International Journal of Scientific Research and Engineering Development-– Volume 4 Issue 6, Nov-Dec 2021

Available at www.ijsred.com

RESEARCH ARTICLE OPEN ACCESS

A Review Study of Pratt Truss & Lattic Truss with Different Section by Using Ansys Software

Bhupendra Shakya *, Anil Rajpoot**, Sonu Singh**

*(Civil Department, Vikrant Institute of Technology & Management, GWALIOR, (M.P.)
Email: bhupendra.civil14@gmail.com
** (Civil Department, Vikrant Institute of Technology & Management, GWALIOR, (M.P.)
Email: ce_anilrajput@vitm.edu.in

----------**********----------

Abstract: A bridge is an important structure in the road transportation network. Its performance during and after an earthquake is quite crucial to provide relief as well as for security purposes. It is also subjected to vibration during the movement of vehicle. The structural steel trusses were optimized in ANSYS utilizing the design optimization tool as a first-order optimization approach, and it was extended to compare the optimum truss geometry for the least amount of weight. To ensure solution convergence, mesh analyses were done on all ANSYS finite element models. The trusses were compared by determining the min margin of safety in all truss members. To ensure a fair assessment, all trusses must have identical geometry and loading situation or pattern. When preparing a truss, the main goal is to see which truss is more efficient. Finally, following optimization, considered that the Warren truss had a higher stiffness to wt. ratio than other trusses. The goal of this study is to focused structural engineers to the potential for distortions during the service life of steel and composite bridges when its subjected to vehicle dynamic activities. For this research work various effort has been made to analyze 2 types of bridge structure i.e. Pratt truss & lattice truss with two different sections (‘I’ Section & ‘C’ Sections) by applying various loads at the nodes of the frame of two trusses. This work focuses on the analysis of truss bridge structure which is most widely used in steel bridge as railway and pedestrian crossings. The primary focus has been on comparing total deformation and direct stresses b/w these 2 types that is 'I' and 'C' sections.

----------**********----------

I. INTRODUCTION

The bridges are the structures, which provide means of communication over a gap and they provide passage for the highway and railway traffic over these gaps. There are several classifications of bridges based on different considerations. Some of the major classifications are based on: material used, makeup of main load carrying elements, the structural layout of the principal load carrying members, floor location, kind of connections, the level of highway and railway track crossing, and the nature of connections, the level of highway and railway track crossing, and the character of bridge movement The scope of this research is limited to steel truss bridges, specifically the truss component. A bridge having spans a canyon, a roadway, a river, a railway, or other obstacles, allowing automobiles,

trains, and pedestrians to travel across safely. A pedestrian railway bridge is an alternative to automobile transit that is designed for pedestrians, bikers, animal site visitors, and horse riders. Pedestrian bridges add to the beauty of the landscape and can be utilised to visually connect beautiful regions or to signify a transaction. For poor rural communities in developing countries, a footbridge is one of the ways to travel to medical clinics, schools, and markets, which would otherwise be inaccessible because to canals that are too high to move.

II. LITERATURE REVIEW

P.L. Bishay et. al. (2020) FEM was initially introduced to engineering students enrolled in a "Computer-Aided Analysis and Design" course through a basic computer code for analysing 2D or 3D trusses in this study. To assist in the modelling and analysis of big and sophisticated trusses, computational tools dubbed "Truss Builder" and "Truss Analyzer" were offered. In two mini projects, students were encouraged to create tasks for their peers that involved assessing two dimensional or three dimensional trusses which simulated real structures or practical applications of their choice.

S. Baisthakur et. al. (2020) The goal of this research is to create a Bayesian Hamiltonian Monte Carlo-based technique for FEM update. The proposed algorithm employs an adaptive prior-based strategy to produce intermediate pdfs. For model updating, numerical analysis is performed with various coefficients of variations of the prior. This study also includes guidelines for their effective selection technique. The effectiveness of the suggested method for updating a steel truss bridge's finite element model is demonstrated using synthetic trials and actual test data. Finally, the algorithm's performance is compared to that of the typical Markov chain Monte Carlo algorithm.

A. Singh. et. al. (2019) The current study used FEA to investigate the vibration behaviour of a Warren truss bridge. CATIA was used to creating CAD model of the bridge that adhered to Indian standards. To give improved structural qualities, the bridge in this study was developed as a composite structure. Model analysis with ANSYS was used to investigate the vibration behaviour of this composite structure. And 500 KN of static load was also applied to the CAD model. The vibration and deformation simulation resulted in a satisfactory result.

K. Senthil et. al. (2017) Using Abaqus application, 3d numerical analyses of railway bridge structures were accomplished. The current study took into account the bridge length of 30 m and a single truck. The application of the JC version in ABAQUS predicted the constitutive and fracture behaviour of chemicals. The JC model's material properties for the bridge contributors are available for inspection. In light of and von-misses stress, the bridge responses were predicted. The examinations were carried out while taking into account the bridge's beauty AA loading. The influence of primary girder intensity was investigated by increasing the intensity to 1600, 1400, 1200, and 1000 mm. Bridge reactions are researched by dividing the span of the bridges in half and using this as a strengthening measure. Furthermore, the reaction of bridges was investigated by removing cross girders from the centre of the bridges in the event that the contributors were damaged owing to corrosion and failure of member.

Darius Bačinskasa et. al. (2017) This study discusses an experimental analysis of behaviour of a glass fibre reinforced polymer area truss bridge variant subjected to static loading. The bridge prototype was put together with the help of fiberline Composites Ltd's GFRP profiles, metal bolts, and GFRP brackets. The wooden bridge deck was erected in order to load the shape. While measuring

the bridge node displacement, and 13.3 KN complete load was applied in four ranges. At each loading stage, the truss shape's flexural behaviour was measured. The outcomes are in perfect agreement, according to comparative analysis. The obtained results show that the planned and tested bridge model has a suitable structural stiffness reserve.

Emdadul Hoque et. al. (2017) As a mechanical structure, the harmonic response of a model of a simply supported steel truss bridge was investigated. ANSYS Workbench 15.0 was used for the geometric modelling and simulations. For first 15th nodes, natural frequencies were identified using modal analysis, while harmonic response was detected using an excitation sinusoidal force of 100N dispersed over the bridge deck with an analysis range of 0-1000Hz. With respect to frequency variation, the bridge-load interactions are graphically and analytically represented in terms of total deformations and equivalent stresses. All of these analytical and graphical results represent the possibility for many degrees of vibration and provide a full geometric optimization to prevent potential resonance in the steel truss bridge model, which would lead to a practical implementation of such a steel truss bridge.

Huili Wang et. al. (2017) Multi-scale FEM is used to check the fatigue performance of a steel truss integral junction. Experimental and numerical results are compared. The fatigue performance of a steel truss integral junction is studied against the backdrop of a sea-crossing suspension bridge. The beam elements and three-dimensional elements are connected using the connection elements. The connection element have 2 joints, first one having 6 DOF, 3 degrees of translation, and 3 degrees of rotation, all of which are connected. The results are exactly same of multi-scale FEM and experimental FEM. To simplify the computation, the multi-scale FEM can give an accurate simulation of primary research portion or ensure the fatigue life forecast. This strategy is both effective and practical. The multi-scale FEM is a new and accurate method for analysing structure fatigue performance.

Jayakrishnan. T J et. al. (2017) ANSYS was used to examine the seismic behaviour of a composite bridge without cross girders. In aerospace, civil infrastructure, and construction, composite structures have a wide range of uses. The response spectrum method is used to investigate the seismic behaviour of composite bridges. The geometry, material, configuration, response spectrum chosen, and system construction elements all influence the structure's response. ANSYS is used to investigate the seismic behaviour of a composite bridge in this study.

Mohamed Ghannam et. al. (2017) studied the effect of post tensioned cables in strengthening double span steel trusses. Different truss's systems (Lattice and N types) are involves in this study. Different post tensioned cable procedures are utilised to strengthen various truss structures. The profile and placements of post tensioning wires are the key differences between these approaches. In order to establish the best post tensioning technique for each truss system, comparisons of various techniques are done. The ANSYS programme was used to conduct the analysis and obtain the findings.

Najla Yas V et. al. (2017) The analysis of a box girder is carried out using the ANSYS programme. Highway flyovers and contemporary elevated light rail transport infrastructure frequently feature box girder bridges. Its often a beam bridge type, it can also be utilised on cable-stayed bridges and other structures. In this study analysis of different shapes of box girder is done A section of box girder is selected with a certain mass. Box girders having various cross sections such as Rectangle, Square, Trapezoidal and curved are modelled. Mass of all sections is kept constant so that it becomes comparable with each other. Analysis of these

sections is done in ANSYS Software for results such as Moment, Stress, Deformation, Frequency and Time period. The results are compared to find a better sectional shape.

Z. Zheng et. al. (2017) They looked into the probability of structural deterioration over time. The results for load instances below it shows that it will be more destructive if impact load arranged, or collapsing will occur if the pillar fails after the crash. ANSYS/LS-DYNA dynamic analysis software was used to creating FE truss model with 6 trusses in this investigation. It acted out scenarios in which structures were hit by a large truck. This study evaluated the impacts of modifying variables such as crash positions, impact load intensity, and structural height-span ratio on truss structure stress and strain. This article also looked at the prospect of structural progressive collapse.

Prashant S Patil et. al. (2017) the research carried out to examine the performance of steel bridge with FRP in accordance with Indian Road Congress (IRC) codes under moving load. The deformation of a bridge caused by vehicles travelling at different speeds is quite difficult, and it has received some attention from the engineering community. Due to its numerous advantages over conventional materials, the use of fibre reinforced polymer (FRP) bridge deck systems is growing quickly all over the world. The FRP bridge deck is lighter, more durable, easier to work with, requires no maintenance, and has a low life cycle cost. ANSYS software creates a FE model of a bridge with FRP based on usual theory on vibration analysis between bridge and cars. The dynamic reaction characteristics of the bridge body are obtained using numerical simulation analysis when a vehicle passes across the bridge at various speeds and frequencies, and the inner force of the bridge is determined. These will serve as a benchmark for improving bridge vibration control methods under moving loads.

Alpesh Jain et. al. (2016) ANSYS software was used to investigate a bridge structure made of four different materials and to perform a modal evaluation of bridge hassle. For each of the four substances, eight node solid elements are chosen and meshing is performed. Each material's material properties are chosen in accordance with the literature database in the ANSYS software programme. The modal evaluation in ANSYS is completed to attain natural frequency or bridge's mode shape in order to avoid the bridge's resonance. It has been determined that the bridge should not be used at received frequencies that are similar to natural frequencies at applied hundreds. Resonances will arise if it is utilised at natural frequencies, and the bridge may hurt or collapse. The future scope includes temporary bridge evaluation and harmonic analysis, as well as the deflection of the bridge over time, which can be derived using FEA software.

Alika Koshi et. Al. (2016) A comparison of a through arch bridge at various arch locations was studied. A pure compression shape is an arch. It may span a large region by solving forces into compressive stresses or releasing tensile tensions in the process. This is sometimes referred to as arch motion. The arch will push at bottom force inside the arch are carried to the floor. The importance of the arch peak within the conveyed forces and stresses cannot be overstated. This study uses a three-dimensional bridge model in the FEA software programme ANSYS to describe the behavioural components of a through arch bridge with particular arch placements and to compare them to the actual structure.

III. CONCLUSIONS

The standard loading system is used in the ANSYS study for this steel truss. The standard value specifies a set of constrained preconditions. We can conclude from this study that the ANSYS analysis

of this truss is really useful. The study looked into the possibilities of analysing and designing truss bridge structures using steel profiles that were readily available in the area. Even though the cost of local production is closer to importing it is still a good option since it helps in the capacity building of local design, fabrication and construction firms, creates job opportunities for many people and is a saving in foreign currency.

IV. REFERENCES

1) Alika Koshi, Laju Kottalil, “Performance Comparison of Through Arch Bridge at Different Arch Positions”, International Journal of Scientific & Engineering Research, Volume 7, Issue 9, September-2016.
2) Alpesh Jain, J.N. Vyas, "Modal Analysis of Bridge Structure Using Finite Element Analysis", IOSR Journal of Mechanical and Civil Engineering, Volume 13, Issue 4 Ver. IV, Jul. - Aug. 2016, PP 06-10.
3) B. Stankiewicz, "Composite GFRP Deck for Bridge Structures", Steel Structures and Bridges, Elsevier, 2012. PP. 423-427.
4) CHEN Shuli, SU Mubiao,LIU Yuhong, WANG Qingmin "Vibration Remote Monitoring System of Continuous Steel Truss girder for the Wuhu Yangtze River Bridge", 14th World Conference on Earthquake Engineering, October 12-17, 2008, Beijing, China.
5) Darius Bačinskasa, Arvydas Rimkusa,b,*, Deividas Rumšysa , Adas Meškėnasa,"Structural analysis of GFRP truss bridge model", Modern Building Materials, Structures and Techniques, Procedia Engineering 172, Elsevier, 2017, PP. 68 – 74.
6) E. Yamaguchi, R. Okamoto, K. Yamada "Post-Member-Failure Analysis Method of Steel Truss Bridge", The Twelfth East Asia-Pacific Conference on Structural Engineering and Construction, Elsevier, 2011.
7) F. Masoumi, A. Mehrabzadeh, "Application of Influence Lines on Static Analysis of Cable-Stayed Bridges", IACSIT International Journal of Engineering and Technology, Vol. 5, No. 6, December 2013.
8) Huili Wang, Hao Gao, Sifeng Qin, "Fatigue Performance Analysis and Experimental Study of Steel Trusses Integral Joint Based on Multi-Scale Fem", Engineering Review, Vol. 37, Issue 3, 257-262, 2017.
9) J. Eckermann, S. Mehmood, H.M. Davies, "Design for Reliability of Half-Bridge Module due to Design Consideration and Material Selection", KES Transactions on Sustainable Design and Manufacturing, 2014, pp.703-719.
10) Jianing Hao, "Natural Vibration Analysis of Long Span Suspension Bridges", 5th International Conference on Civil Engineering and Transportation, ICCET, 2015.

