INTERNATIONAL JOURNAL OF
RECENT TRENDS IN MULTIDISCIPLINARY RESEARCH
Special Issue, March-April 2023, Vol 3 (02), 106-111.
International conference on Innovation towards Sustainable Development Goals-ICISDG'23
ISSN No: 2583 0368

Comparative Study of Structural Members in Truss for Material Handling System Using FEA

Chirag Goriya[1], Akshat Shah[2], Amey Shirodkar[3], Harsh Vaya[4], N. G. Nagothkar[5]
[1,2,3,4,5]*Department of Mechanical Engineering, PCT's A.P. Shah Institute of Technology, Thane, India.*

Article Type: Research

OPENACCESS
Article Citation:
Chirag Goriya[1], Akshat Shah[2], Amey Shirodkar[3], Harsh Vaya[4], N. G. Nagothkar[5], "Comparative Study of Structural Membersin Truss for Material Handling System Using FEA", International Journal of Recent Trends In Multidisciplinary Research, March-April 2023, Vol 3(02), 106-111.

Check for updates

https://www.doi.org/10.59256/ijrtmr.20230402c23

Publishedby**5th Dimension Research Publication.**

Abstract: This paper presents a comparative study on the structural members of truss using Finite Element Analysis. The focus of the work is on optimizing the design of truss structures to improve the overall performance and efficiency. This study demonstrates application of numerical simulations using ANSYS 2021 R2 to evaluate the impact of various loads on the behaviour of different structural members of the truss modelled using Solid Works 2021. The results of the simulations were analyzed and compared to select the structural membersfor optimal structure that provides the best balance between strength, weight, and cost. The findings of the study have shown important implications on the design and fabrication of truss for material handeling system at Evergreen Company Ltd in Vasai and can further contribute to the development of more efficient truss structures in future.

Key Word: Truss structure, Material handling system, FEA, ANSYS 2021 R2, Solid Works 2021.

1. Introduction

A truss is essentially a triangulated system of straight interconnected structural elements. The most common use of trusses is in buildings, providing support to roofs, floors and internal loading such as services and suspended ceilings, are readily provided. The main reasons for using trusses are: long span, Lightweight Reduced deflection (compared to plain members), Opportunity to support considerable loads.[1]

However, the iterative process of determining the optimal cross-sectional members for a truss structure can be complex and time-consuming. The aim of this study is to simplify this process by providing a comprehensive analysis of the iteration of section of truss structures. This study focuses on exploring the effects of different sizes of the cross sectionson the overall strength and stability of truss systems. The methodology involves finite element analysis and testing, allowing for a detailed examination of the behaviour of trusses under various loading conditions.

The results of this study provide valuable insights into the optimal section selection for truss structures, and highlight the importance of considering cross section in the design process. The findings have important implications for the design and construction of truss systems, and can be used to improve the overall efficiency and reliability of these structures.

2. Literature Review

Using a genetic algorithm, Cazacu and Grama (2013) [2] demonstrated a process and software application to improve the topology, size, and form of planar trusses. The major goal of the optimization is to reduce the structure's overall mass while keeping it within the maximum permitted levels of stress and displacement. The assessment process makes use of general algorithms.

Using ANSYS, Ajinkya Karpe et al. (2014) [3] carried out the FEM analysis of the tower crane jib. The components developed in this article included the crane hook and snatch block assembly, wire ropes, moving trolley, tie rods, jib, counterweight side, mast, and slewing ring. Initial comparisons between two Tower Crane jib models focused on axial force and jib member deformation. For further examination, the superior model was chosen.

In the illustration by Pathak and Garg (2015) [4] different types of trusses such as, A-type, Fink, and Howe were analysed. Several load combinations were applied to get the desired outcomes.The study also analysed the effects of purlins on truss members, considering purlin locations and the induced forces. Fixed connectivity was found to provide better strength, but it resulted in an increase in overall truss weight.The "A type" truss was discovered to be ideal as it produced satisfactory results.

A study presented by Arumugam, K. Naveen (2016) [5] illustrated a truss structure bridge employing composite materials which was designed and analysed in order to be both stronger and lighter than an equivalent steel bridge. By comparing the minimal margin of safety in each truss bridge member, the strength was compared. In order to compare which material was more effective while building a truss bridge, both composite material and steel truss bridges had similar geometry. The capacity of the application to reduce weight and enhance strength was used to assess the effectiveness of employing composite materials in the truss members of an industry.

The behaviour of a welded rectangular truss measuring 18000 x 3600 mm was demonstrated by Yaroslav Kovalchuk et al. (2017) [6]. A prototype was created using 100x100x7 mm pairs of rolled, angular steel profiles. According to the authors, gusset plates with circular cut-out designs enable static stress to be reduced by 16.2% and fatigue durability to be improved by 18.4% under cyclic stressors.

The purpose of the study provided by Sharma and Pahwa (2018) [7] was to develop a bridge construction with several elemental parts. The study examined the viability of analysing and designing Truss bridge structures using steel profiles that may be found nearby. Eight node solid components are chosen for sections "I" and "L," and each modal is meshed separately. In order to determine the total deformation and mode forms of the bridge structure, the modal analysis in ANSYS was finished.

3. Methodology

The following methodology was adopted for the study:

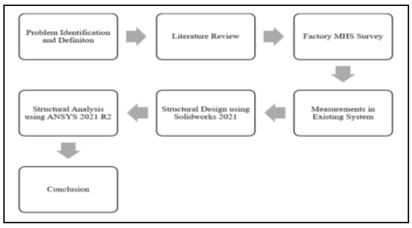


Fig 1: Methodology adopted for design and Fabrication of Material Handling System

Problem identification was the first step in the methodology. Firstly, we identified the problem in the existing model and brainstormed on certain solutions to solve it. After that, we had done literature survey so that we can find some more ways to solve the problem by referring to research and review papers. Then we performed factory Material Handling System (MHS) survey to get to know about the conventional MHS being used in the factories. It was then followed by carrying out the measurements of the existing systems. Later, we developed CAD models of our solution in the Solidworks software which is a renowned and user-friendly designing software. Then we carried out the analysis of our solution using ANSYS software and after getting suitable results, we concluded our solution.

4. Problem Identification

At Evergreen group of companies, the material handling system (MHS) was not up to the expectations. Previously, raw materials and the finished components were transportedmanually by 5 to 6 labours resulting into higher loading and unloading time eventually leading to higher lead time and sometimes damaging the components. To overcome this issue, the company required to develop a new MHS as a part of production planning activities.One of the objectives was to develop the system using a pure mechanical energy. As a part of this project a mechanical MHS was designed and developed with truss, a gearbox, chain- sprocket and trolley as shown in Fig.1

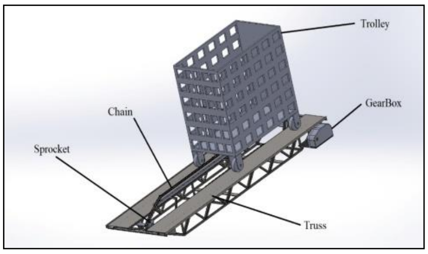


Fig 2: CAD model of the Material Handling System

It is evident from figure 1 that truss is the main structural component of the system and it is necessary to evaluate the stresses induced in the system during unloading of the trolley with payload. Also, the overall cost of the MHS will be determined by the truss as it has the maximum weight. Therefore, detailed study was conducted on truss.

5. CAD Model

Computer-aided design (CAD) is the use of computer software to assist with the design, layout, and technical documentation of products. CAD enables engineers to generate two-dimensional (2D) or three-dimensional (3D) models of an object or system of objects and view those models under a variety of different parameters to simulate and test real-world product conditions.[8] With 3D CAD, we can share, review, simulate, and modify designs easily, opening doors to innovative and differentiated products that get to market fast. The software used for the purpose was SolidWorks 2021 software. Being a user-friendly software, it has all the necessary commands and tools that were used for making the model of the project.

The existing system to load and unload the manufactured components was observed during the problem identification phase. All the crucial dimensions were measured using a measuring tape. Considering the space available at the factory layout crucial dimensions were found out. This work resulted into the overall length of truss as 10 ft with width of 3ft (9 inches) and height as 8 in with an inclination of 20°

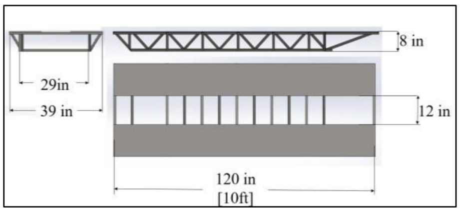


Fig 3: CAD model of truss

Truss can be fabricated using various cross-sectional members available in the market. Therefore, market survey was conducted during this phase of the study. Different cross sections viz. 'C channel', 'I beam', 'L channel', 'Square pipe', 'Circular pipe' with minimum standard sizes available in markets were selected for the modeling and simulation.).

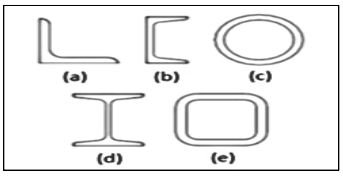


Table 1: Standard sizes of the cross sections

Shape of the section	Length	Width	Thickness	Diameter
L section	1 inch	1 inch	0.14 inch	-
C section	1.5 inch	0.26 inch	3 inches	-
I beam	4 inches	3 inches	0.17 inch	-
Circular tube	-	-	0.14 inch	1 inch
Square section	1 inch	1 inch	0.14 inch	-

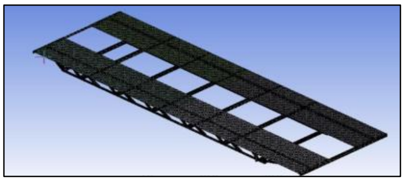
AISI 1008 is a low carbon steel with excellent mechanical properties that make it popular in various industries[8]. It is also readily available in the material libraries of modeling and simulation software. Therefore, it was selected as the material for CAD model. It has a high density of 7.872 g/cm^3, making it suitable for applications that require high mass. Its tensile strength of 340 MPa and high elastic modulus of 210 GPa make it a durable and stiff material suitable for structural components and support beams. Lastly, its Poisson's ratio of 0.3 suggests that it is relatively incompressible, making it ideal for applications that require high resistance to compression or impacts. Properties of AISI 1008 are summarised in table 2. These values were generated on the basis of present market survey. In future, these values may change.

Table 2: Properties of AISI 1008 material

Properties	**Value**
Density	7.872 g/cm^3
Tensile strength	340 MPa
Elastic modulus	210 GPa
Poisson's ratio	0.3

6. Simultion

Meshing is a very crucial step in Finite Element Analysis (FEA) simulations, as it directly affects the accuracy of the results.To achieve accurate results, it is essential to choose an appropriate type of element and its mesh size. Linear tetrahedral element was selected, as it is computationally efficient for larger structures, as compared to other element types. Preliminary simulations were carried out with mesh sizes of 5mm, 10mm and 15mm. Mesh size of 10mm had given successful results with convergent solutions and reasonable computational time. Quality of meshing is as shown in Fig 4.



Applying correct boundary conditions is the second step of simulation. Analysing the model following boundary conditions were applied during structural analysis:

- $Mass\ of\ Trolley\ = 30\ kg$
- $Total\ Mass\ of\ trolley\ with\ Payload = 500\ kg$
- $Total\ weight\ of\ trolley\ and\ Payload = F_t =\ 500 \times 9.81\ =\ 4905N$
- $Distribution\ of\ Weight =\ Left: Right =\ 50: 50$
- $Weight\ Per\ wheel\ F_w\ =\ 4905 \times 0.5\ =\ 2452.5\ N$
- $Truss\ Inclination\ with\ ground;\ \theta\ =\ 20°$
- $Y\ component\ of\ F_w;\ F_w sin\theta\ =\ -2452.5 \times sin20\ =\ \mathbf{-856.5\ N}$
- $X\ component\ of\ F_w;\ F_w cos\theta = \ 2452.5 \times cos20\ =\ \mathbf{2292\ N}$

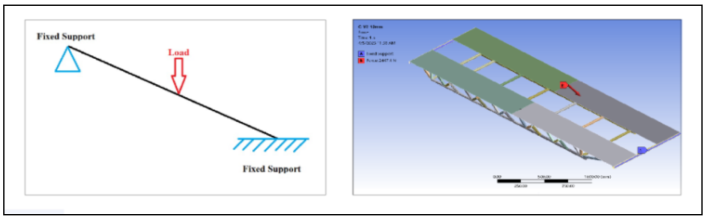


Fig 4: Boundary conditions applied to the truss

A fixed support is the most rigid type of support or connection available. It effectively constrains the member in all translations and rotations.For examining the truss, fixed supports were applied to both the front and rear sides. These supports are denoted as (B) in the figure 4. It is also important to mention that the primary point of application for the load being tested was the midplane of the truss. To achieve this, a force of 4905N was applied to the midplane of the truss using the slice command in ANSYS 2021 R2. This is represented as (A) in the figure 4.

Static structural simulations were performed on all the CAD models using ANSYS 2021 R2. Computer system with 8GB RAMwas used to carry out simulation trials.

7. Result

Results of the simulations can be plotted in terms of stresses developed, deformation and factor of safety. Figure 6 shows the sample result for truss with 60 structural members for deformation.

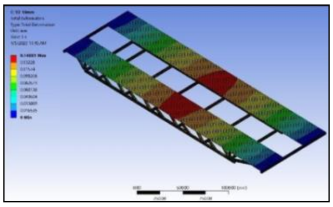


Fig 5: Result of the simulation

It can be observed that the maximum deformation in the truss is 0.14 mm. As far as the application is concerned, this amount of deformation is within acceptable limit. The results of all the simulation trials are summarised in table 3.

Table 3: Parametric values of different cross sections

Profile	Stress (MPa)	Deformation (mm)	FOS	Mass of structure (Kg)	Cost Per kg	Strength to weight ratio	Total Cost
L Angle	206.84	0.324487	1.377	90.93	67	2.27	6092.31
Square tube	81.961	0.14881	3.477	89.42	78	0.92	6974.76
Circular Tube	26.397	0.39127	10.796	134.38	88	0.20	11825.44
I Beam	25.505	0.016236	11.174	434.91	66	0.06	28704.06
C channel	26.821	0.033164	10.626	296.76	85	0.09	25224.6

The comparison of Von Moises stresses in various cross sections of the truss are as shown in figure 7:

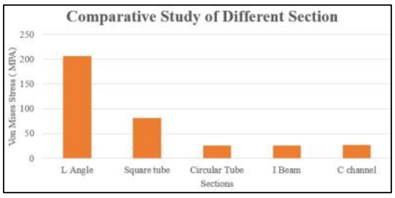


Fig 6: Stress induced in different sections

Careful investigation shows that the stress developed in L angle is 206.84 MPA, maximum among all whereas the stress developed in Circular, C channel, and I beam are minimum25.505 MPA. Square tube shows moderate stress development of 81.691 MPA.

Equivalent masses of the truss structure were evaluated using Solid Works software. The masses developed in different sections are as shown in figure 8. Mass of structure designed using all members as I beam is maximum at 434.91 kg whereas the mass developed in square tube at 89.42 kg are minimum.

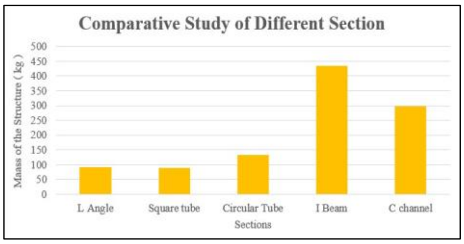


Fig 7: Mass of structure of various sections

In this study, a market survey was conducted to obtain the cost of various cross sections per kg. The prices for each section were recorded in ₹/kg and the survey was completed using local vendors. The purpose of the survey was to obtain accurate market prices for each cross section, which could then be used to inform cost analyses and purchasing decisions. After careful analysis of the market, we concluded that we are selected AISI 1008 as it is a low carbon steel with good weldability.[9] The pricing of various sections is compared as follows:

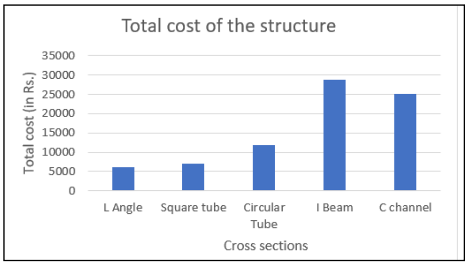


Fig 8: Total cost of various section

Pricing is an important aspect in the project.After careful investigation, we can conclude that the total cost of the I beam at Rs. 28704 is maximum where as the price of the square tube at Rs. 6974 is minimum. C channel, L angle and square tube are moderately priced. This price was calculated after conducting the present market survey. In future, this price may change depending on the demands in the market.

8. Conclusion

The comparative study of different cross sections has demonstrated the significant impact on decisions to select structural members for fabrication of truss. From circular and rectangular pipes to more complex shapes such as I-beams, each shape has its unique advantages and limitations in terms of strength, mass, weight, and cost.

The choice of cross-sectional shape ultimately depends on the specific requirements and constraints of the application, as well as the materials and manufacturing processes involved. From interpretationof data, it can be concluded that square tube is consistent in terms of mass, stress induced and cost as compared to other cross sections.Therefore, square tube has been chosen for the application purpose.

References

[1] Yash Patel, Yashversinh Chhatasia, Shreepalsinh Gohil, Het Parmar, Prof. Tausif Kauswala, "Analysis and Design of Conventional Industrial Roof Truss and Compare it with Tubular Industrial Roof Truss", International Journal of Science Technology & Engineering (IJSTE), Volume: 02, Issue: 10, April-2016, e-ISSN: 2349-784X

[2] Razvan Cazacu, Lucian Grama, "Steel truss optimization using genetic algorithms and FEA",The 7th International Conference Interdisciplinarity in Engineering, 2014, e-ISSN: 339-346

[3] *Ajinkya Karpe, Sainath Karpe, Ajaykumar Chawrai, "Validation of use of FEM (ANSYS) for Structural Analysis of Tower Crane Jib and Static and DynamicAnalysis of Tower Crane Jib Using Ansys", International Journal of Innovative Research in Advanced Engineering (IJIRAE), Volume: 01, Issue: 04, May-2014. e-ISSN: 2349-2163*

[4] *Upendra Pathak, Dr. Vivek Garg, "Optimization and Rationalization of Truss Design", International Research Journal of Engineering and Technology (IRJET), Volume: 02, Issue: 05, Aug-2015. e-ISSN: 2395-0056.*

[5] *Arumugam, K. NaveenStrength, "Failure Analysis on Structural Truss by using Finite Element Analysis",International Journal of Engineering Science and Computing (IJESC), Volume: 06, Issue: 11, Nov-2016*

[6] *Yaroslav Kovalchuk, Natalya Shynhera, Mykola Basara, "Construction of Welded Truss nodes using ANSYS Software Complex", Apr-2017, e-ISSN: 1727-7108.*

[7] *Ankit Sharma, Sumit Pahwa, "Truss bridge structure frame section analysis by using Finite element analysis", International Research Journal of Engineering and Technology (IRJET), Volume: 05, Issue: 04, e-ISSN: 2395-0056*

[8] *C.S. Abima, S.A. Akinlabi, N. Madushele, O.S. Fatoba and E.T. Akinlabi, "Experimental Investigation of TIG-welded AISI 1008 Carbon Steel", International Conference on Engineering for Sustainable World (ICESW 2020), Volume: 1107, Aug-2020*