International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 10 | Oct 2021 www.irjet.net p-ISSN: 2395-0072

COMPARITIVE STUDY OF COMMERCIAL BUILDING BY ANALYTICAL METHOD

Rashmi J[1], Prof Shiva Shankar K M[2]

[1]PG Student, Dept. of Civil Engineering, ACS College of Engineering, Bangalore
[2]Professor, Dept. of Civil Engineering, ACS College of Engineering, Bangalore

---

Abstract - Structural analysis is branch which involves in determining the structural behavior of structure in order to predict the response of the structure and the structural components due to effect of loads .Basically each structural components involves or subjected to different types of loads. The different types of loads are dead load, live load , wind load(IS 875 -1987 Part1,2,3),earthquake load (IS:1893-2016).ETABS (Extended Three Dimensional Analysis of building system) it is a software with all analysis engine like static ,dynamic, linear, non linear and it is used to calculate the forces , Bending moment , shear force , stress , strain , deformation etc by using this analysis of structure is carried out . coming to our project the analysis is carried out for G+4 Frame structure carried out by equivalent static method using Etabs 2018 software by this the response of structure is recorded after all these in the next step ANSYS WORKBENCH software is used to analyze the structure and its components of G+4 Frame structure here the calculation of forces, bending moment, shear force , static, axial force is carried out and in the final the results of Etabs and the Ansys is taken to reference and the response of the structure is seen and difference is calculated .

Key Words: Etabs, Static Analysis, Dynamic Analysis , Non-linear Analysis , Linear Analysis.

1.INTRODUCTION

The building term is used in the civil engineering to know the meaning of structure and the various components of the structure like foundation ,walls, column, beam, floor, door, window, ventilator etc.The structural analysis is carried out to give the proper structure that should be capable to resist all the applied loads without any fail during the service life. Before the analysis it is necessary to check the soil conditions by collecting soil and by the means of geotechnical investigation the information and the condition if the soil is known and the further designing and the construction of the foundation is carried out.

Structural engineers are facing the challenges to obtain most efficient and economical design and with accuracy in design of building and the structure is serviceable and economic to take the loads applied to function in life time. In the recent days many software are available in the market to analyze and design the structure example :Stadd pro, Etabs, SAP, STRUDAL, RISA, RAM etc.

1.1 Objective

1. The Primary objective of the study is to analyze a G+4 commercial structure using ETABS and ANSYS Workbench 15.0.

2. The comparative study of structure using **ETABS** and **ANSYS WORKBENCH** model.

3. Comparison of outcomes obtained from **ETABS** and **ANSYS WORKBENCH.**

4. To obtain stress, strain, deformation, axial force, storey drift from both the software's and to comparative study of the results obtained.

1.2 Methodology

The adoption of the methodology for the experimental research is as following**:**

PART 1

1.Analysis using ETABS.
2.Modeling and analysing the structure .
3.Comparing results.

PART 2

1.Analysis using ANSYS WORKBENCH 15.0.
2.Modeling and analysing the structure.
3.Comparing results.

1.3 Literature review

G.Fabbrocino, G.Manfredi,E.Cosenza(2008) : They studied the structural behavior of the steel -The composite beam depends in the interaction in between the steel beam and the concrete slab which analyzes the connection between the behavior of the beam and the modeling of the structure has been done. The relationship between the slip and interaction force given by the connection and the behavior of the structure of the beam is subjects to the lateral loads .

Barour et al.2019 : He presented the non-linear finite element model to invesrtigate the response of the external strengthened beams he tested under the three point flexure test and the ansys software is used for modeling the solid65,Shell 181,elements has been used to model polymer , steel plate etc.

Jagadish et.al 2019 Presented the non-linear finite element analysis is done to investigate behavior of the steel fiber reinforced concrete beams. By varying percentage of steel the strength of the reinforced concrete beam. And deformation, the ultimate load carrying capacity is been checked.

Uttam kumar Chakravartyet al.2014 presented on the Analysis of the composite beam cross-sections. The computational Technique like FEA has been taken for the investigation .Finally the Difference asymptotic beam section analysis was good.

Jun Deng et al.2011 investigated on the stress analysis of steel beam bonded with carbon fiber material it includes the analytical solution to calculate the stress is thermal load. Finite element analysis has been done to validate the analytical results and the maximum stress is carried out.

Hang et al. 2018 investigated three Rc beams and Analytical study is carried out using the Ansys 15.0 and parameters like stirrup configuration, depth ratio , crack pattern and the load calculated for the first crack has been initiated. And the shear configuration is been studied wrt beam span.

Balaji,U and Selvarasam M.E 2016 Done analysis and design on multi storey building of G+13 storey building under the static and dynamic load condition using ETABS . The non linear analysis is carried out considering the seismic zones and the behavior is observed and different results like deformation ,stress, strain, displacement is drawn.

Chandrasekhar et.al 2015 worked on analysis and design of multi storey building using ETABS .They worked on G+5 storey building under lateral loading condition under wind and the earthquake was considered for the study and analysis. They considered the importance of the fire catch hence fire proof material usage importance is explained.

Varalakshmki V et.al 2014 worked on analysis and the design of the residential building of G+5 storey. It has been researched that service life of the building depends upon the structural configuration .

2. Details of Structural Membranes considered for Analysis

2.1 Material properties

Young's Modulus (E) for M30 Concrete =27.376x10^6 kn/m^2

Density of concrete = 25Kn/m^2

Density of Masonry unit = 20Kn/m^2

2.2 Member properties

1. Beam =150*250mm
2. Column = 300*500mm
3. No of storey G+4
4. Grade of rebar = Fe415&Fe500
5. Bottom storey height = 3.2m
6. Typical storey height = 3m
7. Slab Thickness = 150mm
8. Loadings
9. Importance factor = 1
10. Response reduction factor = 3
11. Dead load = 2.5Kn/m^2
12. Live load = 3Kn/m^2
13. Floor finish = 1Kn/m^2

2.3 PLAN

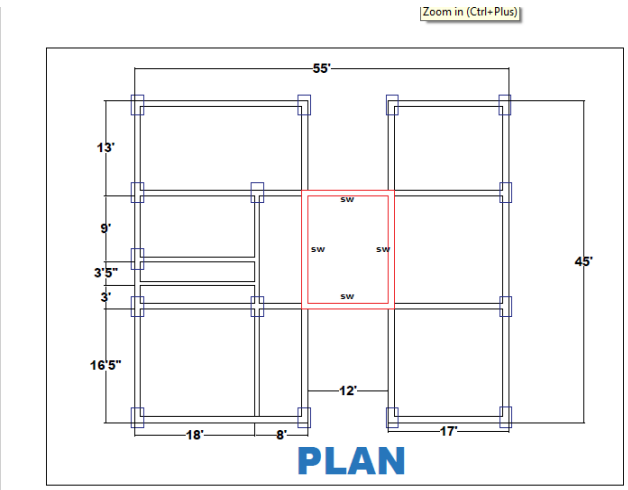


Fig -1: PLAN

2.4 INVESTIGATION

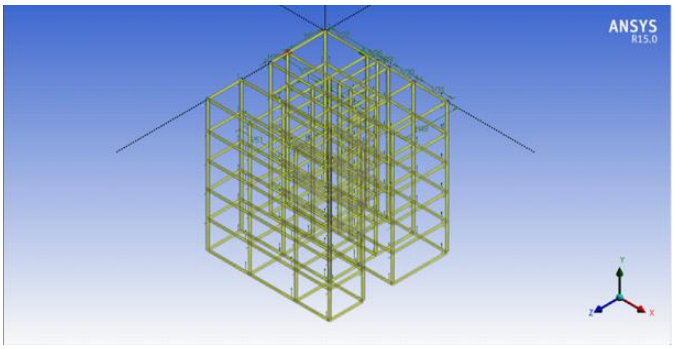


Fig- 2 Geometry

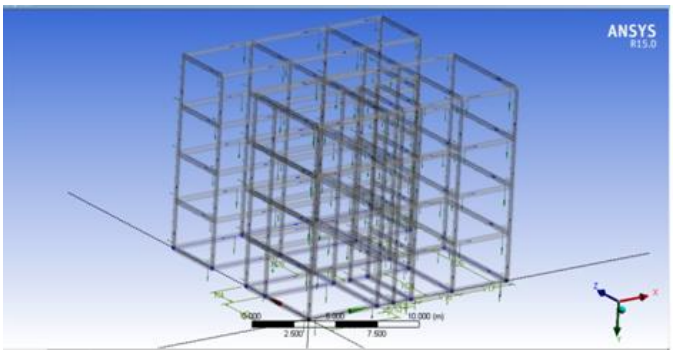


Fig -3 Meshing

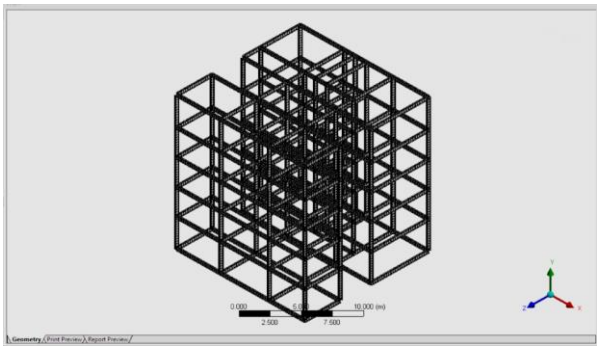
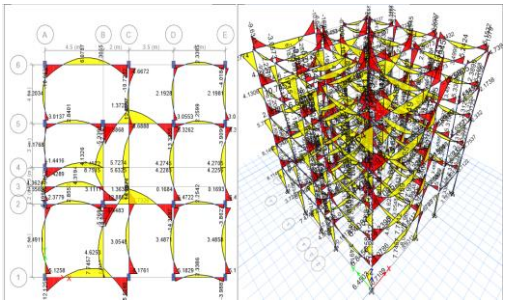


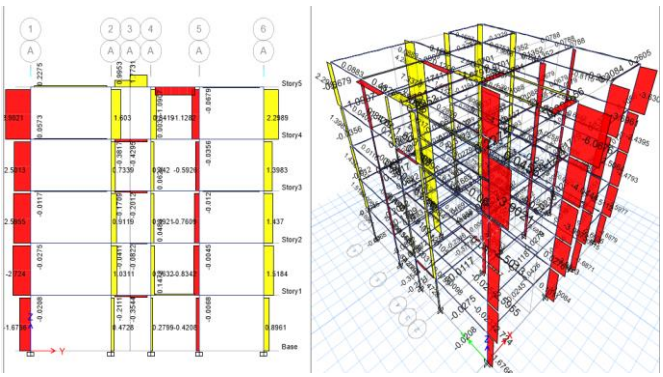
Fig -4 meshing

3. RESULTS

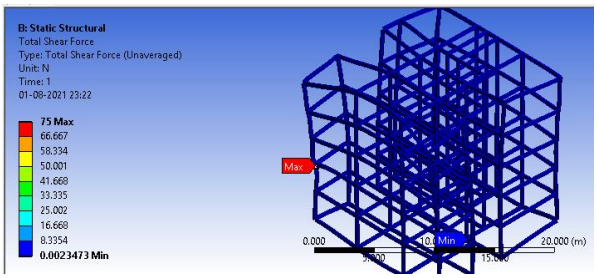
1. Maximum bending moment



2. SHEAR FORCE

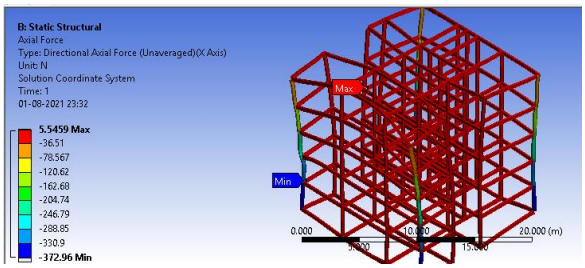


3. TOTALSHEAR FORCE



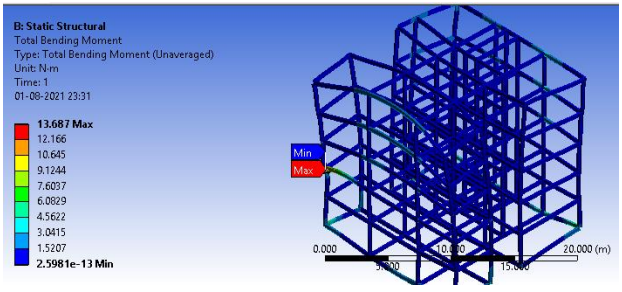
Total shear force is a force which is considered as the over all force which is acting on the structure .As per the above structure the total shear force acts at the middle of the structure and it is indicated in the above fig. The above fig shows how the shear force values has been displayed as per the value in accordance to the color indication

4. DIRECTIONAL AXIAL FORCE



As per the above structure the max directional Axial force acts at the top of the structure and the min directional Axial force acts at the storey 2 of the structure. The above fig shows how the directional deformation values has been displayed as per the value in accordance to the color indication.

5. TOTAL BENDING MOMENT



As per the above structure the max directional Axial force acts at the top of the structure and the min directional Axial force acts at the storey 2 of the structure. The above fig shows how the directional deformation values has been displayed as per the value in accordance to the color indication.

4. CONCLUSIONS

- Comparing between Etabs and ansys workbench 15.0 analysis of G+4 structure it is found that the Maximum bending moment is same 13.687KN.M so there is no difference between the values.
- Comparing between Etabs and Ansys workbench 15.0 analysis of G+4 structure it is found that the Maximum shear force occurs at the mid of the structure .
- Comparing between the software compatibility the results obtained from Etabs are almost similar but flexibility in knowing the minor behavior of the structure the Ansys software is useful in this by animation process we can see the behavior of the structure which is not able to observe through 2D or 3D drawing so the mistakes during the design can be known.

REFERENCES

- Prof.Dr.Tushar Shende "Project report on Deep Beam Analysis using ANSYS" IJSTE JUNE 2018.
- Ahmed Abdullah Mansoor, Wisdom D.Salman "Finite element analysis on reinforced concrete deep beams Strengthened in shear with CFRP"ARPN Journals January 2019.
- P.Leela krishna,G.Santoshi kumari"Analysis and design of G+4 Commercial building " IJIEMR
- Sharadenu shukla, Dr Prabhat kumar sinha "Frequency Analysis of complex composite beam by using Finite element and Ansys workbench"IJARIIE 2017.
- Balaji et.al "worked on analysis and design of multistoried building"
- Ragy jose, Restina Mathew, Sandra Devan "Analysis and design of Commercial building "IRJET June 2017.
- Prof.Dr.Rakesh Patel"A review on crack identification and analysis of RCC beam using Ansys" IJSET 2020.