Vol-05 Issue 12, December - 2021 ISSN: 2456-9348
Impact Factor: 5.004

International Journal of Engineering Technology Research & Management

A COMPARATIVE STUDY OF PRATT TRUSS & LATTIC TRUSS WITH DIFFERENT SECTION BY USING ANSYS SOFTWARE

Bhupendra Shakya
M.Tech Student, Civil Engineering Department, Vikrant Institute of Technology and Management, Gwalior, (M.P.) India
Anil Rajpoot,
Assistant Professor, Civil Engineering Department, Vikrant Institute of Technology and Management, Gwalior, (M.P.) India
Ankur Tyagi
Assistant Professor, Shri Mata Vaishno Devi University, Katra, Jammu and Kashmir, India

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.*

1. 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.

2. INTRODUCTION TO FINITE ELEMENT METHOD

This is one of the valuable tools for calculating numerical solutions to a wide range of construction problems. The process can be applied to any complex shape or geometry, as well as any material, under a variety of limit and stacking circumstances. The comprehensive explanation of the limited component method meets the examination requirement of

iJETRM
International Journal of Engineering Technology Research & Management

today's complicated building frameworks and structures, where closed form arrangements for administering balance conditions are typically unavailable. Furthermore, it is a useful planning tool that allows designers to consider various structural situations (different shapes, materials, loads, etc.) when performing parametric structure calculations. The approach was developed in the aerospace sector to investigate stress in a complex airframe structure. It arose from what was once known as the matrix inspection strategy, which was used in aviation machine design. The approach has grown in popularity among both specialists and professionals. The basic concept behind limited component approach is that a body or structure break down into small, limited-measurement components known as "finite components." The initial body, or structure, is thus viewed as a collection of these components linked together at a small number of joints known as hubs or nodal centres.

3. GENERAL PROCEDURE OF FINITE ELEMENT METHOD

This method is used for piecewise estimation in which the structure or body is partitioned into little components of limited measurements considered limited components and after that the first body or the structure is considered as a gathering of these components associated at limited number of joints called nodal focuses or hubs. Since the genuine variety of field factors like displacement, stress, temperature, pressure or speed inside the continuum are not known, the variety of variable of field inside a limited component can be approximated by a basic capacity. These guess capacities called introduction models are characterized as far as the estimations of the field factors of the hubs.

The nodal estimations of field variable are gotten by settling the field conditions, which are by and large as framework conditions.

The approximation functions determine the field variable across the assemblage of elements after the nodal values are known.

The arrangements of general continuum issues by the finite component technique dependably pursue a precise well-ordered process. The well-ordered system for the static structuralproblem can be expressed as follows:

Step 1: - explanation of design model (Domain). The primary step in finite element process is to separate the structure of result area in to sub divisions or elements.

Step 2: - chosen of proper interpolation form. As the dislocation (field variable) explanation of a difficult structure under any precise load conditions cannot be predicted accurately, we assume some suitable result, within a component to estimate the unknown solution. The assumed result must be easy and it should fulfill certain convergence necessities.

Step 3: - beginning of component stiffness matrices (feature matrices) and load vectors. From the assumed displacement version, the stiffness matrix [K (e)] and the burden vector P (e) of element ‘e’ are to be resultant by using the use of both equilibrium situations and anappropriate Variation precept.

Step 4: - The balancing equations are created by putting together element equations.

Because the structure is made up of many finite elements, the individual element stiffness matrices and load vectors must be built correctly, and overall equilibrium equation must be written as [K] = P.......................... (1)

Where [K] stands for the constructed stiffness matrix, is nodal vector displacement, and P stands for nodal vector pressure for the entire shape.

Step 5: - To find the nodal values of displacement, solve the system equation (subject variable). To account for the problem's boundary conditions, the standard equilibrium equations must be adjusted. The equilibrium equations can be stated as, when the boundary conditions are taken into account.

$$[K]\varphi = P \qquad (2)$$

For linear issues, the vector ‘φ’ can be solved very without problems. But for non-linear problems, the answer has to be received in a series of steps, each step involving the amendment of stiffness matrix [K] and ‘φ’ or the weight vector P.

Step 6: - Computation of detail strains and stresses. From the regarded nodal displacements, if required, the detail lines and stresses may be computed via the use of the essential equations of stable or structural mechanics. In the above steps, the phrases indicated in brackets enforce the general FEM step-through-step manner.

4. MODELING OF BRIDGE STRUCTURE

The use of ANSYS Workbench to model the bridge truss designing has been thoroughly explained. FEA is used to reconstructing the mathematical behaviour of real-world engineering structure. All nodes, elements, material qualities, real constants, boundary conditions, and other features required to characterise the physical system are included in the model. The model will be created first, then particular boundary conditions will be added to the nodes, and finally a final analysis will be performed.

4.1. Layout of ANSYS Workbench

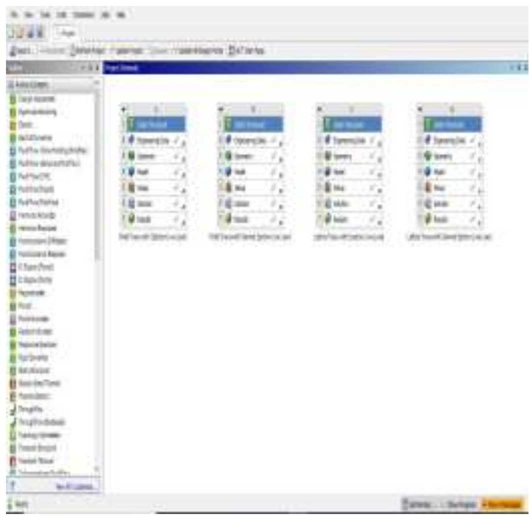


Figure 4.1: Shows ANSYS Workbench layout

5. MODEL GENERATION

First, we click double to the "Static Structural" then small window will open where rename done and its clearly shows 1 no. fig. If anyone wants to displace or interchange the material we will double click on "Engineering Data". For drawing the beam we double click on "Geometry". clicking twice on "Geometry" separate window names "Design Modeler" shows in 4.2 no.fig.

It will come out, when we selecting dimension 1st on which we wants to do work fig 4.3 showing that.

iJETRM
International Journal of Engineering Technology Research & Management

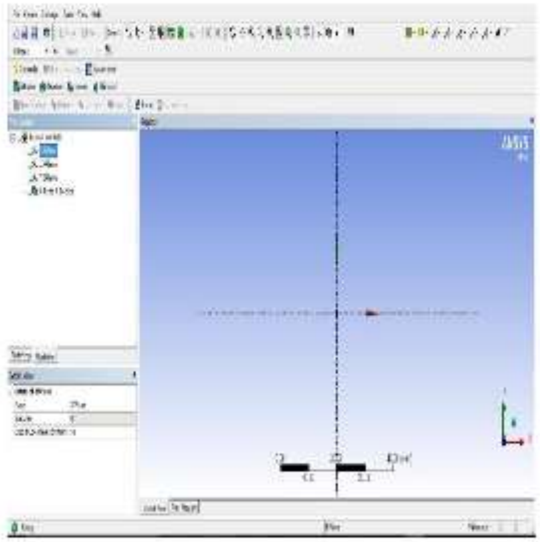


Figure 4.2: Layout of Design Modeler

6. GEOMETRY OF SECTIONS

There are 2 types structures of bridge. The Pratt and lattice types are designed here, as are the bridge sections, namely the 'I' and 'C' sections. The 1st bridge was built with 'I' section beam, and 2nd bridge was built with 'C' section beam. Below is a description of the geometry of the 'I' and 'C' sections. The material used to design the bridge structure is structural steel.

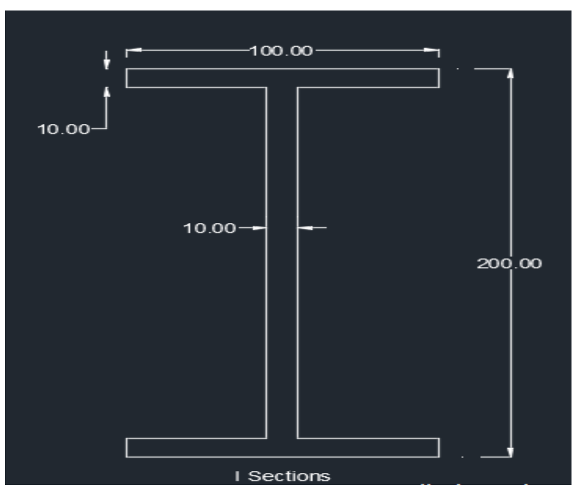
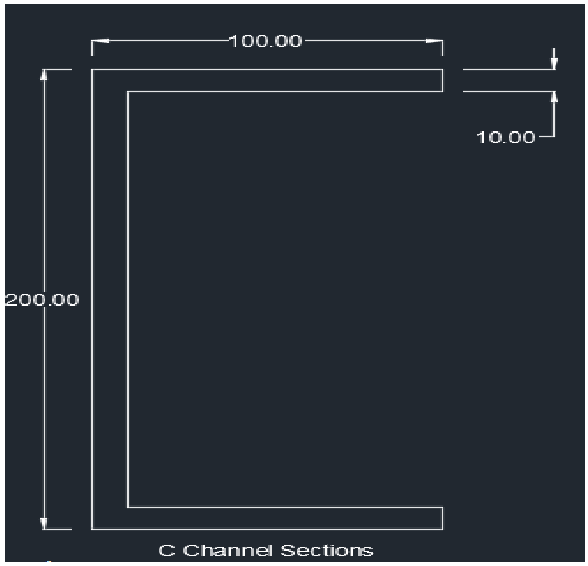


Fig 4.4'I' Section dimensions

Figure 4.4 shows the geometrical layout of 'I' Section element that is used to design a bridge structure.

iJETRM
International Journal of Engineering Technology Research & Management



7. CONCLUSION

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.
Various conclusions can be stated based on the evaluation of the analyses.
At a max 500 KN load applied, deformation occur in Pratt structures with C or I sections is 1.530mm, 17.327 N/mm2 are stresses.
Deformation in a Pratt truss with a 'C' section is 1.526mm, or 17.284 N/mm2 are stresses, according to analysis. We discovered Pratt structure with the 'C' section has fewer deformations and stresses after examining both section data. We discovered Pratt truss with ‘C' type is excellent for designing purposes during this research.
When comparing the results of a Lattice truss construction with 'I' and 'C' section at a max 500 KN load applied, the deformation and stresses of the 'I' section are 1.8577 mm and 19.889 N/mm2, respectively. After that as per analysis results of both sections describe a difference between both sections, as a civil engineer we do not neglect minor variations. According to the study, in both Pratt and lattice bridge structures, the 'C' section has very least deformation or stresses than the 'I' portion.
The result shows ‘I' section deformation of Pratt truss or stress at 500 KN load were 1.530 mm and 17.327 N/mm2, respectively, as compared to Lattice truss with ‘I' section. Also, at 500 KN load, deformation and stresses for Lattice truss with 'I' section are 1.8576mm and 19.889 N/mm2, respectively.
Here, Pratt truss with ‘I' section with above study When comparing Pratt with 'C' section to Lattice 'C,' we found that at 500 KN, Pratt truss with 'C' section deformation and stresses are 1.525 mm and 17.3274 N/mm2, respectively. Deformation and stresses for a Lattice truss with a 'C' section at 500 KN load are 1.8577 mm and 19.835 N/mm2, respectively. According to the above analysis, Pratt truss with 'C' section has the least deformation and stresses. Pratt truss with 'C' section is ideal for designing purposes.

iJETRM
International Journal of Engineering Technology Research & Management

8. SCOPE OF FUTURE WORK

The similar research can be done for RCC and prestressed bridges. Seismic performance paired with moving load can be compared. During the analysis, we can employ a variety of trusses. This study can be used to monitor an old truss construction.

9. REFERENCES

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.

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.

B. Stankiewicz, "Composite GFRP Deck for Bridge Structures", Steel Structures and Bridges, Elsevier, 2012. PP. 423-427.

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.

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.

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.

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.

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.

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.

Jianing Hao, "Natural Vibration Analysis of Long Span Suspension Bridges", 5th International Conference on Civil Engineering and Transportation, ICCET, 2015.