

# STAADPRO

DR. MOHD HILTON BIN AHMAD

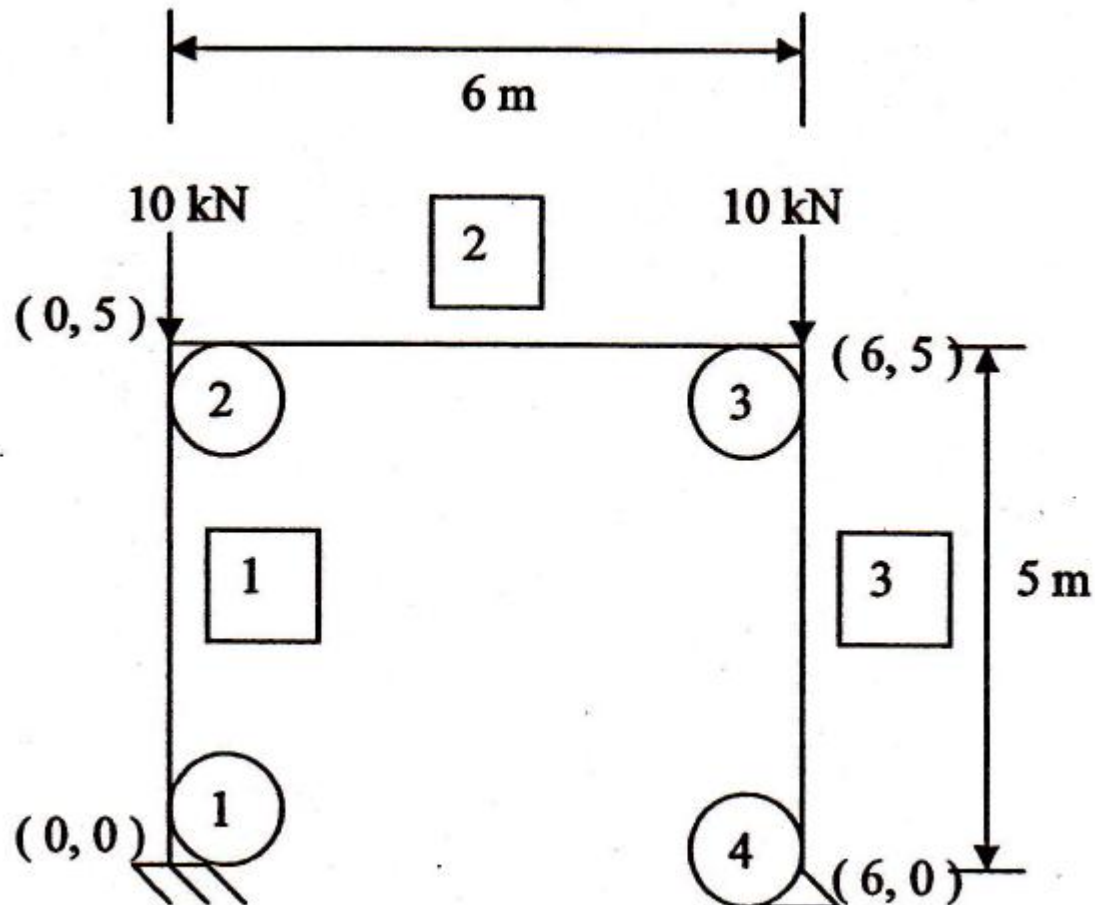
# INTRODUCTION

- STAAD.Pro stands for Structural Analysis and Design for Professional.
- STAAD.Pro comprehensively addresses all aspects of structural engineering – model development, analysis, design, visualization, verification *etc.*
- STAAD.Pro capable to guide user through the process of:
  - ▣ Creating an input file whether graphically or text editor.
  - ▣ Running Staad.Pro to perform analysis and design
  - ▣ Visualization and verification of the model graphically and numerically
  - ▣ Printing the output result as desired.

# STEPS IN STAAD.Pro

- 5 main steps to use this program
  - Modelling
    - To obtain and study the given structure
    - Decide the coordinates and node number
  - Assigning the load
  - Properties
  - Analysis and design
  - Result
- To design using this program there are two methods:
  - Graphic Method
  - Text Method

# GRAPHIC METHOD

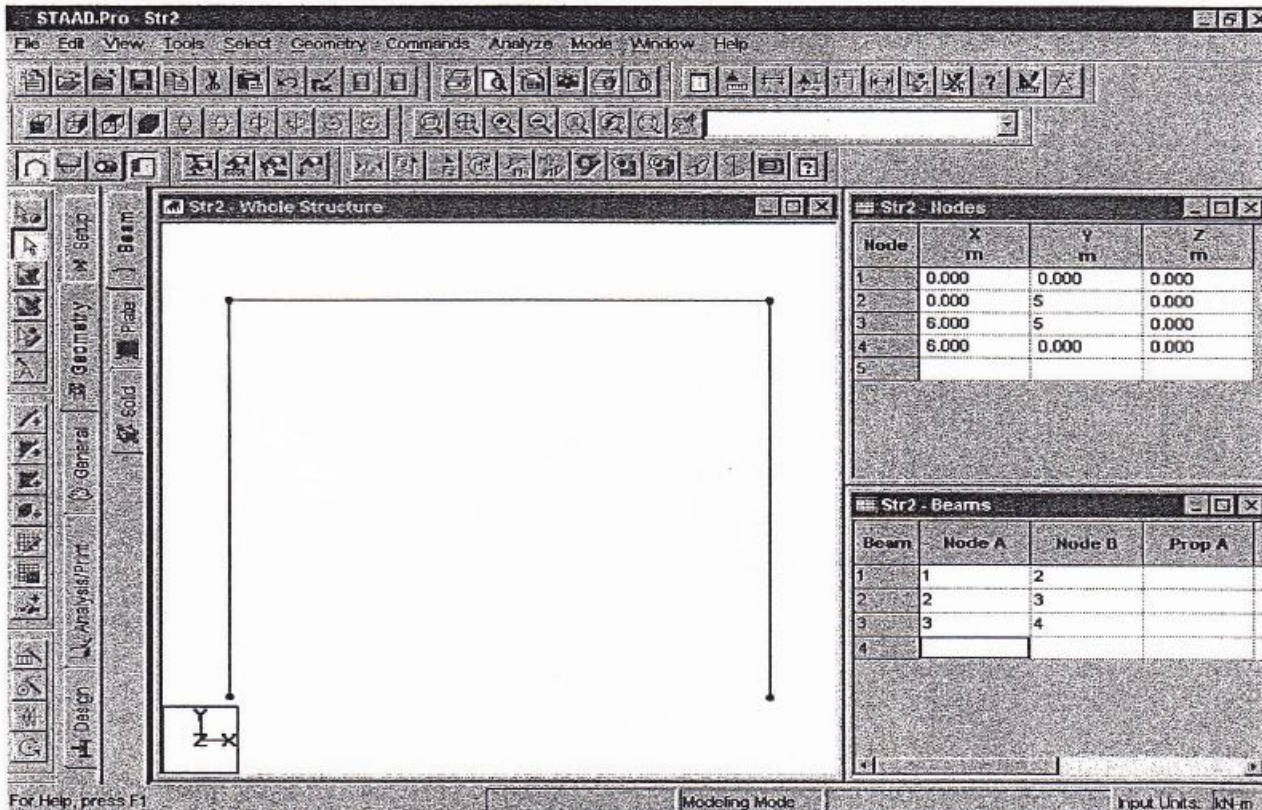


# GRAPHIC METHOD

---

- Steps:
  - ▣ Determine axis coordinates of origin
  - ▣ Set the coordinates
  - ▣ Give the node numbers
  - ▣ Give the members numbers

# GRAPHIC METHOD



The screenshot displays the STAAD.Pro software interface. The main window shows a 2D structure model with a coordinate system (X, Y, Z) at the bottom left. The structure consists of a vertical line on the left, a horizontal line at the top, and a vertical line on the right. The right side of the interface contains two data tables: "Str2 - Nodes" and "Str2 - Beams".

Node	X m	Y m	Z m
1	0.000	0.000	0.000
2	0.000	5	0.000
3	6.000	5	0.000
4	6.000	0.000	0.000
5			

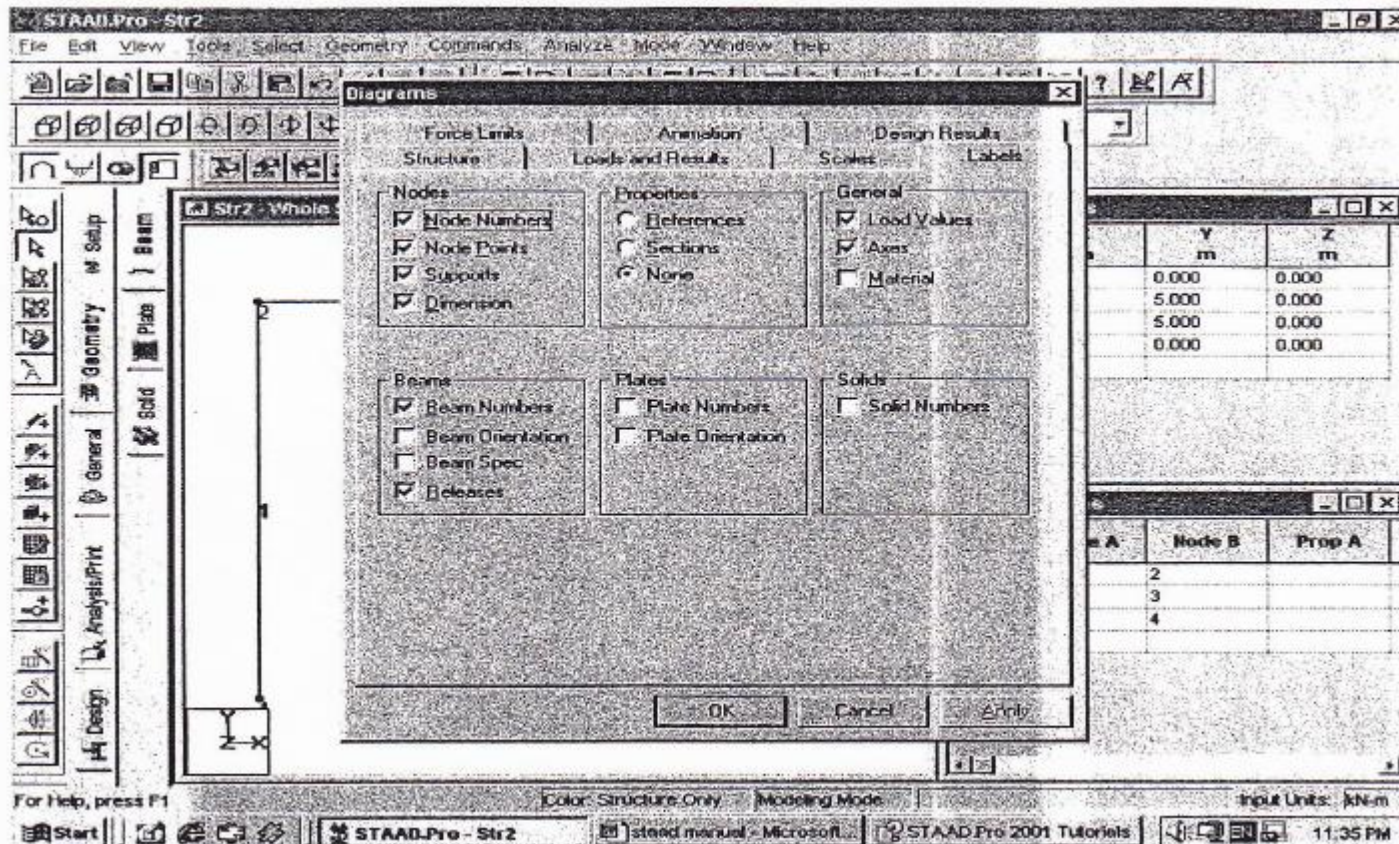
  

Beam	Node A	Node B	Prop A
1	1	2	
2	2	3	
3	3	4	
4			

At the bottom of the interface, the status bar indicates "Modeling Mode" and "Input Units: kN-m".

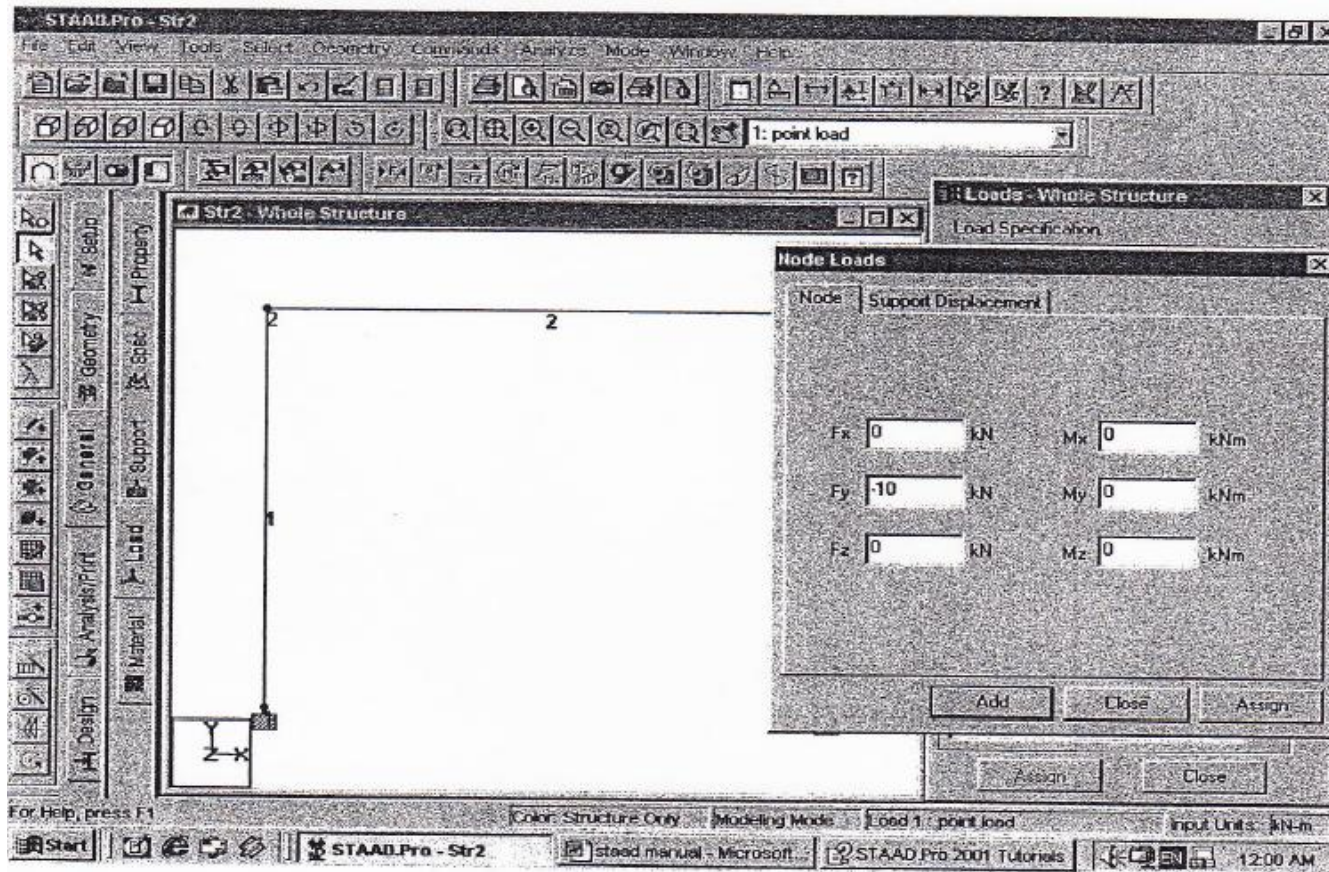
1. Click **GEOMETRY** as shown
2. Fill the nodes and beams in the table given

# GRAPHIC METHOD



Click icon **SYMBOLS AND LABELS** on the top of the screen. Tick the node numbers, node points, supports, beam, etc. Then click **OK**.

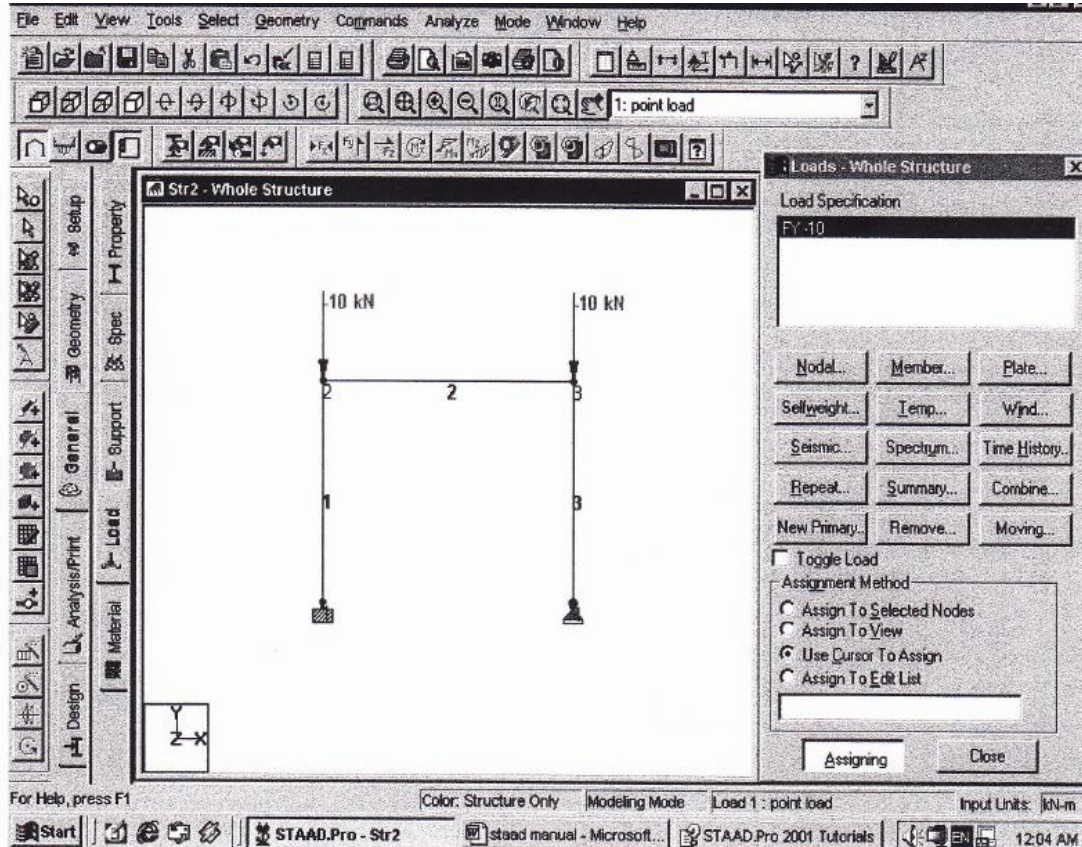
# GRAPHIC METHOD



To insert the load, click to the **LOAD** icon. Choose **NODAL** then insert the load value. Then click **ADD**.

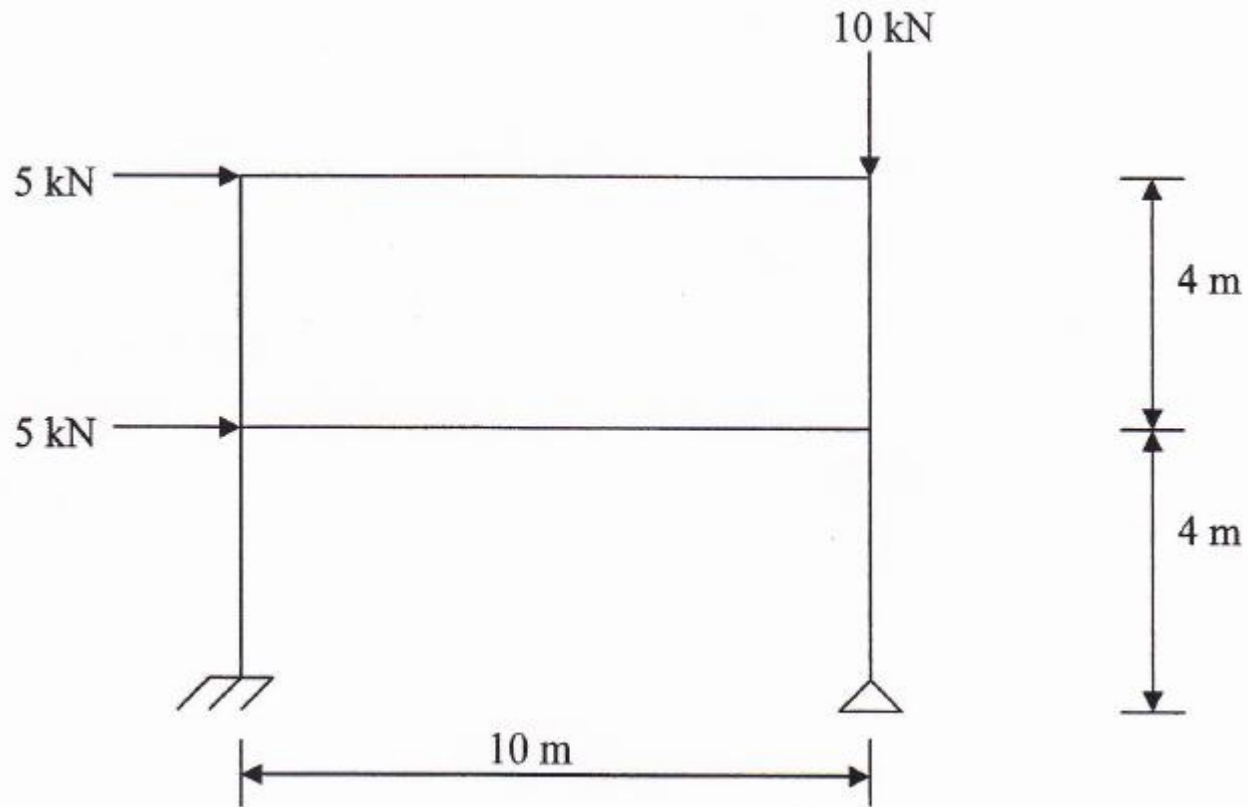


# GRAPHIC METHOD

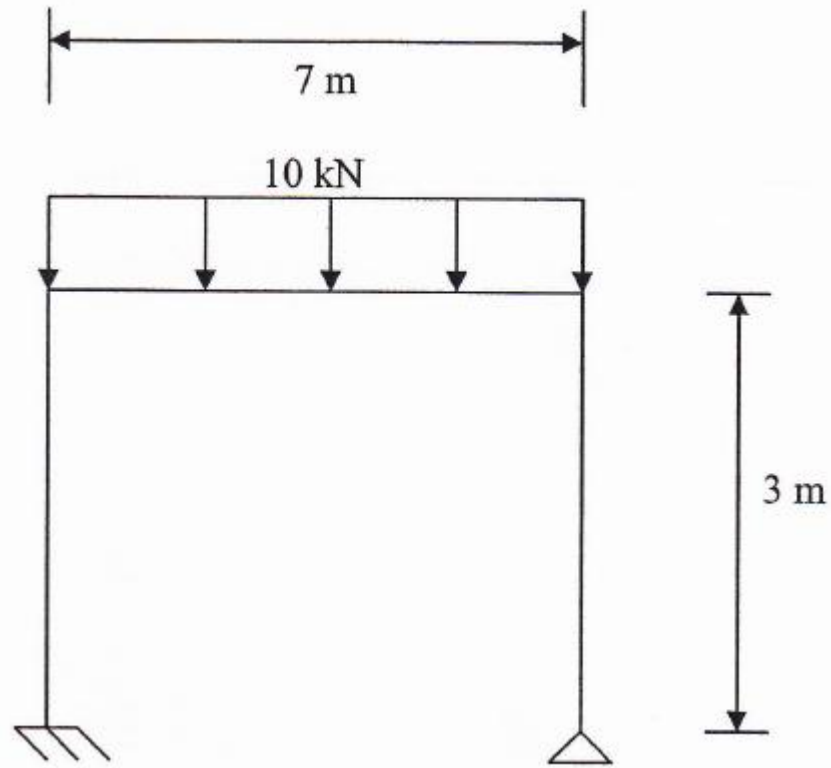


To fill the load onto the diagram, click **USE CURSOR TO ASSIGN**. Then click **ASSIGNING** and drag your mouse to the node point and press the left button of your mouse.

# EXERCISE 1



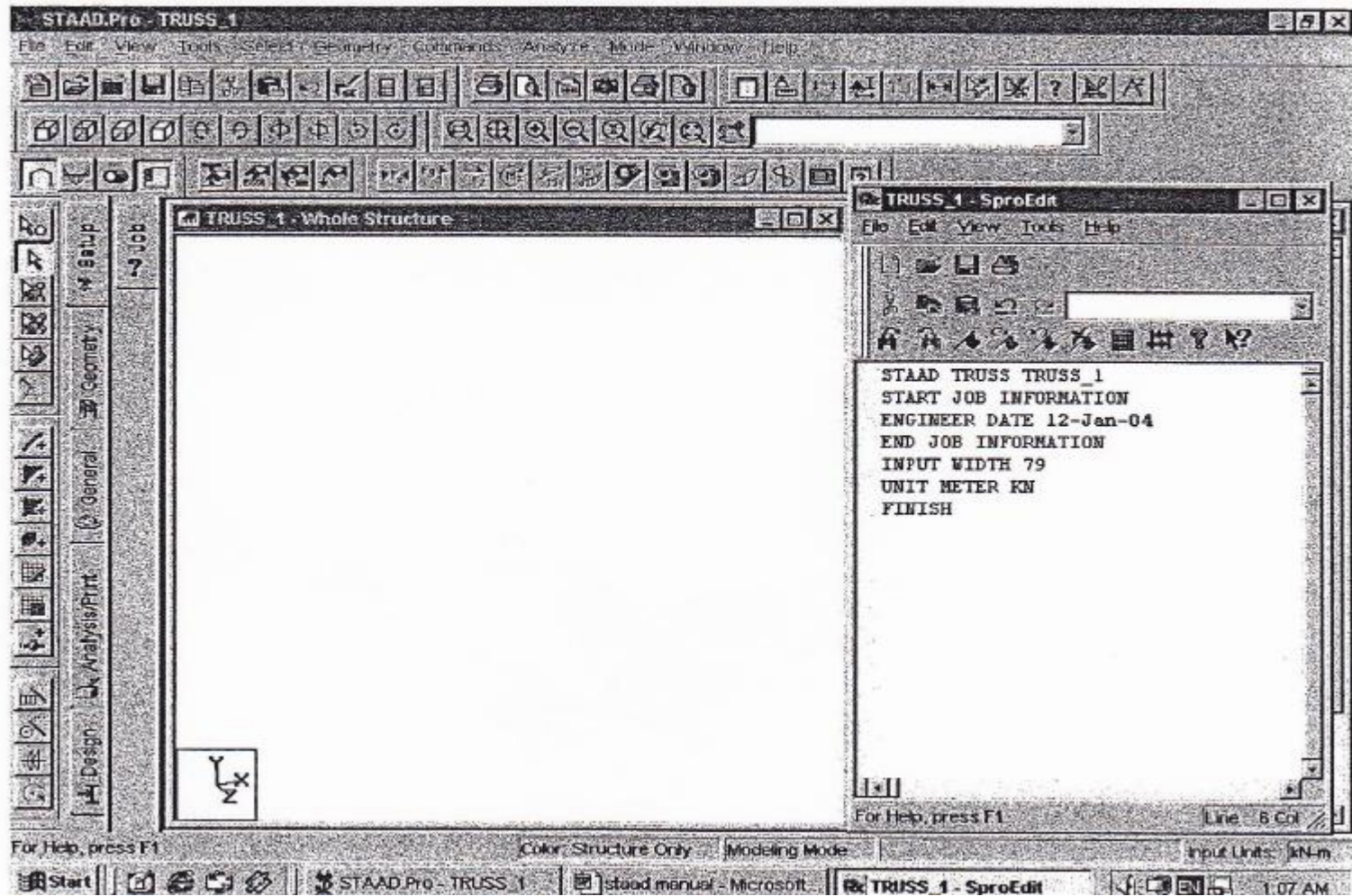
# EXERCISE 2



# TEXT METHOD

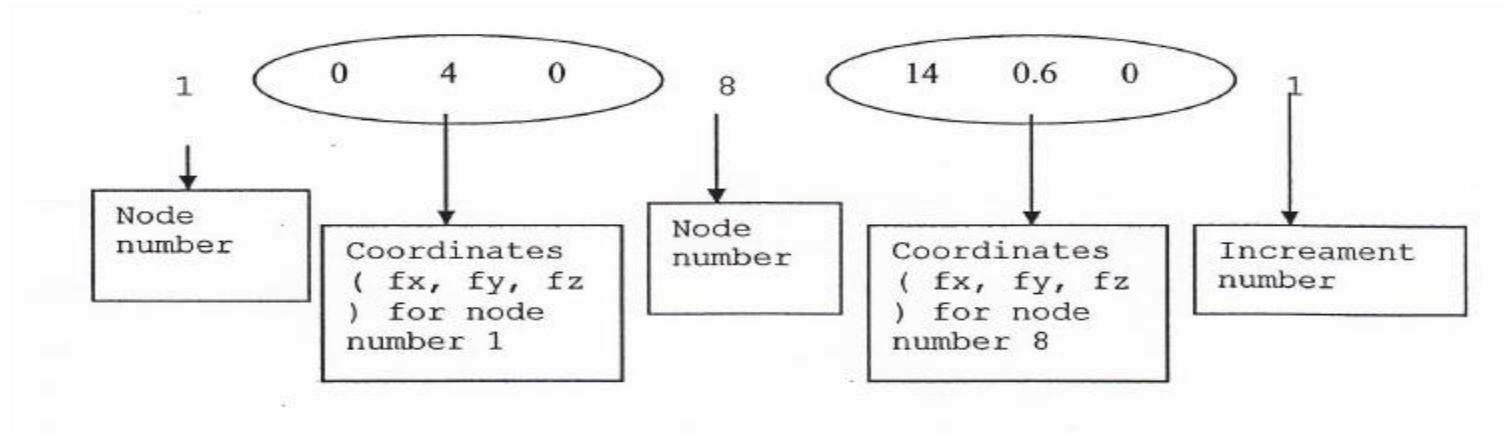
- For text method, we will use the STAAD Editor instead of icon command. STAAD Editor is more likely like AutoCad format. In this method we will see the 2D structure such as truss, plane and frame.
- At the end of this lecture you will find using STAAD Editor is much easier than graphic method.

# TRUSS

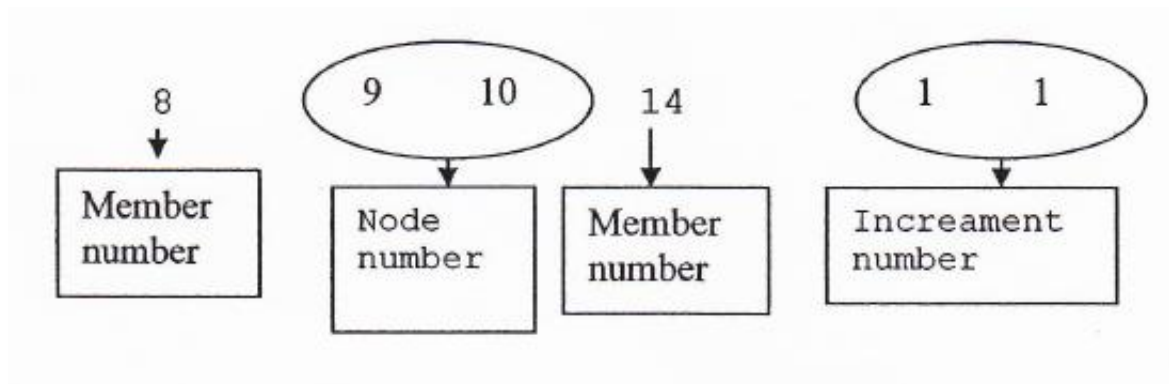


# DESCRIPTION

## □ Joint Coordinates



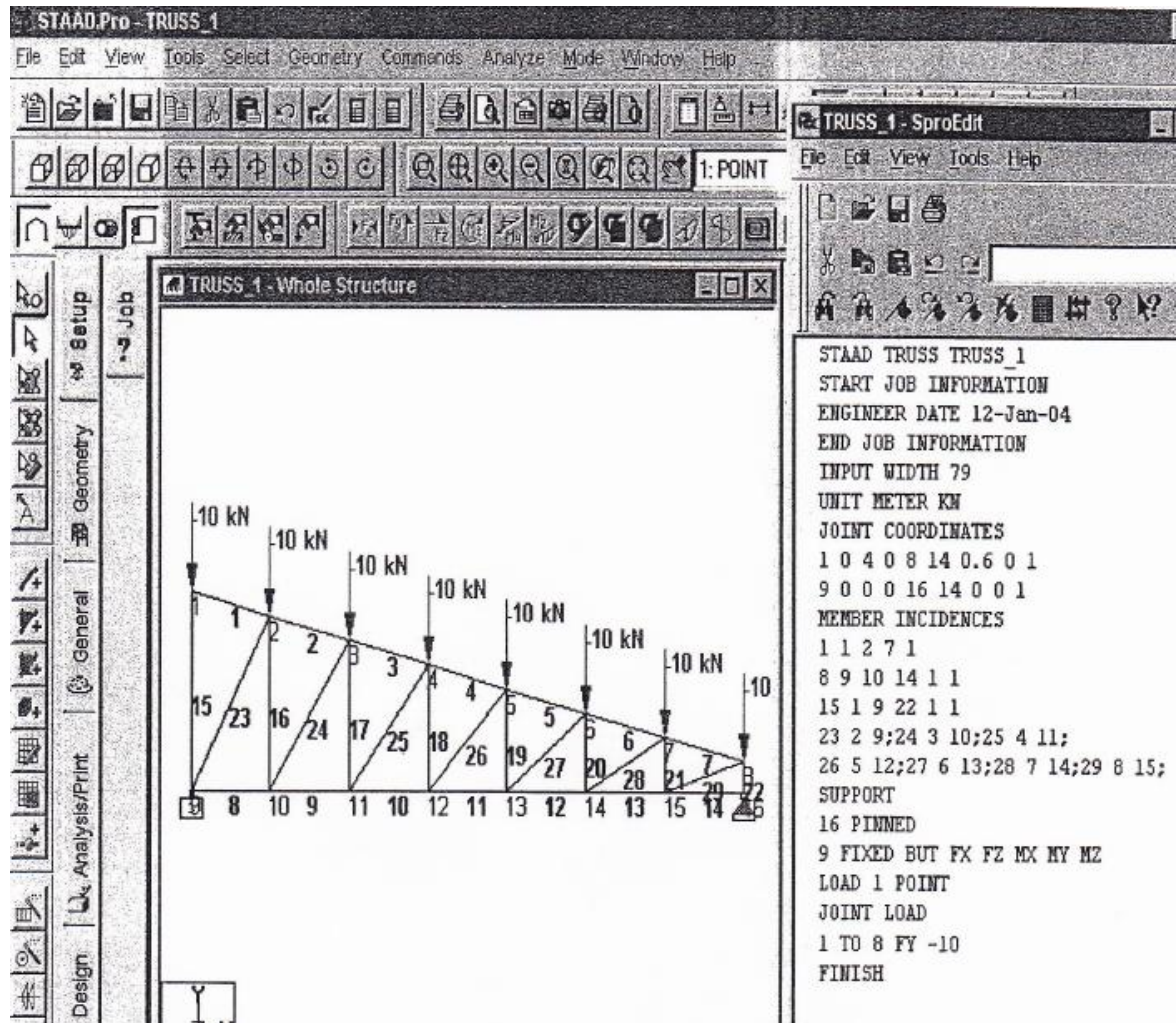
## □ Member Incidences



# DESCRIPTION

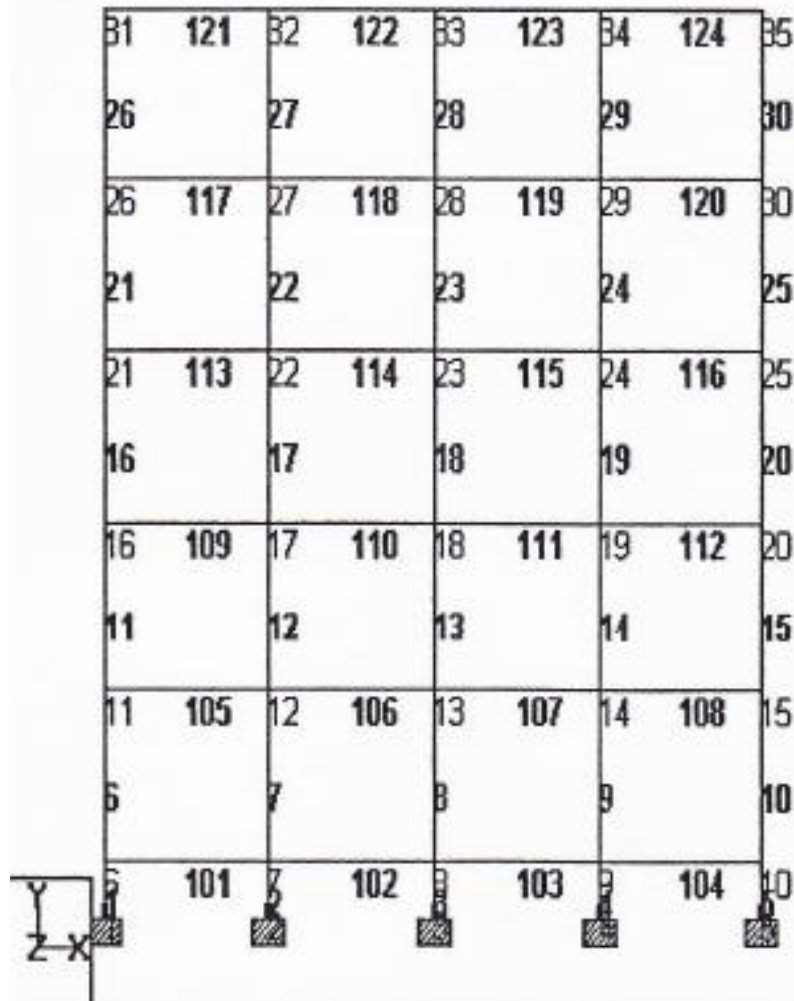
- Support
  - ▣ 16 pinned → shows that the support is located at node number 16
  
- Load
  - ▣ Load 1 point → shows that there is point load node number 1
  - ▣ Joint Load 1 to 8 FY-10 → shows that there are point loads from node number 1 until node number 8.

# TRUSS





# FRAME



# FRAME

The screenshot displays the STAAD.Pro interface for a frame structure. The main window shows a grid of joints and their corresponding member connections. The joints are arranged in a 5x5 grid, with columns numbered 1-5 and rows numbered 1-5. The members are numbered 1-12, connecting adjacent joints in both horizontal and vertical directions. The input data window on the right provides details for the structure, including job information, unit settings, joint coordinates, member incidences for columns and beams, and support conditions.

**FRAME\_1 - Whole Structure**

31	121	32	122	33	123	34	124	35
26		27		28		29		30
26	117	27	118	28	119	29	120	30
21		22		23		24		25
21	113	22	114	23	115	24	116	25
16		17		18		19		20
16	109	17	110	18	111	19	112	20
11		12		13		14		15
11	105	12	106	13	107	14	108	15
6		7		8		9		10
	101		102		103		104	10

**STAAD SPACE FRAME\_1**  
**START JOB INFORMATION**  
ENGINEER DATE 12-Jan-04  
**END JOB INFORMATION**  
INPUT WIDTH 79  
UNIT METER KN  
JOINT COORDINATES  
1 0 0 0 5 24 0 0 1  
R 1 0 1.2 0  
R 5 0 5 0  
**MEMBER INCIDENCES**  
\*COLUMN  
1 1 6 5 1 1  
R 5 5 5  
\*BEAM  
101 6 7 104 1 1  
R 5 4 5  
**SUPPORT**  
1 TO 5 FIXED  
**FINISH**

For Help, press F1      Line: 1 Col: 1

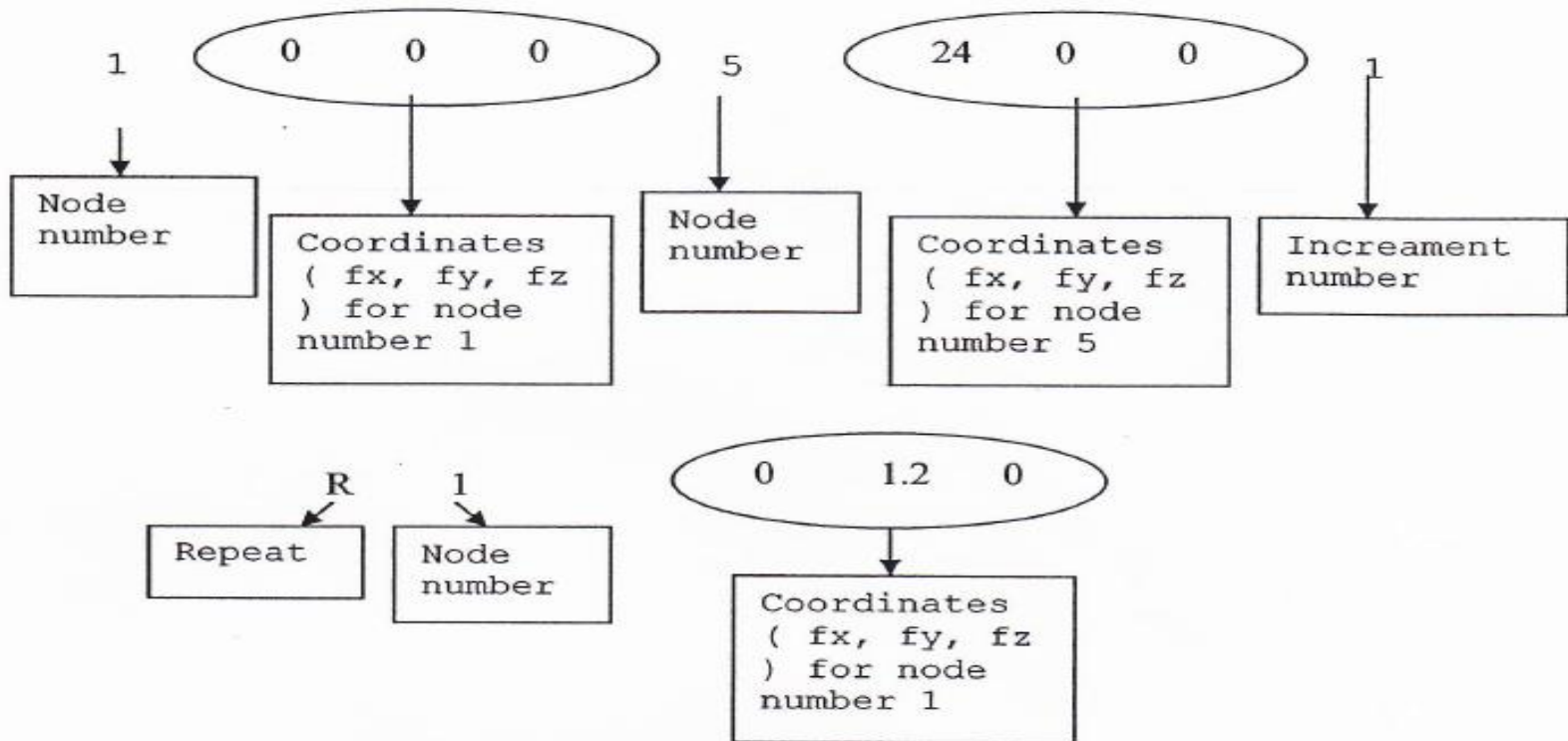
Color: Structure Only    Modeling Mode    Input Units: kN-m

Start    STAAD.Pro - FR    STAAD MANUA...    3 1/2 Floppy (A:)    FRAME\_1 - Spr...    254 AM

# FRAME

## DESCRIPTION:

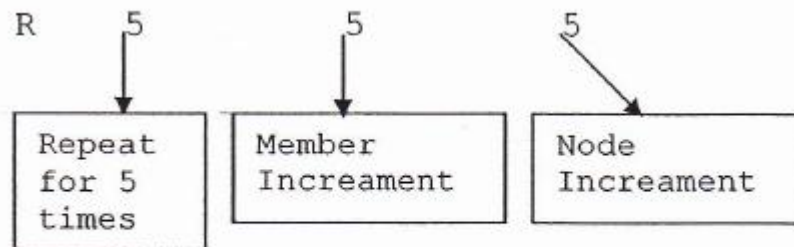
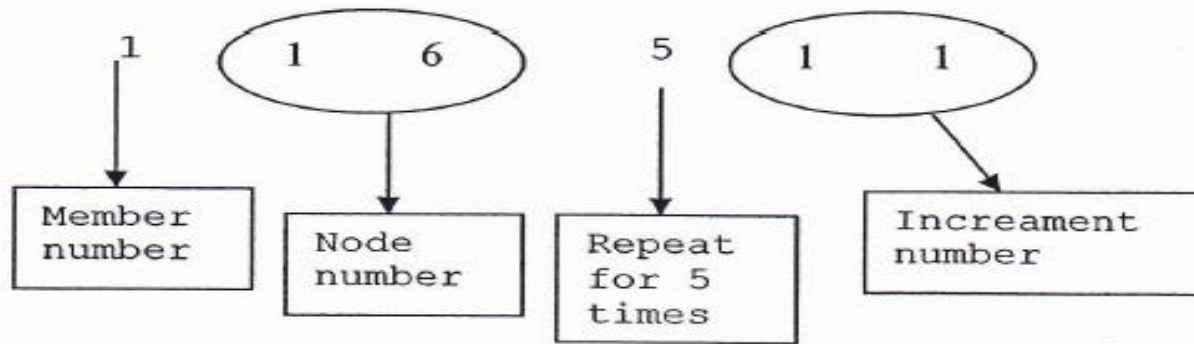
### I. Joint Coordinates



# FRAME

## II. Member Incidences

\* COLUMN

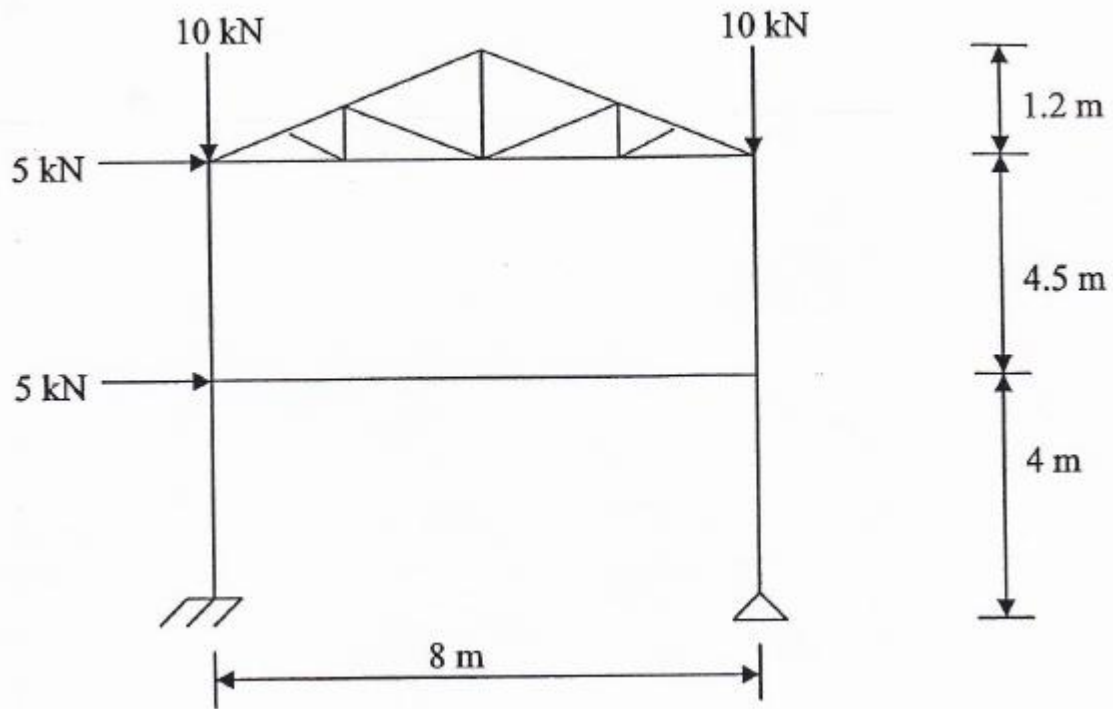


# FRAME

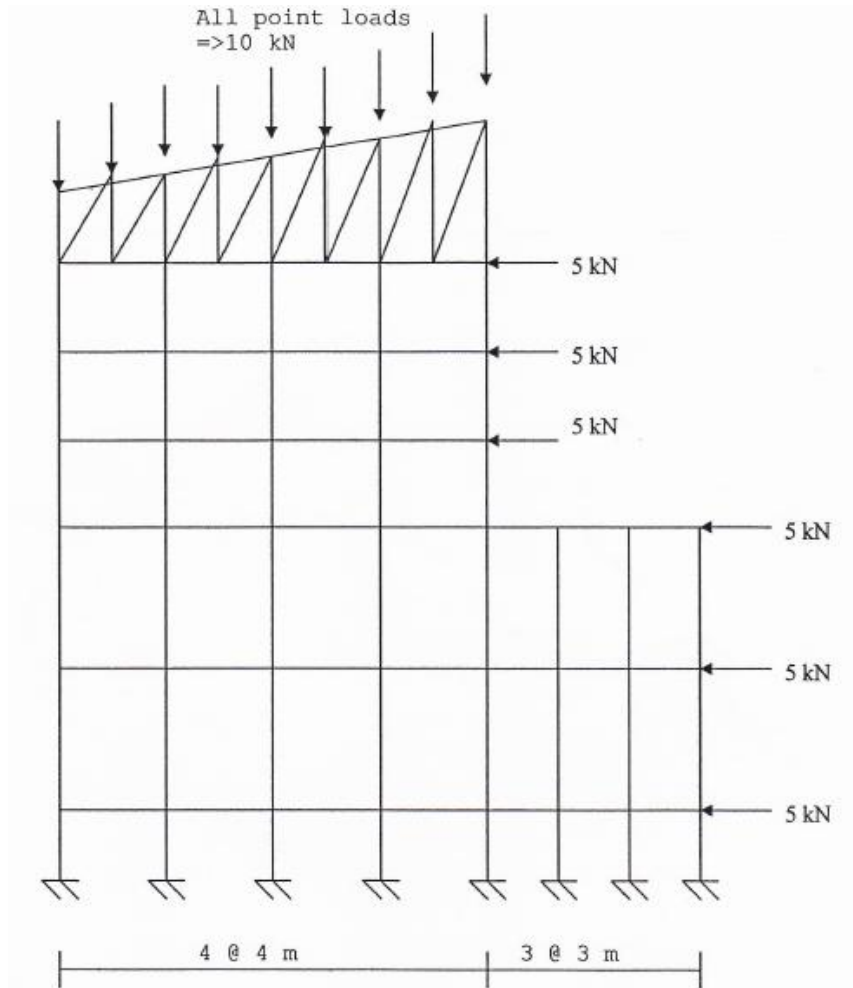
### III. **Support**

1 to 5 fixed => shows that there are fixed support from node number 1 until node number 5.

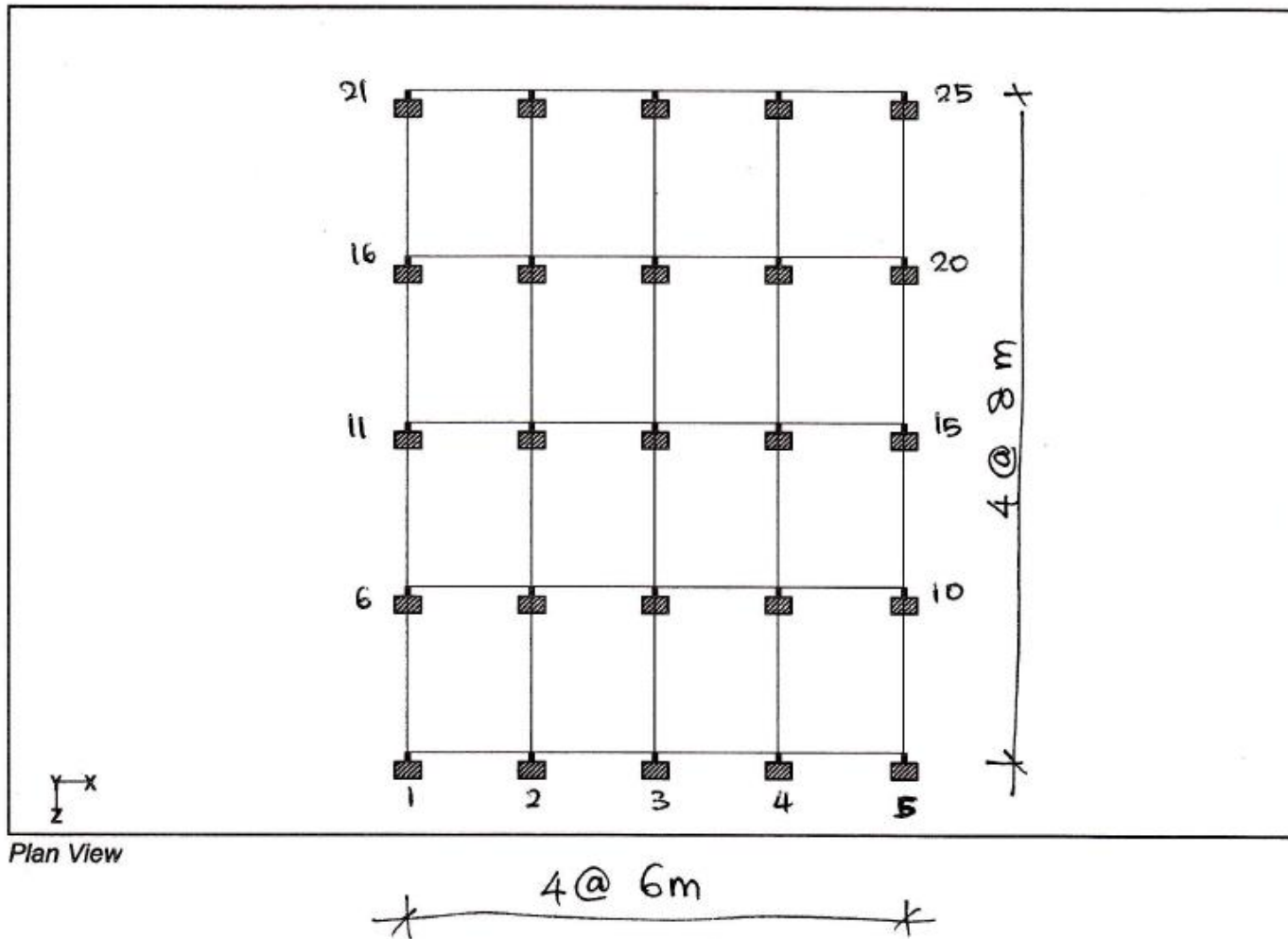
# EXERCISE 1



# EXERCISE 2

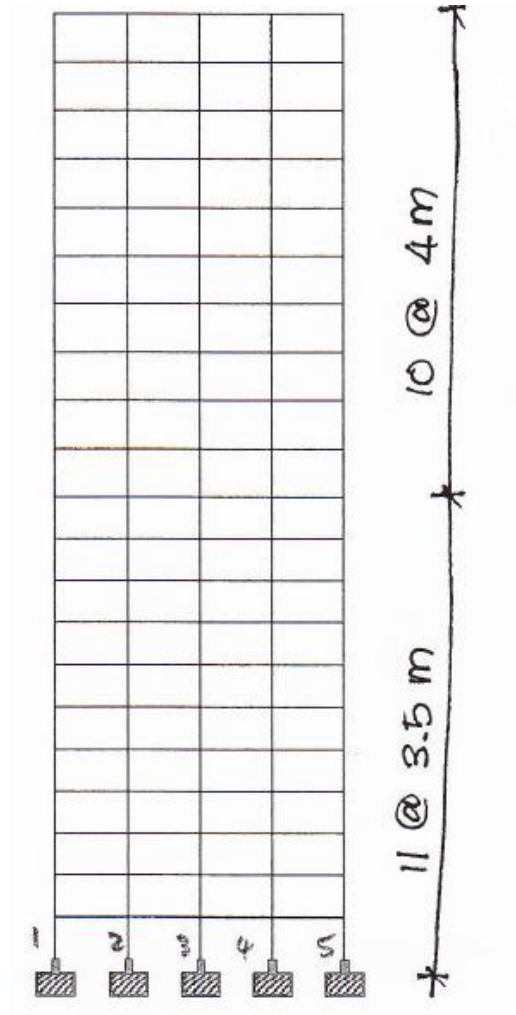


# EXAMPLE 1- 3D SPACE





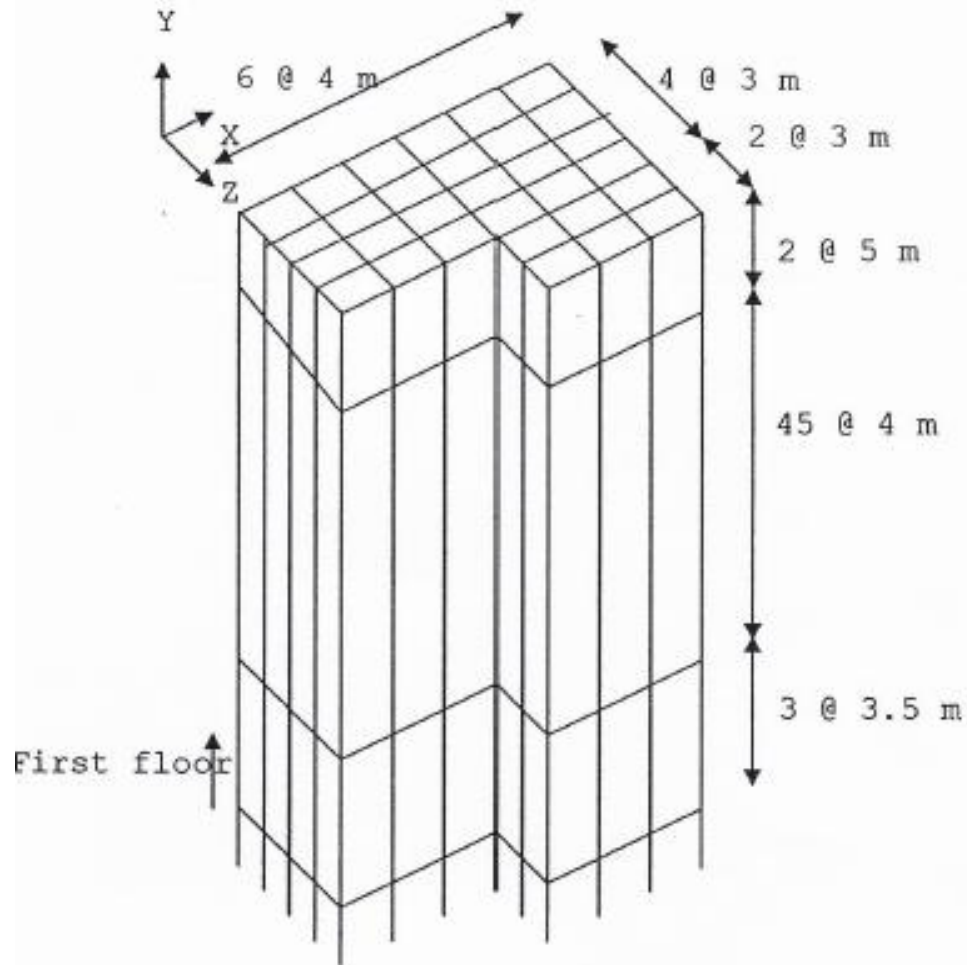
# EXAMPLE 1- 3D SPACE



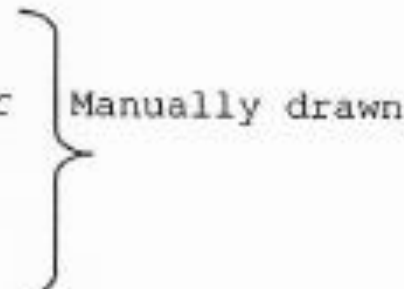
# EXAMPLE 1- 3D SPACE

```
STAAD PLANE TUGASAN 4.1 - 3D MULTISTOREY BUILDING
START JOB INFORMATION
ENGINEER NAME Chen Yee
CHECKER NAME David Yeoh
APPROVED NAME David Yeoh
ENGINEER DATE 24-Jul-01
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER KN
JOINT COORDINATES
1 0 0 5 24 0 0 1
R 4 0 0 -8
R A 11 0 3.5 0
R 10 0 4 0
MEM INC
* COLUMN
1 1 26 25 1 1
R 20 25 25
R A 0 0 0
* BEAM LEVEL 1 (X-DIRECTION)
1001 26 27 1004 1 1
R 4 4 5
* BEAM LEVEL 1 (Z-DIRECTION)
1021 26 31 1024 1 5
R 4 4 1
R A 20 40 25
SUPPORT
1 to 25 FIXED
FINISH
```

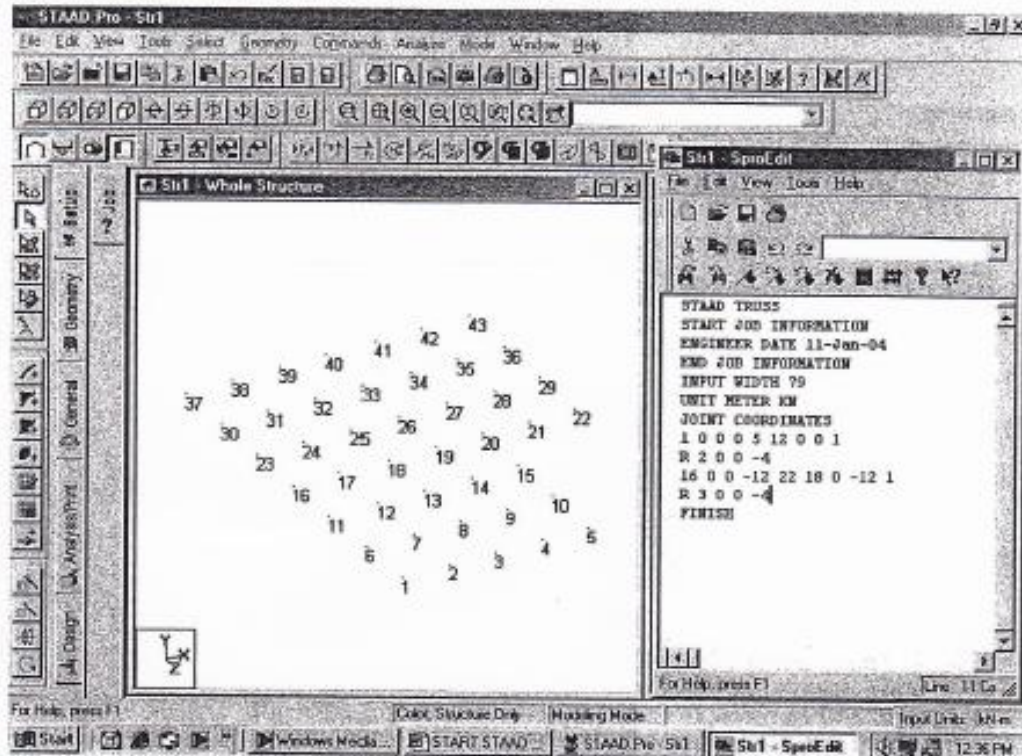
# 3D Space



# 3D Space

- 1 Ground floor plan node numbers
  - 2 Ground floor to first floor column number
  - 3 First floor plan member numbers
  - 4 Roof plan member numbers
  - 5 Print out ground floor plan with node numbers
  - 6 Print out STAAD editor before expending
- \* All supports are fixed
- 

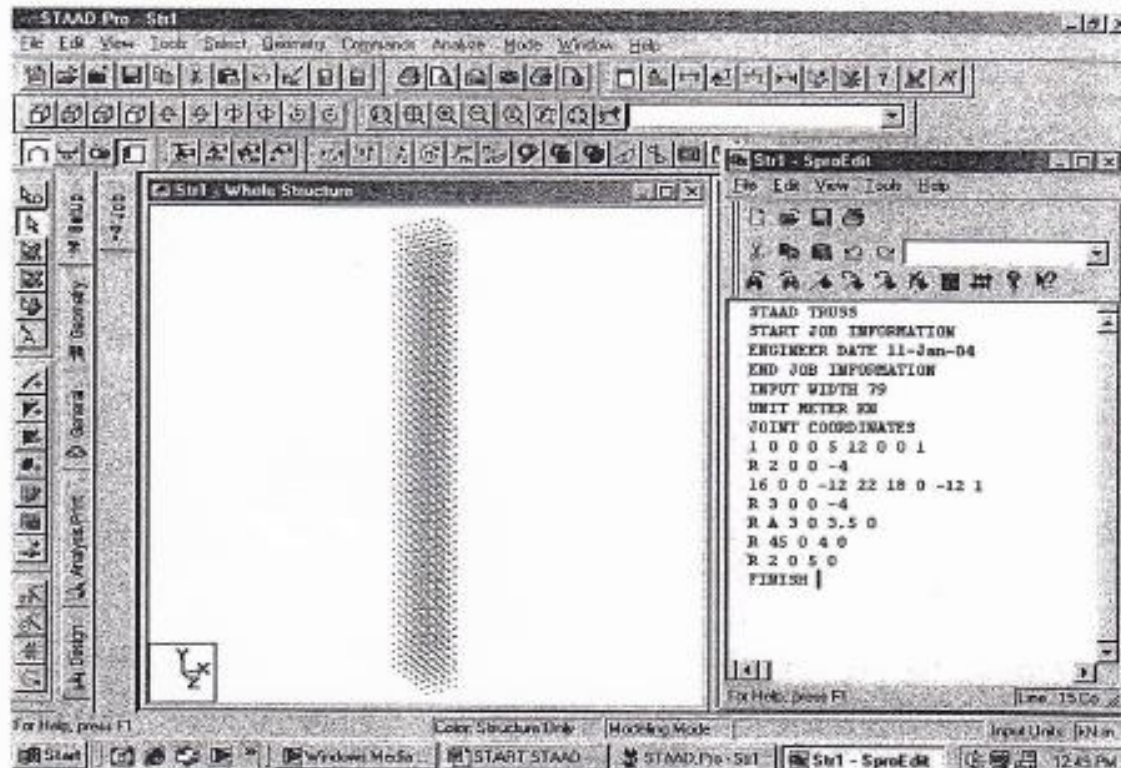
# 3D Space



\* This is ground floor plan with node numbers. Print out this. Go to 'Edit' then 'Take Picture', then go to print preview, current view and then print out.

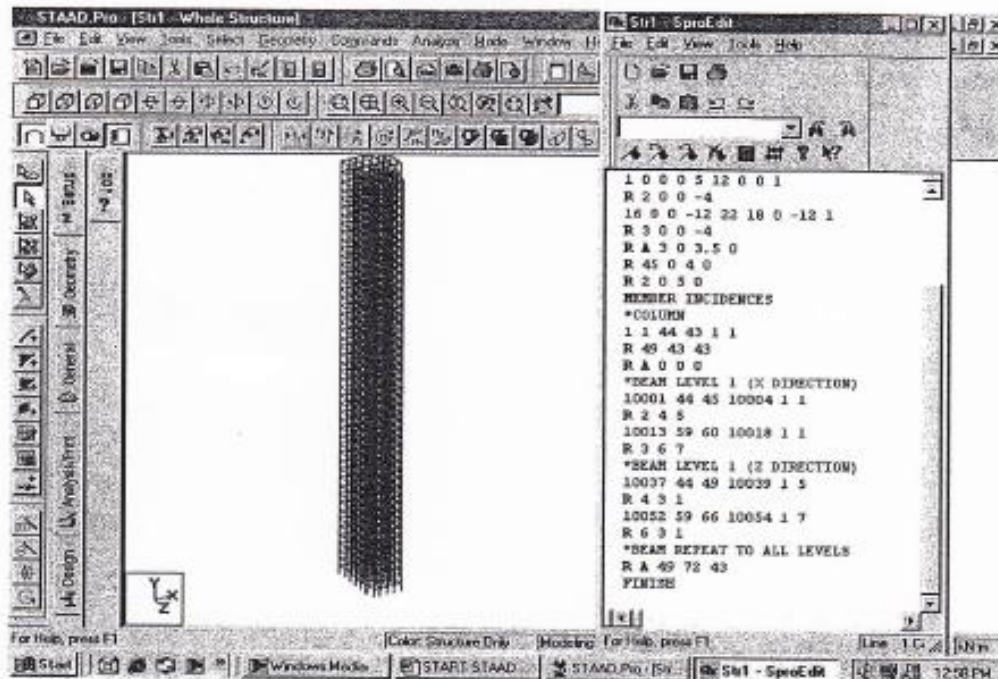
# 3D Space

- 1.8 Type 'JOINT COORDINATES', then save it and close. See the result appear as shown below.



# 3D Space

- 1.9 Then, type 'MEMBER INCIDENCES' including column, beam level 1 (x direction), beam level 1 (z direction) and beam repeat to all levels. Save it and close. See the results appear as shown below.



# 3D Space

1.10 Lastly, type 'SUPPORT'. Save it and close. See the result as shown as below.

