Analysis of Beam and Column for Improving the Integrity Performance

Ravikiran P. Jadhav[1]* Dr. Prashant M. Pawar[2] Shrikrishna A. Gosavi[3] Dr. Vidyarani S. Kshirsagar[4] Dr. Manik G. Deshmukh[5]

[1] PG Student, Civil Engineering Department, SVERI's College of Engineering Pandharpur, Maharashtra

[2] Professor, Civil Engineering Department, SVERI's College of Engineering, Pandharpur, Maharashtra

[3] Assistant Professor, Civil Engineering Department, SVERI's College of Engineering, Pandharpur, Maharashtra

[4,5] Associate Professor, Civil Engineering Department, SVERI's College of Engineering, Pandharpur, Maharashtra

Abstract – In structural Analysis generally the term frame means rigide structures having different component such as slab, Beam, Column, Footing etc. This all Components are casting together to form monolithic construction so it acts as single integral unit. There are many forces are acting on the building frame this acted force are pass through different component of frame. In some of structure extra live loads are act because of changes uses from residential to public or godown. For that the element of frame should be enough strong to sustain loads coming. In this Present work, we were anlysed columns and beams. for different load combination and improve integrity performance by using angles and Steel plate for strengthening. Finite element modelling is carried out for numerical analysis using ANSYS Workbench. Different models of beam and column are made in ANSYS Workbench and studying and comparison is done for Total Deformation and select optimum strengthen location for different loading condition.

Keywords – ANSYS Workbench, Finite Element Model, Loads, Numerical Analysis, Total Deformation

- X -

I. INTRODUCTION

In the industry of construction we consider columns and beams are rigide. Frame consists of different components such as beam and column. All Total load coming in building were going from this comp. so this compo. is enough strong and stable to transfer this load to the ground. Different forces are acted on the building such as vertical load as dead load of structure and live load, lateral loading is due to seismic waves i.e., due to earthquake and torsional force as wind load etc. for force sustainability strong joint is necessary i.e., Bending, lateral and torsional forces generated during loading and transfer of forces from one component to another. Connected members get failed in earthquake becauseof joint failure. For load sustainability col. and beams frequently strengthening is essential.[5] There are multiple numbers of method like batten plates of U shape is used in beams also distinct members were strengthened at nonidenticale points. fibre-reinforced polymer also used for jacket to existing column.

M.I.S. Elmasry et al. (2018) [1] various techniques on cfrp retrofitting in the literature paper were studied by author. Results of his study said that strength of columnbeam connections were improved using jacketing steel by wrapping sheets of cfrp in cross setup. Furthermore, using the proposed technique proved to show better ductile response than other techniques suggested in some previous research related to the same study point.

M.F. Belal et al. (2014) [2] Investigated in his paper about behaviour of strengthened steeljacketing column. The researcher demonstrated that capacity of column were depend upon various schemes of strengthening. shapeof main strengthning using C shape section, angles,platees and their numbers and size.he tested seven column out of which 2 were normal and remaining were strengthened. By using fem behaviour of these columns were studid. He verify his results with experimentals results. He conclude that significant strengthening results were given by batten plates used to C channel section.

Chunli Zhou Jinkun Sun et al. (2019) [3] reviewed in his paper about constructional methods of RC frames. Researcher introduced strengthened of beam by using exterior-warping U-Shaped steel plates. He derived formula brifly to compute carrying load capcity of warping exterior strengthened reinforced cementconcrete beams. Also he use softwere called abaqus to verify results got through theoretical setupe. Lastly he suggest warping of exterior U shape battenplates for rational method of strengthening.

II. OBJECTIVES

1. To analyse beam for two-point bending.
2. To analyse beam for combined torsion and bending.
3. To analyse the column for combined lateral and vertical loadings.

III. METHODOLOGY

Finite Element Approach [4]

The different specimens were modelled by the FEM using ANSYS Workbench 15.0 In order to simulate as closely as possible the actual behaviour of the specimens. From the program menu list open the ANSYS 15.0 folder and select Workbench 15.0.

Step 1 Unit Setting: Now is a good time to set the units that are going to work since the units typically do not change. For modelling a Beam and column of 150X150X2000 MM and 150X150X1000 MM. units tab from project bar menu was selected to units change.

Fig.1 Unit Settings.

Step 2 Creating the Model: To perform a Static structural analysis, we will need to add the correct building block (system) from the Analysis Systems menu. ANSYS run different type of analyse like Fluid interaction thermalanylisis, static structure, buckling and dynamicanylisis, etc. Select the static structural block drag it into green box. For analysis in ANSYS workbench we need to define Engineering Data>Geometry>Model>Solution>Results.

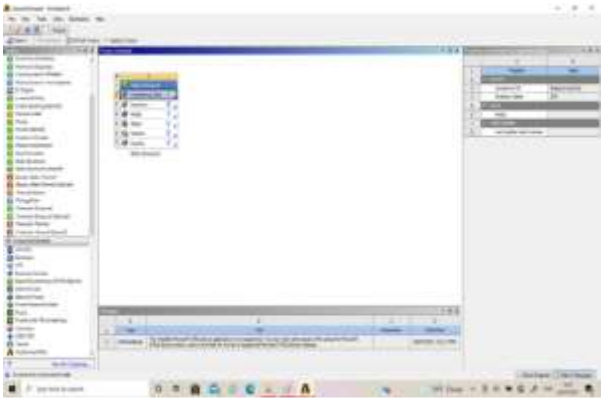


Fig. 2 Static Structure Standalone

Step 3 Engineering Data: To have the correct behaviour modelled, need to define the type of material, along with its mechanical properties, using the Engineering Data cell. The selection of material was done by depending upon the physical and chemical properties of the material. These properties are used for analysis.

Table 1. Material Data

Sr. No	Name of Material	Properties	Value
1	Concrete	Density	2500 kg m^-3
		Compressive Ultimate Strength	25e+007 Pa
		Young's Modulus	3.e+010
		Poisson's Ratio	0.18
2	Reinforcement Bar	Density	7850 kg m^-3
		Tensile Yield Strength	5.e+008
		Young's Modulus	2.e+011
		Poisson's Ratio	0.3
3	Batten Plates	Density	7850 kg m^-3
		Tensile Yield Strength	2.5e+008
		Young's Modulus	2.e+011
		Poisson's Ratio	0.3
4	Angles (Structural Steel)	Density	7850 kg m^-3
		Tensile Yield Strength	2.5e+008
		Young's Modulus	2.e+011
		Poisson's Ratio	0.3

Step 4 Geometry: During this part, geometry was drew by sketch, these steps: Change the Units from the toolbar Units.go in XY Plan, and drawne witha 2D shape. Then extrude this 2D shape into 3D model by extrude command in this we were gave length to that element. Similarly, the reinforcement is drawn by using sketching command.

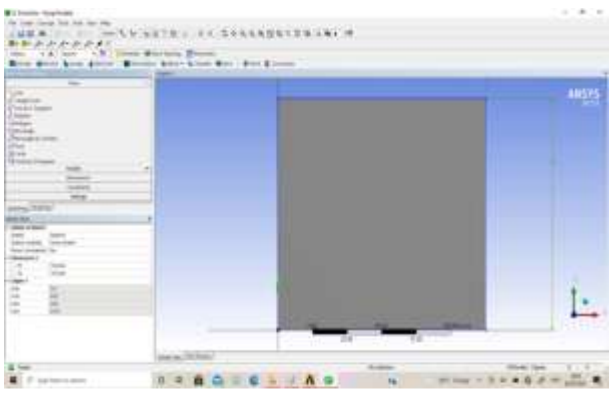


Fig.3 Modelling of element

Step 5 Model: The Model cell is the next one that will want to right-click on and select Edit to work with. Another new window will open that is tagged with a red ball and giant M icon, which stands for Mechanical. This Mechanical window will allow working on the meshing of the flat solid rectangular slab

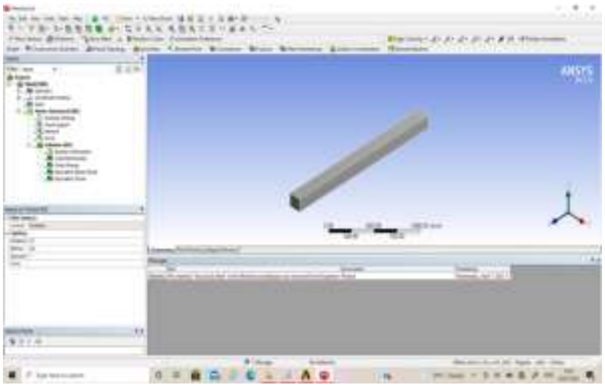


Fig.4 Modelling of Beam

Step 6 Meshing: A mesh is needed to run Finite element analysis (FEA). The mesh takes the 3D part and represents it as many small elements that are connected by nodes. The FEA cannot run without having a mesh defined. In the Mechanical Tree Outline, Rightclick on Meshtab and then select the size of meshing as fine and then right click on Mesh and click on generate mesh the mesh will be generated. There are three types of mesh Course medium and fine out of which fine meshing is selected for more accuracy.

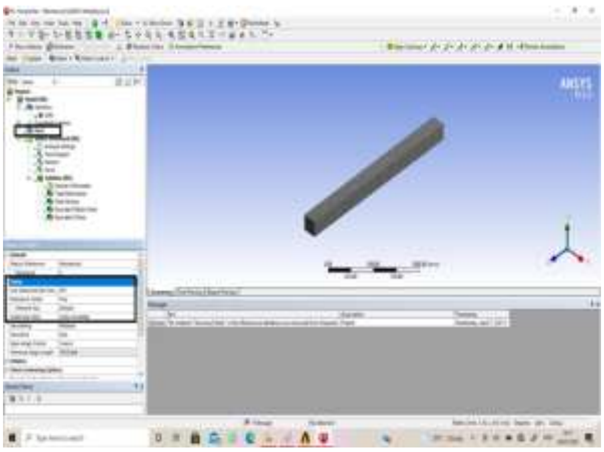


Fig.5 Meshing

Step 7 Setup: Use the Setup cell to launch the appropriate application for that system. Using these options will define loads, boundary conditions, and otherwise configure analysis in the application. Go back to the Project > Setup cell > Edit… Will work again in the same Mechanical model window but will need to right-click on the Static Structural leaf to access the functions needed. Many options will show when select Insert… as shown below. Here we selected different boundary conditions and different loading condition as per problem statement or as per strengthening pattern. We were selected cantilever beam simply supported beam and column with one end free and another fix this boundary condition was selected from this setup cell.

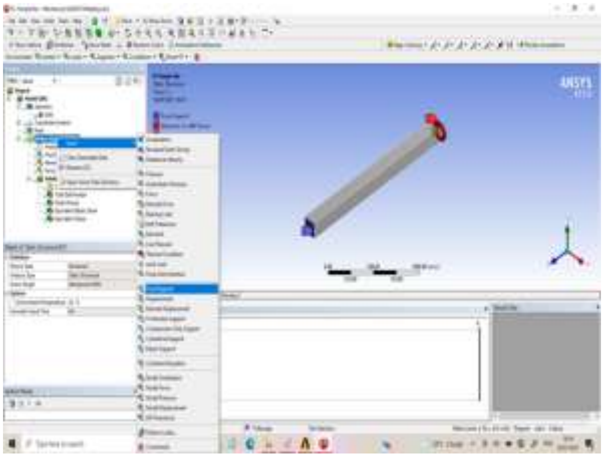


Fig.6 Assigning boundary condition and loads

Step 8 Results: Try to run the analysis now by right-clicking on the Solution leaf in the LHS menu and clicking Solve. To set-up a viewer for the Equivalent Stress we will select Solution>Insert>Stress>Equivalent Stress. For the Total Deformation: To set-up a viewer for the Axial (Normal) Deformation we will select Solution>Insert>Deformation>Total. After inserting require output result right click on solution and click on evaluate all result to obtain different result.

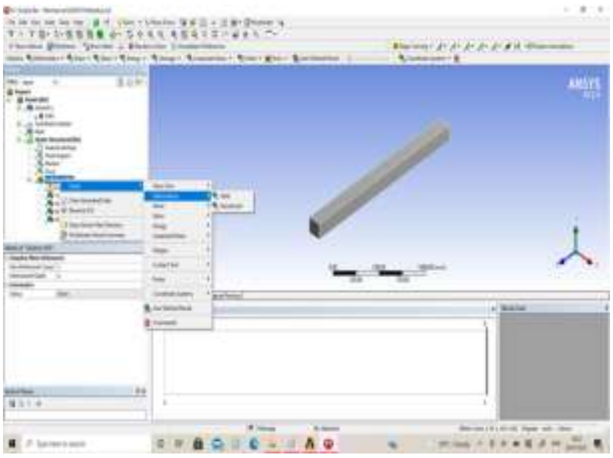


Fig.7 Deformation Tool.

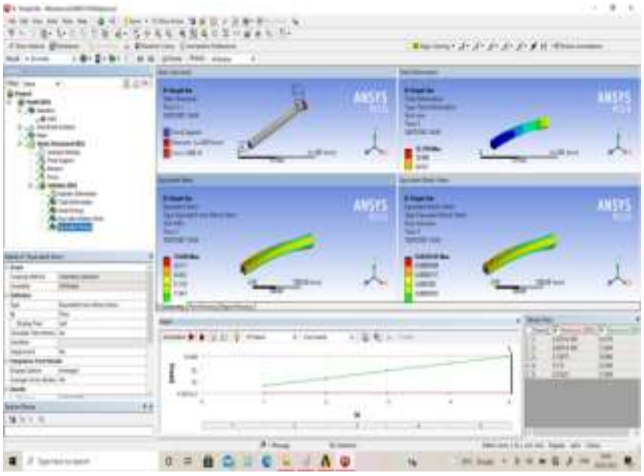


Fig.8 Results window

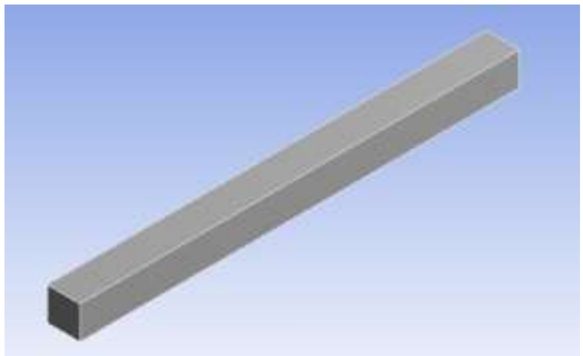
To analyse beam for two-point bending the study was done toinvestigate the effects of different Strengthening technique on beam for improving the bending strength of beam for that simply supported beam was analysed for two-point load of equal magnitude and equally spaced acted on beam and results were recorded for different models. Similarly, to analyse beam for combined torsion and bendingCantilever beam (one end Fixed another end free) were analysed. In this the vertical point load is given on free support and clockwise moment of different magnitude were applied on free end and results of different beam models were observed.

Loads

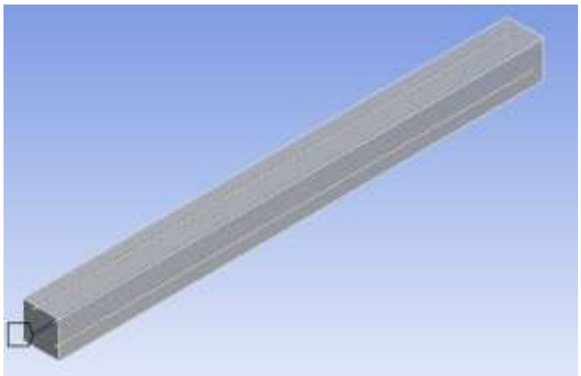
Table 2 Loading for Combined Torsion and Bending of Beam

Steps	Time (Sec)	Force (N)	Moment (N-M)
0	0	0	0
1	1	1000	2000
2	2	2000	4000
3	3	3000	6000
4	4	4000	8000
5	5	5000	10000

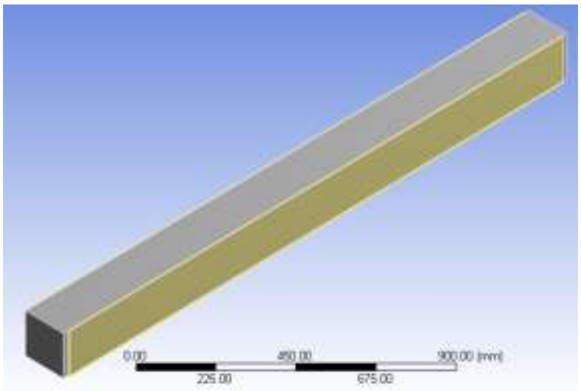
Different models of Beam: Following are the different types of beams analysed in ANSYS15.0 Beam 1 was simple bar Beam 2 was Bar with 10 mm Reinforcing Bar Beam 3 was a beam strengthened with 6mm thick plate to 3 side Beam 4 was beam strengthened with 50 mm batten plates attached to 2 sides of beam and at bottom full plate were attached in Beam 5 only bottom of plate were strengthened with 6 mm thick plate in Beam 6 two equal angle of ISA 50X50X6 were attached to two corners of beam and in Beam 7 two equal angle of ISA 50X50X6 were attached to Middle one third span of beam.



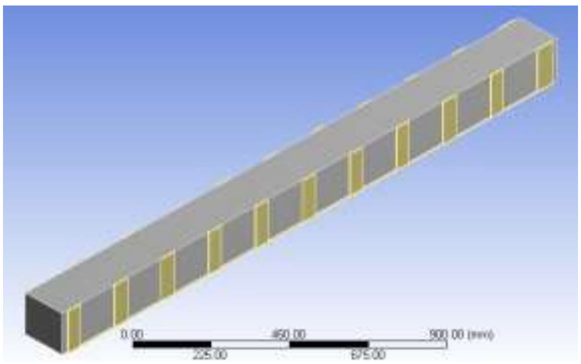
(a) Beam 1



(b) Beam 2

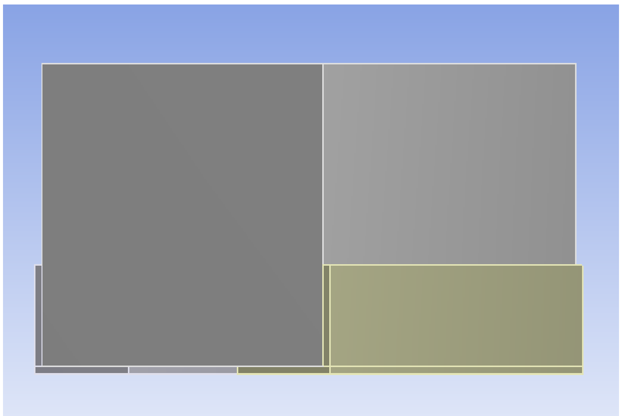


(c) Beam 3

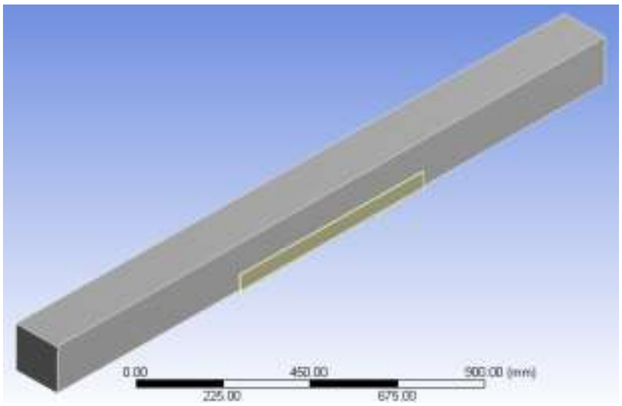


(d) Beam 4

(e) Beam 5



(f) Beam 6



(g) Beam 7

Fig.9 Different Specimens of beams.

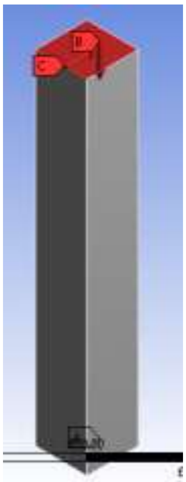
For analyse the column for lateral and vertical loading,vertical as well as lateral load of different magnitude were applied on column and results were recorded after that strengthening technique was applied for avoiding failure of column and analysis will be done and changes in the results were observed.

Load

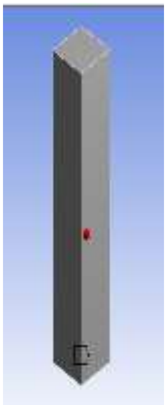
Table 3. load value for column

Time (Sec)	Y-Axis	Z-Axis
0	0	0
1	3000	-2000
2	6000	-4000
3	9000	-6000
4	12000	-8000
5	15000	-10000

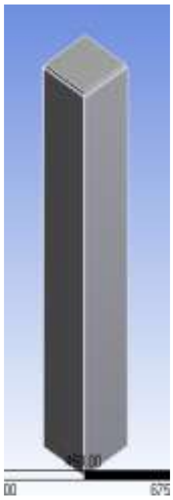
Different models of Column: Following are the different types of Columns were anylized in ANSYS15.0 Col. 1 is simple column without reinforcement Col. 2 was Column with four 10mm Dia bar in Col. 3 6 mm thick plate were attached to all four sides of column in Col. 4 four ISA 50X50X6 mm attached to 4 corners of column and these angles were connected with each other with 50X6 mm batten plate. Col. 5 is same like Col. 4 but in this 100X6 mm batten plate is used for connecting Angles



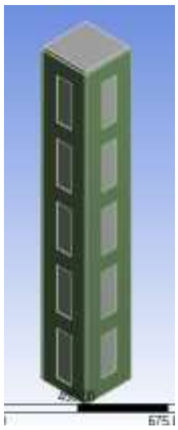
(a) Col. 1



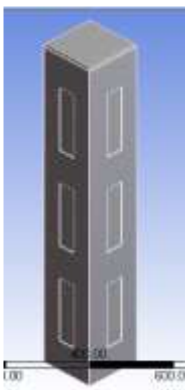
(b) Col. 2



(c) Col. 3



(d) Col. 4



(e) Col. 5

Fig.10 Different Specimens of Columns.

IV. RESULTS:

In order to understand the behaviour of Normal specimen and strengthened specimens under different loading conditions Such as simply supported beam tested for two point bending cantilever beam tested for combined torsion and bending, and column is tested for combined lateral and vertical loads and deformation of each specimen is measured at regular interval of time and graph of deformations v/s time is plotted.

Table. 4 Deformation of Beam For two points Bending.

Deformation (MM) Max							
Time (Sec)	Beam 1	Beam 2	Beam 3	Beam 4	Beam 5	Beam 6	Beam 7
1	6.81E-04	6.42E-04	3.48E-04	4.21E-04	4.36E-04	4.17E-04	9.76E-05
2	1.36E-03	1.29E-03	6.95E-04	8.41E-04	8.72E-04	8.35E-04	1.95E-04
3	2.04E-03	1.93E-03	1.04E-03	1.26E-03	1.31E-03	1.25E-03	2.93E-04
4	2.73E-03	2.57E-03	1.39E-03	1.68E-03	1.75E-03	1.67E-03	3.90E-04
5	3.41E-03	3.22E-03	1.74E-03	2.10E-03	2.18E-03	2.09E-03	4.88E-04

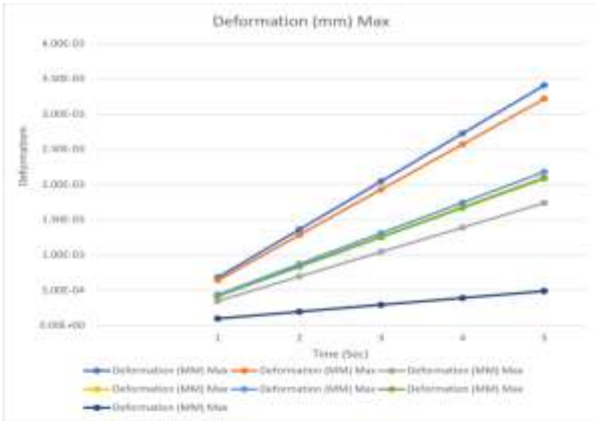


Fig.11 Deformation of Beam For two points Bending

Table 5. Deformation

Deformation (MM)							
Time (Sec)	Beam 1	Beam 2	Beam 3	Beam 4	Beam 5	Beam 6	Beam 7
1	2.47E-03	2.30E-03	1.32E-03	1.60E-03	1.67E-03	1.70E-03	3.69E-04
2	4.94E-03	4.60E-03	2.64E-03	3.20E-03	3.34E-03	3.40E-03	7.37E-04
3	7.42E-03	6.90E-03	3.95E-03	4.81E-03	5.01E-03	5.10E-03	1.11E-03
4	9.89E-03	9.20E-03	5.27E-03	6.41E-03	6.68E-03	6.81E-03	1.47E-03
5	1.24E-02	1.15E-02	6.59E-03	8.01E-03	8.35E-03	8.51E-03	1.84E-03

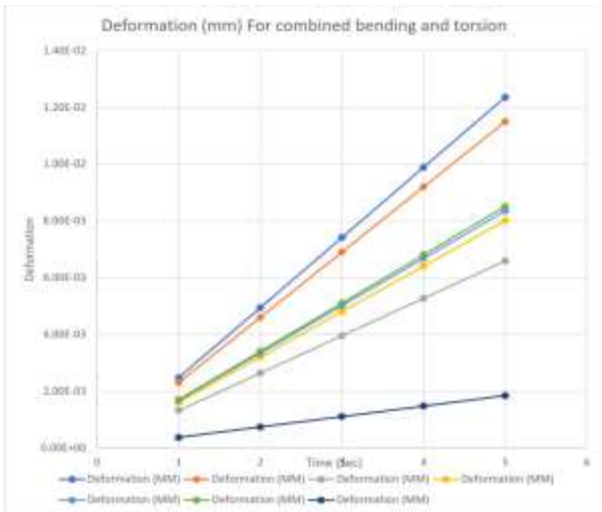


Fig.12 Deformation

Table. 6 Deformation Value

Deformation (MM)					
Time (Sec)	Col. 2	Col. 3	Col. 4	Col. 5	Col. 6
1	5.39E-04	5.03E-04	2.16E-04	2.49E-04	6.79E-05
2	1.08E-03	1.01E-03	4.33E-04	4.98E-04	1.36E-04
3	1.62E-03	1.51E-03	6.49E-04	7.47E-04	2.04E-04
4	2.16E-03	2.01E-03	8.66E-04	9.96E-04	2.72E-04
5	2.70E-03	2.51E-03	1.08E-03	1.25E-03	3.40E-04

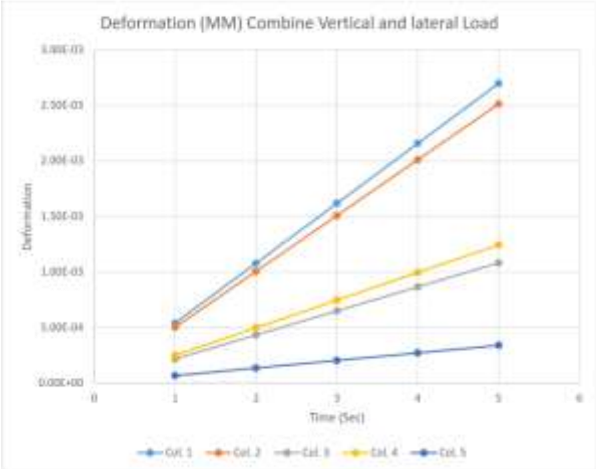


Fig.13 Deformation

V. CONCLUSIONS:

In paper behaviour of different strengthen beam, column models were investigated throughe FEM in ANSYS 15.0. these elements were strengthening by using steel plates and angle section at different location with different loading combinations. The Research focused on the batten plates and angle for enhancing the resistance to deformation from this study following conclusions were drawn.

1. Using batten plates and angle for strengthening of beam and column proved effective since it increases the resistance to deformation.

2. Strengthening element shows greater resistance to deformation as compared to normal element.

3. Angle connected to middle one third span of beam for strengthening prove very effective as compare to other strengthening models for simply supported beam subjected to two-point bending.

4. For cantilever beam subjectede to a combind torsion and bending strengthening with angle to middle one third proven very effective since it will undergo less deformation as compare to other models.

5. Column strengthen with four angles connected to each corner and four batten plates connected to them at equal interval shows greater resistance to deformation as compare to other models.

REFERENCES

[1] Elmasry M.I.S., Abdelkader A.M., Elkordy E.A. (2018) An Analytical Study of Improving Beam-Column Joints Behavior Under Earthquakes. In: Rodrigues H., Elnashai A., Calvi G. (eds) Facing the Challenges in Structural Engineering. GeoMEast 2017. Sustainable Civil Infrastructures. Springer, Cham. https://doi.org/10.1007/978-3-319-61914-9_37

[2] Mahmoud F. Belal, Hatem M. Mohamed & Sherif A. Morad (2015) Behavior of reinforced concrete columns strengthened by steel jacket, HBRC Journal, 11:2, pp. 201-212, DOI:10.1016/j.hbrcj.2014.05.002

[3] Chunli Zhou, Jinkun Sun, Zhaorong Zeng, Jie Liu, "Studies on the RC Beam Strengthened by Using the Exterior-Wrapping U-Shaped Steel Plate and Its Application", Advances in Civil Engineering, vol. 2019, Article ID 2396764, 9 pages, 2019. https://doi.org/10.1155/2019/2396764

[4] ANSYS 15.0. Theory reference. ANSYS Inc.; 2015.

[5] Nasih Habeeb Askandar, Abdulkareem Darweesh Mahmood (2020). "Torsional Strengthening of RC Beams with Near-Surface Mounted Steel Bars", Advances in Materials Science and Engineering, vol. 2020, Article ID 1492980, 11 pages, 2020. https://doi.org/10.1155/2020/1492980

Corresponding Author

Ravikiran P. Jadhav*

PG Student, Civil Engineering Department, SVERI's College of Engineering Pandharpur, Maharashtra

ravikiranjadhav@yahoo.com