ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2015

NUMERICAL ANALYSIS OF THE STRESS THAT OCCURS IN THE CRANE STRUCTURE OF 325 TF METALNA TO MANOEUVERE THE BLOCK SECTIONS OF THE SHIP IN THE DAMEN SA SHIPYARD, GALATI

Mădălina Cristina Onica
"Dunărea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:cristinaa.onica@yahoo.com

Mădălin Marius Costea
"Dunărea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail: costeamadalin18@yahoo.ro

Costel Iulian Mocanu
University"Dunarea de Jos" of Galati,
Faculty of Naval Architecture, Galati, Domneasca
Street, No. 47, 800008, Romania,
E-mail: costel.mocanu@ugal.ro

ABSTRACT

Cranes are mainly used for lifting weights everywhere. When desigining cranes, several factors must to be takan into consideration: weights, shocks, wind and ground. It means that design process requires very accurate calculations and several verifications. To shorten design process, a lot of metohods can be used. The use of the numerical method gives the solutions in short time. The main method used is Finite Element Method (FEM). This method gives designers an exact overview of stress and strain which occur in structures under loads. The aim of the study is to determine the state of stress occurring in the structure of Damen Crane Metalna SA Galati ,which can lift and manoeuvre the block sections of the ship up to 320 tons. This analysis is made up with the help of the FEMAP software which enables us to make evolved finite items – shell-type and beams. As a result, it has been seen that FEM is the most practical and easy to use method during the design process.

Keywords: finite element analysis, crane.

1. INTRODUCTION

Cranes are mainly used for lifting heavy structures. The main task of the overhead crane is to handle and transfer different heavy loads from one position to another. Thus they are used everywhere the uman hands cannot lift loads, even in areas such as automobile plants and shipyards.

Most of the time, the structures resistance is not statically determined. Solving the stress and strain problems is very hard using analytical methods. By using approximative methods, calculation time can be reduced. There are several numerical methods which can be used to solve stress and strain problems. FEM is considered to be one of the most used methods .

In this study, based on a FEM model, the strength of the 320 tonnes crane from Damen Shipyards Galati is checked.

FEM, which uses automatic data processing, allows us to study the stress more accurately. The equation of this method allows us to calculate the displacements of nodes and to know the stress everywhere in structure.

This analysis is made with the help of the method used for producing finite items and the FEMAP calculation software. The results we have obtained give a complete view about the stress and deformation state of the resistance structure of a crane, and highlight all the details regarding the operation loads. Based on those results, we are able to produce some details about the lack of resilience of the resistance structure, and to make some decisions about how to improve the bearing capacity of the equipment, and, if this is the case, to perform the best size optimization. This is possible only if we do not exceed the limits of the acknowledged resistance, in order to redesign the resistance structure of the crane, while the material consumption is highly reduced.

In many engineering applications in the area of field computation, the numerical models are based on the finite element method. From all numerical techniques, FEM is used due to the availability of many user-friendly commercial software programs. These programs have a modular form in accordance with the stages of the method: pre-processing, processing and post-processing. The greatest task in any finite element program is generally the preparation of the input data. There is a large amount of input data, both geometrical and physical. This method can analyse any geometry and solve both stress and displacements. FEM approximates the solution of the entire domain under study.

The FEM model results by assembling the discrete finite elements, connected each other at the nodal points of the structure.

The approximate solution is formulated over each element matrix and thereafter assembled to obtain the stiffness matrix, displacement and force vectors of the entire structure domain. The result is a very big equation system which uses iterative numerical methods to obtain the solution as displacements.

2. NUMERICAL ANALYSIS

FEM witnessed a quick development in tandem with the increase of the computational capacities and it has been enforced as a general numerical method of solving engineering problems from different areas, including the naval domain.

In general, a structural analysis has the following steps: the objectives settlement, the type and the size of the analysis; the modelling of structures and the boundary conditions; the settlement of the type of the elements and the modelling of the loads; the analysis and the evaluation of the results.

The type and the size of the analysis depend on the nature of the structural response that is to be obtained. In general, at the structural analyses the following categories of responses can be obtained: stresses and deformations for a loading case, eigen vibration modes, the behaviour of structural elements at buckling (stability loss).

For the analytic determination of the stresses and static and dynamic deformations, inside the structure, generated by external and thermal loads, a solution of the elasticity theory equations has to be obtained, with the accomplishment of the boundary conditions enforced to the structure. The structural analysis through FEM requires the use of the same equations of the elasticity theory.
FEM fundamental equation is:

$$[K]\{U\}=\{Q\} \tag{1}$$

where $[K]$ represents the rigidity matrix and $\{U\}$ is the displacement vector and $\{Q\}$ refers to loads.

For calculating the resistance structures using this method, it is highly important to use a discretization operation. This analysis is made up with the help of the FEMAP software which enables us to make evolved finite items – shell-type (with three or four nodes per element) and beams. Specialized literature describes that in the case of bi-dimensional structures, if we use triangular finite elements, they provide a lot of possibilities to approximate the geometrical design of the resistance structure; if we use quadrilateral elements, we see they shape up the tension distribution better. In the case of complex resistance structures, the results of the analysis that uses FEM (the values of tensions and shifting) depend mainly on the discretization solution we choose.

In studying the stress that occurs in the crane structure 320, we have followed the following steps : structure mesh, defining loads and boundary conditions, meshing tasks, defining material properties.

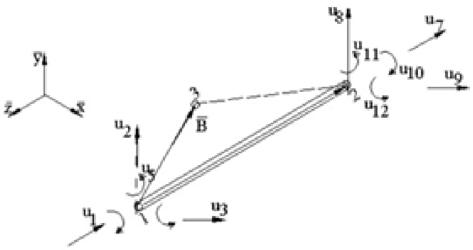

Fig. 1 Beam elements in space

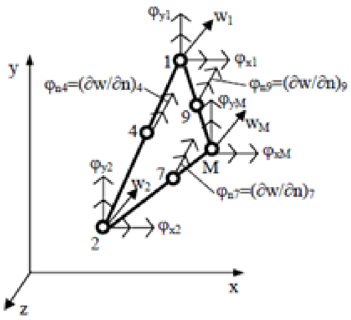

Fig. 2 Triangular finite element of shell type thin plate -release degree presentation

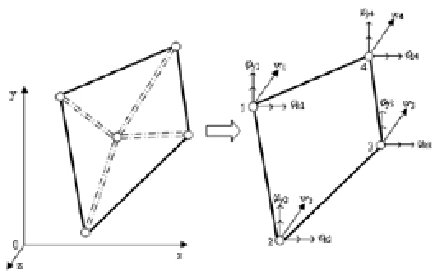

Fig. 3 Quadrilateral finite element

3. APPLICATION OF FEM TO AN OVERHEAD CRANE

This analysis is made up with the help of the FEMAP software which enables us to make evolved finite items – shell-type with three or four nodes per element and beams.

A beam element will be assigned to a straight bar of uniform transversal cross section able to support axial forces, bending moments around the principal axes of sectional transversal plane and torques around the central axis.

The finite element used in the analysis is presented below.

4. CREATING GEOMETRY

The very important stage of the study includes the analysis of the crane using finite element method. For the analysis of the crane structure with this method, firstly the 3D solid model of the crane must be generated in AUTOCAD and then imported in FEMAP. Figure 4 shows the geometry of the model made in AutoCAD and the geometry of FEMAP.

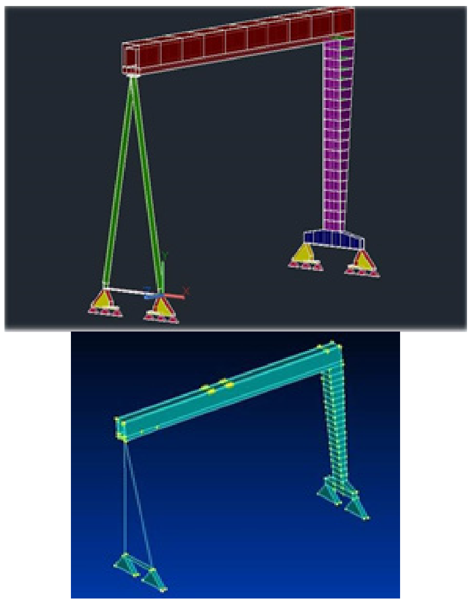

Fig.4 Crane geometry in AutoCAD and Femap

Table 1. Main dimensions

Main dimensions	
Length overall	81.150 mm
Length between legs	76.000 mm
Right leg	51.000 mm
Left leg	54.200 mm

5. ANALYSIS OF THE CRANE USING FEMAP

The models, which were generated via AutoCAD software, were transferred to FEMAP finite element analysis software and prepared for the analysis. In order to define the problem, firstly the models were discretized (meshed) in FEMAP software. The crane structure model consists of 506.278 elements and 472.136 nodes. The parts of crane structure were modelled as surface bodies or shell elements to reduce the elements and nodes number of the model (to reduce finally the calculation time). After the discretization process, the load combinations and the boundary conditions were defined for the model.

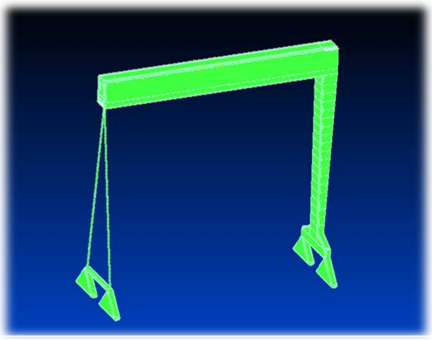

Fig. 5 Discrete structure of the crane

6. DEFINING LOADS AND BOUNDARY CONDITIONS

The structure of the crane was loaded with 10 tonnes (Table small crane to manoeuvre the piece), 20 tonnes (mainly shopping cart manoeuvre) and 300 tonnes (full load lifting).

The structure of the crane was fixed on the ground, through the embedding conditions. After this process, the boundary conditions (external loads) were applied for the crane structure.

Standard earth gravity was also taken into consideration for analysing the effect of the crane's own weight. Taking into consideration the crane's own weight, it is possible to check again the weight of the crane structure. This will be used, eventually, for the dynamic calculation of stress and strain.

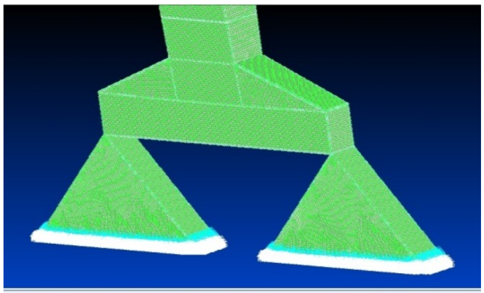

Fig.6 Mesh detail and the external loads applied to the crane

7. DEFINITION OF MATERIAL PROPERTIES

The material used for this numerical simulation is the high strength steel with stress at which the flow begins, $\sigma_c = 325 MPa$, and the stress of 500 MPa at which rupture begins, the modulus of longitudinal elasticity (Young's modulus) E=210 GPa and Poisson's ratio υ=0.3. The density of the material was defined as $\rho = 7.850 kg/m^3$.

8. CALCULUS AND RESULTS

With these boundary conditions and load combinations, the program was launched in execution of the stress analysis of the main frame.

Static analysis is the most common analysis method which is used in engineering. As the loads are assumed to be applied instantly, the effects due to the time variation are neglected.

According to the numerical results, the maximum stresses have occurred at the middle section of the upper side of the boom, with values between 260 MPa and 400 MPa; and at the bottom section of the boom with values between 90 MPa and 230 MPa.

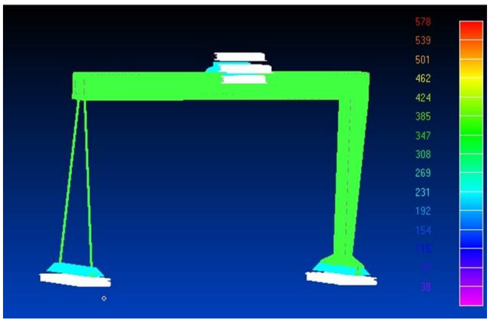

Fig.7 Result of the stress analysis

9. CONCLUSIONS

When using the analytical method in the case of designing this type of cranes, the process takes too long. In the crane structure design process, after the main dimension of the parts are done, the verification using FEM can save time on condition that reliability of the method is checked for the model. The manufacturer may change constructive solutions easily and quickly, and can obtain the new design of the structure. After these modifications, the results are very easily obtained. Finite Element Method is used in most of the applications in all fields of industries. This is the most practical and reliable way to reach the optimum design in terms of strength, weight and cost.

The applied methodology gives useful results, and these results will be used to establish the point where strain gauge transducers will be applied. Using the Tensometry will be determined the strain and stress which occur in Metalna crane structure on manoeuvring a block section in dry docks.

REFERENCES

[1]. **Alkin C., Imrak C.E., Kocabas H**., "*Solid modelling and finite element analysis of an overhead crane bridge*", Acta Polytechnica, Vol. 45, No. 3, 2005.

[2]. **Domnisoru, L**., "*The finite element method in shipbuilding*", Technical Publishing House, Bucharest, 2001.

[3]. **FNN**, Femap NX Nastran Program, 2008.

[4]. **S. S. Bhavikatti,** "*Finite Element Analysis*," New Age International, New Delhi, 2007.

[5]. **[5].Department of Labour of New Zealand**, "*Approved Code of Practice for Cranes*," 3rd Edition, 2009.

Paper received on December 31st, 2015