RESEARCH ARTICLE OPEN ACCESS

Static Analysis of Bridge Structure using Finite Element Analysis Software

Mahendra Tirpathi[1], Ms Lavina Talawali [2]
*[*1,2] Department of Civil Engineering, SKSITS Indore, India*
Corresponding Auther: Mahendra Tirpathi

ABSTRACT: Finite element analysis is an effective method of determining the static performance of structures for three reasons which are saving in design time, cost effective in construction and increase the safety of the structure. Previously, it is necessary to used advanced mathematical methods in analysis large structures, such as bridges, tall buildings and other. More accuracy generally required more elaborate techniques and therefore a large friction of the designer's time could be devoted to mathematical analysis. Finite element methods free designer's from the need to concentrate on mathematical calculation and allow them to spend more time on accurate representation of the intended structure and review of the calculated performance.Furthermore, by using the programs with interactive graphical facilities, it is possible to generate finite element models of complex structures with considerable ease and to obtain the results in a convenient, readily assimilated form. This may save valuable design time. More accurate analysis of structure is possible by the finite element method leading to economics in materials and construction also in enhancing the overall safety.

Keywords: Bridge Analysis, Static Analysis, ANSYS14.0, Maximum Deflection and Stress

---

Date of Submission: 01-06-2018 Date of acceptance:16-06-2018

I. INTRODUCTION

The primary objective of this investigation was to develop a convenient and reliable analysis methodology, specifically; a procedure for developing finite element bridge models that can accurately predict the static and dynamic response of bridges. Much of the previous research concerned with evaluating the dynamic response of bridges required the development of individual finite element models. In these studies, the commands and procedures used to define these models were remarkably similar, even for different bridges. Thus, this paper focused on developing an interactive framework, consisting of a software package using ANSYS 14.0 that would permit bridge engineers to easily Model Bridge.

Karoumi (1998) [12] derived approaches for solving the moving load problem of cable-stayed and suspension bridges. This research interests in an efficient finite element program development, to carry out dynamic analyses of bridges.

Broquet et al. (2004) [4] describe in a paper the dynamic behavior of deck slabs of concrete road bridge. This research interesting in dynamic analyses of bridges. The finite element method was used to study the local dynamic effects of traffic actions on the deck slabs.

Damian Kachlakev, Thomas Miller(2001) Linear and non-linear finite element method models were developed for a reinforced concrete bridge that had been strengthened with fiber reinforced polymer composites.

II. MATERIALS

The dynamic material properties for the road-bridge as well as for the surrounding ground are important information for the study of the road-bridge. Parameters regarding the soil characteristics are taken from the database that was created during previous studies which were aimed of ensuring acceptable vibration levels. By choosing the parameters according to that database it was easier to compare the results with investigations made previously. The database containing material data also contains information about suitable parameters for concrete and other materials such as asphalt. In Table 1.1 the material parameters which are used in the Modeling are shown. The quantities displayed in Table 1.1 are the materials density ρ [kg/m3], Young's modulus E [Mpa], Poisson's ratio v [−],

Table 1

Material	**ρ[kg/m3]**	E**[MPa]**	v**[−]**
Asphalt	2600	500	0.25
Concrete	2400	4000	0.2
Bedrock	2600	8809	0.4

III. MODELING

The road-bridge was created in ANSYS as a homogeneous solid concrete part according to the section which is shown in figure below. In ANSYS modeling terms, one single part is created and thereafter partitioned into the different material and geometry definitions. The reinforcement steel was not taken into account when modeling the road-bridge, since small deformations were assumed.

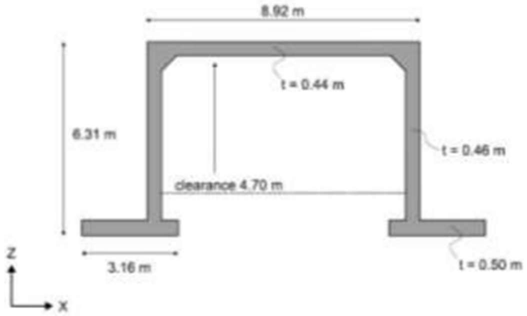

Figure 1 Bridge Structure Drawing

IV. MESHING

Four different material are used to create bridge structure . meshing is done by using solid 185 element and element size is o.1mm .boundary condition are as fallows.

1. 48996N Load Applied at node 563
2. Area 20, 30 are restricted all degree of freedom.
3. 48996N Load Applied at node 424
4. Area 20, 30 are restricted all degree of freedom.
5. 48996N Load Applied at node 209
6. Area 20, 30 are restricted all degree of freedom.

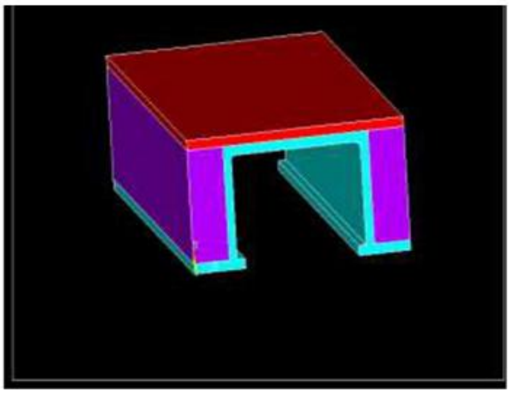

Figure 2 Final Bridge Model

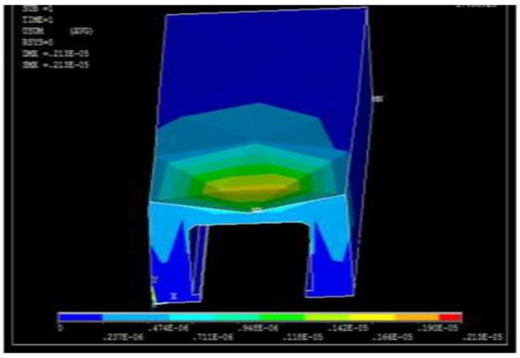

Figure 3 Displacement of bridge when load at 569 node

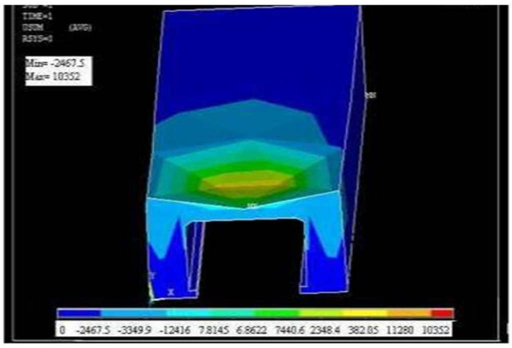

Figure 4 Stress of bridge when load at node 569

Static Analysis

Static analysis of Road Bridge is done aftersetboundarycondition and results are computed by ANSYS like stress,displacement etc.

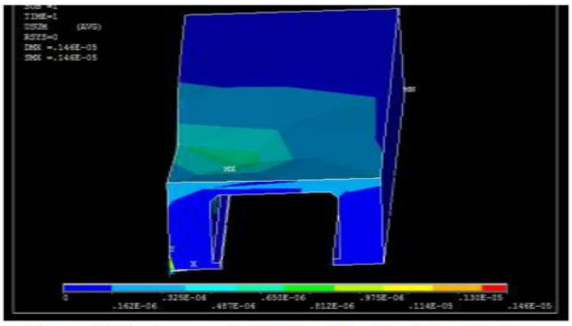

Figure 5 Displacement ofbridge when load at node 424

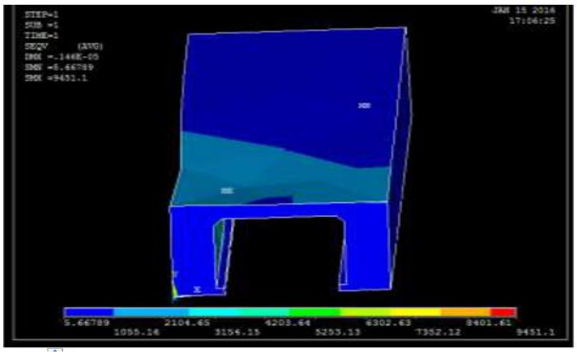

Figure 6 stress of bridge when load at node 424

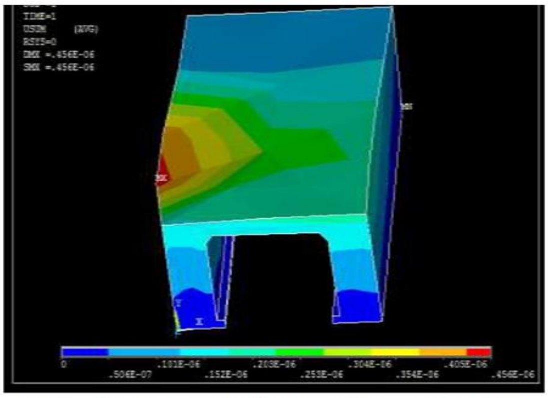
Figure 7 Displacement of bridge when load at node 209

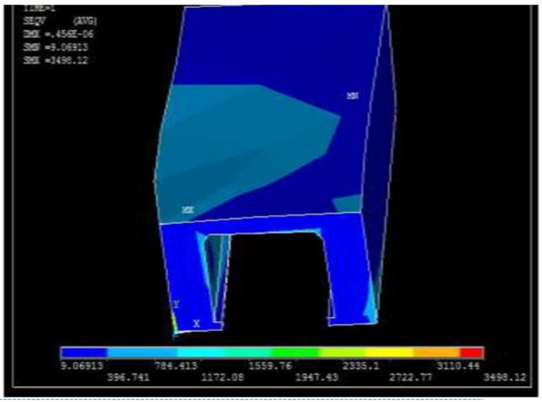
Figure 8 stress of bridge when load at node 209

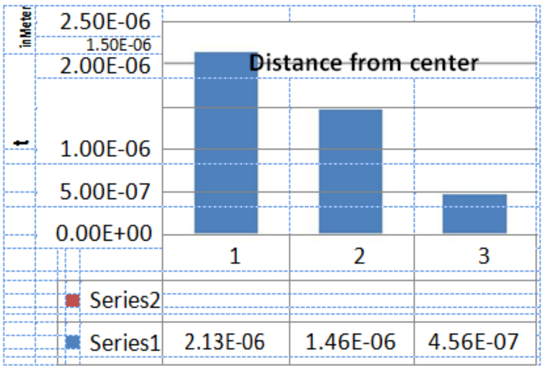

Figure 9 Graph between Displacement and Distance from mid

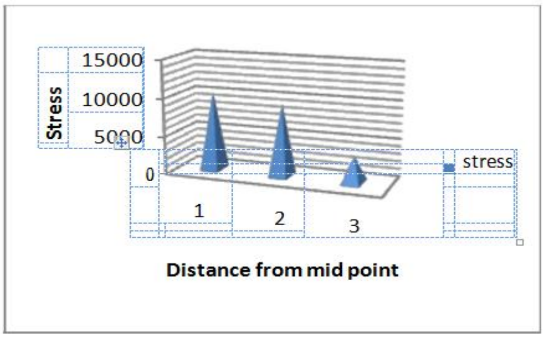

Figure 10 Graph between stress and Distance from mid

V. RESULTS AND DISCUSSION

ANSYS is computed displacement and stress. Above diagram shows that deformation is more towards the mid of the bridge. Stress is also more at mid of the bridge.

VI. CONCLUSION

The following conclusions can be stated based on the evaluation of the analyses.

(1) Bridge modeled quite well using FEA, and the failure load predicted is very close to the failure load calculated.

(2) Deflections and stresses at the zero deflection point and decompression are modeled well using a finite element package.

(3) Middle of the bridge is always at a maximum stress and displacement so that it will fail from middle when load is higher.

REFERENCES

[1]. ANSYS, ANSYS User's Manual Revision 5.5, ANSYS, Inc.,

[2]. Canonsburg, Pennsylvania, 1998.

[3]. Bangash, M. Y. H., Concrete and Concrete Structures: Numerical Modeling and Applications, Elsevier Science Publishers Ltd., London, England, 1989.

[4]. Broquet, C., Bailey, S.F., Fafard, M. And Brühwiler, E., (2004), Dynamic Behaviour of Deck Slabs of Concrete Road Bridges, Journal of Bridge Engineering, Vol. 9, No. 2, ISSN 10084-0702/2004/2-137.

[5]. Concrete Beams, International Journal of Advanced Technology in Civil Engineering, ISSN: 2231 –5721, Volume-1, Issue-3, 2012

[6]. Damian Kachlakev, Thomas Miller,Finite Element Modelingof Reinforced Concrete Structuresstrengthened With Frp Laminates, OregonDepartmentofTransportationResearchGroupandFederalHighway Administration, May 2001

[7]. Doctoral Thesis, Royal Institute of Technology, Stockholm, ISSN 1103-4270.

[8]. Jenmfei BJ. Chang, Ian N. Robenson, 2003, Computer modeling of the proposed Kealakaha stream bridge, Research Report, UHMCEE0343.

[9]. Karoumi, R., (1998), Response of Cable-Stayed and Suspension Bridges to

[10]. Mindess, S. and Young, J. F., Concrete, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1981.

[11]. Moving Vehicles – Analysis methods and practical modelling techniques,

[12]. (Melaku.S, Hongsheng Qiu, 2014) Transient Dynamic Analysis of Simple Railway

Bridge under Train Moving Load, 2014 International Conference on Civil Engineering, Energy and Environment

[13]. Shah, S. P., Swartz, S. E., and Ouyang, C., Fracture Mechanics of Concrete, John Wiley & Sons, Inc., New York, New York, 1995.

[14]. William, K. J. and Warnke, E. P., "Constitutive Model for the Triaxial Behavior of Concrete,"Proceedings, International Association for

[15]. Bridge and Structural Engineering, Vol. 19, ISMES,Bergamo, Italy, pp. 174, 1975.

Mahendra Tirpathi "Static Analysis of Bridge Structure using Finite Element Analysis Software "International Journal of Engineering Research and Applications (IJERA) , vol. 8, no.6, 2018, pp.70-73