STM JOURNALS

Trends in
Transport Engineering and Applications

ISSN: 2394-7284
Volume 12, Issue 2, 2025
May–August
DOI (Journal): 10.37591/TTEA

https://journals.stmjournals.com/ttea

Research TTEA

Static Structural Analysis and Bridge Modeling Using ANSYS

Amol Wankhade[1,*], R.S. Londhe[2]

Abstract

Structural analysis is a crucial discipline within civil engineering that ensures built structures are safe, durable, and capable of performing the functions for which they are intended. This scientific approach is essential for confirming that all safety requirements are met, thereby allowing structures to fulfill their purpose efficiently and economically over their design lifespan. The principles of structural analysis originate from the collaborative efforts of engineers across various domains, all sharing a common objective: to create structures that are not only functional but also aesthetically pleasing and cost-effective. At its core, structural analysis involves the application of mechanical theories and adherence to physical laws to predict and evaluate the behavior of structures under various loads and environmental conditions. The primary focus of this study is to develop comprehensive 2D and 3D models for the design and analysis of a pedestrian bridge spanning 60 m. The research involves conducting a static structural analysis for the bridge under different design configurations, including variations in width and cross-sectional geometry. The aim is to understand how different design parameters influence the bridge's performance, specifically in terms of stress distribution and displacement when subjected to static loads. In addition to static loading conditions, the study also evaluates the thermal effects on the bridge structure by analyzing how temperature variations impact its structural behavior. This dual approach: combining both static and thermal analysis, provides a more complete understanding of the bridge's performance under real-world conditions. The results of this comprehensive analysis will identify the most suitable bridge design by comparing different configurations based on stress concentration, displacement, and thermal resilience. The findings will ultimately contribute to optimizing pedestrian bridge design for improved safety, structural efficiency, and cost-effectiveness in future infrastructure projects.

Keywords: Pedestrian bridge, static structural analysis, thermal analysis, ANSYS, stresses, displacement

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

***Author for Correspondence**
Amol Wankhade
E-mail: amolwankhade76@gmail.com

[1]Student, Department of Applied Mechanics, Government College of Engineering, Aurangabad, Chhatrapati Sambhajinagar, Maharashtra, India
[2]Professor, Department of Applied Mechanics, Government College of Engineering, Aurangabad, Chhatrapati Sambhajinagar, Maharashtra, India

Received Date: March 24, 2025
Accepted Date: April 30, 2025
Published Date: May 04, 2025

Citation: Amol Wankhade, R.S. Londhe. Static Structural Analysis and Bridge Modeling Using ANSYS. Trends in Transport Engineering and Applications. 2025; 12(2): 27–32p.

design development. Any structure can be analyzed with static method or dynamic method. Selection of an appropriate analysis method depends on a number of factors [1–3].

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 [4–6].

METHODOLOGY

After reviewing and deep scrutiny of the literature available in the area of different bridge configurations published by various researchers in Indian as well as international journals, the gaps or problems which are not yet investigated are established and discussed as follows:

1. Most of the analysts utilized a cable cable-stayed and suspension bridges for investigation.
2. There was no study for the investigation of people on foot steel bridge with combination of different setups to analyze the results.
3. While conducting the research no practical perspective was considered for analysis of foot steel bridge using response spectrum analysis [7–9].
4. While executing the bridge design all the researchers studied Static analysis, explicit dynamics, Harmonic Response analysis with ANSYS 21.0 for stresses and displacement; but the researchers missed to carry out thermal analysis for the stresses arising due to thermal coefficients.
5. Similarly, research for Thermal analysis, Response spectrum, and Random Vibration analysis with ANSYS 21.0 for steel bridge using different thickness for deck sheets was missing.
6. Comparison results of each force and displacements and the percentage of difference with static and harmonic response was found missing.
7. In addition to all the parameters, the consideration of Bhuj venture in relation to the reaction range analysis is included in the investigation.
8. To make the structure safe and economical.

Based on the gaps in literature and unstudied problems that are discussed above, the Problem Formulation is prepared for clarifying and investigating the problems and deciding the approach and parameters for analysis. Three dimensional models are arranged utilizing exceptionally presumed Limited Components based program, such as ANSYS (version21.0). ANSYS is chosen since the program specifically planned for investigation of bridges for distinctive parameters. In addition, the computer program is able to perform the analysis and get the results in accordance with Indian Standard Code of practice [10, 11].

MODELING IN SOFTWARE

In this study, a static analysis of the bridge structure—including thermal effects, reaction range analysis, and other parameters—is carried out using ANSYS. The result of this investigation is utilized for considering different basic parameters such as Relocation, stress, increasing speed etc. Program utilized is ANSYS. The work has been carried out for the static investigation of structure. The analysis has been carried out for the Mumbai locale as indicated as per IS 1893-2016, IRC 006-2017. After obtaining parameters like stretch, uprooting, recurrence, number of cycles, thermal stress etc., ANSYS version 21.0 is used for modeling of bridge [12]. The essential exercises which have be carried out are:

1. Model generation (geometry).
2. Assignments of material, stiffness and loads.
3. Analysis (Static, thermal, random vibration, response spectrum).
4. Extraction and interpretation of results for useful findings.

In this, structure is modeled for the following cases:

1. *Model 1: Static structural analysis:* The first model is considered as static; in this model a pedestrian bridge of 60 m in length with 6 m in height is taken for analysis. In this model the value of stress and the amount of deflection experienced by the structure is obtained.

2. *Model 2: Transient thermal analysis:* The second model is considered as thermal model. In this model, the value of sum of coefficient of thermal extension is examined.
3. *Model 3: Harmonic response analysis:* The third model is a Consonant Reaction investigation of pedestrian foot over bridge with 60 m length. The height of bridge is 6 m and the distance between the bridge beams is 4 m. In this model, the graph showing frequency induced in the bridge is obtained.
4. *Model 4: Explicit dynamic analysis:* The fourth model is Explicit dynamic analysis with a pedestrian bridge length of 60 m. The distance between bridge beams is 4 m. In this model, the energy and momentum generated in the structure is summarized.
5. *Model 5: Random Vibration analysis:* The fifth model is Random vibration analysis with a pedestrian bridge length of 60 m. In this model, the PSD data and frequency relation graph results are obtained.
6. *Model 6: Response spectrum analysis:*
   a. This model is Response spectrum analysis with a pedestrian bridge length of 60 m. In this model the number of cycles per moment of frequency with respect to amplitude is opted. In the present research we have used different factors for the analysis like static analysis, thermal analysis, frequency distribution etc.
   b. Model the structure in ANSYS 21.0.
   c. Static analysis is done as per IRC-24-2001.
   d. Running the model in ANSYS 21.0 for static criteria.
   e. Obtaining parameters like stress, strain, displacement, frequency, thermal analysis etc.

STRUCTURAL MODELLING

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

1. Pedestrian bridge with connecting rods.
2. Analysis and conclusion by using static analysis with different widths for stresses.
3. Analysis and conclusion 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 frequency distribution.
6. Analysis and conclusion by taking various loads on structure.
7. Analysis and conclusion by taking various cross sections on Pedestrian Bridge (Table 1 and Figure 1).

Response Spectrum Analysis

Sine vibration testing is performed when it has been given as it is one recurrence at given time moment. Reaction range examination could be a strategy to gauge the basic reaction to brief, nondeterministic, temporal energetic occasions (Figures 2 and 3).

Table 1. Preliminary data.

S.N.	Description	Value
01.	Length of bridge	60 m
02.	Height of bridge	6 m
03.	Distance between bridge beams	4 m
04.	Thickness of slab	80 mm
05.	*Cross sections used:*	
	1. Channel section	160×80×8 mm
	2. I section	160×80×8 mm
	3. Rectangular tube 1	100×8 mm
06.	Width	4 m
07.	Grade of steel	Fe250
08.	Type of support	Roller

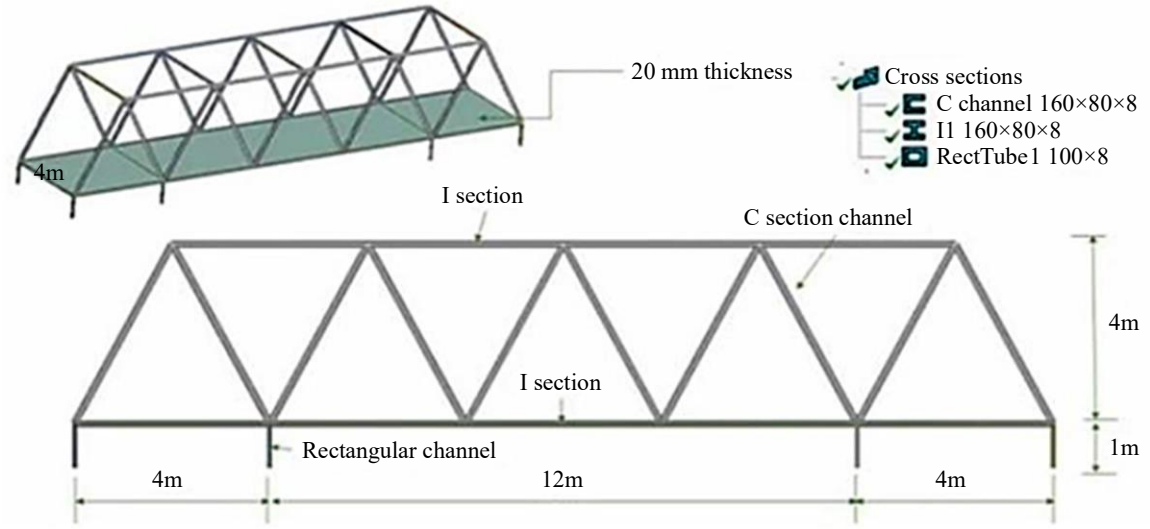

Figure 1. Sketch of design problem.

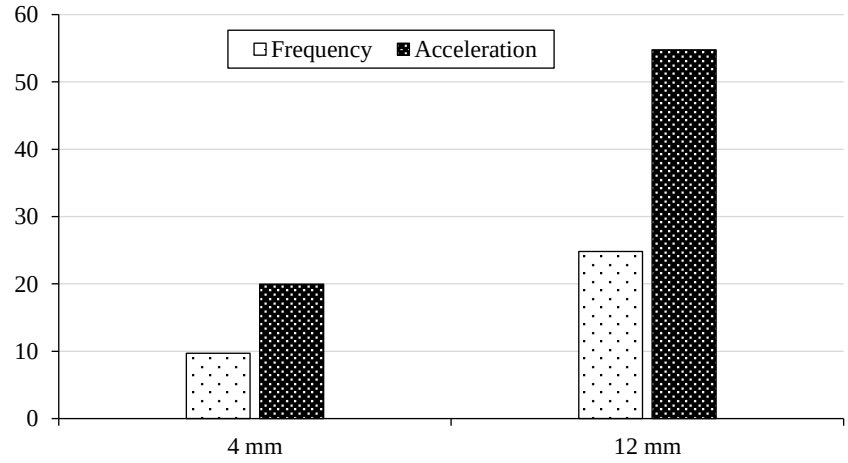

Figure 2. Response spectrum analysis of frequency and acceleration.

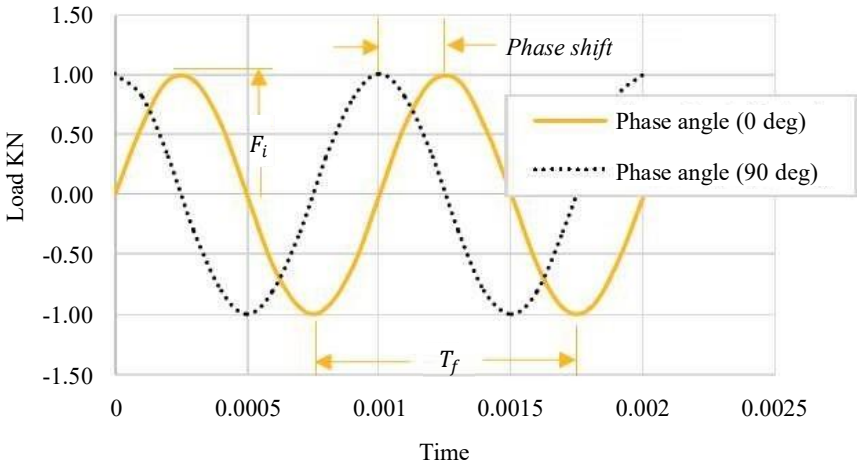

Figure 3. Load vs. time for phase angle 2.

Displacement, Velocity and Acceleration

Displacement, velocity, and acceleration describe an object's position change, rate of motion, and how that rate changes over time.

Table 2. Static analysis comparison between force and support moments (upper member of bridge).

S.N.	Member name	Forces (N)	Support moments (N.mm)
1	U1	425.98	320.1
2	U2	424.6	321.2
3	U3	428.7	325.3
4	U4	430.45	325.45
5	U5	431	328
6	U6	433.6	330.6
7	U7	435.9	332.8
8	U8	437.2	334.9
9	U9	440.3	336.3
10	U10	442.7	339.45

Table 3. Static analysis comparison between force and support moments (lower member of bridge).

S.N.	Member name	Forces (N)	Support moments (N.mm)
1	L1	505.23	420.65
2	L2	520.63	422.9
3	L3	523.9	424.1
4	L4	526.1	426.7
5	L5	528.3	428.3
6	L6	531.7	430.9
7	L7	533.6	433.13
8	L8	537.9	436.03
9	L9	540.31	439.63
10	L10	542	441.3
11	L11	543.21	443

RESULTS AND DISCUSSION

Static Analysis Comparison Between Force and Support Moments for Upper Members of Bridge Section

The upper members mainly experienced compressive forces, with maximum values near the mid-span. Support moments were highest at the fixed ends due to rotational restraints. The force and moment distribution was symmetrical, indicating stable structural behavior (Table 2).

Static Analysis Comparison Between Force and Support Moments for Lower Members of Bridge Section

The lower members were primarily under tension, increasing toward the center of the span. Support moments were lower compared to the upper members, reflecting their role in vertical load support. Overall, the lower members effectively balanced the structural system under static loads (Table 3).

CONCLUSION

1. From the model analysis it is found that model 2 bridge suffers less deformation than model 1 bridge system.
2. The equivalent stresses formed in both the bridges are observed to be within permissible limit.
3. But it was also noted that the results of model 1 and 2 have a slight difference for all the IRC load parameters, looking from the practical point of view, combination of response spectrum and vibration can be used.
4. The selection of updating parameter made by ANSYS tool present satisfactory results.

5. If adopting response spectra analysis for design purpose as compared to harmonic vibration and explicit dynamics, the frequency reduces up to marginally and amplitude will go on higher ranger of values. Thus, adopting structure with response spectra will be more effective.

REFERENCES

1. Kulkarni Shruti G, Pradeep AR, Guruprasad TN. Static Analysis of RC Box Girder Bridge with Skew Angles Using Finite Element Method. Int Res J Eng Technol. 2017 May; 4(5): 2711–2714.
2. Kundan Kumar, Nawaz Ali, Prabir Kumar Biswas. Structural Health Monitoring of Web Girder Bridge Using Vibration Signal Analysis. Int Res J Eng Technol. 2017 May; 4(5): 1422–1427.
3. Yadav Bhola Chunkawan, Siva Subramaniyam R. Static Structural Analysis of Crane Hook. Int Res J Eng Technol. 2017 Jul; 04(7): 2265–2274. e-ISSN: 2395-0056.
4. Herakal SB, Goud C. Structural static analysis of connecting rod. Anveshana's Int J Res Eng Appl Sci. 2016 Mar;1(3):22–6.
5. Pravin Malvia, Farhan Vahora. Effect of Cable Arrangement of Non-linear Static Analysis of Cable Stayed Bridge. Int J Eng Res Technol. 2017 Apr; 4(4): 1725–1740.
6. Ghodke PY, Tupe DH, Rama Rao MV. Modal Analysis of Cracked Continuous Beam Using ANSYS. Int J Eng Res Technol. 2010 Feb; 4(2): 86–93.
7. Prasanna SVSNDL, Suresh Kumar N. Simulation of Flow Behavior Around Bridge Piers Using ANSYS. International Journal of Engineering Science Invention (IJESI). 2018 Sep; 7(9): 13–22.
8. Hao Shuai, Qi Yuming, Hendrik Schlune. Static Analysis of Concrete Pouring Robot Arm Based on Ansys Workbench. Int J Eng Res Technol. 2009 Jan; 7(1): 43–52.
9. Mahendra Tirpathi, Lavina Talawali. Static Analysis of Bridge Structure Using Finite Element Analysis Software. Int J Eng Res Appl. 2018 Jun; 8(6): 70–73.
10. Najad SN, Sudhakar A, Nair SS. Static and dynamic analysis of wind energy building under the action of wind load using ANSYS. Int Res J Eng Technol. 2019 Oct;6(10):150–6.
11. Rajesh Chaturvedi. Analysis of Cable Stayed Bridge for Different Structural Model. Int J Eng Res Technol. 2019 Jul; 8(7): 751–761.
12. Thazneema CN, Soumya Anand. Study of Static Analysis on Conventional and Oblique Prestressed Concrete Bridge Superstructure. Int Res J Eng Technol. 2019 Jun; 6(6): 2830–2836.

