MODEL TESTING LABORATORY

Name of the Student: ……………………………………………………………….

Branch: ………………………………………………………………………………

Roll No: ……………………… …………… …**Academic Year:** ………………

GOKARAJU RANGARAJU

INSTITUTE OF ENGINEERING AND TECHNOLOGY

Nizampet Road, Bachupally, Hyderabad - 500 090

GOKARAJU RANGARAJU
INSTITUTE OF ENGINEERING AND TECHNOLOGY

Nizampet Road, Bachupally, Hyderabad - 500 090

CERTIFICATE

This is to certify that it is a bonafide record of practical work done in the ……………………………………………………………………………laboratory in ……………………… semester of ……………………… year during the academic year ………………by

Name of the Student : ………………………………………..

Roll. No. : ………………………………………..

Branch : ……………………………………….

Section : ………………………………...............

Signature of the Internal Examiner

Signature of the Head of Department

Signature of the External Examiner

Contents

Exercise	Content	Page No	Faculty Signature
I			
II			
III			
IV			
V			
VI			
VII			
VIII			
IX			
X			

INTRODUCTION TO STAAD PRO

ENGINEERING STRUCTURE AND STRUCTURAL DESIGN:

An engineering structure is an assembly of members or elements transferring the load or resisting external actions and providing a form to serve the desired function.

The structural design is a science and art of designing with economy and elegance. A durable structure, which can safely carry the forces and can serve the desired function satisfactorily during its expected service life span.

OBJECT AND BASIC REQUIREMENTS OF STRUCTURAL DESIGN :

- Serviceability
- Safety
- Durability
- Economy
- Aesthetic beauty

SEQUENCE OF STRUCTURAL DESIGN:

- Design of Slab
- Design of Beam
- Design of Column
- Design of Footiong
- Design of Staircase

Design Process:

Engineering is a professional art of applying the science to the efficient conversion of natural resources for the benefit of man. Engineering, therefore, requires above all creative imagination to innovate useful application for natural phenomenon.

The Design of any structure is categorized into two types:

1) Functional Design

2) Structural Design

1) Functional Design:

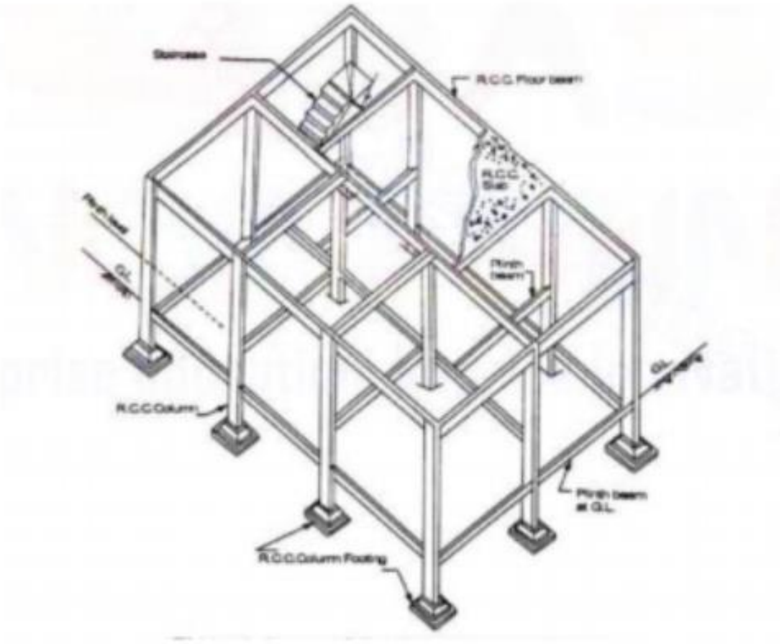
The structure to be constructed should primarily serve the basic purpose for which it is to be used and must have a pleasing look.

The following factors comes under Functional Design Whether the structure should be a load bearing structure or RCC framed structure or a steel structure, whether the roof shall consist of steel roof trusses and girders or RCC folded plates or a RC beam slab construction, Arrangement of rooms, good ventilation, lighting, acoustics, sufficient head room, proper water supply and drainage etc.

2) Structural Design:

Once the form of the structure is selected the structural design process starts.

Structural Design is an art and science of understanding the behaviour of structural members subjected to loads and designing them with economy and elegance to give a safe, serviceable and durable structure.



Structural details of framed structure

The principal elements of a RC building frame consists of:

1) Slab to cover a larger area
2) Beams to support slabs and walls
3) Columns to support beams
4) Footing to distribute concentrated column loads over a larger area of the supporting soil such that the bearing capacity of soil is not exceeded.

In a framed structure the load is transferred from slab to beam, from beam to column and then to foundation and soil below it.

Stages in STRUCTURAL DESIGN:

The process of structural design involves the following stages:

1) Structural Planning
2) Action of forces and Computation of loads
3) Methods of Analysis
4) Member design
5) Detailing, Drawing and Preparation of schedules.

STRUCTURAL PLANNING

After getting an architectural plan of the buildings, the structural planning of the building frame is done. This involves determination of the following:

(a) Positioning and Orientation of Columns
(b) Positioning of Beams.
(c) Spanning of Slabs.
(d) Layout of Stairs
(e) Selecting proper type of Footing.

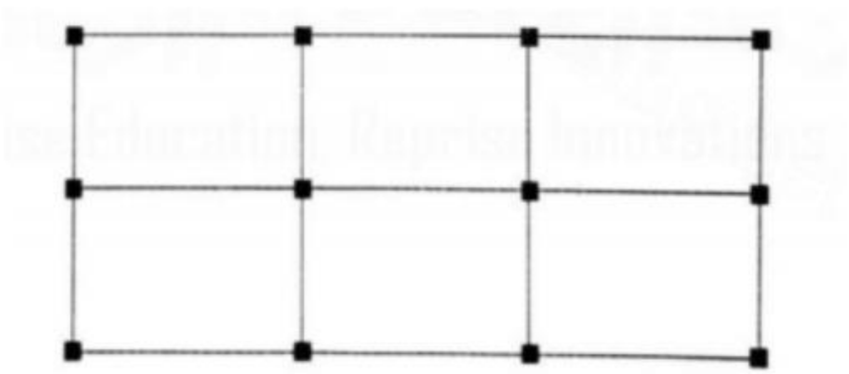
The basic principle in deciding the layout of component members is that the loads should be transferred to the foundation along the shortest path.

Positioning and Orientation of Columns:

Following are some of the guiding principles which help in deciding the column positions

(1) Columns should preferably be located at or near the corners of a building, and at the intersections of beams/walls:

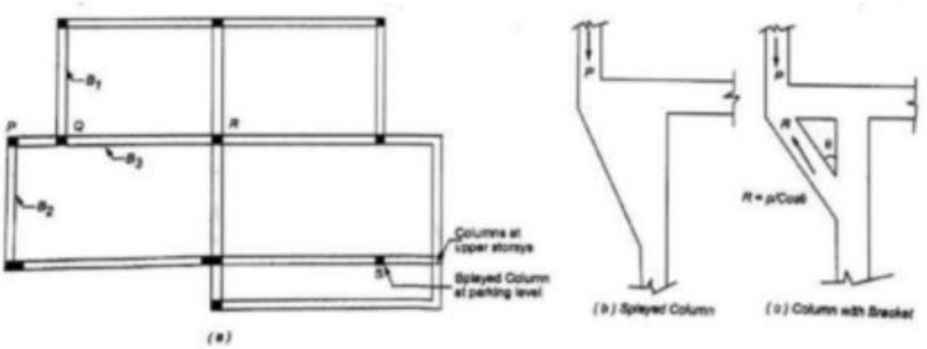
The positioning of columns at the intersection of walls and at the comers of a building is shown in Fig. Since the basic function of the columns is to support beams which are normally placed under the walls to support them, their position automatically gets fixed as shown in the figure. The commercial buildings have normally rectangular pattern of grid type shown in the figure but especially for residential buildings the said type of pattern for columns does not become possible and different problems that arise are discussed further.



Column position for rectangular pattern Building

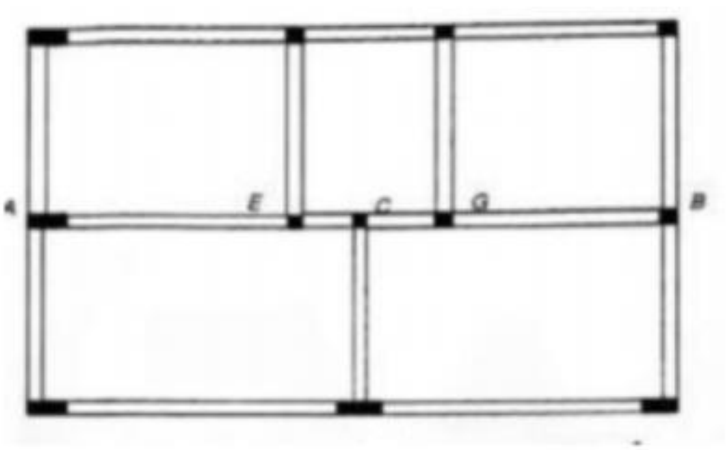
2) Select the position of columns so as to reduce bending moments in beams:

When the locations of two columns are very near (eg. as it occurs when the corner of a building and the point of intersection of walls come very close to each other), then one column should be provided instead of two at such a position so as to reduce the beam moment.



(3) Avoid larger spans of beams.

When the centre to centre distance between the intersection of walls is large or where there are no cross walls, the spacing between two columns is governed by limitations on spans of supported beams, because spacing of columns decides the span of the beam. As the span (and the length) of the beam increases, the required depth of the beam, and hence its self-weight, and the total load on beam increases.



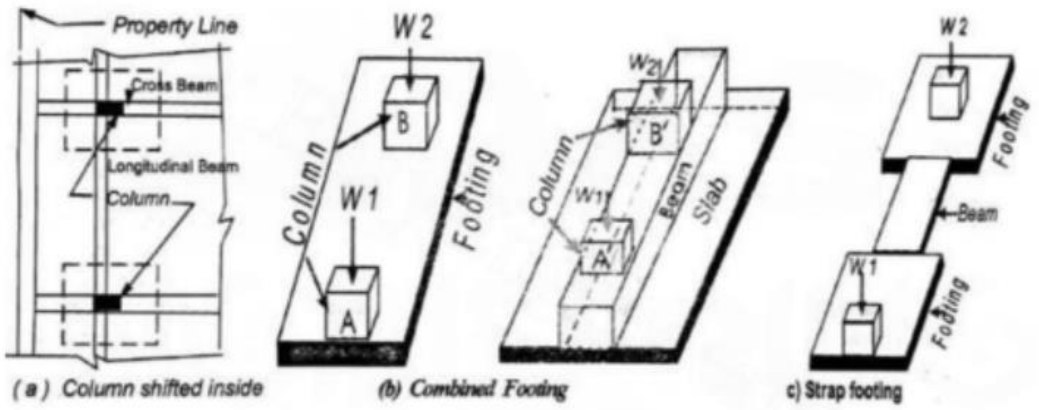
(4) Avoid larger centre to centre distance between columns:

It increases the span and cost of beams and in turn increases the load on the column at each

floor.

(5) Columns on property line:

Shift the column inside along a cross wall or adopt a combined footing or a strap footing.



Columns on Property line

Orientation of Columns:

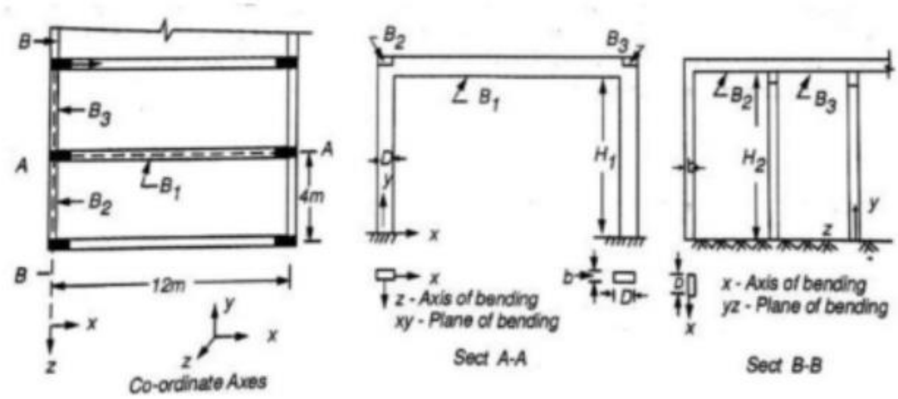
(1) Avoid projection of column outside wall:

When the thickness of wall is less than the width of column adopt a T shaped column at junctions or L shaped column at corners.

(2) Orient the column so that the depth of the column is contained in the major plane of bending or is perpendicular to the major axis of bending.

> Increasing the depth of the column in the plane of bending not only increases the moment carrying capacity but also increases its stiffness, thereby more moment is transferred to the column at the beam column junction.

If the difference in bending moment in two mutually perpendicular directions is not large, the depth of the column may be taken along the wall, provided column has sufficient strength in the plane of large moment.



Orientation of columns from stiffness and Effective Length Criteria.

Positioning of beams:

(1) Beams shall be provided under the walls or below a heavy concentrated load to avoid these loads directly coming on the slabs.

The maximum spacing of beams may be limited to the values of maximum spans of slabs given below(for a LL < 5KPa)

Support Condition	Cantilevers		Simply supported		Fixed/Continuous	
Slab type	One way	Two way	One way	Two way	One way	Two way
Max. recommended span of slabs	1.5m	2m	3.5m	4.5m	4.5m	6.0m

(2) Avoid larger spacing of beams from deflection and cracking criteria.

Deflection varies directly with the cube of the span and inversely with the cube of the depth i.e L^3/D^3. Hence increase in depth D is less than increase in span L which results in greater deflection for large span.

Spanning of Slabs:

It depends on the type of slab i.e., when the supports are only on opposite edges or only in one direction, then the slab acts as a one way supported slab.

When the rectangular slab is supported along its four edges, it acts as a one way slab when Ly/Lx is >2 and as two way slab for Ly/Lx < 2.

Note: A two way slab is generally economical compared to one way slab because steel along both the spans acts as main steel and transfers the load to all its four supports.

INTRODUCTION TO ETABS

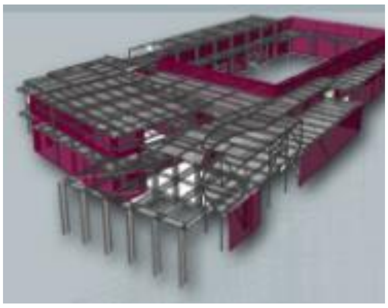
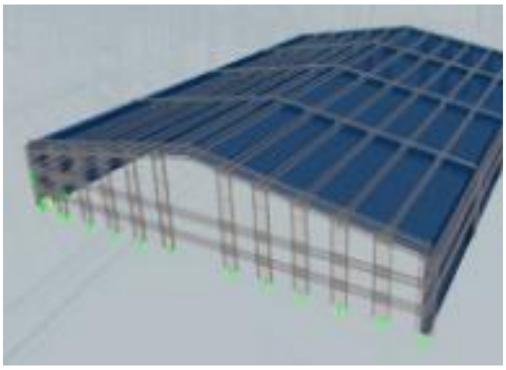
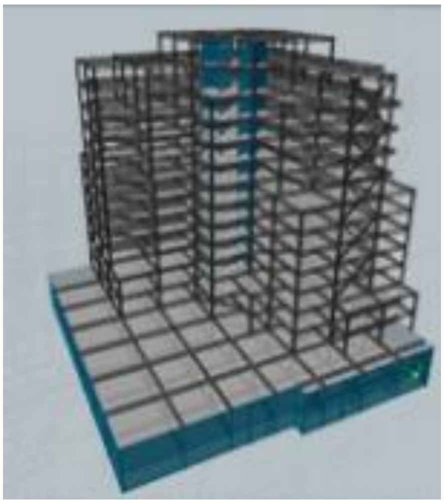
ETABS is an engineering software product that caters to multi-story building analysis and design. Modelling tools and templates, code-based load prescriptions, analysis methods and solution techniques, all coordinate with the grid-like geometry unique to this class of structure. Basic or advanced systems under static or dynamic conditions may be evaluated using ETABS. For a sophisticated assessment of seismic performance, modal and direct-integration time-history analyses may couple with P-Delta and Large Displacement effects.

Nonlinear links and concentrated PMM or fiber hinges may capture material nonlinearity under monotonic or hysteretic behaviour. Intuitive and integrated features make applications of any complexity practical to implement. Interoperability with a series of design and documentation platforms makes ETABS a coordinated and productive tool for designs which range from simple 2D frames to elaborate modern high-rises.

MODELLING OF STRUCTURAL SYSTEMS

Fundamental to ETABS modelling is the generalization that multi-story buildings typically consist of identical or similar floor plans that repeat in the vertical direction. Modelling features that streamline analytical-model generation, and simulate advanced seismic systems, are listed as follows:

- Templates for global-system and local-element modelling
- Customized section geometry and constitutive behaviour
- Grouping of frame and shell objects
- Link assignment for modelling isolators, dampers, and other advanced seismic systems
- Nonlinear hinge specification
- Automatic meshing with manual options
- Editing and assignment features for plan, elevation, and 3D views

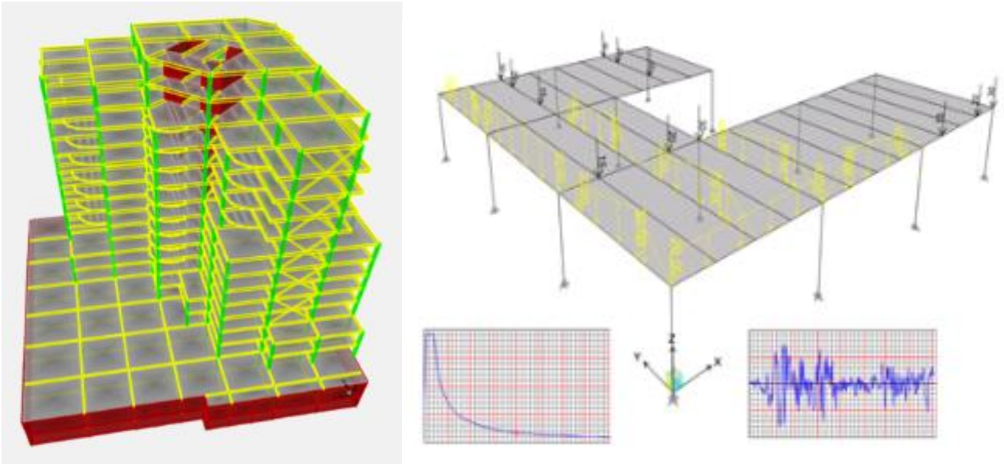
  

LOADING, ANALYSIS, AND DESIGN

Once modelling is complete, ETABS automatically generates and assigns code-based loading conditions for gravity, seismic, wind, and thermal forces. Users may specify an unlimited number of load cases and combinations.

Analysis capabilities then offer advanced nonlinear methods for characterization of static-pushover and dynamic response. Dynamic considerations may include modal, response-spectrum, or time-history analysis. P-delta effect account for geometric nonlinearity.

Given enveloping specification, design features will automatically size elements and systems, design reinforcing schemes, and otherwise optimize the structure according to desired performance measures.

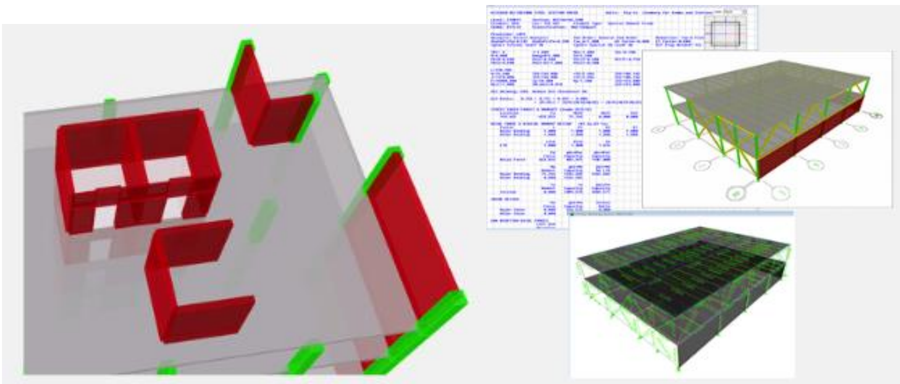


OUTPUT, INTEROPERABILITY, AND VERSATILITY

Output and display formats are also practical and intuitive. Moment, shear, and axial force diagrams, presented in 2D and 3D views with corresponding data sets, may be organized into customizable reports. Also available are detailed section cuts depicting various local response measures. Global perspectives depicting static displaced configurations or video animations of time-history response are available as well.

ETABS also features interoperability with related software products, providing for the import of architectural models from various technical drawing software, or export to various platforms and file formats. SAFE, the floor and foundation slab design software with post-tensioning (PT) capability, is one such option for export. CSI coordinated SAFE to be used in conjunction with ETABS such that engineers could more thoroughly detail, analyse, and design the individual levels of an ETABS model.

While ETABS features a variety of sophisticated capabilities, the software is equally useful for designing basic systems. ETABS is the practical choice for all grid-like applications ranging from simple 2D frames to the most complex high rises.



BASIC MODES, DRAWING TOOLS, MOUSE POINTERS

Select or Draw

- Draw Joint Objects
- Draw Beam/Column/Brace Objects
  - Draw Beam/Column/Brace (Plan, Elev, 3D)
  - Quick Draw Beams/Columns (Plan, Elev, 3D)
  - Quick Draw Columns (Plan)
  - Quick Draw Secondary Beams (Plan)
  - Quick Draw Braces (Elev)
- Draw Floor/Wall Objects
  - Draw Floor/Wall (Plan, Elev, 3D)
  - Draw Rectangular Floor/Wall (Plan, Elev)
  - Quick Draw Floor/Wall (Plan, Elev)
  - Draw Walls (Plan)
  - Quick Draw Walls (Plan)
  - Draw Wall Openings (Plan, Elev, 3D)
- Draw Links

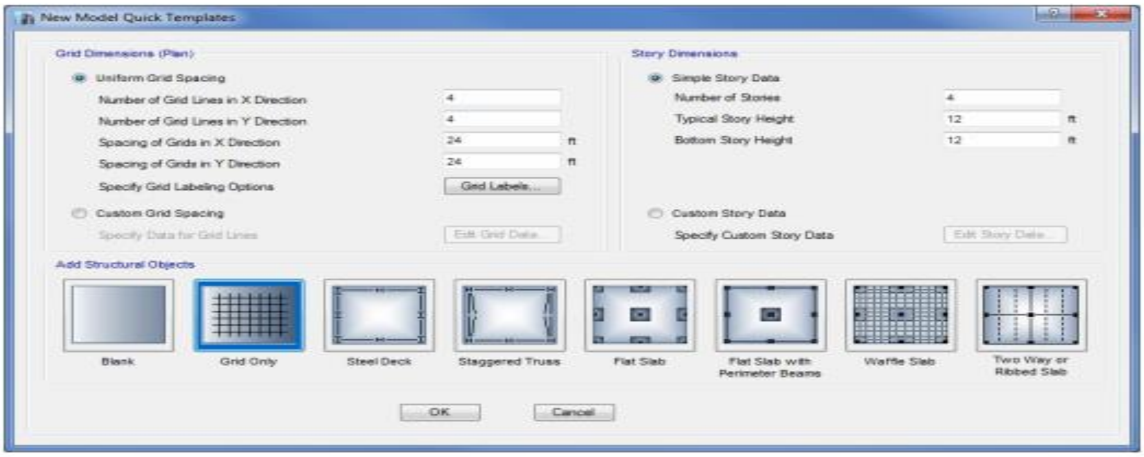
- Draw Tendons
- Draw Design Strips
- Draw Grids
- Draw Dimension Lines
- Draw Reference Points
- Draw Reference Planes
- Draw Section Cut
- Draw Developed Elevation Definition
- Draw Wall Stacks (Plan, Elev, 3D)
- Auto Draw Cladding
- Snap Options

Quick Model Template Forum

Grid Dimensions (Plan) - Define a Grid System Use the Grid Dimensions (Plan) area of the form to define a grid line system. Select from two options for defining the grid line system:

Uniform Grid Spacing. Specify the number of grid lines in the X and Y directions and a uniform spacing for those lines. Note that the uniform spacing in the X and Y directions can be different. This option defines a grid system for the global coordinate system only. Click the Grid Labels button to control how the grids are labelled. **If subsequently necessary, edit the information using the Edit menu > Edit Stories and Grid Systems command.**

Custom Grid Spacing. Define non uniformly spaced grid lines in the X and Y directions for the global coordinate system. After choosing this option, click the Edit Grid Data button to edit the grid system.



STORY DIMENSIONS

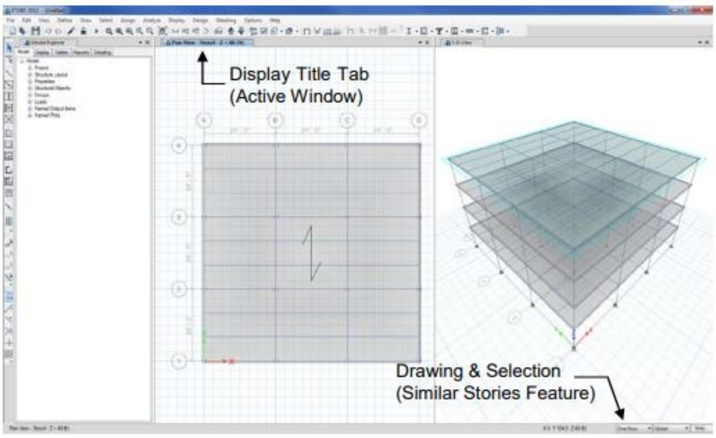
DEFINE STORY DATA

Use the Story Dimensions area of the form to define the number and height of stories. Select from two options for defining the story data:

♣ Simple Story Data: Enter values in the edit boxes to define the number of stories and a typical story height that is used for all story levels except for the bottom story, which is specified separately. The program provides default names for each story level (for example, Story1, Story2 and so on) and assumptions for story level similarity.

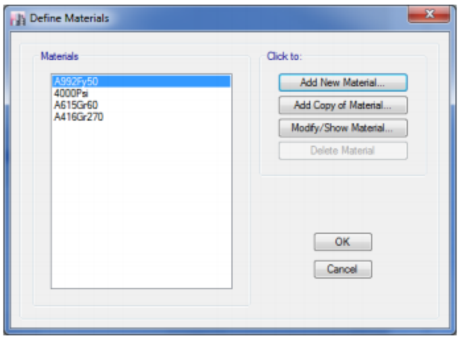
♣ Custom Story Data: After choosing this option, click the Edit Story Data button to access the Story Data form. Enter values in the Story Data form to define your own story names, story levels of non-uniform height and customized story similarity. Story level "similarity" can be significant, e.g., when Story2 is a Master Story, and Story1 is similar to Story2, an object drawn on Story2 typically appears in the same

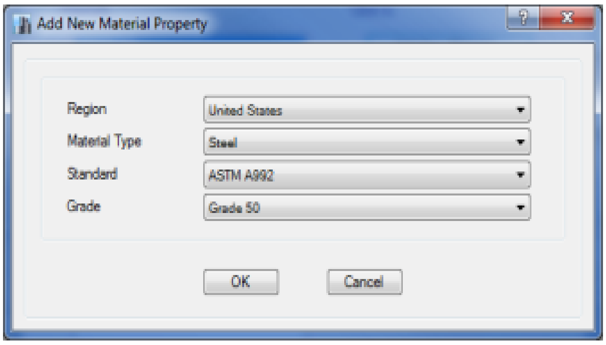
CREATE THE STRUCTURAL MODEL



MATERIAL PROPERTIES

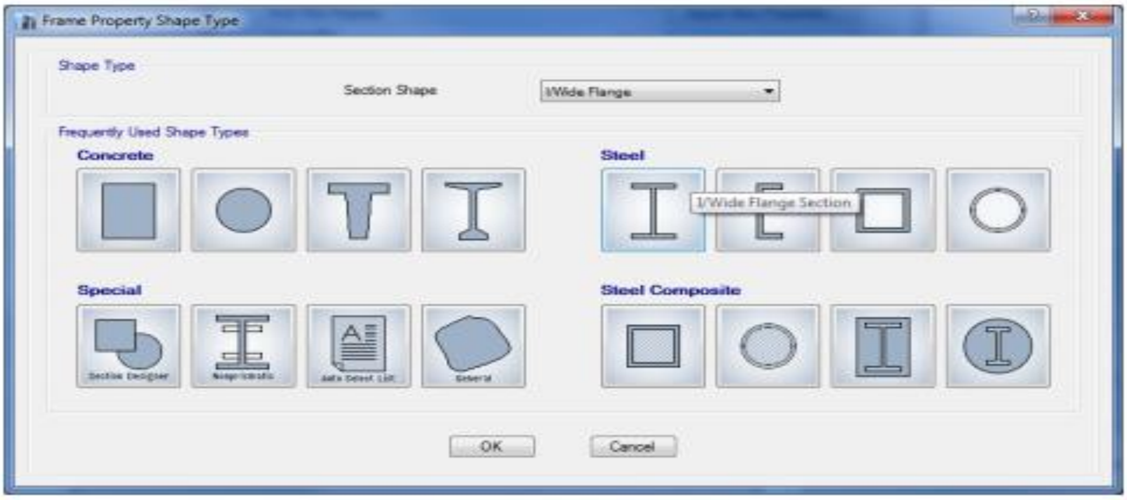
Click the Define menu > Material Properties command to display the Define Materials form or under the Model tab on the Model Explorer expand the Properties branch and then the Materials branch to see a list of the defined material properties (a right-click on the Materials branch will display a context sensitive menu). The Define Materials form allows for the both the review of existing materials, as well as the definition of new properties. To add a new material, click the Add New Material button on the Define Materials form. When the Add New Material Property form appears as shown in Figure 5-4, select a material from the Material Type drop-down list and then a Standard and Grade from their respective drop-down lists.





FRAME SECTIONS

Click the Define menu > Section Properties > Frame Sections command, which will display the Frame Properties form. The Frame Properties form allows for the definition of new sections as well as the review of existing sections. To make steel frame sections from property files available click the Import New Properties button, or to add user defined sections click the Add New Property button, both of which will display the Frame Property Shape Type form.



ASSIGNING THE PROPERTIES

In creating the model, the user draws joint, frame, shell, link, and tendon objects. To enable analysis and design, those objects must be assigned properties, such as material properties, frame sections, wall/slab/deck sections, link properties, tendon properties, and loads, among others. Note that the assign menu lists the various properties that can be assigned.

STRUCTURAL LOADS

The program allows the user to define a variety of structural loads, including dead, live, earthquake and wind loads. The user then assigns the loads to various structural objects in the model. An unlimited number of load patterns can be defined. Note that the steel frame, concrete frame, composite beam, composite column, steel joist, concrete shear wall, concrete slab, and steel connection design manuals describe design combinations in accordance with building codes.

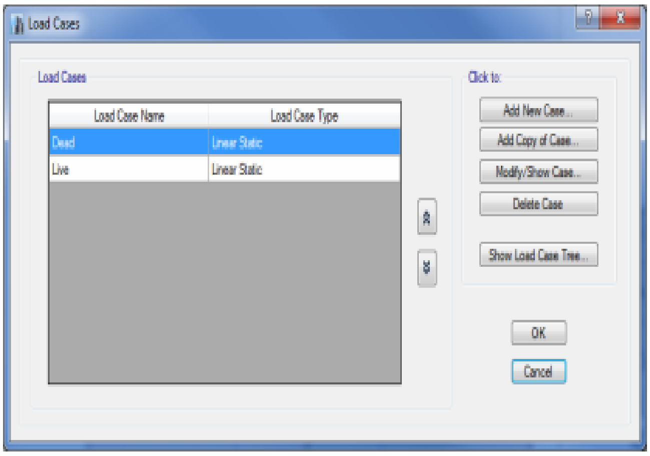
DEFINE THE LOAD PATTERNS

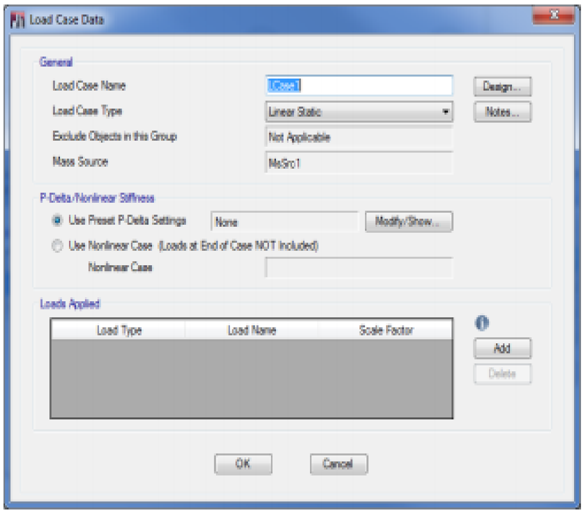
To add a load pattern, click the Define menu > Load Patterns command or expand the tree on the Model tab in the Model Explorer and right click on Load Patterns to access the Define Load Patterns form. Complete the following actions using that form:

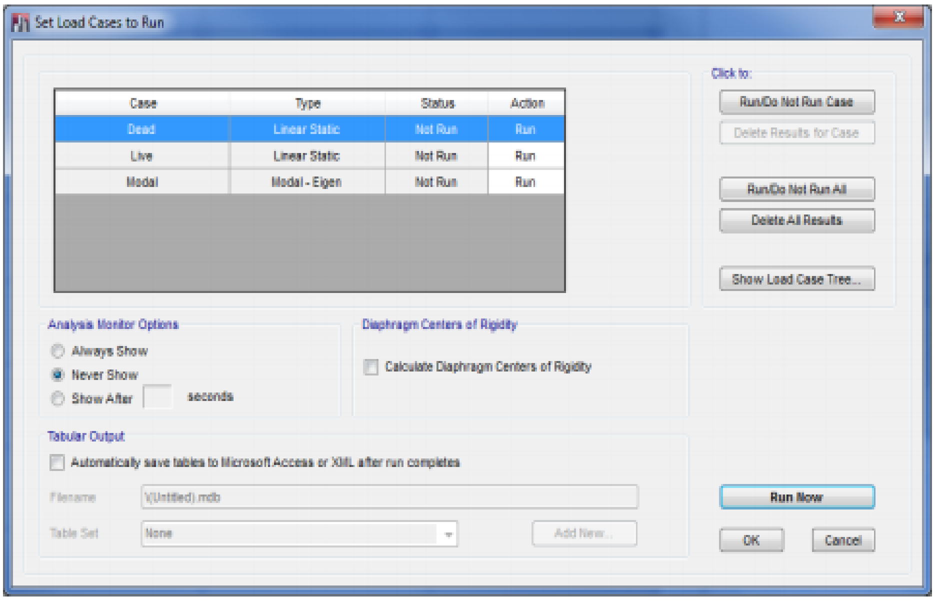
1. Type the name of the load pattern in the Load edit box. The program does not allow use of duplicate names.
2. Select a load type from the Type drop-down list.
3. Type a self-weight multiplier in the Self-Weight Multiplier edit box (see the explanation about the self-weight multiplier that follows).
4. If the load type specified is Seismic or Wind, select an option from the Auto Lateral Load drop-down list.
5. Click the Add New Load button.

REVIEW/CREATE LOAD CASES

A load case defines how loads are to be applied to the structure, and how the structural response is to be calculated. Analyses are classified in the broad sense as either linear or nonlinear, depending on how the model responds to the loading. The results of linear analyses may be superposed, i.e., added together after analysis. The results of nonlinear analyses normally should not be superposed. Instead, all loads acting together on the structure should be combined directly within the nonlinear load case. After all geometry and load input has been specified for a model, review, modify, or add load cases using the Define menu > Load Cases command. load case (ETABS automatically generates a load case for each load pattern defined) and click the Modify/Show Case button to review or modify the load case definition. Click the Delete Case button to delete the highlighted load case.

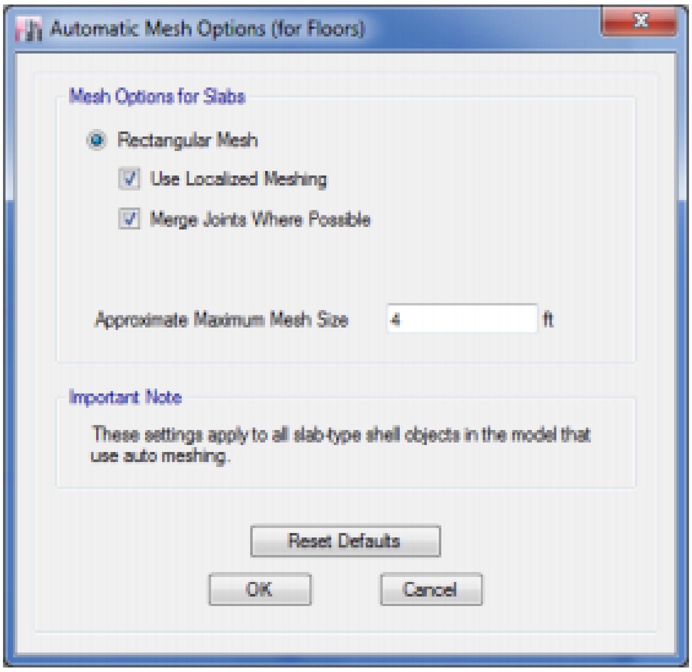






SET THE MESH OPTIONS

If your model contains wall objects, or floor objects that have plate bending behaviour such as cast-in-place slabs, review the meshing options (e.g., maximum mesh size) before running the analysis by using the Analyze menu > Automatic Mesh Settings for Floors or Analyze menu > Automatic Rectangular Mesh Settings for Walls commands. The Automatic Mesh Options (for Floors) form is shown in Figure 11-1. Default mesh settings for floors and walls may be reviewed by using the Assign menu > Shell > Floor Auto Mesh Options and the Assign menu > Shell > Wall Auto Mesh Options commands, respectively.



MODEL ANALYSIS

Prior to running the analysis, verify what load cases are set to run by clicking on the Analyze menu > Set Load Cases To Run command. The Set Load Cases to Run form. To add or remove a load case from the analysis, highlight the load case in the Case column and click the Run/Do Not Run Case button. Both the status and action for each case are shown in their respective columns. This form also allows the user to set how the Analysis Monitor should be displayed - the default setting is that it Never Shows.

To run the analysis, click the Run Now button if the Set Load Cases to Run form is still displayed, otherwise click the Analyze menu > Run Analysis command or the Run Analysis button, . The program will display an "Analysing, Please Wait" window if the Analysis Monitor has been set to "Always Show" or "Show After." Data will scroll in this window as the program runs the analysis. After the analysis has been completed, the program performs a few more “bookkeeping actions” that are evident on the status bar in the bottom left-hand corner of the ETABS window

LOCKING AND UNLOCKING THE MODEL

When the entire analysis process has been completed, the model automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, appears closed. Locking the model prevents any changes to the model that would invalidate the analysis results.

DESIGN THE STRUCTURE

The ETABS design postprocessors include the following:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Composite Column Design
- Steel Joist Design
- Shear Wall Design

- Concrete Slab Design
- Steel Connection Design

To perform the design, first run the analysis then click the Design menu and select the appropriate design from the drop-down menu. The type of design available depends on the type of members used in the model. That is, the user cannot complete a shear wall design if no shear walls have been included in the model, nor could they do a steel connection design if no steel members are present. Similarly, the commands used to execute a design depend on the type of design to be performed. However, each design has commands to address the following:

♣ Review and/or select overwrites.

♣ Review and/or select design combinations.

♣ Start the design or check of the structure.

♣ Perform interactive design.

♣ Display input and output design information on the model.

DETAILING PROCESS

ETABS detailing generates two basic types of drawing output:

♣ Drawing sheet component views of detailed objects, such as steel beam framing plans, steel column schedules, concrete beam elevations and sections, concrete column schedules, shear wall reinforcing sections and elevations, and steel connection tables

♣ Drawing sheets containing the selected component views

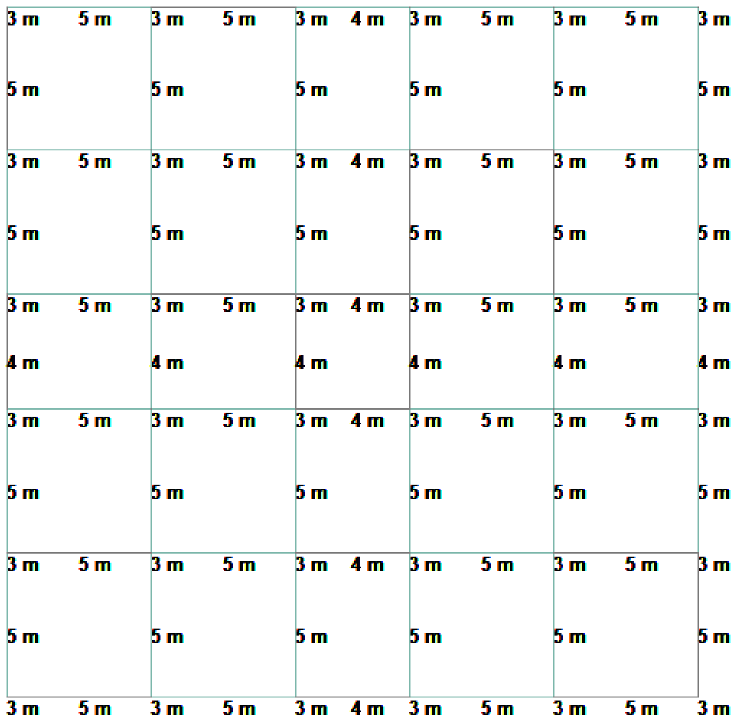
1a. MODELLING OF THE GIVEN STRUCTURE USING STAAD PRO

Aim:

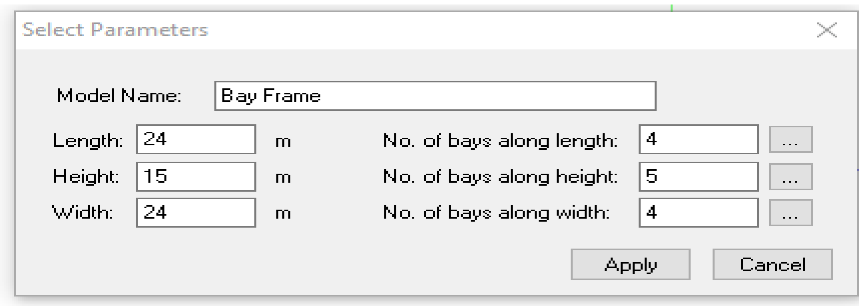
To model the given structure using STAAD pro

Software Used:

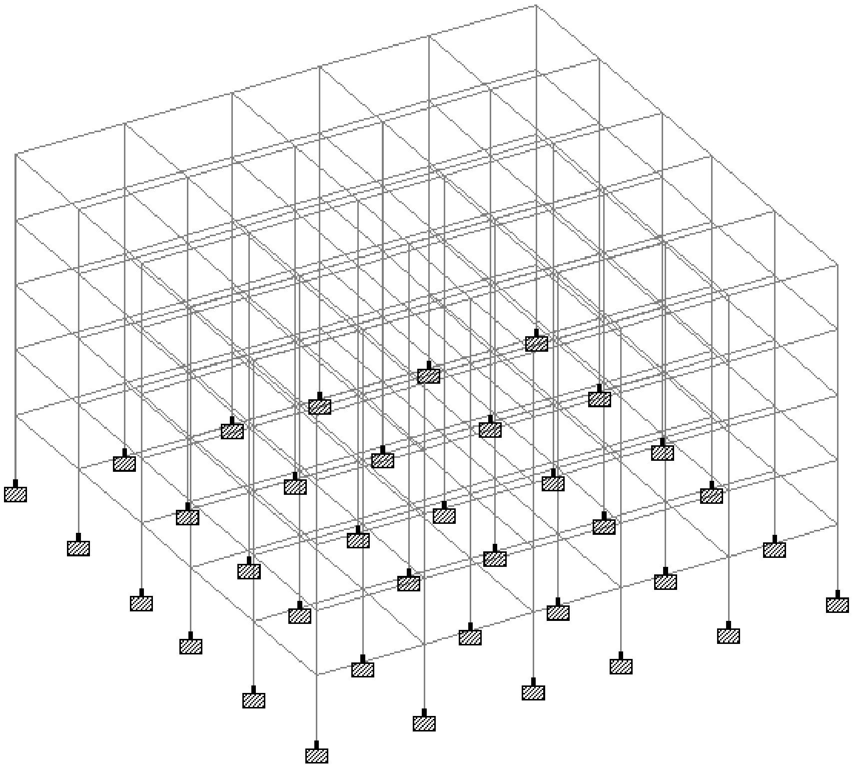
STAAD Pro



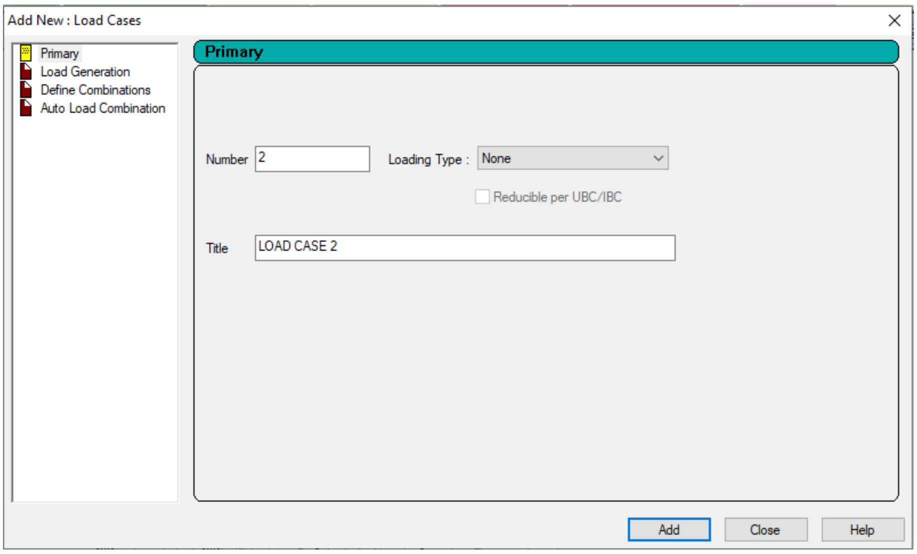
Layout of the building



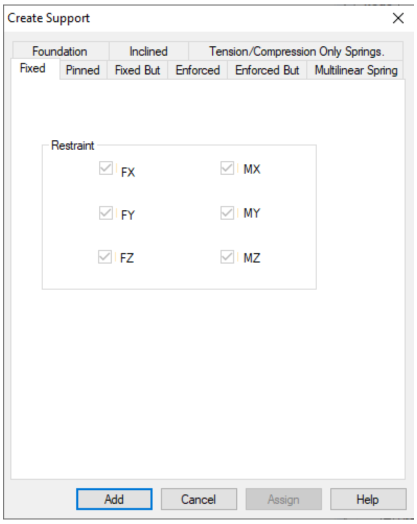
Defining the dimensions of the building



Model of the building



Defining primary loads



Defining supports

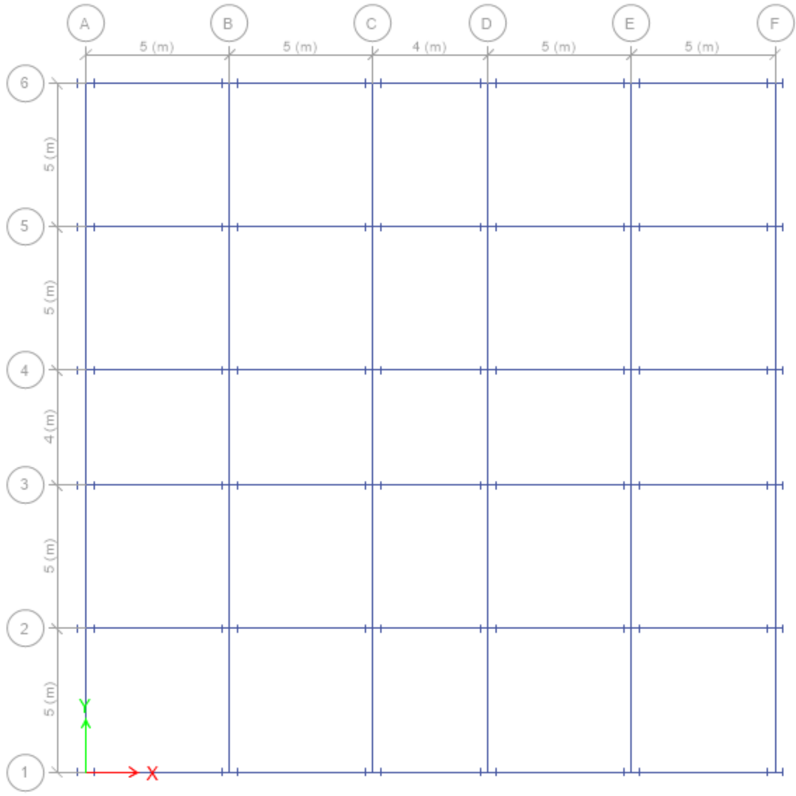
1b. Modelling of the given structure using ETABS

Aim:

To model the given structure using etabs

Software used:

Etabs

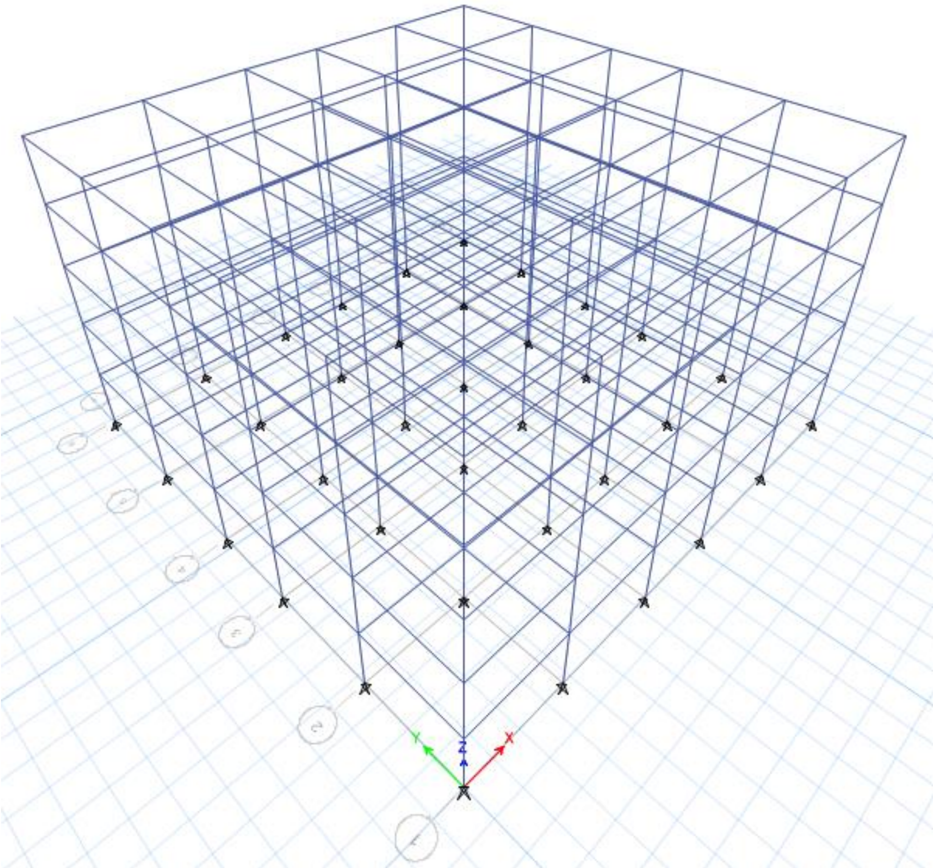


plan of the model

Story Data

	Story	Height m	Elevation m	Master Story	Similar To	Splice Story	Splice Height m	Story Color
▸	Story5	3	15	Yes	None	No	0	
	Story4	3	12	No	Story5	No	0	
	Story3	3	9	No	Story5	No	0	
	Story2	3	6	No	Story5	No	0	
	Story1	3	3	No	Story5	No	0	
	Base		0					

Storey data



Modelling of the given plan

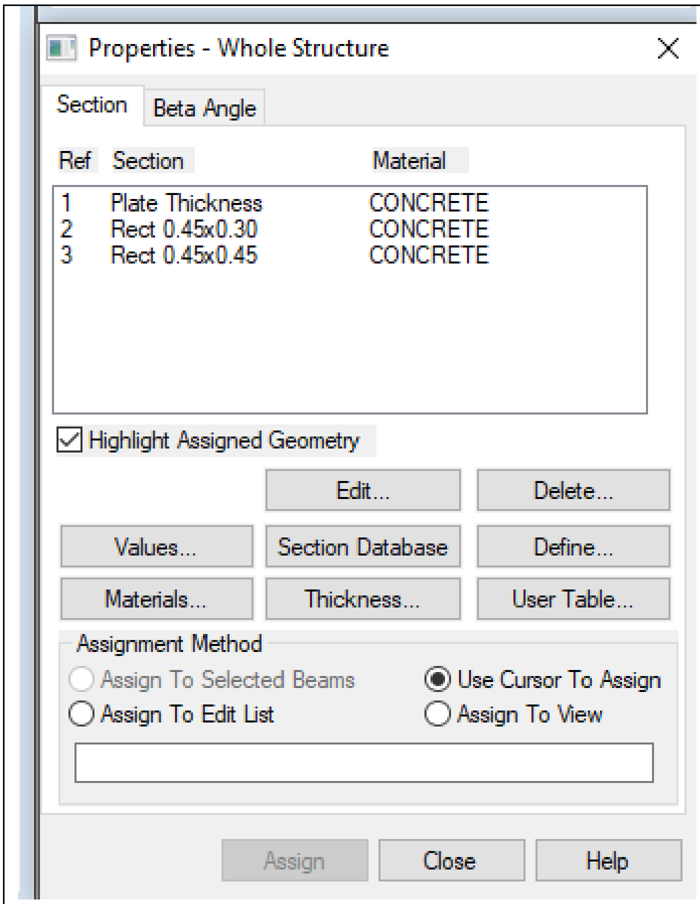
2a. MODEL TESTING FOR FRAMES

Aim:

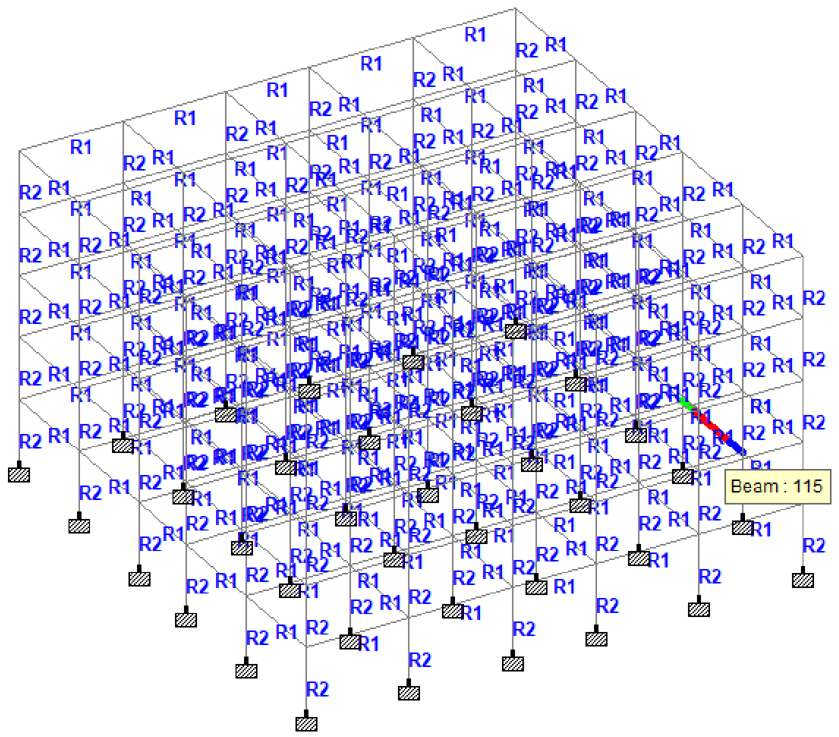
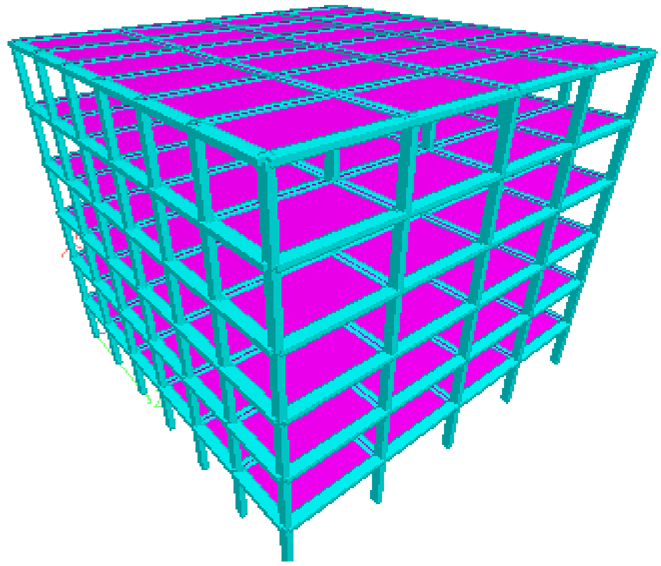
To test the frames of models of the given structure using STAAD pro

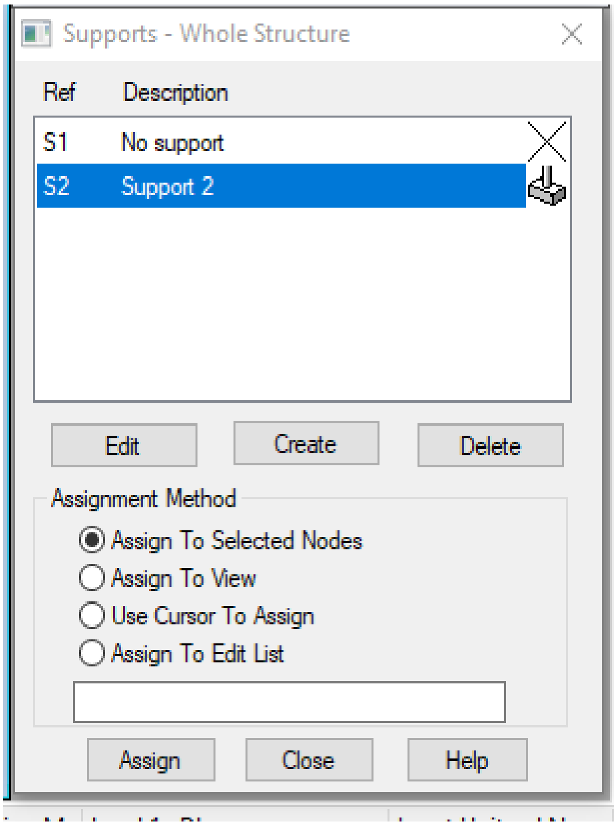
Software Used:

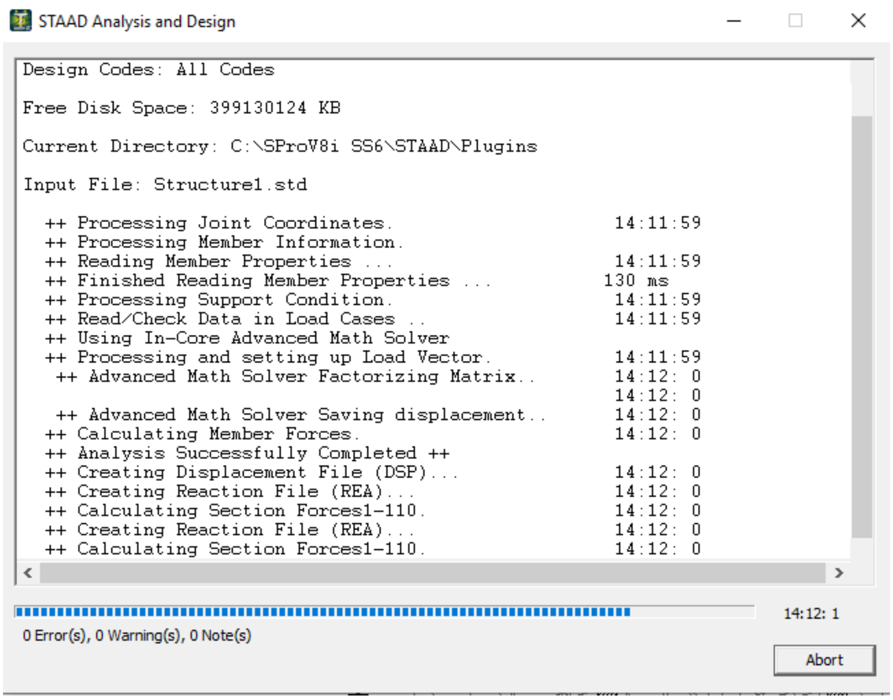
STAAD Pro



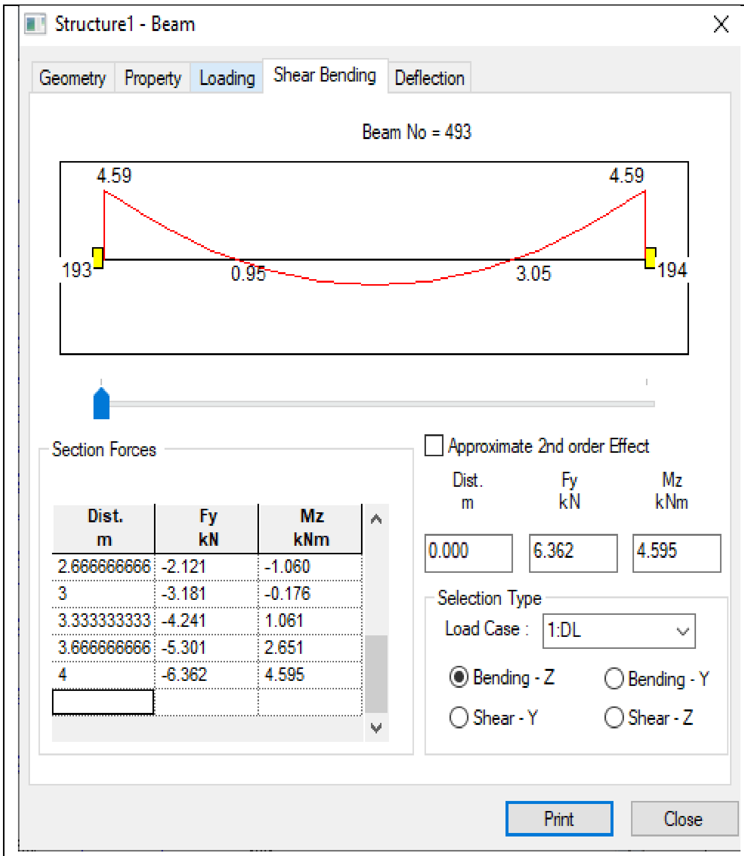
Properties for the frame

	Assigning the member property
	Building with plates

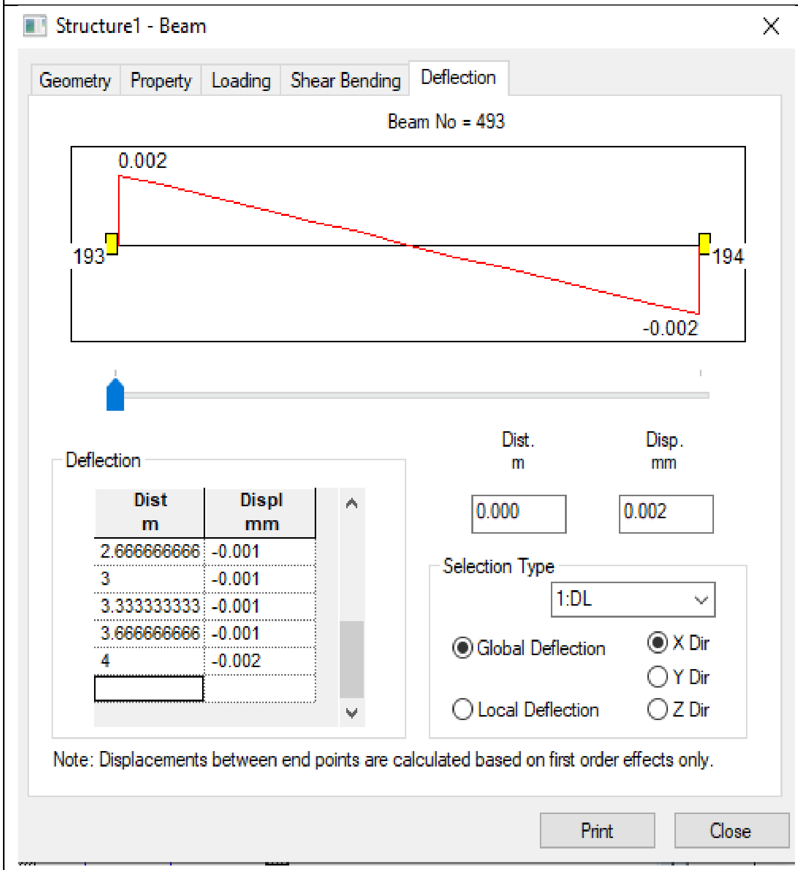
<table>
<tr>
<td>


</td>
<td>Creating support</td>
</tr>
<tr>
<td>


</td>
<td>STAAD analysis</td>
</tr>
</table>



Shear bending of the member



Deflection of the member

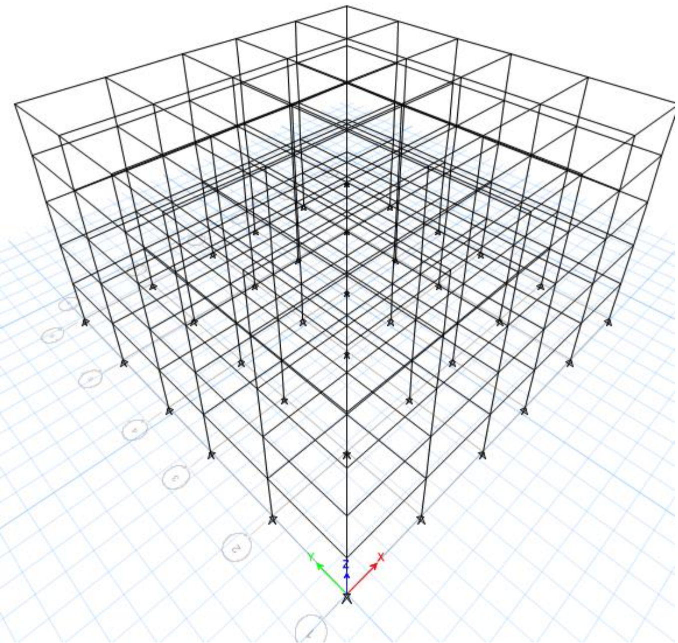
2b. Model testing for frames using ETABS

Aim:

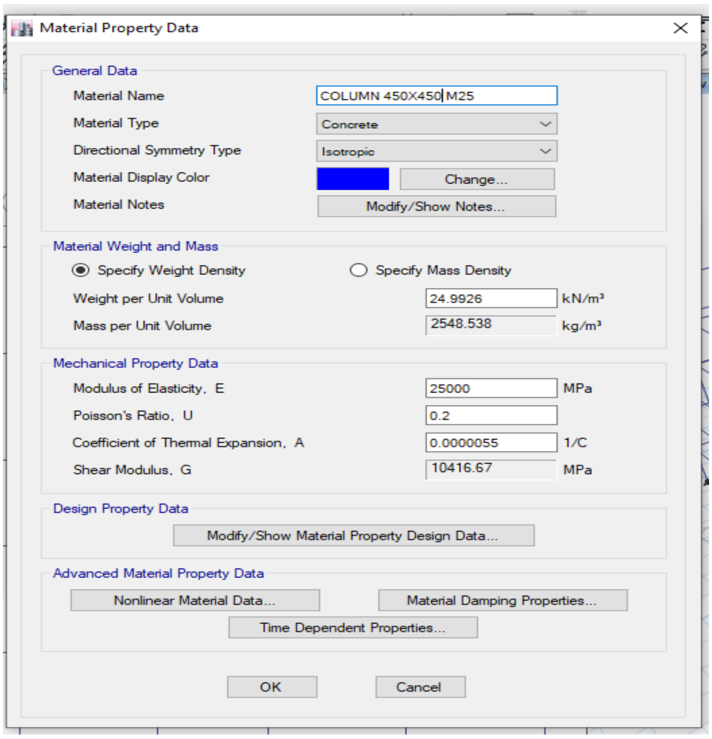
To model the given structure using etabs

Software used:

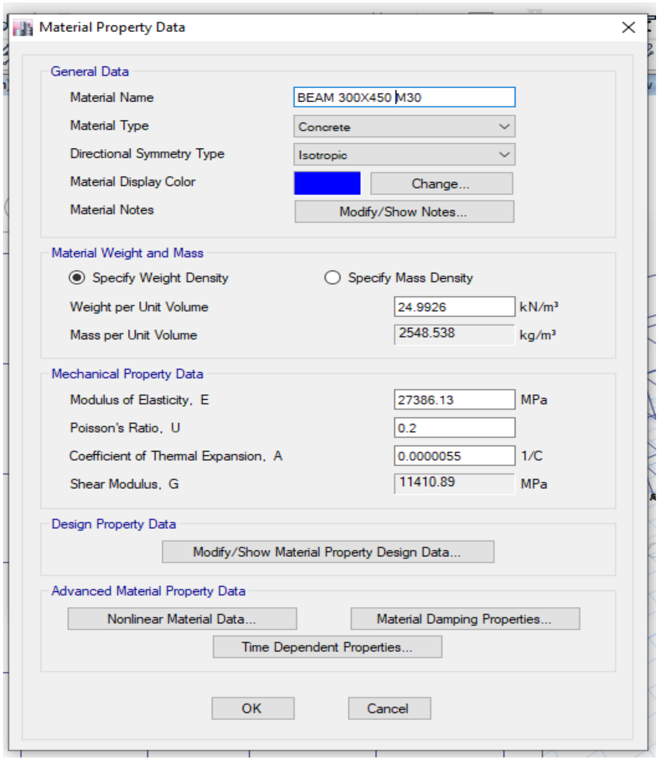
Etabs



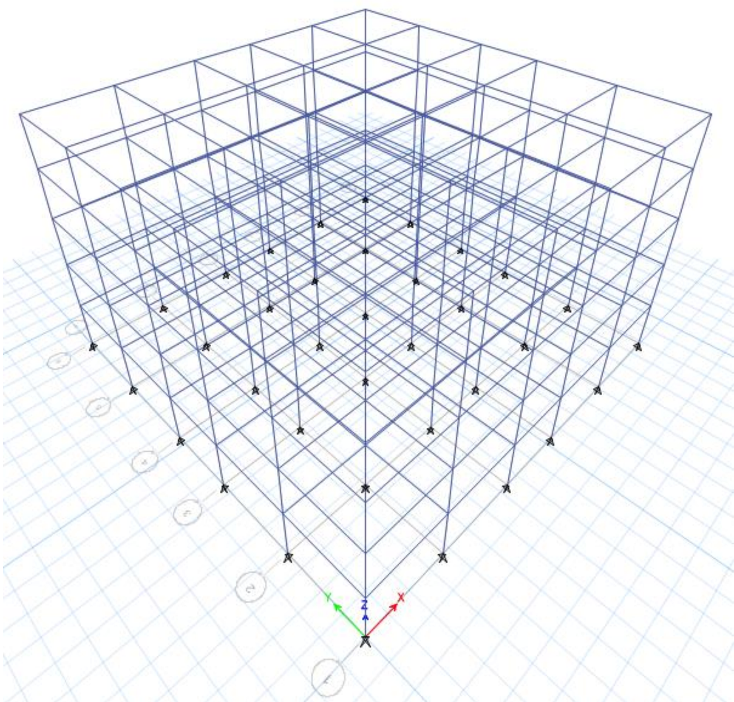
Model of the plan



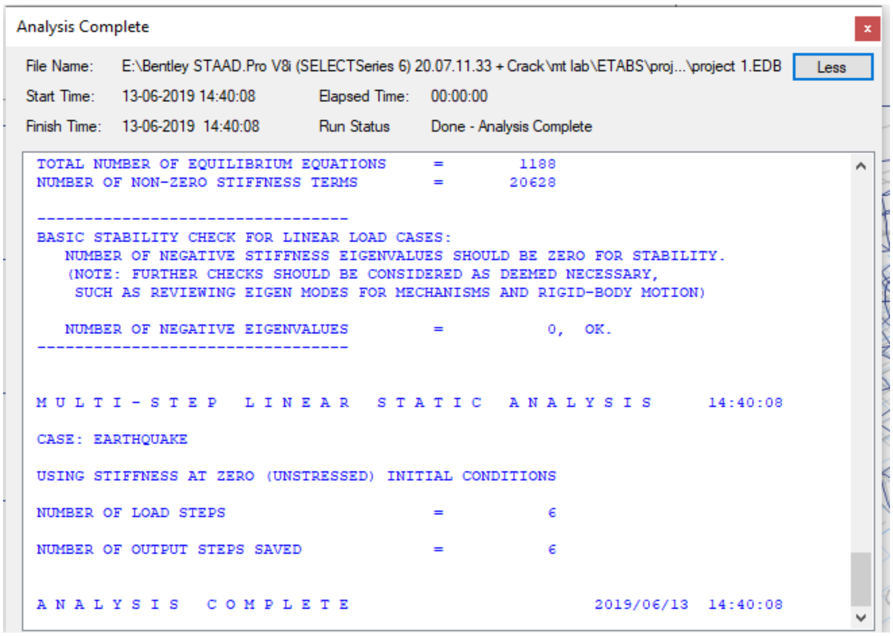
Column property



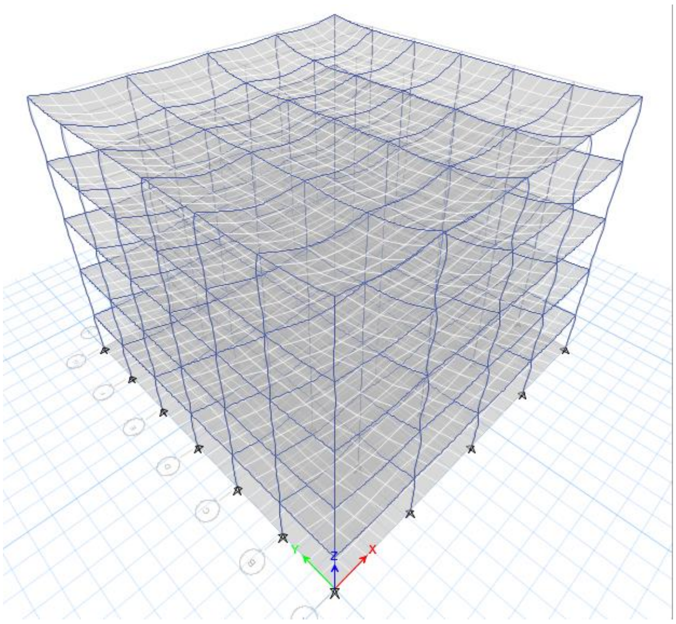
Beam properties



After assigning the loads



Analysis of the building



Deflection

After loadings

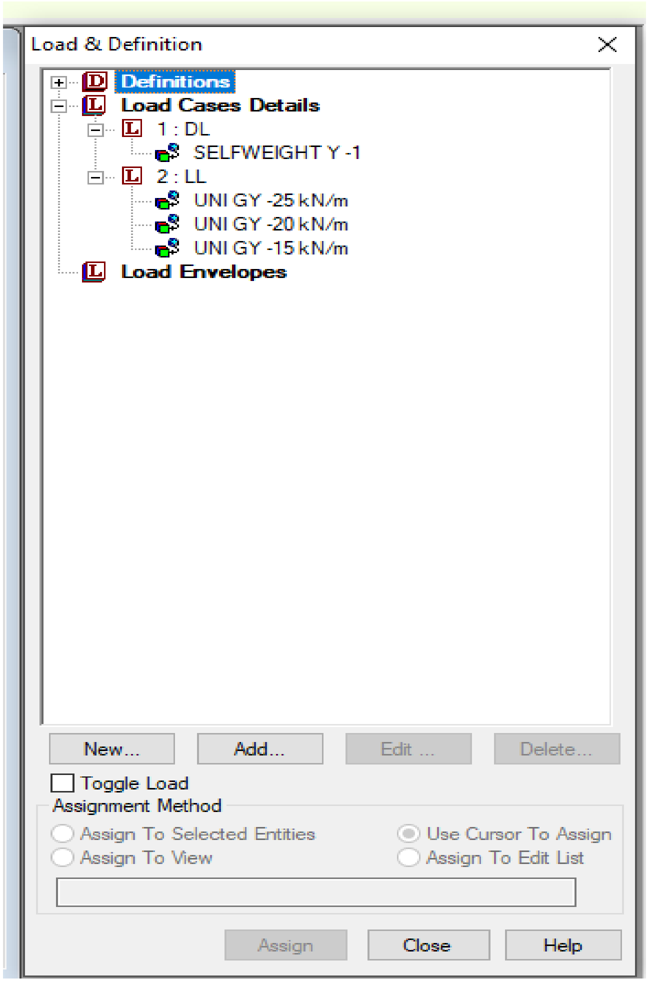
3A. MODEL TESTING OF FRAMES UNDER STATIC LOADING

Aim:

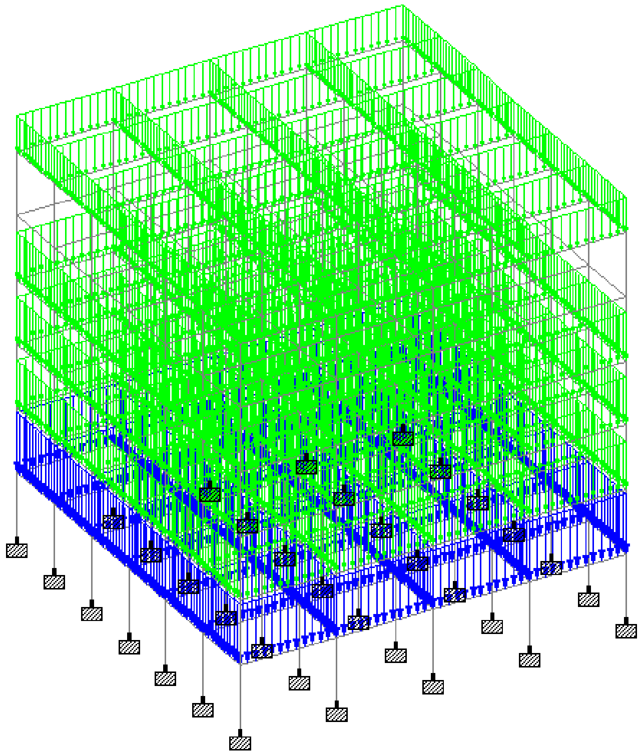
To test the frames of models of the given structure under static loading using STAAD pro

Software Used:

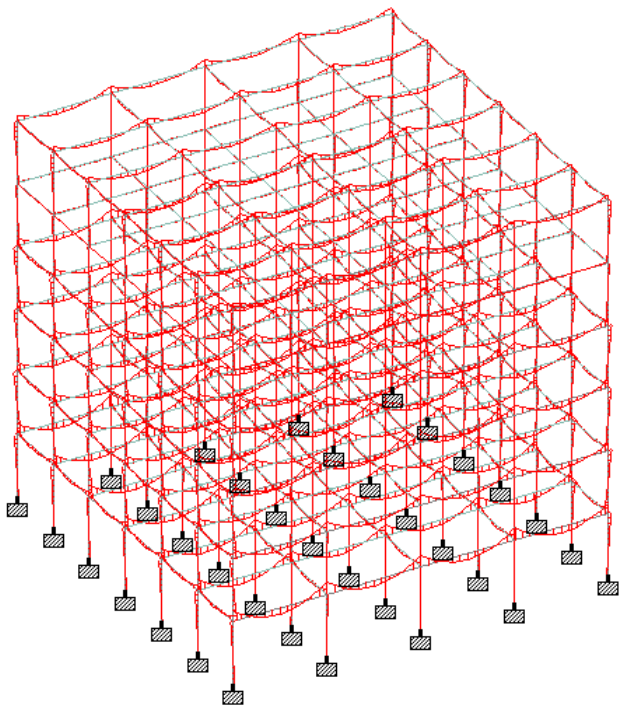
STAAD Pro



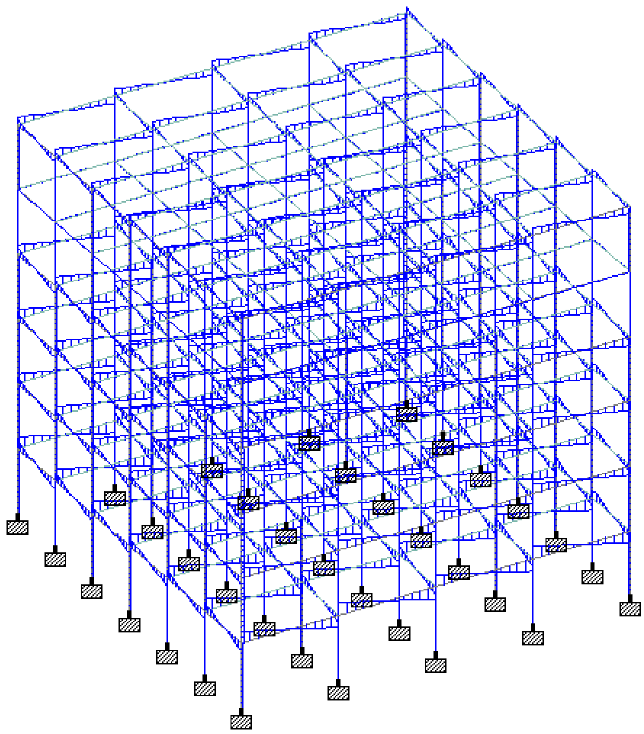
Loads and Defination



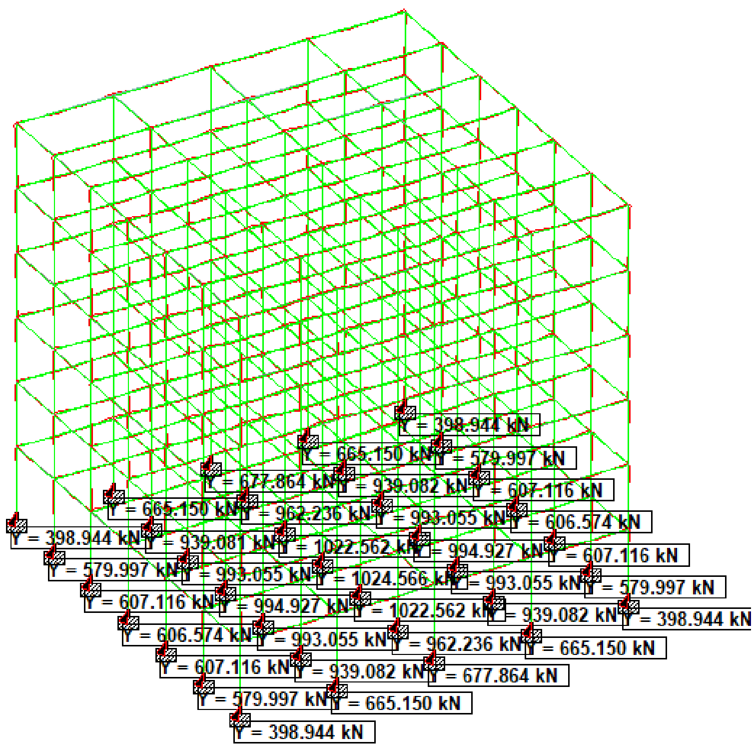
Wall loadings



Bending moment in the members of the building



Shear force in the members of the building



Support reactions

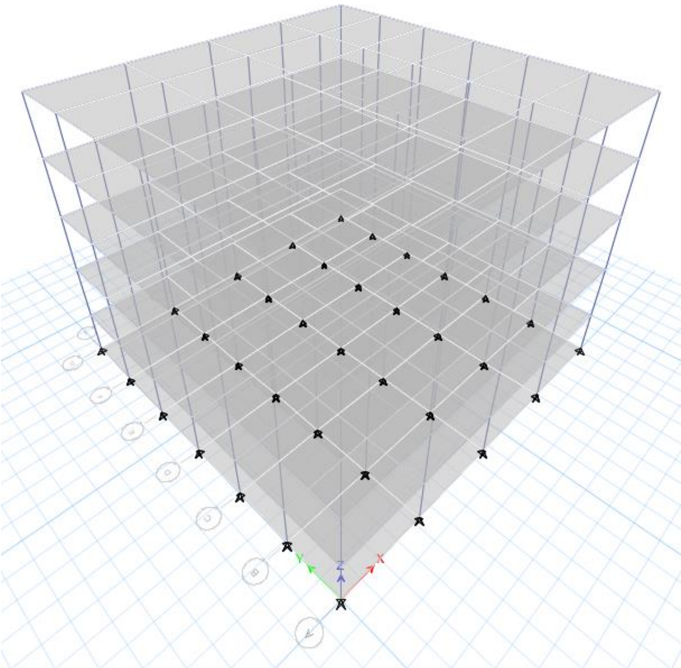
3b. Model testing on frames under static loading

Aim:

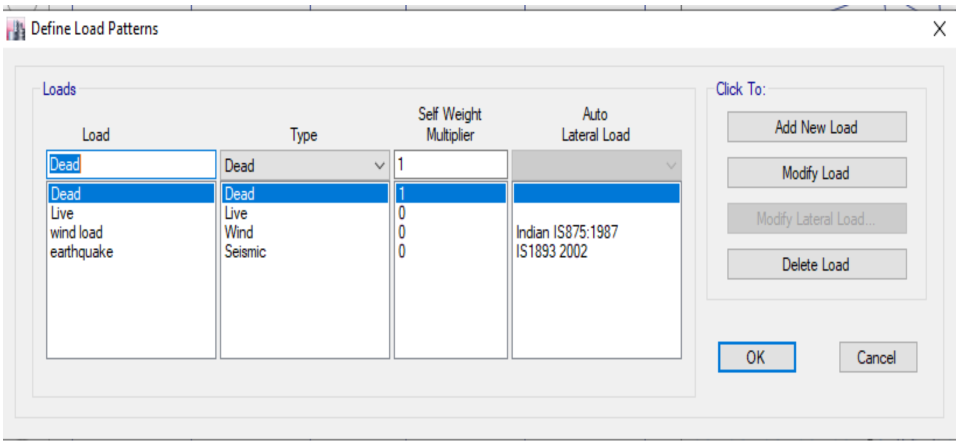
To model the frames under static loading for given structure using etabs

Software used:

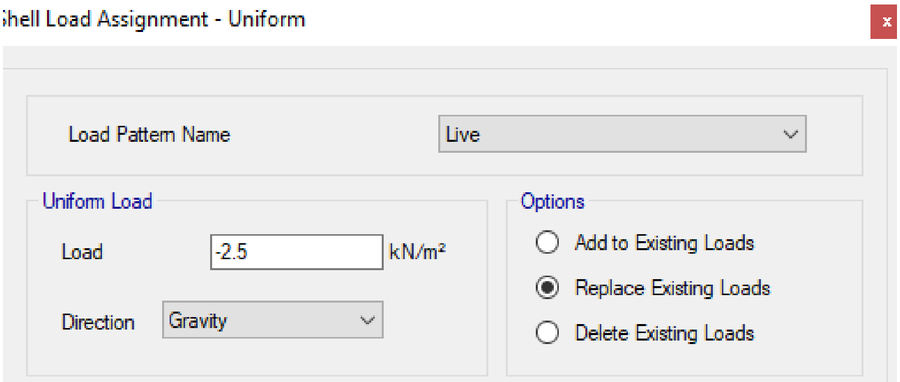
Etabs



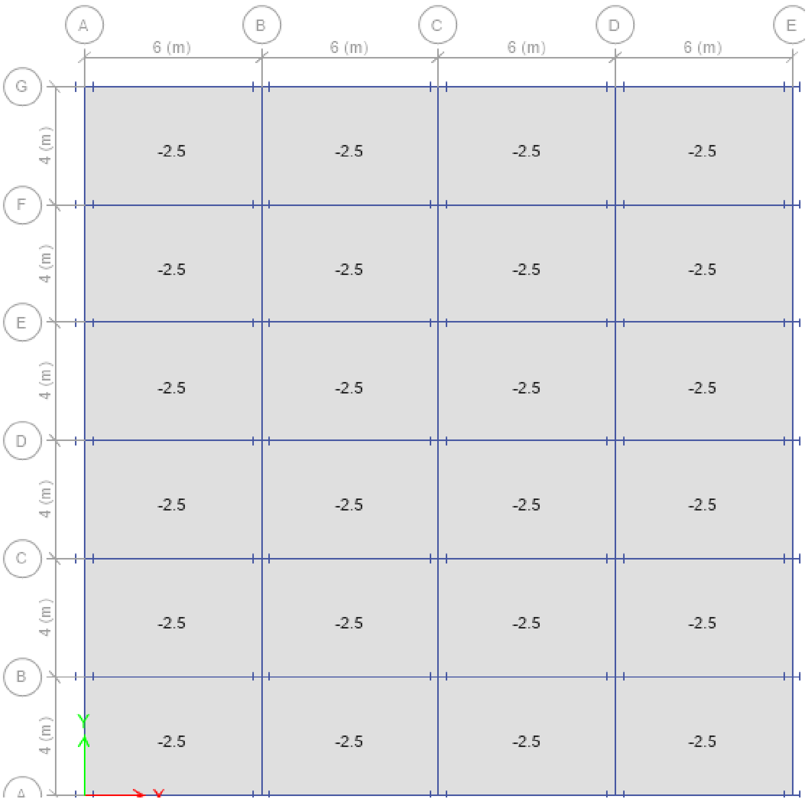
Plan of the model



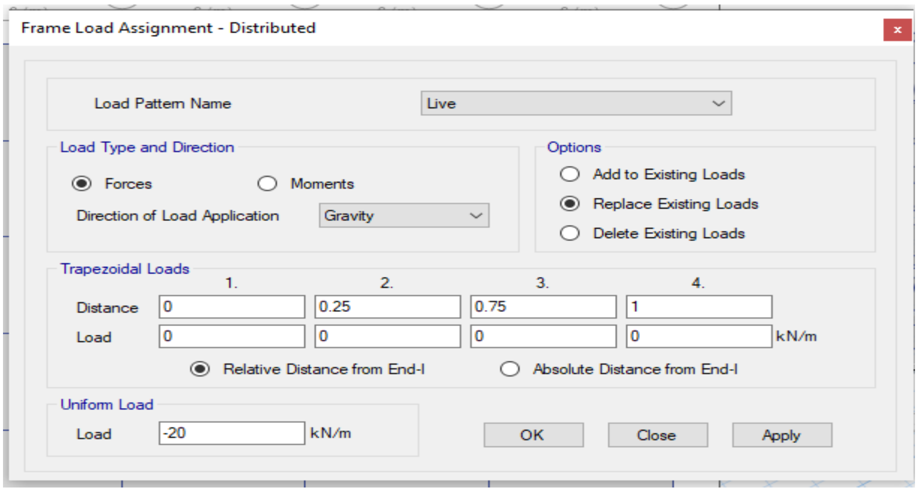
Defining load patterns



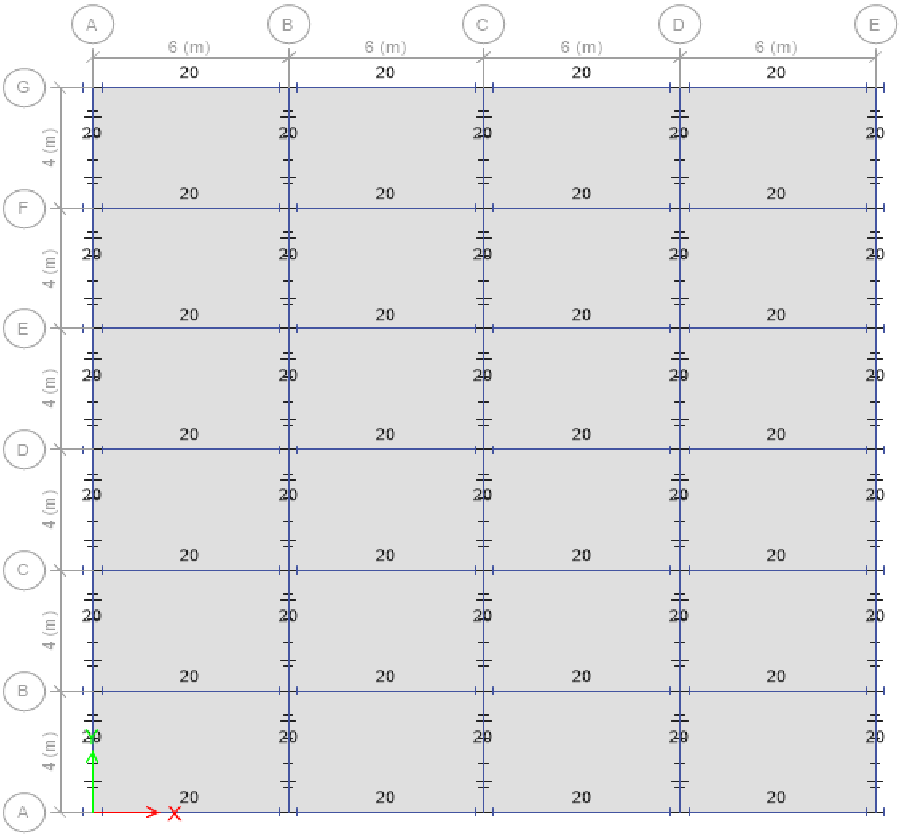
Shell loading assignment



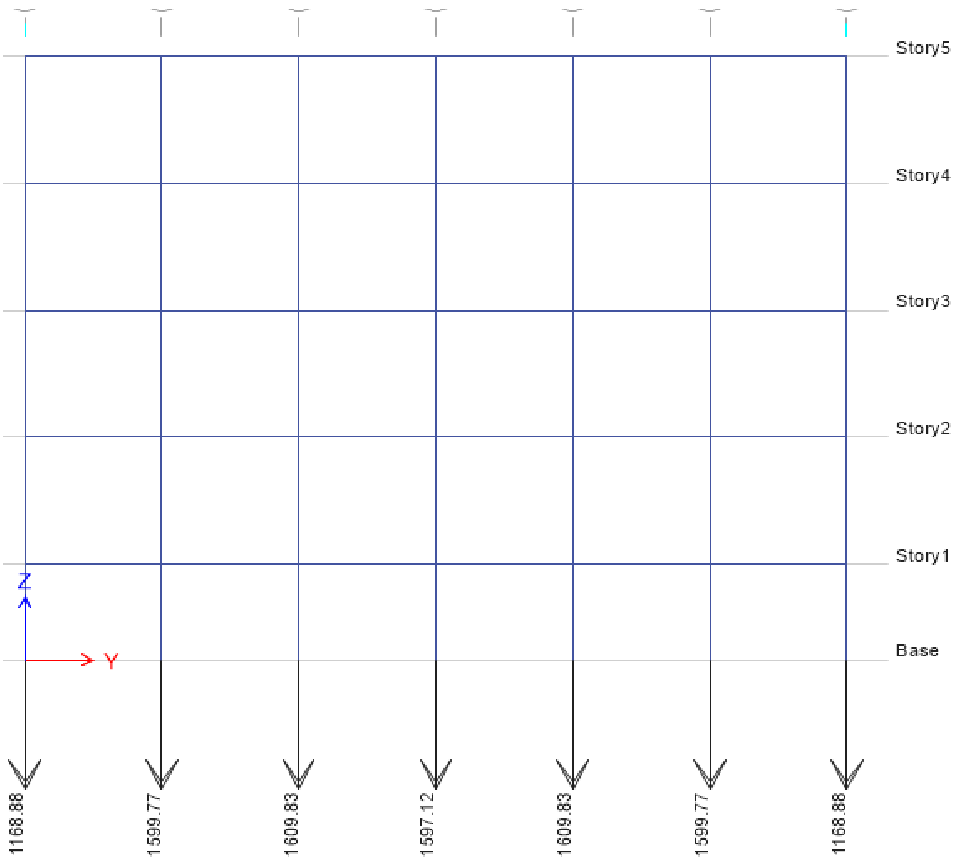
Assigning slab loads



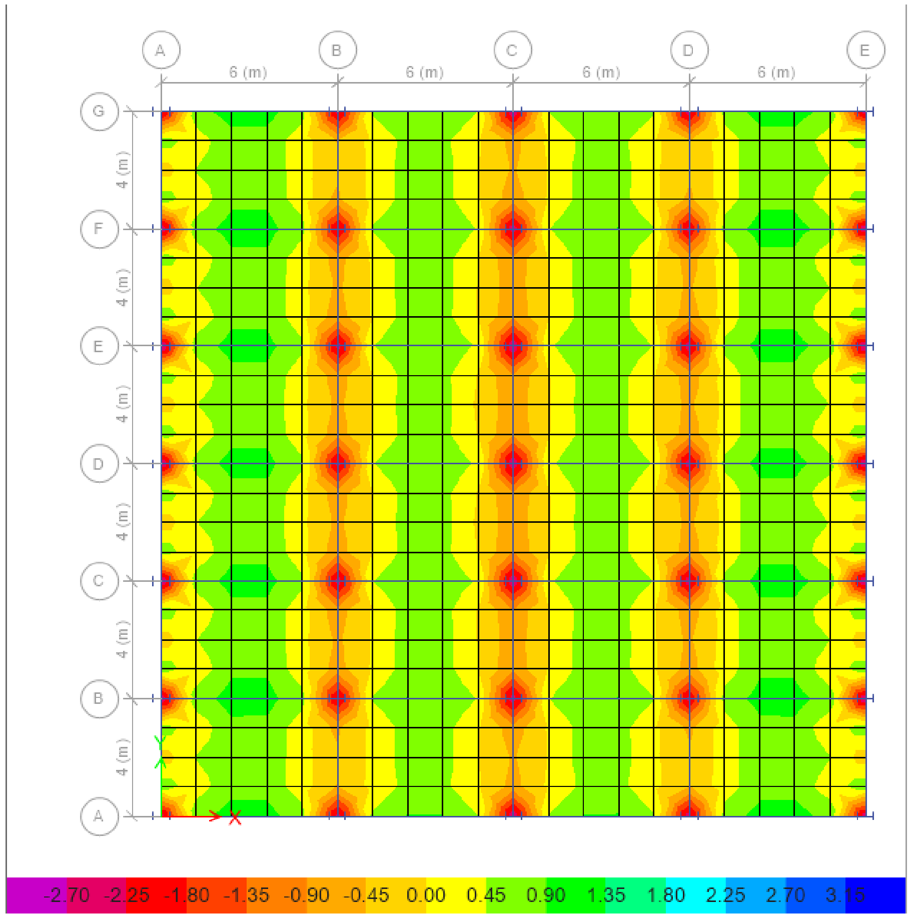
Loadings on frame



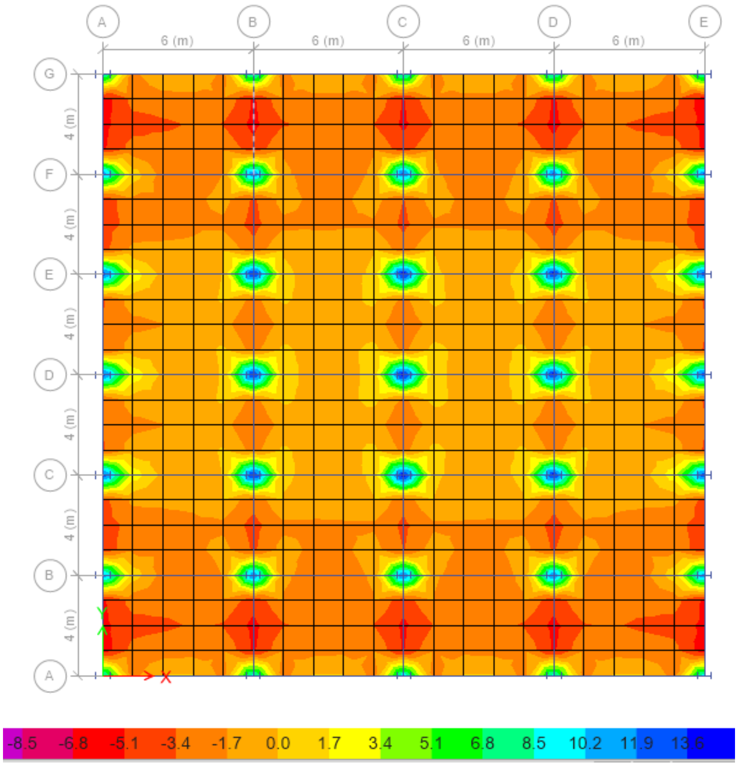
Beam loads



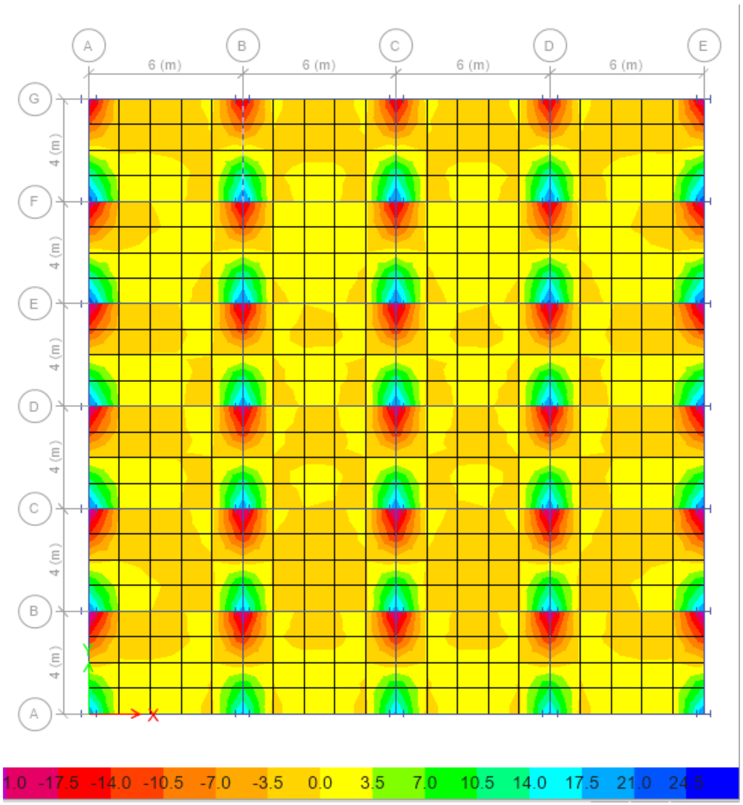
Support reaction along a section



Bending stress on the floor



Bending moments on the floor



Shear force on the floor

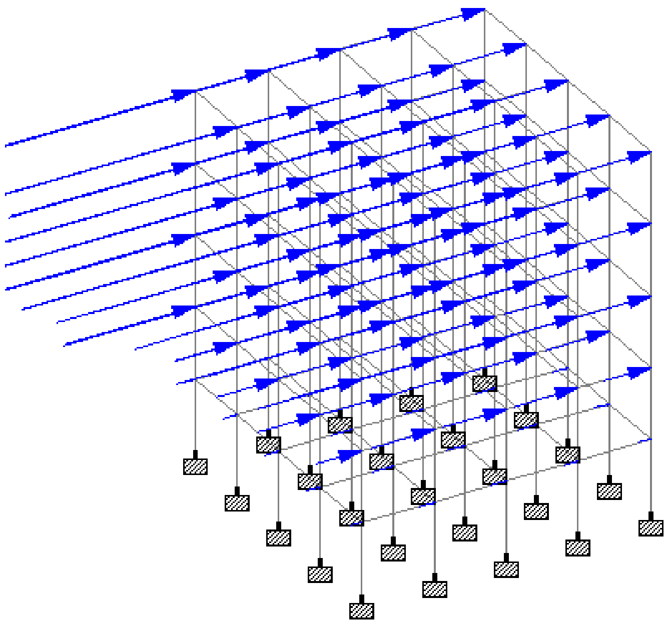
4A. MODEL TESTING OF FRAMES UNDER DYNAMIC LOADING

Aim:

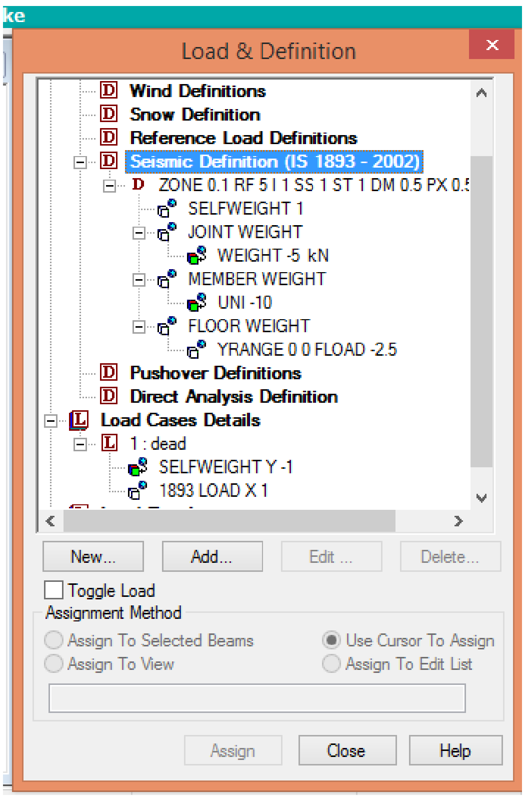
To test the frames of model of the given structure under dynamic loading using STAAD pro

Software Used:

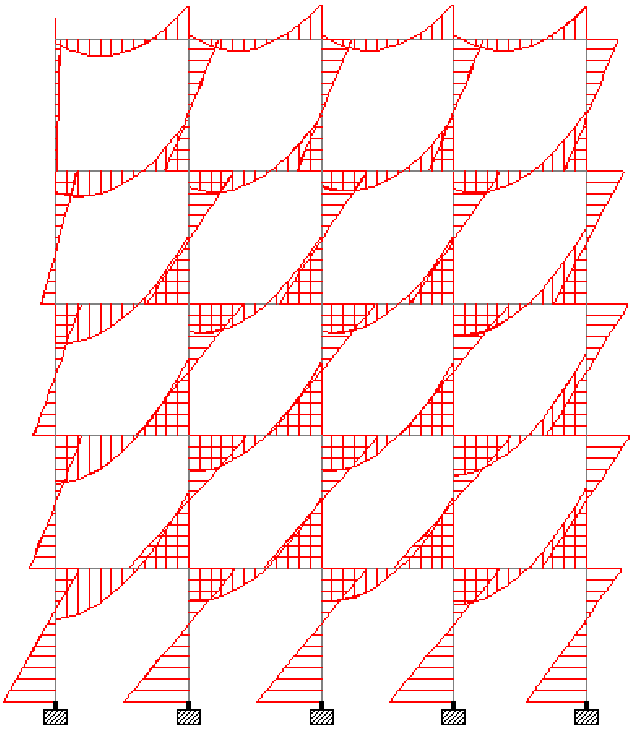
STAAD Pro



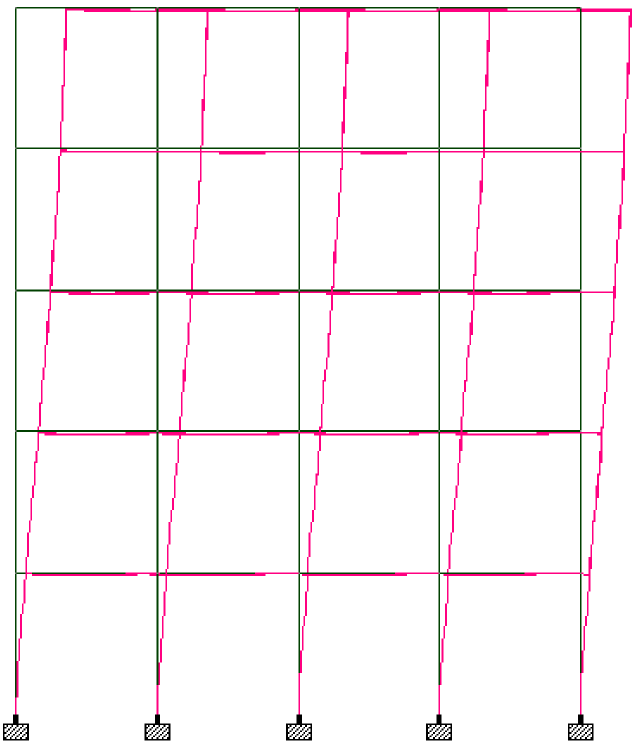
Seismic loading



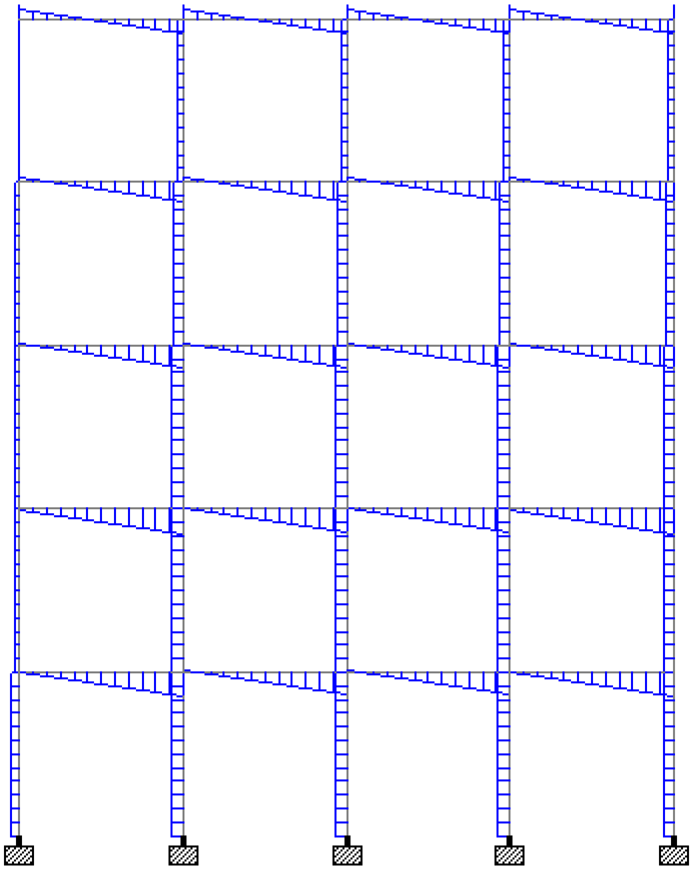
Seismic definitions



Bending moment in the members of the building



Deflection of the members



Shear forceof
the members

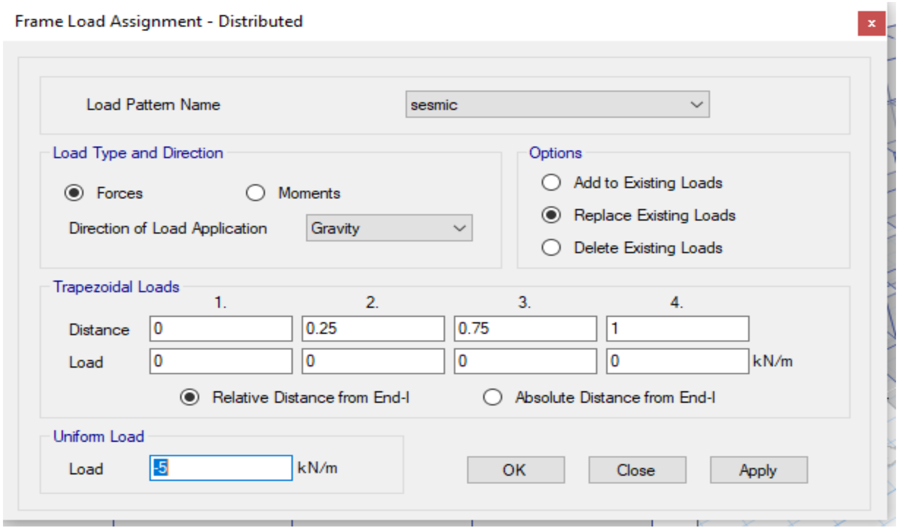
4b. Model testing of frame under dynamic loading

Aim:

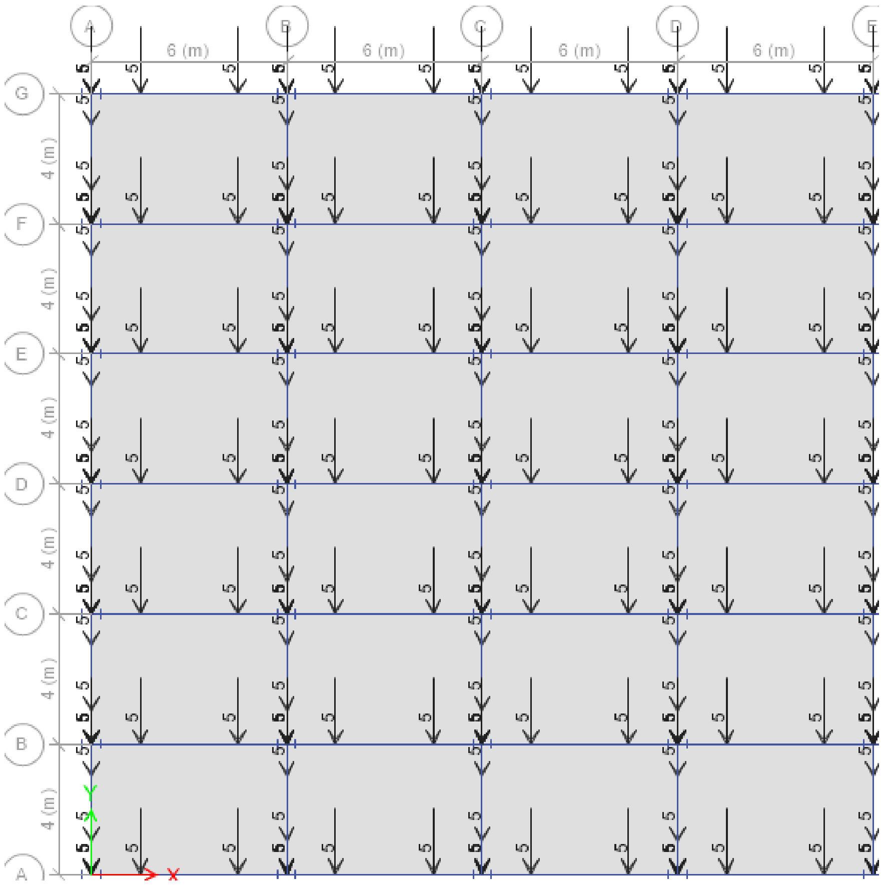
To model the frame under dynamic loading for given structure using etabs

Software used:

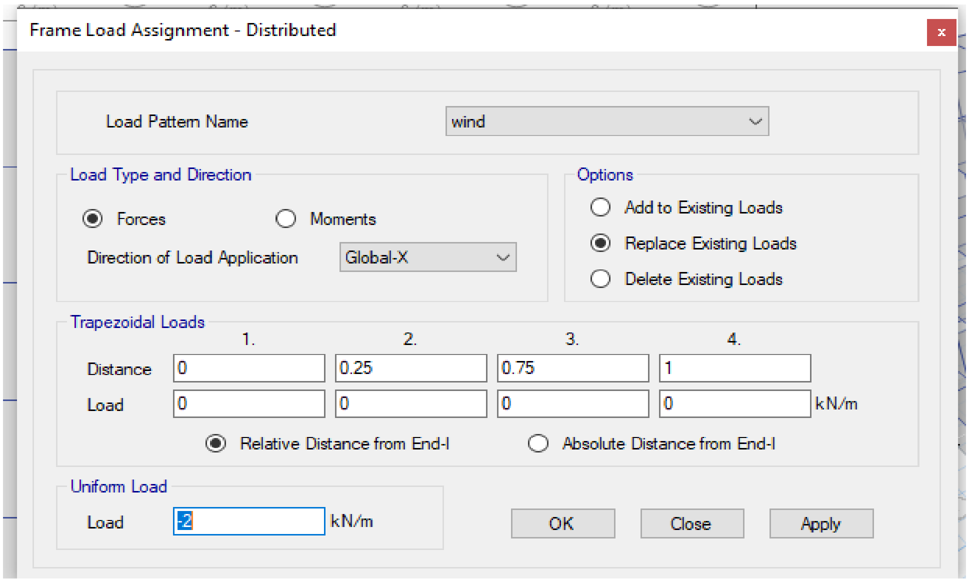
Etabs



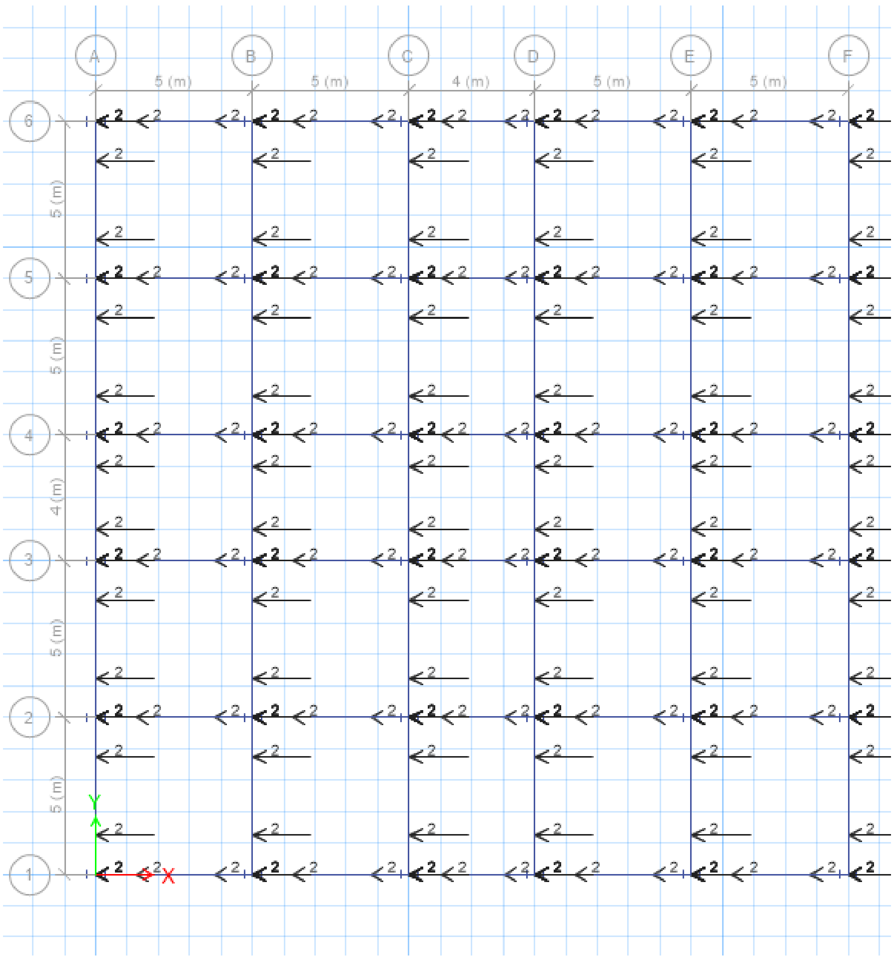
Loadings



Assigning of seismic load



Loading on frame



Assigning of wind load

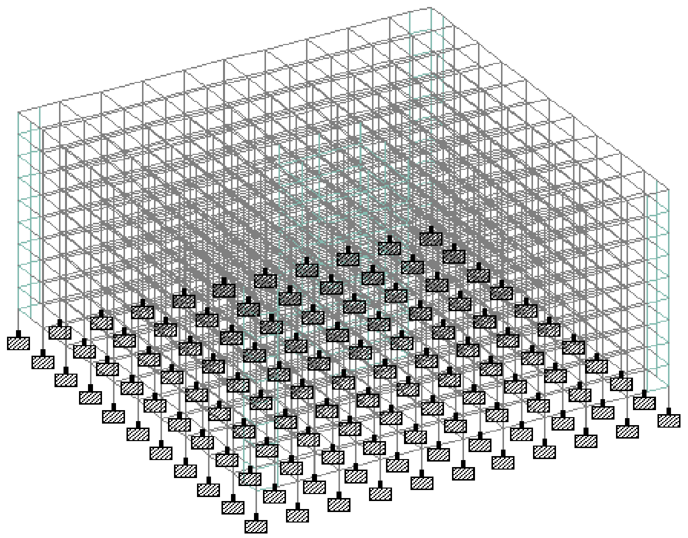
5A. MODEL TESTING OF SHEAR WALL UNDER LATERAL LOADS

Aim:

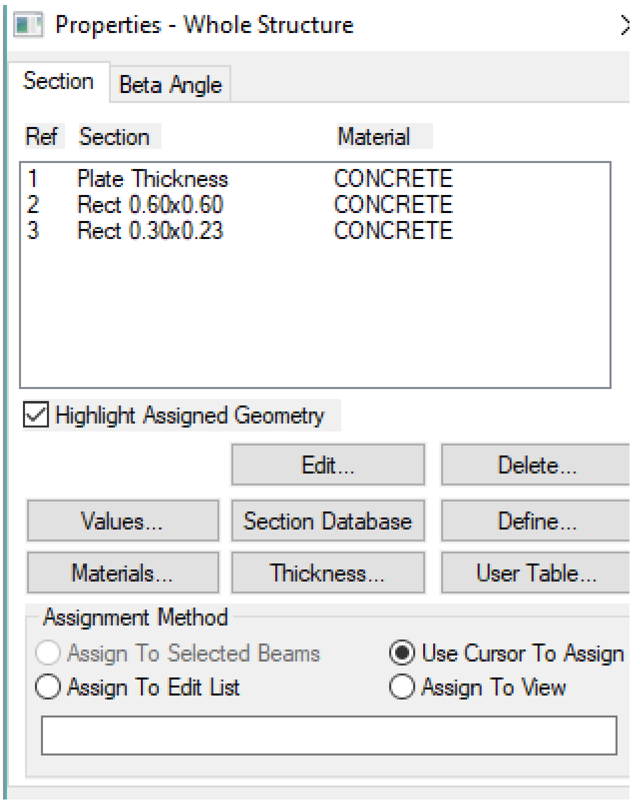
To test the model of shear wall under lateral loads using STAAD pro

Software Used:

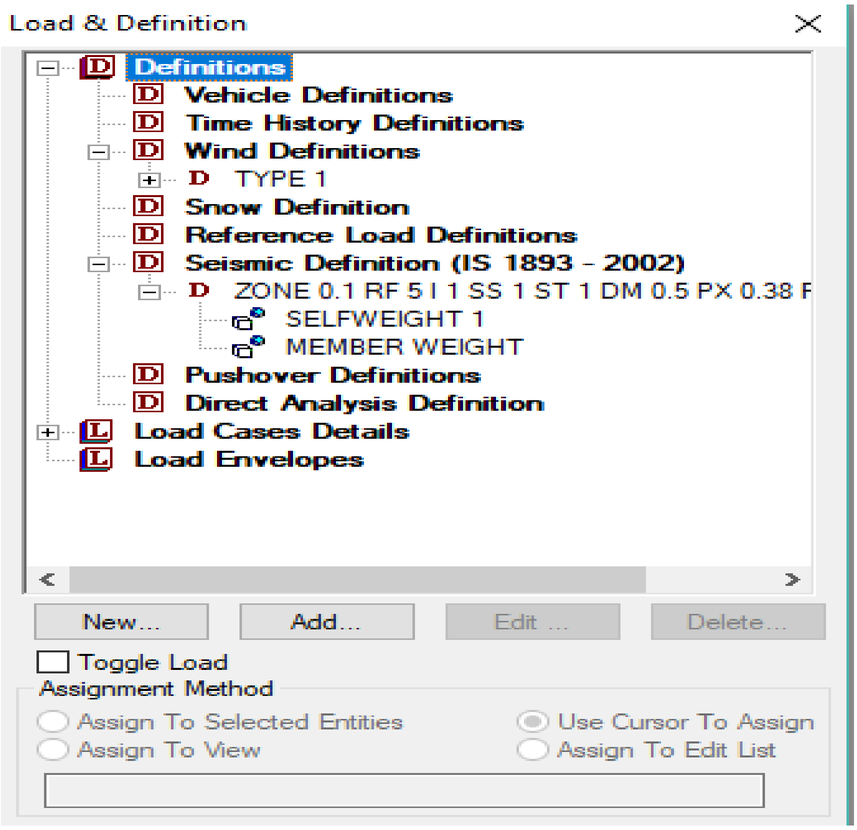
STAAD Pro



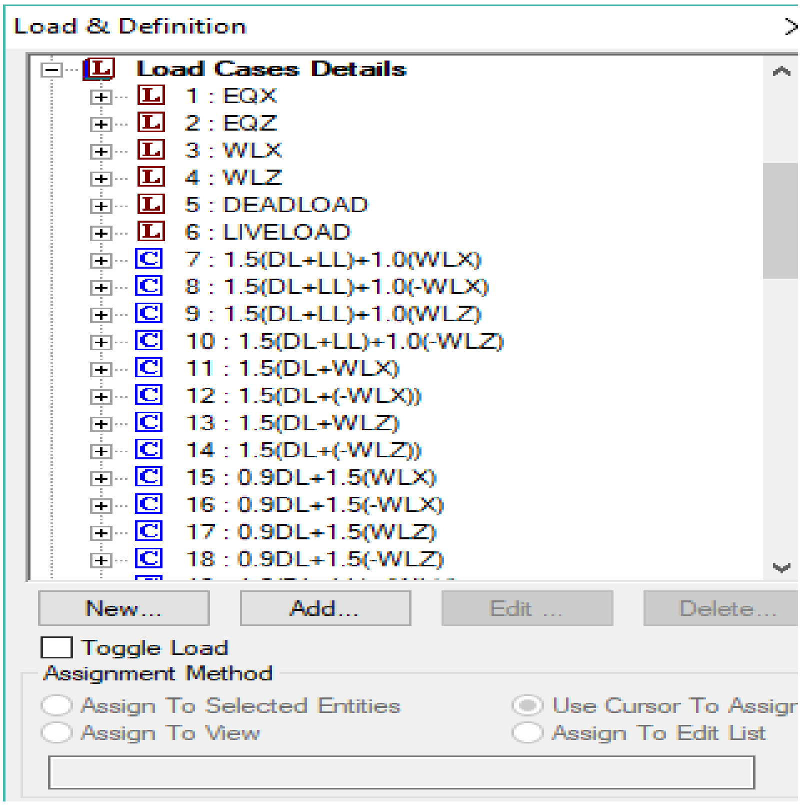
Model of the shear wall building



Member property of building

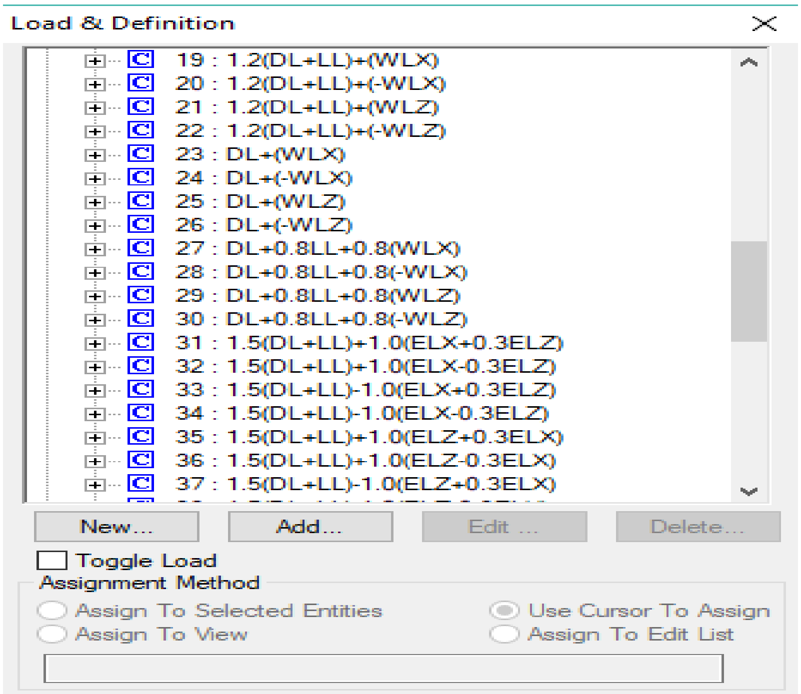


Winds and seismic
Load definitions



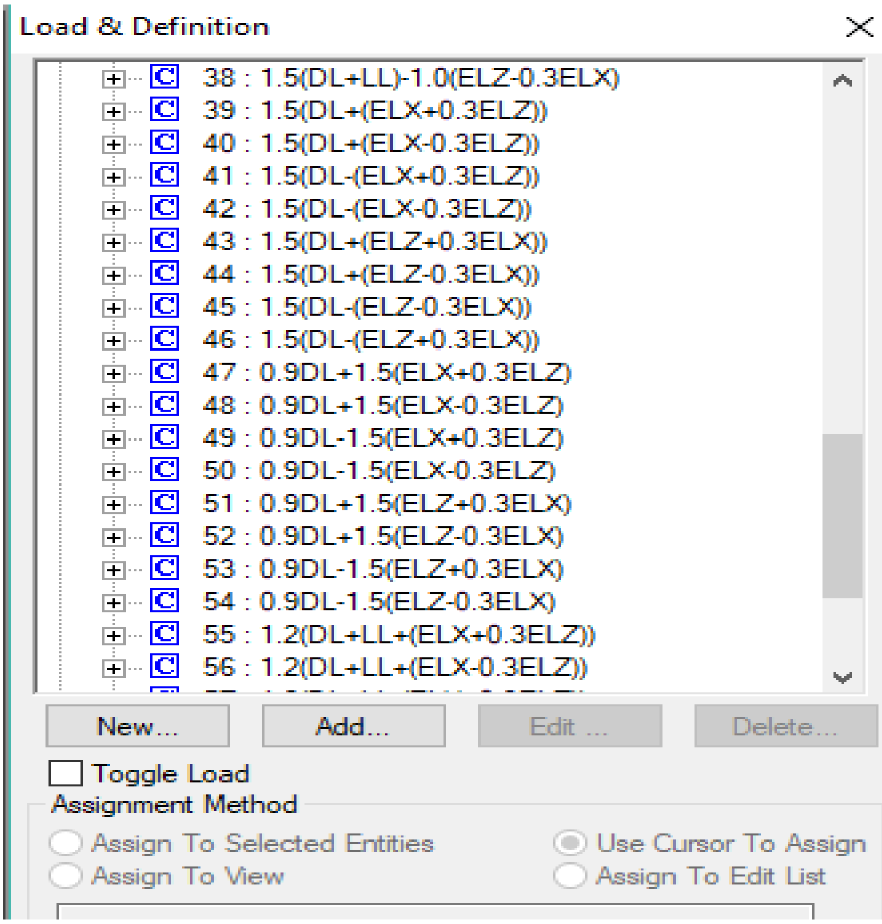
Loading data

Of earthquake loads, wind loads and load combinations



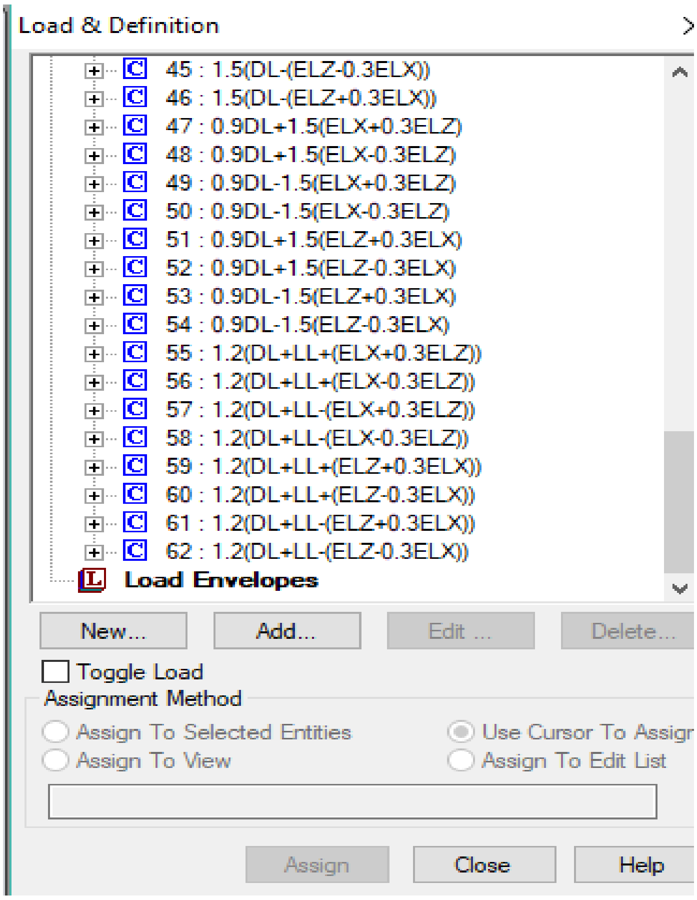
Loading data

Of loading combinations



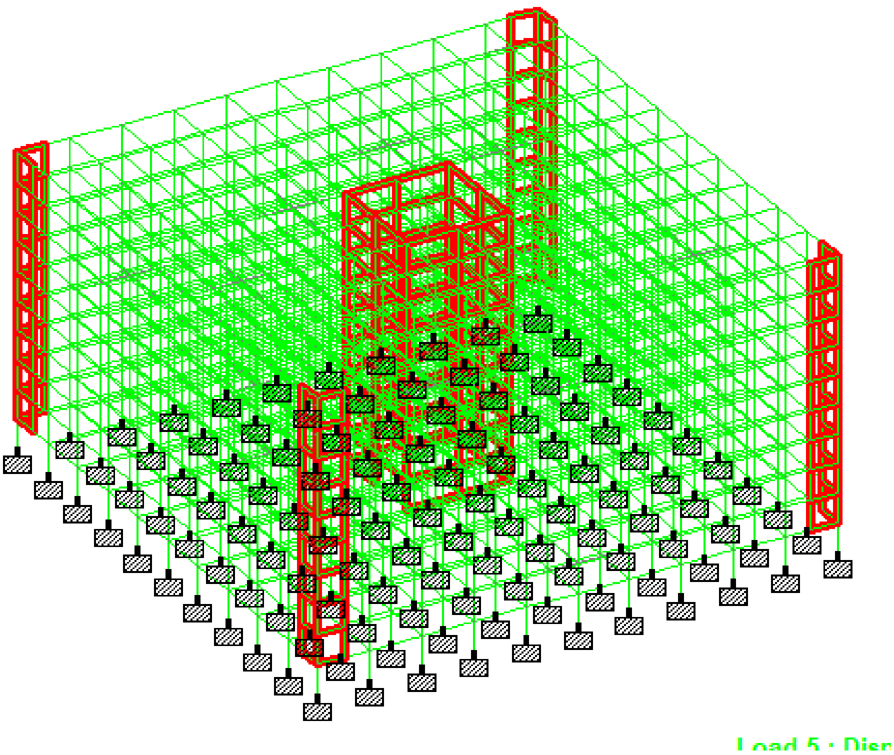
Loading data

Of loading combinations

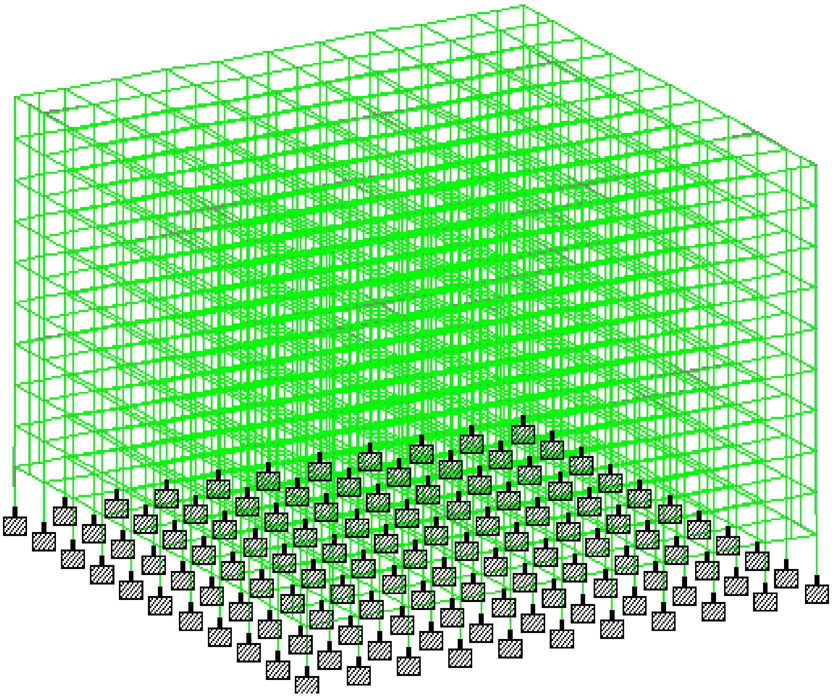


Loading data

Of loading combinations



Positioning of shear wall



Displacements obtained due to loading

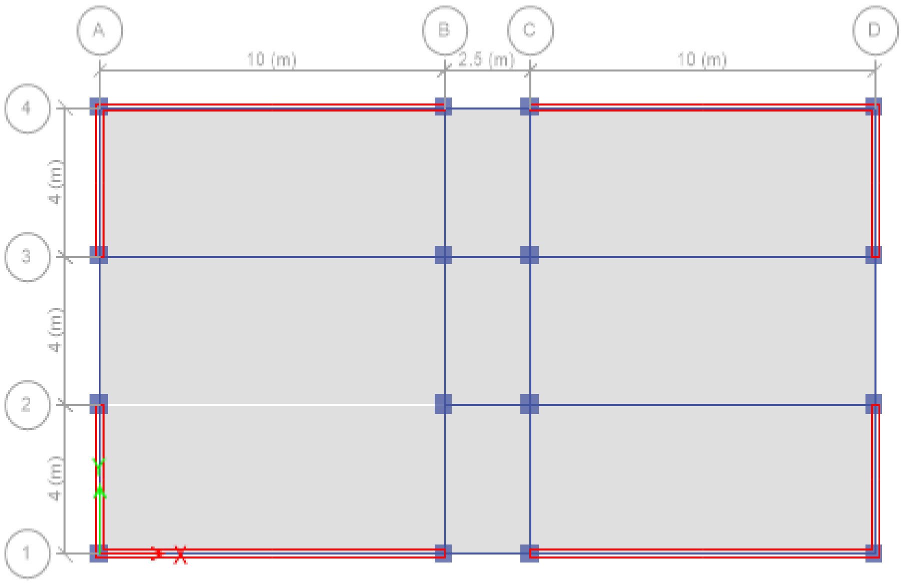
5b. Model testing of shear wall under lateral loads

Aim:

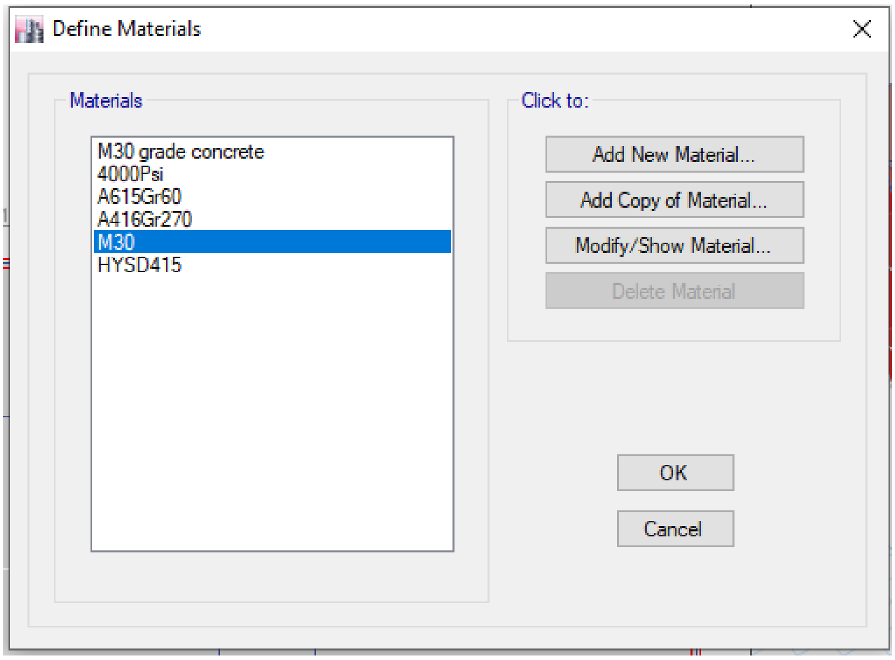
To model shear wall under lateral loads for given structure using etabs

Software used:

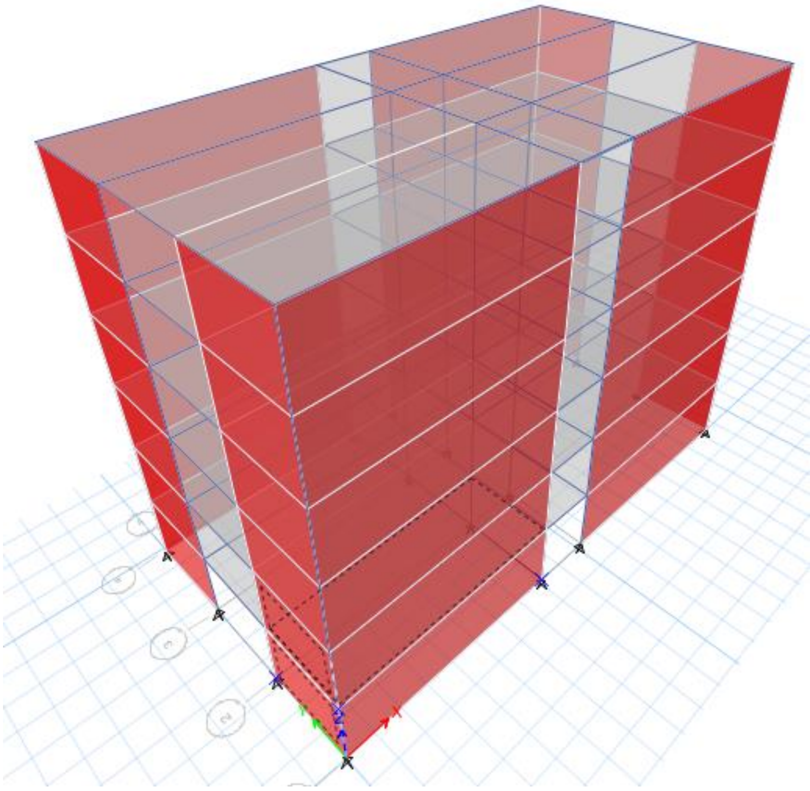
Etabs



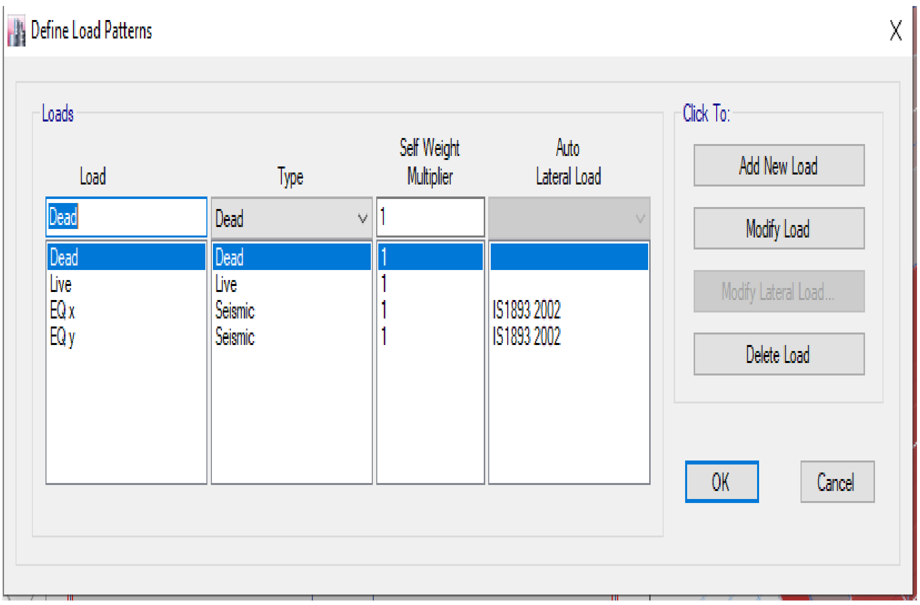
Plan of the given building



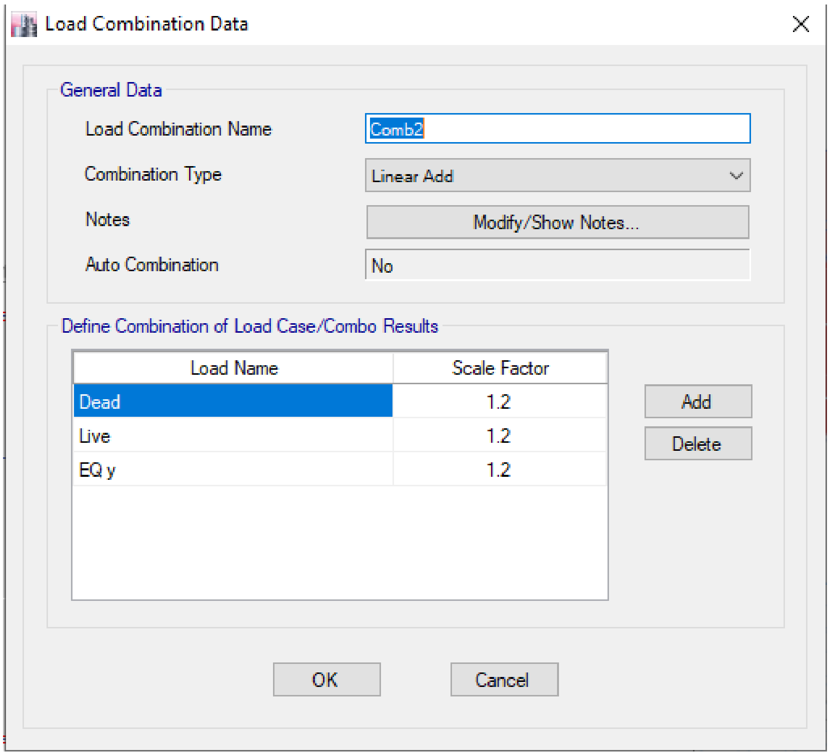
Defining material property



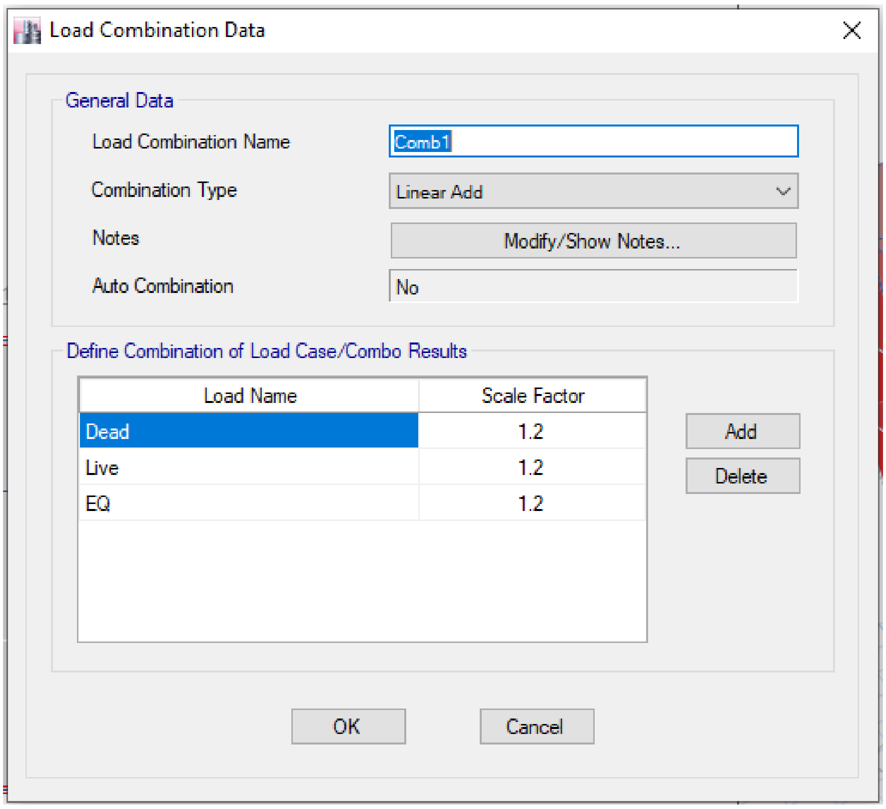
Shear walls on the model



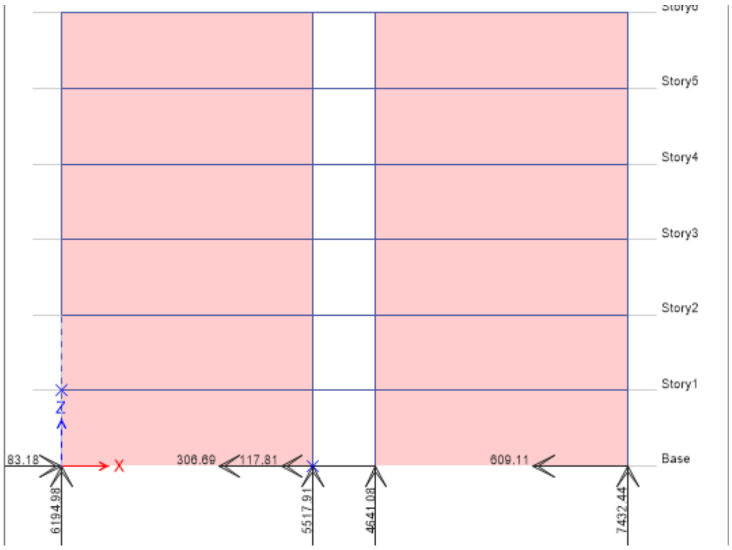
Load combinations



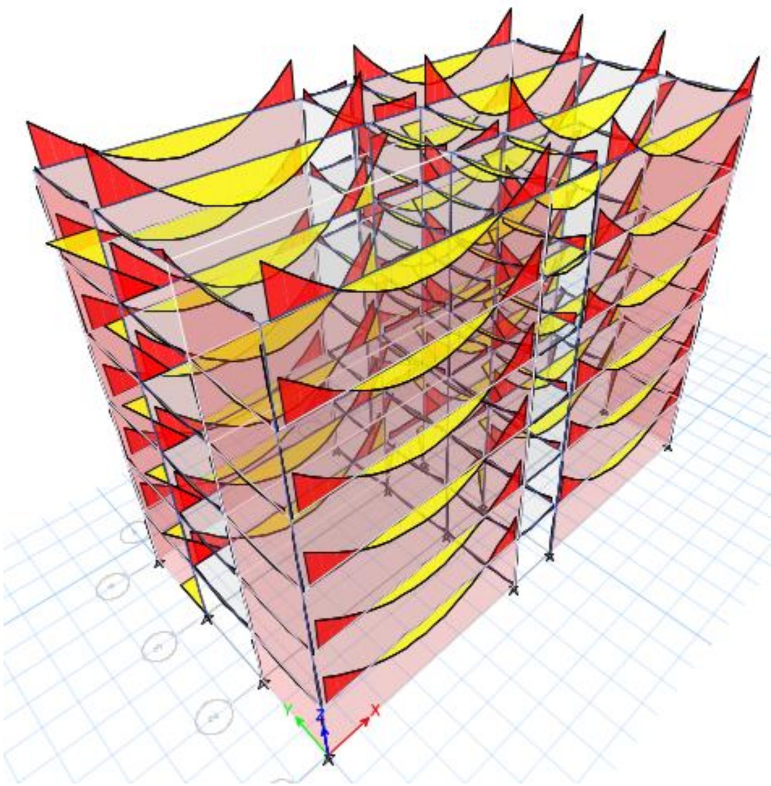
Load combinations



Load combinations

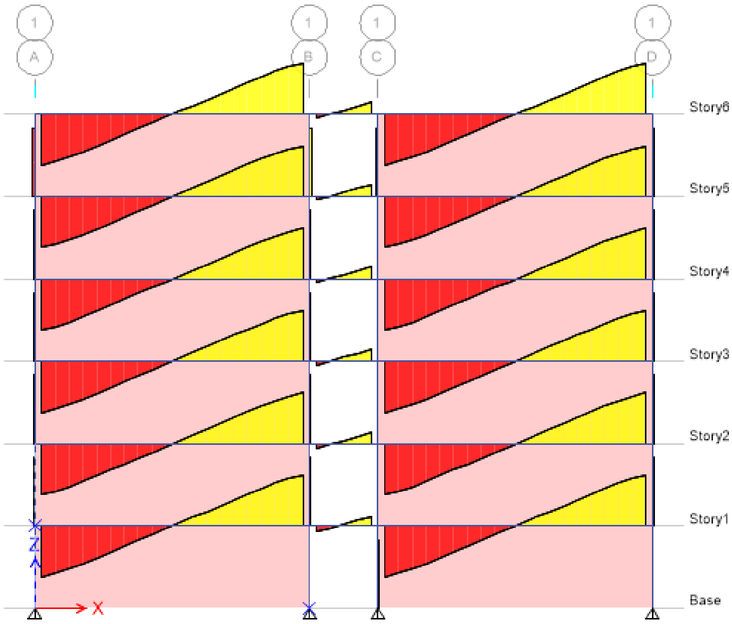


Reactions at supports



Reactions at supports

b. M.


1
A
1
B
1
C
1
D
Story6
Story5
Story4
Story3
Story2
Story1
Base
Z
X

Shear force

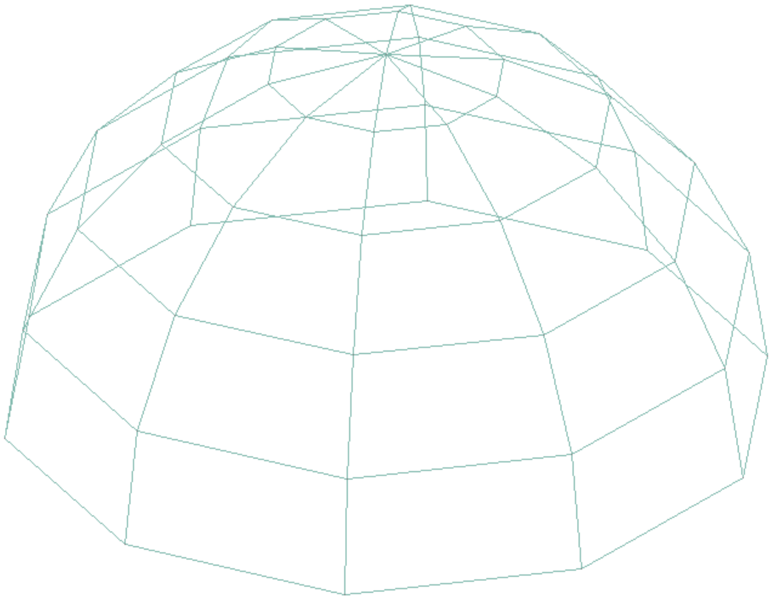
6A. MODEL TESTING OF SHELLS, PLATES UNDER STATIC LOADING.

Aim:

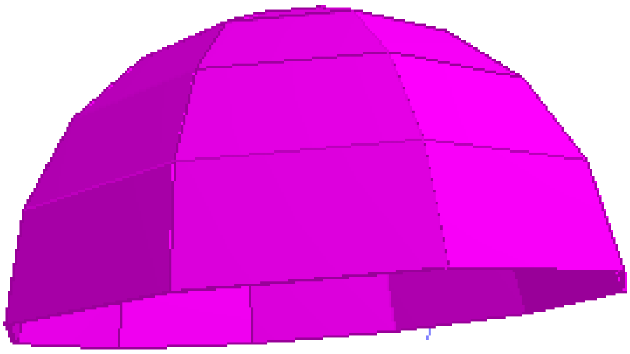
To test the model of shells , plates under static loading using STAAD pro

Software Used:

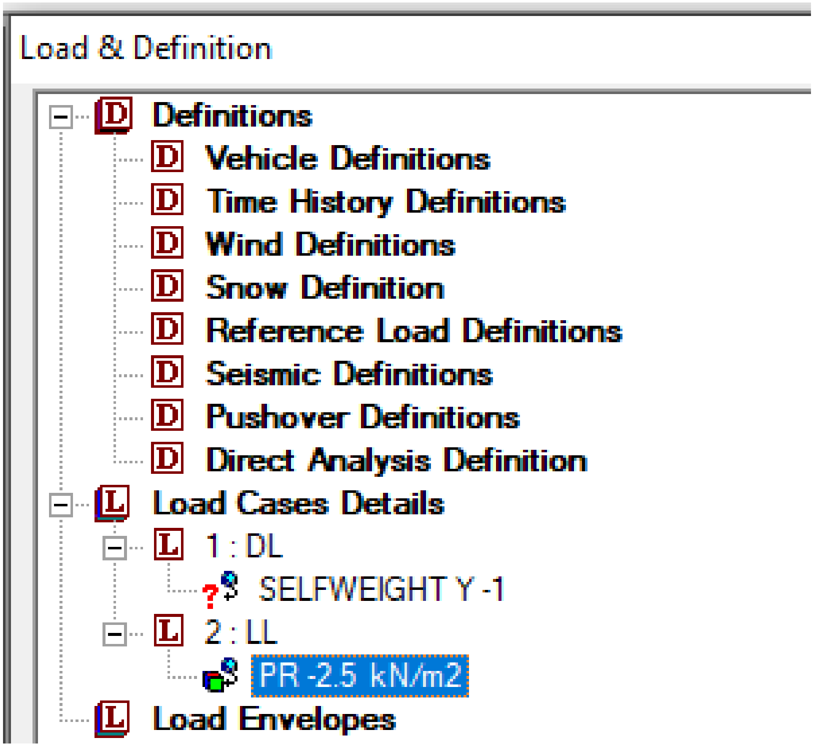
STAAD Pro



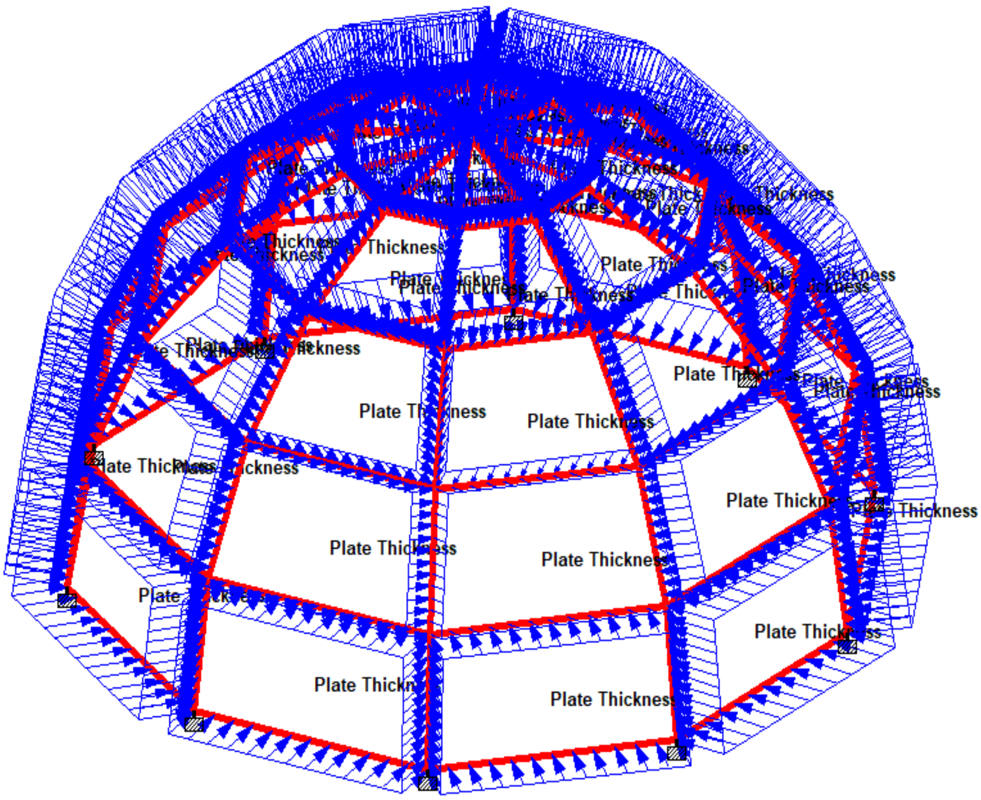
Model of a shell



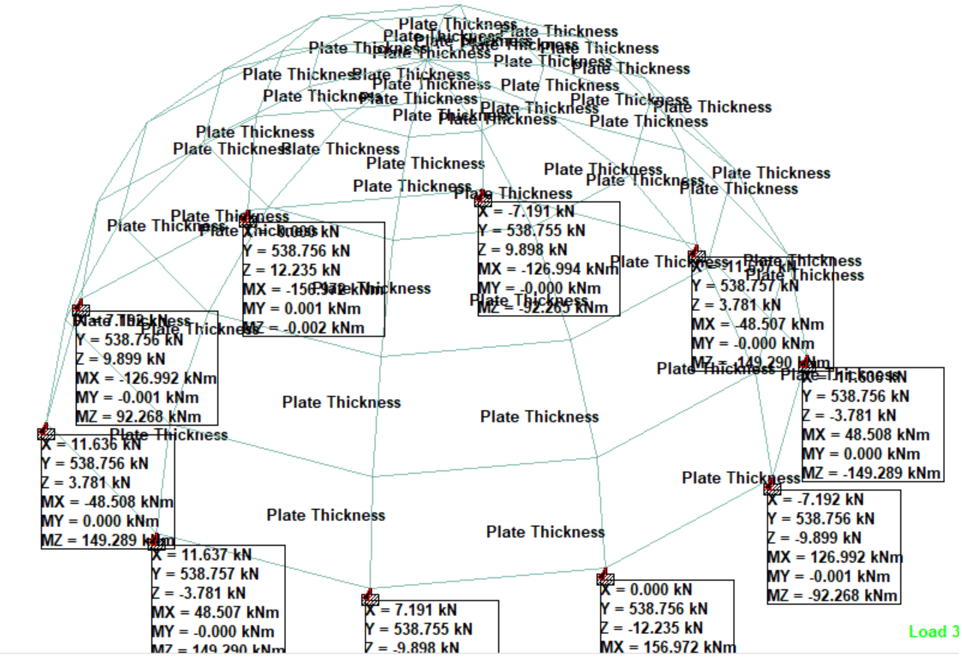
Modelling of the Shell with plate thickness



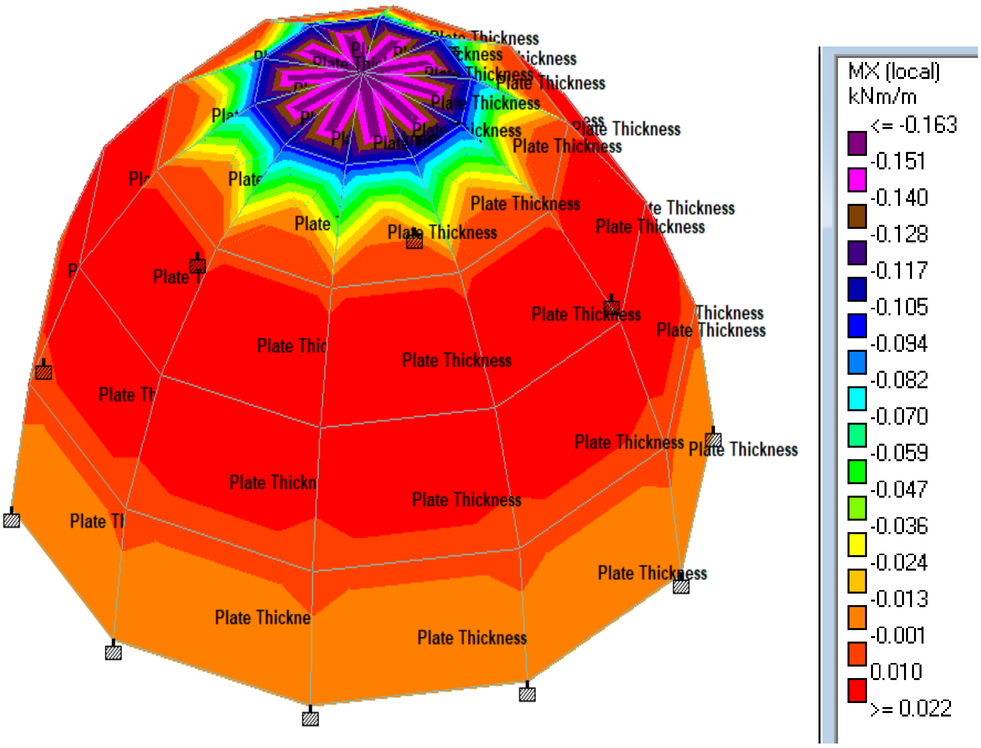
Loads



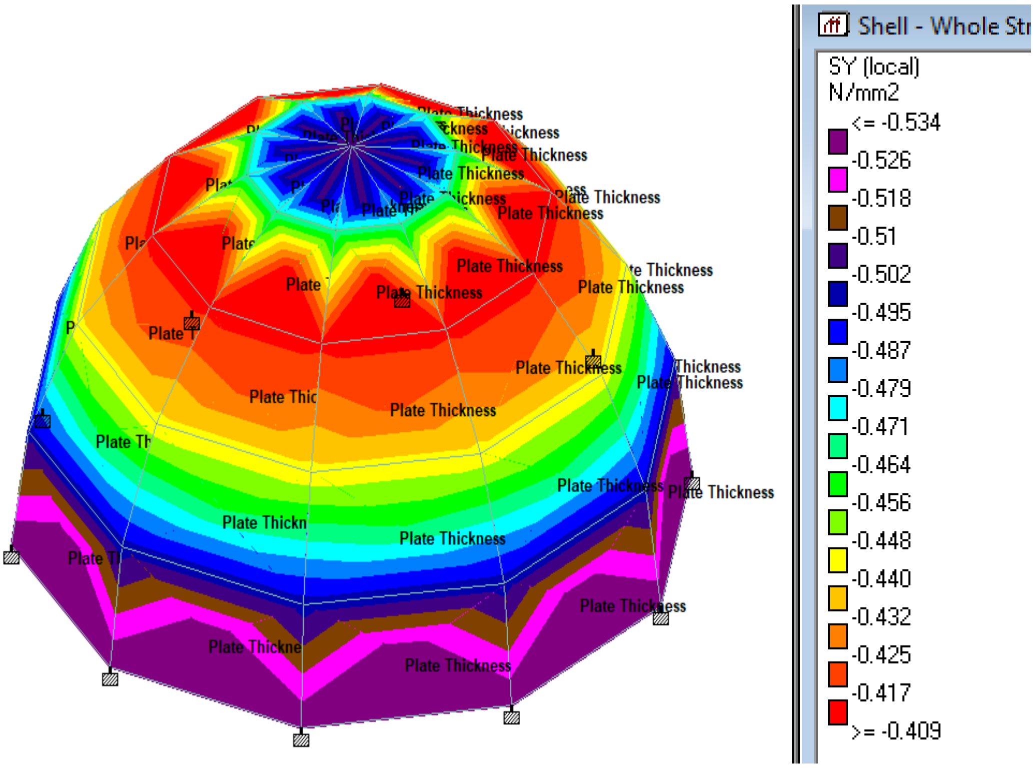
Assigning of load



Reactions at the supports



Bending moment of the plates in the shell



Shear force of the plates in the shell