STRESS ANALYSIS OF MAST STRUCTURE FOR WATER-WELL DRILLING MACHINE

Pritibala B. Pekhale[1], Prof. P.V. Deshmukh[2]

[1] PG Student, Department of Mechanical Engineering,AISSMS COE, SPPU, PUNE, India
[2] Professor, Department of Mechanical Engineering,AISSMS COE, SPPU, PUNE, India

ABSTRACT

This research paper contains stress analysis of the mast structure for water well rotary drilling rig. As an important support and loading structure for main actuating mechanism of a rotary drill, the mast is important component in drilling operation. It plays important role for drilling to ensure normal operation and work quality. According to the extreme condition, load is applied to the mast structure, stress analysis conducted on three different types of mast structure in Ansys software. The Ansys results are verified experimentally. This study presents a finite element model of the entire tower structure of water well drilling rig. 3D graphics software Catia is firstly used to establish solid model of the tower structure of water well drilling rig and then the model is imported into the finite element analysis software Ansys to calculate the working characteristics and to perform the deformation analysis of the tower structure of rotary drilling rig in the Ansys Work bench. The critical buckling stress is calculated by the conventional formula.

Keyword – *Buckling, Derrick Structure, Euler formula, Water well drilling, Ansys workbench etc….*

1.INTRODUCTION

As one of key parts of rotary drilling rig, mast is a large steel structure with box section which loads all kinds of work forces. The stiffness of mast structure can influence directly the construction of rotary drilling rig. If the mast bends excessively, the working device such as drilling tool cannot move along the guide track of mast. Therefore, the study of the mechanical properties of rotary drilling rig mast is urgent and meaningful.

The function of rotary drilling rig mast is shown as the following several aspects.

- The mast structure supports the parts of rig, such as drilling tools, adjusting mechanism, pressure mechanism and so on.
- The mast provides orientation for drilling footage.
- The mast is the lifting arm of hoister.

At present, there has carried out extensive research of rotary drilling rig mast, but there are some deficiencies:

- Due to the large mast structure and limited calculation resources, the mesh density of finite element model is small and the calculation accuracy is not high enough;
- The position optimization of the mast hinge is not used the results of strength and stiffness of the mast structure
- The selection of limiting condition has lacks of adequate theoretical basis, and the affection of various postures of the rig parts to mast is not considered.

The main objective of the paper is to minimize the tower deformation by structural optimization to control its functional operation and improve its performance. Also to relook at derrick design to optimize the structure, redesign derrick structure in terms of improved cost and quality, to upgrade the derrick structure for easy manufacturing and to define the test procedure for validation of new derrick structure.

The tower, as a support device of water well drilling rig and an important element of water well drilling rig, is used to sustain power head and drilling pipe of water well drilling rig. If the excessive deformation of the tower of rotary drilling rig occurs, the positioning accuracy of drilling hole is largely affected. This results in hole-deviation and orientation inaccuracy. The problem of the tower deformation of rotary drilling rig has been recognized as one of the prime causes of deterioration in drilling performance of water well drilling rig.

The mast have many working conditions, such as lifting mast, lifting drill, drilling, soil dumping, transposition, so on. The lifting mast, lifting drill and drilling are the more important working conditions of the mast [3]. So the stress analysis of the lifting drill condition and drilling condition are carried out.

Fig -1 Water well drilling machine with tower structure

2. LITERATURE REVIEW

In Luo jua.b, Li Liang-ganga'Yi weiaLi'Xiaoliangb presents Working Performance Analysis and Optimization Design of Rotary Drilling Rig under on Hard Formation Conditions. The rotary drilling rig power head and drill pipe were optimized design and improved. To verify this they done the productive test of rock pile hole drilling. The rotary drilling rig rapidly popularized in the construction project because of its efficiency, quickness, environmental protection, good hole forming quality and other advantages. But in drilling hard rock, there are some problems, such as low service life of key position, low construction efficiency and others,. In this paper, these existing problems for power head, drill pipe etc. of working device of rotary drilling rig were analyzed. The rotary drilling rig power head removable combined drive key design and rotating seal transition device design which is adapted to drill hard formation put forward the innovation.[1]

In Innovative Drilling Rig Delivers Increased Drilling Performance in Permian Basin J. Gregory Nutter, Reg Layden, Cody Grasmick, Don Eubank Analyzed offset bit records, replaced the standard drilling rig with an innovative drilling rig design that combines both a full top drive rotary rig and a coiled tubing drilling rig into one rig model for cost reduction. [2]

Stipica Novoselac,Todor Ergić,Pavo Baličević in their paper presents a linear and nonlinear buckling and post buckling numerical analysis of a bar with the influence of imperfections. In the first step, they used an approach of analytic and numerical linear buckling analysis of a bar with linear-elastic material. The post buckling behaviour becomes unstable even for a very small value of eccentrically load in nonlinear analysis with elasto-plastic behaviour of material. [3]

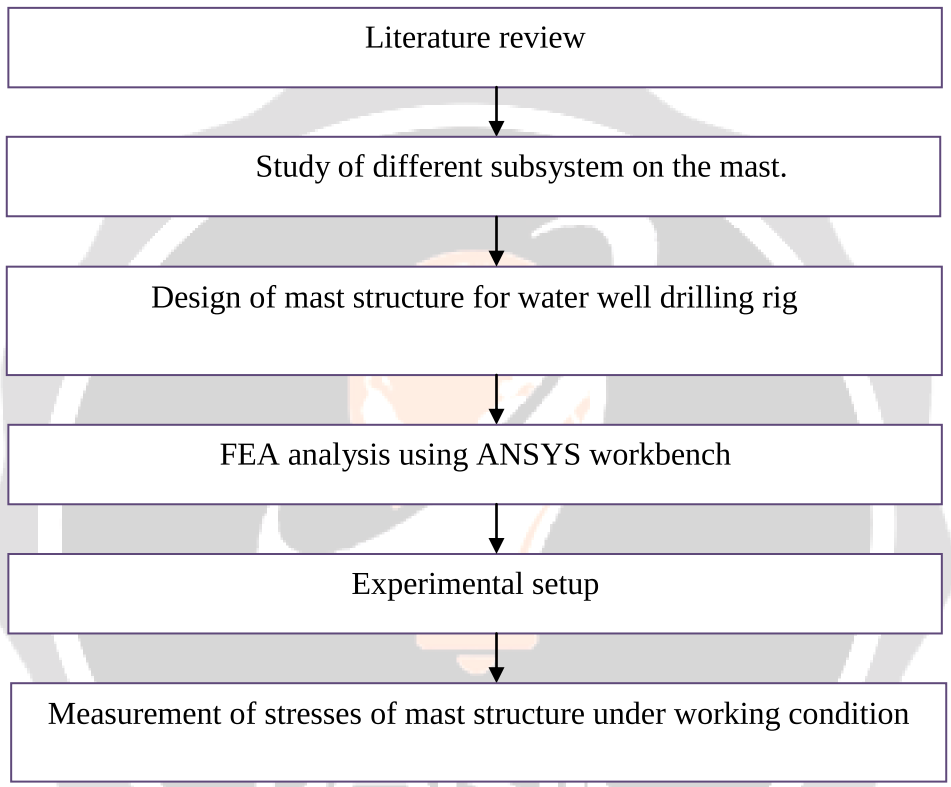
In Numerical Simulation of Cold-Formed Channel Sections with Intermediate Web Stiffeners Undergoing Pure Shea Research Report presents a numerical solution to model a simply supported cold-formed steel beam subjected to the pure shear load case. The modeling procedures including the unique concept of pure shear loads in the Finite Element Method (FEM) are discussed thoroughly. The accuracy of the simulation is confirmed by comparison with the available test data. Based on this model, a number of shear strength analyses were conducted on cold-formed channel members with plain webs and various types of web stiffeners. The outcomes are graphically shown in two formats and related discussions are included. Application to the Direct Strength Method (DSM) of design of cold-formed sections is presented. [4]

Trends in the Analysis and Design of Steel Framed Structures surveys trends in the analysis and design of steel framed structures with reference to design codes such as the US AISC Specification, the UK BS5950, the Australian AS4100, the European EC3, and the Hong Kong Code of Practice. The paper provides a brief timeline of the development of steel design codes over the past 80 years, summarizes the methods of analysis and design now permitted in codes also, discusses some of the shortcomings of present design codes, and suggests future areas for improvement. [5]

L. Brubak , J. Hellesland , E. Steen stated in their research paper present and validate an approximate, semi-analytical computational model for such plates subjected to in-plane loading for that Estimation of the buckling strength is made using the von Mises' yield criterion for the membrane stress as the strength limit. [6]

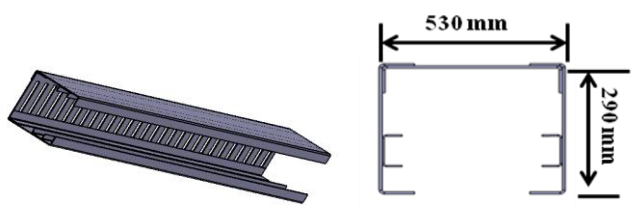
In Buckling of short cold‑formed lipped channels in compression presented by M Dundu, he concluded that Local buckling in the web and the resulting low strength of the lipped channels may have been caused by the increased cross-section distortion and initial geometric imperfections caused by the cutting process of these sections. [7]

3. METHODOLOGY



4. CAD MODELING

For Tower structure is done using CATIA software. Individual parts modeling are performed in Part drawing module and then the parts are assembled using Assembly module in CATIA against different constraints. Following figures shows assembly model of tower structure.Tower height which is equal to 8.8 m. Two inverted c channels are connected through web plates having same thickness as that of c channel, for tower structure. Modeling of tower is as follows:



2

Fig -2 CAD Model of tower

5. CALCALATIONS

For calculations following steps are followed:

1. To determine the actual length of column
The tower structure is fixed at bottom end and free at another end. Hence for buckling calculations total length of tower will be considered.

L = actual length of column. = 8.8 m

2. Determine end fixity factor from the manner of support of the ends.
Here on end is fixed and another end is free here we took,

K = fixity factor = 2

3. Compute the effective length.

Leq = effective length of the tower
= 16360mm

4. To calculate moment of Inertia.

Moment of Inertia about X axis is given by: $I_{xx} = \left[\frac{bd^3}{12} - \frac{b_1 d_1^3}{12}\right]$

I = 149747072

5. To calculate radius of gyration.

r = radius of gyration = $\sqrt{\frac{Ixx}{A}}$

= 129.5

6. To calculate slenderness ratio.

S. R. = $\frac{\text{Leq}}{r}$
= 127.72

7. To calculate critical ratio of the tower

Critical ratio is given by

$$Cc = \sqrt{\frac{2\times\pi^2\times E}{\sigma_y}}$$

Where,
Cc = Critical ratio
E = Modulus of elasticity of column material = 200 GPa
σ_y = Yielding strength = 250 MPa.

Cc = 125.66

8. To calculate Buckling stress
As Cc is greater than S.R. we use Johnsons parabolic formulae
Buckling Stress is given by,

$$\sigma = \sigma_y\left[1 - \frac{\sigma_y \times (S.R)^2}{4E\pi^2}\right]$$

Where,
S. R. = slenderness ratio
Cc = Critical ratio
σ_y = Yielding strength
E = Modulus of elasticity of column material

σ = 62.32861 MPa.

9. To Calculate factor of safety,

$$F.S = \left\{1 - \frac{(S.R)^2}{2{C_c}^2}\right\}\frac{\sigma_y}{\sigma}$$

Where,
S. R. = slenderness ratio
Cc = Critical ratio
σ_y = Yielding strength
σ = Stress on the tower

Using conventional formulae Moment of inertia, factor of safety and critical load are calculated for all three structures. All are tabulated as follow:

TABLE .I CALCULATION SUMMERY

Sr. No.	Moment of Inertia (mm^4)	σ (Buckling stress) MPa	Factor of safety
1	149747072	62.32861	1.99

6. FINITE ELEMENT ANALYSIS

Ansys workbench bucking analysis is carried out, the Factor of safety for the buckling and Deflection of the buckling is calculated.The different loading conditions used for the analysis are as follows:
During the operating condition the tower carries maximum loads, it acts as a column with one end fixed and another end hinged. While in operating conditions Maximum pull back force is generated by the feed system which is mounted on derrick structure. This pull back force also acts on structure. The derricks considered for analysis are of drilling rigs having 110 KN Pull back force. Hence for the buckling analysis we have considered 222KN axial compressive force which is maximum possible load on the structure.

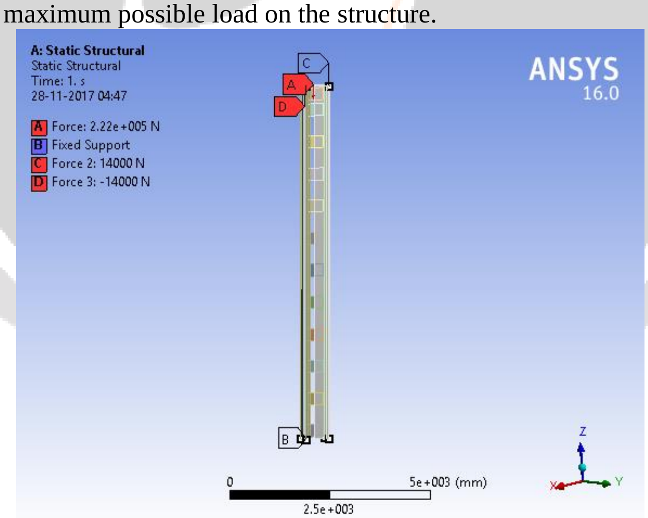


Fig -3 Loading condition on derrick structure

Deflection for above loading condition:

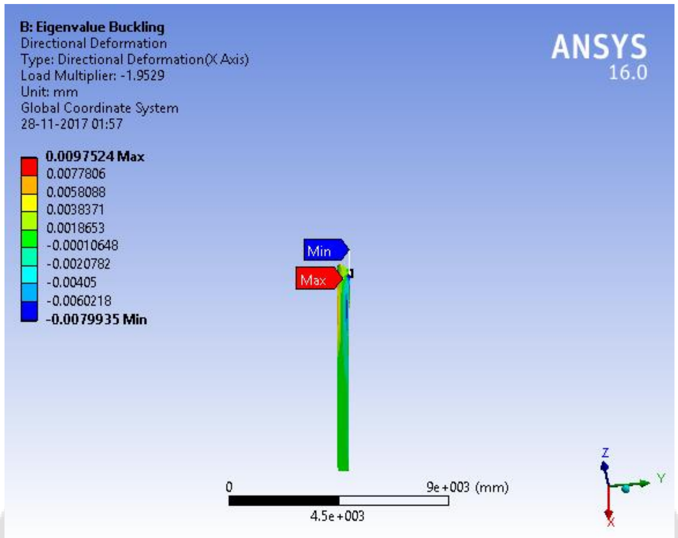


Fig -4 Deflection and FOS results of Structure by Ansys analysis

The above figure indicates the buckling analysis calculated by the Ansys and its results are as follows:
Factor of safety comes out to be 1.9.

7. EXPERIMENTATION

An alternative method of determine the initial stress condition is to use experimental stress analysis techniques. This could be used to analyze, for example, the complex case for an automatic transmission for a car or the housing for an air compressor,. Photo elastic techniques may first be used to determine where the regions of maximum stresses are loaded. The strain gages can be applied in that area to measure more precisely the magnitude of strains in particular directions. If known, the strain gages should be aligned as closely as possible with the directions of principal stresses.

The purpose of the experimental analysis us to verify the results obtained from the analytical calculation and using finite element analysis.

Using FEA analysis we can locate the points of high stresses on the tower so strain gage L13 gives the reading of maximum stress values on the tower which is located at the top of the tower structure.

Fig -5 Experimental set up

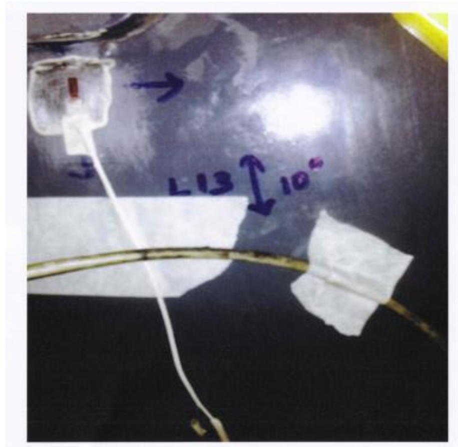


Fig -6 Location of strain gages

7. EXPERIMENTATION RESULTS

Graph shows the stresses at the location L13 which is maximum stress value along the entire tower structure at maximum load condition.

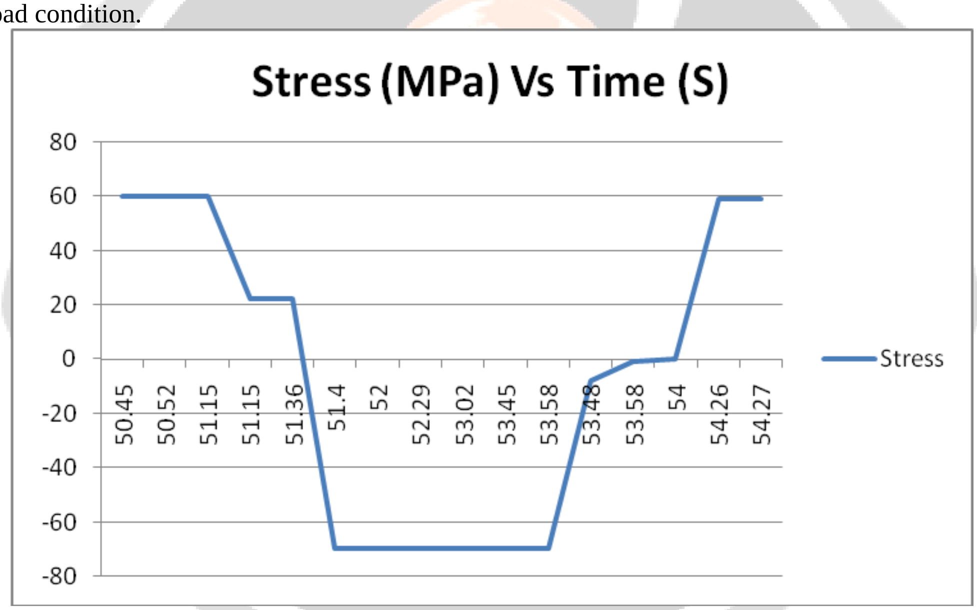


Fig -7 Loading condition on derrick structure

From the above graph it is cleared that maximum stress on the tower is come out to be 65.6 MPa which is approximately equal to the calculated values within 7 % of the experimental value.

9. REFERENCES

[1]. Luo ju,Li Liang-gang, Yi wei,Li Xiaoliang, "Working Performance Analysis and Optimization Design of Rotary Drilling Rig under on Hard Formation Conditions",Elsevier Procedia Engineering 73 (2014) 23 – 28.
[2].J. Gregory Nutter, Reg Layden,Cody Grasmick,Don Eubank, "Innovative Drilling Rig Delivers Increased Drilling Performance in Permian Basin", IADC World Drilling Conference,2009
[3].Stipica Novoselac, Todor Ergić, Pavo Baličević, "Linear and non-linear buckling and post buckling analysis of a bar with the influence of imperfections", Tehnički vjesnik 19, (2012), 695-701

[4].Song Jong Pham, Cao Hung Pham & Gregory Hancock, "Numerical Simulation of Cold-Formed Channel Sections with Intermediate Web Stiffeners Undergoing Pure Shear", University of Sydney, School of civil engineering, Research Report No R930, July 2012.
[5].Nicholas S Trahair, "Trends in the Analysis and Design of Steel Framed Structures", University of Sydney, School of civil engineering, Research Report No R926, February 2012.
[6]. L. Brubak, J. Hellesland , E. Steen," Semi-analytical buckling strength analysis of plates with arbitrary stiffener orientations", ISSN 0809{4403}, December 20052. TianHongyu and Zhang Ziyi (2011).
[7].M Dundu , "Buckling of short cold-formed lipped channels in compression", University of Sydney, School of civil engineering,2014.
[8]Cheng Yu and Benjamin W. Schafer, "Distortional buckling of cold formed steel members in bending", American Iron and Steel Institute, final report Baltimore, Maryland, 2005
[9]Cao Hung Pham & Gregory J Hancock, "Experimental Investigation and Direct Strength Design of High Strength Complex C-Sections in Pure Bending", University of Sydney, School of civil engineering, Research Report No R925, January 2012.
[10]Richard Biyas, "Advanced Strength and Applied Stress Analysis", 2nd edition, Mc-Graw Hill,1999.