Failure Investigation of Rotating Table of Main Substructure Frame

Khaled I. Azzabi[1], Ahmed M. Algadal[2], Mohamed S. Ahmed[3], Tarek Ben Mahmud[4]

[1,2,3] College of engineering technology-Janzour– Tripoli, Libya
[4] University of Calgary, Canada

Abstract

Stress Analysis of Main Substructure Frame is carried out using SOLIDWORKS. The maximum load is considered according to the capacity of the drilling mast of 100 tons. Variant load case scenarios are investigated in order to determine the actual load carrying capacity of substructure. Stresses produced by 100 tons placed on the rotating table section are calculated; it is found that the factor of safety (FOS) in critical areas is less than 1, due to high stress concentration at the surface of contact between the rotating tables and the supporting beams. In order to increase the FOS, the load is reduced to 75tons on rotating table showing improved results with FOS around 1.1 and stress concentration is localized in the contact region. Finally, it is found that the load on rotating table should be limited to the safe working load of 60 tons with FOS greater than 1.5.

Keywords: *Substructure, Rotating Table, 3D model, Load Capacity, Solidworks™, Stress Analysis, Adaptive Mesh.*

1. Introduction

Substructure is designed to handle the loads from setback and drill string, the setback capacity is due to drilling pipe placed on either side of the substructure; the drill string is supported by the rotating table placed at the front section. Stress Analysis of Main Substructure Frame is carried out. The maximum load is considered according to the capacity of the drilling string of 100 tons. Variant load case scenarios are investigated in order to determine the capacity of substructure. The substructure was complex, some auxiliary attachments were omitted for finite element analysis, the body bearing substructure model was only established. The results finite element static analysis of substructure could also be used to adjust the section size of weak parts with the high stress and deformation to reduce stress and deformation and to ensure the safety of the substructure in use [2].

2. Assumptions

Due to lack of design documents, the maximum load to be applied is estimated according to information provided by the owner of drilling rig, where only static load is considered, load is acting perpendicular to contact surface. The substructure is placed on an even solid ground, the lower beams are evenly placed and no moments are present.

3. Model Information

Design documents were not available nor the nameplate of the substructure. Following API Recommended Practice for Use and Procedures for Inspection, Maintenance, and Repair of Drilling and Well Servicing Structures [3], the first step is to reconstruct the mechanical design drawing and generate 3D model; thereafter a finite element analysis is performed to validate the design and give accurate estimation.

SOLIDWORKS®is used to generate the 3D model of substructure, detailed stress and structure finite element analysis performed with Solidworks Simulation using solid elements [4], the substructure statically loaded by setback and/or load on rotating table, load distribution is shown in figure 1. The load on the rotary table is acting at the center of the well.

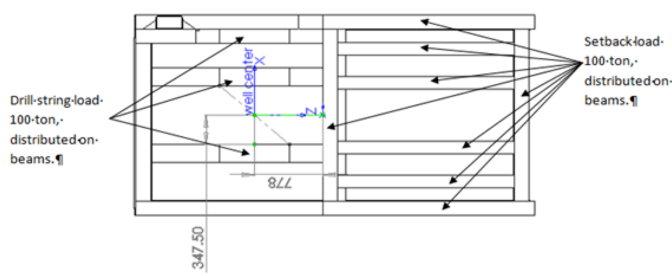

Fig.1, substructure top view load distribution plan.

Table 1 illustrates the FE Solid Mesh information used for load case 3, the time required to complete the mesh depends on the size of elements used and the level of refinement. The analysis of contact stress is also considered using pair contact definition, higher stresses are expected in critical location in the model therefore adaptive mesh technique is used.

Table 1- Mesh Information

Mesh Type:	Solid Mesh
Mesher Used:	Standard mesh
Smooth Surface:	On
Jacobian Check:	4 Points
Element Size:	24.0 mm
Tolerance:	1.2 mm
Number of elements:	577394
Number of nodes:	188748
Time to complete mesh(hh;mm;ss):	00:17:41

Figure 2 shows adaptive mesh used to refine the results; where more elements are generated around singularity points in the vicinity of high stress concentration due to contact stresses on the rotating table support beams

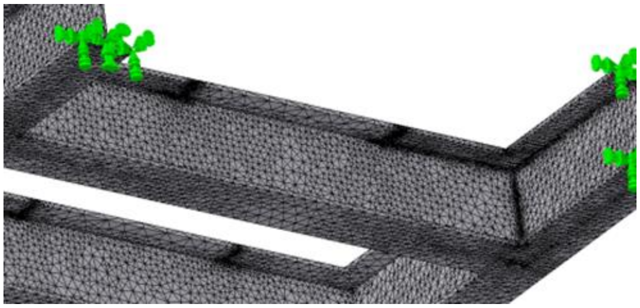

Fig. 2, Adaptive mesh used for rotating table support beams

4. Design Scenario

The analysis of combined loading due to acting static load of drill pipes placed on substructure is considered in three load cases. These load cases are investigated to study the interaction between the applied load and the response of the structure; Load Case 1- Front-side load: 100 tons on rotating table; Load Case2- Rear-side load: 100 tons on setback platform; Load Case3- Front-side load: 100 tons on rotating table and rear-side load: 100 tons on setback platform.

5. Results

The results are illustrated in the following figures demonstrate the effects of load distribution on the outcome; distributed load resembles the actual loading of substructure by the setback and drilling string. The substructure has showed moderate stresses when the load of 100 tons placed on setback section, with factor of safety greater than 2.5; Figure 3 shows results from Case3- The combined loads on setback and rotating table sections increased the stress on the middle vertical beam, which resulted in reducing the factor of safety below acceptable value.

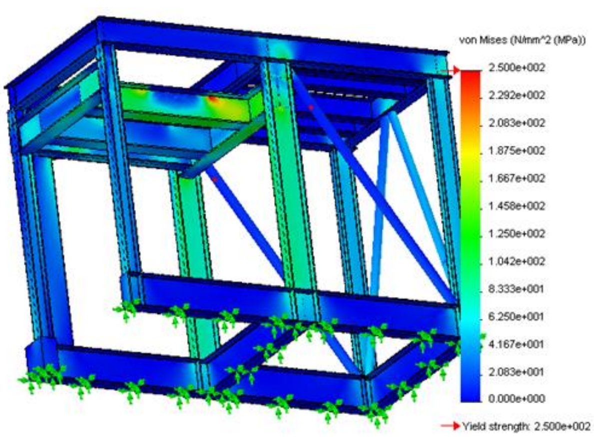

Fig.3, Substructure stress distribution, drill string load of 100 tons and setback of 100 tons

The focus of investigation is shifted to the rotating table where higher stresses are detected, it is obvious that the rest of structure to be removed from the model. Boundary conditions are examined carefully in order to accurately account for regional stress near singularity points due to actual loading. Figure 4 shows the local stress resulted from applied load on the rotating table section; the mesh refinement using adaptive mesh control have improved the accuracy of the results, it clearly shows that the structure have high stress concentration at the contact region, local deformation has occurred and will reduce the load carrying capacity below 100 tons load.

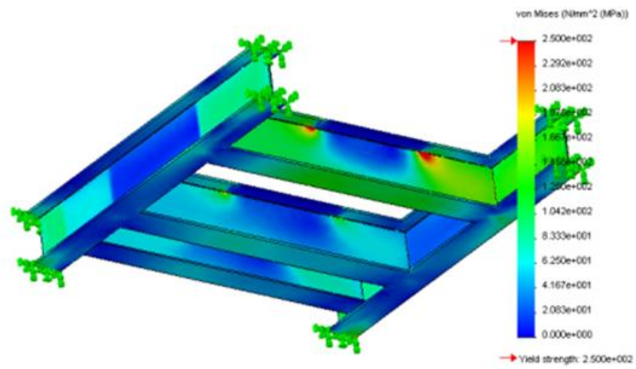

Fig.4, Rotating table frame stress distribution, drill string load of 100 tons

Different load values is applied in order to reduce the stresses to acceptable range, the rotating table load is chosen to be60 tons; figure 5shows improved stress distribution with a factor of safety values ranging from 1.5 to 2.0 in the adjacent areas.

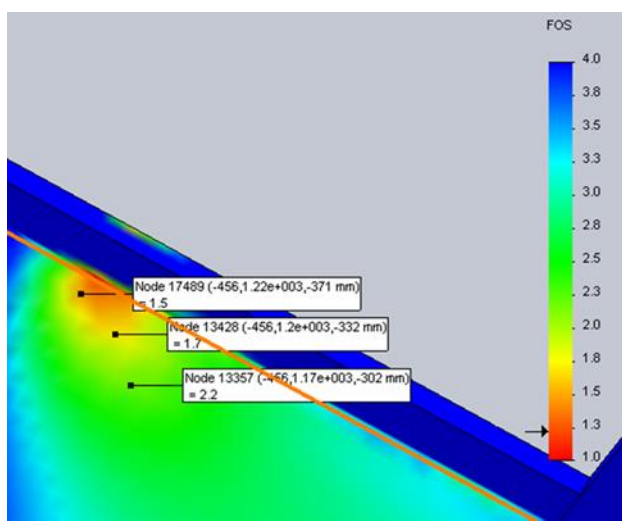

Fig.5, Rotating table frame stress distribution, drill string load of 60 tons

6. Conclusion

Following API Recommended Practice for Use and Procedures for Inspection, Maintenance, and Repair of Drilling and Well Servicing Structures, the inspection of the substructure has shown that the rotary table support beams have local plastic deformation due to the load acting through the rotary table and onto the beams as shown in photo 1. The substructure is designed for a maximum limit load of 75 tons on the rotating table with safe working load of 60 tons; and setback load of 100 tons.

Photo 1- Deformation on single rotary beam

Acknowledgments

We would like to present our gratitude to Aberdeen Drilling Consultant(ADC) and VAOS staff for their assistance to conduct this study.

References

[1] SOLIDWORKS®, SOLIDWORKS Corporation, Waltham, Massachusetts, USA.

[2] Zhang Xiaoguang, etal, “Improvement Design of Vertically-telescopical Substructure of ZJ30 Rig basedon the theory of finite element”, Scholars Journal of Engineering and Technology,2015; 3(4A):351-367

[3] API RECOMMENDED PRACTICE 4G THIRD EDITION, APRIL 2004

[4] Engineering Analysis with COSMOSWorks Professional, Finite Element Analysis with COSMOSWorks 2005, Paul M. Kurowski Ph.D., PEng., SDC Publications.