DESIGN AND ANALYSIS WITH FINITE ELEMENT METHOD OF JIB CRANE

Assist. Prof. Gerdemeli I.¹, Assoc. Prof. Kurt S.¹, Tasdemir B.²
Faculty of Mechanical Engineering Istanbul Technical University - Turkey ¹,²

Abstract: Cranes are transport machines, which generally used in heavy machinery industry, shipyards, seaports, warehouses and construction sector. There are several factors that have to be taken into consideration when a crane being designed. Most important factors are; own weight of the crane, the weight of the bulk which has to be transported and the dynamic loads which occur during the movements. Moreover, for the cranes which operate in open-air, the external loads caused by wind and the other climate conditions have to be considered. In order to prevent possible accidents which can cause enormous losses after manufacturing, all these factors have to be taken into account during the design process. That means crane design process requires repetitive strength calculations. During the design process, time can be saved by handling these calculations with the assistance of Finite Element Method. In this study; results of the analytical calculation and the results that were obtained by finite element method have been compared. In this way, it has been investigated the reliability of the finite element method for JIB crane design. As a result, it has been seen that, F.E.M is the most practical and reliable method which can be utilized during JIB crane design process.

Keywords: JIB CRANE, DESIGN, FINITE ELEMENT METHOD, ANALYSIS

1. Introduction

In this study; JIB cranes, which generally used in ship manufacturing and maintenance processes, have been analyzed. JIB cranes can be considered as a combination of gantry cranes and tower cranes. Because it can move on a rail like a gantry crane, on the other hand has a boom like a tower crane. JIB cranes consist of these components from bottom to top: At the bottom there are bogie groups which collect the crane wheels [1-4]. Legs are the elements between the bogie groups and the main frame. Over the main frame, there is a cylindrical part which is called pedestal. Engine room is placed on this pedestal. In the engine room there are balancing weight and the required mechanism for the rotation. On the engine room there are two drums, one for the boom movements and the other one for lifting the bulk. Also there are two frames on the engine room, one is called back stay and the other one is called A frame. These frames are connected with I-beam profiles which are called arm. And the last part of JIB crane is called boom. Boom has joints at the bottom and also it is connected to the frames with steel wire ropes. The crane which was designed during this study is shown in the figure below (Fig.1).

Crane design process starts with the classification of the crane according to FEM or DIN standards. The main parameters used in the classification are the number of load application cycles and the load spectrum. JIB crane firstly has been classified according to these standards. And the working group of the JIB crane was determined as 3ₘ according to DIN standards and M₆ according to FEM standards [4].

2. Analytical Analysis of the Jib Crane

JIB crane has been analysed analytically, before it was analysed with finite element method. Firstly the static balance of the crane was checked. After that, basic crane elements and their dimensions have been defined. At the beginning, required wire rope diameters were calculated for the maximum load. According to these rope diameters, the pulley and the drum diameters were defined via the coefficients which were taken from the FEM and DIN tables. Also the dimensions of the wheels and the rails for the crane were calculated. After all the dimensions of the JIB crane components were defined, the required electrical motor powers were calculated. For a JIB crane, there are four different electrical motors which have varied capacity. First one is used in bogie group to move the crane along the rails. And the second one is located in the engine room, that one is used to rotate the boom around the rotation axis. The wind and the acceleration loads should be considered for both of these electrical motors. The third motor works for a drum, which changes the boom radius. And the last one is used for the other drum, which lifts the bulk. After all the basic components of the crane were defined; strength of the JIB crane was calculated analytically according to FEM and DIN standards. In this section of the study, stress and deformation of the boom and stress of the crane main frame were calculated, taking into account the wind, acceleration and the braking loads. According to these results, crane model was revised to achieve better strength values.

3. Analysis of the Jib Crane with Finite Element Method

The next stage of the study includes the analysis of the JIB crane with finite element method. To be able to analyse the crane by means of this method, firstly 3D solid model of the crane must be generated.

Fig. 1 JIB crane model.
3.1 Modelling Of the Crane

Boom, basically consist of two components. The bottom side of the boom is a box girder and the rest of it is lattice or truss girder. The box girder is made of sheet metal which has 9 mm thickness. And the lattice girder consists of pipe profile beams. The lattice girder has four main beams, and there are support beams between them which have smaller diameters. The main pipes have 154, 2 mm diameter and 10 mm thickness, while support beams have 73 mm diameter, 10 mm thickness and 51 mm diameter, 8 mm thickness. These support elements don’t carry loads, they are used to hold the main beams together and make the girder more rigid. All the boom elements are made of St 52 structural steel.

The main frame or the carrier body of the crane is a box girder like the bottom side of the boom. But the sheet metals used in the main frame are much thicker than the ones used in boom’s box girder. The thickness of the sheet metals, used for the main frame’s box girders, change between 20 mm and 24 mm. And as material, St 37 structural steel was chosen for the main frame of the crane.

In this study SolidWorks 3D design software has been used for modelling the JIB crane. All the crane components were modelled one by one, and then they were combined in the assembly module of the software.

3.2 Analysis of the Crane Using ANSYS FEA Software

The models, which were generated via SolidWorks CAD software, were transferred to ANSYS finite element analysis software and they were prepared for the analysis. For defining the problem, firstly the models were meshed in ANSYS software. Boom model consists of 486,661 elements and 353,298 nodes. Components of the boom were modelled as surface bodies or shell elements to reduce the element and node number of the model.

After the meshing process, the load combinations and the boundary conditions were defined for the model. Firstly, two springs were defined for representing the wire ropes. One of the springs, which represent the wire rope that hoists the boom, was defined with a high spring constant for limiting the displacement of the boom. And the second spring was defined to represent the wire rope which lifts the bulk. This second spring was defined with a preload which equals to % 25 of the maximum load that desired to be lifted.

Except the wire ropes, joints of the boom have to be defined as a boundary condition during the pre-process. One degree of freedom was defined for these joints which take place at the bottom section of the boom. For these joints, all the movements along X, Y, Z axis and the rotations about Z and Y axis were constrained but, only the rotation about X axis was allowed. After this preparation process, only standard earth gravity, wind loads and the loads caused by acceleration were defined as external loads to the model.

According to this first analysis it has been seen that, maximum stress (65-70 MPa) and displacement (105, 33 mm) caused by own weight, occur at the middle section of the boom.
After the stress and displacement analysis of the boom; the main frame of the crane was analysed using ANSYS software as well. Analysis of the main frame started with meshing process of the model. Main frame consists of sheet metals and has a simple geometry compared to the boom. As a result it can be meshed much easier and quicker according to the boom. The model of main frame of the crane consists of 84,997 elements and 169,675 nodes. Figure 6 illustrates the mesh detail of the main frame.

Afterwards the meshing process, the boundary conditions and external loads were defined for the main frame of the crane. Firstly the model was fixed from the joints where the bogie groups were connected to. And then, weight of the engine room and the components which were located on the engine room were applied on the model. Also the wind load and the loads which were caused by the acceleration were defined for the model. On the other hand, standard earth gravity was defined for taking into account the effect of the own weight of the main frame. And finally the bending moment, which was caused by the weight of the boom and bulk and the lateral loads were applied on the model. All the boundary conditions and the loads defined for the main frame are shown in Figure 7.

With these boundary conditions and the load combinations, program was runned for the stress analysis of the main frame. According to this analysis, maximum stress occurs at the triangular support plates of the legs. The stress value detected at this point equals to 145 MPa which is under the allowable stress 180 MPa for St 37 structural steel. And the stress value at the bottom section of the cylindrical pedestal, changes between 95 MPa and 105 MPa. The stress distribution of the main frame is illustrated in Figure 8.

4. Conclusion

At the end of the study, results of the analytical calculation and the results that were obtained by finite element method have been compared. According to these comparison results, it has been seen that, the error margins were between the acceptable boundaries. This comparison is given in the table below (Table 1).

<table>
<thead>
<tr>
<th>Section / Loading Type</th>
<th>Analytical Calculation</th>
<th>Finite Element Method</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boom Middle Sec. (Self Weight + Stress)</td>
<td>67.7 MPa</td>
<td>65÷70 MPa</td>
<td>% 0.3</td>
</tr>
<tr>
<td>Boom Middle Sec. (Self Weight + Displacement)</td>
<td>106.39 mm</td>
<td>105.33 mm</td>
<td>% 1</td>
</tr>
<tr>
<td>Boom Middle Sec. (Hz)</td>
<td>178.2 MPa</td>
<td>175÷190 MPa</td>
<td>% 2.4</td>
</tr>
<tr>
<td>Boom Middle Sec. (Hz)</td>
<td>101.3 MPa</td>
<td>90÷120 MPa</td>
<td>% 3.6</td>
</tr>
<tr>
<td>Cylindrical Pedestal (Hz)</td>
<td>111.6 MPa</td>
<td>95÷105 MPa</td>
<td>% 10.3</td>
</tr>
</tbody>
</table>

Stress and the deformation results of the boom according to finite element method were fairly similar to analytical calculations. But it has been seen that, there was % 10 difference between the two method's stress values for the cylindrical pedestal. This relatively big error rate comes from the assumption which made during the analytical calculation. Because there are support plates inside the pedestal and they have been ignored during the analytical calculation.

On the other hand, some spots have been detected in the stress analysis with finite element method of the boom, which include high stresses that could not be able to occur in the real system. That kind of spots takes place in the connection areas of the beams. This situation illustrates that, finite element method could be inadequate over some cases and the results should be checked carefully. In this study, these spots were ignored and the stresses which occur several elements away from these spots were taken into consideration. As a result, it has been seen that if it is applied appropriately and the results could be analysed in the right way, finite element method gave proper results for stress and deformation analysis.

During the design process of cranes or the similar structures, it is not enough only system’s being safe in terms of strength. The design must fulfil the minimum safety conditions and should be light and cheap as well. Therefore, to be able to reach the optimum design, system should be modified and revised numerous times.
During all these modifications, calculating the system with analytical method causes design process to take long. In crane design process and the similar studies which require repetitive calculations, designers can save time by using finite element method on condition that checking the reliability of the method for the model. Constructor can change the model in computer environment and get the results of the new design via finite element method without wasting time. And this is the most practical and reliable way to reach the optimum design in terms of strength, weight and cost.

References


