Dynamic analysis of RCC structure using Autodesk Robot Structural Analysis Software

PROJECT SUPERVISOR

Assistant Prof. Dr. Engr. Sajjad Ahmad

BY

Muhammad Hasnain Abid	**2019-CE-03**
Muhammad Hammad Shafiq	**2019-CE-05**
Muhammad Haseeb	**2019-CE-32**

2019-2023

Department of Civil Engineering

Faculty of Engineering & Technology

Bahauddin Zakariya University Multan, Pakistan

A thesis Submitted in partial fulfilment for the requirement of the degree of B.Sc. Civil Engineering.

Supervisor

Dr. Engr. Sajjad Ahmad
Assistant Professor
Civil Engineering Department
Faculty of Engineering & Technology, BZU, Multan

Thesis Supervisor/Internal Examiner's

Signature: ________________ Signed in Hard-Copy ________________

External Examiner's Signature: ______ Signed in Hard-Copy ______

Head of Department Signature: ________ Signed in Hard-Copy ________

UNDERTAKING

It is certified that research work entitled “Dynamic analysis of RCC structure using Autodesk ROBOT Structural Analysis Software” is our own work. The work has not been presented/ submitted elsewhere for assessment. Where material has been used from other sources has been properly acknowledge/ referenced.

Muhammad Hasnain Abid

Roll# 2019-CE-03

Muhammad Hammad Shafiq

Roll# 2019-CE-05

Muhammad Haseeb

Roll# 2019-CE-32

ACKNOWLEDGEMENT

In the name of Allah, the beneficent the most merciful. All praise is to Allah (SWT) Who in His ultimate and bountiful mercy gave me the opportunity to study up to this level. May peace be uponour Holy Prophet Muhammad (SAW), his companions, households, and the generality of the Muslim Ummah.

First and foremost, we would like express special thanks to my supervisor, Asst. Prof. Dr. Engr. Sajjad Ahmad for his guidance and support during our studies here at Civil Engineering Department BZU Multan. He was always there when we needed help and ready to share his professional expertise, which makes us feel better when we are nervous. Working with such respected and inspirational person has been a privilege and has confirmed our interest in advanced Structural Engineering. Our sincere appreciation goes to Asst. Prof. Dr. Engr. Sajjad Ahmad who taught us the basic knowledge of Autodesk ROBOT structural Analysis software.

During his course, and all academic staff of Civil Engineering department, Faculty of engineering & technology BZU Multan for their valuable and commendable helping hand to us. Deep appreciation is extended to all faculty members and staffs of the department for their help and providing us with helpful information during our study.

May Allah reward him in abundance.

ABSTRACT

The rapid expansion of our built environment has necessitated the construction of high-rise multi-storey buildings to accommodate the growing population. In contrast to the simple materials used in early single-story structures, modern construction relies on Reinforced Cement Concrete (RCC) frames, encompassing slabs, beams, columns, and foundations. RCC combines the strength of concrete with steel reinforcement, making it a best choice for structural integrity.

Megastructures face numerous forces i.e., dead loads, occupancy loads, wind, rain, snow and earthquake load which needs to be carried and transferred to the foundation through the structural framing. The numerous loads on the structure and their cumulative effect on the structure as governed by the ACI/ASCE codes needs to be evaluated for each structural member to determine the required sizes and their reinforcements. The exactness of the analysis not only ensures safety but also brings cost effectiveness in the construction. Considering the complexity of the analysis it is very difficult to analyse the large structures through manual calculations. Therefore, various computer programs have been developed overtime to streamline and reduce the cumbersome work of structural analysis, for example Sap2000, Staad pro, Etabs etc., are commonly employed for this purpose. However, one notable limitation of these software is that they lack the Building Information Modelling (BIM) integration.

Therefore, in this research project a relatively new software “Robot Structural Analysis” has been employed to carry out the structural analysis and design of a multi-storey building. The “Robot Structural Analysis” is a product of Autodesk, which is a leading solution provider in the field of civil engineering such as Autocad, Revit, Naviswork etc.

This research focuses to model a 56 ft. high multi-story commercial building of five stories in Zone 2A and with the soil conditions of Multan Pakistan. After that the structure is design and analyse and this software’s analysis is check through manual calculations. Software and manual results were appropriately same. It is observed that this is easy software in learning as it has flat learning curve. We learned it in very better way, it is a very powerful software and due to BIM integration, all its content can be easily transferable to Revit. It is an advantageous software and is recommended to use.

Sustainable Development Goals

Sustainable development goals plays an important role in the construction of accessible, secure, healthy and productive structures while minimizing negative impacts to environment, society and the economy. Some of the main SDGs for this research are as follows.

Industry, Innovation and infrastructure: by exploring and utilizing innovative structural analysis software "Robot structural analysis" to address the pressing infrastructure needs driven by rapid urbanization and papulation growth. By employing advanced technology, the study facilitates the construction of high-rise, multi-story buildings using sustainable materials like RCC.

Sustainable cities and communities: as it aims to create resilient urban environments capable of accommodating increasing urban populations.

Decent work and economic growth: by enhancing the efficiency and cost effectiveness of structural analysis, ensuring the safety and structural integrity of buildings while reducing construction costs. This contributes to the creation of more employment opportunities in the construction sectors and promotes economics development. The integration of Building Information Modelling (BIM) through the "Robot Structural Analysis" software also aligns with SDGs as it promotes technological advancements in the field of civil engineering.

Good Health and Well-Being: The research underscores the significance of accurate structural analysis, which is vital for ensuring the safety and sustainability of buildings. Safe structures foster better living conditions and minimize risks associated with structural failures.

Partnerships for the Goals: Lastly, the integration of BIM in this study is of vital importance as it promotes collaboration and information exchange between different software tools like "Robot Structural Analysis" and Revit, enhancing efficiency and promoting sustainable construction practices.

In summary, this thesis engages with multiple Sustainable Development Goals by promoting innovative solutions for infrastructure development, enhancing economic growth, ensuring safety and well-being, and fostering partnerships in the pursuit of sustainable and resilient urban environments

Table of Contents

List of Figures

List of Tables

LIST OF ABBREVIATIONS

RCC	Reinforced cement Concrete
ACI	American Concrete Institute
ACI318-14	Design and construction of reinforced concrete structure
BS 8110	British Building Code
UBC97	Earthquake Code
RSA	ROBOT Structural Analysis
BIM	Building Information Modelling
SDG	Sustainable Development Goals

ASCE	American Society of Civil Engineers

LIST OF SYMBOLS

I	Importance factor
W_U	Factored Load
DL	Dead Load
LL	Live Load
M_y	Moment
F_z	Shear Force
R	Radius
f	First mode frequency
G	Peak factor
A	Projected area
G	Shear Modulus
W	Wind load
E	Young Modulus
v	Poisson Ratio
F	Force
S	Stress
g	Acceleration due to gravity
U	Displacement
R	Angle in radian
F_y	Flexural strength
F_c	Compressive strength
F_u	Ultimate strength

CHAPTER NO. 1
INTRODUCTION

1.1 Introduction

There is growing responsiveness of multi-storey reinforced concrete structures, to accommodate growing population. Generally, such structures have prismatic sections which are common in developing countries, which resist applied loads without any appreciable deformation of one part relative to another. It is the need to accomplish some function, one of them is to receive loads (usually known as service loads) at certain points & transmit them safely to other points, that prompts the designer to give life to a structure furthermore since it is the need for a safe, serviceable, feasible and aesthetically pleasing fulfillment of a structure. The ultimate aim of structural analysis is to design all the structural elements of a structural system in such a way that they perform their functions satisfactorily and at the same time assist design to become efficient, elegant and economical which helps to choose the right type of sections consistent with economy along with safety of the structure. Many structures are built of reinforced concrete: bridges, viaducts, buildings, retaining walls, tunnels, tanks, conduits, and others .This deals primarily with fundamental principles in the design and investigation of reinforced concrete members subjected to axial force, bending moment, shear, torsion, or combinations of these. Thus, these principles are basically applicable to the design of any type of structure, so long as information is known about the variation of axial force, shear moment, etc., along the length of each member.

Although analysis and design may be treated separately, they are inseparable in practice, especially in the case of reinforced. The multistory building is statically indeterminate structure and there are several methods to analysis this structures such as method three and moment distribution….etc. To analyze and design the multistory building we must analyze and design the elements that combined it, such slabs, beams, columns and footing. Large amounts of concrete are used in the construction industry in Pakistan and most countries due to its availability. Concrete is arguably the most important building material, playing a role in all structures. It has the virtue of versatility, i.e. its ability to be molded to take different shapes n the structural work.

It is also very durable and fire resistant with good control and correct construction procedures are followed.

1.2 RCC structures

The reinforced cement concrete (RCC) is a composite building material consisting of structural concrete reinforced with a reinforcing material like steel. The most common reinforcement used is steel, due to its complimentary properties and it is called steel reinforced cement concrete or simply reinforced cement concrete.

1.3 History of RCC Structures

Leaning Tower of Nevyansk in the town of Nevyansk in Sverdlovsk Oblast, Russia is the first building known to use reinforced concrete as a construction method. François Coignet used iron-reinforced concrete as a technique for constructing building structures. In 1853, Coignet built the first iron reinforced concrete structure, a four-story house at 72 rue Charles Michels in the suburbs of Paris. Joseph Monier, a 19th-century French gardener, was a pioneer in the development of structural, prefabricated and reinforced concrete, having been dissatisfied with the existing materials available for making durable flowerpots. In 1877, Monier was granted another patent for a more advanced technique of reinforcing concrete columns and girders, using iron rods placed in a grid pattern. Ransome, an English-born engineer, was an early innovator of reinforced concrete techniques at the end of the 19th century. One of the first skyscrapers made with reinforced concrete was the 16-story Ingalls Building in Cincinnati, constructed in 1904. The structure was constructed of reinforced concrete frames with hollow clay tile ribbed flooring and hollow clay tile infill walls. In 1908, the San Francisco Board of Supervisors changed the city's building codes to allow wider use of reinforced concrete.

1.4 Importance of civil structures

Civil structures are designed to withstand loads and forces that can cause damage or collapse. They are typically made of materials such as concrete, steel, and masonry, which are strong and durable. Examples of civil structures include buildings, bridges, tunnels, dams, and roads. These structures are essential for modern society to function, as they provide transportation, shelter, and other services that people rely on every day.

Civil engineers are responsible for designing, building, and maintaining these structures to ensure that they are safe and functional. They use advanced technologies and mathematical models to ensure that structures are able to withstand the forces of nature, such as earthquakes, wind, and floods. Overall, civil structures are an essential part of modern society, and they play a critical role in promoting economic growth, public safety, and quality of life.

1.5 Components of RCC structure

A Civil Engineering project compose of the following main components.

1.5.1 Structural member columns

Columns are the important structural member of a frame building. They are the vertical members which carry the load from the beam and upper columns, transferring it to the footings. The design of the columns is more important than the design of the beams and slabs. This is because, if one beam fails, only 1 floor will collapse, but if even a single column fails, it can lead to the collapse of the whole structure.

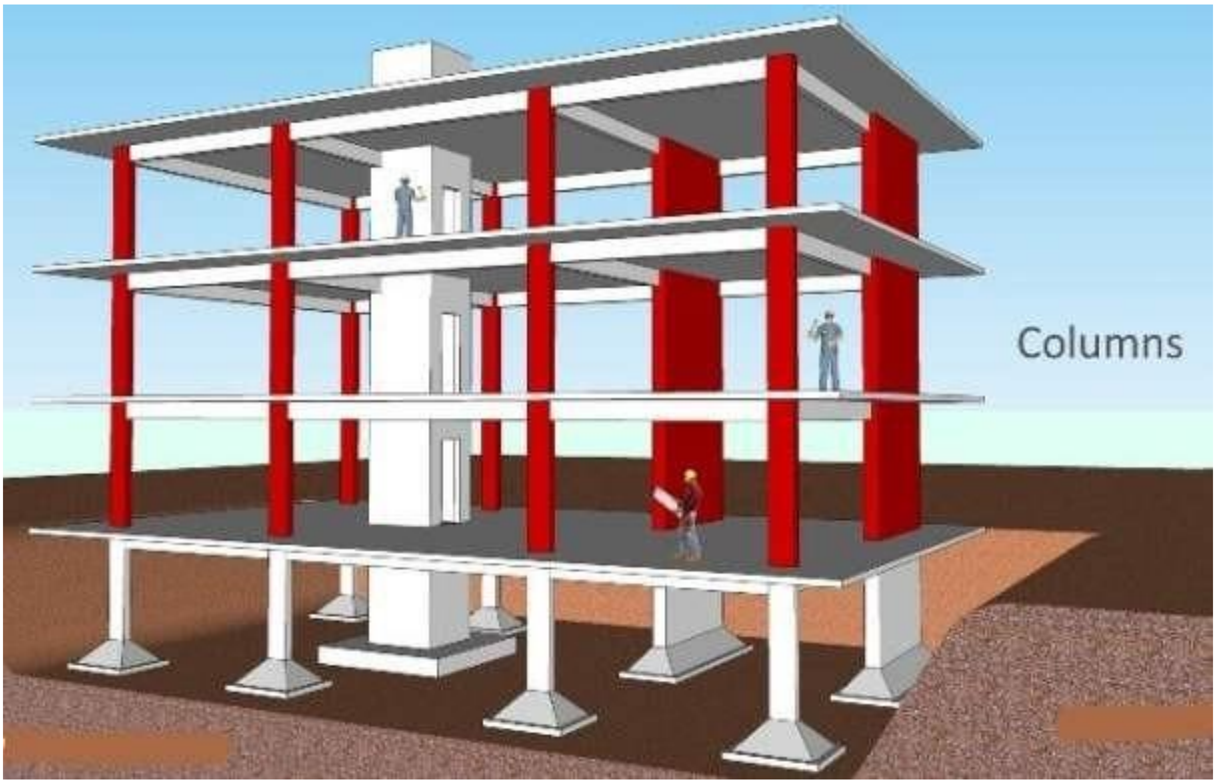


Fig 1.1 Structural member Columns

1.5.2 Structural member Beams

Beams are the horizontal load-bearing members of the framed structure. They carry the load from the slab as well as the direct load of masonry walls and their self-weight. The beams may be supported on the other beams or may be supported by the columns forming an integral part of the frame. These are primarily the flexural members and are classified into 2 types:

a) - Main Beams - Transmitting floor and secondary beam load to the columns.

b) - Secondary Beams - Transmitting floor loads to the main beams.

Fig 1.2 Structural member Beams

1.5.3 Structural member Slabs

A slab is a flat horizontal place that is used for covering the building from above and providing shelter to the inhabitants. This is the plate element and carries the loads primarily by flexure. They usually carry a vertical load. Under the action of a horizontal load and a large moment of inertia, slabs can carry large wind and earthquake forces, and then transfer them to the beam.

Fig 1.3 Structural member Slabs

1.5.4 Structural member Foundation

The sole purpose of the foundation is to transmit the load coming from the above columns and beams to the solid ground. Foundation, a part of a structural system that supports and anchors the superstructure of a building and transmits its loads directly to the earth. To prevent damage from repeated freeze-thaw cycles, the bottom of the foundation must be below the frost line.

Fig 1.4 Structural member Foundations

1.6 Problem Statement

Our built up environment is expanding day by day, in early days there were single story buildings made up of simple types of material i.e. soil, wood and bricks but nowadays as population is increasing rapidly due to which land occupancy is also increasing. So to meet challenges of modern days we have to construct high rise multistory buildings. For the construction of multistory buildings there require RCC frames i.e. Slab, Beam, Column etc.

When a mega structure is construct a number of forces acts on the structure and these forces transmit the load of the structure through RCC frames mechanism that is from slab to beam to column and finally to the foundation.

So, when there is megastructure there must be an analysis of the structural design before the construction that how much and where stresses are produce in the structure. For the analysis of structure various software are used for example Sap2000, Staad Pro, Etabs, Tekla, Revit & Autodesk Robot structural Analysis but mostly Etabs is used however there is a lack in the

Etabs that is BIM integration. Building Information Modeling is the foundation of digital transformation in the architectures, engineering and construction (AEC) industry. As the leader in BIM, Autodesk is the industry's partner to realize better ways of working and better outcomes for business and built the world. Building Information Modeling (hereinafter called BIM) – design concept that provide complex data collection, processing and analysis from each stage in the design process (architecture, construction and etc.) and the use of that information throughout the entire lifecycle of the object. Moreover, the information from the model is interrelated, so, the design collisions can be avoided. BIM has three main advantages. 1. The objects of BIM is not just 3D models. The information provides the ability to create drawings automatically, perform analysis of project, etc. It helps to find the best design decisions. 2. BIM supports collaborative design. Different specialists can use all the information during each phase of design and construction, eliminating the loss of data during the sharing process. 3. Reduced costs and design errors (collisions). That's why we will use Autodesk ROBOT Structural Analysis Software for the static & dynamic analysis of our building analysis as it BIM integrated software.

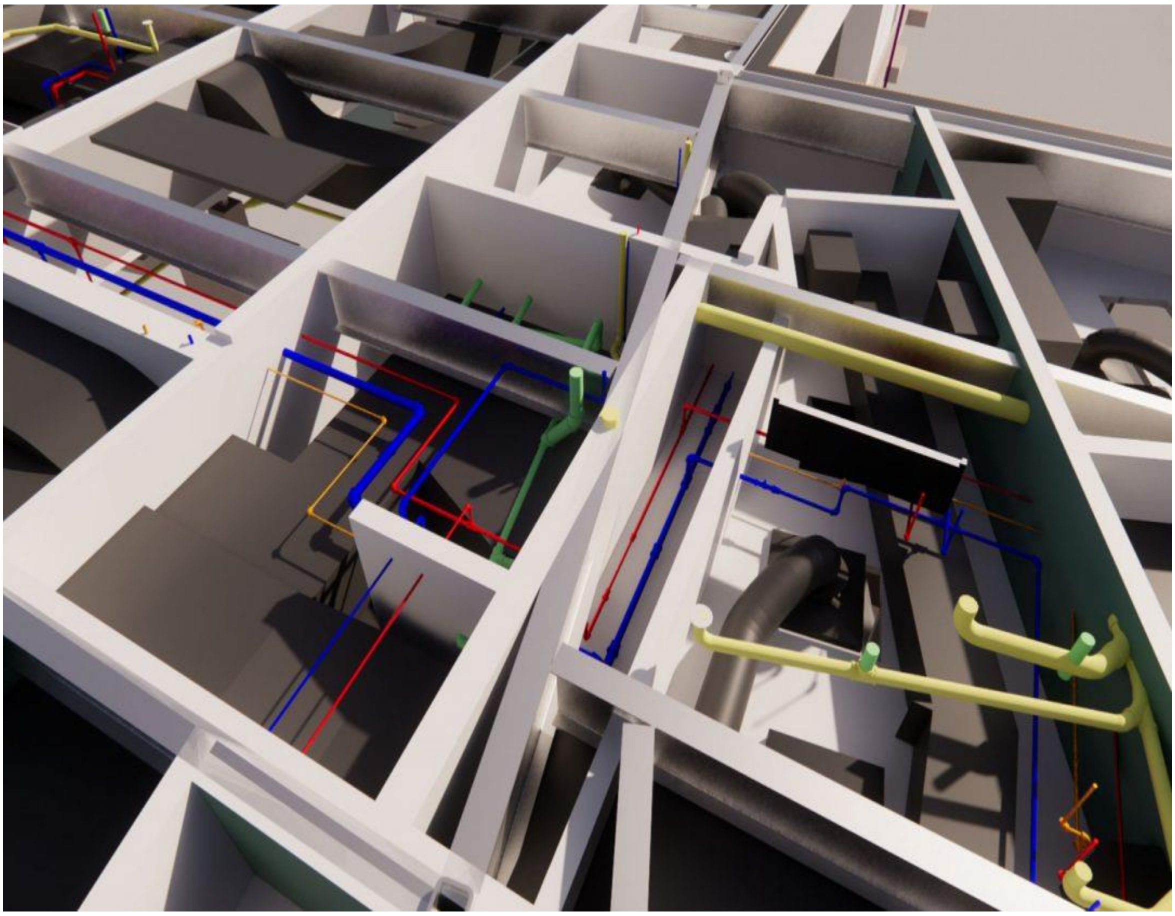


Fig 1.5 Application of BIM Technology

1.7 Objectives

The main objectives of this investigation/research are

1. To brilliantly comprehend the design capability, method & features of Autodesk Robot Structural Analysis Software.
2. To perform the structural analysis of a multistory RCC frame structure using Autodesk Robot Structural Analysis.
3. To validate the Autodesk Robot Structural Analysis results by comparing them with the manual calculations.

1.8 Scope & Relevancy to National needs

The project focuses mainly analysis of the building, the structure is a five storey building. This structure is intended to serve as a commercial building. The main reason why a five-storey structure is being adopted is that it does not involve calculation for the wind load, the code used is ACI and the selected software to use is Robot Analysis.

In this research, we study the dynamic analysis of a RCC structure building comprising 5 storeys & have a covered area of approximately 600 sq. ft. located in the DHA Multan (PAKISTAN) using Autodesk Robot structural analysis software on the intensity of earthquakes of zone 2A & the soil conditions of Multan.

With the passage of time, the construction industry is growing very fast, in early days there were are one story building composed of soil bricks, wood, etc. But nowadays trend is change, there is low space but requirements are very high. One of the most significant benefits of high rise living is the efficient use of limited urban space, as these structures can house many people and businesses in a relatively small footprint. This can help reduce urban sprawl, limit the need for long commutes, and create more walkable neighborhoods.

User wants all facility such as multi story building, air conditions, Fire extinguisher, fall ceiling, plumbing & electricity wiring in this short space. This is only possible by using a software that should be BIM integrated where we can see our structure, plumbing, wiring & many more in one software. In Pakistan, the uses of software is very low that's why there is no high performance related to construction industry.

Fig 1.6 Design of Model for Analysis

That's why we are using Autodesk Robot structural analysis software for this purpose as it is BIM integrated. Also, by using software we can step into the era of technology and advancement and finally this will lead us to the betterment and development of the country in near future.

CHAPTER NO. 2
LITERARATURE REVIEW

2.1 Introduction

The structural analysis focuses on the changes occurring in the behavior of a physical structure under observation when provided with a force or in case of structures & load. Now if this load is quasi (very slow), the inertia forces from the basis of newton's first law of motion can be neglected and the analysis becomes static. The static loads are very slow on the time rate graph. Static structure analysis methods are:

- Linear static analysis
- Non-linear static analysis

Opposite to this, if the force or load applied have a high rate of change of velocity during the process then it comes under dynamic analysis, the cause-and-effect portion comes in to play here. The system under consideration is observed for the development occurring during a course of time and then analyzing the cause of those changes. A dynamic analysis is also related to changes. This dynamic analysis is also related to the inertia forces developed by a structure when it is excited by means of dynamic loads applied suddenly (e.g., wind blasts, explosion, and earthquake)

2.2 Structural Analysis

When a mega structure is construct a number of forces acts on the structure and these forces transmit the load of the structure through RCC frames mechanism that is from slab to beam to column and finally to the foundation. Due to the loading on the structure, strains are developed in the structure which ultimately shows us that there are some stresses present in the structure. For the analysis of the stresses in the structure we can use analytical methods. Similarly, Softwares can also be used for this purpose. A building comprising of a G+7 regular multi-storey is analysed for the zone 5 & concluded that as the effect of earthquake is more in bending moment in column, footing and beam displacement in column joint, if structure is designed for Static case it will not sustain earthquake load (Dynamic). So we have to consider the earthquake load for the analysis and design. The bending moment due to earthquake load in column is highly increasing with the storey height.

So, if earthquake load is not considered for the analysis there will be possibilities for overturning, (Amresh. A. Das, et al.) Shear walls are very important factor in a high-rise multistory building comprising of G+7 stories. For the dynamic analysis, a plan of a multi-storey building is taken, and it has been modelled with different structural elements for minimum story displacement. The dynamic analysis of multi-storey buildings is done using Etabs 2015 by IS and SP codal provisions (Etabs User's Manual, 2015). The multi-storey building is R.C.C. structure with 3 basements + ground floor + 14 upper floors in zone IV with a maximum earth fill of 750 mm on the ground floor for landscape requirements. For important structures, when compared with response spectrum analysis, time history analysis has to be performed as it predicts the structural response more accurately. From the results, it is clear that, shear walls are to be present in the high-rise buildings to control storey displacement, storey drift and centre of mass displacement, (Udaya Bala K, et al.)

Working on Structural softwares need a deep and sound knowledge about the working principles of the softwares otherwise it will be time consuming and headache for someone. The calculations were done on the design of three-story structures elements, which are beams, columns, slabs and foundation. A specific load was applied, and designs were carried by Robot analysis software using ACI code to find the minimum cost and maximum safety of design according to the code. From the study, it was concluded that all design done on software require sound knowledge about design methods and philosophies, (Muhammad Abdul Azeem baig, et al.) Manual calculation for the dynamic analysis of a high-rise building is a time consuming process. The results of the thesis show that for the given applied load on the structure the load combinations by traditional manual calculations is almost impossible to executed due to many possible combinations which leads to difficulties to find out the critical load. This also leads to the overestimation of critical load by introducing a high value for safety factors, resulting in oversizing of structural elements eventually increasing the cost of the structure. Moreover, it is tedious and time consuming to perform the calculation manually which makes using software for structural analysis more beneficial and efficient, (Ardeep Maharjan, 2018) Comparative study has been carried out between static and dynamic analysis. It was found that ELF procedure provides higher displacement, story drift and base shear compared to RS procedure. Based on the findings of the study it is recommended to use dynamic analysis (RS) instead of static analysis (ELF) especially in high rise building, (Ghusen Al-Kafri, et al.)

Similarly, some of the analytical methods for the analysis of a structure are

- Conjugate Beam Method
- Moment Distribution Method
- Slope Deflection Method
- Direct Stiffness Method
- Equivalent Frame Method
- Virtual Work Method
- Finite Element Method
- Response Spectrum Analysis
- Time History Analysis

All the loads are considered dynamic in nature as at some point of time they were nature asat some point of time they were absent and that includes the self-weight of the structure also.The applied action or force’s acceleration is compared with the structures natural frequencyto find weather analysis is static or dynamic in nature. Let us discuss each of following briefly.

1) Conjugate beam method:

A conjugate beam is defined as a fictitious beam whose length is the same as that of the actual beam, but with a loading equal to the bending moment of the actual beam divided by its flexural rigidity, EI.

2) Moment distribution method:

In the moment distribution method, every joint of the structure to be analyzed is fixed so as to develop the *fixed-end moments*. Then each fixed joint is sequentially released and the fixed-end moments (which by the time of release are not in equilibrium) are distributed to adjacent members until equilibrium is achieved.

3) Slope deflection method:

By forming **slope deflection equations** and applying joint and shear equilibrium conditions, the rotation angles (or the slope angles) are calculated. Substituting them back into the slope deflection equations, member end moments are readily determined. Deformation of member is due to the bending moment.

4) Direct stiffness method:

It is a *matrix* method that makes use of the members' stiffness relations for computing member forces and displacements in structures.

5) Response Spectrum Analysis:

Response spectrum analysis is a method to estimate the structural response to short, nondeterministic, transient dynamic events. Examples of such events are earthquakes and shocks

6) Time History Analysis:

Time history analysis is a part of structural analysis and is calculation of the response of a structure to any earthquake.

7) Finite Element Analysis

The static or dynamic analysis of such idealized structures is known as the Finite Element Method (FEM). This is a powerful method for the analysis of structures with complex geometrical configurations, material properties or loading conditions.

Finite Element Analysis. Finite Element Analysis (FEA) is a type of computer program that uses the finite element method to analyze a material or object and find how applied stresses will affect the material or design. FEA can help determine any points of weakness in a design before it is manufactured.

The finite element method (FEM) is the most widely used method for solving problems of engineering and mathematical models. Typical problem areas of interest include the traditional fields of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The FEM is a particular numerical method for solving partial differential equations in two or three space variables (i.e., some boundary value problems). To solve a problem, the FEM subdivides a large system into smaller**,** simpler parts that are called finite elements. This is achieved by a particular space discretization (discretization is the process of transferring continuous functions, models, variables, and equations into discrete counterparts. This process is usually carried out as a first step toward making them suitable for numerical evaluation and implementation on digital computers) in the space dimensions, which is implemented by the construction of a mesh of the object: the numerical domain for the solution, which has a finite number of points. The finite element method formulation of a boundary value problem finally results in a system of algebraic equations. The method approximates the unknown function over the domain.

Finite element analysis (FEA) is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis shows whether a product will break, wear out, or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what is going to happen when the product is used.

8) Equivalent Frame Method

Equivalent Frame Method involves the representation of three-dimensional slab system by a series of two-dimensional frames that are then analyzed for loads acting in the plane of the frames.

9) Virtual Work Method

This method was developed by John Bernoulli in 1717 AD. The Virtual work method also referred to as the method of virtual force or unit-load method, uses the law of conservation of energy to obtain the deflection and slope at a point of structure.

2.3 Structural Design

After the analysis of the structure, we perform the design. Usually there are different softwares for design and drafting but with the use of Building Information Modelling (BIM) integration we can automated our all the system. Cost Controlling, Planning Controlling, and time management become easier with the use of a software that is BIM integrated. A study compared the BIM processes with the traditional processes for project delivery, combining the actual use of BIM, and found the benefits of applying BIM to the project: fast real-time design; accurate 3D visualization with multiple design views; automatic document management; enhanced collaboration and communication; fast and accurate model-based estimating; and optimized 4D scheduling. The application of BIM also meets the standard of sustainability in various ways: shortening the field-cycle time, increasing on-site renewable opportunities, reducing the waste produced through 45 construction, saving energy, increasing labor productivity, etc. In summary, the impact of BIM on time and cost control is all about minimizing cost and accelerating progress. (Dalu Zhang).

2.4 BS 8110 Building Code: Part 1:1997

BS 8110 part 1 gives recommendations for the structural use of concrete building and structures,excluding bridges and structural concrete made with high alumina cement. The aim of design is the achievements of an acceptable probability that structures being design will perform satisfactory during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformation of normal construction and use and have adequate durability and resistance to theeffects of misuse and fire. The structure should be so designed that adequate means exist to transmit the design ultimate dead, wind and imposed loads safely from the highest supported levelto the foundations (British code, 1997).

The design strengths of materials and design loads should be based on the loads and material properties as in the BS 8110 and as appropriate for the serviceability limit state (SLS). The design should satisfy the requirement that no SLS is reached by rupture of any section, by overturning orby buckling under the worst combination of ultimate loads.

2.5 ACI 318 Building Code (ACI 318-14)

The American concrete institute standard 318, building code requirements for reinforced concrete,has permitted the design of reinforced concrete structure in accordance with limit state principles using load and resistance factors since 1963. A probabilistic assessment of these factors and implied safety levels is made, along with consideration of alternate factors values and formats. (Adiscussion of issues related to construction safety of existing structure is included). Working stress principles and linear elastic theory formed the basis for reinforced concrete design prior to 1983, when the concept of ultimate strength design was incorporated in the ACI building code (ACI318-02), (Edward cohen, 1971). Because of the highly nonlinear nature of reinforced concrete behavior,the linear approach was unable to provide a realistic assessment of true safety levels (Andrew Scanlon, 1992).

The developers of ACI 318-14, who introduced the idea of load and resistance factors to account for uncertainties in both load and resistance .Probabilistic methods were developed and refined during the late 1960s in response to the need to consider variability and uncertainty, explicitly and rationally. Proposed formulations include code incorporation of explicit second moment probabilistic procedures.

In such an approach, the designer would select a desired safetyindex “B” and carry out the design utilizing the means standard deviations of the load and resistance variables.

The safety index positions the mean load effect to ensure attainment of the target reliability (American code). The explicit second moment approach was not considered by ACI38 or other major code writing organizations. (Edward Cohen, 1971).

2.6 Optimum Cost of Reinforced Concrete Building

The meaning of the optimum cost of reinforced concrete building with some studies, which it is minimum quantity of concrete and steel in any construction or it is the minimum cost of the construction but the most studies explains the optimum cost by minimum quantity of concrete andsteel in any construction.

Hence, the primary objective of economic analysis is to secure cost-effectiveness for the client. In order to achieve this, it is necessary to identify and to evaluate the probable economic outcome of a proposed construction project. An analysis is required from the viewpoint of the owner of the project when doing the proposal, the analysis can be evaluated the followings (Ashworth A., 1994) to achieve maximum profitability from the project concerned, to minimize construction costs within the criteria set for design, quality and space, to maximize any social benefits, to minimize risk and uncertainty and to maximize safety, quality and public image.

Cost and safety are one of the important factors that will affect method of construction, quality of work, period of the construction and most of all, the success of a project. It seeks to ensure the efficient use of all available sources to construction. Client's requirements, possible effect on the surrounding areas, relationship of space and shape, assessment of the initial cost, the reason for, and method of, controlling costs, the estimation of the life of buildings and material need to be studied so as to improve the efficiency of control in construction (Flanagan R. and TateB., 1997).

2.7 Factors Contributing to the Building Construction

Implementation of a construction projects is a complicated and complex process (Neap H.S and Celik T., 2001). Phases of construction are divided into categories such as material, labor, plant, supervision, All disturbances regarding the cost must be detected periodically (Popescu, 1977).

The collection, analysis, publication and retrieval of designed information are very important to the construction industry. Contractors and surveyors will tend, wherever possible, to use their owngenerated data in preference to commercially published data, since the former incorporate those factors which are relevant to them. Published data will therefore be used for backup purpose. Theexistence of a wide variety of published data leads one to suppose, that it is much more greatly relied on than is sometimes admitted (Ashworth A., 1994)

2.8 Basic principles of cost

Most decision makers recognize that there are only a few variables that have a large influence ona building's costs. Brandon has classified these variables into two categories decisions concerning the size of the buildings and decisions concerning material specifications and buildingconfiguration (Figure 2.1).

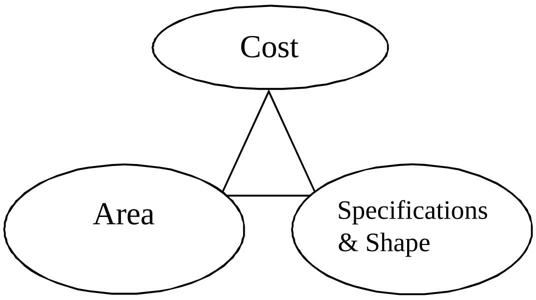


Fig 2.1 Design / Costs relationships

CHAPTER 3
RESEARCH METHDOLOGY

3.1 Use of Software for Dynamic analysis

3.1.1 Introduction

We used Robot Structural Analysis Professional for the dynamic analysis of RCC Structure. Robot Structural Analysis Professional is structural load analysis software that verifies code compliance and uses BIM-integrated workflows to exchange data with Revit. It can help you to create more resilient, constructible designs that are accurate, coordinated, and connected to BIM.

Autodesk Robot structural analysis professional is a software application developed by Autodesk, Inc. It falls under the category of structural analysis and design software. The software is designed to assist engineers and structural designers in simulating and analyzing the behavior of various structures, such as buildings, bridges, towers, and other types of infrastructure.

3.1.2 Salient Features of Software

1) **Advanced analysis capabilities:** The software provides a wide range of analysis options, including linear and nonlinear static analysis, dynamic analysis, buckling analysis, and more. This allows engineers to accurately stimulate behavior of complex structures under various load conditions.

2) **BIM integration:** Robot structural analysis can work seamlessly with Building Information Modelling (BIM) software such as Autodesk Revit. The integration enables better collaboration between architects and engineers by allowing them to exchange information and models more effectively.

3) **Concrete and steel design:** Robot structural analysis offers tools for the design of both concrete and steel structures. It can perform design checks for elements like beams, columns, slabs and more based on the chosen design codes.

4) **Visualization and reporting:** The software provides visualization tools that help engineers understand the behaviors of structures through graphical representation of stresses, deformation, and other analysis results. It also allows generating detailed reports that can be shared with project stakeholders.

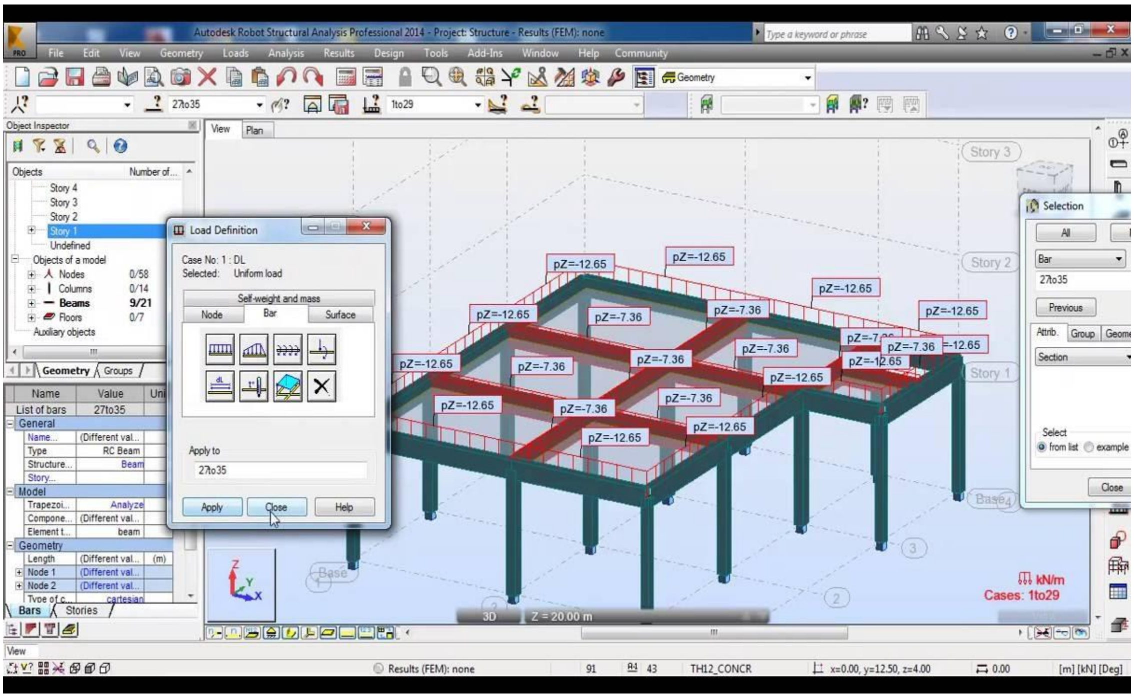


Fig 3.1 Model Analysis on Autodesk Robot Structural Analysis Software

6) Concrete and steel design: Robot structural analysis offers tools for the design of both concrete and steel structures. It can perform design checks for elements like beams, columns, slabs and more based on the chosen design codes.

7) Visualization and reporting: The software provides visualization tools that help engineers understand the behaviors of structures through graphical representation of stresses, deformation, and other analysis results. It also allows generating detailed reports that can be shared with project stakeholders.

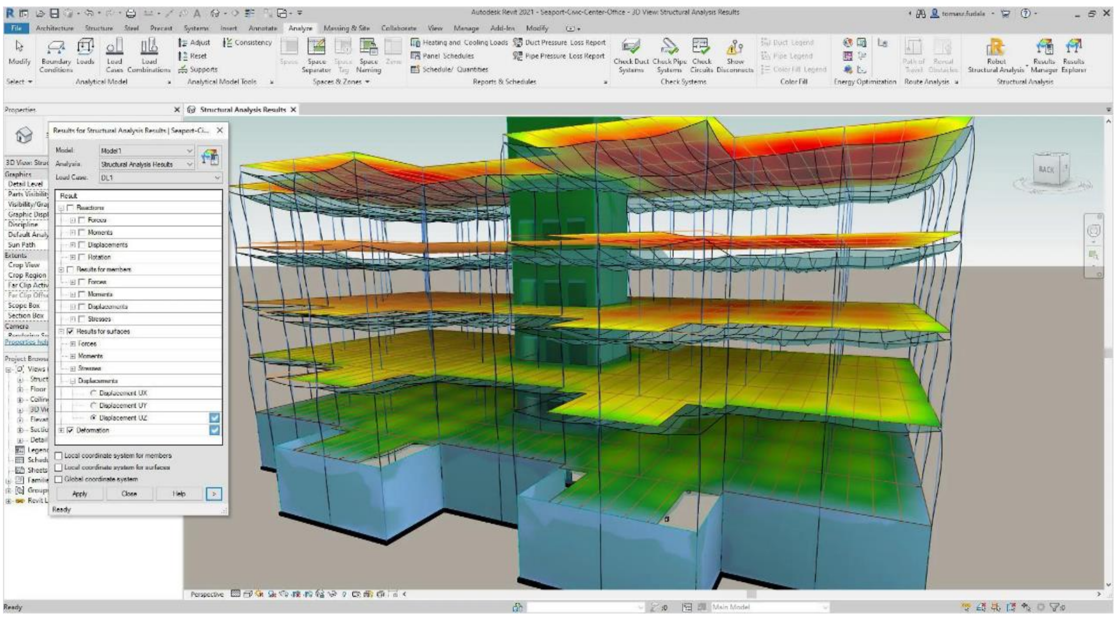


Fig 3.2 Model Analysis on Autodesk Robot Structural Analysis Software

8) Parametric wind loading: The software can automatically generates wind loadings on complex building geometries, simplifying the process of assessing wind-induced effects.

9) Open API: For those who require custom workflows or integrations, ROBOT structural analysis offers open API (Application programming interface) that allows users to extend and customize the software’s capabilities.

10) Cloud Collaboration: Depending on the version and licensing, the software may offer cloud-based collaboration options, allowing engineers to work on projects collectively from different locations

3.2 Selection of RCC Structure for Dynamic analysis

3.2.1 Site Selection

We started our site visit and survey in September 2022 for selection of a multi-storey RCC structure for dynamic analysis, in different locations of Multan. We searched out a commercial building at DHA (Defense Housing Society) Multan. It is under construction and it is built on a very short area of 30’ x 30’ comprising of five stories in Multan region, so we selected this building as it is recommended by our project`s supervisor.

3.2.2 Reconnaissance survey of Building

After the selection of Building, we started our 1st task to take its dimension and drawings for details from relevant authorities of Building and take all geometrical readings manually.

3.2.3 Model of Design

The model that will be designed is a multi-stories reinforced concrete commercial building which has a covered area of 32.764 ft. x 32.604 ft. width as covered area and the building consists of five stories, three stories upon the ground floor with height 10 ft. with ground floor height of 16 ft. Figure 3.3, 3.4, 3.5, 3.6, 3.7, 3.8 & 3.9 shows the elevation & plan of the building.

Fig 3.3 Elevation of Building of building

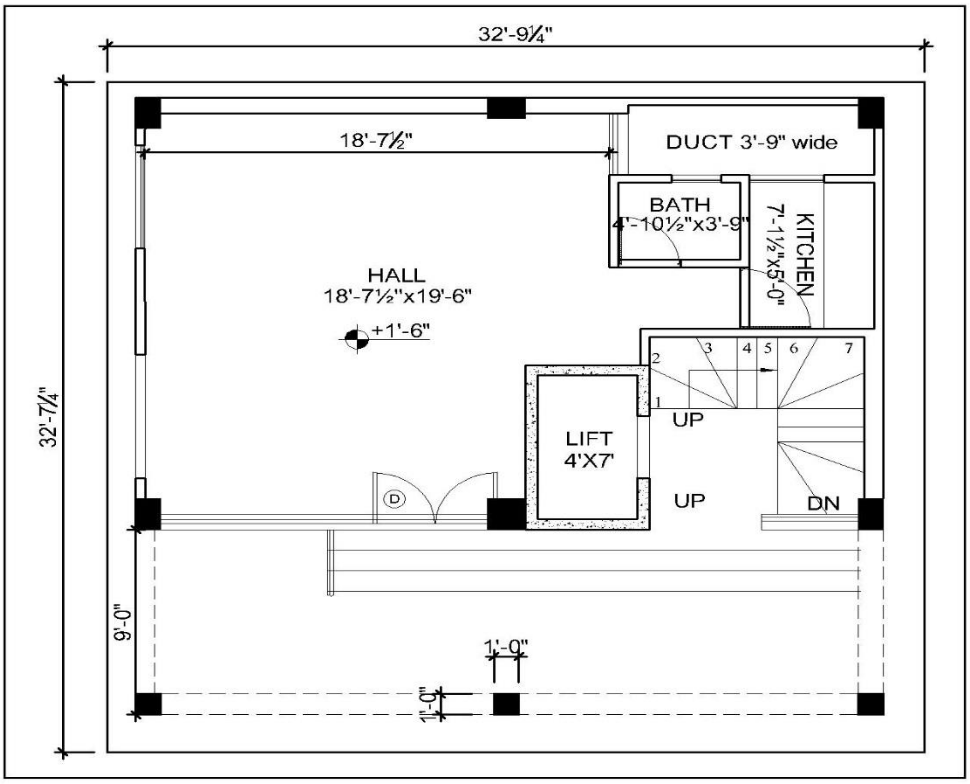


Fig 3.4 Ground Floor Plan of building

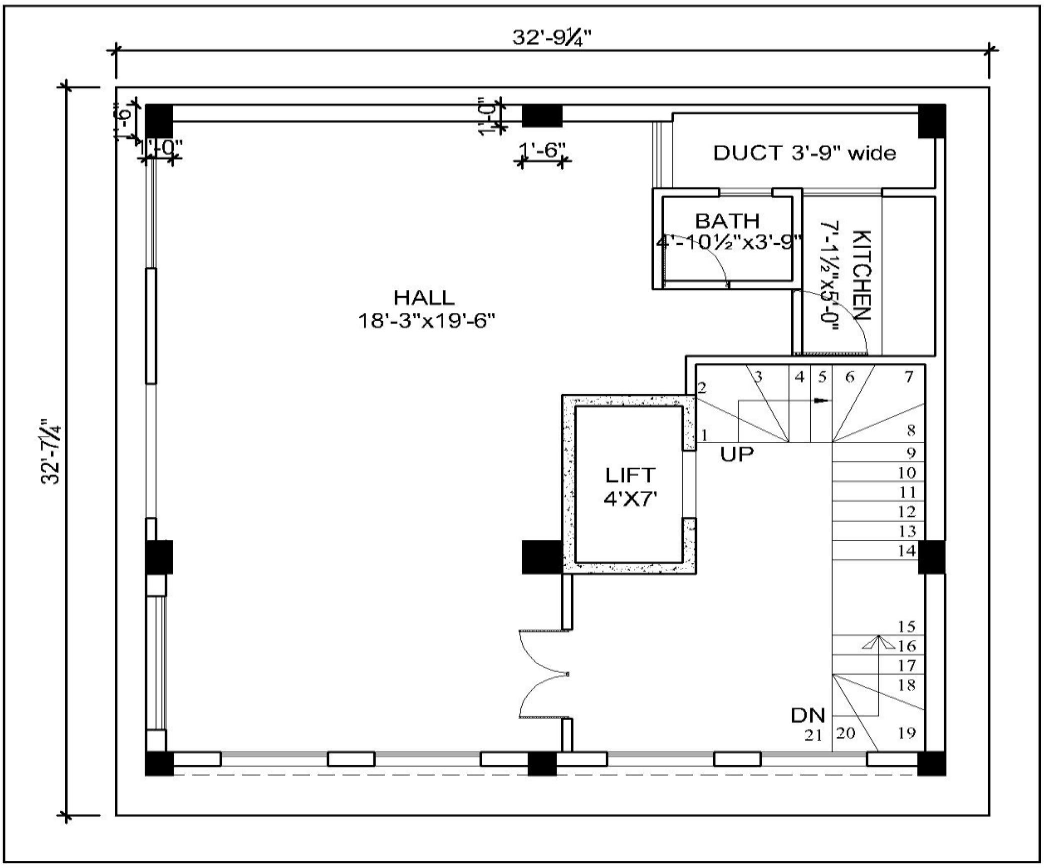


Fig 3.5 1st Floor Plan of building

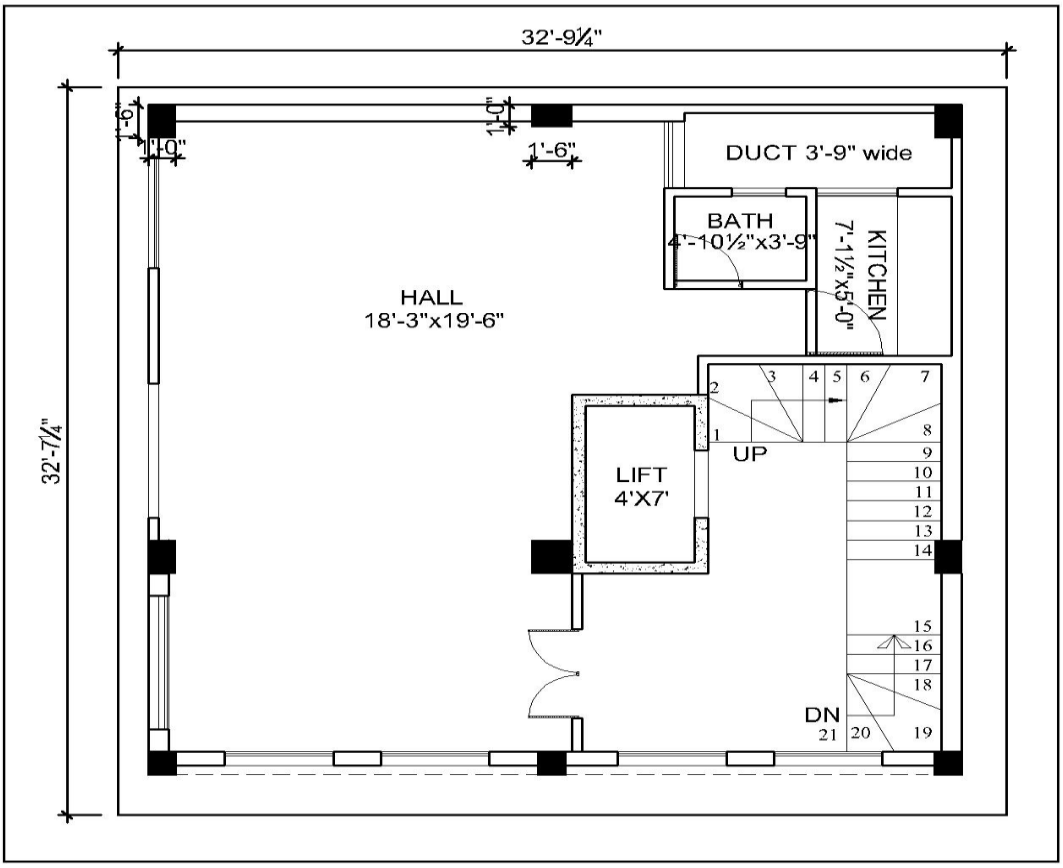


Fig 3.6 2nd Floor Plan of building

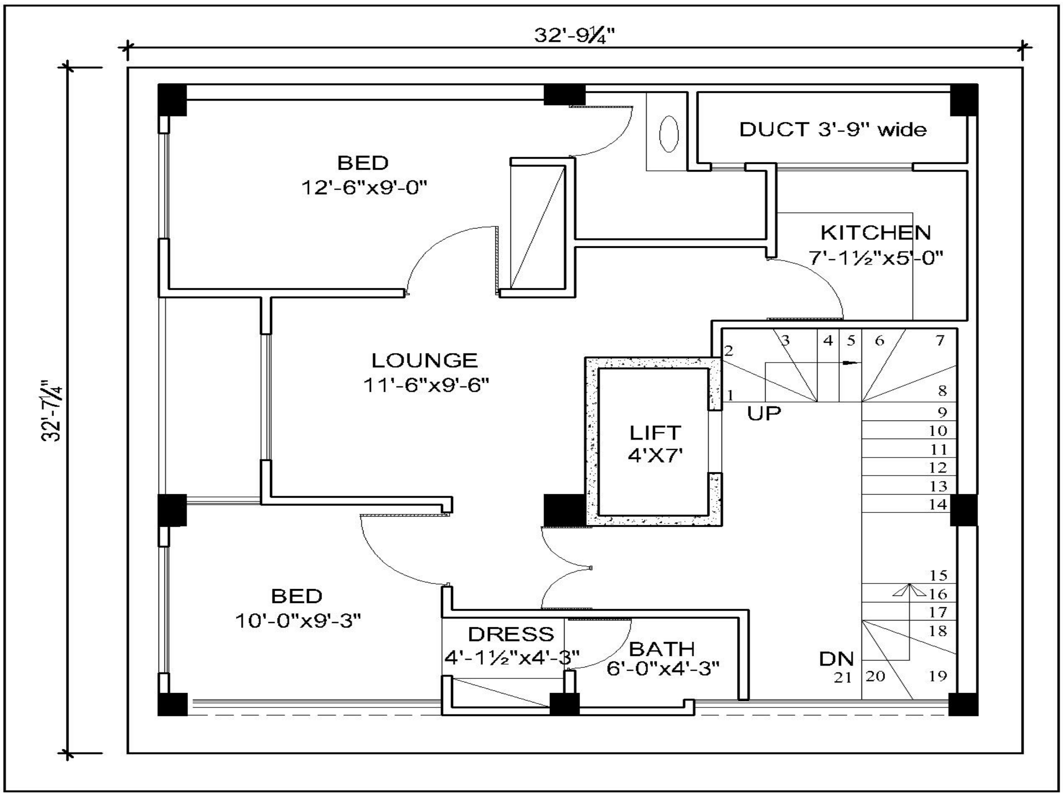


Fig 3.7 3rd Floor Plan of building

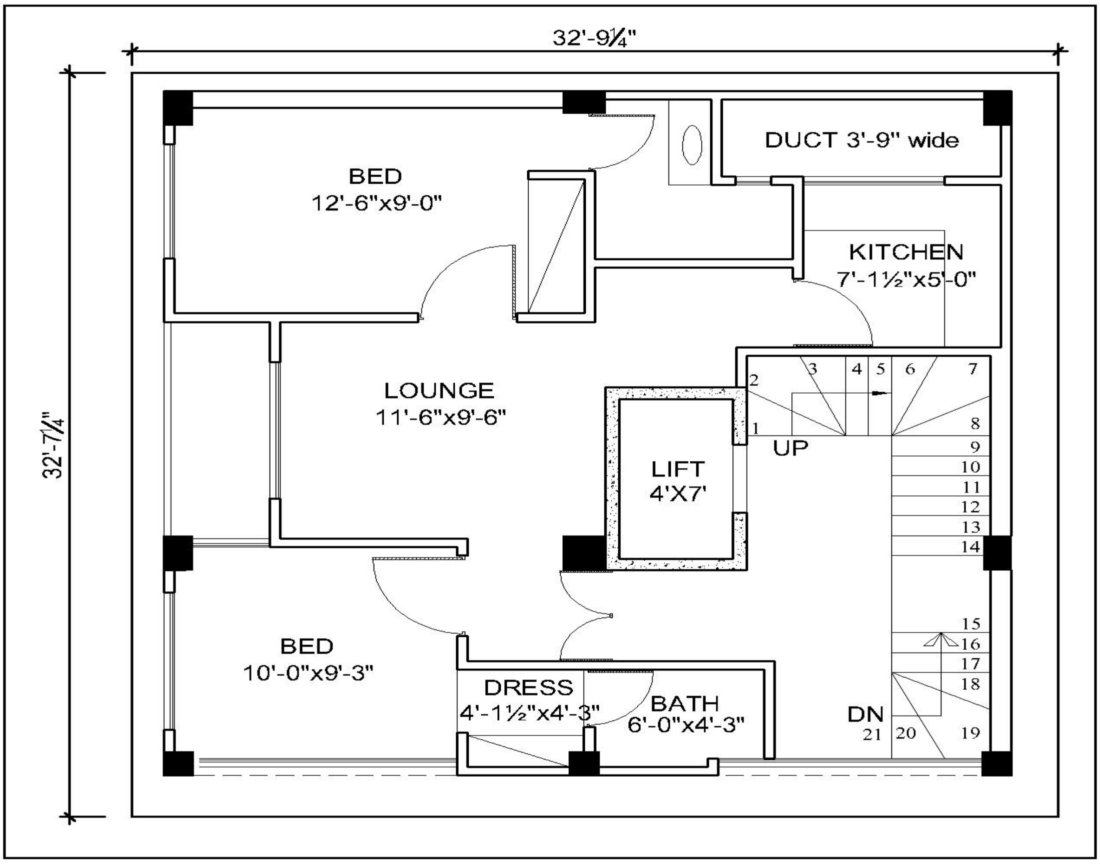


Fig 3.8 4th Floor Plan of building

The dimensions of 1st & 2nd floors are same, similarly the dimensions of 3rd & 4th floors are exact same. All the floor have a height of 10 ft. except ground floor which has a height of 16 ft. and building with total height of 56 ft.

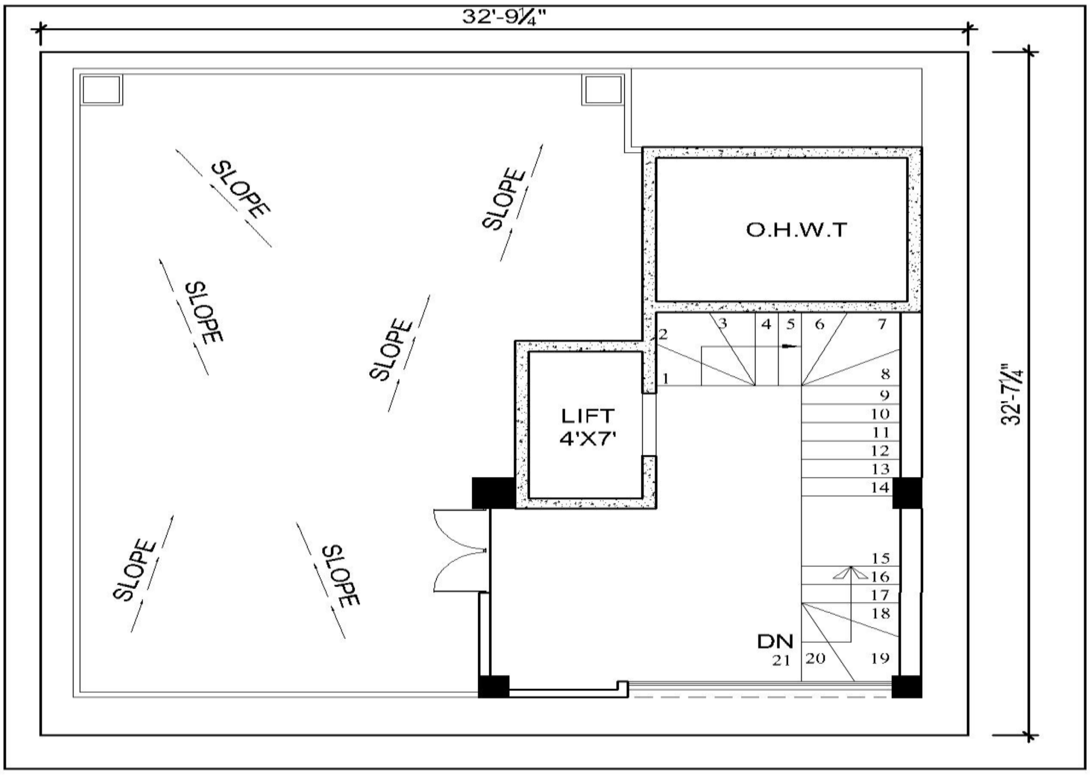


Fig 3.9 Roof of building

3.2.4 Design specification & properties of the structure

The initial sizing of members and specifications of the frame building are shown in Table 3.2. Theinitial sizes of member were checked against the conditions according to serviceability limit stateand ultimate limit state. The sizes were adjusted until the conditions of serviceability limit state and ultimate limit state stated in ACI318-14 were satisfied.

Structural Elements		**Dimensions(Back)**	**Dimensions(Front)**
Columns	Ground floor	12x18 inch & one 18x18 inch	12x12 inch
	1^{th} to 4^{th}	12x18 inch & one 18x18 inch	12x12 inch
Beams	Tie Beam(plinth)	12x18 inch	12x18 inch
	Ground to 4^{th}	12x18 inch	12x18 inch
Slab		6 inch THK.	
No. of stories		5 stories	
Beam to column connection = fixed			
Column to base connection = fixed			

Table 3.1 Initial sizes and specification of the building according to ACI-318 code

Building usage	**Shops**	
Story height	Ground floor	16 ft.
	1^{st} 2^{nd} 3^{rd} & 4^{th}	10 ft.
Length of building		30 ft.
Width of building		30 ft.
Height of building		56 ft.

Table 3.2 Design input detail of the building

3.3 Material properties

Every material has different properties that are simply of their own. Similarly, the material used inthe design of the structure in this research also has different properties and strength. Table 3.4, 3.5list the material properties applied in the preliminary analysis of the design of the structural members (beams, slabs and columns, etc.) The values of compressive strength of concrete, yieldstress of reinforcement, concrete density and modulus of elasticity conforms to ACI 318.

Structure Elements	Parameters
Compressive strength of Slab; *f*cu	3000 psi
Compressive strength of Beam; *f*cu	3500 psi
Compressive strength of Column; *f*cu	4500 psi
Yield stress *f*y	60 Ksi

Table 3.3 Material Properties conform to ACI318 code

3.4 Seismic Zoning of Pakistan

As we are studying the structural dynamic behavior of R.C.C. structures so we must study about the seismic zoning of Pakistan, we must study the seismic history of Multan too, how much earthquake developed in Multan and in its outside region in different time in past.

3.4.1 Seismic provinces and area sources

For the study of earthquake hazard in Pakistan, the country was divided into 19 zones, including some portions of the neighboring countries of Afghanistan, Tajikistan, Iran, and India. The division of the region into these source zones is based on the seismicity, the fault systems, and the stress direction analysis. The division was also based on the data processing of the whole catalogue regarding the seismicity, depth, and the study of research papers. One of the basic principles for the zonation of a region is that the seismicity within a single zone remains uniform and homogeneous, even though this principle clearly is not always fulfilled as judged from the individual catalogue used in the study. The nineteen seismic zones are all having geometric shapes (polygons) and the coordinates of their corners are described below along with a Fig. 3.9

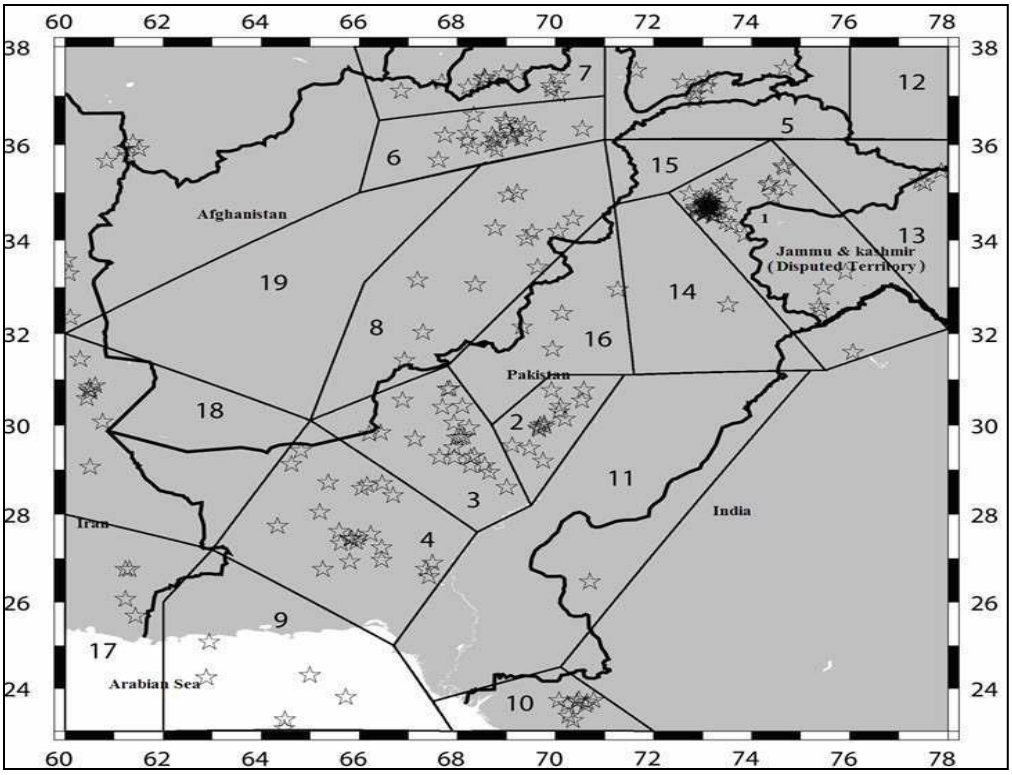


Fig 3.10 Seismic Zones of Pakistan with Seismic Intensity Showing with Stars

Source (Seismic Hazard Analysis and Zonation for Pakistan, Azad Jammu, and Kashmir by PMD & NORSAR JULY 2007)

3.4.2 Multan seismic zone

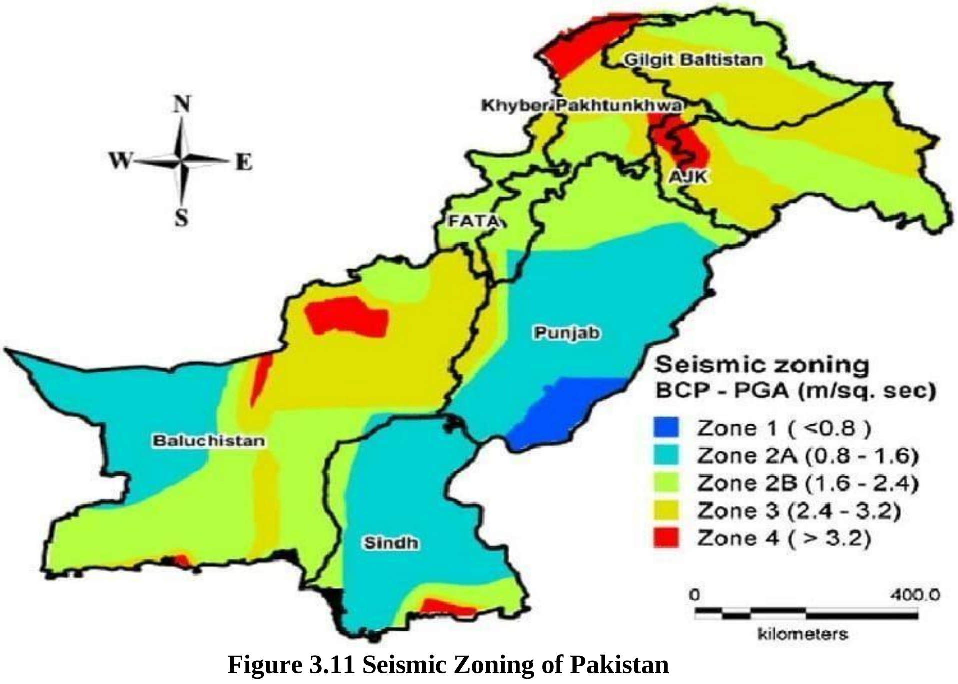


Figure 3.11 Seismic Zoning of Pakistan

Multan is a region of 2A Seismic zone that is peak ground acceleration (PGA) value is 0.8 To 1.6 m/sq.

3.5 Analysis of the Building

3.5.1 Model of the building

The model of the building was selected first as discussed above on which working was to be done. Dimensions & Drawing of the building was taken from the authorities and word was started.

3.5.2 Static Analysis of the building manually

First, static analysis of the building was made by hand. Slab was analyzed by the coefficient method to find the moments of the slab. Similarly beams and columns was also analyzed manually by hand according to ACI code. This analysis was made for a sole purpose to compare the manually result of the analysis with the result obtain from the ROBOT structural analysis software. In this way the accuracy of the software can be checked.

3.5.3 Static Analysis of the building by software

After the manually calculations of the building, the static analysis was performed on the ROBOT structural analysis software. FEM (Finite Element Method) method is used by the software for the analysis of the building. In this way the results obtained from the software were compared with the manually results.

3.5.4 Dynamic Analysis of the building by software

As dynamic analysis of a multi-story building is very complex and time-consuming process. So, we analyses the building dynamically (Seismic Analysis) with the help of ROBOT structural analysis software only. Our zone for the seismic analysis is Zone 2A & according to the soil conditions of the Multan, Pakistan. The code used by the software was UBC 97 international code for the seismic analysis of the building. In this way, we saved a lot of time and avoid our self from the complexity of dynamic analysis of the building.

3.5.5 Results of the analysis

After the analysis of the building that include both static analysis (both by hand and by software) and dynamic analysis the results was collected and was compared to check the accuracy of the results.

CHAPTER 4
RESULTS & DISCUSSION

4.1 Analysis of Slabs (Coefficient Method)

Calculations for the slab/panel no. 173

Slab Thickness

Dimension of the slab = 13’-$1_{1/2}$” x 18’

Assume trial slab thickness = Perimeter /180

= 2(13.125 + 18)/180 = 0.35 ft. = 4.15 in (We will take Slab thickness 6”)

m = la/lb = 13.125/18 = 0.73 > 0.5 (So, it is a two-way slab)

β = 1/m = 1/0.73 = 1.37

Load Calculations

Dead Load = D.L. = 6”/12*150 + 30 (Finishes) + 20 (Piping/Ducting/Plumbing)

= 125 psf = 0.125 ksf

Live Load = L.L. = 60 psf = 0.060 ksf (Commercial Building)

Now,

(Wu) d.l = 1.2 (D.L) = 0.15 ksf

(Wu) l.l = 1.6 (L.L) = 0.096 ksf

(Wu) d + l = 1.2 (D.L) + 1.6 (L.L) = 0.15 + 0.096 = 0.246 ksf

Moment Calculations

Short direction (Discontinuous edge)

M1 = M2/3 = 1/3 (M+ve) = (23.65/3) =7.88 kip-in = 0.65 kip-ft

For Midspan

M2 = [(Ca) d.l x (Wu) d.l + (Ca) l.l x (Wu) l.l] x La^2

= [(0.043) x 0.150 + (0.052) x 0.096] x $(13.125)^2$ = 1.97 kip-ft. = 23.65 kip-in

Short direction (Continuous edge)

M3 = (Ca) neg. x (Wu) x $La^{2\,=}$ (0.076) x 0.246 x $(13.125)^2$ = 3.22 kip=ft. = 38.64 kip-in

Long direction (Discontinuous edge)

M4 = M5/3 = 1/3 (M+ve) = (13.55/3) = 4.52 kip-in = 0.38 kip-ft.

For Midspan

M5 = [(Cb) d.l x (Wu) d.l + (Cb) l.l x (Wu) l.l] x Lb^2

= [(0.013) x 0.150 + (0.016) x 0.096] x $(18)^2$ = 1.13 kip-ft. = 13.55 kip-in

Long direction (Continuous edge)

M6 = (Cb) neg. x (Wu) x Lb^2

(0.024) x 0.246 x $(18)^2$ = 1.91 kip=ft. = 22.95 kip-in

Calculations for the slab/panel no. 172

Slab Thickness

Dimension of the slab = 13'-$1_{1/2}$" x 8'

Assume trial slab thickness = Perimeter /180

= 2(13.125 + 8)/180 = 0.24 ft. = 2.15 in (We will take Slab thickness 6")

m = la/lb = 8/13.125 = 0.61 > 0.5 (So, it is a two way slab)

β = 1/m = 1/0.61 = 1.64

Load Calculations

Dead Load = D.L. = 6"/12*150 + 30 (Finishes) + 20 (Piping/Ducting/Plumbing)

= 125 psf = 0.125 ksf

Live Load = L.L. = 60 psf = 0.060 ksf (Offices)

Now,

(Wu) d.l = 1.2 (D.L) = 0.15 ksf

(Wu) l.l = 1.6 (L.L) = 0.096 ksf

(Wu) d + l = 1.2 (D.L) + 1.6 (L.L) = 0.15 + 0.096 = 0.246 ksf

Moment Calculations

Short direction (Discontinuous edge)

M3 = M2/3 = 1/3 (M+ve) = (11.04/3) =3.68 kip-in = 0.31 kip-ft

For Midspan

M2 = [(Ca) d.l x (Wu) d.l + (Ca) l.l x (Wu) l.l] x La^2

= [(0.053) x 0.150 + (0.067) x 0.096] x $(8)^2$ = 0.92 kip-ft. = 11.04 kip-in

Short direction (Continuous edge)

M1 = (Ca) neg. x (Wu) x La^2 = (0.089) x 0.246 x $(8)^2$ = 1.40 kip-ft. = 16.81 kip-in

Long direction (Discontinuous edge)

M6 = M5/3 = 1/3 (M+ve) = (3.96/3) = 1.32 kip-in = 0.109 kip-ft.

For Midspan

M5 = [(Cb) d.l x (Wu) d.l + (Cb) l.l x (Wu) l.l] x Lb^2

= [(0.007) x 0.150 + (0.009) x 0.096] x $(13.125)^2$ = 0.33 kip-ft. = 3.96 kip-in

Long direction (Continuous edge)

M4 = (Cb) neg. x (Wu) x Lb^2

(0.011) x 0.246 x $(13.125)^2$ = 0.47 kip=ft. = 5.60 kip-in

4.2 Analysis of Beam

Calculations for the beam no. 160

Beam Dimensions

Assume beam of dimensions 12" x 18"

Load Calculations

Beam own weight = 12" x 18"/144 = 225 lb/ft = 0.225 k/ft.

Slab own weight

For Trapezoidal Section = Wlx/2 x (1-1/2β)

=0.125(13.125)/2 x (1-1/2(1.37)) = 0.521 k/ft.

Wall Load = 600 lb/ft = 0.600 k/ft

D.L = 0.225 + 0.521 + 0.600 = 1.346 k/ft

L.L = = 0.060(13.125)/2 x (1-1/2(1.37)) = 0.250 k/ft.

W_u = 1.2 (D.L) + 1.6(L.L) = 2.015 k/ft

For simply fixed beam

Bending Moments

Minimum thickness = H_{min} = L/16 = 18/16 = 13.5"

Maximum (+ve) bending moment = $M_u = W_u l^2/16$

= $2.015(18)^2/16$ = 40.80 k-ft = 489.64 k-in

Maximum (-ve) bending moment = $M_u = W_u l^2/16$

= $2.015(18)^2/11$ = - 59.35 k-ft = - 712.21 k-in

Shear Force

Maximum (+ve) Shear Force = $R_u = +W_u/2$

= + 2.015/2 = + 18.135 kip

Maximum (-ve) Shear Force = $R_u = -W_u/2$

= - 2.015/2 = - 18.135 kip

Calculations for the beam no. 159

Beam Dimensions

Assume beam of dimensions 12” x 18”

Load Calculations

Beam own weight = 12” x 18”/144 = 225 lb/ft = 0.225 k/ft.

Slab own weight

For Triangular Section = Wlx/3

=0.125(13.125)/3= 0.546 k/ft.

Wall Load = 600 lb/ft = 0.600 k/ft

D.L = 0.225 + 0.546 + 0.600 = 1.371 k/ft

L.L = = 0.060(13.125)/3 = 0.262 k/ft.

W_u = 1.2 (D.L) + 1.6(L.L) = 2.064 k/ft

For simply fixed beam

Bending Moments

Minimum thickness = H_{min} = L/16 = 13.126/16 = 9.85”

Maximum (+ve) bending moment = $M_u = W_u l^2/16$

= $2.064(13.125)^2/16$ = 24.22 k-ft = 290.64 k-in

Maximum (-ve) bending moment = $M_u = W_u l^2/16$

= $2.064(13.125)^2/11$ = - 34.35 k-ft = - 412.2 k-in

Shear Force

Maximum (+ve) Shear Force = $R_u = +W_u/2$

= + 2.064/2 = + 13.54 kip

Maximum (-ve) Shear Force = $R_u = -W_u/2$

= - 2.064/2 = - 13.54 kip

4.3 Modelling procedure (Static Analysis)

Next assigning materials and loading to model will be same, as discussed below procedure:

4.3.1 Units

To set the desire units in the software we have to go in the “Tools” in the tool bars then select the option “Job Preferences” then selecting the option “Imperial”. By this way we can set the units to FPS (Foot Pound Second) System.

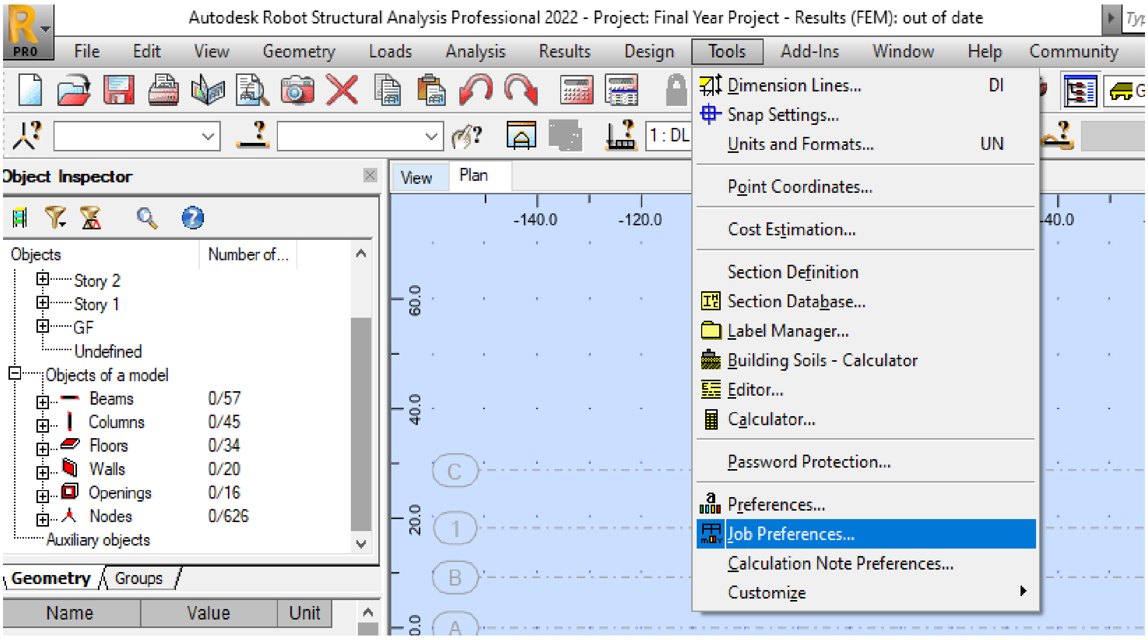


Fig 4.1 Setting Units

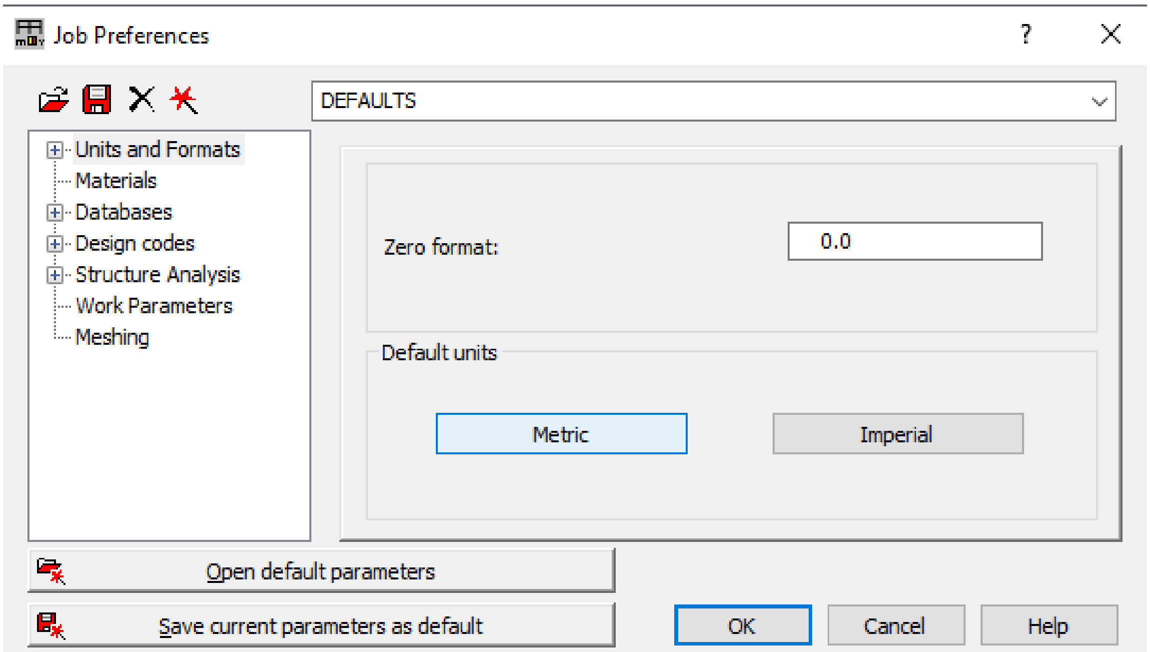


Fig 4.2 Setting Units

4.3.2 Axis Definition

Grids are the aligned reference lines onto which frame sections are subjected First of all, we must make gridlines. After setting the desire units, the next step is to set the grid lines for the required model. For this purpose go to “Geometry” in the Tool Bars and then select the option “Axis Definition” from the option given and the select the Cartesian in the structural axis. We can set the axis or grid space of our requirement by simply adding values in the X, Y & Z axis.

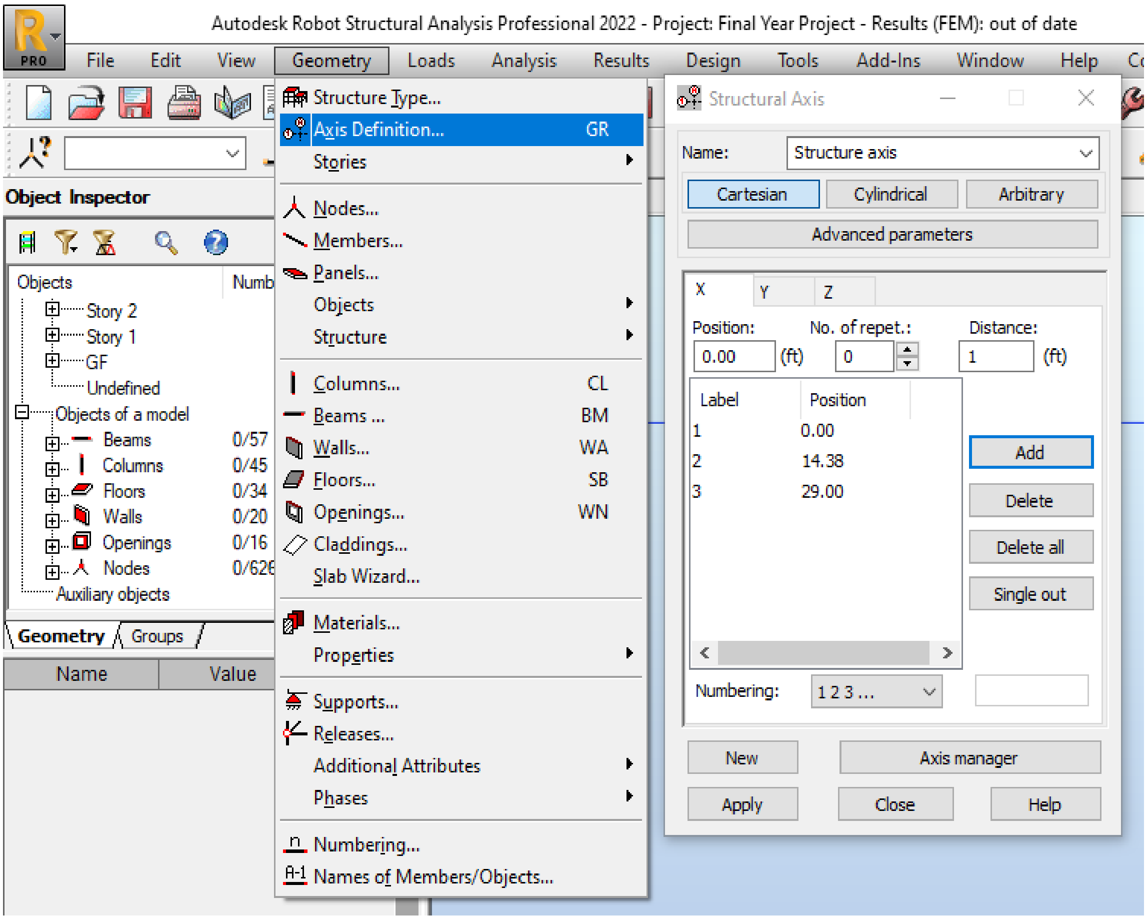


Fig 4.3 Setting Grid Lines/Axis Definition

4.3.3 Define Materials

Materials are defined first, and analysis runs according to specified material, as in our case using concrete of strength 3000psi for slab, 3500psi for beam & 4500 psi for column and steel of grade 60 as properties of materials is defined in ROBOT 2022.As shown in figure.

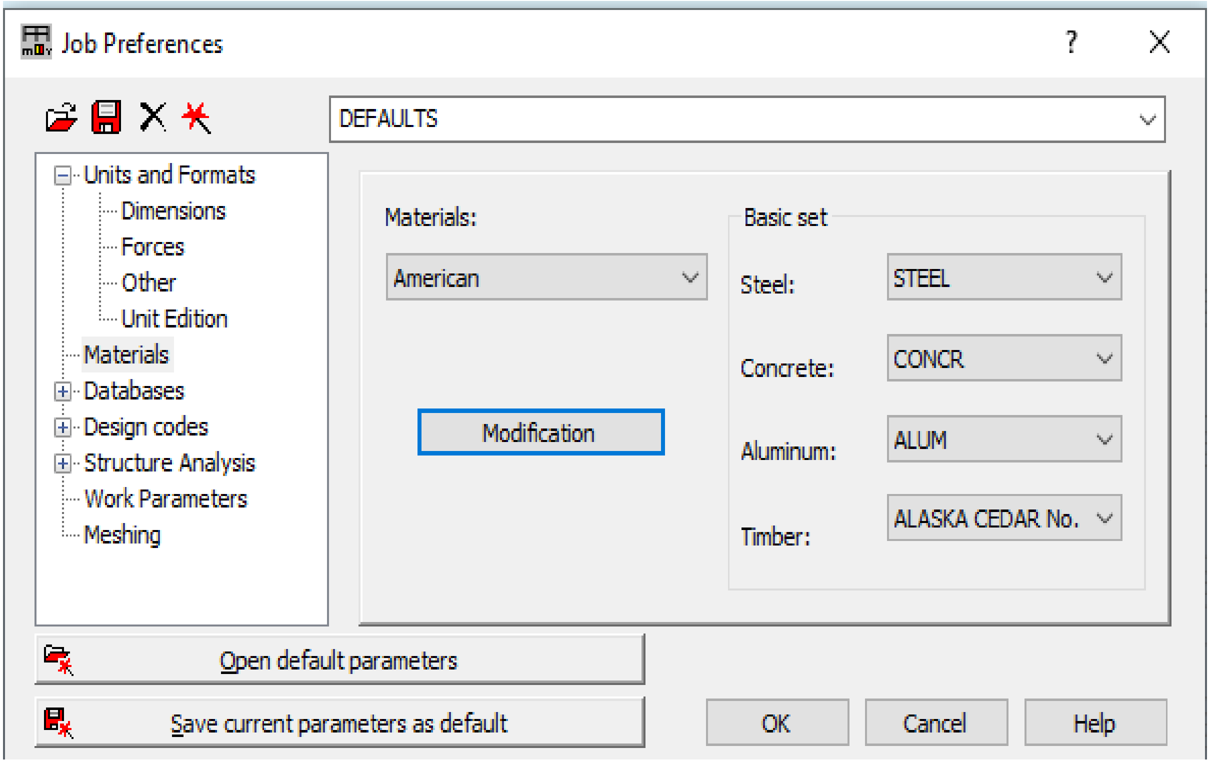


Fig 4.4 Defining Material Properties

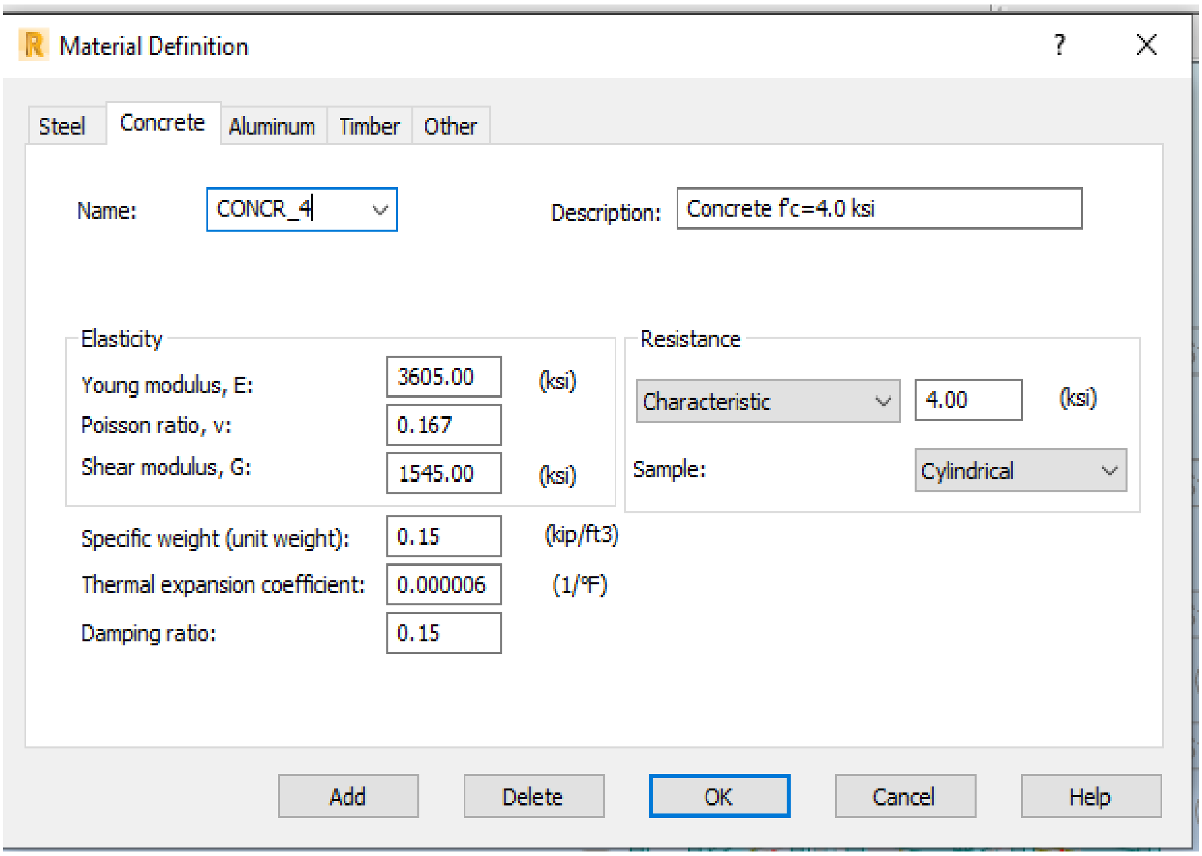


Fig 4.5 Defining Concrete Properties

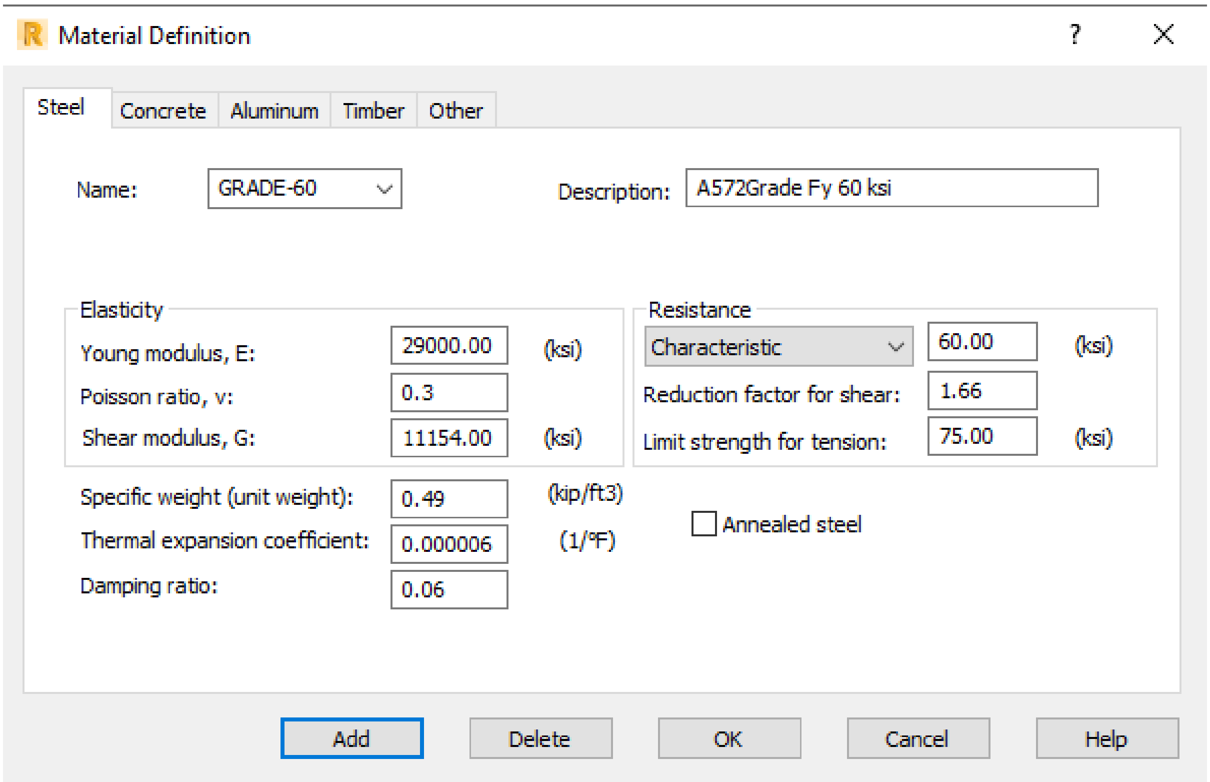


Fig 4.6 Defining Steel Properties

4.3.4 Material Properties

This section provides material property information for materials used in the model.

Member	Material strength F_{c}' (Psi)
Slab	3000
Beam	3500
Column	4500

Table 4. 1 Concrete Properties

Material type	Fy (ksi)	Young Modulus (Y)	Shear Modulus (G)
A572Gr60	60	29000 ksi	11154 ksi

Table 4. 2 Rebars/Steel Properties

4.3.5 Define Sections

According to the model there are different size of columns, beams, floors/slabs & opening. So, we introduced them by using the “Geometry” in the Tool Bars.

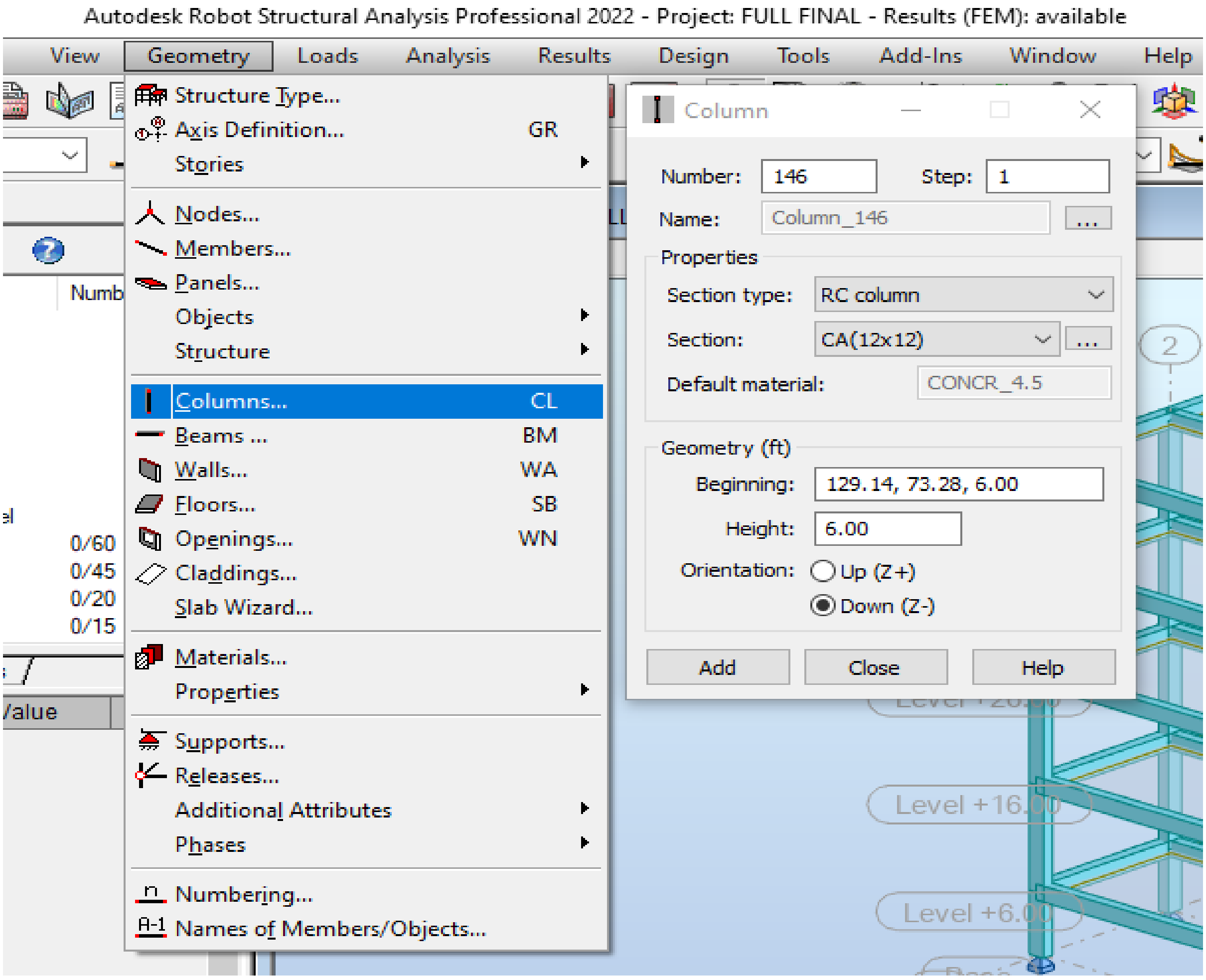


Fig 4.7 Defining Columns

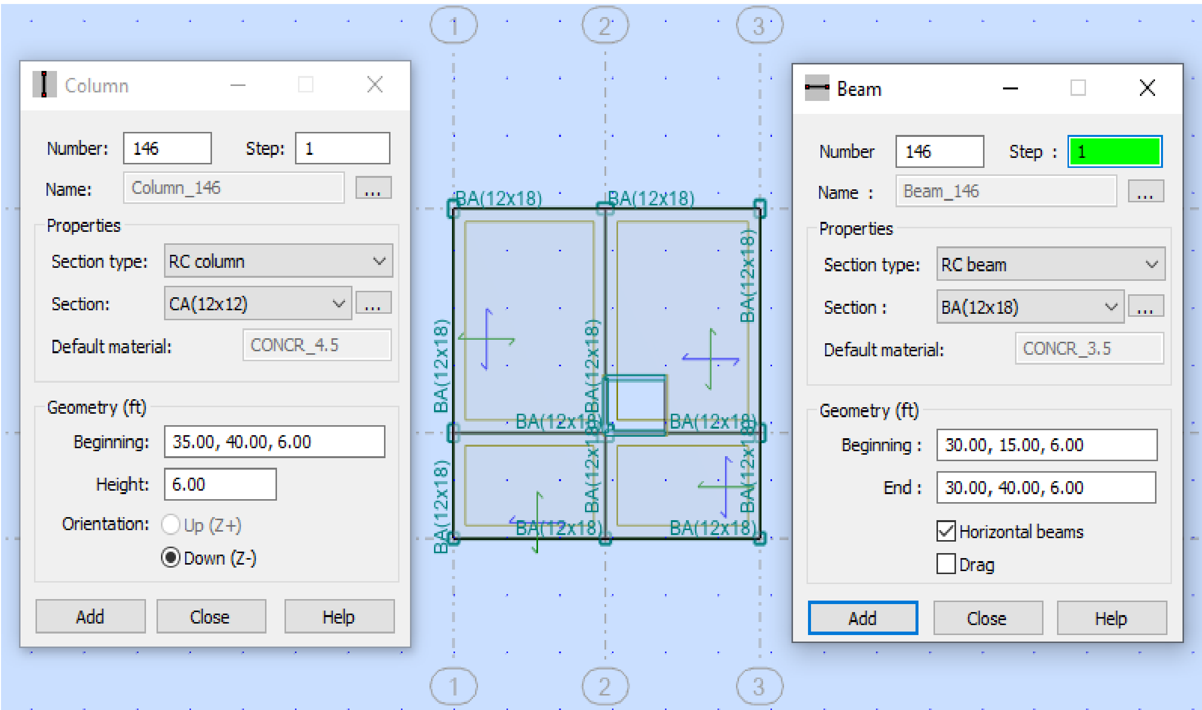


Fig 4.8 Dimensioning of Columns & Beams

As per our model dimensions, we set the columns to CA (12x12), CB (12x18) & CC (18x18). Similarly we set the beams of BA (12x18) & BB (18x18). Also floors/slabs of S1 (6”) shell slab and walls of S1 (6”) shear walls for the lift or shear wall.

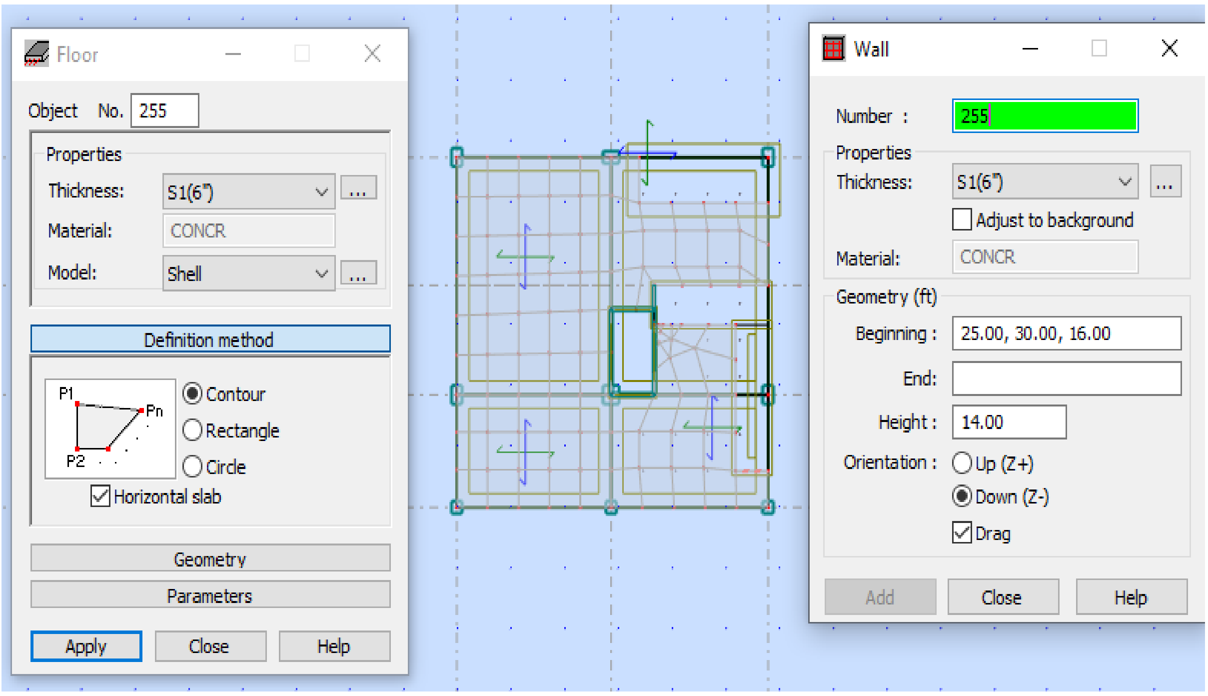


Fig 4.9 Dimensioning of Slabs & Walls

In the same way, using the “Geometry” Tool bars we set the required openings in the model. Opening is provide at the stairs & at the duct as per the given dimensions in the layout of the building for the analysis.

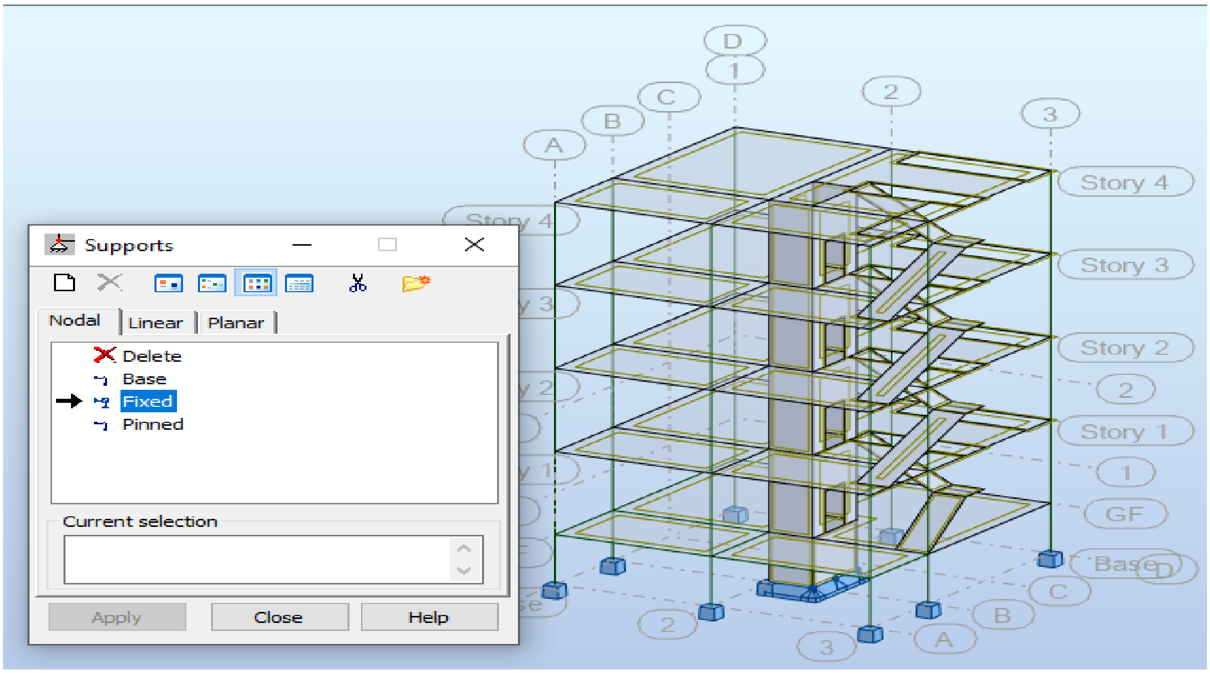


Fig 4.10 Supports

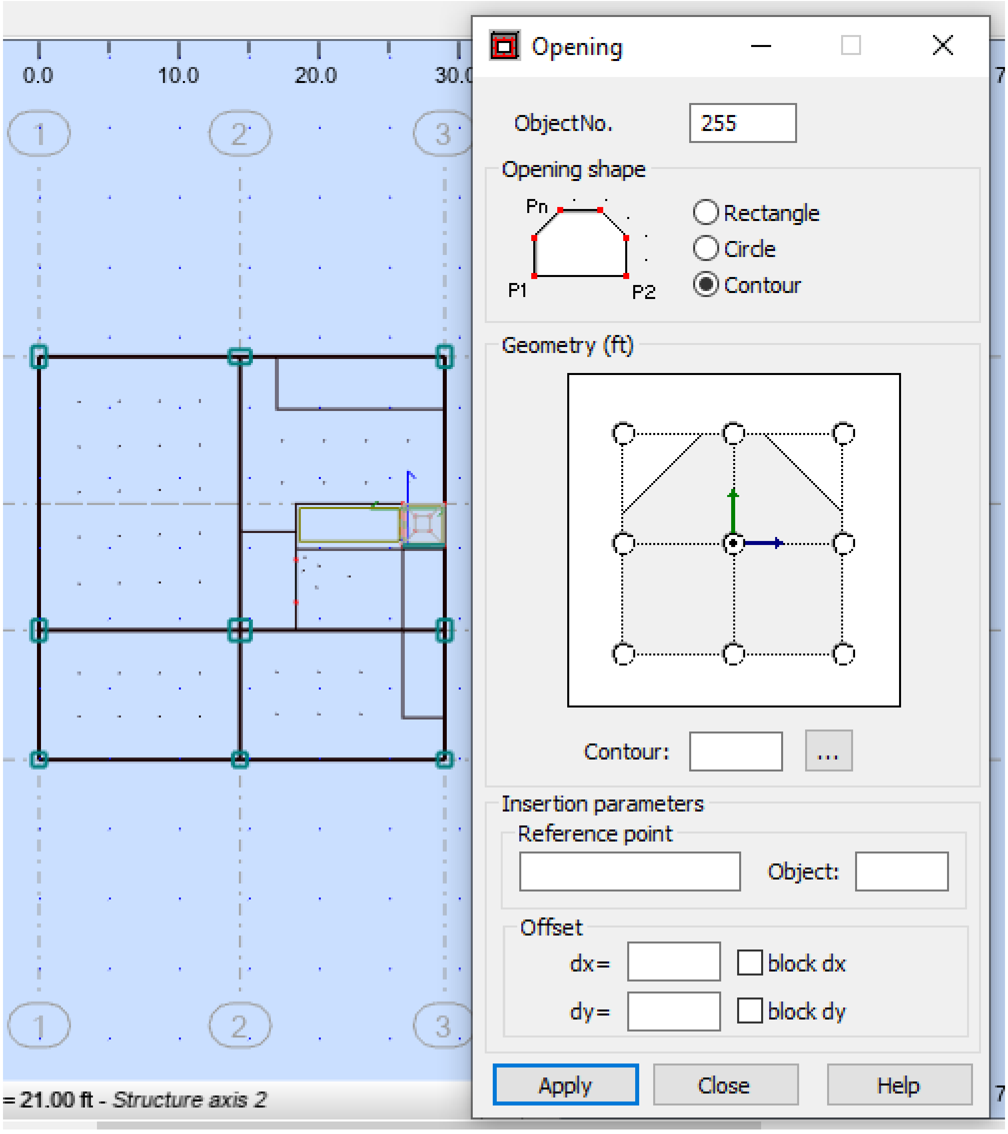


Fig 4.11 Dimensioning of Opening

4.3.6 Defining Loads

The next step is to define the loads for the static analysis of the structure. So, we define the following loads for the analysis. Self-Load, Dead Load & Live Load as shown in the figure below. At this stage we are carrying static analysis only, so we define these loads only now.

s

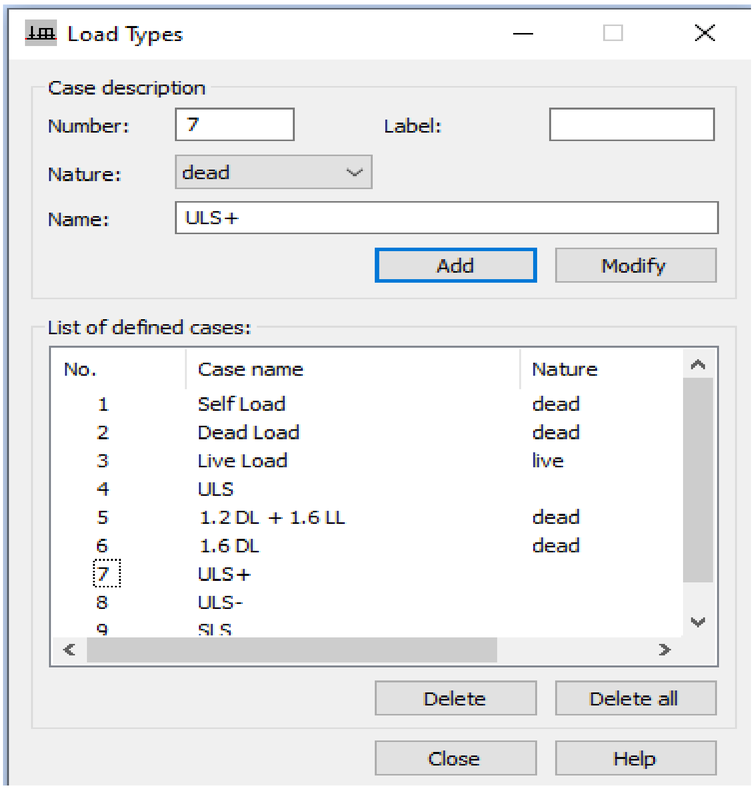


Fig 4.12 Defining Loads

4.3.7 Loads Combination

After defining loads, we make the loads combination for the analysis purpose. We make a combination of (1.2 D. L + 1.6 L. L) & one (1.6 D. L) by going in the "Loads" in the tool bars and selecting the "Manual Combination" as shown below.

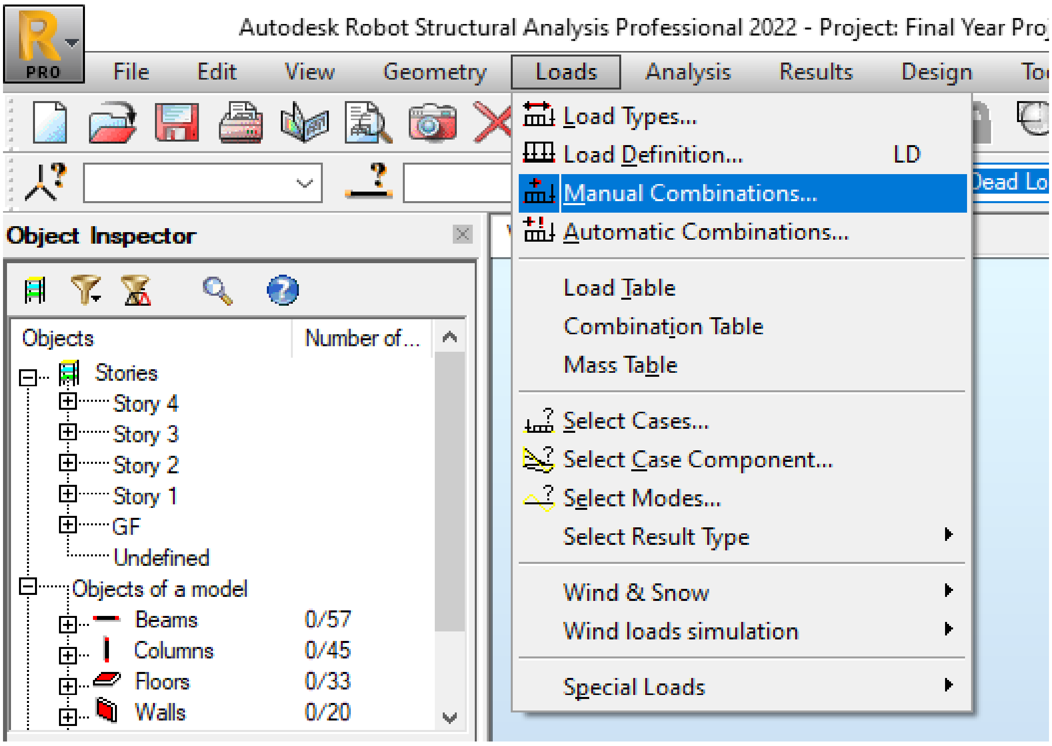


Fig 4.13 Loads Combination

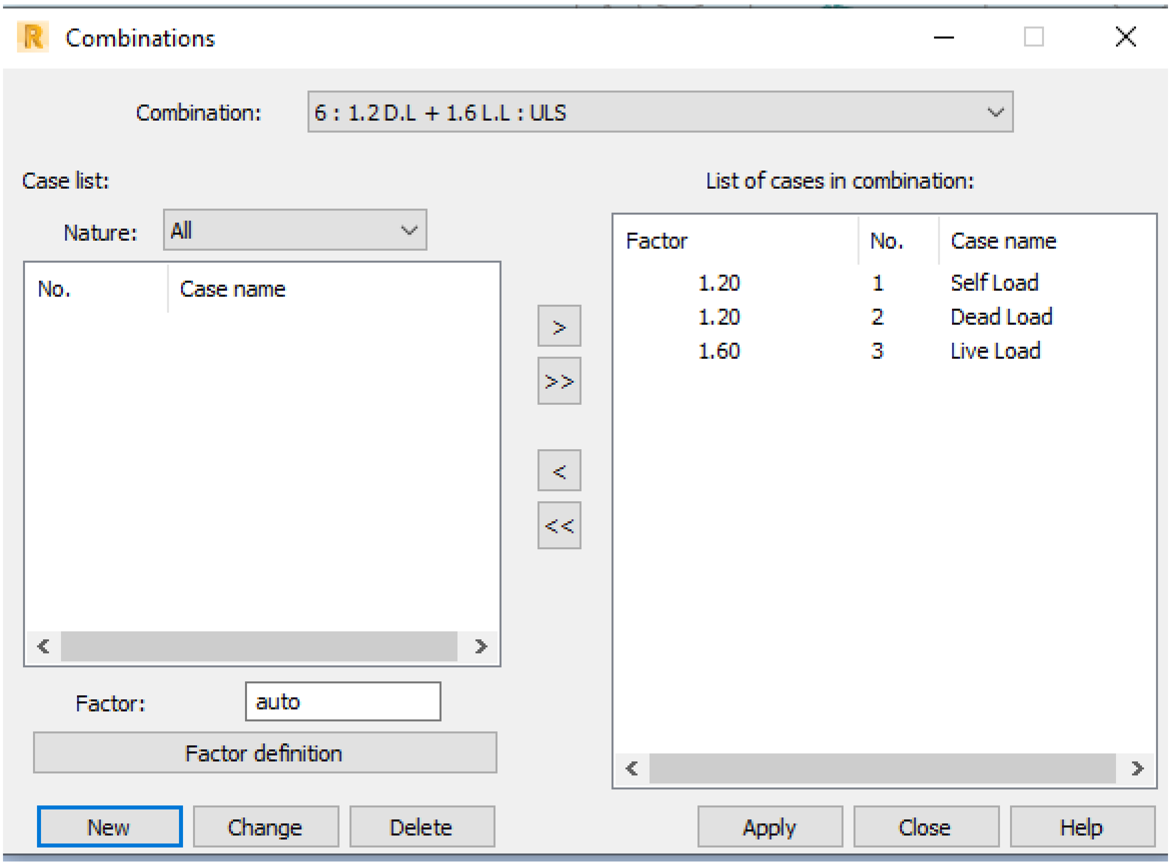


Fig 4.14 Loads Combination

4.3.8 Assigning Loads

Self-Load is taken automatically by the system. So, now we assigned the dead load of 0.050 kip/ft^2 on all the slabs.

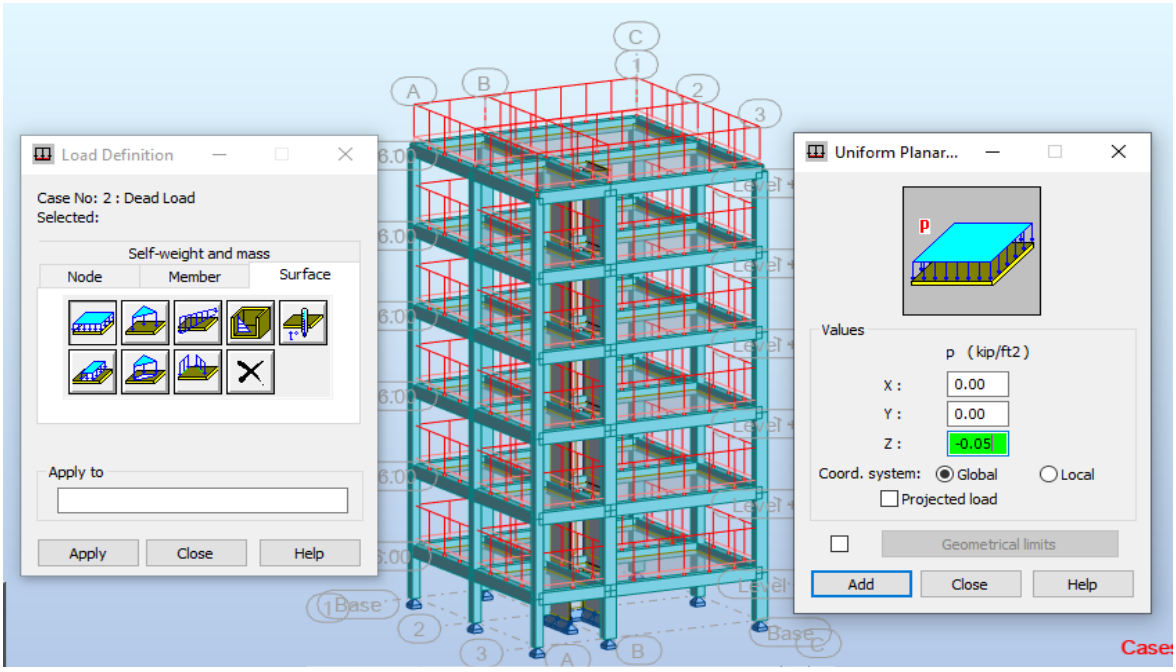


Fig 4.15 Assigning Dead Load to Slabs

After assigning Dead Load now we assigned the Live load of 0.060 kip/ft^2 on all the slabs. As for the commercial building Live Load is 0.060 kip/ft$^{2.}$

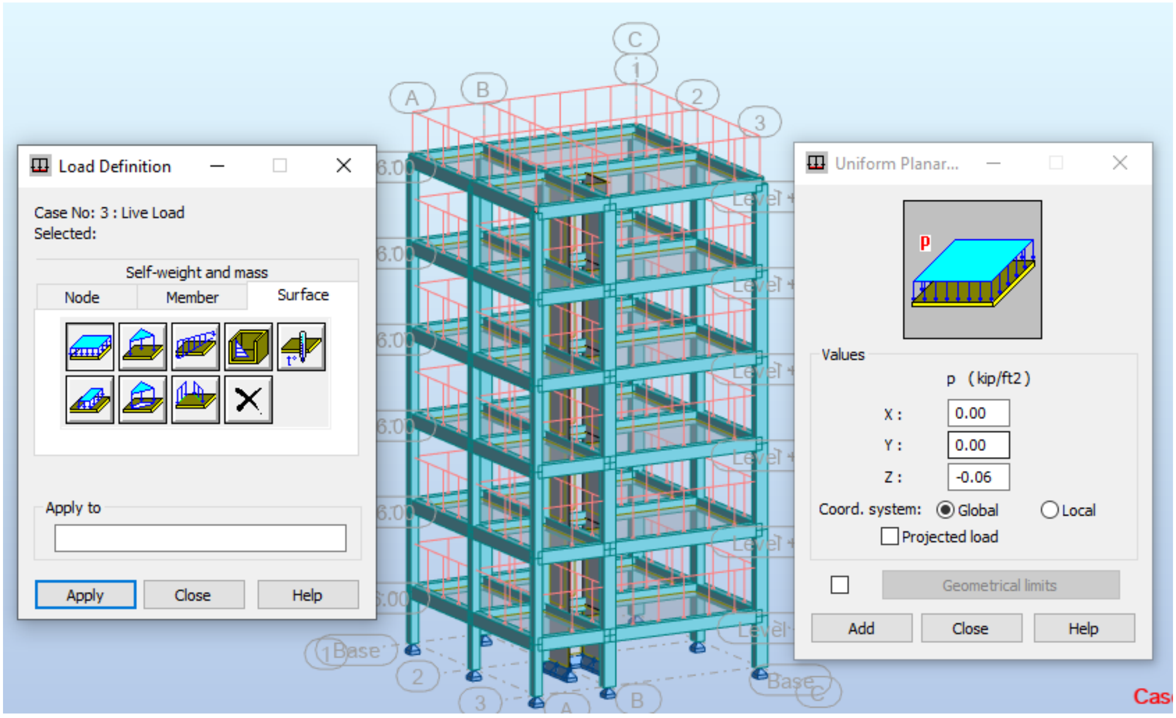


Fig 4.16 Assigning Live Load to Slabs

After assigning Live Load now we assigned the Dead load on the beams of 0.60 kip/ft. on all the beams.

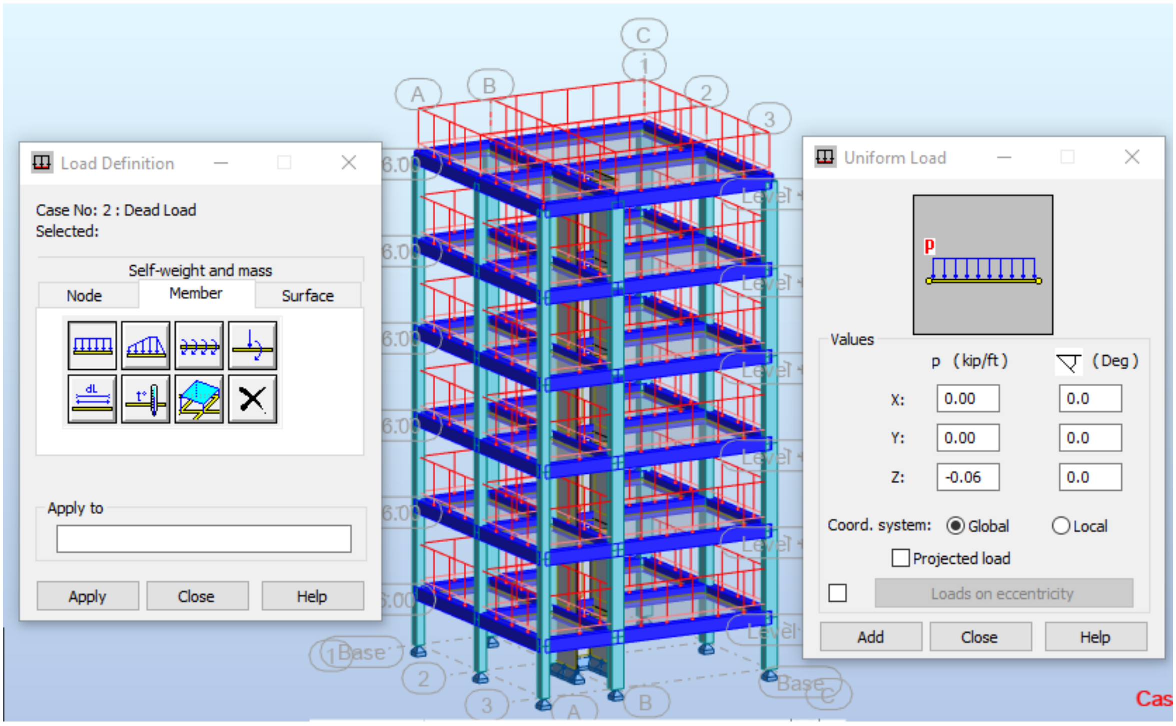


Fig 4.17 Assigning Dead Load on Beams

4.3.9 Calculations & Results

As our loads are assigned and combination is also assigned. So we will now calculate the results. For this purpose, we go in the Analysis and the select Calculations.

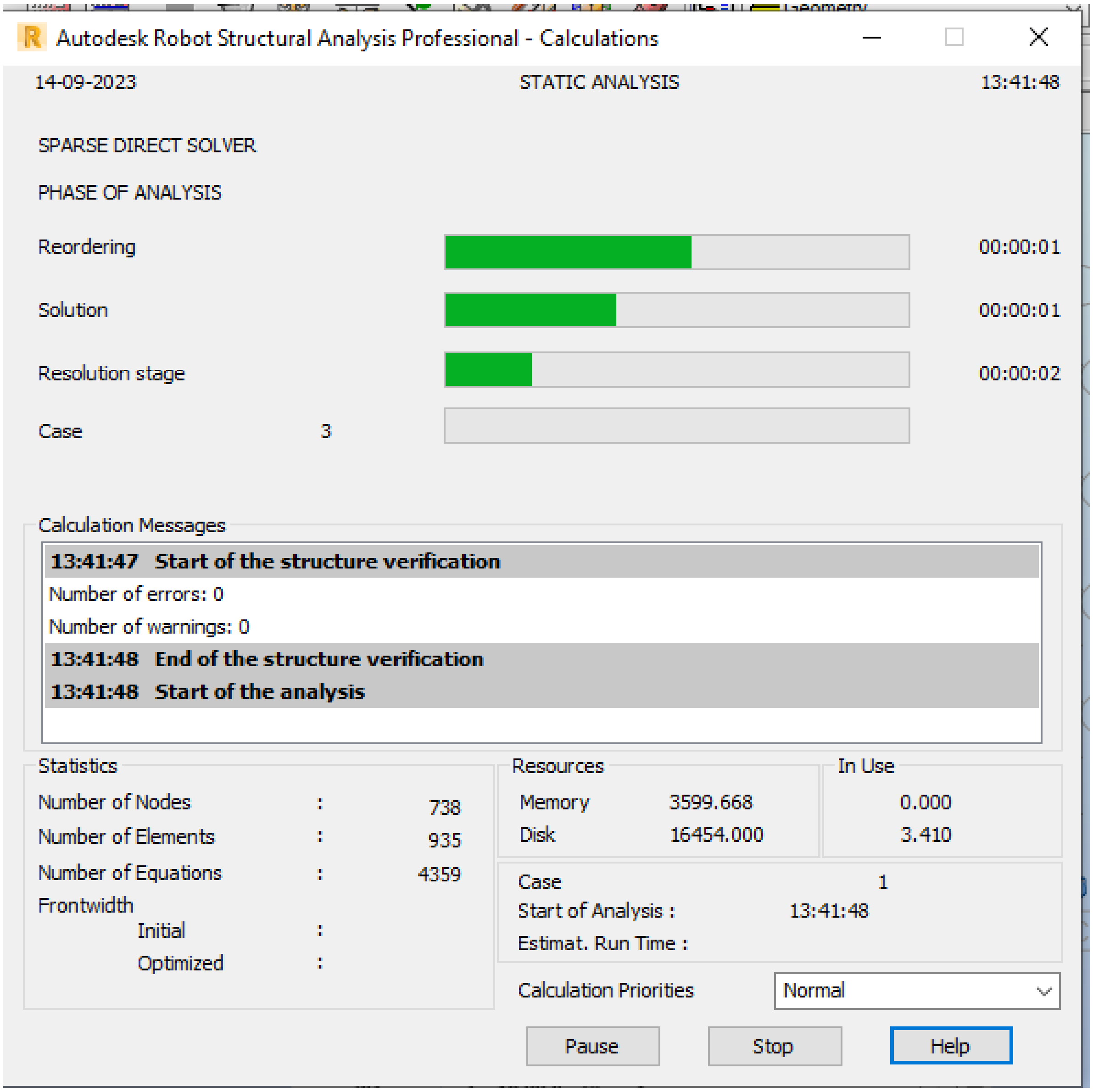


Fig 4.18 Calculations

The Shear force diagram, Reactions & Bending Moment Diagram obtained from the software results the for some Components is shown below.

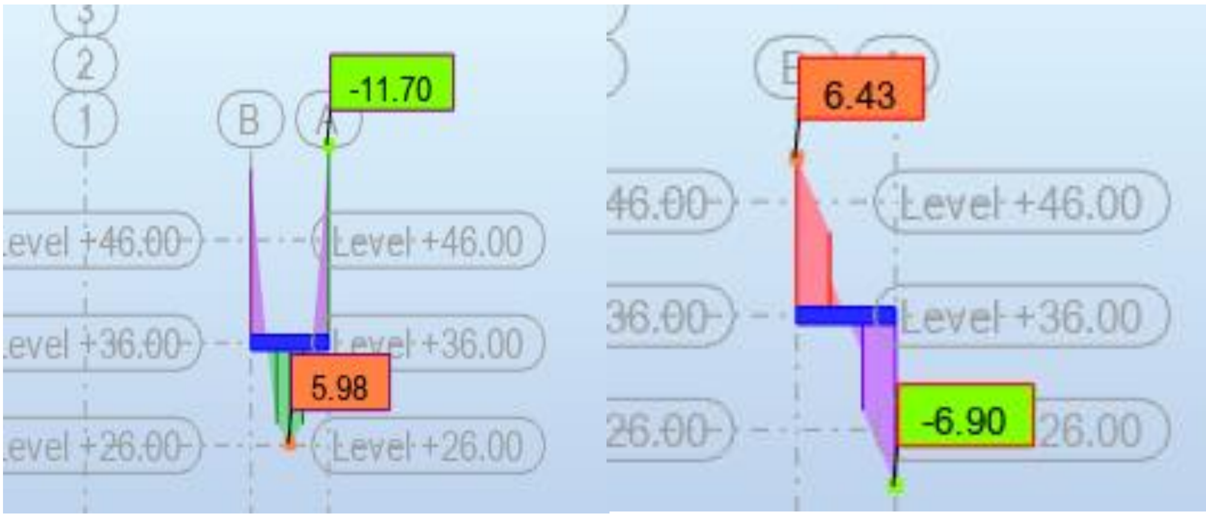


Fig 4.19 BMD & SFD for the beam 103

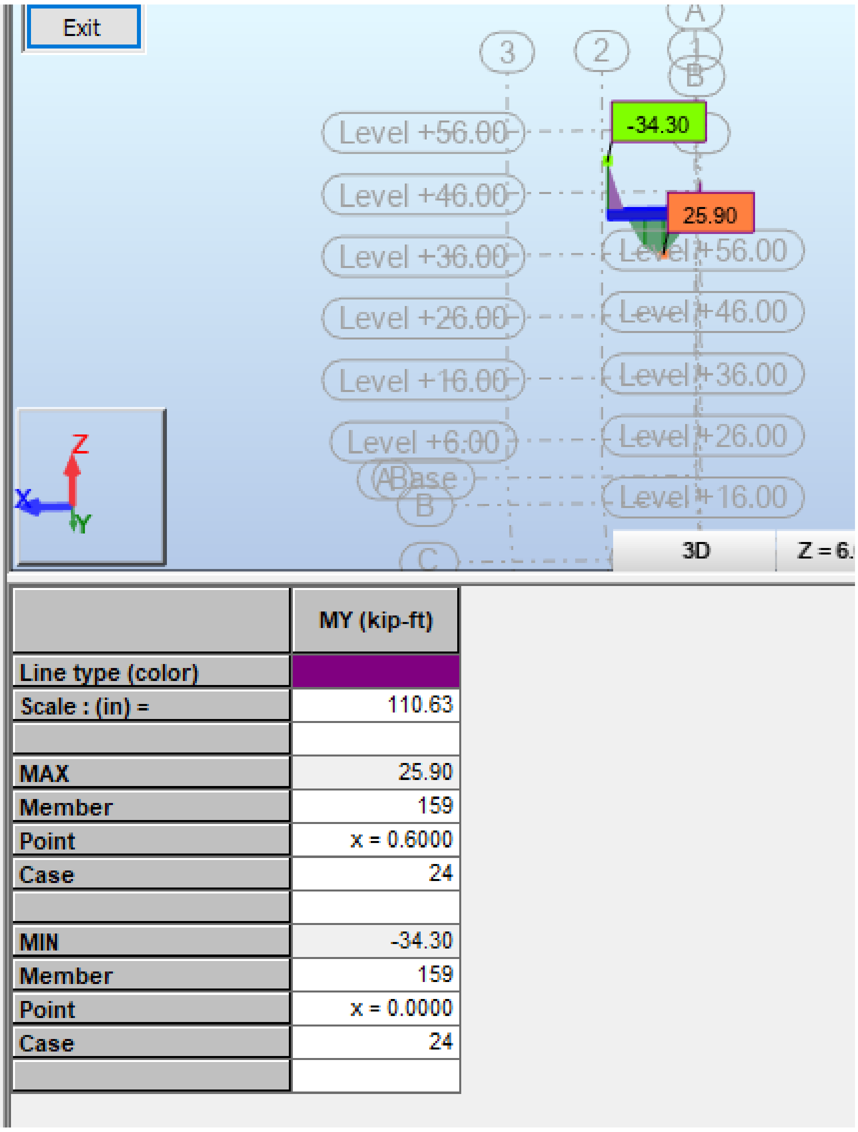


Fig 4.20 BMD for the beam 159

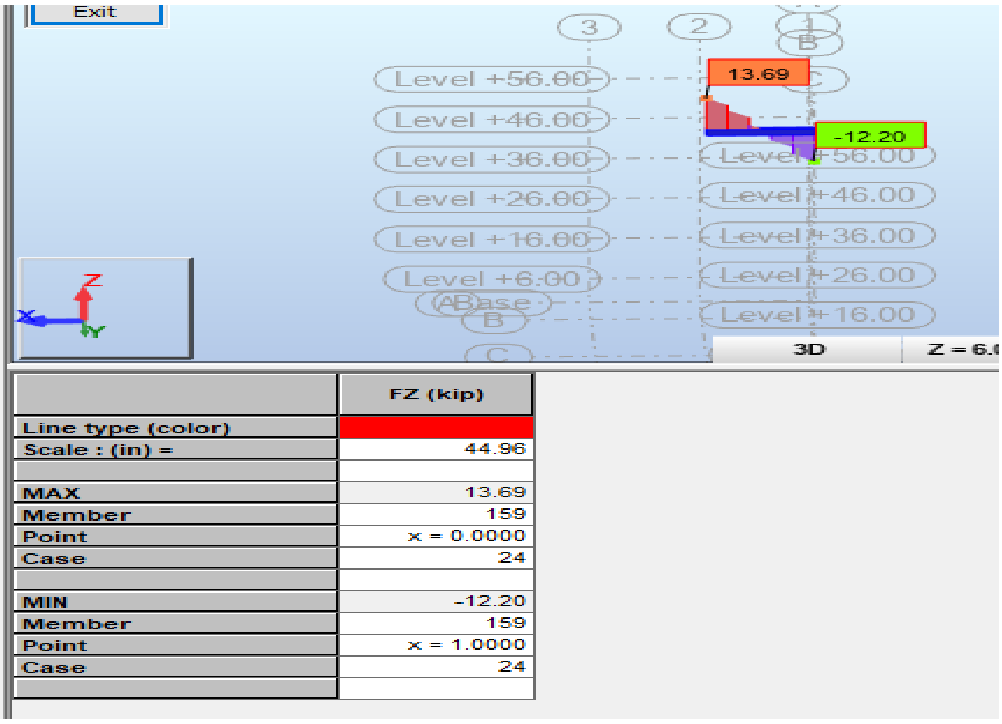


Fig 4.21 SFD for the beam 159

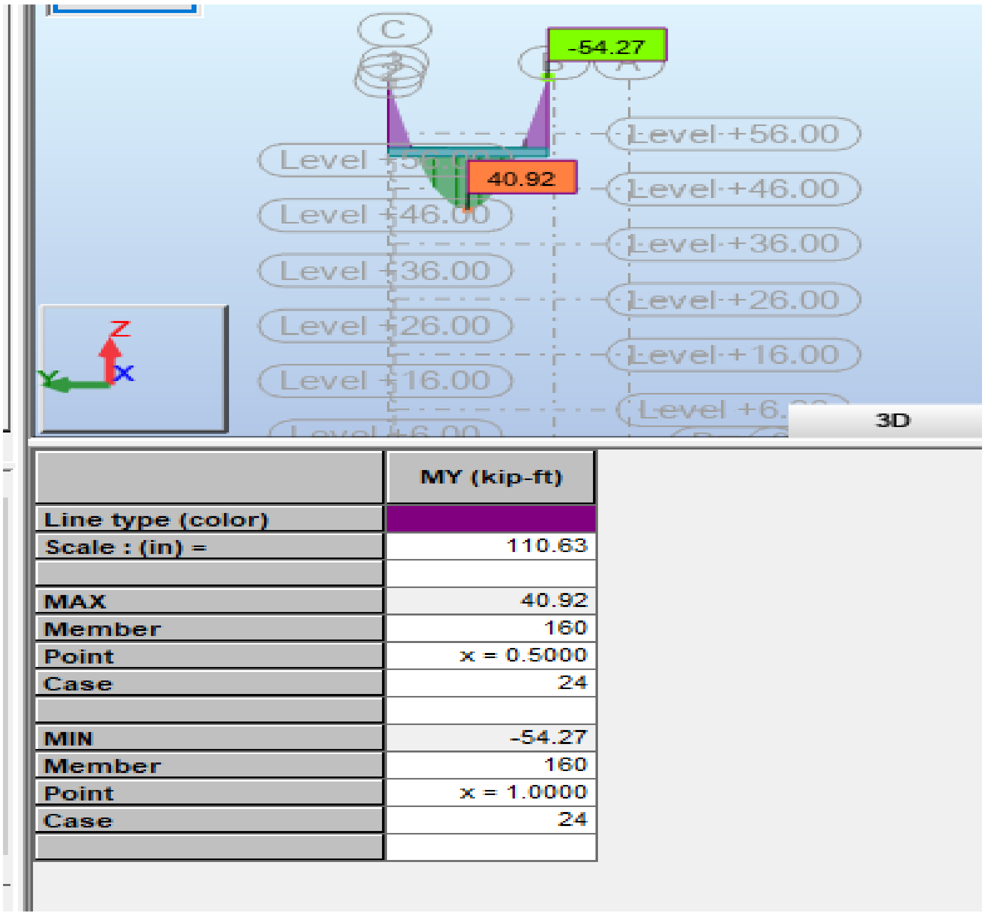
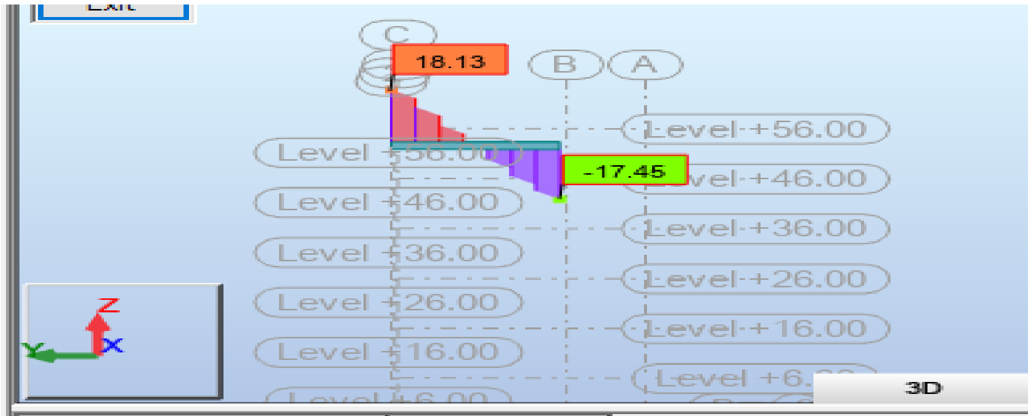


Fig 4.22 BMD for the beam 160



	FZ (kip)
Line type (color)	
Scale : (in) =	44.96
MAX	18.13
Member	160
Point	x = 0.0000
Case	24
MIN	-17.45
Member	160
Point	x = 1.0000
Case	24

Fig 4.23 SFD for the beam 160

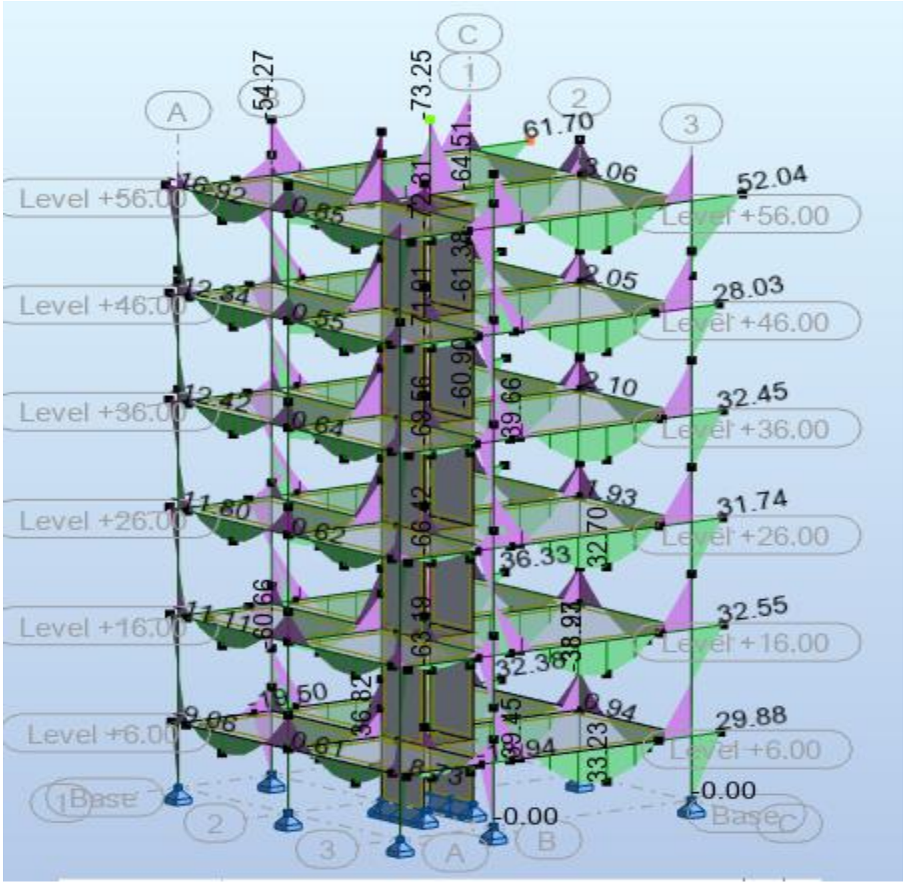


Fig 4.24 BMD for the beams

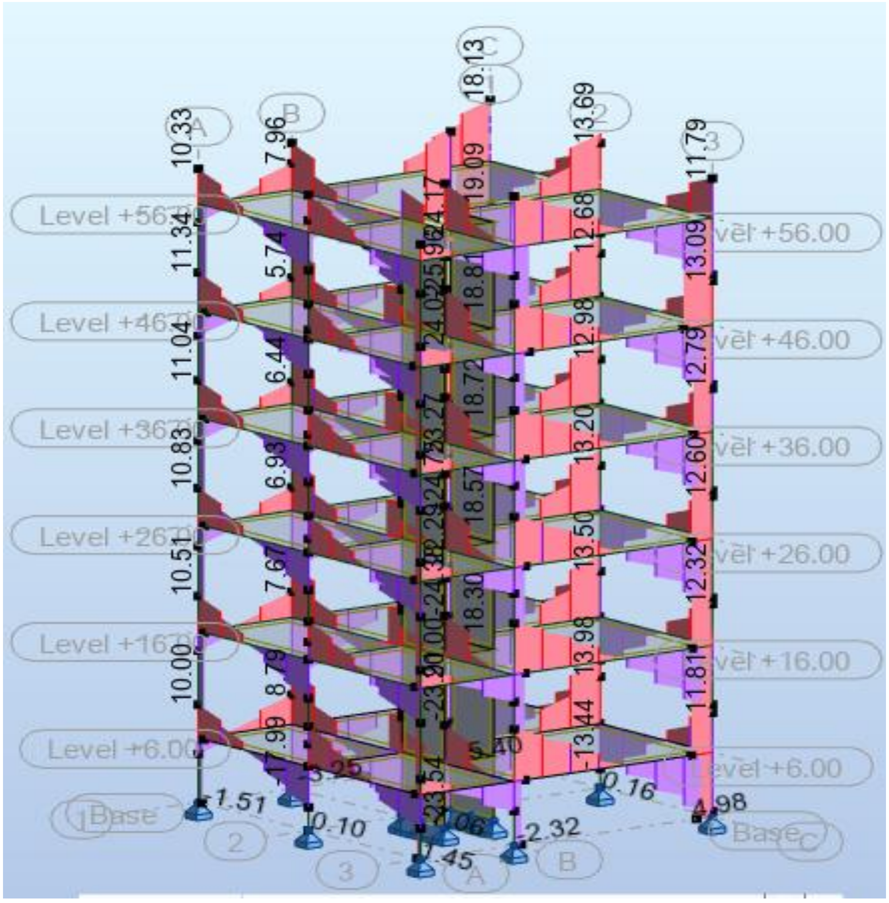


Fig 4.25 SFD for the beams

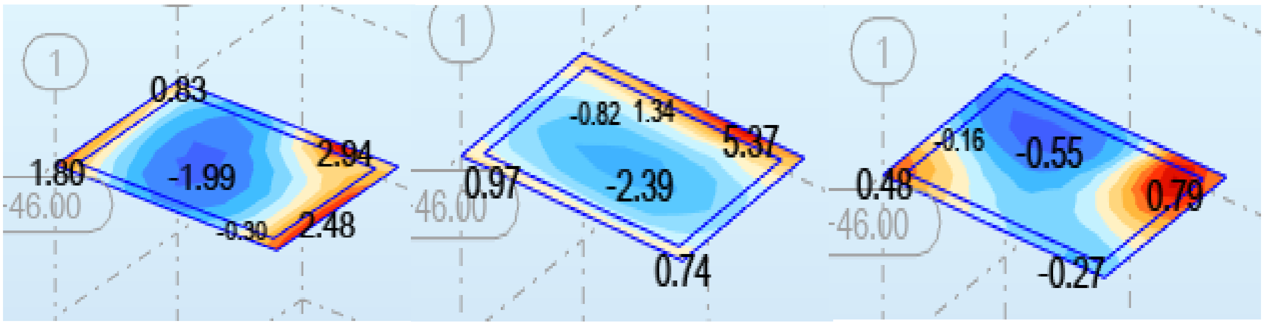


Fig 4.26 Moments for the Slab 173 in XX, YY & XY Direction

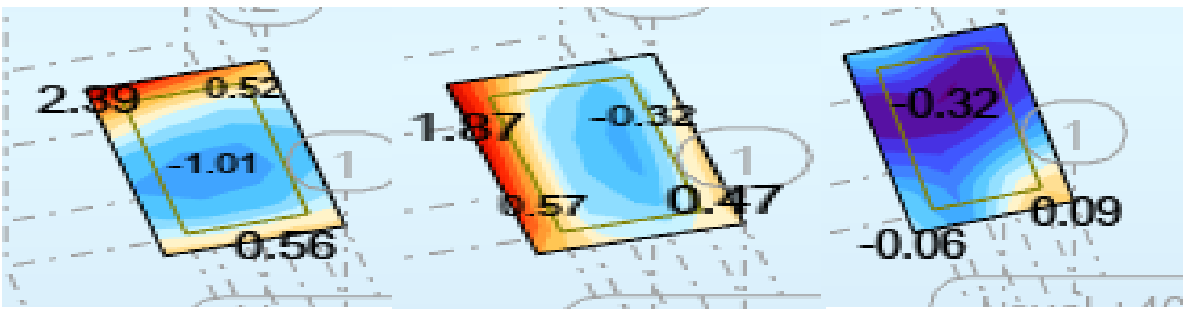


Fig 4.27 Moments for the Slab 172 in XX, YY & XY Direction

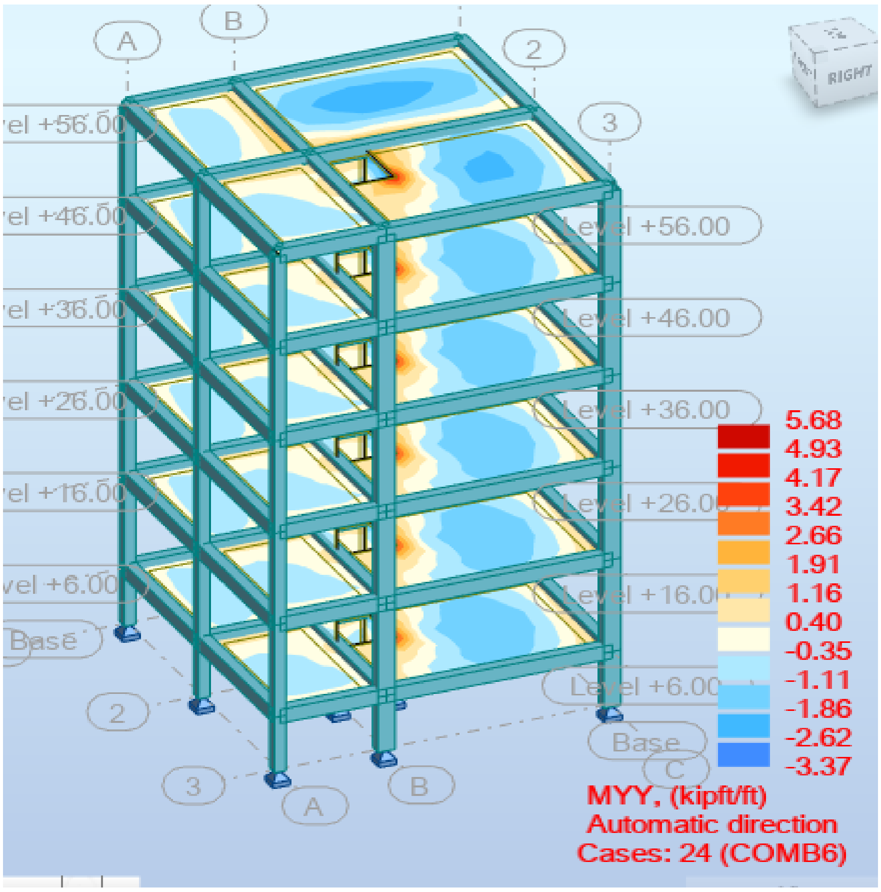


Fig 4.28 Moments for the Slabs in YY Direction

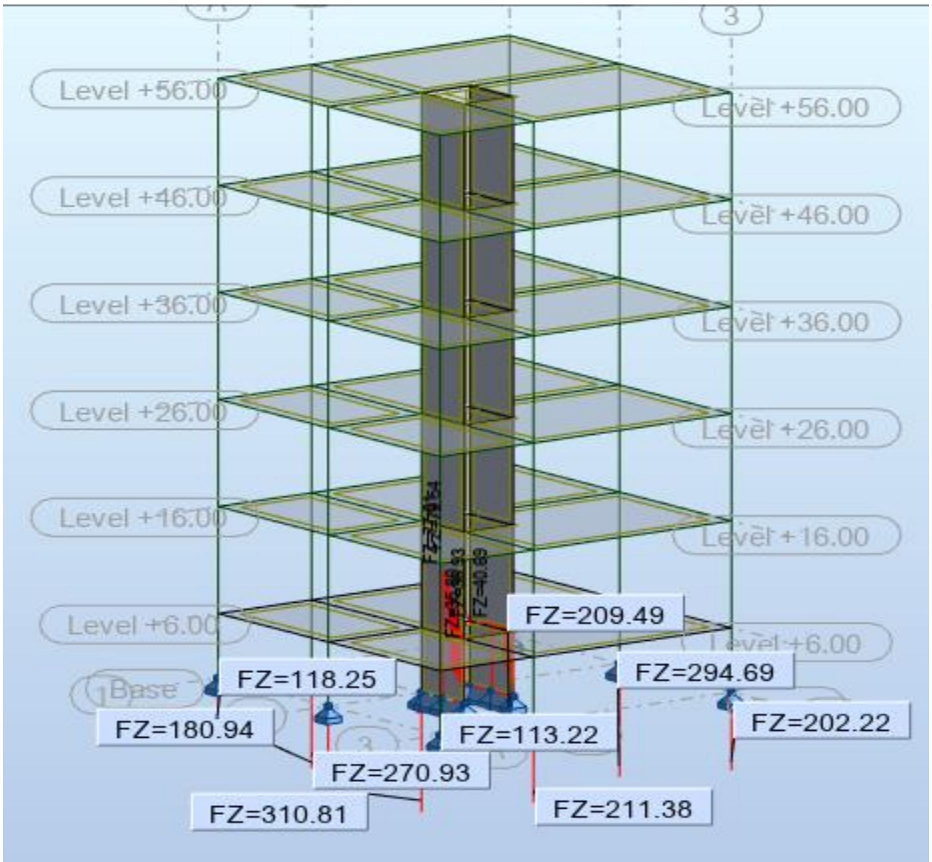


Fig 4.29 Reactions values

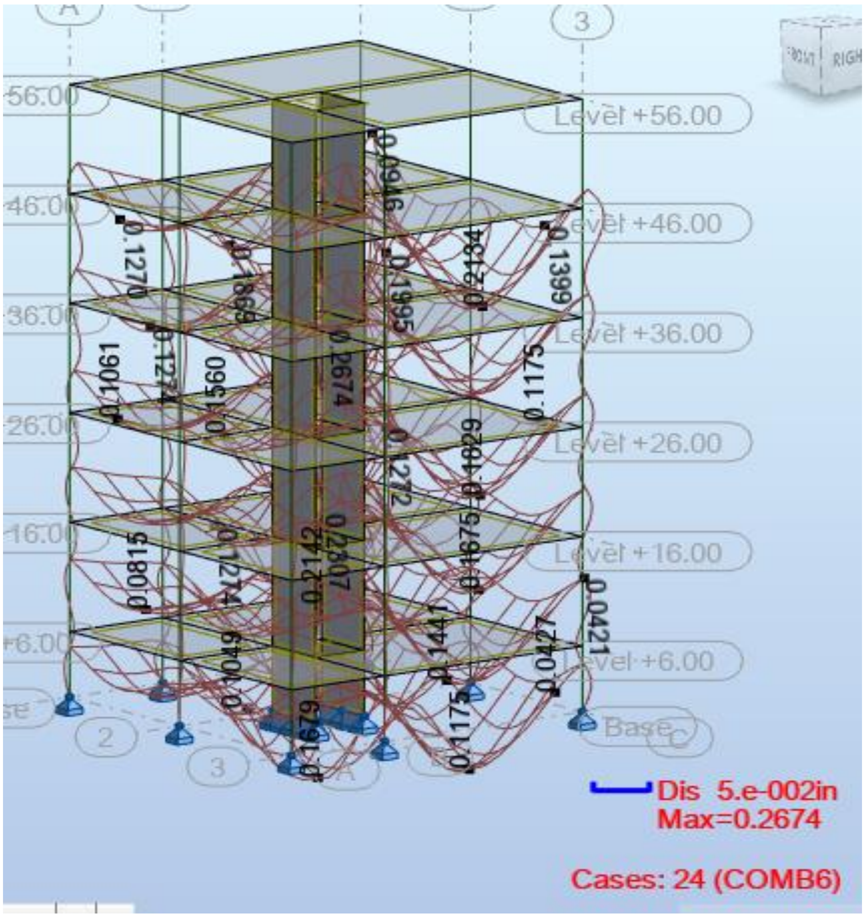


Fig 4.30 Deformations values

Node/Case	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
1/ 24 (C)	-1.45	0.32	113.22	0.0	0.0	0.0
3/ 24 (C)	-0.10	0.45	180.94	0.00	0.00	0.0
5/ 24 (C)	1.51	0.46	118.25	-0.00	0.0	0.0
7/ 24 (C)	-1.10	2.32	211.38	0.00	0.00	0.0
9/ 24 (C)	-2.17	-4.98	202.22	0.00	-0.00	0.0
11/ 24 (C)	-0.16	-3.88	294.69	0.00	0.0	0.0
13/ 24 (C)	2.41	-5.40	209.49	-0.00	0.00	0.0
15/ 24 (C)	2.91	3.25	270.93	-0.00	-0.00	0.0
17/ 24 (C)	21.00	20.65	310.81	0.00	-0.00	0.00
101/ 24 (C)	25.73	-20.53	96.21	-0.00	-0.00	-0.00
105/ 24 (C)	0.30	6.83	95.44	-0.00	-0.00	-0.00
722/ 24 (C)	-28.69	0.60	54.59	0.00	0.00	0.00
727/ 24 (C)	0.88	0.09	114.34	0.00	0.00	0.00
731/ 24 (C)	-25.37	-0.18	50.29	-0.00	0.00	-0.00
736/ 24 (C)	4.31	-0.01	108.20	-0.00	0.00	-0.00
Case 24 (C)	COMB6					
Sum of val.	0.00	-0.00	2431.01	-0.00	0.00	0.00
Sum of reac.	0.00	-0.00	2431.01	33883.12	-35266.04	-0.00
Sum of forc.	0.0	0.0	-2431.01	-33883.13	35266.04	0.0
Check val.	0.00	-0.00	0.00	-0.00	-0.00	-0.00
Precision	3.03792e-04	8.35314e-14				

Table 4.3 Reactions Results

	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
MAX	25.73	20.65	310.81	0.00	0.00	0.00
Node	101	17	17	17	722	727
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)
MIN	-28.69	-20.53	50.29	-0.00	-0.00	-0.00
Node	722	101	731	101	101	736
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)

Table 4.4 Reactions Results (Global Extreme)

	UX (in)	UY (in)	UZ (in)	RX (Rad)	RY (Rad)	RZ (Rad)
MAX	0.0056	0.0097	0.0	0.002	0.001	0.000
Node	998	999	1	717	636	819
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)
MIN	-0.0014	-0.0082	-0.2672	-0.002	-0.002	-0.000
Node	992	842	709	643	703	844
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)

Table 4.5 Displacement Results (Global Extreme)

	UX (in)	UY (in)	UZ (in)
MAX	0.0007	0.0117	0.0095
Member	169	151	169
Case	24 (C)	24 (C)	24 (C)
MIN	-0.0010	-0.0092	-0.1060
Member	166	150	160
Case	24 (C)	24 (C)	24 (C)

Table 4.6 Maximum Deflections Results (Global Extreme)

	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
MAX	294.69	6.58	19.09	8.31	61.70	29.87
Member	6	9	131	13	152	154
Node	11	17	70	10	998	1000
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)
MIN	-12.79	-10.71	-21.03	-9.89	-67.37	-26.40
Member	169	154	167	14	167	35
Node	1000	1000	1000	14	1000	12
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)

Table 4.7 Forces Results (Global Extreme)

	S max (ksi)	S min (ksi)	S max(My) (ksi)	S max(Mz) (ksi)	S min(My) (ksi)	S min(Mz) (ksi)	Fx/Ax (ksi)
MAX	2.09	1.36	1.25	0.81	0.00	0.00	1.36
Member	8	6	167	151	7	6	6
Node	16	11	1000	997	13	11	11
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)
MIN	0.00	-1.72	-0.00	-0.00	-1.25	-0.81	-0.06
Member	111	152	4	5	167	151	169
Node	58	998	7	9	1000	997	1000
Case	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)	24 (C)

Table 4.8 Stresses Results (Global Extreme)

	MXX (kipft/ft)	MYY (kipft/ft)	MXY (kipft/ft)
MAX	4.89	5.62	0.89
Panel	170	144	170
Node	1037	145	662
Case	24 (C)	24 (C)	24 (C)
MIN	-1.97	-3.34	-0.77
Panel	173	156	170
Node	709	1037	634
Case	24 (C)	24 (C)	24 (C)

Table 4.9 Slabs Results (Global Extreme)

4.4 Modelling procedure (Dynamic Analysis)

Next using UBC97 code for the seismic analysis of building, first we will define loads for the seismic analysis.

4.4.1 Defining Loads

The next step is to define the loads for the dynamic analysis of the structure. To define dynamic loads we have to go in the “Load Type” then selecting “New” and selecting “Modal” from the option. After defining modal, we have to repeat the same procedure and now selecting “UBC97” from the new.

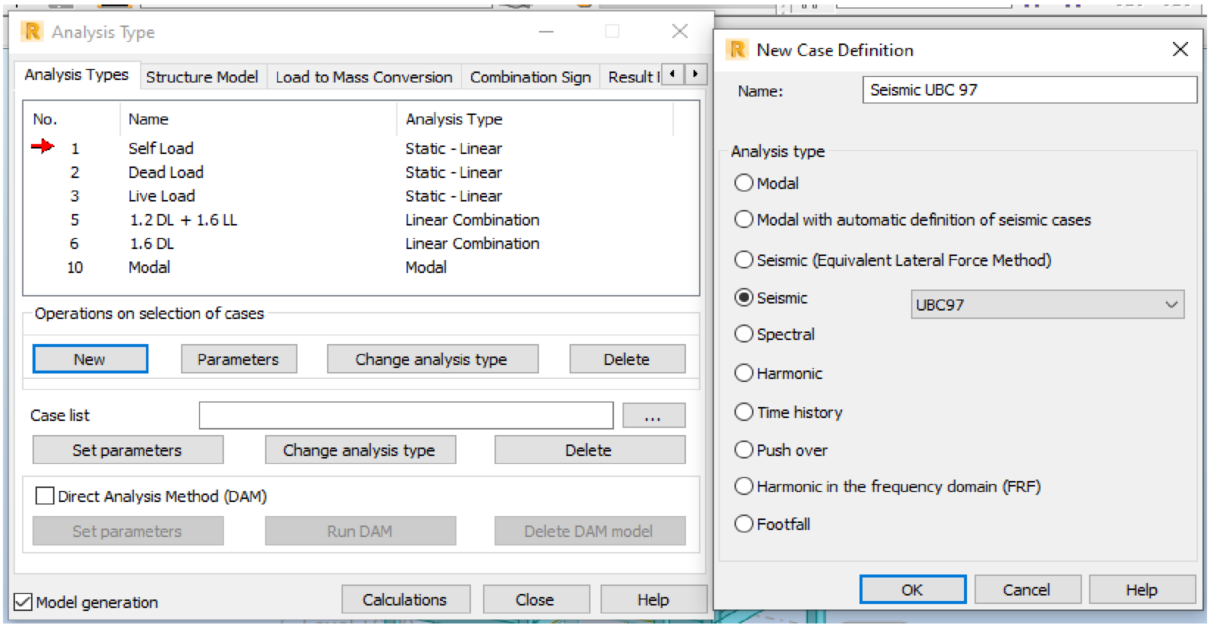


Fig 4.31 Defining UBC 97 Load

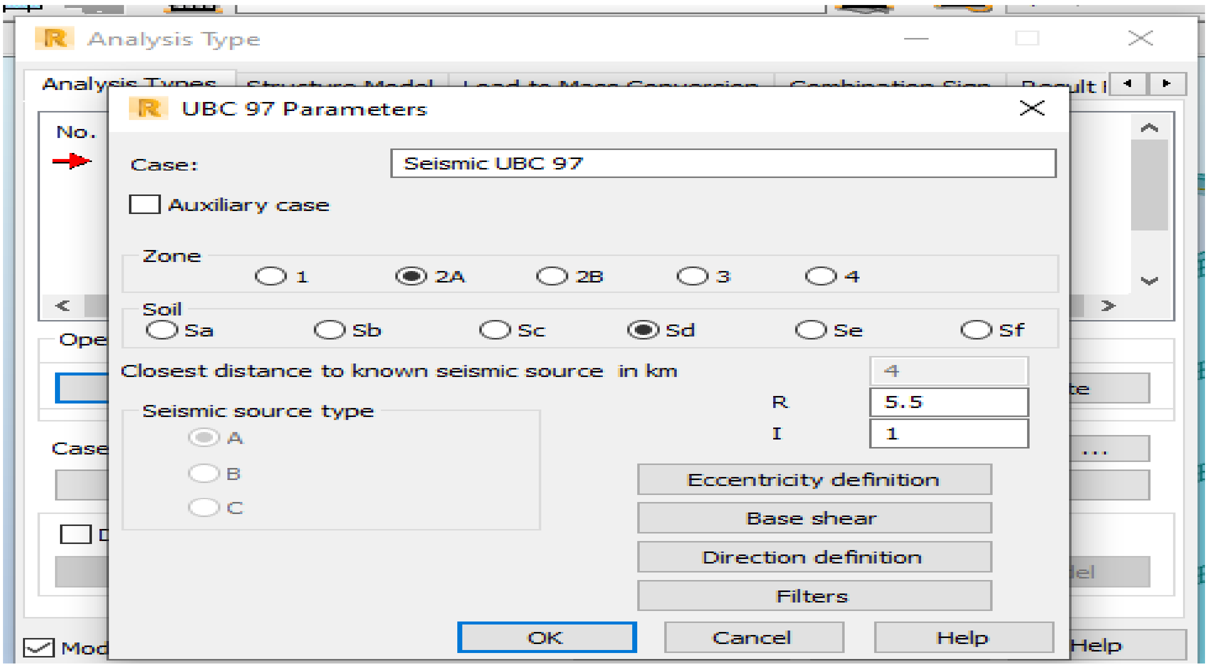


Fig 4.32 Defining zone & soil type

As Multan (Pakistan) lies in the zone 2A for the seismic zone where Zone = 2A, seismic zone coefficient Z = 0.15 and the soil of Multan is stiff so we select the soil Sd as it represent the stiff soil.

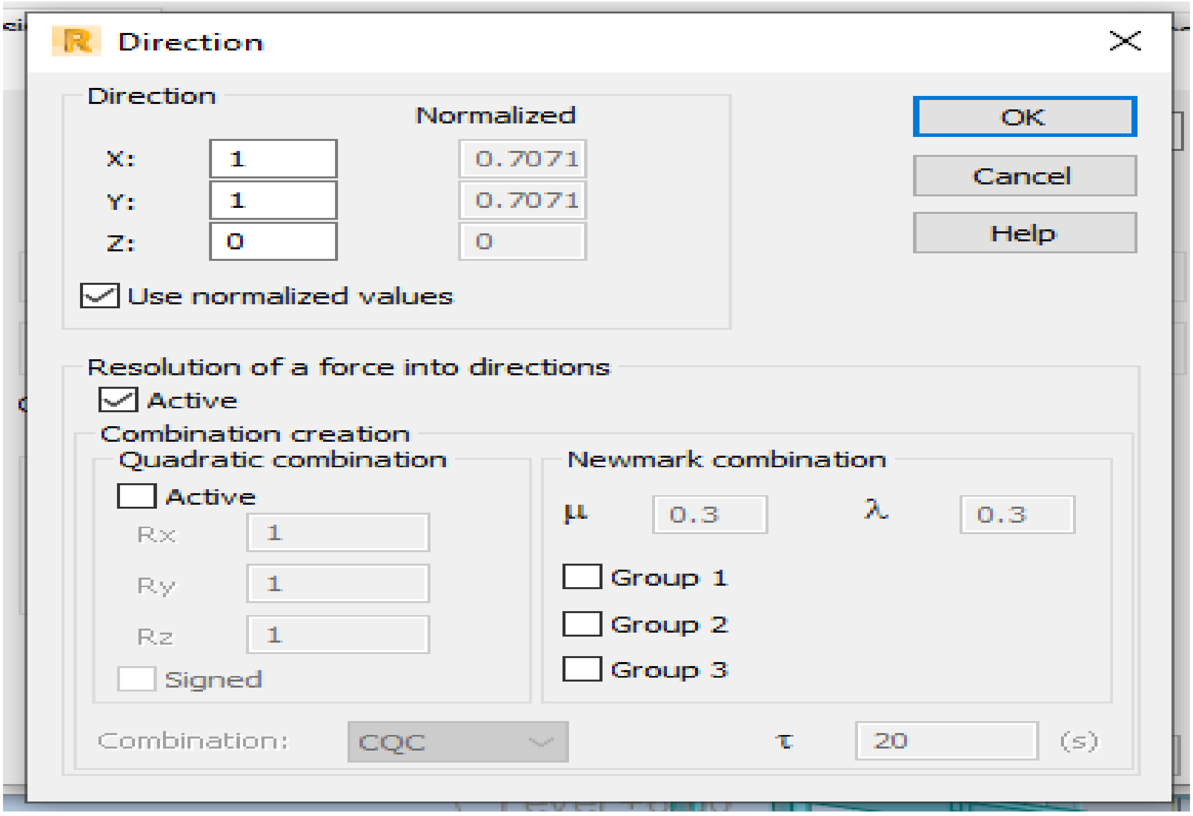


Fig 4.33 Defining direction for the earthquake

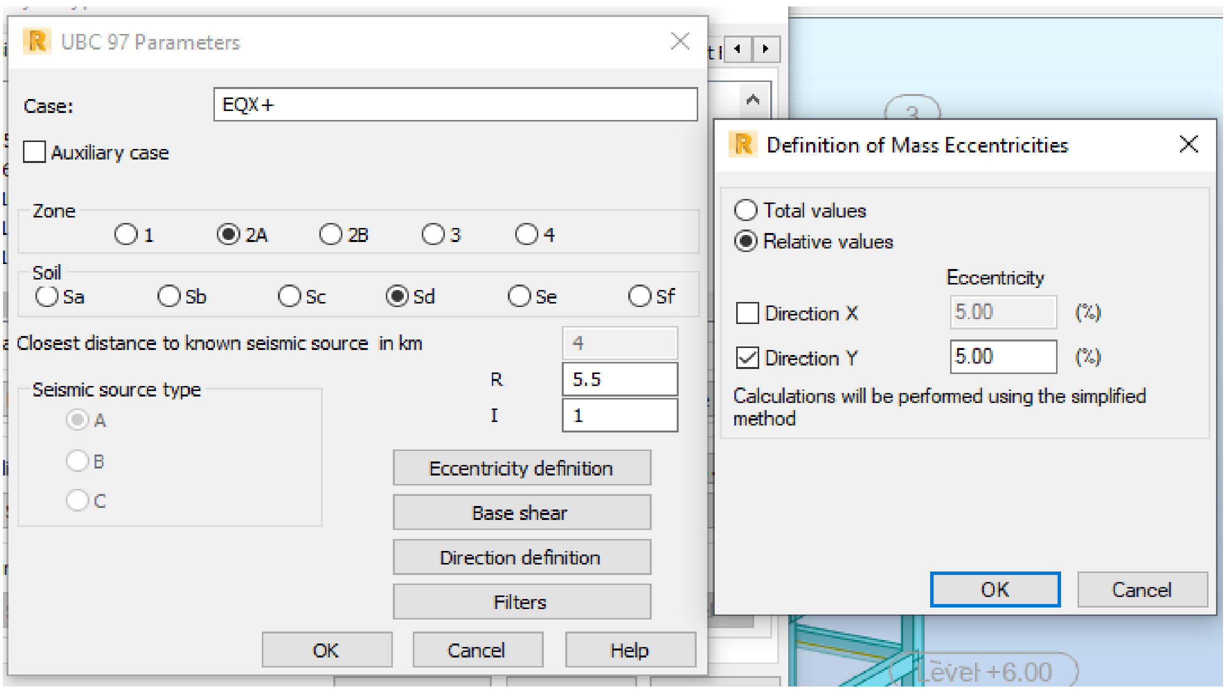


Fig 4.34 Defining EQX+ & EQY+ Load

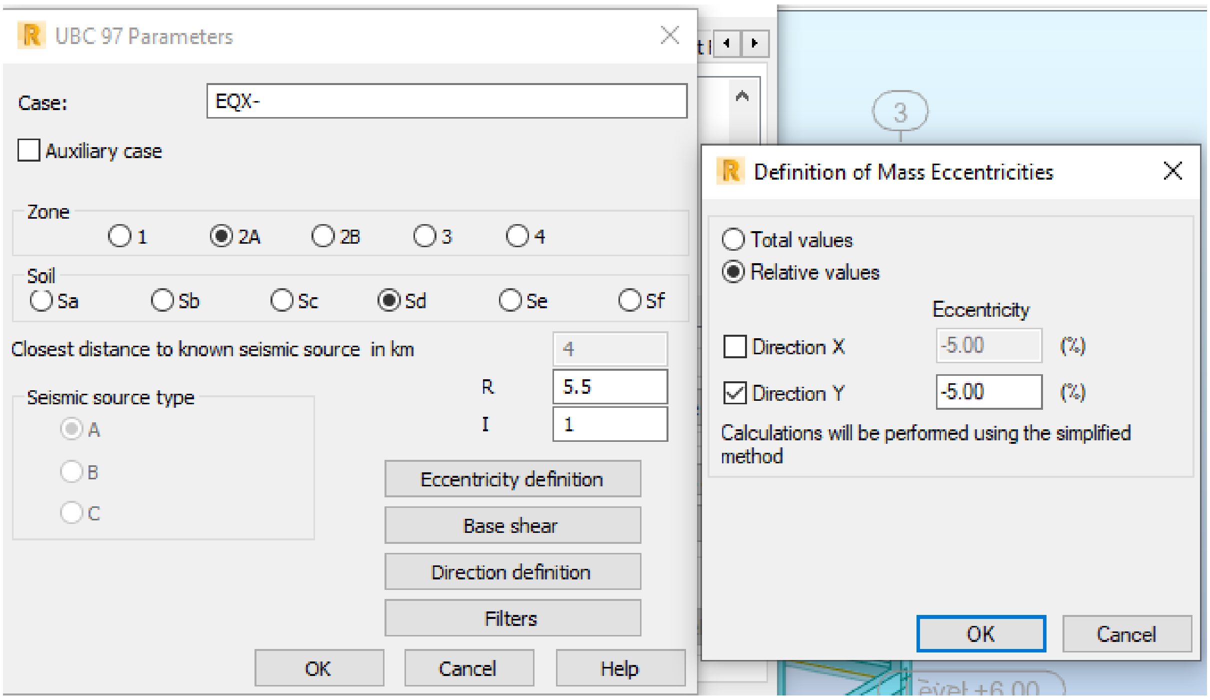


Fig 4.35 Defining EQX- & EQY- Load

4.4.2 Loads Combination

After defining loads, we make the loads combination for the analysis purpose. We make a combination of (1.2 D.L + 1 L.L + 1 EQX+), (1.2 D.L + 1 L.L + 1 EQX-), (1.2 D.L + 1 L.L + 1 EQY+) & one (1.2 D.L + 1 L.L + 1 EQY-) by going in the “Loads” in the tool bars and selecting the “Manual Combination” as shown below

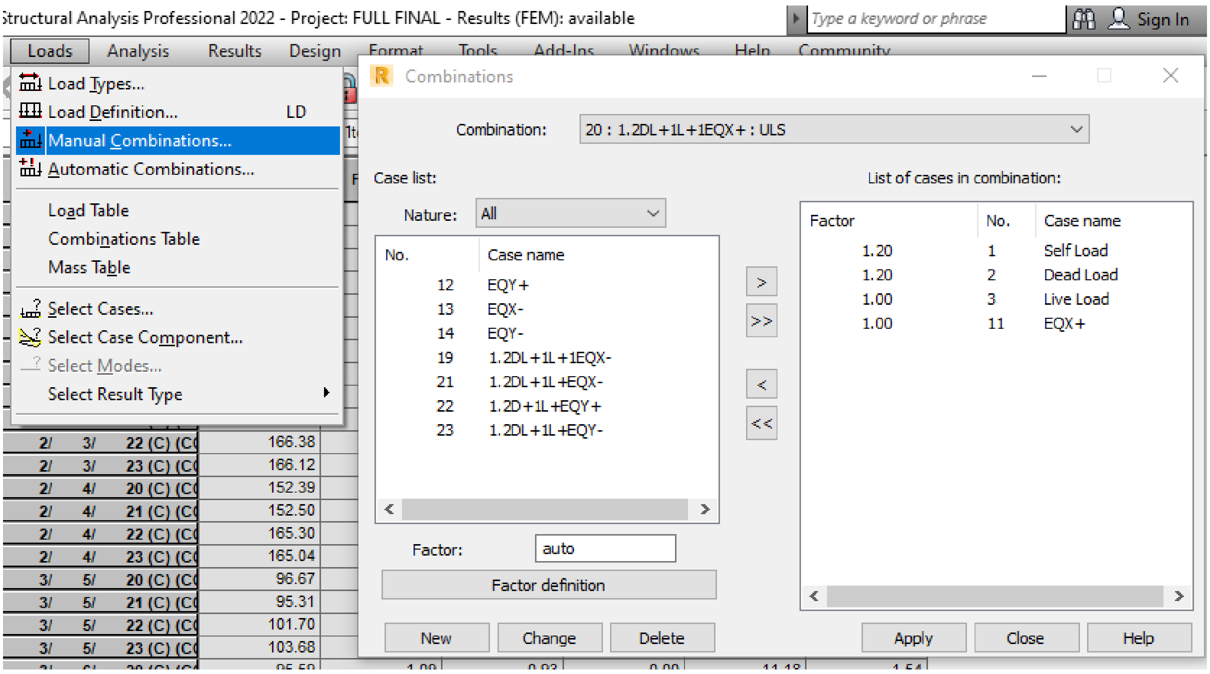


Fig 4.36 Load combination for the seismic analysis

4.4.3 Calculations & Results

As our loads are assigned and combination is also assigned. So, we will now calculate the results. For this purpose, we go in the Results selecting advanced and then selecting Modal results.

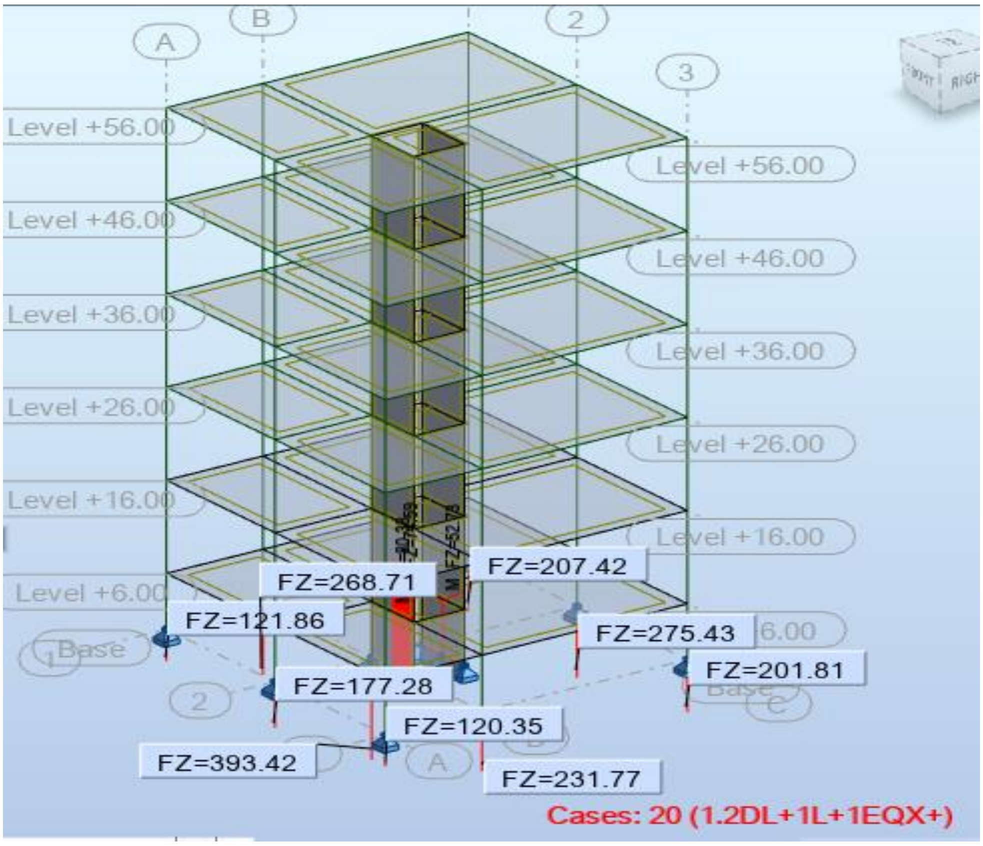


Fig 4.37 Reactions Diagram for the (1.2 D.L + 1 L.L + 1 EQX+)

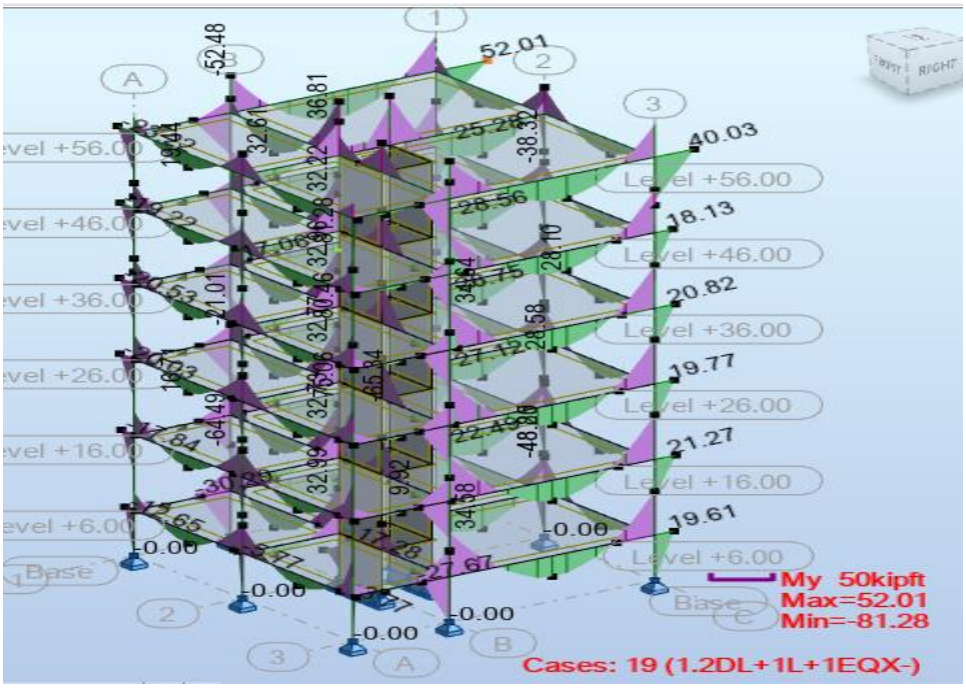


Fig 4.38 Moments diagram for the (1.2 D.L + 1 L.L + 1 EQX-)

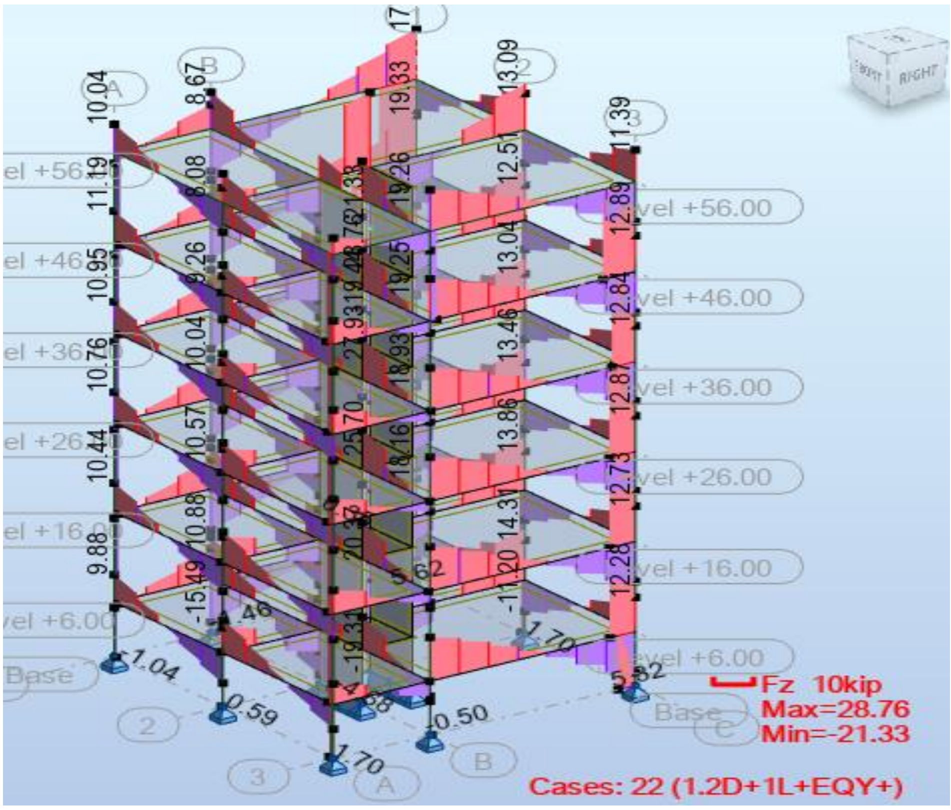


Fig 4.39 Shear Force diagram for the (1.2 D.L + 1 L.L + 1 EQY+)

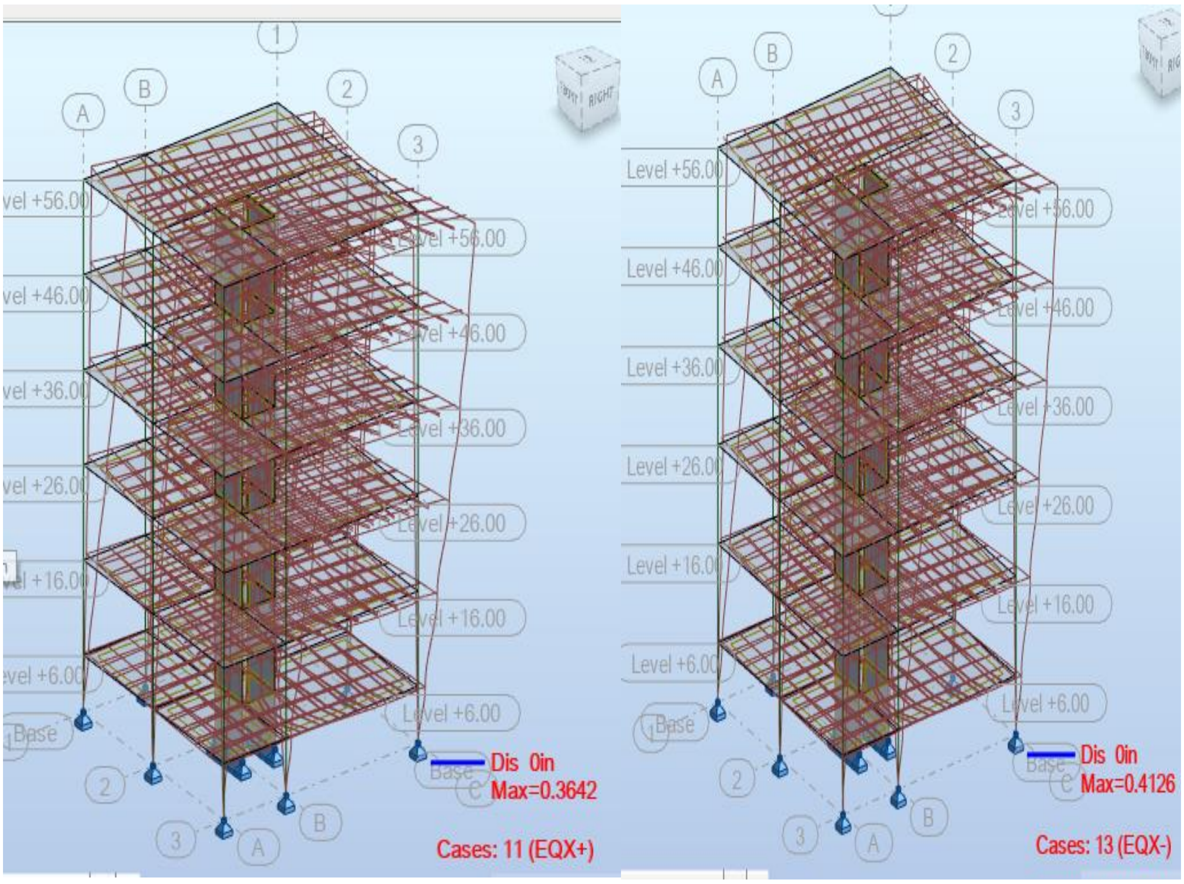


Fig 4.40 Deformation diagram for the EQX+ & EQX-

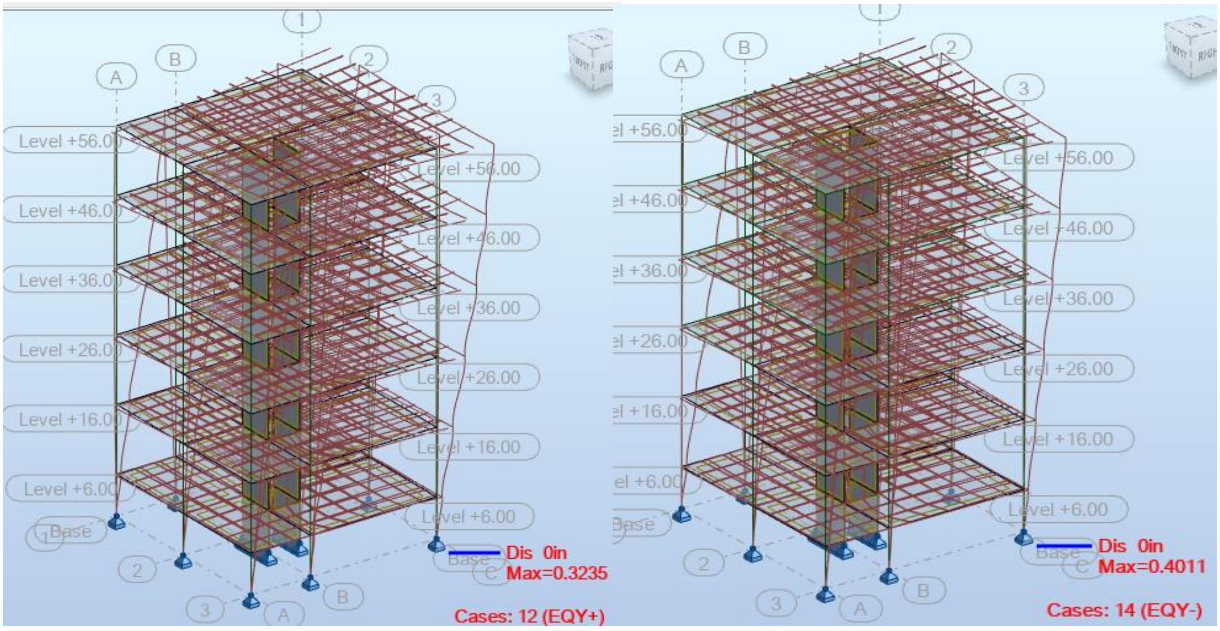


Fig 4.41 Deformation diagram for the EQY+ & EQY-

Case 11 : EQX+
Analysis type: Seismic-UBC 97

Mass eccentricities ex = 5.000 (%) ey = 5.000 (%)
Excitation direction:
X = 1.000
Y = 0.000
Z = 0.000

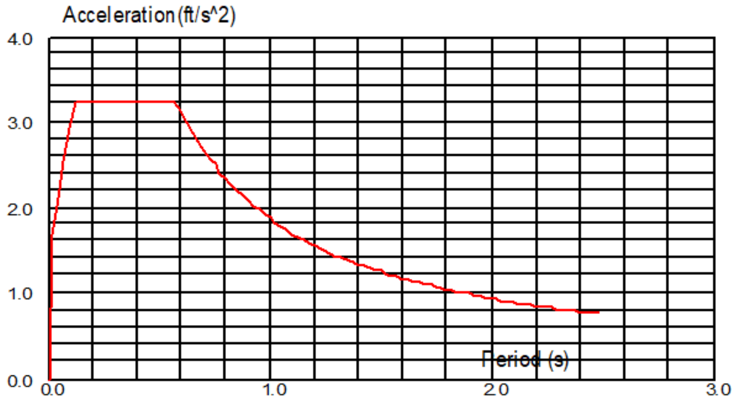


Fig 4.42 Graph for EQX+

	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ (%)	Cur.mas.UX (%)	Cur.mas.UY (%)	Cur.mas.UZ (%)	Total mass UX (lb)	Total mass UY (lb)	Total mass UZ (lb)
MAX	21.36	0.57	91.87	93.82	74.56	48.56	49.19	30.92	765813.04	765813.04	765813.04
Case	13	13	13	13	13	13	13	13	13	13	13
Mode	25	1	25	25	25	3	2	10	1	1	1
MIN	1.75	0.05	10.92	4.26	0.00	0.00	0.00	0.00	765813.04	765813.04	765813.04
Case	13	13	13	13	13	13	13	13	13	13	13
Mode	1	25	1	1	1	20	23	7	1	1	1

Table 4.10 Seismic Analysis values (Global Extreme) for EQX-

Case/Mode	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ (%)	Cur.mas.UX (%)	Cur.mas.UY (%)	Cur.mas.UZ (%)	Total mass UX (lb)	Total mass UY (lb)	Total mass UZ (lb)
11/ 1	1.75	0.57	10.92	4.26	0.00	10.92	4.26	0.00	765813.04	765813.04	765813.04
11/ 2	2.13	0.47	22.53	53.45	0.00	11.61	49.19	0.00	765813.04	765813.04	765813.04
11/ 3	2.20	0.45	71.09	71.87	0.00	48.56	18.43	0.00	765813.04	765813.04	765813.04
11/ 4	5.46	0.18	72.22	72.30	0.00	1.13	0.42	0.00	765813.04	765813.04	765813.04
11/ 5	7.47	0.13	75.31	82.37	0.01	3.08	10.08	0.00	765813.04	765813.04	765813.04
11/ 6	8.12	0.12	85.28	87.57	0.01	9.98	5.20	0.00	765813.04	765813.04	765813.04
11/ 7	9.63	0.10	85.84	87.76	0.01	0.56	0.19	0.00	765813.04	765813.04	765813.04
11/ 8	13.94	0.07	85.85	87.82	0.10	0.00	0.06	0.10	765813.04	765813.04	765813.04
11/ 9	14.96	0.07	86.52	91.41	7.84	0.67	3.59	7.74	765813.04	765813.04	765813.04
11/ 10	15.35	0.07	86.59	92.71	38.76	0.07	1.30	30.92	765813.04	765813.04	765813.04
11/ 11	16.73	0.06	89.29	93.17	45.62	2.70	0.47	6.86	765813.04	765813.04	765813.04
11/ 12	17.35	0.06	91.28	93.59	51.08	1.99	0.42	5.46	765813.04	765813.04	765813.04
11/ 13	17.53	0.06	91.41	93.64	51.19	0.13	0.05	0.11	765813.04	765813.04	765813.04
11/ 14	17.81	0.06	91.44	93.64	51.24	0.03	0.00	0.05	765813.04	765813.04	765813.04
11/ 15	18.30	0.05	91.52	93.64	51.48	0.08	0.01	0.23	765813.04	765813.04	765813.04
11/ 16	18.34	0.05	91.59	93.64	52.06	0.07	0.00	0.59	765813.04	765813.04	765813.04
11/ 17	18.60	0.05	91.68	93.65	52.88	0.08	0.01	0.82	765813.04	765813.04	765813.04
11/ 18	18.78	0.05	91.80	93.65	54.56	0.12	0.00	1.67	765813.04	765813.04	765813.04
11/ 19	19.67	0.05	91.81	93.66	56.05	0.01	0.00	1.49	765813.04	765813.04	765813.04
11/ 20	20.34	0.05	91.81	93.77	68.35	0.00	0.12	12.31	765813.04	765813.04	765813.04
11/ 21	20.36	0.05	91.82	93.80	71.64	0.00	0.03	3.29	765813.04	765813.04	765813.04
11/ 22	20.78	0.05	91.83	93.80	72.04	0.01	0.00	0.40	765813.04	765813.04	765813.04
11/ 23	21.01	0.05	91.86	93.80	72.04	0.03	0.00	0.00	765813.04	765813.04	765813.04
11/ 24	21.08	0.05	91.86	93.81	72.34	0.00	0.01	0.30	765813.04	765813.04	765813.04
11/ 25	21.36	0.05	91.87	93.82	74.56	0.01	0.01	2.21	765813.04	765813.04	765813.04

Table 4.11 Seismic Analysis values for EQX+

Case/Mode	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ (%)	Cur.mas.UX (%)	Cur.mas.UY (%)	Cur.mas.UZ (%)	Total mass UX (lb)	Total mass UY (lb)	Total mass UZ (lb)
12/ 1	1.75	0.57	10.92	4.26	0.00	10.92	4.26	0.00	765813.04	765813.04	765813.04
12/ 2	2.13	0.47	22.53	53.45	0.00	11.61	49.19	0.00	765813.04	765813.04	765813.04
12/ 3	2.20	0.45	71.09	71.87	0.00	48.56	18.43	0.00	765813.04	765813.04	765813.04
12/ 4	5.46	0.18	72.22	72.30	0.00	1.13	0.42	0.00	765813.04	765813.04	765813.04
12/ 5	7.47	0.13	75.31	82.37	0.01	3.08	10.08	0.00	765813.04	765813.04	765813.04
12/ 6	8.12	0.12	85.28	87.57	0.01	9.98	5.20	0.00	765813.04	765813.04	765813.04
12/ 7	9.63	0.10	85.84	87.76	0.01	0.56	0.19	0.00	765813.04	765813.04	765813.04
12/ 8	13.94	0.07	85.85	87.82	0.10	0.00	0.06	0.10	765813.04	765813.04	765813.04
12/ 9	14.96	0.07	86.52	91.41	7.84	0.67	3.59	7.74	765813.04	765813.04	765813.04
12/ 10	15.35	0.07	86.59	92.71	38.76	0.07	1.30	30.92	765813.04	765813.04	765813.04
12/ 11	16.73	0.06	89.29	93.17	45.62	2.70	0.47	6.86	765813.04	765813.04	765813.04
12/ 12	17.35	0.06	91.28	93.59	51.08	1.99	0.42	5.46	765813.04	765813.04	765813.04
12/ 13	17.53	0.06	91.41	93.64	51.19	0.13	0.05	0.11	765813.04	765813.04	765813.04
12/ 14	17.81	0.06	91.44	93.64	51.24	0.03	0.00	0.05	765813.04	765813.04	765813.04
12/ 15	18.30	0.05	91.52	93.64	51.48	0.08	0.01	0.23	765813.04	765813.04	765813.04
12/ 16	18.34	0.05	91.59	93.64	52.06	0.07	0.00	0.59	765813.04	765813.04	765813.04
12/ 17	18.60	0.05	91.68	93.65	52.88	0.08	0.01	0.82	765813.04	765813.04	765813.04
12/ 18	18.78	0.05	91.80	93.65	54.56	0.12	0.00	1.67	765813.04	765813.04	765813.04
12/ 19	19.67	0.05	91.81	93.66	56.05	0.01	0.00	1.49	765813.04	765813.04	765813.04
12/ 20	20.34	0.05	91.81	93.77	68.35	0.00	0.12	12.31	765813.04	765813.04	765813.04
12/ 21	20.36	0.05	91.82	93.80	71.64	0.00	0.03	3.29	765813.04	765813.04	765813.04
12/ 22	20.78	0.05	91.83	93.80	72.04	0.01	0.00	0.40	765813.04	765813.04	765813.04
12/ 23	21.01	0.05	91.86	93.80	72.04	0.03	0.00	0.00	765813.04	765813.04	765813.04
12/ 24	21.08	0.05	91.86	93.81	72.34	0.00	0.01	0.30	765813.04	765813.04	765813.04
12/ 25	21.36	0.05	91.87	93.82	74.56	0.01	0.01	2.21	765813.04	765813.04	765813.04

Table 4.12 Seismic Analysis values for EQY+

	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ (%)	Cur.mas.UX (%)	Cur.mas.UY (%)	Cur.mas.UZ (%)	Total mass UX (lb)	Total mass UY (lb)	Total mass UZ (lb)
MAX	21.36	0.57	91.87	93.82	74.56	48.56	49.19	30.92	765813.04	765813.04	765813.04
Case	14	14	14	14	14	14	14	14	14	14	14
Mode	25	1	25	25	25	3	2	10	1	1	1
MIN	1.75	0.05	10.92	4.26	0.00	0.00	0.00	0.00	765813.04	765813.04	765813.04
Case	14	14	14	14	14	14	14	14	14	14	14
Mode	1	25	1	1	1	20	23	7	1	1	1

Table 4.13 Seismic Analysis values (Global Extreme) for EQY-

19 : 1.2DL+1L+1EQX- 1, CQC

Node/Case	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
1/ 19 (C) (	-0.63	1.40	118.44	0.00	0.00	0.00
3/ 19 (C) (	0.96	0.59	177.46	0.00	0.00	0.00
5/ 19 (C) (	2.11	1.32	119.71	0.00	0.00	0.00
7/ 19 (C) (	-0.18	4.61	235.60	0.00	0.00	0.00
9/ 19 (C) (	-0.76	-3.27	200.74	0.00	0.00	0.00
11/ 19 (C) (	2.40	-3.26	275.42	0.00	0.00	0.00
13/ 19 (C) (	3.40	-3.75	206.18	0.00	0.00	0.00
15/ 19 (C) (	3.04	5.05	271.49	0.00	0.00	0.00
17/ 19 (C) (	28.69	25.50	402.38	0.00	0.00	0.00
101/ 19 (C) (	33.62	-9.92	123.31	-0.00	-0.00	0.00
105/ 19 (C) (	0.79	10.48	119.48	-0.00	0.00	0.00
722/ 19 (C) (	-5.65	0.85	87.69	0.00	0.00	0.00
727/ 19 (C) (	12.54	0.16	132.45	0.00	0.00	0.00
731/ 19 (C) (	5.62	0.09	97.43	0.00	0.00	0.00
736/ 19 (C) (	8.92	0.32	139.31	0.00	0.00	0.00
Case 19 (C) (CQC	1.2DL+1L+1EQX-					
Sum of val.	94.88	30.19	2707.07	-0.00	0.00	0.00
Sum of reac.	48.60	12.39	2257.97	31836.00	-30768.35	570.63
Sum of forc.	48.60	12.39	-2255.53	-30887.84	34741.46	570.70
Check val.	97.20	24.79	2.44	948.16	3973.11	1141.33
Precision	1.48342e-02	4.97664e-04				

Table 4.14 Reactions Results for (1.2 D.L + 1 L.L + 1 EQX-)

20 : 1.2DL+1L+1EQX+ 1, CQC

	UX (in)	UY (in)	UZ (in)	RX (Rad)	RY (Rad)	RZ (Rad)
MAX	0.3097	0.1955	0.0042	0.001	0.001	0.001
Node	998	995	202	717	636	856
Case	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)
MIN	0.0	0.0	-0.2313	-0.002	-0.002	-0.000
Node	1	1	709	643	703	728
Case	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)

Table 4.15 Displacement Results for (1.2 D.L + 1 L.L + 1 EQX+)

19 : 1.2DL+1L+1EQX-

	UX (in)	UY (in)	UZ (in)
MAX	0.0006	0.0111	0.0187
Member	169	152	82
Case	19 (C) (CQC)	19 (C) (CQC)	19 (C) (CQC)
MIN	-0.0009	-0.0378	-0.0955
Member	166	159	160
Case	19 (C) (CQC)	19 (C) (CQC)	19 (C) (CQC)

Table 4.16 Maximum Deflection Results for (1.2 D.L + 1 L.L + 1 EQX-)

22 : 1.2D+1L+EQY+   1, CQC

	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
MAX	288.91	7.96	19.33	9.42	43.79	36.39
Member	6	9	131	42	152	151
Node	11	17	70	23	998	997
Case	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)
MIN	-10.98	-7.67	-18.18	-6.83	-73.35	-18.84
Member	169	154	167	14	131	150
Node	1000	1000	1000	14	70	996
Case	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)	22 (C) (CQC)

Table 4.17 Forces Results for (1.2 D.L + 1 L.L + 1 EQY+)

23 : 1.2DL+1L+EQY-   1, CQC   Level +

	S max (ksi)	S min (ksi)	S max(My) (ksi)	S max(Mz) (ksi)	S min(My) (ksi)	S min(Mz) (ksi)	Fx/Ax (ksi)
MAX	2.25	1.37	1.42	1.01	-0.00	-0.00	1.37
Member	37	2	73	151	1	2	2
Node	26	3	43	997	1	3	3
Case	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)
MIN	0.03	-1.42	-0.00	0.00	-1.42	-1.01	-0.05
Member	139	131	1	2	73	151	169
Node	65	70	1	3	43	997	1000
Case	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)	23 (C) (CQC)

Table 4.18 Stresses Results for (1.2 D.L + 1 L.L + 1 EQY-)

20 : 1.2DL+1L+1EQX+

	MXX (kipft/ft)	MYY (kipft/ft)	MXY (kipft/ft)
MAX	5.49	5.61	0.95
Panel	112	115	170
Node	475	133	662
Case	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)
MIN	-1.70	-2.75	-0.62
Panel	173	156	170
Node	709	1007	634
Case	20 (C) (CQC)	20 (C) (CQC)	20 (C) (CQC)

Table 4.19 Slab Results for (1.2 D.L + 1 L.L + 1 EQX+)

4.5 Discussion

The results obtained from the manual calculations of slabs were something different from the results obtained from the Autodesk Robot structural analysis as we use Coefficient method while software uses the (Finite Element Method) FEM. So, this difference of result was expected in case of slabs.

Whereas, the results obtained for the beams were as given below

Mode	Beam No.	BMD(+ve)	BMD(-ve)	SFD
Manual	159	24.22	-34.55	13.54
Software	159	25.90	-34.30	13.69
Manual	160	40.80	-58.35	18.13
Software	160	40.89	-58.85	18.13

So, the difference is very minor which can be neglected.

CHAPTER NO. 5

CONCLUSION & RECOMMENDATIONS

5.1 Conclusions

In this research, RCC structural model with a height of 56 ft. comprising of five stories located in zone 2A Multan, Pakistan was analyzed using Autodesk Robot Structural Analysis Software. Static analysis was performed using software and manual calculations while dynamic analysis was determined using UBC97 earthquake code in Autodesk Robot Structural Analysis Software. The main conclusions of the research are as follows;

i. The Autodesk Robot Structural Analysis Software helps us in the analysis of a structure more effectively and brilliantly. It is a very advanced software than other software. Due to BIM integrated software it is very powerful.
ii. The maximum and minimum bending moment values for slabs are 4.89 kip/ft and 1.97 kip/ft respectively.
iii. It has seen that maximum and minimum displacement for deformation during the earthquake were obtained as 0.4126 and 0.3235 inch, respectively.
iv. 10% to 16% is higher the value of Moments for Dynamic analysis based on results than the values acquired for Static analysis
v. 12% to 25% is higher the value of Reactions for Dynamic analysis based on results than the values acquired for Static analysis.
vi. It is seen that the results of the BMD & SFD for the beams and columns were appropriately matched with manual and software calculations.

5.2 Recommendations

It is recommended that

i. The structure should be analysis for prestressed concrete using Autodesk Robot Structure Analysis software.
ii. Fiber reinforced concrete should be analysis with Robot Structure Analysis Software.
iii. All Civil Engineers should understand the theory of structures before diving into lengthy calculations and mathematics of design and analysis.

iv. Due to the more advanced features, we recommended to use Autodesk ROBOT structural analysis software for structural analysis.

References

1. **Static and Dynamic Analysis of Multistory Building** by Amresh A. Das, G. B. Bhaskar at Civil Engineering Department, G.H. Raisoni Academy of Engineering and Technology, Nagpur,India

   http://www.ijasret.com/VolumeArticles/FullTextPDF/978_54.74858565.pdf

2. **Dynamic analysis of multi-storey building** by Udaya Bala K, Manish Kumar Gupta and Senthil Pandian M at School of Mechanical and Building Sciences, VIT University, Chennai, India

   https://www.icontrolpollution.com/articles/dynamic-analysis-of-multistorey-building-.pdf

3. **Design & analysis of reinforced concrete multistory commercial building using ACI-318** by Muhammad Abdul Azeem Baig, Imbia abd-el-Salam & Imbia Ammar at Faculty of Civil Engineering University Of Bani-Walid Libya

   https://pdfcoffee.com/download/design-and-analysis-of-reinforced-concrete-multistory-commercial-building-using-aci-318-metric-manually-and-extensive-design-by-robot-analysis-pdf-free.html

4. **Structural analysis of office building with Robot Structural Analysis and Hand calculation** by Ardeep Maharjan at HAMK University of Applied Sciences Construction Engineering

   https://www.theseus.fi/bitstream/handle/10024/154081/Final%20thesis.pdf?sequence=1&isAllowed=y

5. **Static and Dynamic Analysis of Multistory RC Building with Various Heights in High Seismic Zone** by Ghusen Al-Kafri, Md. Shahnewaz Sarkar, Md. Shaizuddin Sarkar at Department of Civil Engineering, College of Engineering, Sirte University, Libya

   https://www.researchgate.net/publication/329634057_Static_and_Dynamic_Analysis_of_Multistory_RC_Building_with_Various_Heights_in_High_Seismic_Zone

6. **Static and dynamic analysis of high rise structure using STAAD Pro Software** by Syed Aynain Wani, Abhishek sharma at Department of structural engineering, 1CBS Group of Institutions, Jhajjar Haryana, India

   https://www.jetir.org/certificatemanager.php?a_rid=236209

7. ACI, 2019. Building Code Requirements for Structural Concrete (ACI318-19) and Commentary (ACI318R-19), American Concrete Institute.

8. Structural Concrete, Theory and Design,7th edition by M.Nadim Hassoun.

9. Other final year projects.

10. Autodesk Robot Analysis, 2023. *structural analysis, design and detailing software*. user manual window version 10.

11. And many other random informative website and resources …