Dynamic Response and Geometry Optimization of EOT Crane Girder under Moving Loads

P. Sanjay kumar[1], T. Vinay[2], R. Ram Saketh[3], K. Archana[4] V. Venkatadri Naidu[5]
K. Srirama Kumar[6]
[1,2,3,4,5]*Students, Avanthi Institute of Engineering and Technology*
[6]*Assistant Professor, Avanthi Institute of Engineering and Technology*

Abstract—**This study focuses on analysing the dynamic response of beams and frames subjected to moving point loads, particularly in EOT crane girders used in industrial applications. These girders experience complex vibrations due to moving crane trolleys and lifted loads. The Finite Element Method (FEM) is used to model the structure accurately, considering material properties, boundary conditions, and load movement. Numerical time integration methods are applied to capture time-dependent structural responses. The study evaluates displacement, stress distribution, and oscillatory behaviour under real operating conditions. It also examines the effect of different load velocities on structural performance and stability. A key parameter analysed is the Dynamic Magnification Factor (DMF), which compares dynamic and static displacements. Higher load speeds are found to increase vibration amplitudes and risk of resonance. Natural frequency analysis is conducted to understand and control these vibrations. Additionally, geometry optimization is performed to enhance strength, reduce weight, and improve overall structural reliability.**

Index Terms—**Dynamic Response, Dynamic Magnification Factor (DMF), EOT Crane Girder, Finite Element Method (FEM), Geometry Optimization, Moving Load, Natural Frequency, Vibration Analysis.**

I. INTRODUCTION

Electric Overhead Travelling (EOT) cranes play a vital role in modern industrial material handling systems by enabling efficient lifting and transportation of heavy loads over large spans. These cranes are widely used in manufacturing plants, shipyards, construction sites, and warehouses due to their high load-carrying capacity and operational flexibility. With the increasing demand for higher productivity, cranes are required to operate at higher speeds and handle heavier loads, which introduces significant dynamic effects on structural components.

The girder is the primary structural element of an EOT crane, responsible for supporting the trolley, hoisting mechanism, and payload. During operation, the girder is subjected to moving loads generated by the trolley and lifted material. These moving loads induce vibrations, deflections, and varying stress distributions, which may affect the structural performance and safety of the crane. Traditional design approaches often consider only static loading conditions, which may not accurately represent real operating conditions.

To address these challenges, dynamic analysis is essential to evaluate the time-dependent behavior of crane girders under moving loads. The Finite Element Method (FEM) is widely used to analyze such complex structural systems with high accuracy. Parameters such as displacement, stress, strain, and natural frequencies are evaluated to understand the structural response. The Dynamic Magnification Factor (DMF) is also considered to compare static and dynamic responses and to assess the influence of moving loads on structural performance.

Material selection plays a crucial role in determining the strength, stiffness, and dynamic behavior of crane girders. Commonly used materials such as structural steel, Carbon Steel 1020, and 34CrMo4 are considered due to their favorable mechanical properties, including high strength, good ductility, and adequate stiffness. Each material exhibits different characteristics in terms of density, modulus of elasticity, and yield

strength, which significantly influence deflection, vibration response, and natural frequencies. Evaluating different materials helps in identifying the most suitable option for improved structural performance and durability.
In addition to material properties, beam geometry and cross-sectional dimensions significantly influence structural behavior. Variations in girder depth and cross-section affect stiffness and vibration characteristics, and higher load speeds may lead to resonance conditions, increasing the risk of structural failure. Therefore, optimization of girder geometry is essential to improve performance, reduce deflection, and enhance stability.

The objective of this study is to analyze the structural and dynamic behavior of EOT crane girders under moving load conditions using numerical simulation techniques. The study focuses on evaluating displacement, stress, strain, natural frequencies, and mode shapes for different materials and girder geometries. Furthermore, modal and dynamic analyses are performed to avoid resonance conditions and ensure safe crane operation while improving efficiency and reliability.

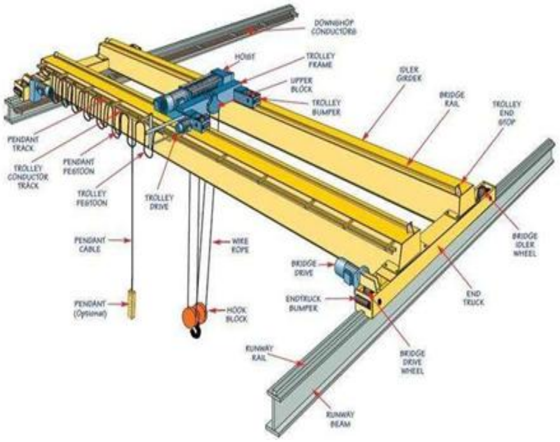

Fig 1.1: EOT crane structure

II. LITERATURE REVIEW

Several researchers have studied the structural and dynamic behavior of overhead cranes using analytical and numerical methods, with particular focus on stress distribution, deformation, and vibration characteristics.

Camelia Bretotean Pinca et al. (2009) applied Finite Element Analysis (FEA) to analyze the resistance structure of an overhead crane bridge. The study evaluated stress distribution using von Mises theory and identified critical regions prone to high stress concentrations. Their work demonstrated that proper stress evaluation helps improve the strength and load-bearing capacity of crane structures.

Ismail Gerdemeli et al. (2010) developed an advanced FEA technique for modeling crane structures. Their study included stress, deformation, and buckling analysis of crane girders and supporting legs using ANSYS. The results showed that numerical analysis provides accurate predictions comparable to theoretical calculations, making FEA a reliable tool for crane design.

C. Alkin et al. (2005) performed finite element modeling of an overhead crane bridge using both solid and shell elements. Their comparison revealed that quadratic shell elements provide more realistic results for stress and deformation analysis. The study also proposed optimization techniques for improving crane girder design.

Vlada Gasic et al. (2010) investigated the dynamic response of bridge cranes under moving load conditions. The moving load was modeled as a time-dependent force, and the study evaluated natural frequencies and dynamic deflections. The results highlighted that standard finite element packages can effectively simulate time-varying loads and predict structural response accurately.

M. Euler and U. Kuhlmann (2011) studied fatigue behavior in crane runway structures using finite element methods. Their work focused on multi-axial fatigue caused by repeated wheel loads and emphasized the importance of fatigue analysis in extending the service life of crane systems.

M. R. Wakchaure et al. (2012) analyzed the behavior of castellated steel beams using FEA. The study showed that beam geometry, especially depth and web openings, significantly affects deflection and stiffness. It was concluded that increasing beam depth improves performance but must be optimized to avoid excessive flexibility.

Recent studies also emphasize the importance of moving load analysis in crane dynamics. The dynamic response of crane structures depends on factors such as load magnitude, speed, material properties, and beam geometry. Due to the difficulty and high cost of experimental testing on full-scale cranes, numerical simulations using FEA have become the primary approach for analyzing crane behavior under realistic operating conditions.

Furthermore, material selection plays a crucial role in crane performance. Common materials such as structural steel, Carbon Steel 1020, and 34CrMo4 have been widely studied for their mechanical properties. Variations in material properties significantly influence deflection, stress distribution, and vibration characteristics, making material optimization essential for efficient crane design.

From the above studies, it is evident that significant research has been carried out on structural analysis, material selection, and dynamic behavior of crane systems. However, limited work has focused on the combined effect of moving loads, material variation, and girder geometry optimization. Therefore, this study aims to address this gap by performing comprehensive structural, modal, and dynamic analysis of EOT crane girders using the Finite Element Method.

III. METHODOLOGY

The methodology adopted in this study involves modeling, analysis, and evaluation of the structural and dynamic behavior of an EOT crane girder under moving load conditions using numerical simulation techniques. Initially, the dimensions of the EOT crane girder are defined based on design specifications, including girder length, cross-sectional properties, and loading parameters such as trolley load, payload, and movement path. The total length of the crane girder considered in this study is 40 m, and the cross-section is square with an initial depth of 1.78 m and thickness of 10 mm. To study the effect of geometry on structural performance, modified models are developed by increasing the depth by 5% (1.87 m) and 10% (1.95 m) while maintaining other parameters constant.

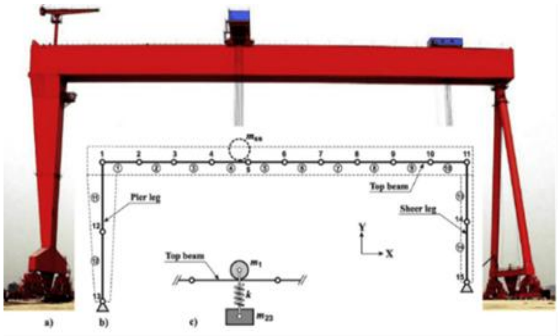

Fig 3.1: EOT Crane Girder Model & Dimensions

A 3D model of the crane girder is developed using CATIA software by defining the beam profile and extruding it along the required span length. Different geometric configurations are created by varying the cross-sectional depth while keeping other parameters constant. The CAD models are then imported into ANSYS Workbench for Finite Element Analysis (FEA), where the geometry is discretized into finite elements using appropriate meshing techniques. Fine meshing is applied in critical regions to capture stress concentration and deformation accurately.

The analysis is carried out using three different materials, namely Structural Steel, Carbon Steel 1020, and 34CrMo4, to evaluate their influence on structural and dynamic performance. To systematically study the combined effect of geometry and material, each girder model is tested with all three materials, resulting in multiple model-material combinations as shown below.

Table 3.1: Model–Material Combinations for Analysis

Name	Model No	Material
M1_MO1	Model 1	structural steel
M1_MO2	Model 1	carbon steel 1020
M1_MO3	Model 1	34 CrMo4
M2_MO1	Model 2	structural steel
M2_MO2	Model 2	carbon steel 1020
M2_MO3	Model 2	34 CrMo4
M3_MO1	Model 3	structural steel
M3_MO2	Model 3	carbon steel 1020
M3_MO3	Model 3	34 CrMo4

From the above combinations, it is possible to compare the performance of each material for different girder geometries. This approach helps in identifying the most suitable material and geometric configuration that provides minimum deformation, reduced stress, and improved dynamic characteristics.

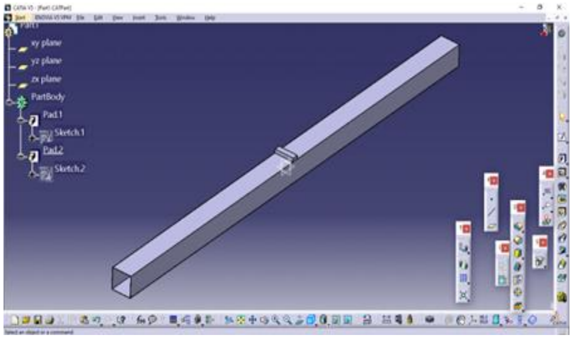

Fig 3.2: 3D view of original model

The girder is modeled as a beam supported at both ends with a moving load of 10 tons representing the trolley and payload traveling along the beam at different speeds. Boundary conditions are applied to simulate real operating conditions. Structural analysis is performed to evaluate total deformation, equivalent stress, and strain, while modal analysis is carried out to determine natural frequencies and mode shapes. Dynamic analysis is conducted to study the time-dependent response under moving loads, and the Dynamic Magnification Factor (DMF) is calculated. Finally, results obtained from different materials and geometric configurations are compared to identify the optimal design with improved structural performance and safety.

IV. FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is carried out using ANSYS Workbench to evaluate the structural and dynamic behavior of the EOT crane girder under moving load conditions. The CAD models developed in CATIA are imported into ANSYS, where the geometry is discretized into finite elements. Appropriate meshing techniques are applied to ensure accurate representation of the structure and to capture stress variations effectively.

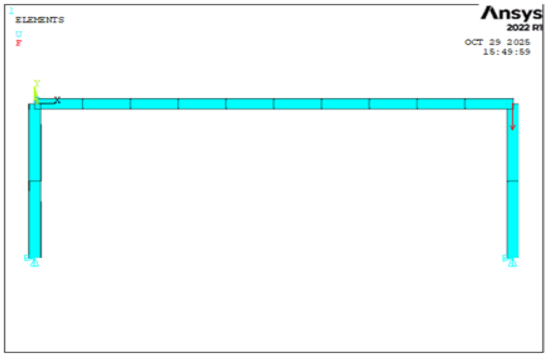

Fig 4.1: Finite element mesh model of EOT crane girder

The mesh consists of nodes and elements that represent the geometry accurately. A finer mesh improves the accuracy of results and captures stress variations effectively.

Boundary Conditions and Loading

The crane girder is subjected to realistic working conditions. Both ends of the beam are supported. A moving load representing trolley and payload is applied

- Load magnitude: 10 tons
- Speeds considered:
  - 2 m/s
  - 4 m/s
  - 6 m/s

These conditions simulate actual crane operation and help in analyzing the dynamic behavior of the structure under varying speeds.

Structural Analysis

Structural analysis is carried out to determine deformation, stress, and strain distribution in the crane girder.

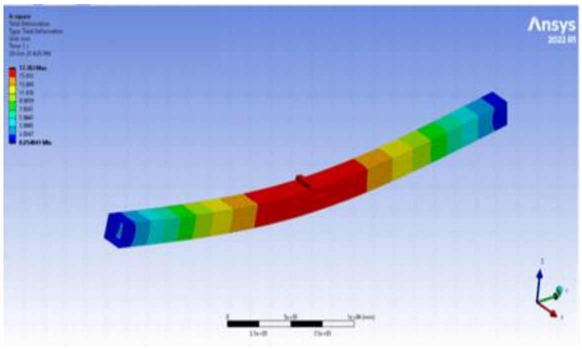

Fig 4.2: Total Deformation of Crane Grider

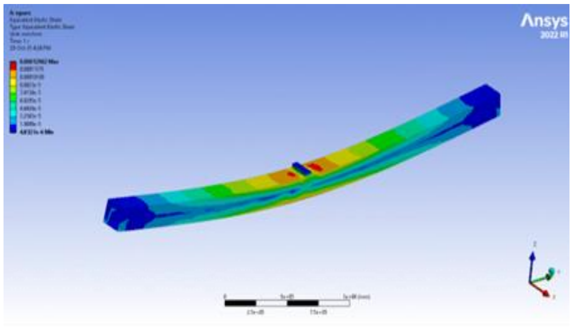

Fig 4.3: Equivalent stress distribution of crane girder

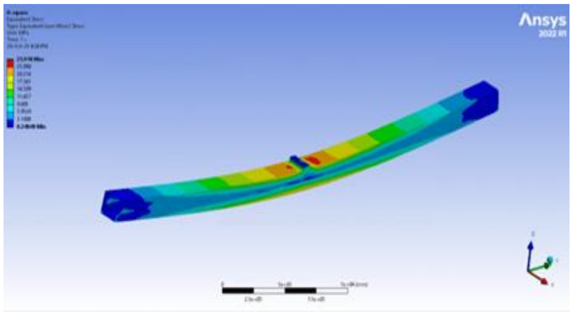

Fig 4.4: Strain distribution of crane girder

The results indicate that maximum deformation occurs at the mid-span of the beam due to bending effects. The stress distribution shows higher concentration near the load application region, which is critical for design consideration. The strain follows a similar pattern, confirming the structural response under applied loading conditions. The structural behavior varies depending on the material properties and geometric configuration of the girder.

Dynamic Analysis (Moving Load Analysis):

Dynamic analysis is performed to evaluate the behavior of the crane girder under moving loads at different speeds.

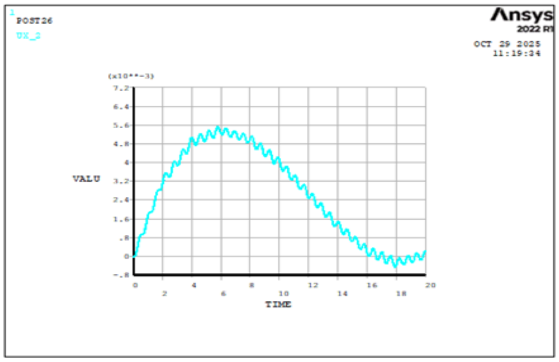

Fig 4.5: Horizontal displacement at 2 m/s

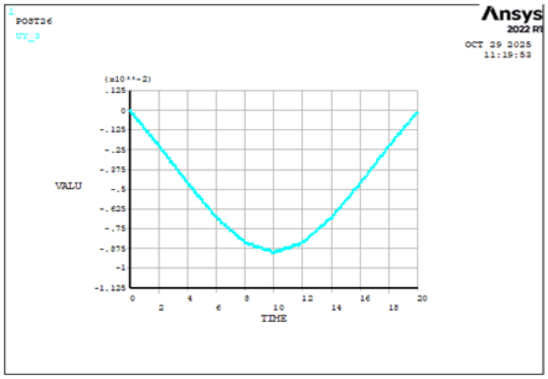

Fig 4.6: Vertical displacement at 2 m/s

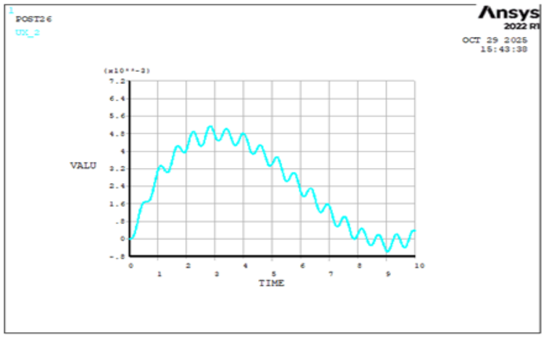

Fig 4.7: Horizontal displacement at 4 m/s

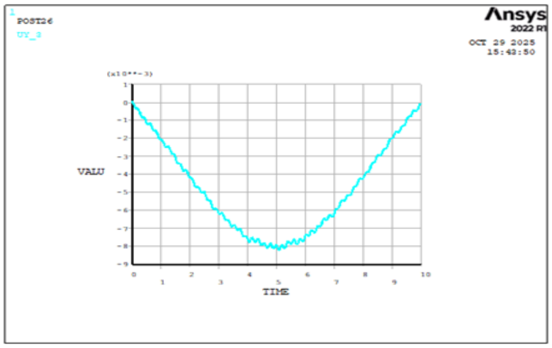

Fig 4.7: Vertical displacement at 4 m/s

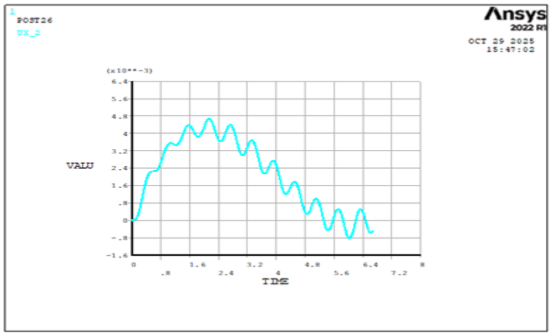

Fig 4.8: Horizontal displacement at 6 m/s

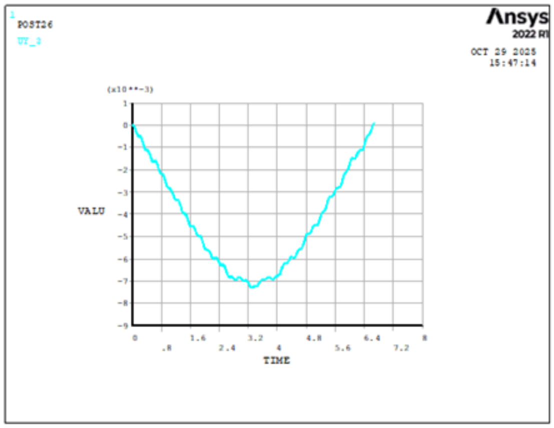

Fig 4.9: Vertical displacement at 6 m/s

The analysis shows that displacement and vibration increase significantly with the increase in load speed. At lower speeds (2 m/s), the response is relatively stable, whereas at higher speeds (6 m/s), the structure exhibits larger oscillations and deflections. This indicates that dynamic effects play a crucial role in crane performance and must be considered during design.

The Dynamic Magnification Factor (DMF) is used to compare dynamic displacement with static displacement. Higher DMF values indicate increased dynamic amplification, which may affect structural safety and service life.

Modal Analysis:
Modal analysis is performed to determine natural frequencies and mode shapes of the crane girder.

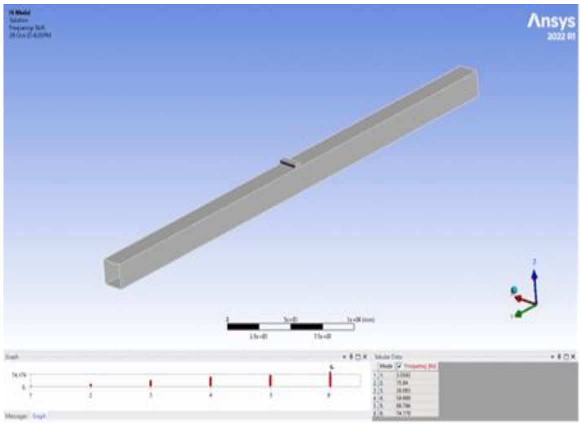

Fig 4.10: Natural frequency of crane girder

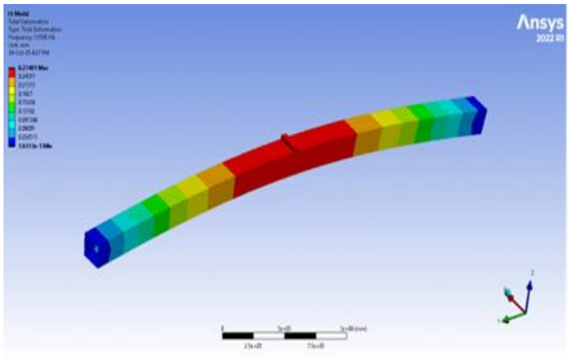

Fig 4.11: First mode shape

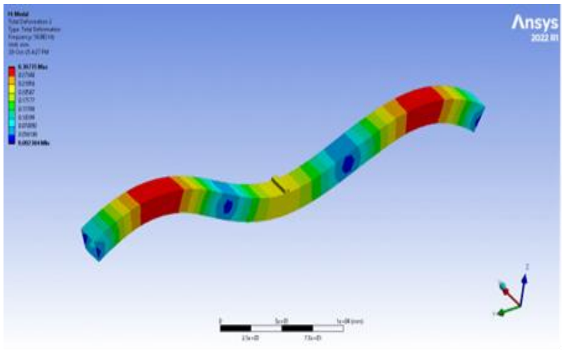

Fig 4.12: Second mode shape

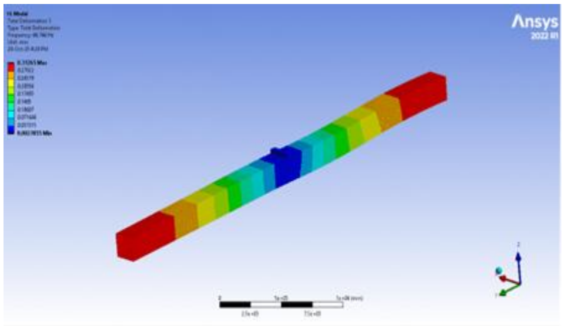

Fig 4.13: Third mode shape

The modal analysis shows that natural frequencies depend on the stiffness and mass distribution of the girder. Higher modes exhibit more complex deformation patterns. It is essential to ensure that the operating frequency of the crane does not coincide with the natural frequency to avoid resonance conditions, which may lead to structural failure.

V. RESULTS AND DISCUSSION

The results obtained from Finite Element Analysis (FEA) are presented and discussed in this section. The analysis includes structural, dynamic, and modal evaluations of the EOT crane girder under moving load conditions. The performance of different models and materials is compared based on deformation, stress, strain, and natural frequency.

Deformation Analysis:

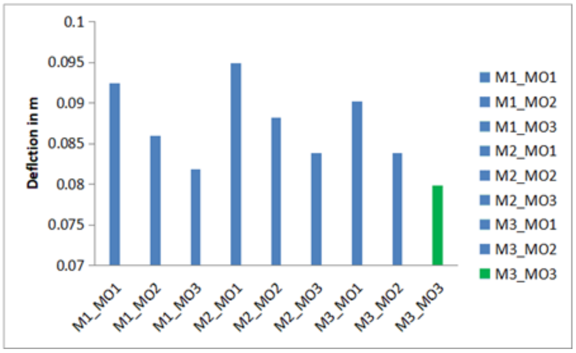

Fig 4.14: Deflection of different models

The deformation results indicate that Model 1 shows higher deflection compared to Models 2 and 3. As the depth of the girder increases, stiffness improves, resulting in reduced deformation. Among all models, Model 3 exhibits the least deformation, indicating better structural performance

Stress Analysis

Stress distribution is analyzed to identify critical regions.

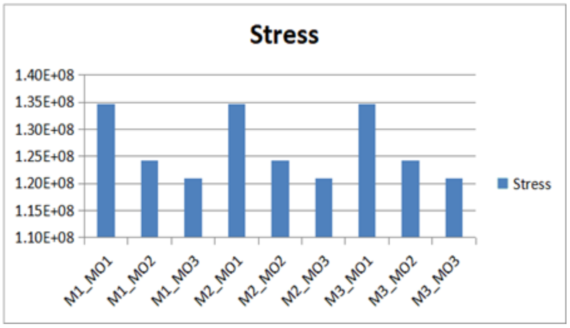

Fig 4.15: stress for different models

The stress analysis shows that maximum stress occurs in Model 1 due to lower stiffness. Modified models (M2 and M3) show reduced stress values, confirming that increasing girder depth helps in minimizing stress concentration.

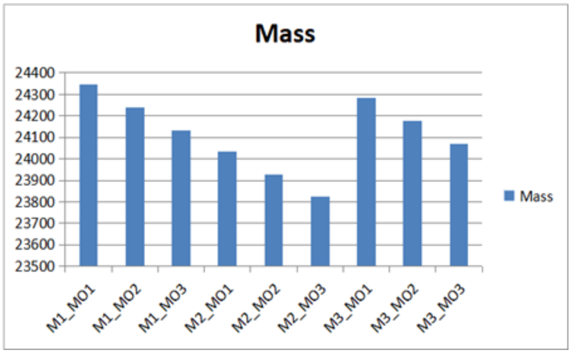

Fig 4.16: Mass for different models

The mass comparison indicates that increasing girder depth leads to an increase in mass. However, this increase in weight is justified by the improved stiffness and reduced deformation, highlighting a trade-off between strength and weight.

Frequency Analysis

Natural frequency results from ANSYS and MATLAB are compared.

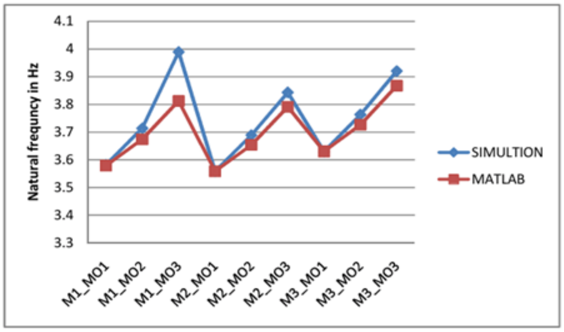

Fig 4.17: Natural frequency 1 for different models for ANSYS simulation and MATLAB

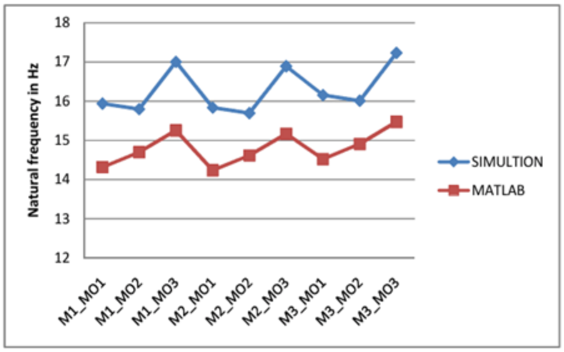

Fig 4.18: Natural frequency 2 for different models for ANSYS simulation and MATLAB

The natural frequency results obtained from ANSYS and MATLAB are in close agreement, validating the accuracy of the simulation. Higher stiffness models exhibit higher natural frequencies, which helps in avoiding resonance conditions.

VI. CONCLUSION

In this study, the structural and dynamic behavior of the EOT crane girder under moving load conditions has been analyzed using Finite Element Analysis (FEA). The analysis was carried out for different beam models with varying depths and materials to evaluate their performance in terms of deformation, stress, strain, and natural frequency.

The results indicate that the deformation of the crane girder is maximum at the mid-span and increases with the increase in moving load speed. It is observed that increasing the depth of the beam significantly Improves Stiffness and Reduces Deformation And stress. Among the materials considered, 34CrMo4 exhibits better performance due to its higher strength and stiffness characteristics. The optimized model (Model 3) demonstrates the best overall performance with reduced deformation, lower stress, and improved dynamic characteristics.

The dynamic analysis shows that higher speeds result in increased vibration and displacement, which may affect the stability and service life of the structure. The Dynamic Magnification Factor (DMF) indicates that dynamic effects significantly influence structural response at higher speeds. Modal analysis reveals that natural frequencies depend on the stiffness and geometry of the beam, and it is essential to avoid resonance conditions for safe operation.

Overall, the study concludes that optimization of beam geometry and proper material selection play a crucial role in enhancing the performance and safety of EOT crane girders. The findings of this work can be effectively used for the design and optimization of safe, efficient, and reliable EOT crane structures in industrial applications.

REFERENCES

[1] Camelia Bretotean Pinca, Gelu Ovidiu Tirian, Ana Josan, "Application of Finite Element Method to an Overhead Crane Bridge", Issue 2, Volume 4, 2009.

[2] Ismail Gerdemeli, Serpil Kurt, Metinyıldırım M.Sc., "Calculations, Modeling and Analysis with Finite Element Method of Rubber Tyred Container Stacking Crane", 14th International Research/Expert Conference, Trends in the Development of Machinery and Associated Technology, 2010.

[3] C. Alkin, C. E. Imrak, H. Kocabas, "Solid Modeling and Finite Element Analysis of an Overhead Crane Bridge", Actapolytechnica Vol. 45, 2005.

[4] Vlada Gasic, Milorad Milovancevic, Zoran Petkovic, "FEA Implementation in Moving Load Problem at Bridge Cranes", Machine design, ISSN 1821 1259, 2010.

[5] M. Euler, U. Kuhlmann, "Crane runways - Fatigue evaluation of crane rail welds using local concepts", International Journal of Fatigue, 2011.

[6] Ozden Caglayan, Kadir Ozakgul, Ovunc Tezer, Erdogan Uzgider, "Fatigue life prediction of existing crane runway girders", Journal of Constructional Steel Research, 2010.

[7] Ismail Gerdemeli, Serpil Kurt, Hasan Onur Alkan, "Main girder beam design and finite element analysis of 2/160 EOT crane",14th International Research/Expert Conference, Trends in the Development of Machinery and Associated Technology, 2010.

[8] Hong-Xia Wan, Mahen Mahendran, "Bending and torsion of hollow flange channel beams" Engineering Structures 84 (2015) 300–312, 2015.

[9] Jiho Moon a, Jong-WonYi b, ByungH.Choi c, Hak-EunLee, "Lateral– torsional buckling of I-girder with corrugated webs under uniform bending", Thin-Walled Structures 47 (2009) 21–30,2009.

[10] Fatimah Denan, Mohd Hanim Osman & Sariffuddin Saad, "The Study of Lateral Torsional Buckling Behaviour of Beam with Trapezoid Web Steel Section by Experimental and Finite Element Analysis", IJRRAS 2 (3) March 2010.

[11] Ismail Gerdemeli, Serpil Kurt, Hasan Onur Alkan, "Main Girder Beam Design and Finite Element Analysis Of 2x160 Ton EOT Crane", Trends in the Development of Machinery and Associated Technology,2010.

[12] M.R.Wakchaure, A.V. Sagade, "Finite Element Analysis of Castellated Steel Beam", International Journal of Engineering and

Innovative Technology (IJEIT) 2, Issue 1, July 2012

[13] R. Divahar, P. S. Joanna, “Lateral Buckling of Cold Formed Steel Beam with Trapezoidal Corrugated Web”, International Journal of Civil Engineering and Technology (IJCIET) Volume 5, Issue 3, March (2014), pp. 217-225.

[14] Limaye A. A, Alandkar P. M, “Strength of Welded Plate Girder with Corrugated Web Plate”Journal of Engineering Research and Applications ISSN: 2248-9622, Vol. 3, Issue 5, Sep-Oct 2013, pp.1925-1930.

[15] Abhinay Suratkar, Vishal Shukla, Dr. K. S. Zakiuddin, “Design Optimization of Overhead EOT Crane Box Girder Using Finite Element Analysis”, International Journal of Engineering Research & Technology (IJERT), Vol. 2 Issue 7, July – 2013.

[16] Sharda P. Siddh, Prof. P.D. Pachpor, “Finite Element Analysis of steel beam with web opening of different shapes”, International Journal of Science and Advanced Technology (ISSN 2221-8386) Volume 1 No 5 July 2011.

[17] Jae-Yuel Oh, Deuck Hang Lee, Kang Su Kim, “Accordion effect of prestressed steel beams with corrugated webs”

[18] Boksun Kim, Andrew Oliver, Joshua Vyse, “Bending Stresses of Steel Web Tapered Tee Section Cantilevers”, Journal of Civil Engineering and Architecture, ISSN 1934-7359, USA, Nov. 2013, Volume 7, No. 11 (Serial No. 72), pp. 1329-1342.

[19] B. Asgarian, M. Soltani, F. Mohri, "Lateral-torsional buckling of tapered thin-walled beams with arbitrary cross-sections", Science Direct, elsevier, Thin-Walled Structures 62 (2013) 96–SSS108.

[20] N.S. Trahair, “Bending and buckling of tapered steel beam structures”, Science Direct, Elsevier, Engineering Structures 59 (2014) 229–237.

[21] H. Ozbasaran, “A parametric study on lateral torsional buckling of European IPN and IPE cantilevers”, World Academy of Science, Engineering and Technology, International Journal of Civil, Architectural, Structural and Construction Engineering Vol:8 No:7, 2014.

[22] R. Drazumeric, F. Kosel, "Shape optimization of beam due to lateral buckling problem" Science Direct, elsevier, InternationalJournal of Non-Linear Mechanics 47 (2012) 65–74.

[23] Gurudatta, K.S.S., Harikiran, V., Raju, M.V.D.K., Rama Krishna, B., Jagadeesh, I. (2023). Design and CFD Analysis of Drone Thrust with Duct. In: Misra, R., Omer, R., Rajarajan, M., Veeravalli, B., Kesswani, N., Mishra, P. (eds) Machine Learning and Big Data Analytics. ICMLBDA 2022. Springer Proceedings in Mathematics & Statistics, vol 401. Springer, Cham. https://doi.org/10.1007/978-3-031-15175-0_44