International Journal of
Emerging Research in Management &Technology
ISSN: 2278-9359 (Volume-4, Issue-3)

Research Article | **March 2015**

FEA Analysis for Customized Overhead Gantry

Ankur Kulshresta
Mechnaical Engg. PAHER, Udaipur,
Rajasthan, India

Anurodh Prashant
M.Tech. Final Year.AHER, Udaipur,
Rajasthan, India

Abstract—

This particular literature review introduces a concept of customized overhead gantry crane being used for very specific use for ISRO but under n-nos. of constraints. It includes the design part of particular overhead beam which undergo the stresses during loading, unloading and transverse movement of the test load and the deflection measured thereafter and reached to the safest gantry.

Keywords— Load Center, Finite Element Analysis, Shear Force bending moment diagrams, Calculation criteria, Deflection.

I. INTRODUCTION

The criteria to publish this paper is for review on gantry structural analysis for load, transverse travel and unload the items by overhead gantry crane, incorporating the various structural components i.e. angles of different sizes, "I" section beams, channels, and quality of the welding with reference to the welding materials being used, particularly by means of Finite Element Analysis (FEA) technique. The overhead crane beam is subjected to various types of load composition of supporting structure, grouting base, wind pressure, travel kinetic energy, static load moment, shear force & bending moment diagram relates to the various components in question. The final load apparent at the girder i.e. "I" Section beam is the ultimately result & strength of all the associated components which ultimately results to the girder as a whole. The present study and the research are related to the correction and improvement to the strength structure of overhead crane girder. The main goal of this study is to avoid resultant failure of the crane as a whole lastly to the girder failure. Finite Element Analysis (FEA) software provides the solutions to all the structural components and the overhead crane girder failure problem. During this study the concluding part is the use of Finite Element Analysis (FEA) technique using different types of approach. Main term: - "I" Section beam-Girder, Overhead, complete Structural analysis & Finite Element Analysis (FEA).The crane in question is mechanical equipment for load, transverse travel and unloads the desired load with ultimate safety and minimum bending stresses. The load shall be moved manual with the help of chain pulley block moving on mongered beam of 400 MM size. A crane with a single girder movable trolley moves to & fro with the desired load up to its specified capacity and the trolley holds the load can raise & lower it as and when required depending upon the requirement, carrying a movable trolley mechanism, and traveling on an overhead fixed runway structure is known as overhead crane. Load lifting & handle to its desired location is an important concept of any manufacturing and supply chain system and the industry involved in manufacturing the items related to this feature keeps the complete track record and the essential data to measure its strength time to time with finite element analysis (FEA) and incorporating it to the design criteria as and when required. The load transportation through the overhead gantry is an important practical aspect in the design of new manufacturing and distribution systems and research into better material handling systems and practices is important. The load handling users are basically innocent and are not in track of using the safety aspects delivered to them, along with the gantry, and even they
are not maintaining the safety instructions passed to them thus it becomes very essential to calculate the FEA very accurately to use the equipment safely at various sites in crucial condition and rigorous loading events. Main component of overhead crane is girder beam which transfers load to its structural member. In past the facility of finite element analysis (FEA) was not available and the safe load calculation were calculated theoretically by shear force and bending moment diagram at every member and then the final stress calculation were made. Now in present era the facility for calculation of the Finite Element Analysis (FEA) technique is available and the process becomes very easy now. Another process for calculation of FEA is the analytical analysis (ANSYS) in which the exact calculation by means of the soft-ware can be done.

II. FINITE ELEMENT ANALYSIS

This method which is used for Finite Element Analysis (FEA) is known as Finite Element Method (FEM).This is the versatile technique among various techniques, Finite Element Method (FEM) is used cause widely available in various user-friendly commercial software, and these programs have modular forms in accordance the stages of method. Any complex geometry can be analyzed by Finite Element Method (FEM).The method can be used for solving both stresses and displacements. Finite Element Method (FEM) provides the approximate the solution of complete domain for the study is being done, The ANSYS structural analysis software is suitable and trusted by all the organizations around the world to solve the complex structural engineering problems rapidly, with ease. FEA analysis (finite element) tools from ANSYS provide the ability to simulate every structural aspect of a product:-

1. Linear static analysis that simply provides stresses or deformations.

© 2015, IJERMT All Rights Reserved

2. Modal analysis that determines vibration characteristics.
3. Advanced transient nonlinear phenomena involving dynamic effects and complex behaviors

All advanced users, from designers to advanced experts, can be benefited from ANSYS structural analysis software. The fidelity & trust of the results is achieved through the wide variety of material models available, the results depends upon the quality of the elements available, the robustness of the solution algorithms, and the ability to model every product — from single parts to very complex assemblies with hundreds of components interacting through contacts or relative motions.

ANSYS FEA tools also offer unparalleled ease of use to help product developers focus on the most important part of the simulation process: understanding the results and the impact of design variations on the model.

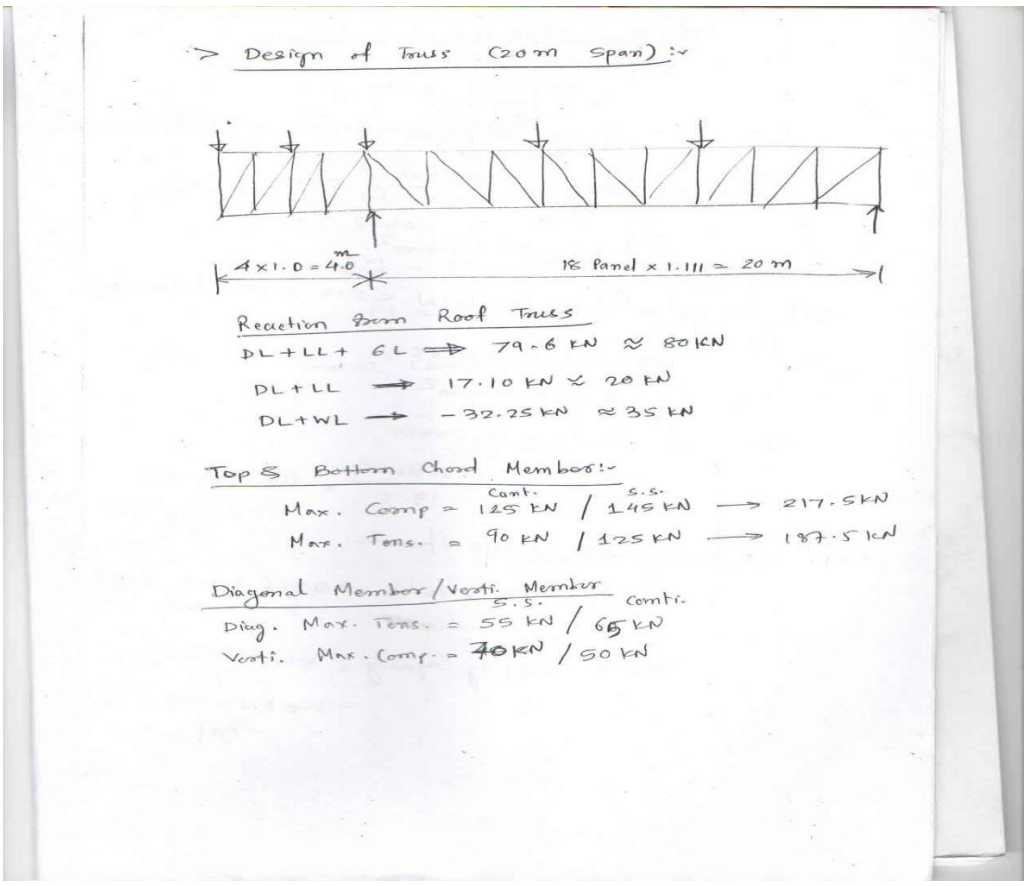
It does not matter whether the industries are increasing the size and the larger models are made available for engineering simulation. Our structural software incorporates parallel algorithms for faster computation time for these large models. For more information about our broad structural analysis capabilities and insight into how customers leverage these tools.

I. As a result the complete assembly of discrete finite elements interconnected at nodal points on element boundaries. The approximate calculation of the solution is fundamentally formulated over each element matrix then it is assembled for obtaining stiffness matrix, Displacement and force vectors of the entire domain can also be obtained. Different types of elements can be used in Finite Element Method (FEM) for acquiring best result and accurate shape function.

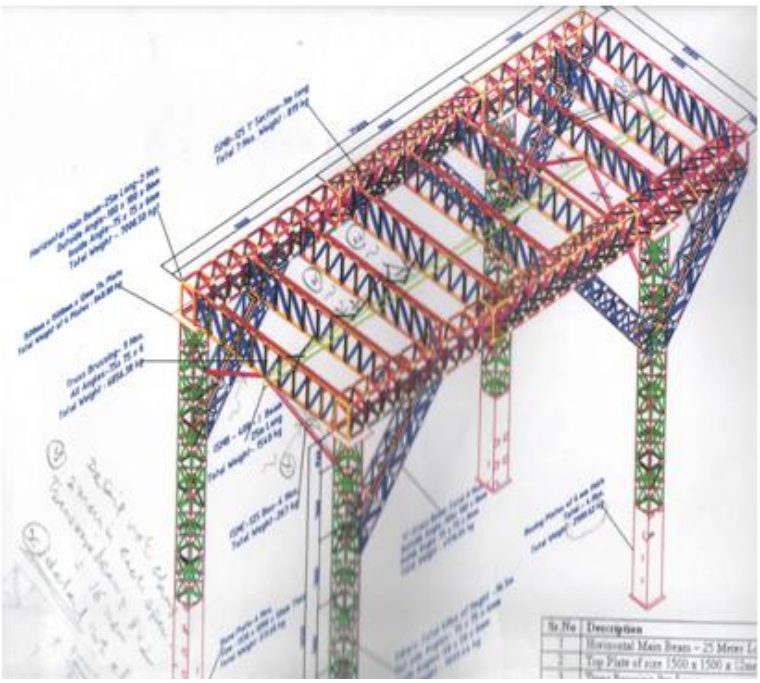
II. LITERATURE REVIEW After the complete studies the study for the literature review is done, the information regarding structural analysis of overhead crane bridge girder can be evaluated. In past also various studies have been done for different structures and different arrangement of the beam girders.

III. Research journals, books, printed or online conference article were the main source in project guides. Camelia Bretotean Pinca, Gelu Ovidiu Tirian and Ana Josan (2009) had used shell type elements with three or four nodes per element in order to find out the best sizes for resistance structure in tension and deformation state.

Apart from many more software here the software used is the cosmos software, for analyzing the tension, bending & deformation conditions for the study of load bearing structure of an overhead crane girder used for the gantry. The composite structure was thoroughly welded with the best material welding rods, for mild steel welded bracing build structural and general steel section of continuous welding. Maximum equivalent tension calculated according to the theory of the specific form modifying energy (the theory of von misses). Ultimately the Finite Element Analysis (FEA) is applied to solid model of load bearing structure. The tension was distributed within the load bearing section of the structure of the crane gantry more accurately. All features causing tensions and deformations in the subject structure were described mathematically by differential equations, The mathematical calculations are also presented within this matter as fig No.1



thus like A Review on Structural Analysis of Overhead Crane Girder Using FEA Technique, A. Prashant Research scholar evaluated the stress supporting areas and pointed out the critical areas, where we have to pay more stress and those areas can be strengthen for solidity and bearing capacity of the strength structure for the load in transverse direction. After comprehensive analysis of the stress fields, the concluding part were decided that there are two critical areas which we have to take into consideration. 1. The connection between the longitudinal beam I and top frame beam. 2. The supporting frame to be strengthens by placing the M.S. plate up to the l/4 height to reduce the slenderness ratio. Fig 2



Discrete resistance structure of Crane load bearing with structural type bracing is subjected to finite element.
Following few scientists has done qualitatively studies for new finite element analysis and the details are described here under:-
Ismail Gerdemeli, Serpil Kurt and Metinyıldırım M.Sc. (2010) had used new techniques rather than using old Finite Element Methods (FEM). The study was carried on a crane being used to lift and handle the rubber tyre container stacking, and the studies were done and the results were summarized and then it was modeled. In addition to this, the study for stress and deformation analysis of the crane bridge girder and buckling analysis was done, there after the crane legs were taken up for detailed study and data were collected. ANSYS workbench was also used for Finite Element Method (FEM) and modeling was done on Autodesk Inventor 2010 program. Comparison of calculations regarding Stress, deformation and buckling analysis were done by the scholar. There is no significant difference between the analysis and calculation result for the stresses and deformations. Therefore analysis result can be taken into consideration. The main aim of this work is to achieve best Finite Element Methods than conventional methods for getting advantage of new methods. The result as achieved by the calculation indicates that the stress values remain under the yield strength of the steel which was used for Crane Bridge and legs. They concluded that it is new method but it gives better result than conventional method. Fig 2 Stress and Deformation analysis result of the bridge girder using Finite Element Analysis (FEA) technique, C. Alkin, C. E. Imrak, H. Kocabas (2005) had taken too many finite elements to get desired result. Experiment was conducted with capacity having 10 Ton and 20 M span length on overhead crane. Here crane was modeled in both solid and surface way. Finite element meshes with 4-Node Tetrahedral and 4-Node Quadrilateral Shell Elements were generated from the solid and shell models, respectively. After a comparison of the Finite Element Analysis (FEA), the conventional calculations and performance of the existing crane, the analysis with Quadratic Shell elements was found to give the most realistic results. As a result of this study, a design optimization method for an overhead crane is proposed. Unlike the other studies carried out previously, Shell elements in finite element modeling of an overhead box girder have been examined. Here author had examined whole overhead crane bridge. The ratio of length to thickness of the element used in modeling overhead crane box girder Is higher than 20.Therefore, In order to show the accuracy of the analysis of the overhead crane bridges.

III. CONCLUSION

The design details received from the FEA is the accurate analysis for design of the safest mode for a customized gantry crane, and the design criteria depends not only the study by FEA but all possible constraints were taken in mid and a final conclusive way was achieved, and ultimately the deflection received was well within the scope of the design and at rated load too. In all, it can be concluded that the developed gantry is useful for customized gantry with unlimited constraints, this activity off course provide a second opinion to all of the manufactures and overcome the disadvantages of general gantry manufacturing.

© 2015, IJERMT All Rights Reserved

ACKNOWLEDGMENT

The authors acknowledge Ankur Kulshresta for his useful guidance in the study of design of the customized overhead gantry. The authors are also grateful to the for providing all information related to Finite Element Analysis.

REFERENCES

[1] A Review on Structural Analysis of Overhead Crane Girder Using FEA Technique.

[2] Overhead and Gantry Cranes (Top-Running Bridge, Single Girder,Underhung Hoist”)] (for latest ASME standards, see http://catalog.asme.org/home.cfm?Category=CS).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.

© 2015, IJERMT All Rights Reserved