Indian J.Sci.Res. 17(2): 37-39, 2017 ISSN: 0976-2876 (Print)
ISSN: 2250-0138(Online)

# FINITE ELEMENT ANALYSIS OF THE DRILL LINE FRAME

[1] Ravi Bhatt, [2]Shalin Marathe, [3]Harit K Raval
[1,2,3] Mech. Engg. Dept., SVNIT,Surat, India, 395007

***Abstract -*** The drum based hoists are normally used in mines for the handling of materials which are extracted from mines. It is supported by the frame during operation. Drill line frame is required to be optimized in order to operate during peak load conditions in any working environment without failure. The standard drill line frame has been selected for the analysis. The design has been analyzed for the two different conditions i.e. the frame is resting and lifted by lugs. The structural steel A-36 used as the frame material and commercial FE tool ANSYS was used for the analysis. It was observed that the von mises stress as well as plastic deformation was found higher while the frame lifted by lugs. Hence, modification in the lug size has been carried out for further analysis. It was observed that the change in dimensions of lifting lug significantly reduces the von mises stress and plastic deformation. Also, additional stiffeners were recommended for better distribution of the load in the modified design.

***Keywords -*** Finite Element analysis, Drill line frame, Equivalent stress, Total Deformation, ANSYS

## I. Introduction

Drill line reel (drum based hoists) are used to moving huge weights specifically for material handling in mines. Such drums are supported by the frame. These frames are resting on ground while operation and lifted on the lifting lugs during relocation of the site. The typical example of the frame is provided in Fig. 1. These are designed to transfer loads from the wire ropes to the shafts and supporting structures (frame). The frame acts also as counterweight during the lifting of loads. The frame is generally fabricated using weld joints based on the fabricator's experience. The actual load bearing capacity, over size etc. are the major issues associated with the drill line frame structure. Hence, in present research an attempt has been made to analyze the frame structure for different loading conditions. Finite element analysis has been carried out as it provides flexibility like reduces costly trial and miss methods, several parameters can be taken care same time, more convenient results without much loss of accuracy (as it works on well established algorithms).

There are number of research has been reported using finite element analysis. Bhatt and Kansara (2012) done the static analysis of filter press using ANSYS to determine the von mises stresses & deformation & it was confirmed by another FE tool AUTODESK/INVENTOR. Chen & Zhu (2011) has done finite element analysis & optimization of design for dump trucks sub frame using ANSYS & found that the deferent stresses are the result of fatigue cracks. This problem has been resolved in optimization of the design & the intensity of the stresses has been reduced. Nam et al (2011) investigated the structural analysis of ITER Tokamak machine which includes the cryostat & components contains therein. The sector sub-assembly tools descried in this paper are main assembly tools to assemble vacuum vessel, thermal shield and toroidal filed coils into a complete 40◦ sector. The 40◦ sector sub-assembly tools are composed of sector sub-assembly tool, including radial beam, vacuum vessel supports and mid-plane brace tools. In this work the strength & stability along with deflection has been analyzed using ANSYS. Rust et al. (2003) analyzed thin walled structures using ANSYS. In this work they have been used the models of keel airship cargo lifter, telescope crane & car roof for non linear limit load analysis. Zhao and Meng (2012) carried out FEA for 3.5 m sheave head. The ProE tool was used for modeling followed by ANSYS for the analysis. It was reported that spoke type sheave head works under alternating stress.

Figure 1: Typical example of drill line frame (courtesy by Sedco Engineers Pvt. Ltd.)

## II. Modeling and Analysis

This section deals with the systematic approach used during modeling and analysis. The modeling was carried out using ProE and analysis was done in ANSYS/Workbench. The analysis was carried out for the two conditions i.e. (1) the frame is resting on its base and (2) the frame is lifted by lifting lugs. The material was selected as ASTM A36 (mild/low carbon steel) as it is widely used in structural applications. The properties are: density = 7,800 kg/m3, young modulas = 200 GPa, yield strength = 220 MPa, ultimate tensile strength = 400-500 MPa and poisson's ratio

[1]Corresponding Author

= 0.26. The weight of drum is considered as 25 tons, half of the weight (12.5 tons) is acting on both end of the frame. The detailed analysis is presented below.

**A. The frame is resting on its base (Case 1)**

In this case, the boundary conditions were applied based on the frame is resting on its base as shown in Fig. 2.

Figure 2: Boundary conditions for Case 1

The von mises stress and plastic deformation were analysed and represented in Fig. 3 and Fig. 4 respectively. It can be seen that the von mises stress was found to be 196.65 MPa with corresponding deformation as 0.2856 mm. The major stress and deformation occurs at the contact zone between frame structure and the drum. The von mises value is lesser than the YS of the material hence it can be said that the design is safe for case 1.

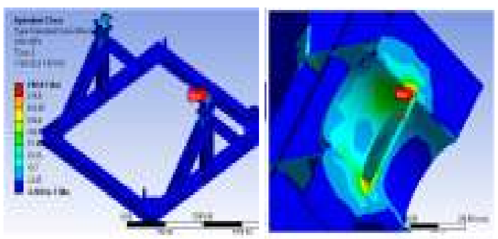


Figure 3: Von mises stress for Case 1

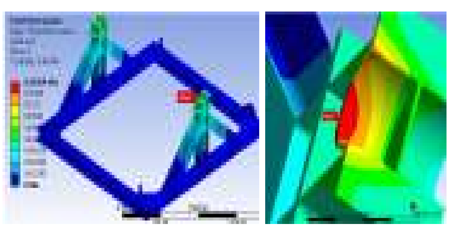


Figure 4: Total deformation for Case 1

**B. The frame is lifted using lifting lugs (Case 2)**

The boundary conditions were applied based on the frame is lifted using lifting lugs as per Fig. 5. The von mises stress and the total deformation were found to be 321.95 MPa (Fig. 6) and 2.6466 mm (Fig. 7) respectively. It can be observed that the highest stress occurs at the joint of ribs which are supporting the column of the drum base.

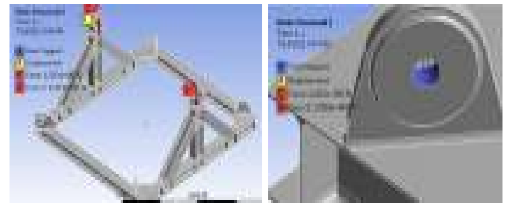


Figure 5: Boundary conditions for Case 2

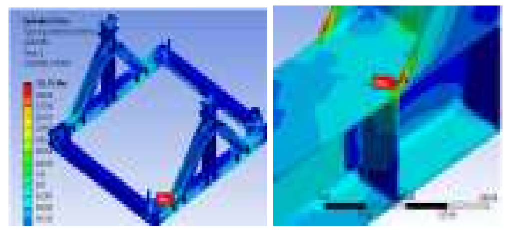


Figure 6: von mises stress for case 2

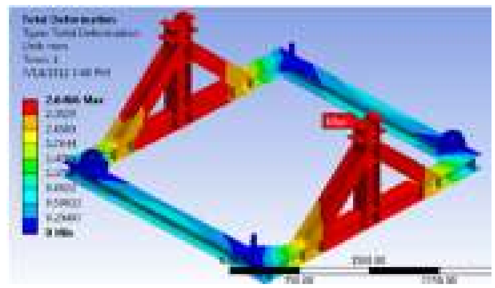


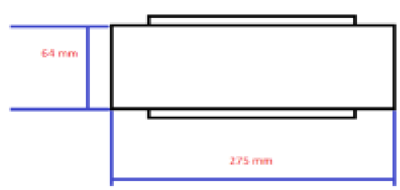
Figure 7: Total deformation for case 2

Also, the vertical members of the frame are having highest deformation. The value of von mises is higher than the YS of the material thus the design is not safe during the lifting of frame on lugs. The design if further modified considering the lug size.

**C. Modified design**

A. As the frame is lifted by the lugs, significant increase in the von mises and plastic deformation took place. Hence, the modification is carried out in the lug size (Fig. 8).

(a)




(b)

Figure 8: lug cross section (a) before (b) after

The size of the stiffener (ribs) in the frame body is also increased by 20 mm (Fig. 9). Later, the analysis is carried out further with the modified design. The von mises stress and total deformation of the modified design is presented in Fig. 10 and Fig. 11 respectively.

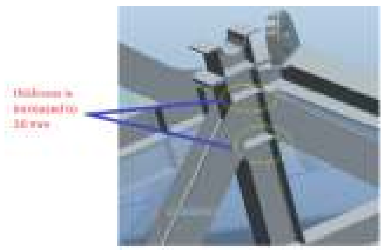
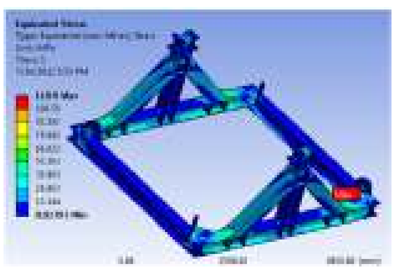
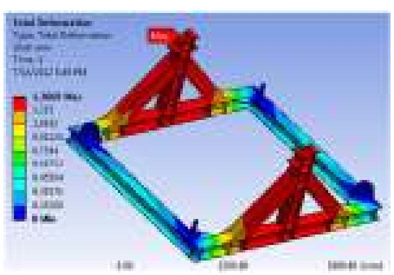


Figure 9: Change of rib size in left and right arm of frame structure

It can be observed that the von mises stress is significantly reduced 66.623 MPa based on color fringes of Fig. 10 (a). Also, the deformation also reduced to 1.3669 mm which is quite small as compared to the size and dimensions of the structure. Therefore, the design is safe for lifting on the lifting lugs.



(a)



(b)

Figure 10: Analysis of modified design (a) von mises stress (b) total deformation

## III. Summary

The FE analysis of the drill line frame has been carried out. Based on the study, it is concluded that the existing design is suitable for resting of the frame on its base only. It may get fail while lifting on the lugs. Thus, modification in the lug and rib size was carried out. The modified design of frame was analyzed again with same loading and boundary conditions. It was observed that the modified design found safe while lifted on lugs.

## References


[1] Bhatt R. J., Kansara K., 2012, “Static Analysis of Filter Press using ANSYS”, Recent Advances in Manufacturing, Proceedings, SVNIT, Surat

[2] Chen Y., Zhu F., 2011, “The Finite Element Analysis and The Optimization Design of the Yj3128-type Dump Truck’s Sub-Frames Based on ANSYS”, Procedia Earth & Planetary Science, pp. 133-138

[3] Nam K. O., Park H. K., Kim D. J., Ahn H. J., Lee J. H., Kim K. K., Im K., Shaw R., 2011, “Structural analysis of ITER sub-assembly tools”, Journal of Fusion Engineering & Design, pp. 1805-1808.

[4] Rust W., Schweizerhof K., 2003, “Finite element limit load analysis of thin-walled structures by ANSYS (implicit), LS-DYNA (explicit) and in combination”, Journal of Thinned Walled Structures, pp. 227-244.

[5] Zhao, X.T. and Meng, J., 2012. 3.5 m Head Sheave Finite Element Analysis. In Advanced Materials Research (Vol. 472, pp. 1823-1826). Trans Tech Publications.

[6] Mehta N. K., 1999, “Tool Design Data book”, Third Edition, pp.18-78.

[7] Ellobody, E., Feng, R. and Young, B., 2013. Finite element analysis and design of metal structures. Elsevier.

[8] ANSYS/Workbench, help manual, (2012)