

Analysis of Bridge vertical column and span with stiffeners Using Finite Element analysis

Prof. Subhash N. Khetre^{1#}, Prof. Nitin Jadhav², Prof. Santosh Pachpute³

¹ Assistant professor, Department of Mechanical Engineering, NMIET, Talegaon, Pune, SPPU, Pune, (India)

² Assistant professor, Department of Mechanical Engineering, JSPMS, RSCOE, Tathawade, Pune, Pune, (India)

³ Assistant professor, Department of Mechanical Engineering, D Y PATIL COE, Akurdi, SPPU, Pune, (India)

ABSTRACT

The Finite element analyses are conducted to model the tensile capacity of steel fiber-reinforced concrete (SFRC). For this purpose bridge with two ends fixed one specimen are casted and tested under direct and uni-axial tension. Two types of aggregates (brick and stone) are used to cast the SFRC and plain concrete. The fiber volume ratio is maintained 1.5 %. Total 8 numbers of dog-bone specimens are made and tested in a 1000-kN capacity digital universal testing machine (UTM). The strain data are gathered employing digital image correlation technique from high-definition images and high-speed video clips. Then, the strain data are synthesized with the load data obtained from the load cell of the UTM. The tensile capacity enhancement is found 182–253 % compared to control specimen to brick SFRC and in case of stone SFRC the enhancement is 157–268 %. Fibers are found to enhance the tensile capacity as well as ductile properties of concrete that ensures to prevent sudden brittle failure. The dog-bone specimens are modeled in the ANSYS 15.0 finite element platform and analyzed to model the tensile capacity of brick and stone SFRC. The SOLID65 element is used to model the SFRC as well as plain concretes by optimizing the Poisson's ratio, modulus of elasticity, tensile strength and stress–strain relationships and also failure pattern as well as failure locations. This research provides information of the tensile capacity enhancement of SFRC made of both brick and stone which will be helpful for the construction industry of Bangladesh to introduce this engineering material in earthquake design. This paper focuses on the numerical analysis and experimental analysis of transverse vibration of fixed free beam and investigates the mode shape frequency. All the frequency values are analyzed with the numerical approach method by using ANSYS finite element package has been used.

Keywords - Tensile capacity of concrete, Steel fiber-reinforced concrete (SFRC), Modal analysis.

1.INTRODUCTION

Concrete shows a low tensile strength in comparison to the compressive strength, which grows only less proportionally with increasing compressive strength; at the same time, the brittleness increases. Therefore, fibers are added to improve ductility and to increase the tensile strength. For many applications, however, conventional reinforcement in the tensile zone is still necessary. Fibers are increasingly being used in concrete structures to compensate for concrete's weak and brittle tensile behavior relative to its compression response. One of the most beneficial aspects of the use of fibers in concrete structures is that non-brittle behavior after

concrete cracking can be achieved with fibers. The tensile stress sustainable in concrete rapidly decreases immediately after cracking. In fiber-reinforced concrete (FRC), on the other hand, fibers crossing the crack interfaces significantly contribute to the load-carrying mechanism so that considerable tensile stress, being the Sum of the tensile resistance provided by fibers and tension softening of the concrete matrix, respectively, can be achieved even with large crack widths. Therefore, the enhanced tensile stress behavior attainable with fibers should be realistically evaluated to accurately predict the post-cracking response of FRC. The bond Resistance of reinforcing bars embedded in concrete depends primarily on frictional resistance and mechanical interlock. The chemical adhesion bond, if any, fails at very small slips. Frictional bond provides initial resistance against loading and further loading mobilizes the mechanical interlock between the concrete and bar ribs. Mechanical interlock leads to inclined bearing forces, which in turn leads to transverse tensile stresses and internal inclined splitting (bond) cracks along reinforcing bars. To this end, this research modeled direct tensile strength of plain concrete and steel fiber reinforced concrete (SFRC) in Finite Element platform and are evaluated based on experimental investigation. The ANSYS 10.0 Finite Element Analysis (FEA) software package is used to analyze the direct tension specimens and introduces a good concrete model for Steel Fiber-Reinforced Concrete (SFRC) as well as plain concrete made of brick and stone aggregate. Two different Poison's ratios for brick and stone concrete are selected by comparing FE output with the stress-strain behavior in tension. A reasonable modeling of concrete using suitable element type, adequate mesh size, appropriate boundary conditions, realistic loading environment and proper time stepping can help to estimate the governing parameters of concrete. Using these governing parameters (i.e. Poison's ratio, tensile strength, and the stress-strain relationship), the dogbane tensile specimens are modeled, analyzed and compared the results gathered from experimental outcomes. After evaluation of this parameter by extensive analysis, Finite Element (FE) models showed a good correlation with the experimental results and also showed similar failure patterns. This investigation is intended to validate the FE models with the experimental results by identifying and using the pertinent parameters of the concrete model as well as to provide a successful FE SFRC model for analyzing future problems on SFRC and in context of Bangladesh.

II.MAIN OBJECTIVES

All the Vehicles, aircraft and home appliances structures are made up of fixed beam with one end free or combination of fixed beams so it becomes necessary to study fixed beam vibration. The following are main objective of yoke design.

1. To Analysis of Bridge with two ends fixed
2. To Analysis of Bridge with two ends fixed using FEA Method.
3. A detailed understanding of function and configuration of Bridge with two ends fixed.

III. NUMERICAL ANALYSIS BY USING ANSYS

3.1 Basic steps of finite element analysis:

There are three basic steps involved in this procedure,

1. Pre Processor (Building the model (or) Modeling)
2. Solution (Applying loads and solving)
3. Post Processor (Reviewing the results)

3.2. Numerical Approach for Transverse Vibration of Fixed Free Beam:

We shall now investigate the free vibration of fixed free beam using the ANSYS program, a comprehensive finite element package. We use the ANSYS structural package to analyse the vibration of fixed free beam. Finite element procedures are presently very widely used in engineering analysis. The procedures are employed extensively in the analysis of solid and structures and of heat transfer and fluids and indeed, finite element methods are useful in virtually every field of engineering analysis.

3.3. Description of the finite element method:

The physical problem typically involves an actual structure or structural component subject to certain loads. The idealization of the physical problem to a mathematical model requires certain assumptions that together lead to differential equations governing the mathematical model. Since the finite element solution technique is a numerical procedure, it is necessary to assess the solution accuracy. If the accuracy criteria are not met, the numerical solution has to be repeated with refined solution parameters until a sufficient accuracy is reached.

3.4. Important features of finite element method

The following are the basic features of the finite element method: Division of whole into parts, which allows representation of geometrically complex domains as collection of simple domains that, enables a systematic derivation of the approximation functions. Derivation of approximation functions over each element the approximation functions are algebraic polynomials that are derived using interpolation theory.

3.5. Boundary Conditions:

The meshed model is then analyzed (static) and the boundary conditions are:

- One end is fixed (All DOF).

3.6 FEA RESULTS:

The dimensions and the material constant for a uniform bridge studied in this paper are: Material of beam = mild steel, Young's Modulus (E) = 210×10^9 , mass density = 7856 kg/m^3 . Poisson Ratio = 0.3. The numerical results were found out by using the ANSYS program as shown in below.

3.6.1. Result of bridge vertical column with different loading:

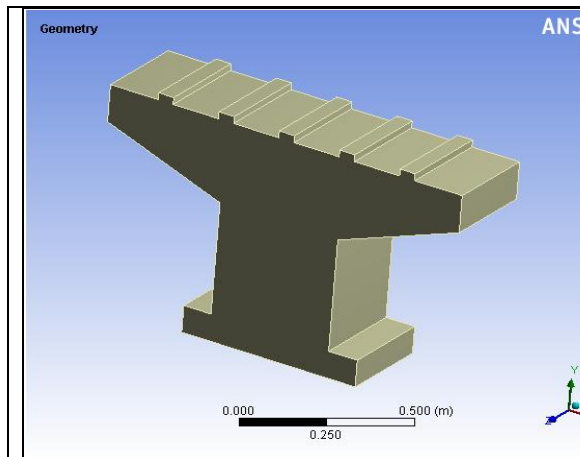


Fig. 1 : (A) Geometry of Bridge vertical column

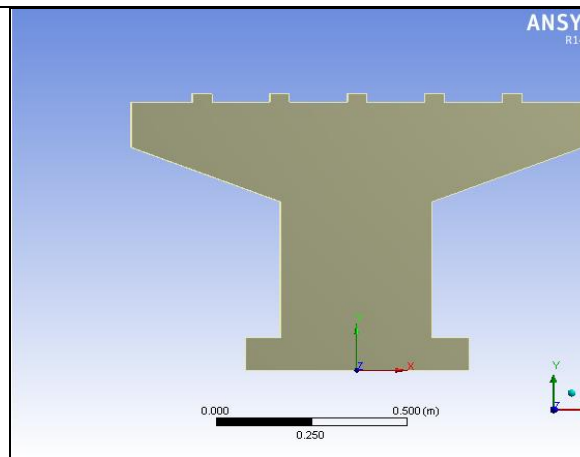


Fig. 2 : (B) Geometry of Bridge vertical column

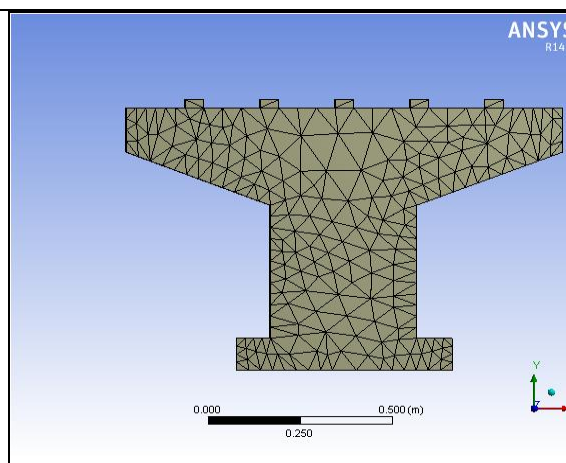


Fig. 3 : Meshing of Bridge vertical column

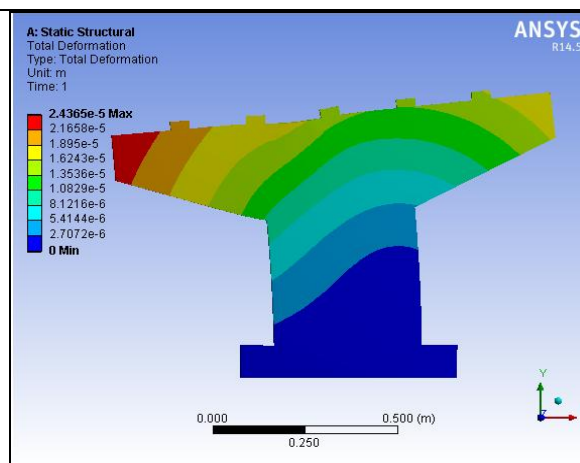


Fig. 4 : Total Deformation (right) vertical column

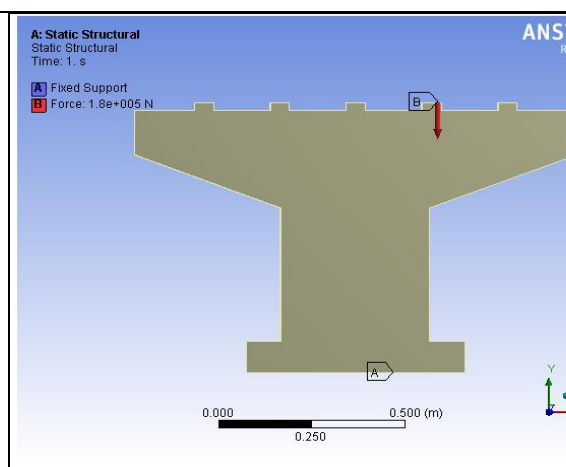


Fig. 5 : Boundary conditions of Bridge (Left) vertical column

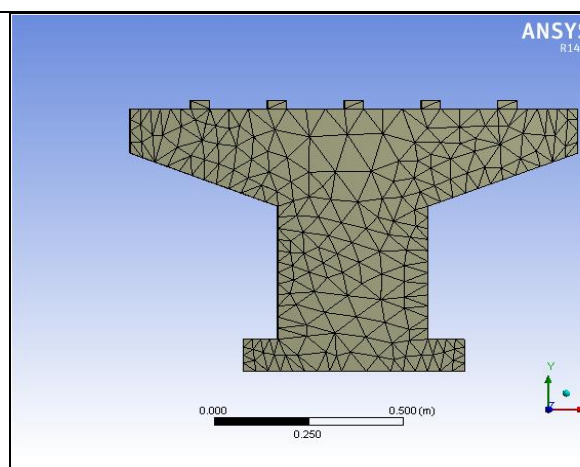


Fig. 6 : (A) Meshing of Bridge With Refinement vertical column

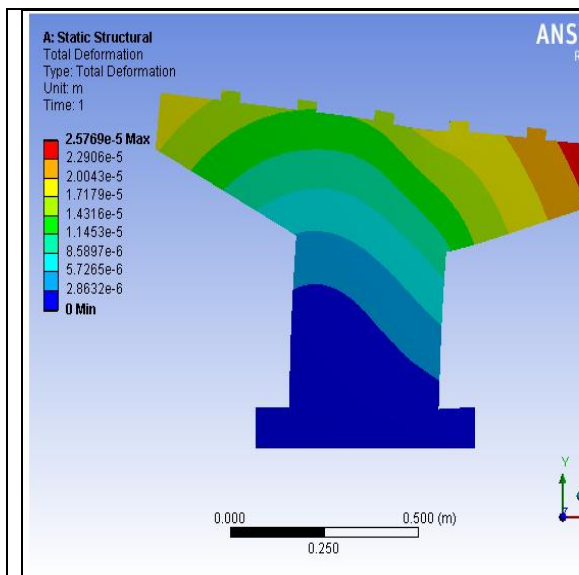


Fig. 7 :Total Deformation (Left) vertical column

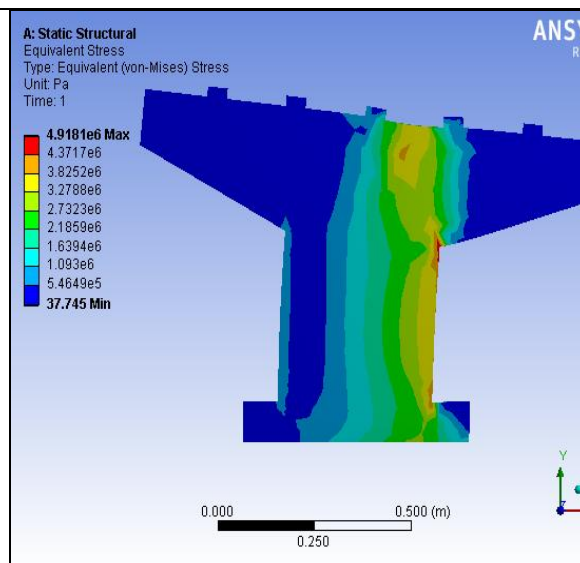


Fig. 8 :Equivalent Stress (Left) vertical column

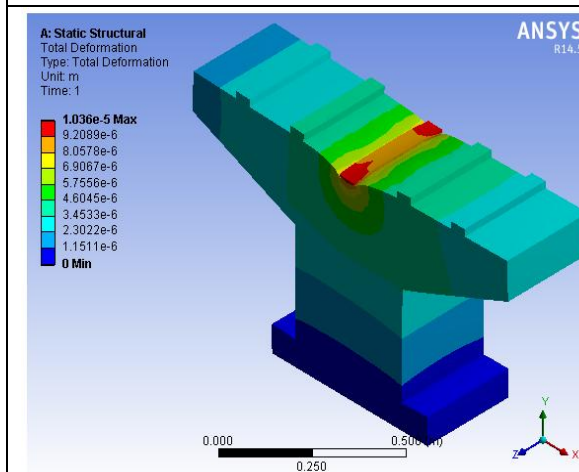


Fig. 9 : (A)Total Deformation (Centrally) vertical column

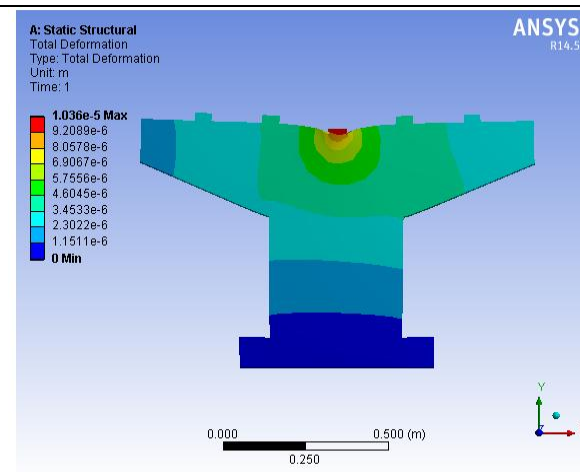


Fig. 10 : (B)Total Deformation (Centrally) vertical column

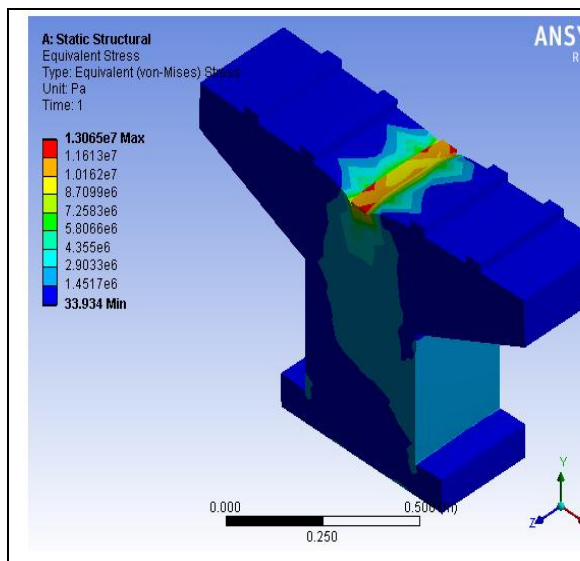


Fig.11:(A) Equivalent Stress (Centrally) vertical column

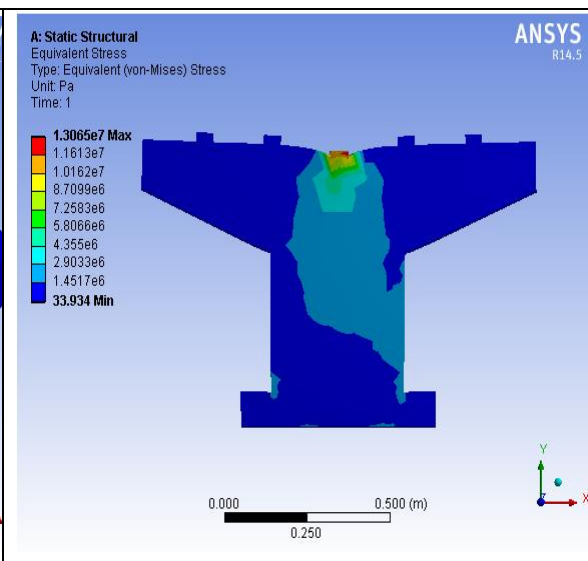


Fig.12 :(B) Equivalent Stress (Centrally) vertical column

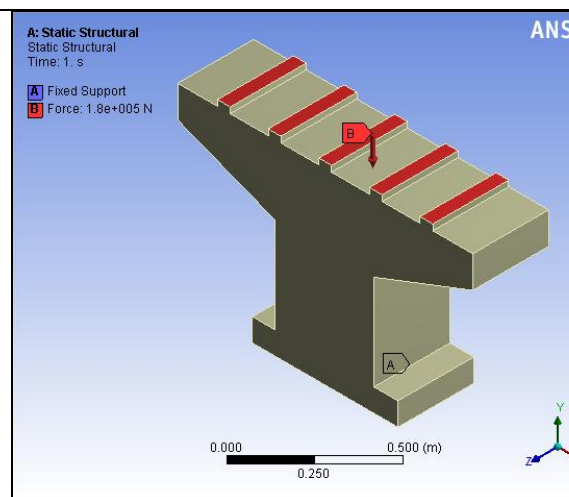


Fig.13: (A) Boundary conditions of Bridge (Centrally) vertical column

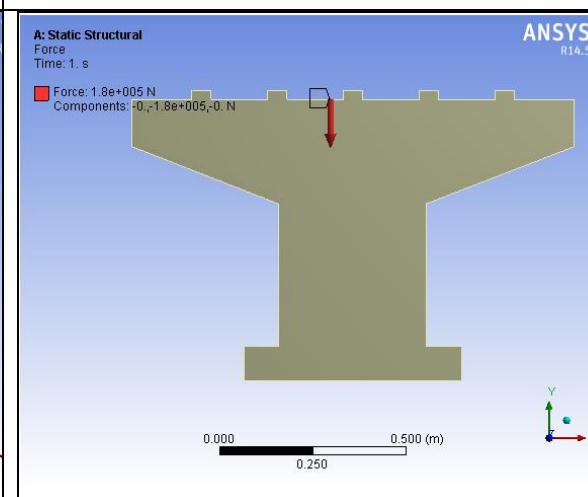


Fig. 14: (B) Boundary conditions of Bridge (Centrally) vertical column

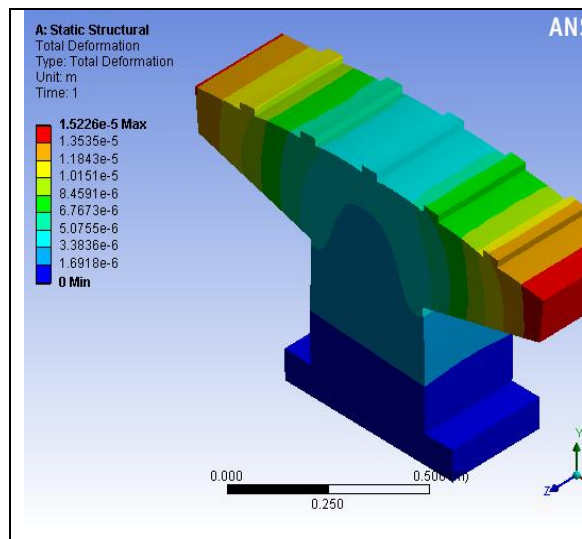


Fig.15:(A)Total Deformation (At Equally distributed load) vertical column

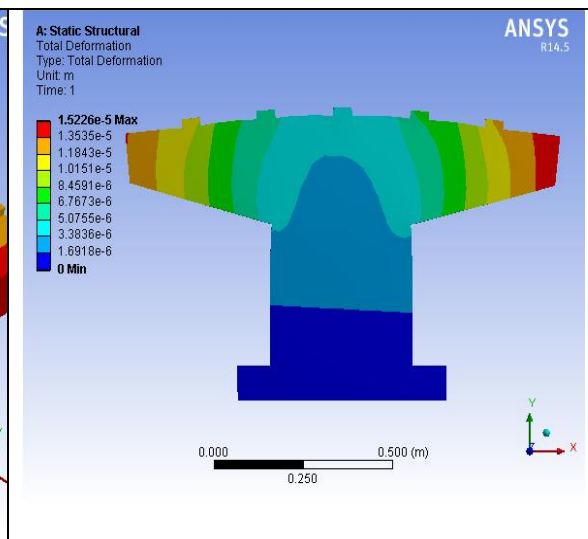


Fig.16:(B)Total Deformation (At Equally distributed load) vertical column

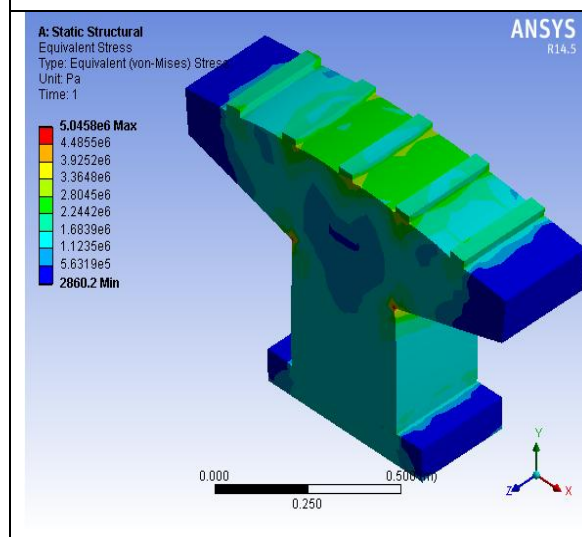


Fig. 17:(A) Equivalent Stress (At Equally distributed load) vertical column

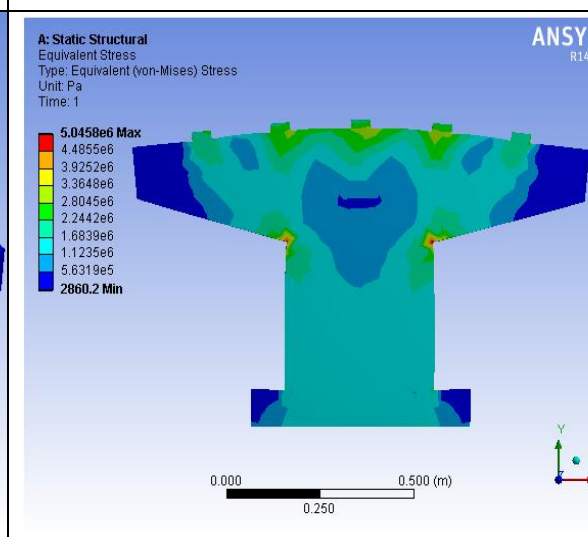


Fig. 18:(B) Equivalent Stress (At Equally distributed load) vertical column

3.6.2. Result of bridge span with two stiffeners:

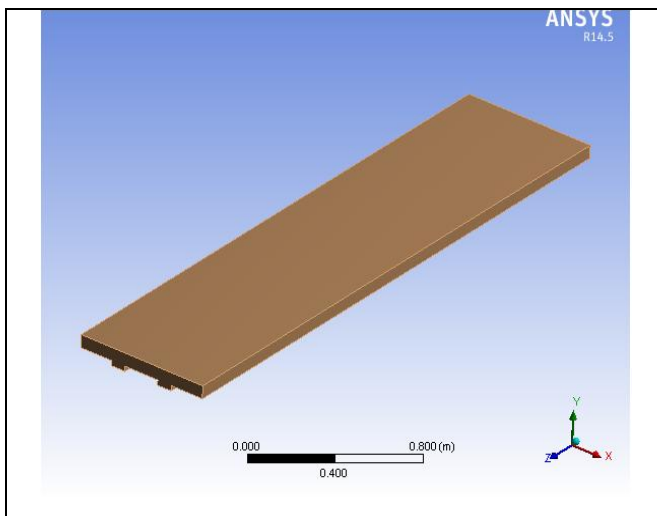


Fig. 19 : (A) Geometry of Bridge span with two stiffeners

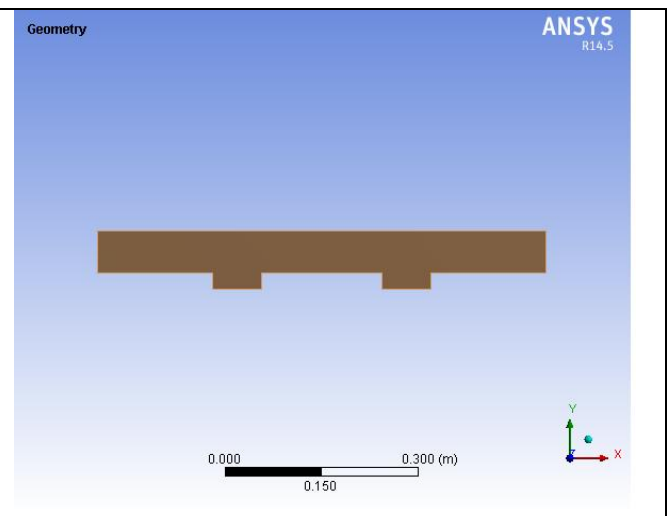


Fig. 20 : (B) Geometry of Bridge span with two stiffeners

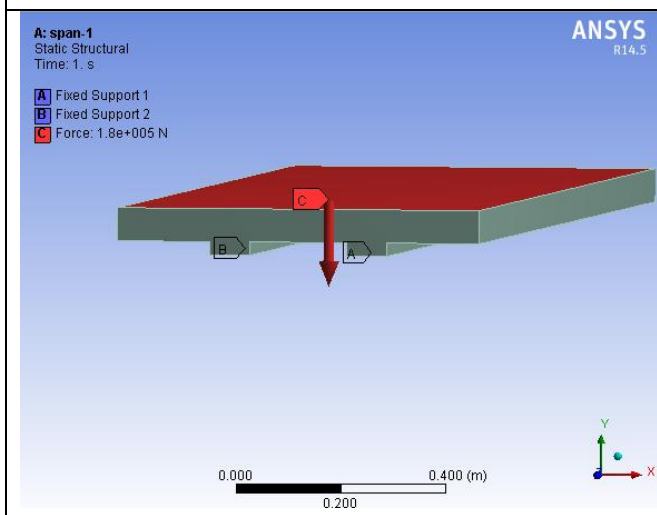


Fig. 21: (A) Boundary conditions of Bridge span with two stiffeners

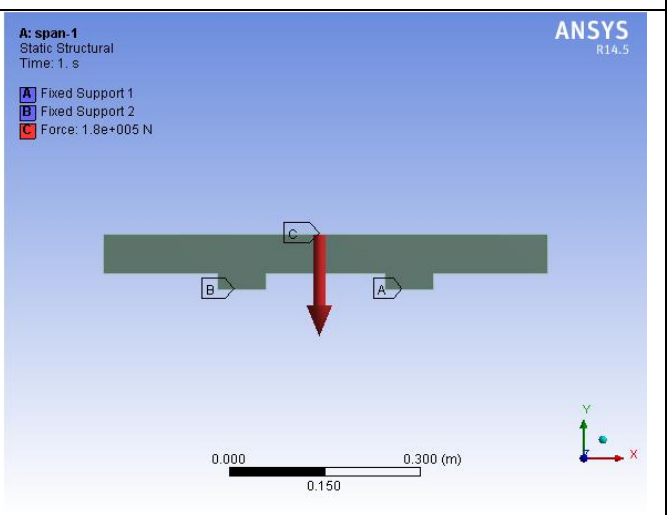


Fig. 22: (B) Boundary conditions of Bridge span with two stiffeners

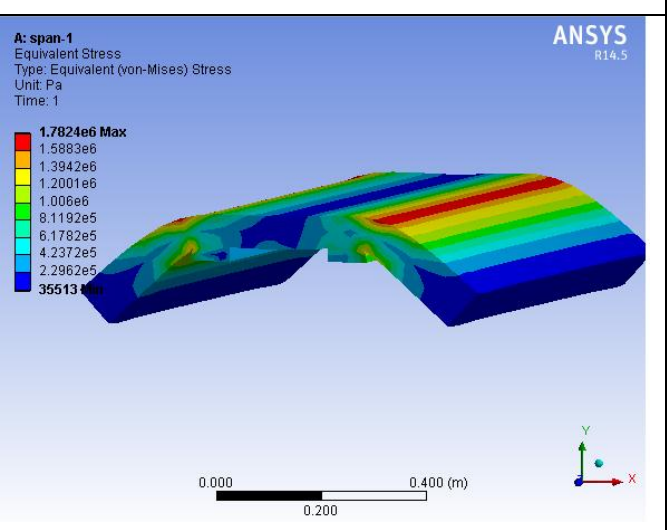
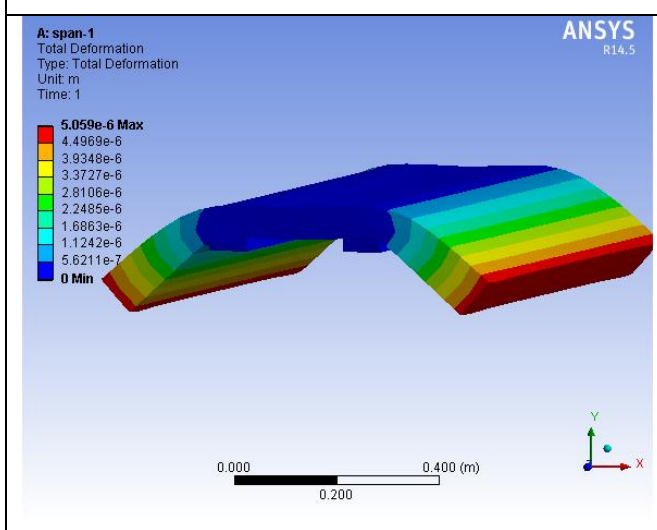


Fig.23:(A)Total Deformation of Bridge span
with two stiffeners

Fig.24: Equivalent Stress of Bridge span
with two stiffeners

3.6.3. Result of bridge span with Tree stiffener:

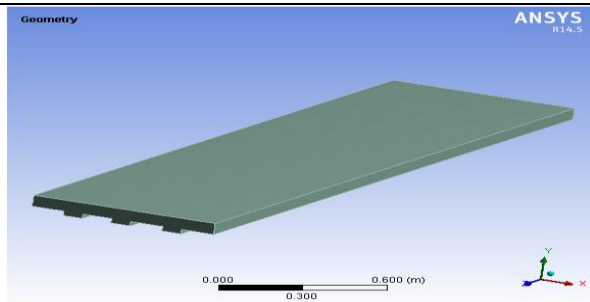
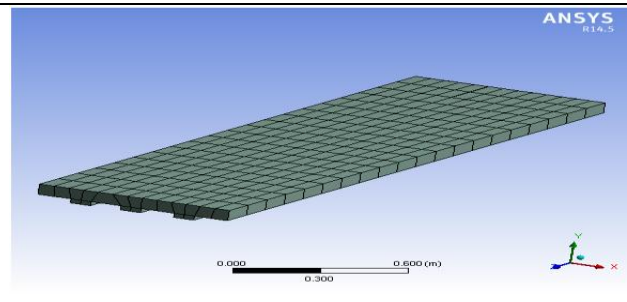


Fig. 25 : (A) Geometry of Bridge span
with three stiffeners



**Fig. 26 : Meshing of Bridge span
with three stiffeners**

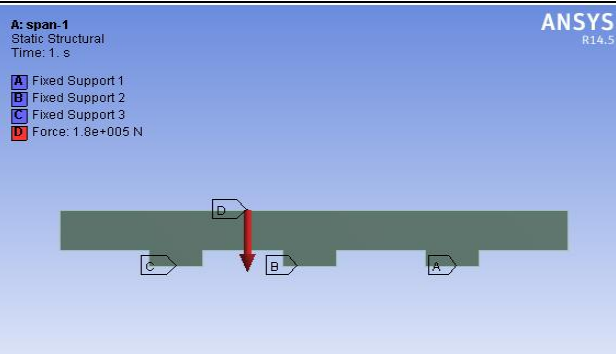


Fig. 27: (A) Boundary conditions of Bridge span
with three stiffeners

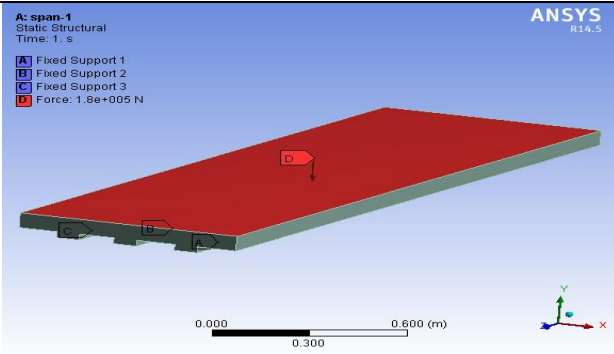


Fig. 28: (B) Boundary conditions of Bridge span
with three stiffeners

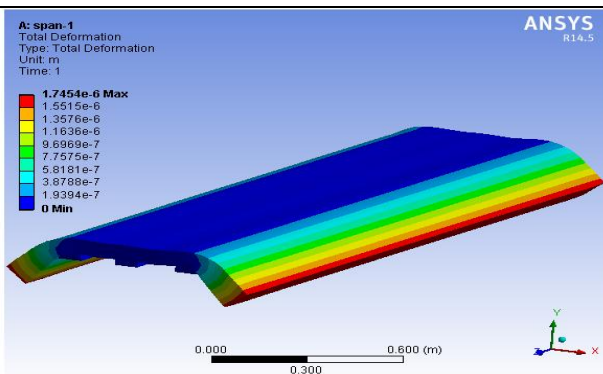


Fig.29:(A)Total Deformation of Bridge span
with three stiffeners

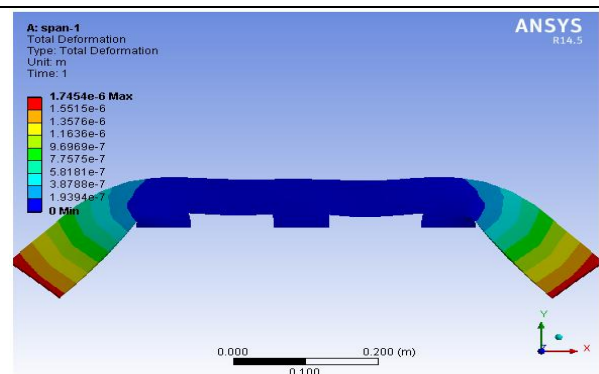


Fig.30:(B)Total Deformation of Bridge span
with three stiffeners

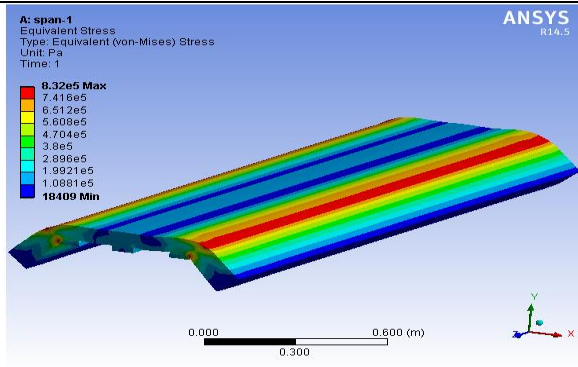


Fig. 31:(A) Equivalent Stress of Bridge span with three stiffeners

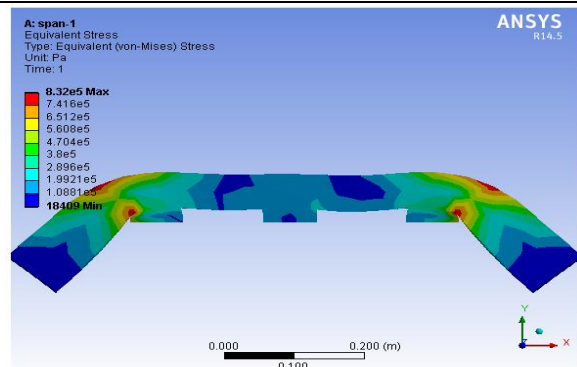


Fig. 32:(B) Equivalent Stress of Bridge span with three stiffeners

3.6.4. Result of bridge span with four stiffeners:

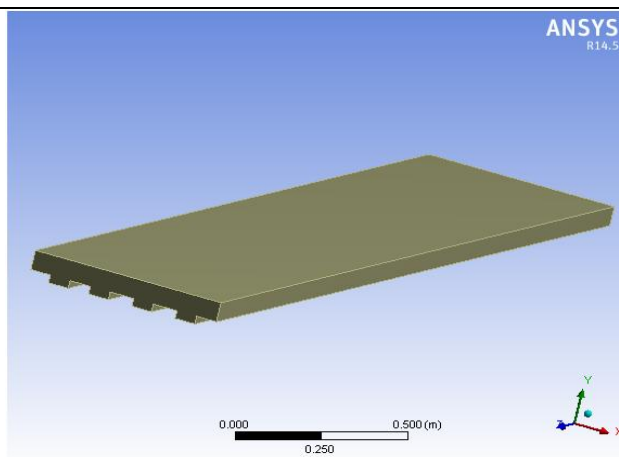


Fig. 33: (A) Geometry of Bridge span with four stiffeners

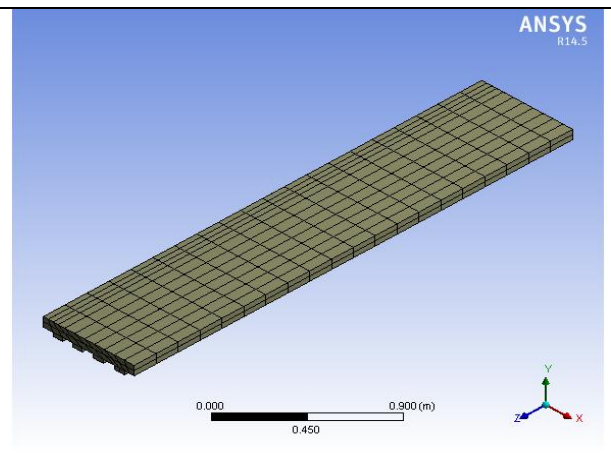


Fig. 34 : Meshing of Bridge span with four stiffeners

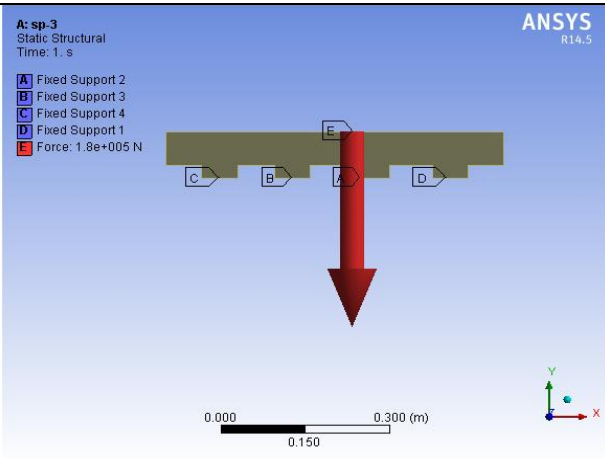


Fig.35: (A) Boundary conditions of Bridge span with four stiffeners

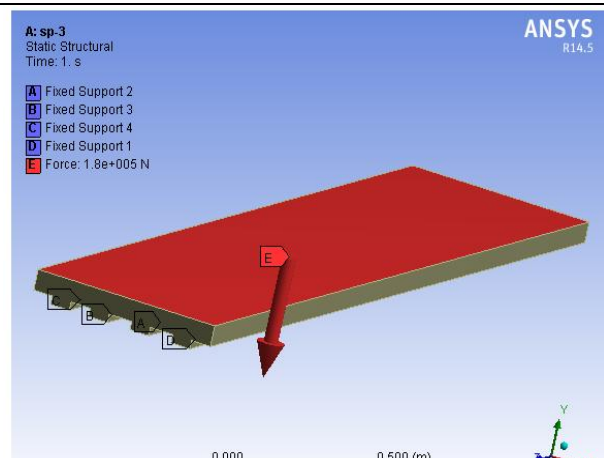


Fig. 36: (B) Boundary conditions of Bridge span with four stiffeners

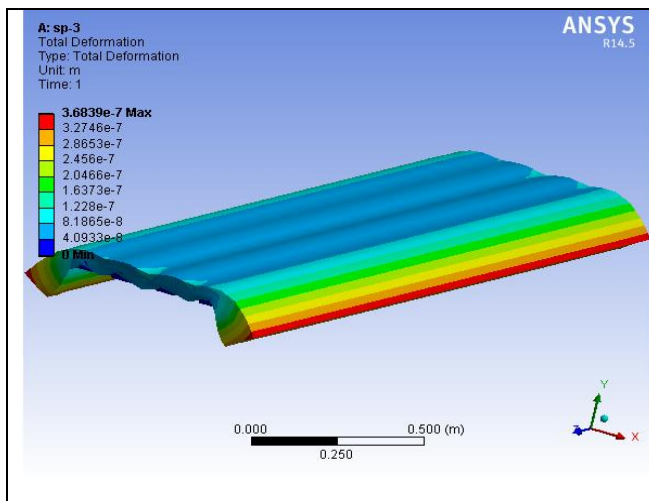


Fig.37:(A)Total Deformation of Bridge span with four stiffeners.

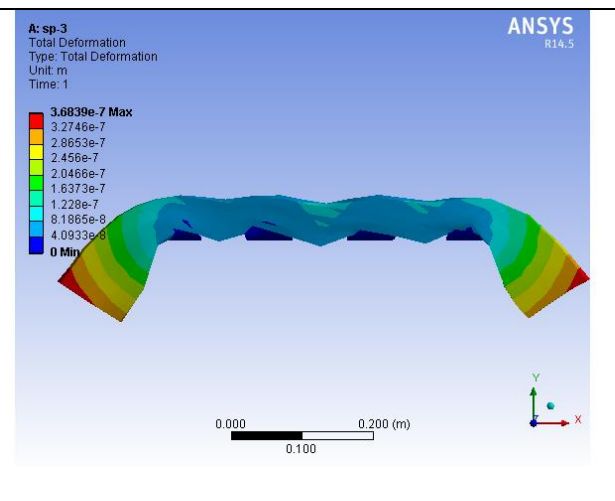


Fig.38:(B)Total Deformation of Bridge span with four stiffeners.

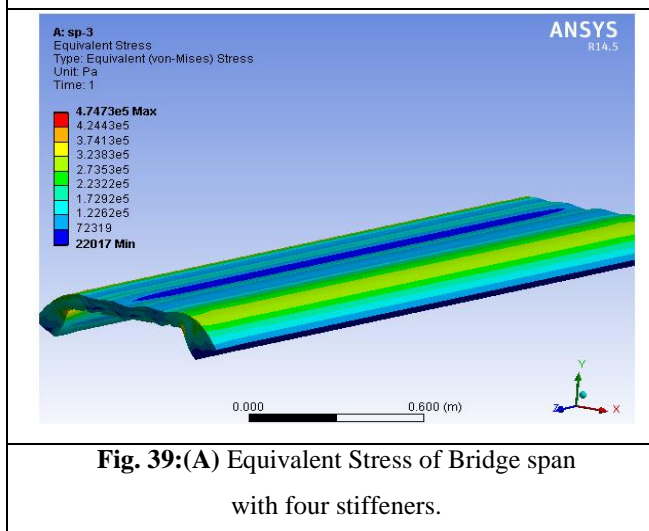


Fig. 39:(A) Equivalent Stress of Bridge span with four stiffeners.

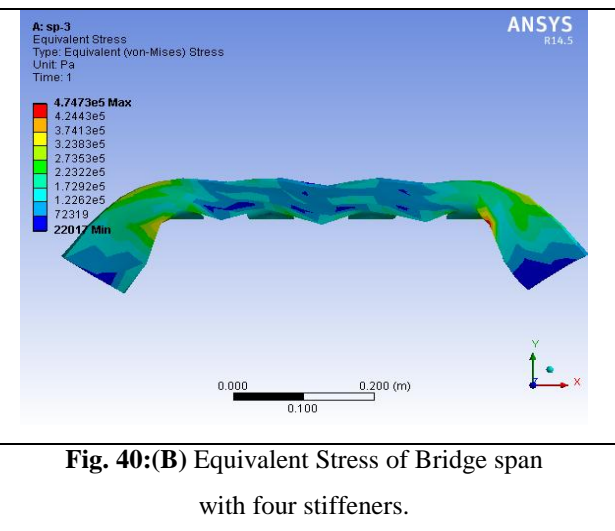


Fig. 40:(B) Equivalent Stress of Bridge span with four stiffeners.

IV.CONCLUSION

The numerical study using the ANSYS program allows investigate the stress analysis of fixed free beam to find out shape with high accuracy. The numerical (FEA) results that are obtained from the ANSYS software. It is observed that the numerical (FEA) results values are in-tuned toward failure values. The procedure and FEA models that are used in this paper are very useful to researchers who willing to work experimental analysis. We have studied the stress and deformation for failure analysis of fixed free beam by using the numerical (FEA) approach using the ANSYS program, it has been found that the relative error between these two approaches are very minute. Firstly we obtained the equivalent Stress of bridge span with four stiffeners for numerically (FEA) and total deformation of bridge span with four stiffeners on the fixed free beam which we were used in this paper.

V.ACKNOWLEDGEMENTS

We gratefully acknowledge department of **mechanical engineering** of **Nutan Maharashtra Institute of Engineering and Technology, Talegaon, Pune (India)** for technical support and providing the research facilities. Their help and dedication toward our work and related research, also their library staff and expert for their directly & indirectly help, support and excellent co-operation.

REFERENCES

Journal Papers:

- [1] D. Ravi Prasad and D.R. Seshu "A STUDY OF DYNAMIC CHARACTERISTIC OF STRUCTURAL MATERIALS USING MODAL ANALYSIS" Asian journal of civil engineering.
- [2] D.Y. Zheng. Free Vibration Analysis Of A Cracked Beam By finite Element Method. Journal of Sound and Vibration, 2004: 457-475.
- [3] Huszar, Zsolt. Vibration of cracked reinforced and prestressed concrete beams Architecture and Civil Engineering, Vol. 6, 2008: 155-164
- [4] G.M.Owolabi, Swamidass, Seshadri Crack detection in beams using changes in frequencies and amplitudes of frequency response functions. Journal of Sound and Vibration, 265(2004) 1-22
- [5] M.Behzad, Ebrahimi, A.Meghdari. A continuous vibration theory for beams with a vertical edge crack. Mechanical Engineering, Vol. 17, 2010: 194-204.
- [6] E. I. Shifrin. "Natural Frequencies Of A Beam With An Arbitrary Number Of Cracks." Journal of Sound and Vibration, 222(3), 1999: 409-423.
- [7] Thin-Lin Horng, "Analytical Solution of Vibration Analysis on Fixed-Free Single-Walled Carbon Nanotube-Based Mass Sensor", Journal of Surface Engineered Materials and Advanced Technology, 2012; 2(1): PP 47-52.
- [8] J S Wu And H-M Chou, "Free Vibration Analysis Of A Cantilever Beam Carrying Any Number Of Elastically Mounted Point Masses With The Analytical-And-Numerical-Combined Method", Journal of Sound and Vibration, 1998, 213(2): PP 317-332.
- [9] C.Chin, A.H.Nayfeh, W. Lacarbonara, two-to-one internal resonance in parametrically excited buckled beam, AIAA , 1997; 97 : 1081
- [10] Haxton RS, Barr ADS. The auto parametric vibration absorber, ASME J.Eng Industry 1972; 94:119-25.
- [11] Brian J. Schwarz & Mark H. Richardson, " EXPERIMENTAL MODAL ANALYSIS", CSI Reliability Week, Orlando, FL, Oct-1999.