

COMPUTER AIDED DESIGN LABORATORY MANUAL

B.TECH – CIVIL ENGINEERING

DEPARTMENT OF CIVIL ENGINEERING



MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

2021-2022



MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

CERTIFICATE

This is to certify that this manual is a bonafide record of practical work in the **Computer Aided Design Laboratory** in **Third Year, Second Semester** of **B.Tech (Civil Engineering)** programme during the academic year **2021-22**. The book is prepared by **Dr.M.Harihanandh**, Associate Professor, Department of Civil Engineering and **Mr.T.JayaKrishna**, Assistant Professor, Department of Civil Engineering of **MLRITM**.

Signature of HOD

Signature of Dean Academics

Signature of Principal

S.No

CONTENT

PREFACE

ACKNOWLEDGEMENT

GENERAL INSTRUCTIONS

SAFETY PRECAUTIONS

INSTITUTE VISION AND MISSION

DEPARTMENT VISION MISSION, PROGRAMME EDUCATIONAL OBJECTIVES AND SPECIFIC OUTCOMES

PROGRAMME OUTCOMES

COURSE STRUCTURE, OBJECTIVES & OUTCOMES

EXPERIMENTS LEARNING OUTCOMES

EXPERIMENTS

- 1 Analysis & Design determinate structures using software.
- 2 Analysis & Design determinate fixed & continuous beam using software.
- 3 Analysis & Design determinate plane frame using software.
- 4 Analysis & Design of space frames subjected to DL & LL
- 5 Analysis & Design of residential building subjected to all loads
- 6 Analysis & Design of Roof Trusses
- 7 Design and detailing of built up steel beam
- 8 Developing a design Programme for foundation using EXCEL Spread Sheet
- 9 Detailing of RCC beam and RCC slab
- 10 Detailing of Steel built up compression member

PREFACE

This book entitled “COMPUTER AIDED DESIGN LABORATORY MANUAL” is intended for the use of first semester (i.e, III Year – II Semester) B.Tech (Civil Engineering) students of Marri Laxman Reddy Institute of Technology and Management, Dundigal, Hyderabad. The main objective of the COMPUTER AIDED DESIGN LABORATORY Manual is to teach the student to impart the practical knowledge in the field to set out any Civil Engineering work, especially in the analysis and design of RCC and Steel structures. This book lays foundation of certain basic concepts and skills that can be repeatedly employed by the students in their future endeavours. The main aim of this book is to develop the habit of scientific reasoning and providing answers to all the doubts that arise during the course of conducting experiments. The book was written as per the new syllabus framed for MLRITM (Autonomous Institution) recommended by the BOS Committee members in a simple language. These experiments will help the students to expertise in the advances in Concrete Technology. Hence, we hope this book serve for better understanding by the student community with all details of experiments

By,

Dr. M.Harihanandh
Associate Professor,
Department of Civil Engineering

ACKNOWLEDGEMENT

It was really a good experience, working at COMPUTER AIDED DESIGN LABORATORY. First, I would like to thank Mr.T.JayaKrishna, Assistant Professor, Department of Civil Engineering, Marri Laxman Reddy Institute of Technology and Management for giving the technical support in preparing the document.

I express my sincere thanks to Mr.K.Murali, Head of the Department, Civil Engineering, Marri Laxman Reddy Institute of Technology and Management, for his concern towards me and gave the opportunity to prepare Advanced Concrete Technology laboratory manual.

I am deeply indebted and gratefully acknowledge the constant support and valuable patronage of Dr.B.Ravi Prasad, Dean Academics, Marri Laxman Reddy Institute of Technology and Management. I am unboundedly grateful to him for timely corrections and scholarly guidance.

I express my hearty thanks to Dr.K.Venkateswara Reddy, Principal, Marri Laxman Reddy Institute of Technology and Management, for giving me this wonderful opportunity for preparing the Advanced Concrete Technology laboratory manual.

At last, but not the least I would like to thanks the entire Civil Department faculties those who had inspired and helped me to achieve my goal.

By,

Dr. M.Harihanandh
Associate Professor,
Department of Civil Engineering

GENERAL INSTRUCTIONS

1. Remove your shoes or wear foot socks before you enter the lab.
2. Observe silence when you are in the laboratory
3. Students are instructed to come to laboratory on time. Late comers are not entertained in the lab.
4. Students should be punctual to the lab. If not, conducted experiments will not be repeated.
5. Students are expected to come prepared at home with the experiments which are going to be performed.
6. Students are instructed to display their identity cards and apron before entering into the lab.
7. Students are instructed not to bring mobile phones to the lab.
8. Breakage amount will be collected from the student for causing damage to the instruments / Equipment's due to wrong operation or carelessness
7. For each lab class, all the students are expected to come with observation note book, record note book, calculator, pencil, eraser, sharpener, scale, divider, graph sheets.
8. Students should update the records and lab observation books session wise before leaving the lab and should get his lab observation book signed by the faculty.
9. Students should submit the lab records 2/3 days in advance to the concerned faculty members in the staff room for their correction and return.
10. Students should not move around the lab during the lab session.
11. If any emergency arises, the student should take the permission from faculty member concerned in written format.
12. The faculty members may suspend any student from the lab session on disciplinary grounds.

SAFETY PRECAUTIONS

1. Report any problems with the computer to the person in charge.
2. Shut down the computer properly.
3. Do not bring any food or drinks in the computer room.
4. Do not touch any part of the computer with wet hands.
5. Do not hit the keys on the computer too hard.
6. Playing games is prohibited in the computer laboratory.
7. Don't damage, remove, or disconnect any labels, parts, cables or equipment.
8. Do not install or download any software or modify or delete any system files on any labcomputers.
9. CD – ROM's and other multimedia equipment are for official work only. Do not use them for recreational activities.
10. If you leave the lab, do not leave your personal belongings unattended.

INSTITUTION VISION AND MISSION

VISION

To establish as an ideal academic institutions in the service of the nation, the world and the humanity by graduating talented engineers to be ethically strong, globally competent by conducting high quality research, developing breakthrough technologies, and disseminating and preserving technical knowledge.

MISSION

To fulfil the promised vision through the following strategic characteristics and aspirations:

- ❖ Utilize rigorous educational experience to produce talented engineers
- ❖ Create an atmosphere that facilitates the success of the students
- ❖ Prudent and accountable resource management.
- ❖ Programs that integrate global awareness, communication skills and leadership abilities.
- ❖ Education and Research partnerships with institutions and industries for preparing the students for interdisciplinary research
- ❖ Successful alumni in their profession at global level.

DEPARTMENT VISION, MISSION , PROGRAMME EDUCATIONAL OBJECTIVES
AND SPECIFIC OUTCOMES

VISION

The Civil Engineering department strives to impart quality education by extracting the innovative skills of students and to face the challenges in latest technological advancements and to serve the society.

MISSION

M-I Provide quality education and to motivate students towards professionalism

M-II Address the advanced technologies in research and industrial issues

PROGRAMME EDUCATIONAL OBJECTIVES

The Programme Educational Objectives (PEOs) that are formulated for the civil engineering programme are listed below;||

PEO-I solving civil engineering problems in different circumstances PEO-II Pursue higher education and research for professional development.

PEO-III Inculcate qualities of leadership for technology innovation and entrepreneurship.

PROGRAM SPECIFIC OOUTCOMES

PSO 1 - UNDERSTANDING: Graduates will have ability to describe, analyse and solve problems using mathematical, scientific, and engineering knowledge.

PSO 2 - ANALYTICAL SKILLS: Graduates will have an ability to plan, execute, maintain, manage, and rehabilitate civil engineering systems and processes.

PSO 3 - EXECUTIVE SKILLS: Graduates will have an ability to interact and work effectively in multi disciplinary teams.

PROGRAMME OUT COMES

- 1. Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
- 2. Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
- 3. Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
- 4. Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
- 5. Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
- 6. The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
- 7. Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
- 8. Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
- 9. Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
- 10. Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
- 11. Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
- 12. Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

COMPUTER AIDED DESIGN LAB
COURSE STRUCTURE, OBJECTIVES & OUTCOMES

COURSE STRUCTURE

Computer Aided Design laboratory will have a continuous evaluation during sixth semester for 30 sessional marks and 70 end semester examination marks.

Out of the 30 marks for internal evaluation, day-to-day work in the laboratory shall be evaluated for 20 marks and internal practical examination shall be evaluated for 10 marks conducted by the laboratory teacher concerned.

The end semester examination shall be conducted with an external examiner and internal examiner. The external examiner shall be appointed by the Principal / Chief Controller of examinations.

COURSE OBJECTIVES

The objectives of the course are to

1. Learn the usage of any fundamental software for design.
2. Create geometries using Pre-Processor.
3. Analyse the interpret the result using processor.
4. Design the structural elements

COURSE OUTCOMES

At the end of the course, the student will be able to:

CE329.1	Understand CAD software and basic functions
CE329.2	Evaluate plans of Single storied building & multistoried buildings
CE329.3	Develop different sections at different elevations
CE329.4	Detailing of building components like doors, windows roof trusses
CE329.5	Develop section and elevation for single and multistoried buildings using CAD software.
CE329.6	Understand development concepts in detailing

COURSE ARTICULATION MATRIX (CO - PO / PSO MAPPING):

Program outcomes	1	2	3	4	5	6	7	8	9	10	11	12	PSO 1	PSO 2	PSO 3
CE329.1	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
CE329.2	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
CE329.3	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
CE329.4	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
CE329.5	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
CE329.6	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1
Total	18	6	6	18	6	6	0	0	0	0	0	18	6	12	6
Average	3	1	1	3	1	1	0	0	0	0	0	3	1	2	1

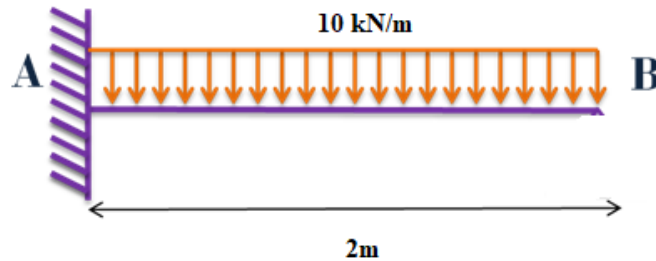
LIST OF EXPERIMENTS

1. Analysis & Design determinate structures using software.
2. Analysis & Design determinate fixed & continuous beam using software.
3. Analysis & Design determinate plane frame using software.
4. Analysis & Design of space frames subjected to DL & LL
5. Analysis & Design of residential building subjected to all loads
6. Analysis & Design of Roof Trusses
7. Design and detailing of built up steel beam
8. Developing a design Programme for foundation using EXCEL Spread Sheet
9. Detailing of RCC beam and RCC slab
10. Detailing of Steel built up compression member

EX.NO: 1

Date:

Analysis & Design determinate structures using software.



Aim: To Analysis & Design determinate structures by using STAAD Pro.

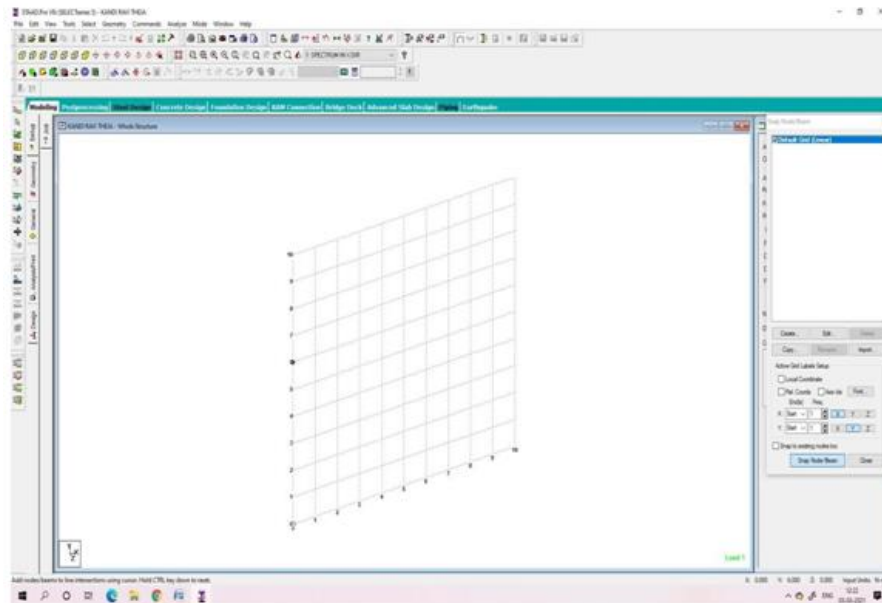
Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

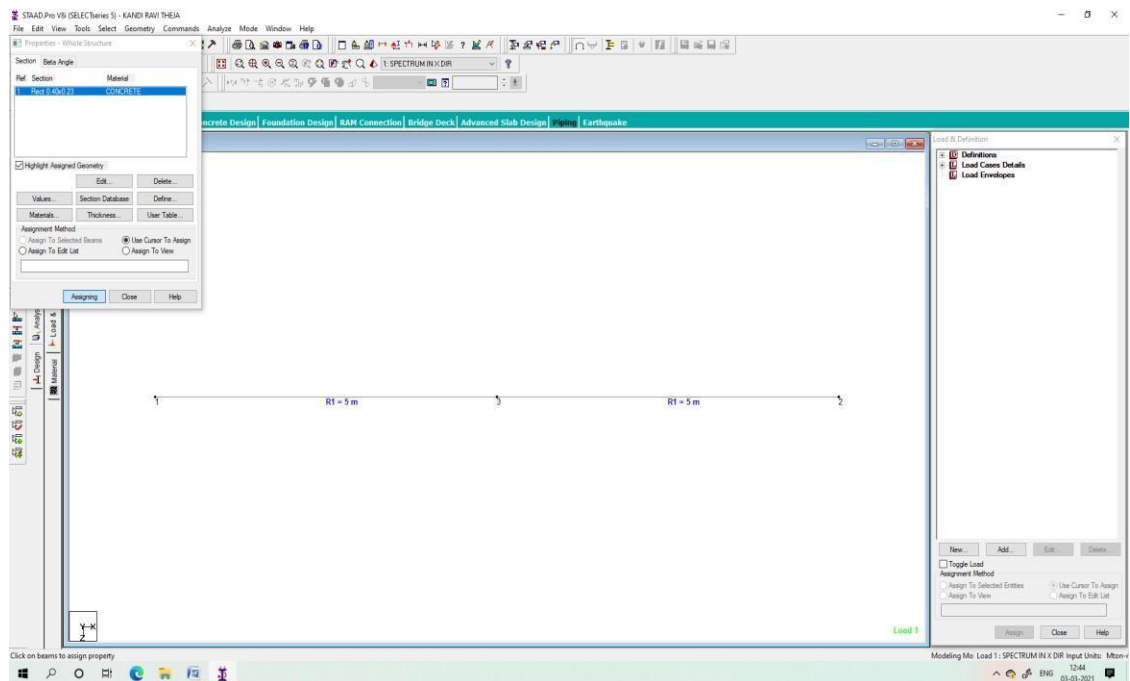
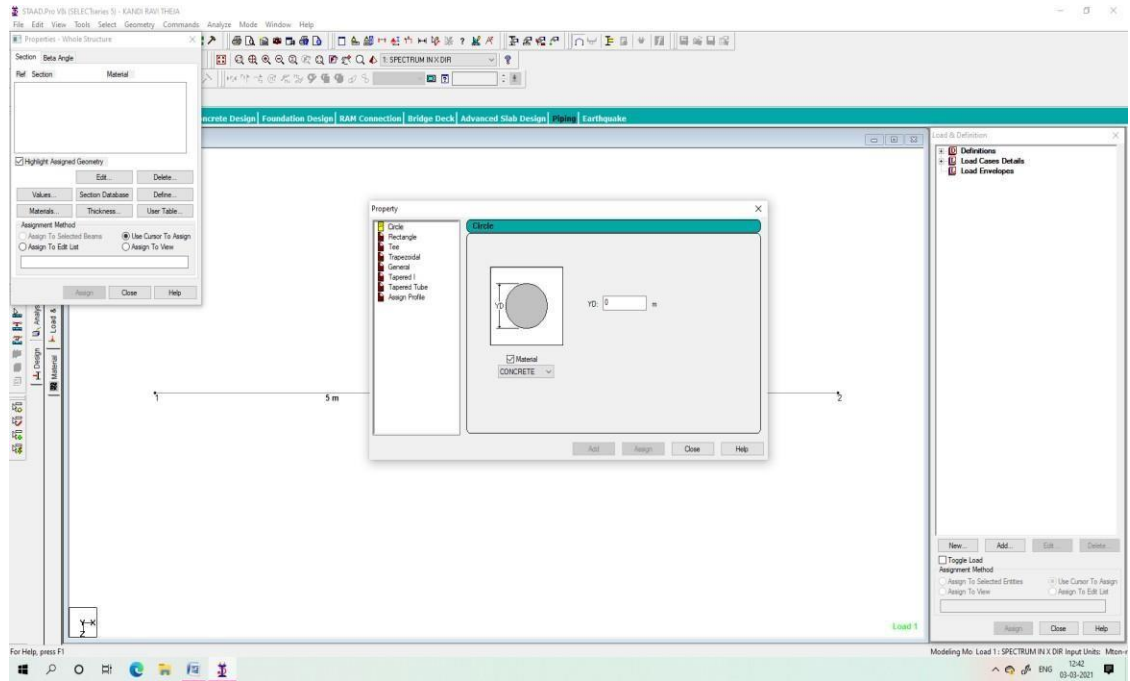
Staad procedure:

1. Click twice on STAAD Icon → New project → Space → Fixed beam → Location → Units(M, kN) → Next → Finish.

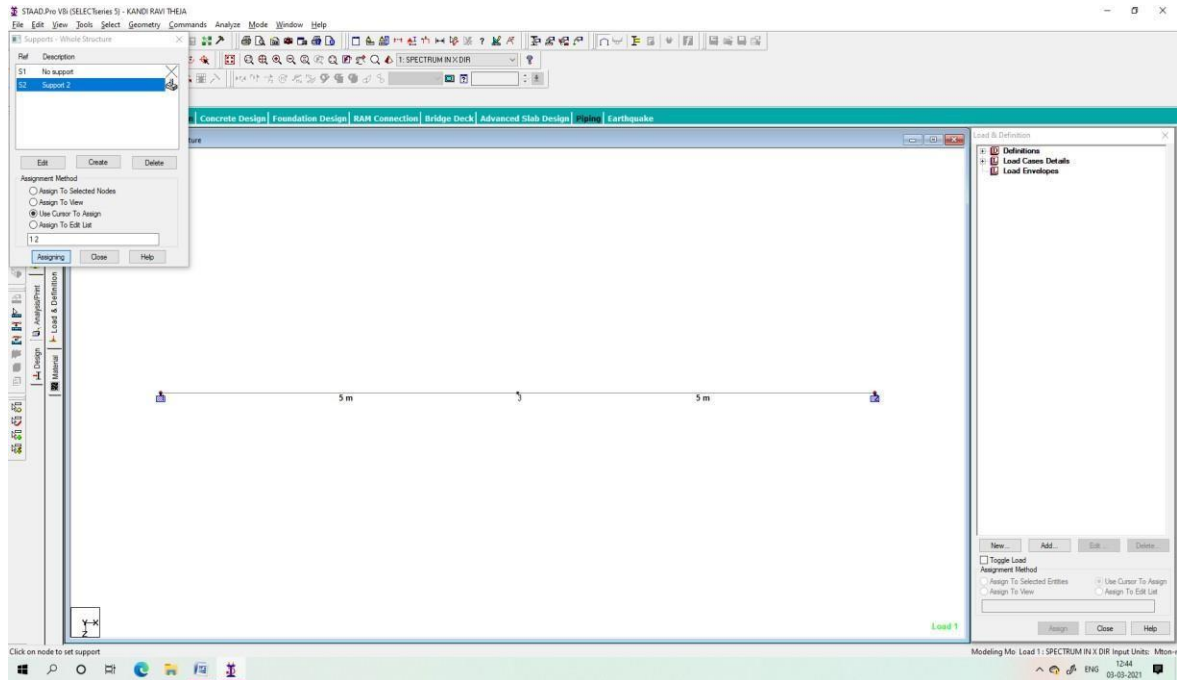


2. Front view(view from +Z) → Click at origin → Click at the 4th grid in X direction → Close grid

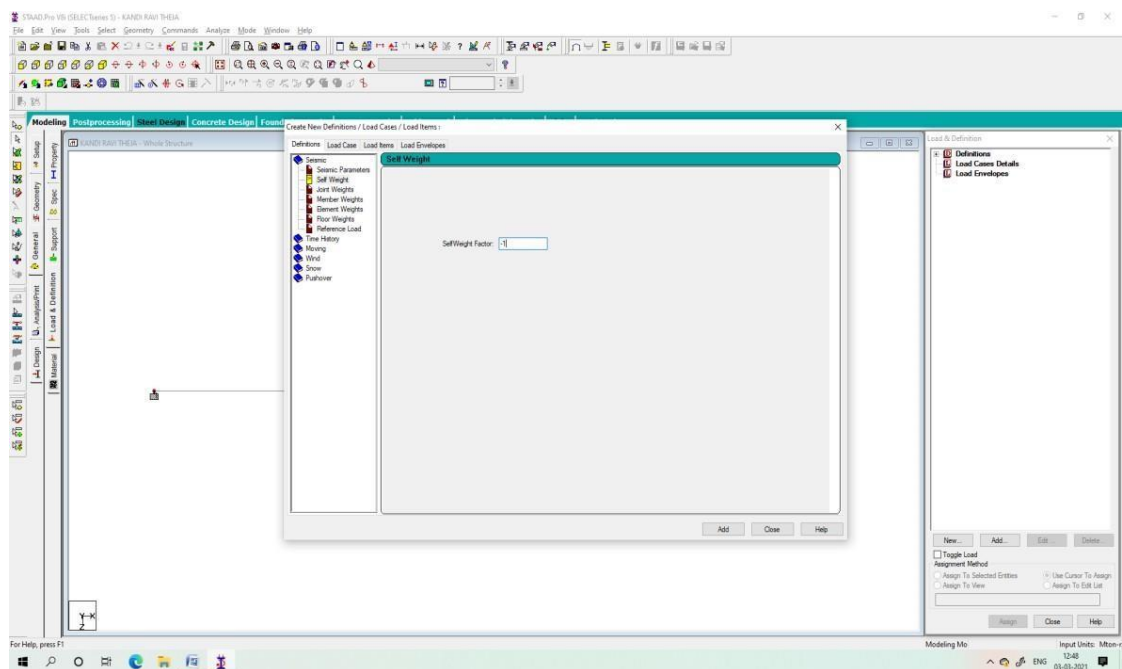
- General → Property → Define → Rectangle $Y_d=0.4$, $Z_d= 0.23$ m → Add → Close → Select(0.4 X 0.23) → Select beam → Assign to selected beams → Assign
 Now the beam has a lable of R1 which indicates reference 1 which means first property is assigned to the beam.



- General → Support → Create → Fixed add → Select S2 support 2 → Select two nodes → Assign to selected nodes → Assign



5. General → load and definition → select 'load case details' → Add → title Load → close select load → add → self weight, Y direction, and factor -1 → add → member load → uniform force, -20 kN/m, GY D1=D2 =D3 = 0, → add → close → select 'self weight' → select beam → assign to select beams → select UNI GY -20 kN/m³ → select beams → Assign to selected beams Assign.



6. Analysis/Print – Click All on Perform Analysis – Add and Close – Click Analyse on Menu – Run Analysis – System will run analysis and display a file – Click on View output file – Done – We can get one output file which have all analysis data.

7. In order to view the results of (Point 6) in diagram format, the following things we have to do.
- 7.1 Click Analyze on Menu – Run Analysis – System will run analysis and display a file – Click on Go to Post Processing Mode – Done – Apply and Ok. On screen we can get Post Processing.
- 7.2 To view deflection, click the deflection icon on the menu - In order to view the values also in diagram, click the results on the menu – view – Beam results – Enable End, Maximum, Midpoint under Deflection - Annotate – To save the result in image format – Click on Take Picture icon, and click ok.
- 7.3 To view shearforce diagram, click on Fy icon on screen – In order to view the values also in diagram, click the results on the menu – view – Beam results – Enable End, Maximum, Midpoint under Shear - Annotate – To save the result in image format – Click on Take Picture icon, and click ok.
- 7.4 To view bending moment diagram, click on My icon on screen – In order to view the values also in diagram, click the results on the menu – view – Beam results – Enable End, Maximum, Midpoint under Bending - Annotate – To save the result in image format – Click on Take Picture icon, and click ok.
- 7.5 If we need all the above results are in PDF format, click on Report (Left side of screen) - Items Available – Click down arrow, select Pictures (We can see the captured pictures list) – Click on Double arrow and Ok. – Report will be available in PDF along with output above diagram.

DESIGN PART:

1. Modeling – Design – Concrete – Select Current code on right side – IS456 – Select Parameters – Click Ok
2. Define Parameters - Click
Clear (Clear Cover) – Assign the value as 0.035m and click add –
Fc (Concrete Grade) - Assign the value as 25000kN/m² and click add –
FyMAIN (Steel Grade) - Assign the value as 500000kN/m² and click add –
FySEC (Steel Grade) - Assign the value as 500000kN/m² and click add –
MAXMAIN (Main Reinforcement maximum Size) - Assign the value as 20mm and click add –
MAXSEC (Secondary Reinforcement maximum Size) - Assign the value as 12mm

and click add –

MINMAIN (Main Reinforcement minimum Size) - Assign the value as 10mm and click add –

MINSEC (Secondary Reinforcement minimum Size) - Assign the value as 8mm and click add – Close

3. On right side space we get question mark (?) near the parameters – It shows those parameters are not assigned. To assign the parameters, select it on the right side space – Click on Assign to view – Click Assign – Yes. So that the (?) will not shown there.
4. Commands – Design Commands, Select DESIGN BEAMS – Add. TAKE OFF (To get the steel and concrete quantity) – Add – Close.
5. Click on Design Beam with (?) on right side space and click on Assign to View – Assign – Yes.
6. Click on Analysis/Print – Click All, under Perform Analysis – Add – Close.
7. Click Analyze on Menu – Run Analyze – View output file – Done – Click on CONCRETE DESIGN - So that we can get entire report in PDF
8. If we need the beam detailing diagram, do double click on beam on screen – CB std Beam – Click Concrete Design - Give Print and Save as PDF.

Applications of Cantilever Beam:

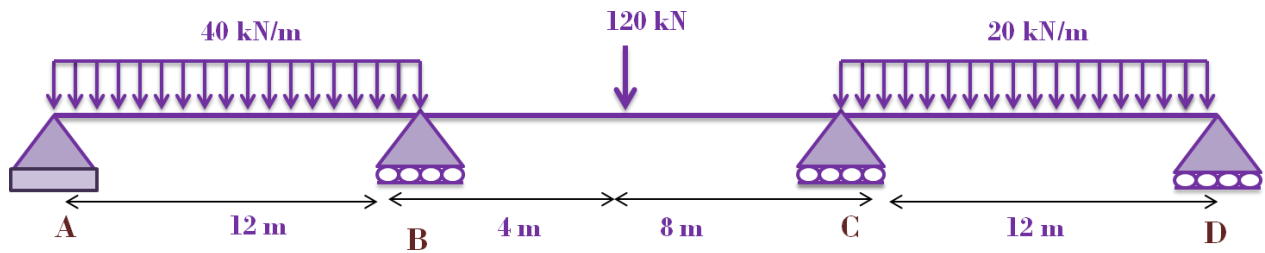
- A good example of a cantilever beam is a **balcony**.
- A balcony is supported on one end only, the rest of the beam extends over open space; there is nothing supporting it on the other side.
- Cantilevers deflect more than most other types of beams because they are only supported from one end.



Fig.1. Cantilever Beam – Building Construction

EX.NO: 2**Date:**

Analysis & Design determinate fixed & continuous beam using software.



Aim: To Analysis & Design determinate fixed & continuous beam by using STAAD Pro.

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

A fixed beam of length 4m is carrying an UDL of 20 kN/m

Staad procedure:

1. Click twice on STAAD Icon → New project → Space → Fixed beam → Location → Units(M, kN) → Next → Finish
2. Front view(view from +Z) → Click at origin → Click at the 4th grid in X direction → Close grid
3. General → Property → Define → Rectangle $Y_d=0.4$, $Z_d= 0.23$ m → Add → Close → Select(0.4 X 0.23) → Select beam → Assign to selected beams → Assign
Now the beam has a lable of R1 which indicates reference 1 which means first property is assigned to the beam.
4. General → Support → Create → Fixed add → Select S2 support 2 → Select two nodes → Assign to selected nodes → Assign
5. General → load and definition → select ‘ load case details’ → Add → title Load → close select load → add → self weight, Y direction, and factor -1 → add → member load → uniform force, -20 kN/m, gy D1=D2 =D3 = 0, → add → close → select ‘self weight’ → select beam → assign to select beams → select UNI GY -20 kN/3 → select beams → assign to selected beams Assign.

Default the scaling of load will be higher and t can be changed automatically by clicking load icon, and it can altered by using” control+ scrolling mouse” inwards or outwards

6. **Analysis/print** → add → close → **analyze** → run analysis → go to post processing → done → ok → maximize window (by default deflection diagram will be displaced in green color, by changing the scale of deflection diagram use “control + scroll the mouse”) → result (menu bar) → view value → beam results → select maximum of bending, shear and displacement → annotate → ok → ok → close
- Note: the magnitude of deflection is visible, for knowing the B.M and SFD with values B.M (MZ) and S.F (FY) icon must be on.

DESIGN PART:

1. Modeling – Design – Concrete – Select Current code on right side – IS456 – Select Parameters – Click Ok
2. Define Parameters - Click
 - Clear (Clear Cover) – Assign the value as 0.035m and click add –
 - Fc (Concrete Grade) - Assign the value as 25000kN/m² and click add –
 - FyMAIN (Steel Grade) - Assign the value as 500000kN/m² and click add –
 - FySEC (Steel Grade) - Assign the value as 500000kN/m² and click add –
 - MAXMAIN (Main Reinforcement maximum Size) - Assign the value as 20mm and click add –
 - MAXSEC (Secondary Reinforcement maximum Size) - Assign the value as 12mm and click add –
 - MINMAIN (Main Reinforcement minimum Size) - Assign the value as 10mm and click add –
 - MINSEC (Secondary Reinforcement minimum Size) - Assign the value as 8mm and click add – Close
3. On right side space we get question mark (?) near the parameters – It shows those parameters are not assigned. To assign the parameters, select it on the right side space – Click on Assign to view – Click Assign – Yes. So that the (?) will not shown there.
4. Commands – Design Commands, Select DESIGN BEAMS – Add. TAKE OFF (To get the steel and concrete quantity) – Add – Close.
5. Click on Design Beam with (?) on right side space and click on Assign to View – Assign – Yes.
6. Click on Analysis/Print – Click All under Perform Analysis – Add – Close.

7. Click Analyze on Menu – Run Analyze – View output file – Done – Click on CONCRETE DESIGN - So that we can get entire report in PDF
8. If we need the beam detailing diagram, do double click on beam on screen – CB std Beam – Click Concrete Design - Give Print and Save as PDF.

Application – Simply supported beam:

- The most common examples of simply supported beams are, *bridge girders, post and lintel structure.*
- You can see in below pictures that the beam is actually placed on the top of the column creating a simply supported structure.
- The bending moments at the end will be zero.



Fig.1. Simply supported beam – Example 1 (Ancient Structure)

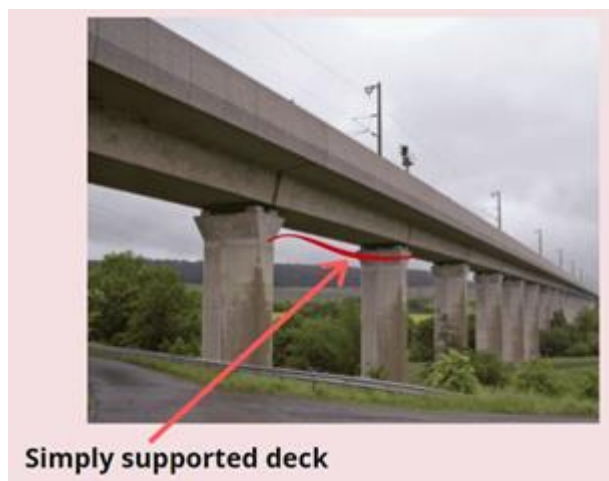


Fig.2. Simply supported beam – Bridge Deck

Applications – Fixed beam:

- A fixed support is the most rigid type of support or connection.
- It constrains the member in all translations and rotations (i.e. it cannot move or rotate in any direction).
- Anything that is more or less horizontal and is not supported at both extremes in order to avoid falling, could be considered a fixed beam.
- If you mean fixed at both ends, then we can find more examples: A walking bridge between two buildings. The section of any bridge that is located between two contiguous supports

**Fig.3 Fixed Beam – Bridge**

The fixed-connected joints of this concrete framework make this a statically indeterminate structure.

Fig.4 Fixed Beam – Building Construction

Application – Continuous beam:

- Continuous steel beams consist of two or more beams that are welded together and supported by other beams to create a stable, yet flexible, component for large-scale structures. For instance, continuous beams are **used in bridges, multi-storey buildings, complex roof structures, and other construction projects.**

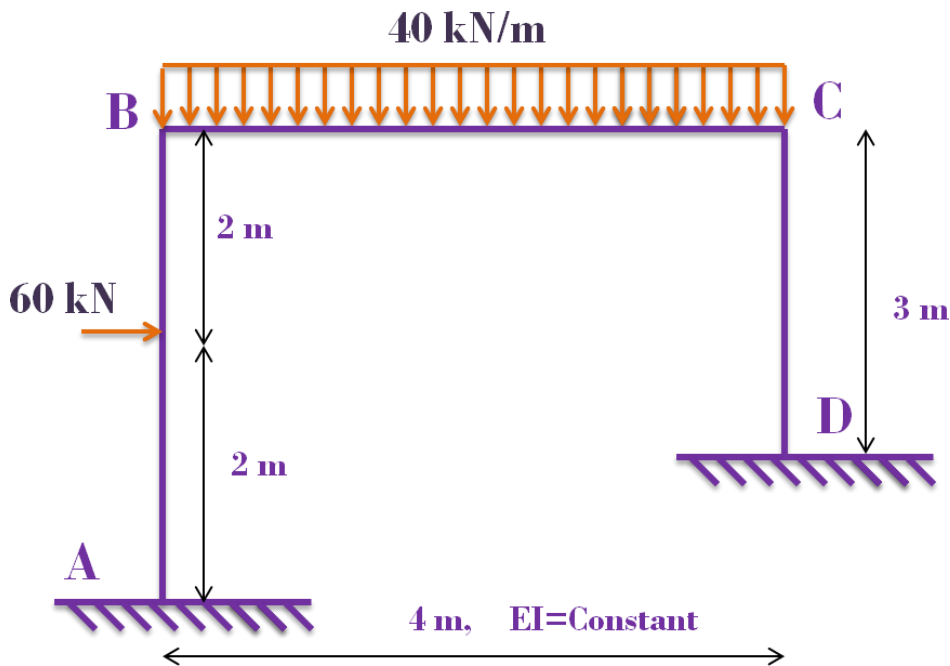


Fig.5 Continuous Beam - Bridge

EX.NO: 3

Date:

Prepare the Simple frame model, and analyze for different loading as shown in fig.



STAAD Procedure

1. Click twice on STAAD Icon → New Project → Space → Frame → Location → Units (M, kN) → Next → Finish.
2. Front View (View from + Z) → Edit → X Right = 1, Spacing = 5, Y Right = 1, Spacing = 3 → O.K → Click at origin → Click at 3rd grid in Y Direction → Click at 5th Grid in X Direction → Click at 0 grid in Y Direction → Close Grid.
3. General → Property → Define → Rectangle Y_d = 0.5 m, Z_d = 0.23 m → Add → Rectangle Y_d = 0.375 m, Z_d = 0.23 m → Add → Close → Select (0.5 X 0.23) → Select Beam → Assign to Selected Beam → Assign → Select (0.375 X 0.23) → Select Two Columns → Assign to selected members → Assign → Close.

Click on modeling window and use Shift + K for nodes display.

Now the members have a label of "R1 & R2" which indicates Reference 1 & 2 Which means first Property is assigned to the members.

4. General → Support → Create → Fixed → Add → Select S2 Support 2 → Select bottom most nodes → Assign to selected nodes → Assign

5. General →Load & Definition →Select “Load Case Details” →Add →Title “DL”
 →Add →Title “LL” →Add →Title “WL XP” →Add →Title “WL XN” →Add
 →Close →Select “DL” →Add →Self weight, Y Direction, and Factor -1 →Add
 →Member Load →Uniform Force, -20kN/m, GY, d1=d2=d3=0 →Add →Select “LL”
 →Member Load →Uniform force, -10kN/m, GY, d1=d2=d3=0 →Add →Select “WL
 XP” →Member load →Uniform force, 5kN/m, GX, d1=d2=d3=0 →Concentrated
 Force, P= -20, d1=2, d2= d3=0, GY →Add →Select “WL XN” →Member
 Load
 →Uniform force, -5kN/m, GX, d1=d2=d3=0 →Add →Close.

6. Select “Self weight” →Assign to view →Assign →Select -20 kN/m, →Select (Menu
 bar) →Select beam →Assign to selected beams →Assign →Select -10 kN/m
 →Select (Menu bar) →Select Beam →Assign to Selected beams →Assign →Select
 5 kN/m →Select Column →Assign to selected beams →Assign →Select -5 kN/m
 →Select Second Column →Assign to selected beams →Assign

*Default the scaling of load will be higher and it can be changed automatically by
 clicking loads icon, and it can be altered by using “control + scrolling mouse”
 inwards or outwards*

7. Analysis/Print →Add →Close →Analyze (Menu Bar) →Run Analysis →Go to
 Post Processing →Done →OK →Maximize window (By Default deflection
 diagram will be displayed in green color, for changing the scale of deflection diagram
 use “control + Scroll the mouse”) →Result (Menu Bar) →View value →Beam
 Results →Select maximum of Bending, Shear and Displacement →Annotate →O.K
 →O.K →Close.

*Now the magnitudes of deflection is visible, for knowing the B.M & S.F.D with values
 B.M (Mz) & S.F (Fy) icons must be on.*

ADDING THE LOAD COMBINATION

8. General →Load & Definition →Select Load case details →Add →Define
 combinations →Name: 1.5 (DL + LL), ai=1.5, transfer DL and LL respectively to
 right side →Add →Name: 1.5 (DL + WL XP), ai = 1.5, transfer DL and WL XP
 respectively to the right side →Add →Name: 1.5 (DL + WL+ XN), ai = 1.5, transfer
 DL and WL XN respectively to the right side → Add →Name: 1.2 (DL+LL+WL
 XN), ai=1.2, transfer DL, LL and WL XN respectively to right side →Add →Close

DESIGN PART:

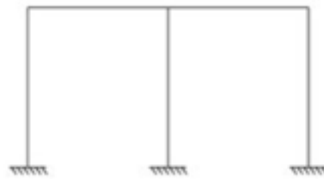
1. Design →Concrete →IS 456 →Select Parameters →transfer all parameters left side and select Fc, Fy main and Fc sec transfer right side →OK →Define Parameters
→Fc = 20000 kN/Sqm →Add →Fy Main = 415000 kN/sqm →Add →Fy sec = 415000 →Close →Commands →Design Beam →Add →Design Column →Add →Take Off →Close
2. Select FC 20000 →Assign to View →Assign →Select Fy Main 415000 →Assign to View →Assign →Select FY sec 415000 →Assign to view →Assign →Select Design Beam →Select Beam →Assign to selected beams →Assign →Select Design Column →Select Columns → Assign to selected beams →Assign
3. Open STAAD EDITOR →Type as “LOAD LIST 5 TO 9” above start concrete design →Close
4. Analyze (Menu bar) →Run Analysis →View output file →Done →Control+ F, Type as “ADEQUATE” and search for the word, if word is found means column is failed otherwise passed, like wise type as “FAIL” and search for the word if the word is found means beam is failed otherwise passed →Close.

EXPLANATION ABOUT LOCAL AXIS, GLOBAL AXIS AND BETA ANGLE

5. General →Property →Beta Angle →Create Beta Angle →Angle in Degrees = 90
→OK→Select Beta 90→Select two Columns →Assign to Selected beams→Assign
6. Analyze (menu bar) →Run Analysis →Go to post processing →Done →OK
→Maximize window (By Default deflection diagram will be displayed in green color,for changing the scale of deflection diagram use “control + scroll the mouse”)
→Results (menu bar) →View value→Beam Results→Select maximum of Bending, Shear and Displacement →Annotate →O.K →O.K →Close

Applications of Frame:

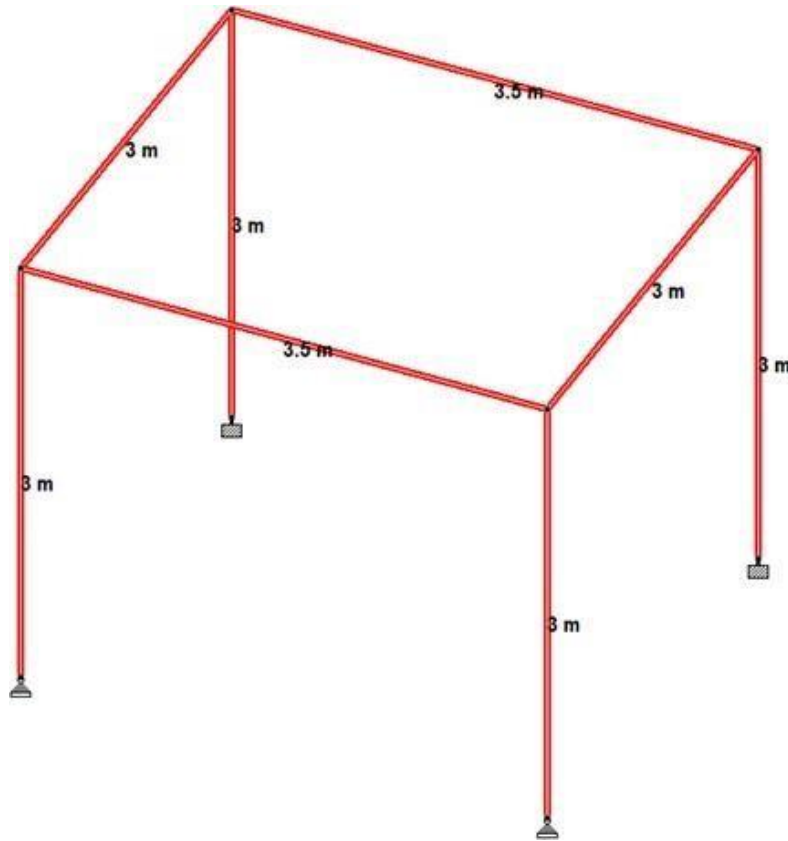
- A frame is a structure with **both vertical members connected by horizontal members.**
- The frames can be braced by diagonal members in which case none of the connections transfer moment, or it can be an unbraced moment frame in which the connections transfer moment.
- The most typical unbraced frames are cast-in-place concrete frames because the beams and columns are cast at the same time transferring the moment as long as the proper reinforcement is included.
- A Frame structure is a structure having the combination of beam, column and slab to resist the lateral and gravity loads. These structures are usually used to overcome the large moments developing due to the applied loading

**Fig.1 Typical plane frame****Fig.2 Typical RCC Frame - Building**

EX.NO: 4

Date:

Analysis & design of space frames subjected to DL and LL



Aim: To Prepare the multi frame model, and analyze for different loading as shown in fig by using STAAD Pro

Software required:

STAAD PRO V8i

STAAD Procedure

1. Click twice on STAAD Icon → New Project → Space ----- → Frame ----> Location
→ Units → (M, kN) → Next → Finish
2. Front View (View from +Z) → Close grid → Geometry (Menu bar) Run Structure wizard → Frame Models → Click twice on “Bay frame” → Length = 15, No. of bays along length = 3 (3 5 7), OK; Height = 15, No. of Bays along height = 5 (3 3 3 3), OK, Width = 0, Apply → Transfer Model → OK
3. General → Property → Define → Rectangle $Y_d=0.3$ m, $Z_d = 0.23$ m, → Add
→ Rectangle $Y_d = 0.5$ m, $Z_d = 0.23$ m, → Add → Rectangle $Y_d = 0.7$ m, $Z_d=0.23$ m
→ Close → Use Plan View → Shift + K → Select (0.3 X 0.23) → Select First Beam

by Clicking at mid of the beam →Assign to Selected Beams →Assign →Select (0.5 X 0.23) →Select second beam by clicking at mid of the beam →Assign of Selected Beams →Assign →Select (0.7 X 0.23) →Select third beam by Clicking at mid of the beam →Assign to selected Beams →Assign →Use Front view →Select (0.375 X 0.23) →Select First and Second Column by dragging selection window →Assign to Selected Beams →Assign →Select (0.45 X 0.23) →Select third and fourth columns by dragging selection window →Assign to Selected beams →Assign

Click on modelling window and use Shift + K for nodes display

Now the members have a label of "R1 & R2" which indicated Reference 1 & 2 Which means first property is assigned to the members.

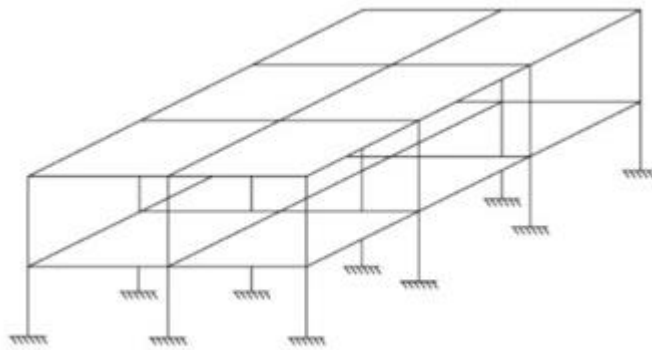
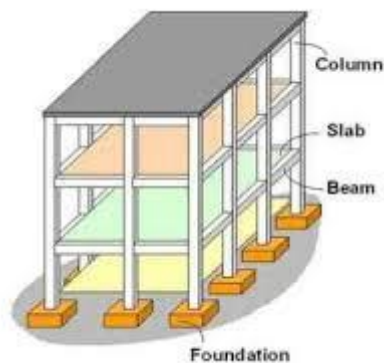
4. General →Support →Create Fixed →Add →Select S2 Support 2 →Select Bottom most nodes →Assign to selected nodes →Assign
5. General →Load & Definition →Select "Load Case Details" →Add →Title "DL" →Add →Title "LL" →Add →Title "WL XP" →Add →Title "WL XN" →Close →Select "DL" →Add Self weight, Y Direction, and Factor -1 →Add →Member Load →Uniform Force, -20 kN/mm, GY, d1=d2=d3= 0 →Add →select "LL" →Member Load →Uniform Force, -10 KN/m, GY, d1=d2=d3= 0 →Add →Select "WL XP" →Member Load →Uniform Force, 5KN/m, GX, d1=d2=d3=0 →Add →Select "WL XN" →Member Load →Uniform Force, -5KN/m, GX, d1=d2=d3= 0 →Add →Close
6. Select "Self weight" → Assign to View → Assign →Select -20 kN/m, (DL) → SELECT (Menu Bar) → Beams parallel to X Direction →Assign to selected Beams →Assign → Select -10kN/m, (LL) → SELECT (Menu Bar) → Beams parallel to X Direction → Assign to selected beams → Assign →Select 5 kN/m,(WL XP) → Select entire first Column except bottom most → Assign to selected Beams → Assign → Select -5 kN/m,(WL XN) → Select entire last column except bottom most column → Assign to selected Beams →Assign → Close

Default the scaling of load will be higher and it can be changed automatically by clicking loads icon, and it can be altered by using "Control + Scrolling mouse" inwards or outwards

7. Open previous Staad file "FRAME" --> Open STAAD Editor --> Copy from "Perform analysis to load list" --> Close --> Open STAAD Editor of Frame file, paste it above "Finish" -->Close
8. Design --> Concrete --> IS 456 --> Select Parameters --> transfer all parameters left side and select FC, FY main and FY sec transfer right side -->OK -->Define parameters
-->FC=20000 kN/sqm -->Add -->Fy_{main} = 415000 kN/Sqm -->Add --> FY_{sec} = 41500
--> Close --> Commands --> Design Beam --> Add --> Design Column --> Add --> Take off --> Close
9. Select Fc 20000 --> Assign to View --> Assign --> Select Fy main 415000 --> Assign to view --> Assign --> Select FY sec 4150000 --> Assign to view --> Assign --> Select Design Beam --> SELECT (menu bar) beams parallel to X direction --> Assign to selected beams --> Assign --> Select Design Column --> SELECT (menu bar) beams parallel to Y Direction --> Assign to selected beams --> Assign
10. Analyze (Menu Bar) --> Run analysis --> View Output File --> Done --> Control + F, Type as "ADEQUATE" and search for the word,if word is found means column is failed otherwise passed, like wise type as "FAIL" and search for the word if the word is found means beam is failed otherwise passed --> Close

Application of space frame:

- In architecture and structural engineering, a space frame or space structure (3D truss) is a rigid, lightweight, truss-like structure constructed from interlocking struts in a geometric pattern.
- Space frames can be used to span large areas with few interior supports.
- Like the truss, a space frame is strong because of the inherent rigidity of the triangle; flexing loads (bending moments) are transmitted as tension and compression loads along the length of each strut.

**Fig.1 Ideal space frame****Fig.2 Typical RCC Space Frame**

Space frame in the different structure:

- Industrial buildings, factories
- Sports halls
- Warehouses
- Swimming pools
- Conference halls and exhibition centers
- Stadiums with long span distance
- Museum and fair houses
- Shopping centers and malls
- Airports and canopy
- Atrium

EX.NO: 5**Date:**

Analysis and design of residential building subjected to all loads (DL, LL, WL, EQL)

Aim: To create and analyze the residential building subjected to all loads (DL, LL, WL, EQL) by using STAAD Pro

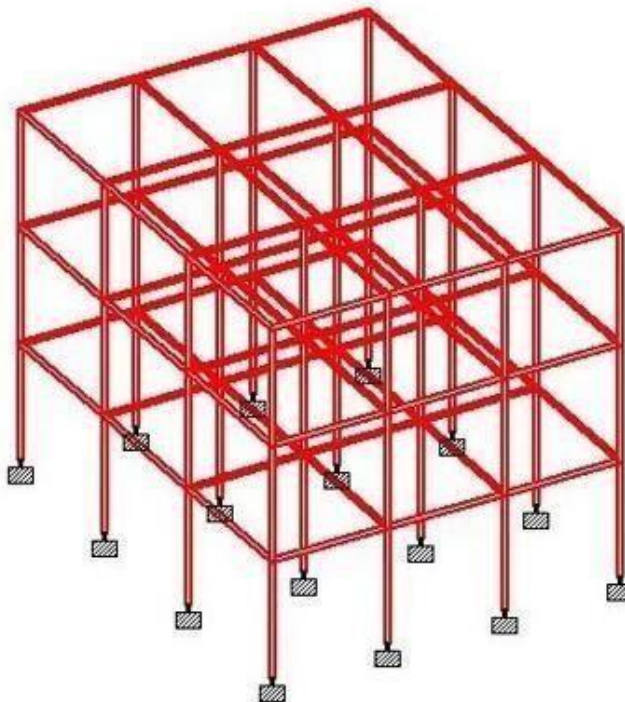
Software required:

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 → 0.35 x 0.23 → add → close
 - b. Select rectangular and assign to view.
3. Support
 - a. 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.
 - g. 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.



Post processing

5. 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.
6. 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.
7. 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.

Design parameters

8. Design concrete→ IS 456→define parameter→ give F_c as 20000 KN/m² for M20 concrete
9. → F_c as 30000 KN/m² for M30 concrete→add→ F_y as 415000 KN/m²→ add→ select→ ratio as 3→add.
10. Then go to commands→design beam→ add→design column→ add→take off → close.
11. Select F_c → select entire structure using cursor→ assign it to selected beams→ similarlydo it for F_y and ratio then assign it.
12. For beams → select the beams parallel to x direction and for columns column parallelto y direction and then assign it.

Analysis:

13. Analysis& print→ add→ close→analyze→ run analyze→ save→ post processing→ done.Post-processing:
14. 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.
15. Results→ view value→ beam results→ maximum displacement→ activate.
16. You can also see the animations of displacement in it.

Application:

- Residential building is the combination of footings, columns, beams and slabs construction.
- Application of space frame will be applicable here.
- Buildings have different types of loading such as dead load, live load, wind load, seismic load, snow load, etc. Based on the load criteria and its combination, how the structural response is to be, that can be found using the analysis software.

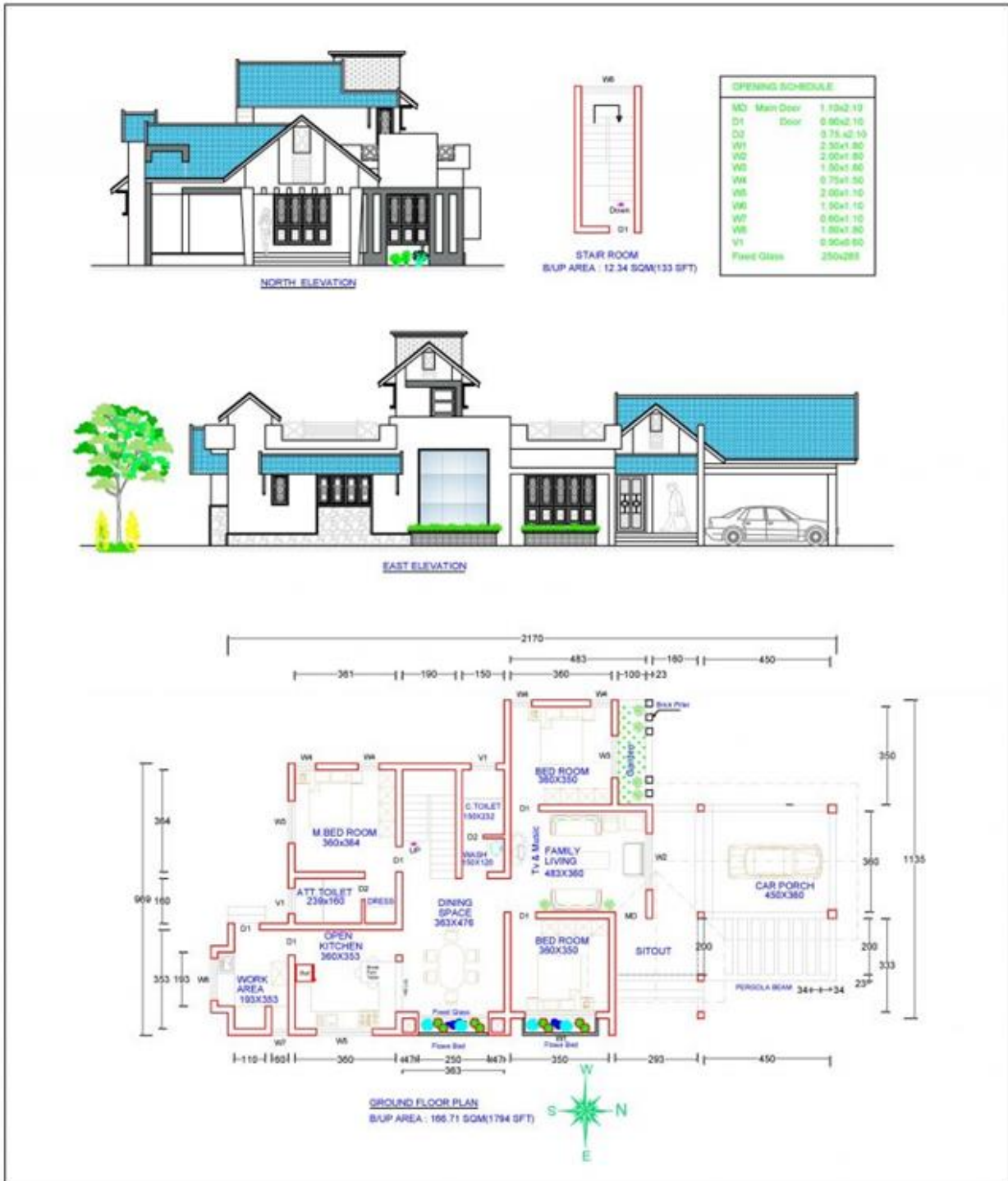
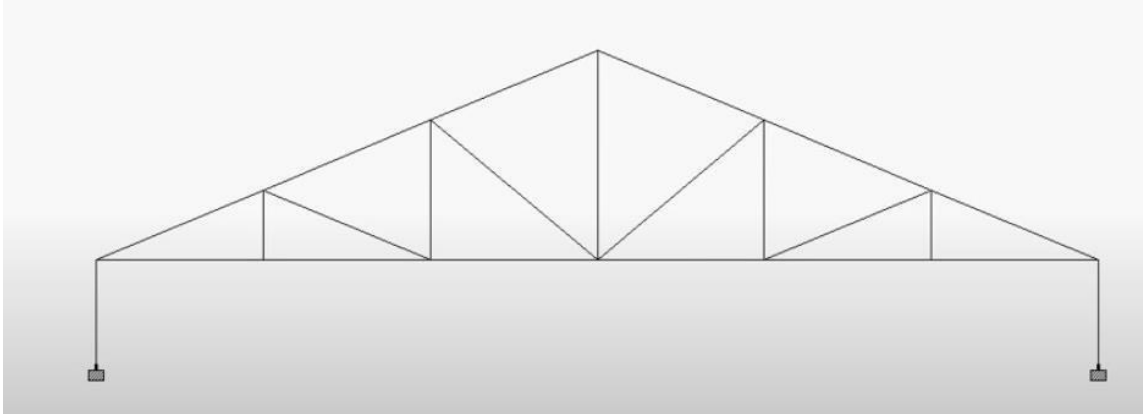


Fig.1 Typical residential building

EX.NO: 6

Date:

Analysis & Design of roof trusses



Aim: To design the roof trusses 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
 - a. Support → fixed support → select the bottom nodes → assign selected nodes.
4. Loads
 - b. Load and definition → load case detail → add → change the title as DL → add → then change LL → add → WL → add → close.
 - c. Select DL → add → member load → uniform load → type W1 as -40kN/m < add.
 - d. Select LL → add → member load → uniform load → type W1 as -20kN/m < add.

- e. 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.
 - f. Similarly, click select → member parallel to x direction → assign selected beams.
 - g. WL → select node cursor → select the particular node → assign selected node.
 - h. For load combinations, go to load case details → add → define combination → type name as
 - i. 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.
5. Post processing
- 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 F_c as 20000 KN/m² for M20 concrete
 - b. → F_c as 30000 KN/m² for M30 concrete → add → F_y as 415000 KN/m² → add → select → ratios 3 → add.
 - c. Then go to commands → design beam → add → design column → add → take off → close.
 - d. Select F_c → select entire structure using cursor → assign it to selected beams → similarly do it for F_y and ratio then assign it.
 - e. For beams → select the beams parallel to x direction and for columns column parallel to y direction and then assign it.
7. Analysis:
- a. Analysis & print → add → close → analyze → run analyze → save → post processing → done. Post-processing:

- b. 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.
- c. Results→ view value→ beam results→ maximum displacement→ activate.
- d. You can also see the animations of displacement in it.

Applications of truss:

- Trusses are used in a broad range of buildings, mainly where there is a requirement for very long spans, such as in airport terminals, aircraft hangers, sports stadia roofs, auditoriums and other leisure buildings.
- Trusses are also used to carry heavy loads and are sometimes used as transfer structures.



Fig.1 Typical truss

Different types of truss based on span:

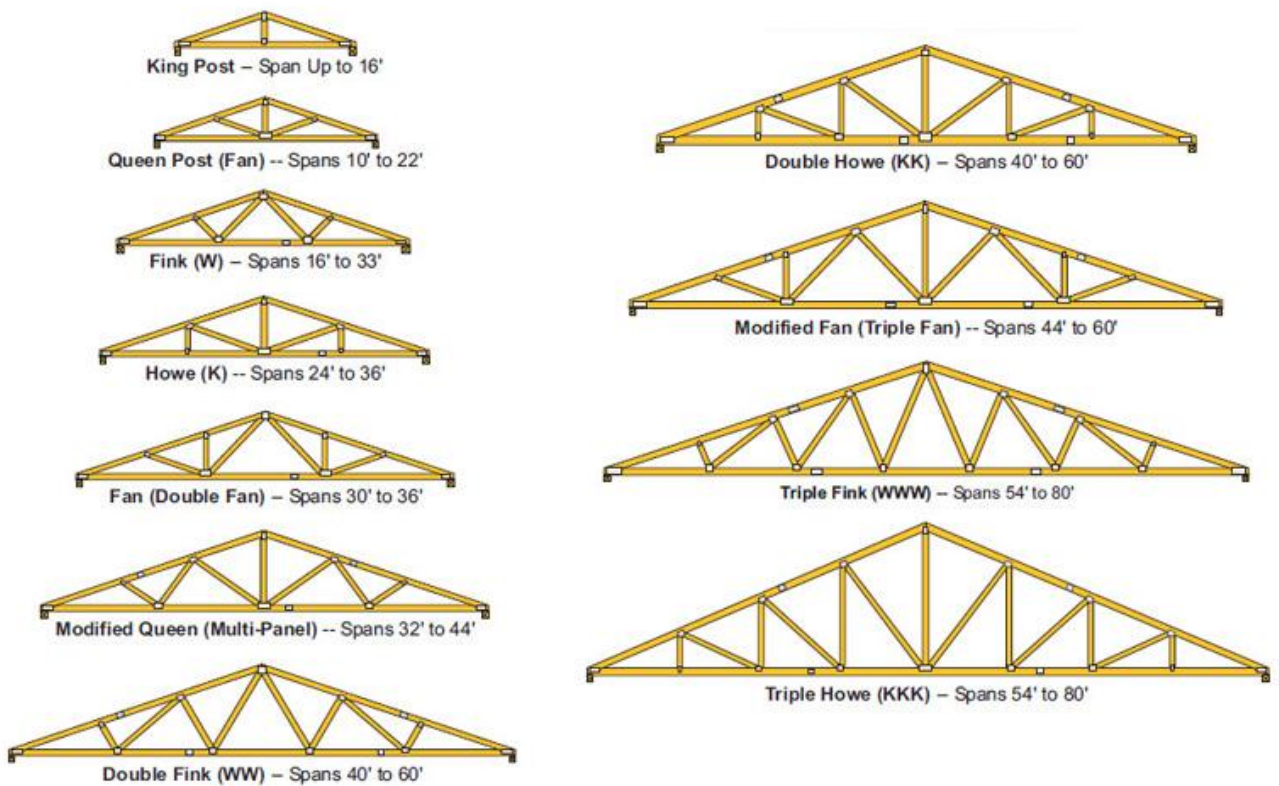


Fig.2 Types of truss – Based on span

Components of King Post Truss:

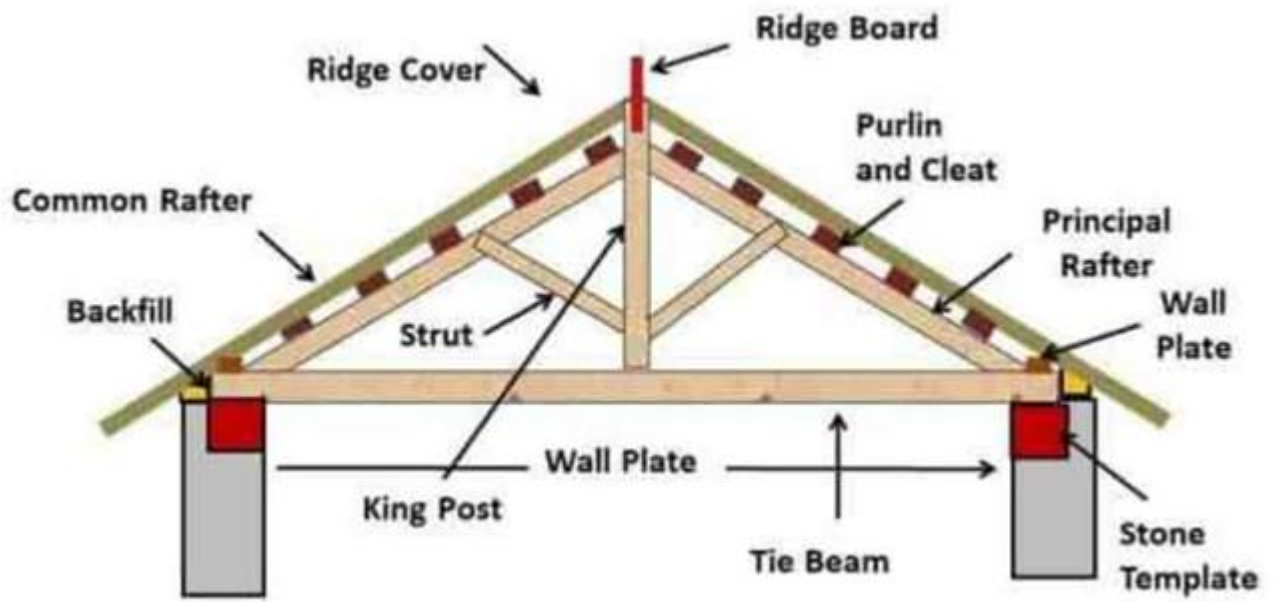


Fig.3 Components of truss

EX.NO: 7

Date:

Design and detailing of built up steel beam

Introduction:

Built-up members are used in many structures, such as compression members of trusses or supporting columns. They are composed from two rolled U-sections or four rolled angles.

A built-up beam is also known as compound beam. The built-up beams are used when the span, load and corresponding bending moment are of such magnitudes that rolled steel beam section becomes inadequate to provide required section modulus. Built-up beams are also used when rolled steel beams are inadequate for limited depth.

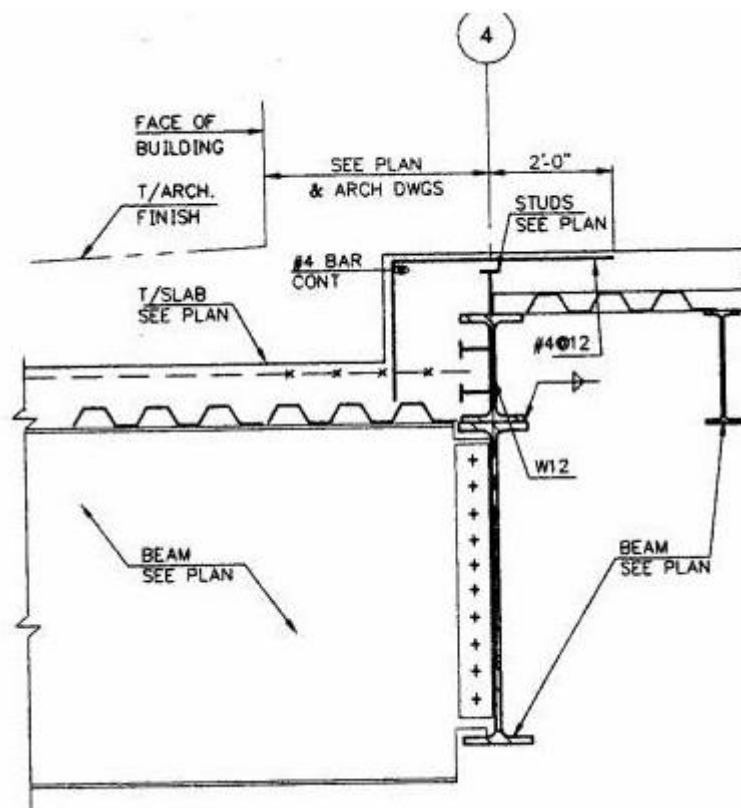
Types of beams such as:

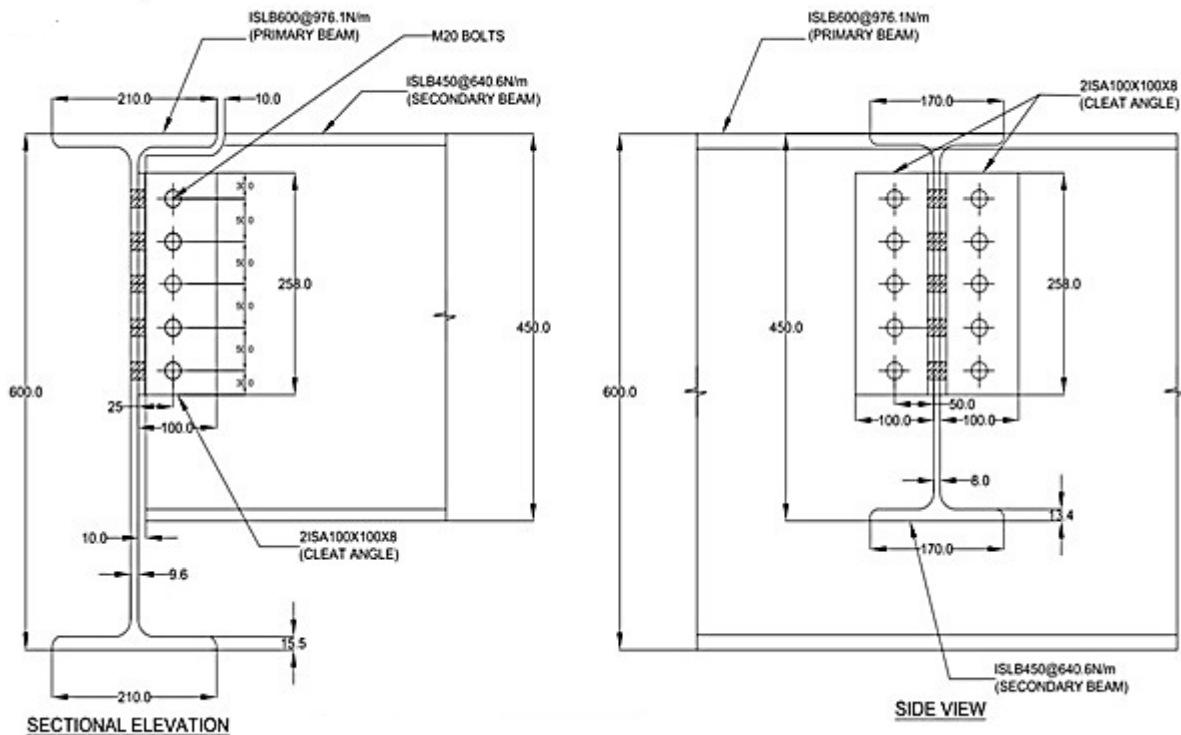
H, T, I, BOX and PLUS type.

Procedure:

A secondary beam ISLB450 is getting connected to the main beam ISLB600 with top flanges at same level. Two angles ISA100X100X8 are used for the connection, 5 bolts M20 diameter are used to connect angle to the web of the secondary beam.

Draw the sectional elevation and side view.





What is built-up beam?

- Steel beams are normally used for shorter spans and usually consists of a readymade available rolled section
- Builtup beams are used for sections, for which readymade available beam sections are not sufficient.
- The additional requirement of moments are compensated by using additional plates connected to the flanges of available I, or channel sections and they are called as built up beams.



Fig.1 Built Beam – I Section with Plates connected on top and bottom flange

Plate Girder:

- Plate girders are used to carry extensively large loads such as bridges.
- Here the various members such as web plates, flanges plates are connected as an assembly by means of suitable angles.
- Moreover since the plate girder is expected to carry huge loads, horizontal and vertical stiffeners are also used to avoid excess buckling and also at places where concentrated loads are acting.



Fig.2 Plate Girder – Typical Built-up beam section along with Stiffeners to resist heavy load in bridges



Fig.3 Plate Girder – Components

EX.NO: 8

Date:

Developing a design Programme for foundation using EXCEL spread sheet.

<u>FOOTING DESIGN FOR COLUMN</u>					
fck (N/mm²)	30				
fy (N/mm²)	500				
Uplift load	0				
Depth of Foundation	2				
Load (P) (in tons)	52				
Moment (M in ton-met)	0	2.9			
Safe Bearing Capacity(SBC) (in ton/m²)	12				
Area required (Areq) (in m²)	6.78				
Provide area (in mts) (A x B)	2.70	2.70	7.29		
Pedestal size (in mm) (p x q)	300	350			
Over burden on footing due to soil	2.70	2.70	1.35	18	17.71
Deduction for area of pedestal	0.3	0.35	1.35	- 18	-0.255
Self weight of footing	2.70	2.70	0.65	25	11.84 6
Over burden + Self weight	29.31				
extra 15 degrees wt for uplift (for considering enter" 1 "(or) enter " 0"	0				
	2.70	0.36	1.35	18	2.37
	2.70	0.36	1.35	18	2.37
	0.00				
Total load on footing	81.31				
Eccentricity in case of uplift (e)	0.04				
length of contact Le = 3((L/2)-e)	3.94	100	%		
pg (in ton/m²) (pg = load/area of footing)	11.15	12.03 7			
Cantilever projection of the footing from face of the pedestal	1200	1175			
©					
Moment (M in ton-met)	7.70	8.67			
Depth required (dreq) (dreq=	167.0	177.2			

$\sqrt{(M \cdot 1.5 \cdot 10^7) / (0.138 \cdot f_{ck} \cdot b)}$	2	0		
Provide total depth(D in mm)	500	500		
Effective depth (d in mm)	450	450		
$(M_u / (b \cdot d^2))$	0.57	0.642		
Refer table -2 of SP-16				
pt in %	0.135	0.152		
A_t in cm^2	6.069	6.843		
provide 16tor @150c/c along longer dir and 16tor @ 150 c/c				
along shorter direction at bottom				
<u>ONE WAY SHEAR</u>				
$V = pg(C-d) \cdot B$ (in tons)	22.58	23.56		
$\tau_v = V_u / Bd$ (in N/mm^2)	0.28	0.29	OK	O K
	$< \tau_c =$	0.28	0.30	
<u>TWO WAY SHEAR</u>				
$V = pg((A \cdot B) - ((p+d) \cdot (q+d)))$ (in tons)	74.61			
$\tau_v = V_u / b \cdot o \cdot d$ (in N/mm^2)	0.80	$<$	1.36 9	
		$\tau_c =$	3	
			OK	

Application of Footing:

- Footing is a **part of foundation** which is constructed with concrete or brickwork masonry and acts as a base to the floor columns and floor walls.
- The main function of footing is to **transfer the vertical loads directly to the soil.**
- The term footing is used in conjunction with Shallow foundation commonly.

Types of footing:

Depending on the depth of the soil in which the foundation is made, there are two types of foundation used in constructing buildings :

- Shallow foundation – These are used for small and light buildings. They are commonly referred to as spread footings or open footings. There are four types of shallow foundation:
 - Individual footing or isolated footing
 - Combined footing
 - Strip foundation
 - Raft or mat foundation
- Deep foundation – These are used for large structures. There are two main types of deep foundation:
 - Pile foundation
 - Drilled Shafts or caissons

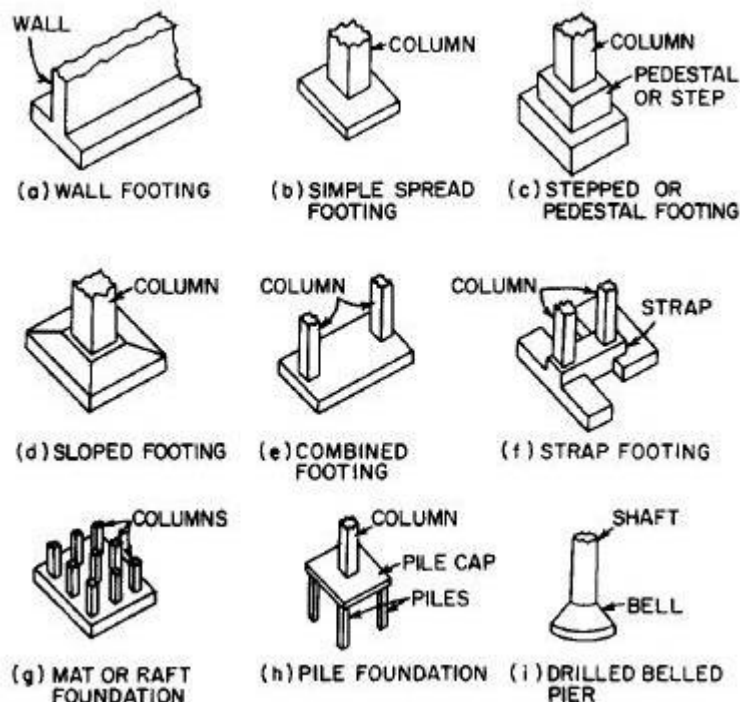


Fig.1 Types of footing



Fig.2 Typical image of isolated footing

EX.NO: 9

Date:

Detailing of RCC beam and RCC slab

Detailing of Slabs:

Design the slab for a hall of 3 m x 7 m clear dimensions, to act as a roof. The Slab rests on brick walls of 230 mm thickness. Adopt M – 20 and Fe – 415.

STEP 1: Dimensions of Slab:

Assuming a slab thickness @ 40mm per meter of Short Span,

$$D = 3.0 \times 40 = 120 \text{ mm}$$

IS: With a clear cover of 15 mm (For a maximum diameter of 12 mm in a mild environment):

refer table 16 of IS 456-2000) and with an ef. Cover of $15 + \frac{12}{2} = 21 \text{ mm}$,

eff. Thickness provided = $120 - 21 = 99 \text{ mm}$.

∴ Eff. Span of the slab is least of

- i. c/c of supports = $3.0 + 0.23 = 3.23 \text{ m}$
 - ii. clear span + d = $3.0 + 0.099 = 3.099 \text{ m}$
- ∴ $l_{eff} = 3.099 \text{ m}$

STEP 2:

Loading of Slab

Self Weight of slab = $25 \times 0.12 = 3.0 \text{ kN/m}^2$

Finishes (@ 1 kN/m^2) = 1.0 kN/m^2

Live Load (From IS: 875, for roof = 1.5 kN/m^2

Slab with access	
	5.5 kN/m ²

Factored UDL = $5.5 \times 1.5 = 8.25 \text{ kN/m}$

STEP 3:

$$8.25 \times 3.098^2$$

Maximum bending moment = $\frac{8.25 \times 3.098^2}{8} = 9.9 \text{ kN-m}$

Critical shear at a distance 'd' from the face of

$$8.25 \times 3.099$$

$$\text{The support} = \frac{\quad}{2} - 8.25 (0.1485) = 11.56 \text{ kN}$$

STEP 4:

Check for effective depth:

$$d_{req} = \sqrt{\frac{9.9 \times 10^6}{0.138 \times 20 \times 1000}}$$

$$= 59.89 \text{ mm} < 99 \text{ mm provided (ok)}$$

* Note that for all slabs, the width is 1 m i.e., 1000 mm

STEP 5:

Design for Flexure

Main reinforcement required,

$$A_{st} = \frac{0.5 \times 20}{(1 - \sqrt{1 - \frac{4.6 \times 9.9 \times 10^6}{20 \times 1000 \times 99^2}})} \times 1000 \times 99$$

$$= \frac{415}{20 \times 1000 \times 99^2}$$

$$= 295.81 \text{ mm}^2 > A_{stmin}$$

$$= \frac{0.12}{100} \times 1000 \times 120 = 144 \text{ mm}^2 \text{ (ok)}$$

$$100$$

Assuming 8 mm Ø bars, spacing required

$$S = 1000 \times \frac{A_{st}}{A_{stmin}}$$

$$\frac{295.81}{144}$$

Where S = spacing of bars, a_{st} = Bar area, A_{st} = required area of steel

$$A_{st} \times 8^2$$

$$1000 \times \frac{\quad}{\quad}$$

$$\therefore S = \frac{1000 \times 295.81}{8^2} = 169.8 \text{ mm}$$

295.81

Maximum spacing of bars is not to exceed least of

- i. $3d = 3 \times 99 = 297$
- ii. 300 mm

∴ Provide 8 mm Ø bars @ 160 mm c/c as main reinforcement and bend alternate bars.

$$\therefore \text{Steel provided} = 1000 \times \frac{50.26}{160} = 314.12 \text{ mm}^2.$$

$$\text{Distribution reinforcement} = \frac{0.12}{100} \times 1000 \times 120 = 144 \text{ mm}^2.$$

28.28

$$\text{Assuming 6mm } \varnothing \text{ bars, spacing required} = 1000 \times \frac{144}{144} = 196.38 \text{ mm}$$

Maximum spacing of distribution steel is not to exceed least of

- i. $5d = 5 \times 99 = 495 \text{ mm}$
- ii. 450 mm

∴ Provide 6mm Ø bars @ 190 mm c/c.

STEP 6:

Check for Shear

Shear stress developed at critical section

$$11.56 \times 10^3$$

$$\tau = \frac{11.56 \times 10^3}{1000 \times 99}$$

$$= 0.166 \text{ N/mm}^2$$

< $\tau_{cr} = 2.8 \text{ N/mm}^2$ from table – 20 of code (ok)

$$\text{For } p_t = \frac{314.12}{1000 \times 99} \times 100$$

$$= 0.32\%$$

and for M-20, from table-19 of code,

$$(0.32 - 0.25)$$

$$p_t = 0.36 + \frac{(0.50 - 0.25)}{100}$$

x (0.48 –
0.36)

$$= 0.3936 \text{ N/mm}^2.$$

For solid slabs, as per clause 40.2.1.1 of the code, the value of σ_{cr} can be enhanced by a factor (k) depending on thickness. In this case for a slab of 120 mm thickness

$$K = 1.3$$

$$\therefore \sigma_{cr} = 1.3 \times 0.3936 \\ = 0.511 \text{ N/mm}^2$$

As σ_{cr} developed is very much less than σ_{cr} , no shear reinforcement is necessary in the slab.

STEP 7:

Check for Bond

At the centre of support, even if 50% of the tension steel is existing;

$$A_{st} = \frac{314.12}{2} = 157.06 \text{ mm}^2. \\ = 7.87 \text{ mm.}$$

$$0.87 \times 415 \times 157.06$$

$$\sigma_{cr} = \frac{\quad}{\quad}$$

$$0.36 \times 20 \times 1000$$

$$< x_{u\max} = 0.45 \times 99 = 47.52 \text{ mm}$$

\therefore Section is under reinforced.

$$\text{Hence } M_u = 0.87 \times 415 \times 157.06 (99 - 0.42 \times 7.87)$$

$$= 5.43 \times 10^6 \text{ N - mm}$$

$$L_d \text{ for high strength deformed bars intension} = 47 \phi$$

$$\text{Shear at the support} = 8.25 \times 3.099/2 = 12.78 \text{ kN}$$

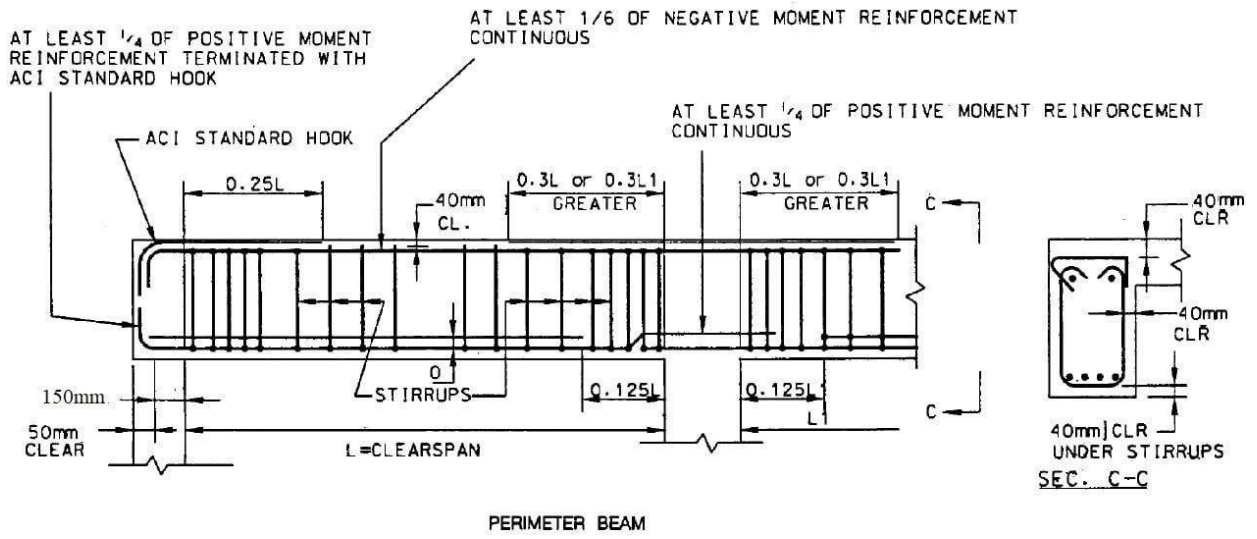
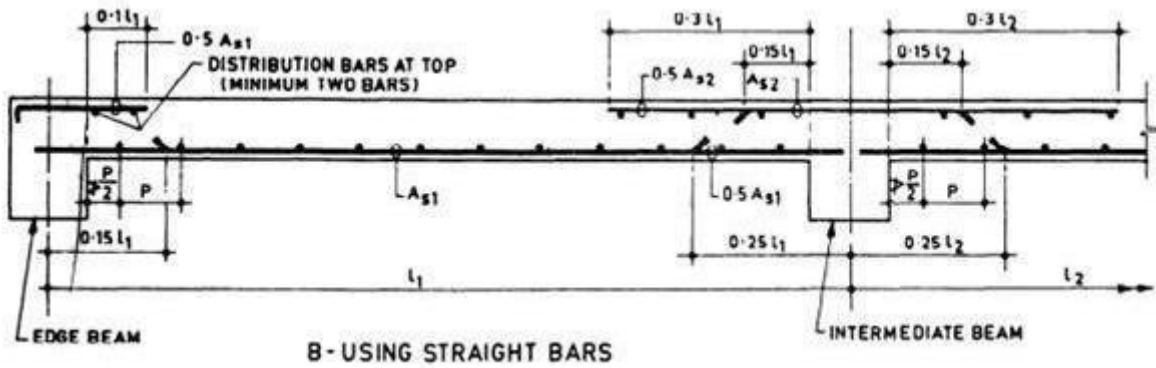
\therefore Assuming L_o initially as zero

$$47\phi \leq \frac{1.3 \times 5.43 \times 10^6}{12.78 \times 10^3} + 0.$$

$$12.78 \times 10^3$$

$$\text{i.e. } \phi \leq 11.75 \text{ mm.}$$

As the bar diameter used is 8 mm which is < 11.75 mm, the bars are safe in bond.



Slab:

- A slab is a structural element, made of concrete that is used to create flat horizontal surfaces such as floors, roof decks and ceilings.
- A slab is generally several inches thick and supported by beams, columns, walls, or the ground.

Beam:

- A beam is a structural element that primarily resists loads applied laterally to the beam's axis (an element designed to carry primarily axial load would be a strut or column).
- Its mode of deflection is primarily by bending.
- The loads applied to the beam result in reaction forces at the beam's support points.

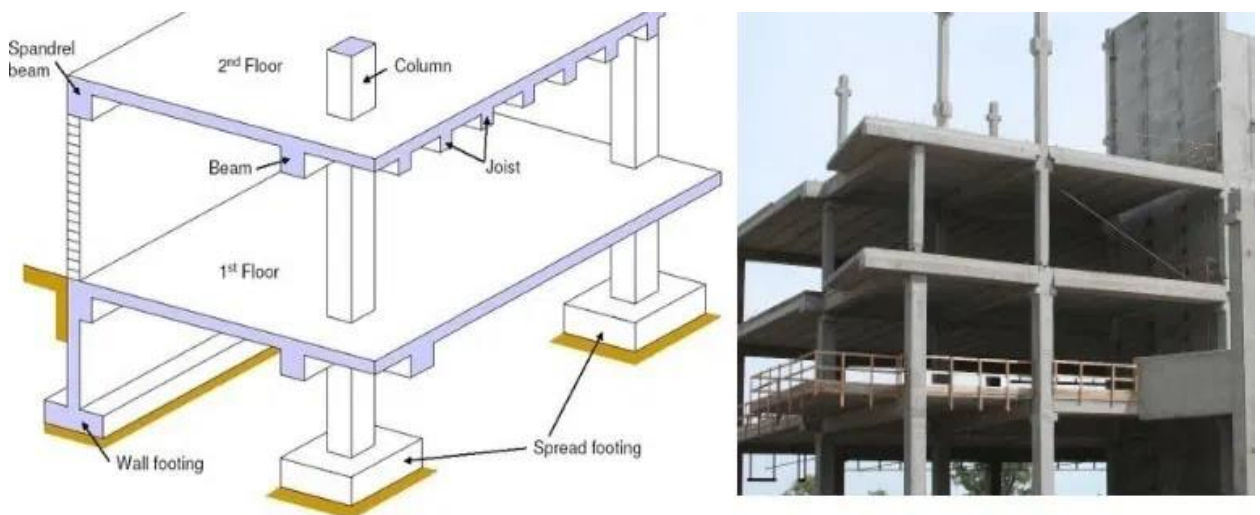


Fig.1 Typical structure with slab and beam

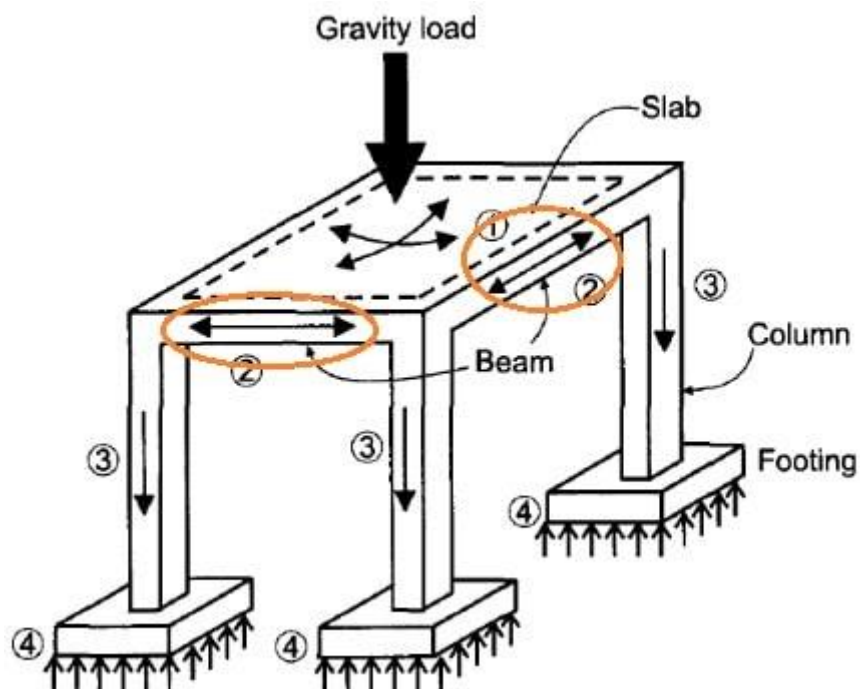
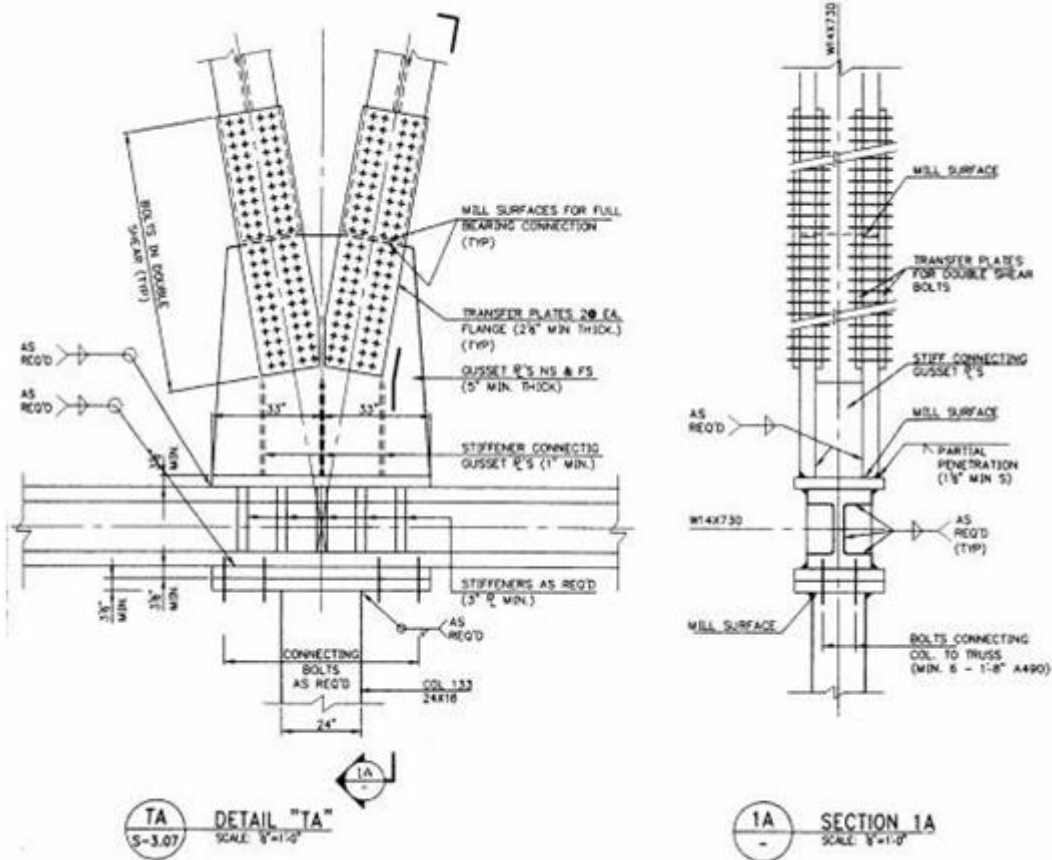


Fig.2 Typical load distribution diagram of structure

EX.NO: 10

Date:

Detailing of steel built up compression member.



What is built-up compression member?

- Built-up members are used in many structures, such as compression members of trusses or supporting columns.
- They are composed from two rolled U-sections or four rolled angles. The distance between them is fixed by lacing or battening.

Laced Columns:

- Lacing consists of **connecting the components of the column by a system of generally flat plates**. (In some cases angles and channels are also used as lacings).
- Lacing plates may be 50 mm to 75 mm wide and 8 mm to 10 mm thick.



Fig.1 Typical laced column

Batten Columns:

- Batten columns are **compression members composed of two or more similar longitudinal components (chords) which are connected at points along their length with batten plates as transverse connectors**.
- These connectors ensure that the column behaves as one integral unit to achieve the maximum axial capacity.



Fig.2 Typical batten column