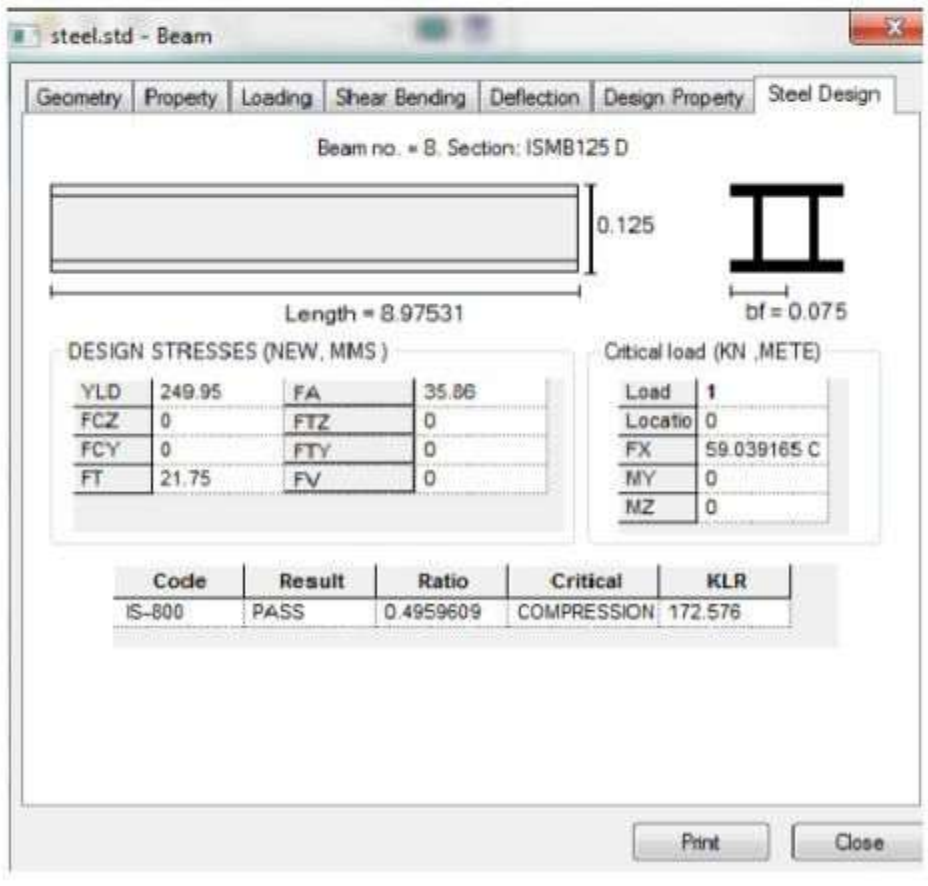
Add → **Assign** → Close

First select beam member's → Click→ Select steel structure

Click → **commands** → Click → **Check code** → Add → **Assign** → Close → **Analysis** → Run analysis → save → Close → View output file → Done → To see the result Steel design result

Click the steel member on mouse to see the → **Steel design**





STEP 7

DESIGN OF STEEL ELEMENTS PASS OR FAIL

- Envelops
- Member design
- Groups and briefs
- Restrains
- Result and report

Steel design → New envelops → **Load envelops** → Click load → **Ok**

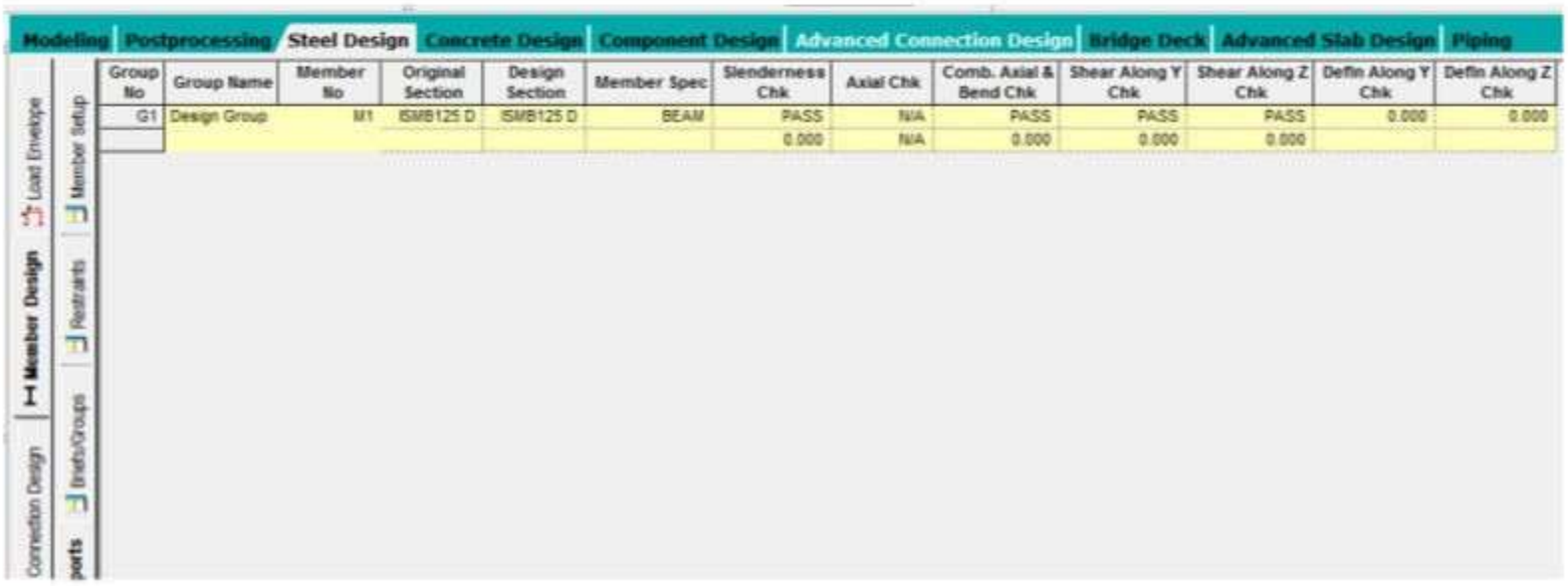
Click → **Member design**→ Click top bar → Member → Design physical members → **Form members**

Click → Group and Briefs → Click → **New brief design** → **Code IS 800** → Ok

Click→ New group → Click → **Group** → Add → **M1** → Transfer left side to right side → Ok

Click → **Restrains** → Select member M1 → Top bar → Member design → Performance group design → **Check code**

Click → **Result and Report**



Group No	Group Name	Member No	Original Section	Design Section	Member Spec	Slenderness Chk	Axial Chk	Comb. Axial & Bend Chk	Shear Along Y Chk	Shear Along Z Chk	Defln Along Y Chk	Defln Along Z Chk
G1	Design Group	M1	ISMB125 D	ISMB125 D	BEAM	PASS	N/A	PASS	PASS	PASS	0.000	0.000
						0.000	N/A	0.000	0.000	0.000		

DESIG OF PLATE SLAB USING STAAD Pro

Step 1

Open the staad pro

New → New file → Floor

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

Add plate → **Finish**

Step 3

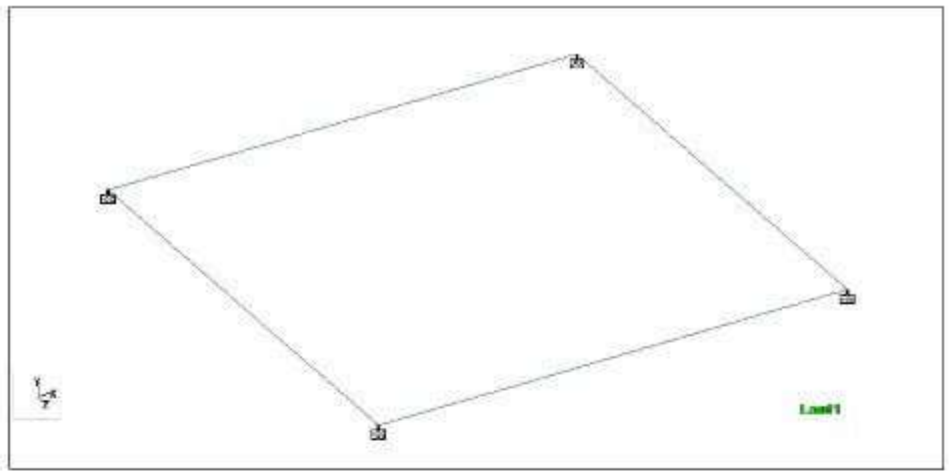
Click → geometry → **Nodes**

Node x y z

1 m m m

Diagram

node	x	y	z
1	0	0	0
2	3	0	0
3	0	0	3
4	3	0	3



Click → Geometry → **Add plate** → Click quad then join the four node point

Step 4

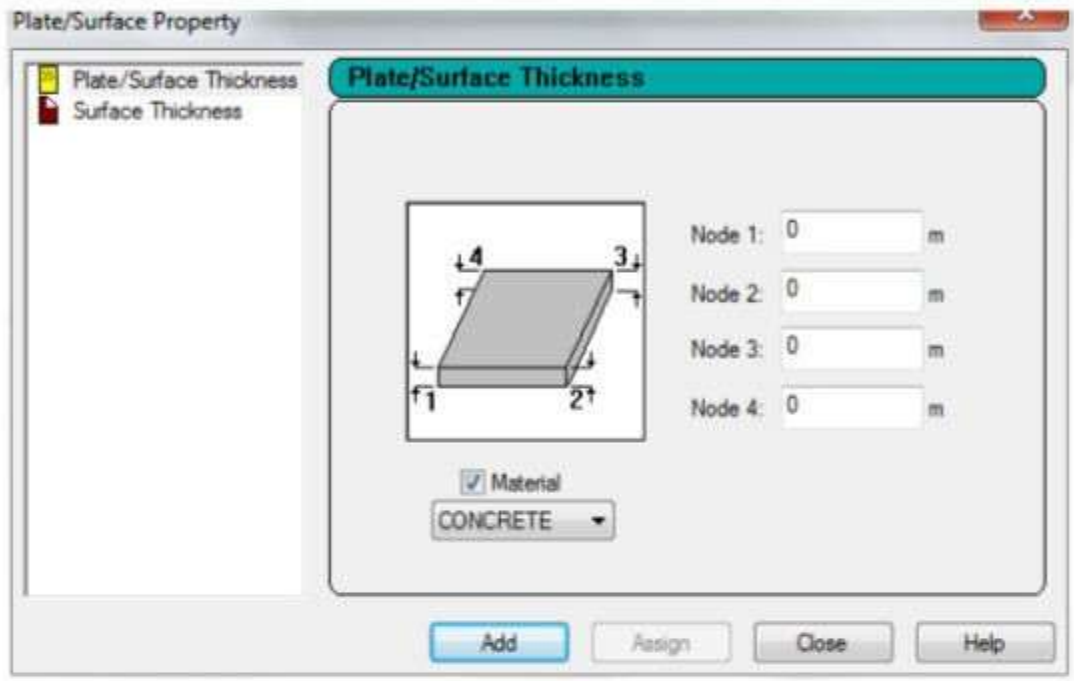
- Material
- Property
- Support
- Load
- Analysis

Material

Click→ Modeling → General→ Material → Material Whole structure → **click Concrete** →Assign to view → **Assign → Yes**

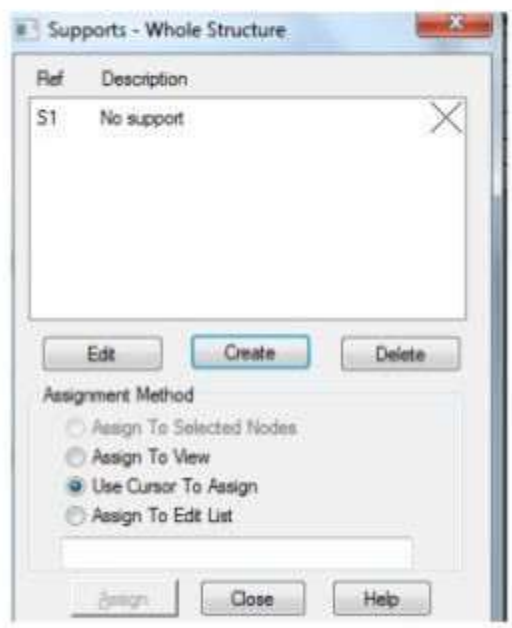
Property

Click → Modeling → General → property → Property Whole structure → **Thickness** → **Plate surface thickness** → 0.120m → Add → Close → Assign to selected plate → **Assign** → Yes → close



Support

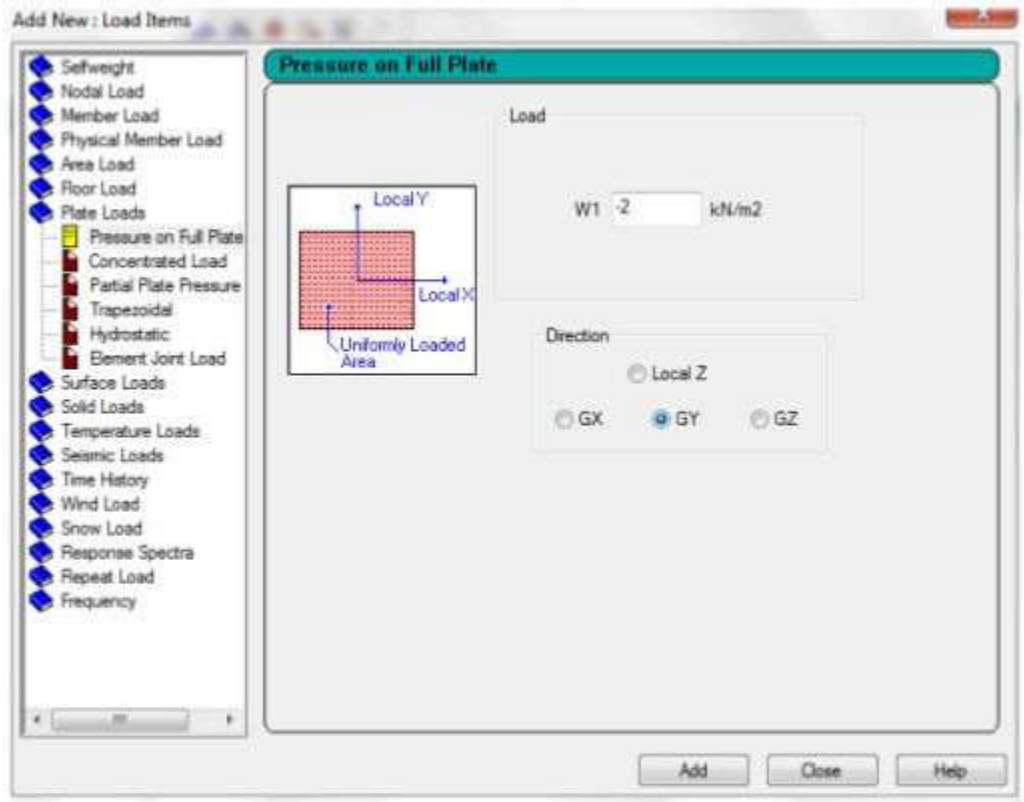
Click → Modeling → General → Support → Support Whole structure → Create → **pinned** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**



Loads

Click → Modeling → General → Load → Click load case details → Add → Add new load cases → **Add – Load case 1** → **Add- Load case 2** → Click Load case 1 Add **self-weight** → Add → Close → Click Load case 2 → plate load pressure on full plate -2kN/m^2 GY →**Add** → **Close**

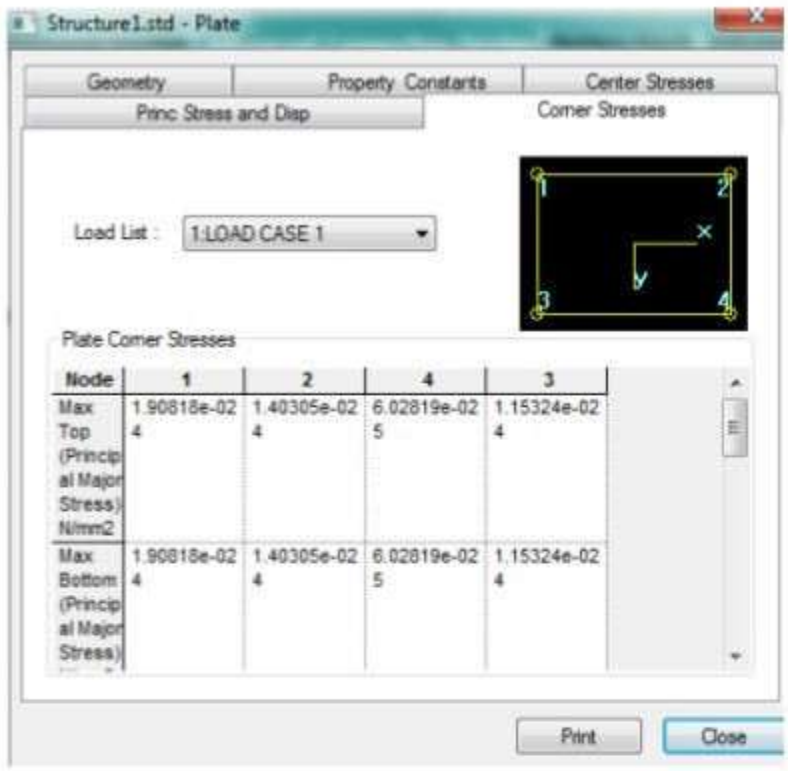
Select load→ PR -2 kN/m^2 →Select →**Assign To View** → **Assign**→ **Yes**



Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

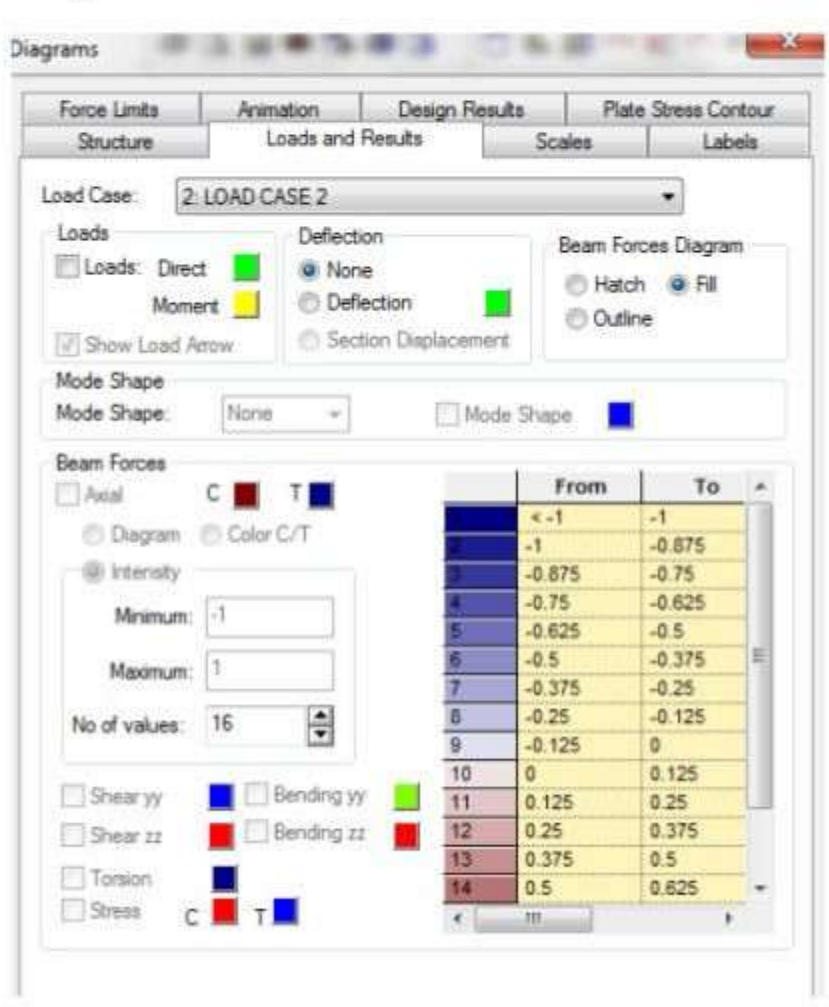
Click the SLAB see the result on slab



POST PROCESSING

Click → Post processing → **Plate** → **Stress type**

Click → Post processing → **Counter**



SLAB DESIGN

whole structure → **Current code IS 456** →Click → Selected parameters

- fc = compressive strength of concrete
- fy = yield strength of concrete

Ok → Then change the unit **kN/m to N/mm²** → Click → Tools → Set current input units → Change unite

Click → **Define parameter**

- fc = compressive strength of concrete = 30 N/mm² → Add
- fy = yield strength of concrete = 415 N/mm² →Add

Assign → close

Click the SLAB member on mouse to see the → SLAB **design**

ELEMENT	LONG. REINF (SQ.MM/ME)	MOM-X /LOAD (KN-M/M)	TRANS. REINF (SQ.MM/ME)	MOM-Y /LOAD (KN-M/M)
8 TOP :	120.	0.00 / 0	120.	0.00 / 0
BOTT:	143.	-5.03 / 2	143.	-5.03 / 2
10 TOP :	120.	0.00 / 0	120.	0.00 / 0
BOTT:	120.	-1.92 / 2	120.	-2.90 / 2
12 TOP :	120.	0.56 / 2	120.	0.00 / 0
BOTT:	120.	0.00 / 0	120.	-1.10 / 2
14 TOP :	120.	2.31 / 2	120.	0.01 / 2
BOTT:	120.	0.00 / 0	120.	-0.24 / 1
16 TOP :	120.	3.26 / 2	120.	0.57 / 2
BOTT:	120.	0.00 / 0	120.	-0.17 / 1
18 TOP :	120.	3.26 / 2	120.	0.57 / 2
BOTT:	120.	0.00 / 0	120.	-0.17 / 1
20 TOP :	120.	2.31 / 2	120.	0.01 / 2
BOTT:	120.	0.00 / 0	120.	-0.24 / 1
22 TOP :	120.	0.56 / 2	120.	0.00 / 0
BOTT:	120.	0.00 / 0	120.	-1.10 / 2

DESIGN OF FOUNATION USING STAAD Pro

Click → STAAD → Foundation

Open→ **Staad foundation** → Click → File new

Click→ **[j]** → New job → Write→ New job name→ Job title→ Isolated → Design code→ **Indian** →Default unit→ **SI** → Ok

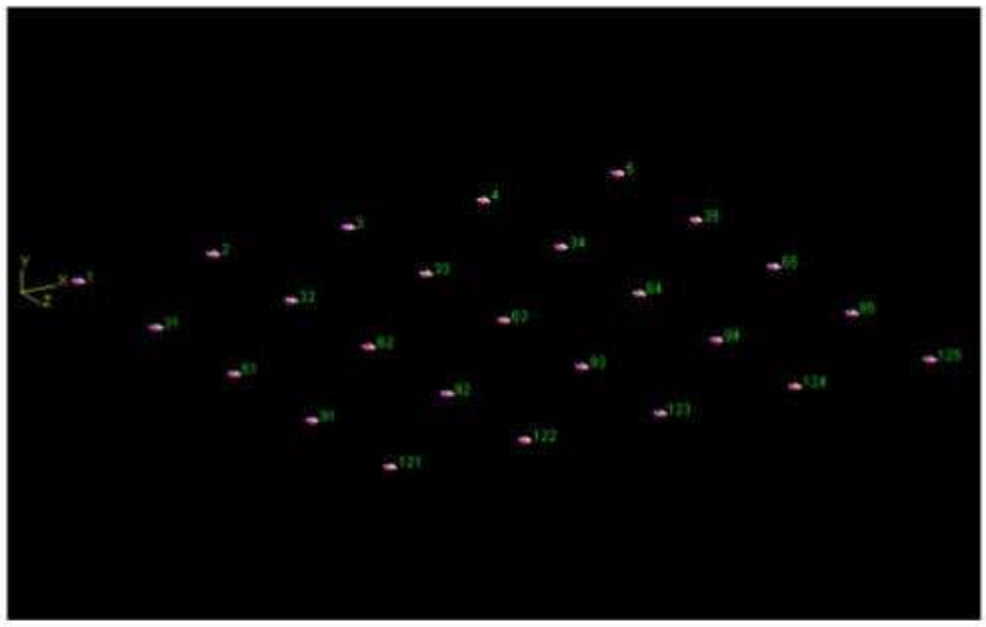
- Foundation Plan
- Load & Factor
- Design Parameter
    - Cover Rebar
    - Cover soil
    - Geometry
- Design
    - Design summary
    - Footing layout
    - Detailed drawing
    - Calculation sheet

Foundation plan

Click → tools → set out put units

Click → Foundation Plan → **Column position**

Click → Foundation Plan→ **Column Dimension**

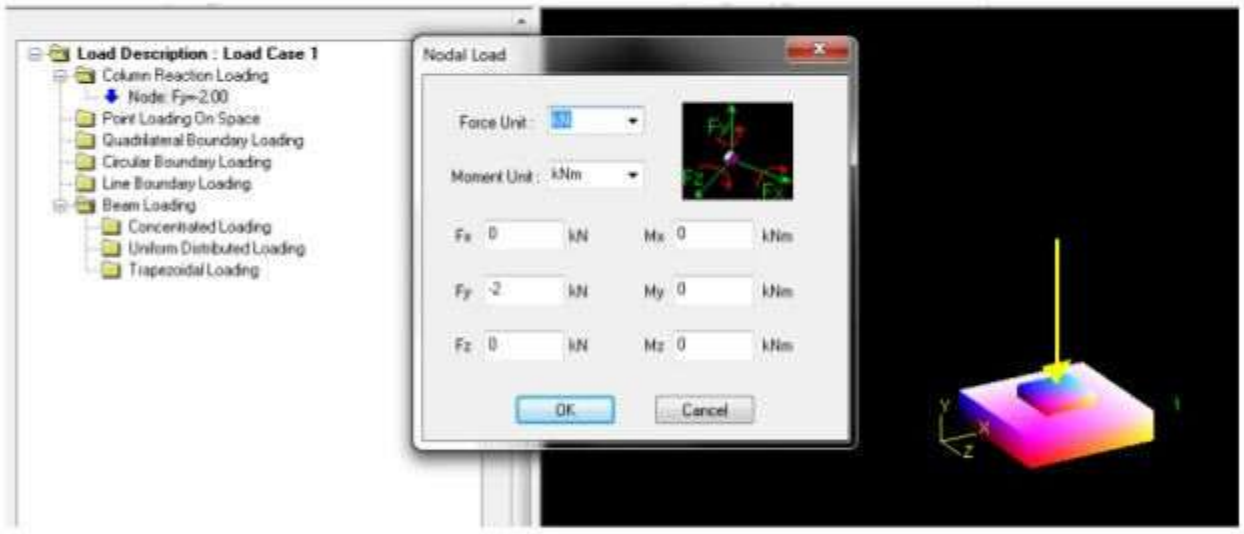


Load & Factors

Click → load &factors → click → new **load case** → create new load case load name→ primary → ok

Click → Column reaction → Node reaction → **fy -2** → **Ok**

Click → Load → Assign to view → **Assign**

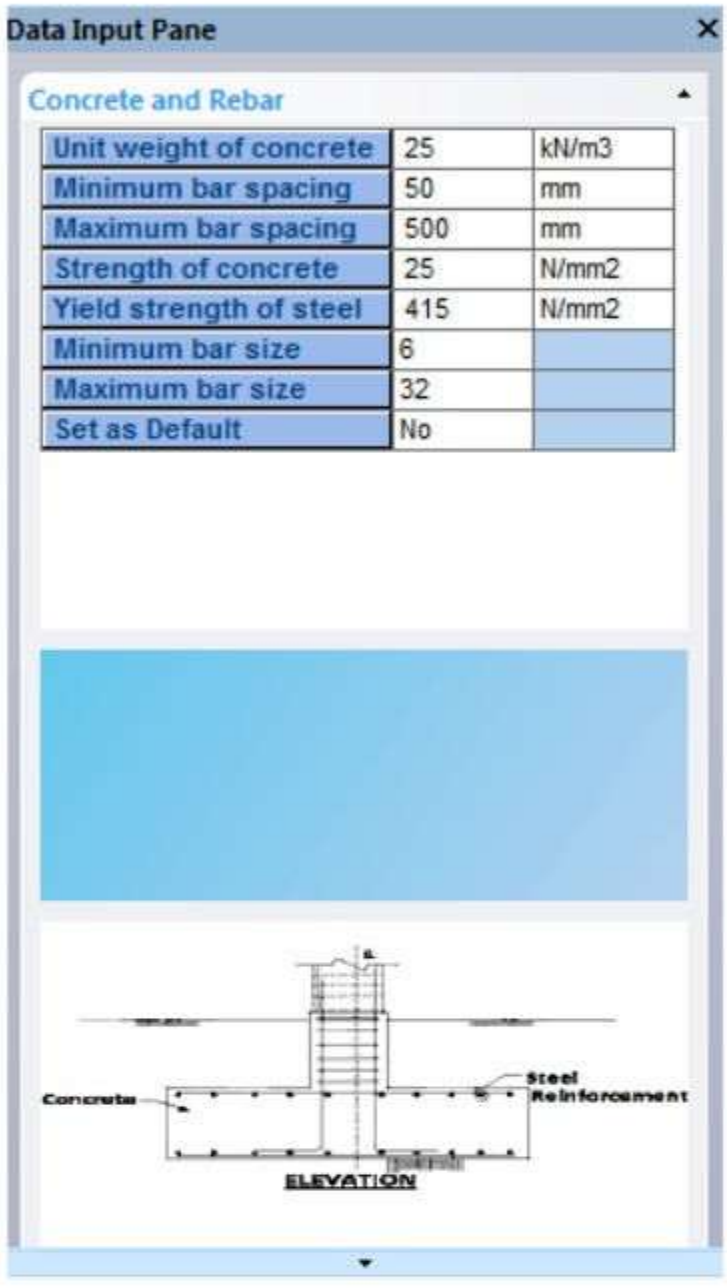


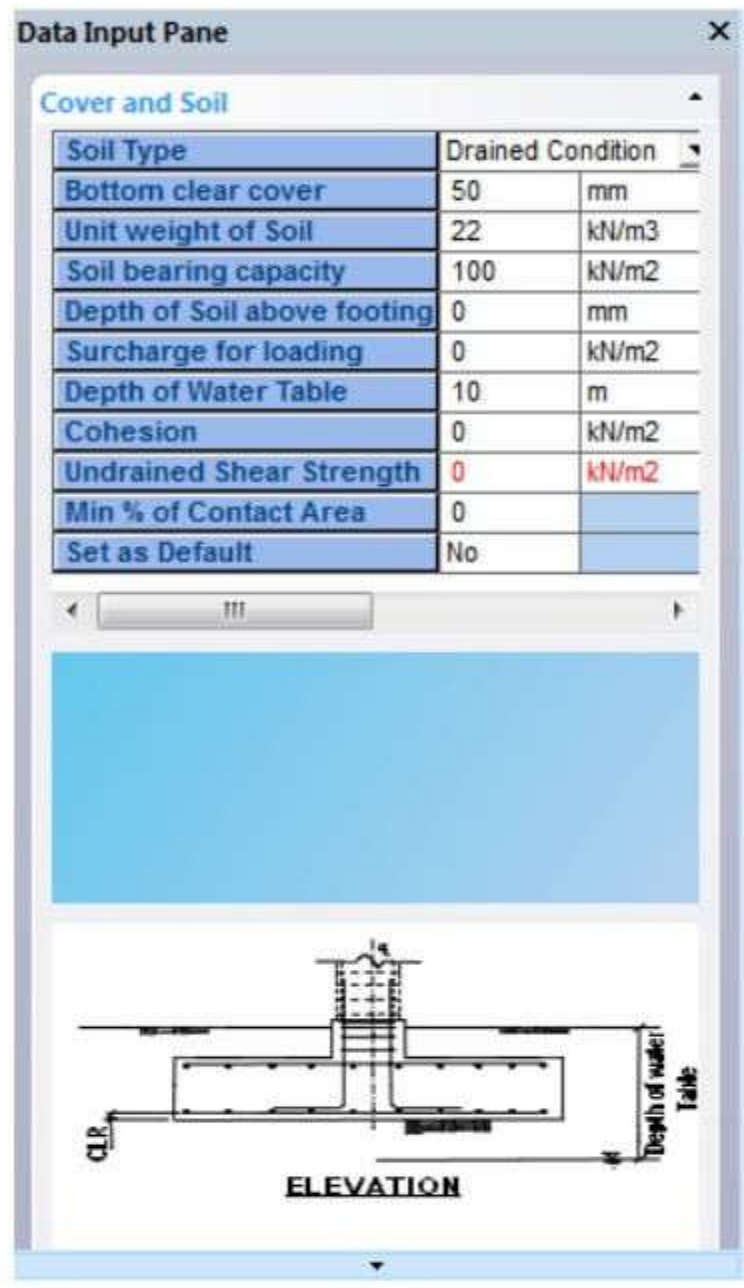
Design Parameter

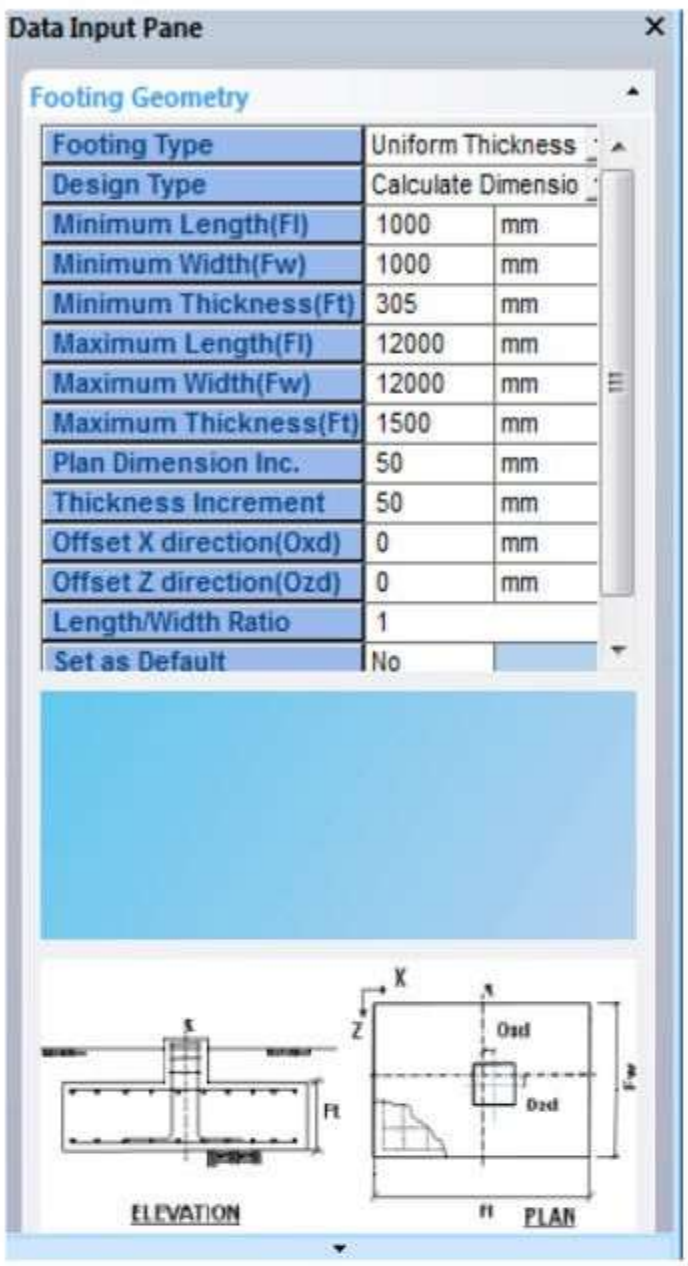
Click→ **Design parameter** → Click→ **Cover rebar** → Change the value click → Set default

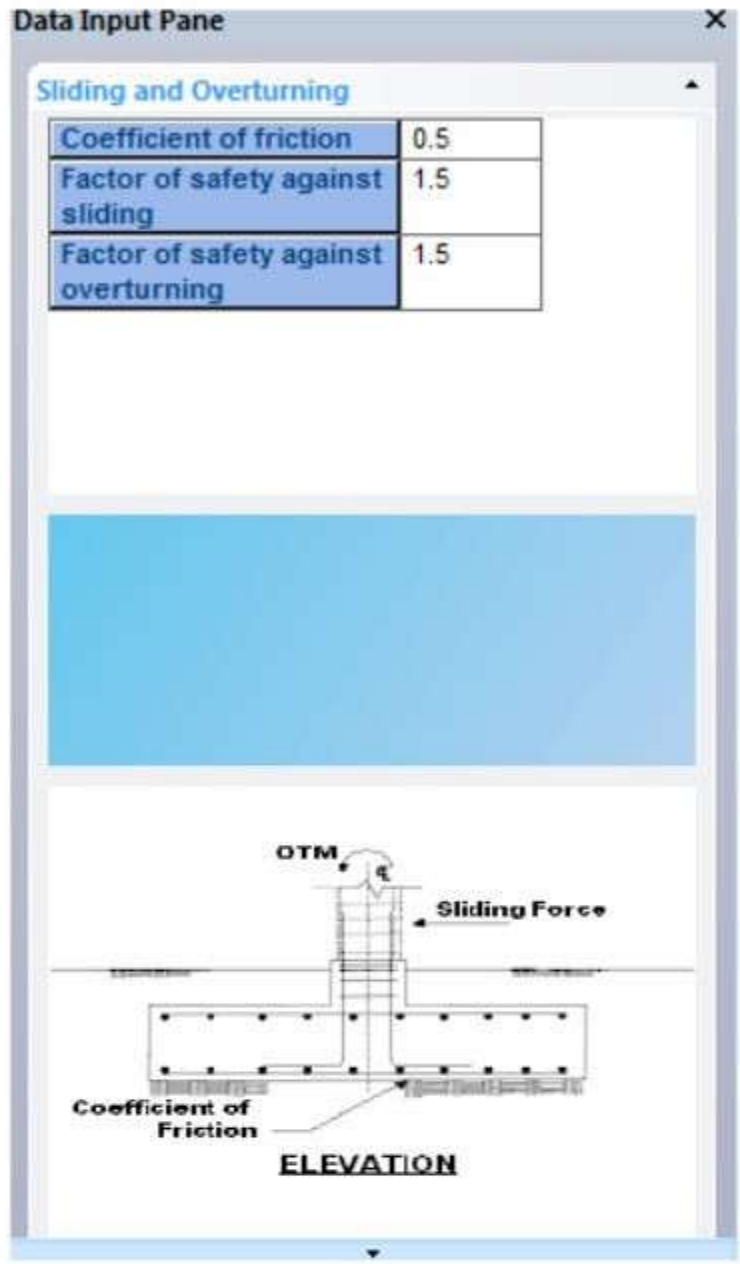
Click → **Cover and soil** → Change the values click → Set defau

Click→ **Geometry** → Change the values click → set default → **ok**







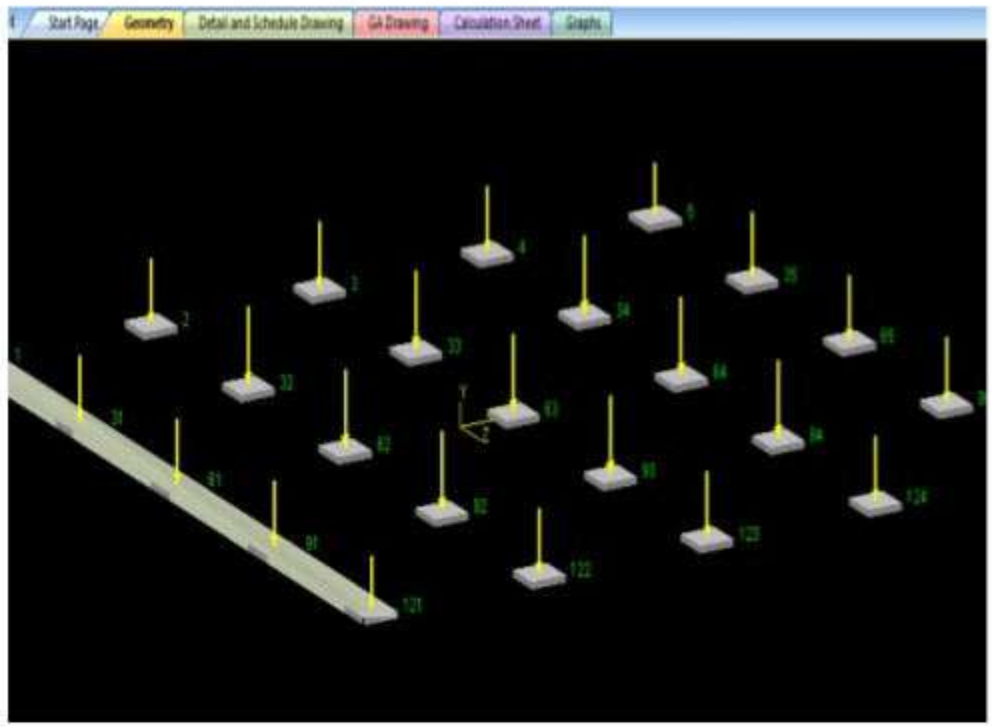


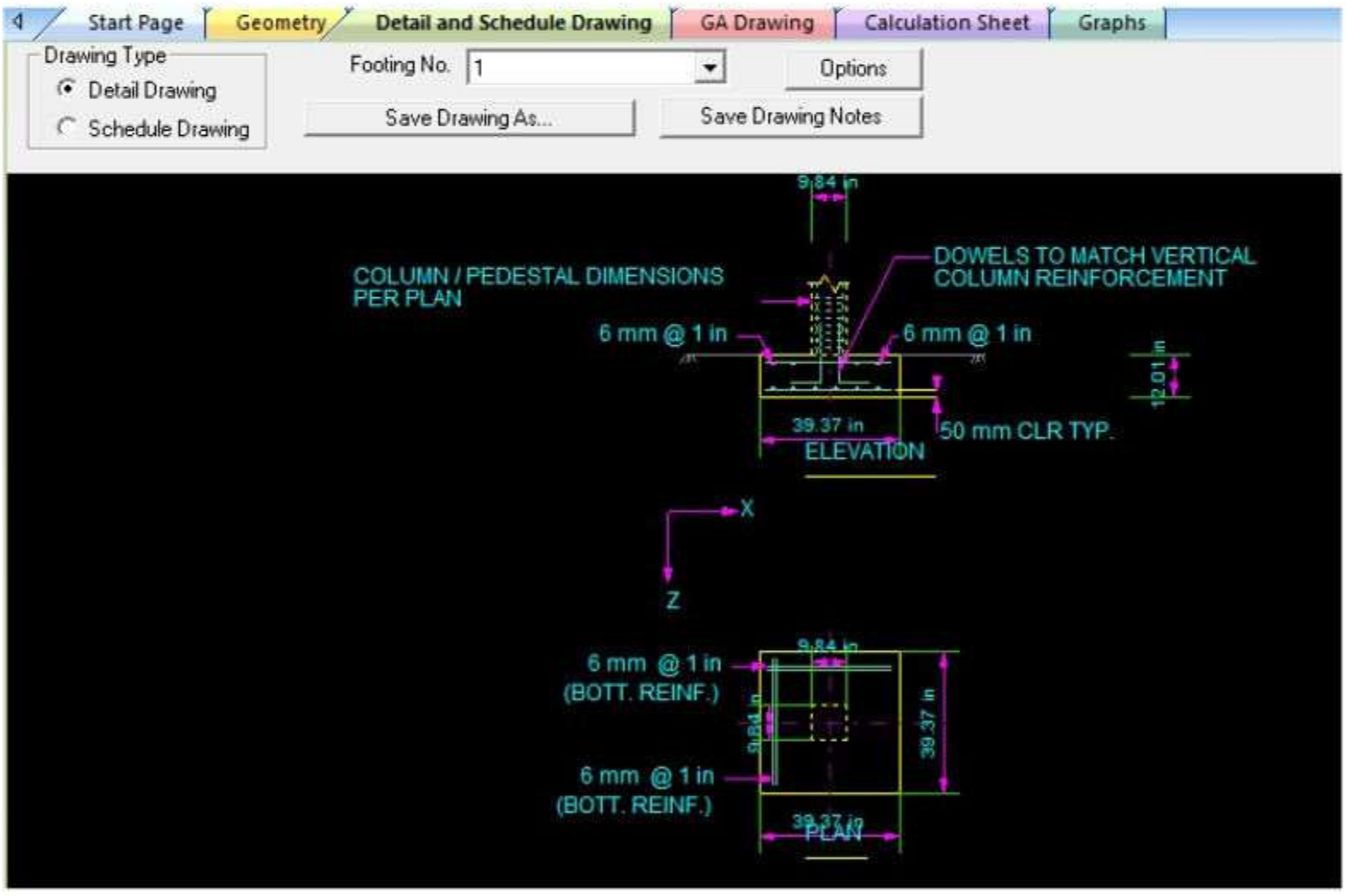
Design

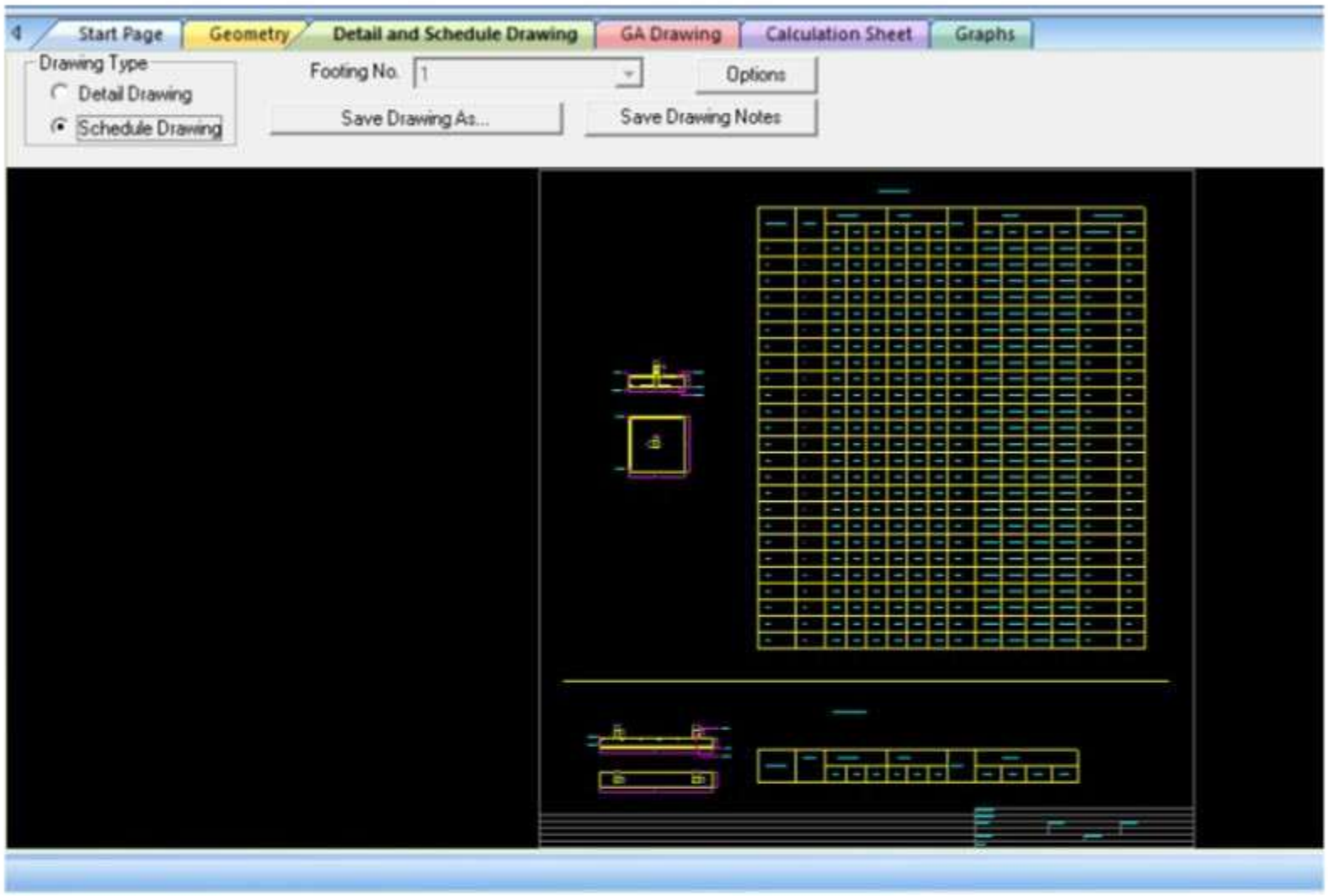
Click → **Design** → ok

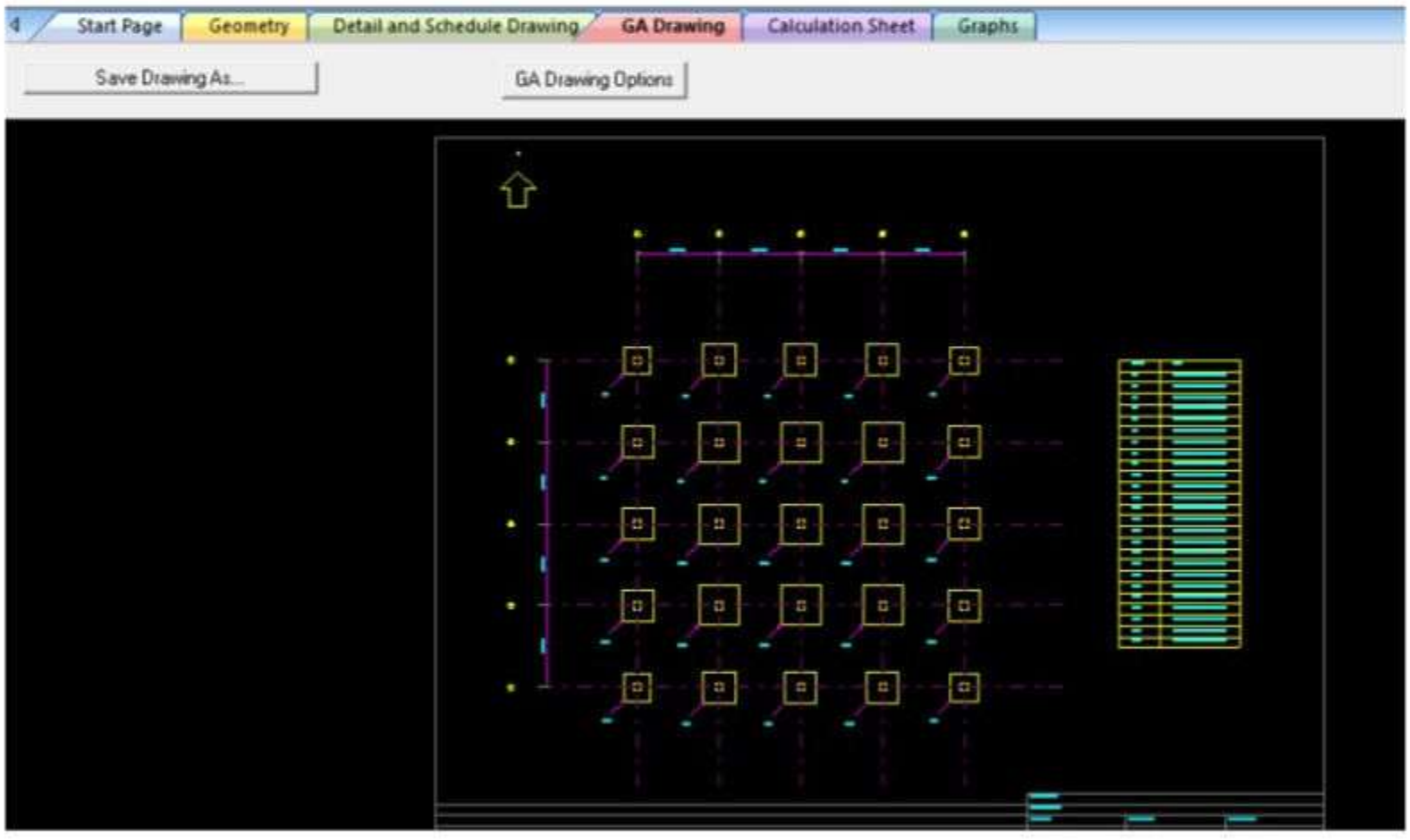
Results Are Taken

- Geometry
- Detailed schemed drawing
- GA drawing
- Calculation sheet


Start Page
Geometry
Detail and Schedule Drawing
GA Drawing
Calculation Sheet
Graphs


Start Page
Geometry
Detail and Schedule Drawing
GA Drawing
Calculation Sheet
Graphs
Drawing Type
Detail Drawing
Schedule Drawing
Footing No.
1
Options
Save Drawing As...
Save Drawing Notes
9.84 in
COLUMN / PEDESTAL DIMENSIONS PER PLAN
DOWELS TO MATCH VERTICAL COLUMN REINFORCEMENT
6 mm @ 1 in
6 mm @ 1 in
12.01 in
39.37 in
50 mm CLR TYP.
ELEVATION
X
Z
6 mm @ 1 in
(BOTT. REINF.)
9.84 in
39.37 in
6 mm @ 1 in
(BOTT. REINF.)
39.37 in
PLAN


Start Page
Geometry
Detail and Schedule Drawing
GA Drawing
Calculation Sheet
Graphs
Drawing Type
Detail Drawing
Schedule Drawing
Footing No.
1
Options
Save Drawing As...
Save Drawing Notes


Start Page
Geometry
Detail and Schedule Drawing
GA Drawing
Calculation Sheet
Graphs
Save Drawing As...
GA Drawing Options

DESIGN OF PILE CAP FOUNATION USING STAAD Pro

OPEN STAAD FOUNDATION

CLICK NEW

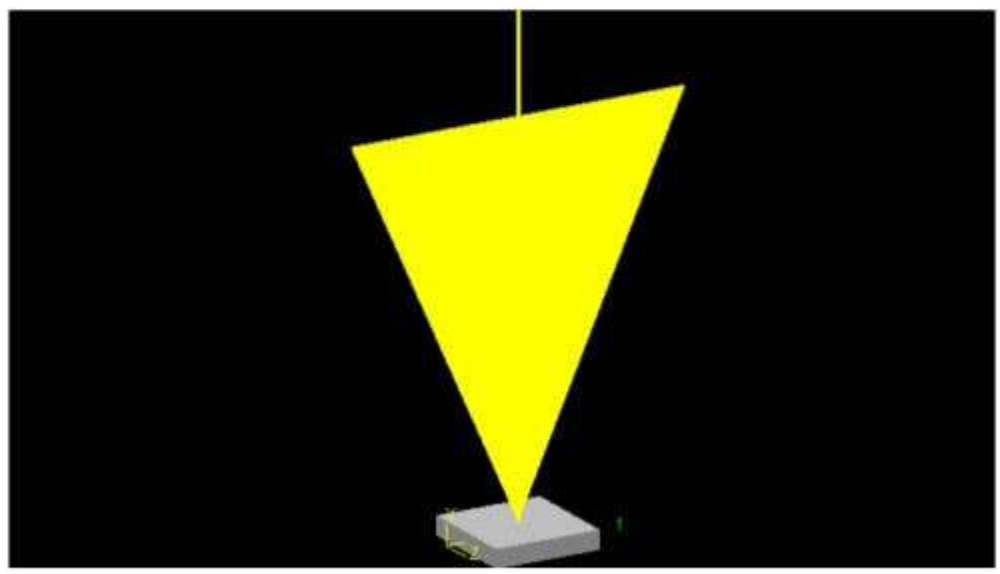
- GENERAL SETUP
- GENERAL FOUNDATION MODE
- COLUMN POSITION
- LOAD AND FACTORS
- JOB SET UP
- DESIGN PARAMETER
- PILE LAYOUT PREDEFINED
- DESIGN

Click → New→ General Setup

Click→ Foundation Mode → Click→ Foundation Plan

Click→ Coolum Position → 0,0,0 → Create The Footing

Click → Column Dimension → Give the Dimension → Ok



Click → Load And Factor → Create New→ Load Case → Load Description→ Dead Load

Load Title = Dead Load

Load Case Type = Service

Loading Type = Dead Add

LIVE Load

Load Title = LIVE Load

Load Case Type = Service

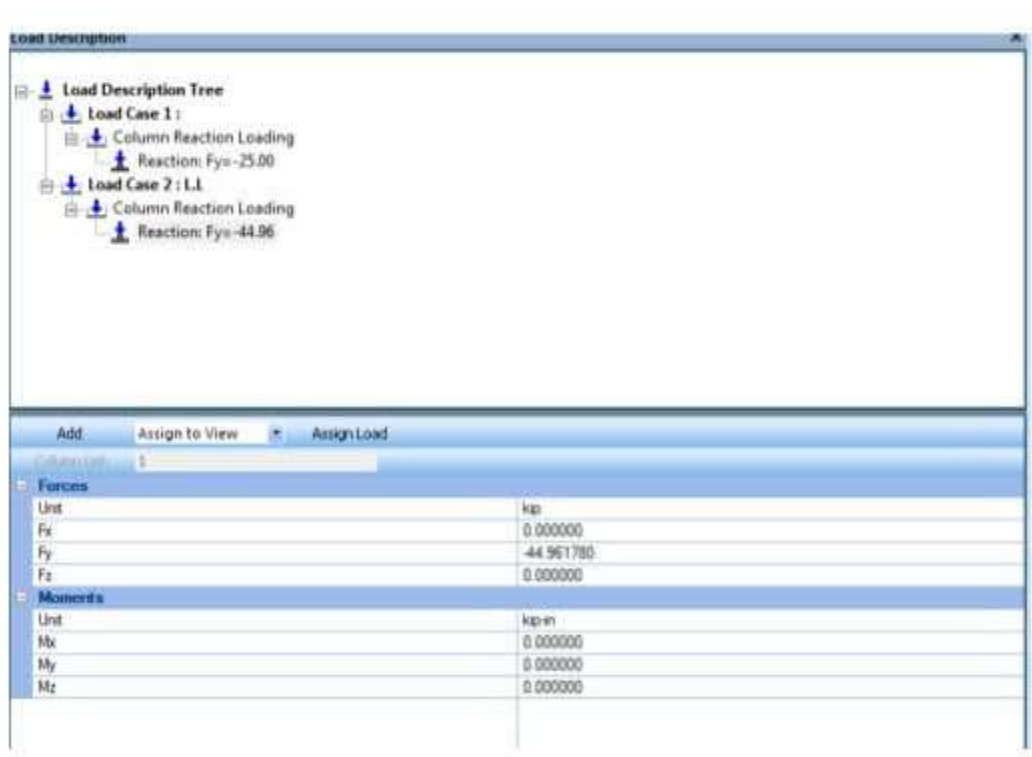
Loading type = live Add

Click → Load Case 1→ Left Click Mouse→ Add Column Reaction→ Load →Dead Load →Click→ Fy -25 →Entre →Add

Live Load → Click → Mouse Left Side → Add Column Reaction → Fy -200 → Entre → Add

Click→ Load→ Assign the Load

See → the Load On Screen



Click → Generate Load Combination → Load Combination → Load Combination →Type →India → Click → Generate Load Combination → Click → Generate Load Combination → close

Click → Job Setup → create → A New Job

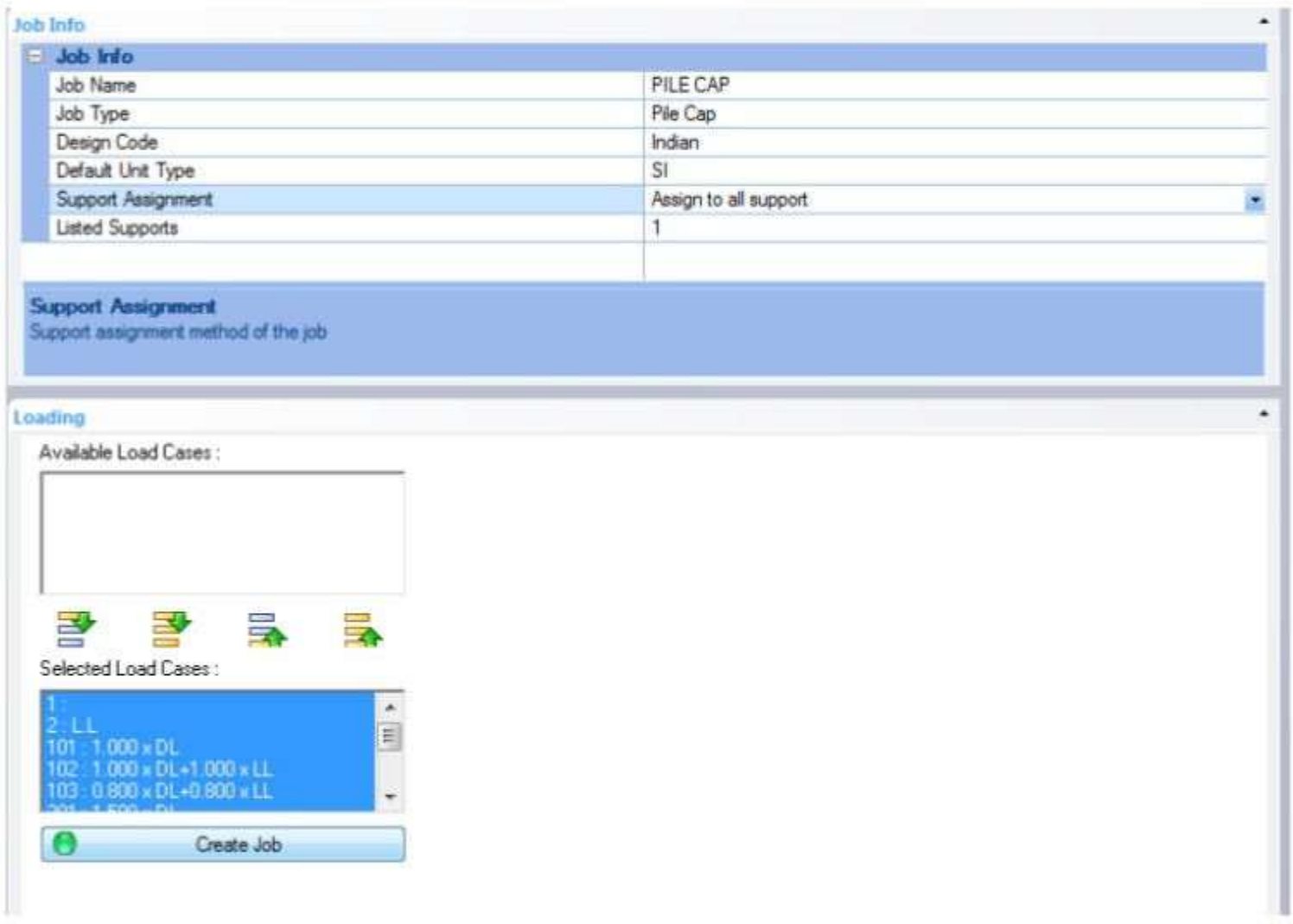
Job Name → Pile Cap

Job Type → Pile Cap

Design Code → Indian

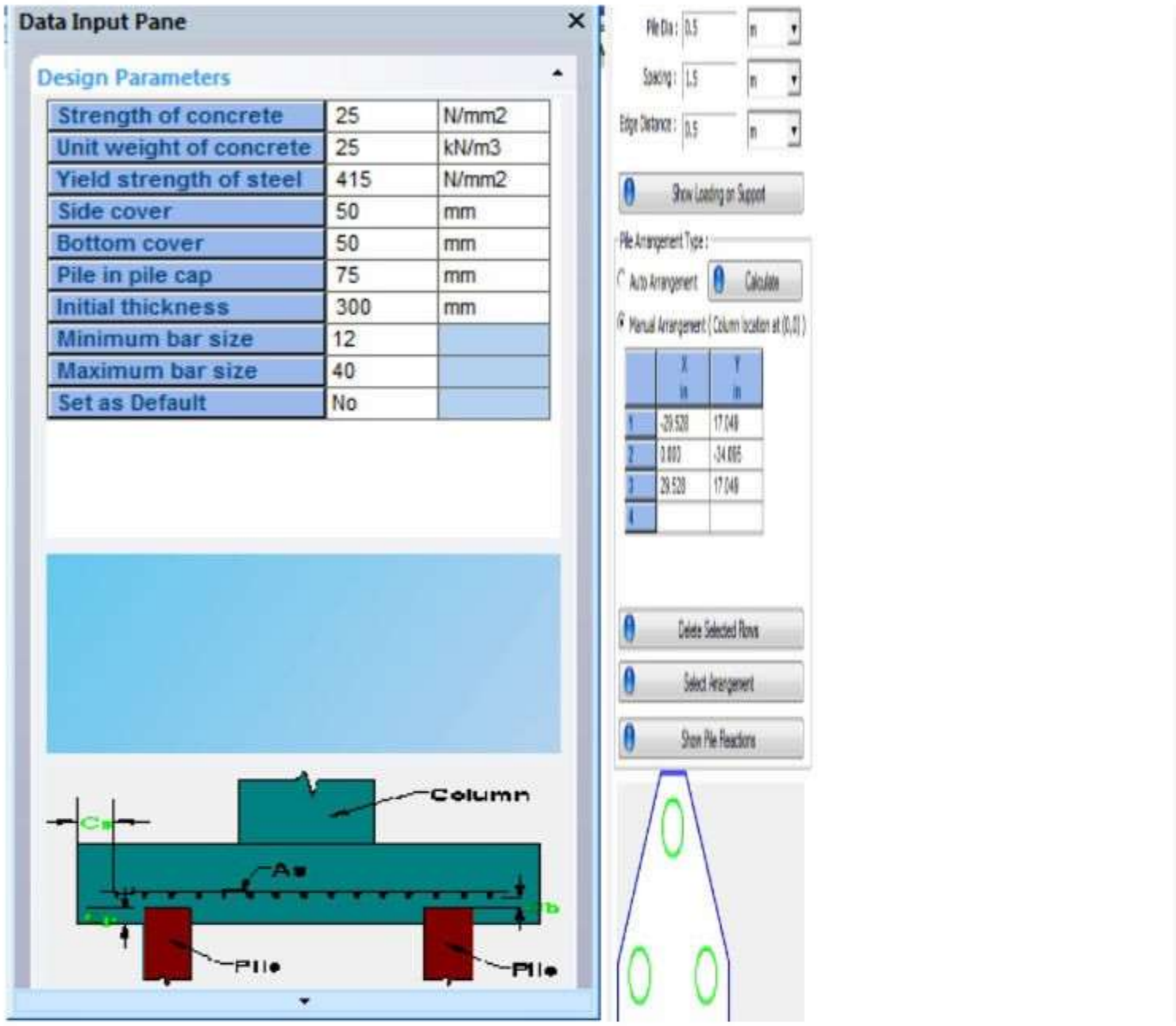
Default Unit → SI

Click→ Available Load Case → Click →Down Load → Create Job → Close → Window



Click→ Pile Cap→ Job→ Click→ Design Parameter→ Values →Changes →Set As Default→ Yes

Click →Pile Layout Predefined →Values Are Changrs→ Click→ Calculate→ Click →3 →Pile Arrangement →Click→ Case1 →Ok→ close

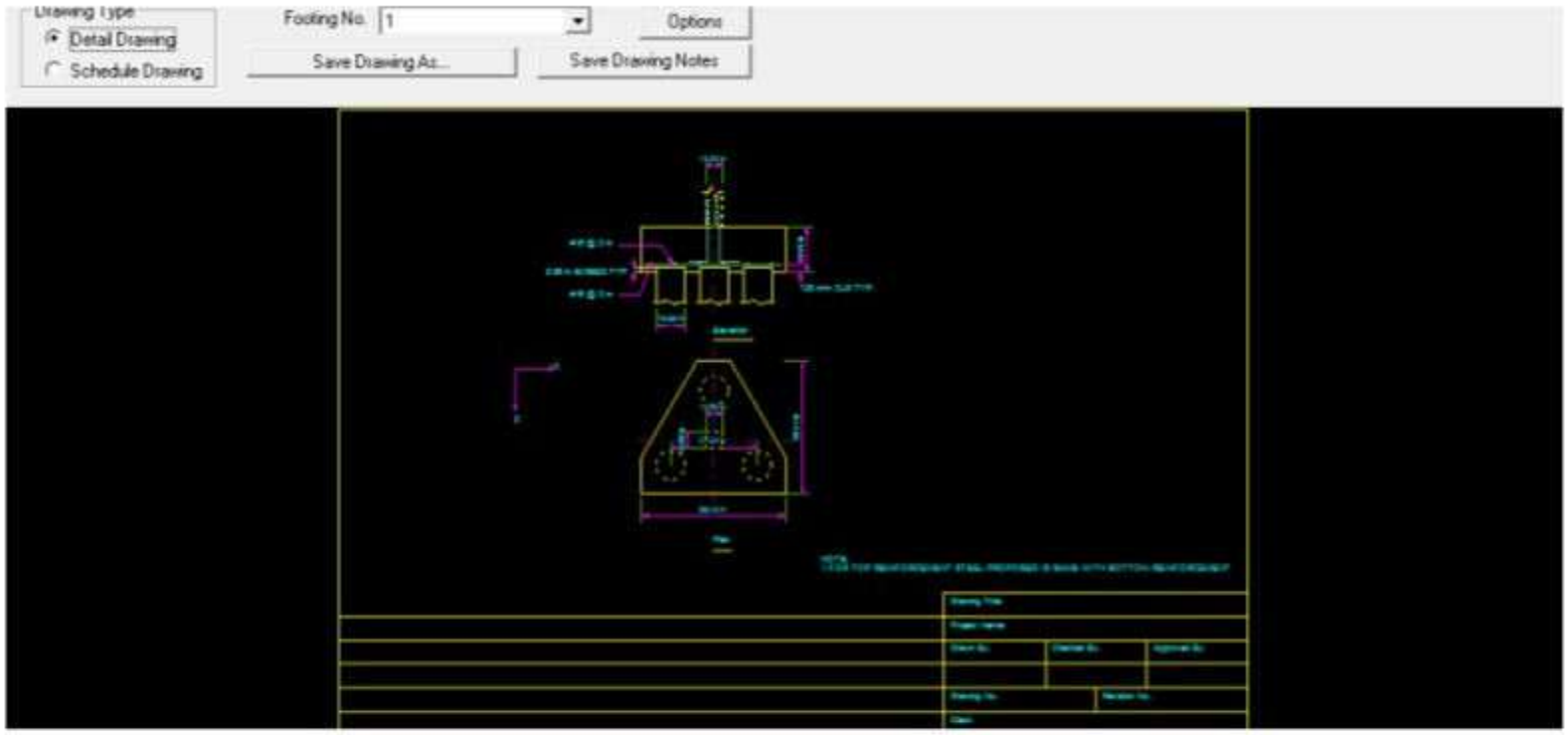


Select→ Design→ Yes→ Ok

Results Are Taken

- Geometry
- Detailed schemed drawing
- GA drawing
- Calculation sheet

Detailed schemed drawing



GA drawing

Calculation sheet

Start Page | Geometry | Detail and Schedule Drawing | GA Drawing | Calculation Sheet | Graphs

Along Alternate Beam

Critical Load Case :

As Per IS 456 2000 Clause 26.5.2.1

Minimum Area of Steel (A_{stmin}) = 2169.796 mm2

As Per IS 456 2000 ANNEX G,G-1.1 b

Area of steel required (A_{stq})= $0.5 \times \left(\frac{f_c}{f_y}\right) \times \left(1 - \sqrt{1 - \frac{4.5977 \times M_u}{f_c \times b \times d \times d}}\right) \times b \times d$ = 2169.796 mm2

Area of steel provided (A_{st}) = = 2169.796 mm2

$A_{stmin} <= A_{st}$ Steel area is accepted

Minimum spacing allowed (S_{min}) = 40 + d_b = 2.20 in

Selected spacing (S) = 8.59 in

S_{min} <= S <= 450 mm and selected bar size < selected maximum bar size... The reinforcement is accepted.

Print Calculation Sheet

DESIGN OF COMBINED FOUNATION USING STAAD Pro

OPEN STAAD FOUNDATION

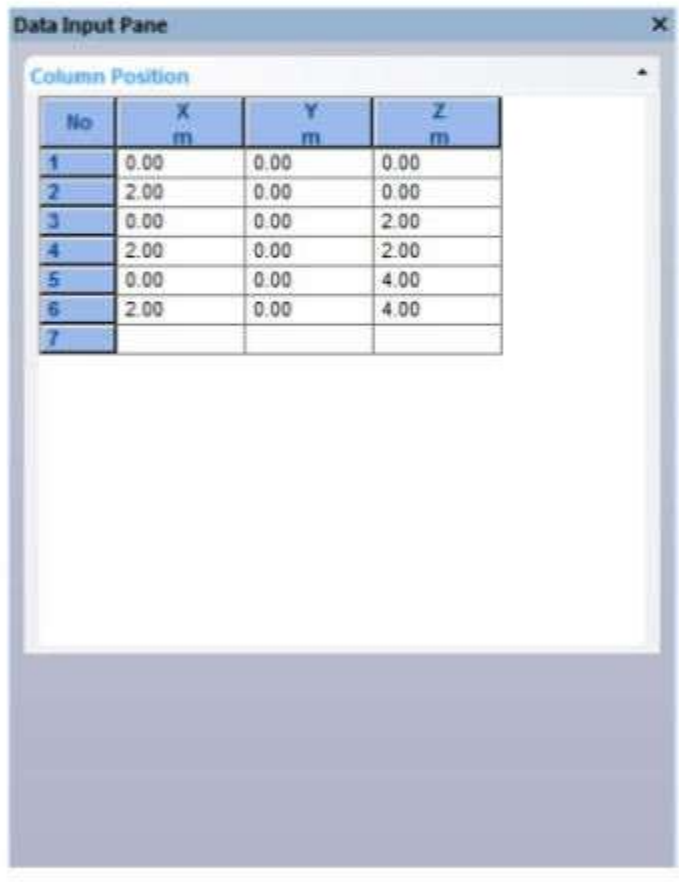
CLICK NEW

- GENERAL SETUP
- GENERAL FOUNDATION MODE
- COLUMN POSITION
- LOAD AND FACTORS
- JOB SET UP
- COMBINED FOOTING DESIGN
- DESIGN

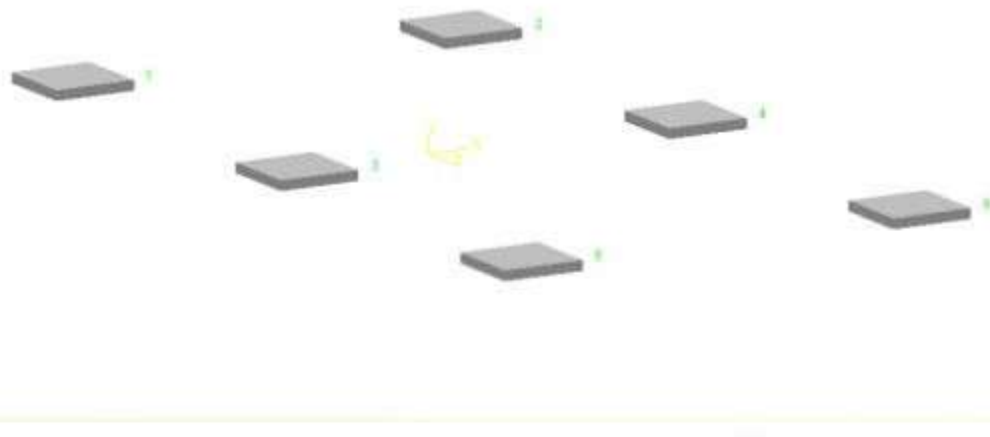
Click → New→ General Setup

Click→ Foundation Mode → Click→ Foundation Plan

Click→ Coolum Position → 0,0,0 → Create The Footing



Click → Column Dimension → Give the Dimension → Ok



Click → Load And Factor → Create New→ Load Case → Load Description→ Dead Load

Load Title = Dead Load

Load Case Type = Service

Loading Type = Dead Add

LIVE Load

Load Title = LIVE Load

Load Case Type = Service

Loading type = live Add

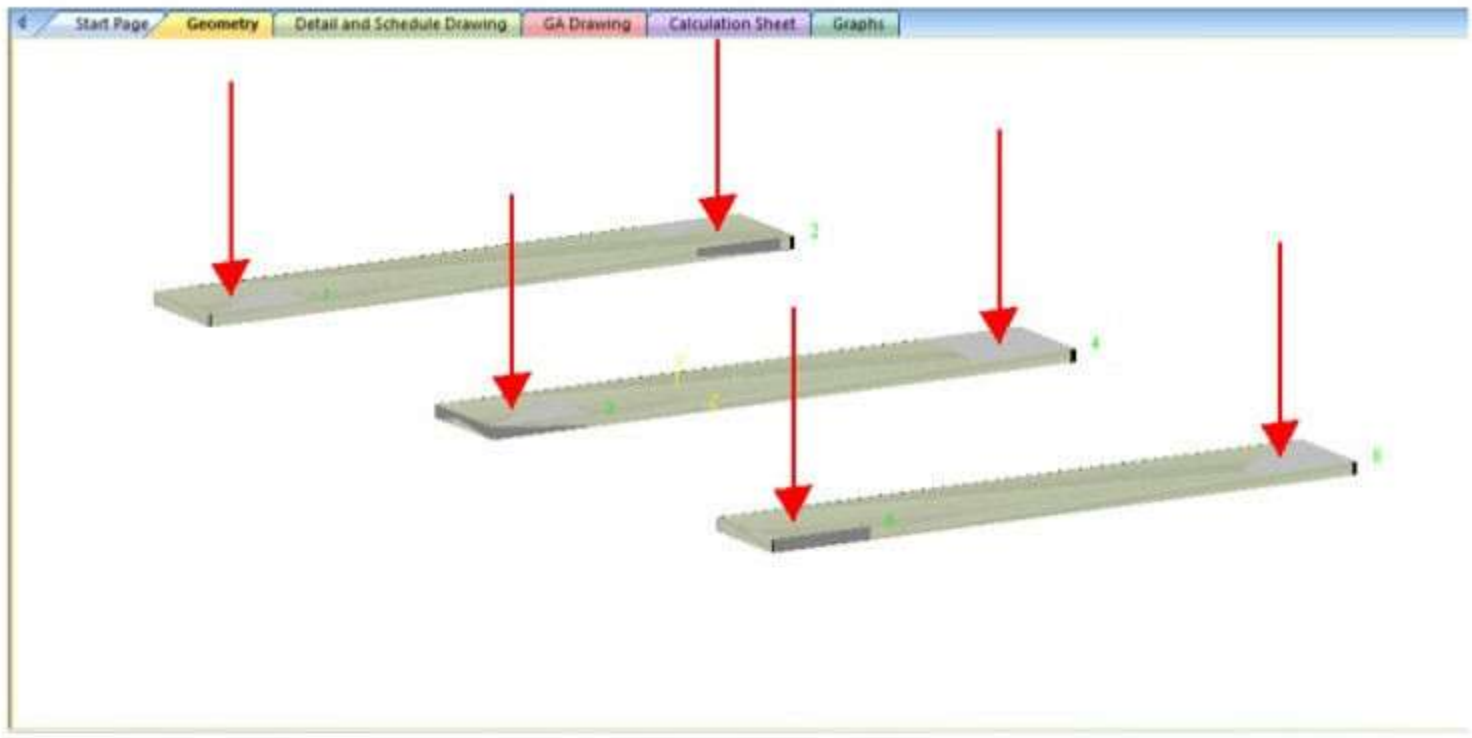
Click → Load Case 1→ Left Click Mouse→ Add Column Reaction→ Load →Dead Load →Click→ Fy -60 →Entre →Add

Live Load → Click → Mouse Left Side → Add Column Reaction → Fy -600 → Entre → Add

Click→ Load→ Assign the Load

See → the Load On Screen

Click → Generate Load Combination → Load Combination → Load Combination →Type →India → Click → Generate Load Combination → Click → Generate Load Combination → close



Click → Job Setup → create → A New Job

Job Name → combined footing

Job Type → combined footing

Design Code → Indian

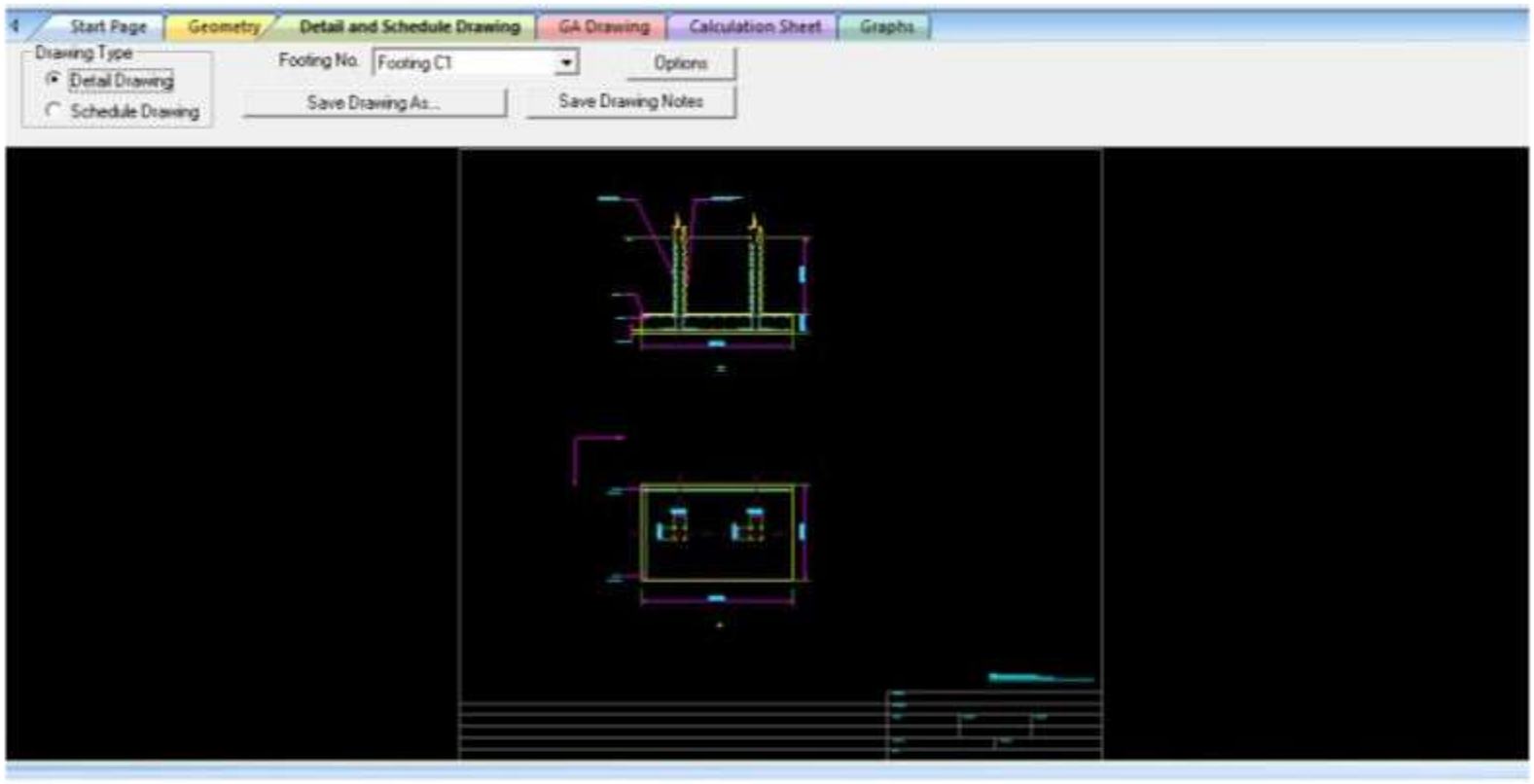
Default Unit → SI

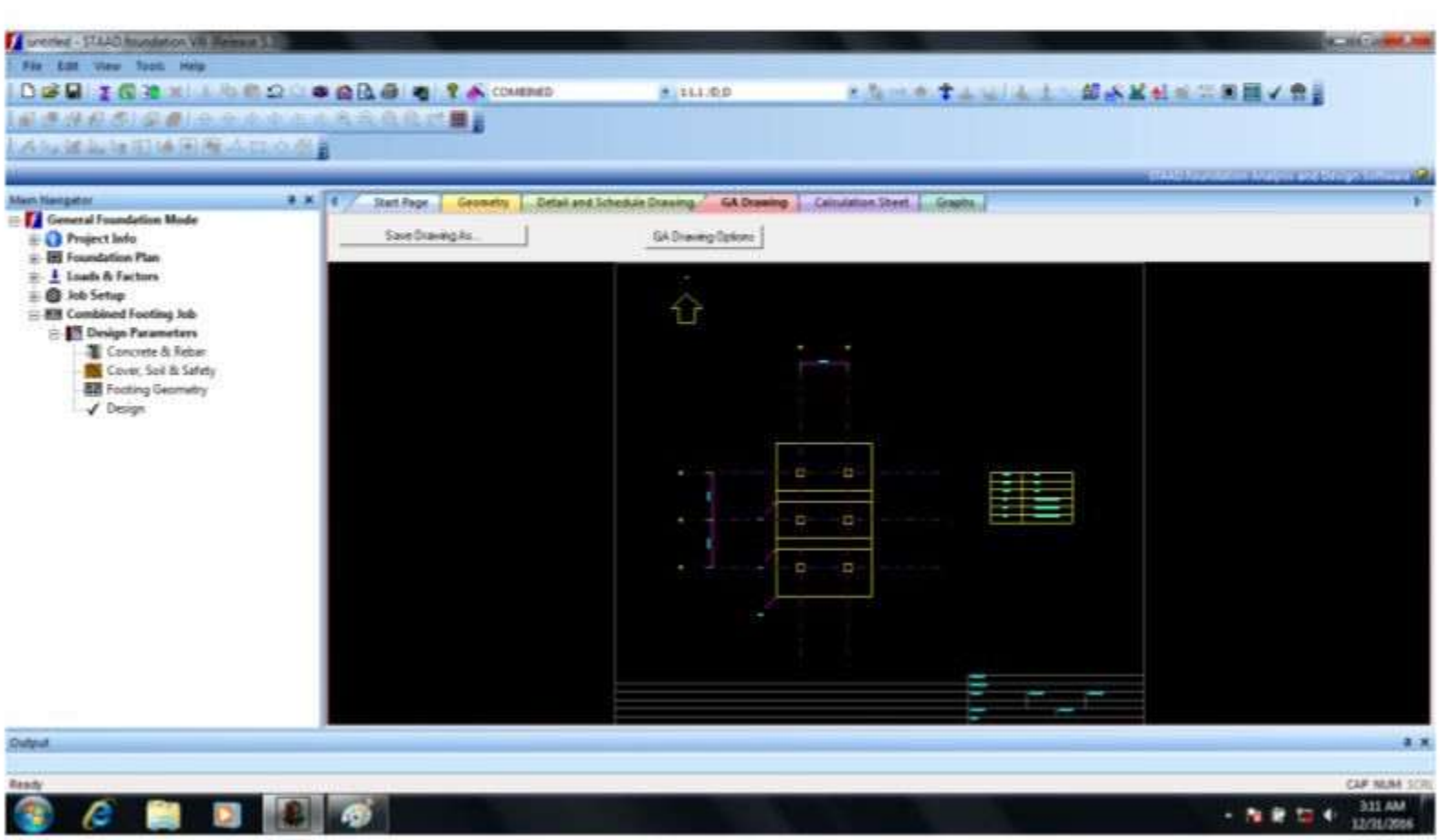
Click→ Available Load Case → Click →Down Load → Create Job → Close → Window

Results Are Taken

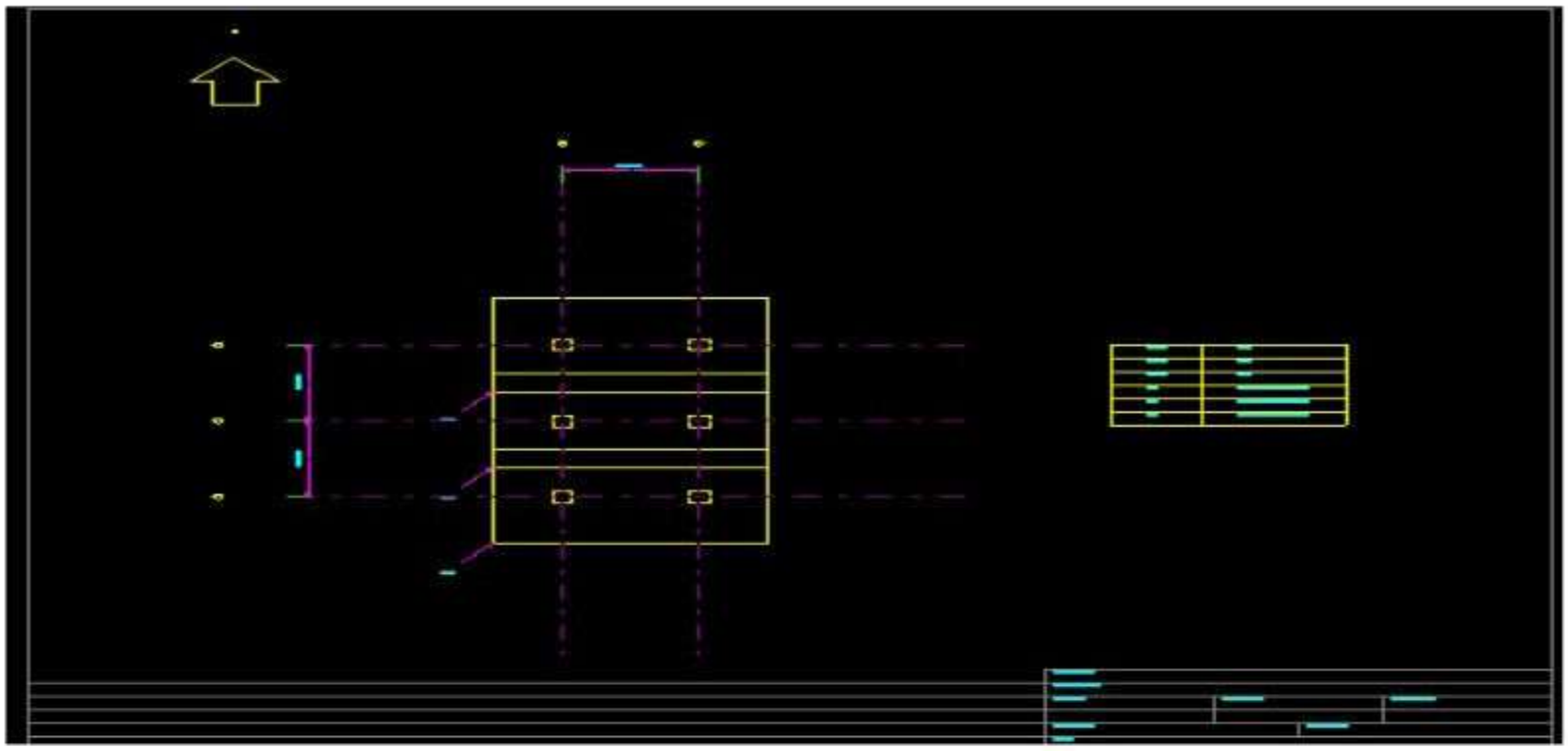
- Geometry
- Detailed schemed drawing
- GA drawing
- Calculation sheet

Detailed schemed drawing





GA drawing



Calculation sheet

Start Page | Geometry | Detail and Schedule Drawing | GA Drawing | Calculation Sheet | Graphs

Based on spacing reinforcement increment; provided reinforcement is

Ø12 @ 175 mm o.c.

Distribution bar no.:	Ø12
Spacing of distribution bars :	185.143 mm

Based on spacing reinforcement increment; provided reinforcement is

Ø12 @ 175 mm o.c.

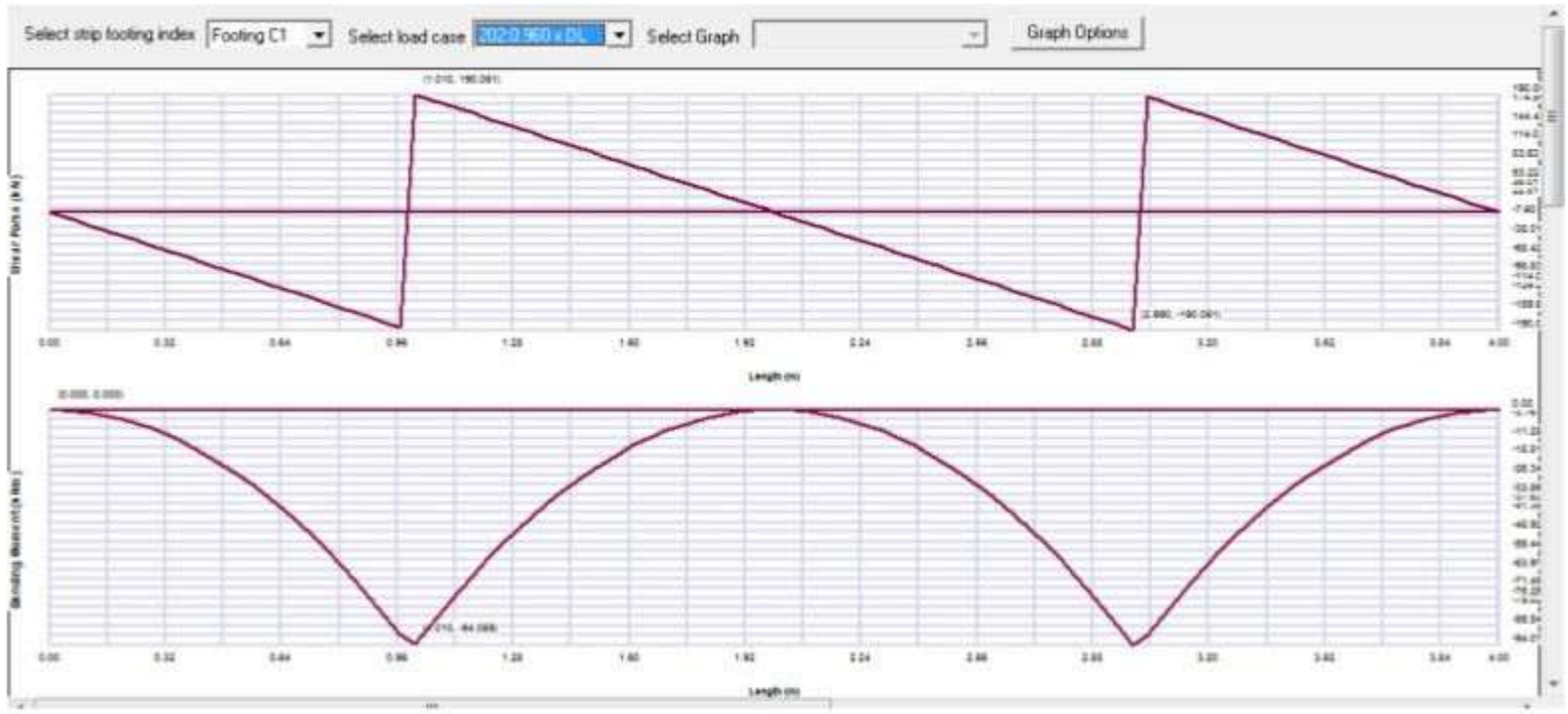
Distribution bar no.(Top):	Ø12
Spacing of distribution bars(Top) :	185.143 mm

Based on spacing reinforcement increment; provided reinforcement is

Ø12 @ 175 mm o.c.

Print Calculation Sheet

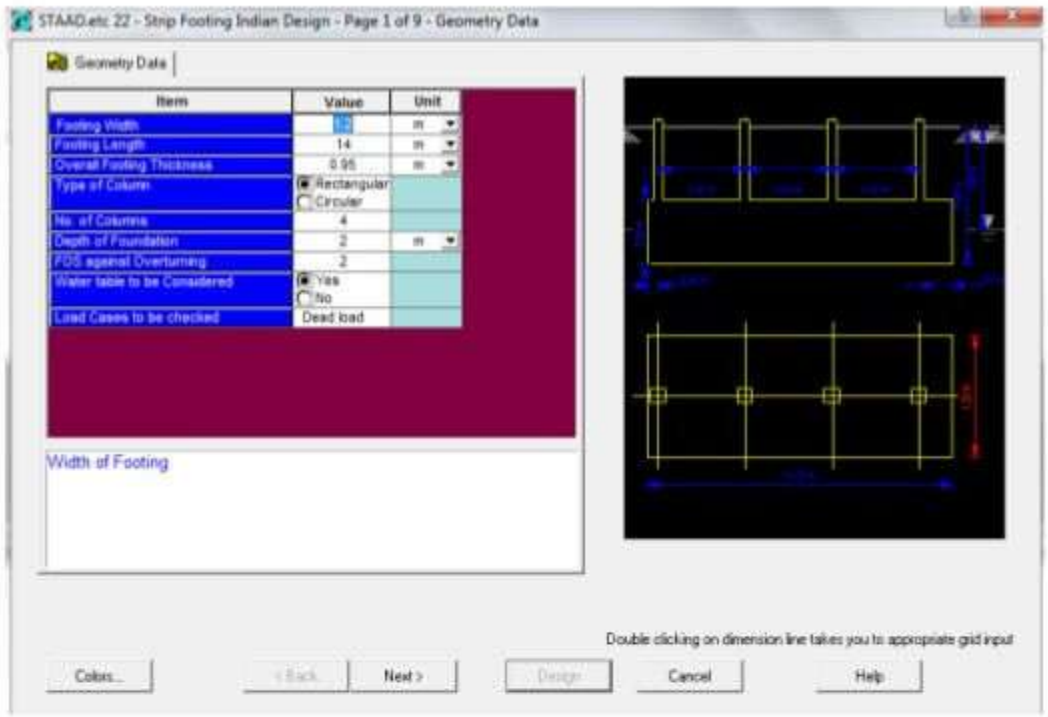
GRAPH

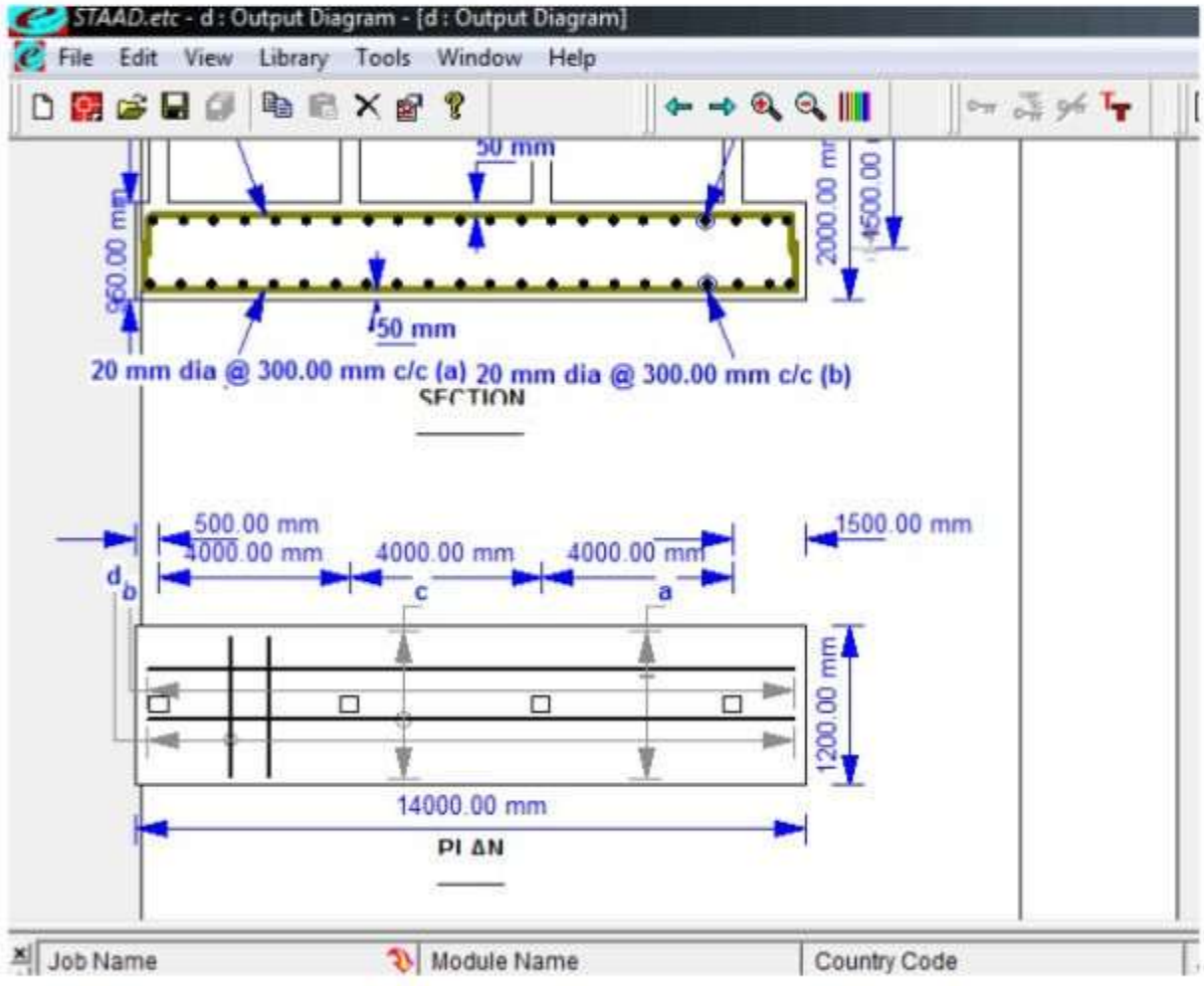

Select strip footing index
Footing C1
Select load case
202:0.960 x DL
Select Graph
Graph Options
Length (m)
Length (m)

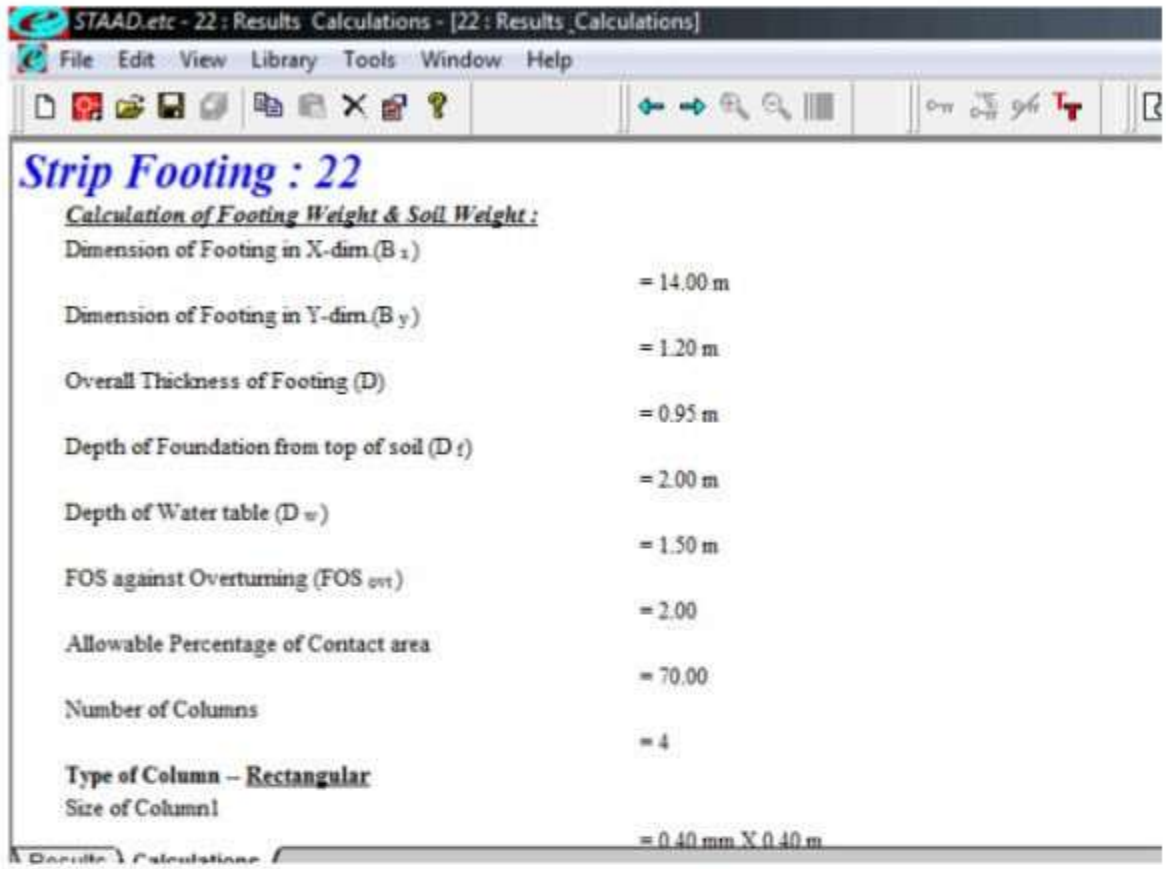
DESIGN OF ETC USING STAAD Pro

STAAD Pro etc

Click →ETC →New job → Click →**Indian** → Click→ Foundation → Click → **Strip Footing** → Fill the job name all → Click → Open → Fill all details given data → Next → **Design** → Input taken → Calculation sheet → Diagram


STAAD.etc 22 - Strip Footing Indian Design - Page 1 of 9 - Geometry Data
Geometry Data
Item
Value
Unit
Footing Width
14
m
Footing Length
14
m
Overall Footing Thickness
0.95
m
Type of Column
Rectangular
Circular
No. of Columns
4
Depth of Foundation
2
m
FOS against Overturning
2
Water table to be Considered
Yes
No
Load Cases to be checked
Dead load
Width of Footing
Double clicking on dimension line takes you to appropriate grid input
Colors...
< Back
Next >
Design
Cancel
Help


STAAD.etc - d : Output Diagram - [d : Output Diagram]
File Edit View Library Tools Window Help
50 mm
2000.00 mm
4500.00 mm
950.00 mm
50 mm
20 mm dia @ 300.00 mm c/c (a) 20 mm dia @ 300.00 mm c/c (b)
SECTION
500.00 mm
4000.00 mm
4000.00 mm
4000.00 mm
1500.00 mm
d
b
c
a
1200.00 mm
14000.00 mm
PLAN
Job Name
Module Name
Country Code


STAAD.etc - 22 : Results Calculations - [22 : Results_Calculations]
File Edit View Library Tools Window Help
Strip Footing : 22
Calculation of Footing Weight & Soil Weight :
Dimension of Footing in X-dim.(B x)
= 14.00 m
Dimension of Footing in Y-dim.(B y)
= 1.20 m
Overall Thickness of Footing (D)
= 0.95 m
Depth of Foundation from top of soil (D f)
= 2.00 m
Depth of Water table (D w)
= 1.50 m
FOS against Overturning (FOS ovt)
= 2.00
Allowable Percentage of Contact area
= 70.00
Number of Columns
= 4
Type of Column -- Rectangular
Size of Column1
= 0.40 mm X 0.40 m

DESIGN OF SHEAR WALL USING

STAAD Pro

Step 1

Open the staad pro

New → New file → Space

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

Add beam → **Finish**

Step 3

Click → geometry → **Nodes**

Node	x	y	z
1	m	m	m

Diagram

SAME PROCEDUR FOR ABOVE DESIGN WATER TANK

Click → create the surface

Click → Geometry → **Add Surface** → Click → the Building → Click the Left Side Mouse → Create the Surface

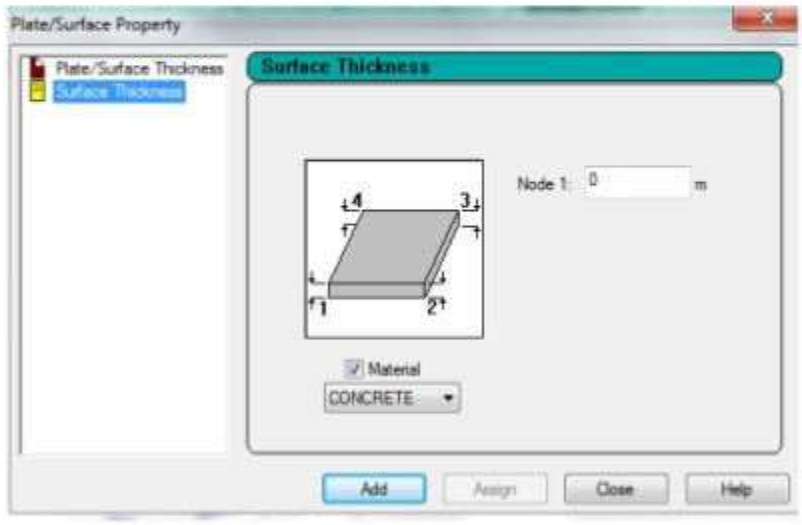
Material

Click→ Modeling → General→ Material → Material Whole structure → **click Concrete** →Assign to view → **Assign** → **Yes**

Property

Click → Modeling → General → property → Property Whole structure → Define → Property → Rectangle YD **.23m ZD .23m** → Add → Close → Assign to view → **Assign** → **Yes**

Click → Modeling → General → Property → Property Whole Structure → **Thickness** → **Surface Thickness 120 mm** → Add → Close → Assign To View →Assign → Yes



Support

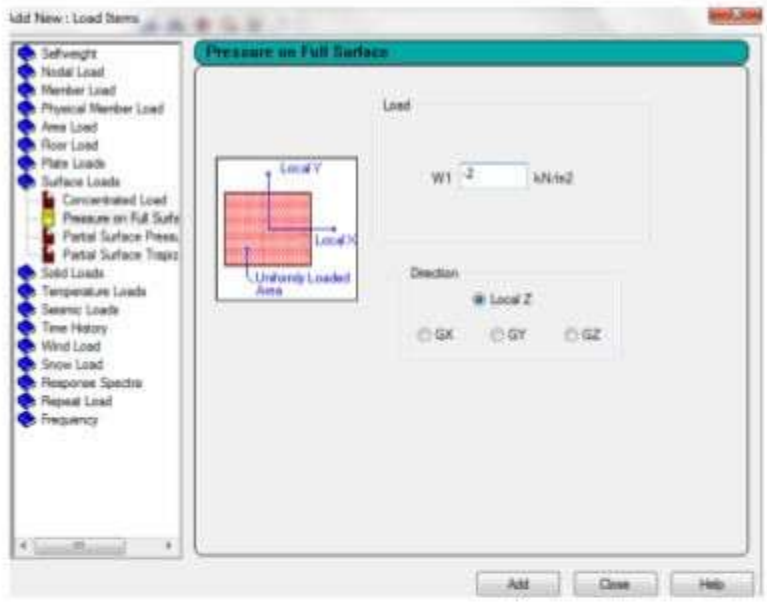
Click → Modeling → General → Support → Support Whole structure → Create → **fixed** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**

Loads

Click → Modeling → General → Load → Click Load Case Details → Add → Add New Load Cases → **Add – Load Case 1** → **Add- Load Case 2** → Click Load Case

1 Add **Self-Weight** → Add → Close → Click Load Case 2 → **Surface Loads** → **Surface Of All Pressure** -2kn/M^2 GY →**Add** → **Close**

Select load→ w1 -2 kN/m^2 →Select →**Assign To View** → **Assign**→ **Yes**



ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

Click the beam see the result on beam

DESIGN OF SHEAR WALL

CLICK → Modeling → Design → **Shear Wall** → Select → Surface → Current **Code Is 456**

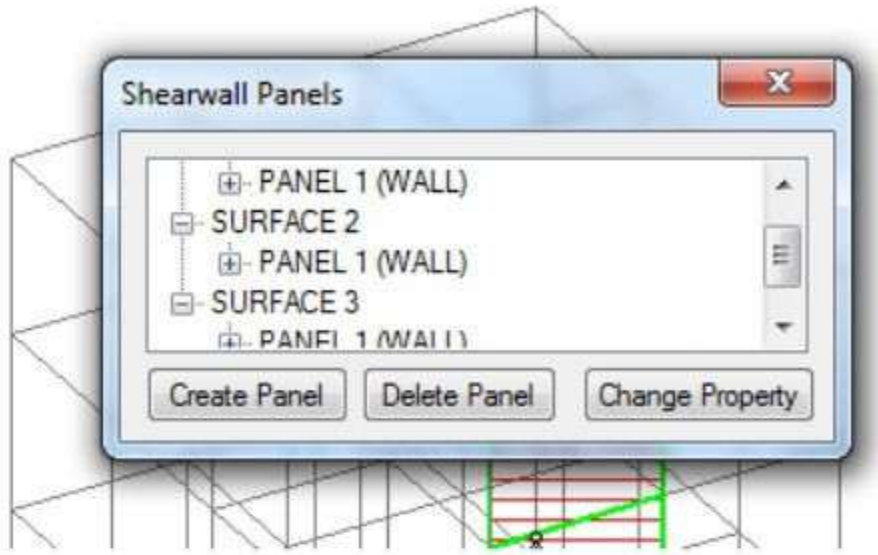
Click → Select → **Parameter Change** → Left Side to Right Side

Click → **Define Parameter** → Enter Value

Click → **Shear Wall** → Click → Commands → Add → Assign → Close

Analysis → Run Analysis → save Result → **Output**

Create → **the shear wall panel**. → Select→ shear wall → draw→ click→ **wall**



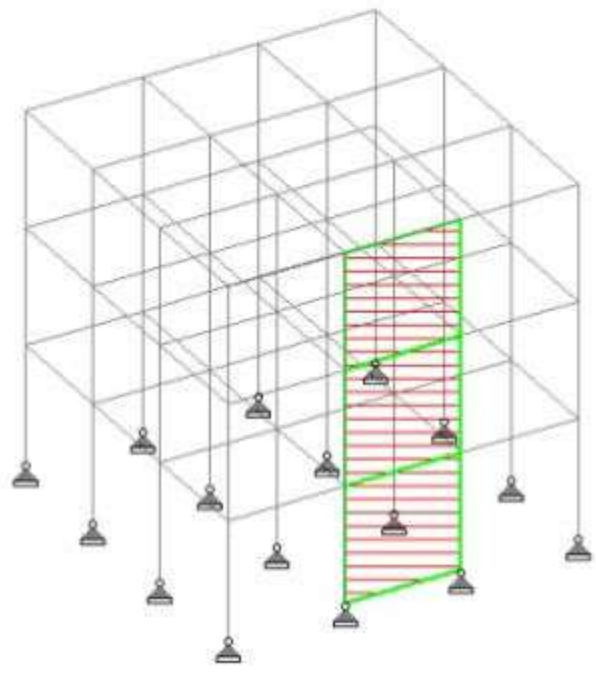
CLICK → Modeling → Design → **Shear Wall** → Select → Surface → Current **Code Is 456**

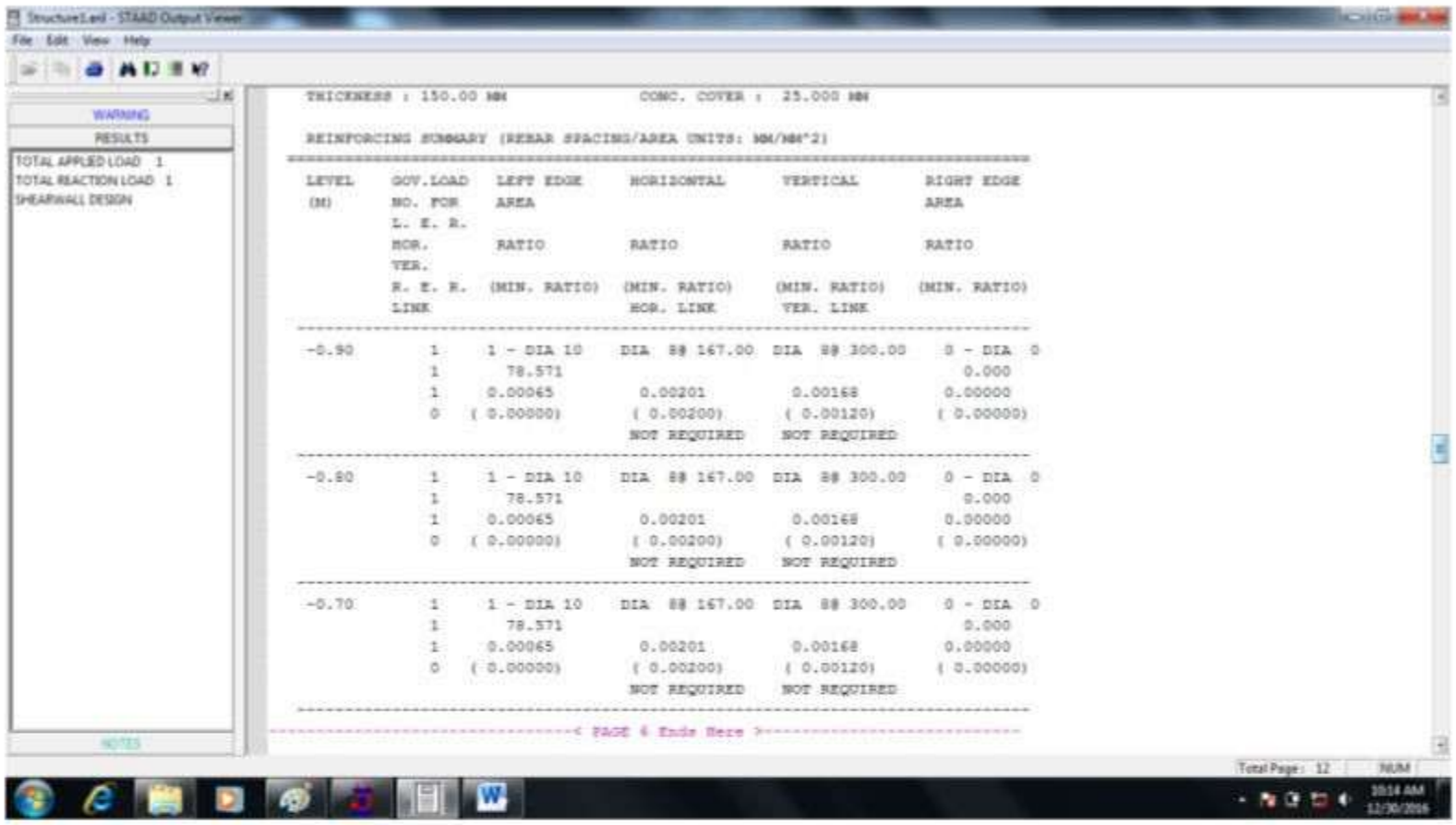
Click → Select → **Parameter Change** → Left Side to Right Side

Click → **Define Parameter** → Enter Value

Click → **Shear Wall** → Click → Commands → Add → Assign → Close

Analysis → Run Analysis → save Result → **Output**




File Edit View Help
WARNING
RESULTS
TOTAL APPLIED LOAD 1
TOTAL REACTION LOAD 1
SHEARWALL DESIGN
NOTES
THICKNESS : 150.00 MM CONC. COVER : 25.000 MM
REINFORCING SUMMARY (REBAR SPACING/AREA UNITS: MM/MM^2)
LEVEL (M) | GOV.LOAD NO. FOR L. E. R. HOR. VER. R. E. R. LINK | LEFT EDGE AREA RATIO (MIN. RATIO) | HORIZONTAL RATIO (MIN. RATIO) HOR. LINK | VERTICAL RATIO (MIN. RATIO) VER. LINK | RIGHT EDGE AREA RATIO (MIN. RATIO)
-0.90 1 1 1 0 | 1 - DIA 10 @ 167.00 78.571 0.00065 (0.00000) | DIA @ 300.00 0.00201 (0.00200) NOT REQUIRED | 0 - DIA @ 0.000 0.00168 (0.00120) NOT REQUIRED | 0 0.00000 (0.00000)
-0.80 1 1 1 0 | 1 - DIA 10 @ 167.00 78.571 0.00065 (0.00000) | DIA @ 300.00 0.00201 (0.00200) NOT REQUIRED | 0 - DIA @ 0.000 0.00168 (0.00120) NOT REQUIRED | 0 0.00000 (0.00000)
-0.70 1 1 1 0 | 1 - DIA 10 @ 167.00 78.571 0.00065 (0.00000) | DIA @ 300.00 0.00201 (0.00200) NOT REQUIRED | 0 - DIA @ 0.000 0.00168 (0.00120) NOT REQUIRED | 0 0.00000 (0.00000)
< PAGE 6 Ends Here >
Total Page: 12
NUM

DESIGN OF SLAB DECKS USING STAAD Pro

Step 1

Open the staad pro

New → New file → Space

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

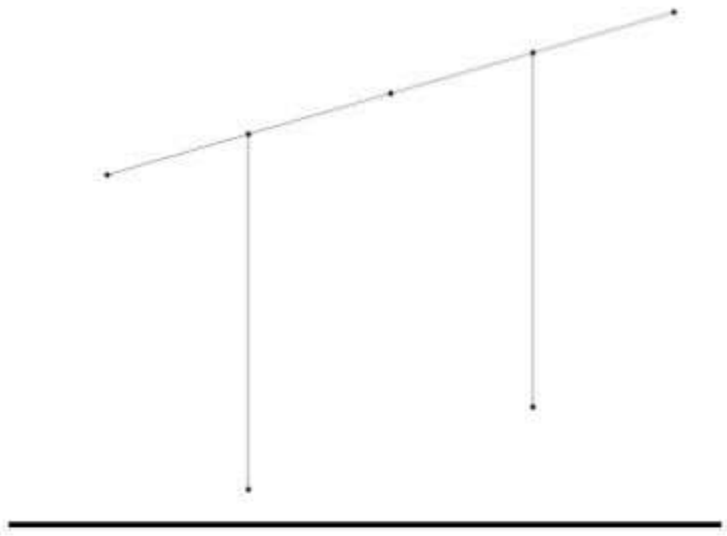
Add beam → **Finish**

Step 3

Click → geometry → **Snap node beam**

	left	**right**	**m**
x	**0**	**4**	**4**
y	**0**	**10**	**1**

Create The Structure → Draw → Close → Click → Centre Of Beam → Left → Click → Mouse →Insert Node → **Add Mid-Point** →Mid-Point → Ok



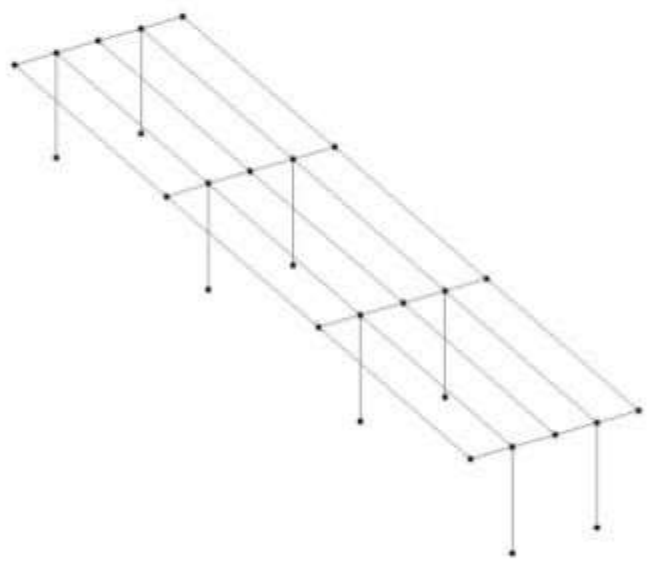
Select → All Members → Click→ Geometry → **Translation Repeat**

Global direction z

No of step 3

Default step spacing 25 m

Click → link steps → click → open base → ok → see → whole structure



Create a Plate Click → Generate The Mesh → Select →The Plate → Left → Click → Generate Mesh → Quaid →Ok →Div. **16 75 16 75**→ Apply

Step 4

- Material
- Property
- Support
- Load
- Analysis

Material

Click→ Modeling → General→ Material → Material Whole structure → **click Concrete** →Assign to view → **Assign** → **Yes**

Property

Click → Modeling → General → property → Property Whole structure → Define → Property → Circular **1 m** → Rectangle YD **.5m ZD .5m** → plate thickness **.3m** → Add → Close → Assign to view → **Assign** → **Yes**

Support

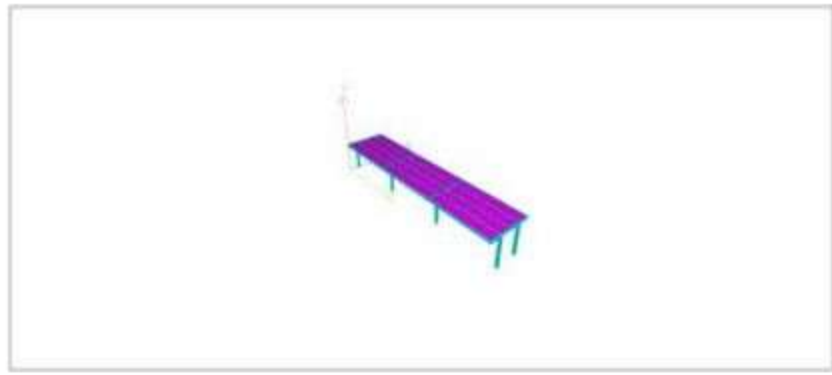
Click → Modeling → General → Support → Support Whole structure → Create → **pinned** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**

Click → Modeling → General → Load → Click load case details → Add → Add new load cases → **Add – Load case 1** → Click Load case 1 Add **self-weight** → Add → close

ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ save → **Output Result**



Step 5

Post processing

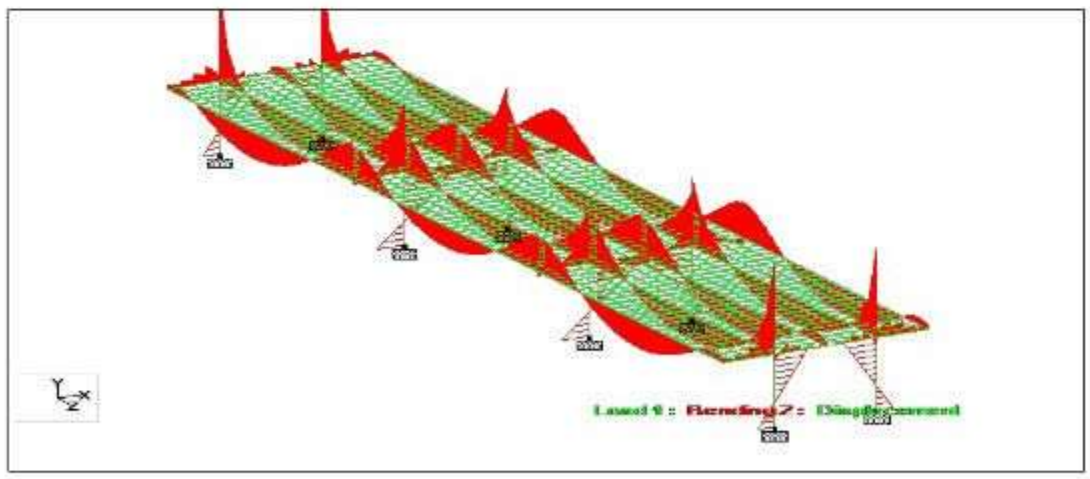
Click → Post processing → Result setup → Select load case → **Ok**

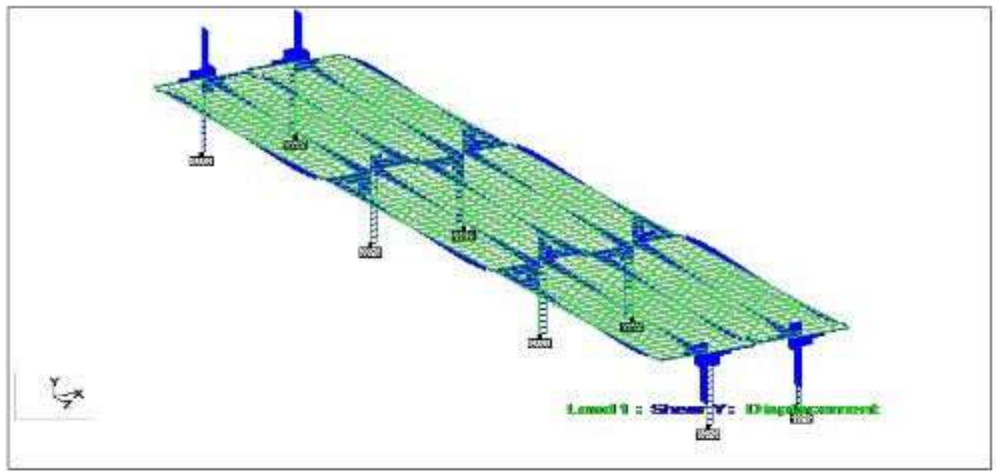
New screen will be displayed →Click → Result → Animation →**Deflection** → **Ok** → F12 to see full screen of deflection

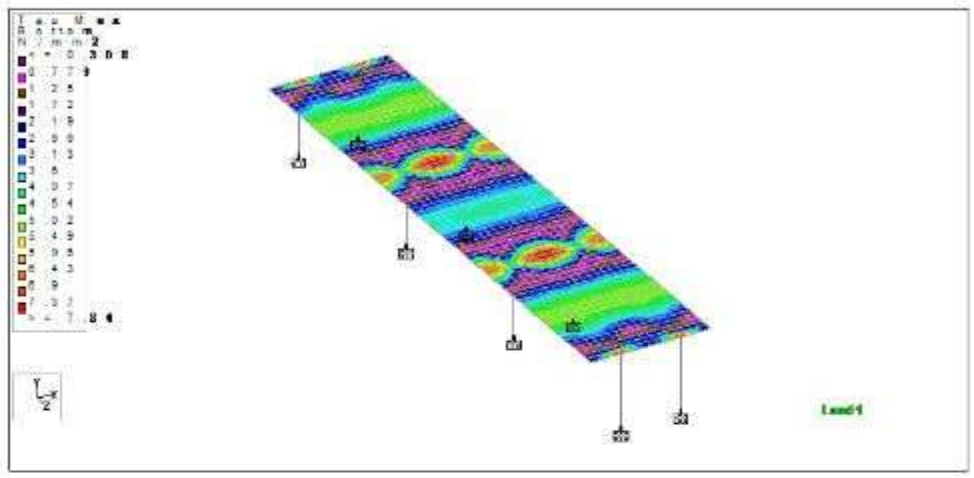
Click→ Result→ **Bending moment** →Scroll the mouse → to see the whole structure bending moment diagram → Value to be noted

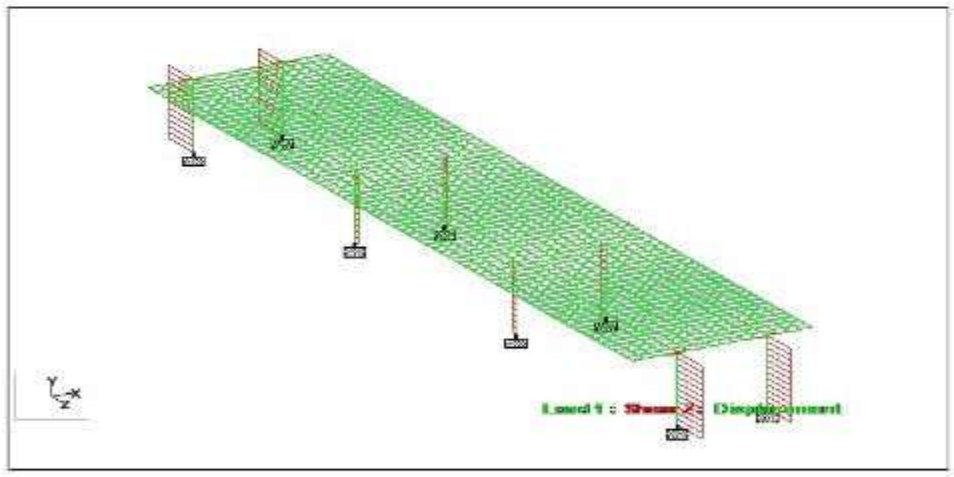
Click→ Result→ **Section displacement** → Scroll the mouse → to see the whole structure section displacement diagram →Value to be noted

Click → Result → **Beam stress** → Click→ Beam stresses → to click any one beam → Open 3d beam stress coutour displayed → Distance to be provided the beam → Add to stress table → Values to be noted





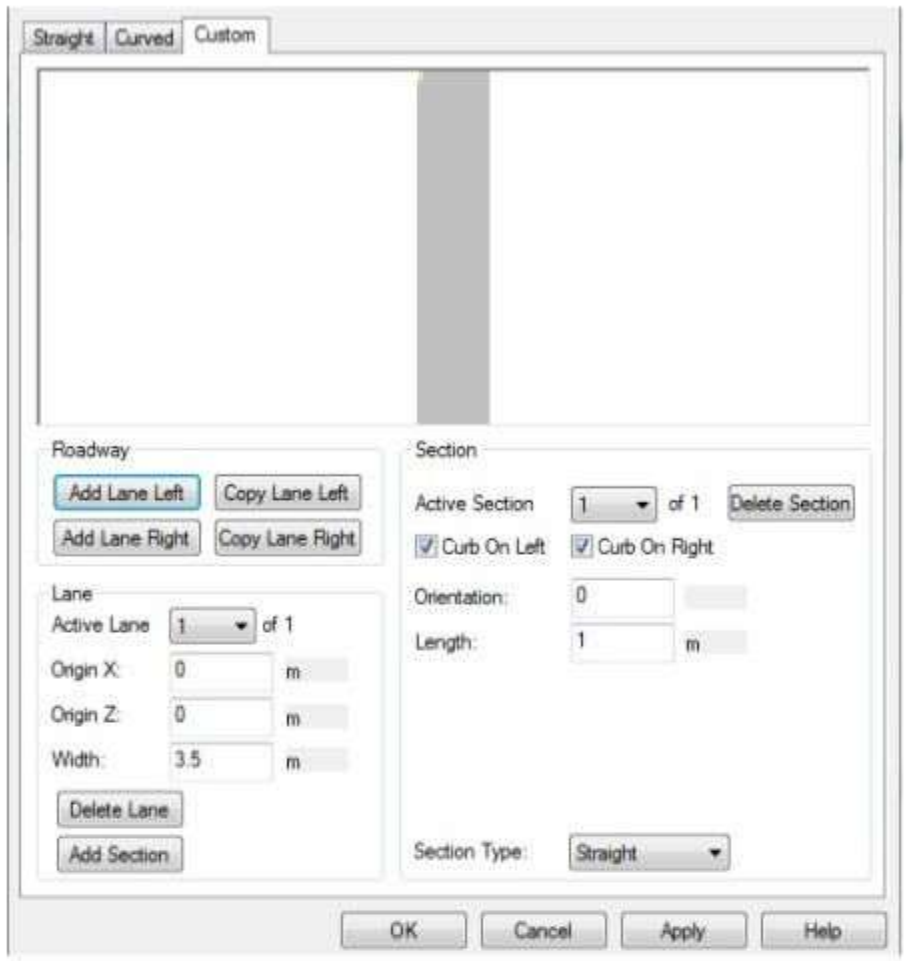




Step 6

Select → Bridge Deck →Select → **The Plate** → Click →**Deck Create** → Deck → Name Bridge → Ok

Click → Deck → **Define Roadway** →Roadways →Select →New Define → Roadways → **Custom Roadway**



1 Select →Add Left →Lane →Origen X 2m → Orientation 90

Length 75 m

Origen Z O m

Width 4 m

→ Apply

2 Select →Add→ Left Lane →Orgine X 2m → Orientation 90

Length 75 m

Origen Z O m

Width 4 M

→Apply

3 Select →Add →Left Lane →Origen X 2m → Orientation 90

Length 75 m

Orgine Z Om

Width 4 M

→ Apply

4 Select →Add Left Lane→ Origen X 2m→ Orientation 90

Length 75m

Orgine Z Om

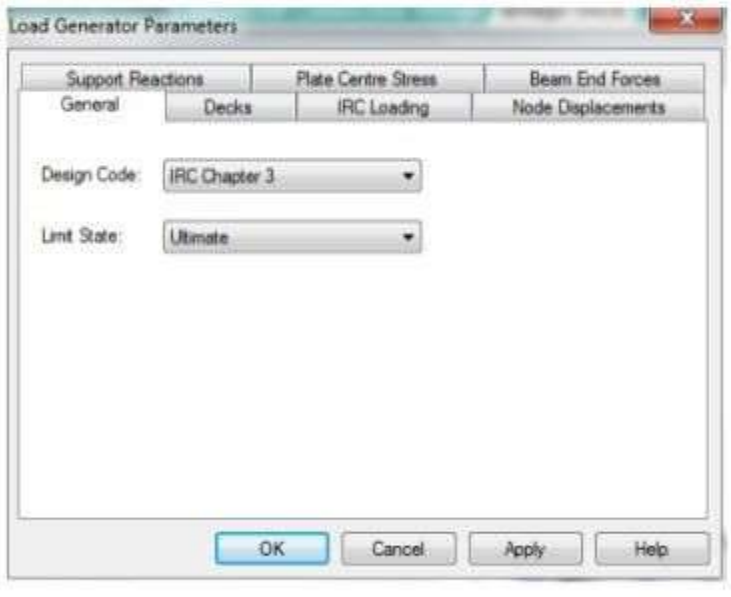
Width 4 M

→Apply

→Close → Close

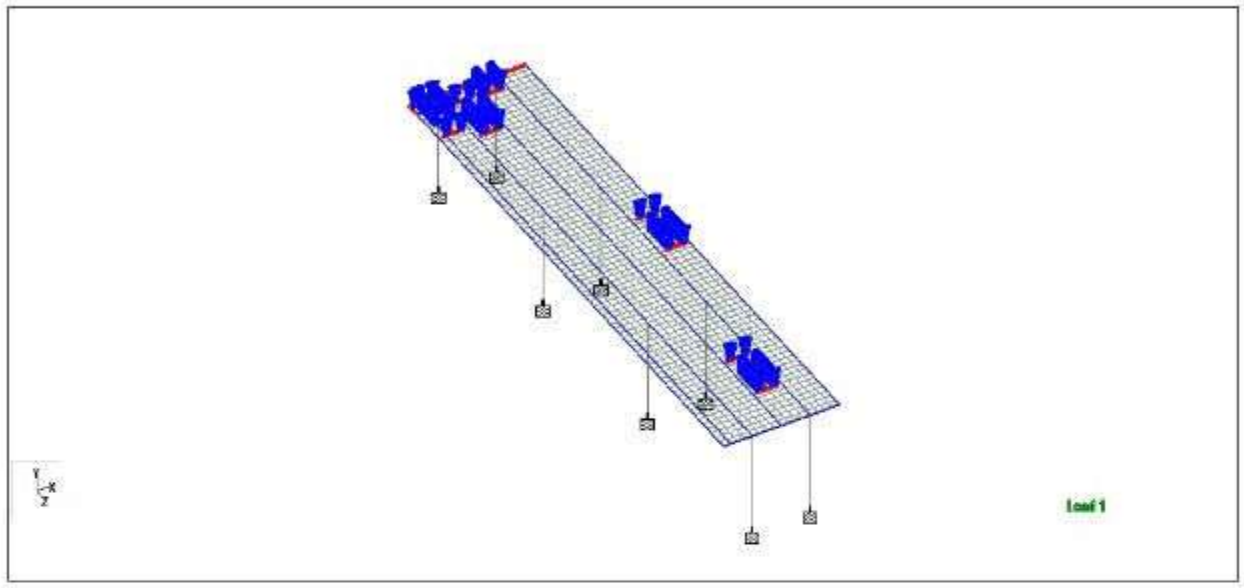
Select →Loading→ Click →**Surface Influence** →Result Noted

Click →Loading →Run Load→ **Generator Load** →Generator Parameter →General



- General
- Decks
- IRC Loading
- Node Displacement

- Apply Ok
- See The The Result
- Click →The Mouse→ Left Side → **Labels** →Decks →Results →Click→ Load →Click →Vehicle → Apply→ Ok



DESIGN OF RAM CONNECTION USING STAAD Pro

Step 1

Open the staad pro

New → New file → Space

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

Step 2

Where do you want to go?

Add beam → **Finish**

Step 3

Click → geometry → **Nodes**

Node	x	y	z
1	m	m	m

Step 4

- Material
- Property
- Support
- Load
- Analysis

Material

Click→ Modeling → General→ Material → Material Whole structure → **click Concrete** →Assign to view → **Assign** → **Yes**

Property

Click → Modeling → General → property → Property Whole structure → Define → Property → Rectangle YD **.23m ZD .23m** → Add → Close → Assign to view → **Assign** → **Yes**

Click → Modeling → General → Property → Property Whole Structure → **Thickness** → **Plate Thickness 120 mm** → Add → Close → Assign To View → Assign → Yes

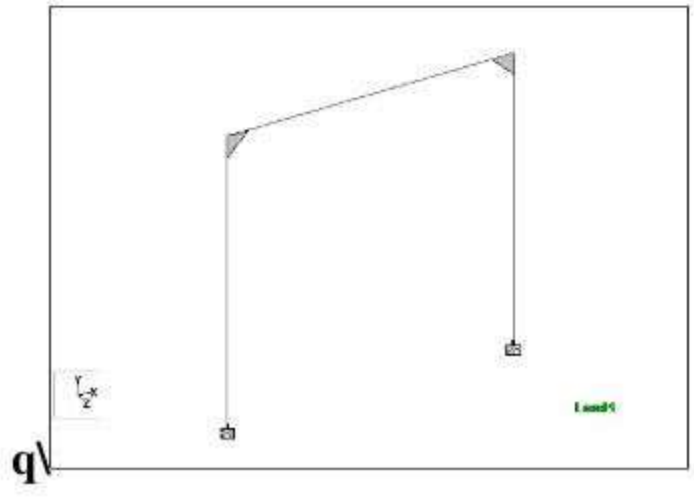
Support

Click → Modeling → General → Support → Support Whole structure → Create → **fixed** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**

Loads

Click → Modeling → General → Load → Click load case details → Add → Add new load cases → **Add – Load case 1** → **Add- Load case 2** → Click Load case 1 Add **self-weight** → Add → Close → Click Load case 2 → plate load → **pressure on full plate -2kN/m^2 GY** → **Add** → **Close**

Select load→ PR -2 kN/m^2 →Select →**Assign To View** → **Assign**→ **Yes**



ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

Step 4

- Design load envelops
- Ram connection setting
- Smart connection
- Connection assignment

Click →Design →Load Envelops→ Select→ Load →Ok

Click → Ram Connection →Setting →Design Code → AISC 360-05 (LRFD) → Ok

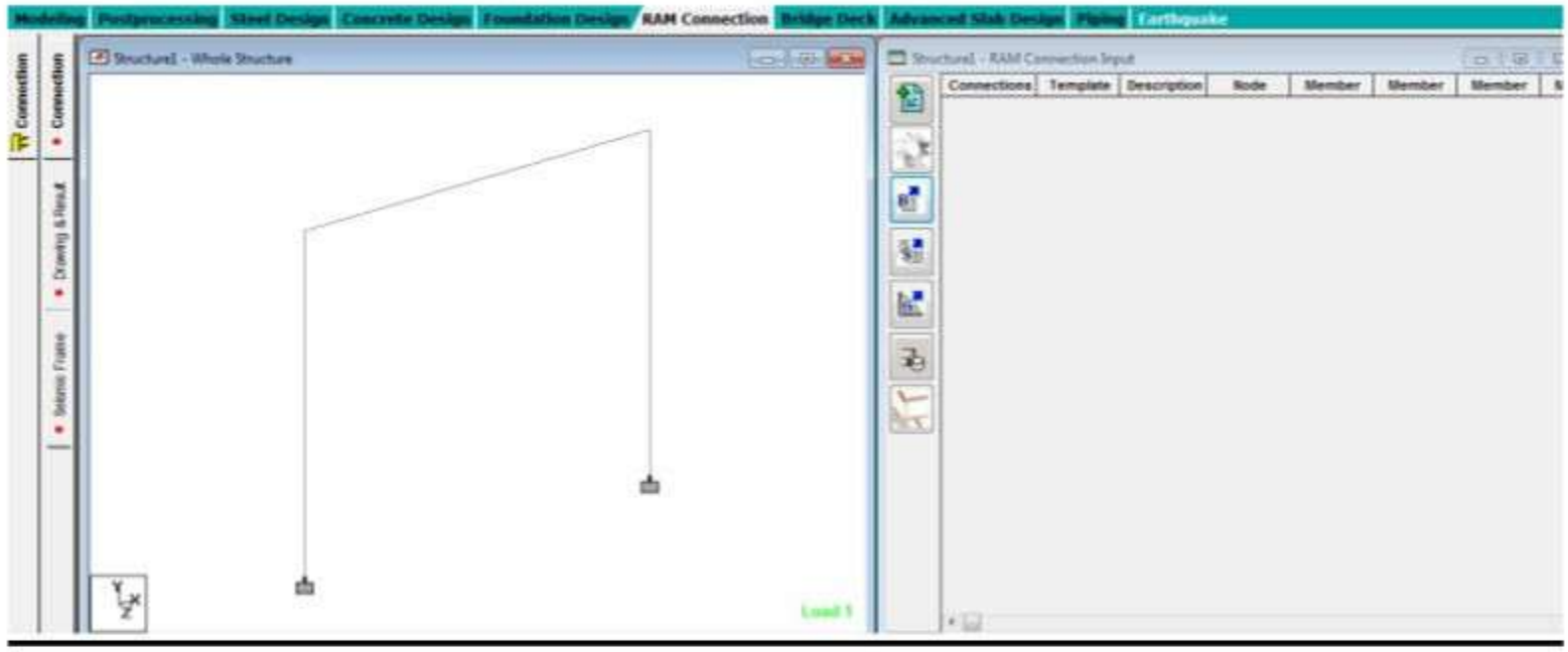
Select→ The Structure Beam Column Number

Click →Smart Connection Clip Angle Bcf →Click →Clip Angle Beam Column Flange →Available→ Left Side To Right Side →Ok

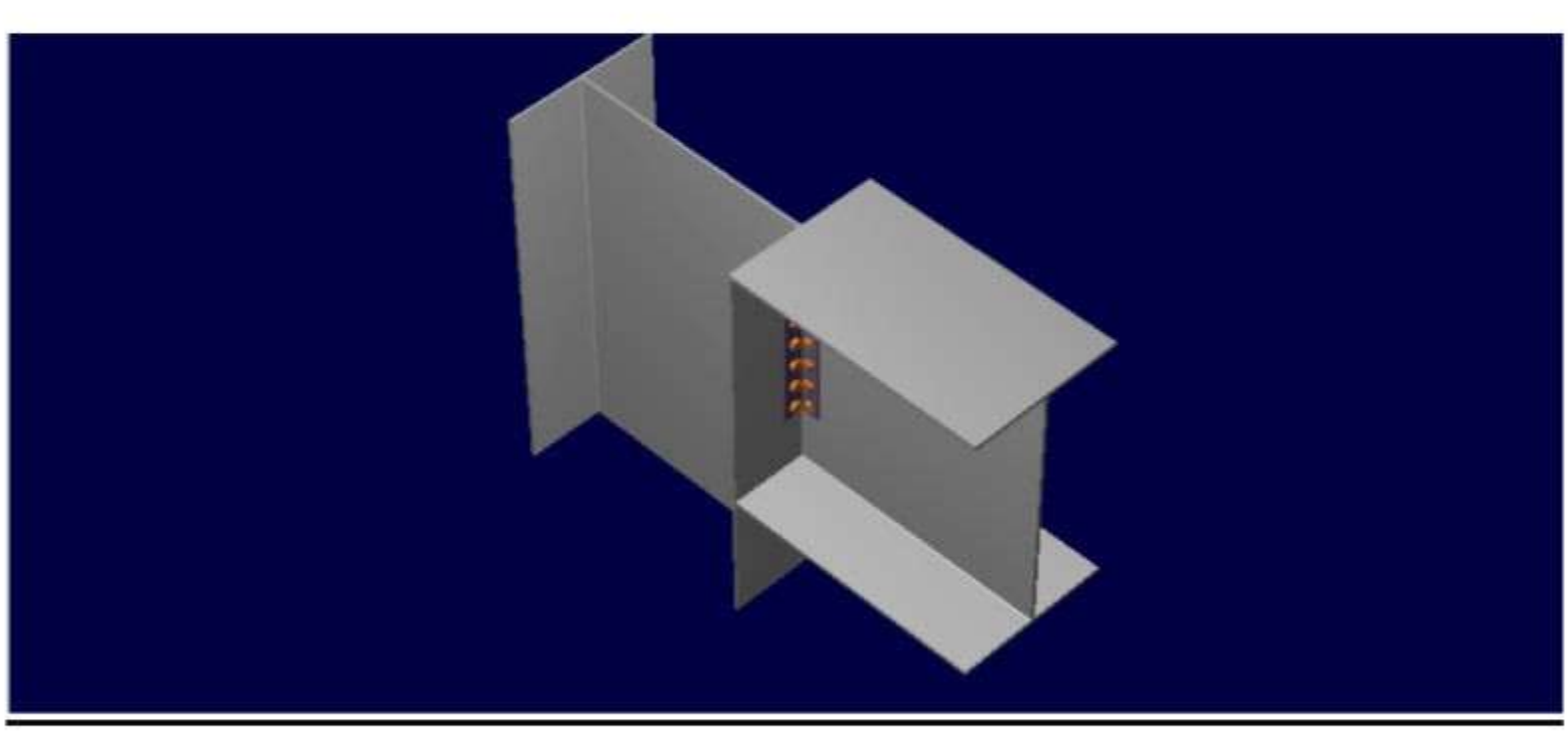
Connection Assignment Has Been Designed →Ok→ Close

Click →Mouse →Select Joint→ Select The Joint

Ram connection



3d view bolts



Click result

(fx) View Formulas | GenerateRTF | Close

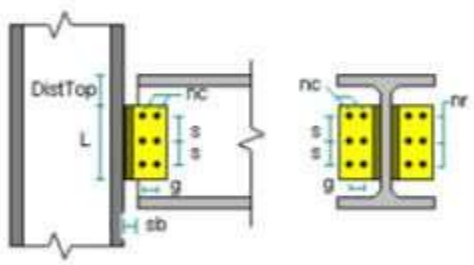
Beam	LC-1	Design	8.92	--	--	--	-1.21

GEOMETRIC CONSIDERATIONS

Dimensions	Unit	Value	Min. value	Max. value	Sta.	References
Angle						
Length	[mm]	444.60	394.00	788.00	✔	p. 10-8
Thickness	[mm]	4.78	--	15.87	✔	p. 10-9
Angle (Beam side)						
Vertical edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.
Horizontal edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.
Vertical center-to-center spacing (pitch)	[mm]	76.20	50.80	114.60	✔	Sec.J3.3,Sec.J.
Angle (Support side)						
Vertical edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.
Horizontal edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.
Vertical center-to-center spacing (pitch)	[mm]	76.20	50.80	114.60	✔	Sec.J3.3,Sec.J.
Beam						
Horizontal edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.
Support						
Horizontal edge distance	[mm]	31.75	25.40	--	✔	TablesJ3.4,J3.

Click data

Connection Template : DA BCF All bolted
Connection ID : BCF - N(2) - M(1,2)
Design Code: AISC-LRFD
Status:: OK



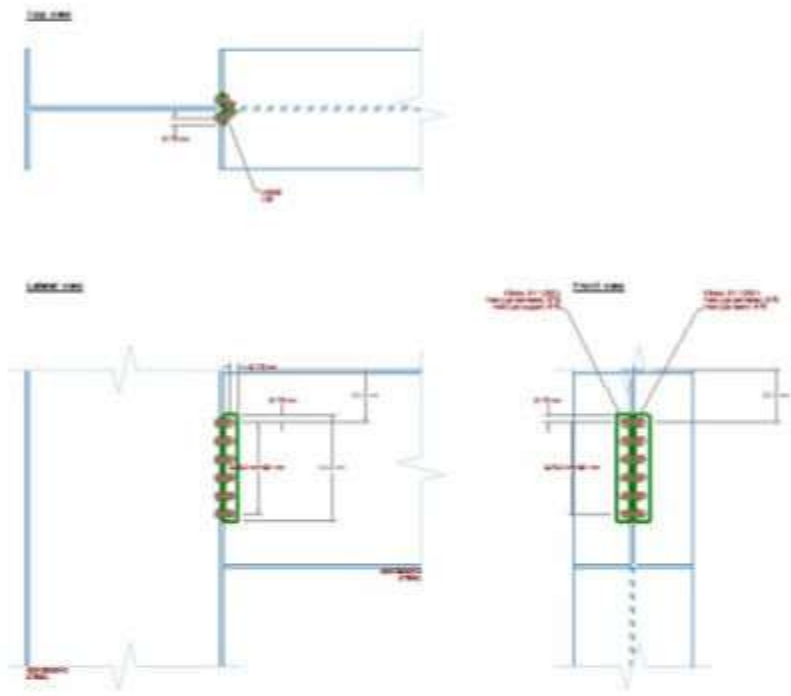
GENERAL DATA

Consider hole deformation in bolts	:	Yes
Is column end	:	Yes
Consider sheared edges in shapes	:	No
Consider sheared edges in shapes	:	No
Corrosive influences	:	No

MEMBERS:

Beam		
Section	=	I80012B50012
Material	=	STEEL

DXF view



DESIGN OF CABLE BRIDGE DECK USING STAAD Pro

Step 1

Open the staad pro

New → New file → Space

Create file name = Name

Location = **E**

Length unit = meter Force unit = kilo newton → **Next**

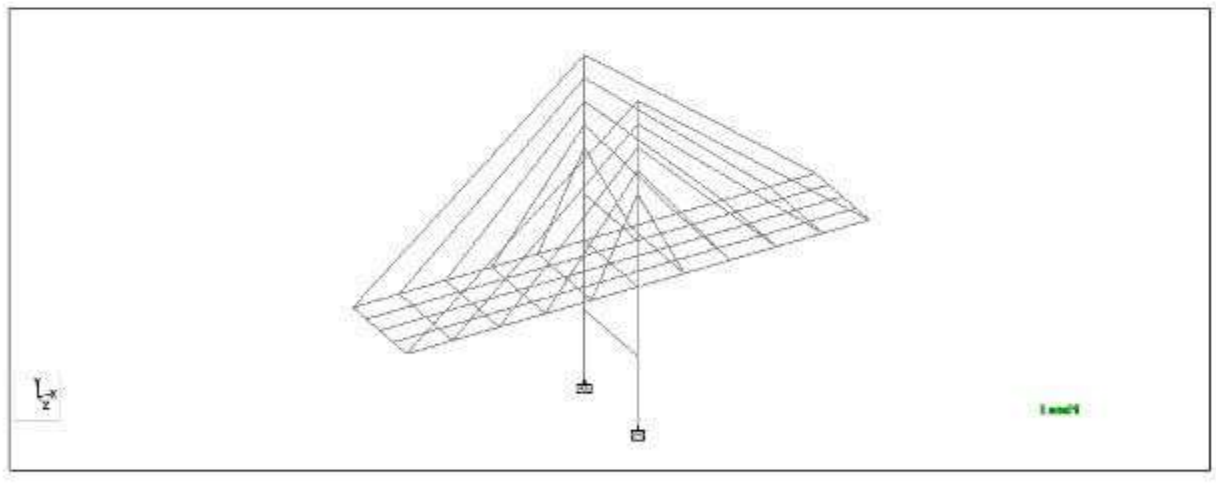
Step 2

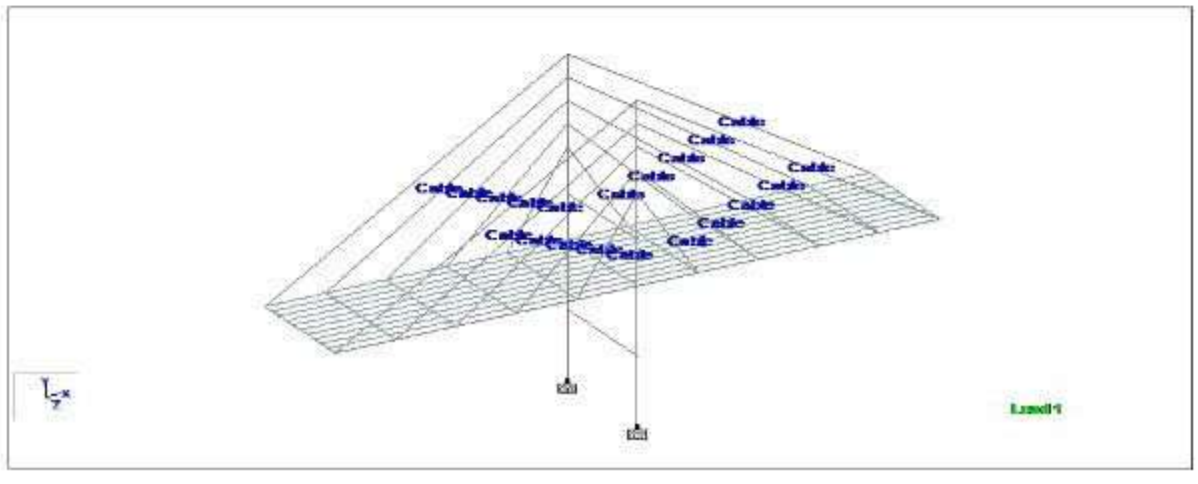
Where do you want to go?

Add beam → **Finish**

Step 3

Click → geometry → **SNOP NODE BEAM**

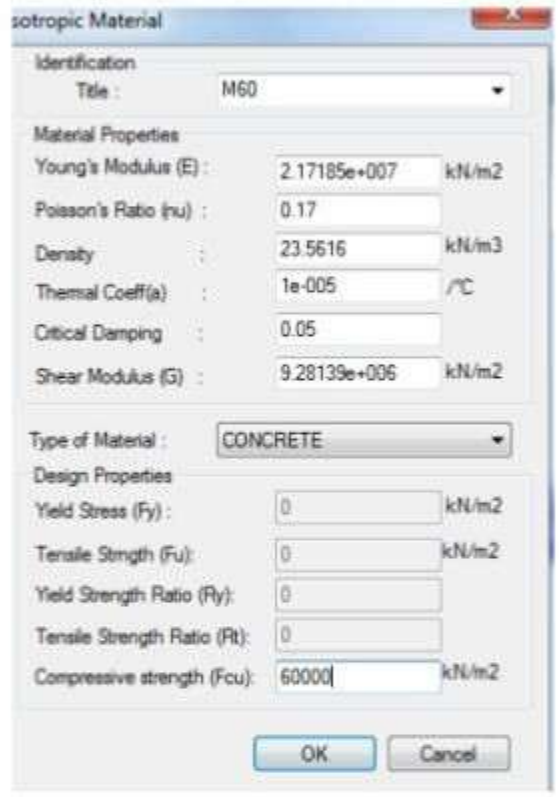


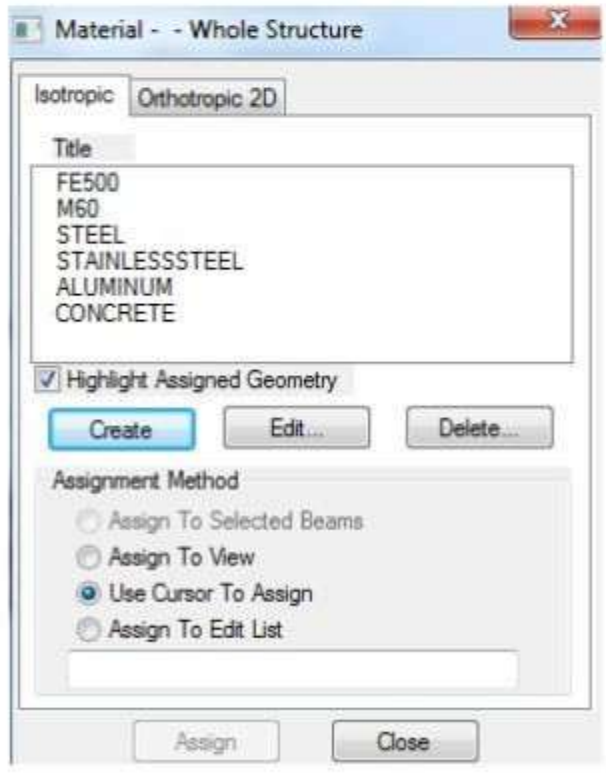


MATERIAL

Click→ Modeling → General→ Material → Material →Whole Structure Create →Select→ Concrete →M60 TITLE→ M60 →Compressive Strength→ **60000**→ Ok →Add

Create→ Select →Steel Fe500→ TITLE →Fe 500 →Yied Strength →**500000** → Ok →Add

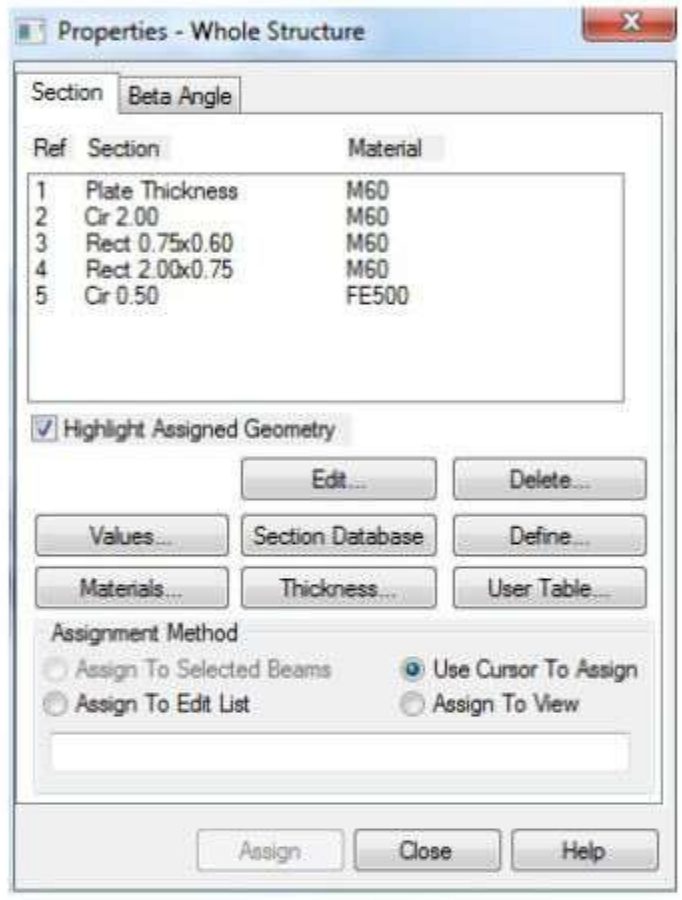


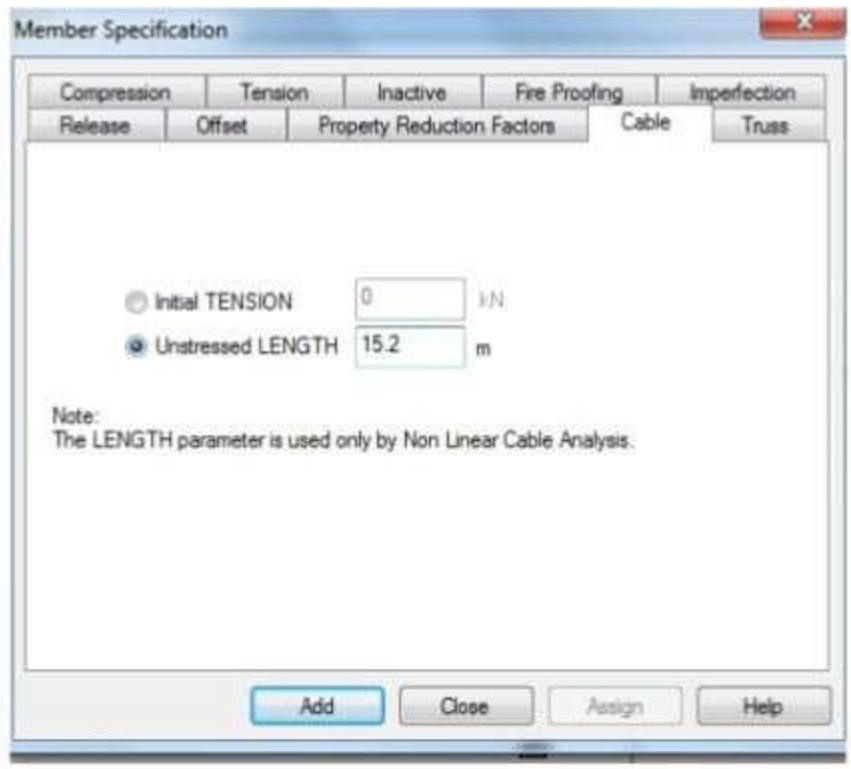


PROPERTY

Click → Modeling → General → property → Property Whole structure → Define → Property → Add → Close → Assign to view → **Assign** → **Yes**

Click → Modeling → General → property → **Spec** → **Beam** → Cable → **unstressed length**→ **Add** → Close → Assign to view → **Assign** → **Yes**





SUPPORT

Click → Modeling → General → Support → Support Whole structure → Create → **fixed** → Add → **Select the support 2** → Select the node point from framed structure → Assign to selected nodes → **Assign** → **Yes**

LOADS

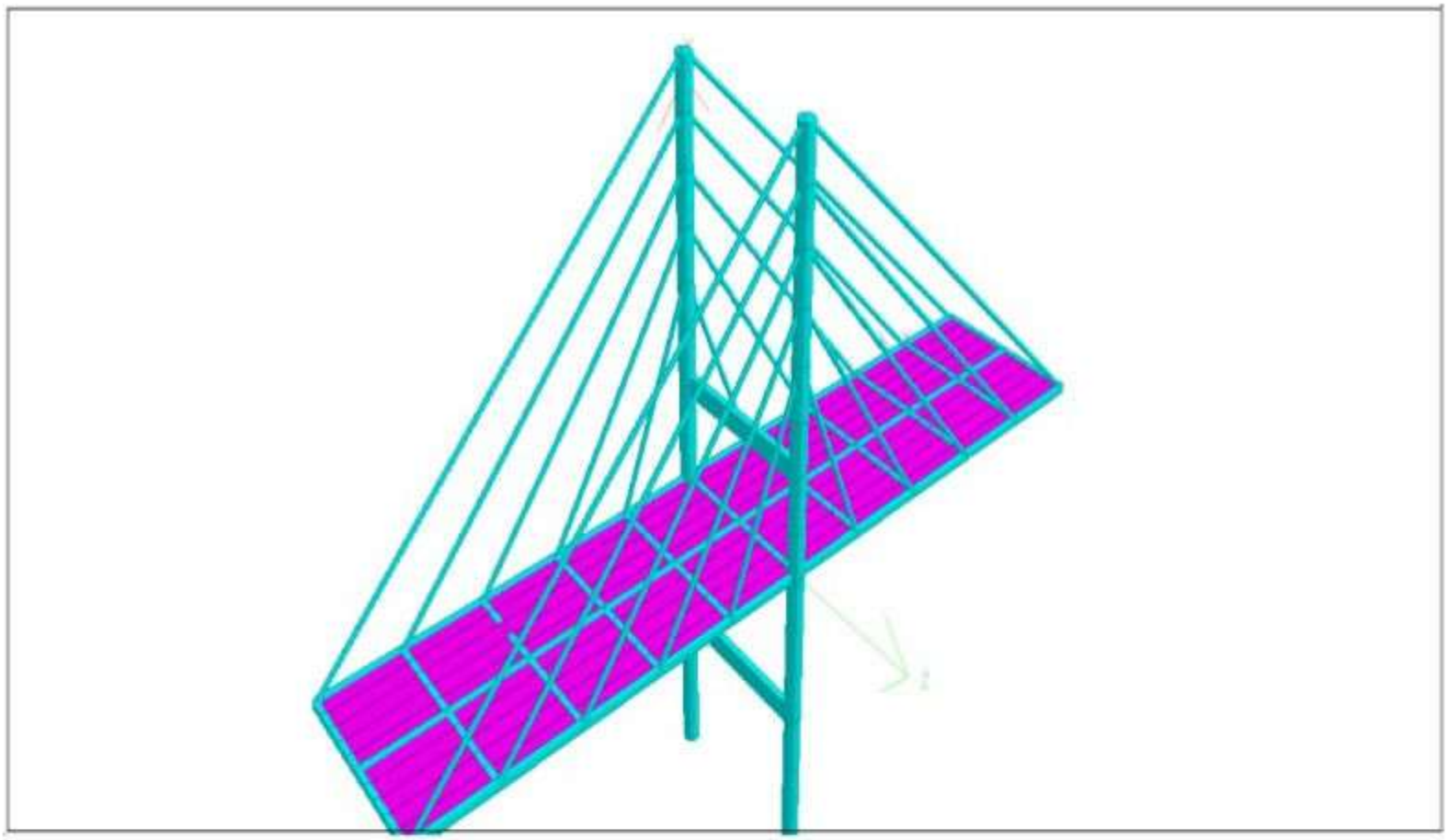
Click → Modeling → General → Load → Click Load Case Details → Add → Add New Load Cases → **Add – Load Case 1** → Click Load Case 1 Add **Self-Weight** → Add → Close

ANALYSIS

Click → Modeling → Analysis→ Print all → **Add** → **close**

Analysis → Run analysis→ Save → **Output Result**

Click the beam see the result on beam



AUTODESK REVIT

C.V. Raman Polytechnic, Bhubaneswar

Chapter-1

WhatisRevitArchitecture?

1. Autodesk Revit software is a BIM (**Building Information Modelling**) application thatutilizes a parametric 3D Model to generate Plan, Section, Elevation, Perspectiveview details and schedules. That all of the necessary instruments to document thedesignofabuilding.
2. Revit gives an opportunity to more easily extract report sand organize our project and data for collaboration with others.

AboutBIM:

- BIM software is **3D design and modeling software that can help optimize theworkofdesigningforarchitecture,construction, plant,civil,andMEP projects**.It does this by: Helping make better design decisions and improve buildingperformance.

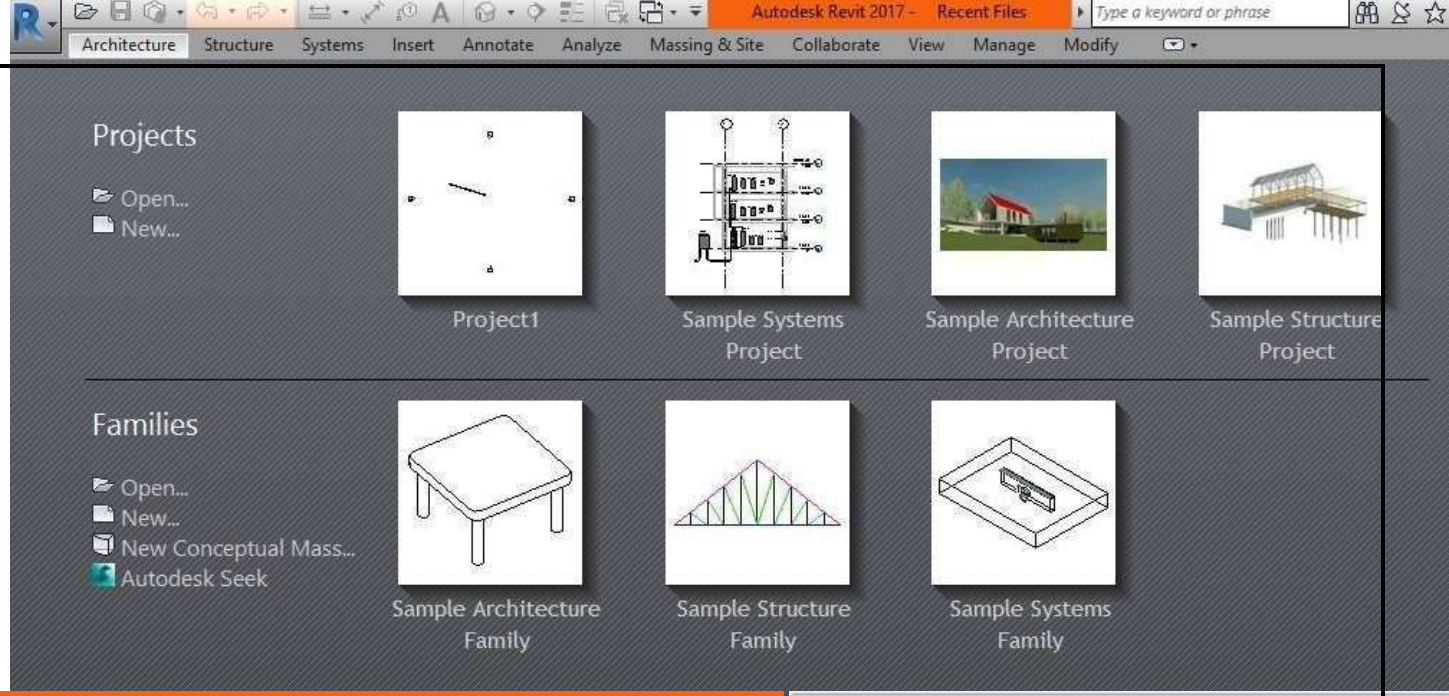
SAMPLE DRAWING:

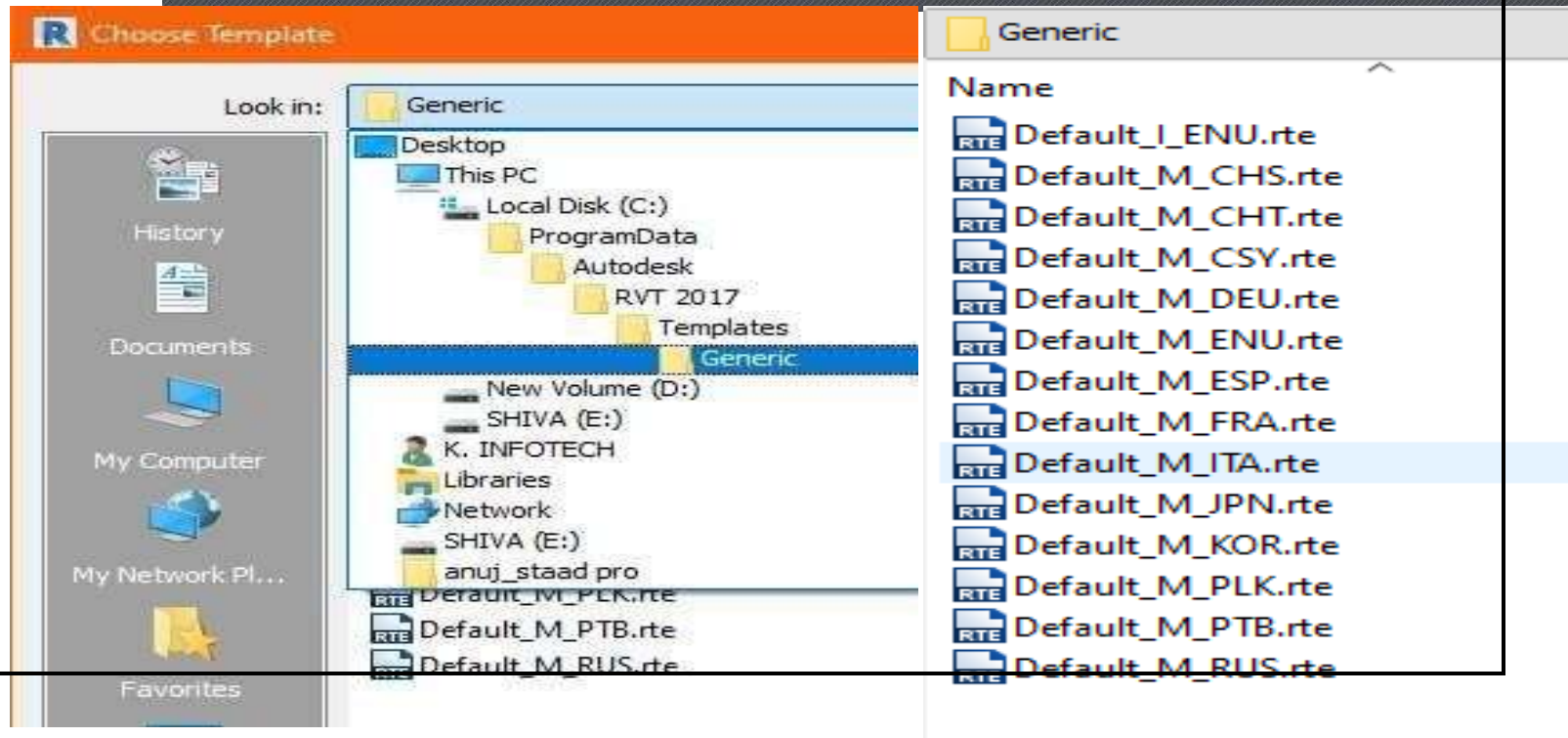
SAMPLEDRAWING:

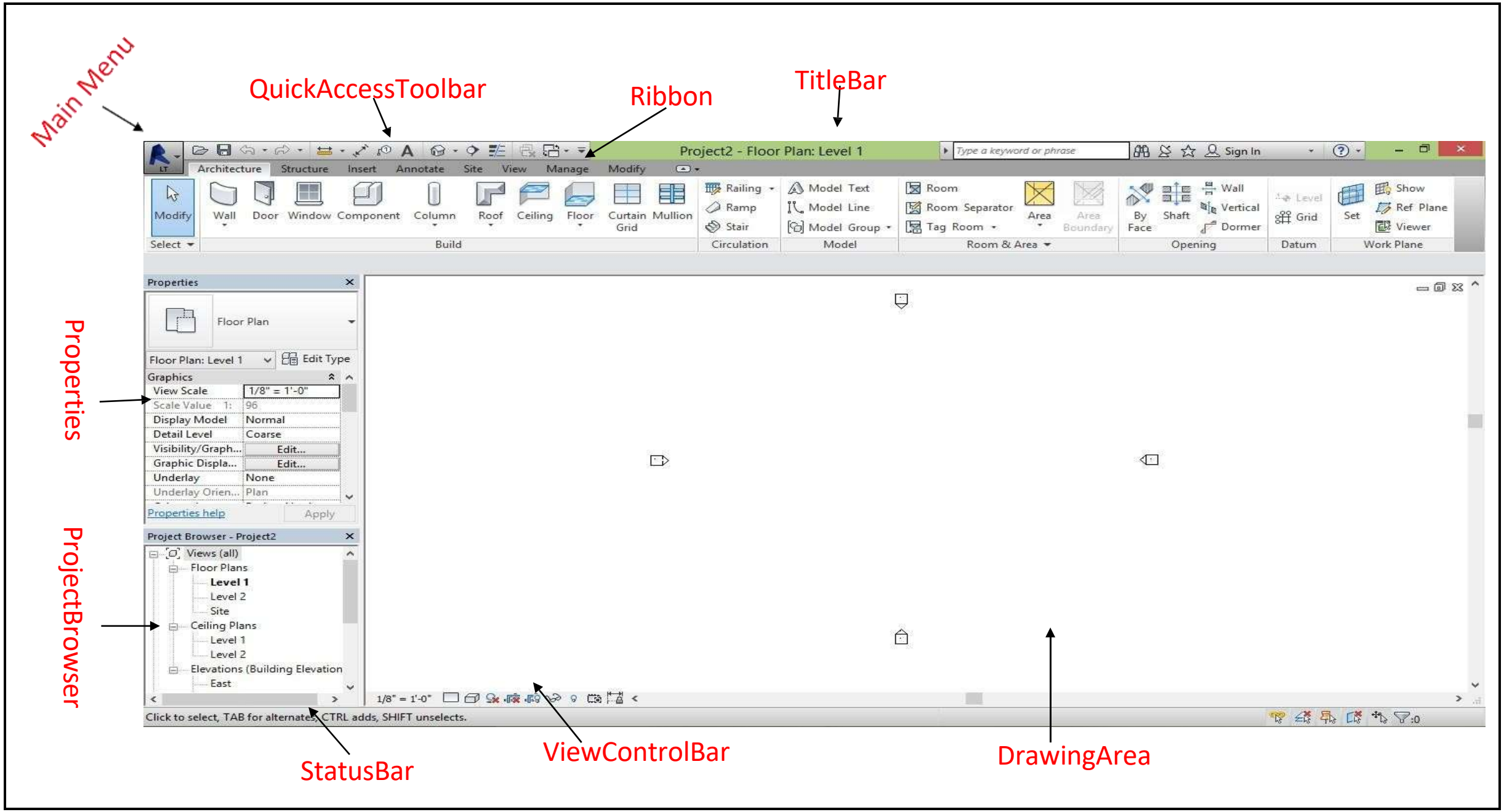
SAMPLE DRAWING:

Class 1 StartingaProject

- Click New underProjects→clickBrowse→Select a templateDefault_I_ENU.rte→Open→Ok





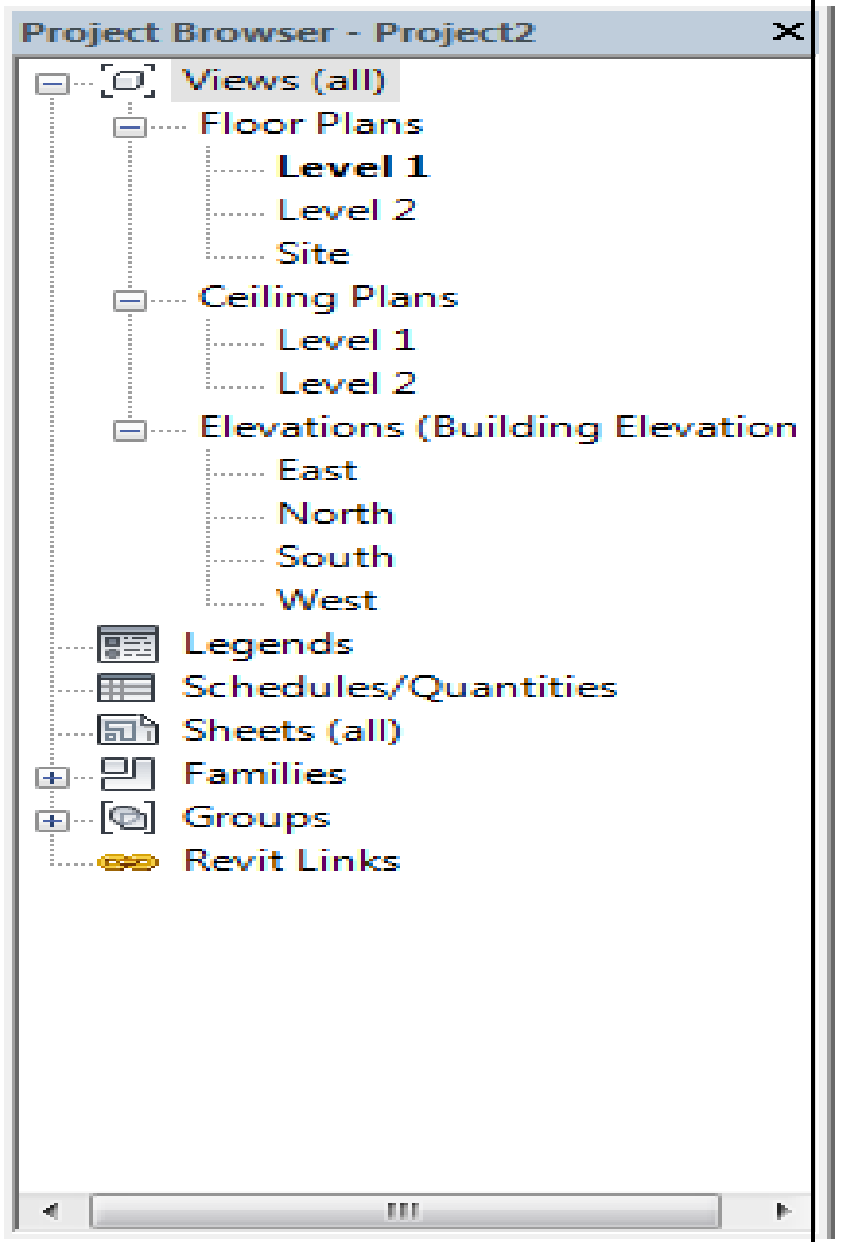

Main Menu
QuickAccessToolbar
Ribbon
TitleBar
Properties
ProjectBrowser
StatusBar
ViewControlBar
DrawingArea
Project2 - Floor Plan: Level 1
Architecture
Structure
Insert
Annotate
Site
View
Manage
Modify
Modify
Wall
Door
Window
Component
Column
Roof
Ceiling
Floor
Curtain Grid
Mullion
Railing
Ramp
Stair
Model Text
Model Line
Model Group
Room
Room Separator
Tag Room
Area
Area Boundary
By Face
Shaft
Wall
Vertical
Dormer
Level
Grid
Set
Show
Ref Plane
Viewer
Select
Build
Circulation
Model
Room & Area
Opening
Datum
Work Plane
Properties
Floor Plan
Floor Plan: Level 1
Edit Type
Graphics
View Scale
1/8" = 1'-0"
Scale Value 1:
96
Display Model
Normal
Detail Level
Coarse
Visibility/Graph...
Edit...
Graphic Displa...
Edit...
Underlay
None
Underlay Orien...
Plan
Properties help
Apply
Project Browser - Project2
Views (all)
Floor Plans
Level 1
Level 2
Site
Ceiling Plans
Level 1
Level 2
Elevations (Building Elevation
East
1/8" = 1'-0"
Click to select, TAB for alternates, CTRL adds, SHIFT unselects.

TheProjectBrowser

One can use the Project Browser to quickly managetheviews, schedules,sheets,reports,families, andgroupsofyourcurrentproject:

- Toopenaview,double-clickitsname
- Expand or collapse the browserlistbyclickingthe“+”or“–”nexttothename

***Note:ToOpenprojectbrowsergo to;**
ViewRibbon→WindowsTab→UserInterface→clickonProjectbrowser



Properties

Shortcutkey:PP

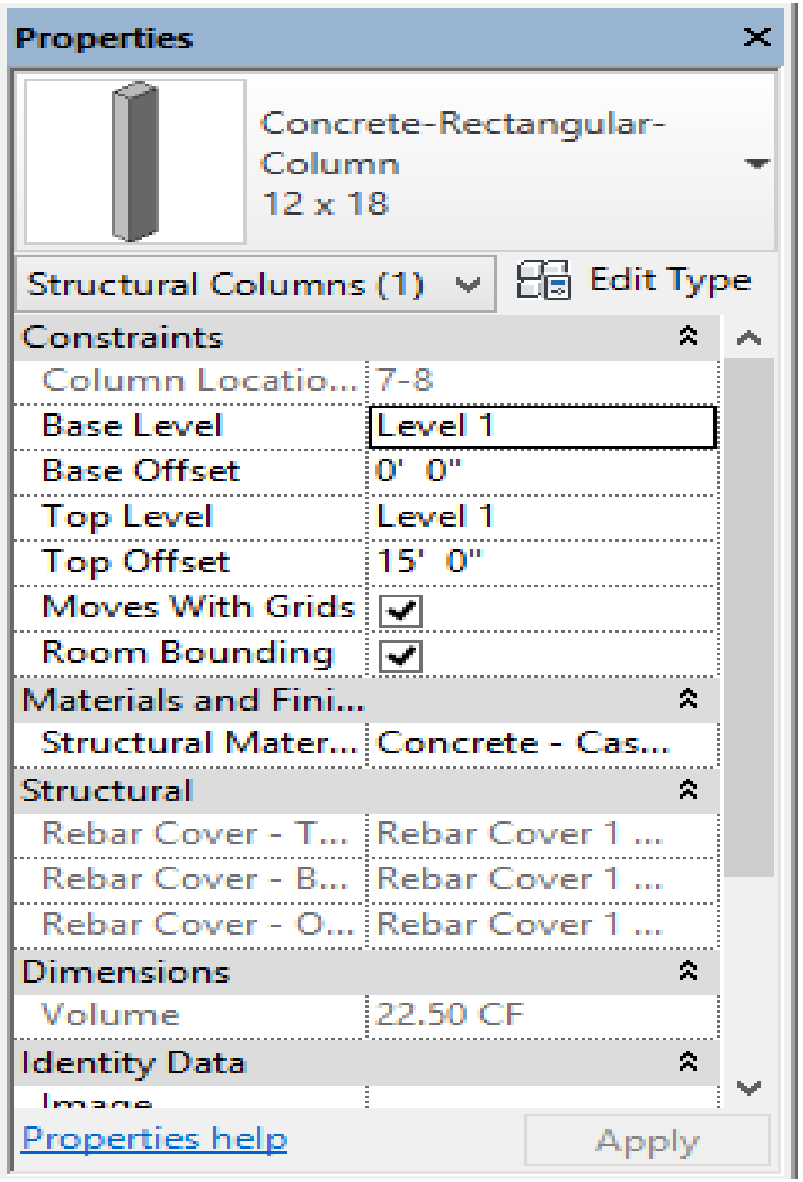
- ❑ Itshowsthepropertiesofcurrentviews
- ❑ Wecanviewandeditthepropertiesofcurrentviews

***Note:ToOpenPropertiesgoto;**

ViewRibbon→WindowsTab→UserInterface→clickonpro perties

OR

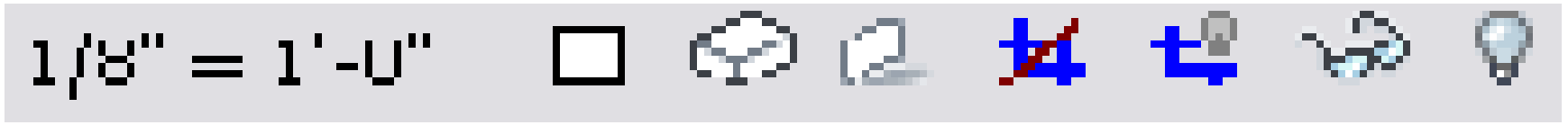
ModifyRibbon→PropertiesTab →selectProperties



ViewControlBar

Itprovidesquickaccesstofunctionsthataffectthecurrentview.

- Viewcommands,namelyscale,detaillevel,graphicsstyle,shadows,cropview,crop region,temporaryhide/isolate,revealhiddenelements



DrawingArea

- ThedrawingareaoftheRevitArchitecturewindowdisplays views(andsheetsandschedules)ofthecurrentproject
- Bydefault,eachtimeyouopenaviewinaproject,theviewdisplaysinthedrawingareaontopofotheropenviews
- YoucanusecommandsontheWindowmenutoarrangethe projectviews

ProjectUnit

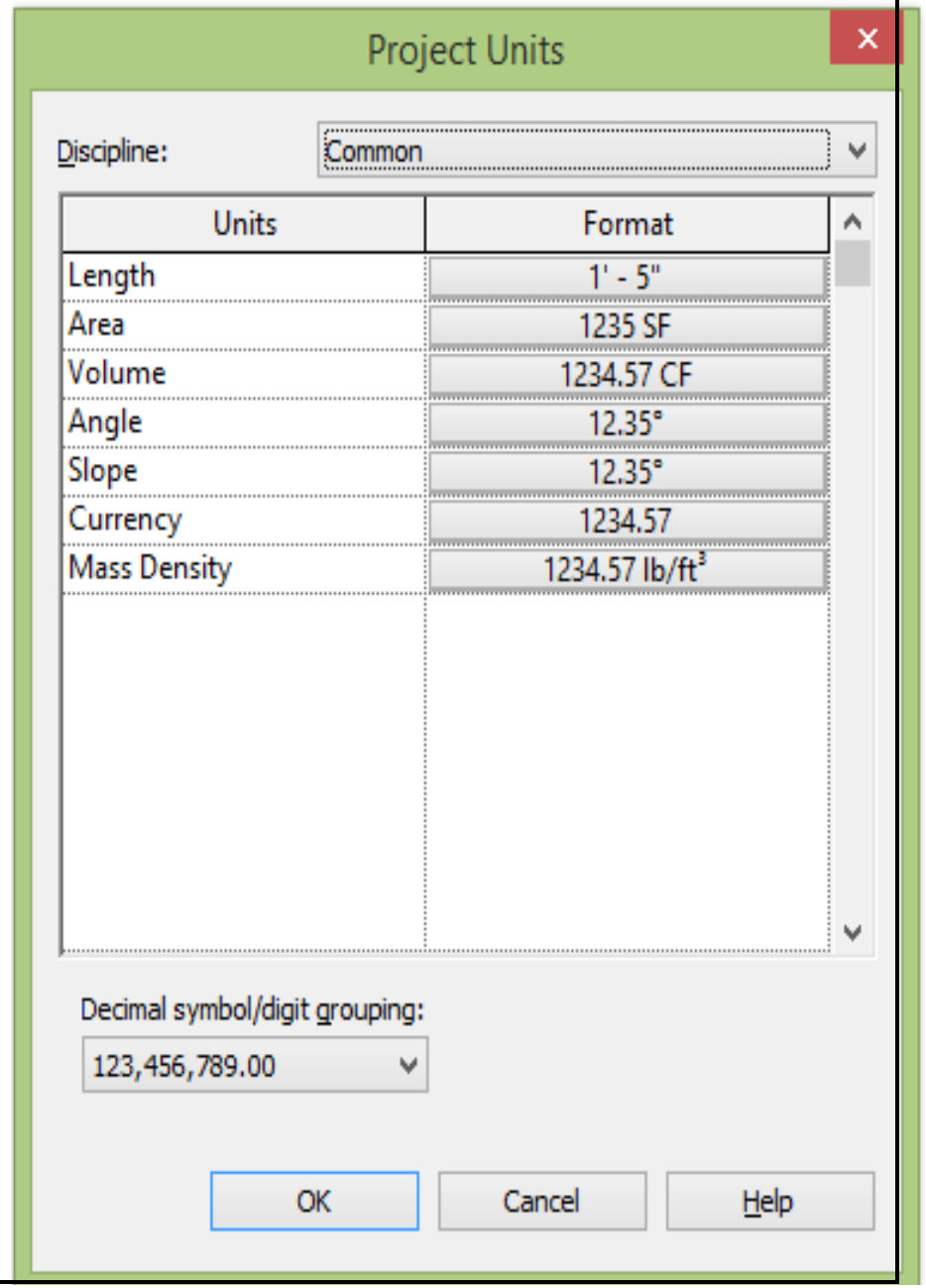
ShortcutKey:UN

ManageRibbon→SettingsTab→projectunits→choose

- ❑ lengthunitas =
  - a) unit–feetandfractionalInches
  - b) rounding–tothenearest1”
  - c) ok
- ❑ Areaunitas=
  - a) unit–Squarefeet
  - b) rounding–2 decimalplaces
  - c) ok
- ❑ Volumeunitas=
  - a) units–Cubicfeet
  - b) rounding–2decimalplaces
  - c) ok

ThenOK

***Note:-TochangeL/A/Vunitclickon(unitspreview),availablein formatpanelof“Projectsunits” windowasshown**

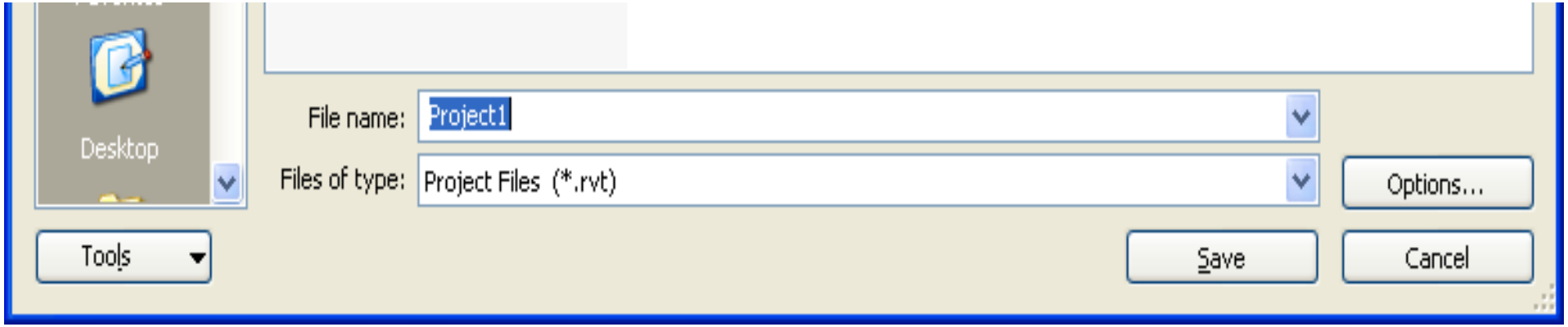


ShortcutKey:CTRL+S

Saveaproject

GotoMainmenu →chosesaveoption

- ❑ Opendesiredlocationtosavefile
- ❑ Givefilename–Selectfileformatas(*.rvt)
- ❑ Clickonsave

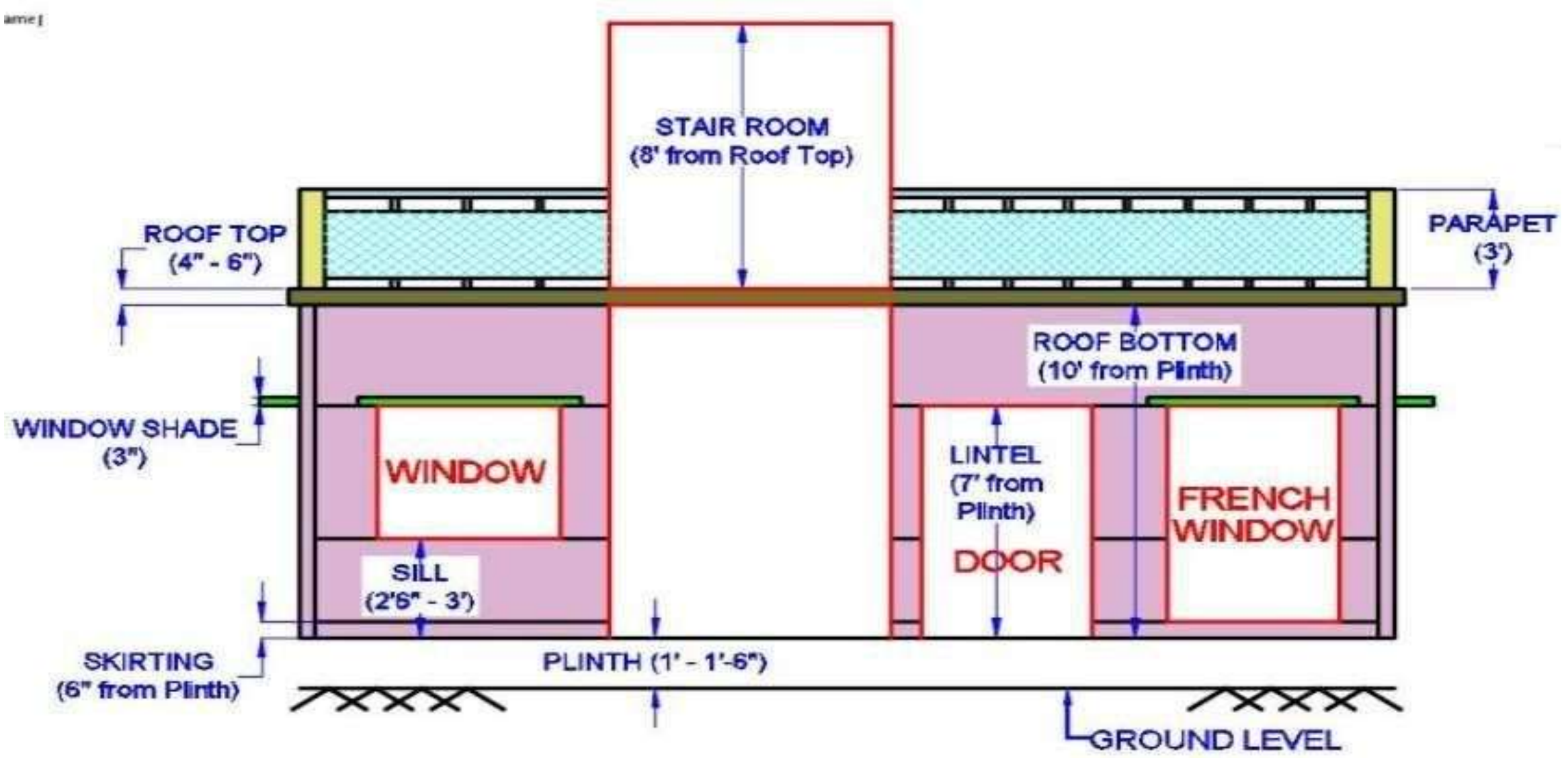


ClosingaProject&ExitingaProject

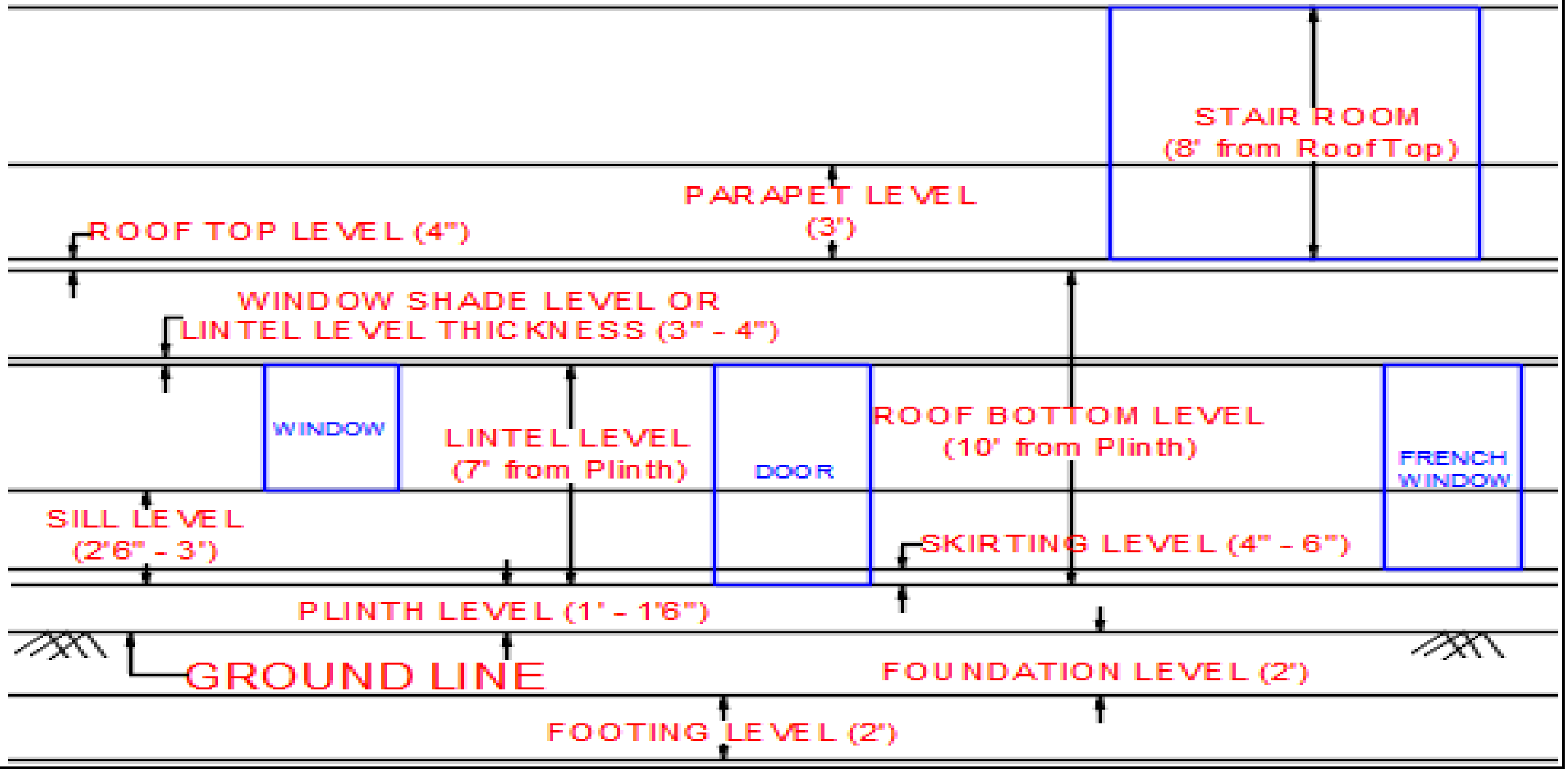
- To Exit from a project, go toMainmenu→clickonClose option

- Click on Close tab available inrightcornerofwindowtoclose anyProject

ELEVATIONOFPLAN



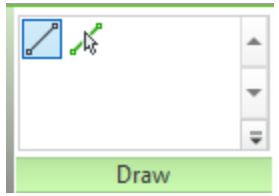
ELEVATIONOFPLAN



Level(LL)

Createlevelsforthebuilding:

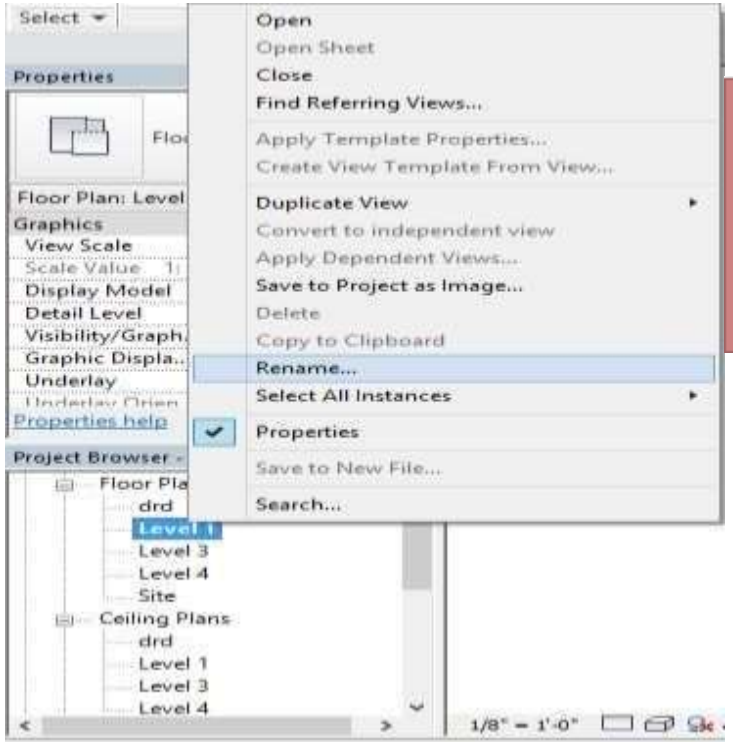
- InthеProjectBrowser→Elevations(BuildingElevation)→double-clickonrequiredElevations(Like:EAST, WEST,NORTH,SOUTH)
- InArchitectureribbon→DatumTab→SelectLevel
- FromDrawTab:
- ClickonDraw→Drawalineforcreatelevel (manually)
- ClickonPickLines→GiveOffsetvalue→Selectreference level tooffset
- Esc→Esc(ToTerminate fromcommand)

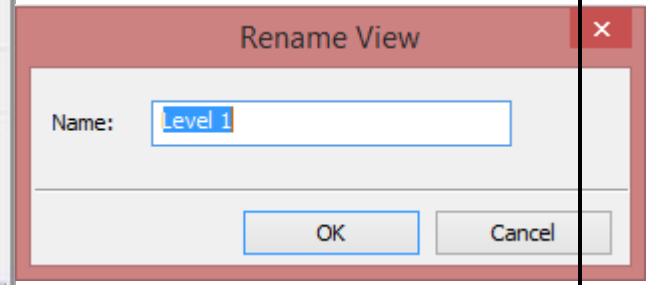


Level(LL)

Rename the new Levels:

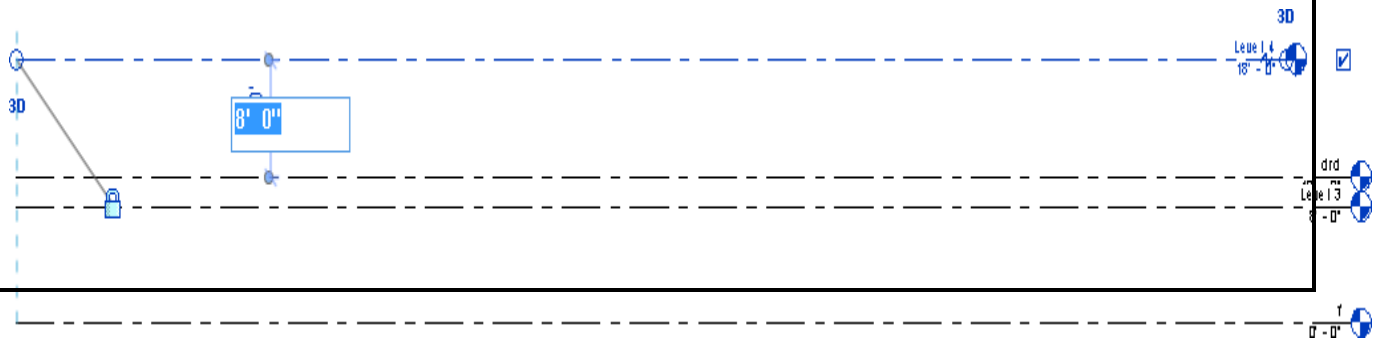
- InthePROJECTBROWSER→FloorPlans→Rightclick on level →Select rename →Give thenameoflevel→Ok →Yes





Changeleveldimension:

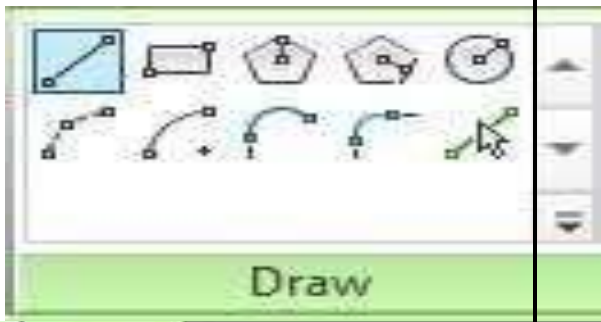
- In the Project Browser →Elevations (BuildingElevation) →double-click on requiredElevations(Like:EAST,WEST,NORTH,SOUTH)
- ClickonRequiredLeveltoChangethe Dimension
- Double-clickonleveldimension→enternew values→enter

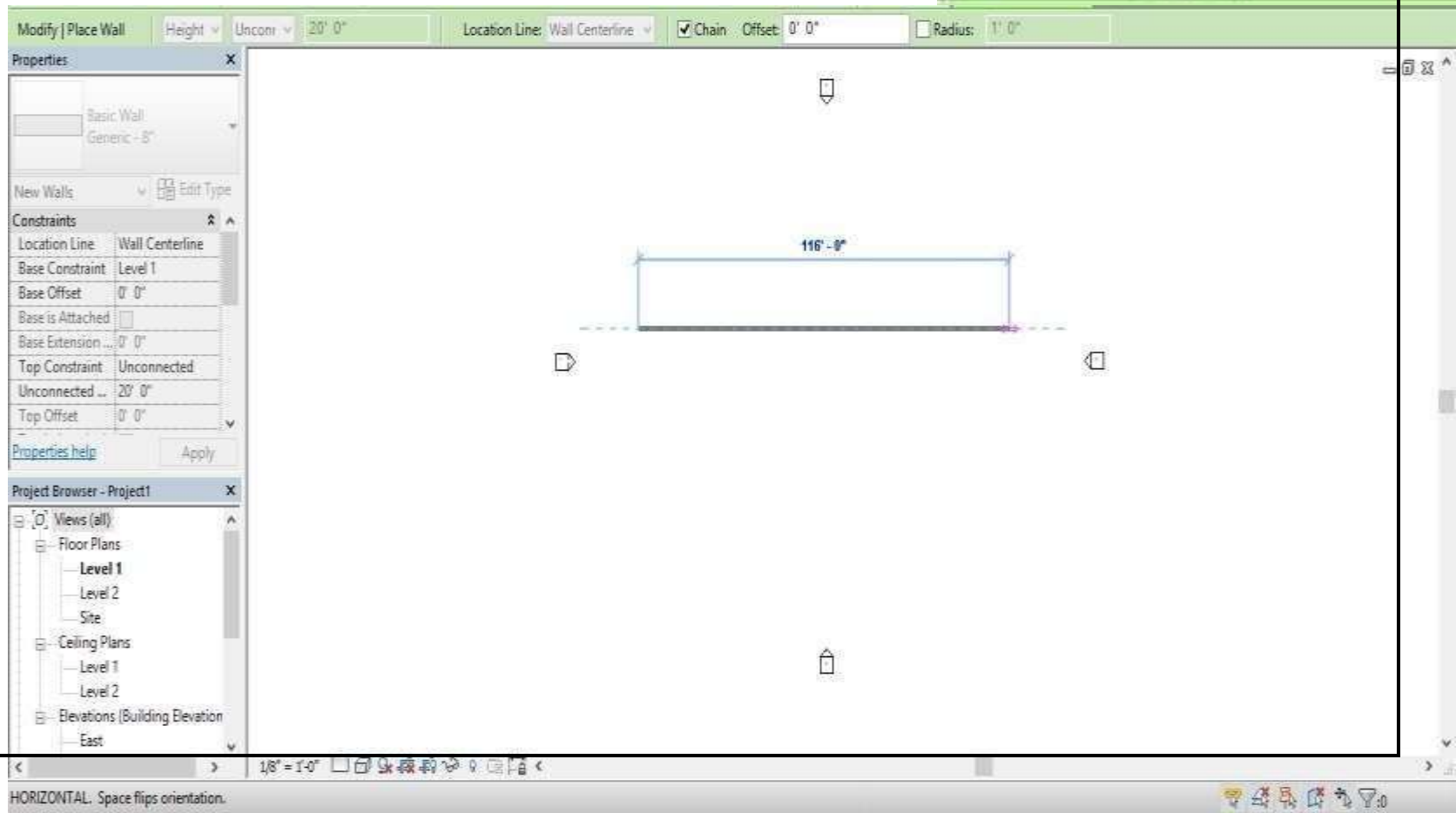


CreatingWalls

ShortcutKey:WA

- ClickonArchitectureRibbon→ BuildTab→SelectWall
- Selecttypeofwall
- Specifyunconnectedheightfor wall
- SelectLocationline
- Select LineOrRectangleoptionto create Horizontal or Verticalwall
- Clickonpointstofixstartanden d position ofwall
- Uncheckchainoptiontodisablec ontinuityofwall.Ifrequired
- Selectarcorcircle optionstocreatecurvewall
- Esc→Esc(ToTerminatefromcommand)





WallProperties

Toopenpropertywindow,

- Select wall→ClickonTypeProperties
- Clickonduplicatetab→giveelemena name→clickonok
- ClickonEditTab

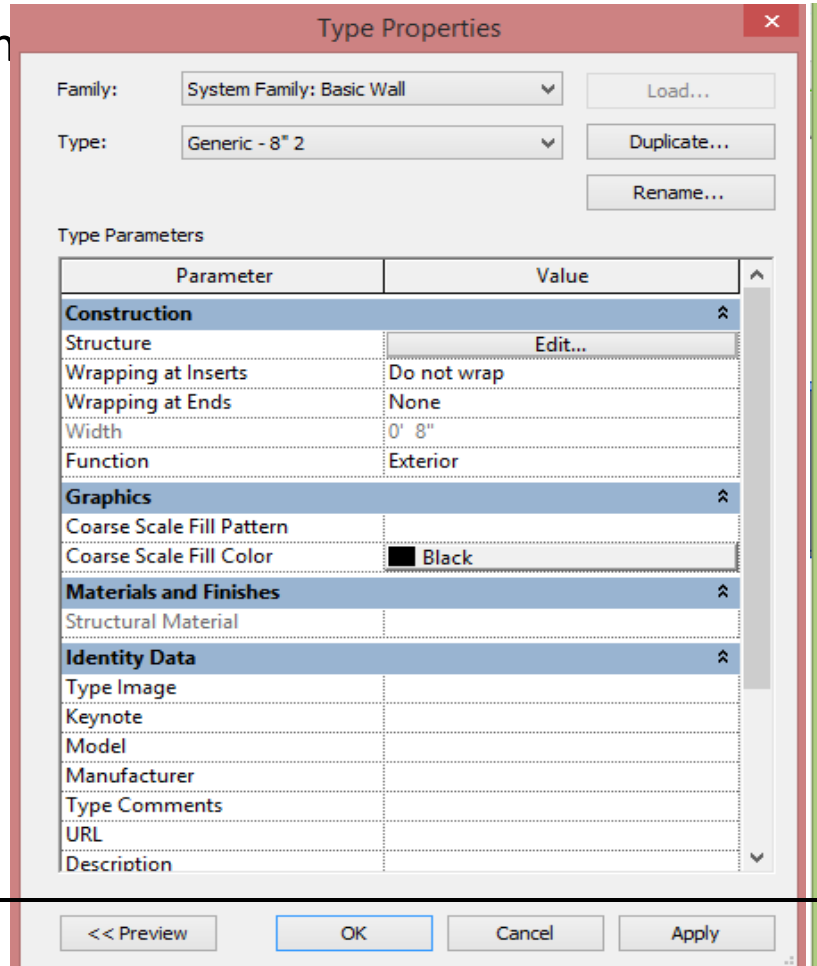
WecanchangeThickness,

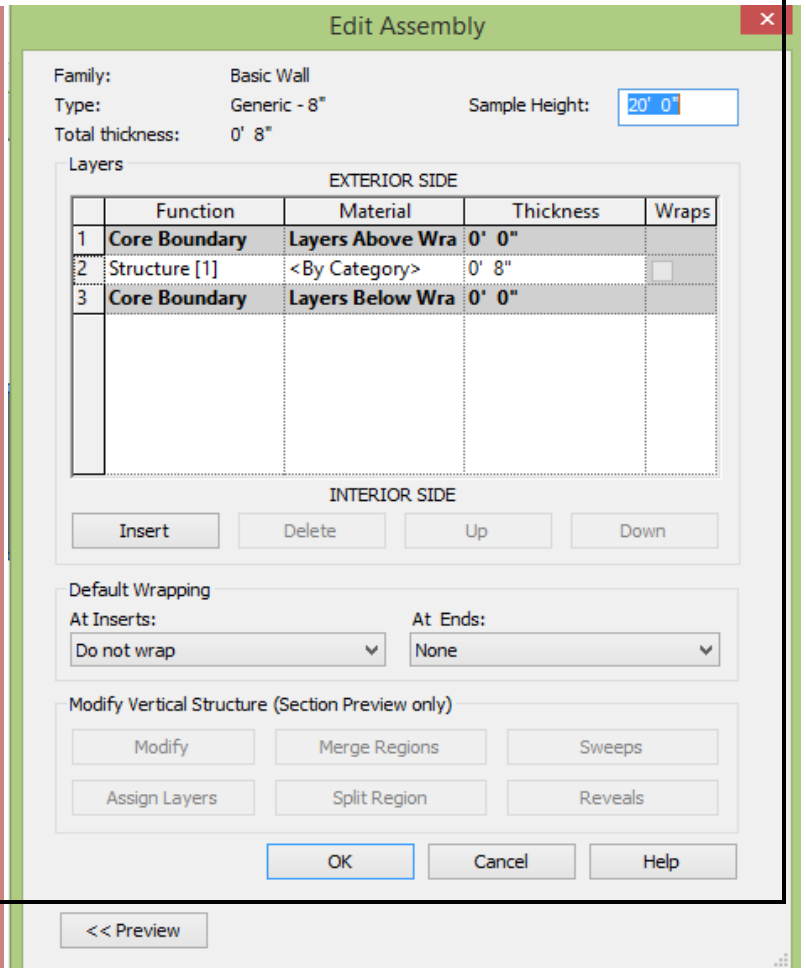
- GiveThicknessvalue
- ClickonOk→Apply→Ok

WecanchangeMaterial,

- Clickon<ByCategory>
- SelectMaterialtype
- ClickonOk →Ok→Apply →Ok

←——TypeProperties

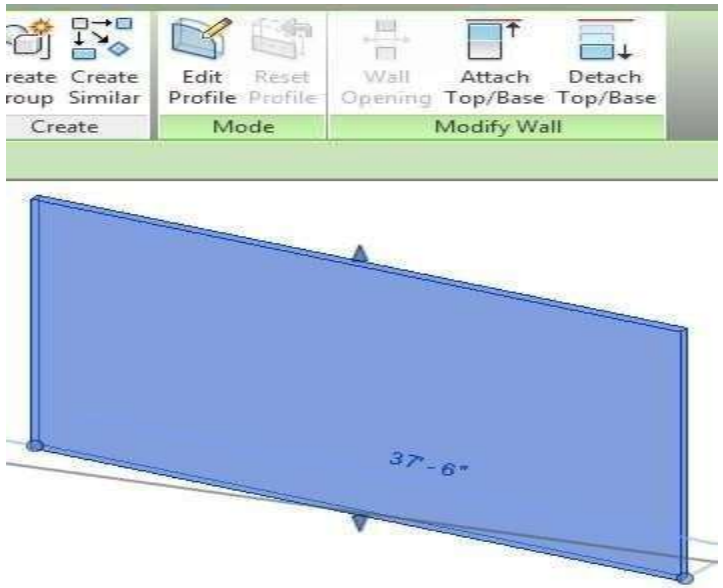


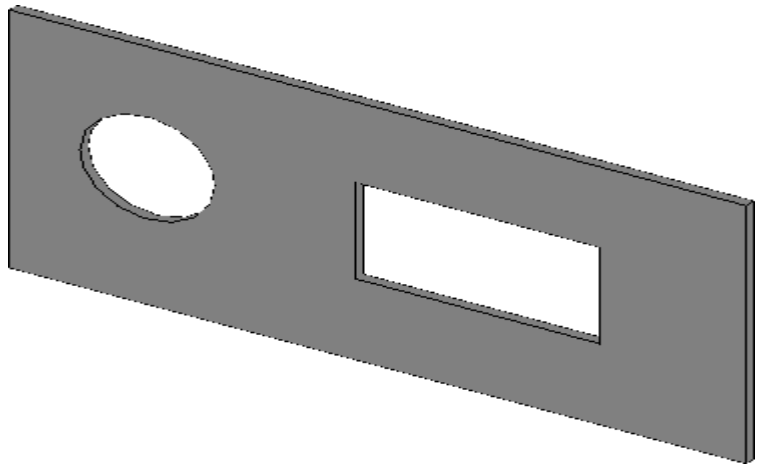


Sketchingwall

Wecanmakeaopeninginwall byusingsuchoption;

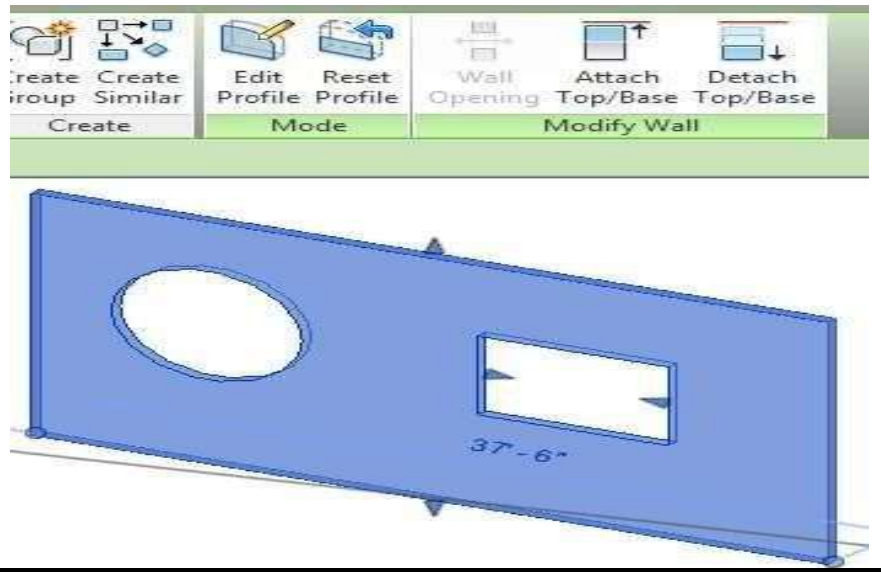
- Selectwall→Clickonedit profile (from Mode Tab)**Note:EditProfileapplieson lyon3D viewandElevations**
  - DrawSketchusingdraw options→Finish Sketch

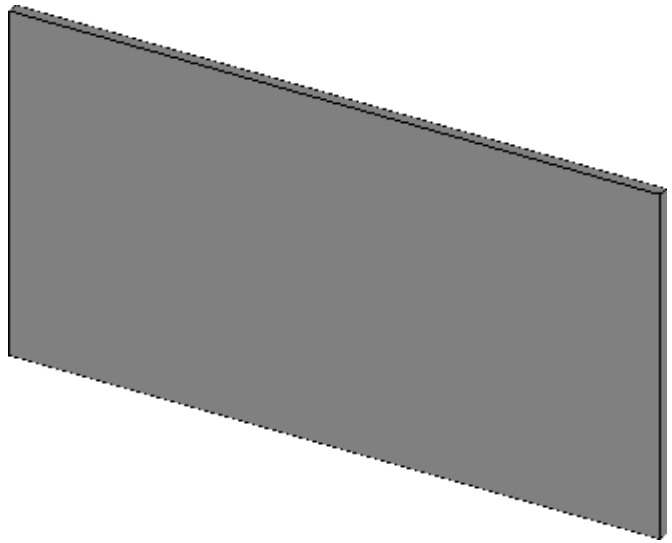




Toremovethesketch;

- Select wall→ResetProfile

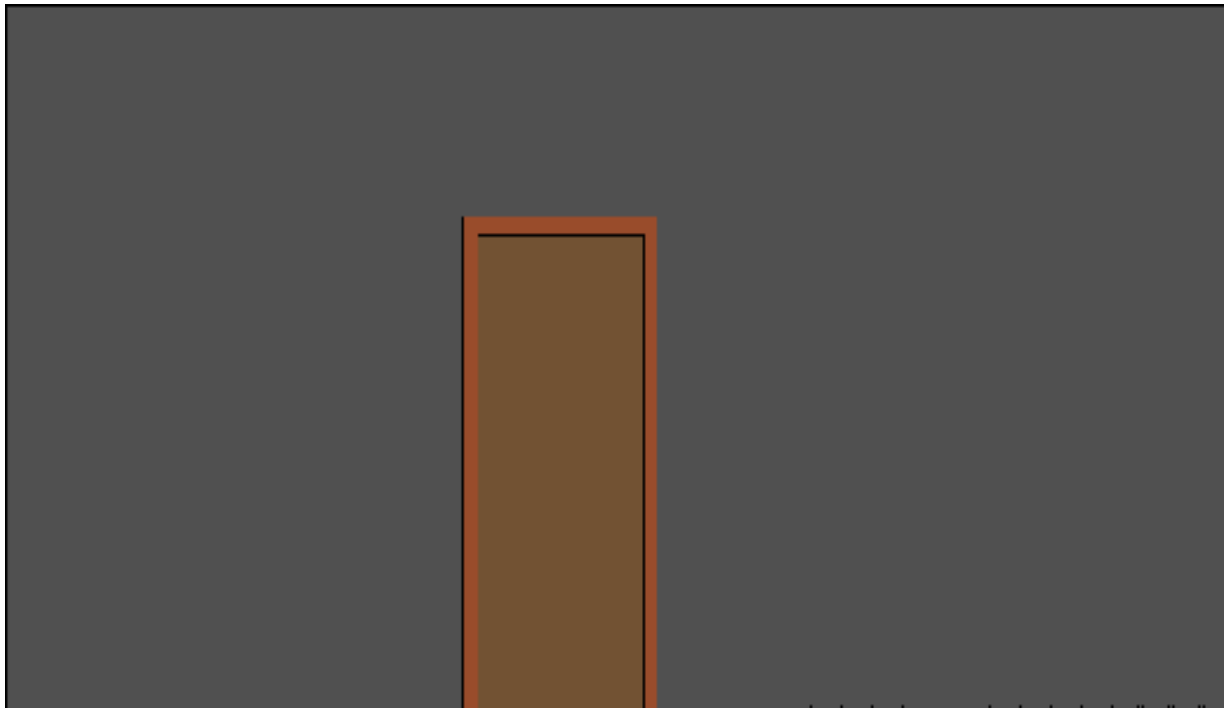




AddingDoors

ShortcutKey:DR

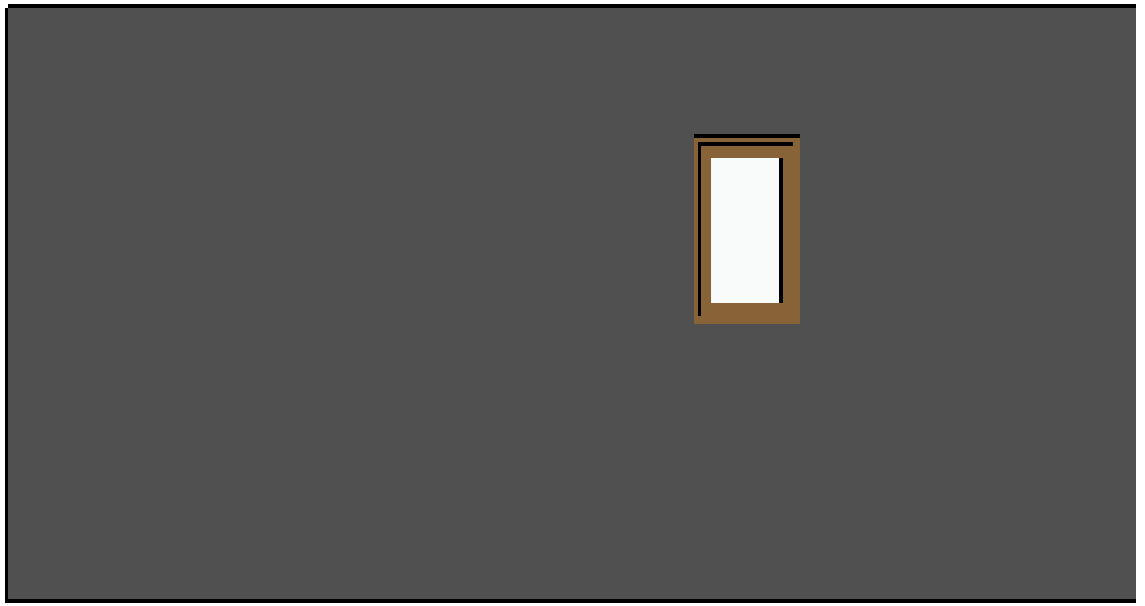
- InthePro jectBrowser,underFloorPlans,double-clickaviewtoopenthatview inthedrawingareawherewewant toplaceDoor
- ClickonArchitectureRibbon→BuildTab→SelectDoor
- ClickonLoadFamilyfrommodetab
- Selectadoor→clickopen
- Placethedoor(itisvalid onlyonwall)
- Esc→Esc(ToTerminatefromcommand)



AddingWindows

ShortcutKey:WN

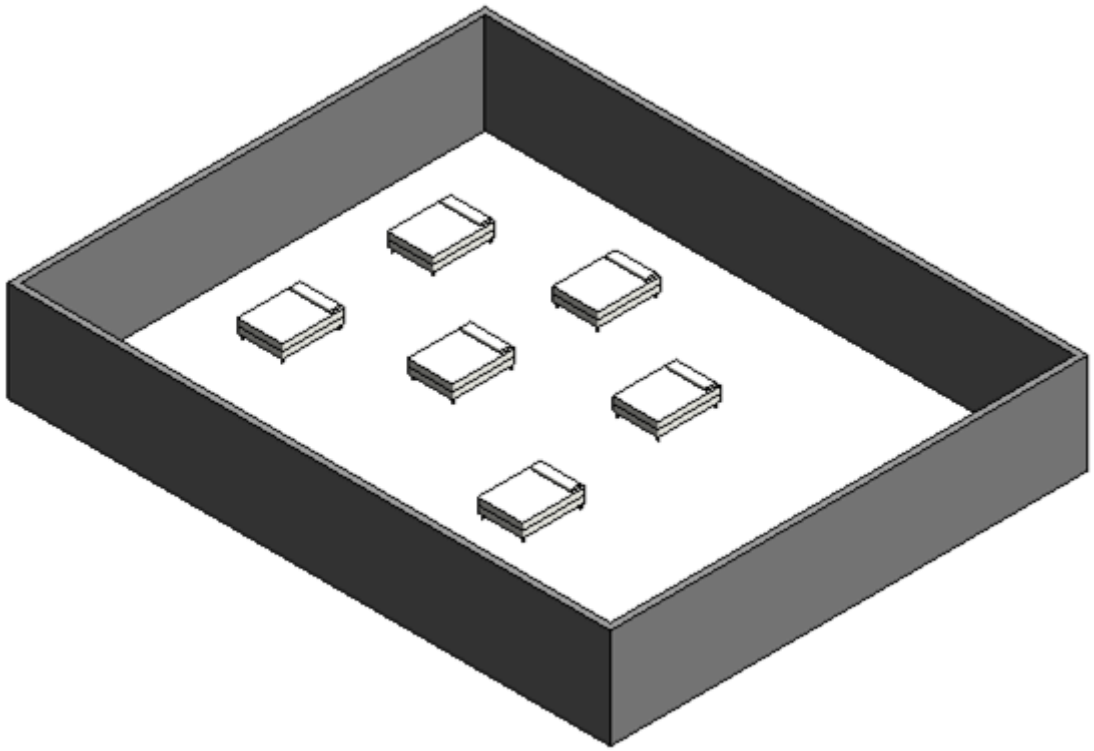
- InthepProjectBrowser,underFloorPlans,double-clickaviewtoopenthatview inthedrawingarea wherewewanttoplaceWindow
- ClickonArchitectureRibbon→BuildTab→SelectWindow
- ClickonLoadFamilyfrommodetab
- SelectaWindow→clickopen
- Placethewindow(it isvalid only onwall)
- Esc→Esc(ToTerminatefromcommand)



AddingComponents

ShortcutKey:CM

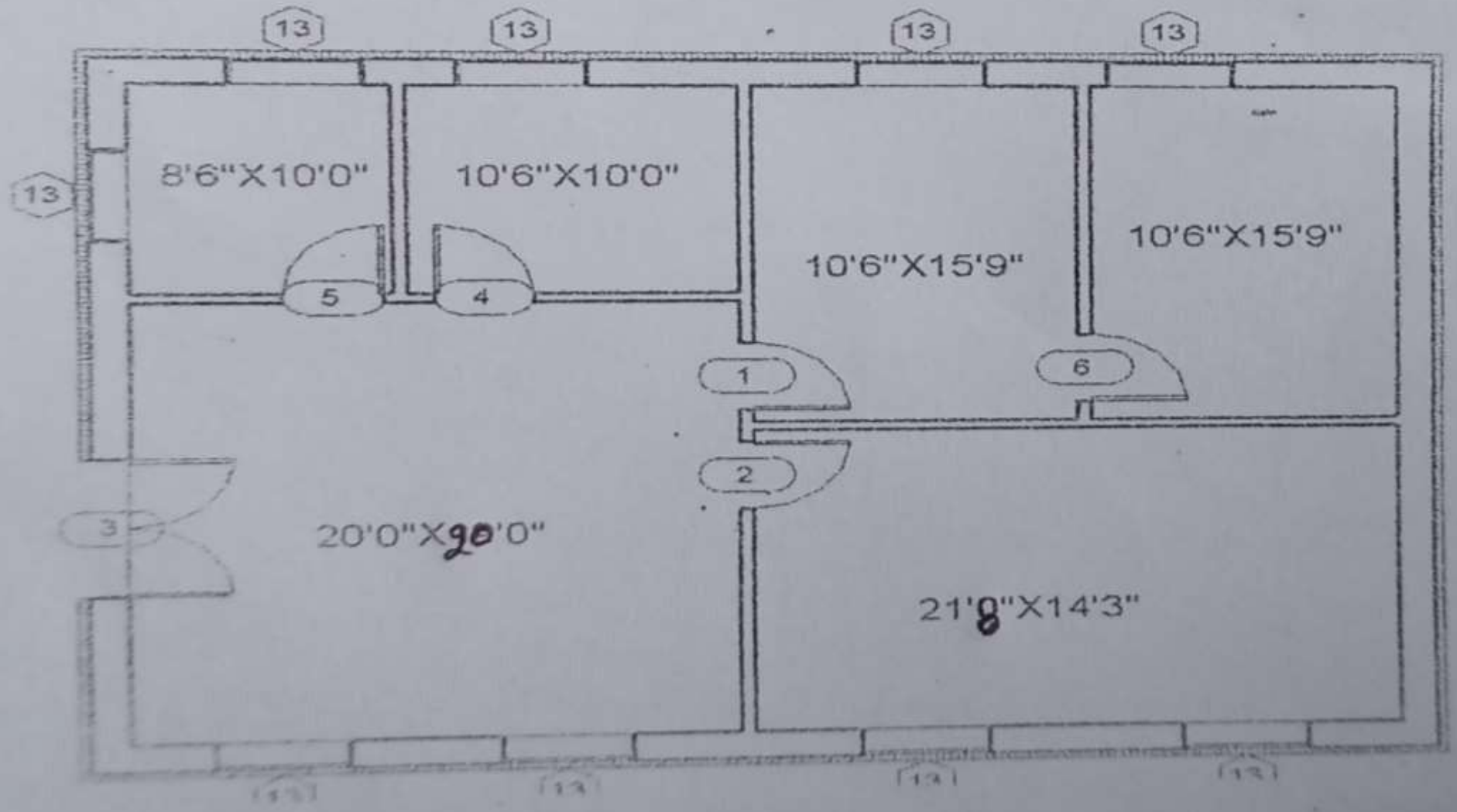
- InthetheProjectBrowser,underFloorPlans,double-click a view to open that view in the drawing areawherewewanttoplaceComponent
- ClickonArchitectureRibbon→BuildTab→ SelectComponent
- ClickonLoadFamilyfrommodetab
- SelectaComponent→clickopen
- PlacetheComponent
- Esc→Esc(ToTerminate fromcommand)



PRACTICAL

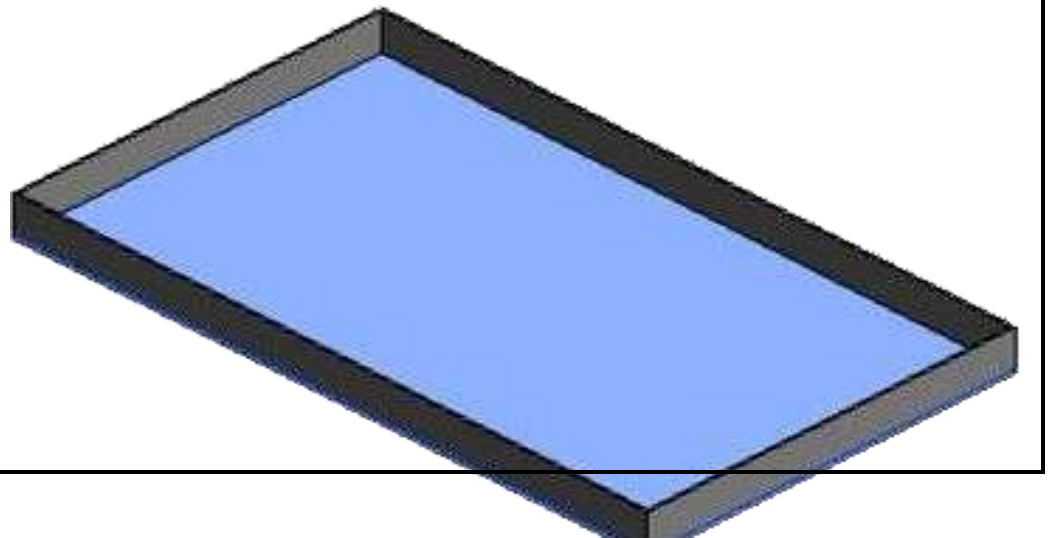
- DOTHEBELOWEXERCISES
  - EX-1

EX-1- Create a sheet and put the floor plan, elevation, sections, 3D view, and sch
to the following sheets.



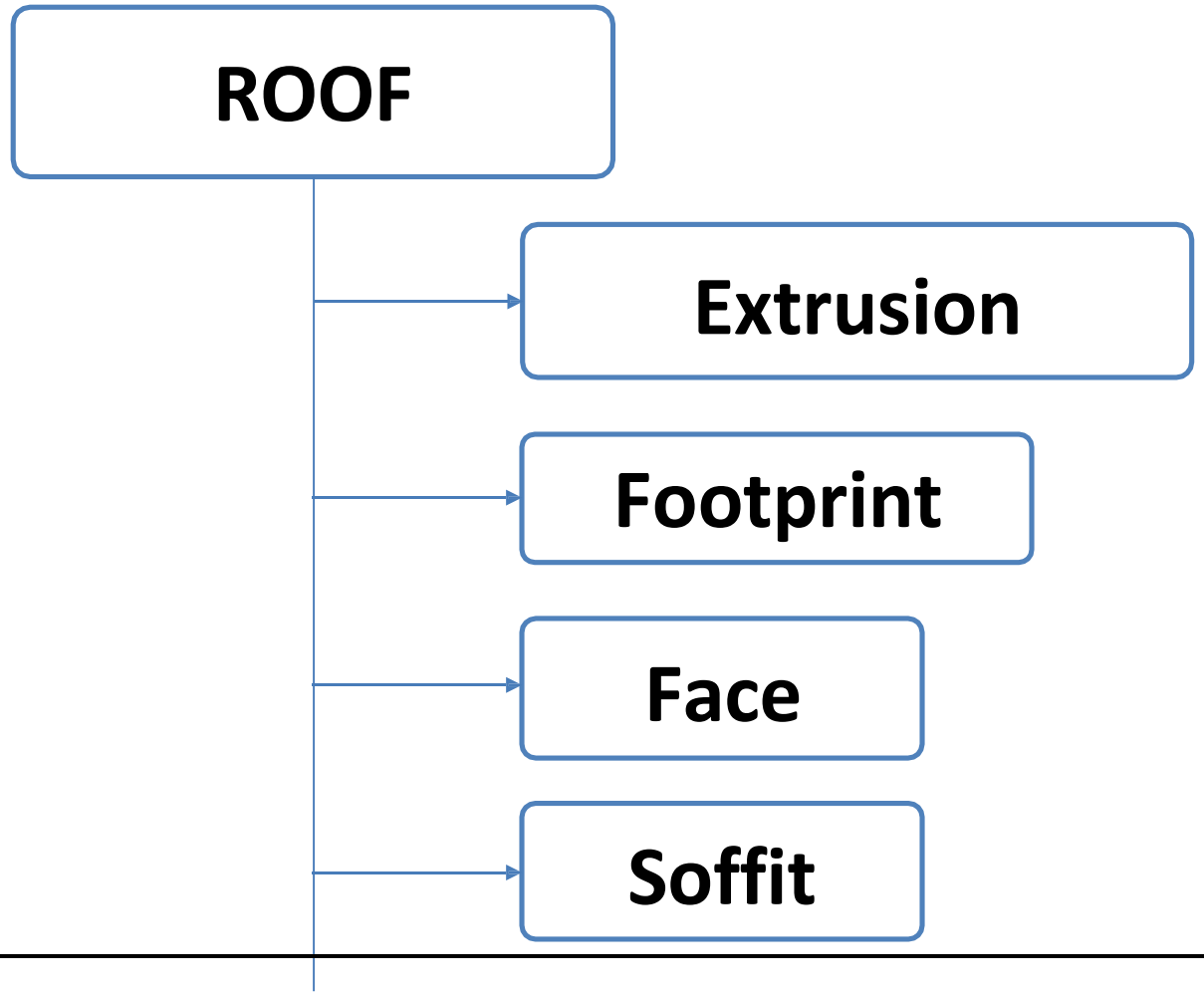
CHAPTER-2
FLOOR

- In the Project Browser →FloorPlansview→double-clickonBaselevel
- ClickonArchitectureRibbon→BuildTab→(fromFloor)SelectFloorArchitectural
- SelectPickWallsoptionfromdrawtab
- Selectwallstocreatefloor
- Finish EditMode



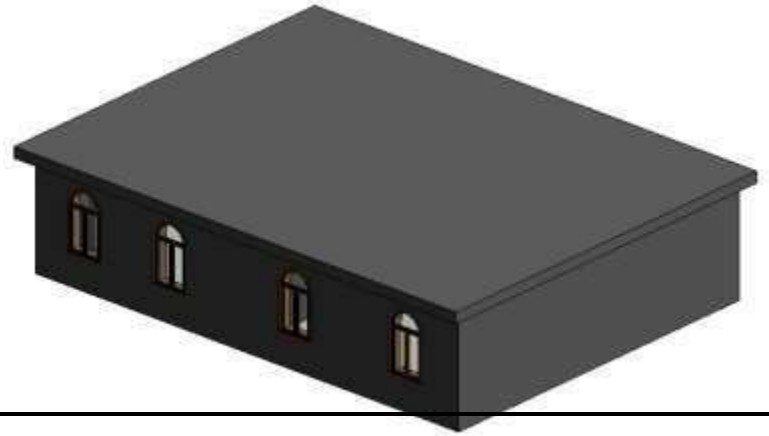
ROOF

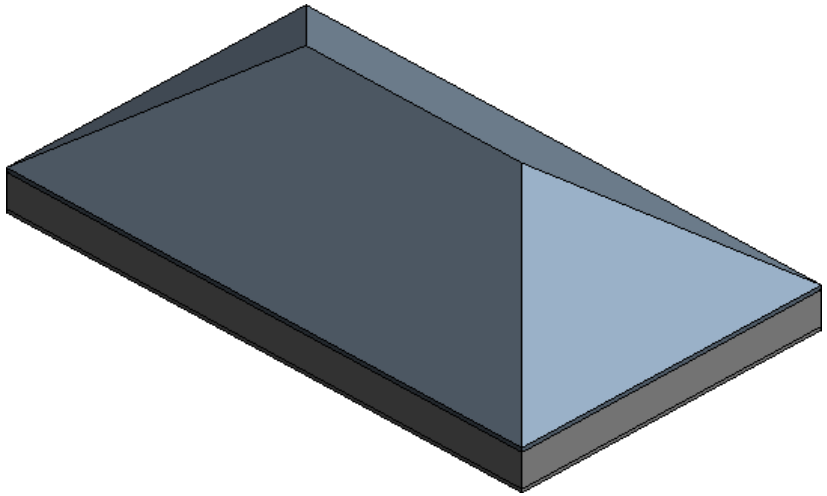
- Select **toplevel** for roof creation from **floorplans viewin projectbrowser**
- Click on Architecture Ribbon → Build Tab → select required Roof



RoofbyFootprint

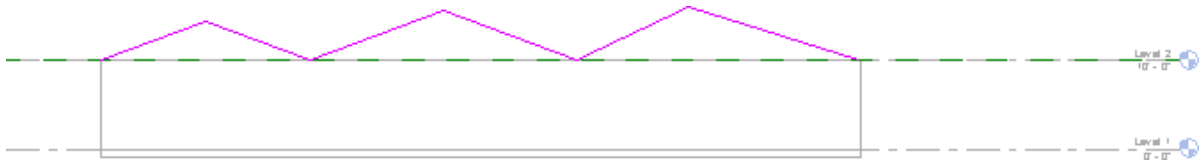
- Choose**RoofByFootprint**option from**Roof**
- SelectPickWallsoptionfromdraw tab
- Selectwallstocreateroof
- FinishEditMode

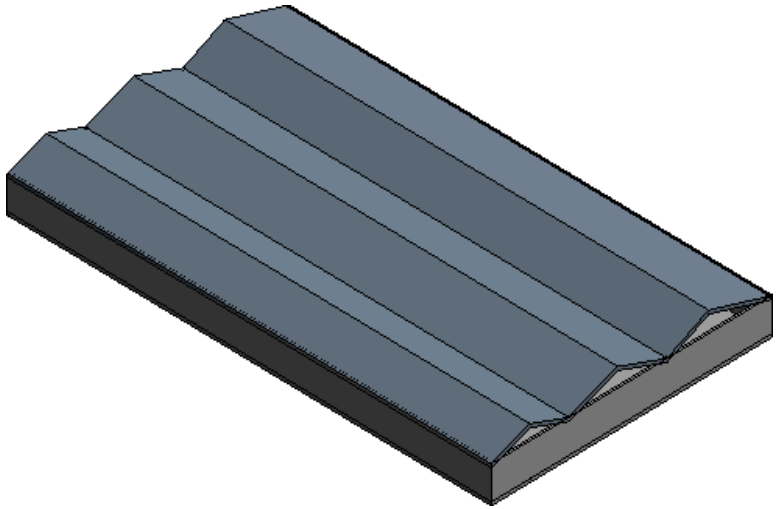




RoofByExtrusion

- ChooseRoofByExtrusionoptionfromRoof
- ClickonPickaplaneoptioninworkplanewindow
- Click onokbutton
- In Roof reference level and offsetdialog box, Select top level →giveoffsetvalueifrequired→Clickonokbutton
- Chooserequireddrawoptionfromdrawtab
- Createroof
- FinishEditMode

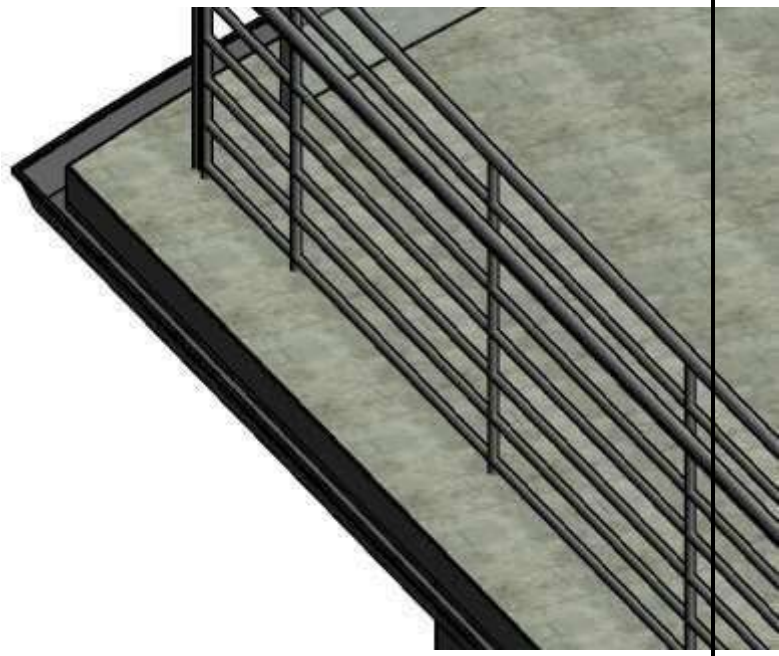




RoofByFace

- Choose **Roof By Face**optionfrom**Roof(**Itus edin **Massing)**
- Selectfacetoaddroof
- Clickon**CreateRoof**
- To terminate fromcommand**“Rightclickthencancel”** again **“Right clickthen cancel”**

RoofBySoffit



- ChooseRoofBySoffitoptionfromRoof
- SelectPickWallsoptionfromdrawtab
- Selectwallstocreateroof
- FinishEditMode

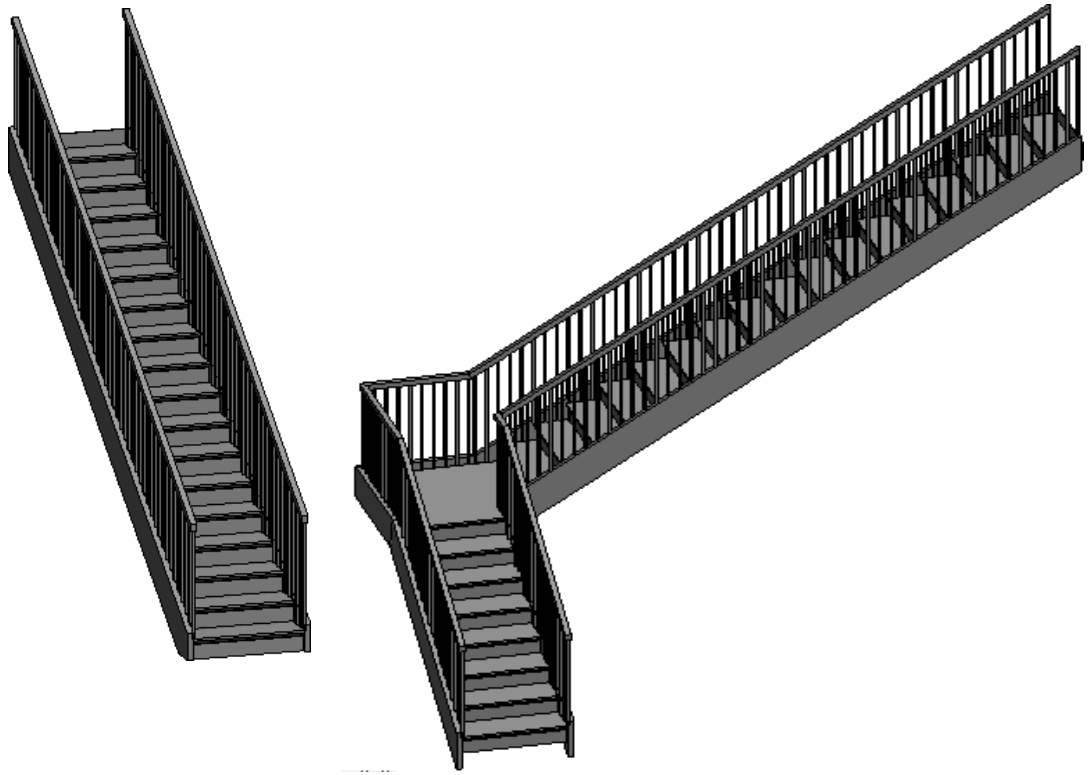
Ceiling

- Select**baselevel**forcellingcreationfrom**cellingplans Viewinprojectbrowser**
- ClickonArchitectureRibbon→BuildTab→select **Celling**
- Select**sketchcelling**
- SelectPickWallsoptionfromdrawtab
- Selectwallstocreatecelling
- FinishEditMode

Note:HeightofCellinglevel=8'fromplinthlevel Thicknessofcellinglevel=6"(itdependsonour requirement)

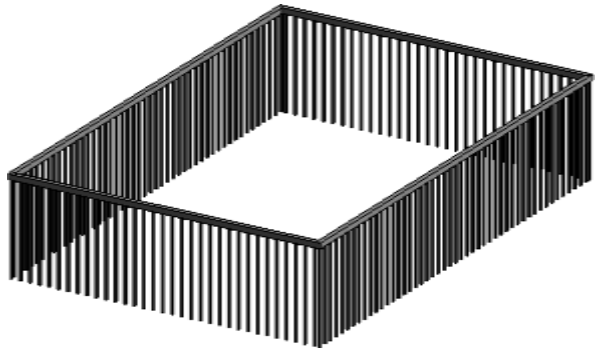
Stairs

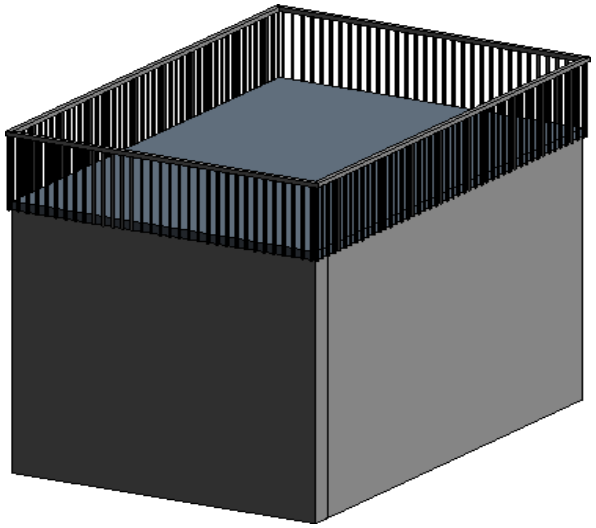
- Select **base level** from **Floor plans viewinprojectbrowserOr3D**view“forStaircreation”
- ClickonArchitectureRibbon→ CirculationTab→select**Stair**
- Selecttypeofstairfromcomponents tab
- Clickthestartpointto startthestair
- Clicktheendpointtofinishthestair(Ifrequiredgivelanding)
- FinishEditMode



Railing

- Select**level**from**Floorplansviewinprojectbrowser**“forRailingcreation”
- ClickonArchitectureRibbon →CirculationTab→select **Railing**→SelectSketchPath
- Selecttypeofrailingfromdrawtab
- FinishEdit Mode





Pin (PN) :

Usethepinpositiontooltolockamodelingcomponent,itcannotbemoved.

Unpin(UP):

✓ Itunpinsalockedelement.

✓ Youcanmoveordeleteitwithoutbeingprompted.

Createsimilar(CS):

It isusedforcreatingsameelementwithsamepropertiesofthe elementthathasbeen selected.

Painttool(PT):

ClickPainttool→IntheTypeSelector,selectthematerialtoapply.Click toapplythe paintintheelement.

PRACTICAL

- DOTHEBELOWEXERCISES
  - Ex

EX-1- Create a sheet and put the floor plan, elevation, sections, 3D view, and sch
to the following sheets.