IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 4, March 2026
Impact Factor: 8.2

Design and Load Analysis of Industrial Monorail Overhead Hoist Crane

Aniket Subhsh Uchit[1], Anuj Umesh Dawnekar[2], Ritesh Prakashrao Shirgire[3], Dipankar Sunil Ghodeswar[4], Prof. Sachin Bende[5]
[1,2,3,4]Students, Jawaharlal Darda Institute of Engineering and Technology, Yavatmal.
*[5] Prof., Jawaharlal Darda Institute of Engineering and technology, Yavatmal.
aniketuchit@gmail.com, adawanekar@gmail.com riteshshirgire@gmail.com
ghodeswardipankar@gmail.com Sachin_bende@jdiet.ac.in

Abstract: *The design and analysis of a monorail girder overhead hoist crane is important for efficient material handling in industries. This project focuses on the structural design, load analysis, and performance evaluation of a monorail girder crane used for lifting and transporting heavy loads. The study includes calculation of stresses, load distribution, and selection of suitable materials to ensure safety and reliability. Modern design methods and analysis tools are used to evaluate the strength and stability of the crane components. The results help in optimizing the crane design for better efficiency, reduced weight, and improved durability. This study contributes to safer lifting operations and enhanced productivity in industrial environments where overhead material handling systems are widely used.*

Keywords*:* Monorail Girder Crane, Overhead Hoist Crane, Structural Design, Load Analysis, Stress Analysis, Material Handling Equipment, Finite Element Analysis (FEA)

I. INTRODUCTION

Cranes are the best way of providing a heavy lifting facility covering virtually the whole area of a building. An overhead hoist crane is the most important materials handling system for heavy goods. The primary task of the overhead hoist crane is to handle and transfer heavy payloads from one position to another. Thus, they are used in areas such as power plants, automobile plants, shipyards and in many industrial applications. Their design features vary widely according to their major operational specifications, such as: type of motion of the crane structure, weight and type of the load, location of the crane, geometric features and environmental conditions. Since the crane design procedures are highly standardized with these components, most effort and time are spent on interpreting and implementing the available design standards

A overhead hoist crane is a type of crane with a hoist in a trolley which runs horizontally along rails, usually fitted underneath a beam spanning between uprights which have wheels so that the whole crane can perform cross travel and long travel. These cranes come in all sizes, and some can carry and move very heavy loads, particularly the extremely large examples are used in power plants, shipyards or industrial installations.

Hoist cranes with a double box girder not only hoist loads but also carry them horizontally. A double beam overhead hoist crane is built of a trolley travelling on bridges, and bridges travelling on rails. The trolley hoists or lowers the loads and carries them on the bridge structure. The monorails (Girder) carry the loads on a rail. As a result, three perpendicular movements are performed. The system is depicted in Fig. 1.1, where the payload of the mass is attached to the bridge with wire ropes. The double box girders are subjected to vertical and horizontal loads by the weight of the crane, the working (hook) load and the dynamic loads. With a double box girder construction, the trolley runs above or between the girders.

II. METHODLOGY

Step1 - Study of drawing and details of monorail girder hoist crane

In this step, the drawing and details of monorail girder hoist crane will be studied. Plan of the crane will be studied properly. All the dimensions, values and parameters will be checked.

Step 2 - Manual Design

Here actual calculations of shear force and bending moment will be carried out considering all the values of various parameters like dimensions, loading on girder i.e. live load and uniformely distributed load acting on girder.

Step 3 – Modeling

Modeling of monorail girder will be carried out by considering all girder dimensions using a CAD software.

Step 4 - Analysis

CAD model created will be analysed in ANSYS software.

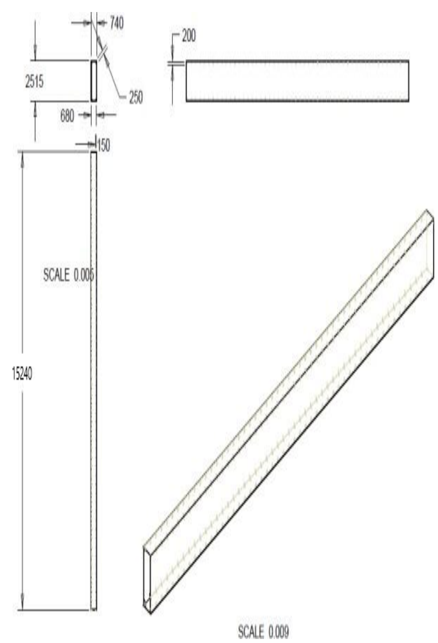
Step 5 - Result and discussion

Based on the various values obtained in the results of analysis, the conclusion will be made.

2.1 DESIGN OF MONORAIL GIRDER

Technical Specifications :

a) Crane Type - Singe Girder EOT Crane
b) Main Hoist safe working Load - 7.5 T
c) Span - 15.24 M
d) Class of Duty - Class II as per IS 807/3177
e) Dead weight of girder including all accessories - 1.5 T
f) Bay Length - 27.43 mtrs
g) Height of lift - 6 mtrs



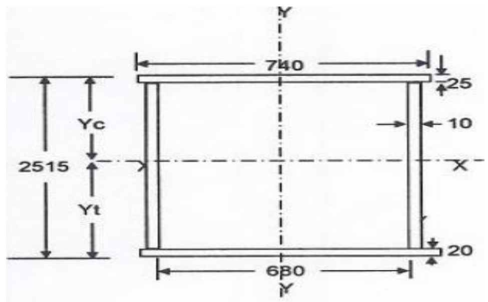
Drawing of Monorail Girder

2 Girder Section Parameters

a) Width : 74 cm
b) Height : 251.5 cm
c) Top thickness : 2.5 cm

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 4, March 2026
Impact Factor: 8.2

d) Bottom thickness : 2 cm
e) Web thickness : 1 cm
f) Web height : 247 cm
g) Distance between web to web : 68 cm
h) Side clearance : 2 cm



Cross section of Monorail Girder

III. COMPUTER AIDED DESIGN

Computer-aided design (CAD), also known as computer-aided design and drafting (CADD) is the use of computer systems to assist in the creation, modification, analysis, or optimization of a design. Computer-aided drafting describes the process of creating a technical drawing with the use of computer software. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print or machining operations. CAD software uses either vector based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects.

CAD often involves more than just shapes.

As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions. CAD may be used to design curves and figures in two- dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space. The challenges before the product designers today are listed below:

- Higher customer quality expectations
- Need to have innovation and originality in design
- Need for global collaboration across and beyond the enterprise among designers, customers and vendors to reduce development lead times
- Need to evaluate feasibility throughout the design process
- Ability to react quickly to design changes as and when change requests are made
- Ability to express the design intent in terms of shape and function using the tools available as well as the ability of the tools to transfer data back and forth seamlessly.

3.1 Role of CAD/CAM in Design:

Computer Aided Design (CAD) is a form of design in which people work with computers to create ideas, models, and prototypes. CAD has been developed to guide people with technical drawing and drafting. Drafting and technical drawing can be very painstaking, and they require some special skills. Using CAD for drafting still requires many of the same skills, but by working with a computer instead of on paper, people can be much more efficient.

Process planning translates design information into the process steps and instructions to efficiently and effectively manufacture products. As the design process is supported by many computer-aided tools, computer-aided process planning (CAPP) has evolved to simplify and improve process planning and achieve more effective use of manufacturing resources.

Computer Aided Design (CAD) involves the use of computer hardware and graphics software to generate design drawings. Modern CAD equipment enables the designer to quickly produce very accurate and realistic images of products to be manufactured. Computer Aided Manufacturing (CAM) is a system of automatically producing finished products by using computer controlled production machines. CAD and CAM work together in that the digital model generated in CAD is inputted to the CAM software package. The CAM software needs to know the physical shape of the product (CAD model) before it can compose a proper set of fabrication instructions to a production machine.

3.2 Various CAD softwares:

The following are some of the best known CAD softwares available in the market

- AutoCAD
- Autodesk Inventor
- Pro/Engineer Wildfire 5.0
- Catia
- SolidWorks
- Solid Edge
- Unigraphics NX
- ANSYS

3.3 CAD Model of Monorail Girder

In this work, CATIA software having Version V5 is used for creating a CAD model of Monorail Girder of Overhead Hoist Crane.

(1) Preprocessing: defining the problem The major steps in preprocessing are

(i) Define keypoints/lines/areas/volumes,

(ii) Define element type and material/geometric properties, and

(iii) Mesh lines/areas/ volumes as required. The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

(2) Solution: assigning loads, constraints, and solving

Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally solve the resulting set of equations.

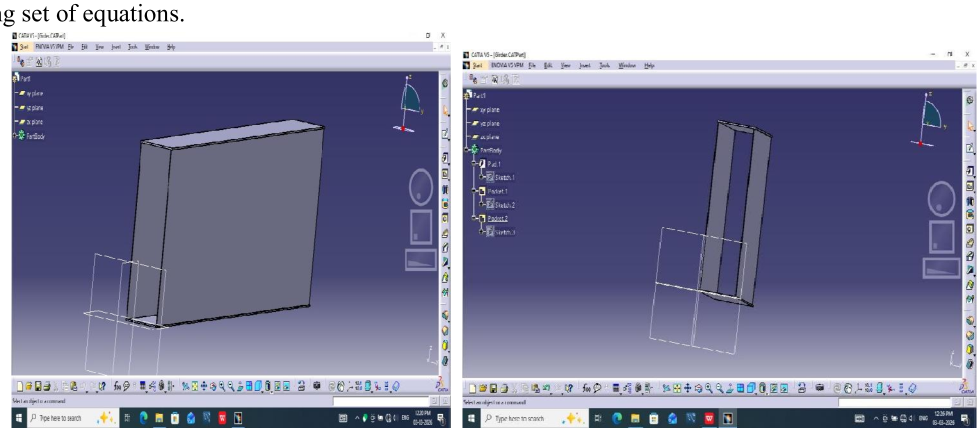


Fig. CAD Model of Monorrail Girder

(3) Postprocessing: further processing and viewing of the results.

In this stage, one may wish to see

(i) Lists of nodal displacements,
(ii) Element forces and moments,
(iii) Deflection plots, and
(iv) Stress contour diagrams or temperature maps.

IV. FINITE ELEMENT ANALYSIS

ANSYS is a general-purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.
In general, a finite-element solution may be broken into the following three stages.

4.1 Preprocessing Stage Building a model

The ANSYS program has many finite element analysis capabilities, ranging from a simple linear static analysis to a complex nonlinear transient dynamic analysis. Building a finite element model requires more time than any other part of the analysis. First, a job name and analysis title have to be specified. Next, the preprocessor is used to define the element types, element real constants, material properties, and the model geometry. It is important to remember that ANSYS does not assume a system of units for intended analysis. Except in magnetic field analyses, any system of units can be used so long as it is ensured that units are consistent for all input data.

4.2 Creating the model geometry

Once material properties are defined, the next step in an analysis is generating a finite element model nodes and element adequately describing the model geometry. There are two methods to create the finite element model: solid modeling and direct generation. With solid modeling, the geometry of shape of the model is described, and then the ANSYS program automatically meshes the geometry with nodes and elements. The size and shape of the elements that the program creates can be controlled. With direct generation, the location of each node and the connectivity of each element is manually defined. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc., are available.

4.3 Applying Load

Loads can be applied using either preprocessor or the SOLUTION processor. Regardless of the chosen strategy, it is necessary to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution.
The analysis type to be used is based on the loading conditions and the response which is wished to calculate. For example, if natural frequencies and mode shapes are to be calculated, then a modal analysis ought to be chosen. The ANSYS program offers the following analysis types: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring. Not all analysis types are valid for all disciplines. Modal analysis, for instance, is not valid for thermal models. Analysis options allow for customization of analysis type.
The word loads used here includes boundary conditions, i.e., constraints, supports, or boundary field specifications. It also includes other externally and internally applied loads. Loads in the ANSYS program are divided into six categories: DOF constraints, forces, surface loads, body loads, inertia loads, and coupled field loads. Most of these loads can be applied either on the solid model (keypoints, lines, and areas) or the finite-element model (nodes and elements).
There are two important load related terms. A load step is simply a configuration of loads for which the solution is obtained. In a structural analysis, for instance, wind loads may be applied in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 4, March 2026
Impact Factor: 8.2

Substeps are incremental steps taken within a load step. They are mainly used for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as time steps which are taken over a period of time.

4.4 Solution Stage

After reviewing the summary information about the model, click SOLVE button to start the solution. When this command is issued, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file and also to the database. The only difference is that only one set of results can reside in the database at one time, while a number of result sets can be written to the results file. Once the solution has been calculated, the ANSYS postprocessors can be used to review the results.

.

4.5 Finite Element Analysis of Monorail Girder

ANSYS is a general-purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

In general, a finite-element solution may be broken into the following three stages.

(1) Preprocessing: defining the problem The major steps in preprocessing are

(i) Define keypoints/lines/areas/volumes,

(ii) Define element type and material/geometric properties, and

(iii) Mesh lines/areas/ volumes as required. The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

(2) Solution: assigning loads, constraints, and solving

Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally solve the resulting set of equations.

(3) Postprocessing: further processing and viewing of the results.

In this stage, one may wish to see

(i) Lists of nodal displacements,

(ii) Element forces and moments,

(iii) Deflection plots, and

(iv) Stress contour diagrams or temperature maps.

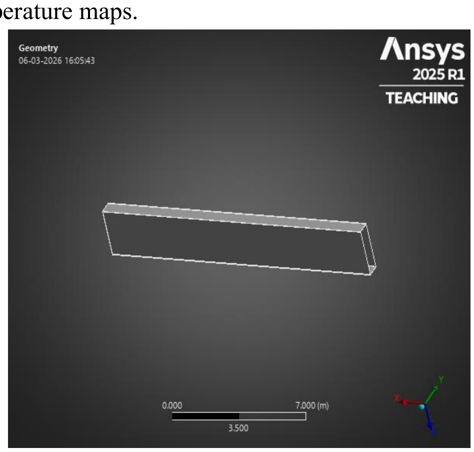


Fig: Geometry

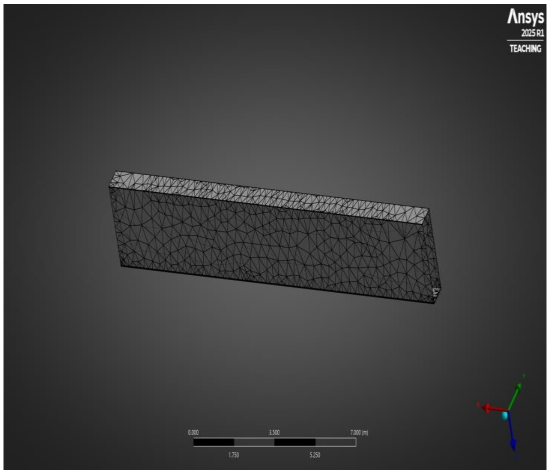


Fig: Meshing

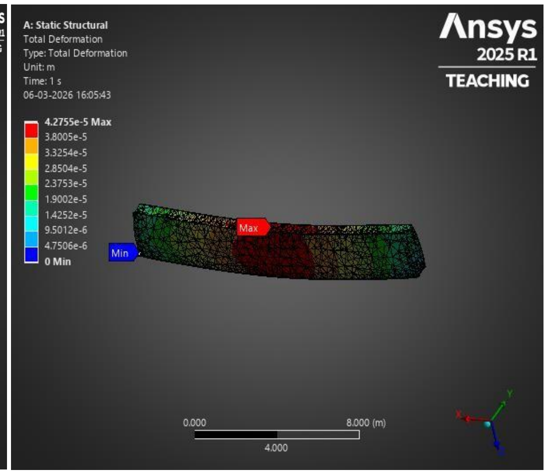


Fig1: Static Structural

V. RESULT

The finite element analysis of Monorail Girder of Overhead hoist Crane is done considering the two loading conditions as without live load acting and with live load acting. The results obtained from the finite element analysis of Girder in ANSYS software are given below.

When live load is not acting i.e. Overhead hoist Crane is carrying no load.

Although Overhead Hoist Crane is carrying no load but considering the self weight and weight of the accessories i.e. dead load as a uniformely distributed load, finite element analysis of Girder is done. The stresses developed and deformation occurred in the girder due to the dead load are shown below.

When live load is acting i.e. Overhead hoist Crane is carrying a load.

When the Overhead Hoist Crane is carrying a load then while performing its analysis, both live load and dead load acting on the girder are considered simultaneously. Live load is applied as a point load and dead load is applied as a uniformly distributed load on the girder simultneously and finite element analysis is carried out in the ANSYS sofwtare.

The results of this analysis are shown below in the form of stresses developed and deformation occurred in the girder.

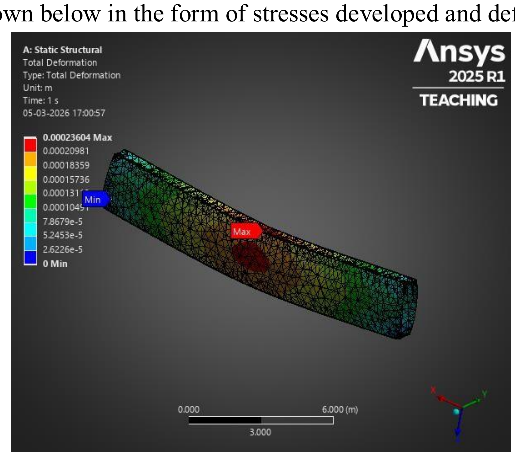


Fig2: Static Structural

IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429

Volume 6, Issue 4, March 2026

Impact Factor: 8.2

REFERENCES

1. International journal on "Shear failure characteristics of steel plate girders" by M.M. Alinia, Maryam Shakiba, H.R. Habashi, , Journal of Thin-Walled Structures 47 (2009) 1498-1506.

2. International journal on "Fatigue life prediction of existing crane runway girders" by Ozden Caglayan, Kadir Ozakgul, Ovunc Tezer, Erdogan Uzgider, Journal of Constructional Steel Research 66 (2010) 1164-1173.

3. International journal on , "Resistance of steel I- sections under axial force and biaxial bending" by A. M. Baptista, Journal of Constructional Steel Research xxx (2011).

International journal on "Shear response of 4.The postprocessor is used to review results. Using this postprocessor contour displays, deformed shapes, Graph plots of results data versus time (or frequency) and tabular listings to review and interpret the results of the analysis can be obtained. It offers many other capabilities, including error estimation, load case combinations, calculations among results data and path operations, arithmetic calculations and complex algebra. tainless steel plate girders" by E. Real, E Mirambell, I. Estrada, Journal of Engineering Structures 29 (2007) 1626-1640.

4. International journal on "I-shaped steel girders subjected to bending moment and travelling patch loading" by Per Granath, Anders Thorsson, Bo Edlund, Journal of Constructional Steel Research 54 (2000) 409-421.

5. International journal on "Elastic buckling of web plates in I-girders under patch and wheel loading" by T. Ren, G.S. Tong, journal of Engineering Structures 27 (2005) 1528–1536.

6. International journal on "Ultimate strength of longitudinally stiffened I-girder webs subjected to combined patch loading and bending" by Carlos Graciano, Euro Casanova, Journal of Constructional Steel Research 61 (2005) 93–111.

7. International journal on "Resistance of longitudinally stiffened I-girders subjected to concentrated loads" by Carlos Graciano, Bernt Johansson, Journal of Constructional Steel Research 59 (2003) 561–586.

8. International journal on "Linear buckling analysis of unstiffened plates subjected toboth patch load and bending moment" by Emanuele Maiorana, Carlo Pellegrino, Claudio Mo dena, Engineering Structures 30 (2008) 3731-3738.

10.International journal on "Solid Modeling and Finite Element Analysis of an Overhead Crane Bridge" by C. Alkin, C. E. Imrak, H. Kocabas Acta, Polytechnica Vol. 45 No. 3/2005.