Computational Investigation on composite beam cross-sections and their influence on structural strength

T. J. Prasanna Kumar[1*], B. Teja[2], B. Chaitanya Naresh[3], B. Lokesh[4], B. Uday Tej[5]

[1*]Assistant Professor, Department of Mechanical Engineering, PVPSIT, Vijayawada, A.P, India.

[2,3,4,5] Student, UG, Department of Mechanical Engineering, PVPSIT, Vijayawada, A.P, India.

Corresponding Author E-mail: tjpk.mech@gmail.com

ABSTRACT

Conventional beams are replaced by composite beams to withstand heavy loads with minimum weight. It is required to consider the type of material, geometrical details, staking sequence and boundary conditions while analyzing a beam for a particular application. The present work addresses the effect of various cross sections under same loading conditions for a cantilevered supported beam using finite element analysis tool ANSYS. A composite beam under different constraints such as simply supported, cantilever supported and fixed conditions are explored by varying number of layers in a composite beam. The finite element models are validated by comparing the deflections of the resulting beam with published results. The linear and transverse deflections, normal and tangential stresses are evaluated by changing the no of layers and stacking sequence of a composite beam under considered boundary conditions. The present work is used for the effective design of composite beams for different applications using finite element method. Composites are frequently applicable in automobile, aerospace, nuclear, biomedical, marine, wind turbine blades because they have permissible fatigue characteristics and fiber orientation can be change to obtain design requirements. Moreover, the composite materials comprise of two or more materials which are stacked together.

Most of the special discretization techniques are based on the finite elements materials (FEM). They basically used for low order basic functions and their accuracy is improved by meshing. There are two main approaches according to laminated beam theory, firstly the approach in lateral(y-direction) where the beam displacement is neglected. cross-ply that is 0 deg and 90deg stacked layers laminated beams, also used this approach where the laminated beams are layered angle-ply (ɵ and -ɵ layers). Secondly From the existing laminated plate theory laminated beam theory is developed where, the stress and moments are ignored. This approach is mainly used for symmetric beams and in laminated beams. this paper figured out the total displacement and stresses on a cantilever supported beam with various cross-sections like I-section, C-section, Rectangular section by applying load at the free end of the beam.

Keywords: Cantilever Beam, stacking, deflection, stress, laminate

1. INTRODUCTION

1.1 Composite beam with various cross sections

Composite beams are composed of two or more materials to improve the material properties such as strength and stiffness at the same time to reduce cost and mass penalty compared to conventional metals. Most common types of composite beams are I- and C-type cross sections where the flanges are made of solid wood and webs are made of plywood material. On the other hand pipe beams which have outer liner made of aluminium.

According to beam theory, the axial load is the simplest loading in composite beams. Even though the stress is discontinuous the strain across the cross section of the beam is continuous. When the two different materials are rigidly attached in composites the normal strains are equal in both material when loaded axially. The hooke's law shows that,

$$\varepsilon_1 = \varepsilon = \sigma_1/E_1$$
$$\varepsilon_2 = \varepsilon = \sigma_2/E_2$$

Eliminating ε gives,

$$\sigma_1/E_1 = \sigma_2/E_2$$

The total load P must equal the stresses times their respective areas, or

$$P = A_1\sigma_1 + A_2\sigma_2$$

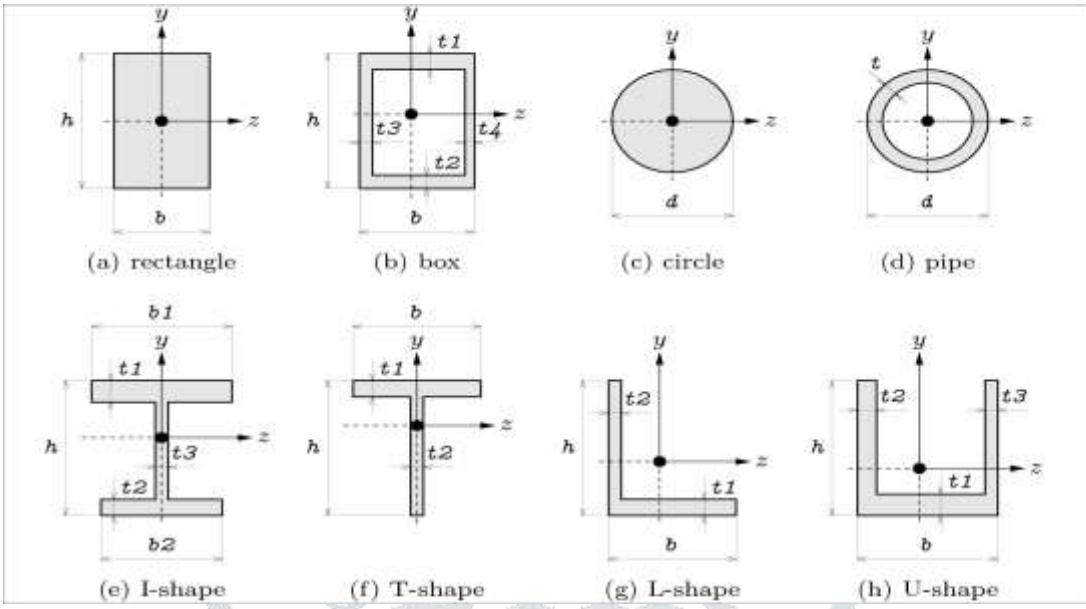


Figure-1 Various Beam Cross Sections

1.2 STRESSES INDUCED IN BEAMS

Beams subjected to Shear force and bending moment may vary from one section to another section. The beam section must develop some internal stresses or induced stresses to resist from the shear force and bending moment. Bending stresses are mostly comes with shear. However, for analysis to be smooth lets ignore the shear effects and consider only moment to find the bending stresses. The stresses due to only bending is known as pure bending.

1.3 FINITE ELEMENT MODELING ANALYSIS

FE analysis is a technique where computerized evaluations are carried out for prediction of how a vibration, real-world forces, fluid and heat flow model will reacts and other physical effects. This analysis will be figured out a breakage of component, wear out based on the way it is designed. Even in product development process, it can predict life of the components and how it really facing the forces externally when it is in operation.

In FEA the real model/ geometrical model divided into a large number of breaking down a real object into a large number (sometimes lakhs of divisions based on the complexity in problem) of finite elements, such as tiny cubes. Individual elements behavior will be predicted by Mathematical equations. The actual object behavior is then predicted by summing all individual behaviors of elements.

However, many physical effects of individual components can be predicted by Finite Element analysis such as:

1. Stresses induced in components
2. Vibrational studies
3. Components effected by Fatigue
4. Linear or non-linear Motion
5. Effects of Heat transfer
6. Fluid flow in a channel/pipe/control volume

1.4 PROCEDURE STEPS FOR FINITE ELEMENT ANALYSIS

In Finite Element Analysis, certain steps in formulating physical problem are similar to all such analysis, whether the problem is related to heat flow, structural, electromagnetic, etc.

The steps are described as follows.

Step-1: Preprocessing phase in FEA in ANSYS

The preprocessing step is, quite generally, described as defining the model and includes Define the geometric problem domain, element type to be used, material properties of the specific elements, geometric properties of the selected elements (volume, length, area), element connectivity's (mesh the model), physical constraints (applying boundary conditions) and the loadings. The preprocessing (model is defined) step is just a time processing. If the problem definition is wrong, the FE solutions will absolutely give no value.

Step-2: Solution phase in FEA in ANSYS

In this phase, finite element Analysis software (ANSYS) combines and the values of unknown field variables will be computed using governing equations of algebraic in matrix form. The element stresses, reaction forces and heat flow are now calculated by substituting the computed values. Some special solution techniques are available to reduce the computation time as well as data storage space as it is familiar that FEA needs thousands of equations to introduce to define a model. For static, linear conditions, a solver with a wave front depended on Gauss elimination is applied.

Step-3: Postprocessing phase in FEA in ANSYS

Evaluation of the solutions from the results obtained from analysis results is known as postprocessing. Postprocessor in Finite Element Analysis is featured with advanced routines used for printing, plotting and sorting obtained results. Some of the advanced features include Plot deformed structural shape , Check equilibrium, Animate dynamic model behavior, Produce color-coded temperature plots Calculate factors of safety Sort element stresses in order of magnitude. In postprocessing solution data may be modified different ways

DESIGN SPECIFICATIONS OF BEAMS WITH 3 DIFFERENT CROSS SECTIONS

Geometrical models of beams are created in ANSYS APDL with various cross-sections like rectangular section, I-section and C-section to determine the displacements and stresses on a cantilever supported beam subjected to a load at free end. The loading conditions are similar for all the 3 cases.

2.1 Specifications of Beam with Rectangular Cross Section:

For Rectangular section:

Width, W = 20 mm,
Length, L = 200 mm, and
Height, H = 10mm.

Figure-2: Rectangular cross section

2.2 Specifications of Beam with I-Section:

S. No	Section	Width	Depth
1.	Flange 1	20	3.75
2.	Web	3.75	13.33
3.	Flange 2	20	3.75

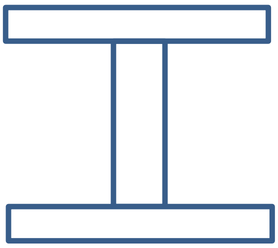


Figure-3: I- section

2.3 Specifications of Beam with C- Section:

S. No	Section	Width	Depth
1.	Flange 1	20	3.75
2.	Web	3.75	13.33
3.	Flange 2	20	3.75

Figure-4: C- section

2.4 LAMINATE STACKING SEQUENCE:

When two or more layers of laminae using similar or different orientation of fibers and stacked to each other is termed as Laminate. The laminae composed of either same or different material with same or variable thickness.

2.5 Fiber Orientation in Stacking Notation with variable thickness:

A special nomenclature is used to designate the laminate. This nomenclature provides the information of orientation of fibers and stacked layers in the laminate. The following table shows the laminate and their designations as per the number of layers. In this analysis the number of layers used are 10 and each layer having a thickness of 1mm.

Table-1: Fiber orientation related to Layer thickness

Layer no.	Thickness	Orientation
1	1.25	0
2	1.25	90
3	1.25	90
4	3.3325	0
5	3.3325	45
6	3.3325	-45
7	3.3325	0
8	1.25	90
9	1.25	90
10	1.25	0

2. STATIC STRUCTURAL ANALYSIS USING FEA:

STEP- 1: The rectangular block is generated in ANSYS with its global co-ordinates.

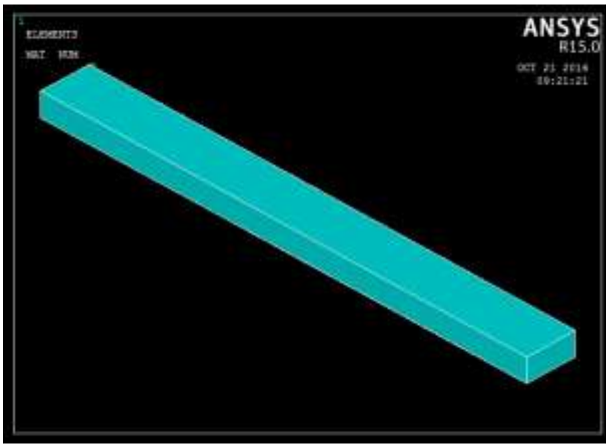


Figure-5: Discretized model one end constrained for a Rectangular Finite Element Model

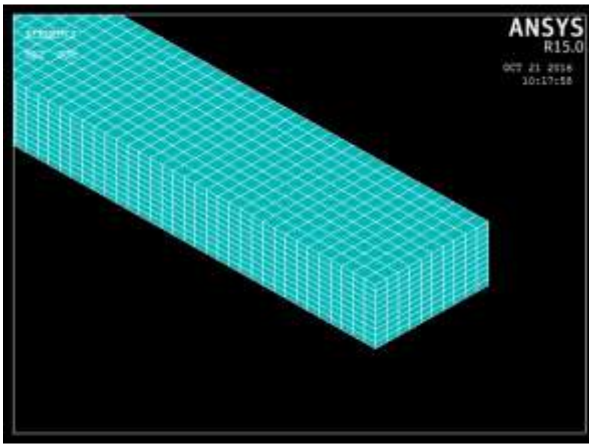


Figure-6: Finite Element Model

The stacking sequence is done according to the notation and the layers are divided into 10. The rectangular block is subjected to point load at free end and constrained at other the end.

STEP-2: The I-Section and C-section is generated in ANSYS with its co-coordinates. The stacking sequence is done according to the notation and the layers are divided into 10. The top and bottom flanges are divided by 3 layers and web are divided by 4 layers.

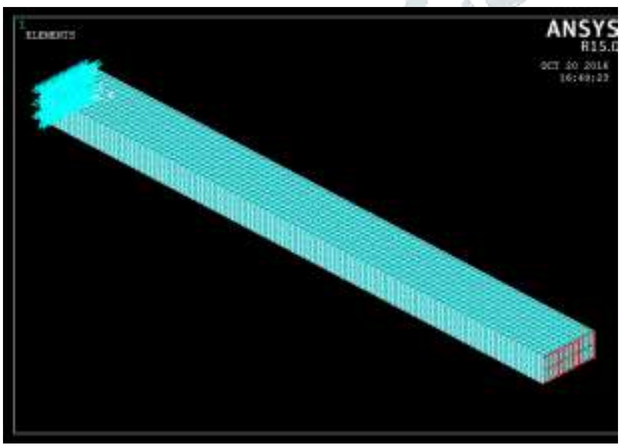


Figure-7: Boundary Conditions and Loads on Rectangular section FEM model

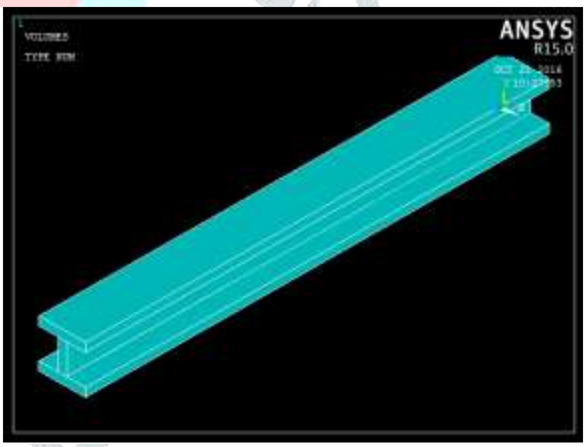


Figure-8: FEM model of I-Section beam

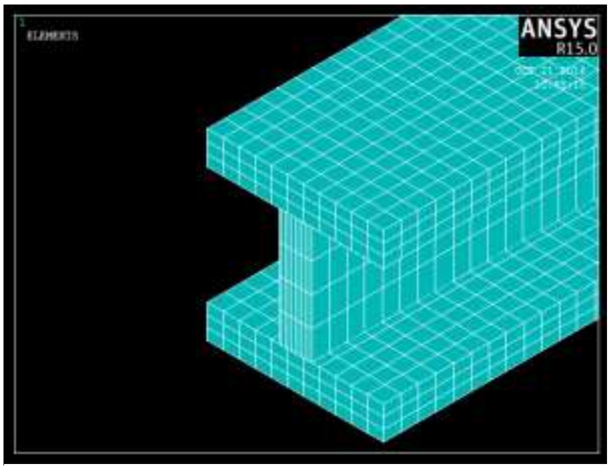


Figure-9: Fem Model of I-section beam

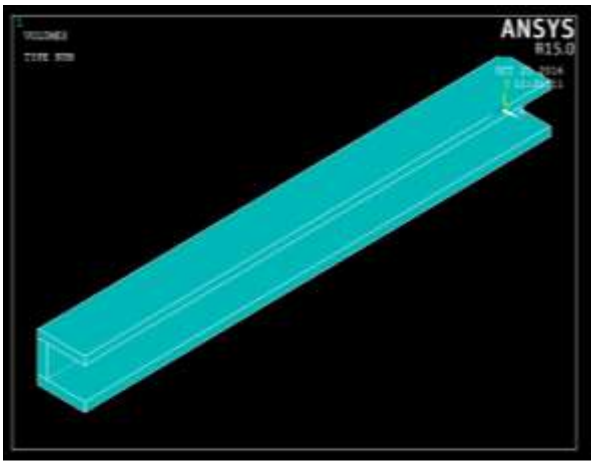


Figure-8: FEM model of I-Section beam

point load at free end

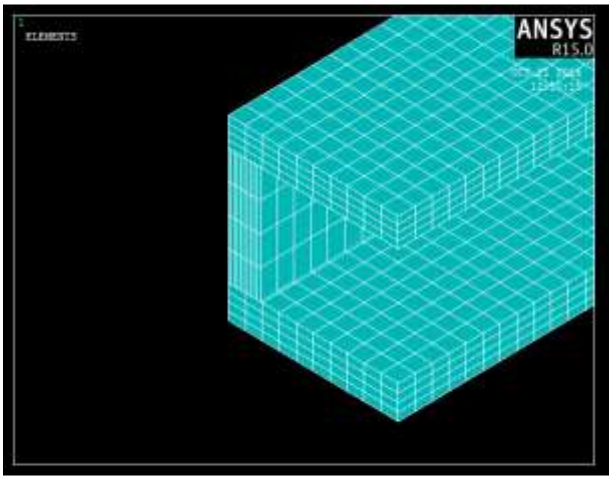


Figure-9: Fem Model of I-section beam point load at free end

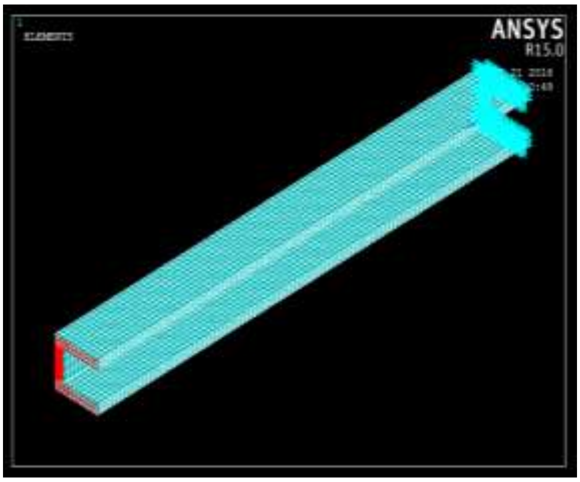


Figure-7: Boundary Conditions and Loads on C-Section FEM model

4. RESULTS AND DISCUSSIONS

4.1 DISPLACEMENT RESUTS:

Contour Plots showing Total Displacement of Cantilever Beam with Rectangular Section:

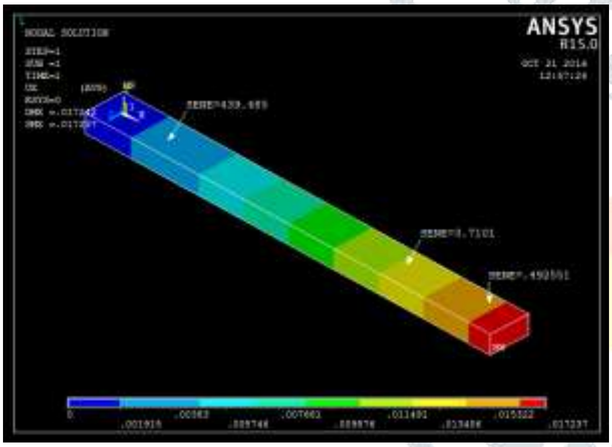


Figure-8: Displacement in X-Direction, Ux

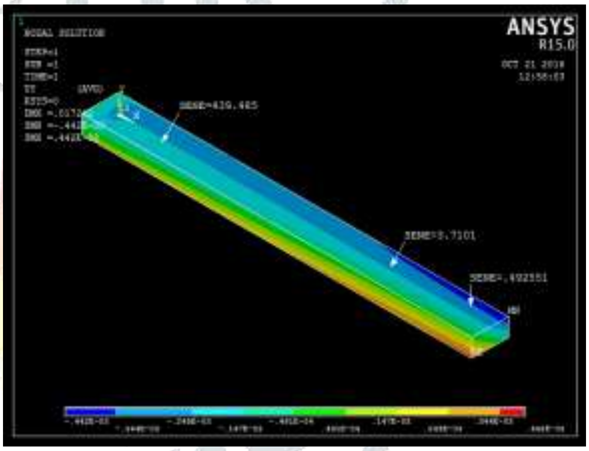


Figure-9: Displacement in Y-Direction, Uy

Contour Plots showing Total Displacement of Cantilever Beam with I-Section:

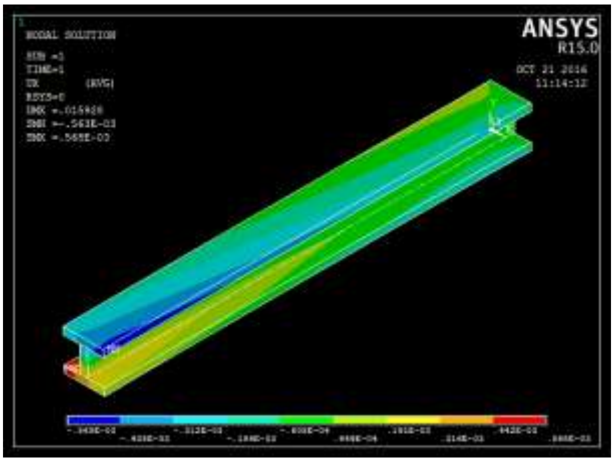


Figure-10: Displacement in X-Direction, Ux

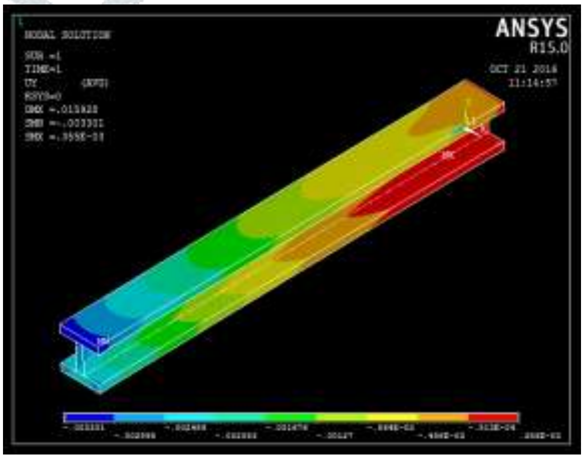


Figure-11: Displacement in Y-Direction, Uy

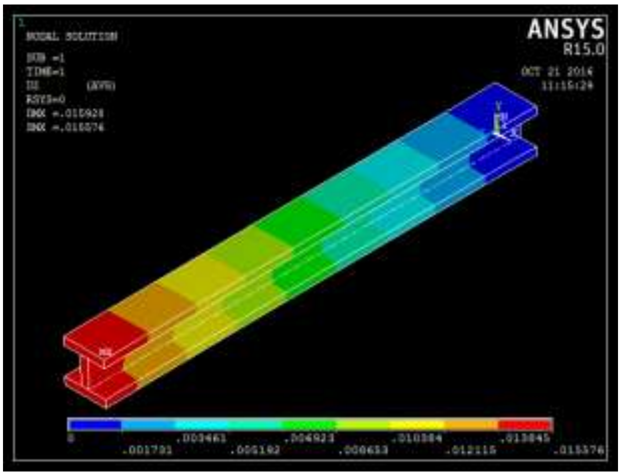


Figure-12: Displacement in Z-Direction, Uz

Contour Plots showing Total Displacement of C-Section beam in X, Y and Z directions:

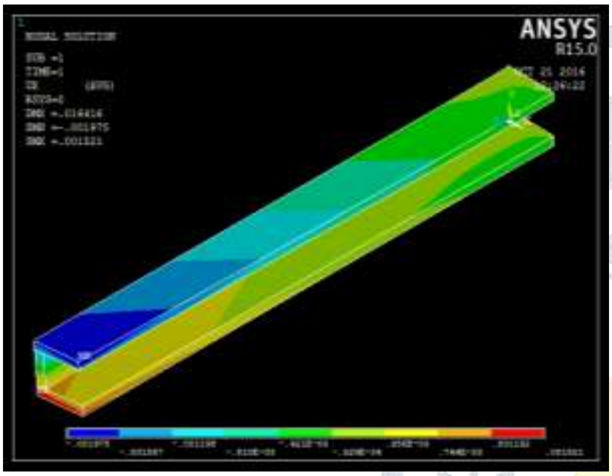


Figure-13: Displacement in X-Direction, Ux

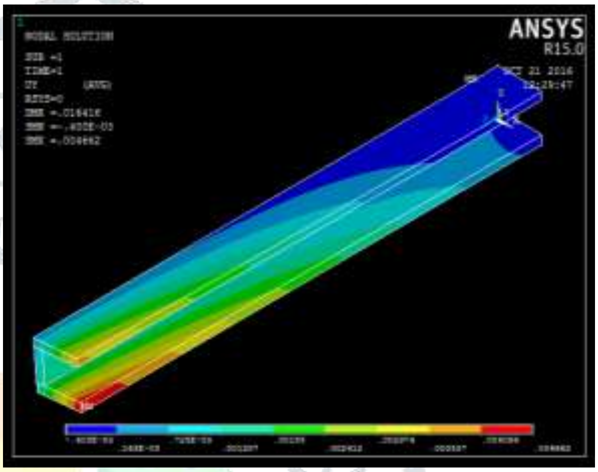


Figure-14: Displacement in Y-Direction, Uy

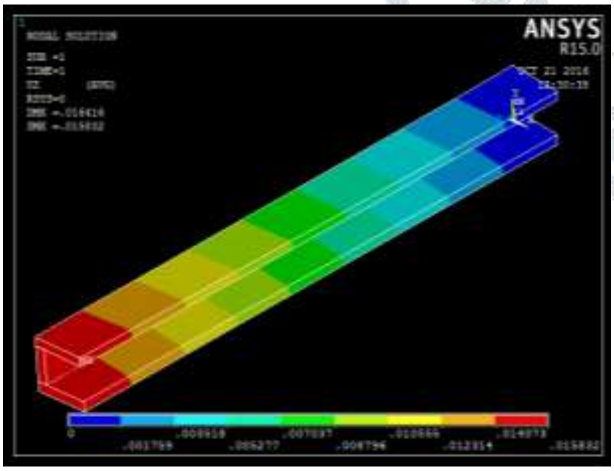


Figure-15: Displacement in Z-Direction, Uz

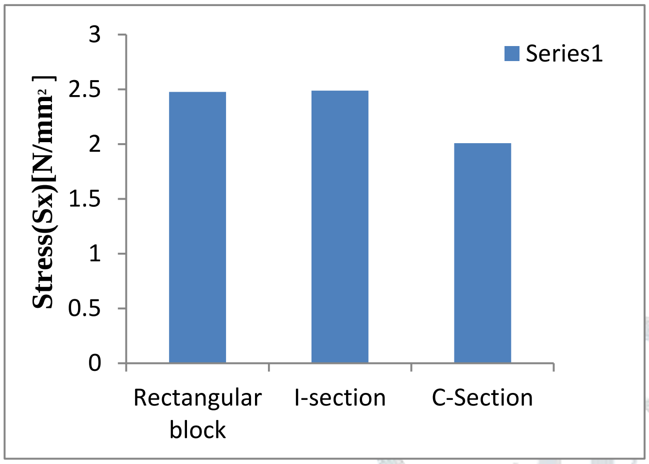
TABLE-2: Comapritive Stress, Shear stress and displacement values in X, Y and Z direction obtained from Post-processing Results

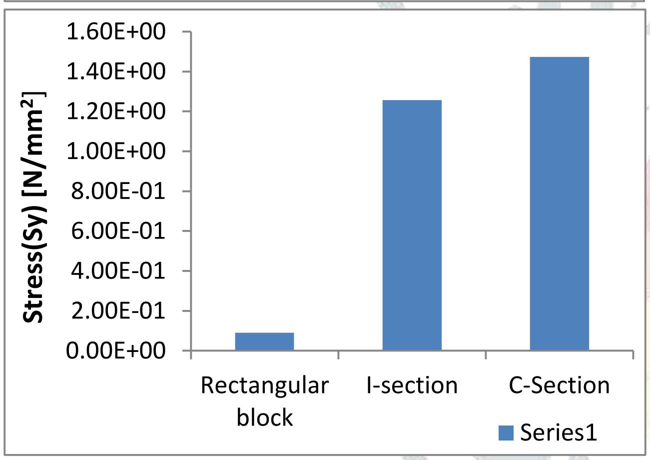
	ux	uy	uz	sx	sy	sz	sxy	syz	szx
Rectangular block	2.84E-03	6.70E-04	3.25E-04	2.4775	8.99E-02	0.10852	0.15784	3.27E-02	6.03E-02
I-section	-4.99E-04	-2.81E-03	3.03E-03	2.4893	1.2569	6.9177	0.3552	1.2334	0.45184
C-Section	1.24E-02	-8.07E-03	4.15E-03	2.0083	1.4736	2.7229	2.71E-01	0.51787	0.59344

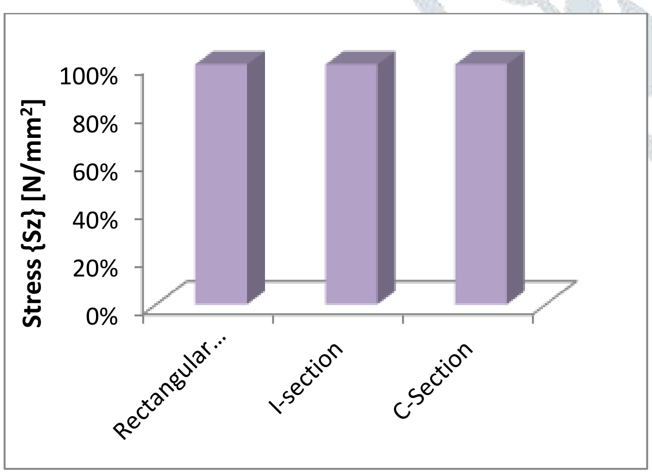
5. COMPARATIVE GRAPHS

5.1 STRESS COMPONENT IN X-DIRECTION:

The stress values plotted for rectangular-section, I-section and C-section in X, Y and Z-direction respectively.

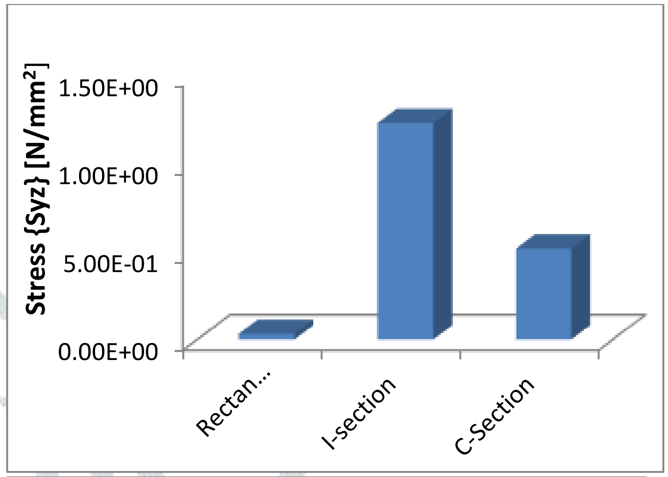


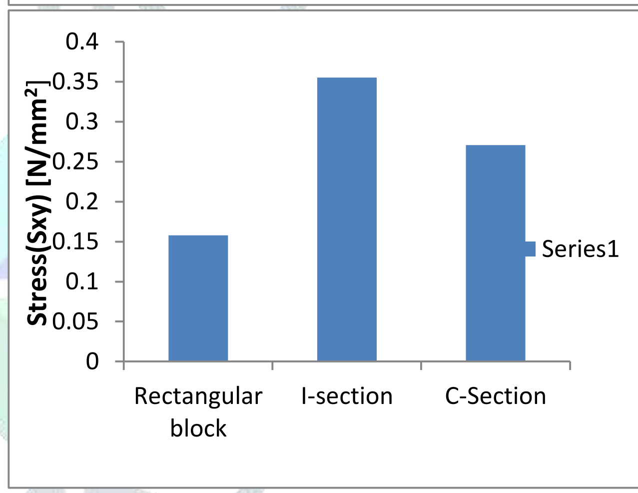


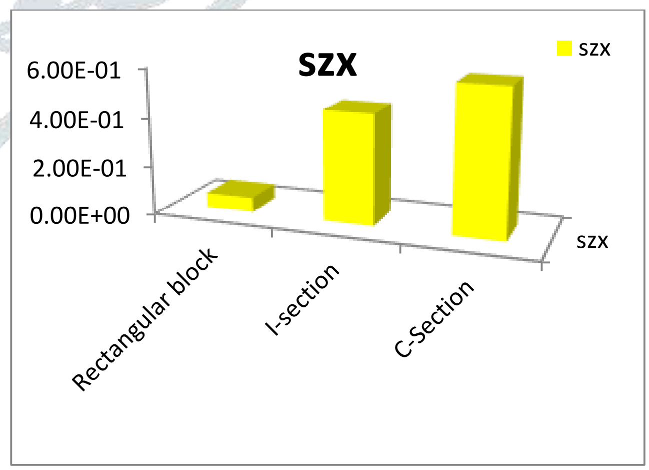


5.2 SHEAR STRESS COMPONENT IN XY, YZ AND XZ-DIRECTION:

The shear stress is plotted for rectangular section, I-section and C-section in XY, YZ AND XZ directions

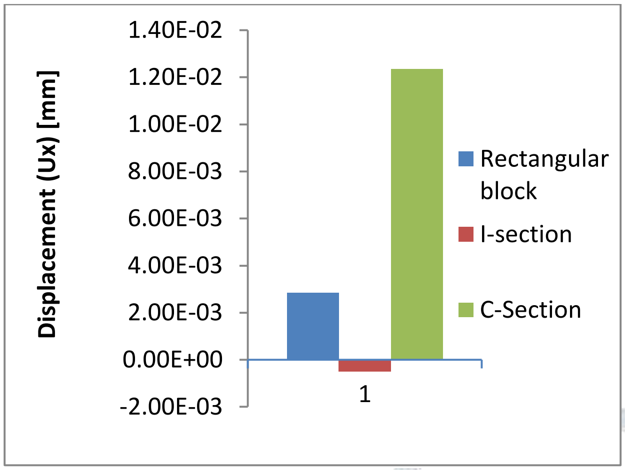


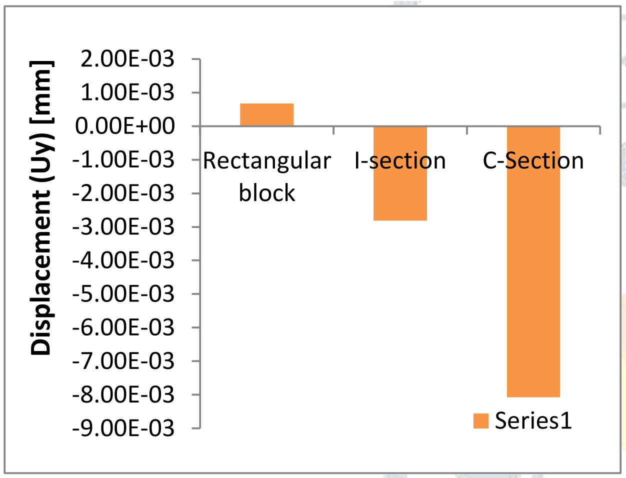


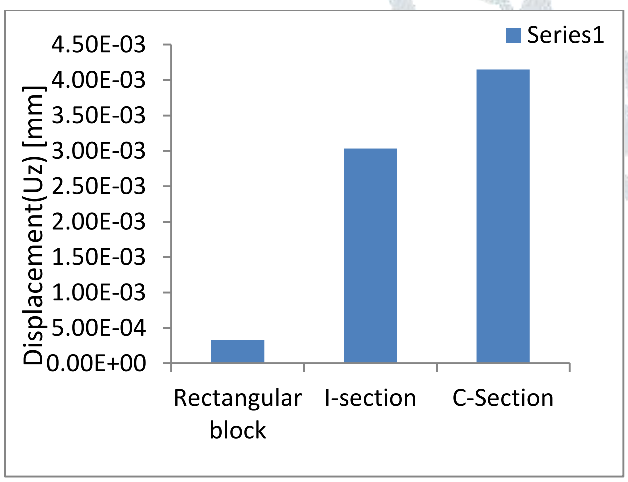


5.3 DISPLACEMENT IN X-DIRECTION:

The displacements plotted for rectangular section, I-section and C-section in X, Y and Z-direction







6. CONCLUSIONS & FUTURE SCOPE

The composite beam with rectangular section, I-section and C- section are analyzed by varying stacking sequence & fiber orientation.

1. From the contour plots and graphs it is observed that the stress component is more for beam with rectangular and I-section and C-section offers more strength in X-direction, whereas rectangular section offers more strength in Y-direction. This is uniform in Z direction.
2. It is also observed that the shear stress component is more for beam with C-section than I and Rectangular section, while rectangular section offers more shear strength
3. it is clearly concluded that the displacement for the composite beam under free end loading of a cantilever arrangement, I-section and C-section beams offers more resistance to displacement in X and Y direction whereas the beam displaced more in Z-direction and Rectangle section offers more resistance to displace in Z-direction.
4. This work can further extend for composite beam with various cross-section such as L-section, T-section and circular section beams & also for un-symmetrical beam.

7. REFERENCES

1. Amer M. Ibrahim, Saad k. Mohaisen, Qusay W. Ahmed, 2012, "Finite element modeling of composite steel-concrete beams with external prestressing", International Journal Of Civil and Structural Engineering, Vol 3, No. 1, pp 101- 116
2. Andrea Dall' Asta, Alessandro Zona, 2002, "Non linear analysis of composite beams by a displacement approach", Computers and structures, Science direct, No. 80, pp 2217-2228.
3. Andrea Dall' Asta, Alessandro Zona, 2005, "Finite element model for externally prestressed composite beams with deformable connection", Journal of Structural Engineering, ASCE, No. 131, pp 706-714.
4. Ashraf Ayoub, Filip C Filippou, 2000, "Mixed Formulation of Nonlinear steel- composite beam element", Journal of Structural Engineering, ASCE, No. 126, pp 371- 381
5. Ciro Faella, Enzo Martinelli, Emidio Nigro, 2003, "Shear connection nonlinearity and deflections of steel- concrete composite beams: a simplified method, Journal of Structural Engineering, ASCE, No. 129, pp 12- 20
6. Deric John Oehlers, George Sved, 1995, "Composite beams with limited slip- capacity shear connectors", Journal of Structural Engineering, ASCE, No. 121, pp 932-938.
7. Dr. D. Vijayalakshmi, Dr. D. Tensing, 2014, "An experimental study on steel concrete composite beams with shear connectors under pure torsion", International Journal of Emerging Technology and

Advanced Engineering, ISO 9001: 2008 certified, Vol 4, Special issue 3, pp 202-209

8. Hamid Saadatmanesh, Pedro Albrecht, Bilal M. Ayyub, 1989, “Analytical Study of prestressed composite beams”, Journal of Structural Engineering, ASCE, No. 115, pp 2364-2381.
9. Jin- Wook Hwang, Ji- Hyun Kwak, Hyo- Gyoung Kwak, 2014, “Finite element model to evaluate nonlinear behaviour of posttensioned composite beams with partial shear connection”, Journal of Structural Engineering, ASCE, pp 1-15
10. Uttam Kumar Chakravarty, “On the modeling of composite beam cross-sections”, Composites: Part B 42 (2011) 982–991
11. I nci Mknz. P. Tum. K. lzgt, Mamak, Ankara, “An Elastic-Plastic Stress Analysis in Silicon Carbide Fiber Reinforced Magnesium Metal Matrix Composite Beam Having Rectangular Cross Section Under Transverse Loading”, KSME International Journal, Vol. 18 No.2, pp. 221- 229, 2004.
12. P.K. Mallick, “Fiber-Reinforced composite Materials”, Third Edition, USA
13. W. S. Chan and K.A. Syed, “Determination of Centroid and Shear Center Locations of Composite Box Beams” the University of Texas at Arlington, Texas, USA.
14. Y.X. Zhang, C.H. Yang, “Recent developments in finite element analysis for laminated composite plates”, Composite Structures 88 (2009).
15. Jun Deng, Marcus, M.K. Lee, Stuart S.J. Moy, “Stress analysis of steel beams reinforced with a bonded CFRP plate” Composite Structures 65 (2004) 205–215.