Mechanical Analysis of Valve Box of New Fracturing Pump based on ANSYS Workbench

Zhen Song, Yaran Hu

School of Mechanical and Electrical Engineering, Southwest Petroleum University, Chengdu, 610500, China

Abstract

There are few researches on the stress variation trend of the new type of fracturing pump valve box without interlocking line structure during operation. Therefore, based on ANSYS Workbench software, numerical analysis is carried out on the valve box of the new fracturing pump with three plunger diameters under different internal pressure loads. The results show that: Under the designed maximum pressure load, the maximum stress of the valve box is σ_{max}=670.9MPa, the yield strength of the material 40CrNiMo is σ=835MPa, $\sigma_{max}< \sigma$, which meets the strength requirements. The maximum stress of the valve box concentrates on the discharge outlet, and the maximum total deformation appears on the discharge chamber. Under different internal pressures, the maximum stress and total deformation decrease with the increase of pump strokes when the plunger diameter remains unchanged. The decreasing rate of the maximum stress and total deformation is significantly lower when the pumping times are 200~300r/min than when the pumping times are 150~200r/min. When the pump stroke is constant, the maximum stress and total deformation decrease with the increase of plunger diameter. Compared with the valves with 139.7mm plunger diameter, the maximum stress and total deformation decrease by 15.1% and 33.3%, respectively, and the total deformation decrease by 15.2% and 33.5%, respectively, for the valves with 46.1mm and 152.4mm plunger diameter. This paper provides a new theoretical basis for the optimization and improvement of the valve box structure of the new fracturing pump, and at the same time reduces the number of tests and research costs.

Keywords

The Fracturing Pump; Valve Box; Finite Element Analysis; Mechanical Properties; ANSYS Workbench.

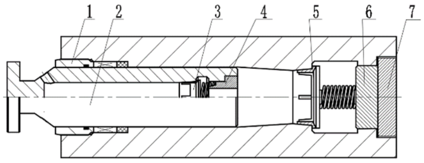
1. Introduction

Fracturing is one of the important measures to improve reservoir production in unconventional energy oil and gas development. The fracturing pump plays a vital role in fracturing operations, and the hydraulic end, as the most critical component of the fracturing pump, has extremely harsh working environment, and the internal components are often damaged and have a limited life[1-2]. The hydraulic end valve box of the traditional plunger pump has complex intersecting lines, gaps, chamfers and other special structures, which are subject to high-intensity alternating loads and stress concentrations during operation, which are prone to fatigue damage. With the development of shale gas exploitation technology in my country, the power requirement of fracturing pump has been continuously improved, which has aggravated the poor working condition of fracturing pump and made its failure more serious[3]. The use of special processes such as chamfering of intersecting lines and valve box self-enhancement can improve the service life of the valve box at the hydraulic end of the traditional plunger pump, but the problem of fatigue failure has not been fundamentally solved structurally. Therefore, a new type of pressure The hydraulic end of the split pump[4-11].

Today, with the rapid development of computer technology, finite element analysis has become very important in the practical engineering applications. ANSYS Workbench is a kind of software developed by ANSYS that integrates various finite element analysis modules such as statics, dynamics, and fluid mechanics. That Finite element analysis software[12]. Compared with the actual experimental research, the finite element analysis method can obtain the force and deformation of the valve box more accurately and intuitively. The author established the finite element mechanical calculation model of the non-intersecting line valve box with different structures. Based on ANSYS Workbench, the new fracturing pump was analyzed. The force characteristics of the valve box at the hydraulic end are analyzed, and the stress cloud diagram and displacement cloud diagram under different working conditions are compared and analyzed, which provides a new theoretical basis for the optimization and improvement of the valve box structure of the new fracturing pump, and reduces the number of tests and research costs at the same time.

2. Model Building and Meshing

The hydraulic end of the new pressure pump solves the problem of accelerated fatigue damage structurally, cancels the intersecting line between the valve box and the high-pressure hydraulic cylinder of the traditional plunger pump, and adopts a straight barrel cylindrical structure without intersecting lines. Its structure is shown in Figure 1. The suction valve is installed on the hollow plunger, and the axis of the discharge valve is collinear with the axis of the valve box. Concentration issues are resolved.



1- seal compression ring; 2- hollow plunger; 3- suction valve; 4- valve stop lock; 5- discharge valve; 6- exhaust hood; 7- fixing nut

Fig 1. Schematic diagram of new hydraulic structure

2.1. Model Building

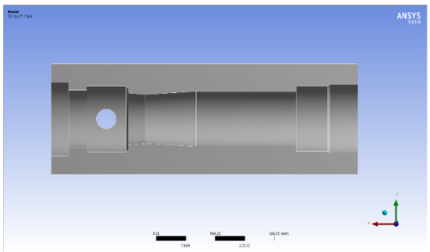


Fig 2. Solid model of valve box

Compared with the DM module that comes with ANSYS Workbench, SolidWorks the characteristics of simple modeling process and easy data modification. Therefore, this paper uses SolidWorks to carry out 3D modeling of valve boxes with 3 different plunger sizes (139.7, 146.1 and 152.4 mm). Then export the format recognized by ANSYS, and then import it into the Static Structure module in ANSYS Workbench. The inner cavity structure of the five valve boxes of this new type of pressure pump is the same as the force, and the single cylinder is a symmetrical structure. In order to improve the calculation speed, under the premise of ensuring the accuracy of the analysis results, this paper takes half of a single cylinder as the research object in the actual analysis [13-14]. Take the maximum working condition of the valve box for finite element analysis, that is, when the pump head body is in the discharge working condition, Figure 2 is the solid model of the valve box.

2.2. Meshing

The established solid model is meshed by the built-in mesh intelligent division method in ANSYS Workbench. The mesh cell size is set to 6 mm, and the mesh element type is selected as the tetrahedral element Solid92. This element has 10 nodes, each node has 3 degrees of freedom, and has plasticity, creep, swelling, stress stiffness, and large deformation. and large resilience [15]. During the analysis, in order to improve the calculation accuracy and reduce the unnecessary calculation amount, the grid is refined at the discharge port position. The grid situation is shown in Figure 3, and the finite element model parameters are shown in Table 1.

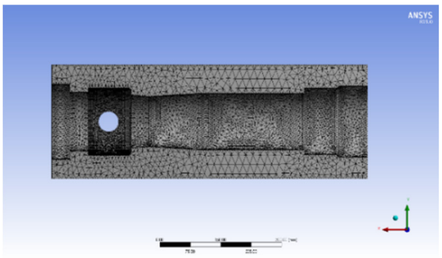


Fig 3. Finite element model of valve box

Table 1. Parameters of the Valve box finite element model

Material Properties and Element Properties	
Material	40CrNiMo
Elastic Modulus	206000MPa
Poisson's ratio	0.3
tensile strength	980MPa
Yield Strength	835MPa
unit type	Solid92
number of nodes	146645
number of units	81208

3. Loading and Constraining

Constraints: According to symmetry conditions, the model does not allow displacement in the z direction on the xOy plane of the symmetry plane; since the valve box is fixed on the fracturing truck, there is no displacement at the bottom of the model in the x, y, and z directions.

Load: In the actual work of the fracturing pump valve box, the change of the plunger diameter and the pump stroke n will cause the working pressure p to change. Therefore, in the finite element analysis and calculation, according to the stroke number n of the valve box with different plunger diameter The changes apply corresponding pressure loads to the inner surface of the valve box, and the loading test parameters are shown in Table 2.

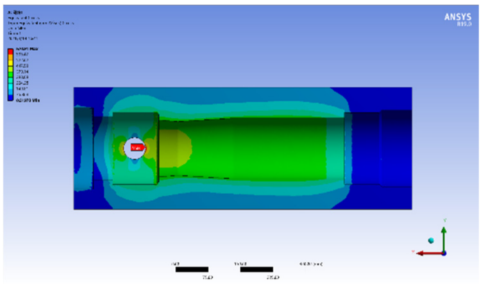
Table 2. Parameters of the loading test

n/min^{-1}	150	200	250	300
p_1/MPa	140	127.5	114.5	101.3
p_2/MPa	119.2	105.3	96.8	86.7
p_3/MPa	98.2	81.6	76.4	63.8

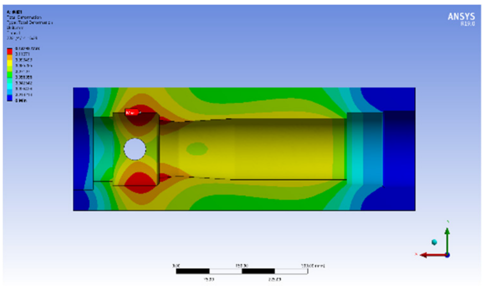
4. Numerical Analysis

4.1. Strength Analysis of Valve Box under Maximum Pressure Load

The finite element analysis results show that: under the discharge condition, the designed maximum pressure load of 140 MPa is applied to the inner surface of the valve box of the new fracturing pump with a plunger diameter of 139.7 mm, and the maximum stress of the valve box is concentrated on the discharge port, σ_{max}= 670.9 MPa, the maximum stress σ_{max}=497 MPa at the installation position of the discharge valve seat, both of which are obviously smaller than the yield strength σ=835 MPa of the material 40CrNiMo, the valve box meets the static strength requirements; the maximum deformation is 0.12793 mm, located on the discharge cavity, the specific position As shown in Figure 4. Due to the valve box structure with no intersecting line, the working cylinder and the discharge chamber always bear high pressure instead of alternating load when working, and the stress value is small and evenly distributed, so there is no obvious alternating stress caused by fatigue problem.



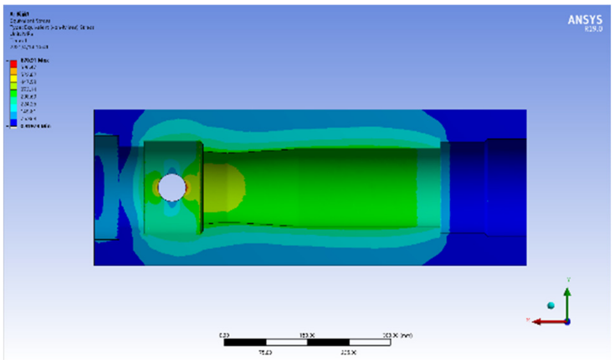
(a) Stress cloud



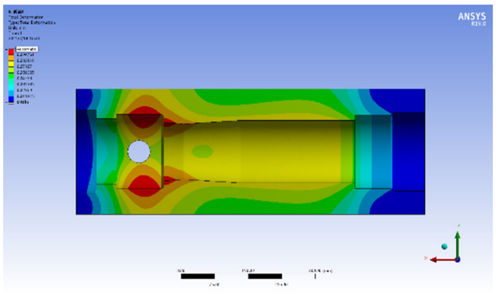
(b) Deformation cloud map

Fig 4. Distribution of the stress and displacement of valve box under 140MPa pressure load

4.2. Influence of Plunger Size on Mechanical Properties of Valve Box



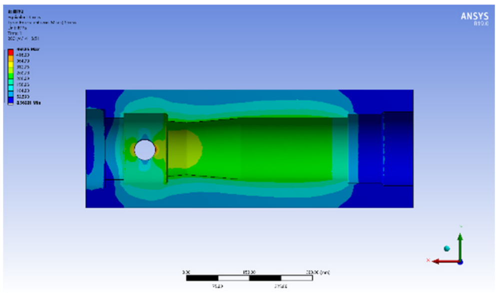
(a) n=150 min^{-1} Stress cloud



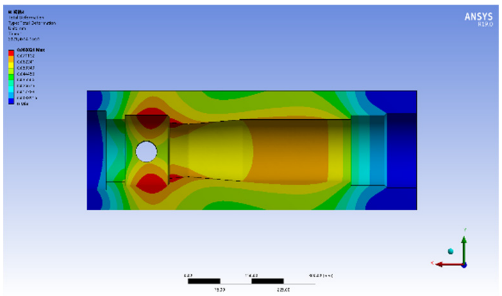
(b) n=200 min^{-1} Stress cloud

Fig 5. Distribution of the stress and displacement of the valve box(139.7mm plunger)

In actual working conditions, the valve box is subject to the greatest stress and deformation in the discharge condition. Therefore, under the discharge condition, the inner surface of the valve box with different structures is applied with a pressure load that changes with the stroke of the plunger. Using the finite element analysis software, The obtained stress nephogram and displacement nephogram are shown in Figures 5 to 7, the maximum stress and total deformation curve are shown in Figure 8, and the mechanical behavior comparison is shown in Table 3.

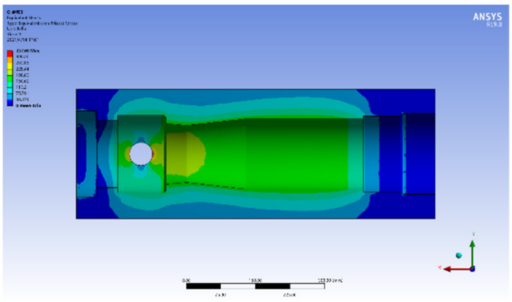


(a) n=200 min^{-1} Stress cloud

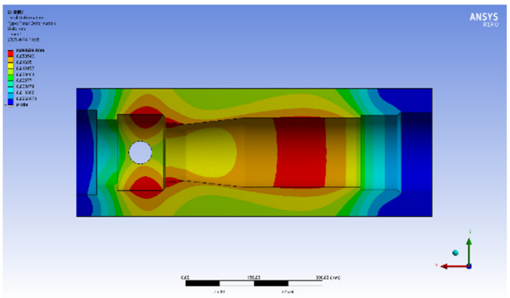


(b) n=250 min^{-1} Deformation cloud map

Fig 6. Distribution of the stress and displacement of the valve box(146.1mm plunger)

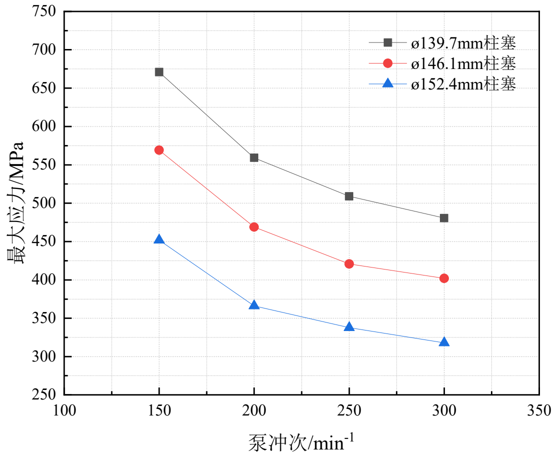


(a) n=250 min^{-1} Stress cloud

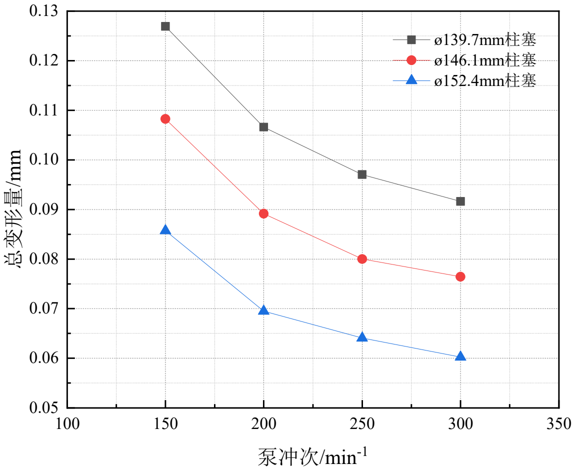


(b) n=300 min^{-1} Deformation cloud map

Fig 7. Distribution of the stress and displacement of the valve box(152.4mm plunger)



(a) Maximum stress as a function of pump strokes



(b) Total deformation as a function of pump strokes

Fig 8. Curves of the maximum stress and total deformation of three valve boxes under different pump strokes

Analysis of the curves of the maximum stress and total deformation of the three valve boxes with the stroke times in Figure 8 shows that: under different internal pressures, when the diameter of the plunger remains unchanged, increasing the pump stroke times, the maximum stress and total deformation The reduction rate of the maximum stress and total deformation of the valve box with the pump stroke of 150~200 r/min is significantly higher than that of 200~300 r/min; when the pump stroke is unchanged When the diameter of the plunger is increased, the maximum stress and total deformation decrease accordingly.

By analyzing Table 3, it can be seen that the other two valve boxes can effectively reduce the maximum stress and total deformation of the valve box compared with the valve box with a plunger diameter of 139.7 mm. The 146.1 mm plunger diameter valve box has an average reduction of 15.1% in the maximum stress value and 15.2% reduction in total deformation compared with the 139.7 mm plunger diameter valve box; The valve box with the plug diameter has an average reduction in maximum stress of 33.3%, and an average reduction in total deformation of 33.5%. When other conditions remain unchanged, increasing the plunger diameter of the valve box can significantly reduce the maximum stress and total deformation on the valve box.

Table 3. Comparison of mechanical behaviors of three valve boxes under different pump strokes

Plunger diameter /mm	n=150 min^{-1}		n=200 min^{-1}		n=250 min^{-1}		n=300 min^{-1}	
	maximum stress / MPa	total deformatio/ mm	maximum stress / MPa	total deformatio/ mm	maximum stress / MPa	total deformation/ mm	maximum stress / MPa	total deformation/ mm
139.7	670.90	0.12793	559.09	0.10661	508.89	0.09703	480.59	0.09164
146.1	569.23	0.10827	468.86	0.89177	420.74	0.08002	401.94	0.07645
152.4	451.77	0.08573	366.20	0.06949	337.68	0.06408	317.91	0.06023

5. Conclusion

(1) Under the designed maximum pressure load of 140 MPa, the maximum stress of the valve box is 670.9 MPa, the yield strength of the material 40CrNiMo is σ=835 MPa, $\sigma_{max}<\sigma$, and the valve box meets the strength requirements.

(2) The finite element analysis results show that the maximum stress of the valve box is concentrated on the discharge port, and the maximum total deformation appears on the discharge cavity. The stress value of the working cylinder and the discharge cavity is small and evenly distributed, so there is no fatigue problem caused by obvious alternating stress.

(3) When the diameter of the plunger remains unchanged, increase the pump stroke, and the maximum stress and total deformation decrease accordingly. The maximum stress and total deformation of the valve box with a pump stroke of 200~300 r/min decrease The small rate is significantly lower than the reduction rate of 150~200 r/min.

(4) When the number of strokes of the pump remains the same, increase the diameter of the plunger, and the maximum stress and total deformation decrease accordingly. Compared with the valve box with the plunger diameter of 139.7 mm, the maximum The mean reduction in stress was 15.1%, the mean reduction in total deformation was 15.2%, and the mean maximum stress reduction was 33.3% for the 152.4 mm plunger diameter valve box compared to the 139.7 mm plunger diameter valve box, and the total deformation The mean reduction in volume was 33.5%.

References

[1] Hu Jinqiu, Wang Qianlin, Zhang Laibin, Wang Haitao, Li Kuiwei, He Weiwei. Fatigue Failure Evolution Law of Fracturing Pump under Variable Pressure Condition [J]. China Petroleum Machinery, 2017,45(04):67-73.

[2] Zhong Gongxiang, Liang Zheng. Analysis and Design of Power End of HQ1400 Fracturing Pump [J]. Oil Field Equipment, 2007(12):24-26.

[3] Gao Yuan, Shan Dongsheng, Li Wenliang, Wang Hongling. Finite Element Analysis of Fatigue Life of Crankshaft of 140MPa Fracturing Pump [J]. Oil Field Equipment,2015,44(02):68-71.

[4] Zhou Sizhu, Zhou Houjun, Hua Jian, Chen Ming, Jing Huabin. Research on the Influence of Material on the Self-reinforcing Performance of the Pump Head of Ultra-high Pressure Fracturing Pump [J]. Machinery Design & Manufacture,2010(07):67-68.

[5] Zhang Si, Zhou Sizhu, Li Ning. Science Technology and Engineering, 2017,17(05):185-190+223. Study on stress field and stress intensity factor at crack tip of fracturing pump head intersecting line [J]. Science Technology and Engineering, 2017,17(05):185-190.

[6] Hua Jian, Liao Jianmin, Zhou Sizhu, Li Ning, Cu-i Hao. Science Technology and Engineering, 2015, 15 (35): 170-174. (in Chinese).

[7] Xu Liangle, Wu Gaoming, Si Tingzhi. Analysis of Cracking Reason and Improvement Measures of Oil Fracturing Pump Valve Box Cross Hole [J]. Heat Treatment of Metals,2018,43(03):233-236.

[8] He Xia, Zhao Min, Chen Linyan, Li Rong. Simulation Research on Self-reinforcing Technology of Hydraulic Fracturing Pump Valve Box [J]. Mechan-ical Science and Technology for Aerospace Engineering, 2014,33(02):208-211.

[9] Wang Guorong, Hu Gang, He Xia, Zhao Min. Fatigue Life Prediction of Steel Used in Hydraulic Fracturing Pump Valve Box Based on Grey System Theory [J]. Hot Working Technology, 2015, 44 (06): 48-52.

[10] Chen Liawang, Cui Xiaoyu, Chen Liaoyang, Mei Yuguang, He Jin. Petroleum & Chemical Equip-ment, 2010, 13(05):20-23. Finite Element Analysis and Design of Valve Box without Intersection Lin-e in Fracturing Pump [J].

[11] MA Yulong. Research on Hydraulic End of 6500HP Fracturing Pump [D]. Lanzhou University of Technology, 2016.

[12] LI Bing, HE Zhengjia, CHEN Xuefeng, ANSYS Workbench Design Simulation and Optimization [M]. Beijing: Tsinghua University Press, 2008.

[13] Tian Qin, Wang Yuanzhong, Liu Wenbao, Zhang Tianlin, Zhu Weidong. Finite Element Analysis of Key Components in Hydraulic End of Fracturing Pump Based on ANSYS Workbench [J]. Manufacturing Automation,2016,38(09):32-34.

[14] Yao Zhiming, Yi Xianzhong, Zhou Yuanhua, Yi Jun, Yin Guangpin, Chen Hui, Song Shunping. Analysis of physical characteristics of pump head of CDJY2500 five-cylinder fracturing pump [J]. China Petroleum Machinery, 2019,47(10):81-87.

[15] Roccatello A, MancòS, Nervegna N. Modelling a variable displacement axial piston pump in a multibody simulation environment [J]. Journal of Dynamic Systems, Measurement and Control, 2007, 129 (4):456-468.