ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 4, July 2013

A Review on Structural Analysis of Overhead Crane Girder Using FEA Technique

Pratik R. Patel, V.K.Patel
Research scholar, Assistant Professor

Abstract:- 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 load. 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.

Index Term: **- Girder, Overhead, Structural analysis, Finite Element Analysis (FEA).**

I. INTRODUCTION

A crane is mechanical equipment for lifting and lowering a load and moving it horizontally, with the hoisting mechanism an integral part of the machine. A crane with a single or multiple girder movable bridge, carrying a movable trolley or fixed hoisting mechanism, and traveling on an overhead fixed runway structure is known as overhead crane. Material handling is a vital component of any manufacturing and distribution system and the material handling industry is consequently active, dynamic, and competitive. Material handling is an important practical consideration in the design of new manufacturing and distribution systems and research into better material handling systems and practices is important. Material handling uses different equipment and mechanisms called Material handling equipment. Main component of overhead crane is girder beam which transfers load to its structural member. In the early stage, there was few software available used Finite Element Analysis (FEA) technique. In present, Structural analysis of girder can be done by different software using Finite Element Analysis (FEA) technique

II. FINITE ELEMENT ANALYSIS

Method which is used for Finite Element Analysis (FEA) is known as Finite Element Method (FEM). Among various techniques, Finite Element Method (FEM) is used cause widely available in various user-friendly commercial software, these programs have modular forms in accordance the stages of method [1]. Any complex geometry can be analyzed by Finite Element Method (FEM). It can solve both stresses and displacements. Finite Element Method (FEM) approximates the solution of entire domain under study as an assemblage of discrete finite elements interconnected at nodal points on element boundaries [3]. The approximate solution is 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.

III. LITERATURE REVIEW

Literature review is one of the scope studies. It helps in a way to get the information regarding structural analysis of overhead crane bridge girder. From the early stage of project various literature studies have been done. 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. Here cosmos software was used for analyzing the tension and deformation state of the resistance structure of an overhead crane bridge. This was performed in iron and steel department of continuous casting. The maximum equivalent tension calculated according to the theory of the specific form modifying energy (the theory of von misses). Finite Element Analysis (FEA) is applied to solid model of resistance structure. They had distributed the tensions within the resistance structure of the crane bridge more appropriately. All features which cause tensions and deformations in resistance structure were described mathematically by differential equations. In this way like

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 4, July 2013
evaluation of stress state and pointing out the critical areas we can increase solidity and bearing capacity of the strength structure for the rolling bridge. After analyzing the stress fields, they were realized by the fact that there are two critical areas that we have to take into consideration. 1. The connection between the longitudinal beam I and the right-end beam. 2. The area of the right-end beam near the global axis system.

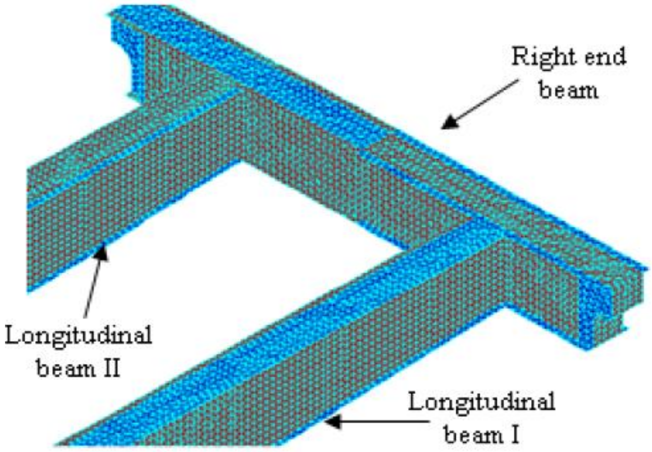

Fig 1 Discrete resistance structure of Crane Bridge with shell type finite element [1]

Ismail Gerdemeli, Serpil Kurt and Metinyıldırım M.Sc.(2010) had carried out the research on developing new Finite Element Analysis(FEA) technique and here they had used new techniques rather than using old Finite Element Methods(FEM). All calculations of elements related rubber tired container stacking crane were done and then it was modeled. In addition of this, they stress and deformation analysis of crane bridge girder and buckling analysis of the crane legs were performed. 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 author. 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. Result shows that 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.

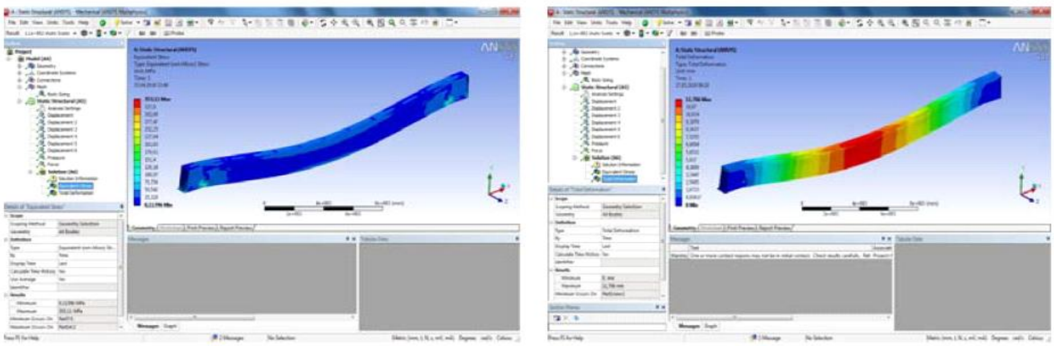

Fig 2 Stress and Deformation analysis result of the bridge girder using Finite Element Analysis (FEA) technique [2]

C. Alkin, C. E. Imrak, H. Kocabas (2005) had taken too many finite elements to get desired result. Experiment was conducted with capacity having 35 Ton and 13 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, A Four-Node Quadratic Shell Element is used instead of the Four-Node Tetrahedral Element for Finite Element Analysis (FEA).

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 4, July 2013

Vlada Gasic, Milorad Milovancevic, Zoran Petkovic(2010) had considered dynamics of an overhead crane with moving hoist with Finite Element Analysis(FEA). The weight of hoist and payload was treated as moving force, ignoring inertia effects. The trolley has velocity which is function in time. Illustration of equivalent nodal forces is applied at uniform discretizied beam subjected to a single load moving at prescribed velocity. It is obtained frequencies and dynamic deflections of structure of bridge crane finite element model. The results show that standard finite element packages, apart from modal analysis and time history analysis, can be used to describe response of structure to time-variant moving loads. This paper deals with moving load problem in dynamic analysis of bridge cranes with Finite Element Analysis (FEA). There is presented accepted technique for using standard finite element package to analyze the dynamic response of structure to time-variant moving load. They are incorporated in FE package SAP 2000 with data from computer program created according to presented algorithm. The aim of this work is to emphasize the technique for describing moving load with standard FE packages. It is convenient to be applied at models of material handling machines with various structural types. But, it can be used only where an inertia effect of moving hoist is neglected which is referred as moving mass problem in structural dynamics.

M. Euler, U. Kuhlmann (2011) had presented in his paper about multi axial fatigue problem using Finite Element Analysis (FEA) technique. Multi axial fatigue problem is created due to wheel load and block rail fastened by longitudinal fillet welds. At the point of wheel load application the top region of the runway girder is subjected to a stress field comprising local stress components induced by the concentrated load. Here author had tested two crane runway girders under proportional multi axial loading, i.e. under flexural bending and stationary wheel load, both simultaneously pulsating. The paper presents the fatigue evaluation of these tests using different local concepts. Generally notch stresses are hardly measurable. So that author had used Finite Element Analysis (FEA) for finding out the notch stresses. Crane runways are subjected to a complex multi axial state of stress due to geometric and metallurgic notch effects and the introduction of concentrated loads. The amplitudes of the stress components do not occur simultaneously resulting in an alternating direction of the shear stress that has been recognized to control the crack initiation.

Ozden Caglayan, Kadir Ozakgul, Ovunc Tezer, Erdogan Uzgider (2010) had studied about fatigue life of crane runway girders. Detailed finite element models of the crane runway girders were prepared using shell and beam elements. Here Finite Element Analysis (FEA) technique is used for calculating remaining fatigue life. Quasi-static load tests were conducted with the help of overhead cranes that travelled with crawling speed. Strain data was collected by using transducers mounted on preselected locations of the crane runway girders. These data were then used to refine the finite element models. Numerical analyses by means of the calibrated finite element models were performed to evaluate the remaining fatigue life. It was found that due to lack of continuity of vertical stiffeners to upper flange, fatigue life of the crane girders is exceeded. To overcome this problem, Fillet welding of the stiffeners to flanges is suggested and this modification is verified by carrying out necessary calculations.

Ismail Gerdemeli, Serpil Kurt, Hasan Onur Alkan (2010) had determined height of the crane, distance between the rails, the lifting height, speed of the crane and speed of the transmission components for gantry crane used in shipyards. Construction geometry was analyzed in Abaqus software program. Three dimensional geometry was created on cad software and then modeled with Finite Element Method (FEM). Then crane is tested under various loads. During them work, they found that if any component has an unpermitable stress value, the thickness of the sheets should be increased or suitable supports should be added. For Finite Element Analysis (FEA), the element type used in the model is 4 node quadratic shell element. Hence by this way they had prevented material waste. After this study, construction is now more reliable, light and durable. This study is very important with respect to aspects like low cost design and low design duration.

M.A.NASEER (2009) had found the fact during his study that crane structures may have many forms of space structures. it is not economical and logical to perform experimental model analysis for such type of structures. Finite Element Method (FEM) was applied to small space structure. In this research Finite element models are used as a tool to investigate barrage structure regarding the crane dynamic load. For avoiding the coincidence between forced and natural frequency of the crane system, Finite Element Analysis (FEA) is capable to anticipate dynamic characteristics and behavior of crane. He had investigated that running velocities of crane must be kept as minimum as possible.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 4, July 2013

IV. CONCLUSION

Finite Element Analysis (FEA) is a 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.

REFERENCES

[1] Camelia Bretotean Pinca, Gelu Ovidiu Tirian, Ana Josan, "Application of Finite Element Method to an Overhead Crane Bridge", Issue 2, Volume 4, 2009.

[2] 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.

[6] Ozden Caglayan, Kadir Ozakgul, Ovunc Tezer, Erdogan Uzgider, "Fatigue life prediction of existing crane runway girders", Journal of Constructional Steel Research, 2010.

[7] Ismail Gerdemeli, Serpil Kurt, Hasan Onur Alkan, "Main girder beam design and finite element analysis of 2/160 gantry crane",14th International Research/Expert Conference, Trends In The Development Of Machinery and Associated Technology, 2010.