Finite Element Analysis of Cellular Beam using Ansys

T.Subramani[1], M.Senthil Kumar[2], K.T.Ajay Bharath [3], S. Rohind Kumar [4], M. Suresh [5]

[1]Professor & Dean, Department of Civil Engineering, VMKV Engineering College, Vinayaka Missions Research Foundation (Deemed to be University), Salem, India

[2] Associate Professor, Department of Civil Engineering, VMKV Engineering College, Vinayaka Missions Research Foundation
(Deemed to be University), Salem, India

[3,4,5]UG Student, Department of Civil Engineering, VMKV Engineering College, Vinayaka Missions Research Foundation (Deemed to be University), Salem, India

ABSTRACT

Castellated beams are fabricated by cutting the web of hot rolled steel (HRS) I section into zigzag pattern and thereafter rejoining it over one another. The openings made in the webs are of generally hexagonal, circular, diamond or square in shape. Therefore, considering structural performance of the beam, the size and shape of openings provided in the web are always an important issue of concern.Cross-section dimension, beam slenderness, opening size and opening spacing are found to affect the stiffness of the analytical load-deflection curve. The parametric study of the FE stiffness and normalized stiffness is conducted. The stress distribution of the finite element (FE) model reveals the strut stress in the web-post contributes to the increasing deflection in addition to the regular bending deflection. The deformation of the web-post and the relative movement between the upper and lower tee-sections due to strut stress is the main reason of the additional deflection in the cellular beams. The effect of the strut stress is found to be significant for the deflection of the short-span beams but less for the long-span beams. In this project the cellular beams with different configurations are analysed using FEM ANALYSIS in Ansys software and the results obtained are discussed

Keywords: Finite Element, Analysis, Cellular beam, Ansys.

1. INTRODUCTION

Many structures are using steel as their constructional element. Castellated beam reduces the cost of structural steel and also increase the stiffness of steel members. Castellated beams are beams having holes or castellation’s on its web portion. It is made by cutting the web portion of the solid beam in a zigzag pattern and then arranging the two halves in such a way that castellation’s are made in the web portion. It is the then welded together to form a castellated beam. In castellated beams, one can increase the depth of the beam without any additional steel. Castellated beams are such structural members, which are made by flame cutting a rolled beam along its centerline and then rejoining the two halves by welding so that the overall beam depth is increased by 50% for improved structural performance against bending. Since Second World War, many attempts have been made by structural engineers to find new ways to decrease the cost of steel structures. Due to limitations on minimum allowable deflection, the high strength properties of structural steel cannot always be utilized to best advantage. As a result, several new methods aimed at increasing stiffness of steel member, without any increase in weight of steel required. Castellated beam is one of the best solutions.

The re-routing of services (or increasing the floor height at the design stage for accommodating them) leads to additional cost and is generally unacceptable. The provision of beams with web openings has become an acceptable engineering practice, and eliminates the probability of a service engineer cutting holes subsequently in inappropriate locations. Beams with web openings can be competitive in such cases, even though other alternatives to solid web beams such as stub girders, trusses etc. are available. This form of construction maintains a smaller construction depth with placement of services within the girder depth, at the most appropriate locations. The introduction of an opening in the web of the beam alters the stress distribution within the member and influences its collapse behavior. Steel structure building are becoming more and more popular due to their many advantages such as the better satisfaction with the flexible architectural, durability, strength, design, low inclusive cost and environmental protect as steel is manufacture to precise and uniform shapes. The two halves are then welded together to produce a beam of greater depth with hexagonal opening in the web. Figure 1 shows the castellated beam.

Figure 1 Castellated beam

The resulting beam has a larger section modulus and greater bending rigidity than the original section without an increase in weight. However, the presence of the holes in the web will change the structural behavior of the beam from that of plain webbed beams. Experimental tests on castellated beams have shown that beam slenderness, castellation parameters and the loading type are the main parameters, which indicate the strength and modes of failure of these beams.

a. Castellate Beams

A castellated beam is a beam style where an I-beam is subjected to a longitudinal cut along its web following a specific pattern. The purpose is to divide and reassemble the beam with a deeper web by taking advantage of the cutting pattern. Figure 2 shows the castellated beam structure.

Figure 2 Castellated beam structure

The recent structures which are built these days follow the traditional design and the components like a beam, columns, etc. are designed in a way that only the loads get transferred easily but are highly uneconomical.

i. Properties

- The castellated beams have a 50% much deeper web than nominal beams.
- It can handle 40% more moment carrying capacity than nominal beams.
- These can be designed of wide span and can be made of wide bay.
- These have less weight of top flange and heavier weight of bottom flange, which makes it better than nominal beams.
- Maximum floor-to-floor height can be attained and these can be easily erected.

ii. Applications

- Used in the construction of bridges
- It is used in the construction of dams

- Used in the construction of high-rise structures

iii. Advantages in using Castellated Beams

Castellated beams refer to the type of beams, which involve expanding a standard rolled steel section in such a way that a predetermined pattern is cut on section webs and the rolled section is cut into two halves.

The two halves are joined together by welding and the high points of the web pattern are connected together to form a castellated beam. The castellated beams were commonly used in Europe in 1950s due to the limited ranges of the available steel rolled section and the cheap labor cost. In terms of structural performance, the operation of splitting and expanding the rolled steel sections helps to increase the section modulus of the beams.

b. Objective

- To study the behavior of Load vs Deflection
- Castellated beams are introduced along the web opening in to minimize the deflection and control failure.
- To study the experimental and analytical investigation of static loading
- To compare analytical and experimental investigation.
- To study different failure mode in a castellated beam

2. METHODOLOGY

Figure 3 shows the methodology of the study.

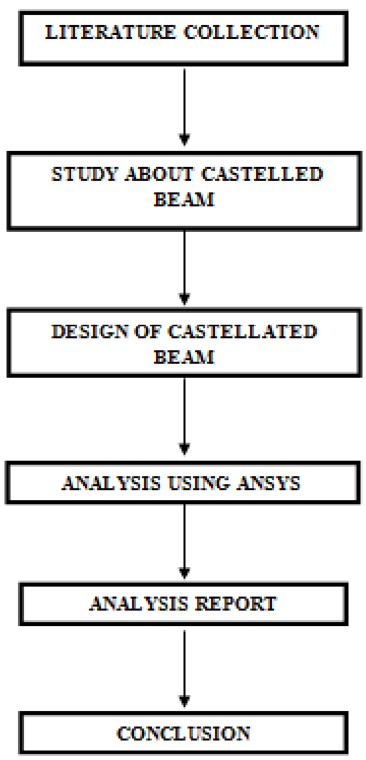

Figure 3 Methodology

3. CASTELLED BEAM

a. Terminology

Throughout this paper, various terms will be used to discuss Cellular beam components and testing results.

i. Web Post

The cross-section of the Cellular beam where the section is assumed to be a solid cross-section.

ii. Throat Width

The length of the horizontal cut on the root beam. The length of the portion of the web that is included with the flanges.

iii. Throat Depth

The height of the portion of the web that connects to the flanges to form the tee section.

iv. Expansion Percentage

The percentage change in depth of the section from the root (original) beam to the fabricated Cellular section.

b. Fabrication of Castellated Beam and Cellular Beam

Fabrication of castellated beams is a comparatively simple series of operations when adequate handling and controlling equipment is used. Structural Steel by burning two or more atA time, depending upon their depth. Splitting is performed by using a component of the oxy-acetylene gas cutter equipment shown in fig. This is an electrically propelled buggy, which runs on a fixed track. The buggy has building burning patterns that can be adjusted to any one of live standard longitudinal "module" dimensions and to any hall-opening height. Castellatedsteel beams fabricated from standard hot-rolled I-sections have many advantages including greater bending rigidity, larger section modulus, optimum self-weight–depth ratio, economic construction, ease of services through the web openings and aesthetic architectural appearance. However, the castellation of the beams results in distinctive failure modes depending on geometry of the beams, size of web openings, web slenderness, type of loading, quality of welding and lateral restraint conditions. The failure modes comprise shear, flexural, lateral torsional buckling, rupture of welded joints and web post buckling failure modes.

Cellular steel beams fabricated from standard hot-rolled I-sections have many advantages including greater bending rigidity, larger section modulus, optimum self-weight–depth ratio, economic construction, ease of services through the web openings and aesthetic architectural appearance. However, the castellation of the beams results in distinctive failure modes depending on geometry of the beams, size of web openings, web slenderness, type of loading, quality of welding and lateral restraint conditions. The failure modes comprise shear, flexural, lateral torsional buckling, rupture of welded joints and web post buckling failure modes. Investigation of these failure modes was previously detailed by Kerdal andNethercot. Also detailed review of the experimental and theoretical investigations on the failure modes of Cellular beams. However, accurate finite element modelling of the buckling behavior of Cellular steel beams is quite complicated due to the presence of the initial geometric imperfections, web openings, lateral buckling restraints and loading conditions.

c. Design Requirements

In the design of beam, two aspects are primary consideration, Strength requirements, that is the beam has adequate strength to resist the applied bending moments and accompanying shear forces and Stability consideration, which is the member is safe against buckling. Because of the opening in the web various failure modes are expected to happen, which need to be checked and designed for. The strength of a beam with various web openings shall be determined based on the interaction of flexure and shear at the web opening. Design constraints include the displacement limitations, overall beam flexural capacitybeams hear capacity, overall beam buckling strength, web post buckling andVierendeel bending of upper and lower tees.

These are the steps followed for designing the castellated beam.

- Calculate moment of resistance (MR)
- Load applied over the section (W)
- Beam is checked for shear (va)
- Beam is checked for deflection
- Check for combined local bending and direct stresses
- Design of Stiffeners Beam is checked for shear (va)
- Beam is checked for deflection
- Check for combined local bending and direct stresses
- Design of Stiffeners

d. Design Of Castellated Beam

- The angle of cut is selected to be 45°. For a good design the depth of stem of the t-section at the minimum beam cross-section should not be less than by 4 of the original beam sections.
- The load over the section from the roof are a curtained and the maximum bending moment are computed.
- The cross-sectional area of the t-section at the open throat is calculated. Neutral axis of the section is determined and moment of inertia about the neutral axis is calculated.
- The moment of resistance of the castellated beam which is the product of the resultant tensile or compressive force and the distance between the centroid of T-section is calculated.

$$M.R.=A\times\sigma at \times d$$

Where
A = area of the T section at open throat
D = distance between the centroid of T section
The moment of resistance of the castellated beam should be more the maximum moment. 5.

- The spacing of castellated beam should not exceed the spacing determined by following equation.

$$S = P/ W\times l$$

Where S= c/c distance between the castellated beam in meter.
P = net load carrying capacity in N
W = design load in N / m^2
l = span of the in meter.

- Stiffeners are designed at the supports and below the concentrated loads.
- The beam is checked in shear. The average shear at ends is calculated from following equation

$$\tau va = R/d'\times t$$

< 0.4 fy Where
R = end reaction in N
d' = depth of the stem of T section
t = thickness of stem

- The maximum combined local bending stress and direct stress in T Segments is also workout and should be less than the permissible bending stress.
- The maximum deflection of T Segment is calculated. This occurs at the mid span is due to the net load carrying capacity load capacity.

Let,
δ1 = deflection due to net load carrying capacity
δ 2 = deflection due to local effects
I = average moment of inertia of the section I
T = moment of inertia of T section
P = number of perforation panels in half span.
δ1 = 5 WL3/384 EI
δ2= VavgP(m+n)3 /24EIT

$$\delta= \delta1 +\delta2 < L/325$$

4. ANALYSIS OF CASTELLATED BEAM

4.1 Introduction of FEM

The FEM is a mathematical performance for outcome nearby experimental solutions to various computational domains. Numerical analysis done using FEM is commonly mentioned as a finite element analysis (FEA). Typical FEA applications consist of structural, thermal, electromagnetic and fluid field problems. Engineers usage it to decrease the several physical models & tests & optimize components in their design segment to improve better products, quicker.
ANSYS is preferred-purpose FEA software. FEA is a mathematical method of discretizing a complicated design into very small pieces (of user-specific length) known as factors. The software Implements equations that govern the performance of these factors and solves all of them growing a comprehensive clarification of how the device acts as an entire. Those consequences can be supplied in tabulated or graphical forms.

i. FEA Works

FEA as useful in engineering possibly will be a machine tool for performance arts engineering analysis. It consists of the operation of mesh creation techniques for dividing a difficult problem into tiny parts, moreover because of the use of software package program coded with FEM rule.

ii. Advantages of FEA Software

- It reduces the amount of prototype testing, thereby saving the cost & time.
- It gives the graphical representation of the outcome of the analysis.
- The finite element modelling & analysis are performed in the pre-processor & solution phases, which if done manually would consume a lot of time & in some cases, might be impossible to perform.
- It helps optimize a design.

5. ANALYSIS OF "I – SECTION"

Finite element analysis (FEA) is the process of simulating the behavior of a part or assembly under given conditions so that it can be assessed using the finite element method (FEM). FEA is used by engineers to help simulate physical phenomena and thereby reduce the need for physical prototypes, while allowing for the optimization of components as part of the design process of a project.

a. Circular Cut Section

Table 1 shows the circular cut section result.

Table 1: Circular Cut Section Result

S. No	Load	Deformation	Stress	Strain	Safety Factor
1	5	0.1701	43.581	2.9866e-004	8.0310
2	10	0.3403	87.162	5.9731e-004	4.0155
3	15	0.5105	130.74	8.9597e-004	2.6770
4	20	0.6807	174.32	1.1946e-003	2.0078
5	25	0.8508	217.90	1.4933e-003	1.6062
6	30	1.0211	261.49	1.7919e-003	1.3385
7	35	1.1912	305.07	2.0906e-003	1.1473
8	40	1.3614	348.65	2.3892e-003	1.0039
9	45	1.5316	392.23	2.6879e-003	0.8923
10	50	1.7018	435.81	2.9866e-003	0.8031
11	55	1.8719	479.39	3.2852e-003	0.7300
12	60	2.0421	522.97	3.5839e-003	0.6692

The circular cut I section model is analysis based on single point load gradually increase (5kN to maximum load) acting with help of Ansys software. To predict the deformation, stress, strain & safety factor in each load condition value mention in the Table 1. Its observed gradually deformation and stress are increased and automatically safety factor is decreased, so maximum load reached at 60kN (deformation 2.0421mm, Stress 522.97Mpa, Safety factor 0.6692) is obtained.

i. Model

Figure 4 shows the circular cut section in cellular beam.

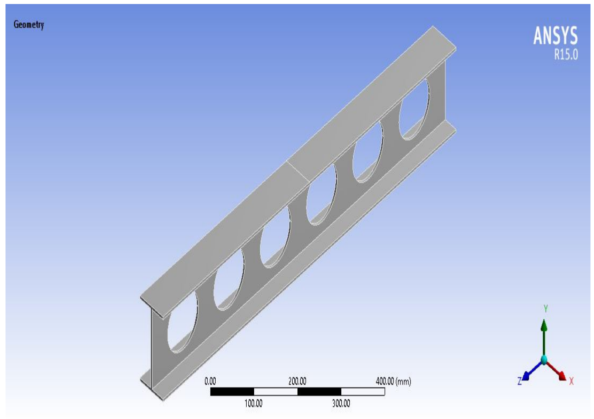

Figure 4 circular cut section in cellular beam

ii. Mesh

Figure 5 shows the circular cut sectionn mesh.

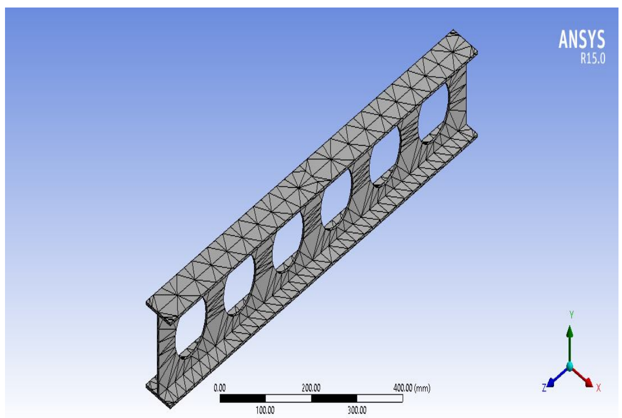

Figure 5 circular cut section in mesh

iii. Deformation

Figure 6 shows the circular cut section in deformation.

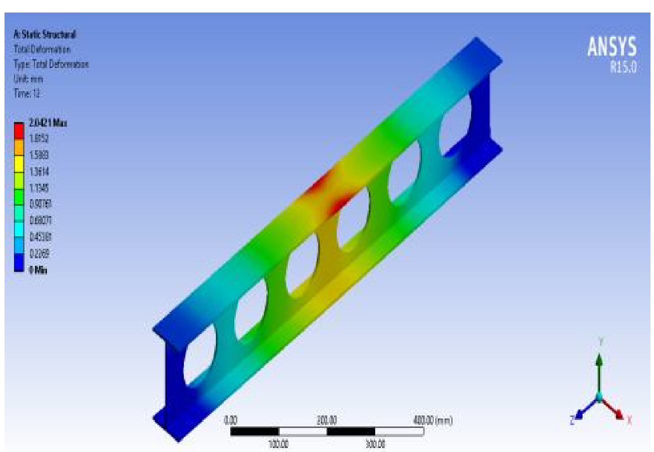

Figure 6 circular cut section in deformation

iv. Stress

Figure 7 shows the circular cut section stress.

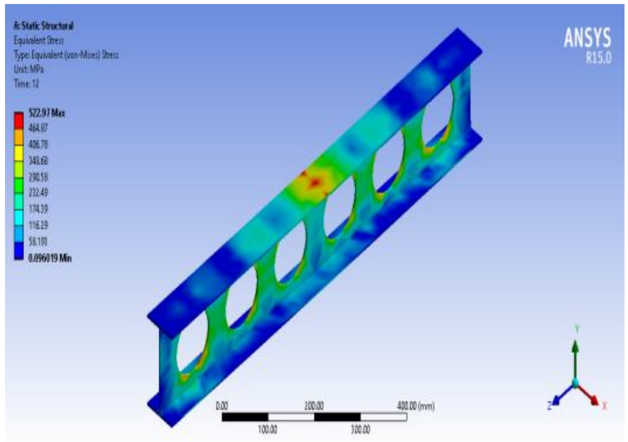

Figure 7 circular cut section stress

v. Strain

Figure 8 shows the circular cut section in strain.

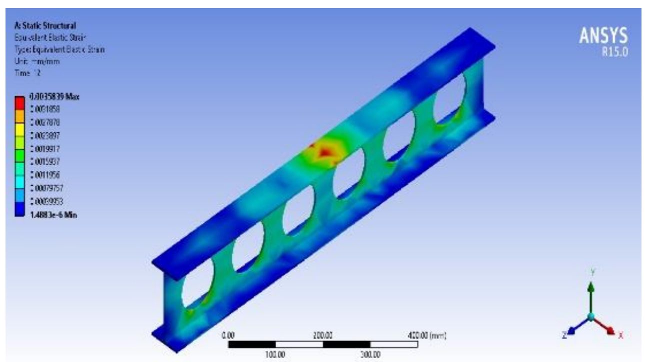

Figure 8 circular cut section in strain

vi. Safety Factor

Figure 9 shows the circular cut section in safety factor.

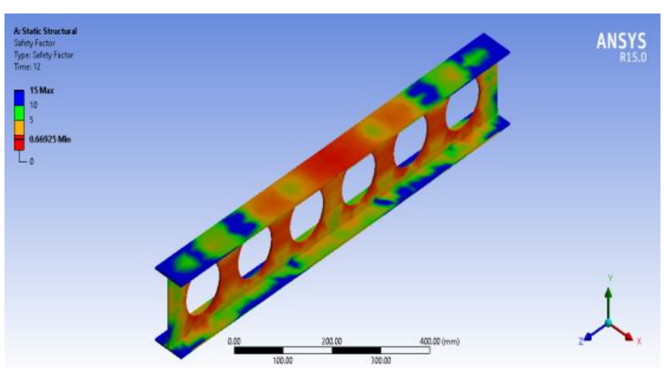

Figure 9 circular cut section in safety factor

b. Hexagonal Cut Section Result

Table 2 shows the hexagonal cut section result.

Table 2: Hexagonal Cut Section Result

S. No	Load	Deformation	Stress	Strain	Safety Factor
1	5	0.1996	59.510	3.0392e-004	5.8813
2	10	0.3993	119.02	6.0785e-004	2.9407
3	15	0.5989	178.53	9.1177e-004	1.9604
4	20	0.7986	238.04	1.2157e-003	1.4703
5	25	0.9982	297.55	1.5196e-003	1.1763
6	30	1.1979	357.06	1.8235e-003	0.9802
7	35	1.3976	416.57	2.1275e-003	0.8401
8	40	1.5972	476.08	2.4314e-003	0.7351
9	45	1.7969	535.59	2.7353e-003	0.6534
10	50	1.9965	595.10	3.0392e-003	0.5881
11	55				
12	60				

The hexagonal cut I section model is analysis based on single point load gradually increase (5kN to maximum load) acting with help of Ansys software. To predict the deformation, stress, strain & safety factor in each load condition value mention in the Table 2. It's observed gradually deformation and stress are increased and automatically safety factor is decreased, so maximum load reached at 50kN (deformation 1.9965mm, Stress 595.10Mpa, Safety factor 0.5881) is obtained.

i. Model

Figure 10 shows the hexagonal cut section in cellular beam.

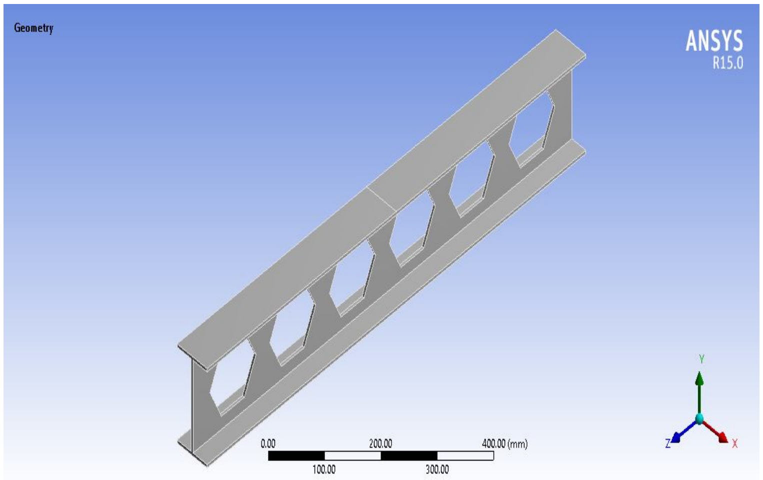

Figure 10 Hexagonal cut section in cellular beam

ii. Mesh

Figure 11 shows the hexagonal cut section in mesh.

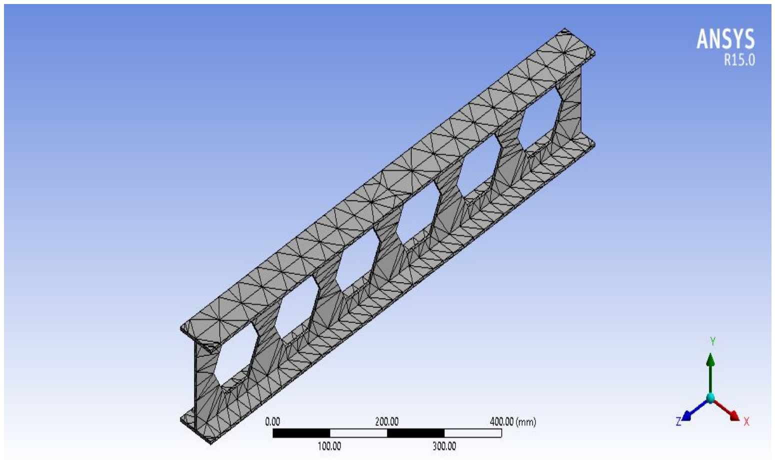

Figure 11 Hexagonal cut section in mesh

iii. Deformation

Figure 12 shows the hexagonal cut section in deformation.

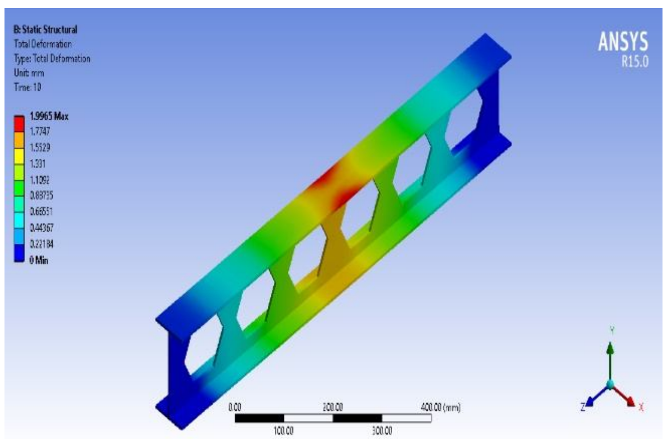

Figure 12 Hexagonal cut section in deformation

iv. Stress

Figure 13 shows the hexagonal cut section in stress.

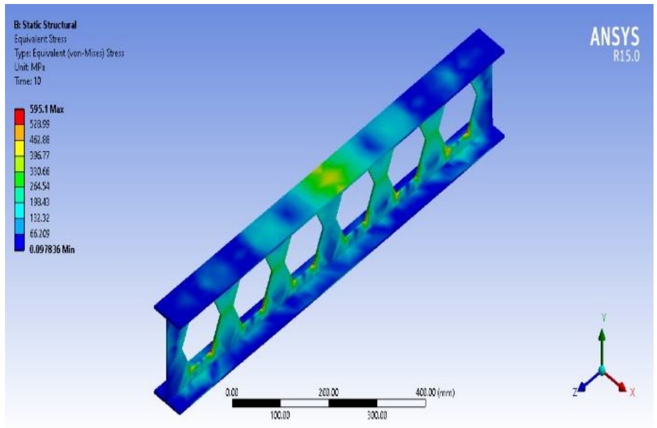

Figure 13 Hexagonal cut section in stress

v. Strain

Figure 14 shows the hexagonal cut section in strain.

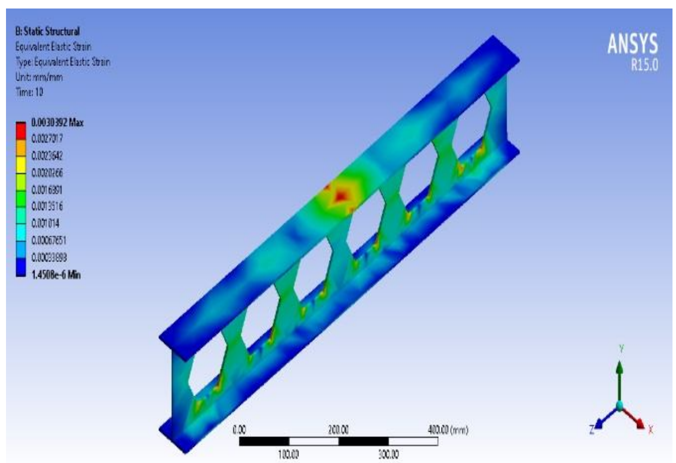

Figure 14 Hexagonal cut section in strain

vi. Safety Factor

Figure 15 shows the hexagonal cut section in safety factor.

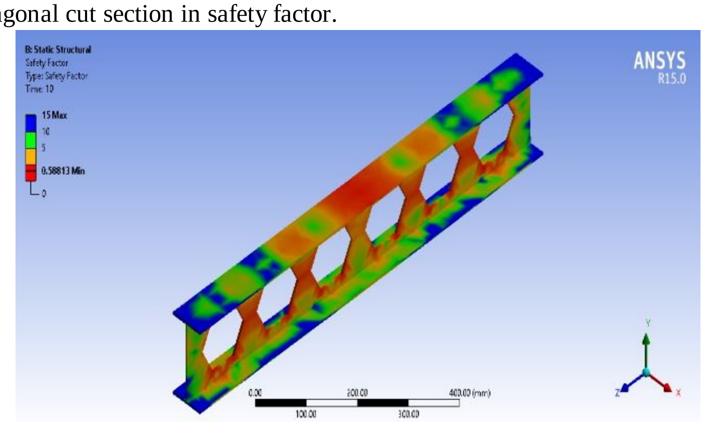

Figure 15 Hexagonal cut section in safety factor

c. Rectangular Cut Section Result

Table 3 shows the rectangular cut section result.

Table 3: Rectangular Cut Section Result

S. No	Load	Deformation	Stress	Strain	Safety Factor
1	5	0.2294	58.484	3.1644e-004	5.9846
2	10	0.4589	116.97	6.3288e-004	2.9923
3	15	0.6883	175.45	9.4933e-004	1.9949
4	20	0.9178	233.93	1.2658e-003	1.4961
5	25	1.1473	292.42	1.5822e-003	1.1969
6	30	1.3767	350.90	1.8987e-003	0.9974
7	35	1.6062	409.38	2.2151e-003	0.8549
8	40	1.8356	467.87	2.5315e-003	0.7480
9	45	2.0651	526.35	2.8480e-003	0.6649
10	50	2.2945	584.84	3.1644e-003	0.5984
11	55	-	-	-	-
12	60	-	-	-	-

The rectangular cut I section model is analysis based on single point load gradually increase (5kN to maximum load) acting with help of Ansys software. To predict the deformation, stress, strain & safety factor in each load condition value mention in the Table 3. It's observed gradually deformation and stress are increased and automatically safety factor is decreased, so maximum load reached at 50kN (deformation 2.2945mm, Stress 584.84Mpa, Safety factor 0.5984) is obtained.

i. Model

Figure 16 shows the rectangular cut section in cellular beam.

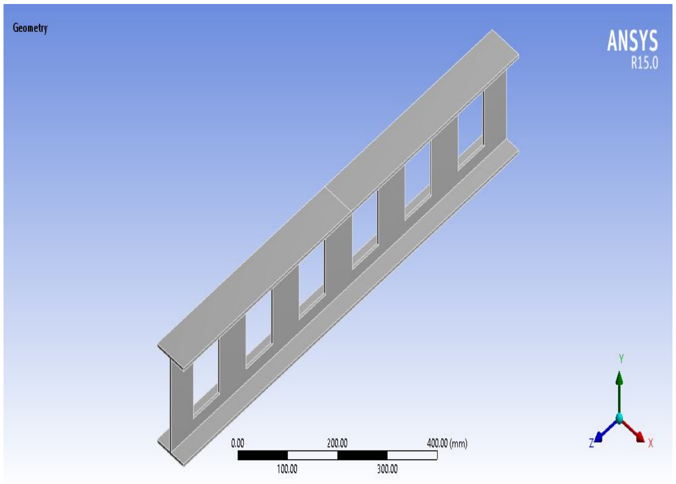

Figure 16 Rectangular cut section in cellular beam

ii. Mesh

Figure 17 shows the Rectangular cut section in mesh.

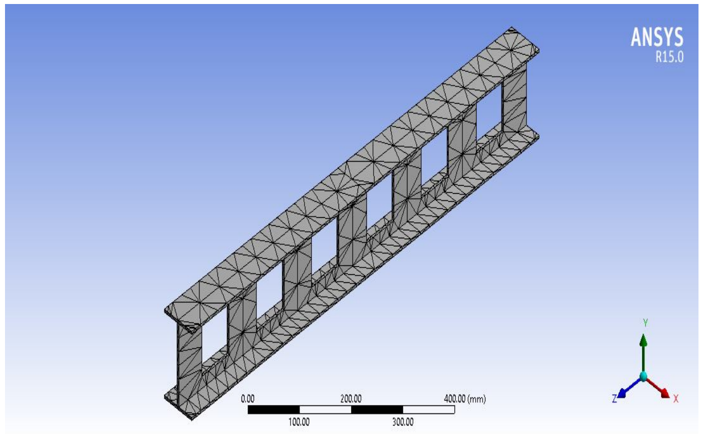

Figure 17 Rectangular cut section in mesh

iii. Deformation

Figure 18 shows the rectangular cut section in deformation.

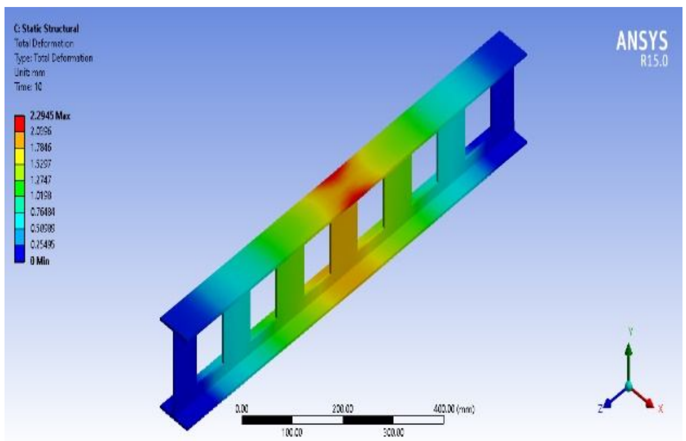

Figure 18 Rectangular cut section in deformation

iv. Stress

Figure 19 shows the rectangular cut section in stress.

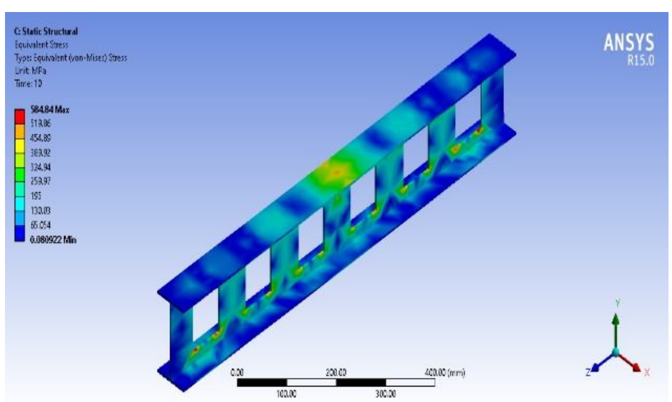

Figure 19 Rectangular cut section in stress

v. Strain

Figure 20 shows the rectangular cut section in strain.

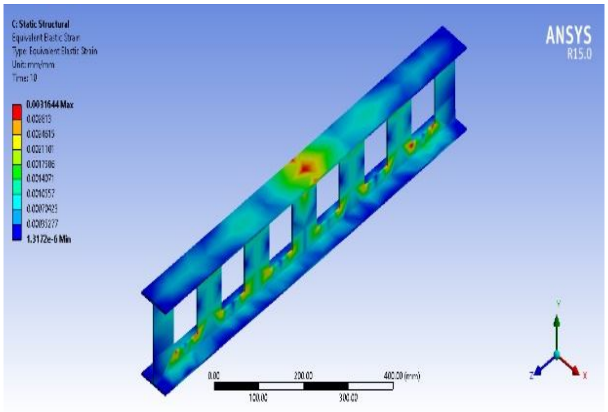

Figure 20 Rectangular cut section in strain

vi. Safety Factor

Figure 21 shows the rectangular cut section in safety factor.

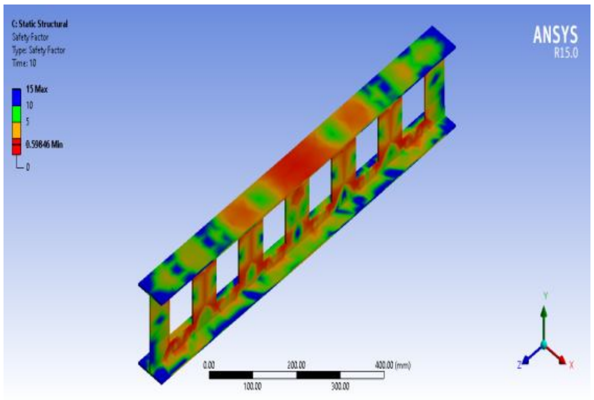

Figure 21 Rectangular cut section in safety factor

6. RESULT & DISCUSSION

Table 4 shows the comparison of all section.

Table 4: Comparison of all Section

Load	Circular		Hexagonal		Rectangular	
	Deformation	Safety	Deformation	Safety	Deformation	Safety
5	0.1701	8.0310	0.1996	5.8813	0.2294	5.9846
10	0.3403	4.0155	0.3993	2.9407	0.4589	2.9923
15	0.5105	2.6770	0.5989	1.9604	0.6883	1.9949
20	0.6807	2.0078	0.7986	1.4703	0.9178	1.4961
25	0.8508	1.6062	0.9982	1.1763	1.1473	1.1969
30	1.0211	1.3385	1.1979	0.9802	1.3767	0.9974
35	1.1912	1.1473	1.3976	0.8401	1.6062	0.8549
40	1.3614	1.0039	1.5972	0.7351	1.8356	0.7480
45	1.5316	0.8923	1.7969	0.6534	2.0651	0.6649
50	1.7018	0.8031	1.9965	0.5881	2.2945	0.5984
55	1.8719	0.7300				
60	2.0421	0.6692				

The comparison of different cellular beam configuration (Circular, Hexagonal, Rectangular cut section of beam), its gradually load apply in the top surface of beam at each 5kN interval. So predicted the deformation & safety factor is comparison in above Table and each cut section graphically represented one by one below. Its comparatively circular cut section cellular model is higher load carrying capacity compared to other model of cellular. Figure 22 shows the comparison of deformation in circular cut section.

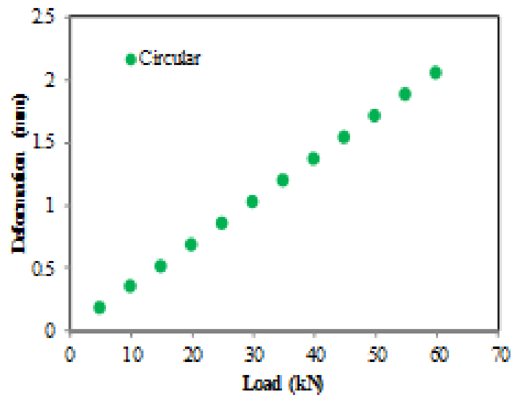

Figure 22 Comparison of Deformation in Circular Cut Section

Figure 23 shows the comparison of deformation in hexagonal cut section.

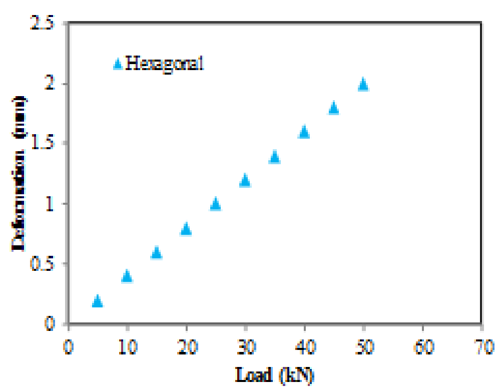

Figure 23 Comparison of Deformation in Hexagonal CutSection

- Its circular cut section cellular beam model is maximum load carrying capacity is 60kN, so to predict the deformation is 2.0421mm
- Its Hexagonal cut section cellular beam model is maximum load carrying capacity is 50kN, so to predict the deformation is 1.9965mm

Figure 24 shows the comparison of deformation in rectangular cut section.

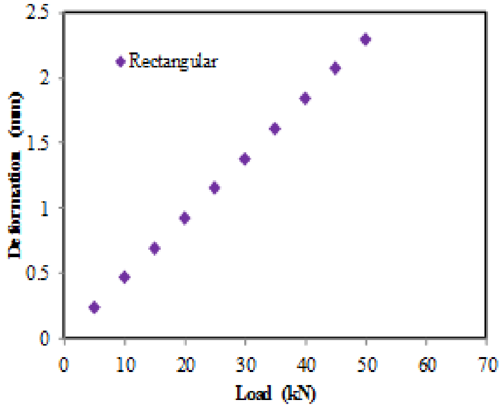

Figure 24 Comparison of Deformation in Rectangular Cut Section

Figure 25 shows the comparison of over all deformation.

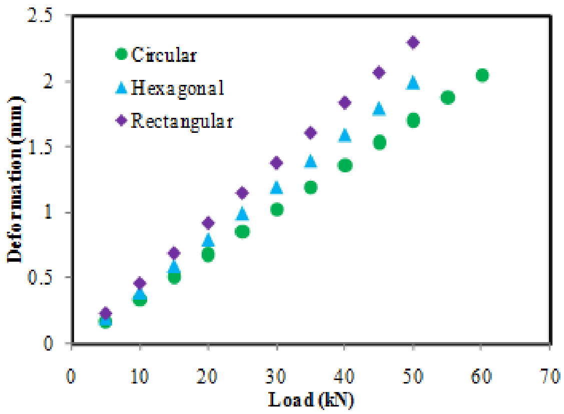

Figure 25 Comparison of Overall Deformation

- Its Rectangular cut section cellular beam model is maximum load carrying capacity is 50kN, so to predict the deformation is 2.2945mm.
- The cellular beam is various cut section is analysis, so its observed circular cut section is reduced the deformation and maximum load carrying capacity.

Figure 26 shows the comparison of overall deformation.

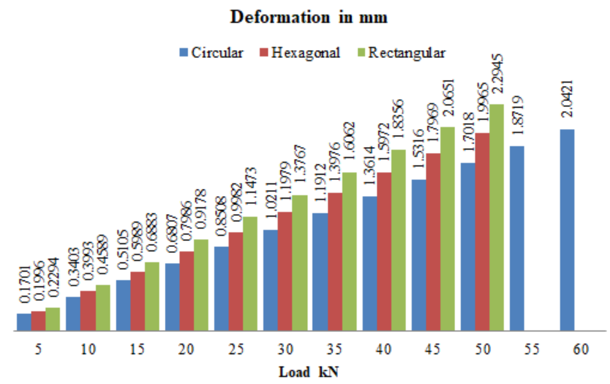

Figure 26 Comparison of overall Deformation

Figure 27 shows the comparison of safety factor.

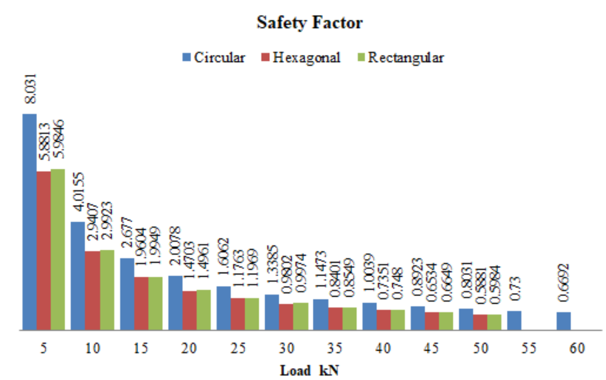

Figure 27 Comparison of Safety Factor

7. CONCLUSION

By comparing the results, it is observed that the finite element analysis with help of Ansys software shows that its circular cut section model achieved better result compare to the other models. All the parameter of maximum load condition (deformation - 2.0421mm, stress – 522.97Mpa, strain – 0.0035839 & safety factor – 0.6692) in circular cut section model. All the model, result data, comparison of table and deformation comparison are graphically represented in the figure.

References

[1] T.Subramani, A.Arul, "Design And Analysis Of Hybrid Composite Lap Joint Using Fem" International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 5), pp 289- 295, 2014.

[2] T.Subramani., J.Jothi,, M.Kavitha "Earthquake Analysis Of Structure By Base Isolation Technique In SAP", International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 5), pp 296 - 305, 2014.

[3] T.Subramani., R.Manivannan.R, M.Kavitha, "Crack Identification In Reinforced Concrete Beams Using Ansys Software" ,International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 133 - 141, 2014.

[4] T.Subramani., Reni Kuruvilla, J.Jayalakshmi., "Nonlinear Analysis Of Reinforced Concrete Column With Fiber Reinforced Polymer Bars" International Journal of Engineering Research and Applications Volume. 4, Issue. 6 (Version 5), pp 306- 316, 2014.

[5] T.Subramani, D.Sakthi Kumar, S.Badrinarayanan. "Fem Modelling And Analysis Of Reinforced Concrete Section With Light Weight Blocks Infill " International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 142 - 149, 2014.
[6] T.Subramani, B.Saravanan., J.Jayalakshmi., "Dynamic Analysis Of Flanged Shear Wall Using Staad Pro", International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 150 - 155, 2014.
[7] T.Subramani, M.Subramani., K.Prasath.,"Analysis Of Three Dimensional Horizontal Reinforced Concrete Curved Beam Using Ansys" International Journal of Engineering Research and Applications, Volume. 4, Issue. 6 (Version 6), pp 156 - 161, 2014.
[8] T.Subramani., K.Bharathi Devi., M.S.Saravanan. , Suboth , Analysis Of RC Structures Subject To Vibration By Using Ansys," International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.45-54, 2014.
[9] T.Subramani., K.Bharathi Devi., M.S.Saravanan., Suboth Thomas, "Analysis Of Seismic Performance Of Rock Block Structures With STAAD Pro International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.55- 68, 2014.
[10] T.Subramani., T.Krishnan., M.S.Saravanan.M , Suboth Thomas, "Finite Element Modeling On Behaviour Of Reinforced Concrete Beam Column Joints Retrofitted With CFRP Sheets Using Ansys" International Journal of Engineering Research and Applications Vol. 4, Issue 12(Version 5), pp.69 -76, 2014
[11] T.Subramani., S.Krishnan., M.S.Saravanan.M, Suboth Thomas "Analysis Of Retrofitting Non-Linear Finite Element Of RCC Beam And Column Using Ansys" International Journal of Engineering Research and Applications ,Vol. 4, Issue 12(Version 5), pp.77-87, 2014.
[12] T.Subramani, J.Jayalakshmi , " Analytical Investigation Of Bonded Glass Fibre Reinforced Polymer Sheets With Reinforced Concrete Beam Using Ansys" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 4, Issue 5, pp. 105-112 , 2015
[13] T.Subramani and M.Kavitha, "Analysis Of Reliability Of Steel Frame Systems With Semi-Rigid Connections Using Numerical Method And Finite Element Analysis", International Journal of Applied Engineering Research (IJAER), Volume 10, Number 38,Special Issues, pp.28240-28246, 2015.
[14] T.Subramani, M.S.Saravanan, "Analysis Of Non Linear Reinforced And Post Tensioned Concrete Beams Using ANSYS", International Journal of Applied Engineering Research (IJAER) International Journal of Applied Engineering Research (IJAER), Volume 10, Number 38 Special Issues, pp.28247-28252, 2015
[15] T.Subramani, K.Balamurugan , " Finite Element Anaylsis Of Composite Element For FRP Reinforced Concrete Slab By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 076-084 , 2016 .
[16] T.Subramani, V.Kanian Poonkundran , " Prefabricated Multistory Structure Exposure To Engineering Seismicity By Using SAP" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 123-131 , 2016 .
[17] T.Subramani, A.Kumaravel , " Analysis Of Polymer Fibre Reinforced Concrete Pavements By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 132-139 , 2016 .
[18] T.Subramani, R.Praburaj , " Pushover Anaylsis Of Retrofitted Reinforced Concrete Buildings By Using SAP" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 140-147 , 2016 .
[19] T.Subramani, M.Senthilkumar , " Finite Element Anaylsis Of RC Beams With Externally Bonded Simcon Laminates By Using ANSYS" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 148-155 , 2016
[20] T.Subramani, R.Vasanthi , " Earth Quake Resistant Building Using SAP" , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 173-181 , 2016 .
[21] T.Subramani, A.Selvam , " Studies On Economical Configuration Of RCC And Prestressed Shell Roofs By Using ANSYS " , International Journal of Application or Innovation in Engineering & Management (IJAIEM) , Volume 5, Issue 5, pp. 182-191 , 2016 .
[22] T.Subramani, A.Anbuchezian , " Experimental Investigation On Flexural Behavior Of Folded Ferro Cement Panels " , International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 6, Issue 3, March 2017 , pp. 045-049 , ISSN 2319 - 4847.

[23] T.Subramani, A. Fizoor Rahman , " An Experimental Study On The Properties Of Pet Fibre Reinforced Concrete " , International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 6, Issue 3, March 2017 , pp. 058-066 , ISSN 2319 - 4847.

[24] T.Subramani, S.Poongothai, S.Priyanka , " Analytical Study Of T Beam Column Joint Using FEM Software " , International Journal of Emerging Trends & Technology in Computer Science (IJETTCS), Volume 6, Issue 3, May - June 2017 , pp. 148-156 , ISSN 2278-6856

[25] T.Subramani, R.Ganapathy,V.Manoharan, M.Balamurugan, R.Murugesan , " Design And Analysis Of Light Weight Concrete Building Using ETAB With Respect To Dynamic Loading " , International Journal of Emerging Trends & Technology in Computer Science (IJETTCS), Volume 6, Issue 3, May - June 2017 , pp. 252-258 , ISSN 2278-6856.

[26] T.Subramani, A.Mohammed Ali, R.Karthikeyan, E.Panner Selvan , K.Periyasamy , " Analytical Study Of T-Beam Using ANSYS " , International Journal of Emerging Trends & Technology in Computer Science (IJETTCS), Volume 6, Issue 3, May - June 2017 , pp. 259-266 , ISSN 2278-6856.

[27] T.Subramani, V.Kalaivanan, S.Priyaranjithkumar, P.Sasikumar, P.Vinoth Kumar , " Design And Analysis Of Multistorey Building With Respect To Seismic Loads Using ETABS " , International Journal of Emerging Trends & Technology in Computer Science (IJETTCS), Volume 6, Issue 3, May - June 2017 , pp. 267-274 , ISSN 2278-6856.