RESEARCHERID
THOMSON REUTERS
[Baghel et al., 11(5), May 2026]

ISSN : 2455-9679
SJIF Impact Factor : 6.190

ijrtsm

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"FINITE ELEMENT ANALYSIS AND STRUCTURAL PERFORMANCE EVALUATION OF RCC T-BEAM AND BOX GIRDER BRIDGES: A COMPREHENSIVE REVIEW"

Sundar Lal Baghel [1], Prof. Satyendra Dubey [2], Anubhav Rai [3]

[1]M-Tech Student Structural Engg. Dept. of Civil Engg, Gyan Ganga Inst. of Tech. & Sciences, Jabalpur M.P. India
[2]Associate Professor, Dept. of Civil Engineering, Gyan Ganga Inst. of Tech. & Sciences, Jabalpur M.P. India
[3]Associate Professor, Dept. of Civil Engineering, Gyan Ganga Inst. of Tech. & Sciences, Jabalpur M.P. India

ABSTRACT

Bridge structures are essential components of transportation infrastructure and require safe, economical, and durable design for long-term serviceability. With the advancement of computational techniques, Finite Element Analysis (FEA) has become a powerful tool for analyzing and designing bridge structures under various loading conditions. This review paper presents a comprehensive study on the structural behavior, analysis methods, and performance evaluation of reinforced cement concrete (RCC) T-beam and box girder bridges. The paper discusses the structural components of bridges, including superstructure and substructure elements, along with the significance of I-section beams in bridge engineering. Various analytical and numerical approaches adopted by researchers using software such as ANSYS, STAAD Pro, SAP2000, and CSI Bridge are reviewed.

Keywords: *Bridge Engineering, Finite Element Analysis, RCC T-Beam Bridge, Box Girder Bridge, ANSYS, STAAD Pro, Structural Analysis.*

I. INTRODUCTION

Bridges are among the most important civil engineering structures used for transportation and connectivity. Modern bridge engineering requires detailed analysis of stresses, strains, deformation, and vibration characteristics to ensure safety under static and dynamic loading conditions. Primary modal analysis plays a pivotal role in evaluating and understanding the dynamic behavior of bridge structures. As essential components of infrastructure, bridges are subjected to a variety of dynamic loads, including vehicular traffic, wind forces, seismic events, and temperature fluctuations. These factors make the dynamic response of bridges a critical concern for structural engineers, as any deficiencies in the design or construction can lead to performance issues or even catastrophic failures. Modal analysis specifically focuses on determining key dynamic characteristics, such as natural frequencies, mode shapes, and damping ratios. These parameters are fundamental in assessing a bridge's overall performance, reliability, and safety, particularly under dynamic loading conditions. In the case of long-span bridges, modal analysis becomes even more critical. Vibrations in these structures can significantly impact their structural integrity, especially in the presence of factors like wind or traffic-induced oscillations. Understanding how a bridge behaves dynamically allows engineers to design structures that can withstand these forces while ensuring safety, durability, and comfort for users. With advances in computational methods, traditional experimental modal analysis has been supplemented or even replaced by computational modal analysis techniques. These computational methods leverage numerical simulations based on finite element analysis (FEA), enabling engineers to model the behavior of bridges under dynamic conditions with greater precision and efficiency. Computational modal analysis offers several advantages over physical testing, including the

ISSN : 2455-9679
SJIF Impact Factor : 6.190

ability to simulate complex bridge geometries, accurately represent diverse material properties, and consider varying loading conditions. This shift towards computational analysis enhances the accuracy of predictions and allows for more cost-effective and time-efficient evaluations compared to experimental methods.

The primary objective of this study is to perform a detailed structural modal analysis of a bridge using computational techniques. By employing modern computational tools, such as finite element software, the research will focus on simulating and analyzing the dynamic response of a selected bridge structure. These computational tools allow for a more thorough understanding of how the bridge will perform under various dynamic loads, helping engineers identify potential vulnerabilities that may affect the structure's stability or longevity. The results of the computational modal analysis will be used to predict the structural performance of the bridge under real-world conditions. This includes identifying natural frequencies that may resonate with external forces, analyzing mode shapes that indicate how different parts of the structure will deform, and calculating damping ratios that determine the rate at which vibrations dissipate. Additionally, the findings will provide insights into possible design improvements and guide decision-making in optimizing the bridge's performance. By using advanced computational techniques, this study will contribute to the development of more robust, efficient, and resilient bridge designs capable of withstanding the diverse dynamic challenges encountered throughout their service life.

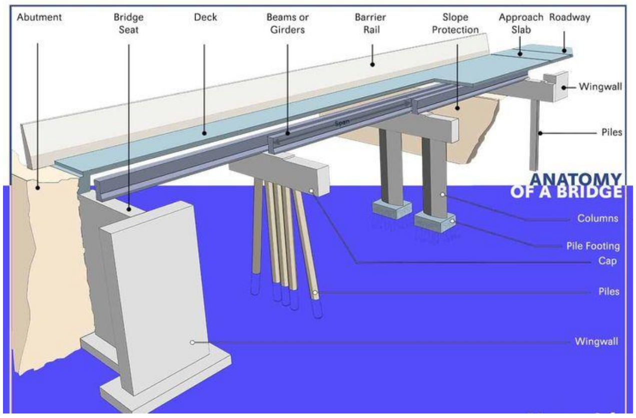

Fig.1 The Many Parts of a Bridge (The Many Parts of a Bridge (2 Illustrated Diagrams) - Homenish

II. Structural Components of Bridges

Bridge structures mainly consist of superstructure and substructure components. The superstructure directly carries traffic loads, while the substructure transfers loads to the foundation. Bridges are important civil engineering structures used to provide safe transportation across obstacles such as rivers, valleys, and roads. The structural components of a bridge are mainly divided into two parts: superstructure and substructure. The superstructure is the upper portion of the bridge that directly carries traffic loads. It includes deck slabs, girders, beams, bearings, parapets, and expansion joints. The deck slab provides the running surface for vehicles, while girders and beams transfer loads from the deck to the supports. Bearings are provided to transfer loads safely and allow thermal expansion and contraction.

The substructure is the lower portion of the bridge that supports the superstructure and transfers loads to the ground. It consists of piers, abutments, wing walls, and foundations. Piers are intermediate supports between bridge spans, whereas abutments are provided at the ends of the bridge to retain soil and support the deck. Foundations transfer the entire bridge load safely to the soil.

Other important components include wearing coats, drainage systems, railings, and approach slabs. Each component plays a vital role in ensuring structural stability, safety, durability, and smooth transportation. Proper design and analysis of bridge components are essential for achieving long service life and efficient performance under various loading conditions.

ISSN : 2455-9679
SJIF Impact Factor : 6.190

III. Finite Element Method (FEM)

The Finite Element Method is a numerical technique used to solve complex structural engineering problems by dividing the structure into smaller finite elements. ANSYS, STAAD Pro, SAP2000, and CSI Bridge are widely used software tools for bridge analysis and design. The Finite Element Method (FEM) is a numerical analysis technique widely used in engineering for solving complex structural, thermal, and fluid flow problems. It is especially useful for analyzing structures with complicated geometry, loading conditions, and material properties where conventional analytical methods become difficult. In FEM, a large structure is divided into smaller and simpler parts called finite elements, which are connected through nodes. The behavior of each element is mathematically represented using equations, and all element equations are assembled to form a complete structural model.

The method helps engineers determine important parameters such as stress, strain, displacement, temperature distribution, and vibration characteristics of structures. FEM is extensively used in bridge engineering, machine design, aerospace, automotive, and civil engineering applications. Software such as ANSYS, ABAQUS, STAAD Pro, and SAP2000 are commonly used for finite element analysis.

The major steps involved in FEM include discretization of the structure, selection of element type, application of material properties, boundary conditions, loading, solution of equations, and result interpretation. FEM provides highly accurate and reliable results, reduces experimental cost, and saves design time. Due to its flexibility and precision, FEM has become one of the most important tools for modern engineering analysis and design.

IV. Literature Review

Sandesh Upadhyaya K (2016) compared deck slab systems of varying span lengths (20m, 24m, and 28m) using conventional design methods. The analysis focused on shear forces and bending moments, with live loads assigned for Class AA wheeled vehicles. The results from the finite element method (FEM) analysis were compared with manual methods, concluding that T-beam slabs are more efficient than ordinary slabs on girders.

Tangudupalli (2017) compared various loading conditions and methods for bridge analysis using STAAD Pro V8i software. Girder analysis was conducted using three rational methods (Hendry Jaegar, Guyon-Massonet, and Courbon's theory) under I.R.C. loadings (Class A, Class AA, Class 70-R, Class B) and international load standards from Saudi Arabia, AASHTO, and British Standards.

Abrar Ahmed (2017) aimed to identify the most suitable section for bridges with varying spans. The analysis, performed using Csibridge software, focused on different sections under various I.R.C. vehicle loadings. The study, validated using the working stress method and Courbon's theory, found that I.R.C. 70-R vehicles had the most significant impact. It also concluded that while T-beam girders are suitable for spans up to 30 meters, box girders are more economical for longer spans.

Manohar R (2018) conducted a study analyzing a T-beam bridge with varying slab dimensions of 3×2, 3.5×2.5, 4×3, 4.5×3.5, and 5×4, and deck depths of 200, 225, 250, 275, and 300 mm using SAP 2000 software. The study focuses on the key bridge components, including the deck slab, cross girders, and longitudinal girders. Various manual methods were also employed for analysis, revealing that the shear force, bending moment, and deflection in the girder increase as the span length increases.

Several researchers have studied the behavior of RCC T-beam and box girder bridges using finite element methods and experimental investigations.

V. RESEARCH GAP

Limited studies are available on optimized beam geometries, curvature effects, and advanced composite materials in bridge engineering. Many researches worked on T-RCC beam by using strip insert in beam, different material used in beam. But no body used different geometry of beam. Here my research curvature effect provides in beam modeling.

ISSN : 2455-9679
SJIF Impact Factor : 6.190

VI. Conclusion

Finite element analysis provides accurate prediction of stresses, deformation, and structural behavior under different loading conditions. The findings help engineers design safer and more economical bridge structures. The major steps involved in FEM include discretization of the structure, selection of element type, application of material properties, boundary conditions, loading, solution of equations, and result interpretation. FEM provides highly accurate and reliable results, reduces experimental cost, and saves design time. Due to its flexibility and precision, FEM has become one of the most important tools for modern engineering analysis and design.

References

[1] R. Manohar, B. Suresh Chandra, "Finite Element Analysis of slabs, cross girders and main girders in RC T-Beam Deck Slab Bridge", International Research Journal of Engineering and Technology (IRJET), Vol 5, Issue 08, 2018.

[2] Abrar Ahmed, Prof. R.B. Lokhande, "Comparative Analysis and Design of T-beam and box girders", International Research Journal of Engineering and Technology (IRJET),Vol 4, July 2017

[3] K. Sandesh Upadhyaya and F. Sahaya Sachin, "A Comparative Study of T- Beam Bridges for varying span lenths", International Journal of Research in Engineering Technology, Vol 5, Jun 2016.

[4] Mahesh Kumar Tangupalli and J Sudhamani, "Analysis of T-Beam Deck Slab Bridge in Different Methods",International Journal for Technological Research in Engineering, (IJTRE), Vol 4, Issue 12,Aug 2017.

[5] M. G. Kalyanshetti and R. P. Shriam, "Study of Effectiveness of Courbons Theory in the Analysis of T Beam Bridges", International Journal of Science and Engineering Research, (IJSER), Vol.4, Issue 3, 2013.

[6] Amit Saxena and Dr. Savita Maru, "Comparative Study of the Analysis and Design of T-beam and box girder Superstructure", International Journal of Research in Engineering and Advanced Technology (IJREAT), Vol 1, Issue 2, April-May, 2013.

[7] S. Soumya and R. Umadevi, "Comparative Study of Courbons Method and Finite Element Method of RC T-Beam and Deck Slab Bridge", International Journal of Engineering and Management Research, (IJEMR) Vol 5 Issue 6, Dec 2015.

[8] N. K. Praful and Balaso Hanumat, "Comparative Analysis of T Beam Bridge by Rational Methodland STAAD Pro", International Journal of Engineering Science & Research Technology (IJESRT), June 2015

[9] Y. Yadu Priya and T. Sujatha , "Comparative Analysis of Post Tensioned T BeamBridge Deck by Rational Method and Finite Element Method", International Journal of Research in IT, Management and Engineering, Vol 6, Issuue 7, Sept 2016.