Finite Element Analysis and Design Validation of High Pressure Gate Valve for a Typical Nigerian Oil Wellhead

ThankGod E. Boye
Department Mechanical Engineering
Federal University of Petroleum Resources,
Effurun, Nigeria.
Boye.thankgod@fupre.edu.ng

Thaddeus C. Nwaoha
Department of Marine Engineering
Federal University of Petroleum Resource,
Effurun, Nigeria
nwaoha.thaddeus@fupre.edu.ng

Adeyemi I. Olabisi
Department Mechanical Engineering
Federal University of Petroleum Resources,
Effurun, Nigeria.
Adeyemi.olabisi@fupre.edu.ng

Corresponding author: ThankGod E. Boye,
Email: boye.thankgod@fupre.edu.ng

***Abstract*— All over the world, there is an urgent need for effective fluid control equipment, such needs can be found in foods, drugs, beverages, power generation, and oil and gas industries. Some of the existing equipment for this purpose has certain limitations for effective control of fluid at high pressure condition. Thus, there is a need for design modification to solve the prevailing problem. This paper majorly addressed the case peculiar to the oil and gas industries, where fluid control failure has cost millions of dollars, hazardous environmental effect and also decline in nation's economy, taking Niger Delta oil spillage as a case study. In this work, computer aided design method (CADM) was utilized to design and perform stress analysis of a high pressure gate valve used on a typical oil and gas wellhead of working pressure up to 15,000 psi. (103.4 MPa). The critical components in the gate valve are the body and gate (disc). Conversional design adopts calculations from analytical method, but in this work, finite element analysis (FEA) was used to determine stresses and deformations distribution on critical components of the high pressure gate valve (HPGV) as they are the main pressure retaining boundaries in the valve. The results obtained from the two methods indicated reasonable convergence after validation. The stress analysis conducted was based on von mises failure criterion which is most suitable for ductile materials. The results obtained from both analytical design calculation and that of finite element analysis indicates reasonable agreements by measuring their percentage performance variance. Therefore, the findings of this research work attests to reliability of the designed HPGV valves in meeting engineering design assessments, and in turn can be optimally advanced in the manufacture of high pressure gate valves.**

***Keywords*— Gate Valve, FEA, Analytical method, Validation, Design and High pressure**

1. INTRODUCTION

Valves are static mechanical equipment that controls the flow and pressure within a system or process. They are essential components of a piping system that conveys liquids, gases, vapors, slurries etc. [1]. They are vital mechanical devices/components used in running day to day oil and gas operations in Nigeria and other oil producing countries. Gate valves like other types of valve such as globe valve, plug valve, ball valve, safety valves, check valve, butterfly valve etc., have been proven effective and efficient in the control of fluid flow operations. Hence, to boost effective production activities in the oil and gas sector of Nigeria, an indigenous design and critical analysis of gate valves is imperative. Thus, failures in valves result in loss of petroleum products and this has direct as well as indirect effect on cost of oil and gas production. Raji and Abejide, 2013 [2] observed that "*one of the major causes of oil spillage in Niger Delta region is as a result of valves failures"* therefore an indigenous design of a Gate valve would be taken into considerations.

Heavy duty gate valves are normally used in oil wellhead configuration and installation. Because of its capability to control and withstand crude oil high flow rate and high pressure that is constantly experienced on the oil wellhead. Typical wellhead operating parameters includes: high pressure, flow rate, specific gravity, etc.

This research work utilized computer aided design method to design and carryout critical stress analysis of a gate valve used on a typical oil and gas wellhead of working pressure up to 15,000 psi (103.4 MPa or 1034.21 bars).

1.1 Gate Valve

A Gate valve can be described as a type of valve that opens by raising a round or rectangular gate/wedge out of the path of the fluid and is operated by means of a threaded stem which connects the actuator (hand-wheel or motor) to the gate [1]. These valves are used for regulating flow, but many are not suited for this purpose, having been designed to be fully opened or closed [1]. Thus, when fully open, the typical gate valve has no obstruction in the flow path, resulting in a very low frictional loss [1] but the opposite become the case when it is fully closed. Hence, high frictional losses occur on the gate (the closing element of the valve) as a result of many obstructions in the fluid flow path. It is important to avoid or minimize the frictional losses at the design stage of the gate valve; therefore, the critical stress analysis of key elements of the gate valve is essentially identified in advance, before manufacturing of the gate valve. Hence, this study is bridging the gap between indigenous technology/engineering and acceptable international design methodology.

Although, it is a serious task for designers to design and estimate accurate stress distributions of any mechanical component; some limitations are encountered in the design. However, to overcome these limitations, computer aided design (CAD) methodology is an alternative. This method enables optimization of the design at the design stage, provides significant results prediction of stress distribution, and minimizes design time and cost, as well as providing efficient product [3].

Valves are designed with global codes and standards, for example the American Petroleum Institute Standards (API), American society of mechanical engineers (ASME), British standard (BS), America society of testing and materials (ASTM) and American National Standards Institute (ANSI) Deutsches Institut für Normung (DIN) etc. Thus, in order to achieve cost effective design, less design time and to produce accurate design performance outcome, CAD tools such as finite element analysis (FEA) is utilized extensively to analyze stress behaviors of the critical valve elements such as the gate, stem, valve body and bonnet. It helps in identifying any possible failures that may occur during operational life of the valve. Figure 1.1 shows the schematic diagram of key elements that make up a gate valve.

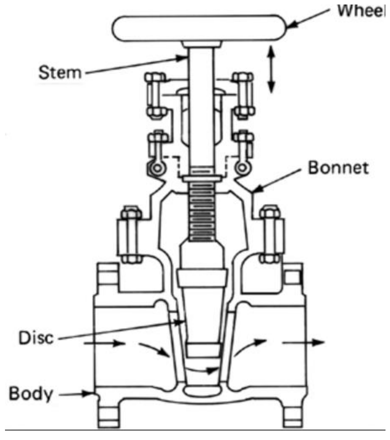


Fig. 1.1 schematic diagram of a gate valve

For effective and efficient fluid (either liquid or gas) flow control in the oil and gas sector, and to reduce oil spillage through components failure, it is therefore, needful to invent a local design of a specific gate valve that can withstand an extremely high pressure flow usually experienced on an offshore oil wellhead. Proficient use of CAD tools will help increase the accuracy of the proposed locally design gate valve in geometry and analysis as well as design time reduction. Though, different size and pressure range of gate valves are found in the market. But constraints such as cost of importation to the local end users, failures in existing design due to old methodologies in design processes such as assumptions made in analytical calculations, manual drafting usually leads to design errors. For that reason, this research work aimed to address aforementioned constraints by proposing an effective, efficient and low cost design for indigenous manufacturers and end users. Upon completion of this design and manufacture accordingly, it will lead to efficient operation in oil and gas industries and will significantly contribute to solving oil spillage problems that are associated with gate valve components failure and reduces cost of operation in Nigeria.

2. Material and method

The materials used in this study consist of two components i.e. hardware and software components. The hardware component involves a high speed performance computer while the software component involves a solid-work CAD tool. Analytical method was used to perform hand calculations for the design in term of sizes, dimensions and critical stresses that would act on the gate valve body, stem, gate (disc), bonnet and flanges. Computer simulation was also conducted to check for convergence of results between the hand design calculations and FEA simulation for the purpose of validating the gate valve optimal design credibility. Figure 1.2 showcased the design procedures employed for this project.

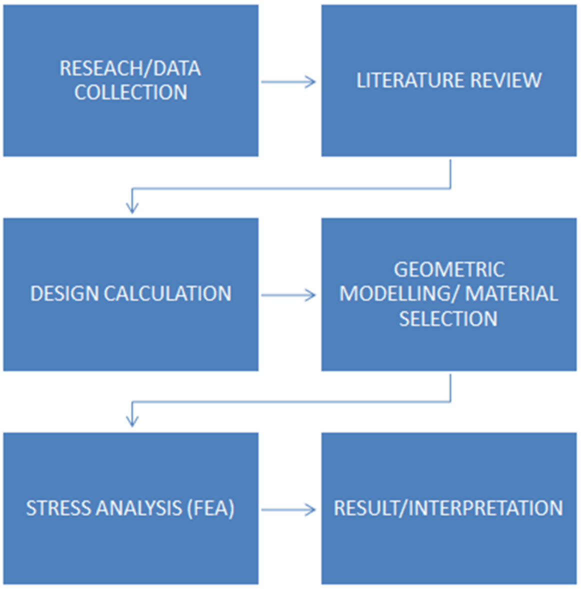


Fig. 1.2 Design flowcharts

2.1 Principle of Operation

Valves principle of operation relies on the driving mechanism or fluids to make the open and closing parts down, sliding, rotating or turning in order to change the size of flow area to work. The basic parameters of the valve are working pressure (PN), temperature (TN) and diameter (DN). Valves designed for industrial pipeline applications, nominal pressure and nominal diameter are considered as the basic parameters. PN means the maximum pressure of certain material valve working in the required temperature. DN refers to the body and the nominal pipe diameter of the end of connection as reported from previous studies [4].

2.2 Parts Design Consideration

In a typical gate valve, the critical parts that experiences pressure of the fluid directly are the body, gate, stem and bonnet. These parts are described briefly with their design principles.

2.2.1 Stem

Gate valves are classified as either rising stem or non-rising stem valves. For the non-rising stem gate valve, the stem is threaded on the lower end into the gate. As the hand wheel on the stem is rotated, the gate travels up or down the stem on the threads while the stem remains vertically stationary. This type of valve will almost always have a pointer-type indicator threaded onto the upper end of the stem to indicate valve position. Figures 2.2 and 2.3 illustrate rising-stem gate valves and non-rising stem gate valves respectively. The non-rising stem configuration places the stem threads within the boundary established by the valve packing out of contact with the environment. This configuration assures that the stem merely rotates in the packing without much danger of carrying dirt into the packing from outside to inside [4].

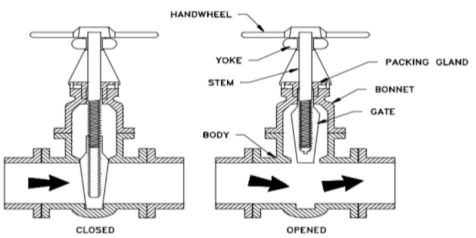


Figure 2.1 Non-rising stems

Rising stem gate valves are designed so that the stem is raised out of the flow path when the valve is open. Rising stem gate valves come in two basic designs. Some have a stem that rises through the hand wheel while others have a stem that is threaded to the bonnet [4].

2.2.2 Body

The body, sometimes called the shell, is the primary pressure boundary of a valve. It serves as the principal element of a valve assembly because it is the framework that holds everything together. The body is the first pressure boundary of a valve, which resists fluid pressure loads from the piping system. It receives inlet and outlet pipes through threaded, bolted, or welded joints. Valve bodies are cast or forged into a variety of shapes [5].

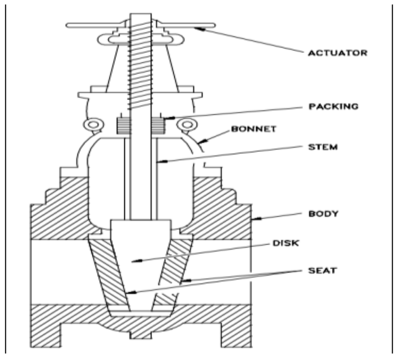


Figure 2.2 Valve body

Narrowing of the fluid passage (Venturi effect) is also a common method for reducing the overall size and cost of a valve. In other instances, large ends are added to the valve for connection into a larger line [5]. When a cylindrical shell of a pressure vessel, hydraulic cylinder, valve and pipe is subjected to a very high internal fluid pressure P_i, then the walls of the cylinder must be made extremely heavy or thick [5]. In the design of thick cylindrical shell, lame's equation is mostly considered; especially when ductile material with close or open ends is to be considered in accordance with the maximum normal stress theory of failures, the stresses σt is given as:

$$\sigma_t(max) \frac{p[(r_o)^2+(r_i)^2]}{(r_o)^2-(r_i)^2} \quad (2.1)$$

The above equation shows the maximum principal stress at the inner surface. The minimum principal stress at the outer surface is given as;

$$\frac{2p(r_o)^2}{2[((r_o)^2-(r_i)^2} \quad (2.2)$$

Maximum shear stress τ is given as;

$$\tau_{\max} = \frac{\sigma_{\text{tmax}}-\sigma_{\text{tmin}}}{2} \quad (2.3)$$

According to Lame's equation, the thickness of the pressure retaining vessel such as valve, is given as

$$t = r_i\left[\sqrt{\frac{\sigma_t+p}{\sigma_t-p}} - 1\right] \quad (2.4)$$

But when taking into consideration theories of failures, maximum energy strain failure criteria, the safe thickness is calculation from the expression;

$$\boldsymbol{\sigma^2} \geq \frac{2\sigma_r^2\left[K^4\left(1+\frac{1}{m}\right)+\left(1-\frac{1}{m}\right)\right]}{\left(K^2-1\right)^2} \quad (2.5)$$

Where K = $\frac{r_2}{r_1}$ from equation above, we can see that if the $P_i = \sigma t$ or $P_i > \sigma t$, then no thickness of the shell will prevent failure.

2.2.3 Bonnet

Since valve is a high pressure retaining vessel, the bonnet which serves as the end flange of the valve sometimes experience high pressure liquid when the gate is open or close in some cases. The bonnet of the valve is commonly bolted to the body. In the design of bonnet, the required thickness is gotten from Grasch of and Bach equation [5] given below:

$$t_1 = K\sqrt{\frac{P}{\sigma t}} \qquad (2.6)$$

Graschof and Bach σt = allowable design stress, K_1 depends on the material and the holding methods, for cast iron, steel as shown in the table below

Table 2.1 Bach constant

Material of the cover plate	*Type of connection*	*Circular plate* k_1
Cast iron	Freely supported	0.54
	Fixed	0.44
Mild Steel	Freely supported	0.42
	Fixed	0.35

2.2.4 Flange End

It is one of the mostly found part at valve ends, it aids connections to the engaging pipe. A flanged joint may be made with flanges cast integral with the pipes or loose flanges welded or screwed. The figure below shows two cast iron pipes with integral flanges at their ends. The flanges are connected by means of bolts. The design calculations are adopted from a recommended manual [7].

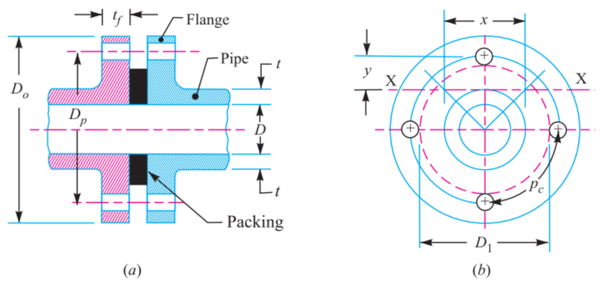


Fig. 2.3 Flange dimensions [7]

The following are the Dimensional consideration of the flange design:

Nominal diameter of bolts, d = 0.75 t + 10 mm

Number of bolts, n = 0.0275 D + 1.6

Thickness of flange, t_f = 1.5 t + 3 mm

Width of flange, B = 2.3 d

Outside diameter of flange, Do= D + 2t + 2B

Pitch circle diameter of bolts, Dp = D + 2t + 2d + 12 mm

2.3 Valve Materials consideration

Valves are constructed from a number of different materials based on the availability and cost of the material, ease of fabrication, resistance to corrosion and pressure, design metal temperature (DMT), and compatibility with the fluid controlled [6] (Luis and Julio, 2002). Materials used in the construction of valves shall conform to the specifications listed in API 600. According to the different types and requirements, the main performances of valves are sealing, strength [5-6]. When designing and selecting a valve, engineers must consider the basic parameters and the performances, the performance of the fluid including the fluid phase state (gas, liquid or contains solid particle), corrosiveness, viscosity, toxicity, flammable explosion hazard and radioactivity [4-6]

2.3.1 Cast Carbon Steel (ASTM A216 Grade WCC)

ASTM A216 Grade WCC is the most popular steel material used for valve bodies in moderate services such as air, saturated or superheated steam, non-corrosive liquids and gases. WCC is not used above 800F (427C) as the carbon rich phase might be converted to graphite. It can be welded without heat treatment unless nominal thickness exceeds 1-1/4 inches (32 mm).

2.3.2 Cast Chromium-Molybdenum Steel (ASTM A217 Grade WC9)

This is the standard Cr-Mo grade. WC9 has replaced C5 as the standard because of superior casting and welding properties. WC9 has successfully replaced C5 in most applications, especially in steam and boiler feed water service. The chromium and molybdenum provide erosion-corrosion and creep resistance, making it useful to 1100F (593C). WC9 requires preheating before welding and heat treatment after welding.

2.3.3 Standards and Codes Used For Petroleum Valve Design

Standards and codes have been developed to provide acceptable, practical, and useful design with ensured quality, safety, and reliability of equipment, practices, operations, or designs [6] (Luis and Julio, 2002). Major standards used for the design and constructions of valves are: American Petroleum Institute Standards (API), American society of mechanical engineers (ASME), British standard (BS), America society of testing and materials (ASTM) and American National Standards Institute (ANSI). These are the standards adopted for the design of the proposed gate valve in this study.

2.4 Design alternatives and required objectives

Design consideration is based on numbers of selected alternatives; optimal selection is drawn from the lists of alternatives with special consideration to the objective functions as outlined below:

- Availability of material
- Environmental consideration
- Conformation to codes and standards
- Strength of materials

Table 2.2 Body and Bonnet materials

Material	Alloy Steel
Density	7850 kg/m^3
Young modulus	200 GPa
Yield strength	6.20422e+008 N/m^2
Tensile strength	7.23826e+008 N/m^2

Table 2.3 Gate material

Material	Alloy steel 1.7014 (17CrS3)
Density	7850 kg/m^3
Young modulus	200 GPa
Yield strength	4.50590e+008 N/m^2
Tensile strength	5e+008 N/m^2

Table 2.4 Stem material

Material	Stainless steel
Density	7850 kg/m^3
Young modulus	200 GPa
Yield strength	4.6e+008 N/m^2
Tensile strength	5e+008 N/m^2

Table 2.5 Nuts And Bolts materials

Material	Carbon steel
Density	7850 kg/m^3
Young modulus	200 GPa
Yield strength	3.05e+008 N/m^2
Tensile strength	5.4e+008 N/m^2

2.4 Design Specifications

The table below describe the design specification as follows:

Table 2.6 Design specification

DESIGN PARAMETERS	***DATA***
Model	*HPGV*
Design pressure P_i	103.4 Mpa (15,000 psi)
Nominal diameter	190.5 mm (7.5 inch)
Net positive suction (NPS)	203.2 mm (8 inch)
Body thickness	40 mm
Working fluid	Crude oil
Flow rate	20,000 bpd
Stem diameter	25 mm
Flange thickness	63.5 mm
No of flange bolt	8
End-Flange diameter	400 mm
End-flange pitch circle diameter. of bolt	340 mm
Body material	Alloy steel
Gate material	Alloy steel 1.7014 (17CrS3)
Corrosion allowance	5 mm
Factor of safety	1.333

The above data were used analytically by hand calculations and numerically by finite element analysis to complete the design of the high pressure gate valve. The result s obtain from the finite element analysis was validated by the analytical method outcomes in order to justify the credibility of the HPGV design.

2.5 Analytical Method

The analytical method constitutes the traditional design hand calculations. Established formula was used to determine stress and deformation values for the design of the HPGV.

2.5.1 Stress / Deformation Analysis

The variation of tensile stress and radial stress in relationship of wall thickness of the pressure retaining boundary of valve's body using lame's equation shown in eq. 2.1, produces the results in Table 2.7, the wall thickness was subdivided into five equal parts.

$$\sigma_t = \frac{p(r_i^2)}{r_o^2 - r_i^2}\left[1 + \frac{r_o^2}{x^2}\right]$$

Where: σ_t= tensile stress, r_i = internal radius, r_o = external radius, p = p_i = internal pressure p_i = 103.44 Mpa (15000 psi)

Using the above equation, for wall thickness from 0 – 40mm, the tangential stress is tabulated below:

Table 2.7 Tangential stress outcome for various valve body thicknesses

Body Wall Thickness (mm)	Tangential stress σ_t (MPa)
0	304.3
10	267.7
20	241.4
30	220
40	203.3

The tensile stress acting on a material is maximum at the inner surface of the pressure retaining body, while radial stress is maximum at the outer radius. However, increasing the thickness will help to prevent bursting failure in HPGV.

2.3.2 Maximum distortion energy theory (Von Misses criteria)

Von misses criterion states that yielding (failure) will occur when the distortional strain energy reaches that value which causes yielding in a simple tension test. Like the maximum shear stress or Tresca failure theory, the maximum distortional energy failure theory addresses ductile, isotropic materials.

2.5.3 Stress analysis of HPGV body

Conferring to this theory, the failure or yielding occurs at a point in a member when the distortion strain energy (also called shear strain energy) per unit volume in a bi-axial stress system reaches the limiting distortion energy (*i.e*. distortion energy at yield point) per unit volume as determined from a simple tension test. Therefore, equation 2.7 below is used to test for yield strength of the Gate valve body i.e. Maximum distortion energy.

$$(\sigma_{t1})^2 + (\sigma_{t2})^2 - 2\sigma_{t1} \times \sigma_{t2} = \left(\frac{\sigma_{yt}}{fs}\right)^2 \qquad (2.7)$$

Where: σ_{yt} = yield strength, σ_{t1}, , σ_{t2} = principal stresses

$$301620.6 + 198737.6 - 489666.7 = 10691.5$$

$$\mathbf{10691.5 < 1716100}$$

From the above failure analysis in line with the von misses failure theory; the body under analysis will not fail when subjected to the design maximum pressure of 103.4Mpa.

2.6 Finite element analysis (FEA) Method

The Finite element analysis of the high pressure gate valve of 15,000 psi (103.4 Mpa) was performed in Simulationxpress domain. This analysis of the gate valve is done in order to study the area of stresses concentration as a result of the extreme high pressure that would be experienced by the valve body and in view minimizes failure during its operational life. The FEA simulation [8] was conducted to ensure optimal results. Hence, the stress concentrated on the valve has been evaluated and identified. These have provided better reference for optimization of the Gate valve. The result obtained is shown and discussed in results section.

Arguably, the most important factor to be considered in designing a high pressure Gate valve is to gain sufficient strength against internal pressure. A valve body cannot be designed without cautious attention given to structural strength. In this study, the structural strength was further analyzed with novel engineering tool such as FEA.

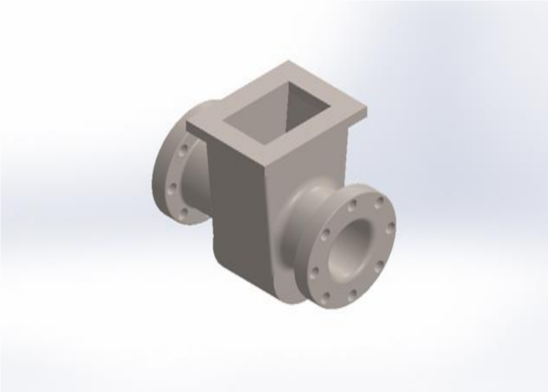
- Boundary condition for the FEA simulation was operating pressure of 15,000 psi (103.4 Mpa) and default mesh size were also adopted.

3. Model Development of the High Pressure Gate Valve

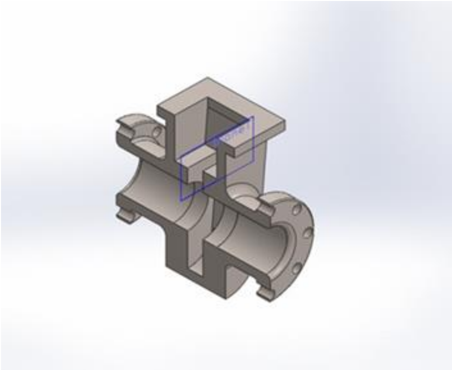
All the components and parts of the High pressure gate valve were carefully developed and modelled using Solidwork workspace. Thereafter, painstaking hand calculations were done to obtain their various dimensions.

3.1 Gate Valve Model in 3D

The following figures (a-d) below show the critical individual component that make up the high pressure gate valve assembly and that was analysed for possible failure. The combination wireframe, surface and solid tools were used to develop these components.

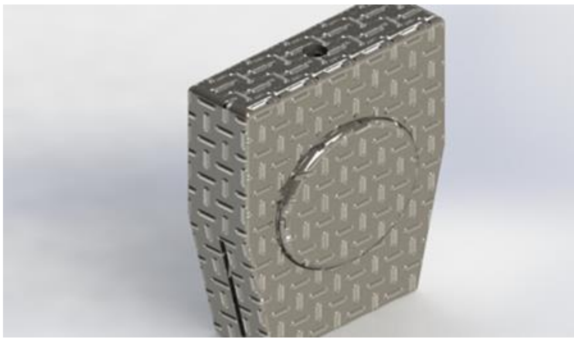


a



b

c



d

Fig. 3.1: (a) valve body, (b) cross-section of valve body, (c) Valve bonnet and (d) Valve Gate

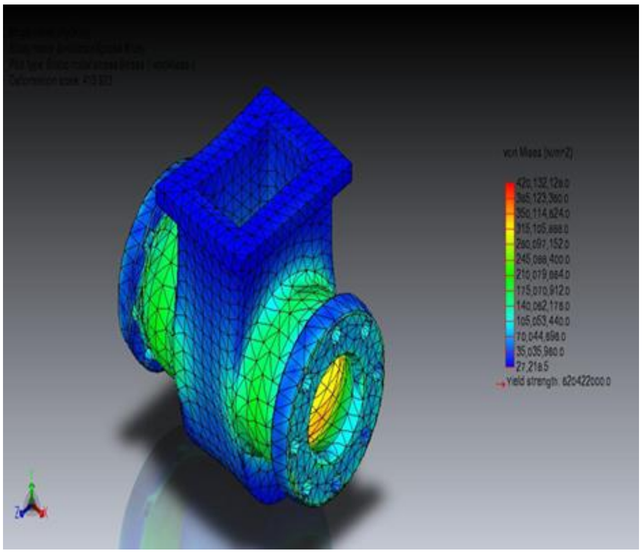
4. Results and Discussions

A gate valve that can safely and reliably control high pressure flow line in the oil and gas industries was design and analyzed.

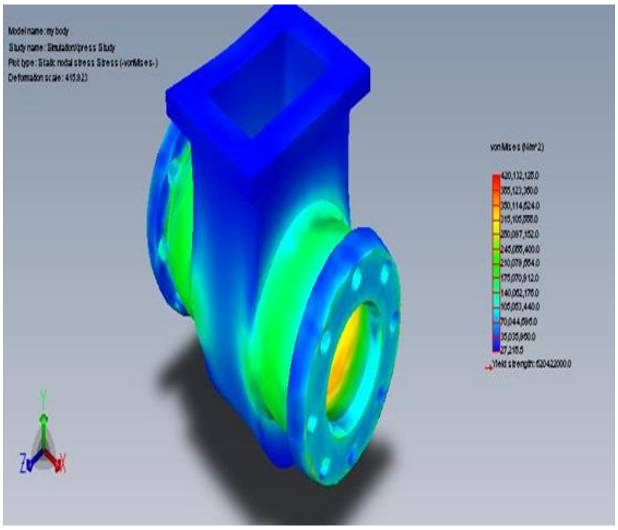
Preliminary decisions taken before the design of the valve includes type of valve, choice of design method, gate type, pressure class, material selection, strength, fluid consideration and corrosion allowance. Class 900 was selected due to the fact that it a preferred option for the design pressure of 15000 psi. The stress analysis method was considered for the design of the gate valve because it is more accurate, reliable, and minimized the cost physical and after-design testing. In the design of HPGV, few objectives were taken into consideration such; strength, availability of design materials to local manufacturers, corrosion resistance etc.

After parametric calculations and standard assumptions were made, the 3D model and engineering drawings were produced for purpose of clarity and analysis as shown in appendix, a pressure retaining body was modeled; the body is made from alloy carbon steel, which possesses high tensile strength. A design pressure of 15000psi, been the maximum pressure experienced in high pressure flow line such as wellhead etc.

4.1. Stress and deformation of the valve body (internal wall surface and seat)

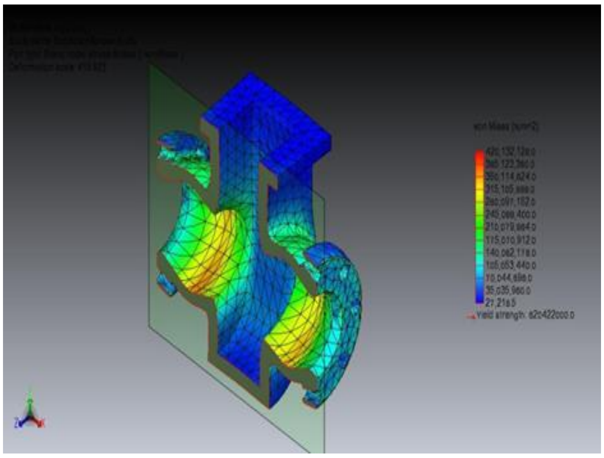


a

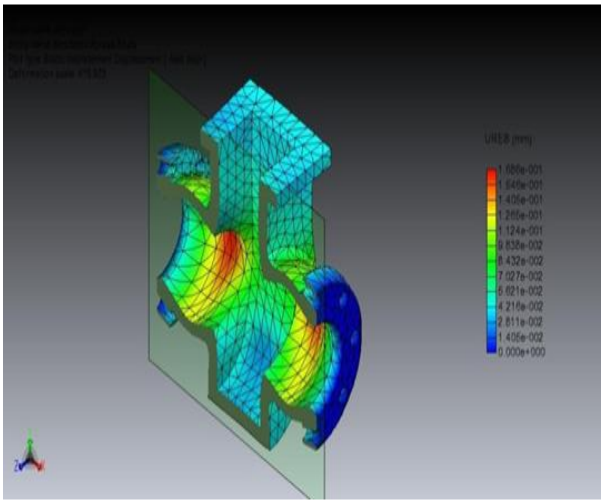


b

Fig. 4.1 (a) Valve body with default mesh and (b) Stress distribution of valve body



a



b

Fig. 4.2 (a) Cross-section of stress distribution (b) and cross-section deformation of vale body

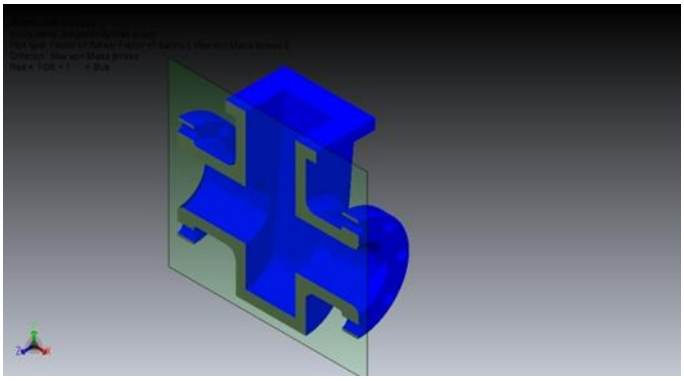
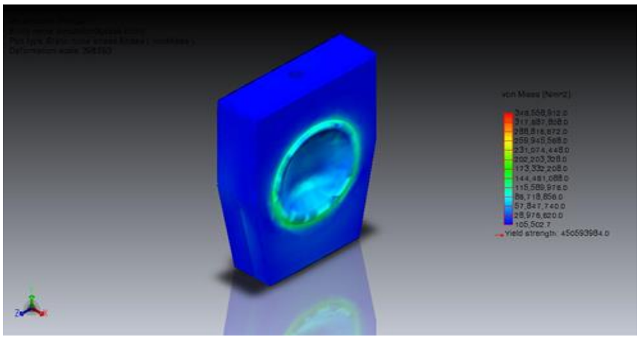


Fig. 4.3 Factor of safety on valve body

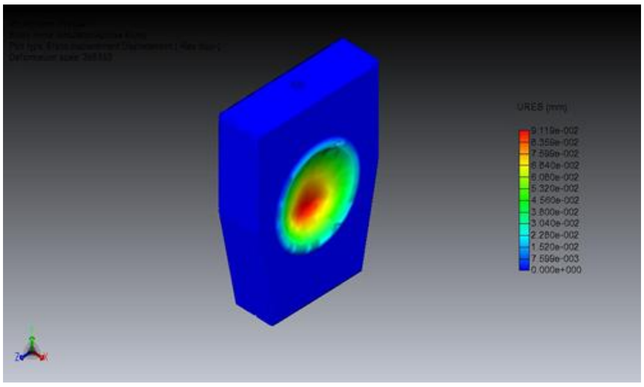
It is clearly shown from the results above in figure 4.1 a and b that maximum stress values experienced inside the valve body reached 420 MPa while the minimum stress recorded was 27MPa. However, the yield strength of the high pressure gate valve material used is 620MPa. Hence, it indicates that the maximum stress is less than the yield strength; therefore the designed valve body could be relied upon to withstand the operating pressure of 15,000 psi (103.4 Mpa).

Moreover, the deformation of valve body reached a maximum value of 0.1666 mm on area of the valve body with thickness of 40 mm; it is also observed that deformation is seen to be minimum on region with higher wall thickness (40 mm) and fillet (Optimized valve body). The minimum deformation recorded is 0.145 mm. additionally the factor of safety shown in figure 4.3 is optimum.

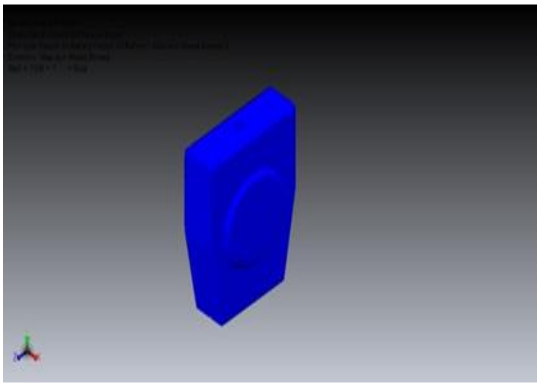
4.2 Stress and Deformation Valve Gate (Disc)



a



b



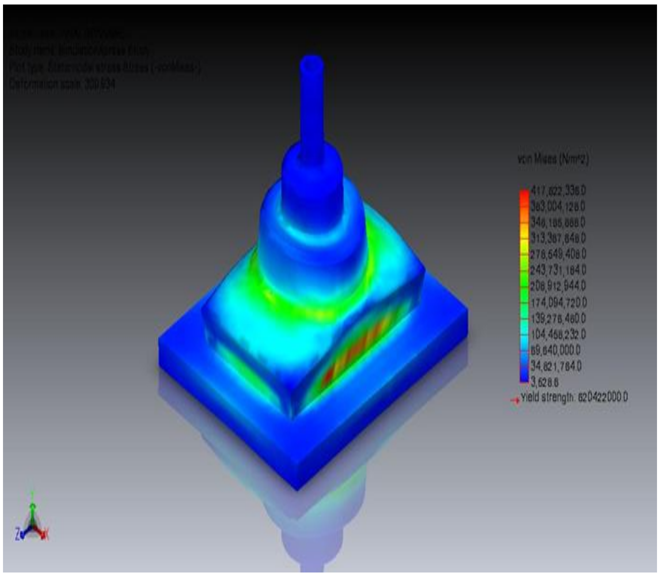
C

Fig. 4.4: Stress distribution on the gate (a), deformation diagram on gate (b) and Factor of safety (c)

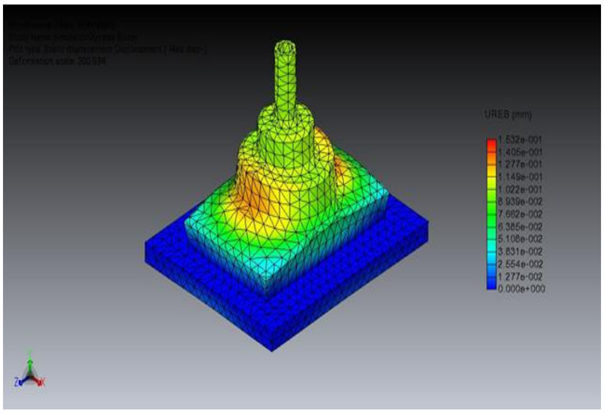
Information deduced from figure 4.4 a and b indicated that maximum stress values of 348 MPa is experienced when high pressure of 15000 psi (103.5 Mpa) acts on its surface inside the valve body and the minimum stress of 28 Mpa is recorded on the gate. The yield strength of the material used is 450MPa. Hence, it indicates that the induced stress is less than the yield strength of the gate material. Therefore, designed gate component is said to be safe from failure.

Moreover, the maximum deformation acts at the center where high pressure concentration is experienced and reached the value of 0.0911 mm, deformation reduces away from the center with value of 0.007599 mm. The factor of safety in the gate analysis is 1.453; this shows that the design is safe to withstand pressure of 103.5MPa.

4.3 Stress and deformation of the valve bonnet



a



b

Fig. 4.5 (a) Stress distribution diagrams of valve bonnet and (b) deformation diagram

Though the bonnet experience little or no fluid pressure, but an assumption was made on the ground that it is exposed to the design pressure of 103.5MP. The result obtained was favorable to accommodate the fluid pressure. The maximum stress experienced on the bonnet is 417 MPa, while the minimum on the pressure retaining area is 34MPa. Comparison of yield strengths of the bonnet material suggested that the bonnet is also safe from possible failure. However, it was noted that, the sharp corners is prone to failure; therefore filleting operation should be done to address excess deformation that may lead to failure. The maximum deformation on the bonnet is 0.1532mm and the minimum deformation is 0.01277mm.

The von-mises failure criterion is used throughout the analysis because it is an acceptable theory for obtaining safe wall thickness against failure. The factor of safety of 1.3333 was adopted in the design. When a design factor of safety is less than 1, the designed component is most likely to fail. In the course of the design, many engineering materials were tested, but the performance towards resistance to high pressure rendered some materials unfit for the design pressure. Alloy steel was found suitable in terms of strength, cost and availability.

4.6 Results Validation of FEA and Analytical Methods.

Upon the finite element analysis method employed in this paper, analytical method was also used to help validate the numerical method of the design. The stress and deformation results obtained from the FEA method analysis was compared with that of the analytical method. Table 4.1 below shows summary of the results of analytical and FEA methods as well as result variation and the percentage difference between the two methods. The purpose is to justify and give more confidence on the designed device (high pressure gate valve).

Table 4.1 Result Summary of FEA and Analytical methods

PART	STRESS			DEFORMATION	
	MAX. VALUE (MPa)		YIELD STRENGTH OF MATERIAL (MPa)	MAX. VALUE (mm)	
	FEA	ANALYTICAL		FEA	ANALYTICAL
GATE	348	307.7	450.59	0.091	0.089
BODY	402.5	304.3	620.42	0.162	0.162

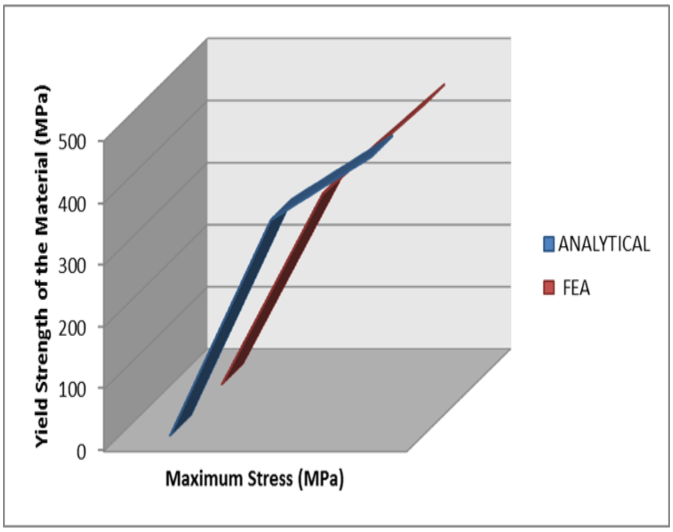


Fig. 4.6 Comparison of analytical Method against FEA method for Stress on the Valve Gate

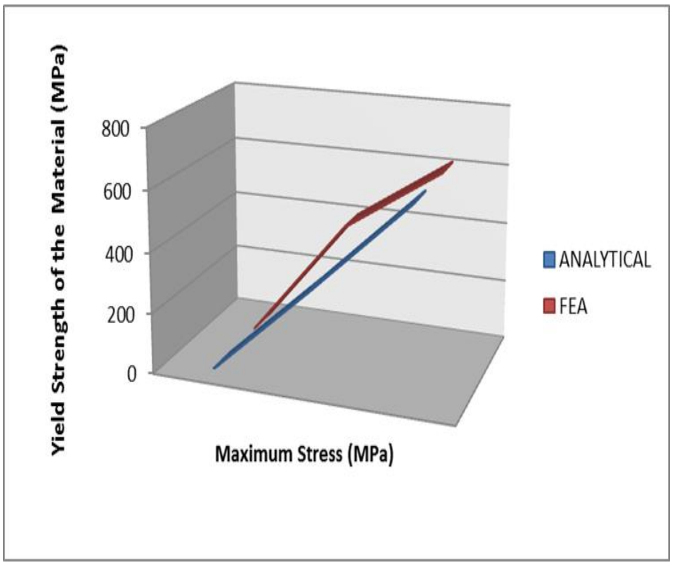


Fig. 4.7 Comparison of analytical Method against FEA method for Stress on the Valve Body

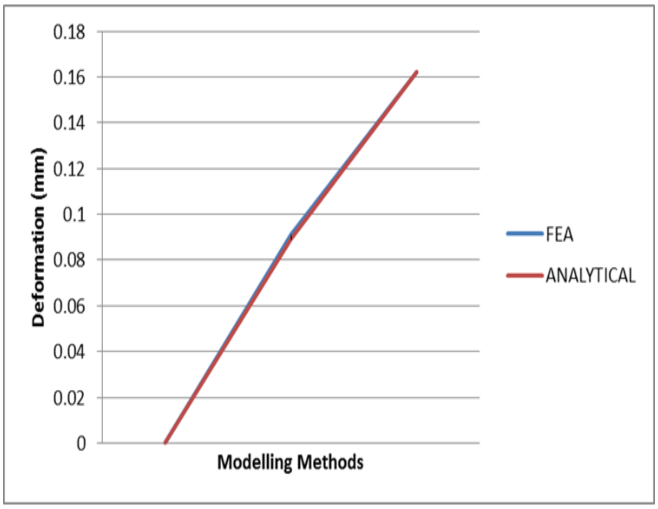


Fig. 4.8 Comparison of analytical Method against FEA method for Deformation on Valve Body and Gate

Table 4.2 Percentage difference in FEA and Analytical results for stress

PART	STRESS		Percentage Difference
	FEA (Mpa)	ANALYTICAL (Mpa)	
GATE	348	307.7	12.3%
BODY	402.5	304.3	27.8%

Table 4.3 Percentage difference in FEA and Analytical results for deformation

PART	DEFORMATION		Percentage Difference
	FEA (mm)	ANALYTICAL (mm)	
GATE	0.16	0.0137	2.2%
BODY	0.168	0.162	3.6%

4.7 Discussion of Validation

The use of digital method to carry out mechanical engineering design is time and cost saving, the analysis of the bonnet is given less concentration as it restricts little or no pressure in the gate valve equipment. Table 4.1 above shows the maximum stresses and deformations experienced on the gate (disc) and the body of the high pressure gate valve using both the analytical and FEA methods. Thus, the values stress and deformation of the valve gate and body on table 4.1 are plotted in figures 4.6, 4.7 and 4.8

respectively. Figure 4.6 and 4.7 shows reasonable agreement in result of the two modelling methods employed in this research paper.

Therefore, the agreements in results of the FEA and Analytical methods are established in table 4.2, which shows percentage difference for both methods. This could mean that both methods validated the credibility of the design of the high pressure gate valve since their percentage differences are within acceptable range. However, the percentage differences of both methods for deformation experienced on the gate (disc) and the body of the high pressure gate valve shown in table 4.3 indicates strong convergence.

The percentage differences of 2.2% and 3.6% for gate and body was recorded on both the FEA and Analytical methods respectively. Hence, the significant convergence deformation results shown in table 4.2(b), of both methods give sufficient integrity to the design and analysis of the designed high pressure gate valve used for a typical oil and gas wellhead of working pressure up to 15,000 psi. (103.4 MPa or 1034.21 bars).

5.0 Conclusion

The importance of the design, analysis and validation of a high pressure gate valve used for a typical oil and gas wellhead of working pressure up to 15,000 psi (103.4 MPa or 1034.21 bars) was justified. In this work, the analytical method and finite element analysis (FEA) was deployed to solve stress distribution on the valve body, and the gate (disc) as they are the main pressure retaining boundaries in the valve. The results obtained from the two methods indicate reasonable convergence from validation. Stress analysis conducted was based on von-mises failure criterion which is most suitable for ductile materials. It is proposed that the high pressure gate valve designed can be relied upon to take engineering design judgment especially in the areas of manufacturing of the gate valve and its design optimization.

REFERENCES

[1] Gurudutt Sahni and Balpreet Singh (2014). To Perform the Structural Analysis on Wedge Type Gate Valve and it's Optimization using F.E.A. *IRJMST*, p. 277-281.

[2] Raji, A. O. Y and Abejide T. S.(2013), An Assessment of Environmental Problems Associated with Oil Pollution and Gas flaring in the Niger Delta Region C 1960-2000 Arabian Journal of Business and Management Review Vol. 3 No 3, Oct 2013 p 115

[3] Pujari A.A and Joshi G.S (2015) Anyalysis And Design Optimization Of 8''- 600# Gate Valve Body Using Fea And Stress Anylasis IJRET: International Journal of Research in Engineering and Technology Volume: 04 Issue: 02 | Feb-2015

[4] Cheng Y. Cheng (2008) Basic Knowledge of Valves. 1st Edition. Publisher: Tongling Education Press, China.p.46-50

[5] Schweitzer, Philip A. (1972). Handbook of Valves, New York Industrial Press, p. 25-56

[6] Luis A. Godoy and Julio C. Mendez-Degro, Introduction to Above Ground Steel Tanks www.efn.unc.edu.ar/investigacion/e-learning/tanques/documentos/.../typestanks.pdf , 18th March 2016

[7] Khurmi R.S. and Gupta J.K. (2005) A textbook of machine design, first multicolour edition, Eurasia publishing house (PVT.) ltd. Ram Nagar, New Delhi-11005, p.261-280

[8] Sonali Mantati and Surde A. N. (2015) Finite Element Analysis of Flexible Wedge Type (2"#150) Gate Valve International Journal of Emerging Technology and Advanced Engineering Volume 5, Issue 6. P.195-198

APPENDIX