AutoCAD

Bentley®
STAAD.Pro
CONNECT Edition
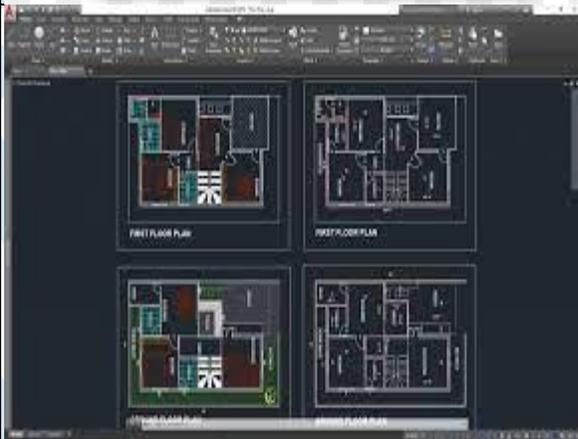
Structural Analysis (Staad.Pro Tool)

BAHAL • HARYANA

- **Prepared by: Mr. ANAND PRAJAPATI**

- **Approved by: Mr. RISHABH SHARMA**
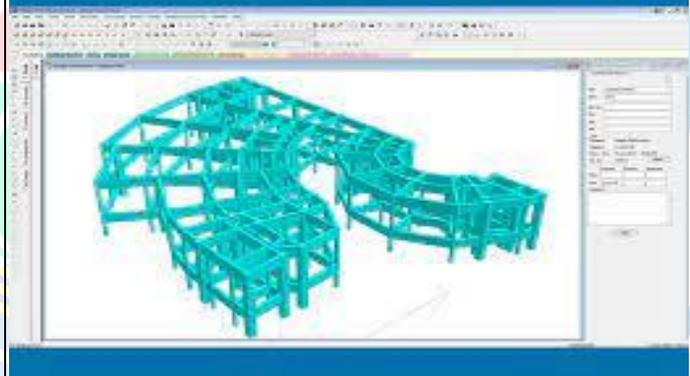
- **Civil Engineering Department**

- **BRCM College of Engg & Technology**

- **Bahal-127 028, Bhiwani**

- **Haryana**

AUTOCAD LAB MANUAL

LIST OF EXPERIMENTS

- **To study the commands used in plans of buildings using AutoCAD.**

- **To prepare 2 D architectural drawing**

- **To prepare 3 D architectural drawing**

- **To prepare the drawing of grillage foundation using Auto cad.**

- **To study the commands used in modelling and design of structure**

  using STAAD- Pro.

- **Analysis of 2D Frames.**

- **Analysis of 3D Frames.**

- **Design of 2D RCC frames and 2D Steel frames according to IS codes.**

- **Design of 3D RCC frames and 3D Steel Frames according to IS codes.**

- **Design of beams.**

- **Analysis of truss frames.**

EXPERIMENT NO - 1

Title: Introduction to AutoCAD Commands

Objective: To learn and practice some of the most important commands in AutoCAD.

Materials:
- AutoCAD software
- Computer
- Mouse
- Keyboard

Instructions:
- Open AutoCAD and create a new drawing.
- Set up the drawing units to the desired format. To do this, type "UNITS" in the command line and press Enter. Choose the appropriate settings for your drawing.
- Start practicing the following commands:
- LINE: This command is used to draw a straight line between two points. To use this command, type "LINE" in the command line and press Enter. Select the first point, then select the second point, and press Enter to finish.
- CIRCLE: This command is used to draw a circle of a specified radius. To use this command, type "CIRCLE" in the command line and press Enter. Specify the center point of the circle, then specify the radius, and press Enter to finish.
- ARC: This command is used to draw an arc of a specified radius and angle. To use this command, type "ARC" in the command line and press Enter. Specify the center point of the arc, then specify the start point, and finally specify the end point or angle, and press Enter to finish.

- POLYLINE: This command is used to draw a series of connected line segments or arcs. To use this command, type "PL" in the command line and press Enter. Specify the first point, then specify additional points as needed, and press Enter to finish.

- ERASE: This command is used to delete objects from the drawing. To use this command, type "ERASE" in the command line and press Enter. Select the objects you want to delete, and press Enter to confirm.

- MOVE: This command is used to move objects to a new location. To use this command, type "MOVE" in the command line and press Enter. Select the objects you want to move, specify the base point, specify the second point, and press Enter to finish.

- COPY: This command is used to create a copy of an object. To use this command, type "COPY" in the command line and press Enter. Select the object you want to copy, specify the base point, specify the second point, and press Enter to finish.

- OFFSET: This command is used to create a parallel line or arc at a specified distance from an existing object. To use this command, type "OFFSET" in the command line and press Enter. Specify the object you want to offset, specify the distance, and press Enter to finish.

- Practice using these commands on your own. Try drawing different shapes and figures, and experiment with different options and settings.

- Conclusion:

- In this lab, you learned and practiced some of the most important commands in AutoCAD. These commands are essential for creating and editing drawings in AutoCAD, and will be useful in many different applications. Keep practicing and exploring the capabilities of AutoCAD to become a proficient user.

Title: Creating a 2D Plan in AutoCAD

Objective: To create a basic 2D plan of a room in AutoCAD.

Materials:

- AutoCAD software
- Computer
- Mouse
- Keyboard

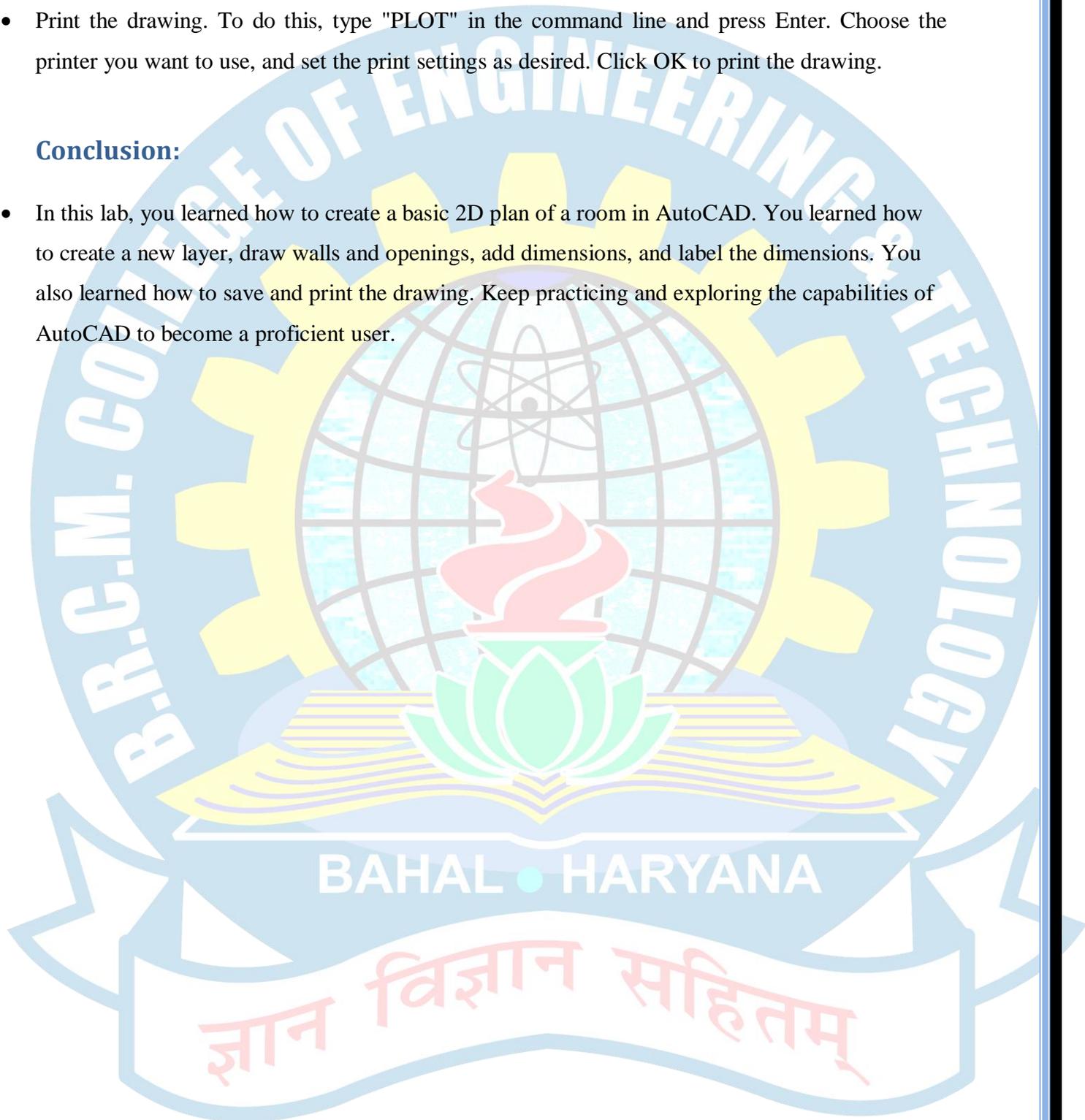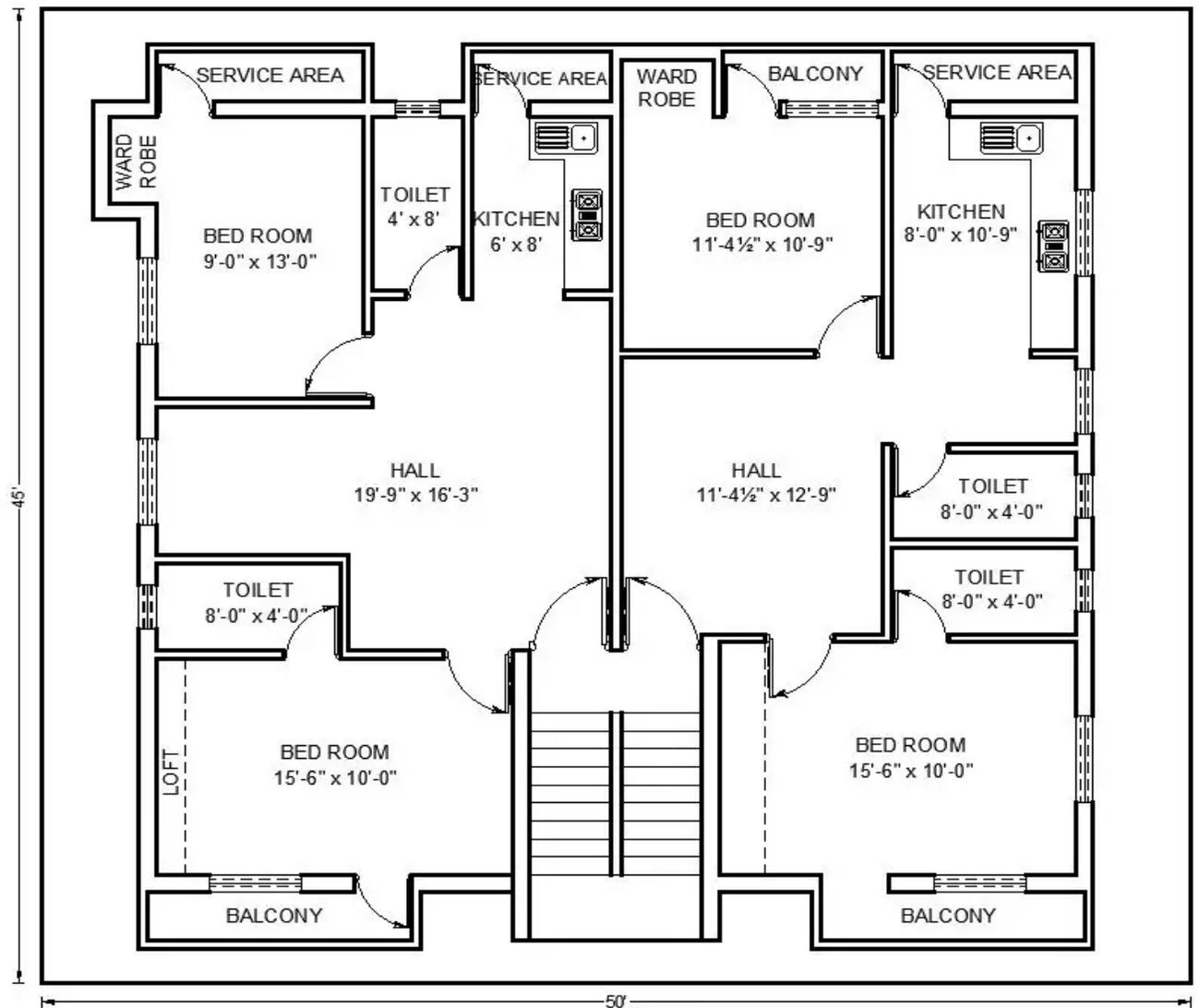## Instructions:

- Open AutoCAD and create a new drawing.
- Set up the drawing units to the desired format. To do this, type "UNITS" in the command line and press Enter. Choose the appropriate settings for your drawing.
- Start by creating a new layer for your plan. To do this, type "LAYER" in the command line and press Enter. Click on the "New Layer" icon, and name the layer "Plan". Set the color of the layer to black, and click OK.
- Start drawing the walls of the room. To do this, use the LINE command. Type "LINE" in the command line and press Enter. Specify the starting point of the first wall, then the ending point of the first wall, and press Enter. Repeat this process for the remaining walls of the room.
- Create openings for doors and windows. To do this, use the LINE command to draw the outlines of the openings, then use the TRIM command to remove the unwanted lines.
- Add dimensions to the plan. To do this, use the DIMENSION command. Type "DIMENSION" in the command line and press Enter. Select the two points you want to dimension, and click to place the dimension line. Repeat this process for all the dimensions you want to add.
- Label the dimensions. To do this, use the TEXT command. Type "TEXT" in the command line and press Enter. Click on the point where you want to place the label, and type in the label. Repeat this process for all the dimensions.

- Save the drawing. To do this, type "SAVE" in the command line and press Enter. Choose the location where you want to save the drawing, and give it a name.
- Print the drawing. To do this, type "PLOT" in the command line and press Enter. Choose the printer you want to use, and set the print settings as desired. Click OK to print the drawing.

Conclusion:

- In this lab, you learned how to create a basic 2D plan of a room in AutoCAD. You learned how to create a new layer, draw walls and openings, add dimensions, and label the dimensions. You also learned how to save and print the drawing. Keep practicing and exploring the capabilities of AutoCAD to become a proficient user.

SERVICE AREA

WARD ROBE

SERVICE AREA

WARD ROBE

BALCONY

SERVICE AREA

BED ROOM
9'-0" x 13'-0"

TOILET
4' x 8'

KITCHEN
6' x 8'

BED ROOM
11'-4½" x 10'-9"

KITCHEN
8'-0" x 10'-9"

HALL
19'-9" x 16'-3"

HALL
11'-4½" x 12'-9"

TOILET
8'-0" x 4'-0"

TOILET
8'-0" x 4'-0"

TOILET
8'-0" x 4'-0"

LOFT

BED ROOM
15'-6" x 10'-0"

BED ROOM
15'-6" x 10'-0"

BALCONY

BALCONY

45'

50'

FIRST FLOOR PLAN

Title: Creating a 3D Plan in AutoCAD

Objective: To create a basic 3D plan of a room in AutoCAD.

Materials:

- AutoCAD software
- Computer
- Mouse
- Keyboard

Instructions:

- Open AutoCAD and create a new drawing.
- Set up the drawing units to the desired format. To do this, type "UNITS" in the command line and press Enter. Choose the appropriate settings for your drawing.
- Start by creating a new layer for your plan. To do this, type "LAYER" in the command line and press Enter. Click on the "New Layer" icon, and name the layer "Plan". Set the color of the layer to black, and click OK.
- Start drawing the walls of the room. To do this, use the LINE command. Type "LINE" in the command line and press Enter. Specify the starting point of the first wall, then the ending point of the first wall, and press Enter. Repeat this process for the remaining walls of the room.
- Create openings for doors and windows. To do this, use the LINE command to draw the outlines of the openings, then use the TRIM command to remove the unwanted lines.
- Switch to 3D mode. To do this, type "VSCURRENT" in the command line and press Enter. In the "Current Viewport Control" dialog box, choose "3D Modeling". Click OK.
- Extrude the walls. To do this, use the EXTRUDE command. Type "EXTRUDE" in the command line and press Enter. Select the wall you want to extrude, and specify the height of the extrusion. Repeat this process for all the walls.

- Create openings in the walls. To do this, use the SUBTRACT command. Type "SUBTRACT" in the command line and press Enter. Select the wall you want to subtract from, and then select the opening you want to use to subtract. Repeat this process for all the openings.

- Add a floor to the room. To do this, use the 3DFACE command. Type "3DFACE" in the command line and press Enter. Specify the four corners of the floor, and press Enter.

- Add dimensions to the plan. To do this, use the DIMENSION command. Type "DIMENSION" in the command line and press Enter. Select the two points you want to dimension, and click to place the dimension line. Repeat this process for all the dimensions you want to add.

- Label the dimensions. To do this, use the TEXT command. Type "TEXT" in the command line and press Enter. Click on the point where you want to place the label, and type in the label. Repeat this process for all the dimensions.

- Save the drawing. To do this, type "SAVE" in the command line and press Enter. Choose the location where you want to save the drawing, and give it a name.

- Print the drawing. To do this, type "PLOT" in the command line and press Enter. Choose the printer you want to use, and set the print settings as desired. Click OK to print the drawing.

Conclusion:

- In this lab, you learned how to create a basic 3D plan of a room in AutoCAD. You learned how to create a new layer, draw walls and openings, extrude the walls, subtract openings, add a floor, and add dimensions and labels. You also learned how to save and print the drawing. Keep practicing and exploring the capabilities of AutoCAD to become a proficient user.

Title: Creating a Grillage Foundation in AutoCAD

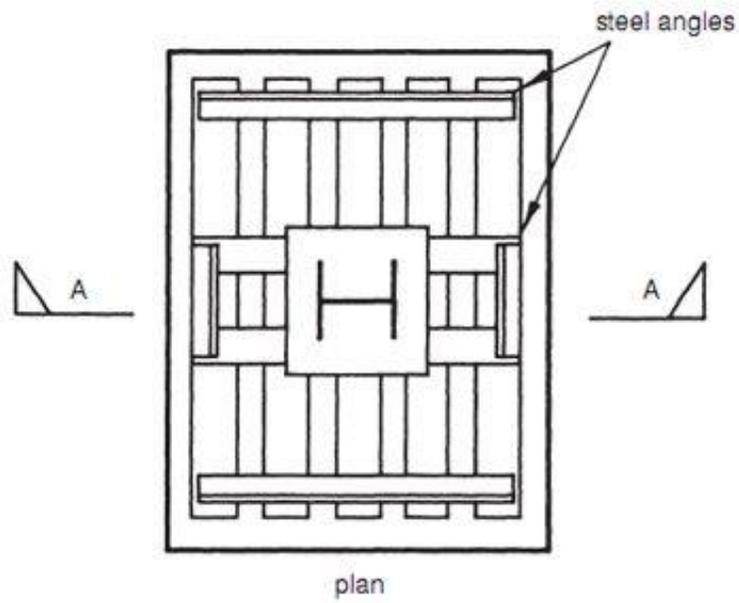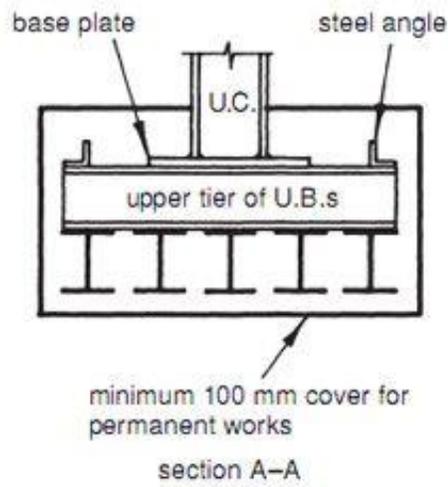**Objective**: To create a grillage foundation using AutoCAD.

Materials:

- AutoCAD software
- Computer
- Mouse
- Keyboard

Instructions:

- Open AutoCAD and create a new drawing.
- Set up the drawing units to the desired format. To do this, type "UNITS" in the command line and press Enter. Choose the appropriate settings for your drawing.
- Create a new layer for the foundation. To do this, type "LAYER" in the command line and press Enter. Click on the "New Layer" icon, and name the layer "Grillage Foundation". Set the color of the layer to black, and click OK.
- Draw the outline of the foundation. To create the foundation shape, use the RECTANGLE command. Type "RECTANGLE" in the command line and press Enter. Specify the starting point of the foundation, then the width and height of the foundation, and press Enter. This will create a rectangle representing the foundation.
- Draw the grid of lines for the steel beams. To create the grid of lines, use the LINE command. Type "LINE" in the command line and press Enter. Specify the starting point of the first vertical line on the left side of the foundation, then the ending point of the first vertical line, and press Enter. Then, specify the starting point of the next vertical line, and the ending point of the same line. Repeat this process until you reach the right side of the foundation. Then, repeat the process for the horizontal lines at the top of the foundation.
- Extrude the vertical lines to create the steel beams. To do this, use the EXTRUDE command. Type "EXTRUDE" in the command line and press Enter. Select the first vertical line you want

to extrude, and specify the height of the extrusion as the height of the steel beam. Repeat this process for all the vertical lines in the grid.

- Extrude the horizontal lines to create the cross beams. To do this, use the EXTRUDE command. Type "EXTRUDE" in the command line and press Enter. Select the first horizontal line you want to extrude, and specify the height of the extrusion as the height of the cross beam. Repeat this process for all the horizontal lines in the grid.

- Add dimensions to the foundation. To do this, use the DIMENSION command. Type "DIMENSION" in the command line and press Enter. Select the two points you want to dimension, and click to place the dimension line. Repeat this process for all the dimensions you want to add. The dimensions should be accurate and should represent the overall dimensions of the foundation, as well as the dimensions of the steel beams and cross beams.

- Label the dimensions. To do this, use the TEXT command. Type "TEXT" in the command line and press Enter. Click on the location where you want to place the text, and type in the label for the dimension. The labels should be clear and easy to read, and should represent the dimensions accurately.

- Create a sectional plan of the foundation. To do this, create a new layer named "Section". Use the LINE command to draw a line representing the cut section of the foundation. This line should be perpendicular to the long axis of the foundation. Use the SECTIONPLANE command to create a section view of the foundation. The section plane should be perpendicular to the cut section line.

- Add dimensions and labels to the sectional plan. To do this, use the DIMENSION and TEXT commands as in steps 8 and 9. The dimensions should accurately represent the dimensions of the section, and should be labeled accordingly.

- Add hatching to the sectional plan. To do this, use the HATCH command. Type "HATCH" in the command.

- Save the drawing. To do this, click on the "Save" icon in the toolbar, or type "SAVE" in the command line and press Enter. Choose a file name and location for the drawing, and click Save.

base plate   steel angle

U.C.

upper tier of U.B.s

minimum 100 mm cover for
permanent works

section A–A

steel angles

A   H   A

plan

Title: To study the commands used in modelling and design of structure using STAAD- Pro.

Objective: Commands used in STAAD- Pro.
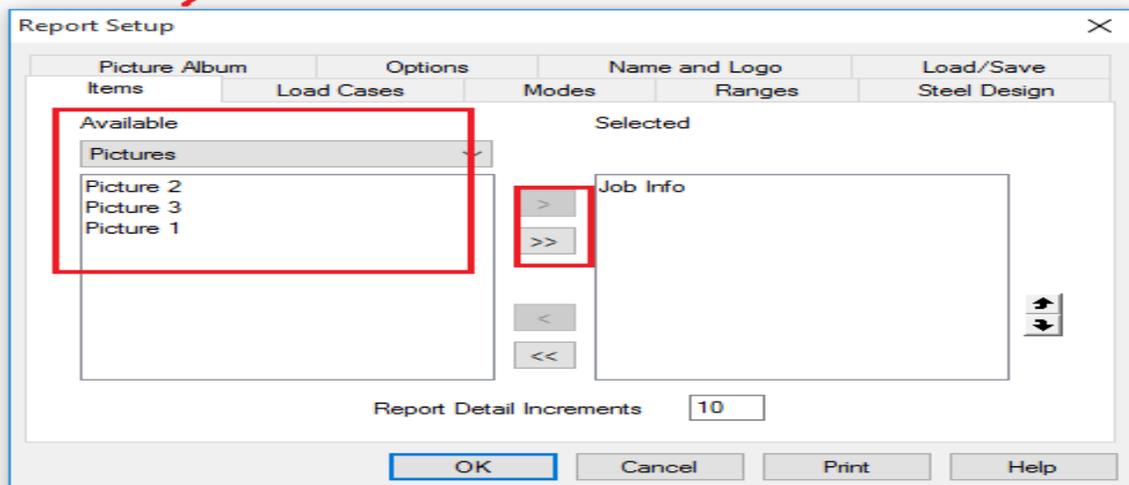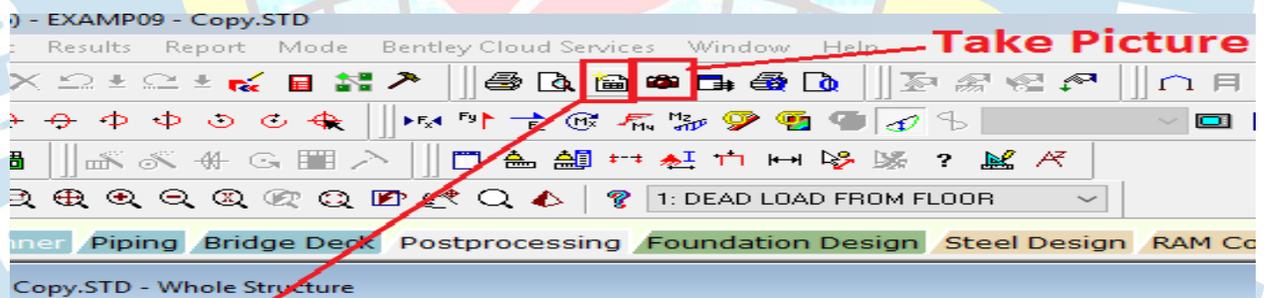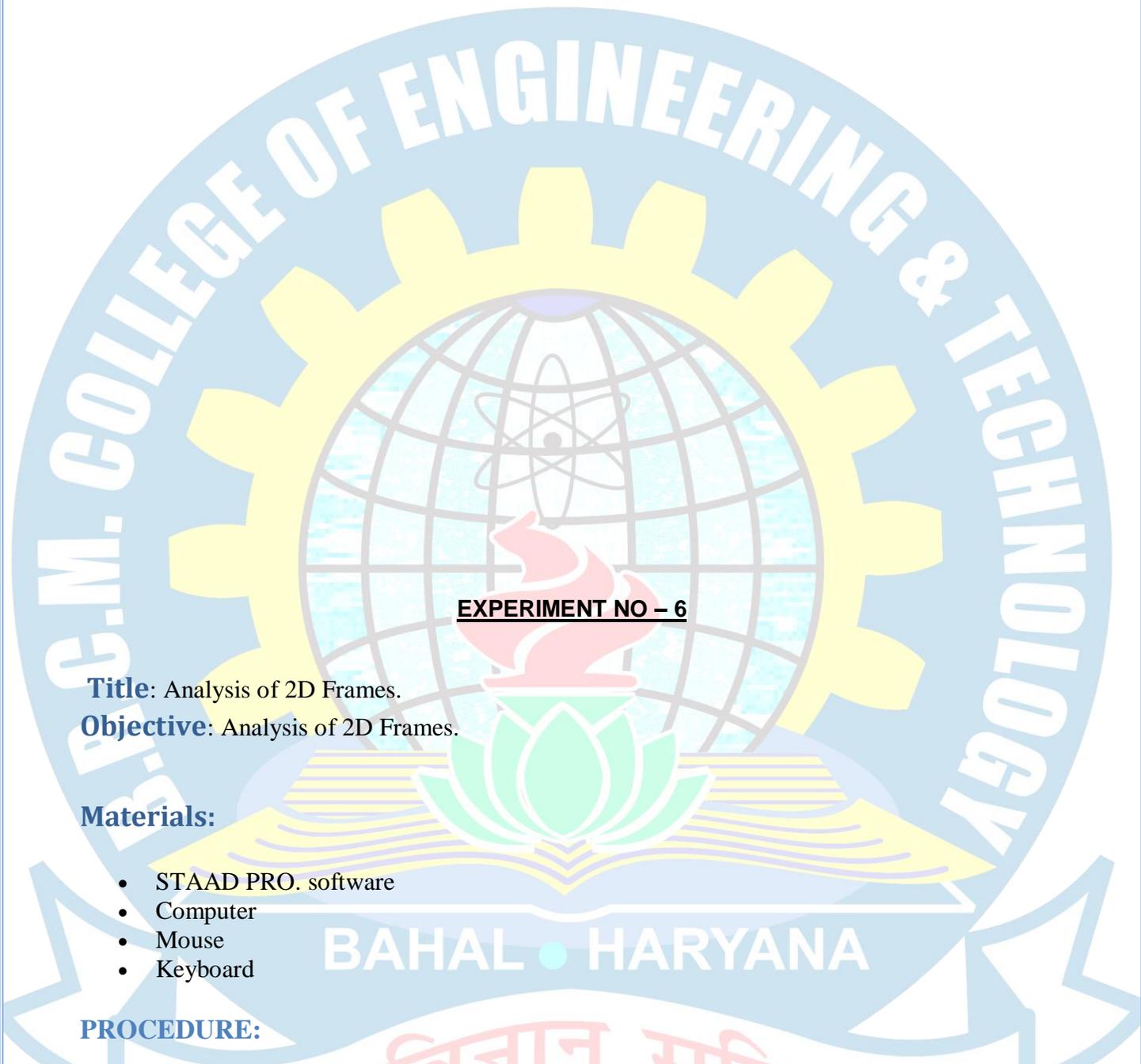
Materials:

- STAAD PRO. software
- Computer
- Mouse
- Keyboard

PROCEDURE:

- UNIT: This command is used to set the unit system for the analysis. Syntax: UNIT <SYSTEM> where <SYSTEM> is the unit system you want to use (e.g. METER, KILOGRAM, SECOND, NEWTON).

- JOINT COORDINATES: This command is used to input joint coordinates for the structure. Syntax: JOINT COORDINATES <JOINT #> <X> <Y> <Z> where <JOINT #> is the number of the joint, and <X>, <Y>, <Z> are the coordinates of the joint.

- MEMBER INCIDENCES: This command is used to specify which joints are connected by which members. Syntax: MEMBER INCIDENCES <MEMBER #> <I> <J> where <MEMBER #> is the number of the member, and <I>, <J> are the joint numbers connected by the member.

- DEFINE MATERIAL: This command is used to define the material properties of the structure. Syntax: DEFINE MATERIAL <MATERIAL #> E <MODULUS OF ELASTICITY> POISSON <POISSON RATIO> where <MATERIAL #> is the number assigned to the material, <MODULUS OF ELASTICITY> is the elastic modulus of the material, and <POISSON RATIO> is the Poisson's ratio of the material.

- DEFINE SECTION: This command is used to define the cross-sectional properties of the members. Syntax: DEFINE SECTION <SECTION #> TYPE <SECTION TYPE> <SECTION PARAMETERS> where <SECTION #> is the number assigned to the section, <SECTION TYPE> is the type of section (e.g. RECTANGLE, TUBE), and <SECTION PARAMETERS> are the parameters required to define the section.

- **SUPPORTS:** This command is used to define the supports for the structure. Syntax: SUPPORTS <JOINT #> <DEGREES OF FREEDOM> where <JOINT #> is the number of the joint, and <DEGREES OF FREEDOM> are the degrees of freedom to be restrained (e.g. UX, UY, UZ, RX, RY, RZ).

- **LOADS:** This command is used to apply loads to the structure. Syntax: LOADS <LOAD TYPE> <LOAD PARAMETERS> where <LOAD TYPE> is the type of load (e.g. CONCENTRATED, UNIFORM), and <LOAD PARAMETERS> are the parameters required to define the load (e.g. magnitude, location).

- **ANALYSIS:** This command is used to perform the structural analysis. Syntax: ANALYSIS where the program will run the analysis based on the input data.

- **PRINT:** This command is used to print out the analysis results. Syntax: PRINT <PRINT TYPE> where <PRINT TYPE> is the type of results to be printed (e.g. FORCES, DISPLACEMENTS, STRESSES).

- **POSTPROCESS:** This command is used to visualize the analysis results. Syntax: POSTPROCESS <POSTPROCESS TYPE> where <POSTPROCESS TYPE> is the type of post-processing to be performed (e.g. DEFLECTIONS, MOMENT DIAGRAMS, SHEAR DIAGRAMS).

EXPERIMENT NO – 6

Title: Analysis of 2D Frames.
Objective: Analysis of 2D Frames.

Materials:

- STAAD PRO. software
- Computer
- Mouse
- Keyboard

PROCEDURE:

- Start by launching STAAD Pro and creating a new project. You can do this by clicking on "New Project" on the start-up screen, or by going to "File" > "New" in the menu bar.

- Define the units for the analysis using the "UNIT" command. This ensures that all input data is entered in a consistent unit system. For example, you can use the command "UNIT METER KILOGRAM SECOND NEWTON" to set the unit system to the SI system.

- Define the geometry of the 2D frame by specifying the coordinates of the joints using the "JOINT COORDINATES" command. For example, you can use the command "JOINT COORDINATES 1 0 0 0" to specify the coordinates of joint 1 as (0,0,0).

- Specify the connectivity between the joints by defining the member incidences using the "MEMBER INCIDENCES" command. For example, you can use the command "MEMBER INCIDENCES 1 1 2" to connect joints 1 and 2 with member 1.

- Define the material properties of the members using the "DEFINE MATERIAL" command. For example, you can use the command "DEFINE MATERIAL 1 E 200000000 POISSON 0.3" to define a material with elastic modulus 200 GPa and Poisson's ratio 0.3.

- Define the cross-sectional properties of the members using the "DEFINE SECTION" command. For example, you can use the command "DEFINE SECTION 1 TYPE RECTANGLE 0.1 0.3" to define a rectangular section with dimensions 0.1 m x 0.3 m.

- Define the supports for the frame using the "SUPPORTS" command. For example, you can use the command "SUPPORTS 1 PINNED" to define joint 1 as a pinned support.

- Apply loads to the frame using the "LOADS" command. For example, you can use the command "LOAD 2 CONCENTRATED FORCE 0 -1000 0" to apply a concentrated downward force of 1000 N at joint 2.

- Perform the analysis using the "ANALYSIS" command. This command will run the analysis and generate the results.

- Review the results of the analysis using the "PRINT" and "POSTPROCESS" commands. The "PRINT" command can be used to display the numerical results of the analysis, while the "POSTPROCESS" command can be used to generate graphical output such as shear and moment diagrams.

- If necessary, refine the model by adjusting the input parameters and repeating the analysis until satisfactory results are obtained.

EXPERIMENT NO – 7

Title: Analysis of 3D Frames.

Objective: Analysis of 3D Frames.

Materials:

- STAAD PRO. software
- Computer
- Mouse
- Keyboard

PROCEDURE:

- Launch STAAD Pro and create a new file.

- Define the units for the analysis using the UNIT command.

- Input the joint coordinates for the structure using the JOINT COORDINATES command.

- Define the members of the structure using the MEMBER INCIDENCES command.

- Define the material properties of the structure using the DEFINE MATERIAL command.

- Define the cross-sectional properties of the members using the DEFINE SECTION command.

- Define the supports for the structure using the SUPPORTS command.

- Define the loads to be applied to the structure using the LOADS command.

- Define the solid elements for the 3D analysis using the SOLID ELEMENTS command.

- Define the properties of the solid elements using the SOLID PROPERTIES command.

- Define the solid boundary conditions using the SOLID BOUNDARY command.

- Define the analysis specifications using the ANALYSIS SPEC command.

- Run the analysis using the ANALYSIS command.

- View the analysis results using the POSTPROCESS command.

- Check the results for any errors or warnings and make any necessary changes to the input data.

- Print the analysis results using the PRINT command.

- Save the file for future reference.

EXPERIMENT NO – 8

Title: Design of 2D RCC frames and 2D Steel frames according to IS codes.

Objective: Design of 2D RCC frames and 2D Steel frames according to IS codes.

Materials:

- STAAD PRO. software
- Computer
- Mouse
- Keyboard

PROCEDURE:

- Launch STAAD Pro and create a new file.

- Define the units for the analysis using the UNIT command.

- Input the joint coordinates for the structure using the JOINT COORDINATES command.

- Define the members of the structure using the MEMBER INCIDENCES command.

- Define the material properties of the structure using the DEFINE MATERIAL command.

- Define the cross-sectional properties of the members using the DEFINE SECTION command.

- Define the supports for the structure using the SUPPORTS command.

- Define the loads to be applied to the structure using the LOADS command.

- Define the design parameters for the structure using the DESIGN command.

- Specify the design code to be used using the DESIGN CODE command, for example, IS 456:2000.

- Define the design parameters for the RCC frame using the DESIGN PARAMETER command, such as the minimum reinforcement ratio, minimum concrete grade, and minimum steel grade.

- Run the analysis using the ANALYSIS command.

- Check the analysis results for any errors or warnings.

- Generate the design results using the DESIGN command.

- Check the design results for any errors or warnings.

- View the detailed design results using the POSTPROCESS command.

- Check the results for any warnings or issues, and make any necessary changes to the input data.

- Print the detailed design results using the PRINT command.

- Save the file for future reference.

Title: Analysis of truss frames.
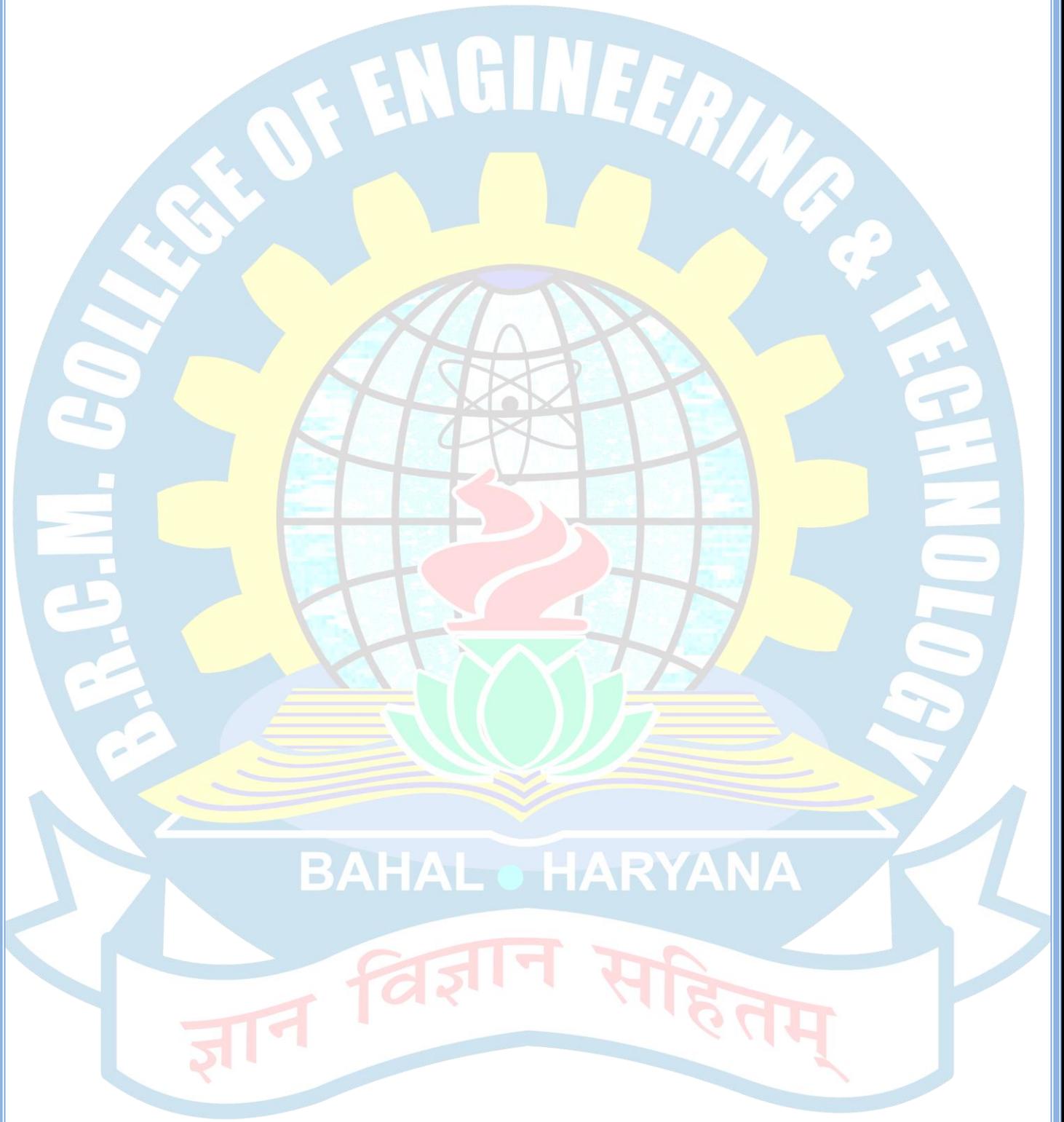
Objective: Analysis of truss frames.

Materials:

- STAAD PRO. software
- Computer
- Mouse
- Keyboard

PROCEDURE:

- Launch STAAD Pro and create a new file.

- Define the units for the analysis using the UNIT command.

- Input the joint coordinates for the truss using the JOINT COORDINATES command.

- Define the members of the truss using the MEMBER INCIDENCES command.

- Define the material properties of the truss using the DEFINE MATERIAL command.

- Define the cross-sectional properties of the members using the DEFINE SECTION command.

- Define the supports for the truss using the SUPPORTS command.

- Define the loads to be applied to the truss using the LOADS command.

- Specify the analysis type to be used using the ANALYSIS TYPE command. For a truss, the analysis type would be STATIC.

- Specify the load combination to be used for the analysis using the LOAD COMBINATION command.

- Run the analysis using the ANALYSIS command.

- Check the analysis results for any errors or warnings.

- View the analysis results using the POSTPROCESS command.

- Check the results for any warnings or issues, and make any necessary changes to the input data.

- Print the analysis results using the PRINT command.

- Save the file for future reference.

- It is important to note that there are many different options available for each command, and the specific details will vary depending on the specific truss being analyzed. It is recommended to study the user manual and familiarize oneself with the available options to use the software effectively. Additionally, it may be helpful to have a basic understanding of structural mechanics and finite element analysis to interpret the analysis results.