

ATRIA INSTITUTE OF TECHNOLOGY
Affiliated to VTU



LABORATORY MANUAL
18CVL66- SOFTWARE APPLICATION
LABORATORY

2020-21

DEPARTMENT OF CIVIL ENGINEERING
ATRIA INSTITUTE OF TECHNOLOGY
Adjacent to Bangalore Baptist Hospital
Hebbal, Bengaluru-560024

B. E. CIVIL ENGINEERING
Choice Based Credit System (CBCS) and Outcome Based Education (OBE)
SEMESTER - VI

SOFTWARE APPLICATION LABORATORY

Course Code	18CVL66	CIE Marks	40
Teaching Hours/Week(L:T:P)	(0:2:2)	SEE Marks	60
Credits	02	Exam Hours	03

Course Learning Objectives: This course will enable students to

1. Use industry standard software in a professional set up.
2. Understand the elements of finite element modeling, specification of loads and boundary condition, performing analysis and interpretation of results for final design.
3. Develop customized automation tools.

Module -1

Use of civil engineering software's:

Use of software's for:

1. Analysis of plane trusses, continuous beams, portal frames.
2. 3D analysis of multistoried frame structures.

Module -2

1. **Project Management- Exercise on Project planning and scheduling of a building project using any project management software:**

- a. Understanding basic features of Project management software
- b. Constructing Project: create WBS, Activities, and tasks and Computation Time using Excel spread sheet and transferring the same to Project management software.
- c. Identification of Predecessor and Successor activities with constrain
- d. Constructing Network diagram (AON Diagram) and analyzing for Critical path, Critical activities and Otherton Critical paths, Project duration, Floats.
- e. Study on various View options available
- f. Basic understanding about Resource Creation and allocation
- g. Understanding about Splitting the activity, Linking multiple activity, assigning Constrains, Merging Multiple projects, Creating Baseline Project

1. **GIS applications using open source software:**

- a. To create shape files for point, line and polygon features with a map as reference.
- b. To create decision maps for specific purpose.

Module -3

Use of EXCEL spread sheets:

Design of singly reinforced and doubly reinforced rectangular beams, design of one way and two way slabs, computation of earthwork, Design of horizontal curve by offset method, Design of super elevation.

Course Outcomes: After studying this course, students will be able to:

use software skills in a professional set up to automate the work and thereby reduce cycle time for completion of the work

Question paper pattern:

- The question paper will have 6 questions under 3 modules.
- There will be two full questions (with a maximum of three subdivisions, if necessary) from each module.
- Each full question shall cover the topics under a module.
- Module-1: 40 Marks, Module-2: 30 Marks, Module-3: 30 Marks.
- The students shall answer three full questions, selecting one full question from each module.

Reference Books: Training manuals and User manuals and Relevant course reference books

LIST OF OPEN SOURCE SOFTWARE'S USED



(FOR VI SEMESTER B.E.)

**AS PER NEW SYLLABUS
PRESCRIBED BY**

VISVESVARAYA TECHNOLOGICAL UNIVERSITY

Academic Year - 20 -20

Name _____

USN _____

Semester _____

Section _____

Branch _____

Faculty

Head of the Department

EXP.NO.	EXPERIMENT NAME	PAGE. NO.
	MODULE 1	
1	Analysis of plane trusses	
2	Continuous beams	
3	Portal frames	
4	3D analysis of multistoried frame structures	
	MODULE 2	
5	Understanding basic features of Project management software.	
	Constructing Project: create WBS, Activities, and tasks and Computation time using Excel spread sheet and transferring the same to Project management software.	
	Identification of Predecessor and Successor activities with constrain.	
	Constructing Network diagram (AON Diagram) and analyzing for Critical path, Critical activities and Other non-Critical paths, Project duration, Floats.	
	Study on various View options available	
	Basic understanding about Resource Creation and allocation	
	Understanding about Splitting the activity, Linking multiple activity, Assigning Constrains, Merging Multiple projects, Creating Baseline Project.	
6	To create shape files for point, line and polygon features with a map as Reference.	
7	To create decision maps for specific purpose.	
	MODULE 3	
8	Design of singly reinforced rectangular beams.	
9	Doubly reinforced rectangular beams.	
10	Design of one way slabs.	
11	Design two way slabs.	
12	Computation of earthwork.	
13	Design of horizontal curve by offset method.	
14	Design of super elevation.	

Module I

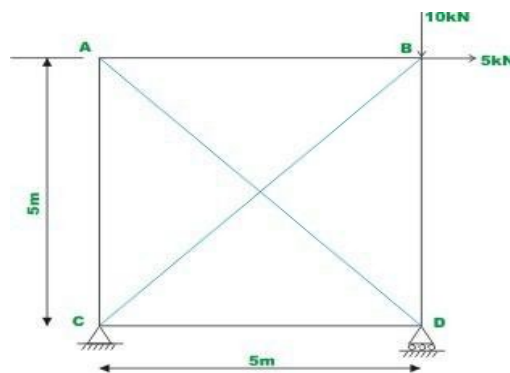
Structural Analysis Using STAAD

Introduction to STAAD pro: It is a structural analysis and design computer program originally developed by Research Engineers International at Yorba Linda, California in 1997. In late 2005, Research Engineers International was bought by Bentley Systems. The commercial version, STAAD.Pro, is one of the most widely used structural analysis and design software products worldwide. It supports several steel, concrete and timber design codes.

Experiment no: 1

Analysis of plane trusses:

1. (a) Analyze the given truss using STAAD pro software.



Problem study: The given truss has a 5m span between two supports and a height of 5m. Two forces (vertical and horizontal forces) are acting at point B.

Aim: To find the reactions at the supports and draw the bending moment, shear force, and axial force diagrams using STAAD pro.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, and Printer.

Procedure:

1. Open STAAD software.
2. A new project is started with unit's m & KN and structure type as Plane.
3. The given structure is drawn in the workspace using the graphical user interface of the software according to given dimensions.

4. Define the material property as steel, assign defined property to drawn truss using commands.
5. Define Supports and assign to specified nodes of the structure.
6. Define load as nodal load and assign to specified nodes in trusses member by using node cursor.
7. Perform analysis command was given.
8. The file was saved and run analysis command was executed.
9. It is made sure that there is no error is indicated in the output window.
10. Post processing mode was selected.
11. The results were properly arranged using the tools.
12. Printouts were taken.

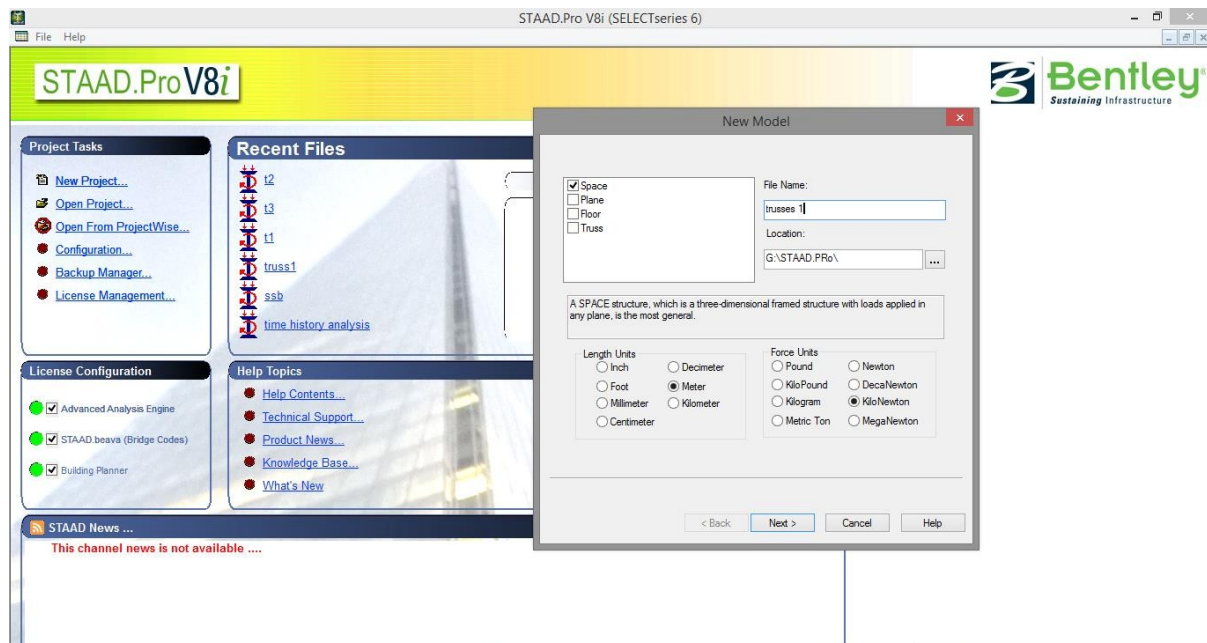
Results:

1. Reaction at support A =
2. Reaction at support B =
3. Shear force, bending moment and member force diagrams are attached.

E link- <https://youtu.be/h83KKRtbXwM>.

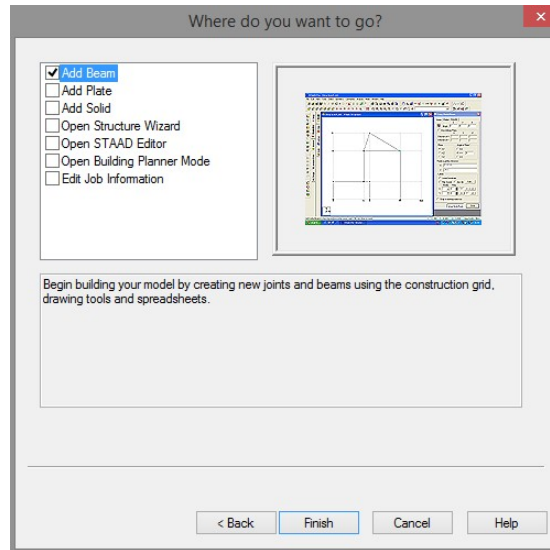
Step-by-step operation on software:

Step: 1 Open STAAD pro software and set units.



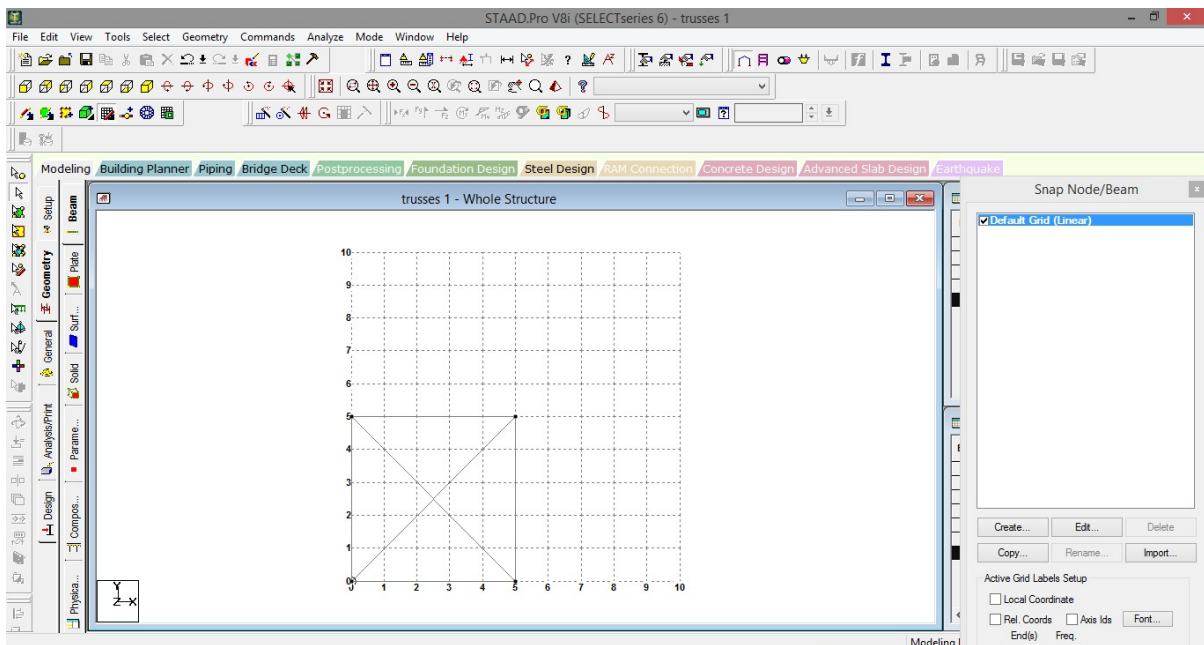
Start with new project > select space option in new model window > Type file name > choose storage location > set units to meter and kilo newton.

Step: 2 Choose add beams option.



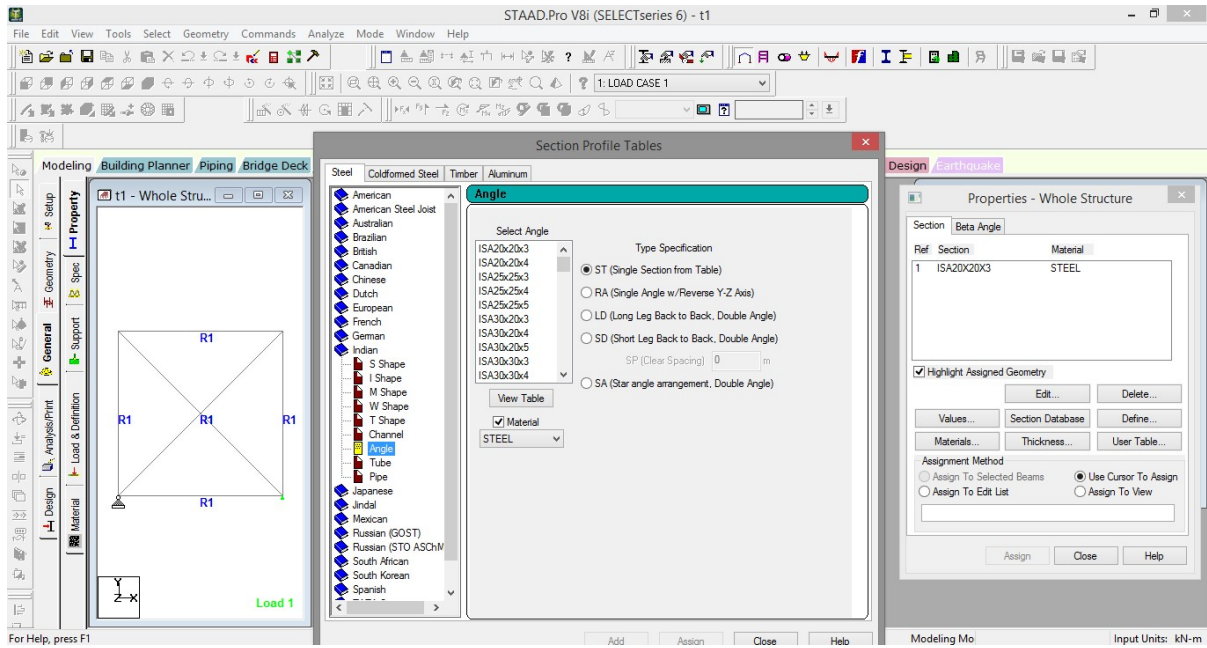
Choose add beam options to draw model.

Step: 3 Open STAAD pro to make model by using snap node or beam option.



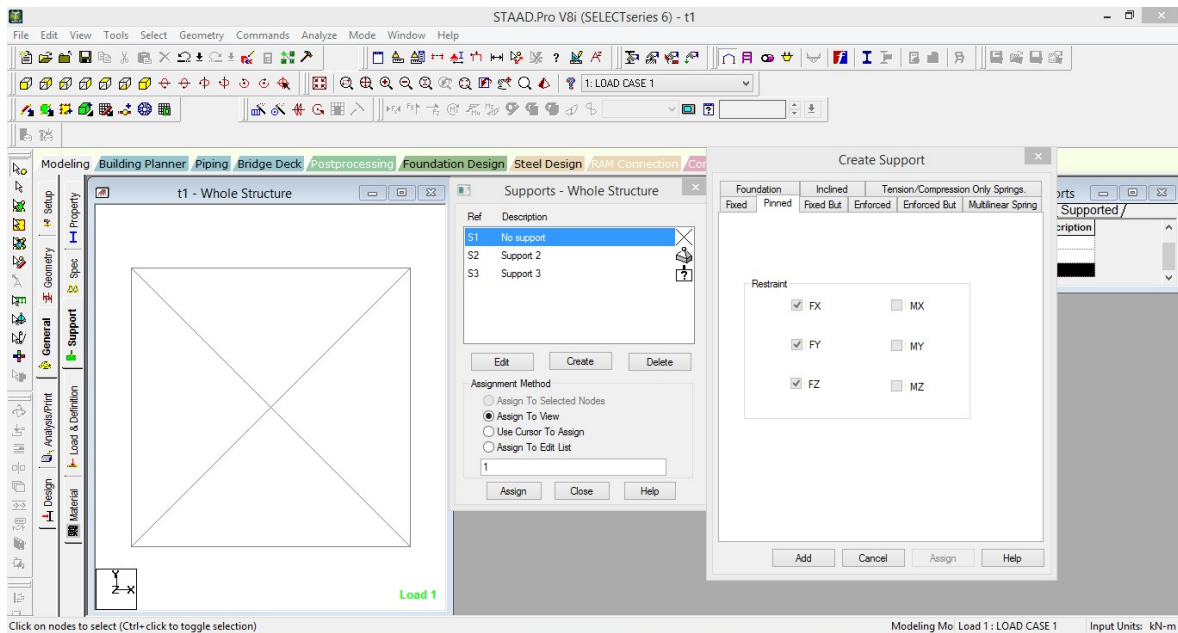
The above figure shows that the truss drawn according to given question. If you want to increase grid spacing use edit command.

Step: 4 Defining of material and assign material to truss.



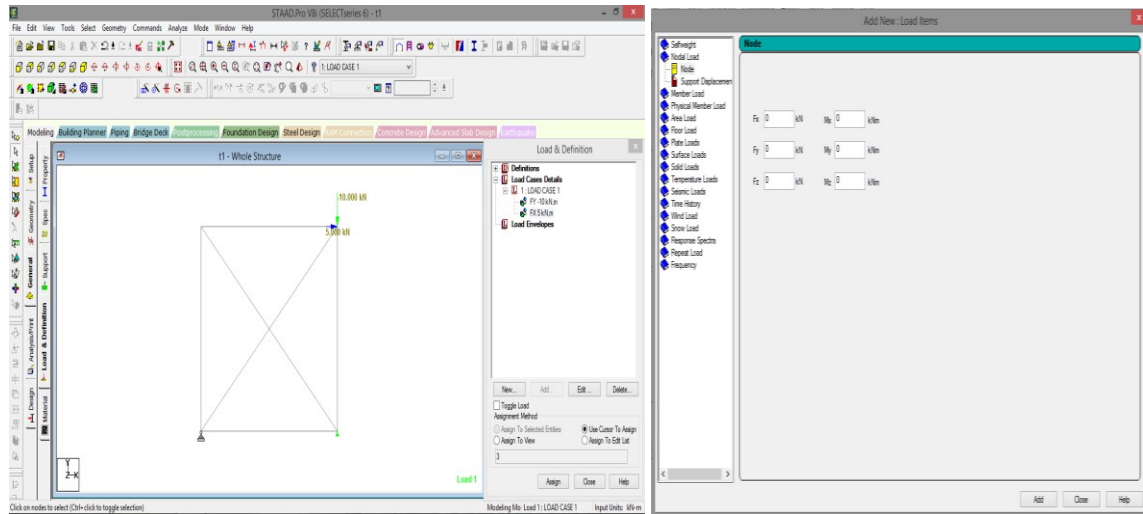
Go to General > property > section database > select Indian angle > assign by using assign to view option

Step: 5 Create support condition and assign to nodes.



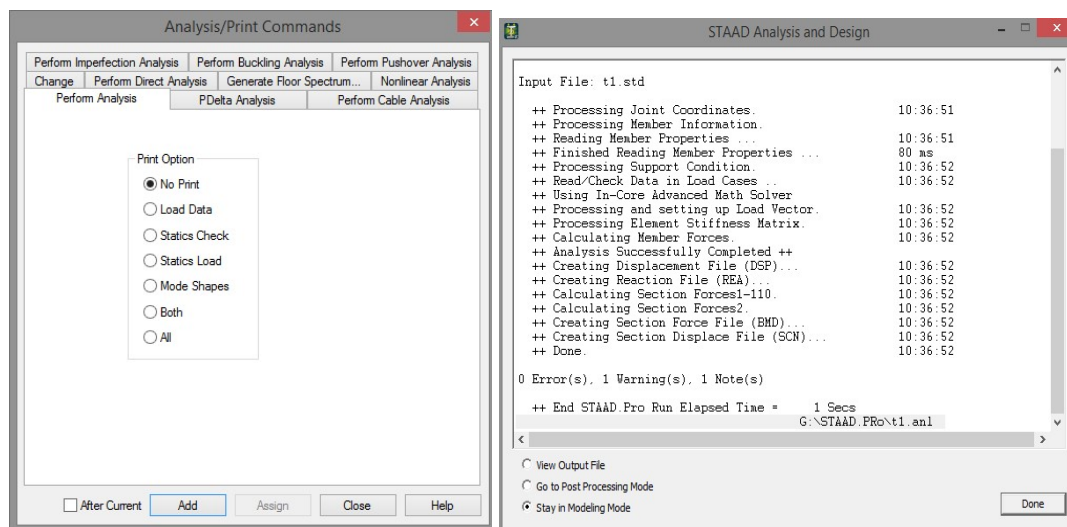
Go to support > create support as pinned (MX, MY, MZ are released), and roller support (FX, FZ, MX, MY, MZ are released) by considering fixed but option > assign support to model by using node cursor and assign to selected node option.

Step: 6 Define load case and assign load cases.



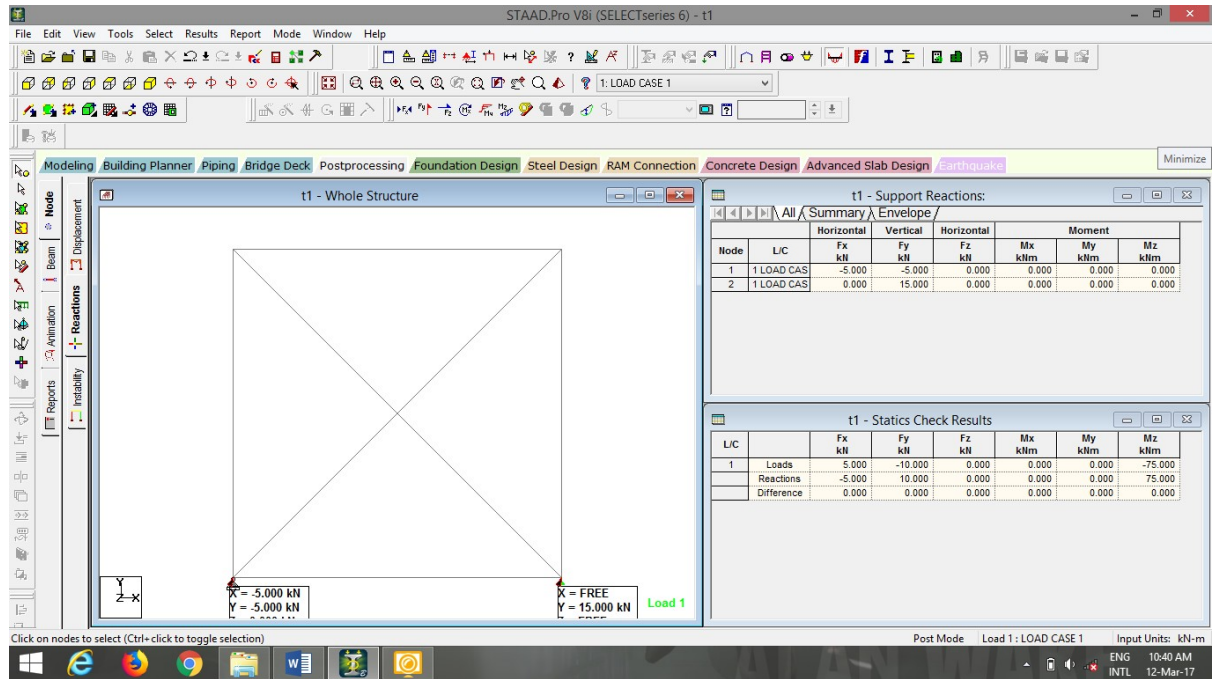
Go to loads and definition > Click on load case details add load case1 > Click on load case 1 add nodal load as FY= -10kN, FX= 5kN add > select defined loads and assign to truss.

Step: 7 Analysis/print.

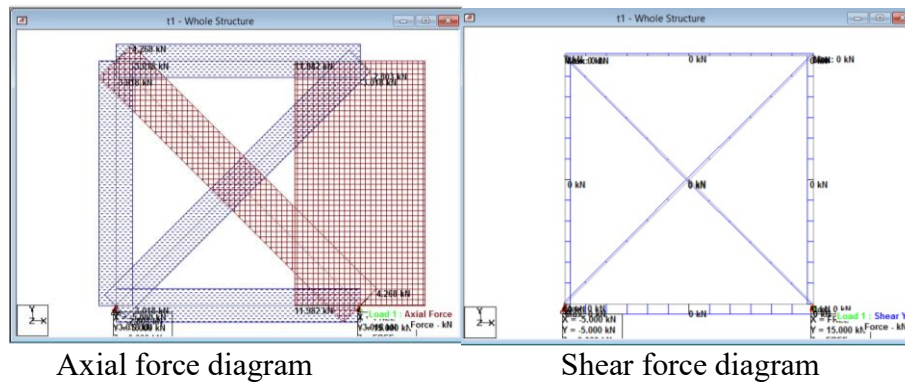


Go to analysis/print > select all add > go to analyze > run analysis > If no error found go to post processing mode.

Step: 8 Results.

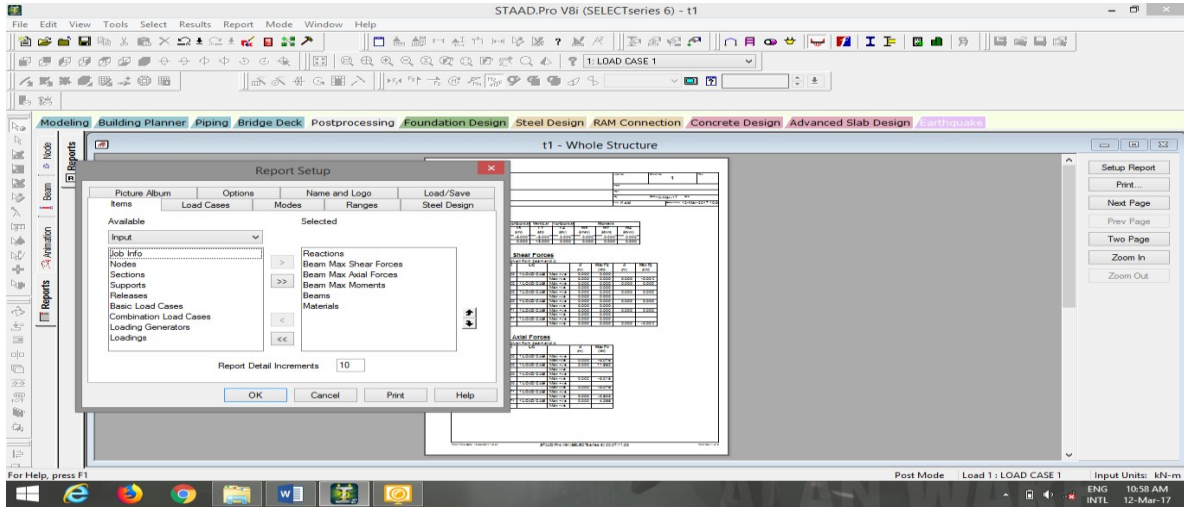


In post processing mode > go to reactions to get support reactions (in this problem the support reaction is -5 , -5 at fixed support and 15 at pinned support).



Bending moment diagram

Step: 9 Report extraction.



In Post processing mode go to report > select input and output parameters should be extracted > ok > go to file > export report > MS WORD.

Reactions

Node	L/C	Horizontal	Vertical	Horizontal	Moment		
		FX (kN)	FY (kN)	FZ (kN)	MX (kNm)	MY(kNm)	MZ(kNm)
1	1:LOAD CASE 1	-5.000	-5.000	0.000	0.000	0.000	0.000
2	1:LOAD CASE 1	0.000	15.000	0.000	0.000	0.000	0.000

Beam Maximum Shear Forces

Distances to maxima are given from beam end A.

Beam	Node A	Length (m)	L/C		d (m)	Max Fz (kN)	d (m)	Max Fy (kN)
1	1	5.000	1:LOAD	Max		0.000		
				Max -ve		0.000		-0.000
2	2	5.000	1:LOAD	Max		0.000		0.000
				Max -ve		0.000		
3	3	5.000	1:LOAD	Max		0.000		0.000
				Max -ve		0.000		
4	4	5.000	1:LOAD	Max		0.000		0.000
				Max -ve		0.000		
5	1	7.071	1:LOAD	Max		0.000		0.000
				Max -ve		0.000		
6	4	7.071	1:LOAD	Max		0.000		
				Max -ve		0.000		-0.000

Beam Maximum Axial Forces

Distances to maxima are given from beam end A.

Beam	Node A	Length (m)	L/C		d (m)	Max Fx (kN)
1	1	5.000	1:LOAD CASE 1	Max +ve		
				Max -ve	0.000	-3.018
2	2	5.000	1:LOAD CASE 1	Max +ve	0.000	11.982
				Max -ve		
3	3	5.000	1:LOAD CASE 1	Max +ve		
				Max -ve	0.000	-3.018
4	4	5.000	1:LOAD CASE 1	Max +ve		
				Max -ve	0.000	-3.018
5	1	7.071	1:LOAD CASE 1	Max +ve		
				Max -ve	0.000	-2.803
6	4	7.071	1:LOAD CASE 1	Max +ve	0.000	4.268
				Max -ve		

Beam Maximum Moments

Distances to maxima are given from beam end A.

Beam	Node A	Length (m)	L/C		d (m)	Max My (kNm)	d (m)	Max Mz (kNm)
1	1	5.000	1:LOAD CASE 1	Max +ve	0.000	0.000	5.000	0.000
				Max -ve	0.000	0.000	0.000	-0.000
2	2	5.000	1:LOAD CASE 1	Max +ve	0.000	0.000	0.000	0.000
				Max -ve	0.000	0.000	5.000	-0.000
3	3	5.000	1:LOAD CASE 1	Max +ve	0.000	0.000	0.000	0.000
				Max -ve	0.000	0.000	5.000	-0.000
4	4	5.000	1:LOAD CASE 1	Max +ve	0.000	0.000	0.000	0.000
				Max -ve	0.000	0.000	5.000	-0.000
5	1	7.071	1:LOAD CASE 1	Max +ve	0.000	0.000	0.000	0.000
				Max -ve	0.000	0.000	7.071	-0.000
6	4	7.071	1:LOAD CASE 1	Max +ve	0.000	0.000	7.071	0.000
				Max -ve	0.000	0.000	0.000	-0.000

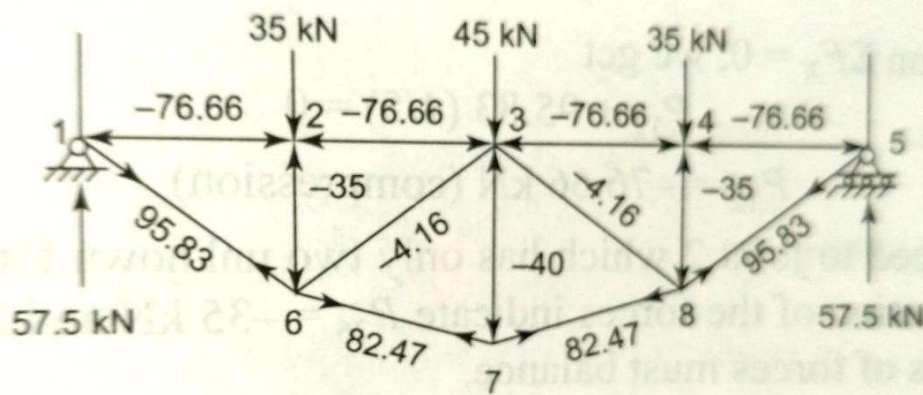
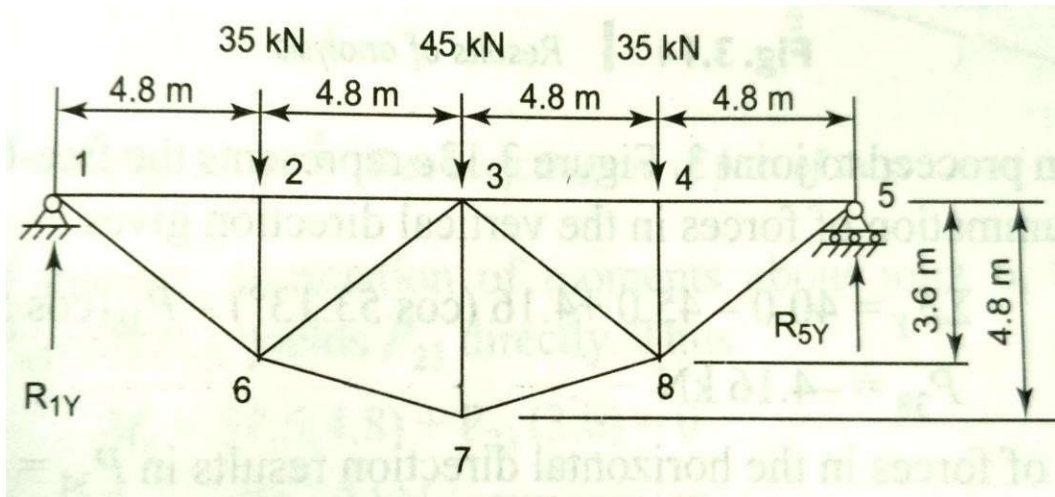
Beams

Beam	Node A	Node B	Length (m)	Property	□ (degrees)
1	1	2	5.000	1	0
2	2	3	5.000	1	0
3	3	4	5.000	1	0
4	4	1	5.000	1	0
5	1	3	7.071	1	0
6	4	2	7.071	1	0

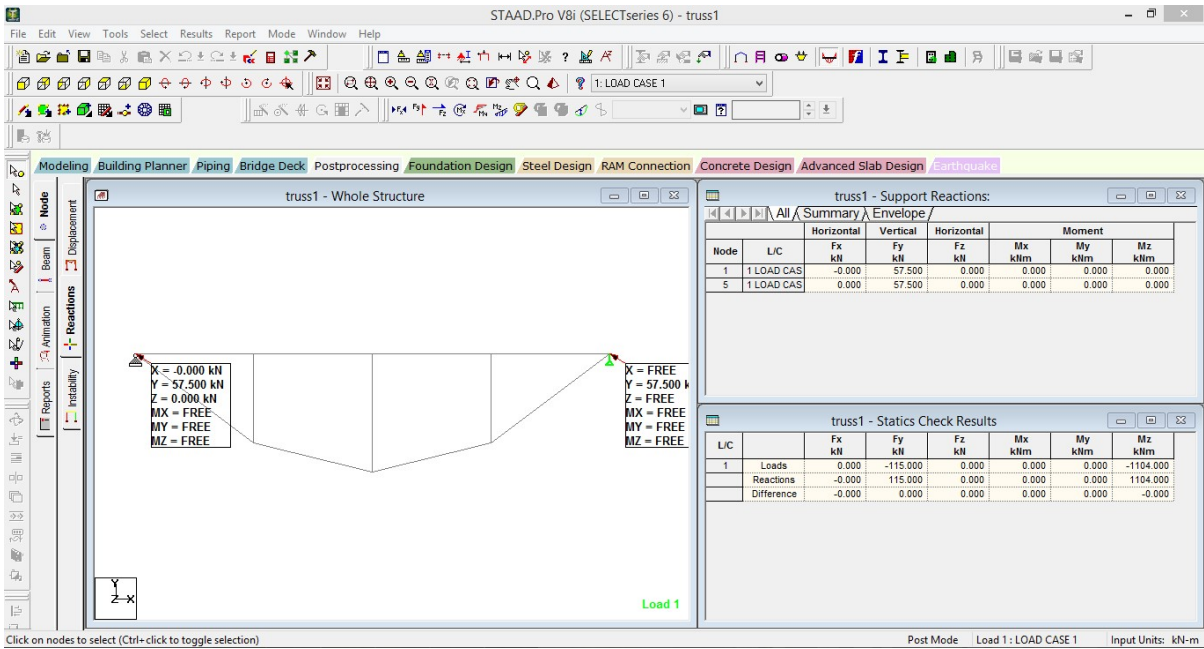
Materials

Mat	Name	E (kN/mm ²)	ν	Density (kg/m ³)	α (/°C)
3	STEEL	205.000	0.300	7.83E 3	12E -6
4	STAINLESSSTEEL	197.930	0.300	7.83E 3	18E -6
5	ALUMINUM	68.948	0.330	2.71E 3	23E -6
6	CONCRETE	21.718	0.170	2.4E 3	10E -6

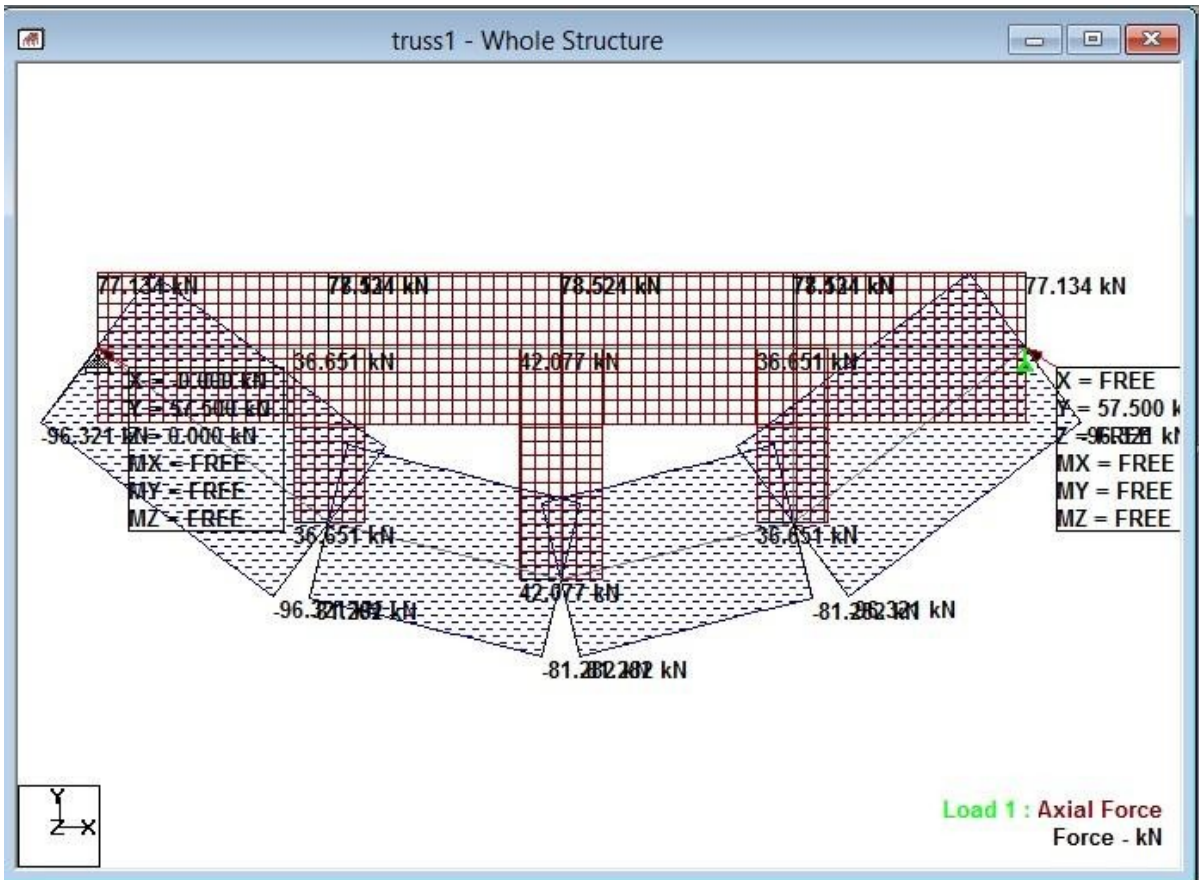
(b) Analyze the given truss using STAAD pro software.



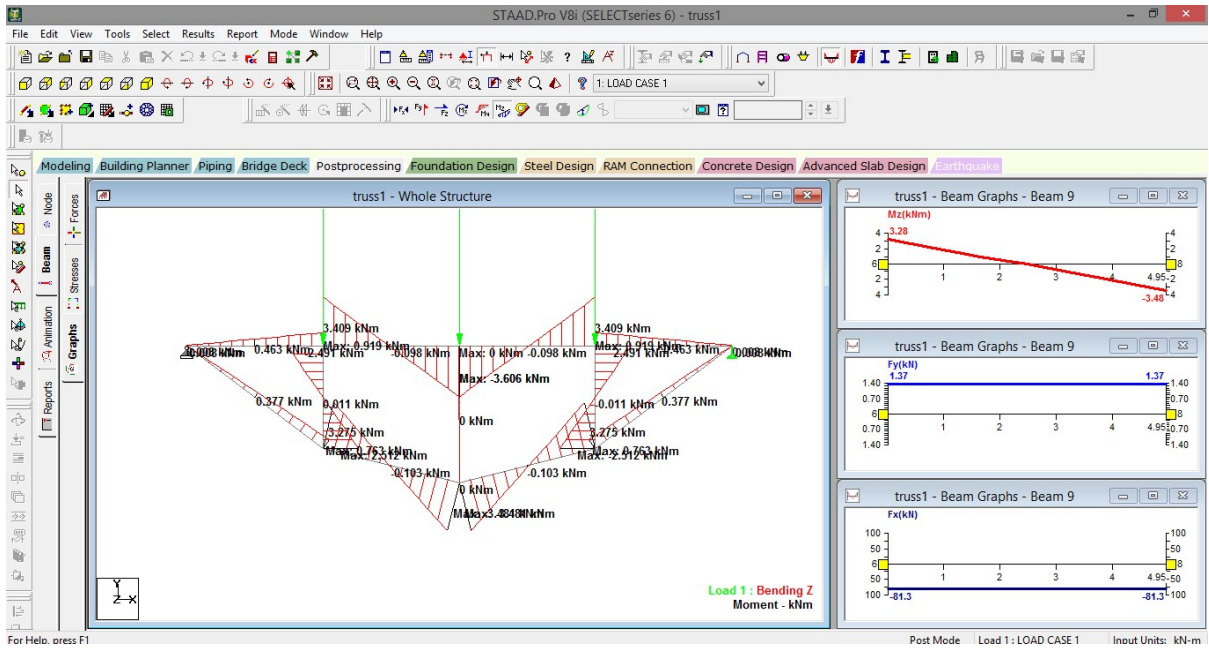
Forces in members obtained from manual calculation.



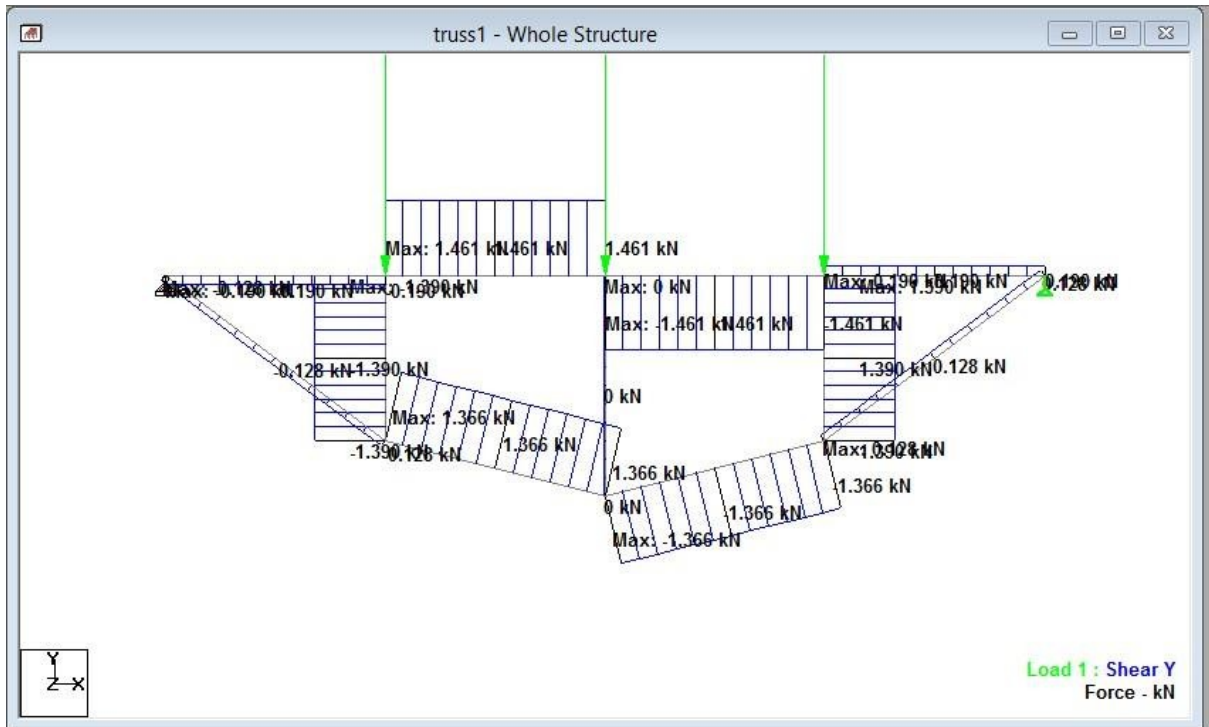
Support reactions obtained in software is matching with manual calculations.



Forces in members obtained from software.



Bending moment diagram with graphs

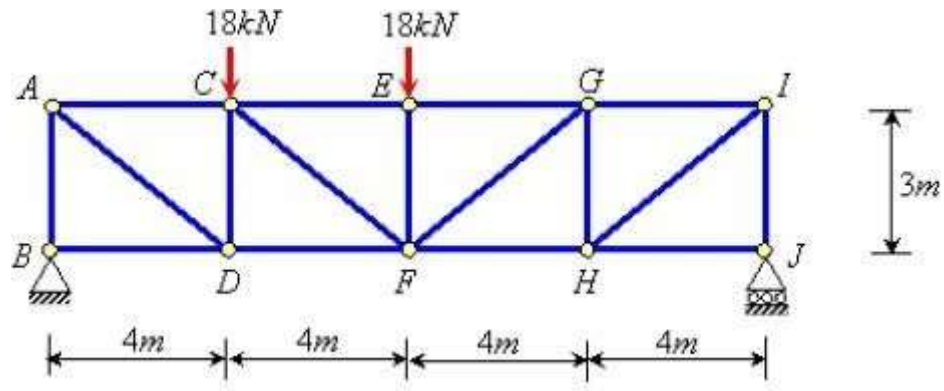


Shear force diagram.

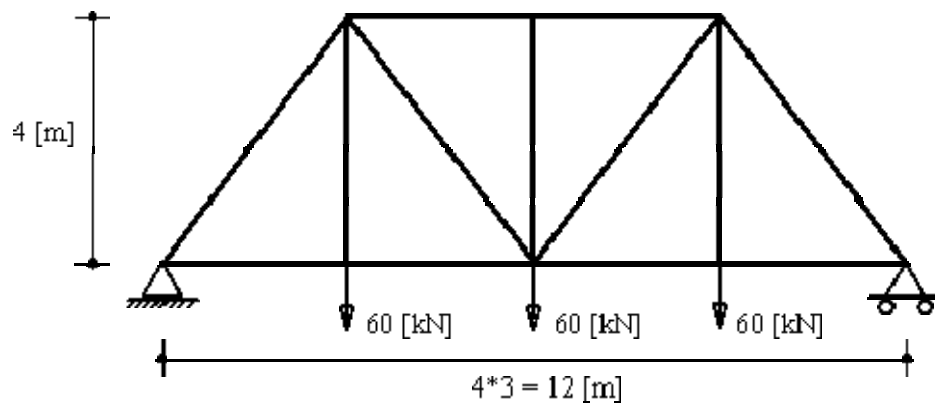
E link: - <https://youtu.be/FzOKmn2xG20>

Examples to solve.

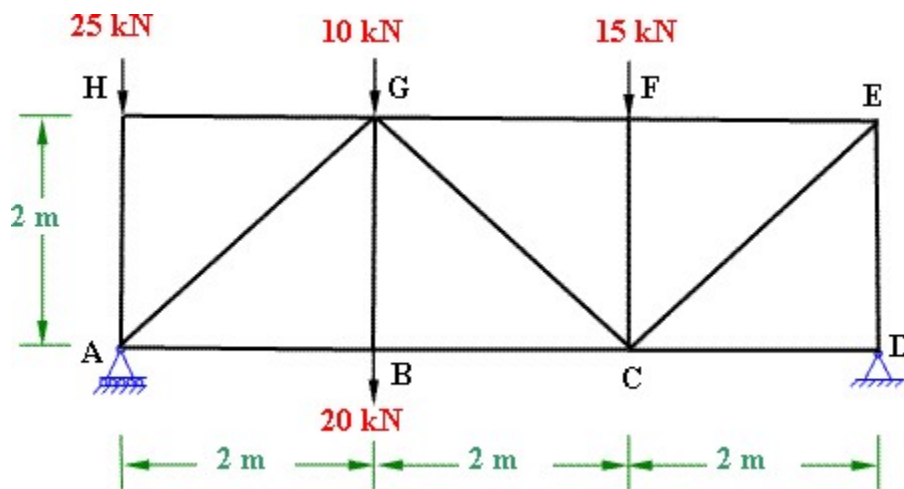
1.



2.

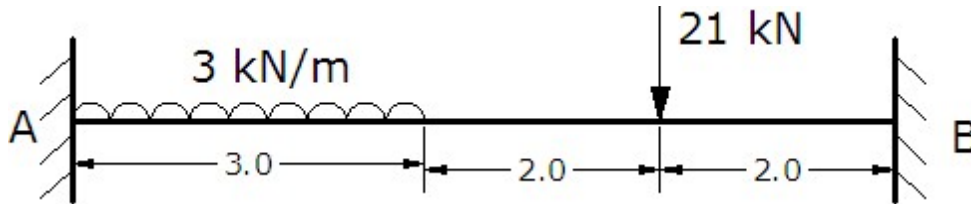


3.



E link: - <https://youtu.be/gPKFfU3iTyQ>

2 (a) Analysis of a Fixed Beam Using STAAD pro software.



Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

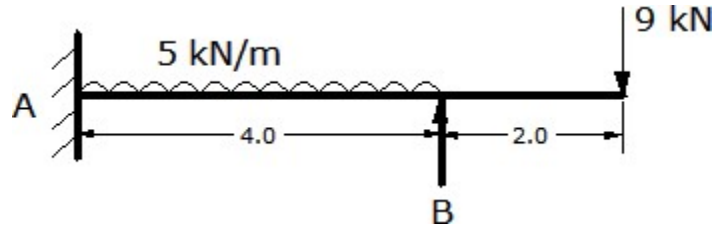
- 1) Open STAAD pro software.
- 2) Start with new project set unit's to m & KN and structure type as Plane.
- 3) The given structure is drawn in the workspace using the graphical user interface of the software.
- 4) Define section property as rectangular assume suitable dimension assigned to the members drawn by using assign to view option.
- 5) Create support condition as fixed assign to the beam at required nodes.
- 6) Define load cases in a single load case, as concentrated force of 21kn and uniform force of 3kn/m and assign on beam member.
- 7) Perform analysis command was given, the file was saved and run analysis command was executed.
- 8) It is made sure that there is no error is indicated in the output window.
- 9) Post processing mode was selected.
- 10) The results were properly arranged using the tools.
- 11) Printouts were taken.

Results:

1. Reaction at support A =
2. Reaction at support B =
3. Shear force and bending moment diagrams are attached.

E link: - <https://youtu.be/mx3di5cGO6s>

(b) Analysis of a Propped Cantilever Beam Using STAAD



Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

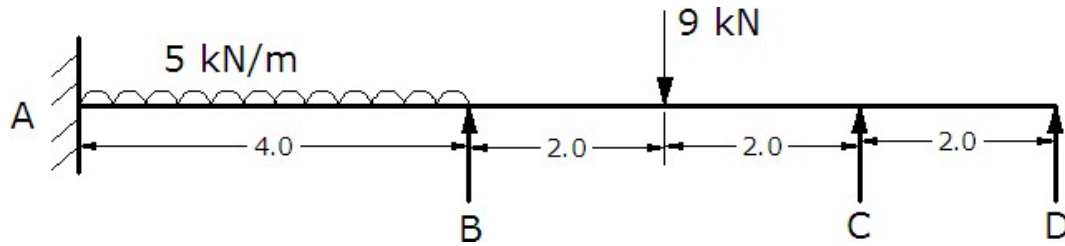
- 1) Open STAAD pro software.
- 2) Start with new project set unit's to m & KN and structure type as Plane.
- 3) The given structure is drawn in the workspace using the graphical user interface of the software.
- 4) Define section property as rectangular assume suitable dimension assigned to the members drawn by using assign to view option.
- 5) Create support condition as fixed and simply supported assign to the beam at required nodes.
- 6) Define load cases in a single load case, as concentrated force of 9kn at 6m distance and uniform force of 5kn/m and assign on beam member.
- 7) Perform analysis command was given, the file was saved and run analysis command was executed.
- 8) It is made sure that there is no error is indicated in the output window.
- 9) Post processing mode was selected.
- 10) The results were properly arranged using the tools.
- 11) Printouts were taken.

Results:

1. Reaction at support A =
2. Reaction at support B =
3. Shear force and bending moment diagrams are attached.

E link: - <https://youtu.be/GkBQUI6waFQ>

(c) Analysis of a Continuous Beam Using STAAD



Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

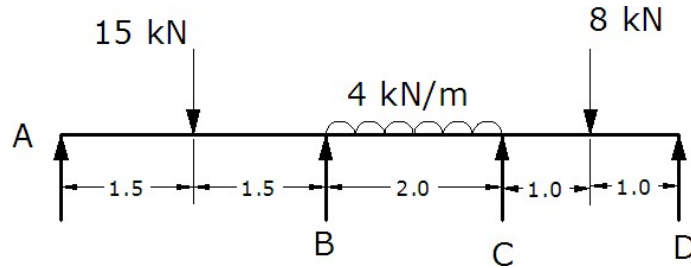
- 1) Open STAAD pro software.
- 2) Start with new project set unit's to m & KN and structure type as Plane.
- 3) The given structure is drawn in the workspace using the graphical user interface of the software.
- 4) Define section property as rectangular assume suitable dimension assigned to the members drawn by using assign to view option.
- 5) Create support condition as fixed and simply supported assign to the beam at required nodes.
- 6) Define load cases in a single load case, as concentrated force of 9kn at 6m distance and uniform force of 5kn/m and assign on beam member.
- 7) Perform analysis command was given, the file was saved and run analysis command was executed.
- 8) It is made sure that there is no error is indicated in the output window.
- 9) Post processing mode was selected.
- 10) The results were properly arranged using the tools.
- 11) Printouts were taken.

Results:

1. Reaction at support A =
2. B=..... ,
3. C=..... ,
4. D=.....
5. Shear force and bending moment diagrams are attached.

E Link: - <https://youtu.be/a5VIPG1C8js>

d) Analysis of a Continuous Beam Using STAAD



Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

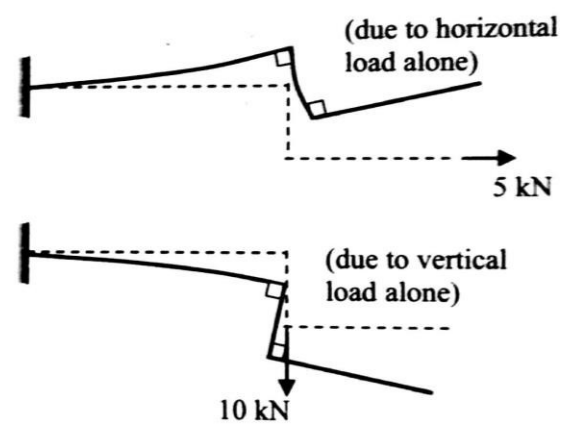
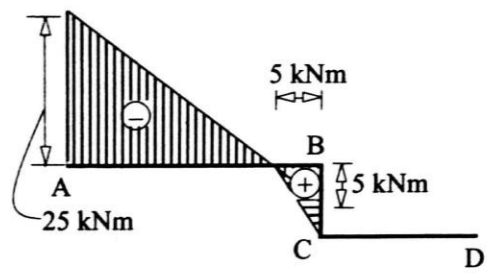
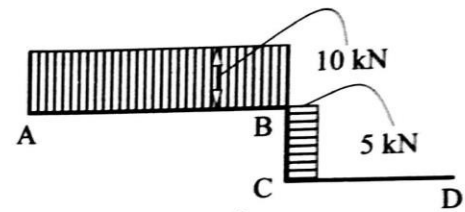
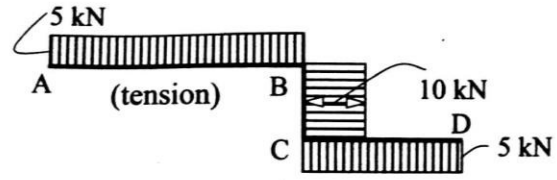
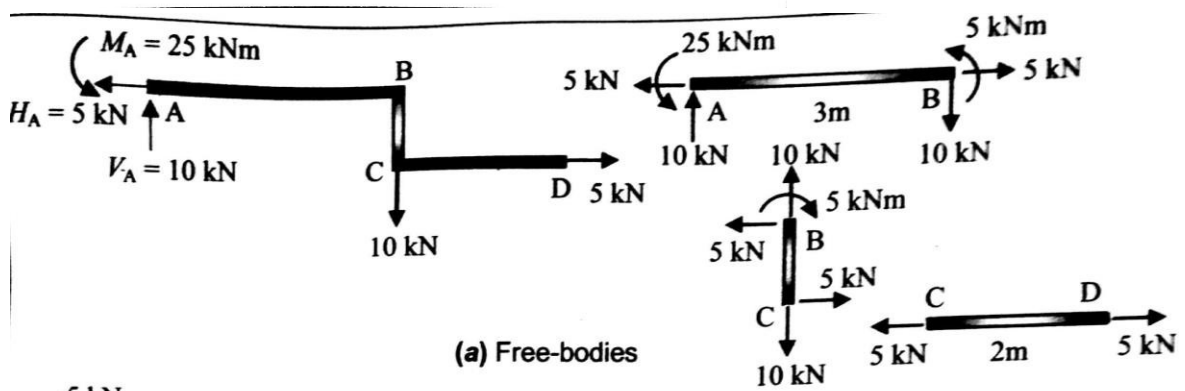
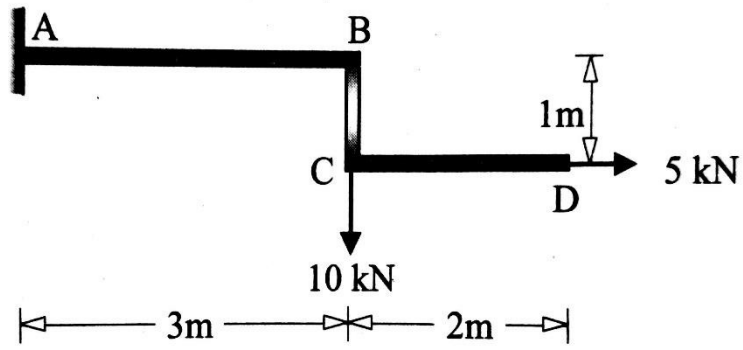
Procedure:

- 1) Open STAAD pro software.
- 2) Start with new project set unit's to m & KN and structure type as Plane.
- 3) The given structure is drawn in the workspace using the graphical user interface of the software.
- 4) Define section property as rectangular assume suitable dimension assigned to the members drawn by using assign to view option.
- 5) Create support condition as fixed and simply supported assign to the beam at required nodes.
- 6) Define load cases in a single load case, as concentrated force of 9kn at 6m distance and uniform force of 5kn/m and assign on beam member.
- 7) Perform analysis command was given, the file was saved and run analysis command was executed.
- 8) It is made sure that there is no error is indicated in the output window.
- 9) Post processing mode was selected.
- 10) The results were properly arranged using the tools.
- 11) Printouts were taken.

Results:

1. Reaction at support A =
2. B=..... ,
3. C=..... ,
4. D=.....
5. Shear force and bending moment diagrams are attached.

E link:-<https://youtu.be/RwXcH-2wE-A>



E link: -<https://youtu.be/w9t7Ht9HcGg>

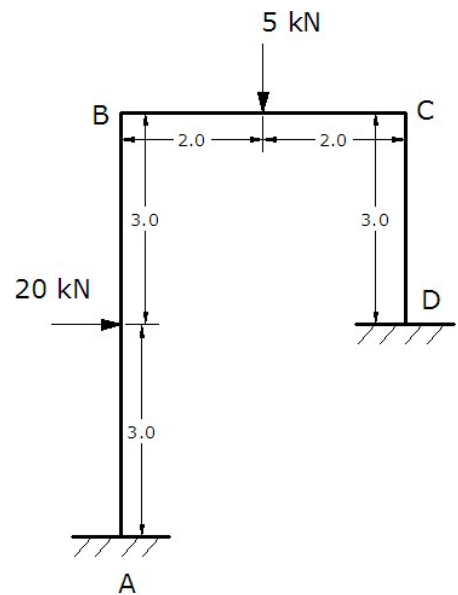
3(a) Analysis of a Single storied 2D Portal Frame Using STAAD

Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

1. Open STAAD Pro software.
2. Start with new project set unit's to m & KN and structure type as Plane.
3. The given structure is drawn in the workspace using the graphical user interface of the software.
4. Define section property as rectangular concrete section with suitable dimensions assumed..
5. Create Supports as fixed and assign to required node.
6. Loading is defined in a single load case and concentrated force of 5kn and 20kn assign to the frame at suitable distance.
7. Perform analysis command was given
8. Post analysis print options were selected suitably.
9. The file was saved and run analysis command was executed.
10. It is made sure that there is no error is indicated in the output window.
11. Post processing mode was selected.
12. The results were properly arranged using the tools.
13. Printouts were taken.



Results:

1. Reaction at support A =
2. Reaction at support D =
3. Shear force and bending moment diagrams are attached.

E link: - <https://youtu.be/FoFmtz0PwE4>

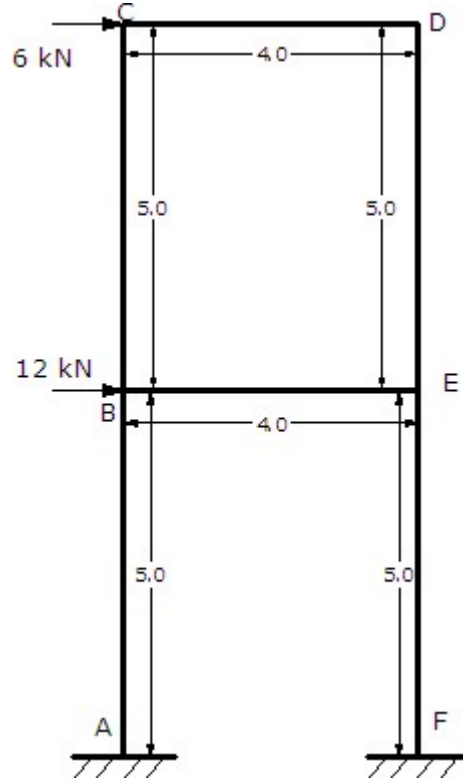
(b) Analysis of a Double storied 2D Portal Frame Using STAAD

Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

1. Open STAAD Pro software.
2. Start with new project set unit's to m & KN and structure type as Plane.
3. The given structure is drawn in the workspace using the graphical user interface of the software.
4. Define section property as rectangular concrete section with suitable dimensions assumed.
5. Create Supports as fixed and assign to required node.
6. Loading is defined in a single load case and concentrated force of 6kn and 12kn assign to the frame at suitable distance.
7. Perform analysis command was given
8. Post analysis print options were selected suitably.
9. The file was saved and run analysis command was executed.
10. It is made sure that there is no error is indicated in the output window.
11. Post processing mode was selected.
12. The results were properly arranged using the tools.
13. Printouts were taken.

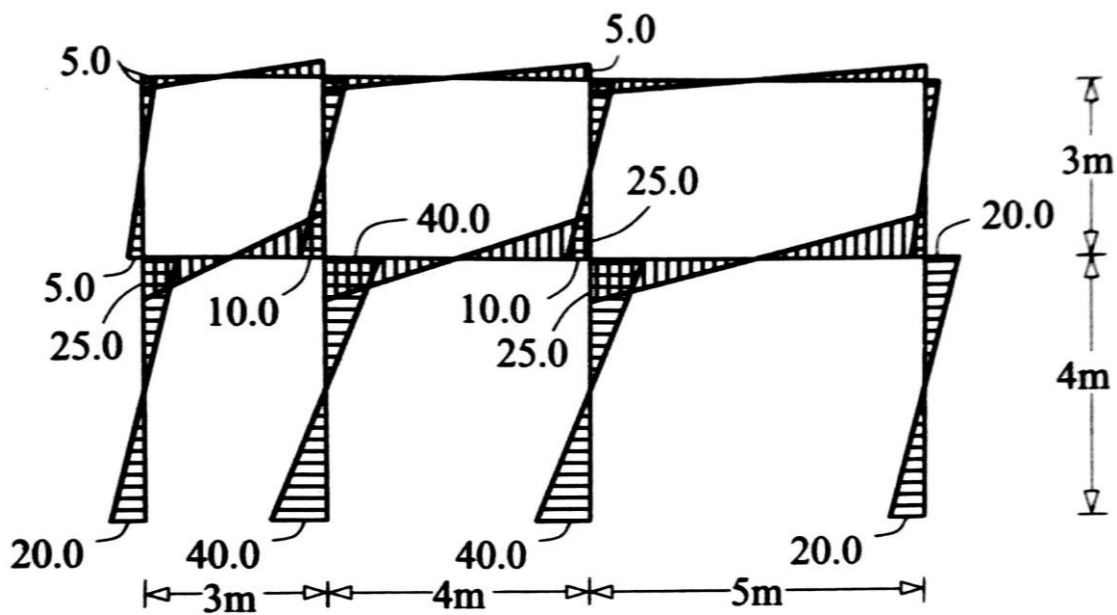
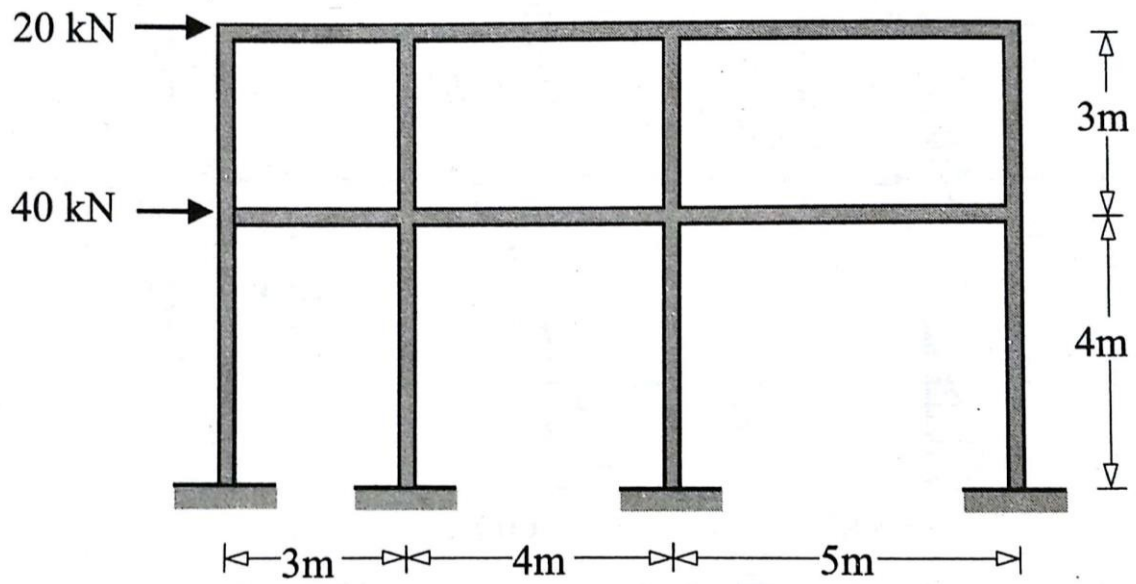


Results:

1. Reaction at support A =
2. Reaction at support F =
3. Shear force and bending moment diagrams are attached.

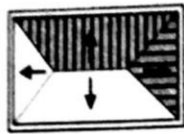
E link: -<https://youtu.be/6Yo8tmlC0mc>

C. Analysis of a Double storied 2D Portal Frame Using STAAD

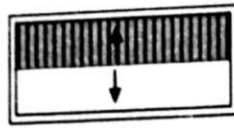


E link:-<https://youtu.be/0apI52PVi3Q>

Load patterns and structural details in 3D framed structures.

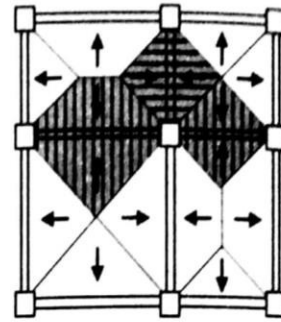


TWO-WAY SLAB

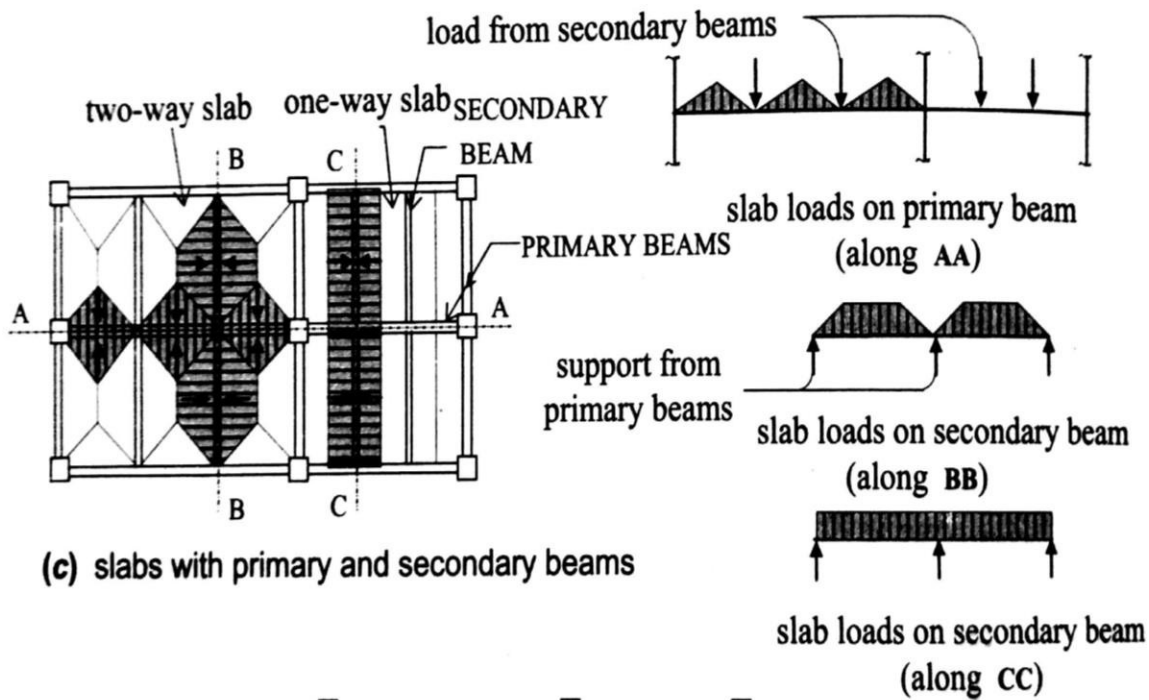


ONE-WAY SLAB

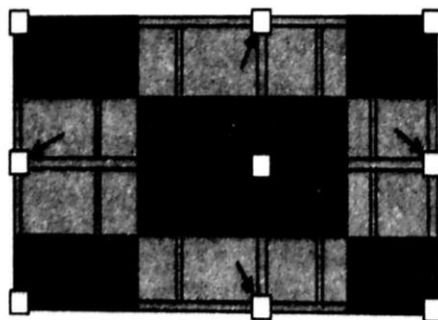
(a) Slabs



(b) Two-way slabs with primary beams



(c) slabs with primary and secondary beams



(d) column tributary areas

4. Analysis of a 3D Portal Frame Using STAAD.

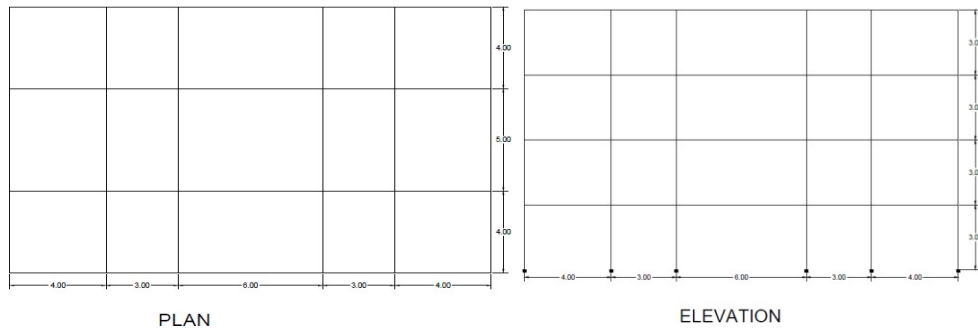
a. Find bending moment shear force and support reactions for a given plan.

Consider beam size as 230X300mm, column size as 230X350.

Apply dead load on slab =3.125Kn/m².

Live load on slab =4Kn/ m².

Main wall load = 10Kn/ m², Support condition is fixed.



Aim: To find the reactions at the supports and draw the bending moment and shear force diagram using STAAD.

Tools Required: STAAD Pro Software, Microsoft Word Software, Paint Software, Printer

Procedure:

- 1) Open STAAD pro software.
- 2) Start with new project set unit's to m & KN and structure type as Plane.
- 3) Select structural wizard to develop a plan according to given problem. The given structure is drawn in the workspace using the graphical user interface of the software.
- 4) Define section property as rectangular according to given dimension assigned to the members drawn by using assign to view option / assign to selected beams .
- 5) Create support condition as fixed and assign to nodes at base of model.
- 6) Define load cases and assign to the members.
- 7) Perform analysis command was given, the file was saved and run analysis command was executed.
- 8) It is made sure that there is no error is indicated in the output window.
- 9) Post processing mode was selected.
- 12) The results were properly arranged using the tools, Printouts were taken.

Results:

- 1) Reaction at support
- 2) Bending moments
- 3) Shear forces
- 4) Report

E link: - <https://youtu.be/omrr9osx4dM>

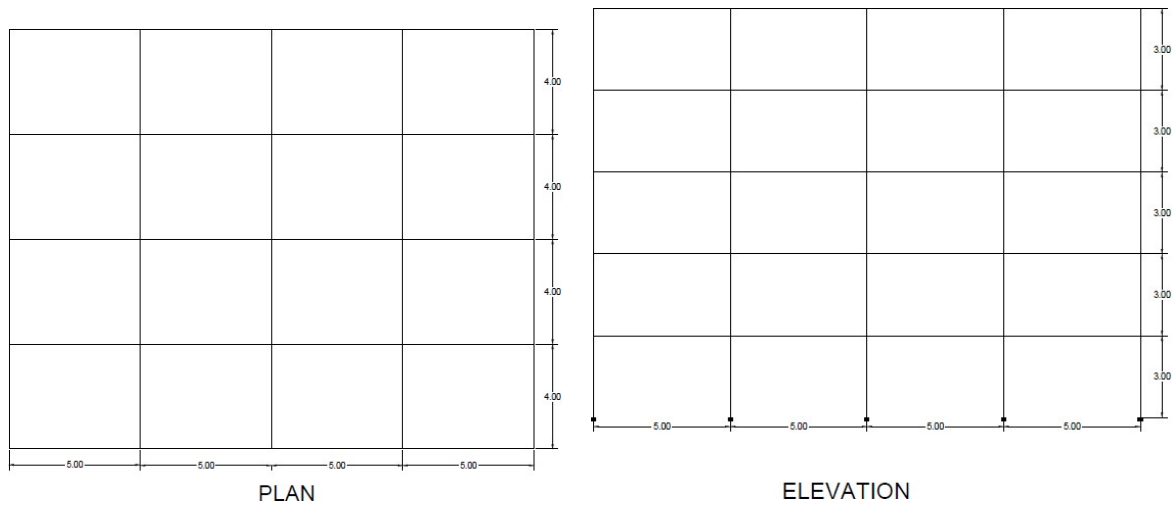
B. Find bending moment shear force and support reactions for a given plan.

Consider beam size as 230X300mm, column size as 230X350.

Apply dead load on slab =3.125Kn/m².

Live load on slab =4Kn/ m².

Main wall load = 10Kn/ m², Support condition is fixed.



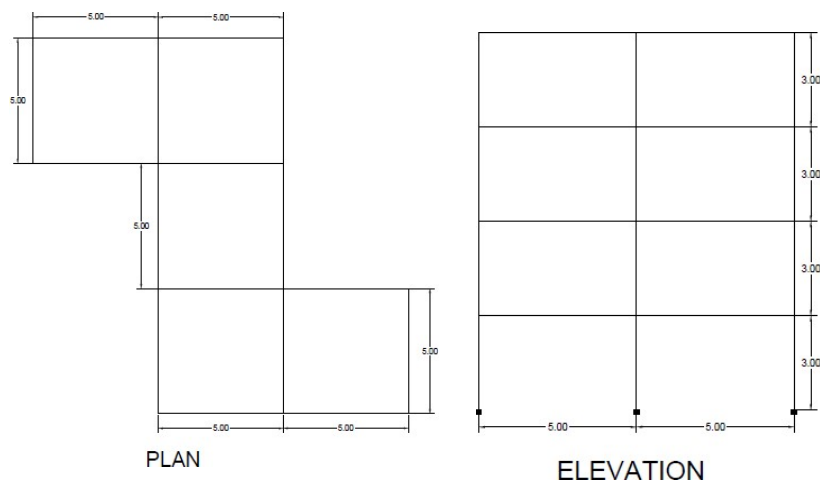
C. Find bending moment shear force and support reactions for a given plan.

Consider beam size as 230X300mm, column size as 230X350.

Apply dead load on slab =3.125Kn/m².

Live load on slab =4Kn/ m².

Main wall load = 10Kn/ m², Support condition is fixed.



E Link:-<https://youtu.be/g4OO8yH20bc>