Research Article
Ghanashyam Chendke, IJPRET, 2013; Volume 1(11): 9-15

ISSN: 2319-507X
IJPRET

INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

STATIC STRUCTURAL ANALYSIS OF PISTON VALVE BONNET

GHANASHYAM MAHALING CHENDKE[1],

Prof. M. M. MIRZA[2]

IJPRET-QR CODE

PAPER-QR CODE

1. M.Tech II, Mechanical Dept., Rajarambapu Institute of Technology, Islampur, Maharashtra, India.
2. Professor, Mechanical Dept., Rajarambapu Institute of Technology, Islampur, Maharashtra, India.

Accepted Date:
30/07/2013
Publish Date:
01/08/2013

Keywords

Microstrip Transmission Line
Ultra Wide Band
Planar Antenna

Corresponding Author

Mr. Ghanashyam Mahaling Chendke

Abstract

The piston valve is an on-off valve, used to allow/restrict the flow of fluid from pipe by means of the linear motion of piston within a chamber or cylinder. The valve bonnet is the main component which is exposed to the high pressure & high temperature, so the design of this component should be safe to make the valve leak-proof. For the Structural analysis, 10 Noded structural tetrahedral element is used .For the analysis, advantage of half symmetry is taken as the component and boundary conditions are symmetric about one axis.The static structural analysis is carried at 76 bar g pressure and the Mesh sensitivity analysis is done. Mesh Sensitivity analysis finalizes the element size for the analysis. The 4mm element size is the best for this component as the results for 3mm element size are nearly same and the time required to solve the analysis with 4mm element size is less than the 3mm element size. This 4mm element size is used for further analysis with this component as it is time saving. The analysis revealed that the component is safe as maximum von misses stress is 136 MPa (Factor of safety-1.86) and the component can sustained maximum pressure of 145 bar g (Factor of safety-1.02). The maximum total deformation at 76 bar g occurs at the flange is 0.049367mm.From the results, it is concluded that the component is safe at the pressure and maximum Total deformation is also negligible, valve bonnet can sustain all these conditions safely. The location of maximum Von Misses Stress is near to the junction where cylindrical parts meets flat surface section and maximum total deformation occurs at the end of flanges. Both these locations will not arise any functional issue.

1. INTRODUCTION :

A 'piston valve' is a device used to control the motion of a fluid along a tube or pipe by means of the linear motion of a piston within a chamber or cylinder. Piston valve is a zero leak isolation valve. It is an on-off valve. Valve bonnet (Fig.1) contains the other components which are either supports the mechanism or arrest the leak. Valve bonnet is the cast component (Cast Steel) which is machined to fulfill other condition of assembly and proper function.

2.THE NEED FOR ANALYSIS:- The valve bonnet is the main component which is exposed to the high pressure & high temperature, so the design of this component should be safe to make the valve leak-proof and hence it important to check the model for both static structural analysis to finalize/modify the design.

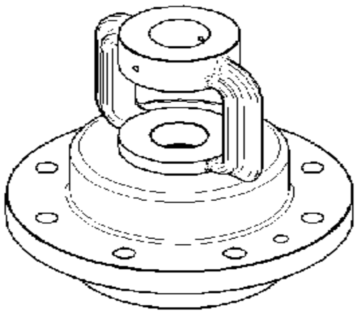


Fig.1 Valve Bonnet (Isometric View)

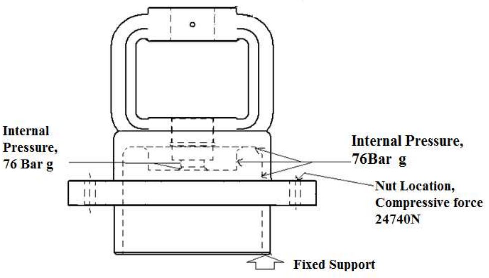


Fig.2 Bonnet Boundary conditions

3.

STATIC STRUCTURAL ANALYSIS:

3.1 Use of Half symmetry: The valve bonnet is having the symmetry about one axis and boundary conditions are also same about this axis. Hence the advantage of half symmetry has been taken to reduce the no. of elements intern reduce the no of equations to solve. It also reduces meshing & solving time.

3.2 The boundary conditions (fig.2) are

i. Fixed support at the bottom

ii. Internal pressure -76 bar g.

iii. Compressive force (preload) by Nut -24740N.

Torque required to tighten the Nut* [1] (T) = thread coefficient of friction [2] (c) x Nominal bolt diameter* (d) x preload force (F)

This preload force is compressive in nature. The properties of cast steel [3] are used for analysis.

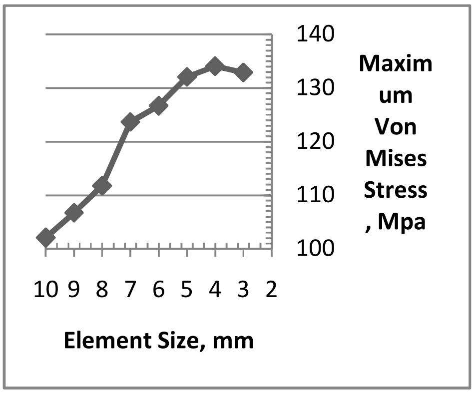
3.3 Mesh sensitivity analysis: Mesh [4] is done using Ansys 'Solid 187' [5] (Fig.3) - 10 Noded structural tetrahedral element. Mesh Sensitivity analysis finalizes the element size for the analysis and this size is used for further analysis. The element size is fixed by comparing the results of analysis for the two different, consecutive element sizes. If the result are same then the element with higher value is finalized as it will reduce the no. of equations to solve and will lower the solving time.

Mesh Sensitivity Analysis (Table 1)is carried out to obtain the optimum element size. The results obtained are given below which shows 4 mm element size is best, as the results are converging at this element size. The element size 3 mm will not only consume more time for meshing, solving but the results are also same when element size 4 mm is used. This element size is used for further analysis to obtain the results at various design conditions and to obtain the maximum possible pressure that can be sustained by the valve body.

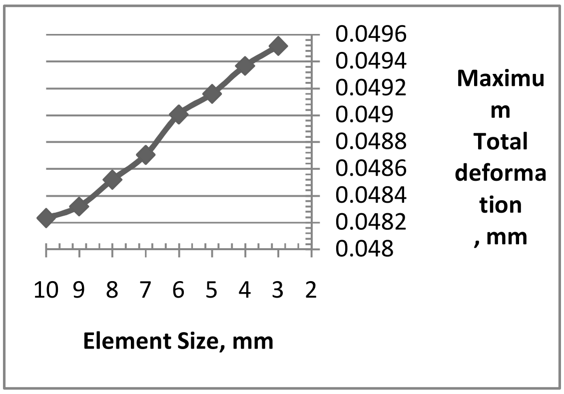
3.4 Results: The results (Table 2) obtained for 76 bar g (Maximum Designed Pressure) are within the safe limit. The Von Mises stress is 136.06 MPa (Fig.5) with factor of safety 1.86. The total maximum deformation is 0.049367 mm (Fig.6) and this deformation is at the free end which is negligible. The component can sustained the maximum pressure of 145 bar g pressure at this wall thickness. From the 100 bar g pressure, the location of maximum total deformation occurs where the gland stack rests (Fig.7).

Table 1. Mesh Sensitivity analysis

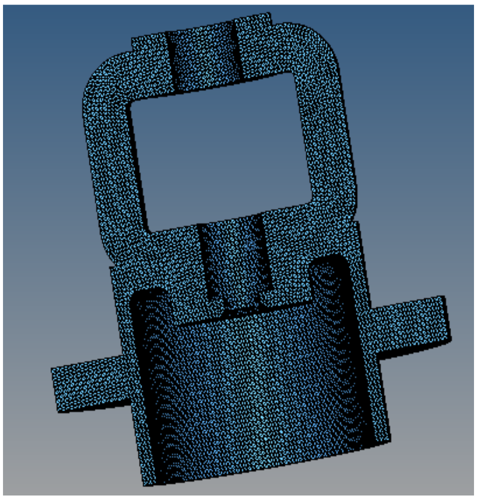
Sr. No.	Element Size, mm	No. of Elements	Maximum Von Mises Stress , Mpa	Maximum total deformation, mm	Percentage increase or decrease in the result from Previous		Time to solve, Min.
					Stress	Deformation	
1	10	33577	102.08	0.048233	------	------	0.41
2	9	45425	106.76	0.04832	4.5846395	0.18004967	0.55
3	8	63710	111.79	0.048518	4.7115024	0.40809596	0.73
4	7	95698	123.67	0.048703	9.6062101	0.3798534	1.1
5	6	150858	126.67	0.049005	2.3683587	0.61626365	1.72
6	5	259462	132.09	0.049158	4.1032629	0.3112413	2.95
7	*4*	*505345*	*134.06*	*0.049367*	*1.4694913*	*0.42335973*	*7*
8	3	1202337	132.9	0.049516	-0.872837	0.30091284	30.12



Graph 1.Element Size vs. Von Misses Stress



Graph 2.Element Size vs. Total deformation



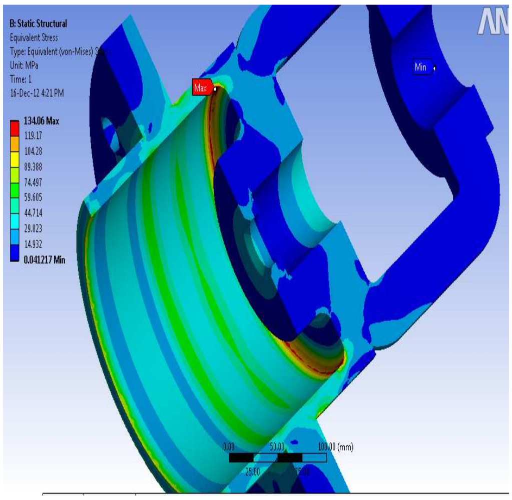


Fig.4 .Mesh (Element size 4mm)

Fig.5 Maximum Von Mises stress at 76 Bar g pressure

Table 2. Results of Static Structural Analysis

Sr no.	Applied Pressure, bar g	Maximum Von-Misses Stress, Mpa	Maximum Total Deformation, mm	Factor of Safety
1.	76	134.06	0.049367	1.8648
2.	83	145.13	0.047981	1.7220
3.	100	172	0.0558	1.4535
4.	110	187.82	0.0626	1.3311
5.	130	219.45	0.076273	1.1392
6.	140	235.26	0.0830	1.0626
7.	145 *	243.17	0.086	1.0281

* = Maximum pressure that can be sustained at this wall thickness.

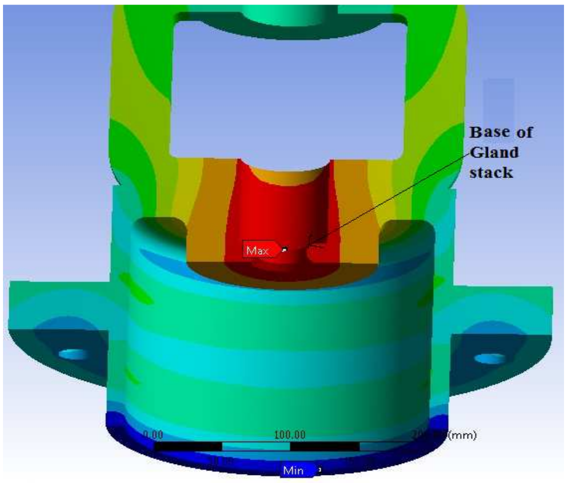


Fig.6 Max. total deformation- 76 bar g pressure

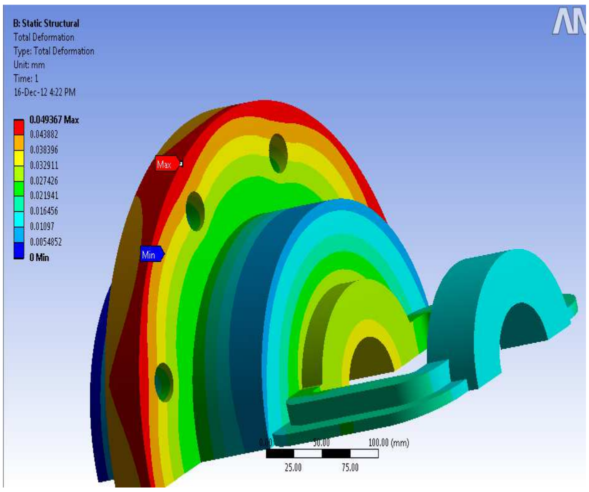


Fig.7 Max. total deformation – 145 bar g pressure

4

. CONCLUSION:

From the above results, it is concluded that the valve bonnet is safe at maximum designed pressure 76 bar g as the maximum Von misses stresses are 136 MPa (Factor of safety-1.86) and the component can sustained maximum pressure of 145 bar g (Factor of safety-1.02). The maximum total deformation occurs at the flange is 0.049367mm which is negligible and does not create any stress on other components as it is at free location. In further analysis, the analysis will be done by reducing the minimum thickness of the component.

5. ACKNOWLEDGEMENT:

I wish to thank Spirax Marshall Pvt. Ltd., Pune for giving me an opportunity to work in this field. The guidance, cooperation, practical approach & inspiration given by company especially by Mr.M.M.Pingale (*R & D Manager)* provided me the much needed impetus to hard work.

6. REFERENCES

1. Machinary Handbook, ANSI B.18.22M

2. Spaenaur, Technical Data, pdf d48

3. ASME SEC.II Part D,2004 Edition ,pp.301

4. Nitin S. Gokhale, Sanjay S. Despande, Dr. Anand N. Thite , *Practical Finite Element Analysis*, 1st Edition, Finite to Infinite, Pune

5. Ansys, Element Reference, Release12.0, April 2009,pp 1357.