International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 6, June - 2014

Structural Analysis of Gate Valve Body Using F.E.A.

Mr. Pradip Bhaskar Patil
Assistant professor, Mechanical engineering department, A.D.C.E.T., Ashta

Prof. V. R. Gambhire
Professor, Mechanical engineering department, T.K.I.E.T., Waranangar

***Abstract*—Gate valves are machine elements which are commonly used to control the flow of volatile, often toxic, liquids and gases and keep them from being emitted into the atmosphere or spilled on the ground or into the water. This gate valve is widely used in various industries like refineries, petro-chemical complexes, fertilizer plants, power generation plants (hydro - electric, thermal and nuclear) steel plants and allied industries etc. for a various process. This paper gives basic methodology of gate valve body design by using cad technologies and FEA at maximum operating pressure. The main purpose is structural analysis to be carried out for determining stresses and strains developed in the valve body**.

Keywords—gate valve body; structural analysis; FEA;

I. INTRODUCTION

The first design of a product that has been in the head of the engineers is illustrated in computers and then produced by computer-aided machines. The analysis processes of a product of which solid modeling have been accomplished can be carried out by ANSYS and similar analysis programs in digital environment, before starting manufacturing. These processes give the companies flexibility in the development of a new product by providing design capability with minimum cost and minimum waste of time. ANSYS program, which has been used in this study, can make the analysis of the substances in many subjects including static, dynamic, thermal, harmonic and electromagnetic.

A lot of research has been done on gate valve other than its valve body is described here in brief. Kim, Park, Kang, Kim [1] have analyzed rate of loading phenomenon generated during the operation of a motor operated gate valve under fluid pressure conditions for performance evaluation. Dynamic & static test were performed to analyze the ROL effect for flexible wedge gate valve. Duda, Dawid Rzasa [2] has applied new numerical method to determine optimum operating parameters i.e. heating & cooling operations of steam gate valve during start up and shut down phase based on thermal stress analysis. Punitharani,Murugan,Shivagami [3] has studied the effect of establishment of temperature distribution, distortion and residual stress field developed during plasma arc hard facing over low alloy steel gate valve by FEA using ANSYS parametric design language code. Zaimović-Uzunović, Ajan et al [4] analyzed butterfly–valve body at working loads, max. pressure & temperature using CAD technologies and FEM. Analysis of CAD model of stress strain state was performed by defining the input elements like loadings, boundary conditions and material properties which show the information to make possible corrections. Peng Guangjie,Wang Zhengwei, et al [5] analyzed centrifugal dredge pump case for stress concentrations using FEA. ANSYS was used in 3D FE analysis of several pump case designs to calculate the distortions and stresses for the service load.
From this reviews of research papers it is seen that there is a scope for structural analysis of gate valve body by using methodologies like finite element analysis.

Gate valve & its basic parts:

A gate valve or sluice valve, as it is sometimes known, is a valve that opens by lifting a round or rectangular gate/wedge out of the path of the fluid. A gate valve is used to control the flow of volatile, often toxic, liquids and gases and keep them from being emitted into the atmosphere or spilled on the ground or into the water. A gate valves are sometimes used for regulating flow, but many are not suited for that purpose, having been designed to be fully opened or closed.

A gate valve controls system or process fluid flow and pressure by performing any functions such as stopping and starting fluid flow, varying (throttling) the amount of fluid flow, controlling the direction of fluid flow, regulating downstream system or process pressure. The basic parts of expanding gate valve assembly with the constructional detail shown in fig.1 are explained as below*:*

Valve body: An expanding gate valve is a Bi-directional valve i.e. they can be used for flow of fluid from either directions. However, it is preferable that it should be installed such that flow is directed towards the minor segment. The valves can be made of carbon and alloyed steel using gravity casting method.

Major & minor gate segment (Expanding gate): An expanding gate valve with steel bodies and bonnets comprises of two gate segments, referred to as major gate and minor gate segment, are confined by the seats and gate guides such that they can move in vertical direction only. The expanding gate design provides a tight mechanical seal which is normally unaffected by pressure variations.

Spring with pin: The major (driving) gate segment is connected by thread to the operating stem. The minor (driven) gate segment nests to the major segment on a bi-angular surface and they are held together by leaf springs on each side.

Seat assembly: Gate valve is equipped with removable seats which are fitted into the valve body. The seats along with teflon coating are press fit into the body to affect a metal-to-metal body-seat seal.

Stem: Gate valves are characterized as having either a rising or a non rising stem. Rising stems provide a visual indication of valve position because the stem is attached to the gate such that the gate and stem rise and lower together as the valve is operated.

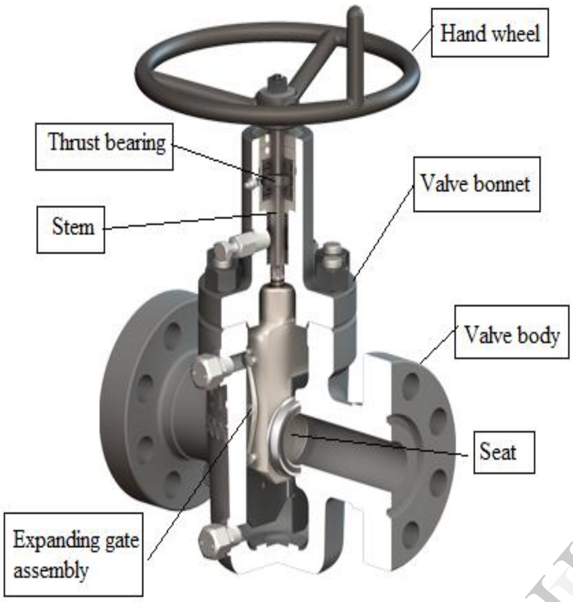

Fig.1: Sectional view of Gate Valve

Bonnet with bearing box: Bonnets provide leak proof closure for the valve body. Gate valves may have a screw-in, union, or bolted bonnet. Screw-in bonnet is the simplest, offering a durable, pressure-tight seal. Union bonnet is suitable for applications requiring frequent inspection and cleaning. It also gives the body added strength. Bolted bonnet is used for larger valves & higher pressure applications.

Hand wheel: The hand wheel operated gate valve is closed by turning the hand wheel in the clockwise direction and opened by turning the hand wheel in the counter clockwise direction.

II. DESIGN METHODOLOGY

The structural modeling of the gate valve body needs to be developed by using computer-aided design (CAD) software CATIA. Descritisation of valve body is to be done by importing CAD model into Hyper Mesh 10 software and begin the meshing on the valve body. Meshed valve body imported for structural analysis into computer aided engineering software. The solver tool is used to solve the analysis equation from ANSYS. The finite element analysis (FEA) then carried out on the valve body.

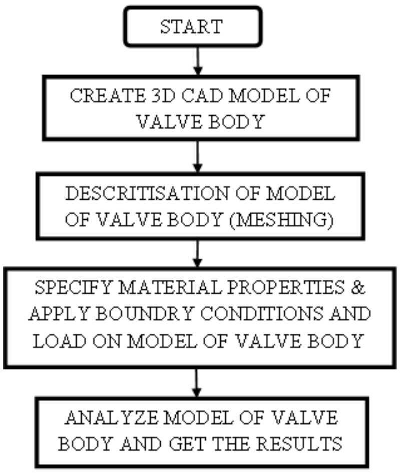

Fig.2: Flow chart of design methodology

III. FINITE ELEMENT ANALYSIS

This paper comprises the development of structural modeling with CAD software, finite element modeling using hyper mesh software and analysis by using solver tool such as ANSYS software. The finite element analysis (FEA) then carried out on the valve body.

A.3D Modeling of valve body:

In ANSYS it's very difficult to model the part with parametric modeling as compared with the available modeling software. The structural modeling of the gate valve body is developed by using computer-aided design (CAD) software CATIA V5. The created 3D model of gate valve body is as shown in fig.3.

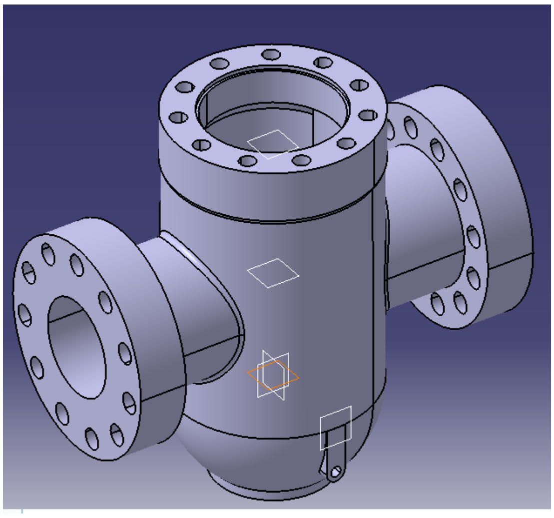

Fig.3: CAD model of valve body

B. Meshing of valve body:

In this step it is required to divide the total volume into small simple regular volumes, so that it can be easily meshed. Mesh generation is the process of dividing the analysis continuum into a number of discrete part or finite elements. Descritisation of valve body is to be done by importing CAD model into hyper mesh 10 software and begin with the meshing on the valve body. For geometry interfacing and cleaning up hyper mesh is used as robust tool to clean (mend) imported geometry containing surfaces with gaps, overlaps and misalignments which prevent auto meshing and high quality mesh generation.

Basic finite element shape- Once the geometry was cleaned, the design space volume is filled with finite elements using the auto-mesh features of hyper mesh. We can idealize the valve body by using the three dimensional as well as higher order elements. For the discretization of problems involving curved geometries, finite elements with curved sides are useful. Finite elements with curved sides are called higher order elements. SOLID45 has a quadratic displacement behavior and is well suited to model irregular meshes. Hence, model is meshed by using SOLID 45 element as shown in fig.4.

Size of finite element- If the size of the elements is small, the final solution is expected to be more accurate. But computational time is required more for analysis of valve body. For that purpose analysis is done by varying element size like 7, 8, 9 and 10 so that they will give results closer. For analysis of gate valve body to get better result and lesser computation time, finite element size is taken as 10.

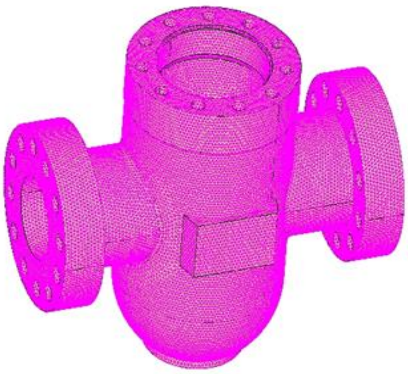

Fig.4: Meshed model of valve body

C. Material properties, boundary conditions and load:

Material properties- Stresses and strains of the valve body are essentially depend of the properties of the material of valve has been made. The material specification for casting of valve body is used as ALLOY STEEL, API 6A-75K with reference of ASTM A487 CLASS 4D modified. Modulus of elasticity E = 2.03×10^4 N/mm^2, Poisson ratio μ = 0.29, mass density 0.278 lb/inch^3 and yield strength 560 N/mm^2.

Boundary conditions- Pressure is uniformly distributed all over the body by flowing fluid. The two end flow line connection or flange and bonnet side will be restricted to prevent displacements and pouring of the fluid. All degree of freedoms of bottom of valve body is restricted and it is shown with the help of blue color in the fig.5. The model is fixed by the restraints to prevent all possible motions to the system of reference.

Load: Structural loading means applying internal hydraulic pressure to valve body. Internal pressures 34.5 MPa as operating pressure and 56.5 MPa as test pressure are applied on the all inner surface of a valve body, which is shown by red colored arrows in the fig.5.

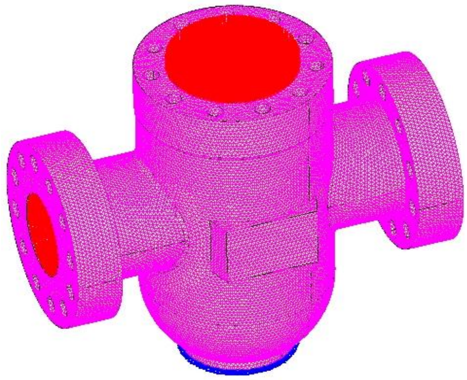

Fig.5: Valve body with Boundary conditions & loading constraints

D. Results and their physical interpretation:

The finite element method (FEM) solves for the nodal displacement and from this the element strain and stresses are calculated. It gives the results that are reviewed for the analysis done, by obtaining graphic displays such as stress plots.

Results of FEA for 34.5 MPa internal pressure condition are shown in following figures:

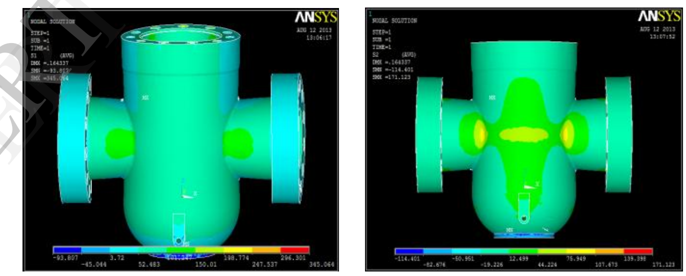

Fig.6: 1st & 2nd principal stress plot

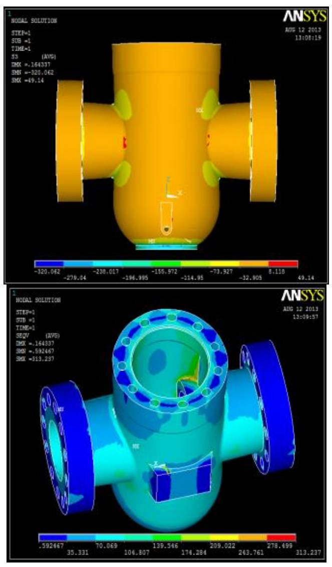

Fig.7: 3[rd] principal stress and von mises stress plot

From above obtained results it is clear that the maximum principal stress (1^{st} principal stress) 345.06 N/mm^2 founds in the inner wall of end line flow connection at 34.5 MPa internal pressures. While minimum principal stress (3^{rd} principal stress) 49.14 N/mm^2 found at outer surface i.e. connection between bowl and flow connector for 34.5 MPa internal pressures. And von mises stress plot gives maximum stress 313.24 N/mm^2.

Results of FEA for 56.5 MPa internal pressure condition are shown in following figures:

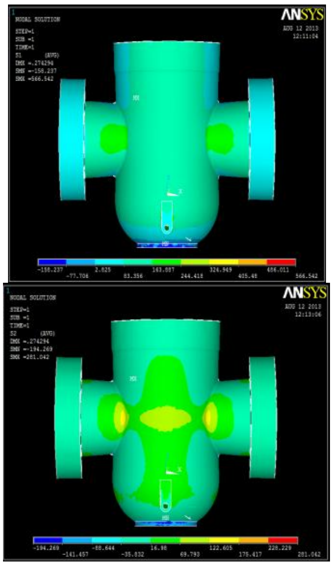

Fig.8: 1^{st} & 2^{nd} principal stress plot

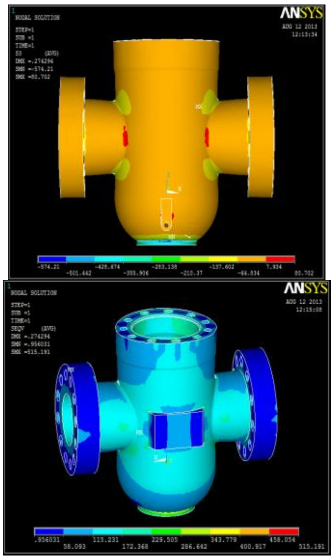

Fig.9: Von mises stress plot

From above obtained results it is clear that the maximum principal stress (1^{st} principal stress) 566.54 N/mm^2 founds in the inner wall of end line flow connection at 56.5 MPa internal pressures. While minimum principal stress (3^{rd} principal stress) 80.70 N/mm^2 found at outer surface i.e. connection between bowl and flow connector for 56.5 MPa internal pressures. And von mises stress plot gives maximum stress 515.19 N/mm^2.

IV. CONCLUSION

From FE analysis, it is clear that the maximum principal stress (1st principal stress) founds in the inner wall of end line flow connection at operating as well as testing pressure. While minimum principal stress (3rd principal stress) found at outer surface i.e. connection between bowl and flow connector at operating as well as testing pressure. As the internal pressure acts on the internal effective pressurizing area of valve body, results to expand the valve body. As the internal pressure increases stresses in the valve body increases linearly. And von mises stress plot gives maximum stress values 313.24 N/mm^2 and 515.19 N/mm^2 are found within yield strength limit.

V. REFERENCES

[1] Dae-woong kim,Sun-Geun Park,Sin-Cheul Kang,Yang-Suk Kim "A study on the phenomenon of rate of loading in motor operated gate valves" Nuclear Engineering & Design 240 (2010) pp 957–962

[2] Piotr Duda,Dawid Rzasa "Numerical method for determining allowable medium temperature during operation of steam gate valve" Mechanics and control,Vol 29 No.3 (2010) pp 102-109.

[3] K punitharani, N murugan, S M shivagami "FEA of residual stresses & distorsion in hard faced gate valve" Journal of scientific & industrial research vol. 69 (2010) pp 129-134.

[4] Nermina Zaimović-Uzunović, Ejub Ajan, Samir Lemeš "Weight optimization of the butterfly valve housing", 10th International Research/Expert Conference "Trends in the Development of Machinery and Associated Technology", Barcelona-Lloret de Mar, Spain, 11-15 September, 2006

[5] Peng Guangjie , Wang Zhengwei , Yan Zongguo, Liu Ruixiang "Strength analysis of a large centrifugal dredge pump case"Engineering Failure Analysis 16 (2009) pp 321–328.

[6] Mona Golbabaei Asl, Rouhollah Torabi, S.Ahmad Nourbakhsh "Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing" , Engineering Failure Analysis 16 (2009) pp 1996-2003.

[7] Pravin Narale, Prof.P.S.Kahare "Structural analysis of Nozzle attachment on pressure vessel design",International journal of engineering research and applications,Vol.02, Issue 4, july-August (2012) pp 1353-1358.

[8] Deokar Vinayak, D.S.Chavan, "Optimization of 16"Plug Valve Body Using FEA And Experimental Stress Analysis Method", International journal of Mechanical engineering, Vol-1, Issue 1,(2011) pp 79-83.