International Journal of Engineering and Management Research
e-ISSN: 2250-0758 | p-ISSN: 2394-6962
https://ijemr.vandanapublications.com

Peer Reviewed & Refereed Journal
Volume-13, Issue-4 (August 2023)
https://doi.org/10.31033/ijemr.13.4.16

Finite Element Analysis of Material Handling System Components

Aarbaz Mutwalli[1], Jamil Dabir[2] and Maruti Khot[3]
[1]M-Tech Student, Department Mechanical Engineering Walchand College of Engineering, INDIA
[2]Assistant Professor, Department of Mechanical Engineering, Walchand College of Engineering, INDIA
[3]Assistant Professor, Department of Mechanical Engineering, Walchand College of Engineering, INDIA

[1]Corresponding Author: aarbaz.mutwalli@walchandsangli.ac.in

Received: 20-07-2023 Revised: 03-08-2023 Accepted: 24-08-2023

ABSTRACT

In any industry, a transport system is always required. Lifting and moving heavy goods over great distances has traditionally been done by workers. This may be economically beneficial to the employer but certainly is counter-productive in a way, considering the time required for the displacement as well as the health risks associated due to prolonged work. This work focuses on the design and analysis of an overhead Monorail system. The design analysis of overhead monorail crane with a single beam has been proposed. In this paper the design analysis of hoist track, support column and hook has been done and study of results of finite element analysis of crane components with 1 ton capacity and 27 m span length has been conducted. The optimum design was selected using various materials handling system method and further tested for its structural performance using finite element analysis. The 3D model prepared and finite element analysis was also used to confirm the structural integrity of the new concept design.

Keywords**--** Overhead Monorail Crane, Structural Analysis, Finite Element Analysis

I. INTRODUCTION

Crane systems are a typical type of product handling technology. Monorail crane enable businesses to safely move items from one area to another without or with minimal human intervention. Overhead Monorail crane use a single rail track, whether manually or mechanically driven, from which the conveying mechanisms and load handling occur across work areas. Overhead crane can be constructed to follow nearly any continuous course, changing direction horizontally and vertically. A single path can be several thousand feet long if numerous drives are used. Rail track, Carrier, hook, Overhead rails, and structure are the essential components of an overhead conveyor. A conveyor system is a type of mechanical handling equipment that transports things from one area to another.to a different. The fundamental definition of the problem is conveyor design and design optimization. Design of material handling system component for working load under 9.3KN. The primary goals are as follows:

- Improve the plant's efficiency.
- Reduce the amount of time spent on handling.
- Structure design with modeling software.
- Structural analysis using the Ansys finite element software.

II. LITERATURE REVIEW

The design of the runway beam, support positions, supporting structure, and connections are all part of the overall monorail structure. Previous researchers have only produced a small number of works on the design of the entire monorail system. As a result, the entire system is broken down into many separate parts, such as the monorail straight I Beam, girder, and design of supporting Beam and columns. In this review, the research works in the area of monorail system and its components-parts have been presented. This may be useful to those working in this field.

Nenad Zrnićet al. I-beam runway beams were given loading capacity curves based on capabilities related to the strength of the bottom flange and lateral buckling of the top flange. The strength of a particular I beam relies on the beam's span and the I beam's wheel location. When creating the load capacity curve, the torsional impact brought on by lateral load is not taken into account [1].

Rajpandian R. developed a system with 50 KN capacity EOT crane's structural analysis. He used ANSYS for structural analysis and the "Indian Standard Code for Steel Design" for his theoretical study. The buckling resistance moment must be greater than the gravity loads' calculated moments, including impact. The top compression flange is thought to be the only one to take on the horizontal bending moment. It applies the overall buckling check. The girder's web is examined for buckling and shear strength. It is also employed to check for local compression under the wheels. The crane girder is examined for fatigue as well as vertical deflection caused by static wheel loads and horizontal surge caused by crane surge. [2]

C. Alkin, C. E. Imrak, H. Kocabas Unlike prior works, shell elements in finite element modeling of an overhead box girder were investigated in this study. To demonstrate the correctness of the overhead crane bridge analysis, a four-node quadratic shell element is employed instead of a four-node tetrahedral element for finite element analysis. [3]

Pratik R. Patel, V.K. Patel the aim of this paper is for review on structural analysis of overhead crane girder using Finite Element Analysis (FEA) technique. Overhead crane girder is subjected to various types of loads. Girder is the critical assembly component of overhead crane. Currently research is being carried out to improve the strength structure of overhead crane girder. These efforts help to overcome overhead crane girder failure. Finite Element Analysis (FEA) software offers inexpensive solutions to overhead crane girder failure problem. In this study the researchers used Finite Element Analysis (FEA) technique using different types of approach. Finite Element Analysis (FEA) is an essential tool for helping us in determining the cause of problems. it also recommends the solutions. Finite Element Analysis (FEA) of structural failure should be adopted as standard tool in failure analysis. If engineer is trained, then Finite Element Analysis (FEA) is very quick methodology. It is also easy to deploy. With exponential increase in computing power, Finite Element Analysis (FEA) is easy to carry out. It is widely available with user friendly commercial software [4]

III. FINITE ELEMENT ANALYSIS

The Finite Element Method (FEM) is a technique used in Finite Element Analysis (FEA). The Finite Element Procedure (FEM) is one of many techniques that are employed because it is widely accessible in a variety of user-friendly commercial software and has modular forms that correspond to the stages of the procedure [1]. Finite Element Method (FEM) analysis is capable of handling any complex geometry. It can resolve displacements as well as stresses. As a collection of discrete finite elements joined at nodal points on element borders, the Finite Element Method (FEM) approximates the solution of the entire domain under study [3]. The stiffness matrix is created by first assembling the approximate solution over each element matrix. The displacement and force vectors for the full domain can also be derived. For the best results and precise shape functions, the Finite Element Method (FEM) can employ a variety of element types

IV. MATERIAL PROPERTIES

a. Young's Modulus = E = 2×10^5 Mpa
b. Poisson's Ratio = $\mu = 0.3$
c. Shear Modulus = G = 7.5×10^{10} Mpa
d. Density, ρ = $7850\ KG/m^2$
e. Yield Strength = $\sigma_y = 250Mpa$
f. Ultimate Strength = $\sigma_u = 460Mpa$
g. Thermal conductivity = k = 16 W/mK

V. STATIC ANALYSIS

Static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes). Static analysis involves both linear and nonlinear analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper -elasticity, contact surfaces, and creep.

a) FEA Model

Following 3D model is design under Ansys workbench as per design calculation.

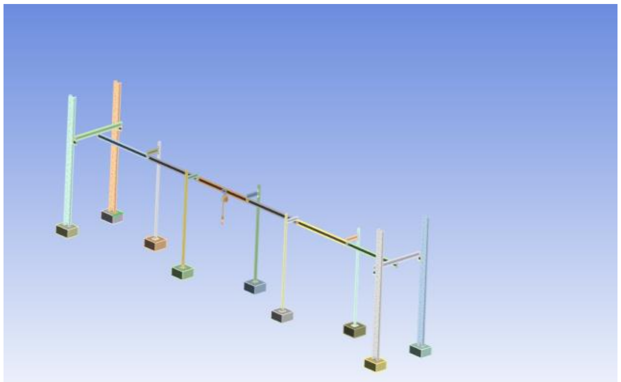


Figure 1: FEA model

VI. STATIC STRUCTURAL ANALYSIS

a) Hoist Track

Preprocessing: Steel properties are as Yield strength = 250 Mpa, Ultimate strength = 410Mpa, Poisson ratio = 0.3, Material density = $7850\ \mathrm{Kg/m^3}$ is added in the preprocessing.

Meshing: For better accuracy sizing is selected as proximity and curvature. Meshing is selected as auto generates fine mesh. Nodes 64 and 21element.

Boundary condition: Hoist track is fixed at one end and line pressure is $2.03N/mm^2$ applied to whole surface.
Solution: Fig.2 shows the maximum total deformation of the hoist track maximum deformation occurs at the whole surface 0.579mm is the maximum deflection which is acceptable. Fig.3 and Fig.4 show the equivalent stress and bending stress are 29.67Mpa and 37.67Mpa respectively. Yield stresses are 250Mpa so the design is safe.

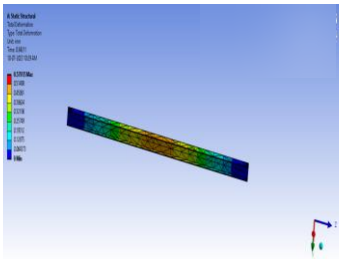


Figure 2: Deformation

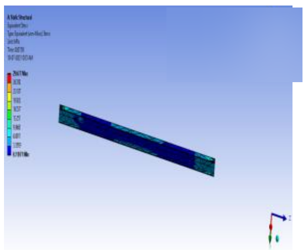


Figure 3: Equivalent stress

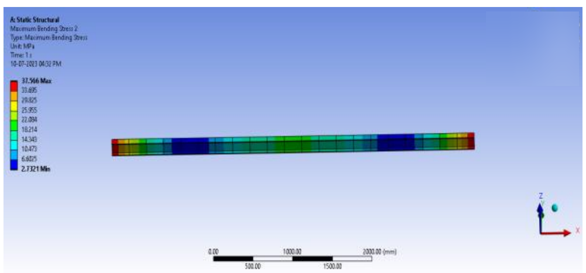


Figure 4: Bending stress

b) End Support Bbeam
Preprocessing: Steel properties are as Yield strength =250Mpa, Ultimate strength = 410Mpa, Poisson ratio = 0.3, Material density = $7850\ \mathrm{Kg/m^3}$ is added in the preprocessing.
Meshing: For better accuracy sizing is selected as proximity and curvature. Meshing is selected as fine mesh.
Boundary condition: Hoist track is fixed at one end and Force is 9300N applied to center of beam surface.
Solution: Fig.5 shows the maximum total deformation of the end support maximum deformation occurs at the whole surface 1.12mm is the maximum deflection which is acceptable. Fig.6 and Fig.7 show the equivalent stress and bending stress are 45.43 Mpa and 16.002 Mpa respectively. Yield stresses are 250pa so the design is safe.

Figure 5: Deformation

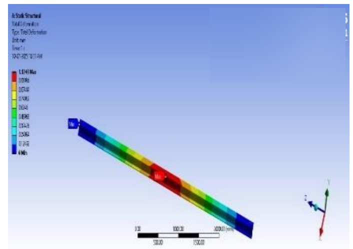
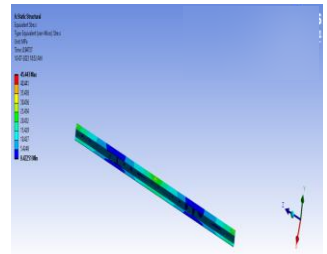


Figure 6: Equivalent stress



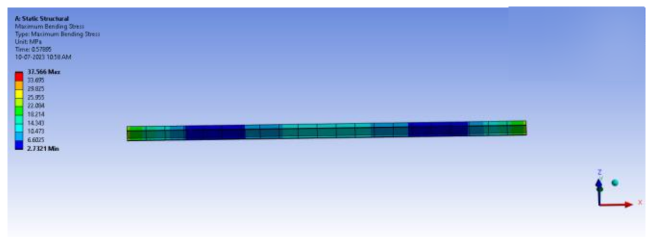


Figure 7: Bending stress

c) Cantilever Supports for Hoist Track
Preprocessing: Steel properties are as Yield strength =250Mpa, Ultimate strength = 410Mpa, Poisson ratio = 0.3, Material density = $7850\ \mathrm{Kg/m^3}$ is added in the preprocessing.
Meshing: For better accuracy sizing is selected as proximity and curvature. Meshing is selected as fine mesh.
Boundary condition: Cantilever beam is fixed at one end and Force is 9300N applied to end of beam surface.
Solution: Fig.8 shows the maximum total deformation of the cantilever beam maximum deformation occurs at the whole surface 2.80mm is the maximum deflection which is acceptable. Fig.9 and Fig.10 show the equivalent stress and bending stress are 238.82 Mpa and 105.44 Mpa respectively. Yield stress is 250Mpa so the design is safe

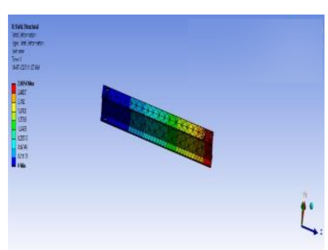


Figure 8: Deformation

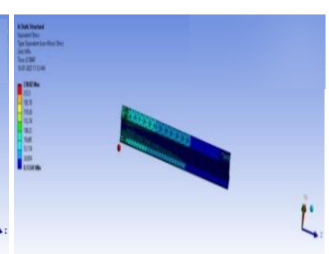


Figure 9: Equivalent stress

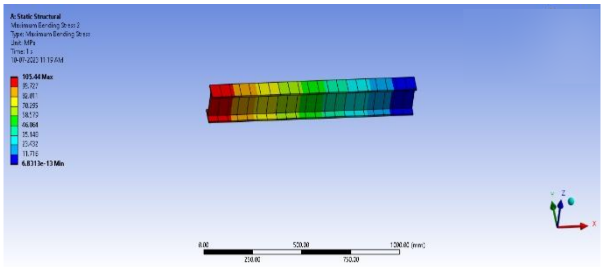


Figure 10: Bending stress

d) Cantilever Support of End Column

Preprocessing: Steel properties are as Yield strength =250Mpa, Ultimate strength = 410Mpa, Poisson ratio = 0.3, Material density = $7850\ \mathrm{Kg/m^3}$ is added in the preprocessing.

Meshing: For better accuracy sizing is selected as proximity and curvature. Meshing is selected as fine mesh. Boundary condition: Cantilever beam is fixed at one end and Force is 4659N applied to end of beam surface

Solution: Fig.11 shows the maximum total deformation of the cantilever beam maximum deformation occurs at the whole surface 0.018mm is the maximum deflection which is acceptable. Fig.12 and Fig.13 show the equivalent stress and bending stress are 10.72 Mpa and 6.44 Mpa respectively. Yield stresses are 250Mpa so the design is safe.

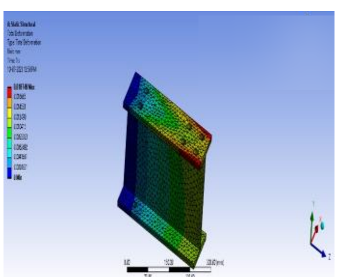


Figure 11: Deformation

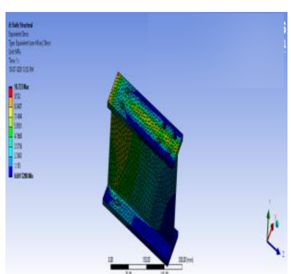


Figure 12: Equivalent stress

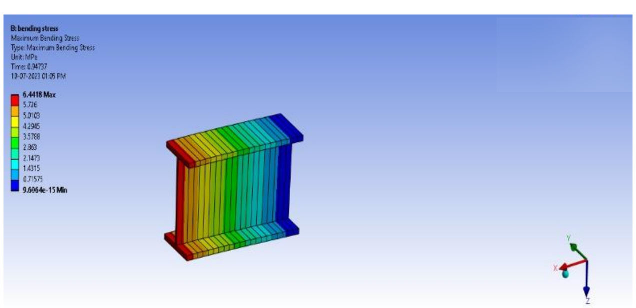


Figure 13: Bending stress

e) Crane Hook

Preprocessing: Mild Steel properties are as Yield strength =250Mpa, Ultimate strength = 410Mpa, Poisson ratio = 0.3, Material density = $7850\ \mathrm{Kg/m^3}$ is added in the preprocessing.

Meshing: For better accuracy sizing is selected as proximity and curvature. Meshing is selected as fine mesh. Boundary condition: Crane hook is fixed at one end and Force is 7848N applied to end of beam surface.

Solution: Fig.14 shows the maximum total deformation of the crane hook maximum deformation occurs at the hole surface 0.288mm is the maximum deflection which is acceptable. Fig.15 and Fig.16 show the equivalent stress and shear stress are 158.03 Mpa and 74.37 Mpa respectively. Yield stress is 250Mpa so the design is safe.

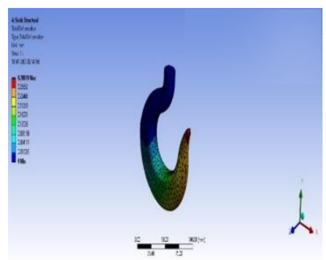


Figure 14: Total deformation

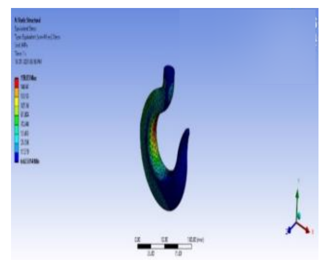


Figure 15: Equivalent stress

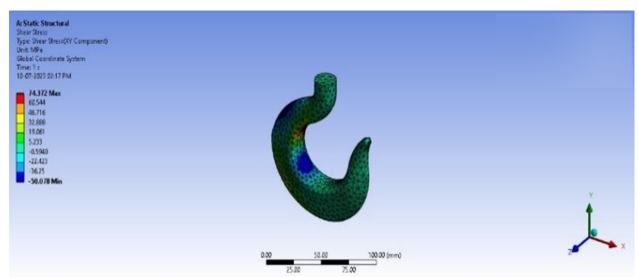


Figure 16: Shear stress

VII. ANSYS RESULTS OF VARIOUS COMPONENTS

Component	Ansys
Girder track	$\sigma = 29.67$ Mpa
	$\sigma_b = 37.54$Mpa
	$\delta = 0.579$ mm
End support beam	$\sigma = 45.44$Mpa
	$\sigma_b = 37.56$ Mpa
	$\delta = 1.12$ mm
Cantilever supports for hoist track	$\sigma = 238.82$ Mpa
	$\sigma_b = 105.44$ Mpa
	$\delta = 2.80$mm
Cantilever support of end column	$\sigma = 10.2$Mpa
	$\sigma_b = 6.44$ Mpa
	$\delta = 0.018$mm
Crane hook	$\delta = 0.288mm$
	$\tau_{max} = 74.37$ Mpa
	$\sigma = 158.03$Mpa

REFERENCES

[1] Nenad ZRNIĆ, Srđan BOŠNJAK & Vlada GAŠIĆ. (2022). Loading capacities curves for i-section runway beams according to bottom flange bending and lateral buckling. *University of Belgrade, Faculty of Mechanical Engineering*.

[2] Raj Pandian R. Design and analysis of overhead crane structural columns. Journal *of Electronics and Communication Engineering*, 69-75.

[3] C. Alkin, C. E. Imrak & H. Kocabas. (2005). Solid modeling and finite element analysis of an overhead crane bridge. *Actapolytechnica, 45*.

[4] Pratik R. Patel & V.K. Patel. (2013). A review on structural analysis of overhead crane girder using fea technique. *International Journal of Engineering Science and Innovative Technology (IJESIT), 2*(4).

[5] Shivaram Sullad & Prof Nagaraj Kelageri. (2015). Design and analysis of rail bracket in material handling process. *International Research Journal of Engineering and Technology (IRJET), 02*(07).