Structure-strength of Hook with Ultimate Load by Finite Element Method

Yu Huali, H.L. and Huang Xieqing

Abstract - Researching and analyzing the static characteristic of the hook that functions at the limited load has an important meaning to design larger tonnage hook correctly. Academic analysis structure-strength is implemented on the hook of drillwell DG450. Firstly, based on the characteristic modeling technology, the 3-D entity model of the hook is built used Pro/E. Secondly, the static analysis on three dangerous work condition at ultimate load of the hook is proceeded by FEM software Ansys. The stress and displacement are gained to specify the stress status at the different work condition of the hook. And find out the most dangerous place where the stress is the largest. Important academic elements and data for study and design are provided. Which supplies the clear direction for the improvement of the weak structure-strength.

Key words: Hook, FEM, Statics

1. Introduction

The structure-strength is the key index to response the load-bearing ability of the elevating equipment. With the fast development of the international economy, the demand of the petroleum is higher and higher. The output of the petroleum depends on the drillwell technology. The competition of the drillwell technology is essentially the competition of the equipment technology. The drillwell hook, which is used to elevate the drill pipe, casing and plug-removal is the elevating equipment of the petroleum drillwell and one of the eight components of the drillwell. [1-2] DG450 drillwell hook is the key equipment manufactured for the depth 7000 meters drillwell, which has the largest tonnage in petroleum drillwell manufacturing industry and whose largest load is 4500 kN. Researching and analyzing the static characteristic of the hook that functions at the limited load has an important meaning to design larger tonnage hook correctly.[3]

Yu Huali is with the Department of Mechanical Engineering, Lanzhou Polytechnical College, Lanzhou, China,730050 (Email: Hualiyu836@sina.com)

H.L. Huang Xieqing is with the Mechanical Engineering College, Xi'an Jiaotong University, Xi'an, China, 710049

2. Finite-element Modeling

1) Hook Entity 3-Dimensional Modeling in Pro/E

When modeling, some simplification shall be conducted on the physical model.

So-called simplification is to make proper reduction on physical model, for instance: omit or reduce the characteristics having no or few influence to study. Primarily, the reduction shall be faithful to critical mechanical properties, meanwhile, both the geometric types between each unit and whole structure, and the mechanical property transmitted between each unit shall be consistent.

For bail, bail support, shank, shank nut, barrel and hook body, their entity 3-dimensional modeling may be made as per original design. but, for hook body, due to its complex structure (one major hook hanging swivel during drilling operation, and two minor hooks hanging on to elevator links), in addition, a hole bored horizontally through the upper body, and a slot on the back, is difficult for modeling.

When a load imposed on a hook is so great that a major load is delivered into the shank nut connecting with the shank; then the load hands down to the bail support via shank, finally, via the bail pin, the load arrives at the bail for support.

The load loss during the transmission is too small to be considered. The loading imposed upon each component of the hook shall be consistent with that upon the whole structure, according to the hook, structural and load carrying features, the following reductions are made:

（1） The screw connection between barrel and body, shank and nut may be regarded as surface connection.

（2）The hook has a complex shape and internal structure, there are many small-angle connections and radii, so the cast radius on the hook body having few influences on stress, may be ignored. [4]

With these simplification above described, both difficulty and numbers to divide grid cells was decreased, misshapen grid cell was avoided, and the calculating time was shortened; however, the statics and dynamic characteristics of the whole structure is equivalent.

Table 1 **Material Property**

Item	Grade	E/ GPa	μ	ρ/ Kg/m^3	σ_s/ MPa	σ_b/ MPa
Bail	T-217B	226	0.3	7.86×10^3	650	790
Support, bail	T-217B	226	0.3	7.86×10^3	650	790
Pin, bail	40CrNi2Mo	226	0.3	7.82×10^3	655	795
Shank and nuts	35CrMo	214	0.286	7.80×10^3	515	690
Body	T-217B	226	0.3	7.86×10^3	650	790

2) Material Mechanical Properties

The bail, bail support, hook body are cast from alloy steel. Bail pin and shank nut are made of intermediate carbon high alloy steel. See Table 1 for details.

3) Set up for Software Interface

In order to make sure the compatibility between Pro/E and ANSYS so that the modeling made in Pro/E is able to introduce into ANSYS, an interface should be set as followings:

(1) Select "Program" – ANSYS 10.0 – Unilities – ANSADMIN, open by clicking on ANSYS Manager, click "OK", then select the link to Pro/E. Click "Yes". Subsequently select "3D" from "display options" in the pop-up dialogue box. Finally, input the Pro/E installation route in this computer into Pro/E installation information dialogue box. Click "OK", the interface setting is ok.

(2) Open Pro/E, if there are two new options in its menu manager: "ANSYSConfig" and "ANSYSGeom", it indicates that ANSYS has been integrated into Pro/E.

(3)After modeling in Pro/E, click "ANSYSGeom", ANSYS is opened automatically, and the current modeling will be led in ANSYS, at the time, click display command, the physical displays. See Fig.1 for "Introducing the Finite-element Model in ANSYS"

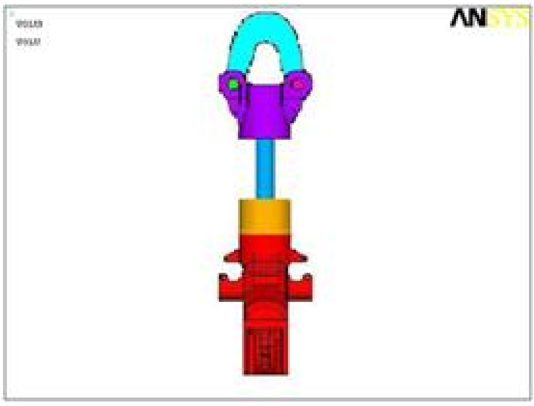


Fig.1 Lead-in Finite-element Model in ANSYS

4) Mesh Dividing

After comparing, the Free Mesh Dividing is applied herein; we use SOLID45 tetrahedron 8-node 3-dimentional physical model liner unit, SMART method, LEVEL 6; thus, the total number of mesh dividing is 211628 and the nodes are 46252.

3. Loading analysis & Boundary Condition Treatment

1) Load Treatment

In GB1806-86 "THE TYPES AND BASIC PARAMETERS FOR OIL DRILLING RIGS", the maximum hook load is defined as the largest load on weight indicator when run-in casing or unload elevators with a specified number of lines strung to the travelling block. Herein, we assume the maximum hook load is 450t (approximately 4500kN) (without regard to load composed of the hook), then conduct the calculation depending on the following three working conditions:

Case 1: major hook carries total downward pulling force 4500kN, the two minor hooks carries nothing.

Case2: two minor hooks respectively carries 2250kN, major hook carries nothing.

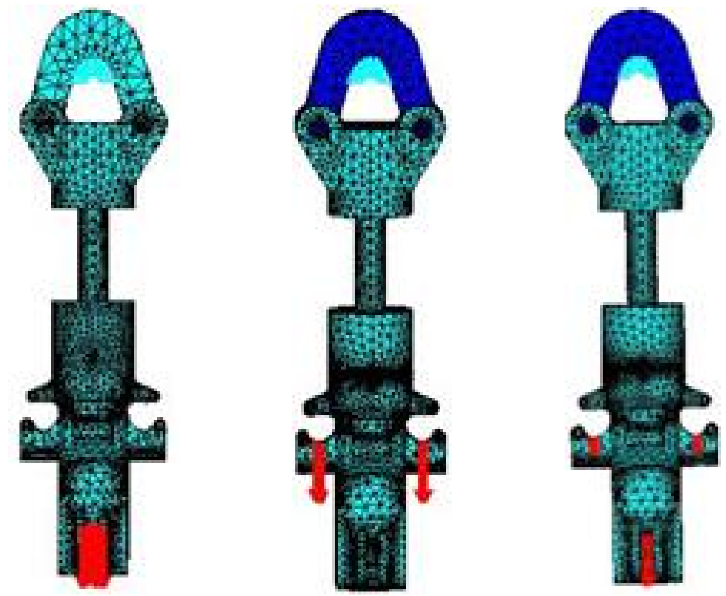
Case 3: the major hook and minor hooks carry same downward pulling force (2250 kN);

2) Constraint Condition Treatment

In the calculation in all three cases, the zero displacement of the contact between bail and traveling block is assumed. Loading condition shall be determined as far as possible by load transmission relationship between parts under loading and servicing cases. Displacement condition shall utilize symmetry condition as far as possible; when adding constraint condition, the place causing the minimum counterforce shall be identified as a displacement constraint location in order to avoid deviation in calculated result. Mesh Dividing and Load for Models in Three Cases see Fig.2.

3) Configuration & Software for Finite Element Calculation

In this text, we calculate the result by means of ANSYS 10.0 Structure. The computer operating system is Windows XP. PC configuration: CPU Pentium 4; frequency of CPU: 3.0G; RAM: 2.0G; GB: 160G. SI Units (force: N; length: m; stress: Mpa; Time: Second (S). Refer to coordinate system of this finite-element Model, considering the hook is like a column type, so set the origin on the center of bottom, Y-axis is in parallel with the rotation axis of the hook, and the direction is consistent with force action.



（a）Case 1 major hook force 4500kN （b）Case 2 two minor hooks carries 2250kN（c）Case 3 every hooks force 2250 kN

Fig.2 Mesh Dividing and Load for Models in Three Cases

3. Conditions recognition

From Fig. 3, 5, 7 show the equivalent stress in three servicing conditions, it can learn the stress in pin connecting bail and support (marked with MX) is the maximum (respective stress in bail, support and pin is 827 MPa、933 MPa、956MPa), and each value is greater than material yield strength. On the whole, the pin is in a dangerous condition because its strength is the weakest in whole hook structure. In case 1, the stress in bail support or minor hooks is relatively smaller than hook mouth radius. The higher stress here results from: 1) here is the load-carrying path of the hook body; 2) loading. In case 2, the stress in bail support or major hook mouth is relatively smaller, however the stress in barrel wall (upper) and minor hooks mouths are larger. In case 3, the stress in major and minor hooks mouths and barrel wall (upper) are relatively larger.

In analyzing, we also find that the stress in the barrel inner wall is larger than that in the outer wall, mainly because of the stress concentration in the inner wall. In addition, the stress in the bore horizontally going through the upper hook body for locking assembly is relatively concentrated, so the stress here is relatively larger. For these stress concentration locations, some modification shall be made in structure.

Fig. 4、6、8 show the equivalent displacement in three servicing conditions, the displacement where the bail and support contacts is smaller than that between lower barrel and hook body; the maximum displacement is at the major mouth; in servicing condition 3, the maximum displacement is 5.835mm**.**

4. Conclusions

Analyzing structure-strength of DG450 Hook using finite element method. Firstly, make the entity physical model of integrally assembling DG450 hook with Pro/E and conduct some necessary simplification. After Mesh Dividing, impressing constraint and loading condition in ANSYS, obtained the static analysis on hook under three cases; finally, got the Stress nephogram and Displacement nephogram, learned the stress condition in different servicing conditions. So, from the angle of strength, the further study on modifying and perfecting hook components structure is available.

The way of the structure-strength analysis on DG450 drillwell hook is given in this thesis. And it also illuminates the instructional meaning and engineering application value to the design and development of the larger tonnage drillwell hook in this thesis.

References

[1] Wang Jinquan , “Current situation of China's oil drilling rig technology and consideration of its research and development”- *Petroleum Machinery*, 2006 (1), pp. 47~50

[2] Zhang Lianshan (translator) , “Current situation and development of foreign oil drilling rig lifting and rotation systems”- *Foreign Petroleum Machinery*, 1997 (8): pp. 1~9

[3] T.D , “Automated systems increase safety and efficiency,” [J].*Drilling Contrator*.2002(5), pp.10～15

[4] Gong Shuguang – “*ANSYS engineering application and its analysis*” (Book style) Beijing China Machine Press, 2003.4 pp. 35–85.

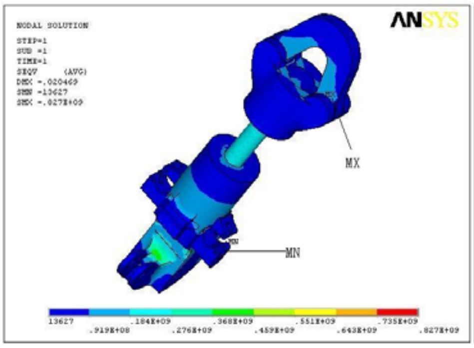


Fig.3 Equivalent Stress of hook of case I

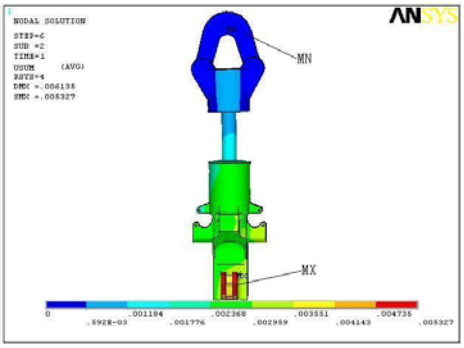


Fig.4 Displacement of hook of case I

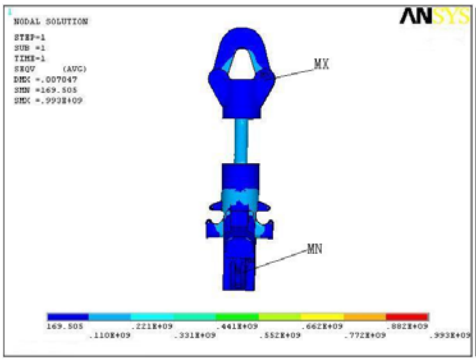


Fig.5 Equivalent Stress of hook of case II

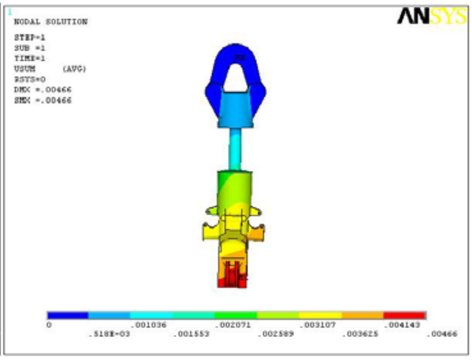


Fig.6 Displacement of hook of case II

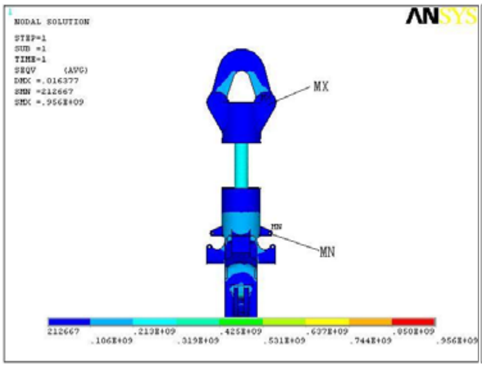


Fig.7 Equivalent Stress of hook of case III

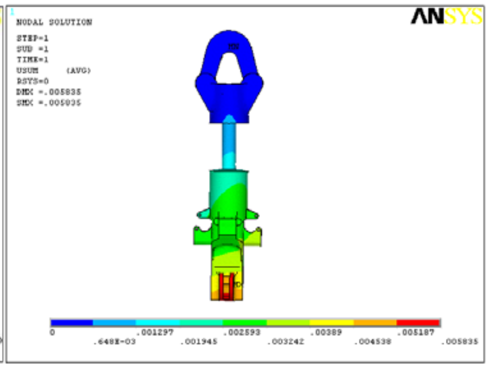


Fig.8 Displacement of hook of case III

ISBN: 978-988-17012-7-5