

e-ISSN: 2319-8753 | p-ISSN: 2320-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 10, Issue 4, April 2021

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 7.512

Static Structural Analysis and Bridge Modeling Using ANSYS

Gayatri Belsare¹, Trupti Narkhede², P.J.Salunke³

P.G. Student, Department of Civil Engineering, M.G.M.'S College of Engineering and Technology, Navi Mumbai, Maharashtra, India¹

Assistant Professor, Department of Civil Engineering, M.G.M.'S College of Engineering and Technology, Navi Mumbai, Maharashtra, India²

Associate Professor, Department of Civil Engineering, M.G.M.'S College of Engineering and Technology, Navi Mumbai, Maharashtra, India³

ABSTRACT: Static structural analysis calculates the effect of steady (or static) loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. Since structures are constantly subjected to varying loads, the possibility of earthquakes, high winds, water currents and other loadings, they must be designed with massive strength and load carrying capacity. Ideal purpose of analysis is to obtain an accurate measure of expected structural response for a given earthquake. In this paper, a 20m pedestrian bridge is analyzed by using ANSYS for static analysis, thermal analysis and frequency distribution for different widths of the bridge. The final result and conclusion will be based on behavior of bridge to the loads and different cross sections acting with a static nature and comparative parameters like stress, displacement, strain etc.

KEYWORDS: Pedestrian Bridge, Static structural analysis, Thermal analysis, Frequency distribution, ANSYS, Stresses, Displacement.

I. INTRODUCTION

Structural analysis is a science that ensures that the structures are safe and fulfil the functions for which they were built. Structural analysis comprises the set of mechanics theories that obey physical law required to study and predict behavior of structures. Bridges are usually constructed with substantial safety margins and compensation for surplus loading conditions. For enhanced viability and sustainability over several hundred years, the design needs to be robust and analyzed from every aspect of physics. For ordinary structures static analysis is sufficient, but for important structures dynamic analysis should be carried out. The ease and fineness in designing of bridge structure by generating 3D model and testing it in a model based environment for physical conditions, empower structural engineers with quick and accurate decisions. Designing of such large structures involves 3D model generation and analysis using software simulation for conceptual design and design development. Any structure is analyzed with static method or dynamic method. Selection of an appropriate analysis method depends on a number of factors.

II. LITERATURE REVIEW

The present work is aimed at static structural analysis and bridge modelling using ANSYS. Various studies have been carried out till date for cable stayed and suspension bridge but none of them compared static and thermal analysis for pedestrian bridge.

T.Subramani et al (2014) Analyzed this paper demonstrates how sophisticated computational techniques can help to understand better the behavior of cable-stayed bridges and what kind of tools are available for bridge engineers to

handle even the most extreme situations. Since one of the key elements in the success of this structural system lies in its construction method, the main focus of this paper is on the simulation of the free-cantilever construction process and the determination of cable forces during the phases of the erection in order to achieve the desired shape and internal force distribution in the final state of the structure. The paper, using ANSYS/Civil/FEM, a software package developed for advanced civil engineering analyses, presents different numerical techniques (based on both linear and non-linear analysis) to determine the final state of the bridge and the backward process analysis to conduct a step-by-step analysis of the construction.

Muhammad Habib et al (2017) Studied the dynamic behavior of a cable stayed bridge. The response of the cable stayed bridge is examined by carrying out non-linear static and dynamic (earthquake) analyses. Modes and the natural frequency of the bridge is studied by Modal analysis based on the Block Lanczos method. ANSYS is used to simulate the 3D model based on the Finite Element Method (FEM). Cables are modeled by single element approach instead of multiple elements to reduce the data usage and computational time.

Mahendra Tirpathi et al (2018) Studied 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.

Rajesh Chaturvedi et al (2019) Studied for longer span, cable stayed bridges are the first choice and to study its behavior under static and vehicle loading is important. Therefore, it becomes essential that the modelling of cable stayed bridge is more realistic and the analysis results are more satisfactory.

Kishan Gautam et al (2020) Studied that for the analyses of skew bridges by the finite element method using eight noded isoperimetric brick element (serendipity), shell elements and some different elements are also attempted, and has seen that brick elements give better results for shear. ANSYS software (version 7) has been used for the finite element analysis. The modeling which is done is applicable for any skew angle and aspect ratio of slab bridges. In this analysis, different types of I.R.C. loadings (Class AA Tracked and Class A) have been built in the modeling and also variable live load positions have been considered, to obtain maximum stresses development on the slab. The parametric study and detailed analysis has given a useful understanding for analysis of skew bridges.

III. METHODOLOGY

In this study, a static structural analysis is done for bridge of 20 m span, for different loads and sections. The result of this analysis will include stress, strain, frequencies and displacement; the conclusion will be findings of most suitable case depending on their analytical results.

1. In present research we have used different factors for the analysis like static analysis, thermal analysis, frequency distribution etc.
2. Model the structure in ANSYS 21.0
3. Static analysis is done as per IRC-24-2001
4. Running the model in ANSYS 21.0 for static criteria.
5. Obtaining parameters like stress, strain, displacement, frequency, thermal analysis etc.

6. Observation and comparison of result.

7. Conclusion

IV. STRUCTURAL MODELLING

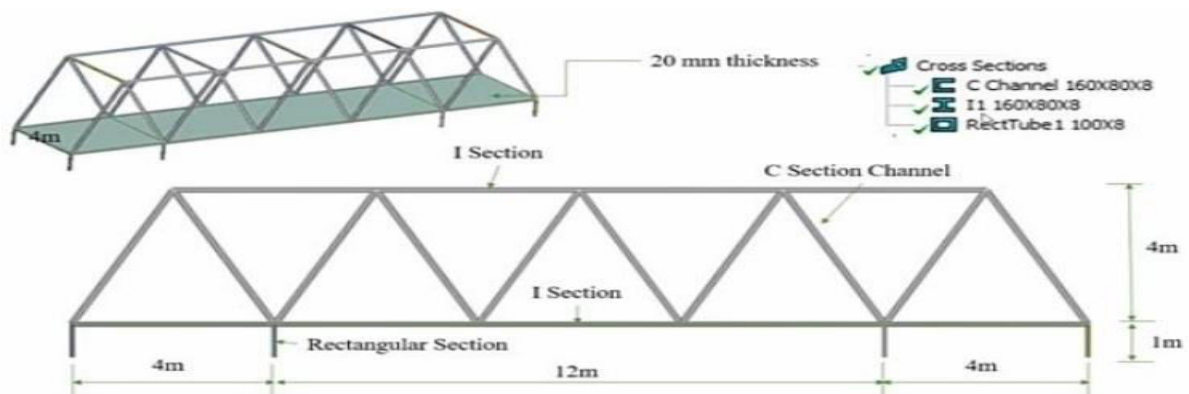
In this study, structure will be model for the following cases:

1. 20 m pedestrian bridge with connecting rods.
2. Analysis and conclusion of same structure by using static analysis with different widths for stresses.
3. Analysis and conclusion of same structure by using static analysis with different widths for displacements.
4. Analysis and conclusion of same structure by using thermal analysis with different widths for displacements.
5. Analysis and conclusion of same structure by using thermal analysis with different widths for stresses.
6. Analysis and conclusion of same structure by using frequency distribution.
7. Analysis and conclusion by taking various loads on structure.
8. Analysis and conclusion by taking various cross sections on Pedestrian Bridge.

V. PRELIMINARY DATA

Following are the data assumed for carrying out this study-

1. Length of bridge=20m.
2. Height of bridge= 5m.
3. Distance between bridge beams=4m.
4. Thickness of slab=20mm
5. Cross sections used:
 1. Channel[C] section 160x80x8
 2. I section [I1] 160x80x8
 3. Rectangle Tube 1 100x8
6. Width=4m.



A) SKETCH OF DESIGN PROBLEM



VI. CONCLUSION

As by reviewing literature it can conclude that different analysis and modellings have been done for Cable stayed and suspension bridges as used for various purposes because it is having long span lengths. The common practice until recently was to use simple 2D-models for static bridge design calculations. The 3D-model should catch the complex relation between the transfer of longitudinal and transversal load effects. Using pedestrian bridge with different loadings and cross sections is analyzed for the static and thermal criteria.

The potential of combining finite element (FE) analysis with on-site measurement through finite element model updating is indisputable. Linear and non-linear finite element method models were developed for a reinforced concrete bridge that had been strengthened with fibre reinforced polymer composites.

REFERENCES

- [1] Shruti G Kulkarni, Pradeep A R, Guruprasad T N “Static Analysis of RC Box Girder Bridge with skew angles using finite element method”, International Research Journal of Engineering and Technology, Vol .4 Issue 5 May 2017.
- [2] Kundan Kumar and Prabir Kumar Biswas “Structural Health Monitoring of Web Girder Bridge using Vibration Signal Analysis”, International Research Journal of Engineering and Technology, Vol .4 Issue 5 May 2017.
- [3] Yadav Bhola Chunkawan, Dr.R.Siva Subramaniam —Static Structural Analysis of Crane Hook”, International Research Journal of Engineering and Technology, Vol.04 Issue 7 July 2017.
- [4] Pravin Malvia, Farhan Vahora “Effect of cable arrangement of non-linear static analysis of cable stayed bridge”, International Journal of Engineering Research & Technology, Vol.4, Issue 4, Apr 2017.
- [5] Prasanna S V S N D L, Suresh Kumar “Simulation of flow Behaviour around bridge piers using ANSYS”, Vol.7, Issue 9 Sep 2018.
- [6] Hao Shuai, Qi Yuming “Static analysis of Concrete Pouring Robot Arm based on Ansys Workbench”, International Journal of Engineering Research & Technology, Vol.7 Issue 1 January 2018.
- [7] Mahendra Tirpathi, Ms Lavina Talawali —Static Analysis of Bridge Structure using Finite Element Analysis Software” International journal of Engineering research and application, Vol.8 Issue 6 June 2018.
- [8] Thazneema CN, Soumya Anand “Study of Static analysis on conventional and oblique prestressed concrete bridge superstructure”, International Research Journal of Engineering and Technology Vol.6 Issue 6 June 2019.
- [9] Shashikant Shrivastava “Skew Bridge Analysis using ANSYS” International Journal of Engineering Research & Technology, Vol.9 Issue 6 June 2020.
- [10] Premsai.T “Structural analysis and optimization of bridge pier using ANSYS” International Journal of Engineering sciences and research technology, Vol.4 March 2020.



**Impact Factor:
7.512**



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 **9940 572 462**  **6381 907 438**  **ijirset@gmail.com**



www.ijirset.com

Scan to save the contact details