MANUAL ON CEE330L: STRUCTURAL ANALYSIS & DESIGN LAB STAAD.Pro

Department of Civil and Environmental Engineering

North South University

TABLE OF CONTENTS

CHAPTER-I

INTRODUCTION TO STAAD Pro. ENVIRONMENT

STAAD Pro. Stands for Structural Analysis And Design Program

All structural analysis software generally consists of three parts:

- Pre Processing: Generates the model, assembles and organizes all data needed for the analysis.
- Processing: Calculates displacements, member forces, reactions, stresses, etc.
- Post Processing: Displays the results.

STAAD.Pro Workflow Process:

The process of modeling and designing in STAAD.Pro can be summarized into the following general workflow process, which is suggested inherently by the on-screen organization of the tabs within the program:

1. Basic Geometry: Define the basic geometry of the structure using beams, columns, plates and/or solid elements.

2. Section Properties: Define the sizes of members by width, depth, cross sectional shape, etc.

3. Materials Constants: Specify material such as timber, steel, concrete, or aluminum to define Poisson's Ratio, Coefficient of Thermal Expansion, density, etc.

4. Member Specifications: Define member orientations, member offsets, member releases where moment transfer is to be limited or eliminated, and conditions that only allow a partial transfer of certain types of forces such as tension-only.

5. Supports: Define support locations and boundary conditions including moment fixity, support stiffness, and support angle.

6. Loads: Assign loads such as self-weight, dead, live, wind and seismic, and define load combinations.

7. Analysis Instructions: Indicate the type of analysis to be performed (regular analysis, P-delta, Buckling, Pushover, etc.) and define associated options.

8. Post Processing Commands: Extract analysis results, review deflected shapes, prepare shear and moment diagrams, generate tables to present results, etc.

9. Design Commands: Specify (for steel, concrete, timber, etc.)

The STAAD.*Pro* Start Page is displayed as following

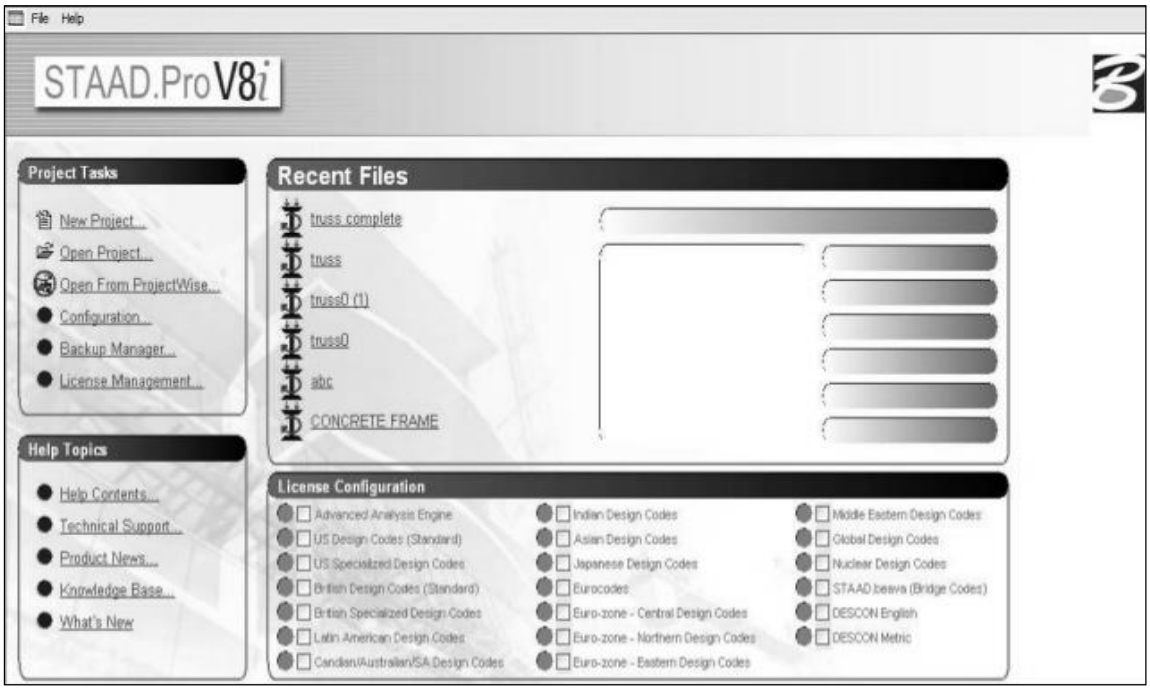


Fig 1.1: Start Page of STAAD Pro.

STAAD Pro. key features:

(A) = Title bar
(B) = Menu bar
(C) = Toolbars
(D) = Mode bar
(E) = Page Control
(F) = View window
(G) = Data area
(H) = Status bar

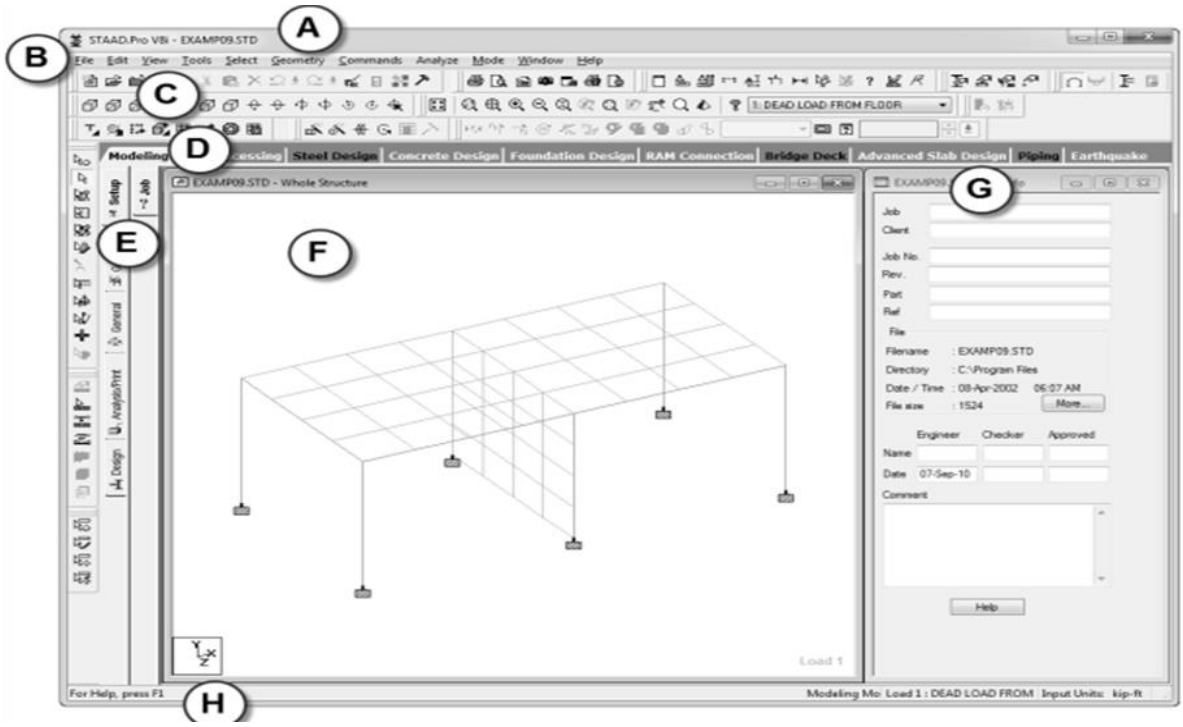


Fig 1.2: For STAAD Pro. Key features

Some most Usable Toolbars and icon views:

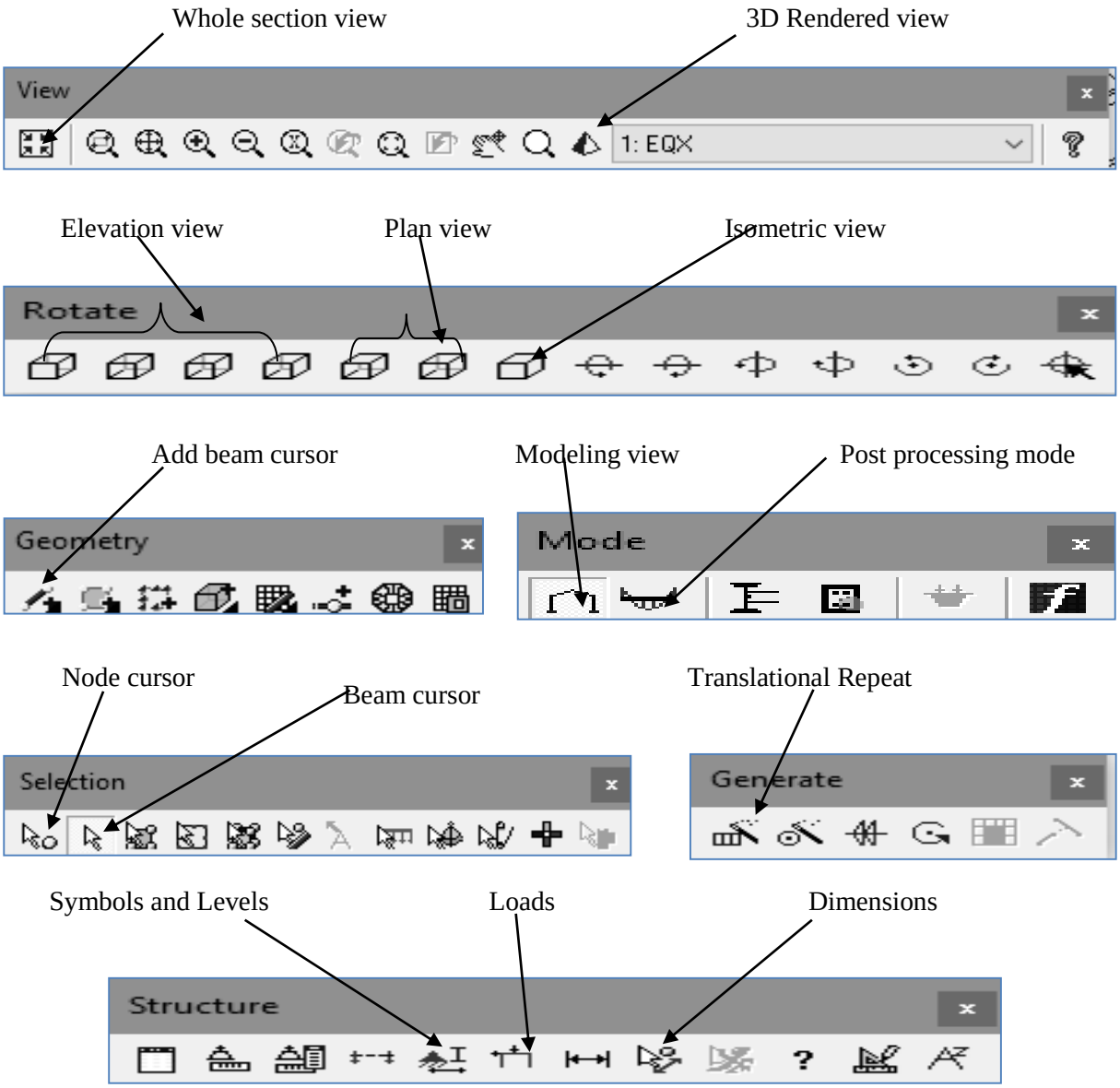


Fig 1.3: STAAD Pro. Toolbars and icon views

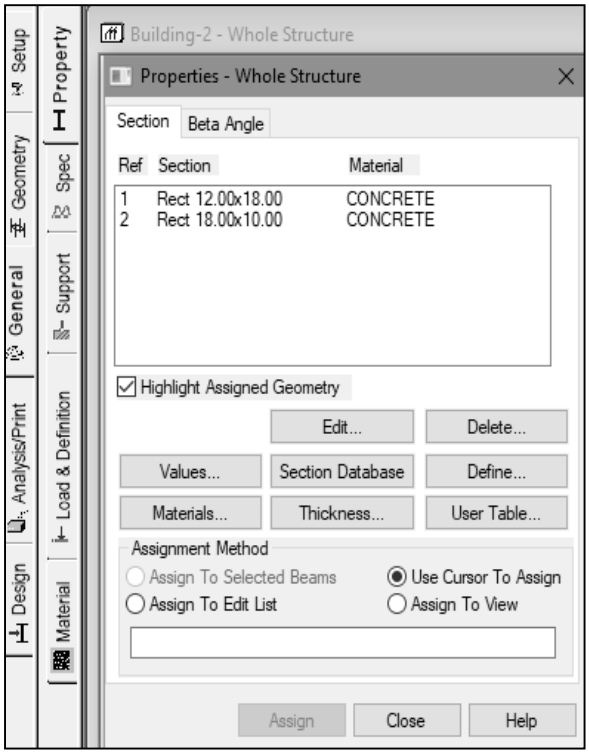


Fig 1.4: Properties Page

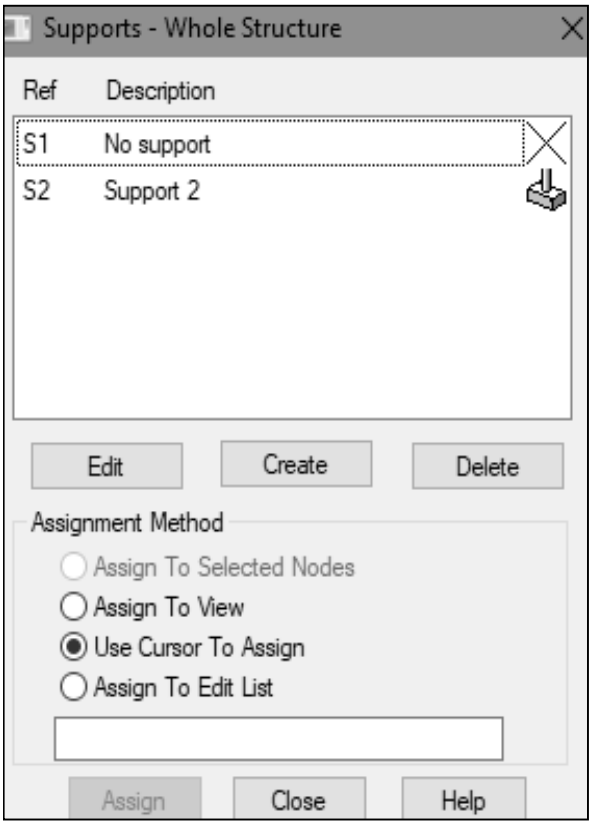


Fig 1.4: Supports Page

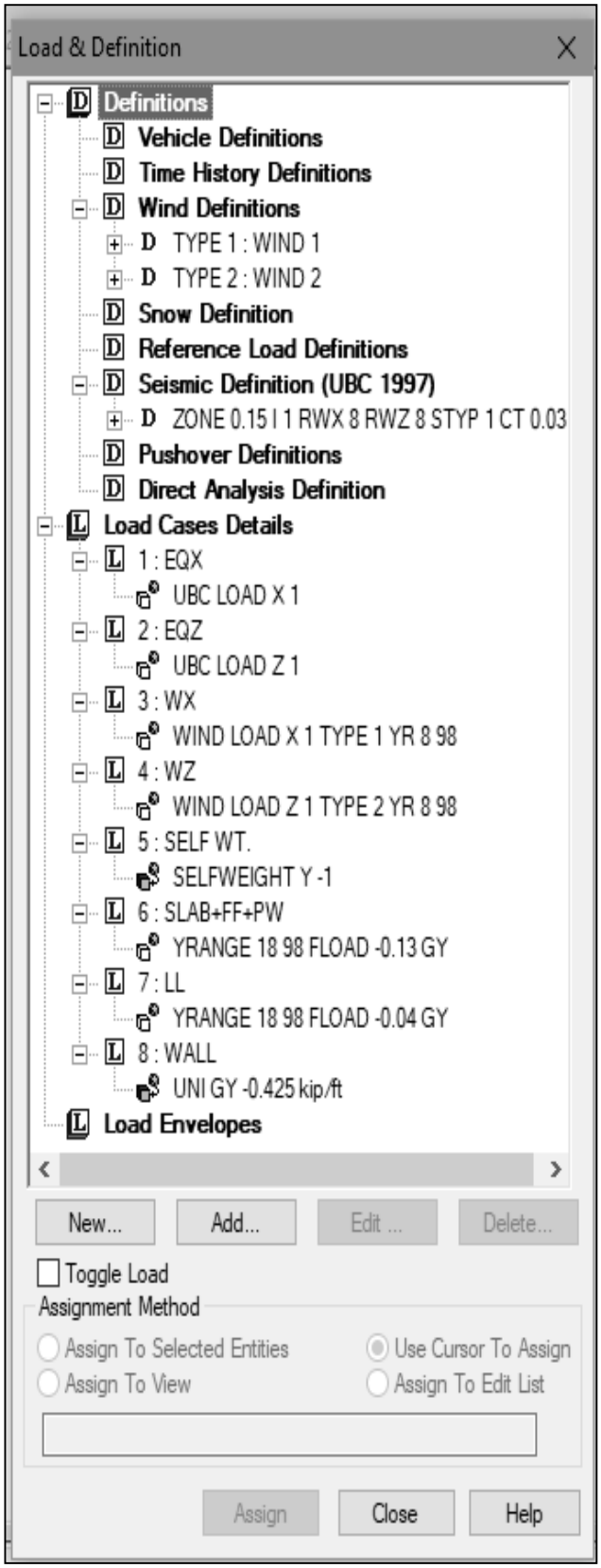


Fig 1.4: Load & Definition Page

CHAPTER-II

CROSS SECTIONAL SHAPES FOR VARIOUS FINITE ELEMENTS

General

To design a building structure, bridge structure or any other structures use multiple elements that can be characterized as beams, columns, trusses etc. These structural elements have some cross-section size and shape to build up the total shape of the structures. The behavior of a structural member is dictated by its material, cross sectional size and shape of the elements and its geometry. Depending on different materials they have different cross sectional shape;

- Rectangular Section
- T-Section
- I-Section
- Channel-Section
- Circular-Section
- Triangular Section
- Wide flanged Shape
- Standard Channel
- Angle
- Structural Tee etc.

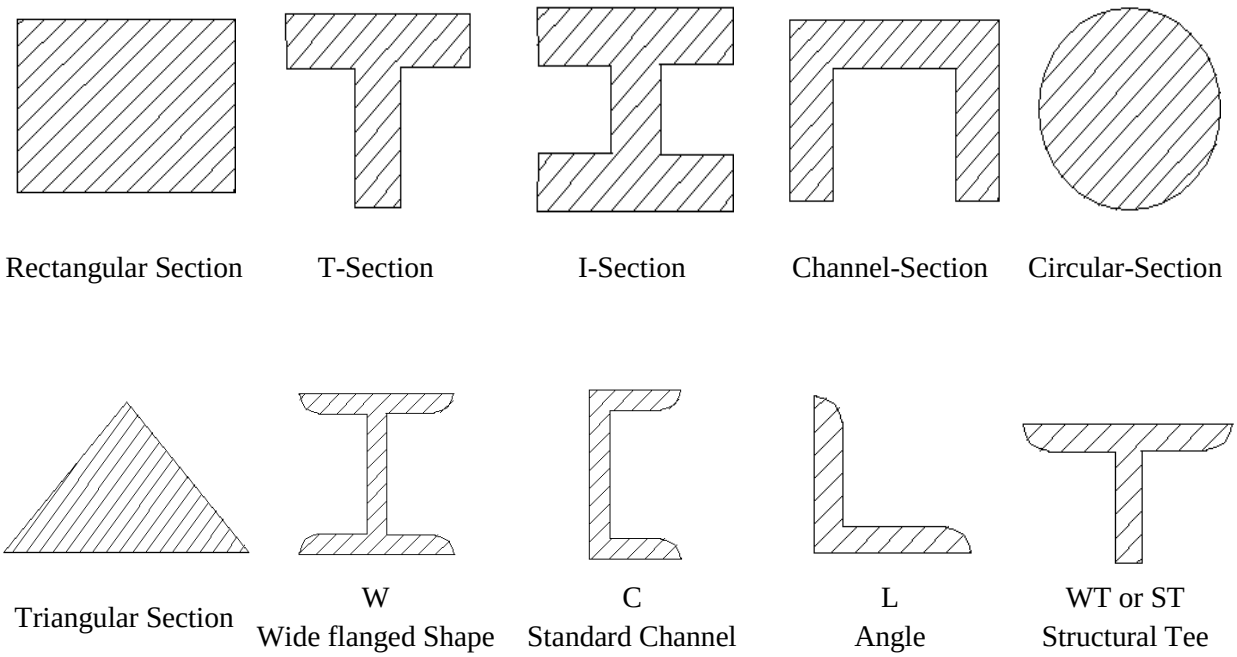
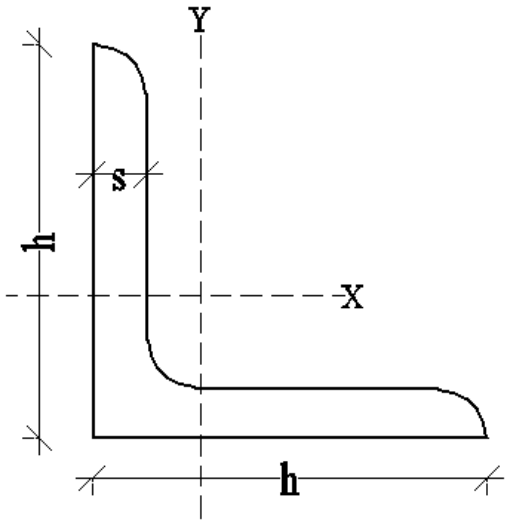


Fig: 2.1 Two dimensional cross section of various finite elements

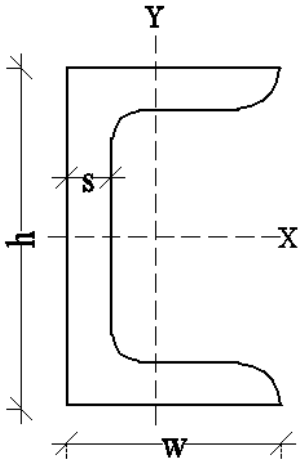
- Some American Standard Steel cross-sections and their properties

1. Angles - Equal Legs



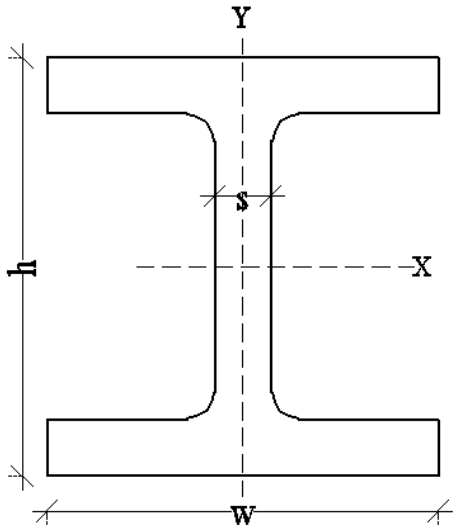
Size (in x in)	Dimensions				Static Parameters
	Depth(h) (in)	Thickness(s) (in)	Sectional Area (in^2)	Weight (lb_f/ft)	Moment of Inertia(I_x) (in^4)
12 x 12	12	1 3/8	30.9	105	410.0
	12	1 1/4	28.3	96.4	377.5
	12	1 1/8	25.6	87.2	344.1
	12	1	22.9	77.8	310.4
10 x 10	10	1 3/8	25.6	87.1	232.1
	10	1 1/4	23.5	79.9	215.1
	10	1 1/8	21.2	72.3	196.2
	10	1	19.0	64.7	177.3
	10	7/8	16.7	56.9	157.6
	10	3/4	14.4	49.1	137.2
8 x 8	8	1 1/8	16.7	56.9	98.0
	8	1	15.0	51.0	89.0
	8	7/8	13.2	45.0	79.6
	8	3/4	11.4	38.9	69.7
	8	5/8	9.6	32.7	59.4
	8	9/16	8.7	29.6	54.1
	8	1/2	7.8	26.4	48.6
6 x 6	6	1	11.0	37.4	35.5
	6	7/8	9.7	33.1	31.9
	6	3/4	8.4	28.7	28.2
	6	5/8	7.1	24.2	24.2

2. American Standard Steel Channels:



Designation	Dimensions					Static Parameters			
						Moment of Inertia		Elastic Section Modulus	
Imperial (in x lb/ft)	Depth - h - (in)	With - w - (in)	Web Thickness (s) (in)	Sectional Area (in^2)	Weight (lb_f/ft)	I_x (in^4)	I_y (in^4)	S_x (in^3)	S_y (in^3)
C 15 x 50	15	3.716	0.716	14.7	50	404	11.0	53.8	3.78
C 15 x 40	15	3.520	0.520	11.8	40	349	9.23	46.5	3.37
C 15 x 33.9	15	3.400	0.400	9.96	33.9	315	8.13	42.0	3.11
C 12 x 30	12	3.170	0.510	8.82	30	162	5.14	27.0	2.06
C 12 x 25	12	3.047	0.387	7.35	25	144	4.47	24.1	1.88
C 12 x 20.7	12	2.942	0.282	6.09	20.7	129	3.88	21.5	1.73
C 10 x 30	10	3.033	0.673	8.82	30	103	3.94	20.7	1.65
C 10 x 25	10	2.886	0.526	7.35	25	91.2	3.36	18.2	1.48
C 10 x 20	10	2.739	0.379	5.88	20	78.9	2.81	15.8	1.32
C 10 x 15.3	10	2.600	0.240	4.49	15.3	67.4	2.28	13.5	1.16
C 9 x 20	9	2.648	0.448	5.88	20	60.9	2.42	13.5	1.17
C 9 x 15	9	2.485	0.285	4.41	15	51.0	1.93	11.3	1.01
C 9 x 13.4	9	2.433	0.233	3.94	13.4	47.9	1.76	10.6	0.96
C 8 x 18.75	8	2.527	0.487	5.51	18.75	44.0	1.98	11.0	1.01
C 8 x 13.75	8	2.343	0.303	4.04	13.75	36.1	1.53	9.03	0.85

3. American Wide Flange Beams:



Designation		Dimensions					Static Parameters			
							Moment of Inertia		Elastic Section Modulus	
Imperial (in x in x lb/ft)	Metric (mm x mm x kg/m)	Depth - h - (mm)	Width - w - (mm)	Web Thickness - s - (mm)	Sectional Area (cm^2)	Weight (kg/m)	I_x (cm^4)	I_y (cm^4)	S_x (cm^3)	S_y (cm^3)
W 4 x 4 x 13	W 100 x 100 x 19.3	106	103	7.1	24.7	19.3	475.9	160.6	89.9	31.2
W 5 x 5 x 16	W 130 x 130 x 23.8	127	127	6.1	30.4	23.8	885.5	311	139.5	49
W 5 x 5 x 19	W 130 x 130 x 28.1	131	128	6.9	35.9	28.1	1099	381.4	167.7	59.6
W 6 x 4 x 9	W 150 x 100 x 13.5	150	100	4.3	17.3	13.5	685.5	91.8	91.4	18.4
W 6 x 4 x 12	W 150 x 100 x 18.0	153	102	5.8	22.9	18	915.9	125.9	122.1	25.4
W 6 x 4 x 16	W 150 x 100 x 24.0	160	102	6.6	30.6	24	1342	182.6	167.8	35.8
W 6 x 6 x 15	W 150 x 150 x 22.5	152	152	5.8	28.6	22.5	1206	386.6	158.6	50.9
W 6 x 6 x 20	W 150 x 150 x 29.8	157	153	6.6	37.9	29.8	1714	555.5	218.4	72.6
W 6 x 6 x 25	W 150 x 150 x 37.1	162	154	8.1	47.4	37.1	2220	706.8	274.1	91.8

- Modulus of Elasticity of Concrete:

According to ACI Code the modulus of elasticity of concrete E_c can be calculated by the formula given below:

$$E_c = 33\, W_c^{1.5} \sqrt{f'_c} \text{ psi (Ib/in}^2\text{)}$$

$$\text{Or, } E_c = 0.04\, W_c^{1.5} \sqrt{f'_c} \text{ MPa (N/mm}^2\text{)}$$

With normal-weight, normal-density concrete these two relations can be simplified to

$$E_c = 57000 \sqrt{f'_c} \text{ psi (Ib/in}^2\text{)}$$

$$E_c = 4700 \sqrt{f'_c} \text{ MPa (N/mm}^2\text{)}$$

Where,

Ec = Modulus of elasticity of concrete (Ib/in^2) or MPa

f'_c = Specified 28-day compressive strength of concrete (Ib/in^2) or MPa

Wc = Concrete Weight.

CHAPTER-III

TWO DIMENSIONAL PORTAL FRAMES

Objective: Analyze the following Two Dimensional Portal frame (Fig: 3.1) under vertical and horizontal loads and find out the following values;

1. Support reactions
2. Shear Force and Bending moment on member CD
3. Displacement (Deflection) of point B, C, D, E

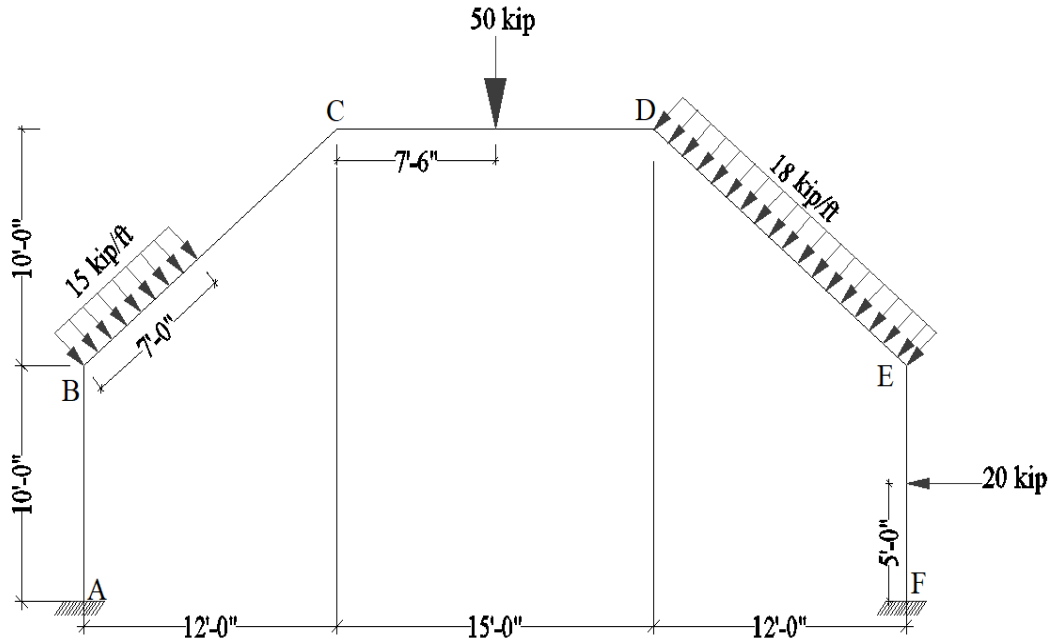


Fig: 3.1

Properties:
Materials= Concrete
Section Size = 15"X 15"

Procedure:

1. Geometry (Model creating):
   - 1.1 Open STAAD Pro. software →File → New → Click on Space →Write the File Name and select Location→Length Units = Foot, Force Units = Kilo Pound → Next →Add Beam →Finish.(Fig: 3.2)
   - 1.2 Now close the Default Grid window → Input the coordinates for point A (X=0, Y=0,Z=0), B (X=0, Y=10, Z=0), C (X=12, Y=20, Z=0), D (X=27, Y=20, Z=0), E (X=39, Y=10,Z= 0), F (X=39, Y=0,Z=0), Then click on Geometry →Add Beam →Add Beam from point to point. (Fig: 3.3) & (Fig: 3.4)

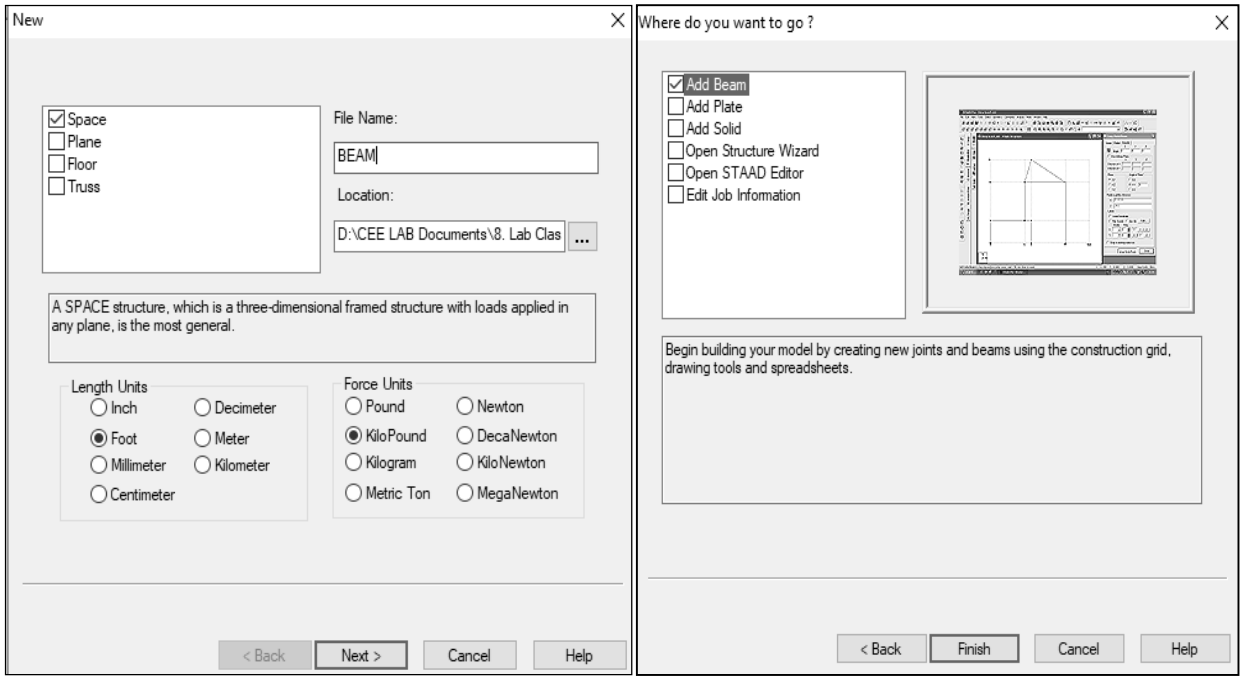
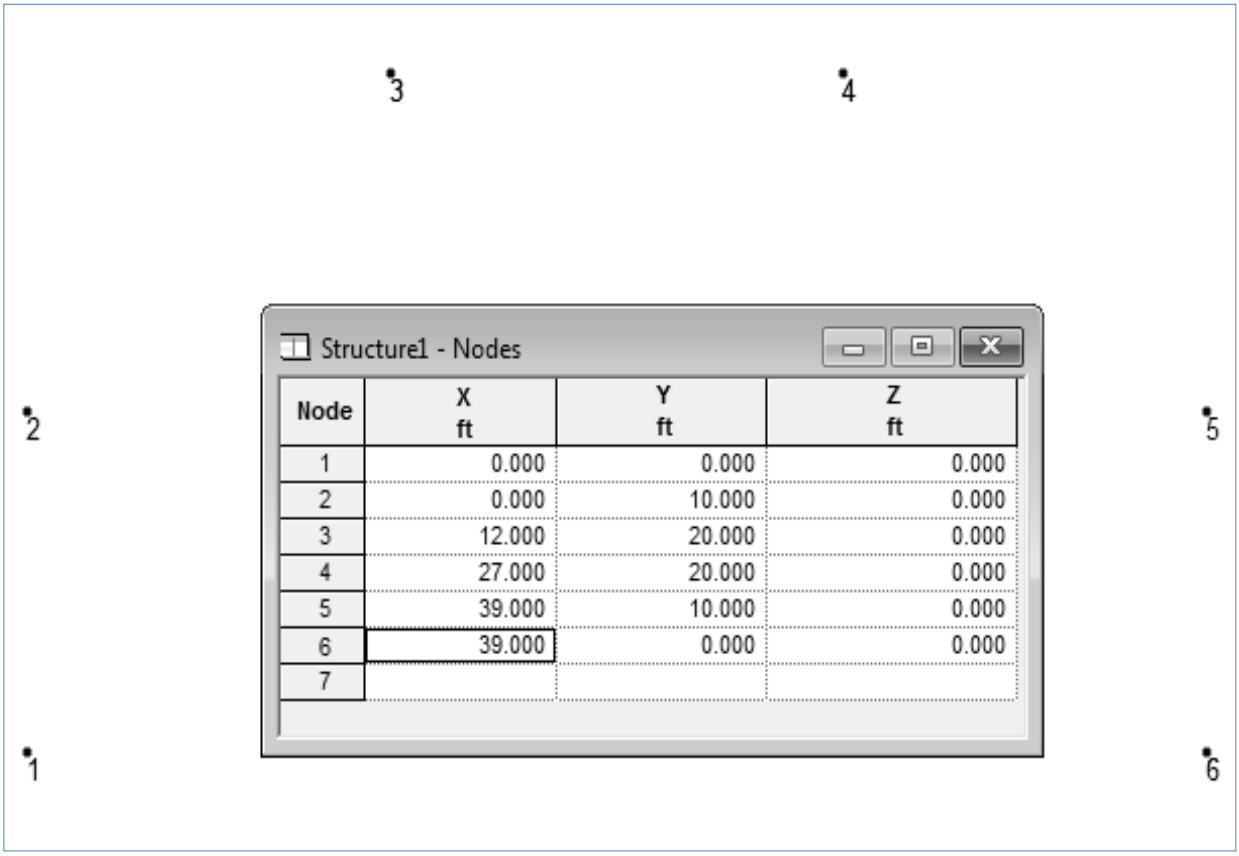


Fig: 3.2



Node	X ft	Y ft	Z ft
1	0.000	0.000	0.000
2	0.000	10.000	0.000
3	12.000	20.000	0.000
4	27.000	20.000	0.000
5	39.000	10.000	0.000
6	39.000	0.000	0.000
7			

Fig: 3.3

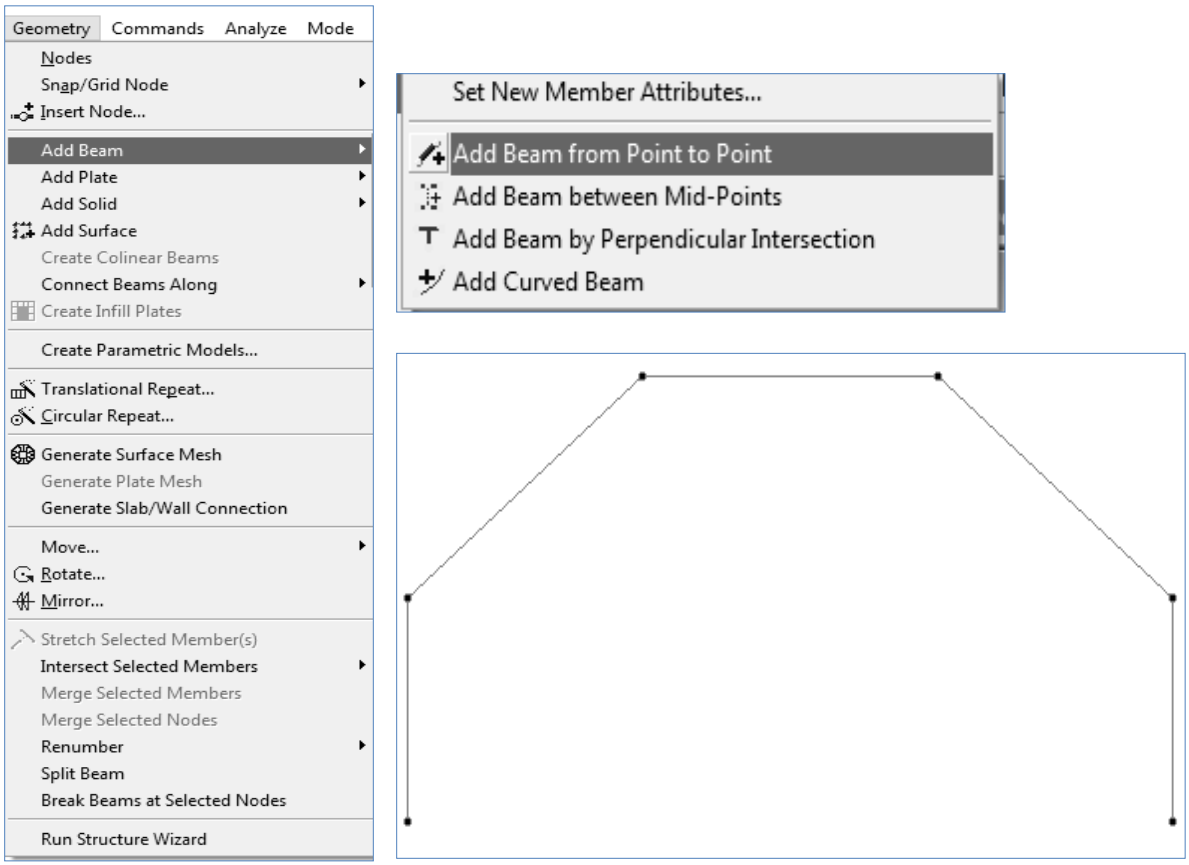


Fig: 3.4

2. General (Define & Assign):

2.1 Property: Define →Rectangle →Material = CONCRETE → YD = 1.25 ft (15”), ZD =1.25 ft (15”) → Add →Close then for Assign select the property and click on Assign to View →Assign → Yes. (Fig: 3.5)

2.2 Support: Create →Fixed →Add. (Fig: 3.6)
Now for Assign click on the Support type →Select the Support point in Beam →Assign to Selected Nodes →Assign →Yes.

2.3 Load & Definitions: Load Cases Details → Add → Loading Type = Dead →Title = Dead Load or DL → Add → Again Loading Type = Live →Title = Live Load or LL → Add → Close. (Fig: 3.7)

- DL →Add →Self weight →Direction = Y, Factor = -1→Add →Close. Then SELFWEIGHT Y-1 →Assign To View →Assign →Yes. (Fig: 3.8)
- For Given loads: Again Live Load or LL →Add →Member Load →Uniform Force →W1 = -15 kip, d1= 0 ft, d2=8 ft →Direction = Y(Local) →Add →Close, then click on defined force and select the required Beam → Assign to selected Beams →Assign →Yes. (Fig: 3.9 & Fig: 3.10)
- The same process follow for other Trapezoidal, Concentrated and Uniform distributed forces.
- Load Combination: Load Cases Details → Add →Define Combinations →Name = DL+LL →Default ai = 1, then select DL, LL and click on >> to send right side from left side. (Fig: 3.11)

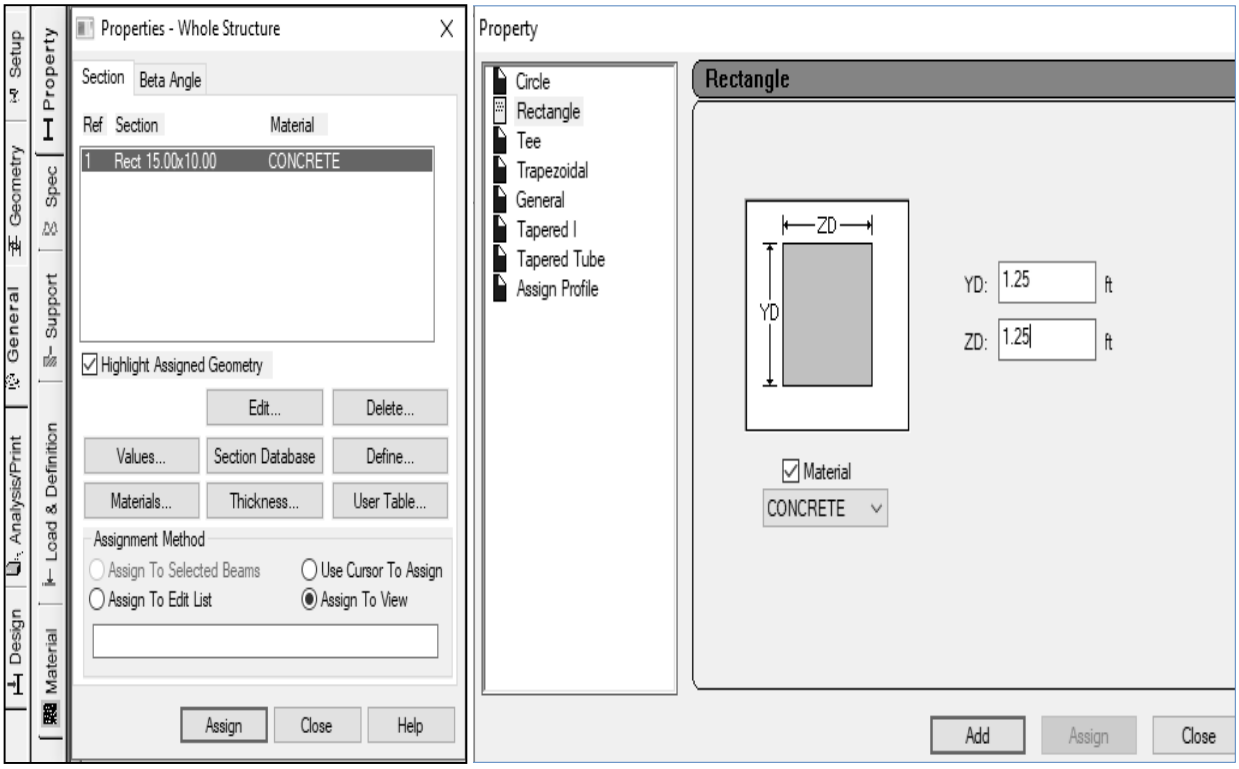

Setup
Geometry
General
Analysis/Print
Design
Property
Spec
Support
Load & Definition
Material
Properties - Whole Structure
Section
Beta Angle
Ref
Section
Material
1
Rect 15.00x10.00
CONCRETE
Highlight Assigned Geometry
Edit...
Delete...
Values...
Section Database
Define...
Materials...
Thickness...
User Table...
Assignment Method
Assign To Selected Beams
Use Cursor To Assign
Assign To Edit List
Assign To View
Assign
Close
Help
Property
Circle
Rectangle
Tee
Trapezoidal
General
Tapered I
Tapered Tube
Assign Profile
Rectangle
ZD
YD
YD:
1.25
ft
ZD:
1.25
ft
Material
CONCRETE
Add
Assign
Close

Fig: 3.5

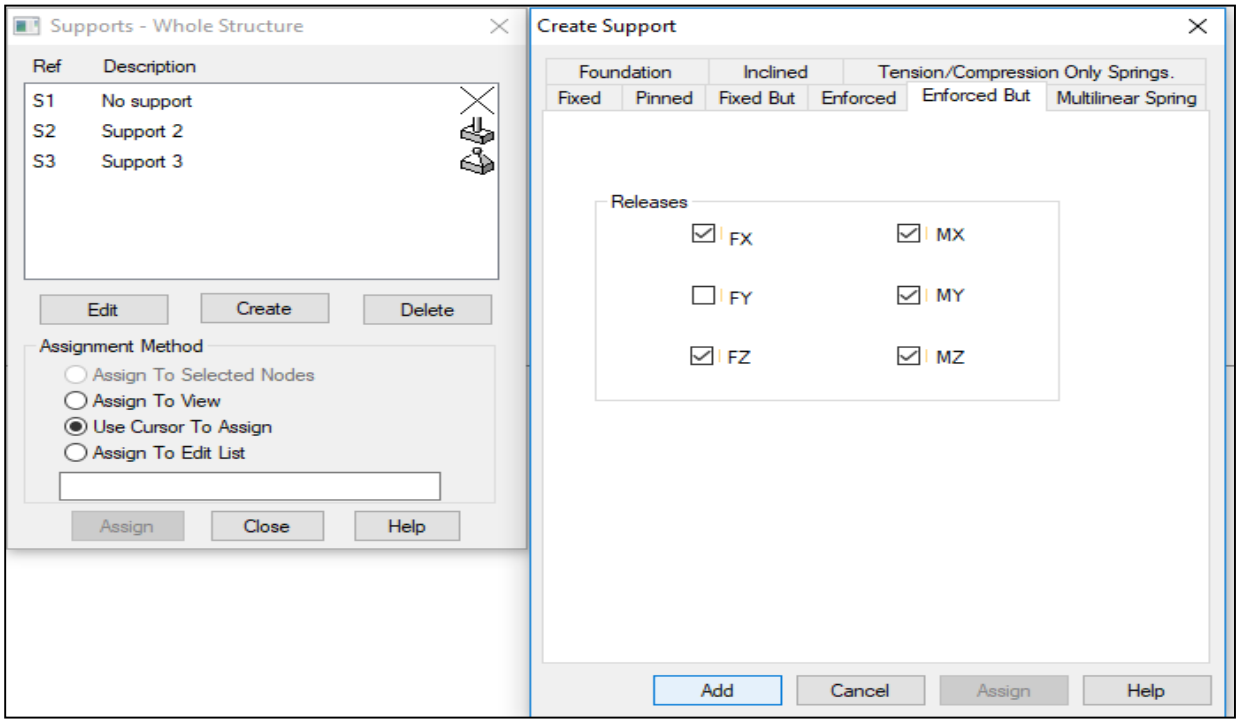

Supports - Whole Structure
Ref
Description
S1
No support
S2
Support 2
S3
Support 3
Edit
Create
Delete
Assignment Method
Assign To Selected Nodes
Assign To View
Use Cursor To Assign
Assign To Edit List
Assign
Close
Help
Create Support
Foundation
Inclined
Tension/Compression Only Springs.
Fixed
Pinned
Fixed But
Enforced
Enforced But
Multilinear Spring
Releases
FX
MX
FY
MY
FZ
MZ
Add
Cancel
Assign
Help

Fig: 3.6

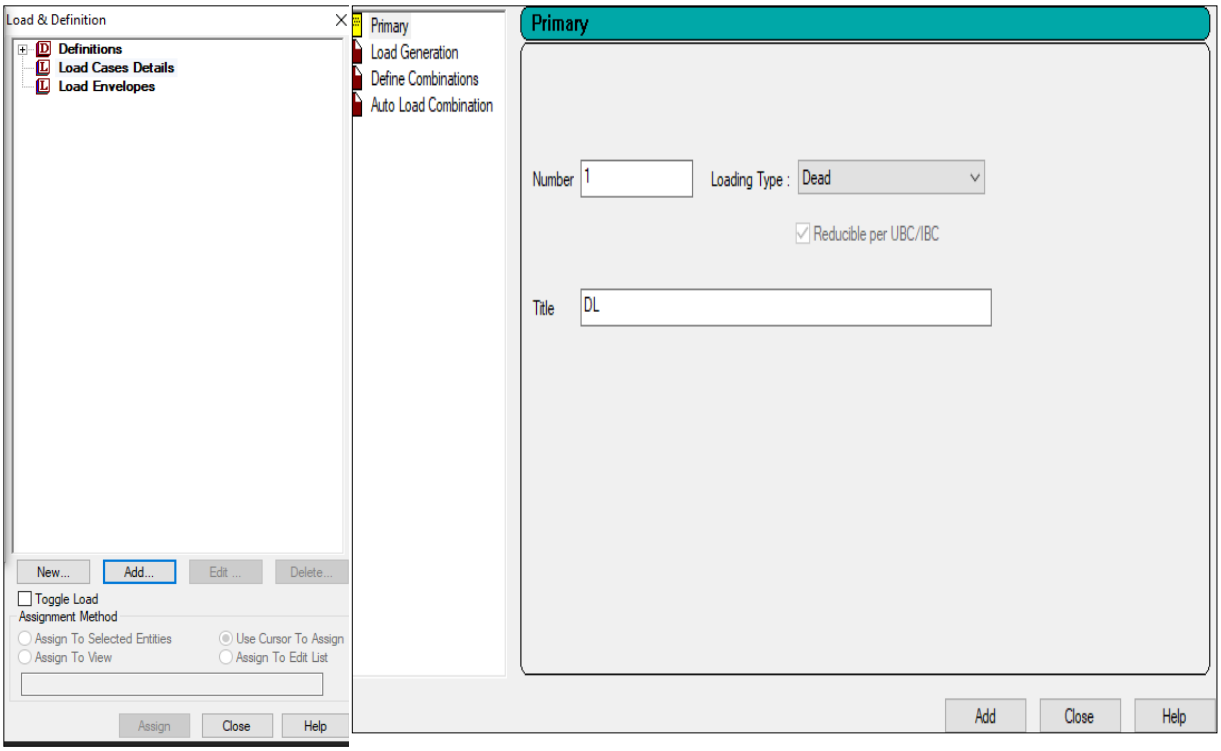

Load & Definition
Definitions
Load Cases Details
Load Envelopes
New...
Add...
Edit ...
Delete...
Toggle Load
Assignment Method
Assign To Selected Entities
Use Cursor To Assign
Assign To View
Assign To Edit List
Assign
Close
Help
Primary
Load Generation
Define Combinations
Auto Load Combination
Primary
Number 1
Loading Type : Dead
Reducible per UBC/IBC
Title DL
Add
Close
Help

Fig: 3.7

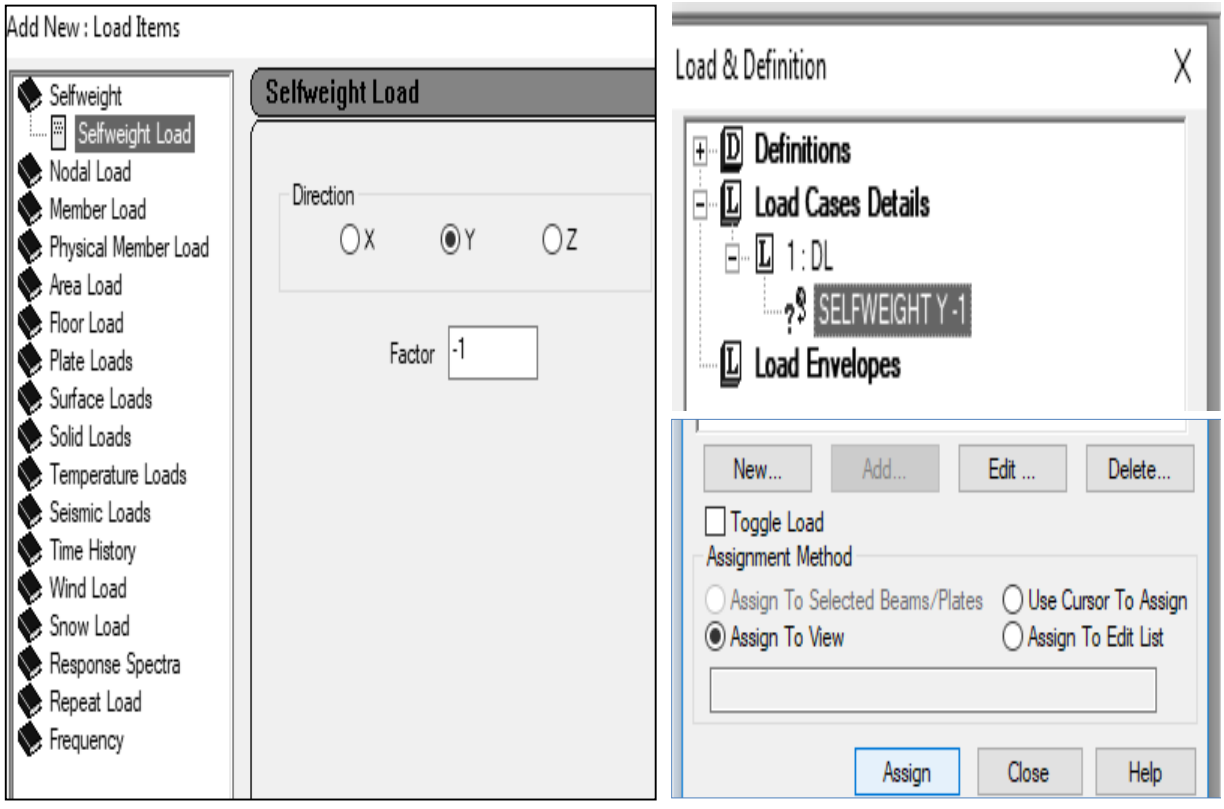

Add New : Load Items
Selfweight
Selfweight Load
Nodal Load
Member Load
Physical Member Load
Area Load
Floor Load
Plate Loads
Surface Loads
Solid Loads
Temperature Loads
Seismic Loads
Time History
Wind Load
Snow Load
Response Spectra
Repeat Load
Frequency
Selfweight Load
Direction
X
Y
Z
Factor -1
Load & Definition
Definitions
Load Cases Details
1 : DL
SELFWEIGHT Y -1
Load Envelopes
New...
Add...
Edit ...
Delete...
Toggle Load
Assignment Method
Assign To Selected Beams/Plates
Use Cursor To Assign
Assign To View
Assign To Edit List
Assign
Close
Help

Fig: 3.8

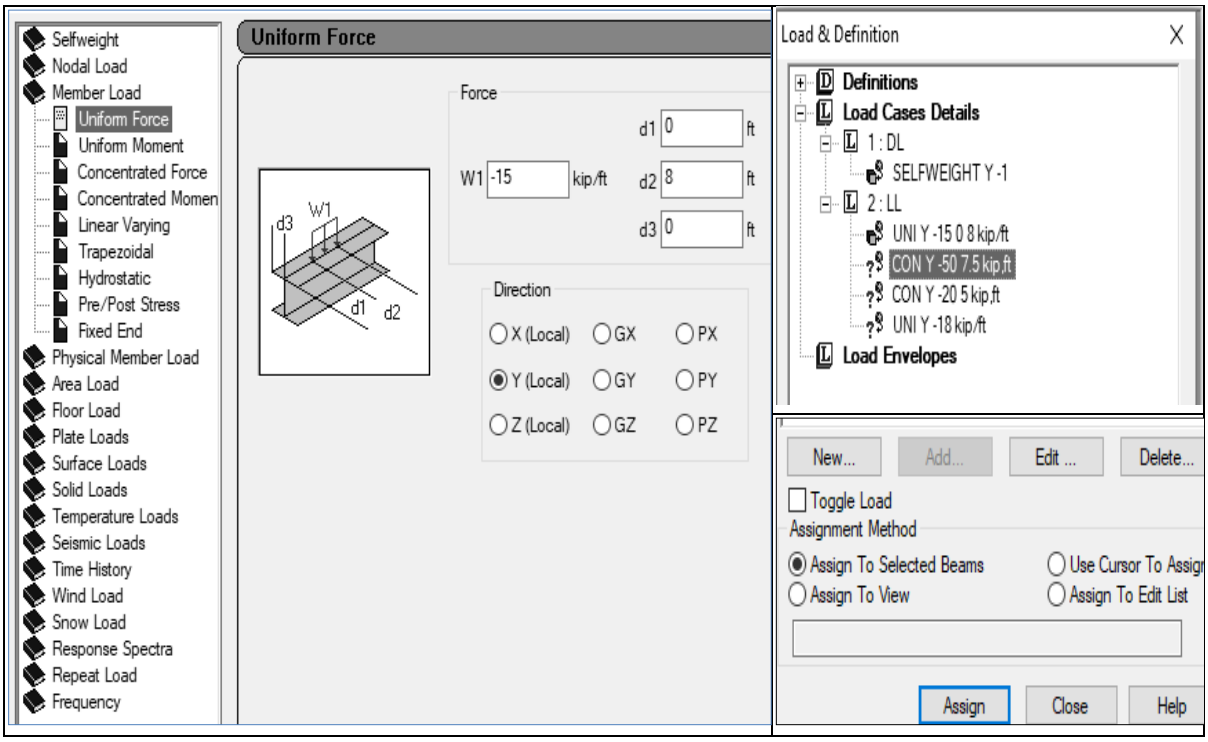


Fig: 3.9

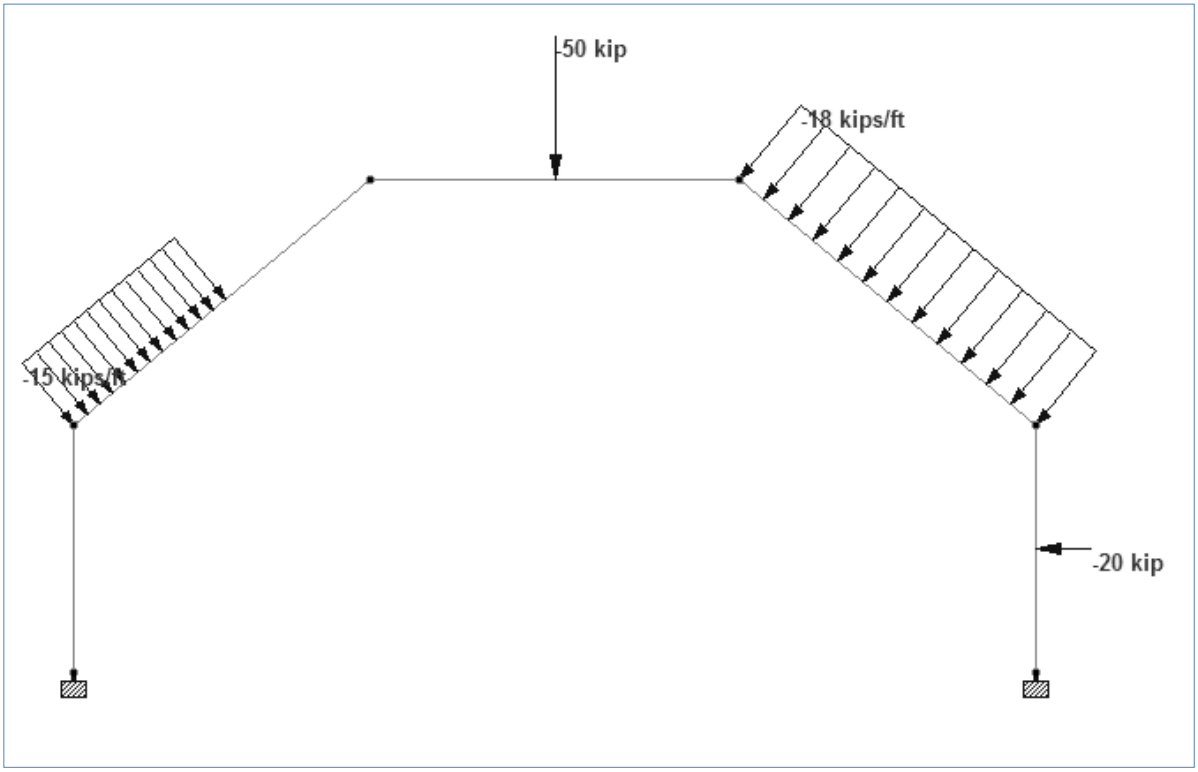


Fig: 3.10

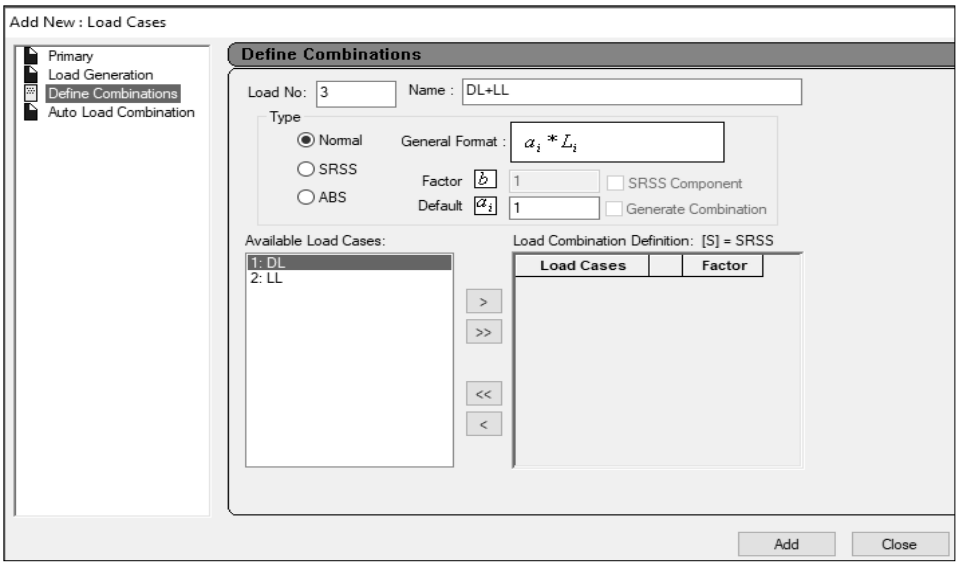


Fig: 3.11

3. Analysis and Result:

- From left side click on Analysis/Print →Static Check or All →Add →Close
- At Menu bar →Analyze →Run Analysis →Go to post processing mode →Done →Selected load cases = DL+LL →Apply →OK. (Fig: 3.12 & Fig: 3.13)
- For Support Reactions use node cursor and double click on the support point →Reactions. Then get the Table for all Support Reactions. (Fig: 3.14)
- For Beam Forces: From left side click on Beam →Graphs the find out Bending moment, Shear force, and Axial force by clicking on required Beam from the following. (Fig: 3.15)
- For Displacement (Deflection) of point go to Result (from Menu bar) →Deflection. use node cursor and Double click on the required point by using Node cursor →Displacements. Then get the Table for all Node Displacements. (Fig: 3.16)

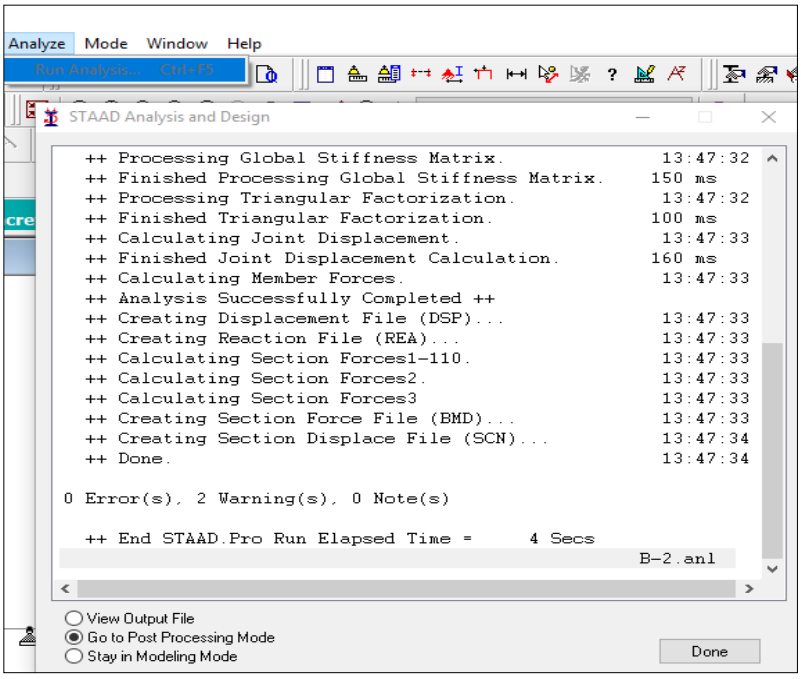


Fig: 3.12

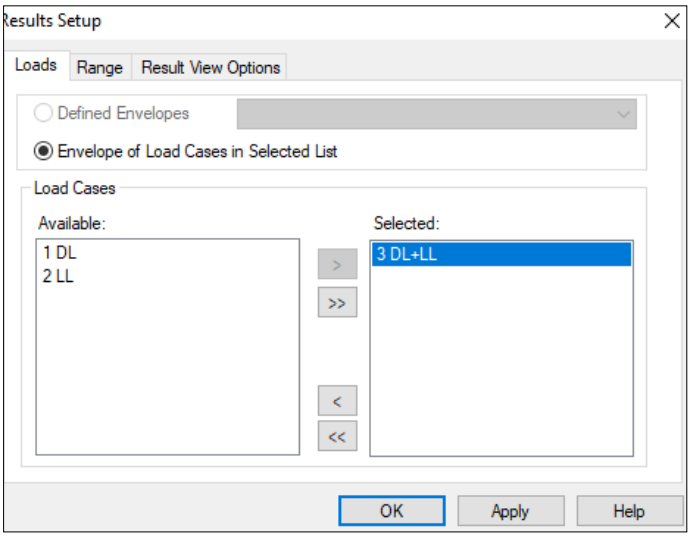


Fig: 3.13

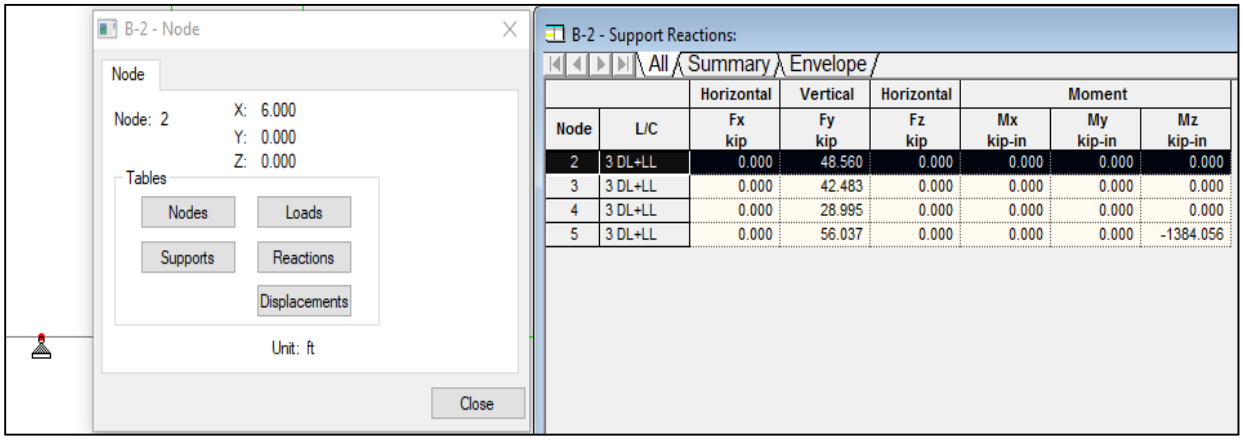


Fig: 3.14

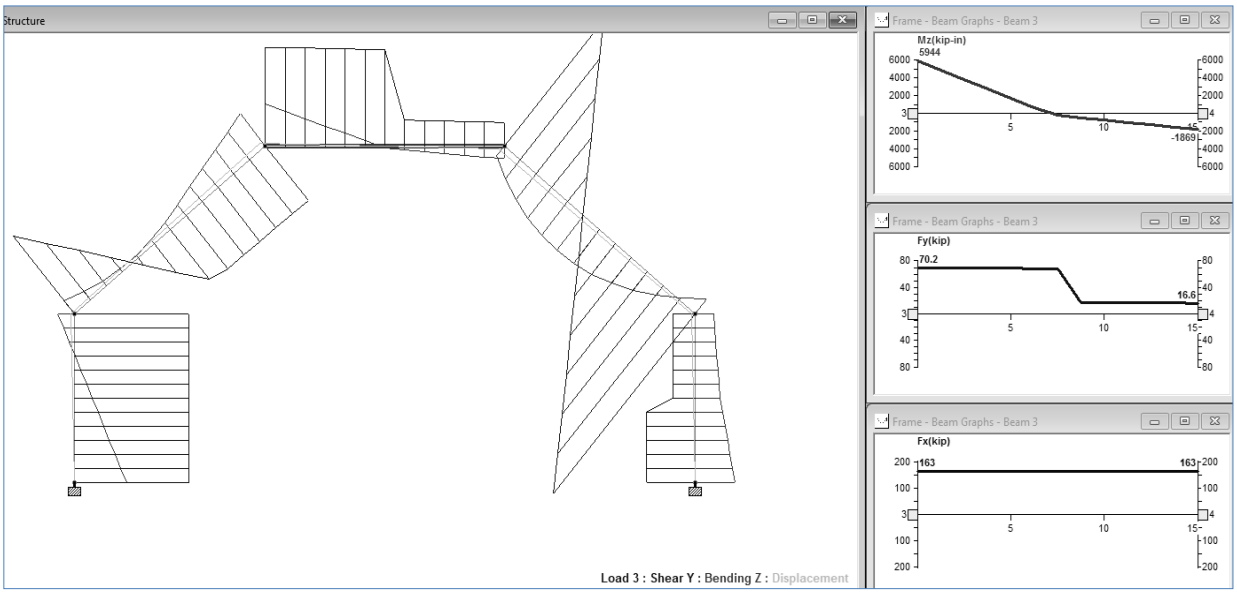
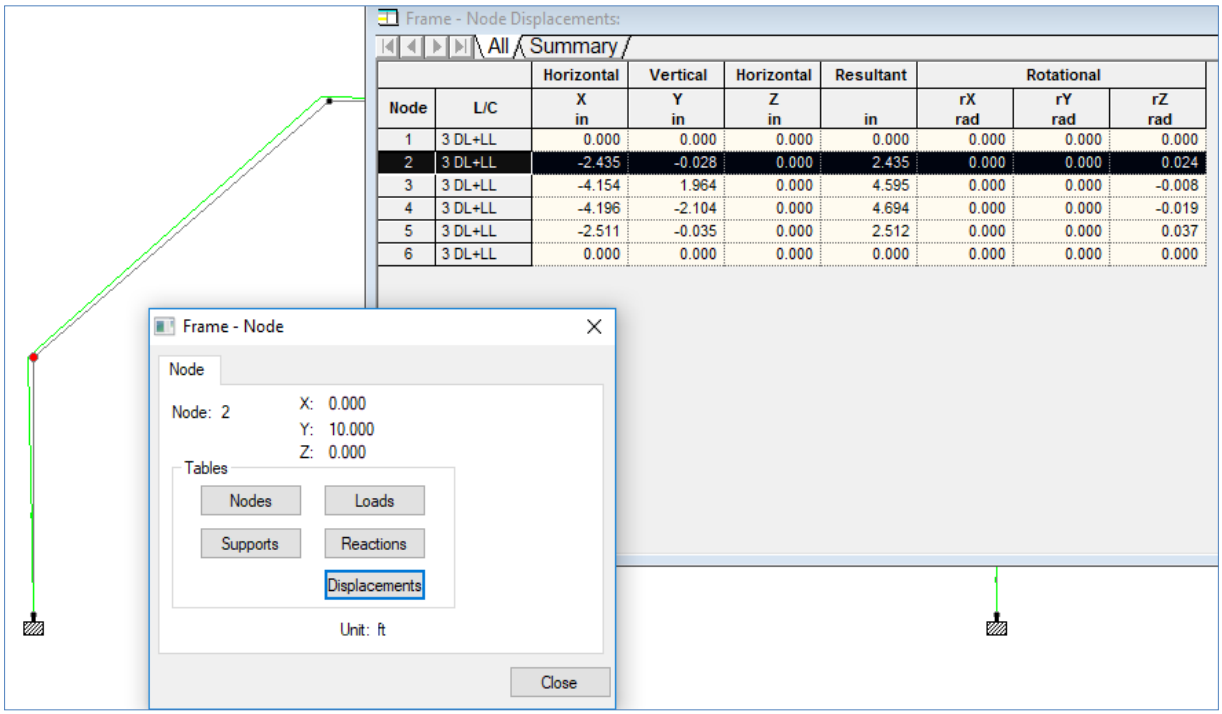


Fig: 3.15



Node	L/C	Horizontal X in	Vertical Y in	Horizontal Z in	Resultant in	Rotational rX rad	rY rad	rZ rad
1	3 DL+LL	0.000	0.000	0.000	0.000	0.000	0.000	0.000
2	3 DL+LL	-2.435	-0.028	0.000	2.435	0.000	0.000	0.024
3	3 DL+LL	-4.154	1.964	0.000	4.595	0.000	0.000	-0.008
4	3 DL+LL	-4.196	-2.104	0.000	4.694	0.000	0.000	-0.019
5	3 DL+LL	-2.511	-0.035	0.000	2.512	0.000	0.000	0.037
6	3 DL+LL	0.000	0.000	0.000	0.000	0.000	0.000	0.000

Fig: 3.16

.....******.....

Assignment-01:

Analyze the following 2D Portal frame under vertical and horizontal loads and find out the following values;

a) Support reactions
b) Displacement (Deflection) of point B, C, D
c) Shear Force and Bending moment on member BC

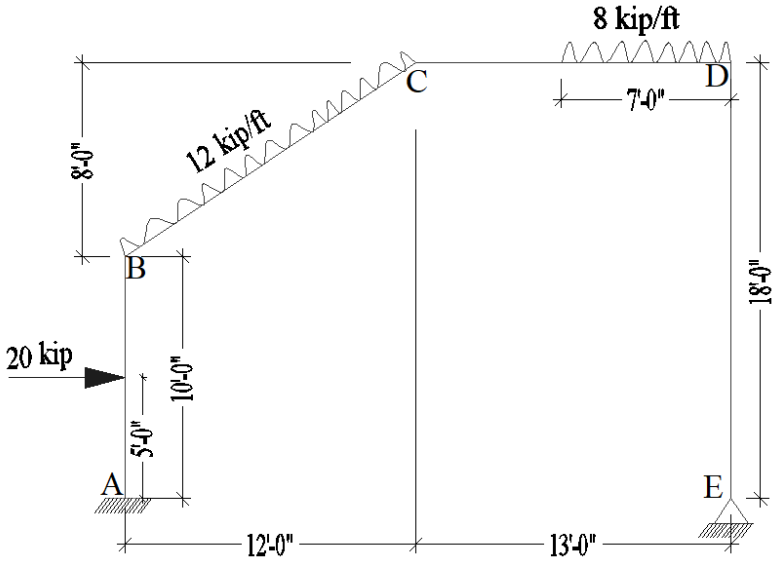


Fig: 3.17

Properties:
Materials= Concrete
Section Size = 12"X 12"

Assignment-02:

Analyze the following 2D Portal frame under vertical and horizontal loads and find out the following values;

a) Support reactions
b) Displacement (Deflection) of point B, C, D
c) Shear Force and Bending moment on member BC

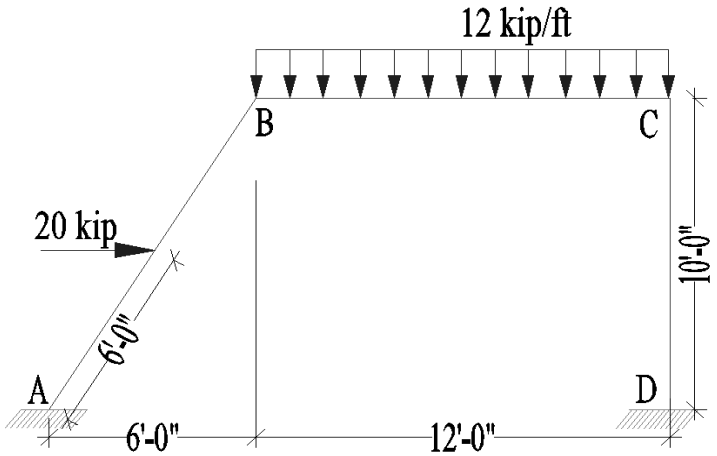


Fig: 3.18

CHAPTER-IV

TRUSS ANALYSIS

A. Roof Truss

Objective: Analyze the following Roof Truss (Fink Type) and find out the following values;

1. Support reactions
2. Forces for the members a, b, c, d, e, f

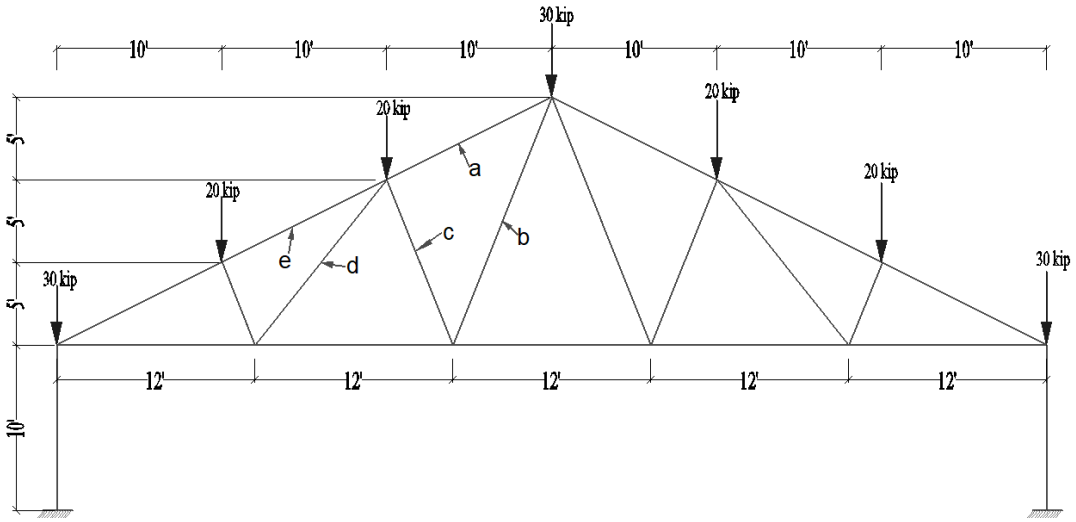


Fig. 4.1: Two Dimensional View of the Roof Truss (Fink Type)

Properties:

Materials= Steel

All members are Japanese Angle = L 60X60X4

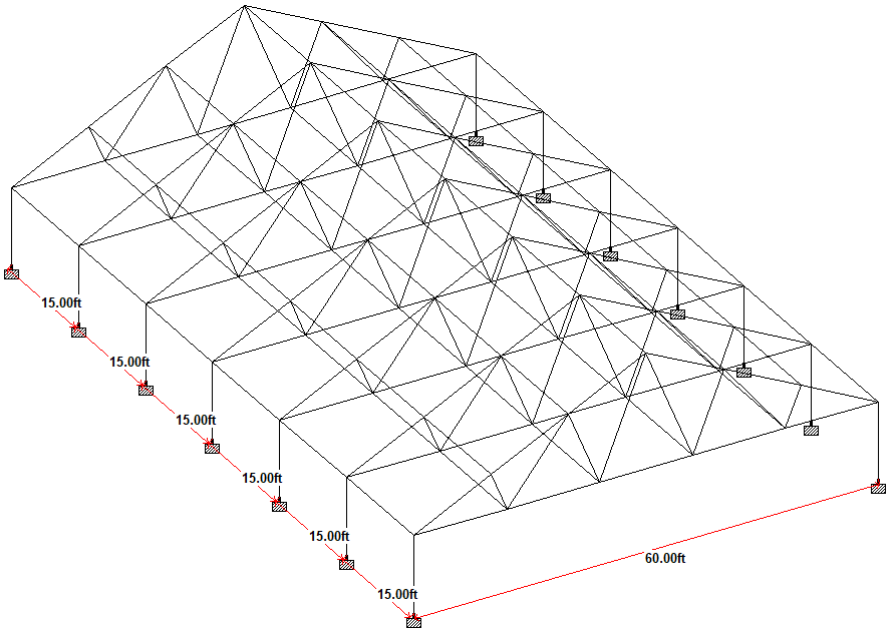


Fig. 4.2: Three Dimensional View of the Roof Truss (Fink Type)

Procedure:

1. Geometry (Model creating):

1.1 Open STAAD Pro. software →File → New → Click on Truss →Write the File Name and select Location→Length Units = Foot, Force Units = Kilo Pound → Next →Add Beam →Finish.(Fig: 4.3)

1.2 Now close the Default Grid window → input coordinates value for all nodes like 1st Node point (X=0, Y=0,Z=0), 2nd Node point (X=10, Y=5,Z=0), 3rd Node point (X=12, Y=0,Z=0), 4th Node point (X=20, Y=10,Z=0), 5th Node point (X=24, Y=0,Z=0) and similarly others nodes Then click on Geometry →Add Beam →Add Beam from point to point. (Fig: 4.4) & (Fig: 4.5)

1.3 Now Select the Nodes 1 & 11 (in View window) by using node cursor →Translational Repeat → Global Direction =Y → No. of Steps = 1 → Spacing= -10(-ve) → Click on Link Steps → OK (Fig: 4.6)

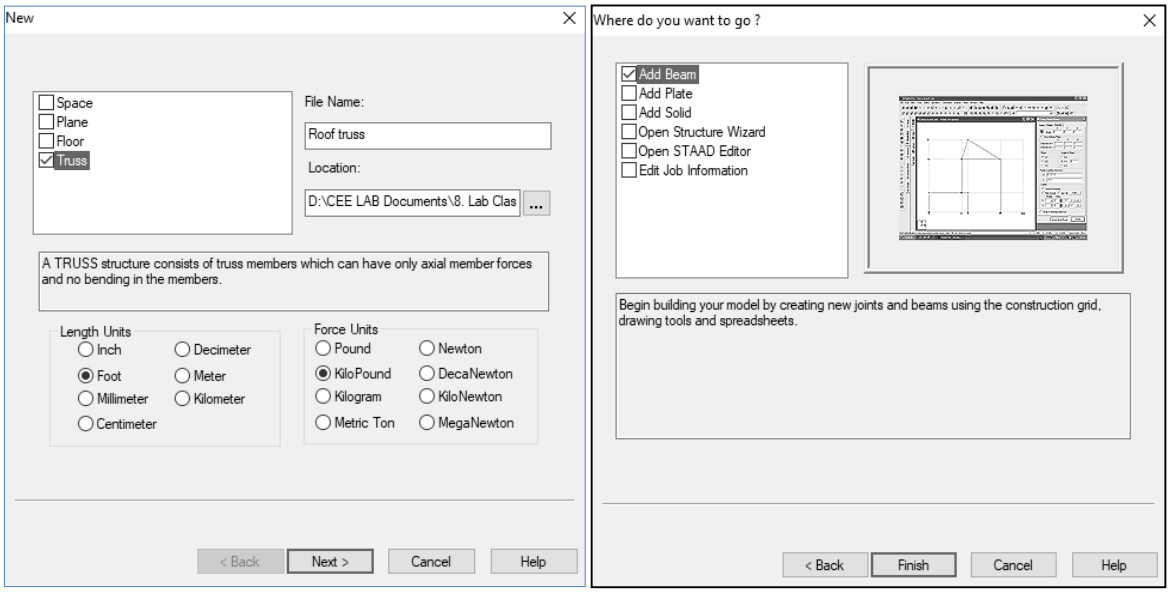


Fig: 4.3

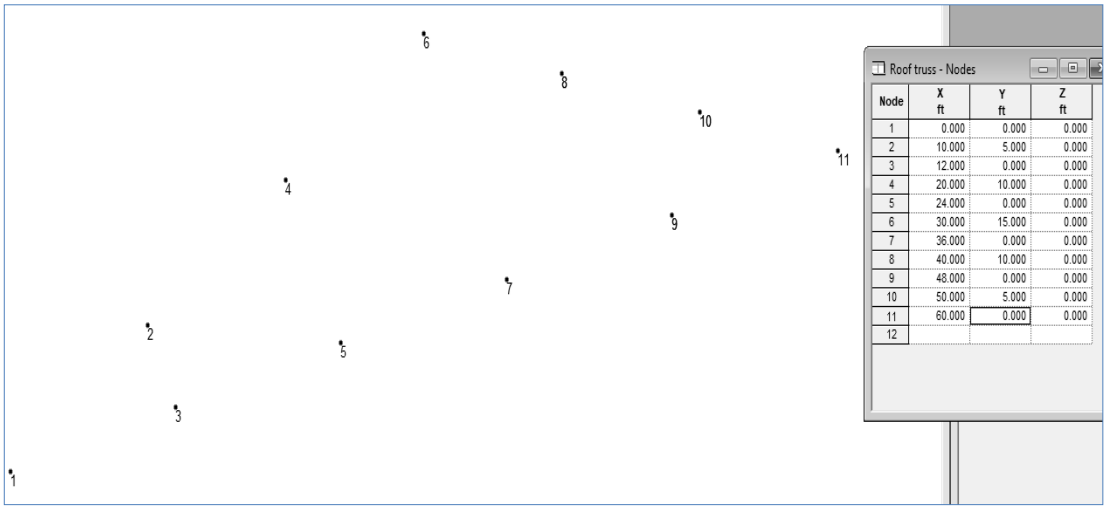


Fig: 4.4

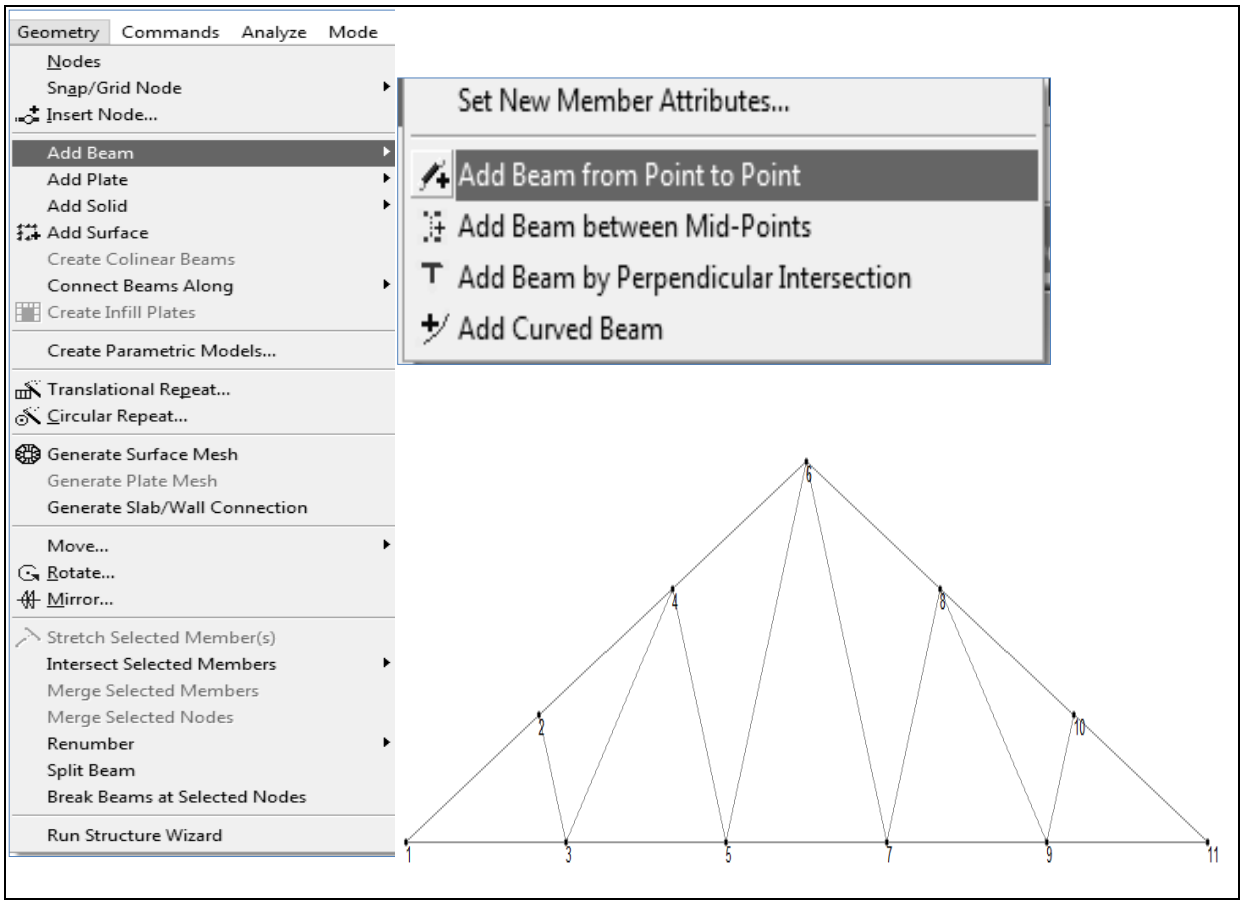

Geometry
Commands
Analyze
Mode
Nodes
Snap/Grid Node
Insert Node...
Add Beam
Add Plate
Add Solid
Add Surface
Create Colinear Beams
Connect Beams Along
Create Infill Plates
Create Parametric Models...
Translational Repeat...
Circular Repeat...
Generate Surface Mesh
Generate Plate Mesh
Generate Slab/Wall Connection
Move...
Rotate...
Mirror...
Stretch Selected Member(s)
Intersect Selected Members
Merge Selected Members
Merge Selected Nodes
Renumber
Split Beam
Break Beams at Selected Nodes
Run Structure Wizard
Set New Member Attributes...
Add Beam from Point to Point
Add Beam between Mid-Points
Add Beam by Perpendicular Intersection
Add Curved Beam
1
2
3
4
5
6
7
8
9
10
11

Fig: 4.5

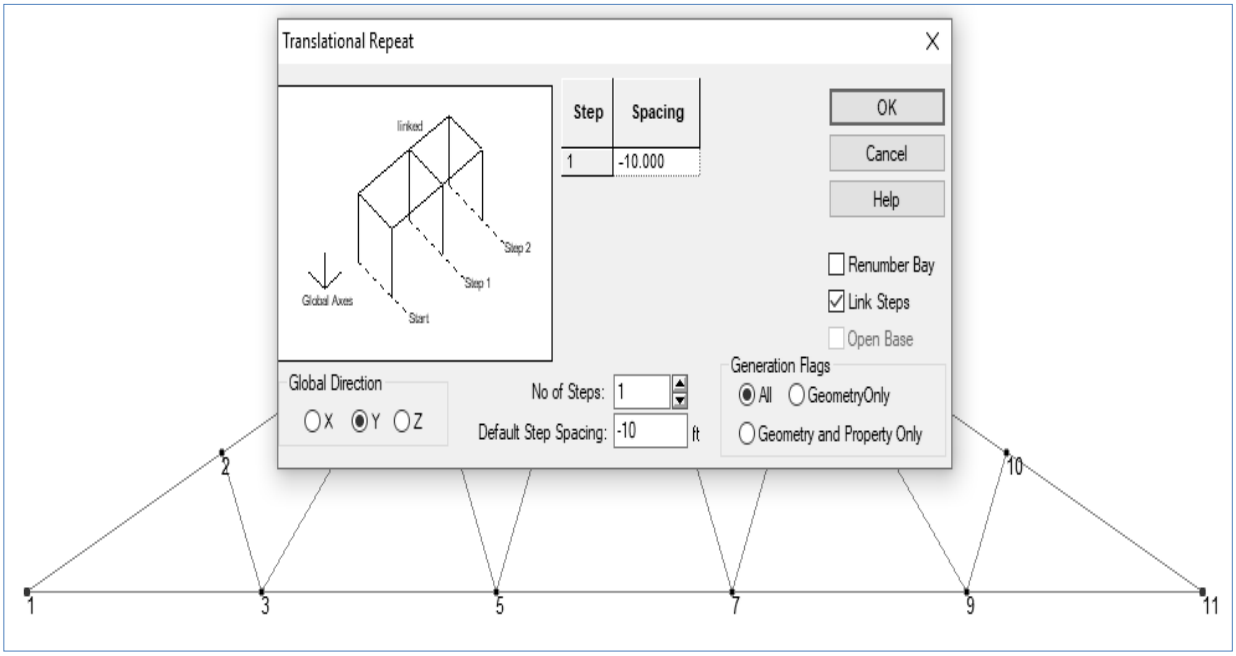

Translational Repeat
linked
Step 2
Step 1
Start
Global Axes
Step
Spacing
1
-10.000
OK
Cancel
Help
Renumber Bay
Link Steps
Open Base
Global Direction
X
Y
Z
No of Steps:
1
Default Step Spacing:
-10
ft
Generation Flags
All
GeometryOnly
Geometry and Property Only
1
2
3
5
7
9
10
11

Fig: 4.6

2. General (Define & Assign):

2.1 Property: Section Database →Japanese→ Angle→ L60X60X4 →Material = STEEL→ Add →Close then for Assign select the property and click on Assign to View →Assign → Yes. (Fig: 4.7)

2.2 Support: Create →Fixed →Add → Now for Assign click on the Support type →Select the Support point in Truss →Assign to Selected Nodes →Assign →Yes.

2.3 Translational Repeat: Now Select the whole structure (in View window) by using beam cursor →Translational Repeat → Global Direction =Z → No. of Steps = 6 → Spacing= 15 → Click on Link Steps → OK and then delete extra beams (Fig: 4.8)

2.4 Load & Definitions: → Load Cases Details → Add → Loading Type = Dead →Title = Dead Load or DL → Add → Loading Type = Live →Title = Live Load or LL → Add → Close.

- DL →Add →Self weight →Direction = Y, Factor = -1→Add →Close. Then SELFWEIGHT Y-1 →Assign To View →Assign →Yes. (Fig: 4.10)
- For Given loads: Again Live Load or LL →Add →Nodal Load → Fy = -30 kip →Add → Fy = -20 kip→Add →Close, then click on defined force and select the required Nodes → Assign to selected Nodes →Assign →Yes. (Fig: 4.11)

2.5 Load Combination: Load Cases Details → Add →Define Combinations →Name = DL+LL →Default ai = 1, then select DL, LL and click on >> to send right side from left side. (Fig: 4.12)

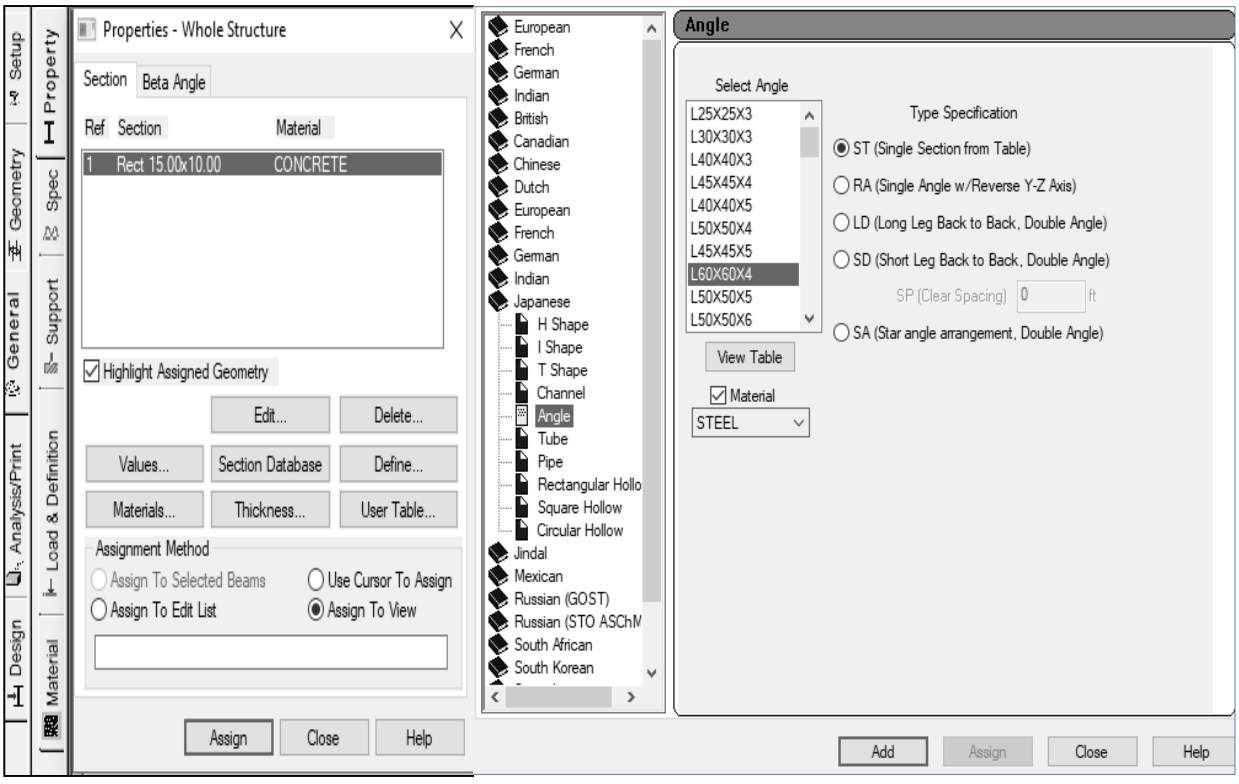


Fig: 4.7

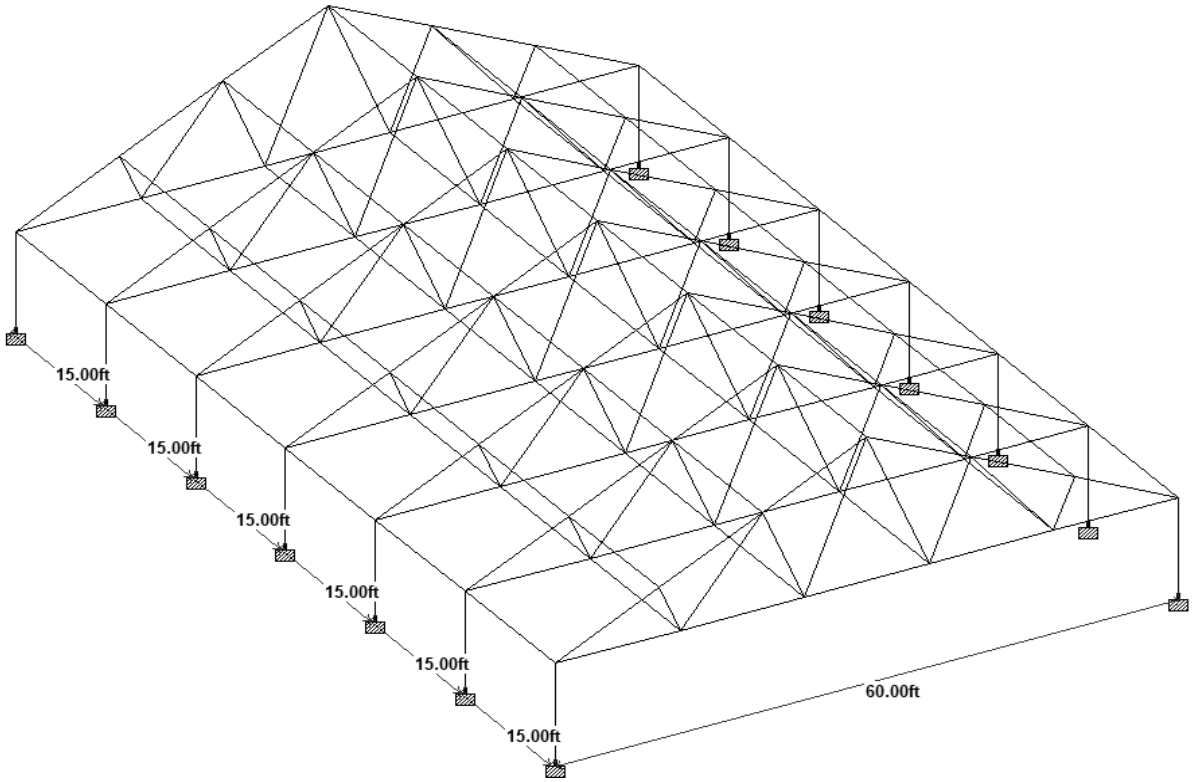


Fig: 4.8

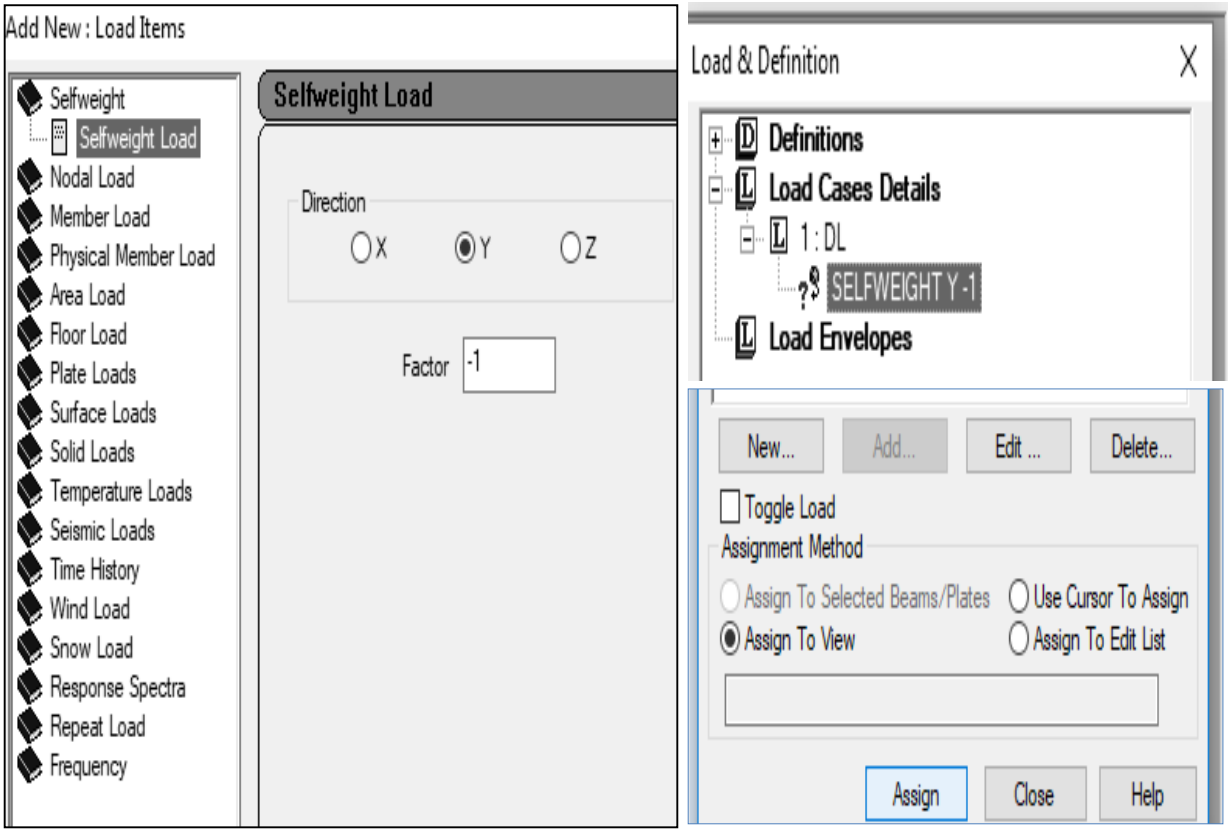


Fig: 4.10

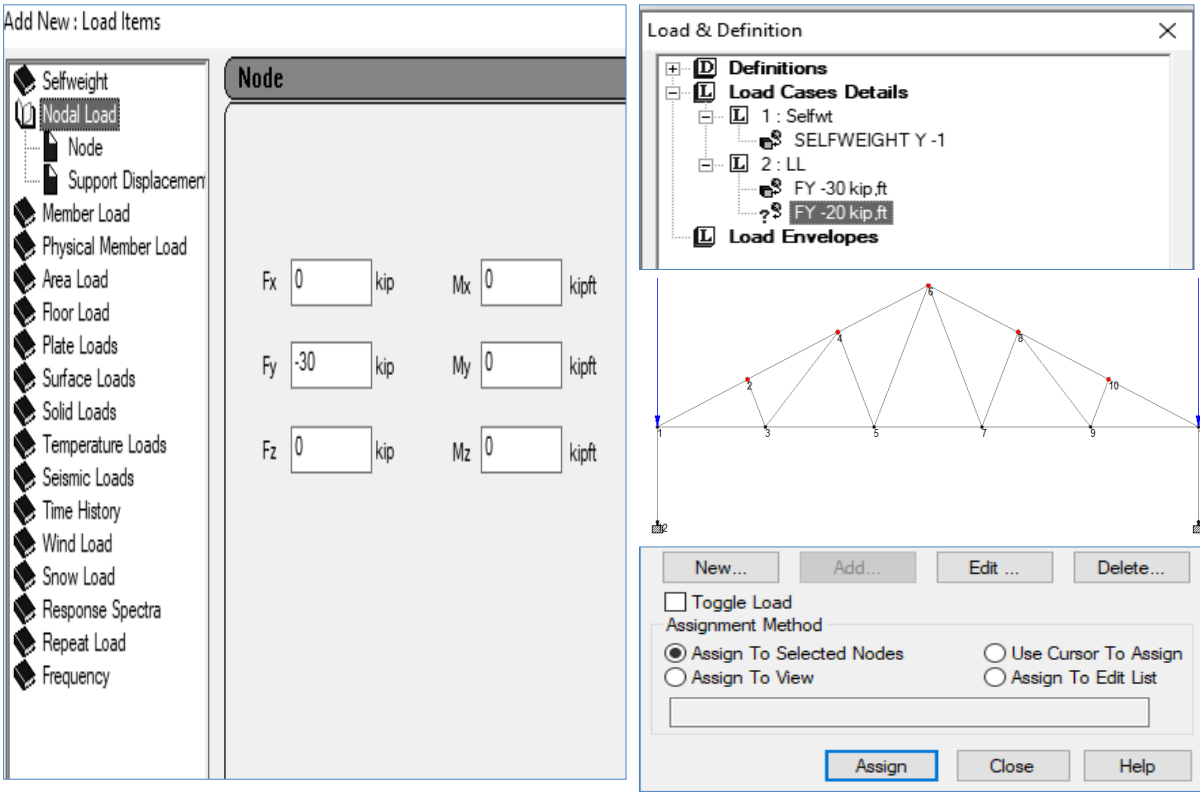


Fig: 4.11

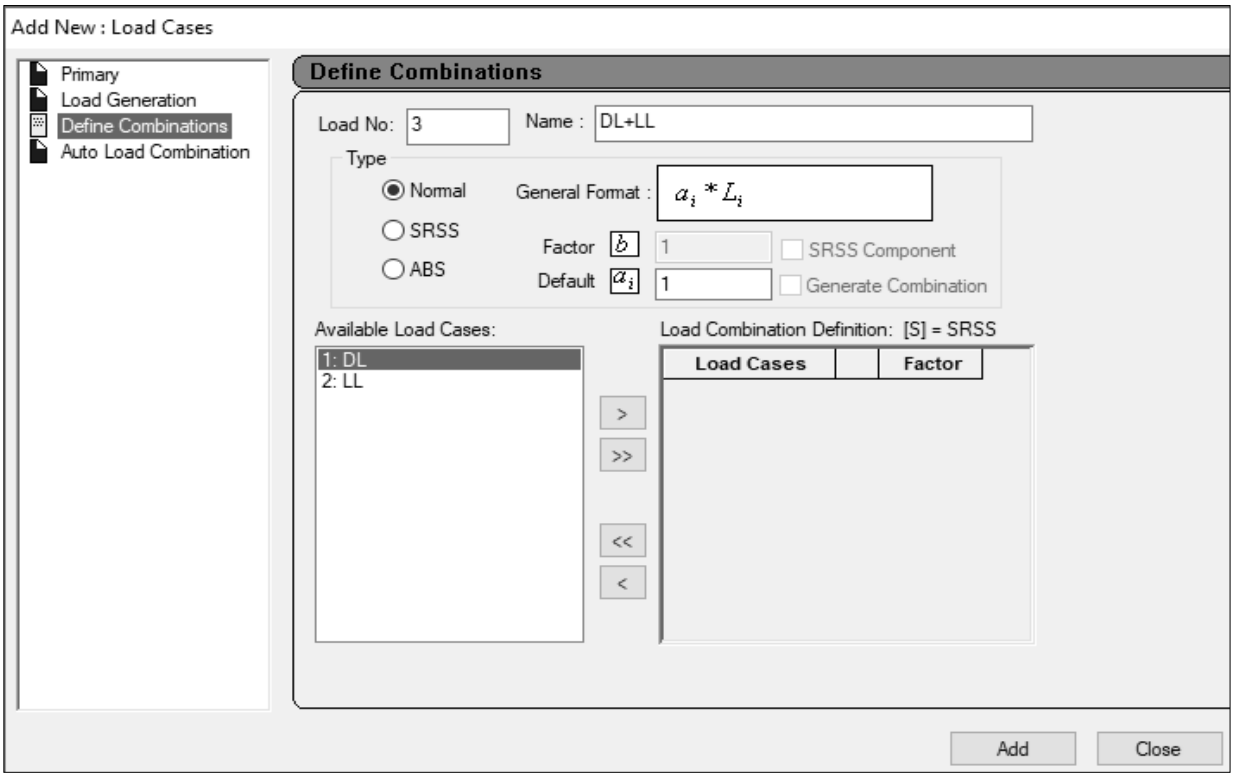


Fig: 4.12

3. Analysis and Result:

- From left side click on Analysis/Print →Static Check or All →Add →Close
- At Menu bar →Analyze →Run Analysis →Go to post processing mode →Done →Selected load cases = DL+LL →Apply →OK. (Fig: 4.13 & Fig: 4.14)
- For Support Reactions use node cursor and double click on the support point →Reactions. Then get the Table for all Support Reactions. (Fig: 4.15)
- For Beam Forces: From left side click on Beam →Graphs the find out Axial force by clicking on required Beam from the following (Fig: 4.16)

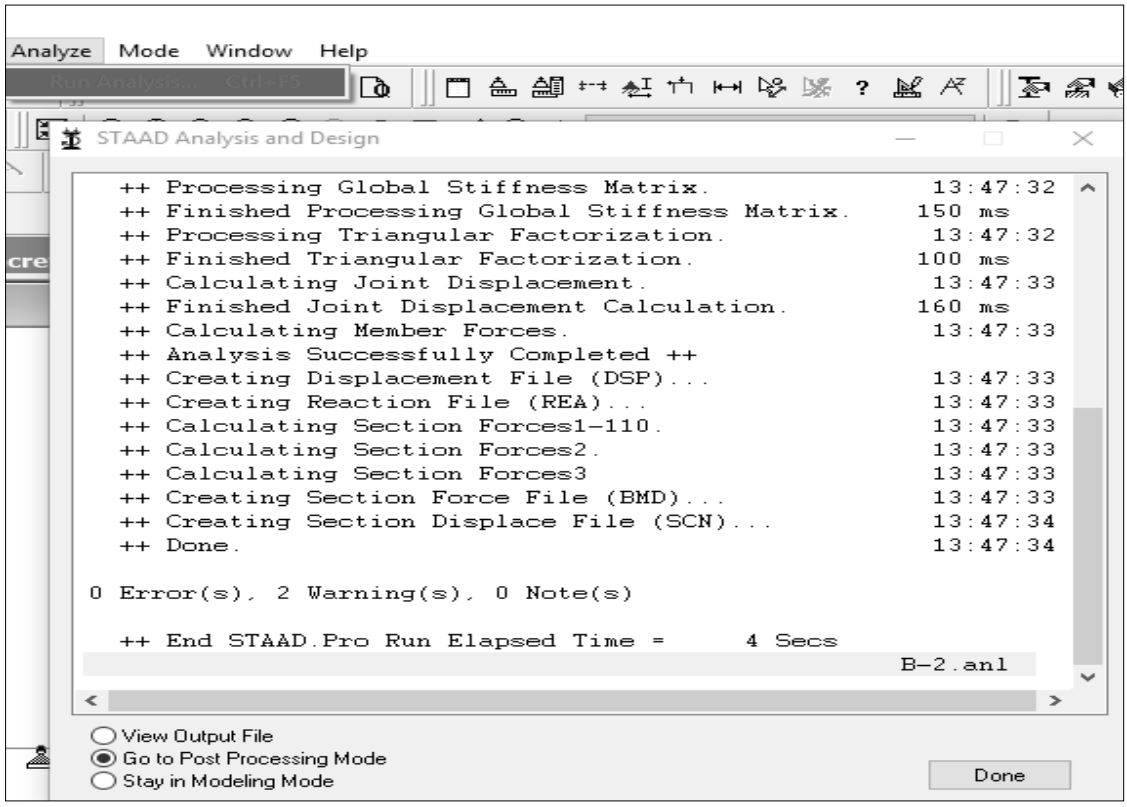


Fig: 4.13

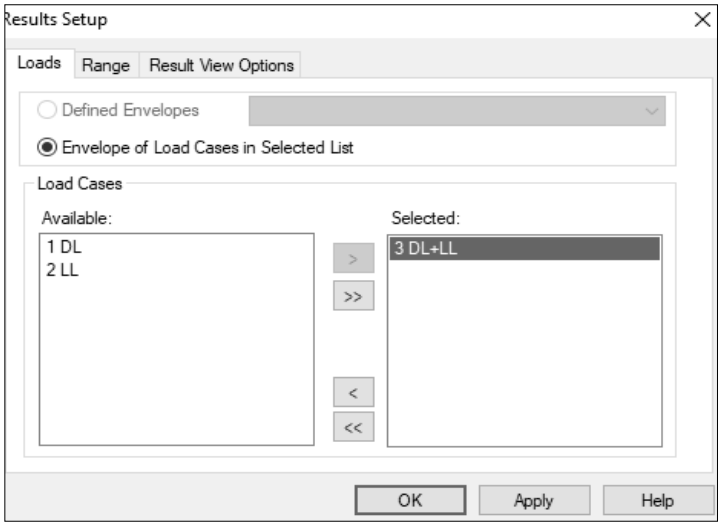
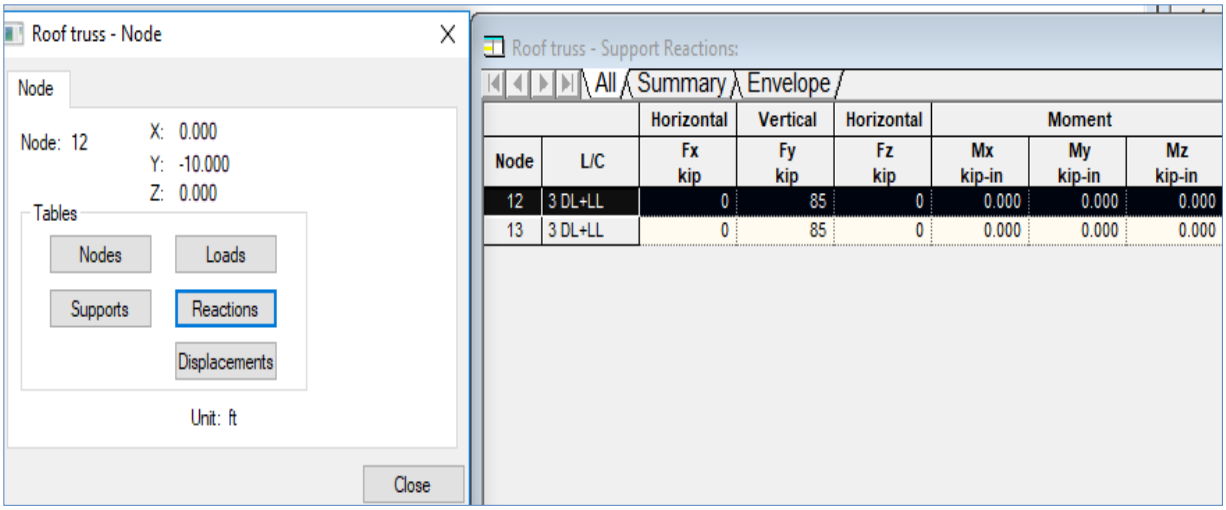


Fig: 4.14



Node	L/C	Horizontal Fx kip	Vertical Fy kip	Horizontal Fz kip	Moment Mx kip-in	My kip-in	Mz kip-in
12	3 DL+LL	0	85	0	0.000	0.000	0.000
13	3 DL+LL	0	85	0	0.000	0.000	0.000

Fig: 4.15

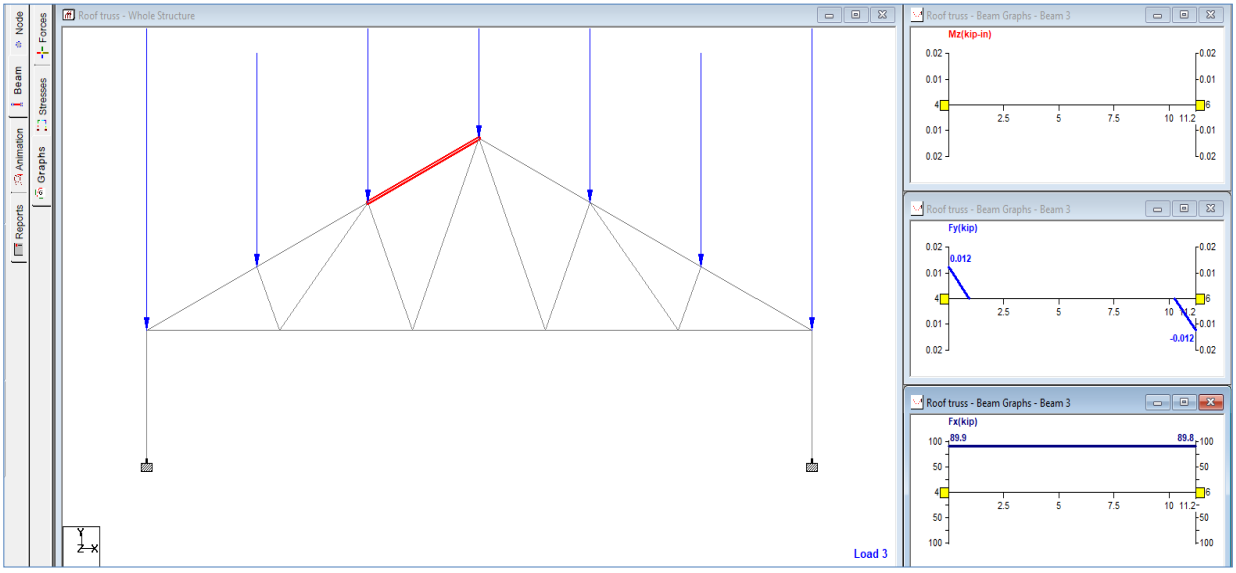


Fig: 4.16

B. Bridge Truss

Objective: Analyze the following Bridge Truss (Pratt Type) and find out the following values;

1. Support reactions
2. Forces for the members a, b, c

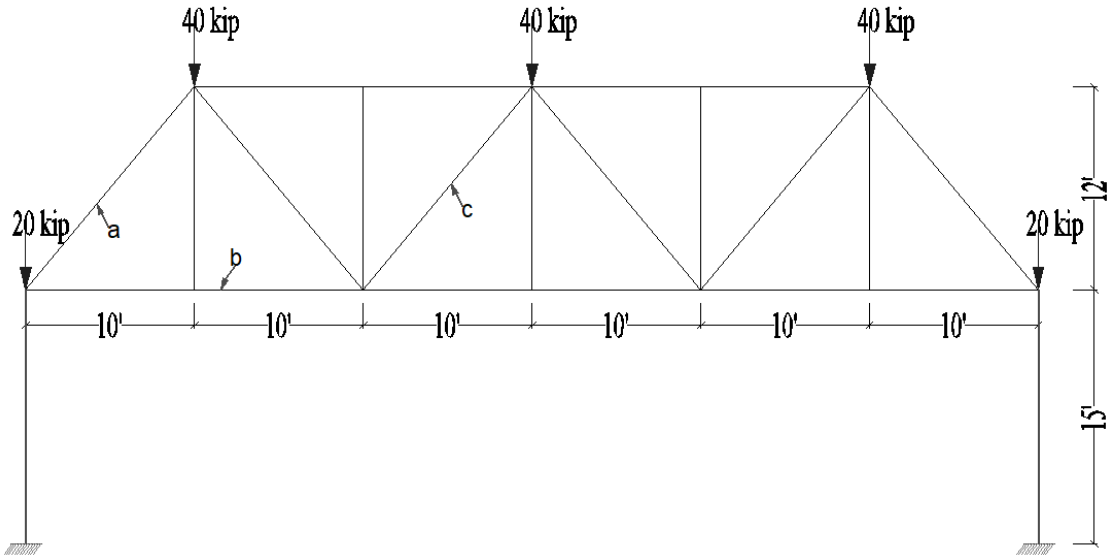


Fig. 4.17: Two Dimensional View of the Bridge Truss (Pratt Type)

Properties:

All members are Japanese I shape = I 150X75X6

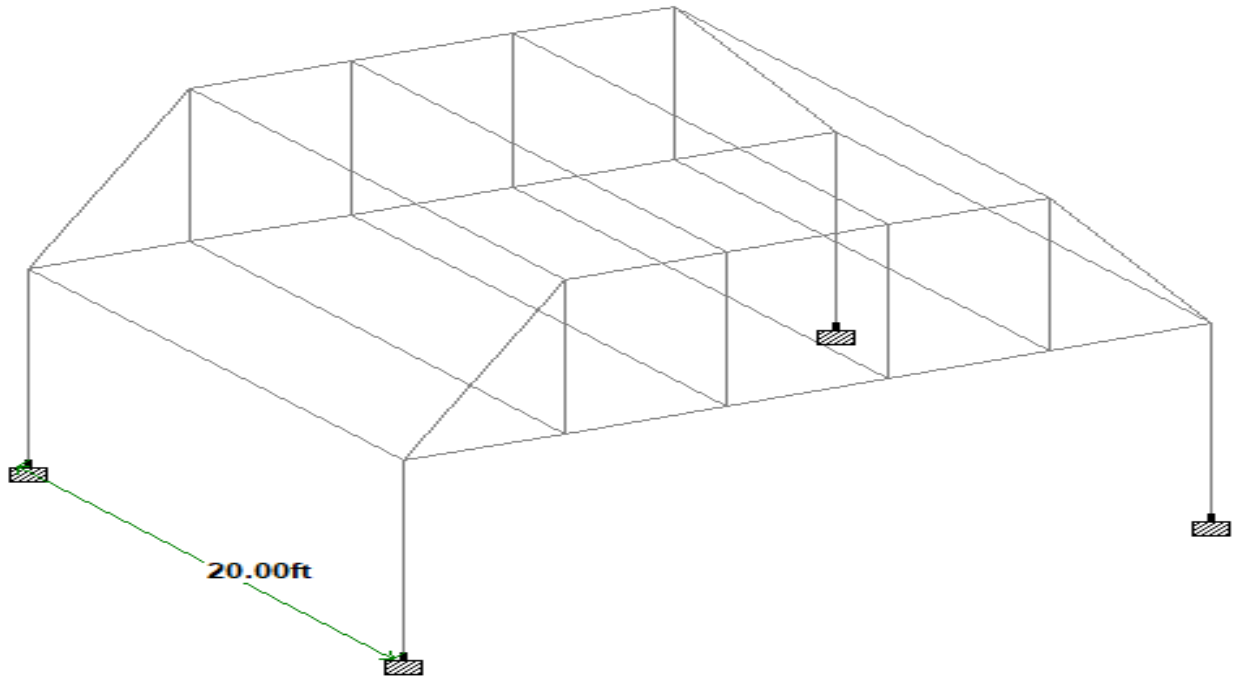


Fig. 4.18: Three Dimensional View of the Bridge Truss (Pratt Type)

Procedure:

1. Geometry (Model creating):

1.1 Open STAAD Pro. software →File → New → Click on Truss →Write the File Name and select Location→Length Units = Foot, Force Units = Kilo Pound → Next →Add Beam →Finish. (Fig: 4.19)

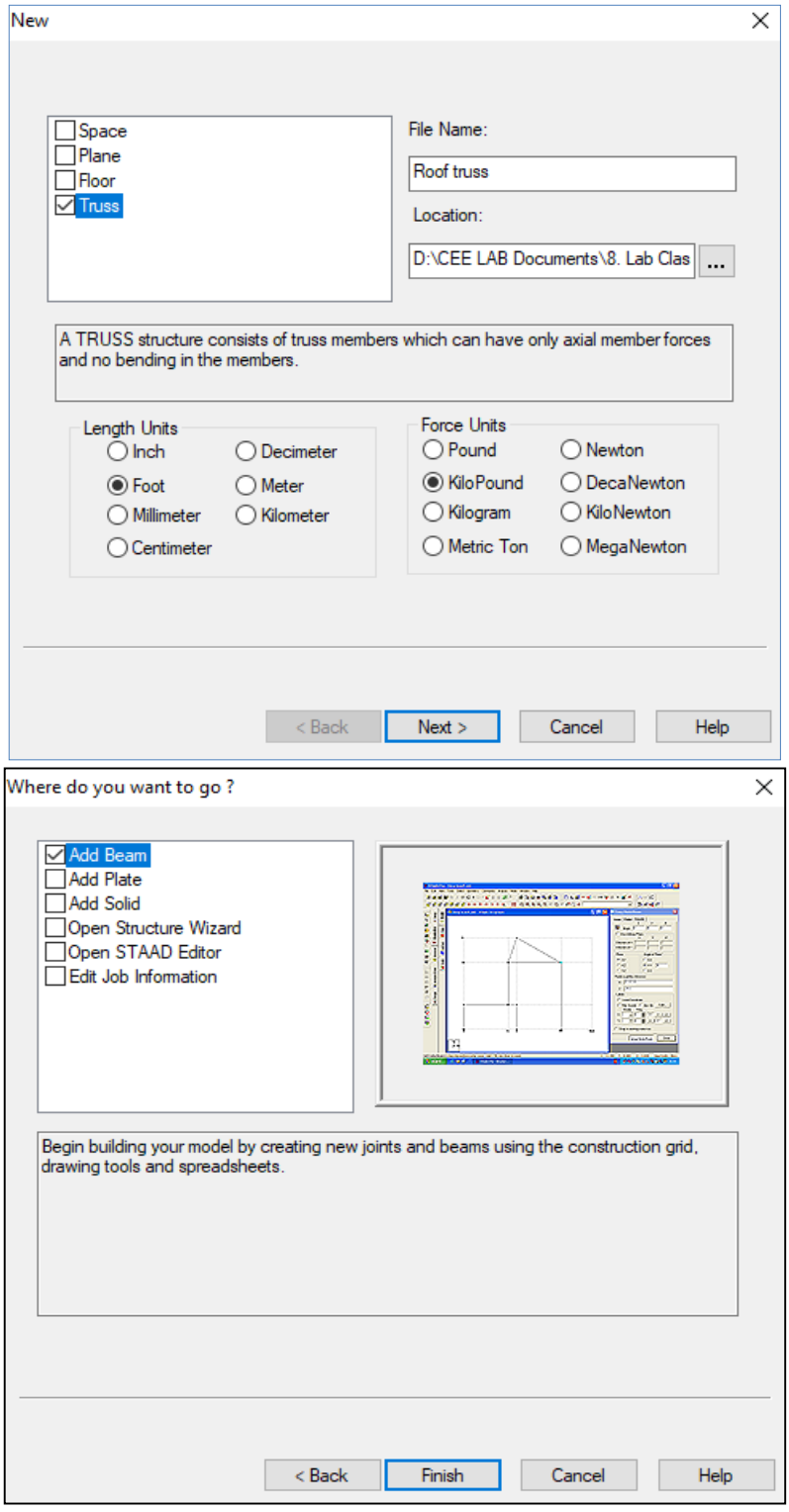


Fig: 4.19

1.2 Now close the Default Grid window → For 1st Node point input (X=0, Y=0,Z=0) → Select Node (in View window) by using node cursor →Translational Repeat →Global Direction =X →No. of Steps = 6 → Spacing=10 →Click on Link Steps →OK. Again select the members 2 to 5→Translational Repeat →Global Direction =Y →No. of Steps = 1 → Spacing=12 →Click on Link Steps →OK. (Fig: 4.20)

1.3 Add beam from point 1 to 7 and 10 to 6. (Fig: 4.21)

1.4 Now Select the Nodes 1 & 6 (in View window) by using node cursor →Translational Repeat → Global Direction =Y → No. of Steps = 1 → Spacing= -15 (-ve)→ Click on Link Steps → OK (Fig: 4.22). For 3D select the whole structure by using beam cursor →Translational Repeat → Global Direction =Z → No. of Steps = 1 → Spacing= 20→ Click on Link Steps → OK

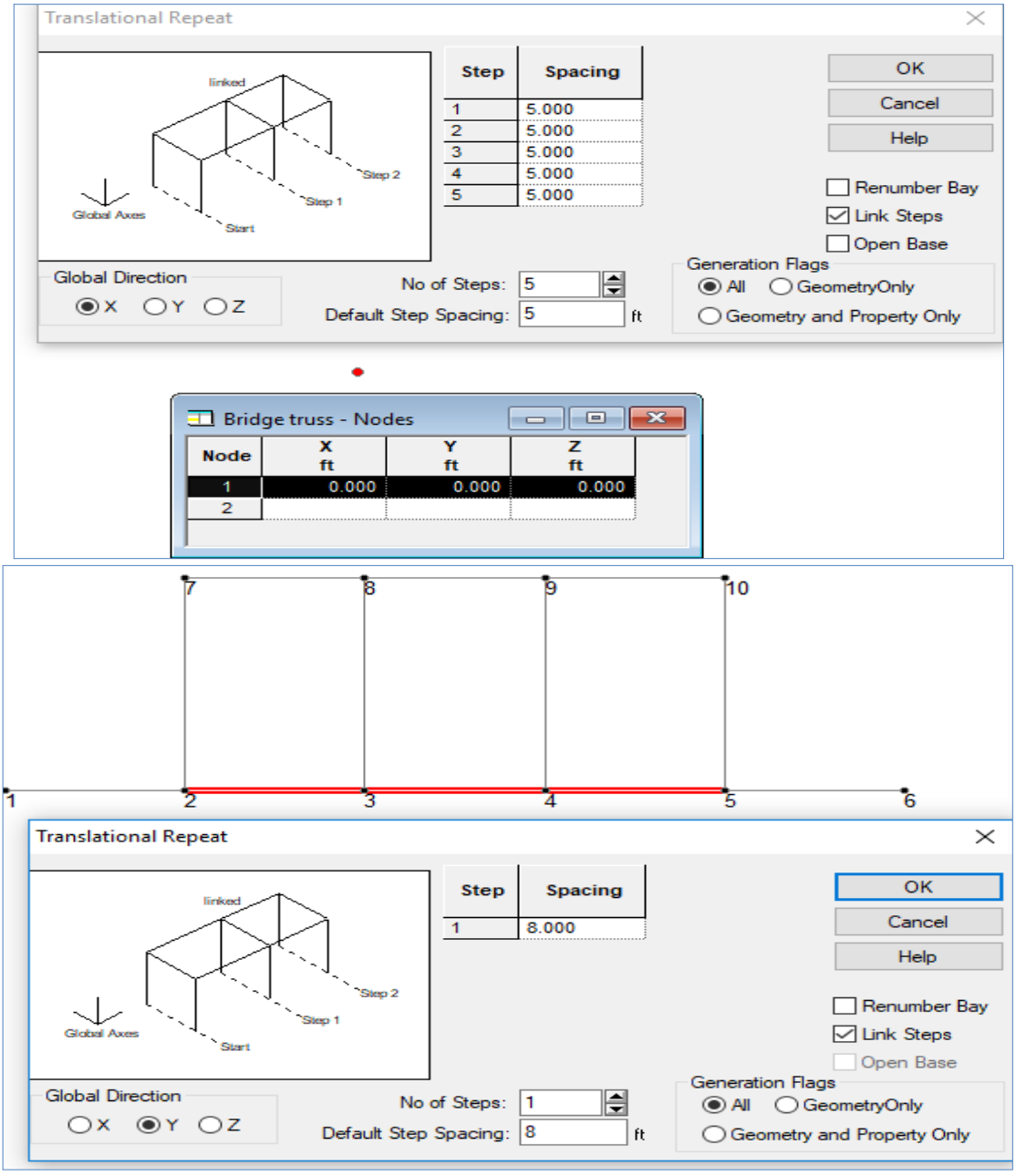


Fig: 4.20

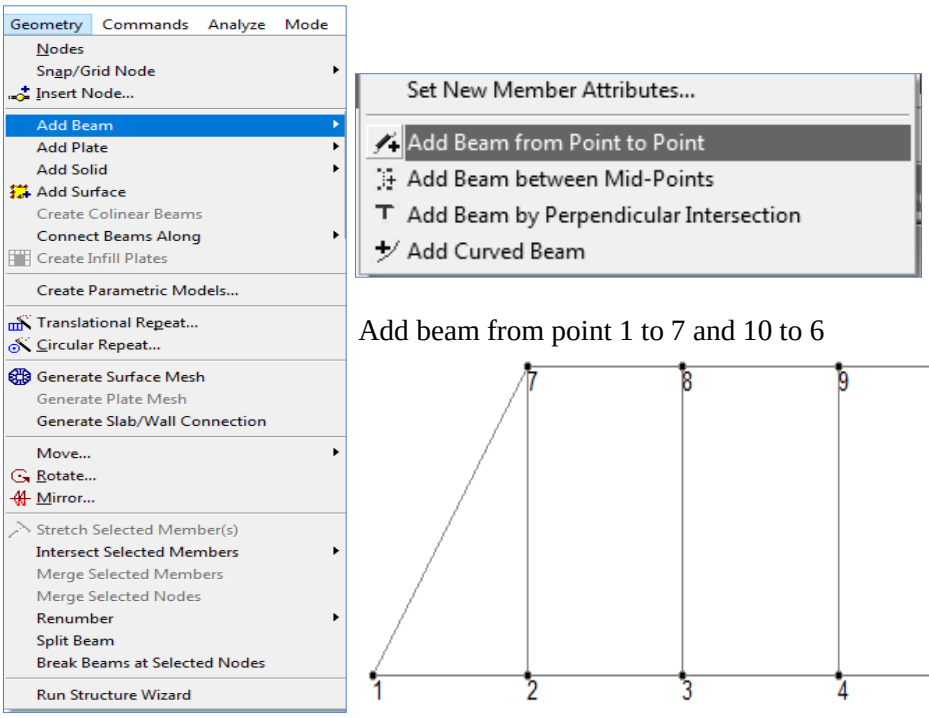


Fig: 4.21

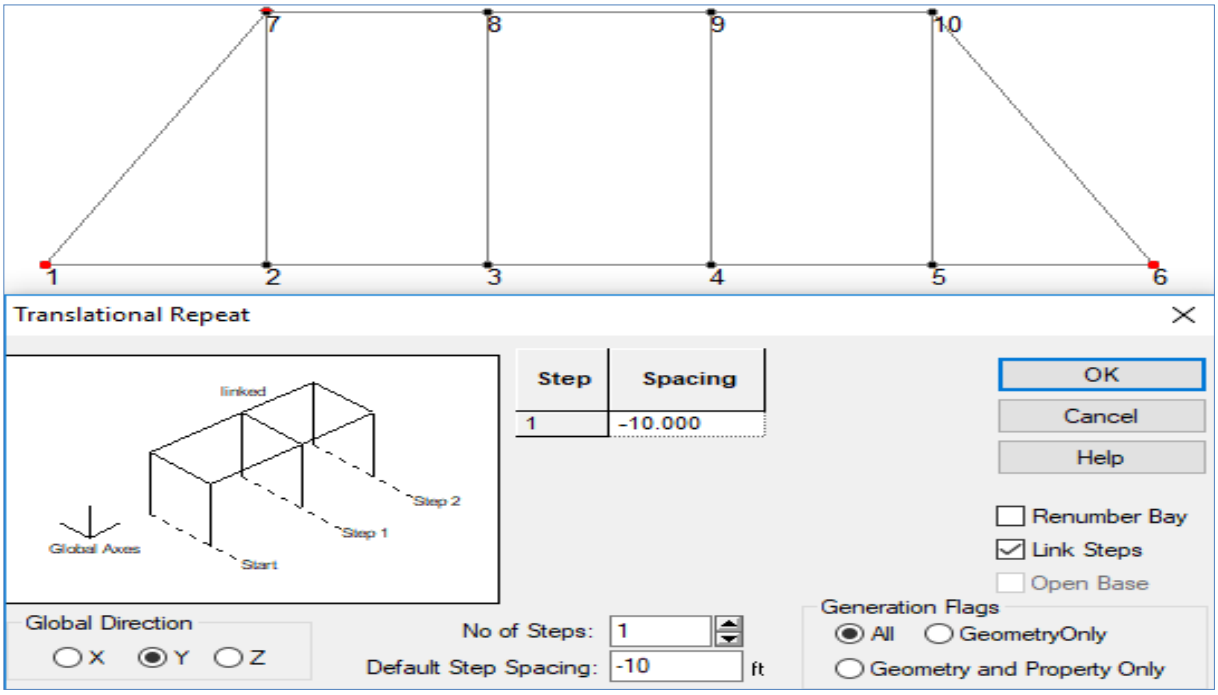


Fig: 4.22

2. General (Define & Assign):
   2.1. Property → Section Database →Japanese→ I Shape→ I 150X75X6 →Material = STEEL→ Add →Close then for Assign select the property and click on Assign to View →Assign → Yes. (Fig: 4.23)
   2.2. Support → Create →Fixed →Add → Now for Assign click on the Support type →Select the Support point in Truss →Assign to Selected Nodes →Assign →Yes.

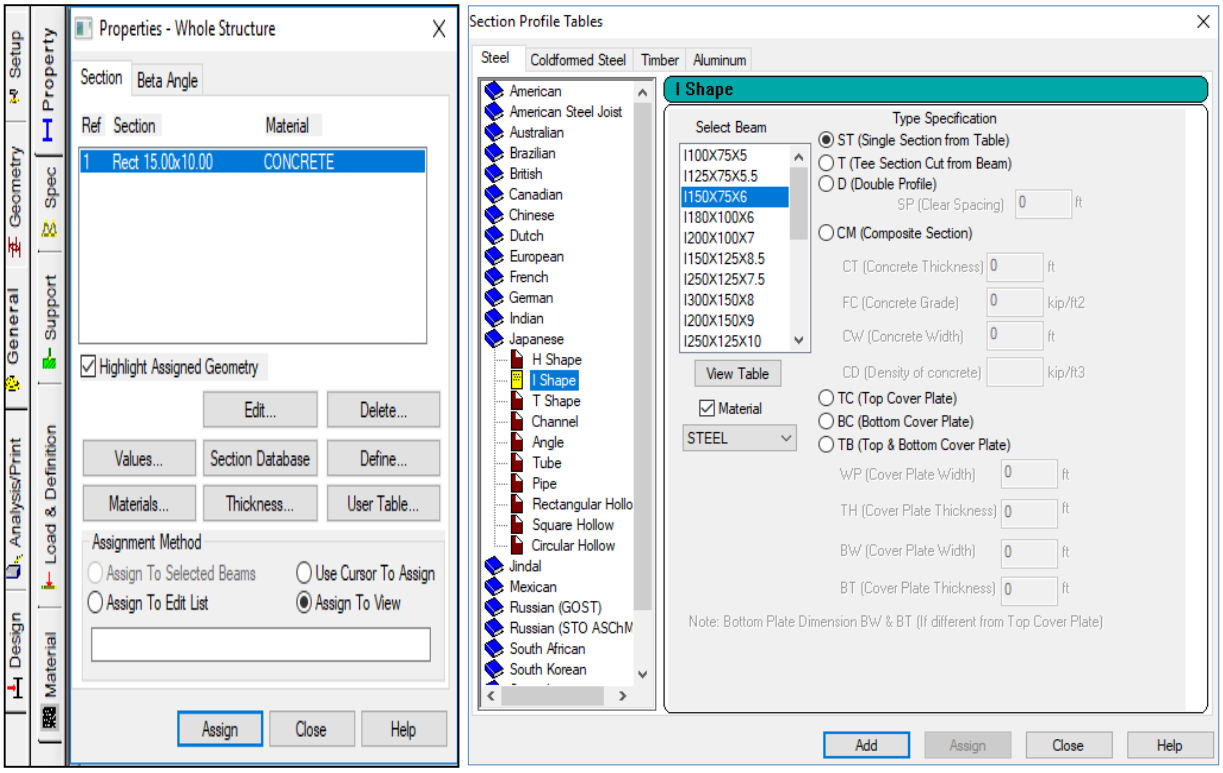


Fig: 4.23

2.3 Now Select the whole structure (in View window) by using beam cursor →Translational Repeat → Global Direction =Z → No. of Steps = 1 → Spacing= 20 → Click on Link Steps → OK and then delete extra beams (Fig: 4.24)

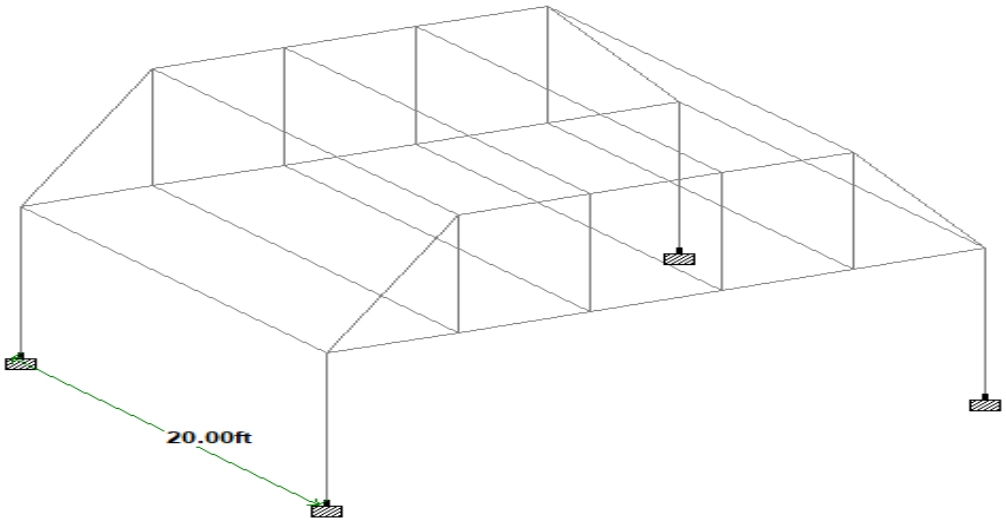


Fig: 4.24

2.4 Load & Definitions → Load Cases Details → Add → Loading Type = Dead →Title = Dead Load or DL → Add → Loading Type = Live →Title = Live Load or LL → Add → Close. (Fig: 4.25)

- DL →Add →Self weight →Direction = Y, Factor = -1→Add →Close. Then SELFWEIGHT Y-1 →Assign To View →Assign →Yes. (Fig: 4.26)

- For Given loads: Again Live Load or LL →Add →Nodal Load → Fy = -20 kip →Add → Fy = -40 kip→Add →Close, then click on defined force and select the required Nodes → Assign to selected Nodes →Assign →Yes. (Fig: 4.27)

2.5 Load Combination: Load Cases Details → Add →Define Combinations →Name = DL+LL →Default ai = 1, then select DL, LL and click on >> to send right side from left side. (Fig: 4.28)

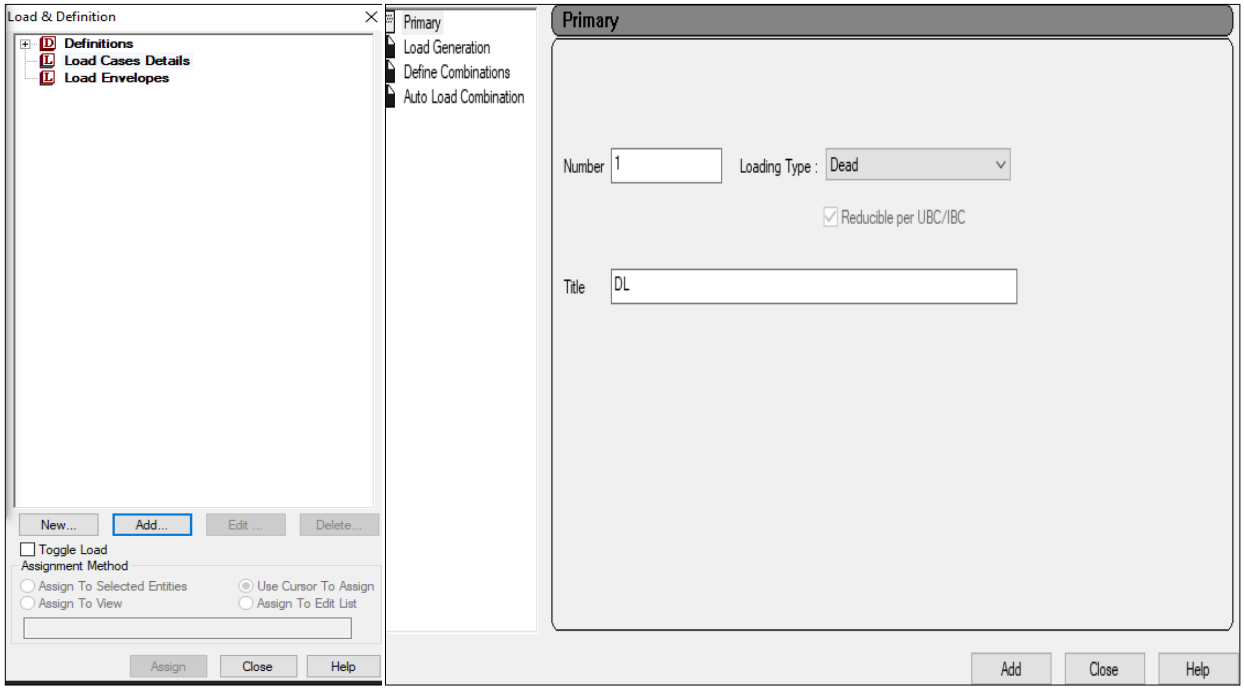


Fig: 4.25

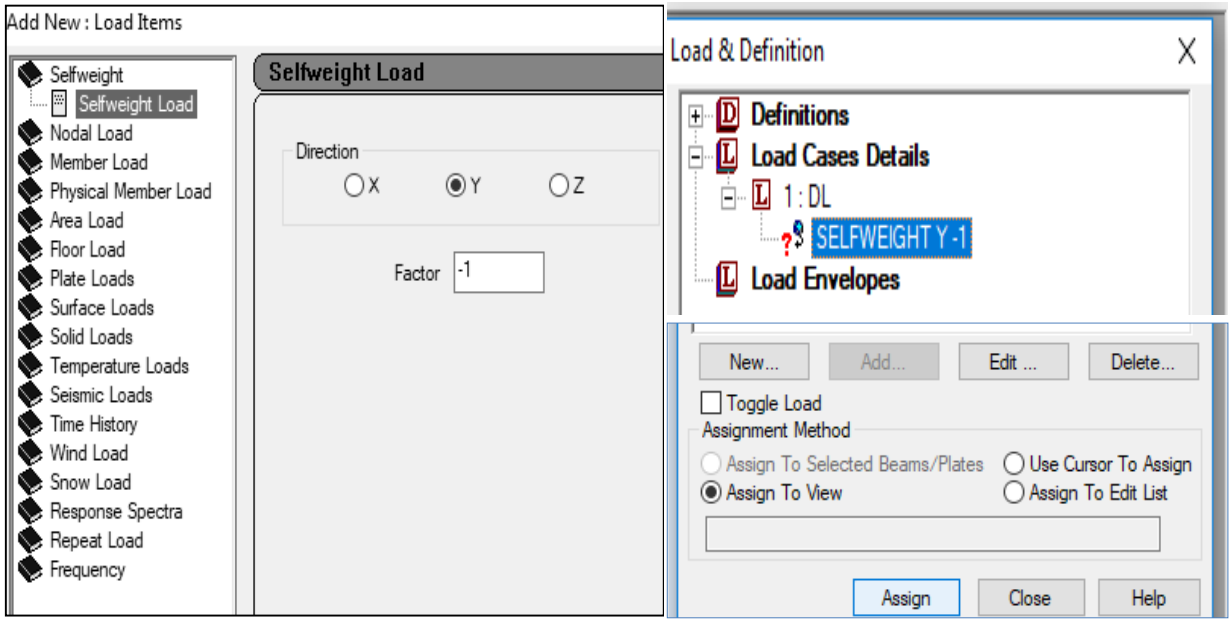


Fig: 4.26

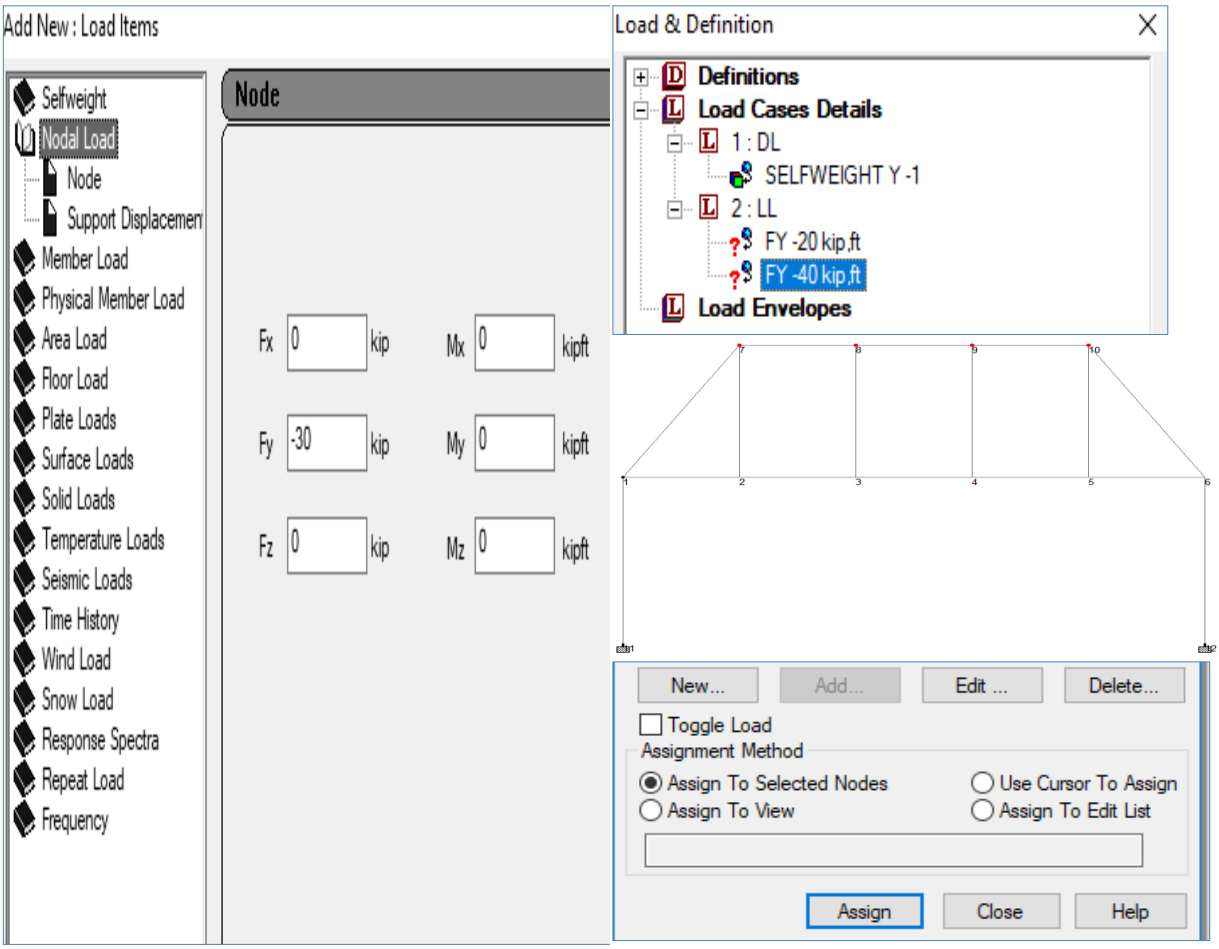


Fig: 4.27

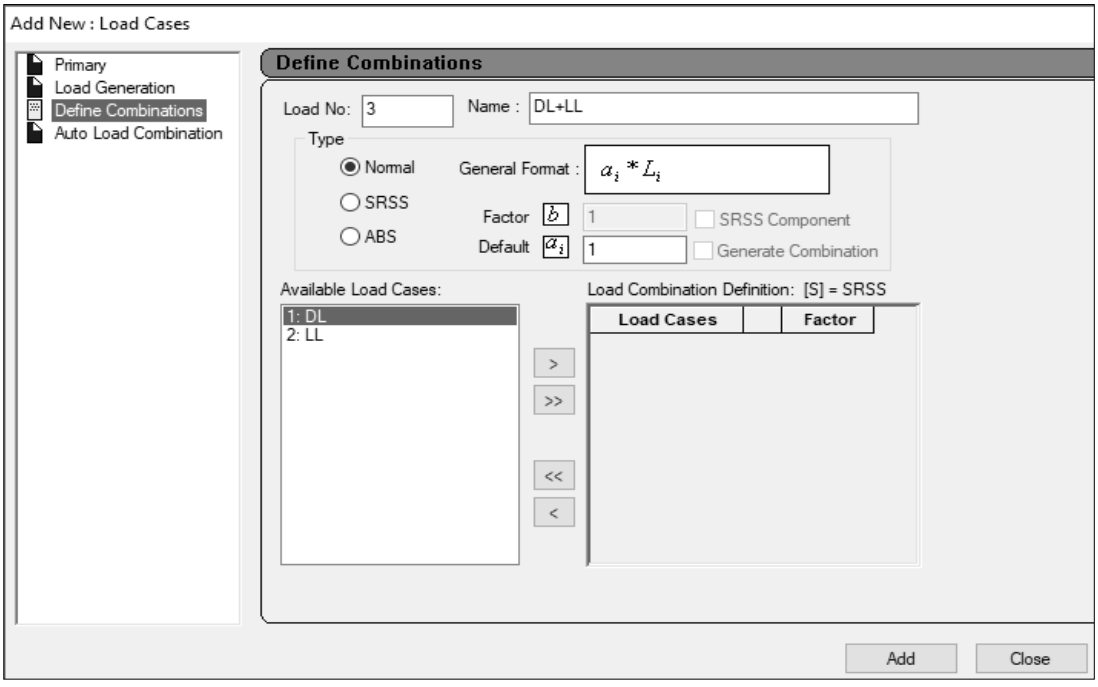


Fig: 4.28

3 Analysis and Result:

- From left side click on Analysis/Print →Static Check or All →Add →Close
- At Menu bar →Analyze →Run Analysis →Go to post processing mode →Done →Selected load cases = DL+LL →Apply →OK. (Fig: 4.29 & Fig: 4.30)
- For Support Reactions use node cursor and double click on the support point →Reactions. Then get the Table for all Support Reactions.
- For Beam Forces: From left side click on Beam →from Graphs find out Axial force by clicking on required Beam.

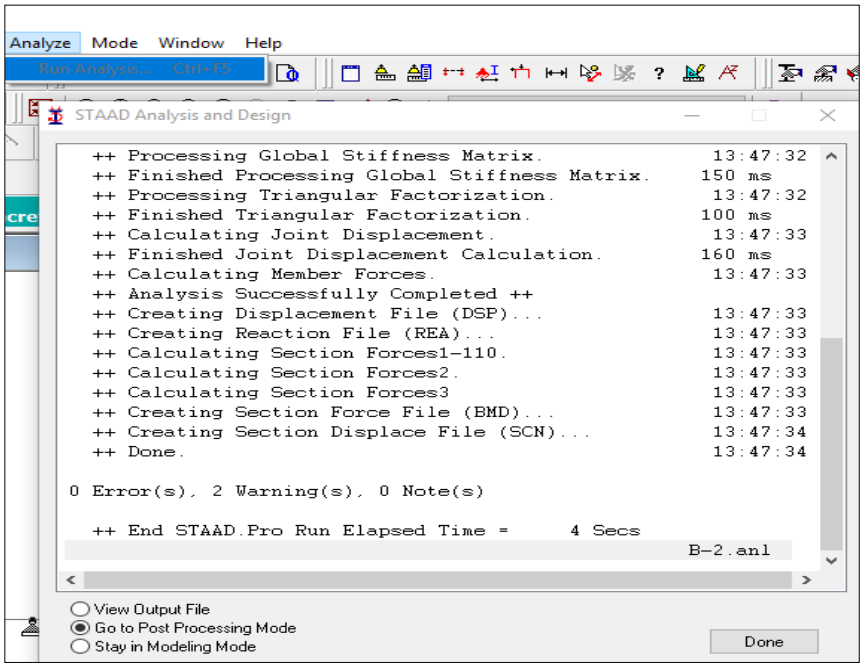


Fig: 4.29

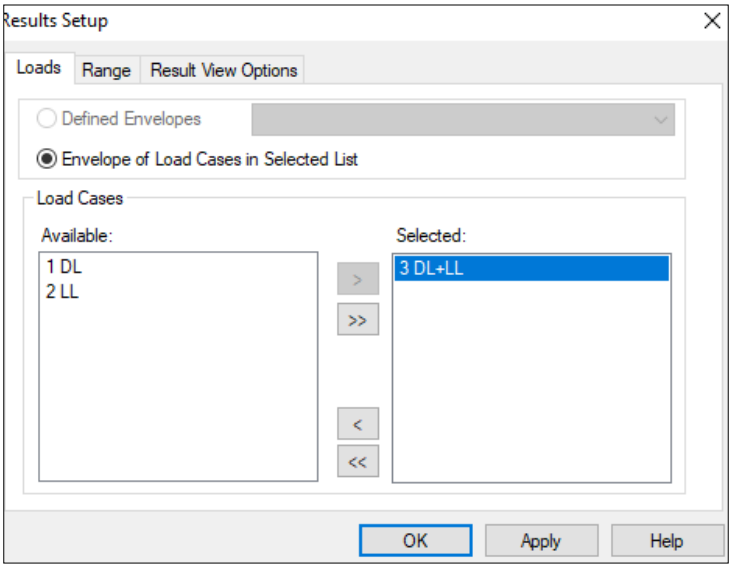


Fig: 4.30

CHAPTER-V

MULTI STORIED BUILDING FRAME UNDER ALLLOADS

Objective: Analyze the following 7-Storied residential building under all loads and Columns, Beams forces and support reactions for foundation design.

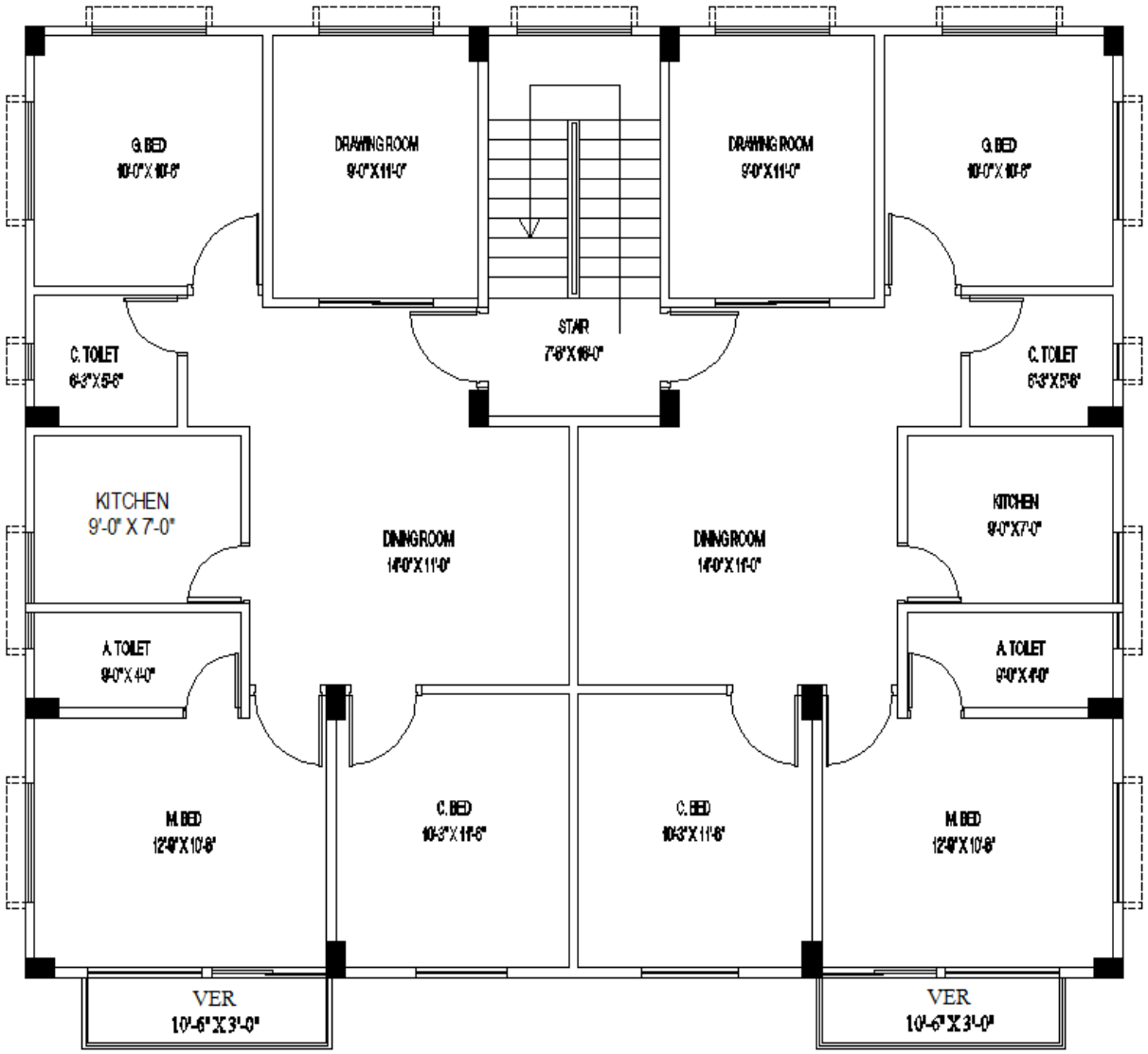


Fig: 5.1 Typical Floor Plan

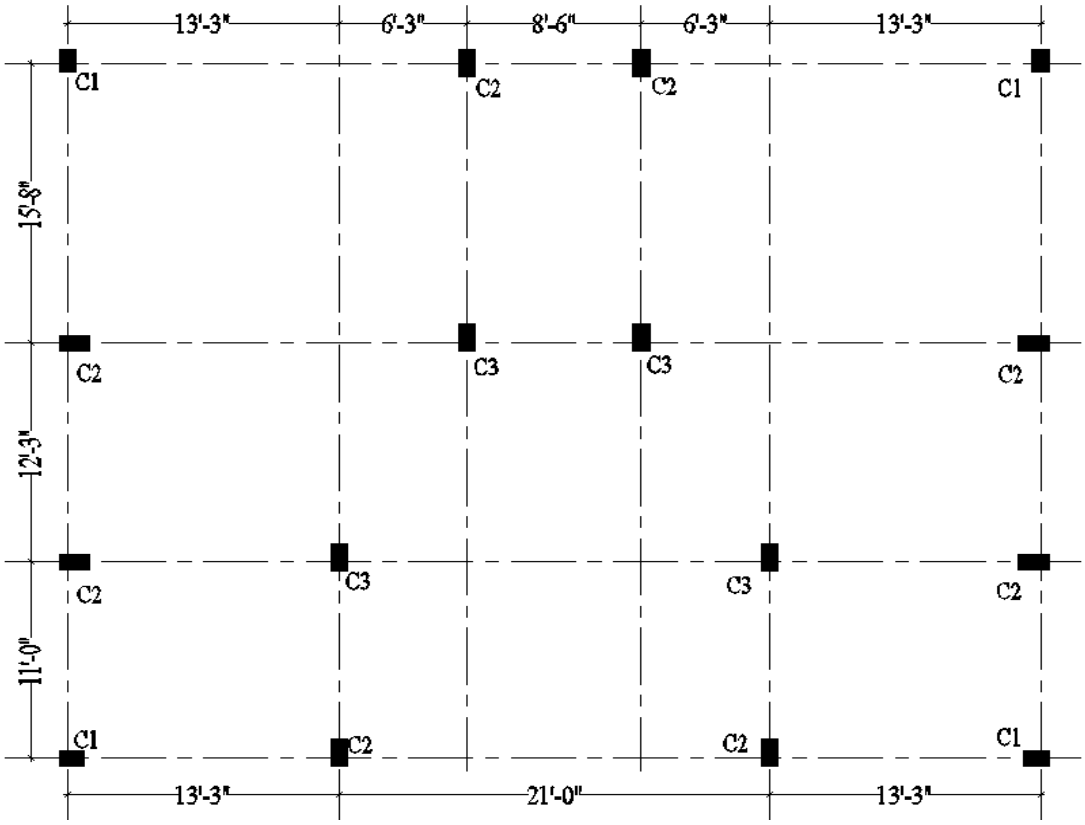


Fig: 5.2 Column LayoutPlan

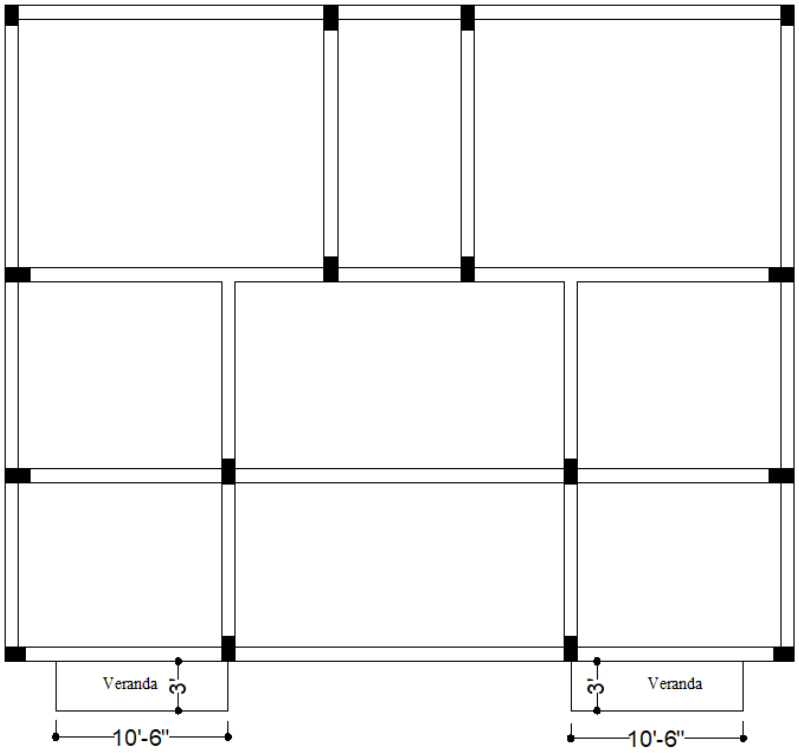


Fig: 5.3 Beam LayoutPlan

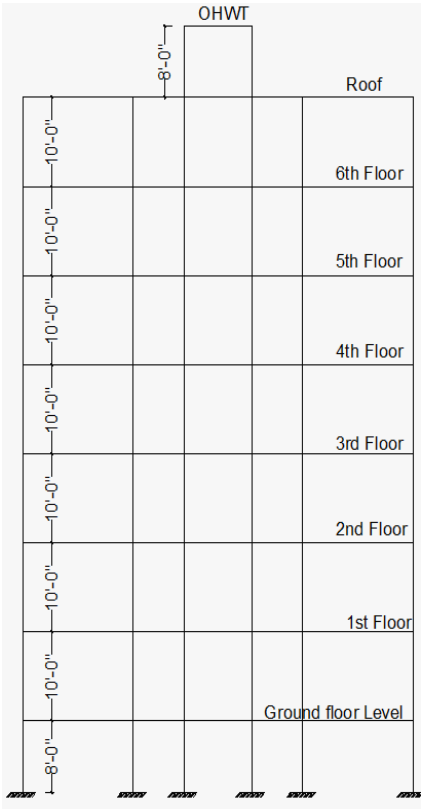


Fig: 5.4 Elevation View

Table 5.1 Geometry and Loads:

Properties:	Load Definitions:
Column: C1= 12”X15” C2=12”X18” C3=12”X20” Beam: GB = 10”X18” FB = 10”X20” Varenda Beam: 6” X 6” • All supports are fixed support • Bottom story height = 8’-0” • Typical story height = 10’-0” • Top story for lift & stair = 8’-0”	1. Seismic Definition: (Dhaka zone) EQx & EQz 2. Wind Definitions: (for Dhaka) Wx & Wz *Wind speed for Dhaka zone = 210 km/hr = 130 mile/hr 3. Dead Load: Self weight (Factor=1) Slab weight = 75 psf (For slab thickness= 6”) Floor Finish (FF) = 30 psf Partition wall load (PW) = 25 psf Wall load on beams (W) = 416 Ib/ft (for 5” brick wall) 4. Live Load: LL = 40 psf
	Load Combinations:
	UFL = DL+LL FDL = 1.2*DL+1.6*LL FDLEQx = 0.9*DL+1.2*LL+1.32*EQx FDLEQz = 0.9*DL+1.2*LL+1.32*EQz FDLWx = 0.9*DL+1.2*LL+1.2*Wx FDLWz = 0.9*DL+1.2*LL+1.2*Wz

Procedure:

1. Geometry (Model creating):

1.1 Open the STAAD Pro. software and click on New Project →Space →File name →Location (select your file location to save) →Length unit select Foot and KiloPound →Next →Add Beam →Finish.

1.2 Column & Beam Layout:

1.2.1 Close the default Grid system and at the right side input your first node point coordinates as (X Y Z)=(0 0 0) (Figure: 5.2). After than select the node by using node cursor →Geometry →Translational Repeat →Select Global Direction = X → No of Steps = 3 → now write down the column spacing from your Column Layout Plan as (Step1 =19.5, Step2 =8.5, Step3 =19.5) →Link Steps →OK. (Figure: 5.2)

1.2.2 After than select the total beam by using Beam cursor →Geometry →Translational Repeat →Select Global Direction = Z → No of Steps = 1 → now write down the column spacing from your Column Layout Plan as (Step1 =15.667 ft) →Link Steps →OK. (Figure: 5.3)

1.2.3 Now select the column node no 5 & 8 from the 2nd row →Geometry →Translational Repeat →Select Global Direction = Z → No of Steps =2 → now write down the column spacing from your Column Layout Plan as (Step1 =12.25, Step2 =11) →OK (Figure: 5.4). Again select the column node no 9 & 10 by using node cursor →Geometry →Translational Repeat →Select Global Direction = X → No of Steps =3 → now write down the column spacing from your Column Layout Plan as (Step1 =13.25, Step2 =21, Step1 =13.25) →OK (Figure: 5.5)

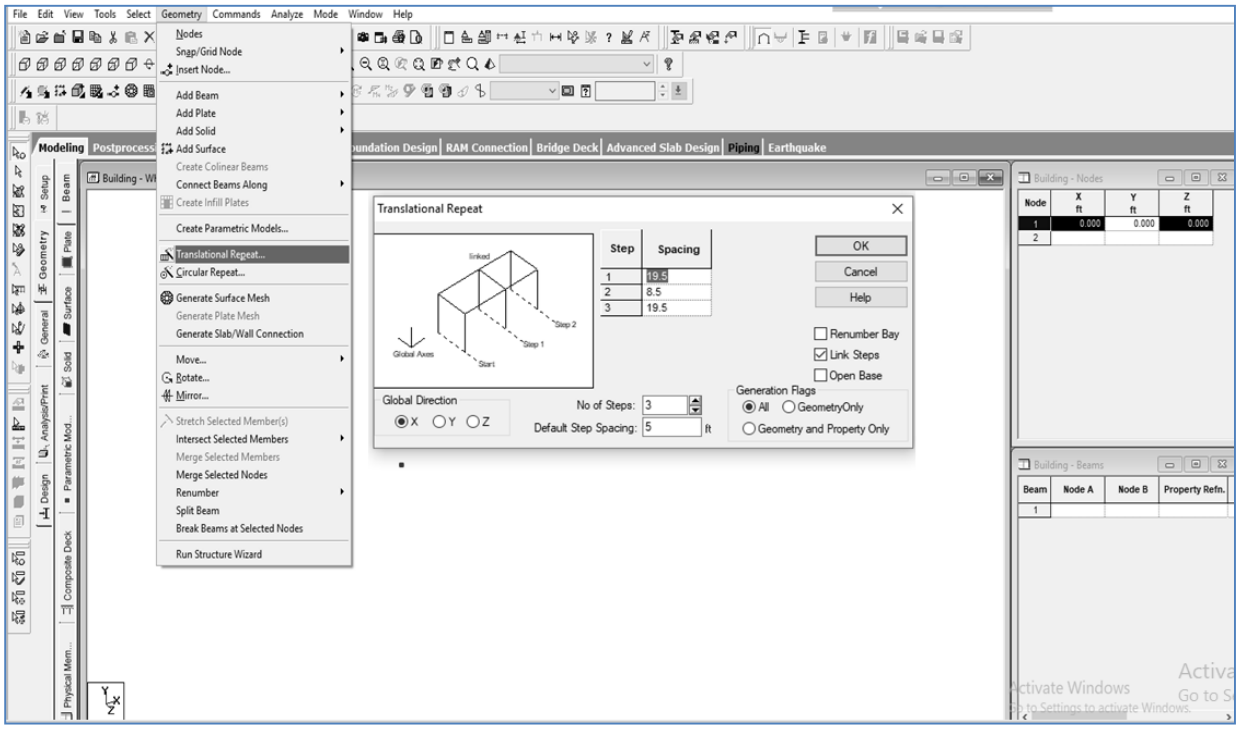

File Edit View Tools Select Geometry Commands Analyze Mode Window Help
Nodes
Snap/Grid Node
Insert Node...
Add Beam
Add Plate
Add Solid
Add Surface
Create Colinear Beams
Connect Beams Along
Create Infill Plates
Create Parametric Models...
Translational Repeat...
Circular Repeat...
Generate Surface Mesh
Generate Plate Mesh
Generate Slab/Wall Connection
Move...
Rotate...
Mirror...
Stretch Selected Member(s)
Intersect Selected Members
Merge Selected Members
Merge Selected Nodes
Renumber
Split Beam
Break Beams at Selected Nodes
Run Structure Wizard
Translational Repeat
Step
Spacing
1 19.5
2 8.5
3 19.5
OK
Cancel
Help
Renumber Bay
Link Steps
Open Base
Global Direction
X Y Z
No of Steps: 3
Default Step Spacing: 5 ft
Generation Flags
All GeometryOnly
Geometry and Property Only
Building - Nodes
Building - Beams

Fig: 5.2

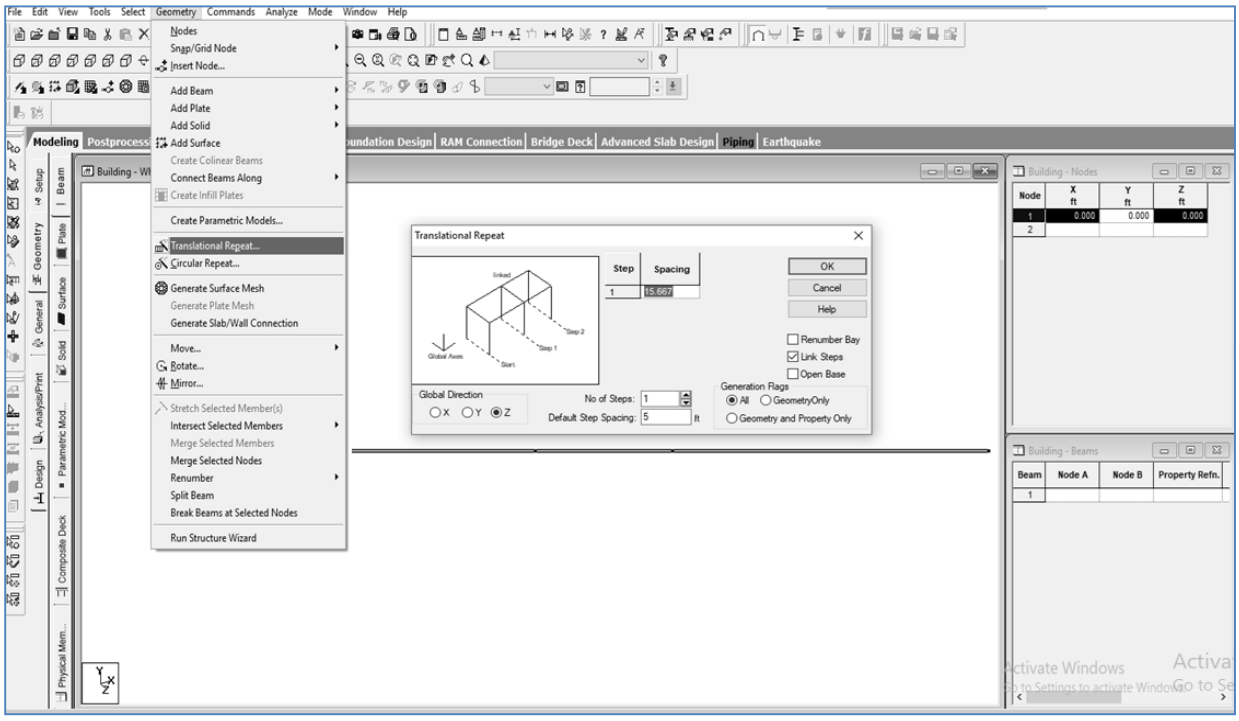

File Edit View Tools Select Geometry Commands Analyze Mode Window Help
Translational Repeat...
Circular Repeat...
Translational Repeat
Step
Spacing
1 15.667
OK
Cancel
Help
Renumber Bay
Link Steps
Open Base
Global Direction
X Y Z
No of Steps: 1
Default Step Spacing: 5 ft
Generation Flags
All GeometryOnly
Geometry and Property Only
Building - Nodes
Building - Beams

Fig: 5.3

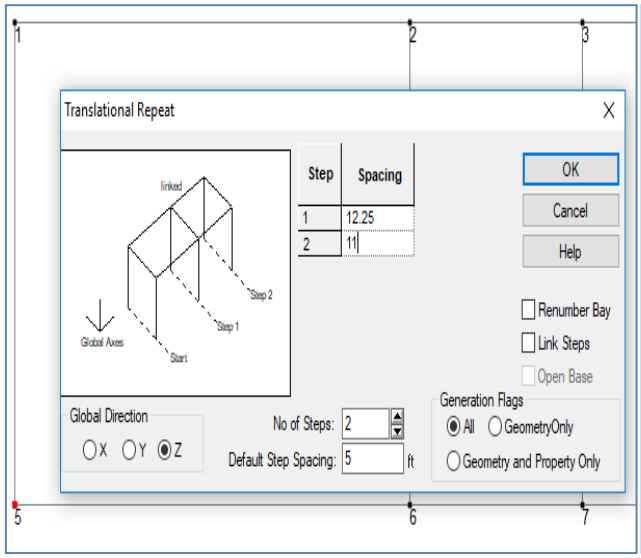


Fig: 5.4

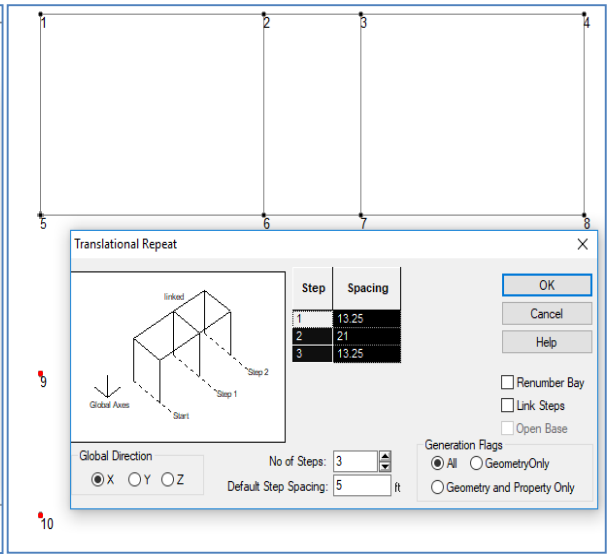


Fig: 5.5

1.2.4 For create beam layout: Go to Geometry →Add Beam →Add Beam from Point to Point and then connect the nodes points each other as the given Beam layout plan. Again for interesting beam go to Geometry →Add Beam →Add Beam by Perpendicular Intersection. (Figure: 5.6 and 5.7)

1.2.5 Veranda Create: Now select the node no 12 →Geometry →Translational Repeat →Select Global Direction = X→ No of Steps =1 → now write down the Veranda length as (Step1 = -10.5) →OK (Figure: 5.8).

1.2.6 Again select the beam from node no 12 to 19 by using beams cursor →Geometry →Translational Repeat →Select Global Direction = Z → No of Steps =1 → now write down the now write down the Veranda width as (Step1 =3) →Click on Link Steps →OK (Figure: 5.9). And similarly create another Veranda.

1.2.7 Now Story Create: Select whole structure →Geometry →Translational Repeat →Select Global Direction = Y→ No of Steps =8(7 stoy+1 is bottom story for base)→ Default Step Spacing = 10(Typical story height) and Step1 = 8) →Click on Link Steps →OK (Figure: 5.10).

1.2.8 Then delete the unnecessary columns from veranda and inside of the plan by selecting them (use Delete button from keyboard).

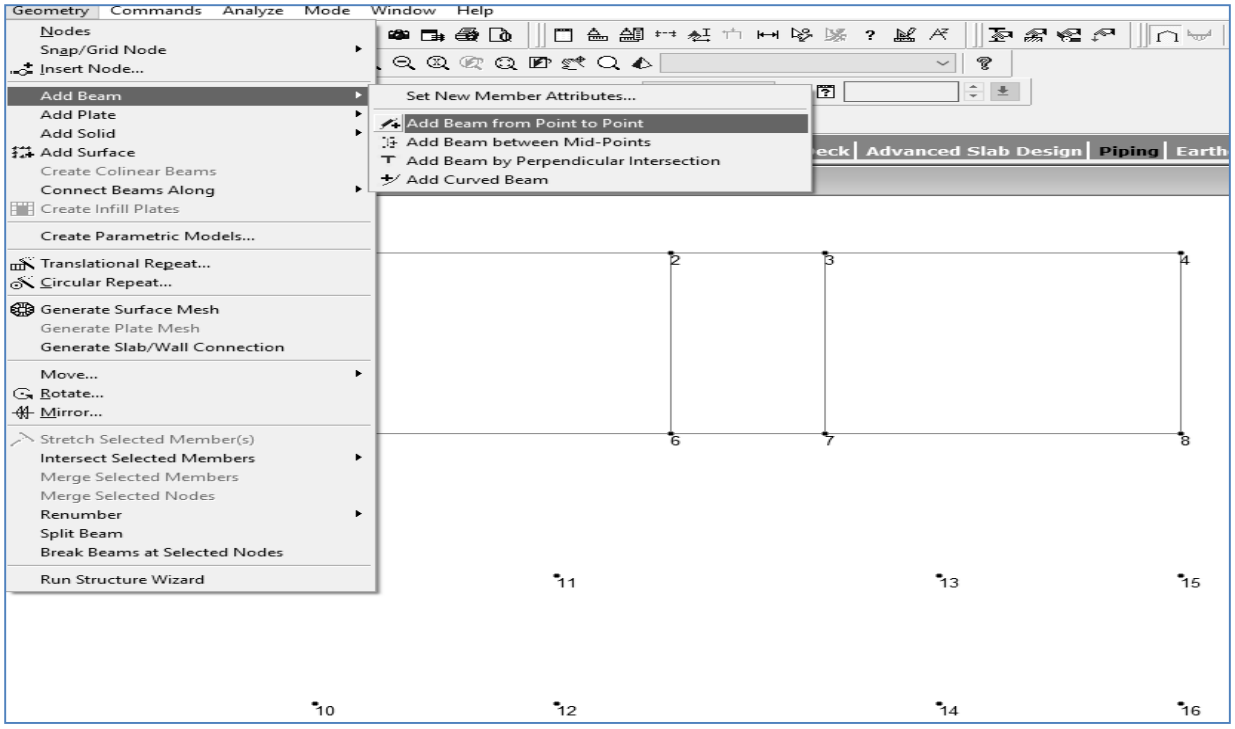

Geometry
Commands
Analyze
Mode
Window
Help
Nodes
Snap/Grid Node
Insert Node...
Add Beam
Add Plate
Add Solid
Add Surface
Create Colinear Beams
Connect Beams Along
Create Infill Plates
Create Parametric Models...
Translational Repeat...
Circular Repeat...
Generate Surface Mesh
Generate Plate Mesh
Generate Slab/Wall Connection
Move...
Rotate...
Mirror...
Stretch Selected Member(s)
Intersect Selected Members
Merge Selected Members
Merge Selected Nodes
Renumber
Split Beam
Break Beams at Selected Nodes
Run Structure Wizard
Set New Member Attributes...
Add Beam from Point to Point
Add Beam between Mid-Points
Add Beam by Perpendicular Intersection
Add Curved Beam
Advanced Slab Design
Piping
Earth
2
3
4
6
7
8
11
13
15
10
12
14
16

Fig: 5.6

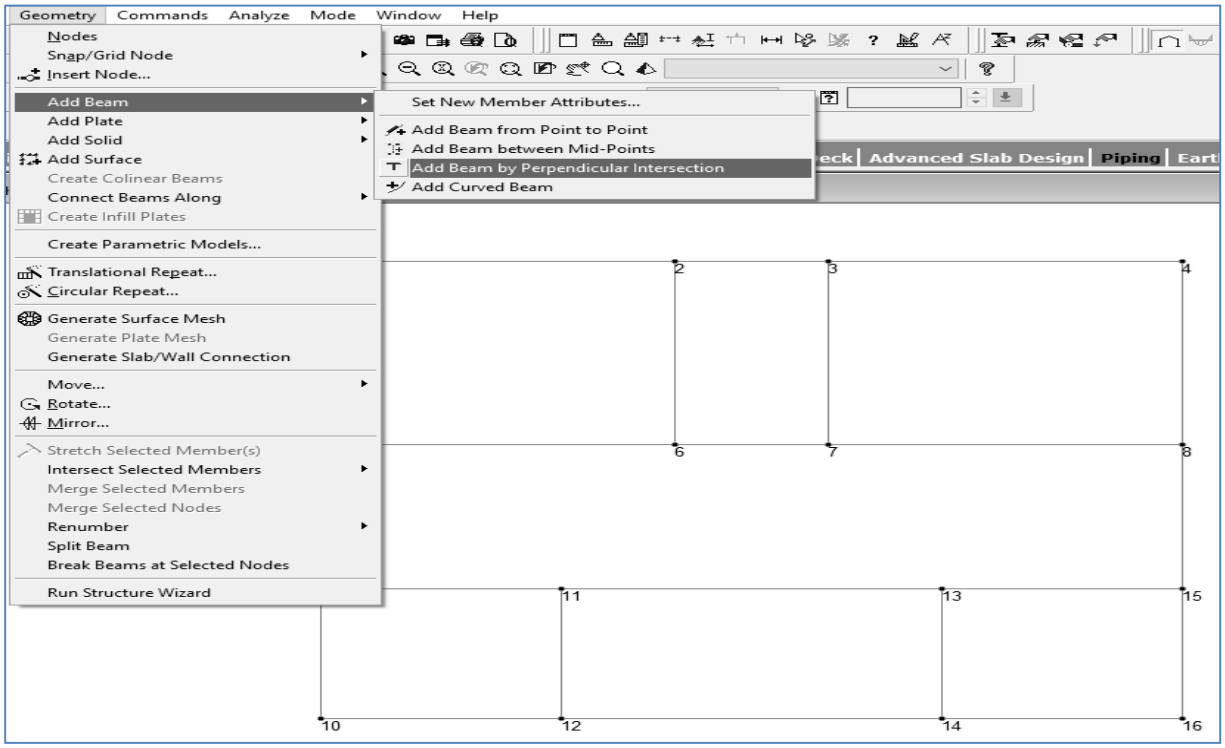

Geometry
Commands
Analyze
Mode
Window
Help
Nodes
Snap/Grid Node
Insert Node...
Add Beam
Add Plate
Add Solid
Add Surface
Create Colinear Beams
Connect Beams Along
Create Infill Plates
Create Parametric Models...
Translational Repeat...
Circular Repeat...
Generate Surface Mesh
Generate Plate Mesh
Generate Slab/Wall Connection
Move...
Rotate...
Mirror...
Stretch Selected Member(s)
Intersect Selected Members
Merge Selected Members
Merge Selected Nodes
Renumber
Split Beam
Break Beams at Selected Nodes
Run Structure Wizard
Set New Member Attributes...
Add Beam from Point to Point
Add Beam between Mid-Points
Add Beam by Perpendicular Intersection
Add Curved Beam
Advanced Slab Design
Piping
2
3
4
6
7
8
11
13
15
10
12
14
16

Fig: 5.7

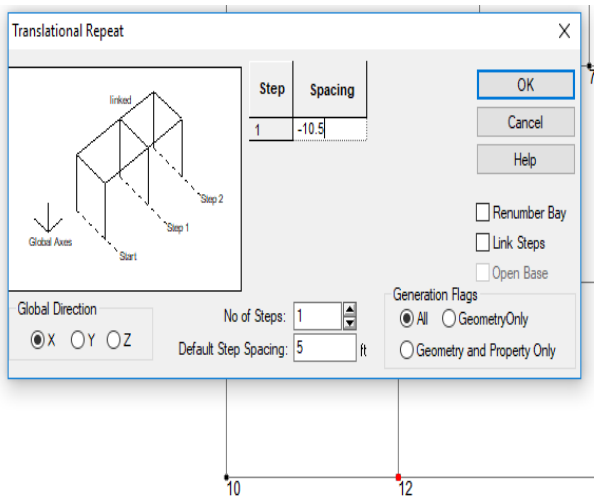


Fig: 5.8

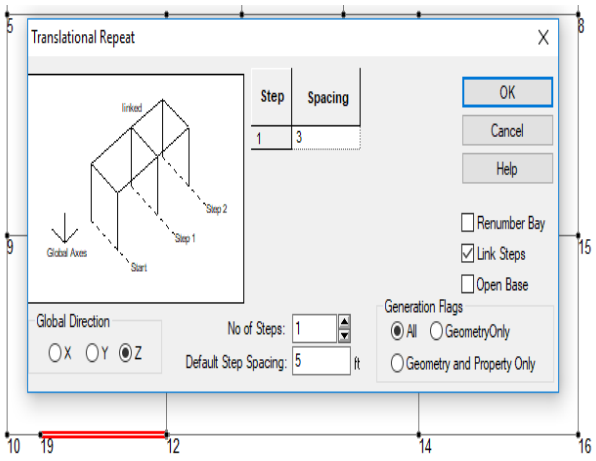


Fig: 5.9

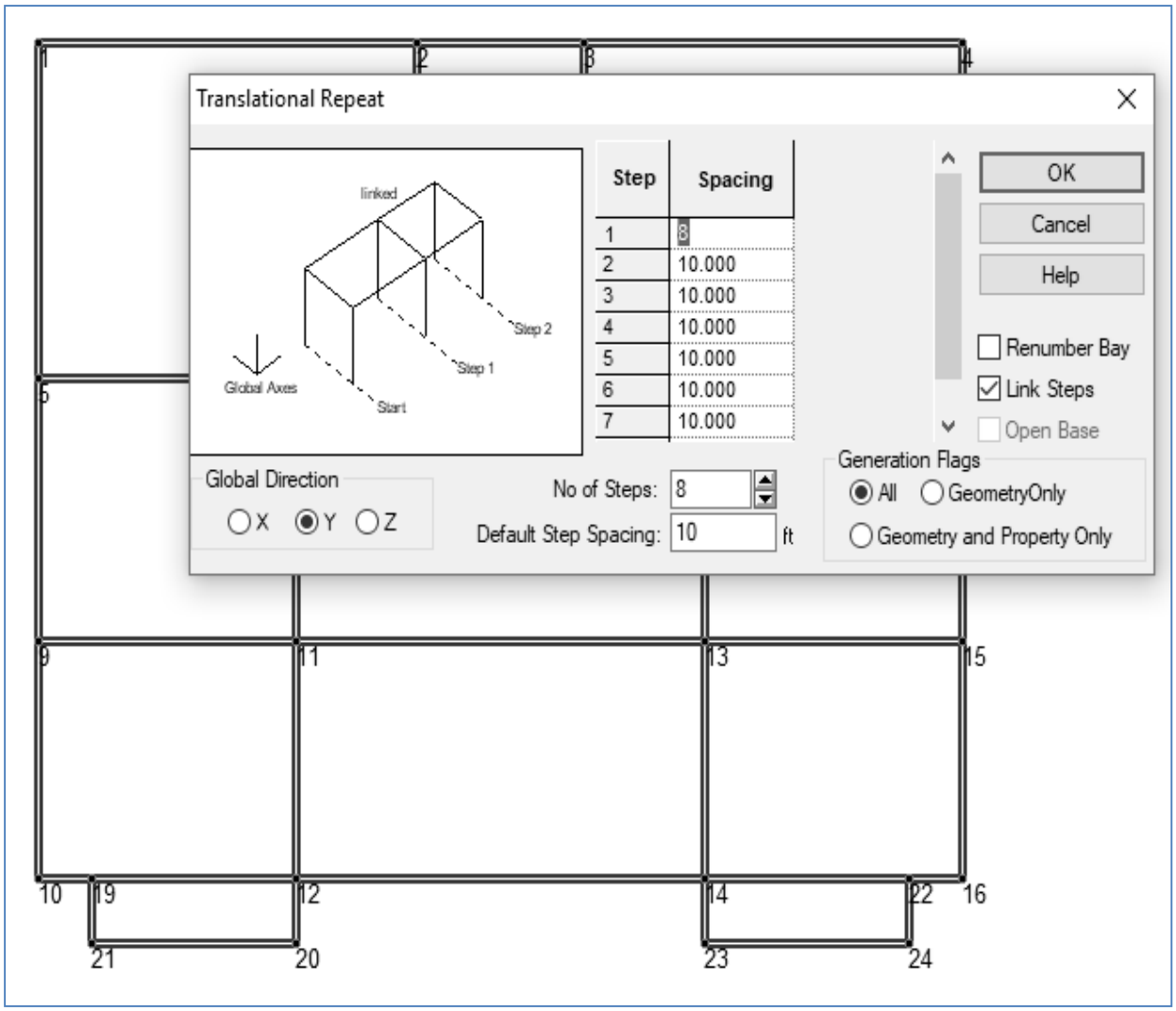


Fig: 5.10

2 General (Define & Assign):

2.1 Support Create and Assign:

2.1.1 Click on View from +Z →Select the all bottom Story beams → Delete →Ok →Yes

2.1.2 From left side Click on General →Support →Create → Fixed →Add. (Fig: 5.11)

2.1.3 Select S2 Support 2 →Select all bottom Nodes by using Node Cursor →Assign to Selected Nodes →Assign →Yes. (Fig: 5.12)

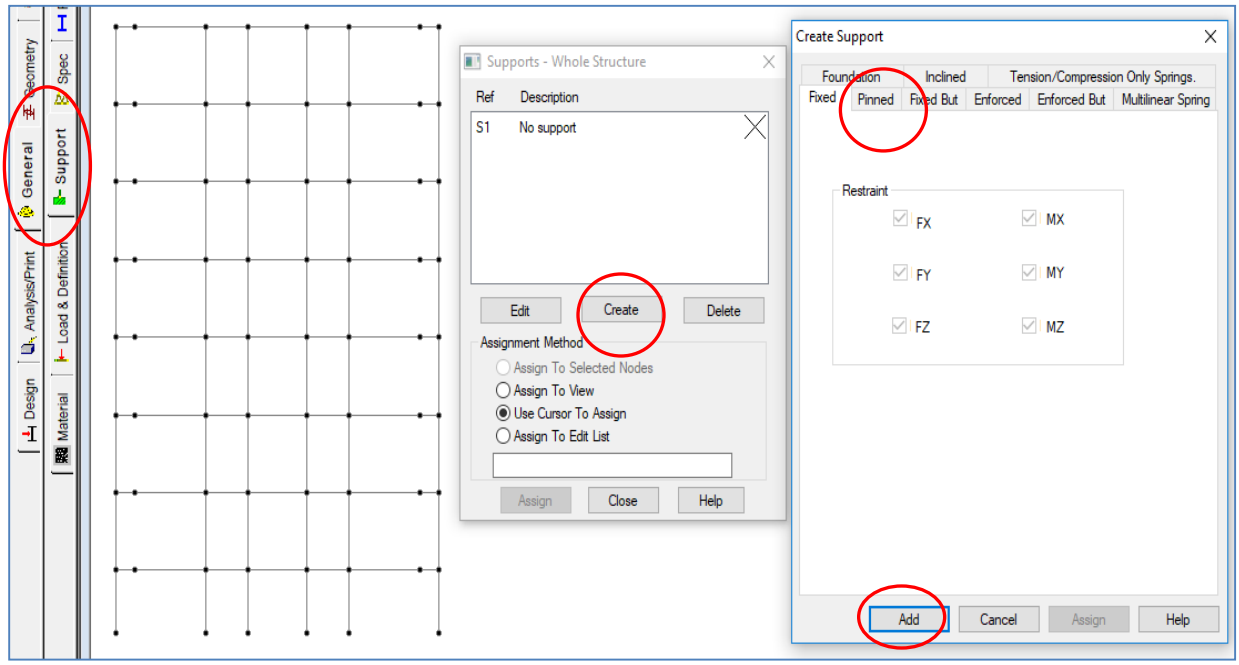


Fig: 5.11

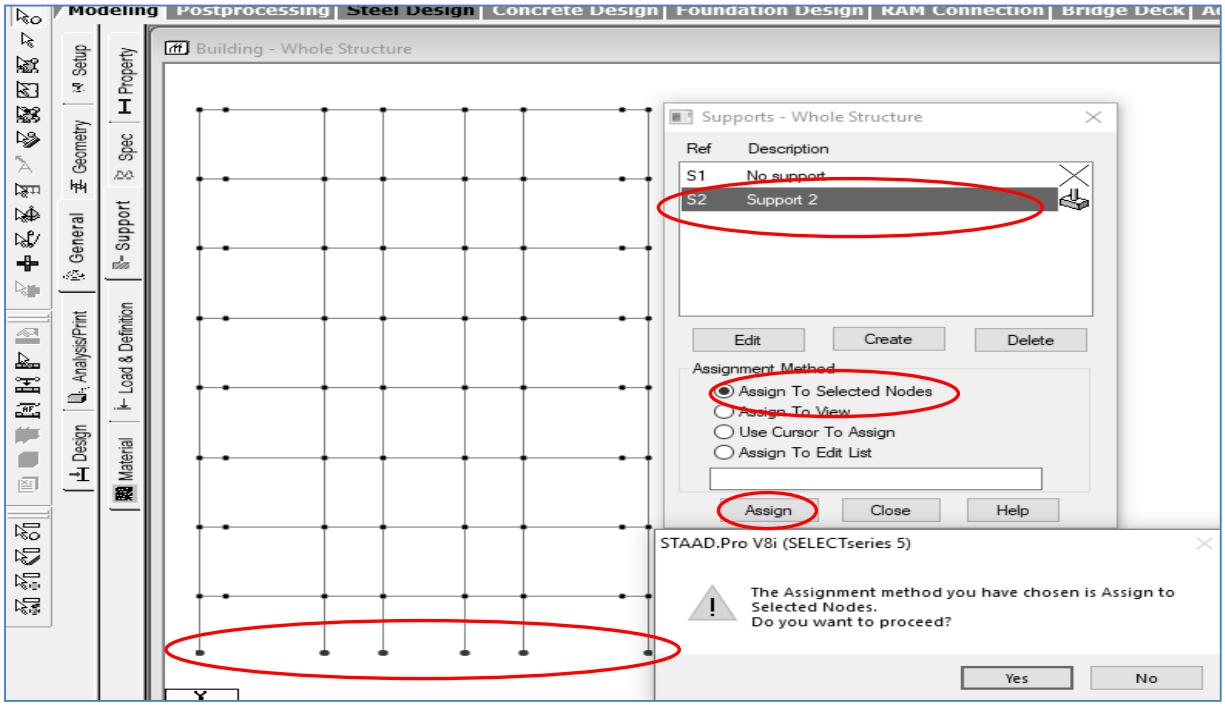


Fig: 5.12

2.2 Property (Beam, Column creates):

2.2.1 Same processes follow for other columns and Beams.

2.2.2 Column & Beam Define: From left side Click on General →Property →Define →Rectangle →YD=1.25 ft, ZD=1 ft (For column C1) →Add. (Fig: 5.13)

2.2.3 Same processes follow for other columns and Beams.

2.2.4 (Here YD is depth of Beams and columns and ZD is width of Beams and columns)

2.3 Column & Beam Assign: Click on each of the property →Select the member from view as your given layout →Assign to selected beams →Assign →Yes.

2.4 Load & Definition: In STADD Pro. must be follow the sequence as 1. Earthquake, 2. Wind load, then 3. Dead load and 4. Live load.

2.4.1 Earthquake Definition: Load & Definition → Definitions →Seismic Definitions →Add →Type: UBC 1994 →Then write down the parameter values as your given data building category →Add→Self weight Factor = 1→Add Then click on Floor Weights →Pressure = -0.17(Slab weight 75+FF 30+ PW 25+LL 40 =170 psf= 0.17 ksf) and Y range 0 to 78 →Add →Close. (Fig: 5.14 and Fig: 5.15)

2.4.2 Wind Load Definitions: Load & Definition → Definitions →Add →Type:1, Comments: X-Direction →Add → TYPE 1, Comments: Z-Direction → Add→ Close.

a. TYPE1: X-Direction→Add →Calculate as per ASCE-7→Input the Common data→Apply→Main building data according to your Project→Apply →Ok Add →Close. (Fig: 5.16 and Fig: 5.17)
b. Same process follow for TYPE2: Z-Direction.

2.4.3 Load Cases Details:

Add →Number =1, Loading Type = Seismic, Title = EQx →c
Number =2, Loading Type = Seismic, Title = EQz →Add
Number =3, Loading Type = Wind, Title = WX →Add
Number =4, Loading Type = Wind, Title = WZ →Add
Number =5, Loading Type = Dead, Title = DL →Add
Number =6, Loading Type = Live, Title = LL →Add →Close

2.4.4 Assign all loads as

- EQx →Add →Seismic Load →X-direction, Factor =1
- EQz →Add →Seismic Load →Z-direction, Factor =1
- WX →Add →Wind Load →X-direction, Factor =1 →Y Range, Minimum =8, Maximum = 78 →Add→Close
- WX →Add →Wind Load →Z-direction, Factor =1 →Y Range, Minimum =8, Maximum = 78 →Add→Close
- DL →Add→Self weight → Factor= -1 →Add→Close →?SELF WEIGHT Y-1 →Assign to view →Assign
- DL →Add→Floor Load → Pressure (input total floor load with negative sign) then Y Range, Minimum =8, Maximum = 78 →Add→Close
- LL →Add→Floor Load → Pressure (input total floor load with negative sign) then Y Range, Minimum =8, Maximum = 78 →Add→Close

2.5 Load Combinations: Load cases details →Add → Define Combinations →Name: DL+LL or other combinations as required then select the load name from the Available load cases and send from left to right side →Input necessary factors →Add→Close. (Fig: 5.21). Otherwise use the Auto Load Combinations according to ACI Code and Add.

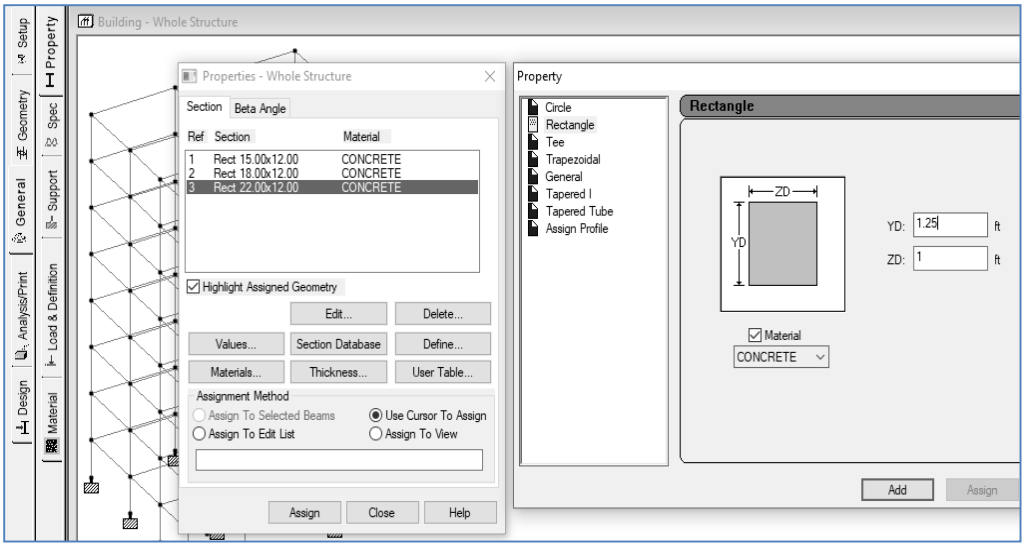


Fig: 5.13

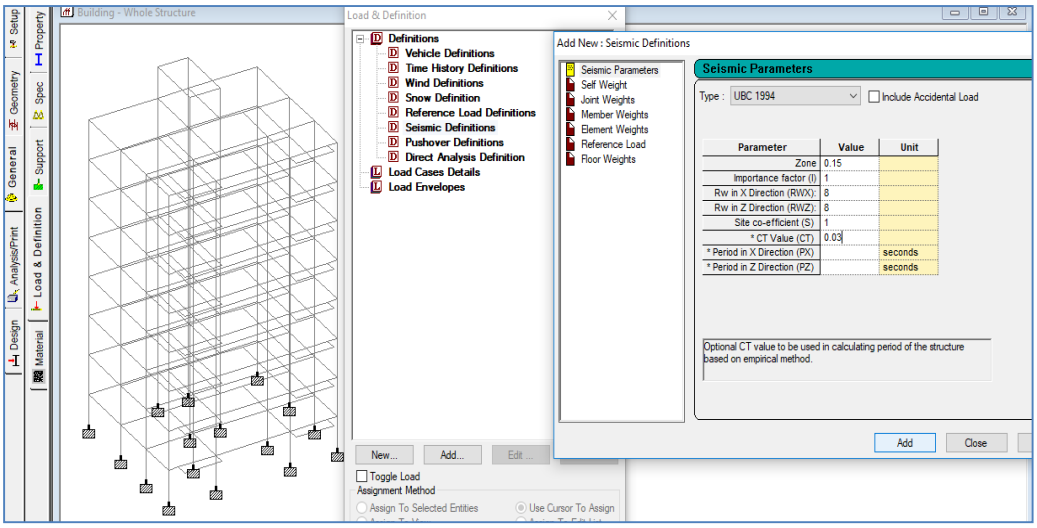


Fig: 5.14

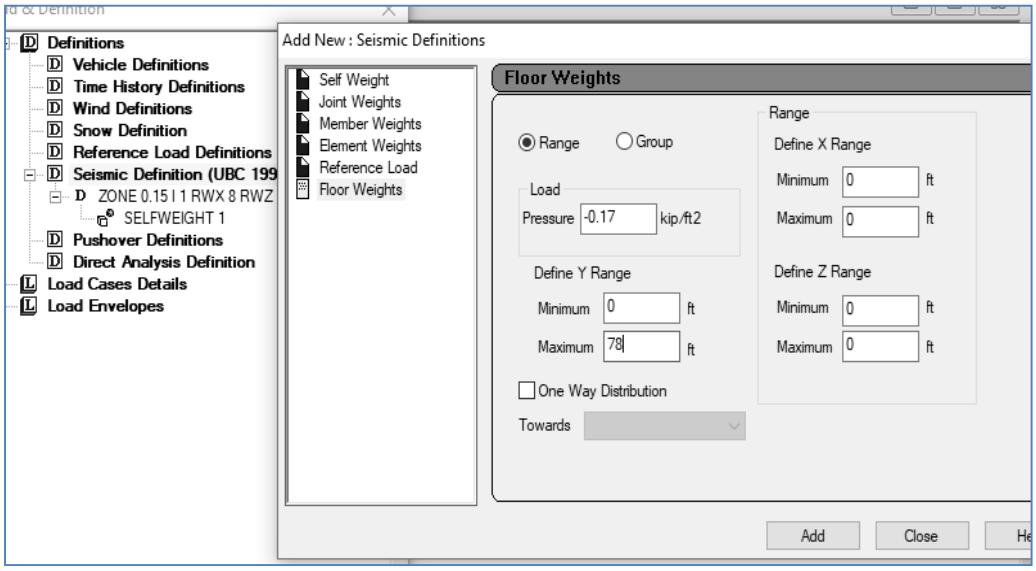


Fig: 5.15

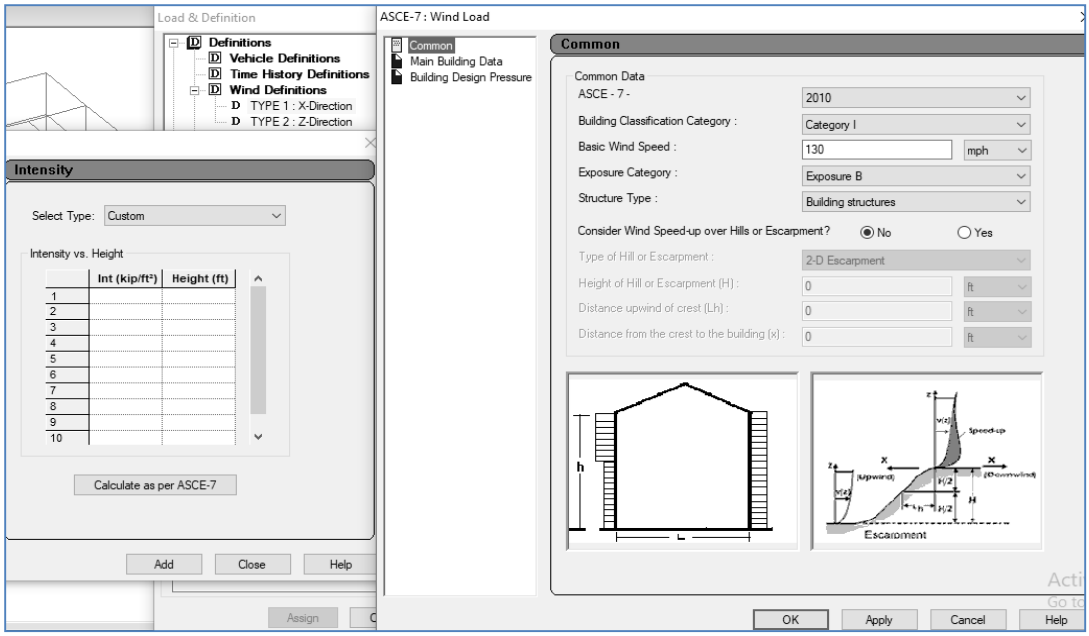


Fig: 5.16

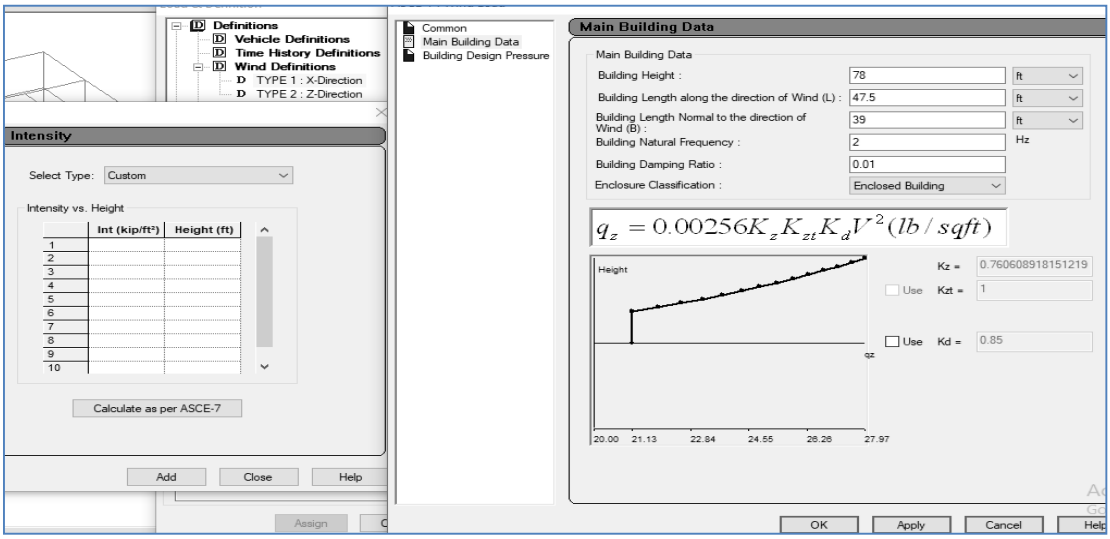


Fig: 5.17

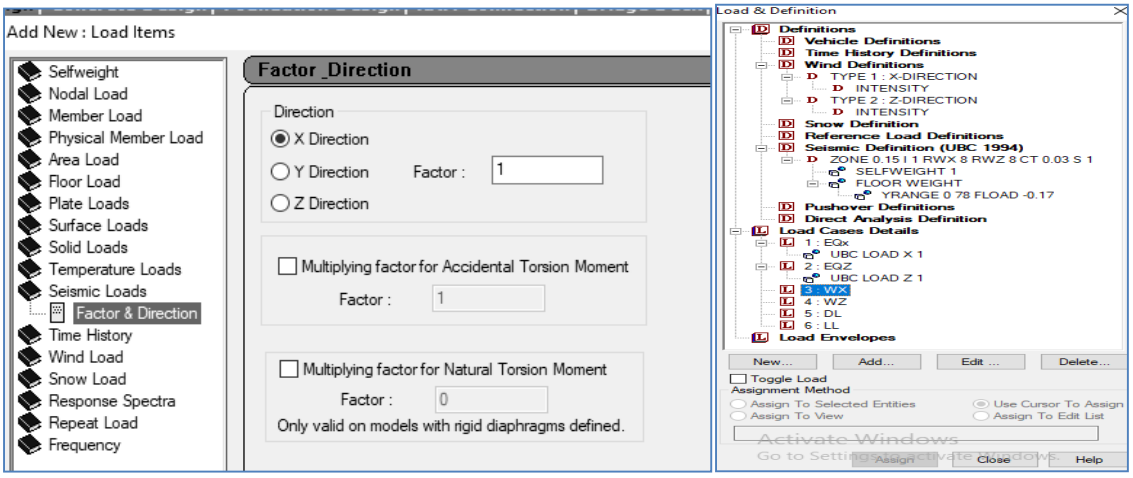


Fig: 5.18

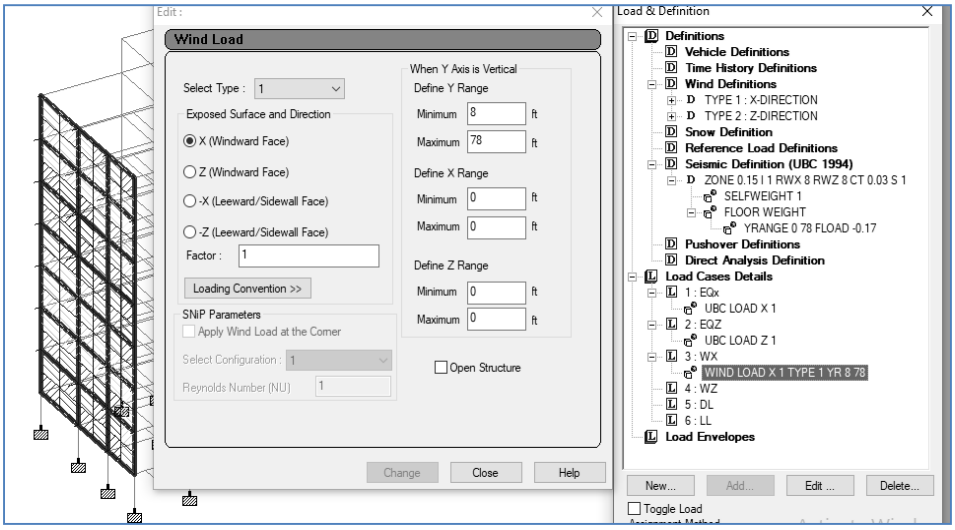


Fig: 5.19

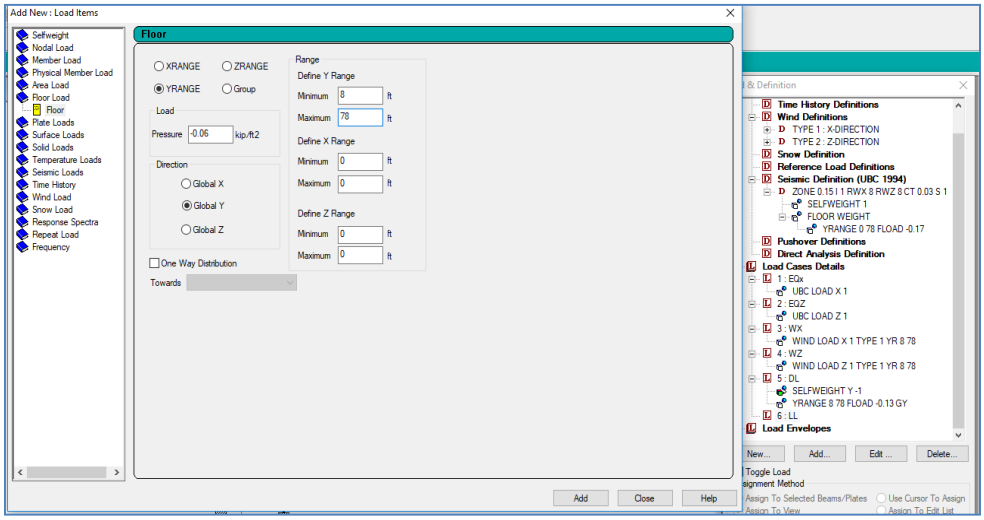


Fig: 5.20

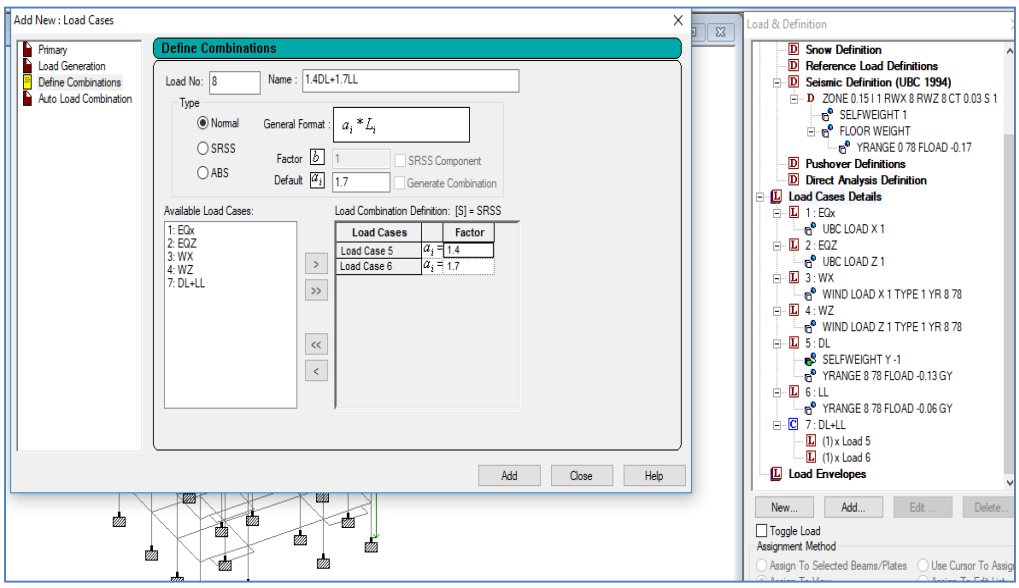


Fig: 5.21

3 Analysis and Result:

- From left side click on Analysis/Print →Static Check or All →Add →Close
- At Menu bar →Analyze →Run Analysis →Go to post processing mode →Done →Selected load cases = DL+LL →Apply →OK. (Fig: 4.29 & Fig: 4.30)
- For Support Reactions use node cursor and double click on the support point →Reactions. Then get the Table for all Support Reactions. (Fig: 4.31)
- For Beam Forces: From left side click on Beam →from Graphs find out Axial force by clicking on required Beam.

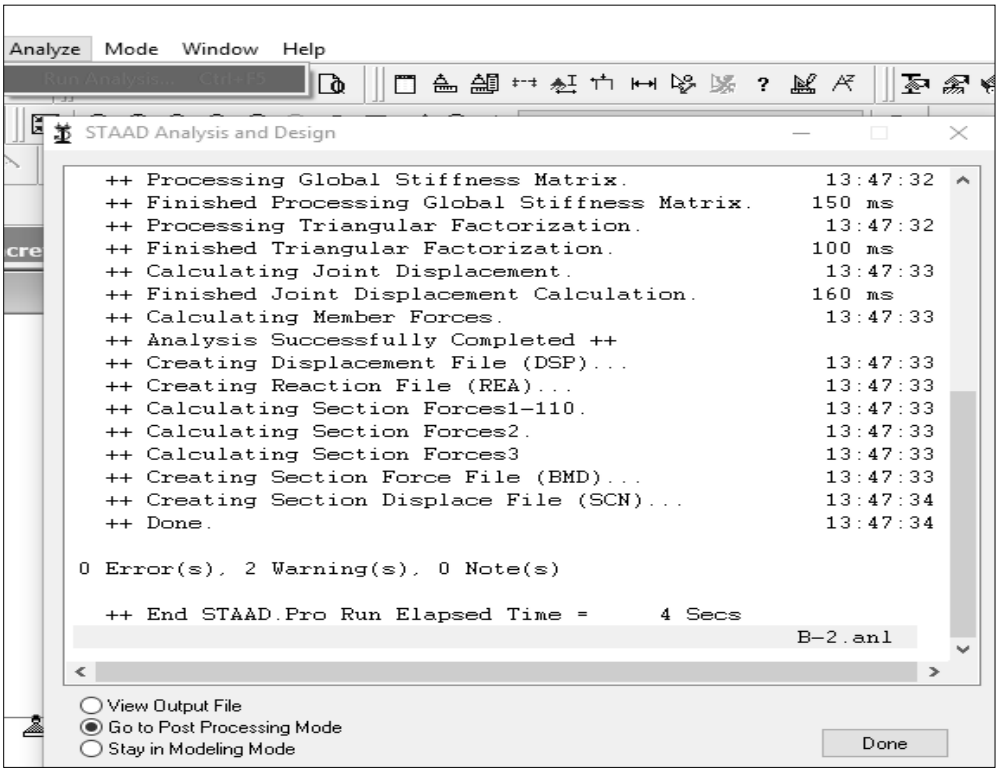


Fig: 5.22

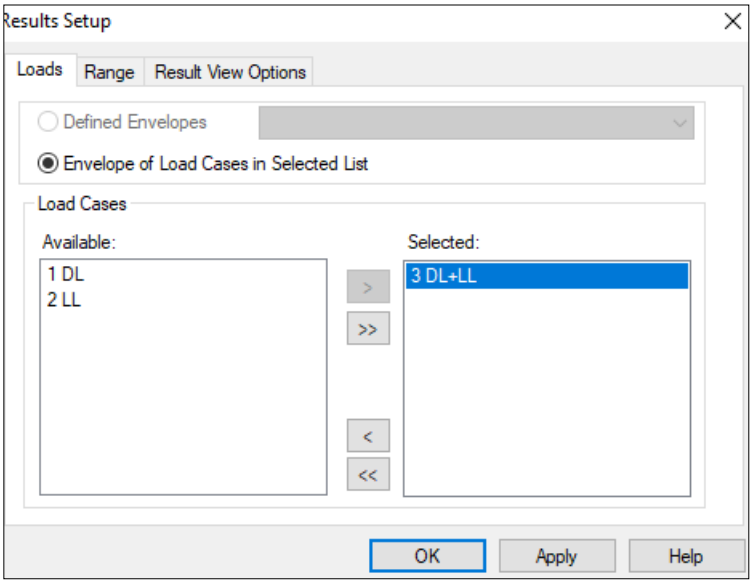


Fig: 5.23

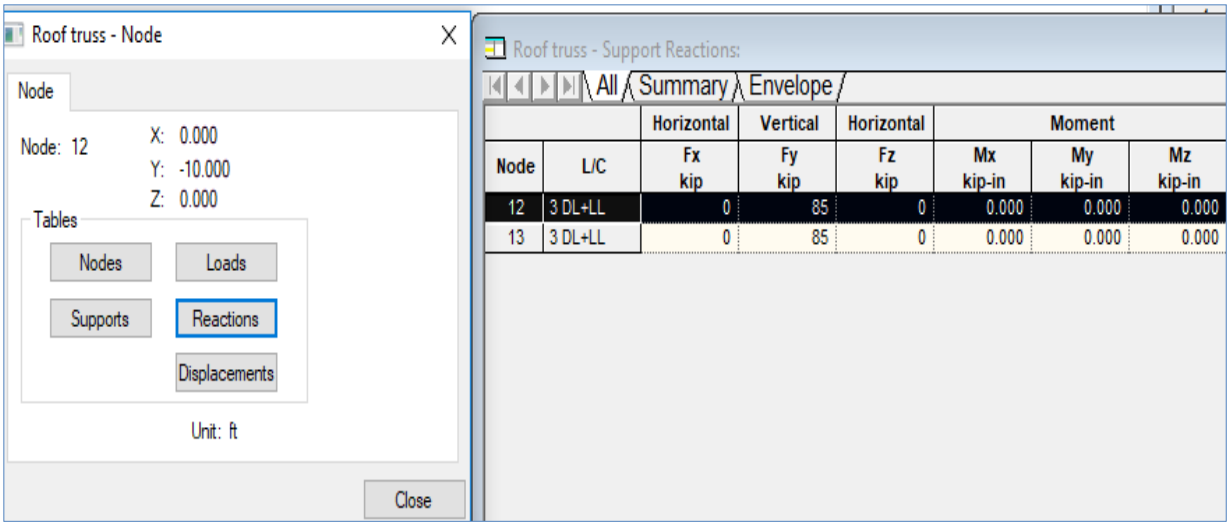

Roof truss - Node
Node
Node: 12
X: 0.000
Y: -10.000
Z: 0.000
Tables
Nodes
Loads
Supports
Reactions
Displacements
Unit: ft
Close
Roof truss - Support Reactions:
All
Summary
Envelope
Horizontal
Vertical
Horizontal
Moment
Node
L/C
Fx kip
Fy kip
Fz kip
Mx kip-in
My kip-in
Mz kip-in
12 3 DL+LL 0 85 0 0.000 0.000 0.000
13 3 DL+LL 0 85 0 0.000 0.000 0.000

Fig: 5.24

**********END**********