Simulation Analysis and Optimal Design of Back Clamp Device

Ping YU, Si-Jie KANG[a*], Yan-Jiao LI, En-Chao JIN

Mechanical Science and Engineering Institute of Jilin University, Changchun, China

[a]Kang sijie@ 163.com

[*]Corresponding author

Keywords: back clamp device, dynamic analysis, orthogonal optimizing design.

Abstract. Back clamp device is the key equipment of the top drive, which is required operation reliable and compact structure. This paper introduces the working process and working principle of the back clamp, According to the operation scheme and performance parameters of the back clamp, Dynamics simulation analysis was carried out. Obtain the contact force between the jaw and drill pipe, as well as force and other performance parameters of clamp teeth. The basic idea of orthogonal optimization method is to optimize the parameters of tooth profile, tooth height and tooth pitch of the jaw, and the parameter optimization combination is gained. Finally, Using the ANSYS Workbench for the finite element statics analysis of the back clamp, the results show that the design and functional requirements of the application are met.

Design and Existing Problems of Back Clamp Device

Due to the overall floating of the back clamping device in the process of clamping and loosening, the whole float is highly required. Therefore need to ensure that the floating precision of the back clamp floating mechanism, Schematic diagram of the back clamp is shown in the Fig.1~Fig.2, The configuration of the spring has a very important role in the back clamping device, One is to be able to limit the position, and the other is to ease the inertia of the hydraulic cylinder caused by the rapid movement. The outer shell body of the clamping mechanism adopts the split structure of the left clamp body and the right clamp body. The split type structure design is convenient to disassemble and reduce the auxiliary time to maintenance of the equipment, and improve work efficiency.
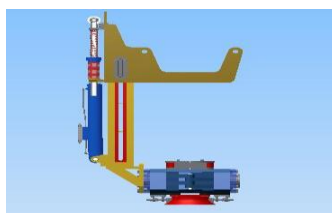
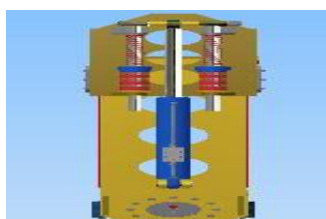

Fig.1 Back clamp device

Fig.2 Back clamp floating mechanism

The whole structure of the existing back clamping mechanism is compact and complete, and the operation is flexible, Safe and reliable, But the actual operation of the process also found that the number of discarded clamp tooth and drill pipe is relatively large, The main reason for the failure is the wear or fracture of the jaw, The jaw is a part of the back clamp device, which is directly under

the pressure and the friction force. The quality of the jaw is directly related to the working efficiency and the service life of the drilling tools.
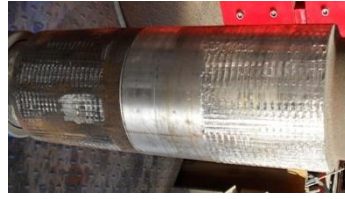

Fig.3 Figure of drill pipe failure

Fig.4 Wear map of jaw

Dynamic Simulation Analysis of the Back Clamping Mechanism

Sets The Model Simulation Parameters. Setting the contact force parameters between the jaw and the drill pipe joint:

1. Normal contact force:

$$F_N = k.\,\delta^\varepsilon + step(\delta, 0, 0, d, c).\frac{d\delta}{dt} \tag{1}$$

Where: Generalized normal contact force, N; k(stiffness):Colliding stiffness on the side of the component; δ: penetration depth Penetration stiffness in the contact point, mm; e (force exponent):Force index, which is also the contribution factor of the stiffness term. Force-Displacement curve shape can be determined. d: Maximum allowable penetration depth, mm; c(damping)Maximum loss factor, N.mm/sec;

2. Tangential force-friction

The contact tangential force is the product of the friction coefficient and the normal force, The Kunlun friction model is used to deal with the contact tangential force in ADAMS, The friction coefficient of contact force is in accordance with the relative sliding velocity between the contact objects. Choice of dynamic friction coefficient μ_d or static friction coefficient μ_s, According to the actual working condition of the drill pipe joint and the jaw, Setting μ_d=0.35, μ_s=0.30.

Simulation Data Output and Analysis. In order to be able to simulation the hydraulic cylinder thrust into four symmetrical distribution of the jaw, and effective clamping of the drill pipe, Set the simulation time to 110 seconds, Set the number of simulation steps for the 11000 step, Set the maximum number of iterations in the ADAMS/solver to 6, The integral polynomial order is set to 2, This can effectively improve the speed and accuracy of the contact impact force.

Force Analysis of the Jaws. The contact pressure information of the four teeth on the left jaw is shown in Fig.5 and Fig.6;
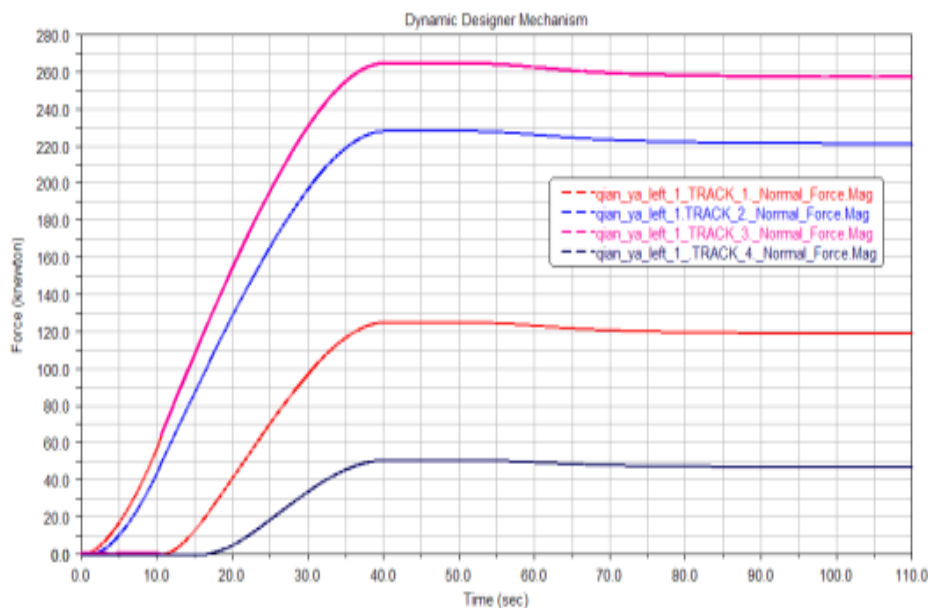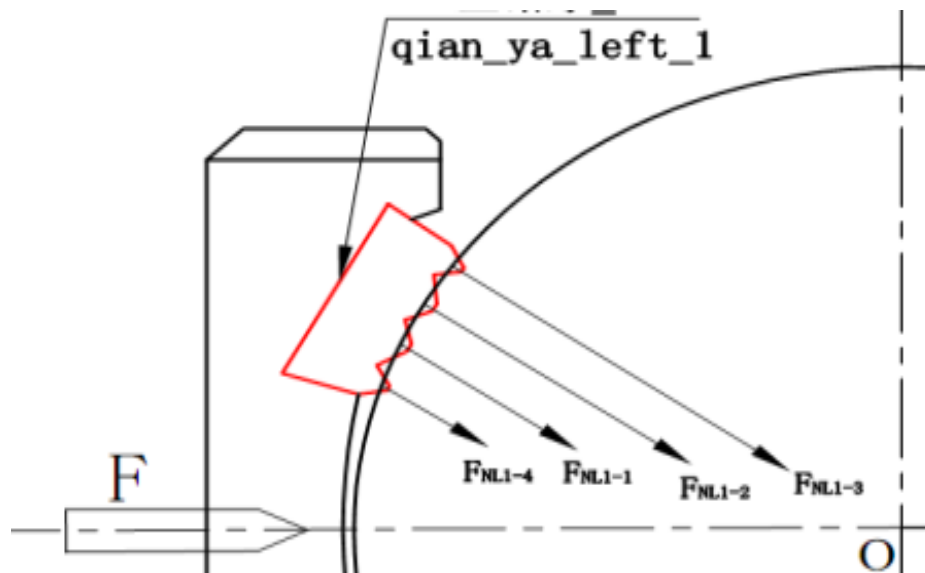
Fig.5 Contact pressure of the jaws

Fig.6 Contact pressure

From Fig.5 we can see that there is a sequence of contact between the four teeth on the left jaw and the surface of drill pipe, what is in accordance with the actual, The contact force of each tooth reached their peak value in the interval of 40 seconds to 50 seconds, FNL1−1=125.31KN, FNL1−2=228.48KN, FNL1−3=264.56KN. According to the contact pressure state diagram, Contact pressure of the teeth at the center of the horizontal line is the largest, which closest from the horizontal line is the least. This helps increase the range of the clamping and basically meet the design requirements.

Eat Into The Depth Analysis Of The Jaws. Eat into the depth of the drill pipe joint that bite by jaws in the process of Make-up is shown in the Fig.7, Combined with the simulation curve to analyze eat into the depth, We can see the bite depth of each teeth gradually increased from zero to maximum values. And then enter the stage of equilibrium and stability. Hydraulic thrust to maximum value in 40 seconds to 50 seconds, Bite depth of the jaw is gradually increased to the maximum, Fig.8. Schematic diagram of the jaws bite depth, Following results can be obtained with the combination of Fig.7 and Fig.8, δL1−1 =-0.2541mm, δL1−2=-0.3792mm, δL1−3=-0.4182mm, δL1−4=-0.1395mm. Bite depth variation of the jaw can be used as foundation of drill pipe joints

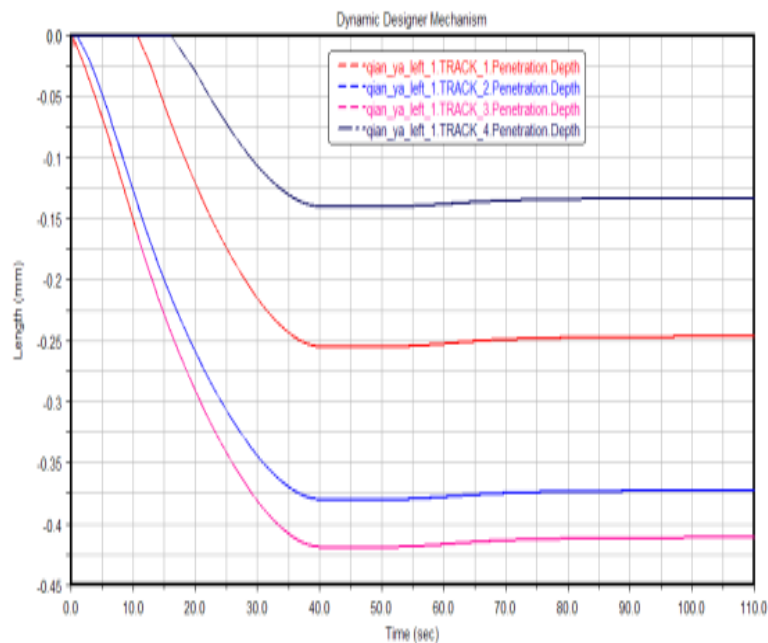plastic deformation. Provide a reference for jaw section size optimization.
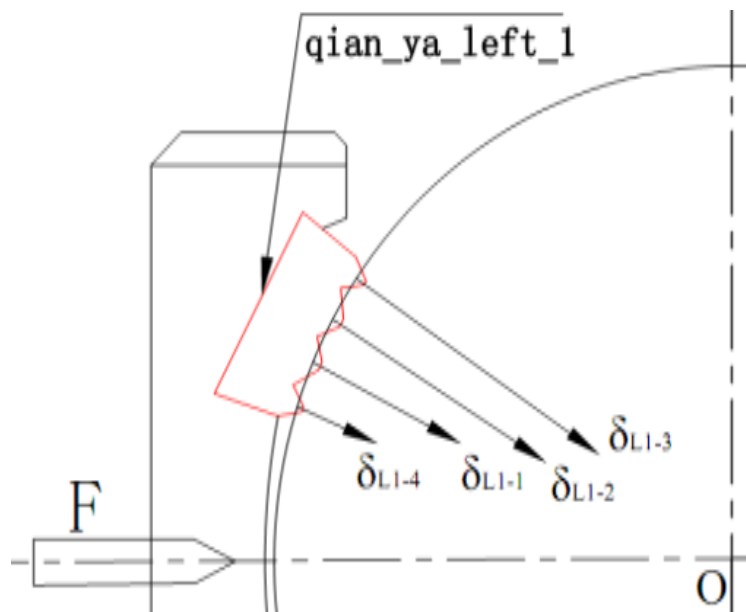

Fig.7 Eat into the depth

Fig.8 Schematic diagram of eat into the depth

Key Parameters Optimization Design of the Jaw

Through the analysis of the jaws, we obtained the relationship between the key parameters and the influence on working condition of the device when parameters variation, a back clamp device with good performance should ensure that the drill pipe and the jaw must have enough friction co-efficient. To ensure that there will be no slip phenomenon, then, as far as possible to ensure that the damage to the outer wall of the drill pipe is small, the orthogonal method is used to optimize the main parameters of the jaw.

Design Variable. In this paper, the tooth height h, pitch p and thread angle of the three cross section parameters are optimized.

Constraint Condition. According to the actual working background and design experience, the

limiting conditions: thread angle $80° \sim 120°$, space of the tooth p $2mm \sim 8mm$, tooth height h $1mm \sim 4mm$.

Optimization Indexes. The purpose of this optimization is to select the optimal value of each parameter by controlling the depth of bite and the equivalent friction coefficient, There are two aspects of the optimization indexes, The bite depth: $t \leq 1mm$, equivalent friction coefficient $f_v \geq f = 0$. The combination can be classified to multi- objective programming problem, Two objective functions are as follows:

$$t = \frac{p}{kmnr\delta_s} \times \frac{1}{\frac{p}{h} + 2\sqrt{\frac{p^2}{4h^2} + (\frac{p}{h}-1)\frac{1}{\tan\alpha}}} \tag{2}$$

Within a certain range, the smaller value of the bite depth function (2) is, the better the results are.

$$f_v = \frac{Q}{P} = \frac{\mu\left\{\frac{P}{2h} + \sqrt{\frac{p^2}{4h} + (\frac{p}{h}-1)\frac{1}{\tan\alpha}}\right\} + 1}{\mu - \frac{p}{2h} - \sqrt{\frac{p^2}{4h} + (\frac{p}{h}-1)\frac{1}{\tan\alpha}}} \tag{3}$$

Within a certain range, the bigger value of the equivalent friction coefficient function (3) is, the better the results are.

The Optimization Design. Optimization of the main parameters by orthogonal method, comprehensive consideration of theoretical analysis, manufacturing requirements and design experience, the design variables of thread angle were selected as 85, 95, 105, 110, 120, the tooth height were selected as 1.5, 2, 2.5, 3, 4, and value of the pitch were selected as 3,4,5,6,8, select standard orthogonal table L25(56), table line number of the table is 25, 25 tests can be carried out, table column number is 6, up to 6 factors can be placed, This study does not consider interaction, A total of three factors and five level are involved, from the standard orthogonal table L25 (56), we can see occupy three columns and idle three columns, For intuitive convenience can be recorded as L25(53), the level factor table is shown below.

Tab.1 Factor level

Level / Factors	A thread angle α ((°)	B Tooth height h(mm)	C Pitch P (mm)
1	85	1.5	3
2	95	2	4
3	105	2.5	5
4	110	3	6
5	120	4	8

After calculating the objective function, the statistical data is needed, in this paper, we choose the range analysis method to process the data, the result analysis table is show in Tab.2, among them, Ks is the sum of the function results that the level number in each column of the Tab.2 for the s. In this paper, s=1, 2, 3, 4, 5; ks is the arithmetic mean value of the function results that the level number in each column of the Tab.2 for the s. ks=Ks/n, n is the number of levels, n=5R is the range, in each column, R=max ks-min ks.

Tab.2 Result analysis

		A thread angle	B tooth height h	C pitch P
Eat into the depth t	K1	3.6415	2.05	4.8408
	K2	1.589	2.6676	4.9959
	K3	4.1323	3.5891	2.9184
	K4	4.7227	4.9633	3.2971
	K5	3.9662	4.7817	1.9995
	k1	0.7283	0.41	0.96816
	k2	0.3178	0.53352	0.99918
	k3	0.82646	0.71782	0.58368
	k4	0.94454	0.99266	0.65942
	k5	0.79324	0.95634	0.3999
	R	0.62674	0.58266	0.59928
	order	ACB		
f_v		A thread angle	B tooth height h	C pitch P
	K1	4.0804	3.0872	5.5713
	K2	4.9626	3.5345	4.347
	K3	2.5071	4.4606	3.7545
	K4	4.8723	4.3208	4.1134
	K5	4.4126	5.5909	3.1482
	k1	0.81608	0.61744	1.11426
	k2	0.99252	0.7069	0.8694
	k3	0.50142	0.89212	0.7509
	k4	0.97446	0.86416	0.82268
	k5	0.88252	1.11818	0.62964
	R	0.4491	0.50074	0.48462
	order	BCA		

Optimization Results. How to co-ordinate various factors with level is the best, this optimization has two objective functions. For the bite depth, the thread angle is the main influencing factors, level 110 is the most preferred, The optimal combination parameter $\alpha = 110, h = 1.5, p = 4$; and for the equivalent friction coefficient, the main influencing factors are the tooth height, level 2 is the most preferred, The optimal combination parameter: $\alpha = 110$, $h = 2$, $p = 3$. According to the requirements of the actual work and theoretical analysis, the design parameters of the final jaw that $\alpha = 110$，$h = 2$, $p = 4$ is the best option.

Finite Element Analysis of the Jaw

To a great extent, Strain and stress magnitude of the back clamp device that under static load, what is affects the safety and reliability of top drive system, Therefore, it is necessary to check the strength of the back clamp, The maximum stress and the total deformation were observed, check whether the device is reliable.

Add Model Material Properties. The back tongs material selection for 20CrMnTi, Material properties are shown in Tab.3:

Tab.3 Material properties of 20CrMnTi

Allowable stress σMPa	Elastic modulus EGPa	Density Kg/m^3	Poisson's ratio	Yield limitMPa
310	207	7.810^3	0.25	835

Divide and Refine the Grid. A three dimensional model of the jaw was built by the 3D software of Inventor, export this model into ANSYS workbench, Add the material properties of jaw as 20CrMnTi, and the mesh division of jaw is shown in Fig.9.
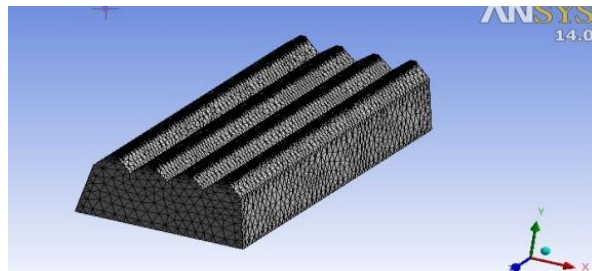
Fig.9 meshing of clamp teeth

Added load and did the FEM. Combined with the specific situation of the model, reasonable boundary conditions are added to it, first of all, to add a fixed constraint to the bottom surface of the clamp, add the load to the four teeth of jaw, then, add the appropriate size of contact pressure P and shear stress Q in each tooth, the load size is provided by the simulation results.

Static analysis of the jaw, the corresponding stress distribution and deformation results are obtained, the total deformation is shown in Fig.10, and the equivalent stress is shown in Fig.11.
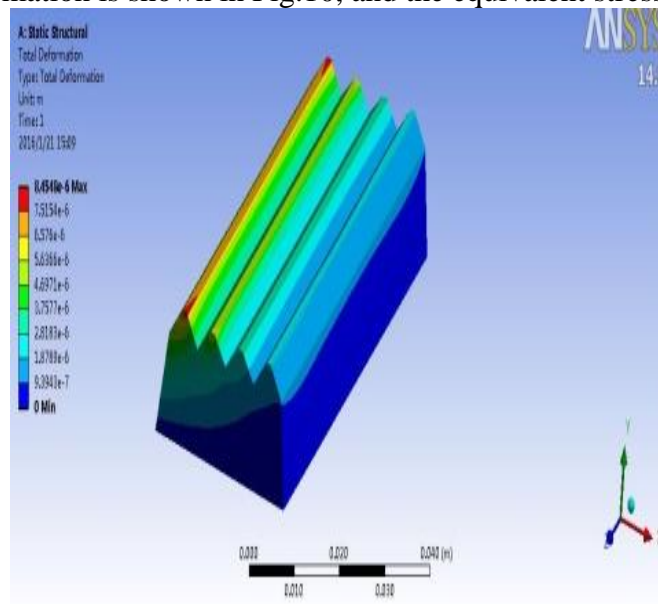

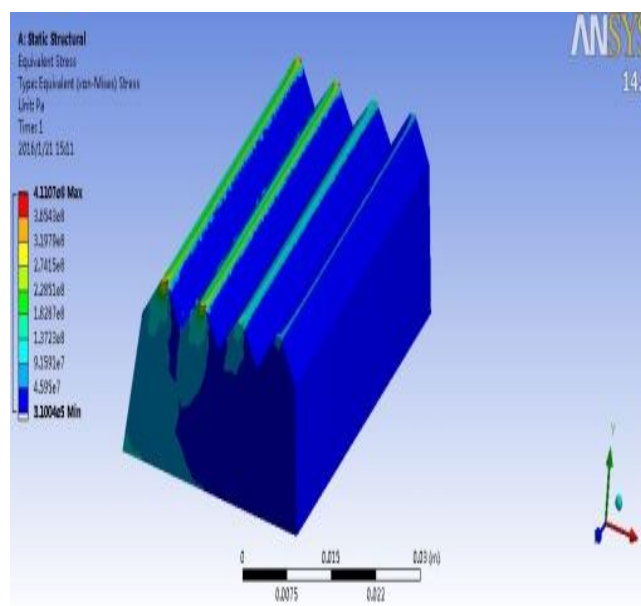**Fig.10** Contour of total deformation

Fig.11 Stress contour of clamp teeth

From Fig.10 and Fig.11 we can see the following conclusions:

Tab.4 Finite element analysis results

category	minimum value	Maximum value
displacement	0	0.00845mm
stress	0.31MPa	411MPa

The maximum stress of jaw is 411Mpa, and the yield stress of jaw is 835Mpa, the maximum displacement of jaw was 0.00845mm, and the deformation is relatively small, so we can conclude that the stress and deformation of jaw are meet the requirements of the use.

Conclusions

The theoretical analysis and optimization design of the back clamp device are carried out, we selected the section parameters of the jaw , thread angle$\alpha = 110$, the tooth height $= 2$, pitch $p = 4$, this scheme ensures that the equipment work process reliable, reduces the wear of jaws and the damage to the drill, prolong the service life of the equipment as well.

Statics analysis of the jaws, the maximum stress and deformation shows that jaws strength meets the application requirements; clamping process of the device is stable and reliable.

Acknowledgement

References

[1] Zhang Feiyu. Dynamic simulation analysis of MPR-70A type full hydraulic automatic drainage pipe [D]. Jilin University, 2013.

[2] Kverneland H. 2009. Electrical Cranes and Winches for Improved Safety and Better Operational Performance for use in Extreme Weather Conditions [C]. Society of Petroleum Engineers, 1(5): 137-149.

[3] Sun Mingxing. Finite element analysis and evaluation of bearing capacity of drilling derrick [D]. Lanzhou University of Technology,2010.

[4] Latorre R. Ship hull drag reduction using bottom air injection [J]. Ocean Engineering, 1997, 24(2): 161-175.

[5] Ke Wang, Huai Chen, Wei Wang et al. 1997. Modal Analysis of Oilfield Derrick Structure [C]. Proceedings of the 1997 15th International Modal Analysis Conference, 1871-1877.

[6] Xiao Wensheng.2004. Dynamic analysis of top drive drilling device and Research on Virtual Prototyping Technology [D]. Huazhong University of Science and Technology.

[7] Yuan Qinghong.2004.TDS Research and practice of virtual prototype system of drilling rig [D]. Huazhong University of Science and Technology. [8] Shuanlu Lu, Yaorong Feng, Faqian Luo, Changyi Qin, Xinhu Wang. Failure anslysis of IEU Drill Pipe Wash out. International Journal of Fatigue[J]. 2005, (27):1360-1365.

[8] Dickinson III B W O, Dickinson R W, Nordlund R. Multiple lateral hydraulic drilling apparatus and method: U.S. Patent 6,206,112 [P]. 2001-3-27.

[9] Ke Wang, Huai Chen, Wei Wang et al. 1997. Modal Analysis of Oilfield Derrick Structure [C]. Proceedings of the 1997 15th International Modal Analysis Conference, 1871-1877.

[10] Dickinson III B W O, Dickinson R W, Nordlund R. Multiple lateral hydraulic drilling apparatus and method: U.S. Patent 6, 206, 112 [P]. 2001-3-27.

[11] Jan Pinka, Jozef Lumtzer, Jamil Badran. 1996. TDS-Top Drive System, new drilling technology. Acta Montanistica Slovaca, (4): 89-295.

[12] Vittorio Gusella, Ostilio Spadaccini, Andrea Vignoli. 1996. In-Service Dynamic Behavior of a Drilling Derrick on a Jacket Platform. International Journal of Offshore and Polar Engineering, 6(7): 184-194.