

STRUCTURAL DESIGN LABORATORY

LAB MANUAL

Academic Year : 2019 - 2020
Subject Code : BSTB09
Regulations : IARE – R18
Class : I Semester (STE)

Prepared by
Dr. J S R PRASAD
Professor



DEPARTMENT OF CIVIL ENGINEERING
INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal – 500 043, Hyderabad



INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

DEPARTMENT OF CIVIL ENGINEERING

Program: Master of Technology (M. Tech)

VISION OF THE DEPARTMENT

To produce eminent, competitive and dedicated civil engineers by imparting latest technical skills and ethical values to empower the students to play a key role in the planning and execution of infrastructural & developmental activities of the nation.

MISSION OF THE DEPARTMENT

To provide exceptional education in civil engineering through quality teaching, state-of-the-art facilities and dynamic guidance to produce civil engineering graduates, who are professionally excellent to face complex technical challenges with creativity, leadership, ethics and social consciousness.



INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad - 500 043

DEPARTMENT OF CIVIL ENGINEERING

Program: Master of Technology (M. Tech)

PROGRAM OUTCOMES (PO's)	
PO 1	Independently carry out research / investigation and development work to solve practical problems.
PO 2	Write and present a substantial technical report / document.
PO 3	Capable to apply the core, multidisciplinary knowledge for understanding the problems in structural engineering and allied fields.
PO 4	Apply appropriate techniques, resources, modern engineering and Information Technology (IT) tools including predictions, modeling of complex structural engineering activities.
PO 5	Able to identify and analyze the impact of Structural Engineering in development projects and find a suitable solution from number of alternatives.
PO 6	Conceptualize and design civil engineering structures considering various socioeconomic factors.
PO 7	Ability to demonstrate in-depth knowledge of Structural Engineering and build capability to apply that knowledge to real problems.

STRUCTURAL DESIGN LABORATORY SYLLABUS

Exp. No.	List of Experiments
1.	Introduction to Staad Pro
2.	Analysis of Continuous beam
3.	Analysis of Single storey frame
4.	Analysis of Multi-storey frame
5.	Design of Multi-storey frame
6.	Analysis of Multi-storeyed building
7.	Design of Multi-storeyed building
8.	Wind load analysis on RCC building
9.	Analysis and design of Steel truss
10.	Analysis and design of Isolated footing
11.	Analysis and design of Combined footing
12.	Analysis of Bridge deck
13.	Revision

ATTAINMENT OF PROGRAM OUTCOMES (POs)

Exp. No.	Name of Experiment	Program Outcomes Attained
1.	Introduction to Staad pro	PO1, PO3,PO5
2.	Analysis of continuous beam	PO1, PO5
3.	Analysis of single storey frame	PO1, PO3
4.	Analysis of multi-storey frame	PO1, PO5
5.	Design of multi-storey frame	PO1, PO3
6.	Analysis of multi-storeyed building	PO1, PO5
7.	Design of multi-storeyed building	PO1, PO3
8.	Wind load analysis on RCC building	PO1, PO5
9.	Analysis and design of steel truss	PO1,PO3, PO5
10.	Analysis and design of isolated footing	PO1, PO3,PO5
11.	Analysis and design of combined footing	PO1, PO3, PO5
12.	Analysis of bridge deck	PO1, PO3, PO5
13.	Revision	

MANDATORY INSTRUCTIONS

1. Students should report to the labs concerned as per the timetable.
2. Record should be updated from time to time and the previous experiment must be signed by the faculty in charge concerned before attending the lab.
3. Students who turn up late to the labs will in no case be permitted to perform the experiment scheduled for the day.
4. After completion of the experiment, certification of the staff in-charge concerned in the observation book is necessary.
5. Students should bring a notebook of about 100 pages and should enter the readings/observations/results into the notebook while performing the experiment.
6. The record of observations along with the detailed experimental procedure of the experiment performed in the immediate previous session should be submitted and certified by the staff member in-charge.
7. Not more than FIVE students in a group are permitted to perform the experiment on a set up.
8. The group-wise division made in the beginning should be adhered to, and no mix up of student among different groups will be permitted later.
9. The components required pertaining to the experiment should be collected from Lab- in-charge after duly filling in the requisition form.
10. When the experiment is completed, students should disconnect the setup made by them, and should return all the components/instruments taken for the purpose.
11. Any damage of the equipment or burnout of components will be viewed seriously either by putting penalty or by dismissing the total group of students from the lab for the semester/year.
12. Students should be present in the labs for the total scheduled duration.
13. Students are expected to prepare thoroughly to perform the experiment before coming to Laboratory.
14. Procedure sheets/data sheets provided to the students groups should be maintained neatly and are to be returned after the experiment.

15. DRESS CODE:
 - a. Boys - Formal dress with tuck in and shoes.
 - b. Girls - Formal dress (Salwarkameez).
 - c. Wearing of jeans is strictly prohibited

CONTENTS

S. No.	Description	Page No.
1	Vision and Mission of the department	02
2	Program Outcomes (PO's)	03
3	Syllabus	04
4	Attainment of Program Outcomes (POs)	05
5	Mandatory Instructions	06
6	Contents	07
7	Experiment No 1: Introduction to Staad Pro	08
8	Experiment No 2: Analysis of continuous beam	12
9	Experiment No 3: Analysis of single Storey frame	19
10	Experiment No 4: Analysis of multi-Storey frame	26
11	Experiment No 5: Design of multi-Storey frame	30
12	Experiment No 6: Analysis of multi-storeyed building	36
13	Experiment No 7: Design of multi-storeyed building	39
14	Experiment No 8: Wind load analysis on rcc building	41
15	Experiment No 9: Analysis and design of steel truss	44
16	Experiment No 10: Analysis and design of isolated footing	59
17	Experiment No 11: Analysis and design of combined footing	64
18	Experiment No 12: Analysis of bridge deck	67
19	Experiment No 13: Revision	

EXPERIMENT NO. 1

INTRODUCTION ON STAAD PRO

Aim: To study the introduction about STAAD Pro

Overview: STAAD- Standard Analyze and Design

Company name – Bentley

Version- STAADV8i.

STAAD or (STAAD Pro) is a structural analysis and design computer program originally developed by Research Engineers International in Yorba Linda, CA. In late 2005, Research Engineer International was bought by Bentley Systems. An older version called STAAD-III for windows is used by Iowa State University for educational purposes for civil and structural engineers. The commercial version STAAD Pro is one of the most widely used structural analysis and design software. It supports several steel, concrete and timber design codes. It can make use of various forms of analysis from the traditional 1st order static analysis, 2nd order p-delta analysis, geometric nonlinear analysis or a buckling analysis. It can also make use of various forms of dynamic analysis from modal extraction to time history and response spectrum analysis.

In recent years it has become part of integrated structural analysis and design solutions mainly using an exposed API called Open STAAD to access and drive the program using a VB macro system included in the application or other by including open STAAD functionality in applications that themselves include suitable programmable macro systems. Additionally STAAD Pro has added direct links to applications such as RAM Connection and STAAD Foundation to provide engineers working with those applications which handle design post processing not handled by STAAD Pro itself.

Advantages:

1. Easy to use interface,
2. Conformation with the Indian Standard Codes
3. Versatile nature of solving any type of problem
4. Accuracy of the solution.

STAAD Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From

model generation, analysis and design to visualization and result verification, STAAD Pro is the professional's choice for steel, concrete, timber, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

STAAD Pro consists of the following:

The STAAD Pro Graphical User Interface:

It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

The STAAD analysis and design engine:

It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, timber and aluminum design.

To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. In the initial phase of our project we have done calculations regarding loadings on buildings and also considered seismic and wind loads.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural elements shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to

design for durability. Design, including design for durability, construction and use in service should be considered as a whole. The realization of design objectives requires compliance with clearly defined standards for materials, production, workmanship and also maintenance and use of structure in service.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

Structural engineering is the branch of civil engineering that deals with the analysis and Design of structure that will support or resist the stresses, forces and loads. Throughout the history, Civil engineers have made an important contribution to improve the environment by designing and Construction major structures and facilities. The profession requires a lot of accuracy because most of the time; the final result of any project will directly or indirectly affect people's lives and safety. As a Result, software usage in this industry reduces the complexity of loss of challenging projects.

STAAD (Structural Analysis and Design Program) is a general structural and design program With applications primarily in the building industry-commercial buildings, bridges and highway Structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, Culverts and other embedded structures [23]. STAAD supports several steel, concrete and timber Design codes. It offers various analysis methods such as the traditional 1st order static analysis, 2ndOrder P-delta analysis, geometric nonlinear analysis and the buckling analysis. It can also analyze Various type of dynamic analysis starts from model extraction to time history and response spectrum analysis. STAAD is fast, efficient, easy to use and accurate platform, conformation with many Standard codes and qualified to solve any type of problem. These features had made the STAAD the Professional's choice. Today, STAAD is one of the most popular structural engineering software Products.

Methods in STAAD Pro:

- Snap and grid method
- Structural wizard

- Excel method
- DXFmethod
- STAAD editor method

Snap and grid method:

Open STAAD Pro> snap node/ beam > edit. Plane X-Y

Angle of plane: Y-Y Grid origin

X	Y	Z
0	0	0

In this you can change the values and spacing of the coordinates. Snap and grid method>snap node beam> to create.

Structural wizard:

Geometry>run structural wizard>model type (frame model)> bay frame> transfer> yes> close.

Excel sheet method:

To create node and member

DXF method:

To import the file from AutoCAD format to STAAD format

a) Draw in AutoCAD> save> file type: AutoCAD 2007/ LT 2007 DXF*(dxf)> save> close.

b) Open STAAD Pro> file> import> select required file> ok.

Result: Thus the introduction about STAAD Pro is studied successfully.

EXPERIMENT NO. 2

ANALYSIS OF CONTINUOUS BEAM

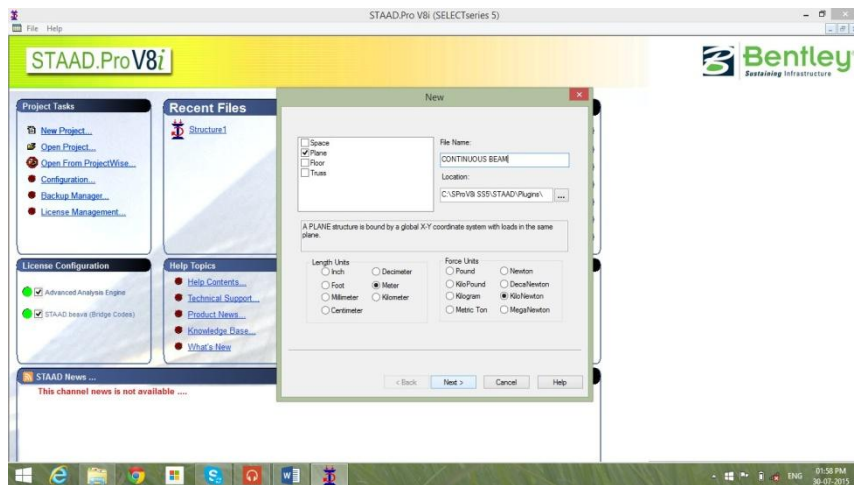
Aim: To analyze the continuous beam using staad.pro

Software: STAAD Pro.

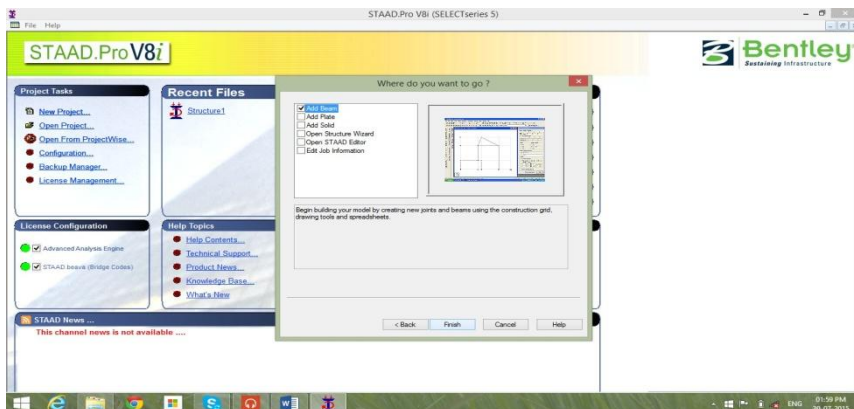
Procedure:

Following are the step wise procedures for Staad Pro Problems (CONTINUOUS BEAM)

1. Double click the Staad Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next

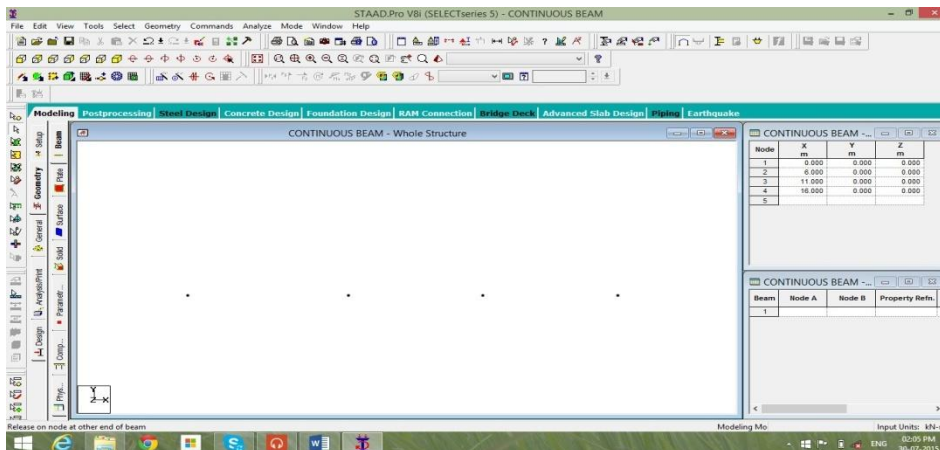


3. Add beam , Finish



4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner.

5. Enter Coordinates of each joint of above beam

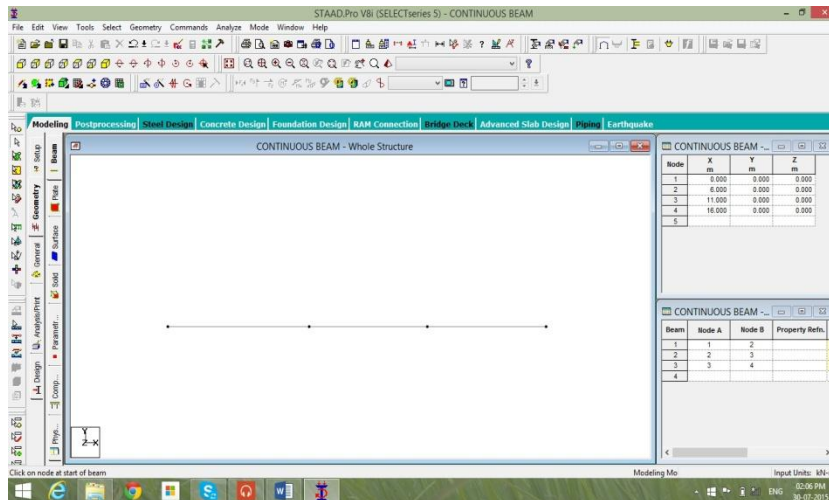


Cut 'Node Screen', Cut 'Structure screen', Expand working screen

- 1 0 0 0
- 2 6 0 0
- 3 11 0 0
- 4 16 0 0

6. Connect all joints by beam element, join nodes by nodes individually

Now the screen should appear like a 3 span beam with nodes clearly seen at the end of each span This step is very important.

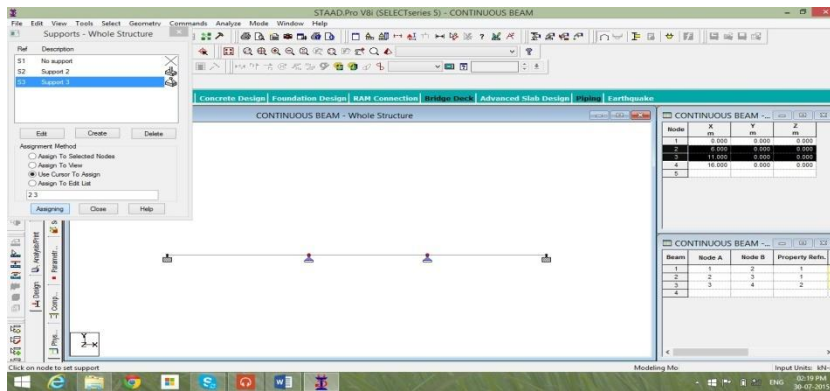


7. Now select Commands. Under this select Member Property\Prismatic and click Rectangle.

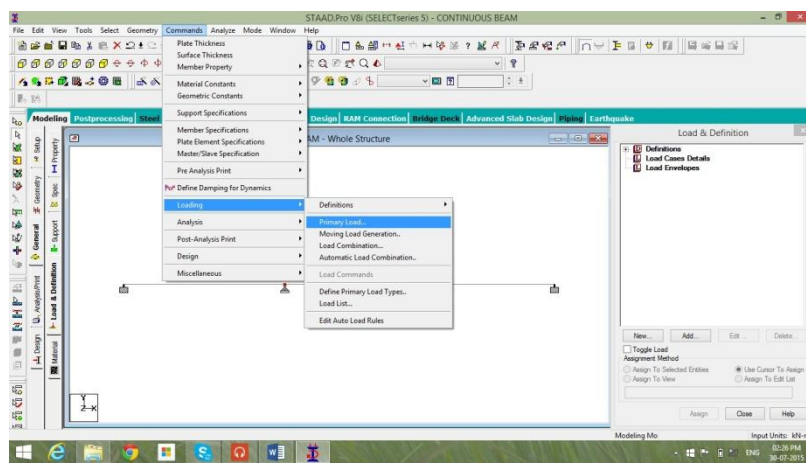
Enter following details YD = 0.45

ZD= 0.30

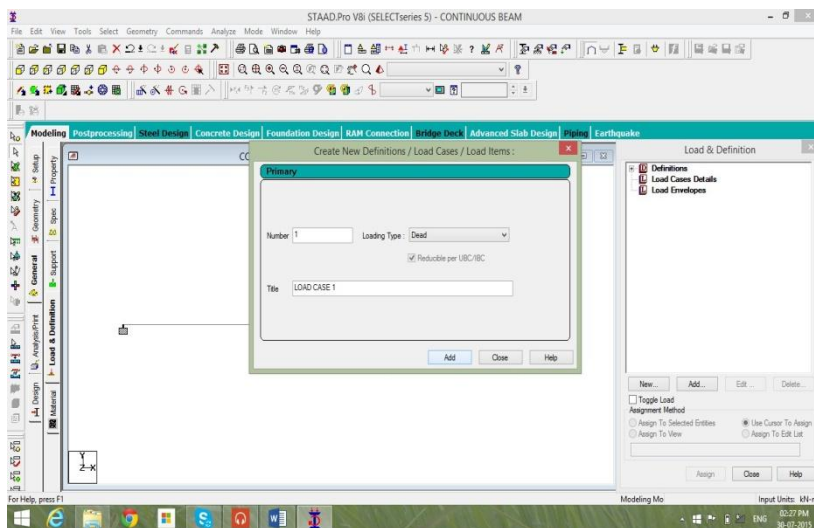
8. Now repeat above steps and assign Pin support to remaining two central supports.



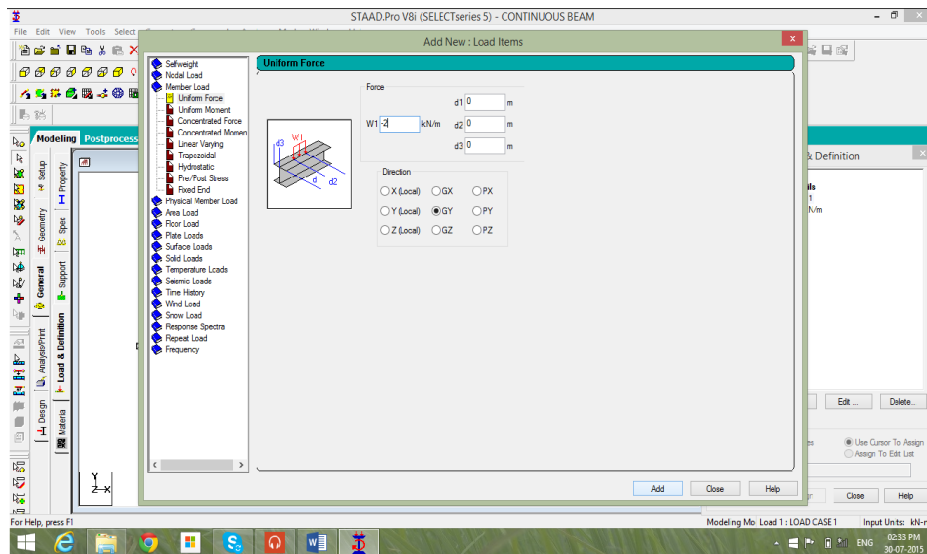
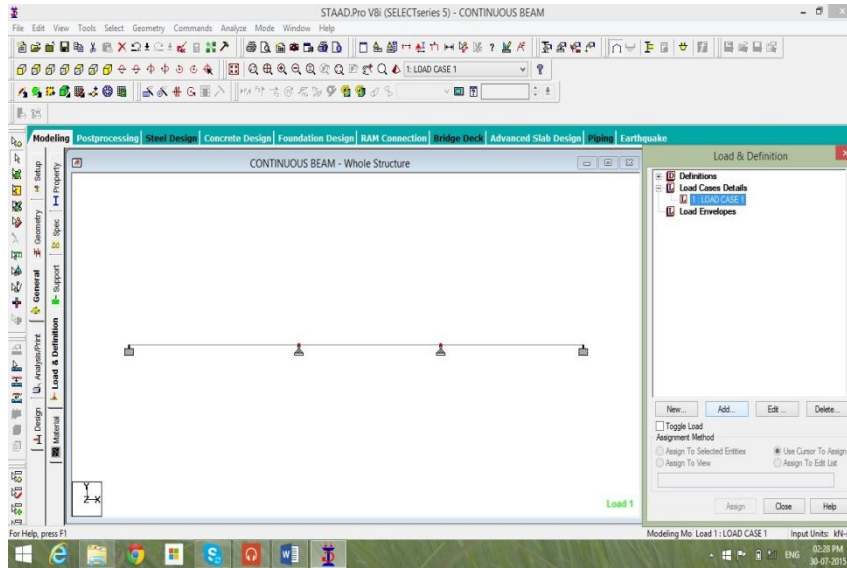
9. Select Commands\Loading\Primary Load



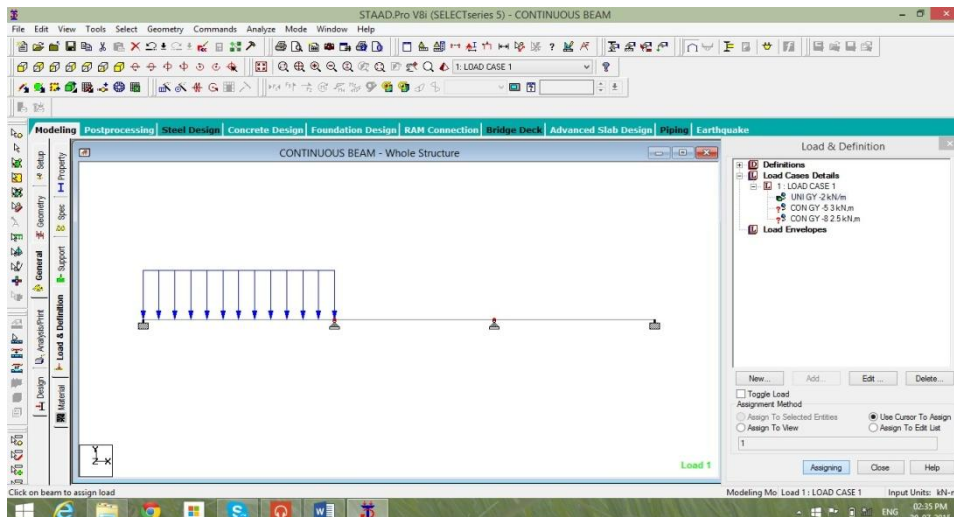
Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close



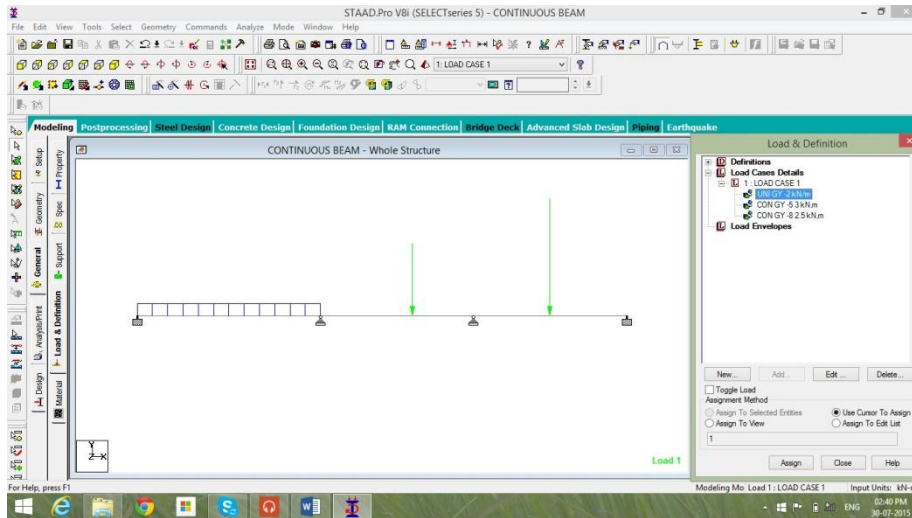
Now click on ' Load case 1 ', New , Load Items , Member load , Uniform Load , W1 , -2 knm Add



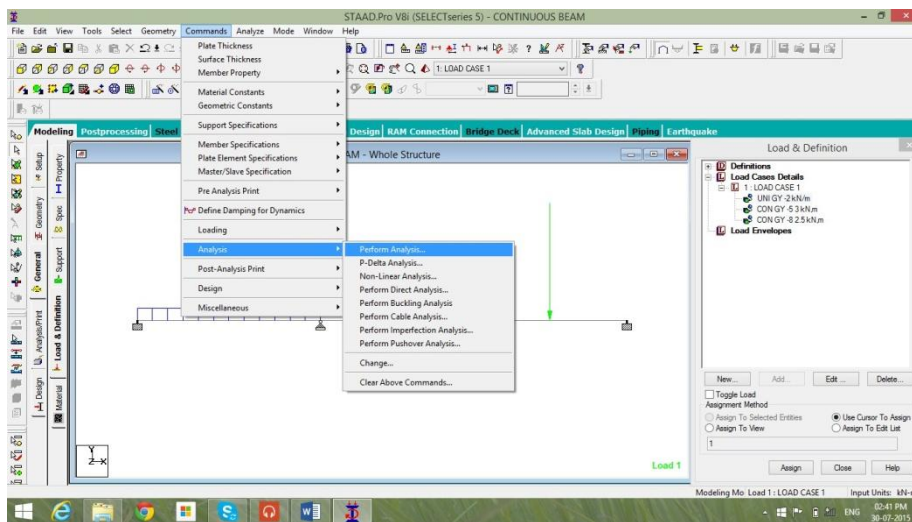
Then Concentrated force P -5 d1 3 Add , then P -8 d1 2.5 Add close Now click on UNI GY -2 kn/m 'use cursor to Assign, Click on first Span.



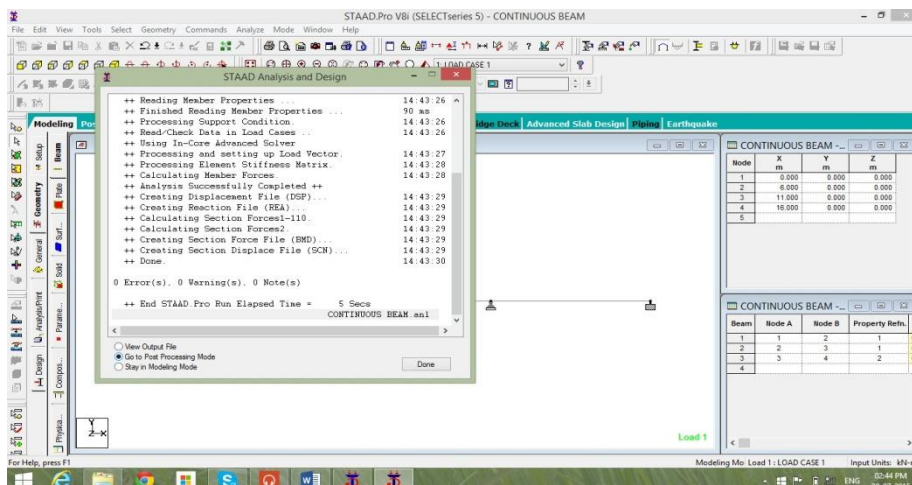
Similarly assign 5kn and 8Kn on remaining two spans and then click close
 To verify whether all loads are assigned correctly or not Click on Load details menu on menu bar And click load view, now load diagram with values should be seen on main diagram.



10. Now Commands\Analysis\Perform Analysis, select all, ok.



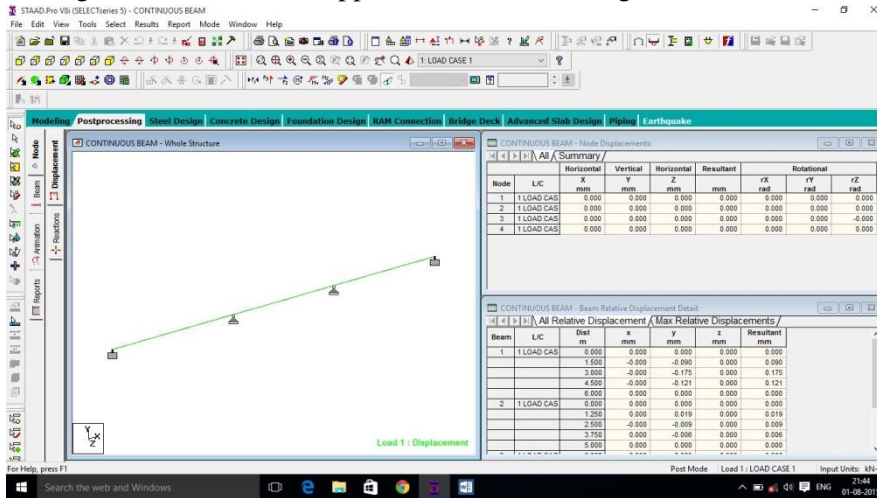
11. Now Click Analysis\Run analysis Click Staad Analysis
 After a time, display will show number of errors & warning messages if anythen



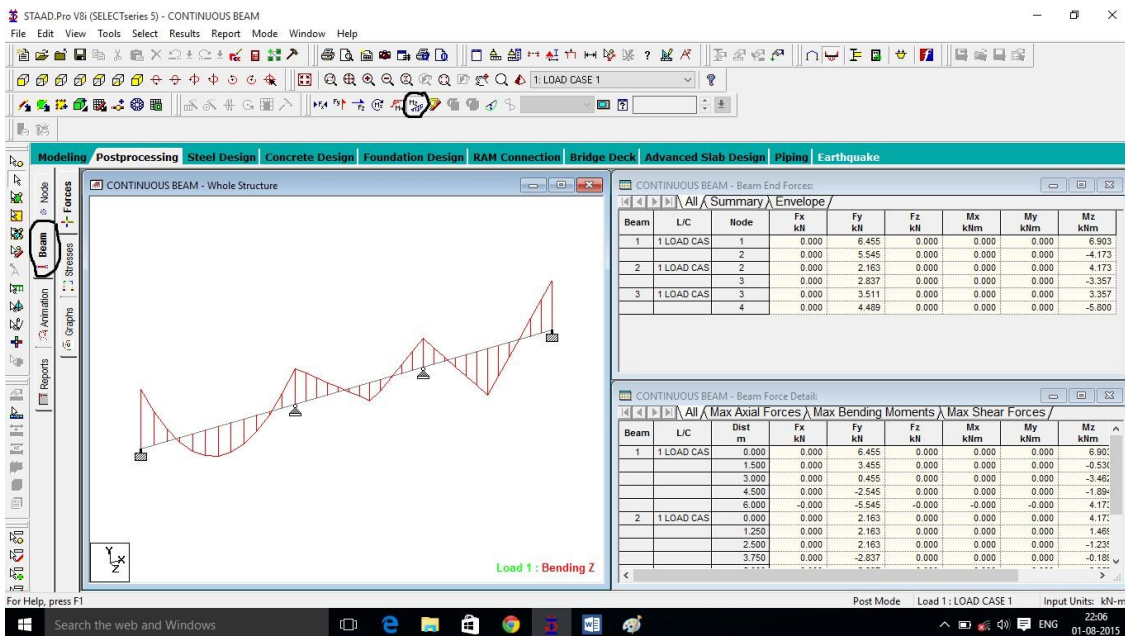
Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode.

Click ‘Go to Post Processing mode ‘ Similarly ‘Done ‘ ‘Apply ‘ ‘OK’

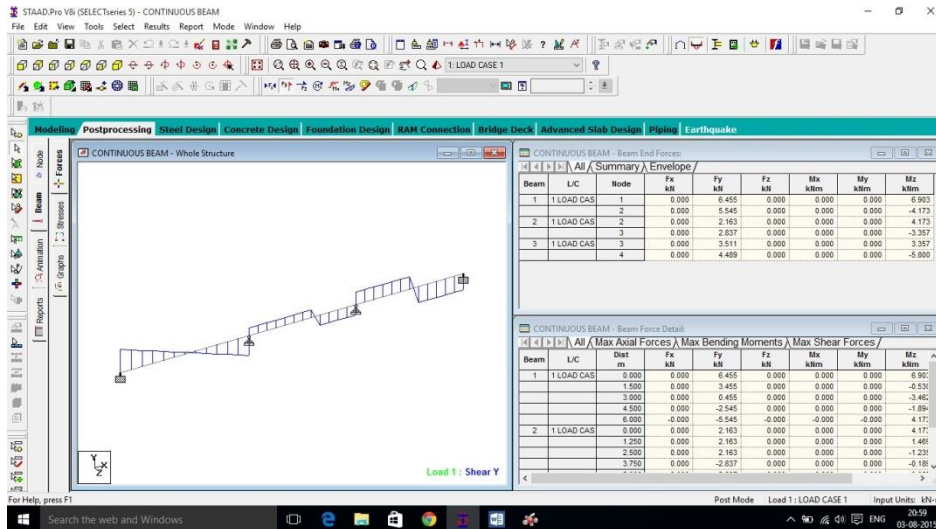
The post processing mode window will appear as shown in the fig. Below.



- To see the bending moment diagram of the analysed beam click on the **beam** tab at left side of the working window. Click on **Mz** button on the menu bar at top of the working window. You should see the BMD as shown in the figure below (Holding down the **ctrl** button scroll the **mouse wheel** to adjust the scale to your convenience). Go to **reactions** menu >> **view value** >> **beam results** and check the boxes **ends**, **maximum** and **midpoint** under **bending** to see the value of **BMD** at respective locations.



13. To see the shear force diagram click on the **beam** tab at left side of the working window. Click on **Fy** button on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the **ctrl** button scroll the **mouse wheel** to adjust the scale to your convenience). Go to **reactions** menu >> **view value** >> **beam results** and check the boxes **ends, maximum and midpoint** under **shear** to see the value of SFD at respective



ocations.

14. To see the displacement of the beam click on the **nodes>displacement** tab in the menu on left side of the working area. You should see the diagram as shown in the figure below (Holding down the **ctrl** button scroll the **mouse wheel** to adjust the scale to your convenience). To view the displacement value you can go to **reactions** menu >> **view value** >> **beam results** and check the box **Max resultant** under **displacement**.
15. To see the reactions at the support of the beam go to **nodes>>reactions** tab on left side of the working area.

Result: The analysis of continuous beam is done by using Staad Pro.

EXPERIMENT NO: 3

ANALYSIS AND DESIGN OF SINGLE-STOREY FRAME

Aim: To analyze the single storeyed space frame using STADD pro

Hardware & software required:

Computer with specified configuration with installed STADD Pro Software

PROCEDURE AND ANALYSIS STEPS:

Staad foundation:

Staad foundation is a powerful tool used to calculate different types of foundations. It is also licensed by Bentley software's. All Bentley software's cost about 10 lakhs and so all engineers can't use it due to heavy cost.

Analysis and design carried in Staad and post processing in staad gives the load at various supports. These supports are to be imported into these software to calculate the footing details i.e., regarding the geometry and reinforcement details.

This software can deal different types of foundations

SHALLOW ($D < B$)

1. Isolated (Spread) Footing
2. Combined (Strip) Footing
3. Mat (Raft) Foundation

DEEP ($D > B$)

1. Pile Cap
2. Driller Pier
1. Isolated footing is spread footing which is common type of footing.
2. Combined Footing or Strap footing is generally laid when two columns are very near to each other.
3. Mat foundation is generally laid at places where soil has less soil bearing capacity.
4. pile foundation is laid at places with very loose soils and where deep excavations are required.

So depending on the soil at type we have to decide the type of foundation required. Also lot of input data is required regarding safety factors, soil, materials used should be given in respective units. After input data is given software design the details for each and every footing and gives the details regarding

1. Geometry of footing
2. Reinforcement

3. Column layout

4. Graphs

5. Manual calculations

These details will be given in detail for each and every column.

Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

Column Position

Column Shape

Column Size

Load Cases

Support List

The multistoreyed frame represents the center line diagram of the building in staad pro. Each support represents the location of different columns in the structure. This structure is used in generating the entire structure using a tool called transitional repeat and link steps. After using the tool the structure that is created can be analyzed in staad pro under various loading cases.

Below figure represents the skeletal structure of the building which is used to carry out the analysis of our building. All the loadings are acted on this skeletal structure to carry out the analysis of our building. This is not the actual structure but just represents the outline of the building in staad pro.

A mesh is automatically created for the analysis of these building.

PROCEDURE AND ANALYSIS STEPS:

Staad foundation:

Staad foundation is a powerful tool used to calculate different types of foundations. It is also licensed by Bentley software's. All Bentley software's cost about 10 lakhs and so all engineers can't use it due to heavy cost.

Analysis and design carried in Staad and post processing in staad gives the load at various supports. These supports are to be imported into these software to calculate the footing details i.e., regarding the geometry and reinforcement details.

This software can deal different types of foundations

SHALLOW ($D < B$)

1. Isolated (Spread) Footing

2. Combined (Strip) Footing

3. Mat (Raft) Foundation

DEEP ($D > B$)

1. Pile Cap

2. Driller Pier

1. Isolated footing is spread footing which is common type of footing.

2. Combined Footing or Strap footing is generally laid when two columns are very near to each other.

3. Mat foundation is generally laid at places where soil has less soil bearing capacity.

4. pile foundation is laid at places with very loose soils and where deep excavations are required.

So depending on the soil at type we have to decide the type of foundation required. Also lot of input data is required regarding safety factors, soil, materials used should be given in respective units. After input data is given software design the details for each and every footing and give the details regarding

1. Geometry of footing

2. Reinforcement

3. Column layout

4. Graphs

5. Manual calculations

These details will be given in detail for each and every column.

Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

Column Position

Column Shape

Column Size

Load Cases

Support List

The multistoreyed frame represents the center line diagram of the building in staad pro. Each support represents the location of different columns in the structure. This structure is used in generating the entire structure using a tool called transitional repeat and link steps.

After using the tool the structure that is created can be analyzed in staad pro under various loading cases.

Below figure represents the skeletal structure of the building which is used to carry out the analysis of our building. All the loadings are acted on this skeletal structure to carry out the analysis of our building. This is not the actual structure but just represents the outline of the building in staad pro.

A mesh is automatically created for the analysis of these building.

PROCEDURE AND ANALYSIS STEPS:

Staad foundation:

Staad foundation is a powerful tool used to calculate different types of foundations. It is also licensed by Bentley software's. All Bentley software's cost about 10 lakhs and so all engineers can't use it due to heavy cost.

Analysis and design carried in Staad and post processing in staad gives the load at various supports. These supports are to be imported into these software to calculate the footing details i.e., regarding the geometry and reinforcement details.

This software can deal different types of foundations

SHALLOW ($D < B$)

1. Isolated (Spread) Footing
2. Combined (Strip) Footing
3. Mat (Raft) Foundation

DEEP ($D > B$)

1. Pile Cap
2. Driller Pier

1. Isolated footing is spread footing which is common type of footing.
2. Combined Footing or Strap footing is generally laid when two columns are very near to each other.
3. Mat foundation is generally laid at places where soil has less soil bearing capacity.
4. pile foundation is laid at places with very loose soils and where deep excavations are required.

So depending on the soil at type we have to decide the type of foundation required. Also lot of input data is required regarding safety factors, soil, materials used should be given in respective units. After input data is given software design the details for each and every footing and gives the details regarding

1. Geometry of footing

2. Reinforcement

3. Column layout

4. Graphs

5. Manual calculations

These details will be given in detail for each and every column.

Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

Column Position

Column Shape

Column Size

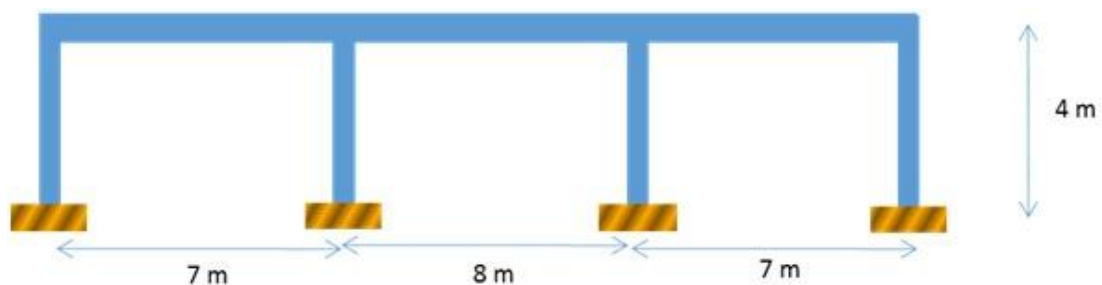
Load Cases

Support List

The multistoreyed frame represents the center line diagram of the building in staad pro. Each support represents the location of different columns in the structure. This structure is used in generating the entire structure using a tool called transitional repeat and link steps. After using the tool the structure that is created can be analyzed in staad pro under various loading cases.

Below figure represents the skeletal structure of the building which is used to carry out the analysis of our building. All the loadings are acted on this skeletal structure to carry out the analysis of our building. This is not the actual structure but just represents the outline of the building in staad pro.

A mesh is automatically created for the analysis of these building.



Load Conditions and Structural System Response :

The concepts presented in this section provide an overview of building loads and their effect on the structural response of typical wood-framed homes. As shown in Table, building loads can be divided into types based on the orientation of the structural action or

forces that they induce: vertical and horizontal (i.e., lateral) loads. Classification of loads are described in the following sections.

Building Loads Categorized by Orientation:

Types of loads on an hypothetical building are as follows.

Vertical Loads

Dead (gravity)

Live (gravity)

Snow (gravity)

Wind (uplift on roof)

Seismic and wind (overturning)

Seismic (vertical ground motion)

Horizontal (Lateral) Loads:

Direction of loads is horizontal w.r.t to the building.

Wind

Seismic (horizontal ground motion)

Flood (static and dynamic hydraulic forces)

Soil (active lateral pressure)

Dead load calculation

Weight = Volume x Density

Self weight floor finish = $0.12 \times 25 + 1 = 3 \text{ kn/m}^2$

The above example shows a sample calculation of dead load.

Dead load is calculated as per **IS 875 part 1**

Generally for any structure live load is taken as **25 N/mm** for design.

Live loads are calculated as per **IS 875 part 2**

Wind loads:

In the list of loads we can see wind load is present both in vertical and horizontal loads. This is because wind load causes uplift of the roof by creating a negative (suction) pressure on the top of the roof. After designing wind load can be assigned in two ways

1. collecting the standard values of load intensities for a particular heights and assigning of the loads for respective height.
2. calculation of wind load as per **IS 875 part 3**.

We designed our structure using second method which involves the calculation of wind load using wind speed. In Hyderabad we have a wind speed of 45 kmph for 10 m height and this value is used in Calculation.

Floor load:

Floor load is calculated based on the load on the slabs. Assignment of floor load is done by creating a load case for floor load. After the assignment of floor load our structure looks as shown in the below figure.

The intensity of the floor load taken is: **0.0035 N/mm²**

-ve sign indicates that floor load is acting downwards

Load combinations:

All the load cases are tested by taking load factors and analyzing the building in different load combination as per **IS456** and analyzed the building for all the load combinations and results are taken and maximum load combination is selected for the design

Load factors as per **IS456-2000**

When the building is designed for both wind and seismic loads maximum of both is taken.

Live load	dead load	wind load
1.5	1.5	0
1.2	1.2	1.2
0.9	0.9	0.9

Because wind and seismic do not come at same time as per code.
Structure is analyzed by taking all the above combinations.

Result: The analysis of single-storey frame is done by using staad pro.

EXPERIMENT NO: 4
ANALYSIS OF MULTI-STOREY FRAME

Aim: To analyze the Multi storeyed space frame using STADD pro

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed STADD Pro Software

PROCEDURE AND ANALYSIS STEPS:

Staad foundation:

Staad foundation is a powerful tool used to calculate different types of foundations. It is also licensed by Bentley software's. All Bentley software's cost about 10 lakhs and so all engineers can't use it due to heavy cost.

Analysis and design carried in Staad and post processing in staad gives the load at various supports. These supports are to be imported into these software to calculate the footing details i.e., regarding the geometry and reinforcement details.

This software can deal different types of foundations

SHALLOW ($D < B$)

1. Isolated (Spread) Footing
2. Combined (Strip) Footing
3. Mat (Raft) Foundation

DEEP ($D > B$)

1. Pile Cap
2. Drilled Pier
1. Isolated footing is spread footing which is common type of footing.
2. Combined Footing or Strap footing is generally laid when two columns are very near to each other.
3. Mat foundation is generally laid at places where soil has less soil bearing capacity.
4. pile foundation is laid at places with very loose soils and where deep excavations are required.

So depending on the soil at type we have to decide the type of foundation required. Also lot of input data is required regarding safety factors, soil, materials used should be given in respective units. After input data is given software design the details for each and every footing and gives the details regarding

1. Geometry of footing

2. Reinforcement

3. Column layout

4. Graphs

5. Manual calculations

These details will be given in detail for each and every column.

Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

Column Position

Column Shape

Column Size

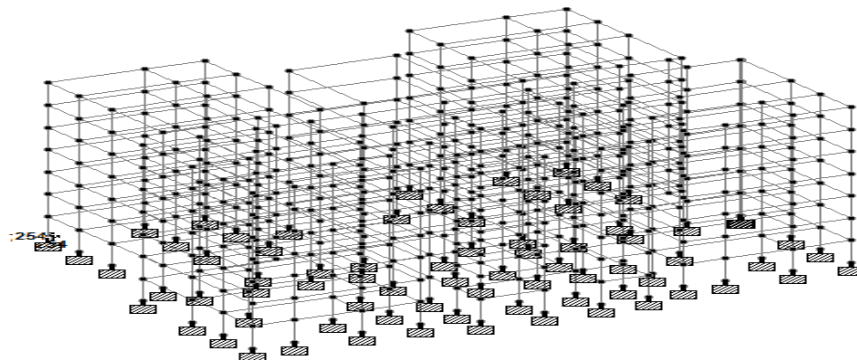
Load Cases

Support List

The multistoreyed frame represents the center line diagram of the building in staad pro. Each support represents the location of different columns in the structure. This structure is used in generating the entire structure using a tool called transitional repeat and link steps. After using the tool the structure that is created can be analyzed in staad pro under various loading cases.

Below figure represents the skeletal structure of the building which is used to carry out the analysis of our building. All the loadings are acted on this skeletal structure to carry out the analysis of our building. This is not the actual structure but just represents the outline of the building in staad pro.

A mesh is automatically created for the analysis of these building.



Load Conditions and Structural System Response :

The concepts presented in this section provide an overview of building loads and their effect on the structural response of typical wood-framed homes. As shown in Table, building loads can be divided into types based on the orientation of the structural action or forces that they induce: vertical and horizontal (i.e., lateral) loads. Classification of loads are described in the following sections.

Building Loads Categorized by Orientation:

Types of loads on an hypothetical building are as follows.

Vertical Loads

Dead (gravity)

Live (gravity)

Snow (gravity)

Wind (uplift on roof)

Seismic and wind (overturning)

Seismic (vertical ground motion)

Horizontal (Lateral) Loads:

Direction of loads is horizontal w.r.t to the building.

Wind

Seismic (horizontal ground motion)

Flood (static and dynamic hydraulic forces)

Soil (active lateral pressure)

Dead load calculation

Weight = Volume x Density

Self weight floor finish = $0.12 \times 25 + 1 = 3 \text{ kn/m}^2$

The above example shows a sample calculation of dead load.

Dead load is calculated as per **IS 875 part 1**

Generally for any structure live load is taken as **25 N/mm** for design.

Live loads are calculated as per **IS 875 part 2**

Wind loads:

In the list of loads we can see wind load is present both in vertical and horizontal loads. This is because wind load causes uplift of the roof by creating a negative (suction) pressure on the top of the roof. After designing wind load can be assigned in two ways

1. collecting the standard values of load intensities for a particular heights and assigning of the loads for respective height.
2. calculation of wind load as per **IS 875 part 3**.

We designed our structure using second method which involves the calculation of wind load using wind speed. In Hyderabad we have a wind speed of 45 kmph for 10 m height and this value is used in calculation.

Floor load:

Floor load is calculated based on the load on the slabs. Assignment of floor load is done by creating a load case for floor load. After the assignment of floor load our structure looks as shown in the below figure.

The intensity of the floor load taken is: **0.0035 N/mm²**

-ve sign indicates that floor load is acting downwards

Load combinations:

All the load cases are tested by taking load factors and analyzing the building in different load combination as per **IS456** and analyzed the building for all the load combinations and results are taken and maximum load combination is selected for the design. Load factors as per **IS456-2000**

When the building is designed for both wind and seismic loads maximum of both is taken.

Live load	dead load	wind load
1.5	1.5	0
1.2	1.2	1.2
0.9	0.9	0.9

Because wind and seismic do not come at same time as per code. Structure is analyzed by taking all the above combinations.

Result: The analysis of multi-storey frame is done by using staad pro.

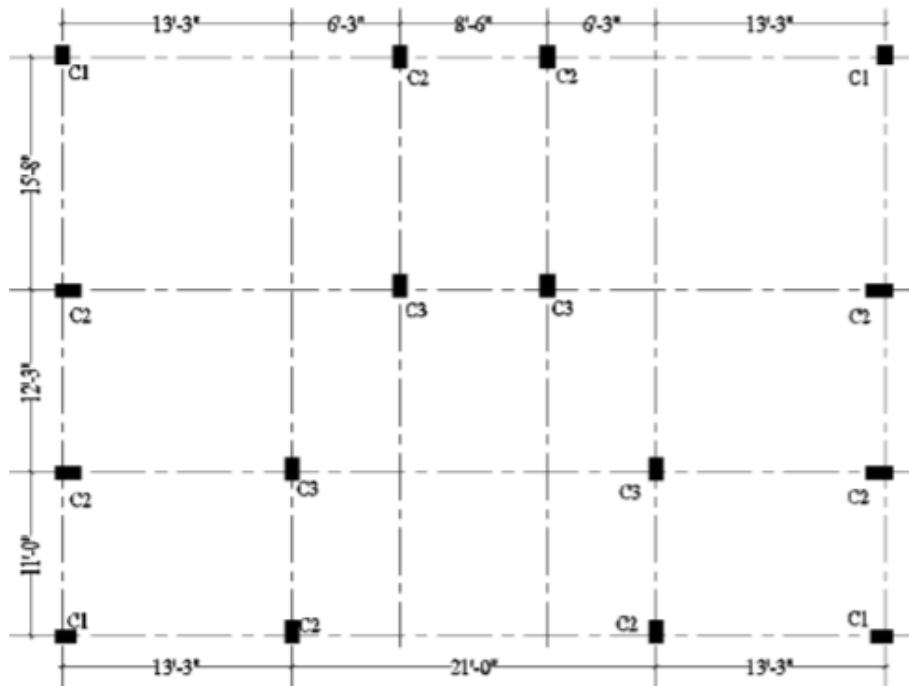
EXPERIMENT NO: 5

DESIGN OF MULTI-STOREY FRAME

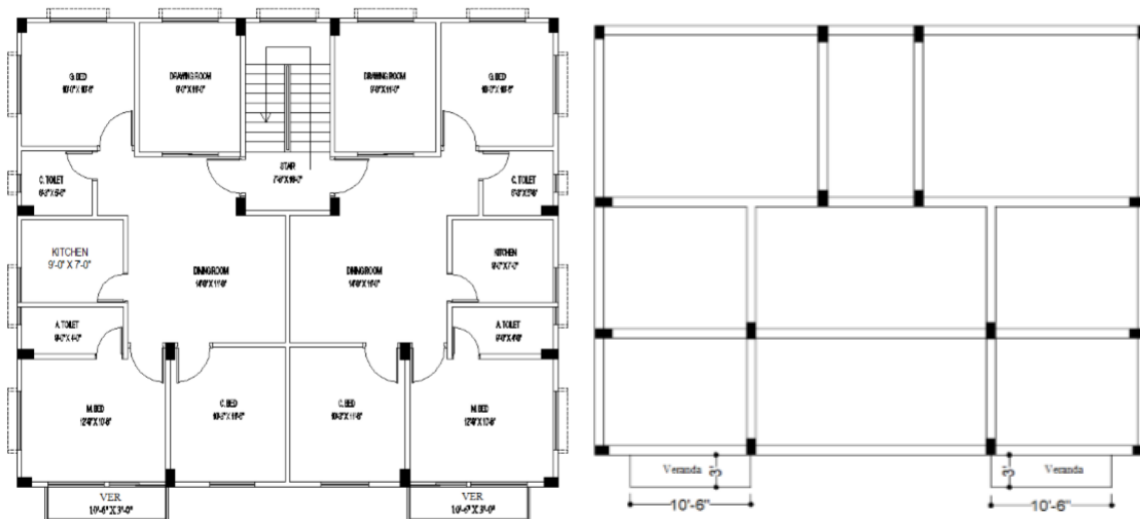
Aim: Analyze the following 7-Storeyed residential building under all loads and Columns, Beams forces and support reactions for foundation design

Software required:

STAAD PRO V8i



floor plan



column layout plan and beam layout plan



Elevation view

Geometry and Loads:

Properties:	Load Definitions:
Column: C1= 12"X15" C2=12"X18" C3=12"X20" Beam: GB = 10"X18" FB = 10"X20" Varendra Beam: 6" X 6" <ul style="list-style-type: none"> • 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 FDL _{EQx} = 0.9*DL+1.2*LL+1.32*EQx FDL _{EQz} = 0.9*DL+1.2*LL+1.32*EQz FDL _{Wx} = 0.9*DL+1.2*LL+1.2*Wx FDL _{Wz} = 0.9*DL+1.2*LL+1.2*Wz

Procedure:

Geometry (Model creating):

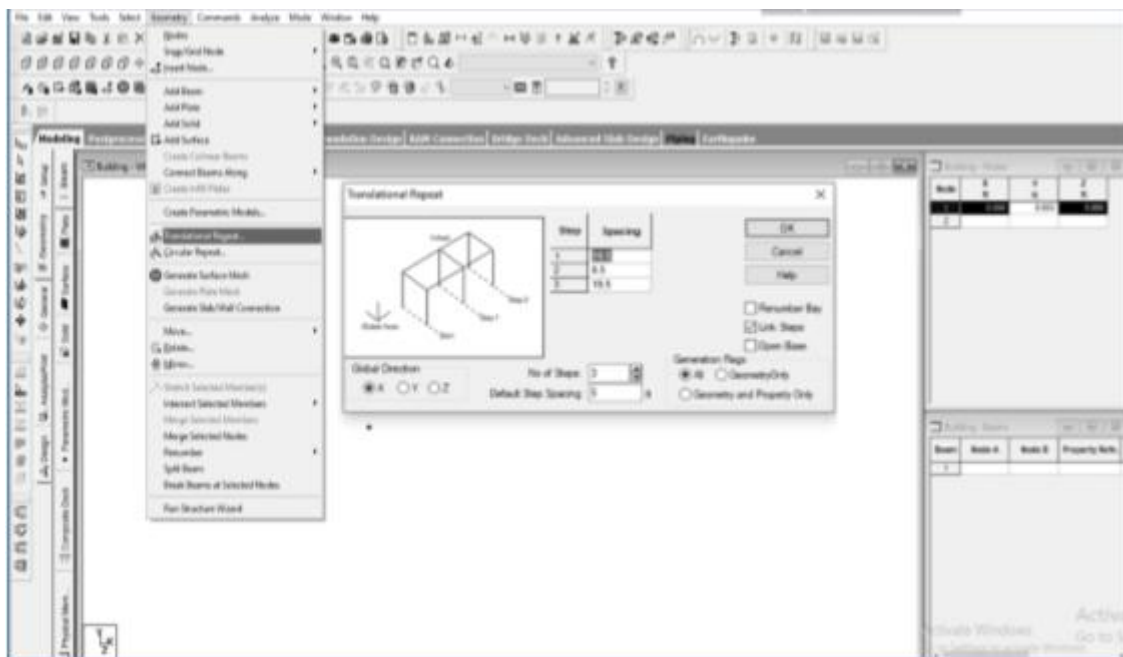
Open the STAAD Pro. software and click on New Project -Space >File name >Location (select your file location to save) >Length unit select Foot and KiloPound >Next >Add Beam >Finish. 1.2 Column & Beam Layout: 1.2.1 Close the default Grid system and at the right side input your first node point coordinates as (X Y Z)=(0 0 0)

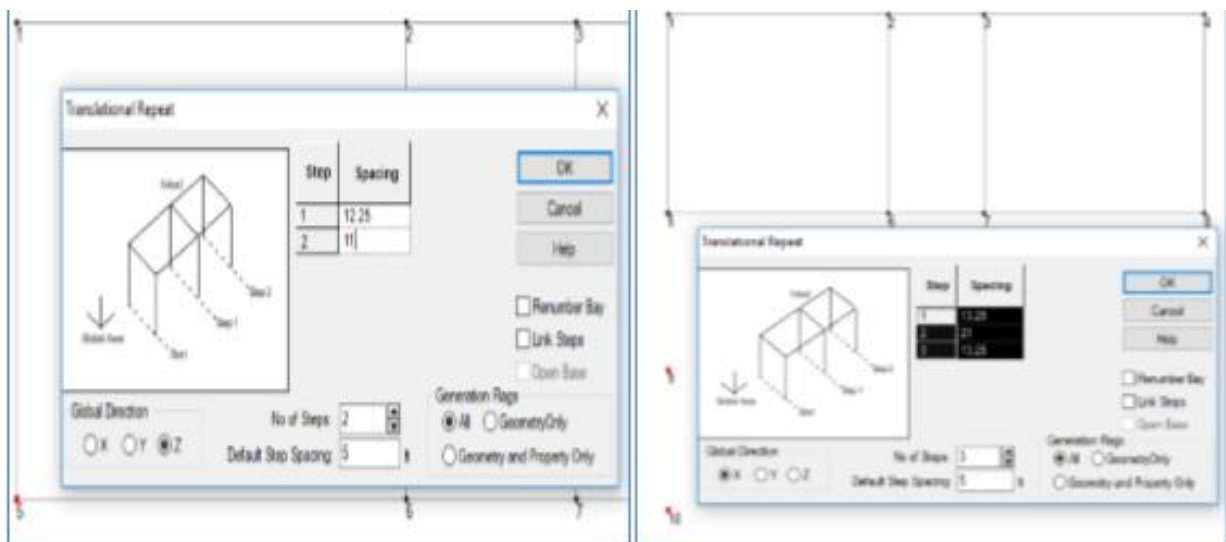
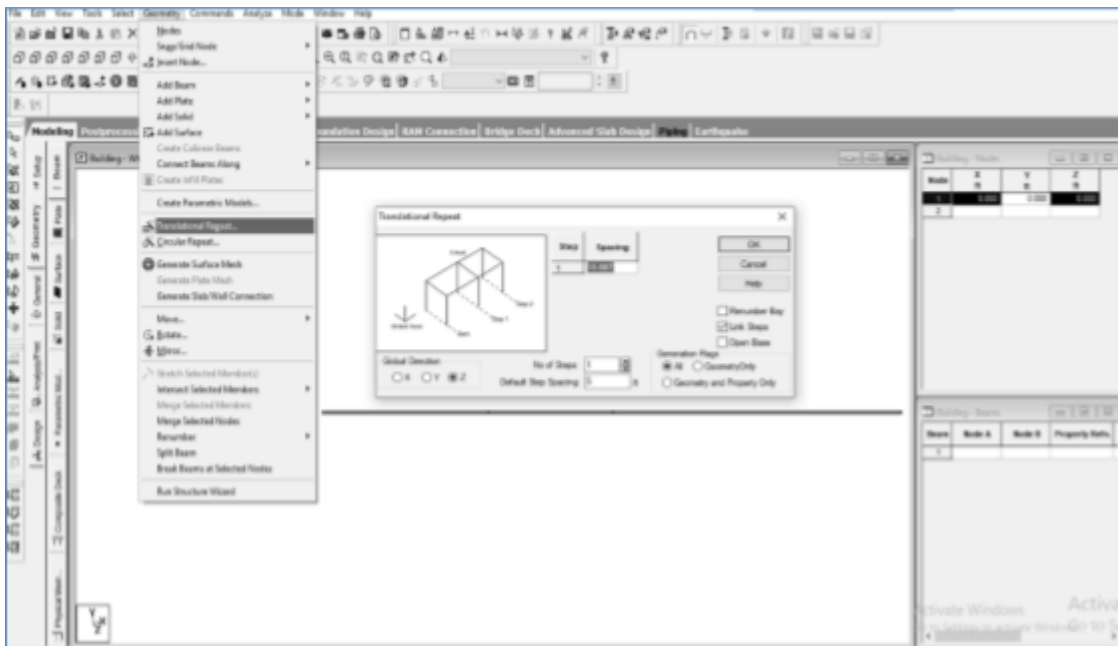
After than select the node by using node cursor >Geometry >Translational Repeat >Select Global Direction = X > No of Steps = 3 >now write down the column spacing from your Column Layout Plan as (Step1 =19.5, Step2 =8.5, Step3 =19.5) >Link Steps >OK.

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.

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

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





- For create beam layout: Go to Geometry >Add Beam >Add Beam from Point top 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
- Veranda Create: Now select the node no 12 >Geometry >Trtranslational Repeat Select Global Direction = X> No of Steps =1 > now write down the Veranda length as (Step1 = -10.5) >OK
- 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.

- 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

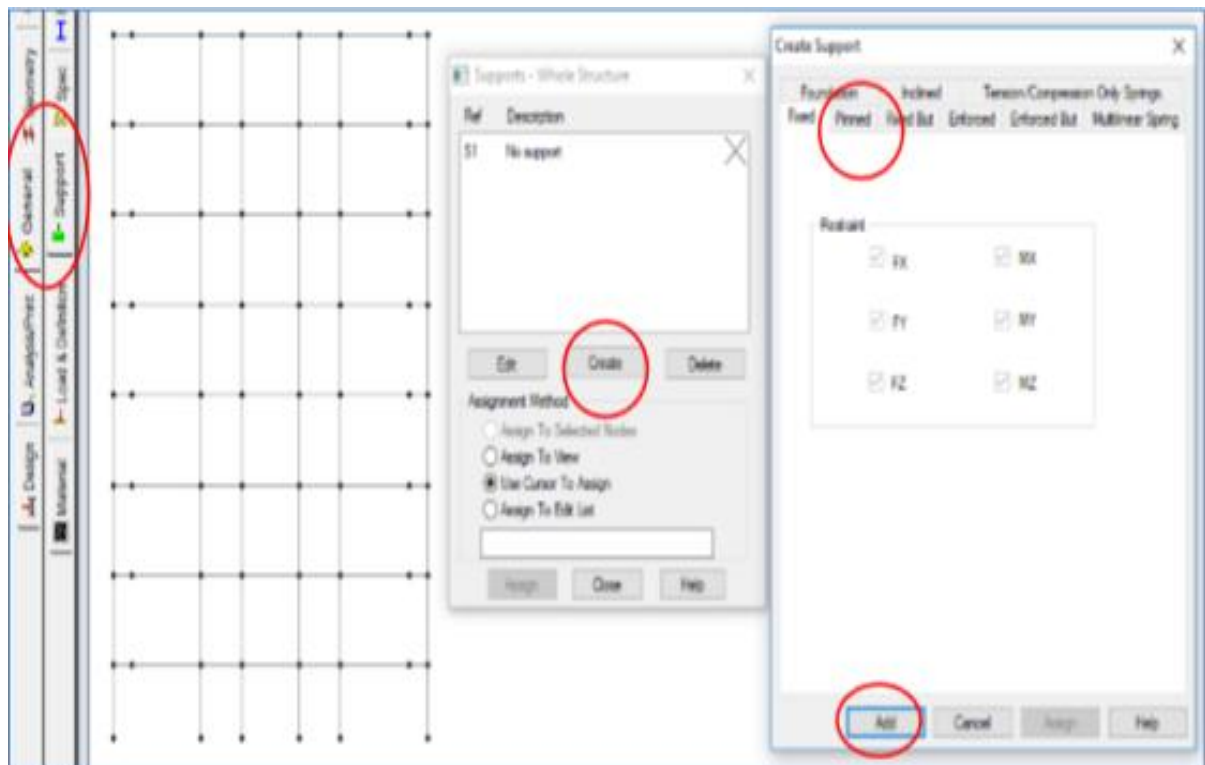
General (Define & Assign):

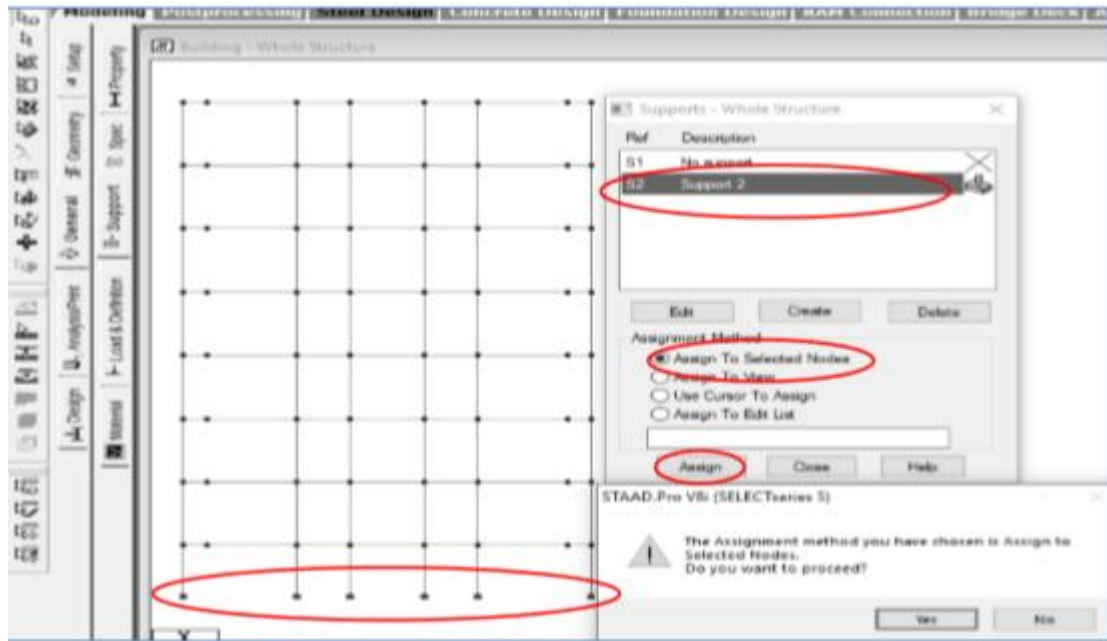
Support Create and Assign:

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

From left side Click on General >Support >Create > Fixed >Add

Select S2 Support 2 >Select all bottom Nodes by using Node Cursor >Assign to Selected Nodes >Assign >Yes.





Result: The multi-Storeyed building is designed by using Staad. Pro software

EXPERIMENT NO: 6

ANALYSIS OF MULTI-STOREY BUILDING

Aim: To create and analyze the multi storey building by using STAAD Pro

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

Preprocessing:

1. *Create model*

- a) New project> plane> length units as m> force units as KN>type the file name as multi storey building>next> add beam.
- b) Geometry> run structural wizard> frame model> bay frame>type the length as 12.6, no.of bays along length as 3, height as 12.6, no. of bays along height as 4> apply & transfer the model to STAAD Pro screen.

2. *Property*

- a) General> property> define> rectangular>0.35x 0.23> add> close
- b) Select rectangular and assign to view.

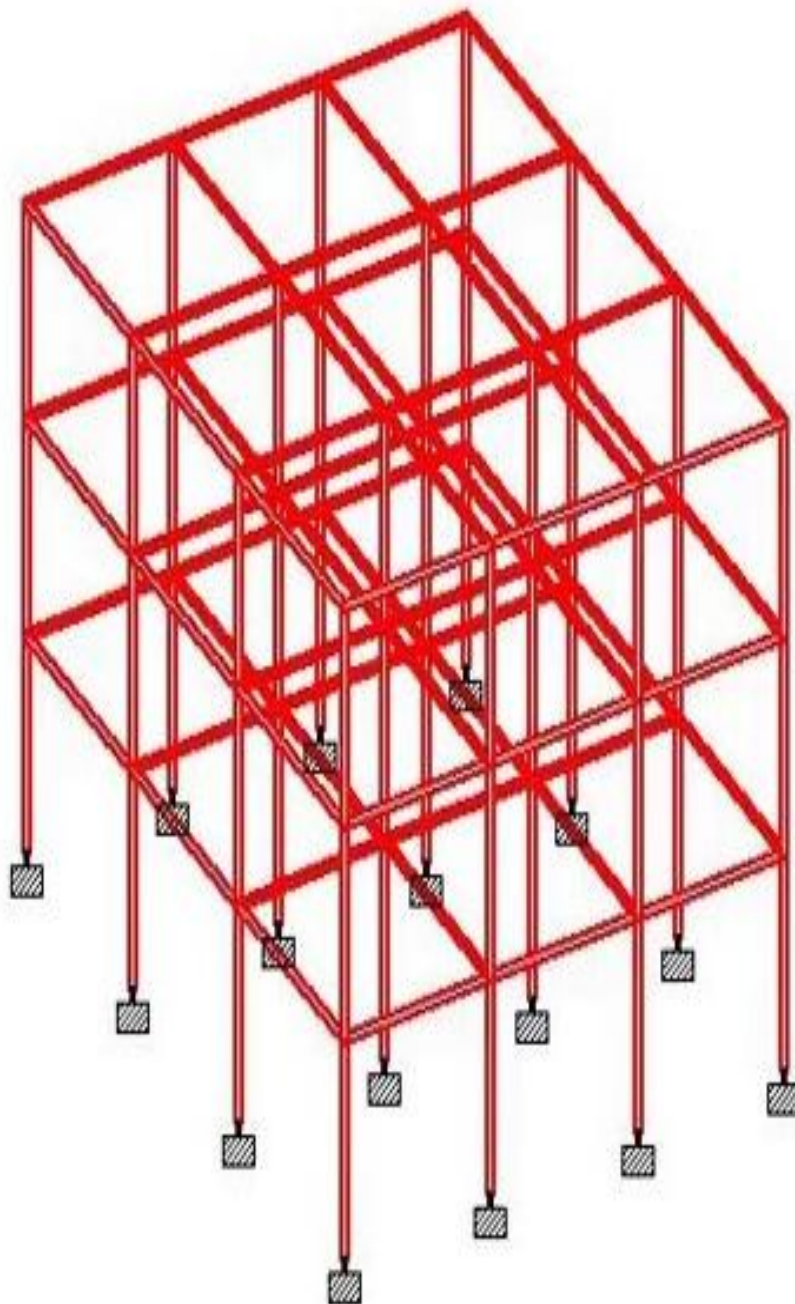
3. *Support*

Support> fixed support> select the bottom nodes> assign selected nodes.

4. *Loads*

- a) Load and definition> load case detail> add>change the title as DL>add> then change LL> add> WL> add> close.
- b) Select DL> add> member load> uniform load> type W1 as -40KN/m < add.
- c) Select LL > add> member load> uniform load> type W1 as -20KN/m < add.
- d) Select WL > add> nodal load> type X as 25> add> type X as 20> add> type X as 15> add> type X as 10> add>close>Select DL> click select > member parallel to x direction> assign selected beams.
- e) Similarly, click select > member parallel to x direction> assign selected beams.
- f) WL> select node cursor> select the particular node> assign selected node.

For load combinations, go to load case details> add> define combination> type name as DL+LL> FOS as 1.5>in available load case, select DL & LL and send it other side> add. Select DL & LL and send it other side> add.



Postprocessing

- g) Then similarly type name as DL+LL+WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.
- h) Then similarly type name as DL+LL-WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL & LL and send it other side> add. After that select WL and send it other side> add.
- i) Then similarly type name as DL+LL+WL> FOS as 1.2>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.

5. Design parameters

- a) Design concrete> IS 456>define parameter> give Fc as 20000 KN/m² for M20 concrete >Fc as 30000 KN/m² for M30 concrete>add> FY as 415000 KN/m²> add> select> ratio as 3>add.
- b) Then go to commands>design beam> add>design column> add>take off > close.
- c) Select Fc> select entire structure using cursor> assign it to selected beams> similarly do it for FY and ratio then assign it.
- d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

6. Analysis:

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

- a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.
- b) Results> view value> beam results> maximum displacement> activate.
- c) You can also see the animations of displacement in it.

Result:

Thus the multi storey building is created and analyzed by using STAAD Pro.

EXPERIMENT NO: 7

DESIGN OF MULTI-STOREY BUILDING

Aim:

To design the multi storey building by using STAAD Pro

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

Preprocessing:

1. Create model

- a) New project> plane> length units as m> force units asKN>type the file name as multi storey building>next> add beam.
- b) Geometry> run structural wizard> frame model> bay frame>type the length as 12.6, no.of bays along length as 3, height as 12.6, no. of bays along height as 4> apply & transfer the model to STAAD Pro screen.

2. Property

- a) General> property> define> rectangular>0.35x 0.23> add> close
- b) Select rectangular and assign to view.

3. Support

Support> fixed support> select the bottom nodes> assign selected nodes.

4. Loads

- a) Load and definition> load case detail> add>change the title as DL>add> then change LL> add> WL> add> close.
- b) Select DL> add> member load> uniform load> type W1 as -40KN/m < add.
- c) Select LL > add> member load> uniform load> type W1 as -20KN/m < add.
- d) Select WL > add> nodal load> type X as 25> add> type X as 20> add> type X as 15> add> type X as 10> add>close>Select DL> click select > member parallel to x direction> assign selected beams.
- e) Similarly, click select > member parallel to x direction> assign selected beams.
- f) WL> select node cursor> select the particular node> assign selected node.

For load combinations, go to load case details> add> define combination> type name as DL+LL> FOS as 1.5>in available load case, select DL & LL and send it other side> add. Select DL & LL and send it other side> add.

Postprocessing

- a) Then similarly type name as DL+LL+WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.
- b) Then similarly type name as DL+LL-WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL & LL and send it other side> add. After that select WL and send it other side> add.
- c) Then similarly type name as DL+LL+WL> FOS as 1.2>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.

6. Design parameters

- a) Design concrete> IS 456>define parameter> give Fc as 20000 KN/m² for M20 concrete >Fc as 30000 KN/m² for M30 concrete>add> FY as 415000 KN/m²> add> select> ratio as 3>add.
- b) Then go to commands>design beam> add>design column> add>take off > close.
- c) Select Fc> select entire structure using cursor> assign it to selected beams> similarly do it for FY and ratio then assign it.
- d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

6. Analysis:

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

- a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.
- b) Results> view value> beam results> maximum displacement> activate.
- c) You can also see the animations of displacement in it.

Result:

Thus the multi storey building is designed by using STAAD Pro.

EXPERIMENT NO: 8
WIND LOAD ANALYSIS ON RCC BUILDING

Aim: To analyze the RCC building by using STAAD Pro.

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

Geometry (Model creating):

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.

Column & Beam Layout:

Close the default Grid system and at the right side input your first node point coordinates as (X Y Z)=(0 0 0)

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.

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.

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

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

Properties:	Load Definitions:
Column: C1= 12"X15" C2=12"X18" C3=12"X20" Beam: GB = 10"X18" FB = 10"X20" Varendra Beam: 6" X 6" <ul style="list-style-type: none"> • 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

- For create beam layout: Go to Geometry >Add Beam >Add Beam from Point top 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
- Veranda Create: Now select the node no 12 >Geometry >Trtranslational Repeat Select Global Direction = X> No of Steps =1 > now write down the Veranda length as (Step1 = -10.5) >OK
- 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.
- 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

General (Define & Assign):

Support Create and Assign:

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

From left side Click on General >Support >Create > Fixed >Add

Select S2 Support 2 >Select all bottom Nodes by using Node Cursor >Assign to Selected Nodes >Assign >Yes.

Result: The RCC building is analyzed by using Staad. Pro software

EXPERIMENT NO: 9

ANALYSIS AND DESIGN OF STEEL TRUSS

Aim: To analyze the Truss using STADD pro.

Software:

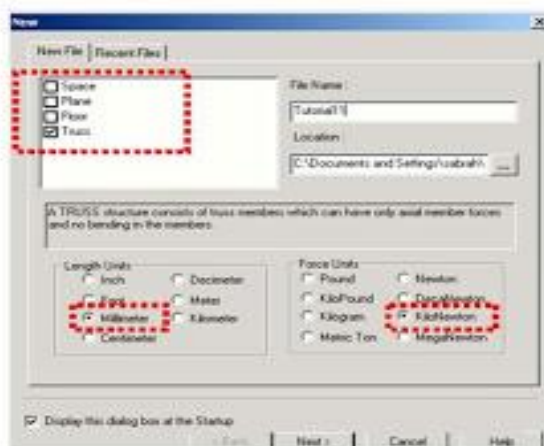
STADD pro software, compute,

PROCEDURE:

1. Entering Job information.
2. Truss model geometry
3. Defining member properties ,sections
4. Assigning Loads(Load Cases and combination)
5. Defining Pre-Analysis Print out, analysis type and Post-Analysis print out
6. Defining Design Requirements

How to Start Staad Pro:

The Pic below shows how to start Staad Pro



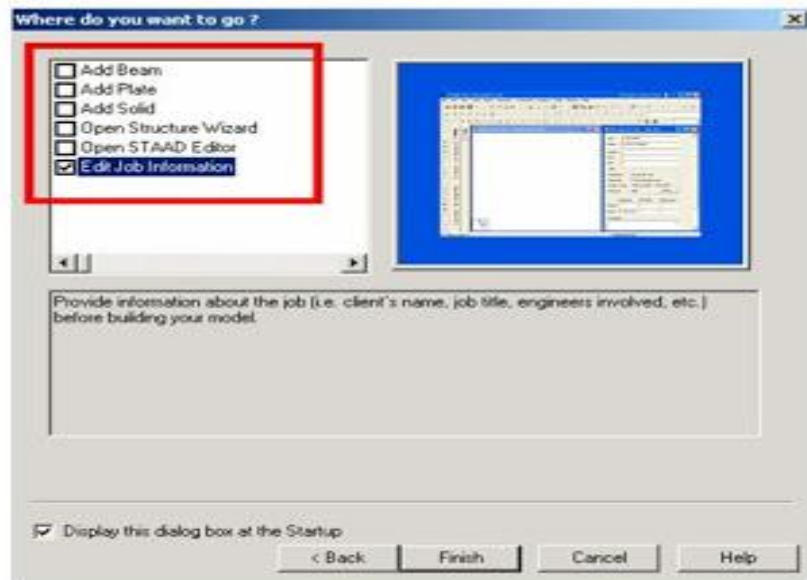
For a new file, select the type of structure you want to solve. Here select truss type.

Select from here the units, that we will deal with.

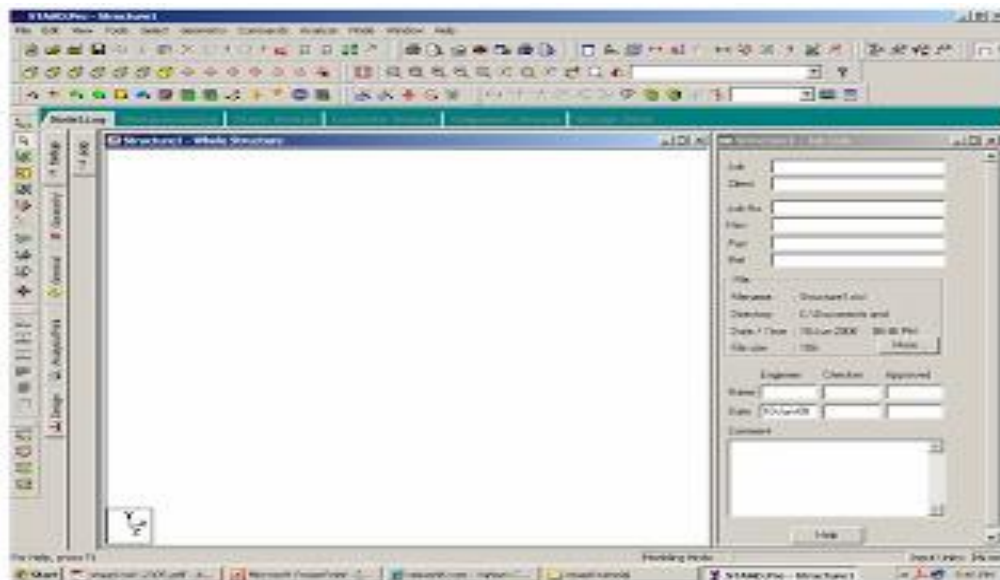
Assume that we have mm, kN.

In New File select

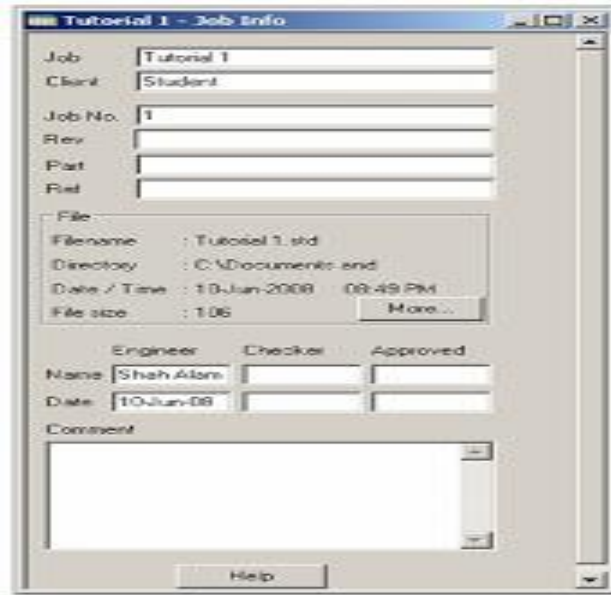
1. Truss
2. Units (here KN and mm in this example)



The staad Graphical interface will appear as shown in below picture



1. Entering Job Information

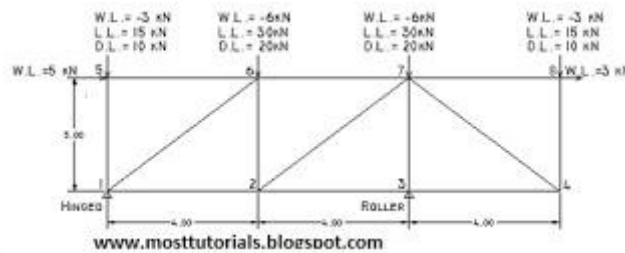
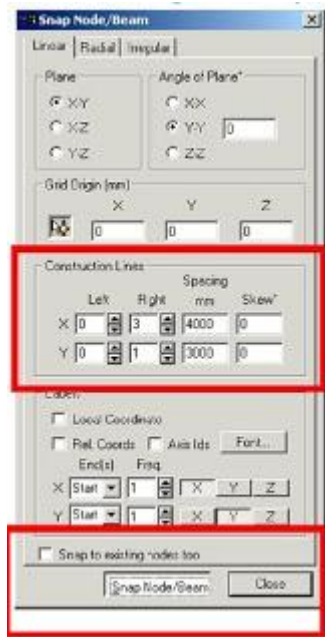


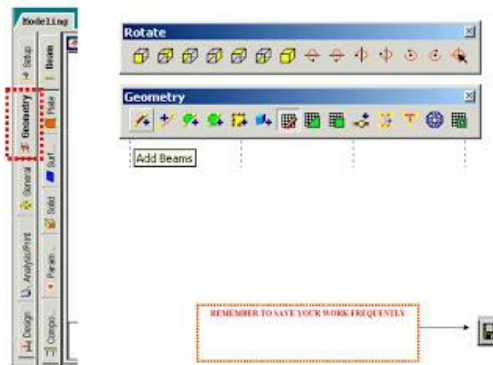
Building Model (structure) Geometry

- Defining Truss Geometry

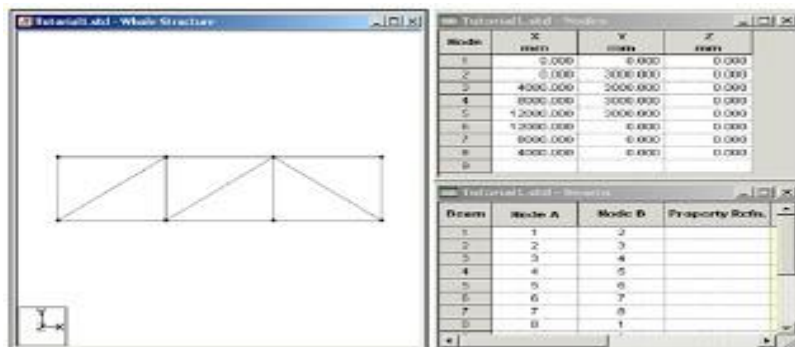
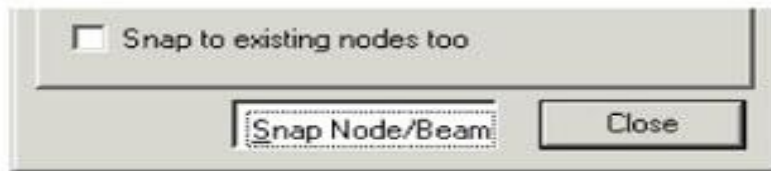
One of the methods that you can construction lines and then draw on these lines the truss members

Noting that the no. of construction lines is excluding 1st line..

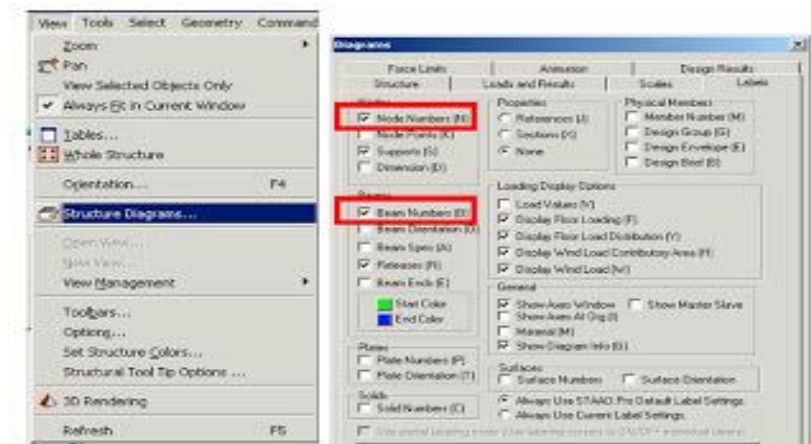


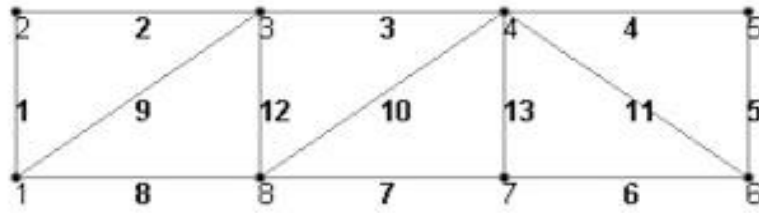


After clicking Snap Node/Beam, use the mouse and connect between nodes created at the intersections of construction lines



How to see Diagram Labels (Nodes Numbers, Beam Numbers etc)





Node and beam labels are a way of identifying the entities we have drawn on the screen, and very useful when dealing with the output results

3. Defining member properties, sections



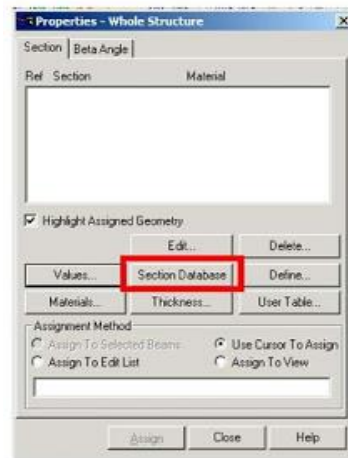
Property

In which we can define or choose sections properties of the members of the truss.

- Spec.: In which we can define or choose members' specifications.
- Support: In which we can define the supports properties (restraints).
- Load: In which we can define the applied loads, load cases, load combinations.
- Material In which we can define the material properties (ex. E, density, etc.)

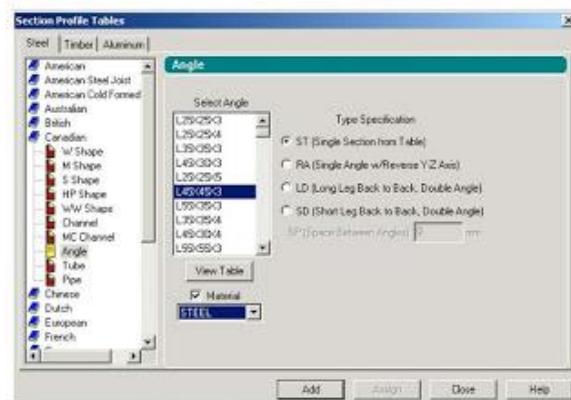
Assume Preliminary

- Sections:

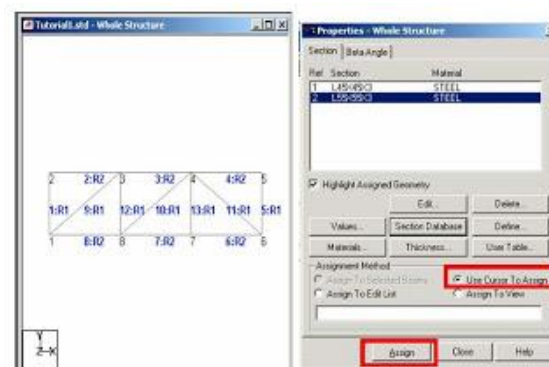


All Top and Bot chords are one size **L 55*55*3**

All Diagonals and verticals are one size **L 45 * 45* 3**

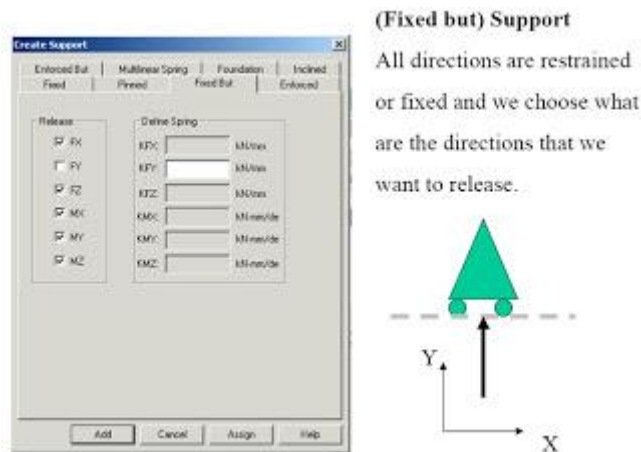
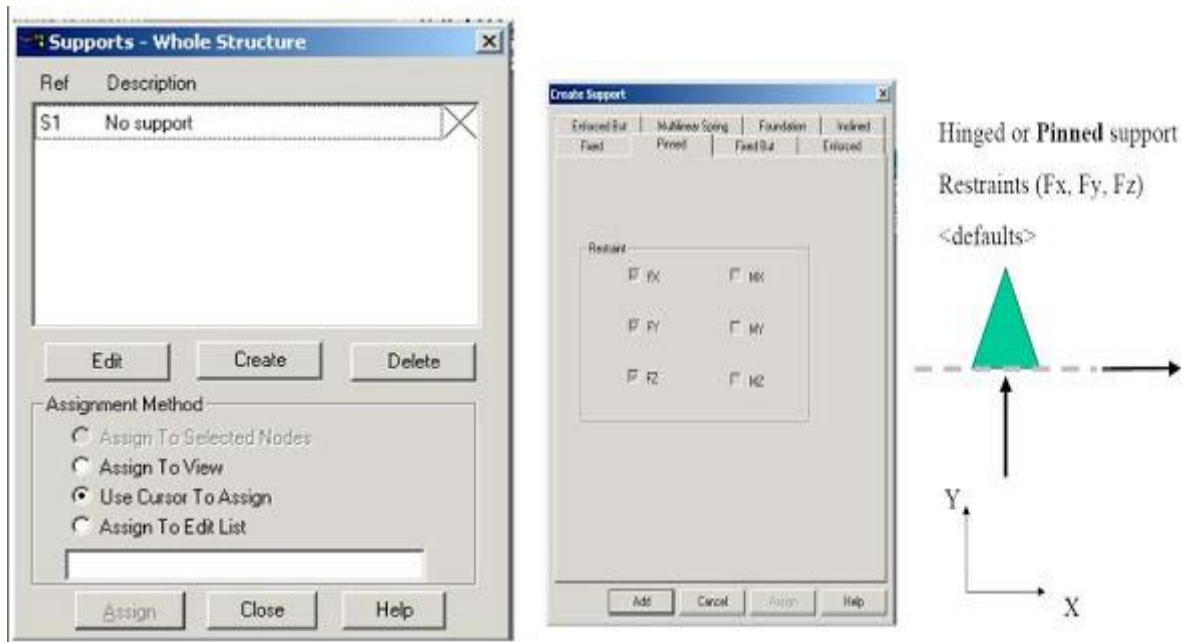


Assigning the sections created to the model

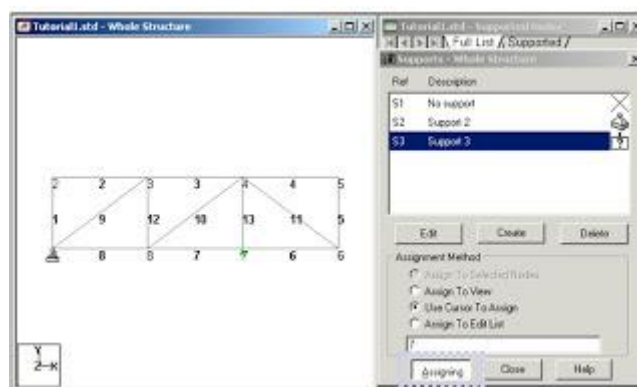


- Supports

By clicking on the support icon the shown window will appear. We have to create new kinds of supports



Use the mouse and click on the nodes according to its support type as shown below

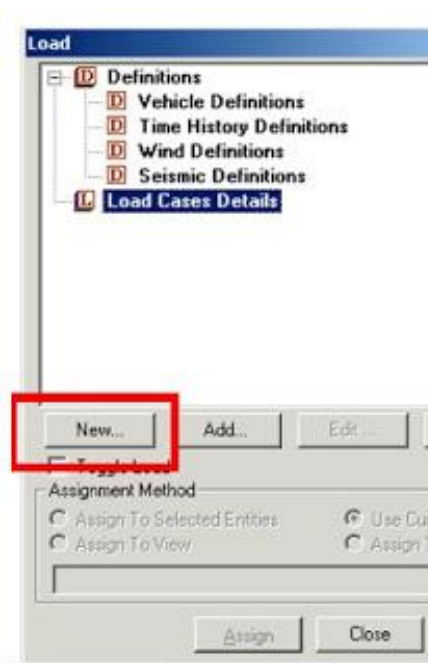


- **4. Defining Loads**

The creation and assignment of load cases involves the following two steps:

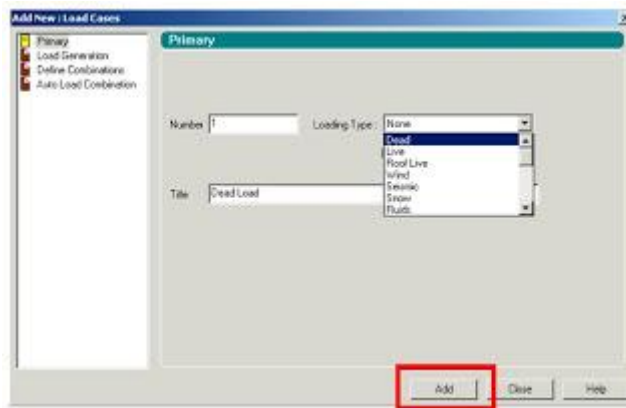


1. First, we will be creating all 3-load cases.
2. Then, we will be assigning them to the respective members/nodes

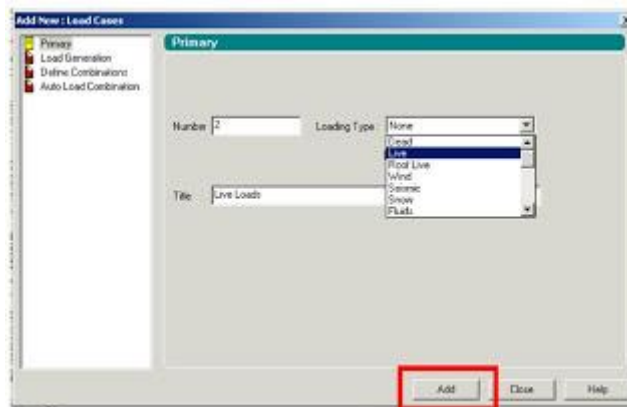


For example

Dead Load Case can be Load Case No.1



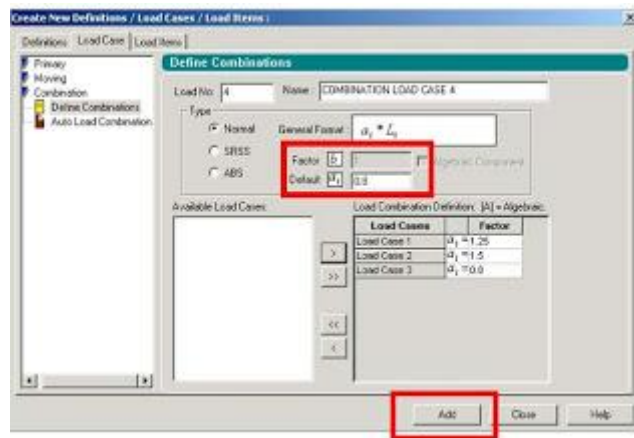
Live Load Case can be Load Case No. 2 and Wind Load is the Case No.3



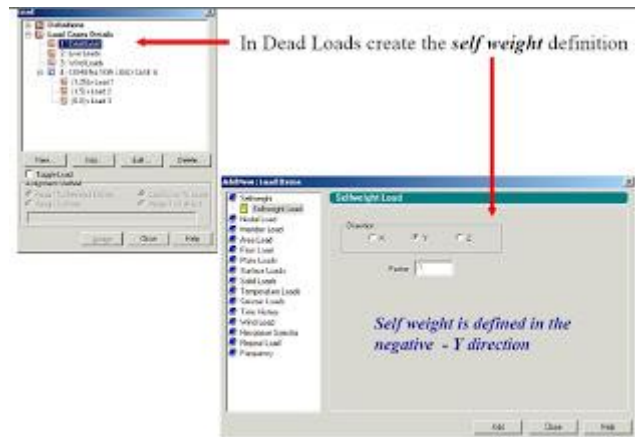
Also, we can define **load combinations** according to required.

For example, we can create a load combination

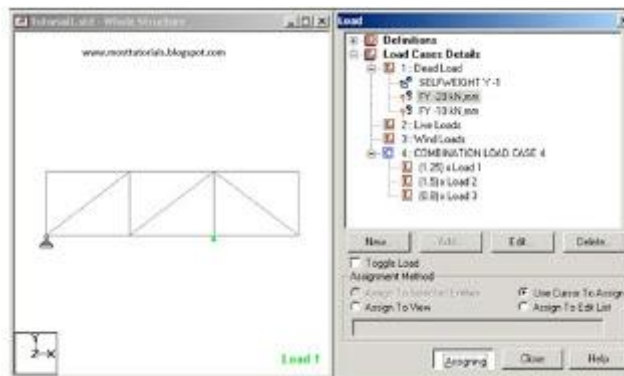
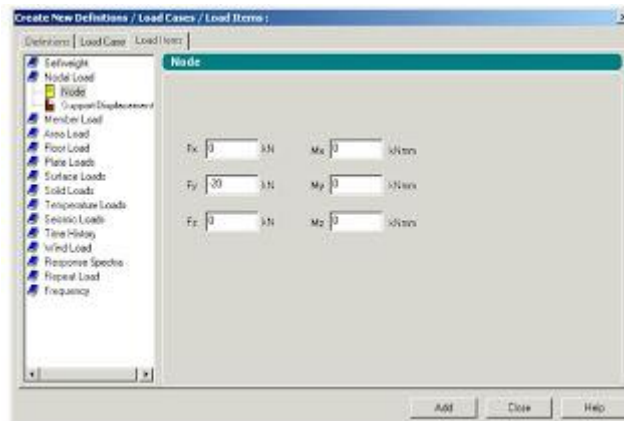
$$1.25 \text{ D.L.} + 1.5 \text{ L.L.} + 0.8 \text{ W.L.}$$

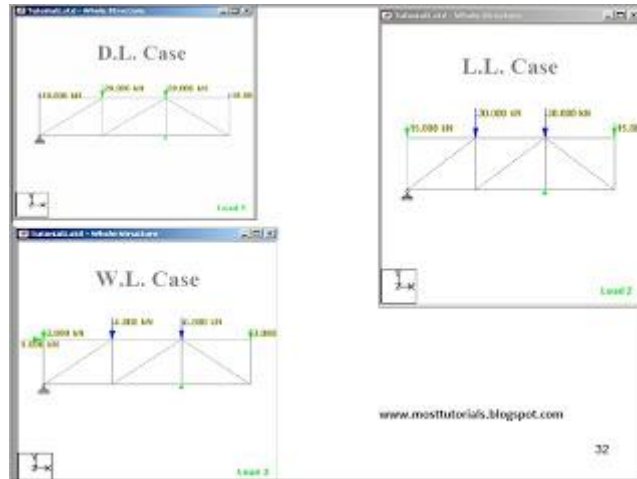


And In Dead Load Create the Self weight as

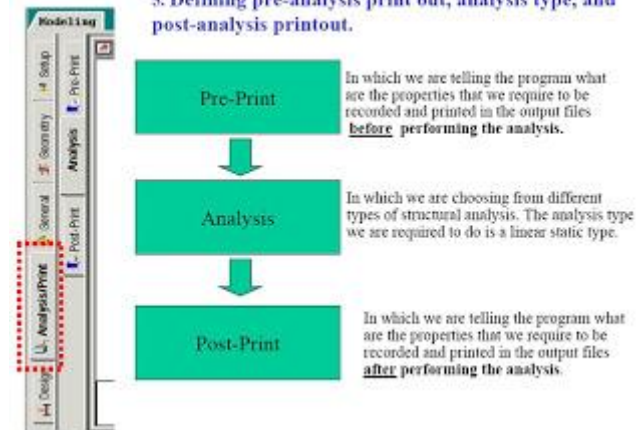


In each Load Case Create the Nodal Loads on truss as specified before. Make sure of the direction of forces according to global coordinates





5. Defining pre-analysis print out, analysis type, and post-analysis printout.



To specify the Analysis command, first go to **Analysis/Print** Page from the left side of the screen.

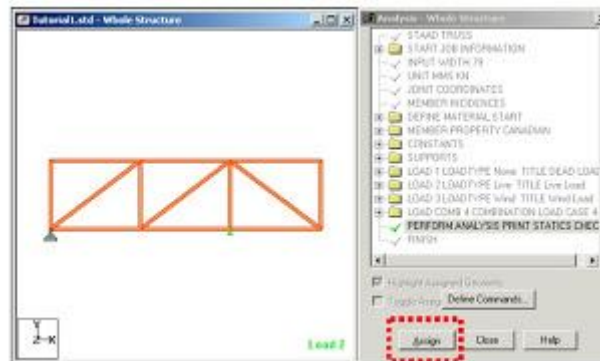
Then, click on the Analysis sub-page from the second row of pages as shown below.

Choose Statics Check and Click **Add** then **Close**.

After Creating the command of (Perform Analysis-Check); We have to assign the members that want to be this type of analysis for it.

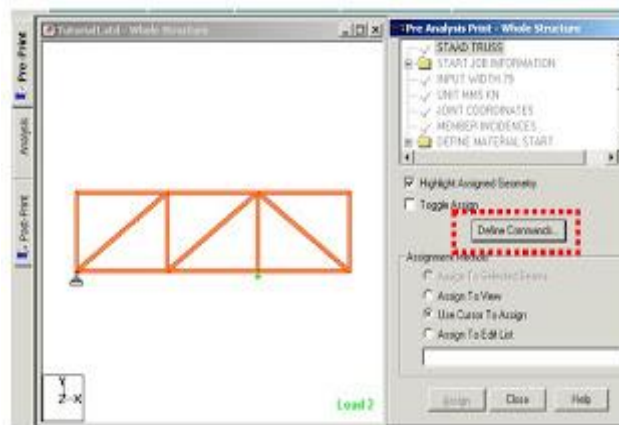
Click **Assign** and then use the cursor and choose all the members, thus all the members should

be highlighted as shown below.

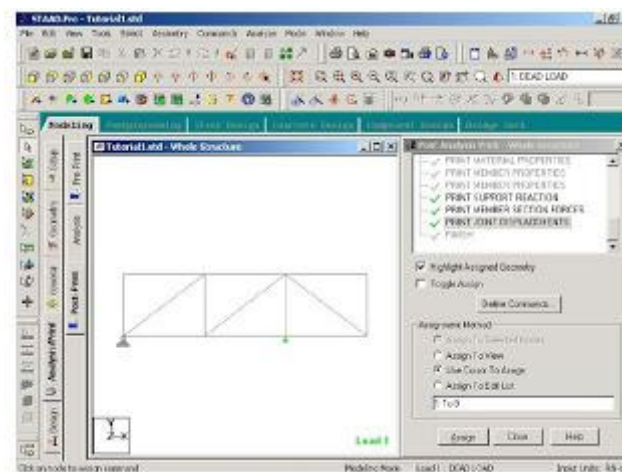


Pre-Print

To add the Pre-Print Command click " Define Commands"



For example here we have chosen to print the support reactions,member forces,joint displacements

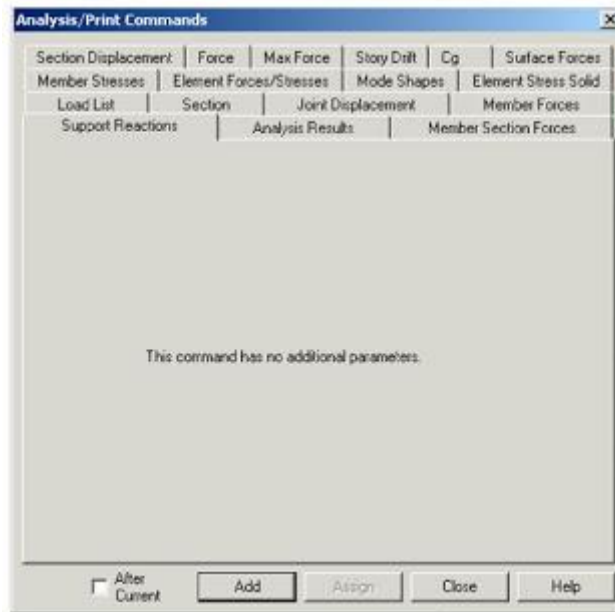


Post Print Commands

- Support reactions
- Analysis results

- Member Forces
- Max Forces

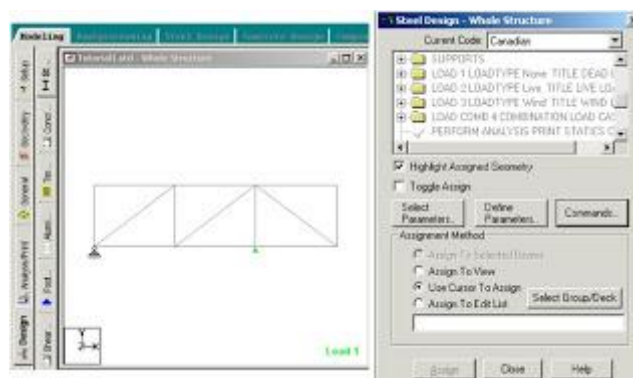
Add which property you need and then use cursor to ASSIGN to which member in the truss



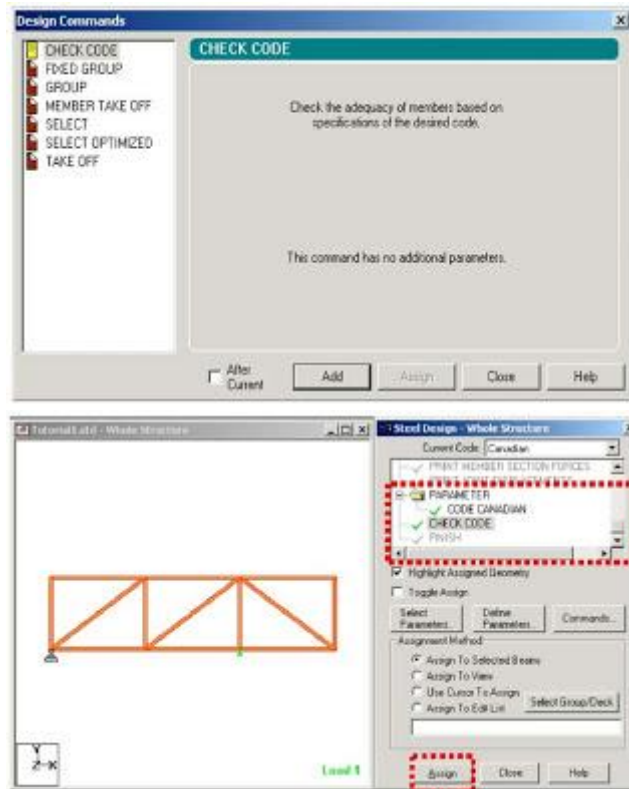
6. Defining Design Requirements

Steps

1. To Specify steel design parameters, go to Design/Steel page from the left side of the screen. Make sure that under the Current Code selections on the top right hand side ,Canadian is selected.

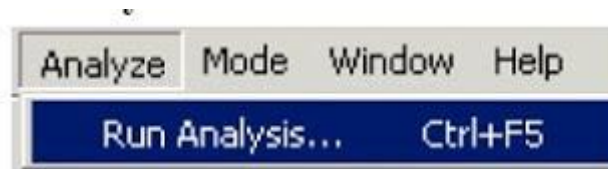


There are many Design Commands in the STAAD Design subroutine. Here, we will use only to **Check Code**, regarding adequacy of members.



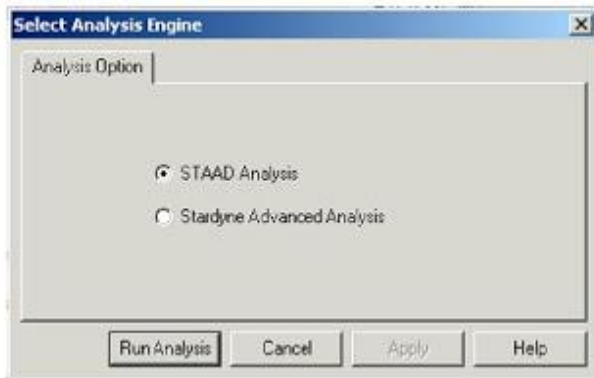
7-Analysis & Viewing Results

STAAD Performs Analysis and Design simultaneously. In order to Perform Analysis and Design, select the Run Analysis option from the **Analyze** menu

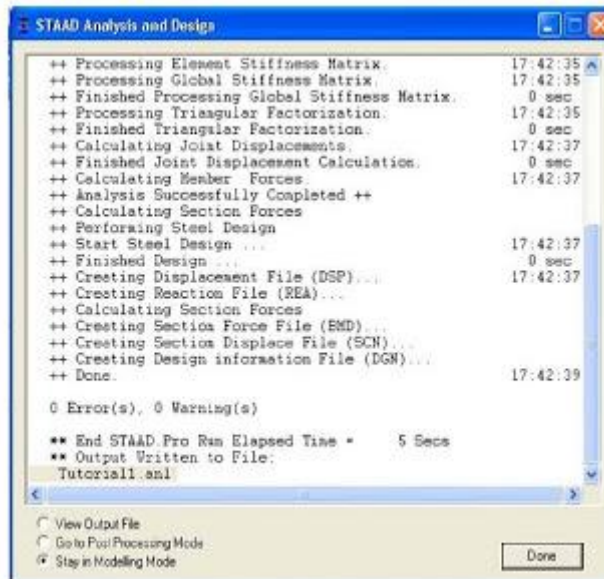


When you select the Run Analysis option from the Analyze menu, the following dialog box appears. We are presented with the choice of 2 engines. The **STAAD engine** and the **STARDYNE Advanced Analysis engine**. The **STARDYNE** Analysis engine is suitable for advanced problems such as **Buckling Analysis, Modal Extraction** using various method ,etc

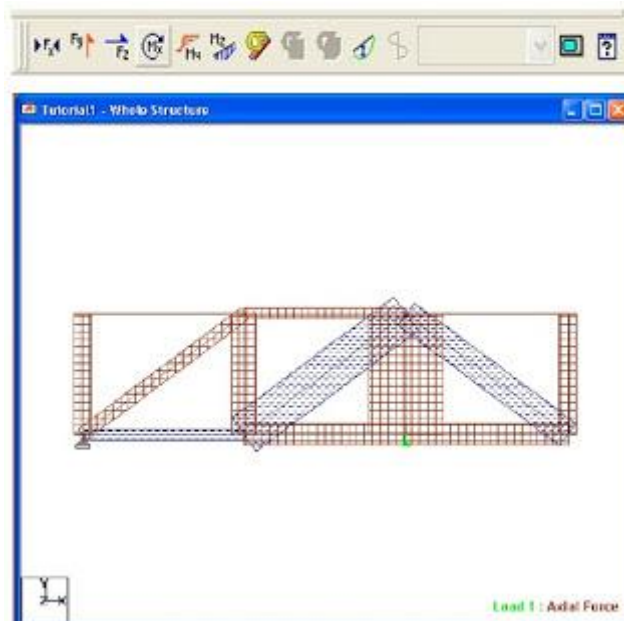
STAAD engine is suitable for this tutorial. Click on th **Run Analysis** button.



The solving process is shown in pop up screen.



Visualization of Some Results



EXPERIMENT- 10

ANALYSIS AND DESIGN OF ISOLATED FOOTING

Aim: To analyze and design isolated footing

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed Microsoft Office

Reinforced Concrete Footings:

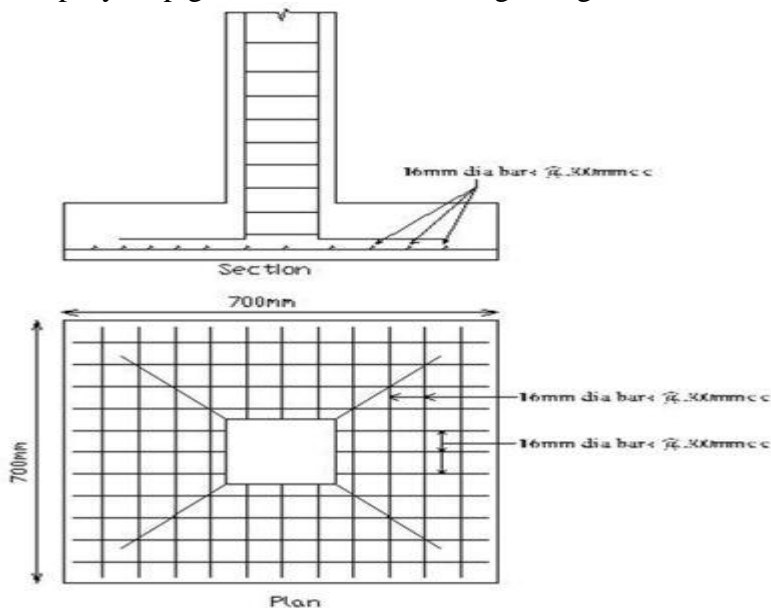
Footing comprises of the lower end of a column, pillar or wall which is enlarged with projecting courses so as to distribute load.

Footings shall be designed to sustain the applied loads, moments and forces and the induced reactions and to ensure that any settlement which may occur shall be as uniform as possible and the safe bearing capacity of soil is not exceeded.

In sloped or stepped footings, the effective cross-section in compression shall be limited by the area above the neutral plane, and the angle of slope or depth and location of steps should be such that the design requirements are satisfied at every section.

Design Procedure of Column Footings | Foundation Design

Here is a step-by-step guide to Column Footing Design:



Step 1

Area required for footing

$$\text{Square} = B = (w+w_1)/P_0$$

Where, P_0 = safe bearing capacity of soil

w_1 = self weight of footing

w = self weight of footing

For Rectangle = $b/d = B/D$

$$A = b \times d$$

Net upward pressure on the footing

$$q/p = W/A$$

Step 2

Bending Moment

Critical section for maximum bending moment is taken at the face of the column

For a square footing,

$$M_{xx} = q \times B/8 (L - a)^2$$

$$M_{xx} = q \times L/8 (B - b)^2$$

$$M_{yy} = q \times B/8 (L - a)^2$$

Step 3

To fix the depth of the footing shall be greater of the following:

Depth from bending moment consideration

$$d = \sqrt{(M/Qb)}$$

where, Q = moment of required factor

Depth from shear consideration

Check for one way shear

Check for two way shear or punching shear

Critical shear for one way shear is considered at a distance 'd' from face of the column.

$$\text{Shear force, } V = qB [\frac{1}{2}(B - b) d]$$

Nominal shear stress, $T_v = k \cdot T_c$

$$T_c = 0.16\sqrt{f_{ck}}$$

Step 4

Check for two way shear

Critical section for two way shear is considered at a distance at a distance d/2 from all the faces of the column.

$$\text{SF, } V = q [B^2 - (b + d)^2]$$

$$\text{SF, } V = q [L \times B - (a + d)(b + d)]$$

Nominal shear stress, $T_v = V/2((a+d)(b+d)d)$ ——— { for a rectangle

$T_v = V/4((b+d)d)$ ——— { for a square

$$T_v = k \cdot T_c$$

$$k = 0.5 + \beta > 1$$

; [Beta β = ratio of sides of the column

$$T_c = 0.16\sqrt{f_{ck}}$$

$$\text{Area of steel, } A_{st} = M/((\sigma)stjd)$$

ISOLATED FOOTING WITH BIAXIAL MOMENT

CALCULATION FOR BOTTOM STEEL

Mu about X1 X1 = (pe max x length²/2)=

$$A_{st} = \frac{0.5 f_{ck}}{f_y} \left[1 - \sqrt{1 - \frac{4.6 M_u}{f_{ck} b d^2}} \right] b d$$

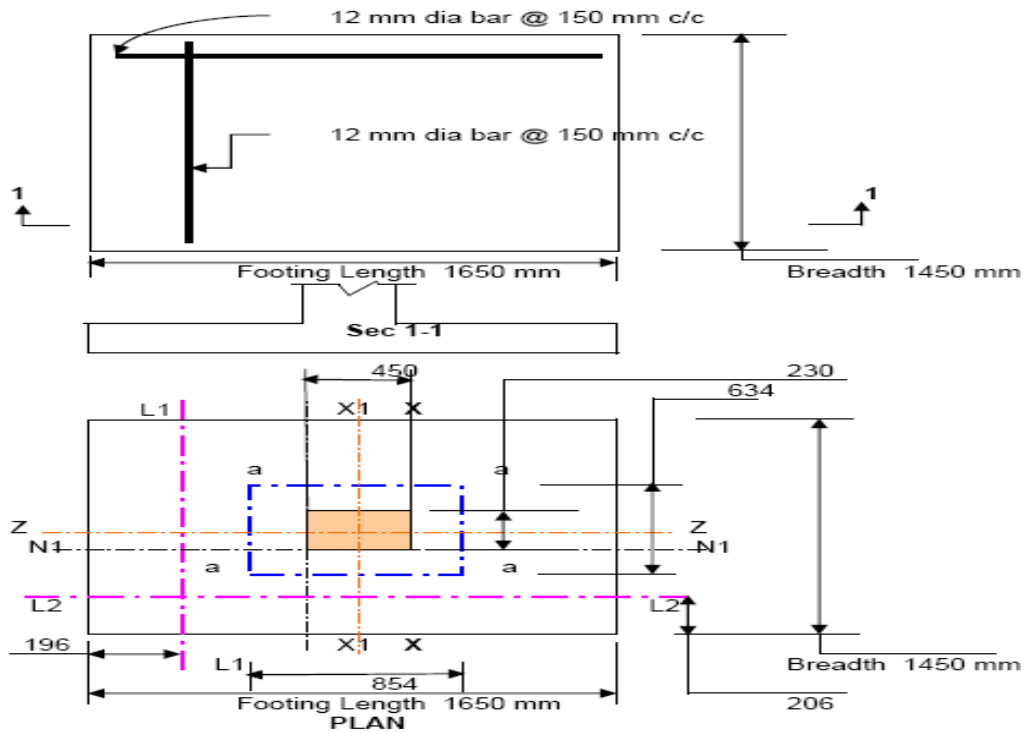
90.27 KN-m per meter
 Mulimit = 450.81 KN-m per meter
 The section is singly reinforced

Hence, Ast = 640.194 mm²
 Min Ast = 484.800 mm² (0.12 % for slab, cl 26.5.2.1)
 Spacing required = 176.66 mm (considering max of above two calculated values of Ast)
 Spacing provided = 150 mm
 pt provided = 0.19 %
 pt required = 0.16 %
 provide a maximum : 0.19 %
 Hence provide 12 mm dia bar @ 150 mm c/c pallel to length of footing (|| to Z)

Mu about N1 N1 = (pe max x length²/2)=

Calc. Ast = 662.496 mm² The section is singly reinforced
 Min Ast = 484.8 mm² (0.12 % for slab, cl 26.5.2.1)
 Spacing = 170.71 mm (considering max of above two calculated values of Ast)
 Spacing provided = 150 mm
 pt provided = 0.19 %
 pt required = 0.16 %
 provide a maximum : 0.19 %
 Hence provide 12 mm dia bar @ 150 mm c/c pallel to breadth of footing (|| to X)

Arrangement of bottom reinforcement as per above design is shown below



CHECK FOR ONE WAY SHEAR :

One way shear at critical section L1- L1

Distance of critical sec. from edge of footing = **0.196 m**
 Shear force $V_s = p_e \max \times 0.196 \times 1\text{m width of footing} =$ **98.289 KN**
 Shear stress $\tau_v = V_s/bd =$ **0.243 N/mm²**
 $\tau_c =$ **0.32 N/mm²**

tv < tc hence O.K.

One way shear at critical section L2- L2

Distance of critical sec. from edge of footing = **0.206 m**
 Shear force $V_s = p_e \max \times 0.206 \times 1\text{m width of footing} =$ **103.304 KN**
 Shear stress $\tau_v = V_s/bd =$ **0.256 N/mm²**
 $\tau_c =$ **0.32 N/mm²**

tv < tc hence O.K.

CHECK FOR TWO WAY SHEAR

Ref. cl 34.2.4 and cl.31.6.3 of IS 456 : 2000

Allowable shear stress $\tau_{v \text{ allowable}} =$ $k_s \tau_c$

$k_s = (0.5 + b/c) =$ **1.011111111 >1**

Hence, $k_s =$ **1**

$\tau_c = 0.25 (f_{ck})^{0.5} =$ **1.118033989 N/mm²**

$\tau_{v \text{ allowable}} = k_s \times \tau_c =$ **1.118033989 N/mm²**

Shear force $V_s = 501.476 (1.65 \times 1.45 - 0.854 \times 0.634) =$ **928.26 KN**

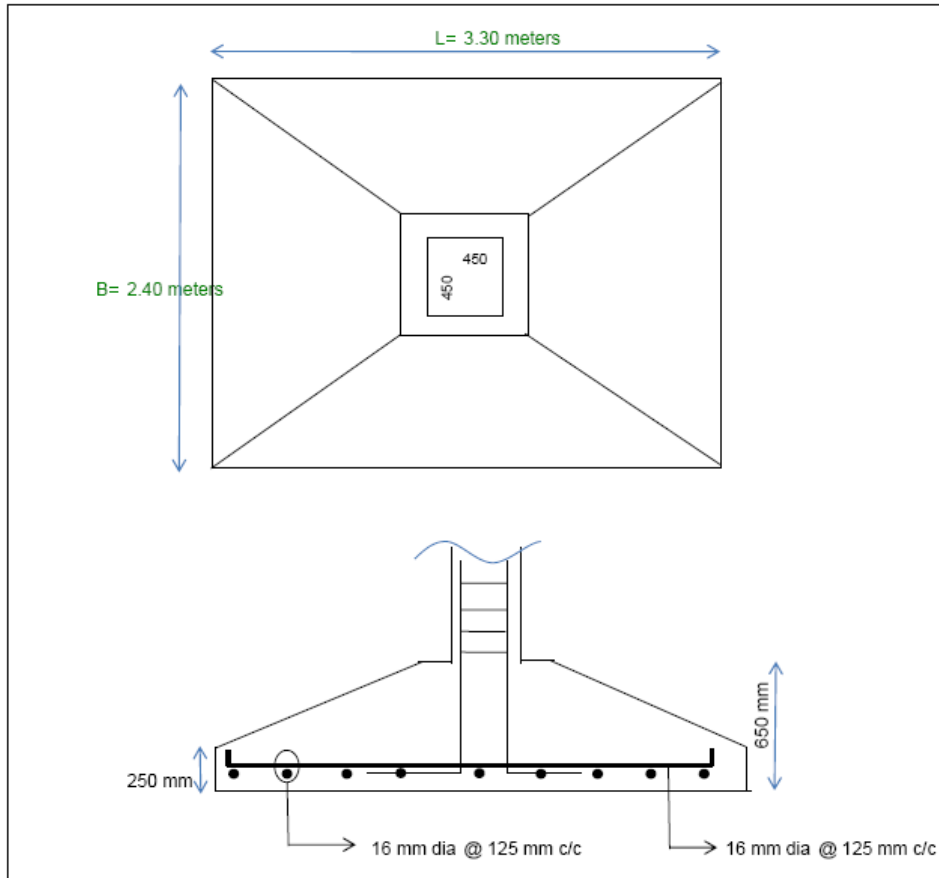
Length of critical section = $2 \times (854 + 634) =$ **2976 mm**

Area of the critical section (length of critical sec x eff. d) = **1202304 mm²**

Hence shear stress $\tau_v =$ **0.772 N/mm²**

tv < allowable hence O.K.

1 Footing Size Design		
Load	Pu	1500 KN
Design Load	P	1100 KN
Moment in x dir	Mux	30 KN-m
Moment in y dir	Muy	30 KN-m
Column size	cx	450 mm
	cy	450 mm
SBC	q	150 KN/sqm
Footing Size required	A req	7.33 sqmm
Footing Size Provided	L	3.30 meters
	B	2.40 meters
Area Provided	A prvd	7.92 meters
	Zx	3.17
	Zy	4.36
Net upward pressure	Nup	150 KNm2
Footing Size OK		



Result: The analysis and design is done by using Staad.Pro.

EXPERIMENT NO: 11

ANALYSIS AND DESIGN OF COMBINED FOOTING

Aim:

To analyze and design the combined Footing using STADD pro

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed STADD Pro Software.

Procedure:

1. Entering Job information.
2. Truss model geometry.
3. Defining member properties ,sections.
4. Assigning Loads(Load Cases and combination).
5. Defining Pre-Analysis Print out, analysis type and Post-Analysis print out.
6. Defining Design Requirements.
7. The combination of STAAD.pro and STAAD.beava can make your bridge design and analysis easier. STAAD.pro is first used to construct the bridge geometry and STAAD.beava is used to find the AASHTO 2002 load positions that will create the maximum load response. The maximum load response could be any of the following:
 1. Maximum plate stresses, moment about the local x axis of a plate (M_x), moment about the local y axis of a plate (M_y) etc. used to design for concrete deck reinforcement.
 2. Maximum support reactions to design isolated, pile cap, and mat foundations.
 3. Maximum bending moment or axial force in a member used to design members as per the AASHTO code.
 4. Maximum deflection at mid span.
17. These loads that create the maximum load responses can be transferred into STAAD.pro as load

18. cases to load combinations for further analysis and design. Figure 1 shows the bridge design
19. procedure discussed above.

Step 1

Area required for footing
 Square = $B = (w+w1)/P0$
 Where, P_0 = safe bearing capacity of soil
 $w1$ = self weight of footing
 w = self weight of footing
 For Rectangle = $b/d = B/D$
 $A = b \times d$
 Net upward pressure on the footing
 $q/p = W/A$

Step 2

Bending Moment
 Critical section for maximum [bending moment](#) is taken at the face of the column
 For a square footing,
 $M_{xx} = q \times B/8 (L - a)^2$
 $M_{xx} = q \times L/8 (B - b)^2$
 $M_{yy} = q \times B/8 (L - a)^2$

Step 3

To fix the depth of the footing shall be greater of the following:
 Depth from bending moment consideration
 $d = \sqrt{(M/Qb)}$
 where, Q = moment of required factor
 Depth from shear consideration
 Check for one way shear
 Check for two way shear or punching shear
 Critical shear for one way shear is considered at a distance 'd' from face of the column.
 Shear force, $V = qB [\frac{1}{2}(B - b) d]$
 Nominal shear stress, $T_v = k \cdot T_c$
 $T_c = 0.16\sqrt{f_{ck}}$

Step 4

Check for two way shear
 Critical section for two way shear is considered at a distance at a distance $d/2$ from all the faces of the column.
 SF, $V = q [B^2 - (b + d)^2]$
 SF, $V = q [L \times B - (a + d)(b + d)]$
 Nominal shear stress, $T_v = V/2((a+d)(b+d)d)$ ——— {for a rectangle

$$T_v = V/4((b+d)d) \quad \text{———} \quad \{ \text{for a square} \}$$

$$T_v = k \cdot T_c$$

$$k = 0.5 + \beta > 1 \quad ; \text{ [Beta } \beta = \text{ratio of sides of the column} \text{]}$$

$$T_c = 0.16\sqrt{f_{ck}}$$

$$\text{Area of steel, } A_{st} = M/((\sigma)stjd)$$

Analysis is done as explained in combined footing

Result: The analysis and design of combined footing is done by using Staad.Pro

EXPERIMENT NO: 12

ANALYSIS OF BRIDGE DECK SLAB

AIM: To analyze the Bridge Deck Slab using STADD pro

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed STADD Pro Software.

PROCEDURE AND ANALYSIS STEPS

The combination of STAAD.pro and STAAD.beava can make your bridge design and analysis

easier. STAAD.pro is first used to construct the bridge geometry and STAAD.beava is used to

find the AASHTO 2002 load positions that will create the maximum load response.

The

maximum load response could be any of the following:

1. Maximum plate stresses, moment about the local x axis of a plate (M_x), moment about the local y axis of a plate (M_y) etc. used to design for concrete deck reinforcement.
2. Maximum support reactions to design isolated, pile cap, and mat foundations.
3. Maximum bending moment or axial force in a member used to design members as per the AASHTO code.
4. Maximum deflection at mid span.

These loads that create the maximum load responses can be transferred into STAAD.pro as load

cases to load combinations for further analysis and design. Figure 1 shows the bridge design

procedure discussed above.

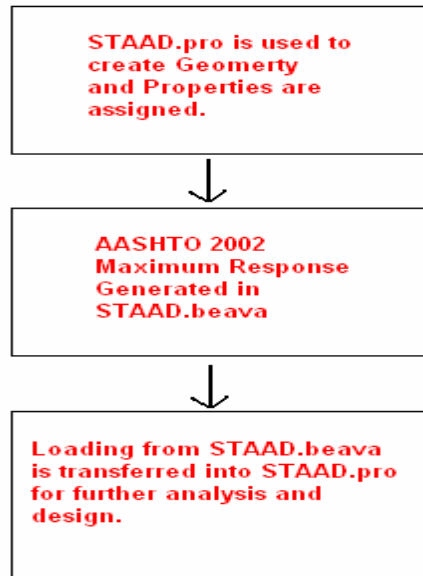


Figure 1: Bridge Design process in STAAD.pro

Creating the Bridge Geometry/Structural Analysis

Figure 2 shows a bridge with the dimensions.

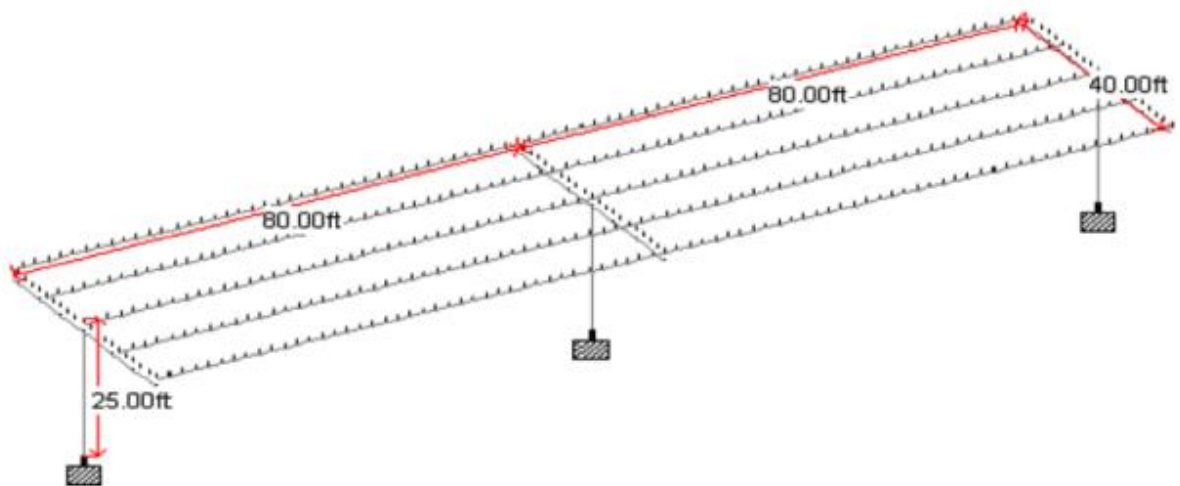


Figure 2: Bridge Dimensions

Step By Step Procedure

1. Open STAAD.pro with the default units of *Kip-Ft* and use the *Space* option.
2. Click the *Next* button and select the *Add Beam* mode. Click *Finish*.
3. The goal of the next few steps is to draw the stick model of the Bridge Structure (i.e. the beams and the girders). Select the *X-Z* grid option. Create two grid lines in the *x* direction at 80ft spacing. Create four grid lines in the *z* direction at 10ft spacing.
4. Click on the *Snap Node/Beam* button and draw the beams and the girders as shown below. First draw five 160ft girders. Then draw the 40ft beams in the *z* direction.
5. Click on *Geometry->Intersect Selected Members->Highlight*. STAAD.pro will highlight all the beams that intersect each other no common nodes. To break these beams at the intersection point, click on *Geometry->Intersect Selected Members->Intersect*
6. The beams have been created. The columns will now be created using the translational repeat command. Select nodes 6, 14, and 7 using the nodes cursor as shown below. The node numbers may vary depending upon how the model was constructed. Select *Geometry->Translational Repeat* from the menu. Select the *y* direction for the translational repeat and select enter a *Default Step Spacing* of *-25ft*
7. The deck of this bridge structure will be created in the following steps using the *Generate Surface Meshing Tool*. In reality, spacing of beams etc. may not be regular and hence it may become difficult to create the deck of the bridge using the *Generate Surface Meshing Tool*. The *Parametric Meshing Mode* could become very useful in these circumstances.
8. Select the *Geometry->Generate Surface Meshing* tool from the menu. Select four nodes that outline the 160ft x 40ft deck. Simply click on node 1, 3, 11, and 10 and select node 1 again to complete the command. Select the *Quadrilateral Meshing* option.
9. 2ft x 2ft element size is adequate for this type of model. Hence input the parameters as shown below.
10. The mesh will be created. To view the mesh properly, you will need to click on the *Setup Control Tab* on your left.
11. You should note that the girders and beams are automatically broken down into smaller elements. In reality, the girders are physically attached to the deck hence it is ok to mesh them. The concrete beams parallel to the *z*-axis are not attached to the deck. The load from the deck is transferred to the 40ft steel girders. The load from the girders is then transferred to the concrete beams. Hence, we should merge the beams in the *z* direction. Select the *Select->Beam Parallel To->z* from the menu.

12. Select *Geometry->Merge Selected Members* to merge the split concrete beams. Select each entry and press the *Merge* button.
13. By merging the beams together, the concrete column to beam connectivity is lost. Hence, click on *Geometry->Intersect Selected Members->Highlight*. STAAD.pro will highlight all the beams that intersect each other no common nodes. To break these beams at the intersection point, click on *Geometry->Intersect Selected Members- >Intersect*.
14. The geometry has been created. The properties and specifications have to be assigned.

Concrete Deck	Plate elements - 12in thick Material - Concrete
Steel Girders – Parallel to x axis	40ft Beam elements – W24X103 Material – Steel Offset = 0.5 x Depth of Beam + 0.5 thickness of slab = 0.5 x 2.044167 ft + 0.5 x 1ft = 1.5220835 ft at both ends.
Concrete Beam – Parallel to the z axis	2ft x 2ft Sections Material – Concrete Offset = 0.5 x Depth of Concrete Beam + Depth of Steel Beam + 0.5 thickness of slab = 0.5 x 2ft + 2.044167 ft + 0.5 x 1ft = 3.544167 ft at both ends.
Concrete Columns	2ft Circular Sections Material – Concrete Offset = Depth of Concrete Beam + Depth of Steel Beam + 0.5 thickness of slab = 2ft + 2.044167 ft + 0.5 x 1ft = 4.544167ft at the end connected to the concrete beams.

15. Click on the *General->Property* control tab on your left and click on the *Section Database* button.
16. Select the *W24x103* section from the American *W shape* database and click on the *Add* button.
17. Select the *W24x103* section that has been created. Click on the Select the *Select- >Beam Parallel To->X* from the menu and click on the *Assign* button on the right.
18. Select the *Define* button on the right and select the *Circle* section profile. Input a 2ft diameter and press the *Add* button.
19. Select the *Rectangle* section profile and input 2ft in the *YD* and *ZD* input boxes. Press the *Add* button. Press the *Close* button.
20. Select the *Cir 24* section that has been created. Click on the Select the *Select->Beam Parallel To->Y* from the menu and click on the *Assign* button on the right.
21. Select the *Rect 24.00x24.00* section that has been created. Click on the Select the *Select->Beam Parallel To->Z* from the menu and click on the *Assign* button on the right.
22. Press the *Thickness* button the right and input 1ft in the *Node 1* input box and press the *Add* button.

23. Select the newly create *Plate Thickness* entry in the *Properties* dialog box. Select all the plates using the *Plates Cursor* and press the *Assign* button.
24. Select the *Spec* sub-control tab on the left.
25. Press the *Beam* button. Select the *Offset* tab. Select the start option from the *location* selection box. Select the local option from the *Direction* selection box. Input an offset of -1.5220835 in the *Y* input box. Press the *Add* button.
26. Press the *Beam* button. Select the *Offset* tab. Select the *End* option from the *Location* selection box. Select the *Local* option from the *Direction* selection box. Input an offset of -1.5220835 in the *Y* input box. Press the *Add* button.
27. Assign these specifications to the steel girders parallel to the x-axis. Select the *START LOCAL 0 – 1.52208 0* specification that has been created. Click on the *Select->Beam Parallel To->x* from the menu and click on the *Assign* button on the right.
28. Select the *END LOCAL 0 -1.52208 0* specification that has been created. Click on the Select the *Select->Beam Parallel To->x* from the menu and click on the *Assign* button on the right.
29. Repeat Steps# 25 to 28 for the concrete beams but use a y-offset of -3.544167 ft at both ends. For Step# 28, you will need to select beams parallel to the z-axis.
30. For the columns assign a local x-offset of 4.544167ft at the end connected to the concrete beams.
31. Click on the *View->3D Rendering* in the menu.
32. Select the *General->Support* tab on the left and click on the *Create* button in the right hand side *Data Area*. Click on the *Add* button (i.e create a fixed support entry).
33. Select the newly created *S2 Support 2* entry and using the nodes cursor select nodes 16, 17, and 18. Click on the *Assign* button.
34. Select the *General->Load* tab on the left and click on *Load Case Details* on the right hand side in the *Data Area*.
35. Click on the *Add* button on the right. Input the “*Dead Load*” in the *Title* input box. Select “*Dead*” in the *Loading type* selection box. Press the *Add* button. Click the *Close* button.
36. Select the newly created *1: Dead Load* entry in the data area. Press the *Add* button. Select the *Selfweight* item and press the *Add* button.
37. Click on the *Analysis/Print* control tab item on the left and press the *Add* button.
38. Click on the *Analyze->Run Analysis* menu. Use the *STAAD Analysis* option and click on the *Run Analysis* button.

39. If the analysis completed successfully, you should look at the exaggerated deflected shape of the bridge under the action of selfweight. Try to find out any connectivity problems etc. You can go to the *Post-Processing* mode by clicking on *Mode- >Postprocessing* command in the menu.
40. You can look at the bending moment diagram for the bridge by clicking on the *Beam->Forces* control tab on your left in the *Post-Processing* mode.
41. You can look at the stress distribution diagram for the bridge by clicking on the *Plate* control tab on your left. Select the *Max Absolute* stress type from the *Stress Type* selection box and click on the *Ok* button.

Result: The analysis of bridge deck slab done by using Staad.Pro