Digital Manufacturing & Automation III

Part 2

Edited by
Yonghong Tan

TTP TRANS TECH PUBLICATIONS

Chapter 4: Deep Sea Mining Equipments, Complex Equipment Design and Extreme Manufaturing

Applied Mechanics and Materials Vols. 190-191 (2012) pp 832-835

doi:10.4028/www.scientific.net/AMM.190-191.832

Structural Finite Element Analysis and Optimization of Underwater High Pressure Valve

Yi Xianzhong [1, a], Jiang Shengzong [2,b], Zhang Junfeng [3,c], Feng Ding[4,d] and Gao Deli[5,e]

[1] School of Mechanical, Engineering Yangtze University, Jingzhou, Hubei 434023, China

[2] Cavaville Energy Services Ltd, Beijing, Beijing 100028, China

[3] School of Mechanical, Engineering Yangtze University, Jingzhou, Hubei 434023, China

[4] School of Mechanical, Engineering Yangtze University, Jingzhou, Hubei 434023, China

[5] China University of Petroleum, Key Laboratory of Petroleum Engineering, Ministry of Education, China University of Petroleum, Beijing 102249, China

[a]yxz@yangtzeu.edu.cn, [b]ken.jiang@cavaville.com, [c]zjflove7788@yahoo.com.cn

[d]fengd0861@sina.com, [e]gaodeli@cup.edu.cn

Keywords: Ship side flap valve finite element analysis, Valve body.

Abstract. The ship side flap valve underwater works with high pressure. In this paper, the valve body is analyzed using seamless connection between the finite element analysis software and Pro/E software. In the analysis, multiple load cases applied to the valve body are as follows: open condition, closed condition and seal test. The analysis result shows the maximum stress value occurs when the flap valve is open. Moreover, stress value is approximately 65 percent the range of the minimum yield strength, but the strength requirement of the valve body is satisfied. Finally, this paper presents a method for weakening the stress concentration effect by increasing the internal transitional fillet radius between the two hollow and intersectant cylinders of the valve body.

Introduction

Flapper type drain valve is a critical component for large ships and submerged submarines. It works in harsh environment and bears very high pressure. If the strength or stiffness of the valve is not enough, it will result in fluid leakage and other serious accidents. This paper adopts the finite element numerical analysis method to do mechanical behaviors analysis when the valve is under different working conditions. The purpose of the analysis is to judge the stress concentration area and the weak links. Finite element analysis is an effective method for strength analysis and safety assessment of high building, submarine, spacecraft, and other engineering structures [1,2,3].

Operating Characteristics of Ship Side flap Valve

Ship side flap valve underwater is a mechanical plug valve, which is operated manually. The flap valve belongs to the high pressure products according to the pressure rating classification of vessels. The normal operating pressure is about 10MPa.The relative attributes of the valve is shows in Table 1.

Table 1 The relative attributes of the valve

material	ultimate tensile strength	ultimate compressive strength	applied situation	characteristic feature
ZCuAl18Mn13 Fe3Ni2	645MPa	280MPa	sea water, fresh water, oils and other media	meet GB600-91 technical conditions

In the process of the finite element analysis, it will apply Pro/E software to create three-dimensional solid model and integrate ANSYS software in Pro/E [4]. The solid45 element is adopted according to the structural feature and loading type.

The symmetry plane only exists deformation in the direction of X and Y. So it has to choose the symmetry of the model and to apply load in the X direction.

Finally, add completely restraint at the marked location (Fig. 1 and Fig. 2), referring to the principle of the valve. Note: It only shows one radius of the constraints at the corner to be R5. Other constraints at the curvature radii are all the same.

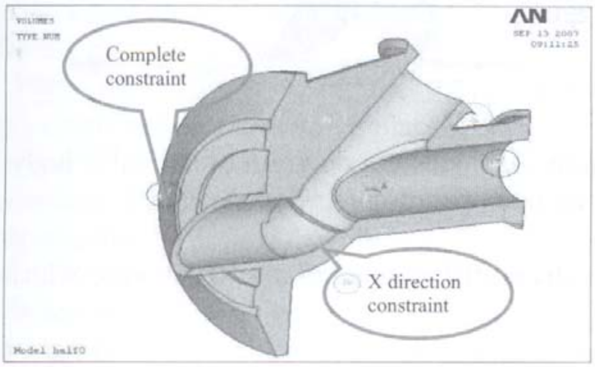


Figure 1 Constraint condition of the valve body (R=5mm)

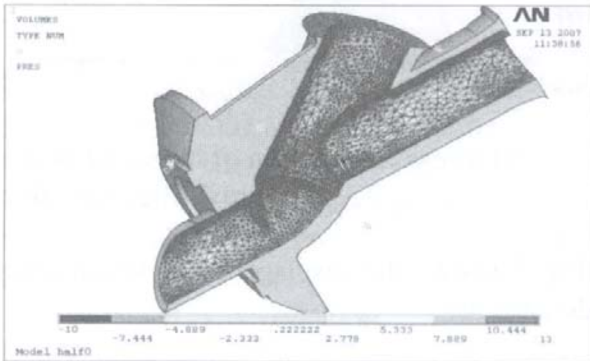


Figure 2 Load applied to the open valve (R=5mm)

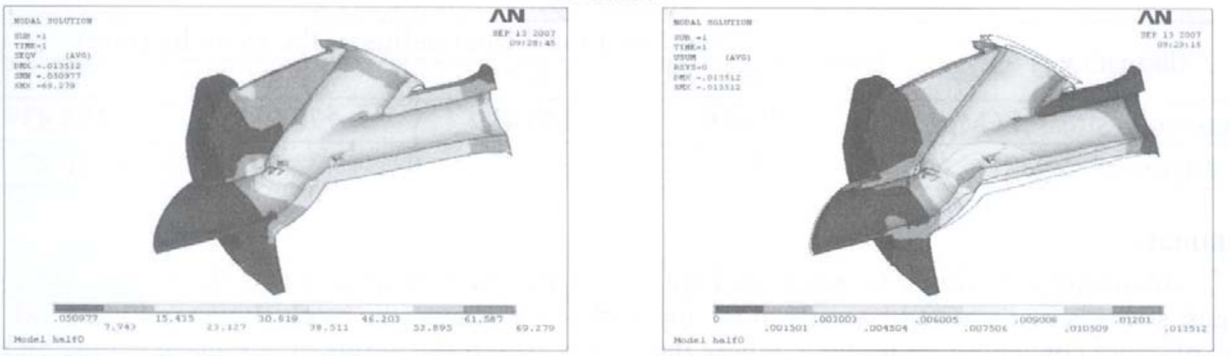
Finite Element Analysis of Typical State of the Valve

Isoparametric element and standard element of Solid 45 can be described as the hexahedral structure, the displacement function of the elements can be described as Eq. 1:

$$\begin{cases} u(\xi,\eta,\zeta)=\sum_{1}^{n} N_i(\xi,\eta,\zeta)u_i(\xi,\eta,\zeta) \\ v(\xi,\eta,\zeta)=\sum_{1}^{n} N_i(\xi,\mu,\zeta,)v_i(\xi,\eta,\zeta) \\ w(\xi,\eta,\zeta)=\sum_{1}^{n} N_i(\xi,\eta,\zeta)w_i(\xi,\eta,\zeta) \end{cases} \tag{1}$$

Where, $N_i(\xi,\eta,\zeta)$ is the shape function; c, $u_i(\xi,\eta,\zeta)$, $v_i(\xi,\eta,\zeta)$, $w_i(\xi,\eta,\zeta)$ are the displacement component of the node; n is the numbers of the nodes; $u(\xi,\eta,\zeta)$, $v(\xi,\eta,\zeta)$, $w(\xi,\eta,\zeta)$ are the displacement of the element.

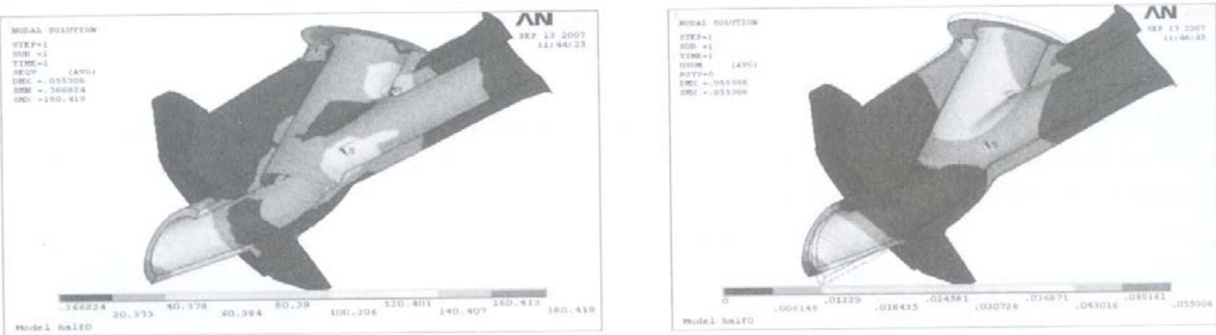
Fig. 3 shows the Displacement, deformation and stress distribution of the flap valve which is closed.



(a) Stress distribution of the valve body (b) Displacement diagram of the valve body

Figure 3 The flap valve is in a closed position (R=5mm)

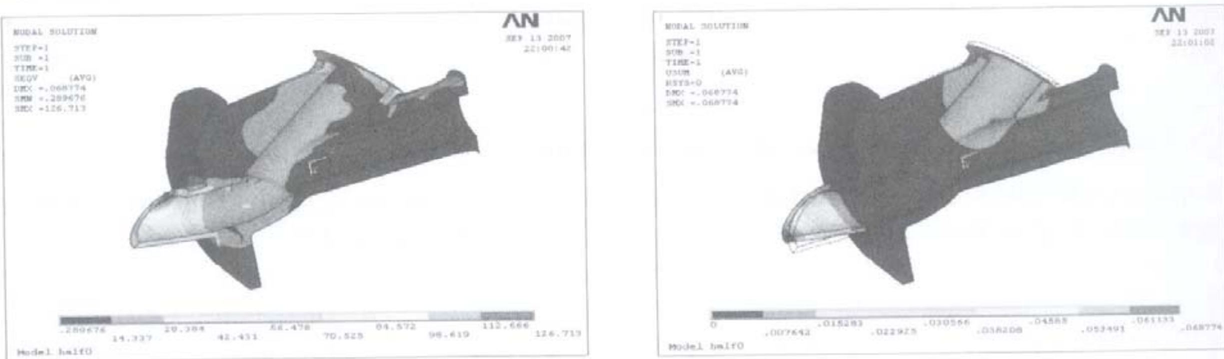
Fig. 4 shows the displacement, deformation and stress distribution of the flap valve which is open.



(a) Stress distribution of the valve body (b) Displacement diagram of the valve body

Figure 4 The flap valve is in an open position (R=5mm)

Fig. 5 shows the displacement, deformation and stress distribution of the flap valve which is under seal test.



(a) Stress distribution of the valve body (b) Displacement diagram of the valve body

Figure 5 The flap valve is in a seal test position (R=5mm)

The Relationships Between the Transitional Fillet Radius and the Maximum Stress of the Open Valve. Table 2 makes a list of the maximum stress of the open valve with the transitional fillet radius of the two oblique and hollow and intersectant cylinders R, the thickness of the valve S_B, the relative fillet radius of the inner wall of different valve R0.R_0 can be written as:

$$R_0=R/S_B. \tag{2}$$

Tables 2 Relationships between the transitional fillet radius and the maximum stress of the valve which is open.

flap valve is open	the transitional fillet radius of the valve R_0 (mm)			
	0	0.5	0.8	1.0
maximum stress （Mpa）	220.886	180.419	170.513	184.939
relative change（%）	1	18.32	22.80	16.27

Summary

1. Compared with Figure 6a and Figure5(a), when the valve is open, it gets the largest stress. The largest stress is just situated laterally at the top part of the seams (the mark shows in Figure 7a). The area of stress concentration is small, where the design size of the radius of curvature is only 5mm. If the internal pressure reaches to 10MPa, the maximum stress at the stress concentration areas reaches 180.419Mpa, but the ultimate compressive strength σ_s is only about 280Mpa. So the large plastic deformation will not appear and the results indicate the design can meet the strength requirements.

If the valve is closed, the maximum stress appears at the sealing surface between the valve plate and the valve body. The maximum stress is about 69.279Mpa. It is only about 38.4 percent of the maximum stress when the open valve bears 10MPa pressure. It meets the strength requirement.

2. When the valve is under seal test, the deformation value is greater compared with the values when the valve is closed or open. But the test displacement value is about 10mm and it is less than 0.7 percent of the thickness of the wall. It meets the stiffness requirement.

When the valve is closed, the max displacement appears at the contact segment between the valve plate and the valve body. The stress value is only 0.0135mm, which is very small compared to the thickness of the wall, and it is less than 0.2 percent of the thickness of the wall. It satisfies to the stiffness requirement well.

When the valve is open, the valve works under 10Mpa pressure. The stress of the contact component between the valve plate and valve body is released, and the stress distribution condition changes. The analysis shows that the maximum stress appears at the top part of the joint and the maximum displacement appears at the exit surface because the total bending deformation is restricted by the boundary. The maximum displacement is about 0.055mm and it is less than 0.6 percent of the thickness of the wall. The stiffness requirement is satisfied.

When the valve is under seal test, the maximum displacement is restricted by the boundary and the inner pressure is 12.5MPa. The maximum displacement appears at the outlet and the maximum value is 0.069mm. The value is small amount to the thickness of the wall, and it only reaches to 0.7 percent of thickness of the wall. The stiffness requirement is satisfied.

3. It can improve the stress concentration after changing the transitional radius of the two hollow and intersectant cylinders.

The analysis result shows that when the valve is open, it bears the large stress. It can improve the stress concentration formally by increasing the transitional radius R. Table 1 shows that if the relative transitional radius R_0=0.8, the stress concentration will drop by 22.8 points of percentage. Therefore, it recommends the best transitional radius is about 0.5-0.8 times the thickness of the wall. The value is only for information for the valve design.

Acknowledgment

This research is partially supported by National Nature Science Foundation of China (No. 51174035, 50974023 and 50874019), National Science and Technology Major Project (No. 2011ZX05009-005), and, by Science and Technology Research Program of Hubei Provincial Department of Education (No. CXY2009A007).

References

[1] Zhou Sizhu ,Yuan Xinmei and Luo Yingping: Analysis of well head control valve using finite element method compared with the simplified calculation [J]. Journal of Oil and Gas Technology. Vo. l27, No (2), (2005), p. 256-257.

[2] Wangmin, Liu Jinliang and Wang Jiaqing: Analysis of supercritical globe valve body using finite element method [J]. Valve. No (3) (2006), p. 20-23.

[3] Cao Guoqiang and Bao Mingyu: Structural finite element analysis of throttle valve [J].General Machinery. No (1), (2002), p. 51-53.

[4] Zhangbo and Sheng Hetai. In: ANSYS finite element analysis theory and engineering applications [M]. Publishing, Tsinghua University PRESS, Beijing (2005), in press.

9 783037 854624

ISBN-13: 978-3-03785-462-4
Applied Mechanics and Materials Vols. 190-191, Part 2
Electronically available at *http://www.scientific.net*