Finite element analysis of annular BOP shell

Ling Feng
School of Southwest Petroleum University, Sichuan 610500, China
895720417@qq.com

Abstract

Through Creo 3D software modeling, combined with ABAQUS software, the finite element analysis of annular BOP shell is carried out to obtain its load distribution, which can determine the dangerous area with high stress in the structure of annular BOP. Taking FH35-35 / 70 annular blowout preventer as an example, this paper analyzes the Mises stress and displacement distribution under its working condition and hydrostatic pressure.

Keywords

Annular BOP; Stress; Displacement Nephogram; Finite Element Analysis.

1. Introduction

According to the working principle and failure analysis results of the annular BOP, the failure sensitive components of the annular BOP are rubber core and shell. The rubber core is a vulnerable part but can be replaced. Once the shell fails or the strength is not enough, the whole BOP can not be used again. Therefore, the structural safety of the annular BOP shell is the focus of our research.

The shape of the annular BOP shell is relatively simple, but the size gradient changes greatly. In practice, the inner cavity of the annular BOP is filled with high-pressure oil and gas. Under the action of fatigue damage and impact load, different types of defects, such as crack, may appear in the stress concentration position of the shell. Therefore, it is necessary to analyze the performance of BOP shell, especially the stress concentration phenomenon of BOP under working condition.

2. Mechanical model of annular BOP

The annular blowout preventer is a thick walled pressure vessel. The influence of the local size of its shell can be ignored. The shell is simplified as a thick walled cylinder model. The mechanical model is shown in Fig. 1, and the following assumptions are made:

1) There are no defects in the structure and the material of BOP is incompressible, which is an ideal elastic-plastic material;

2) The material constitutive model is a linear elastic model with local yield;

3) The thick walled cylinder is only subjected to internal pressure and can be regarded as an axisymmetric plane strain problem.

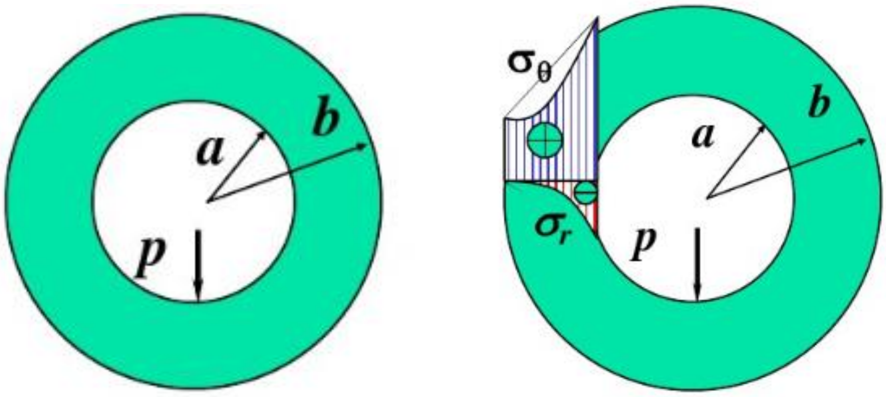


Fig. 1 Mechanical model of annular BOP

The model is a plane stress problem in elasticity, and the stress solution on its cross section can be calculated by using the linear elastic calculation method. The stress boundary conditions are as follows:

Inner boundary

$$\begin{cases}\sigma_{r|r=a} = -p \\ (\tau_{r\theta})_{r=a} = 0\end{cases} \tag{1}$$

Outer boundary

$$\begin{cases}\sigma_{r|r=b} = 0 \\ (\tau_{r\theta})_{r=b} = 0\end{cases} \tag{2}$$

Stress component

$$\left.\begin{aligned}\sigma_r &= \frac{A}{r^2} + 2C \\ \sigma_\theta &= -\frac{A}{r^2} + 2C \\ \tau_{r\theta} &= \tau_{\theta r} = 0\end{aligned}\right\} \tag{3}$$

Combined with the above formulas, the solution is as follows:

$$A = \frac{a^2b^2p}{a^2 - b^2}, \quad C = \frac{a^2p}{b^2 - a^2}$$

The Lame´ solution as follows：

$$\begin{cases}\sigma_r = \dfrac{a^2p}{b^2 - a^2}\left(1 - \dfrac{b^2}{r^2}\right) \\ \sigma_\theta = \dfrac{a^2p}{b^2 - a^2}\left(1 + \dfrac{b^2}{r^2}\right)\end{cases} \tag{4}$$

The solved displacement component as follows:

$$\begin{cases}u = \dfrac{1}{E}\left[-(1 + \nu)\dfrac{A}{r} + Cr(1 - \nu)\right] \\ \nu = 0\end{cases} \tag{5}$$

$$u = \frac{1}{E}\frac{a^2p}{b^2 - a^2}\left[(1 + \nu)\frac{b^2}{r} + (1 - \nu)r\right] \tag{6}$$

According to formula (4), the stress distribution along the wall thickness of the annular blowout preventer can be obtained. At the same time, there are axial surface cracks or buried cracks. The tensile stress can also be calculated according to (4), and the transverse strain displacement of the annular BOP can be calculated according to formula (6).

3. Finite element analysis of annular BOP

3.1. Establishment of model

3.1.1. Structural parameters

The shell of FH35-35 / 70 annular BOP is composed of a top cover and a lower shell. The three-dimensional model of FH35-35 / 70 annular BOP is established by reasonable simplification in reference [1]. The key to simplify the complex model is to take the top cover and lower shell of the BOP as a whole, and omit some small structures, such as lifting lug, sealing groove, connecting thread between top cover and shell, etc. The BOP shell is a symmetrical rotating body. In the analysis, 1 / 4 of its structure can be taken for modeling and analysis, and the model shown in Fig. 2 can be obtained.

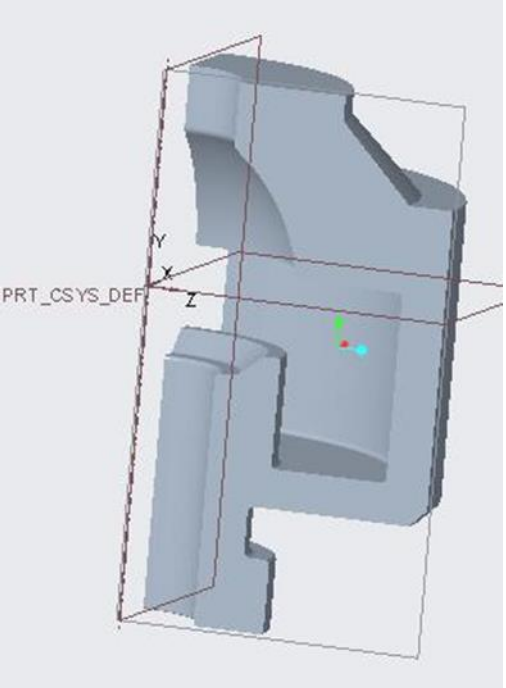


Fig. 2 1 / 4 model of annular BOP

According to the data and calculation method provided by API 16a, the structural parameters of annular BOP are determined (as shown in table 1) .

Table 1 Technical parameters of FH35-35/70 annular BOP

Diameter	346mm
Dimensions	$\phi 1270 \times 1225$mm
wall thickness	101mm
working pressure	35MPa
Strength test pressure	51MPa

3.1.2. Material properties

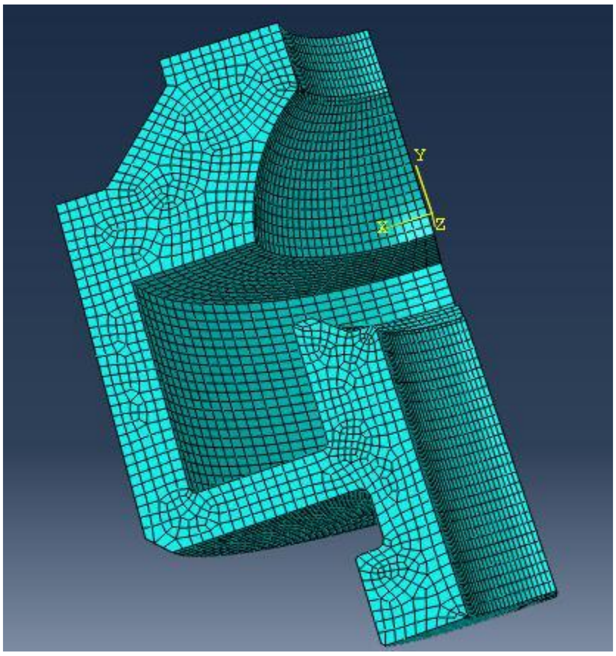
The material of blowout preventer is 25CrNiMo carbon structural steel, which is cast, with high hardness and good machinability. After quenching and tempering, the surface of annular BOP still has good wear resistance, while the toughness and plasticity of low carbon steel are still maintained in the central part. The material parameters required for finite element analysis are shown in table 2.

Table 2 Material parameters of 25CrNiMo

Material name	Elastic modulus (GPa)	Poisson's ratio	Yield strength (MPa)	Tensile strength (MPa)
25CrNiMo	216	0.3	421	724

3.2. Meshing and load setting

The model drawn in Creo is imported into ABAQUS. After the model is assembled, it is meshed. As shown in Fig. 3, the hexahedral shape element is used to divide the mesh. The global size is set to 20, and a total of 29820 nodes and 25461 elements are divided.



摘要

结点总数: 29820

单元总数: 25461

详细资料

	实例名称	单元类型	单元	结点
1	fpq-1		25461	29820
2		C3D8R	25461	

Fig. 3 Mesh generation model of annular BOP

As shown in Fig. 4, the following boundary constraints are applied to the meshed model: constraints are imposed on the flanges of the BOP top cover and lower shell to limit their rotation and displacement; symmetrical constraints in X and Z directions are applied to the two cutting planes of the model. During the finite element analysis, the working condition of hydrostatic test is simulated. The internal pressure of BOP shell is uniformly distributed, and the pressure is perpendicular to the surface of BOP cavity, and the applied load is 35Mpa.

3.3. Result analysis

According to the analysis of the stress nephogram in Fig. 5, under the working pressure of 35MPa, the stress concentration appears at the corner of the lower shell, the inner cavity and arc-shaped inner wall of the annular BOP, and the maximum stress occurs at the corner, which is 274.4Mpa. If fatigue failure occurs, it should appear in these positions first. And the maximum equivalent stress 274.4Mpa is less than the yield strength of 420MPa, which indicates that under the rated working pressure of 35MPa, the annular BOP shell material is in elastic state, the safety factor is 1.53, and the structural strength of the annular BOP meets the requirements.

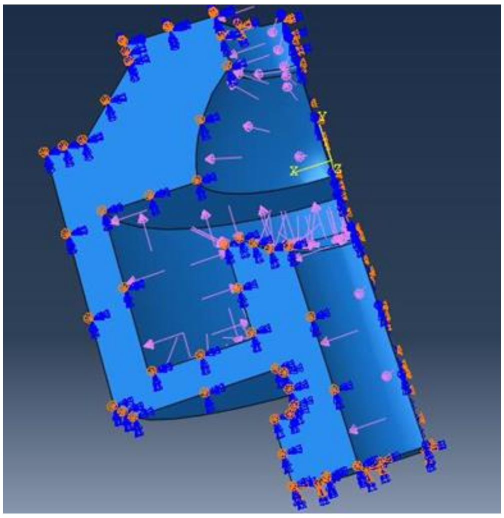


Fig. 4 Stress and restraint model of annular BOP shell

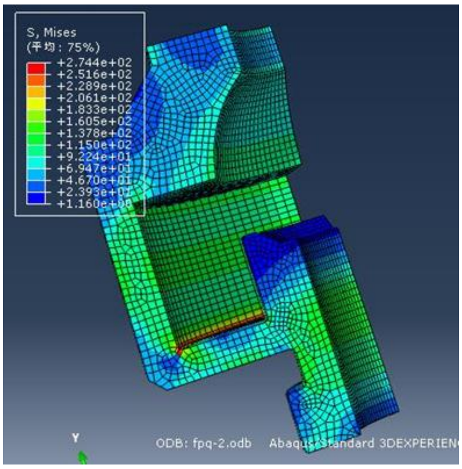
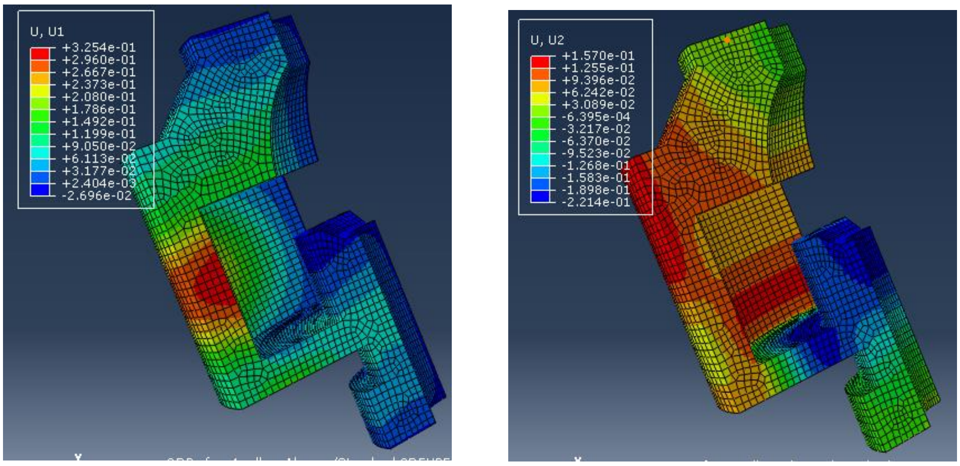
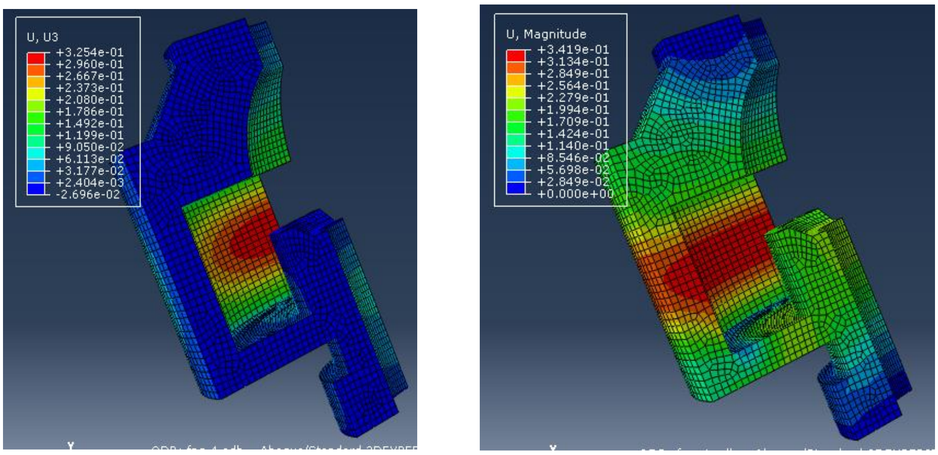


Fig.5 Stress distribution nephogram of annular BOP shell

Through the analysis of the displacement nephogram in Fig. 6, it can be seen that the maximum displacement of the shell occurs in the middle of the annular cavity of the shell. The maximum deformation of the shell is 0.32mm in the X and Z directions, 0.15mm in the Y direction, and the total deformation is 0.34mm. It can be seen that the total deformation of the shell is almost provided by the deformation in the X and Z directions, that is, the circumferential deformation. The stress and deformation are within the allowable range, which proves that the BOP structure is safe and meets the requirements.



a) X-direction displacement nephogram b) Y-direction displacement nephogram



c) Z-direction displacement nephogram b) Total displacement nephogram

Fig.6 Cloud chart of annular BOP shell displacement

4. Summary

In this paper, the mechanical model of the annular BOP shell is established, and the three-dimensional modeling of the annular BOP shell is carried out by Creo. The finite element analysis is carried out in ABAQUS. The maximum equivalent stress and maximum deformation of the annular BOP shell under hydrostatic pressure are calculated

(1) The mechanical model of simplified annular BOP shell is thick wall cylinder, and the relative error of stress value of fh35-35 annular blowout preventer calculated theoretically is less than 5%, which verifies the correctness of the finite element model;

(2) The maximum equivalent stress of the BOP shell occurs at the corner of the lower shell of the BOP, and the maximum displacement of the shell occurs in the middle of the annular cavity of the shell. The stress and deformation meet the safety requirements of the BOP.

References

[1]. Zhao Jun, Tang Yang. Finite element analysis and experimental study of the annular blowout preventer shell[J]. Petroleum Machinery, 2013, 41(002): 87-90.

[2]. Wang Dongxue, Wu Zhanwei, Qiaoying, et al. Finite element analysis of FHZ48-70 annular blowout preventer for underwater use[J]. Petroleum Field Machinery, 2013(09):17-20.

[3]. Yan Jinlin, Fan Chunming, Li Zhonghua, et al. Research on the strength analysis method of ram blowout preventer[J]. Mechanical Engineer, 2020, No.347(05):147-149.

[4]. Rassenfoss, S. (2012, August 1). The Keywords For Blowout Preventers Are Trust But Verify. Society of Petroleum Engineers. doi:10.2118/0812-0040-JPT

[5]. Wang Juanhu. Defect evaluation research and safety evaluation software system development of in-service annular blowout preventer shell[D]. Southwest Petroleum University, 2013.