DESIGN, ANALYASIS AND OPTIMIZATION OF JIB CRANE BOOM

Govinda Chaudhari[1], T.D. Garse[2]

[1] M.Tech (Machine design) J T Mahajan College of Engg. Faizpur Maharashtra, India.
[2]Asst.Professor(Mechanical) J T Mahajan College of Engg.Faizpur, Maharashtra, India.

Abstract - The structural action of a rectangular cantilever beam under loading is predominantly bending, with other effects such as warping, rotation, and lateral torsional buckling the study includes an investigation of stress level for load carrying capacity and bending of regular I section cantilever beam of jib crane subjected to self-weight and eccentric point load at the free end. A new design is proposed in this study to tackle the bending and increase the strength of the crane. The corrugated plate is a widely used structural element in many fields of application because of its numerous favourable properties related to resistance in out of plane twisting.

Design Modules of cranes are defined from the developed component tree of the cranes based on the available design procedures. Access to the "F. E. M. Rules" from any design procedure is fully automated by using a systematic approach of parametric modelling. The parametric model can be used for various jib crane design cases as well as further for optimization.

Finite element analysis is carried out to analyse the effect of geometrical parameters of various web shapes. Structural analysis is done to examine the influence of the section dimensions due to eccentric point load at the free end on cantilever.

Key Words: stress Analysis, Complex structure, FEA (ANSYS), Stress Bluntness, CATIA

1. INTRODUCTION:-

A jib crane is in effect a monorail that is cantilevered from its supporting members and pivoted at one end. The horizontal beam provides the track for the hoist trolley. Jib crane have three degrees of freedom. They are vertical, radial, and rotary. However, they cannot reach into corners. They are usually used where activity is localized. Lifting capacity of such cranes may vary from 0.5 ton to 200 ton and outreach from a few meters to 50 meters. Such cranes find various applications in port area, construction site and other outdoor works. For handling general cargo, lifting capacities usually 1.5 ton to 5 ton with maximum out reach of 30 meters. Jib crane provided with grabbing facilities have usually a capacity ranging from 3 ton operating 50 to 100 cycles per hour. Lifting heights may be 30 meter or more.

Jib cranes used in ship yards for lifting heavy machinery equipment, weighing 100 to 300 tons, are usually mounted on pontoons. Frequently, these cranes are provided with two main hoisting winches which can be employed singly or together to lift a load. For handling light loads may hand auxiliary arrangement localized, such as in machine shops. Column mounted jib cranes are commonly used in packaging industry. The size of the crane can be visualized from the height of the operator.

The use of variable cross section beam has been increasing in the steel construction industry. This is because of their ability to increase stability of the structure, and sometimes to satisfy architectural and functional requirements in many engineering structures. Tapered beams are widely used in the modern constructions mainly due to their structural efficiency. At present the web tapered thin walled I beam is one of the most popular tapered beams used in practice. The strength of laterally unrestrained thin walled beam is frequently governed by the lateral buckling failure, and hence extensive studies were focused on the lateral buckling of thin walled beams.

Cantilever I section beam is a component of the jib crane used in Diesel Loco shed which is located in Shivajinagar, Pune. This component is liable to fail due to lateral torsional buckling, the design of the crane for industry aims to ensure the safety for selected components.

2. PROJECT OVERVIEW:-

Jib crane boom is subjected to bending stresses mostly. Due to these stresses jib crane booms tends to fail under bending. To reduce the load on the jib crane boom we have changed the sections of web of the I beam of the Jib crane boom. In this study we have worked on the Jib Crane Boom specimens. Dimensions of the Jib Crane Boom will be found through references and catalogues.

Initially CAD model of an existing Jib crane boom is created on CATIA V5, then it is analysed through finite element software (ANSYS). After analysing the existing Jib Crane boom, analysis of different web sections is carried out.

After getting suitable results, model of the Jib crane boom is fabricated and then it is tested under UTM subjected to specific boundary conditions, tested results were compared with the FEA results.

3. METHODOLOGY:-

We can know how the CAD model which is to be prepared. The conditions required for applying various constraints and how the loads are applied is briefed about in the technical papers referred.

Getting input data on dimensions of I-beam from market/available resource/journal papers etc.

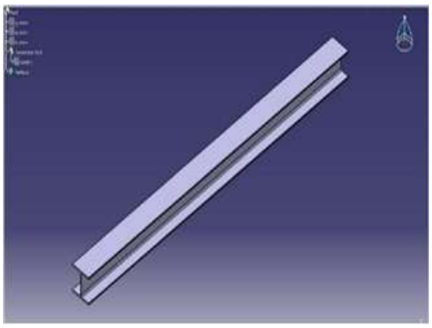
Figure3.1: CAD Model

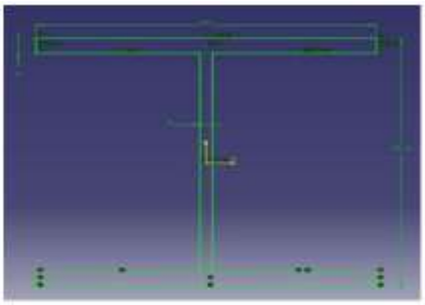
Figure3.2: Sketch of Model

Determination of loads:

- Studying various loads that are acting on the component.
- Determining the magnitude and direction of loads.

a. Capacity of crane (W)= 500Kg
b. Load acting on the boom (F) = W x g= 500 x 9.81=4905 N in downward Direction. i.e. (-Z) direction over 100 mm span at free end.
c. Fixed support = Master Dia (E)= 8" = 203.2mm ~ 204mm.

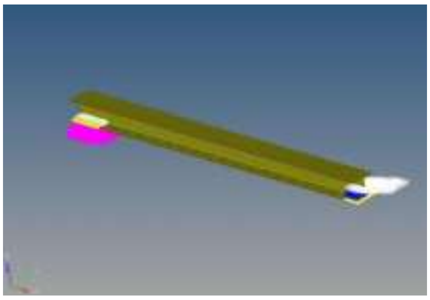
Figure3.3: Boundary Conditions of Boom

4. EXPERIMENTAL SETUP:-

The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. If an extensometer is not fitted, the machine itself can record the displacement between its cross heads on which the specimen is held. However, this method not only records the change in length of the specimen but also all other extending / elastic components of the testing machine and its drive systems including any slipping of the specimen in the grips.

Once the machine is started it begins to apply an increasing load on specimen. Throughout the tests the control system and its associated software record the load and extension or compression of the specimen.


Figure4.1: Universal Testing Machine

For testing we have fixed one end of the I beam to a rigid block and then as it's a cantilever loading, load is applied on the another end as per calculations, as the load incresed from zero to the design load; Load versus deflection graph is generated by data acquisition system sensed through load sensors in UTM the deformation of I Beam is then validated through FEA Results.


Figure 1.2: Data acquisition system


Figure4.3: Test Bed

4.1 Experimental Validations:-
Manufacturing Considerations of Proposed Design:-

As we are aware that manufacturing of I Beam is tough extrusion process but in out Prototype model we are making a scale down model because of testing rig bed size limitations of 800mm, for the I sectioned beam we have considered the following assumptions. For the scale down model accordingly loads will be reduced on the basis of scale down ratio. The middle I segment of beam is made rectangular profile by milling also by maintaining the dimensions as per drawing and then we have done the welding on the upper and lower plate.

For testing we have fixed one end of the I beam to a rigid block and then as it's a cantilever loading, load is applied on the another end as per calculations, as the load incresed from zero to the design load; Load versus deflection graph is generated by data acquisition system sensed through load sensors in UTM the deformation of I Beam is then validated through FE Results.

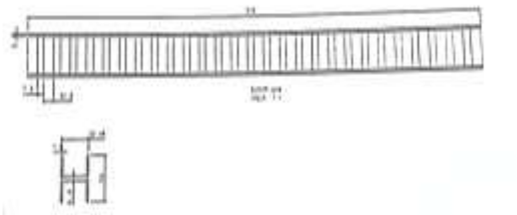

Figure4.1.1: Proposed scale down model of boom

Experimental Test Results:-

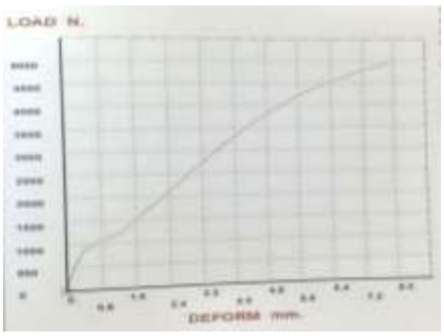

Figure4.1.2: Experimental Test Results

Experimental Results:-

Sr. No	Design	Deformation (mm)
1	Original Design	8.18
2	Proposal 4	7.7

Table4.1.1: Experimental Results

5. FINITE ELEMNT MODELLING OF BOOM:-
Finite Element Method:-

The finite element method (FEM) sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

Following are the results displayed for stress and deformation (SS):

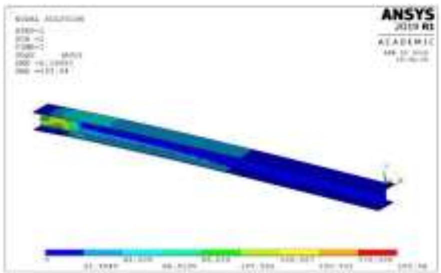

Figure5.1: von-mises stress for boom

Stress value for boom is 193.54 N/mm^2 which is well below the critical value. Hence, design is safe.

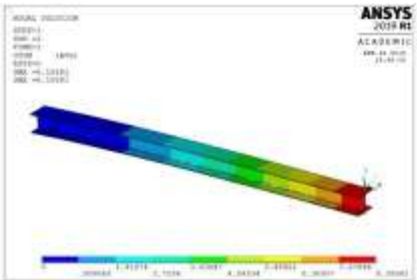

Figure5.2: Displacement result for boom

Finite element analysis summary:-

The maximum stress value for I section boom is coming out to be 193.54 N/mm^2 which is within the safety limit; we have scope for optimizing the topology without affecting its structural behaviour rather increasing its load bearing capacity. The maximum displacement value is 8.188mm which we must refer.

Finite Element Results:-

Stress and Deformation plot for Proposal 1:-

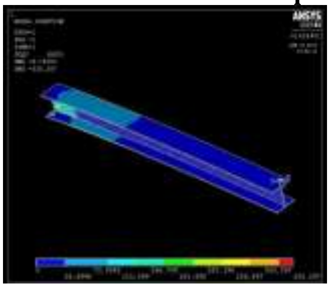

Fig5.3: Stress plot for 1

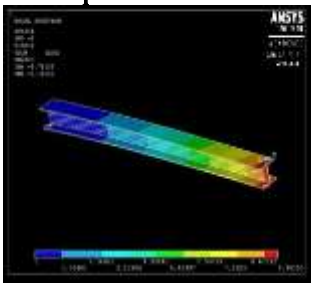

Fig5.4: Deformation plot for 1

Stress and Deformation plot for Proposal 2:-

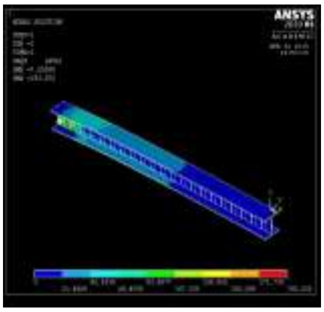

Fig5.5: Von-mises stress for 2

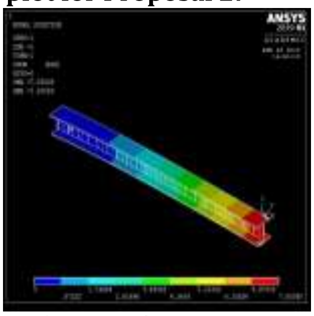

Fig5.6: Displacement result for 2

Stress and Deformation plot for Proposal 3:-

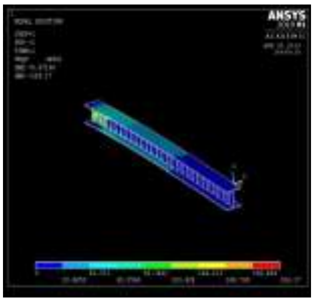

Fig5.7: Von-mises stress for 3

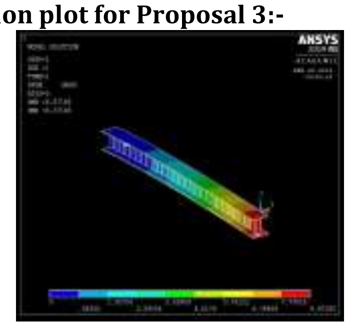

Fig5.8: Displacement result for 3

Stress and Deformation plot for Proposal 4:-

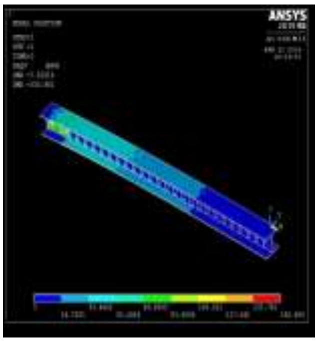

Fig5.9: Von-mises stress for 4

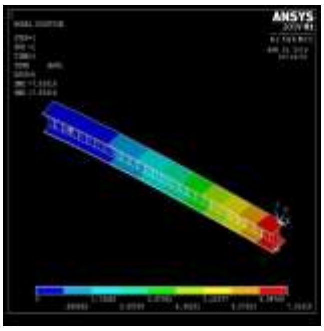

Fig5.10:Displacement result for 4

Stress and Deformation plot for Proposal 5:-

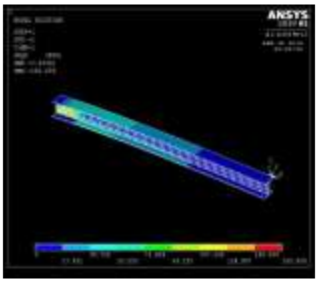

Fig5.11: Von-mises stress for 5

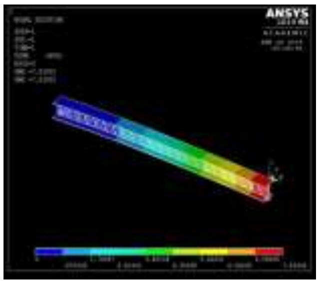

Fig5.12: Displacement result for 5

Results obtained from the ANSYS:-

Sr. No.	Design	Stress (Mpa)	Deformation (mm)
1	Original Design	193.5	8.18
2	Proposal 1	330.297	9.76
3	Proposal 2	193.202	7.85
4	Proposal 3	186.17	8.67
5	Proposal 4	150.484	7.81
6	Proposal 5	160.65	7.83

Table5.1:Results obtained from the ANSYS

The above results are showing stress and deformation of Jib Crane from all iterations. It is observed that the proposal 4 having lesser values for stress and deformation compared to all the proposals and original design.

The maximum stress value is coming out to be 193. 54 N/mm^2 which is within the safety limit. We have scope for optimizing its topology without effecting its structural behaviour rather increasing its load bearing capacity. The maximum displacement value is 8.188 mm which we must reduce.

As it has been seen from the Finite Element Results that the Proposal 4 design is having lesser stress values for the same loadings also the deformation is lesser compared to all other proposals and original design.

Comparison of Finite Element Results with the Experimental Results:-

Sr.No.	Deformation (mm)		Error(%)
	FEA	Experimental	
1	7.81	7.7	3.5

Table 5.2: Comparison of Results

5. CONCLUSION AND RESULT:-

A New design approach of the beam shape has been proposed to reduce the deformation and stresses generated due to direct loading, we had scope for optimizing its topology without affecting its structural behaviour rather increasing its load bearing capacity. So we have made possible topological shape changes in the web, various web profiles has been designed for the boom, keeping the analysis on the original design and five other proposed designs, we concluded that the proposed model 4 i.e. rectangular web section design of boom is the best amongst all. It showed 150.484 Mpa stress along with 7.81mm deformation.

Now the experimental validation is carried out using UTM and the graph thus produced gives us 7.7mm deformation. Thus generating 23% less stress and 5% less deformation as compared to the original web design. This value is close to the obtained values of stress and deformation from the FE Results, we obtained the 3.5% error in FE Results and Experimental results.

Therefore, from the experimental and Analysis we conclude that the rectangular web section design increases the load bearing capacity of the boom with lesser deformation and it is best amongst all the proposed designs.

REFERENCES:-

1. Mr.Faud Hadzikadunic, "An Analysis of Jib Crane Constructive Solution In Exploatation", 12th International Conference TMT 2008, Istanbul, Turkey, 26–30 August, 2008.
2. Gerdemeli I., K. Kurt S"Design and Analysis with Finite Element Method of Jib Crane", Faculty of ME Istanbul Technical University – Turkey.
3. Chirag A. Vakani, "Analysis and Optimization of 270° Jib Crane Deflection", IJSRD,Vol.2, Issue 10, 2014, ISSN (online): 2321-0613.
4. Subhash N.Khetre "Modelling and Stress Analysis of Column Bracket for Rotary Jib Crane"(IJMET), ISSN 0976–6340(Print), ISSN 0976–6359(Online),Volume5, Issue 11, November (2014), pp. 130-139
5. Ajinkya Karpe"Validation of Use of Fem (Ansys) For Structural Analysis of Tower Crane Jib and Static and Dynamic Analysis of Tower Crane Jib Using Ansys"(IJIRAE) ISSN: 2349-2163, Volume 1 Issue 4 (May 2014).