Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 04, April-2016

Design and Analysis of Hydro Test Rig for GGCB Forged Valve with 4 Stations

Rajat S. Shinde [1], Saurabh B. Gharge [2]
[1, 2] Department of Mechanical Engineering,
NBN Sinhgad School of Engineering, Ambegaon (BK),
Pune, India

Rajat B. Shinde [3], Akshay S. Shinde [4]
[3, 4] Department of Mechanical Engineering,
NBN Sinhgad School of Engineering, Ambegaon (BK),
Pune, India

***Abstract* - Hydro testing as the name suggests using water as the working fluid. It is used to detect leakages present in the valves such as Gate, Globe, Check and Ball valves (GGCB Valves). The water is used as testing as well as working fluid which gives the exact results. Since water is incompressible it can be pressurised at high pressures without significant change in its volume. Valve to be checked is subjected to the high pressure when it is clamped to the hydro test rig. The pressure for testing is verified by the ANSI standard used for manufacturing the valve. It includes two types of leakage tests namely 'body leakage test' and 'seat leakage test'. The main components of hydro test rig are design by theoretical method and the same design is checked by finite elemental analysis method using ANSYS 15.0 software.**

Keywords — Hydro test rig, GGCB Valves, Finite Elemental Analysis, body leakage test

I. INTRODUCTION

The Gate, Globe, Check and Ball valves are important in order to regulate the flow of fluids. The GGCB valves are used in Textile Industries, Chemical Industries, Biopharm Industries, Desalination, Marine Industries, Oil Industries, Food Processing Industries, Fertilizers Industries, Waste Processing Industries, Diary Industries Explosion prevention in grain elevators, Pulp and Paper Industries, at different pressure and temperature.

As GGCB valve is made up of forged Mild steel by forging process, due to defects in the manufacturing process there may be leakage in the body. To detect these leakages after assembling valves are tested. For Testing and Inspection of this GGCB valves 4 stations Hydraulic Rig can be efficiently used.

It is used to detect leakages present in the pressure vessels such as gas cylinders, High Pressure Intensifiers, fuel tanks, Pipelines and valves. .There are two methods of leak detection namely seat leakage test and body leakage test. For the body leakage test, valve is completely filled with water, it is pressurised by using high or low pressure pneumatic pump to testing pressure the water is kept in the pressurised condition for three minutes. If leakage occurs, it is noticed by visual inspection or drop in pressure observed on the pressure gauge. The difference between body and Seat leakage test is that in Seat leakage the valve is at completely closed position whereas in body leakage testing valve is at opened condition. The pressure and the time for both the tests are same.

II. METHODOLOGY

There are mainly two methods which are used to obtain the results and to achieve the goal of this study. The methods used are theoretical calculation by using ANSI standard and Finite Element Analysis (FEA). Theoretical calculation is carried out by calculating the various dimensions of the rig with the help of concepts in Strength of Materials and Machine Design. Finite Element Analysis is done by using ANSYS V15.0 software where we performed computer based analysis of the various variable like Stress, Strain and Deflection etc. of the component.

A. Design by Theoretical Method

The structure of the test rig is designed like a high pressure hydraulic press which consists of top and bottom plate connected by three tie rods. For testing a specimen valve it is located between locators present on two plates. So the main components to be designed theoretically are Tie rods, top plate and bottom plate .Tie Rod Diameter is found out by using column theory. For this the following material properties are used.

B. Specifications of Material used

Material used for manufacturing the machine: Medium Carbon steel
Carbon content in the mild steel: 0.40 % of carbon
Density: 7801Kg/m^3
Ultimate tensile strength of the mild steel: 650 N/mm^2
Young's modulus: 210 GPa
Yield Strength: 250 N/mm^2
Poisson's ratio: 0.30

C. Design Calculations

Valve testing specifications provided by authority.

The sheet has provided with job size (DN), Flange rating, Hydro test pressure. O- ring ID which is to be consider while actual calculations of forces.

From this given data we have calculated the design forces for each job size and hydro test pressure using following method.

Working pressure, P=396 kg/cm^2
i.e. P = 38.83 MPa
Design Pressure = Working Pressure x FOS
Considering FOS = 1.
P_D = 38.83 x 1.2
=46.59 MPa
P_D=46.59 x 10^6 N/ m^2

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 04, April-2016

Area of valve,

$$A = \frac{\pi}{4} \times d^2$$

$$= \frac{\pi}{4} \times (54)^2$$

$A = 0.2290\ m^2$

Bending force on the plate-

$F = P_D \times A$

$= 46.59 \times 10^6 \times 0.2290$

$= 106.661$ KN

$F \approx 106.67$ KN

D. Design of the Top and Bottom plate-

Material of the plate:- Medium Carbon Steel

Young's Modulus of MS (E) = 210GPa

From Fig.1

b = 200 mm

L = 1500 mm

t =?

Considering Problem statement,

Allowable deflection of plate = 0.5mm

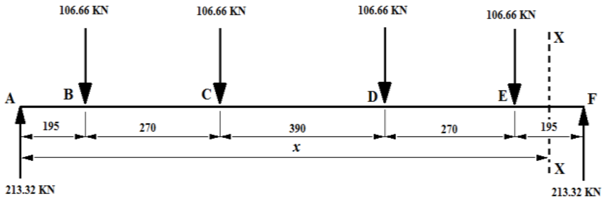

Fig 1. Plate as Simply Supported Beam

Using Macaulay's Method,

$$EI\frac{d^2y}{dx^2} = M$$

$-213.32x + 106.66(x-0.195) + 106.66(x-0.465) + 106.66(x-0.855) + 106.66(x-1.125)$ ……..(a)

Integrating equation (a),

$$EI\frac{dy}{dx} = -213.32\frac{x^2}{2} + C_1 \left| +106\frac{(x-0.195)^2}{2} \right| \left| +106\frac{(x-0.465)^2}{2} \right| \left| +106\frac{(x-0.855)^2}{2} \right| \left| +106\frac{(x-1.125)^2}{2} \right|$$

……….(b)

Integrating equation (b),

$$EIy = -\frac{213.32}{2}\frac{x^3}{3} + C_1x + C_2 \left| +\frac{106}{2}\frac{(x-0.195)^3}{3} \right| \left| +\frac{106}{2}\frac{(x-0.465)^3}{3} \right| \left| +\frac{106}{2}\frac{(x-0.855)^3}{3} \right| \left| +\frac{106}{2}\frac{(x-1.125)^3}{3} \right|$$

Applying Boundary Conditions,

At

$x = 0$

$y = 0$

$\therefore C_2 = 0$

At

$x = 1.320\text{m}$

$y = 0$

$$0 = -106\frac{(1.320)^3}{3} + C_1(1.320) + \frac{106}{2}\frac{(1.125)^3}{3} + \frac{106}{2}\frac{(0.855)^3}{3} + \frac{106}{2}\frac{(0.465)^3}{3} + \frac{106}{2}\frac{(0.195)^3}{3}$$

$C_1 = 32.7017$

Putting values of C_1 and C_2 in equation,

$EIy = -35.33x^3 + 32.7071x + 17.66(x - 0.195)^3 + 17.66(x - 0.465)^3 + 17.66(x - 0.855)^3 + 17.66(x - 1.125)^3$

We know that Maximum bending moment occurs between section C and D.

Therefore, x =660 mm from A.

Allowable deflection (y) = 0.5 mm = 5×10^{-4} m.

$EIy = -35.33(0.66)^3 + (32.7071 \times 0.66) - 17.66\ (0.465)^3 - 17.66\ (0.195)^3 + 17.66\ (0.195)^3 + 17.66\ (0.465)^3$

$EI \times 5 \times 10^{-4} = 11.42493$

But, E= 210 GPa ,

Therefore,

$I = 1.08808 \times 10^{-7}\ m^4$.

But $I = \frac{bt^3}{12}$

$I = 1.08808 \times 10^{-7} = \frac{bt^3}{12}$

$$t^3 = \frac{12 \times 1.08808 \times 10^{-7}}{0.2}$$

t = 18.68 mm≈ 50mm

After employing trial and error method for finding value of plate thickness in ANSYS 15.0 and considering safety the plate thickness selected is 50 mm.

E. Design of Tie Rod –

Tie rod act as column so we can use Rankine formula for design of tie rod

Considering both ends of the Tie Rod fixed

$$L_e = \frac{\text{length of tie rod}}{2}$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 04, April-2016

$L_e = \frac{650}{2} = 325$ mm

Radius of gyration

$K = \sqrt{\frac{I}{A}}$ but, $I = \frac{\pi}{64} d^4$, $A = \frac{\pi}{4} d^2$

Therefore K= 0.25d

By Rankine Method, (for buckling load)

$$P_c = \frac{\tau_c \times A}{1+\alpha\left(\frac{Le}{k}\right)^2}$$ but for mild steel $\tau_c = 320$ MPa

$$213320 = \frac{320 \times \frac{\pi}{4} \times d^2}{1+\frac{1}{7500}\left(\frac{325}{0.25d}\right)^2}$$

$213320\ d^2 + (48.0681x10^{6}) = 251.32\ d^4$

By solving equation,

d = 32.1527mm ≈ 50mm.

After employing trial and error method for finding value of diameter of Tie Rod in ANSYS 15.0 and considering friction between the Tie Rod and plates, and aesthetic point of view the diameter of Tie Rod is taken as 50mm.

III. FINITE ELEMENTAL ANALYSIS OF TEST RIG -

The purpose of the FEA is to check the design for failure before manufacturing stage. This helps to eliminate the defects in the design and reduced the cost and also helpful aesthetically and ergonomically. For Finite element analysis of the test rig components, the components are modeled in CATIA V5R19and assembly of components is done. For analysis purpose ANSYS v15.0 WORKBENCH is used.

A. Analysis of top plate

After entering the material properties, we started with top plate. The plate is imported in geometry then, the boundary conditions loads are applied to the plate. In this plate both ends are fixed and force is applied on one face is 35.5 KN. After solving, the maximum deflection in plate is 0.1532 mm.

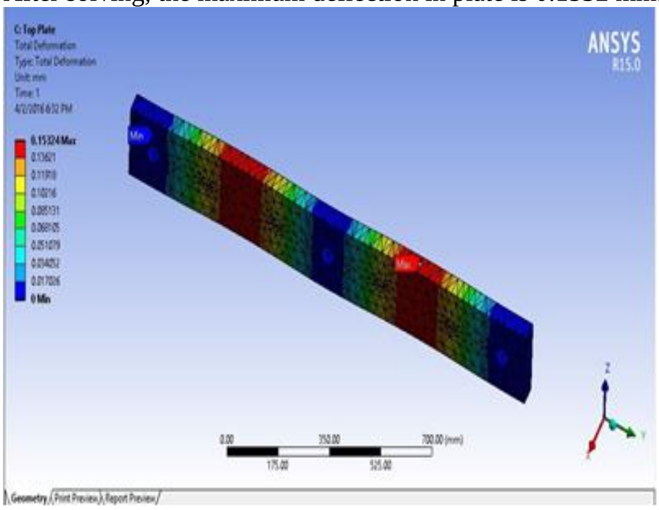

Fig.2. Total deformation of Top plate

B. Analysis of Bottom Plate

Similarly the analysis of bottom plate is performed. The force of value 71.10 KN is applied on one face of bottom plate and both end fixed. The maximum deflection as 0.30647mm which is again less than the 0.5mm and stress induce is also less than yield stress hence design of bottom plate is safe.

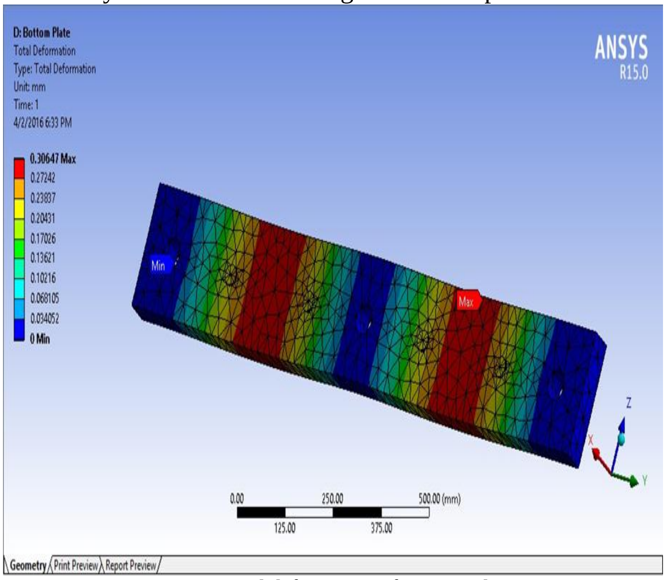

Fig.3. Total deformation of Bottom plate

C. Analysis of Tie Rod

Third component is tie rod. After applying the load on its both ends the linear buckling test is performed and we got the linear buckling of tie rod the deflection observed in it is 0.1229mm which is equal to the expected value i.e. 0.5mm and stress induced is also less than the yield stress. Hence the design of tie rod is safe.

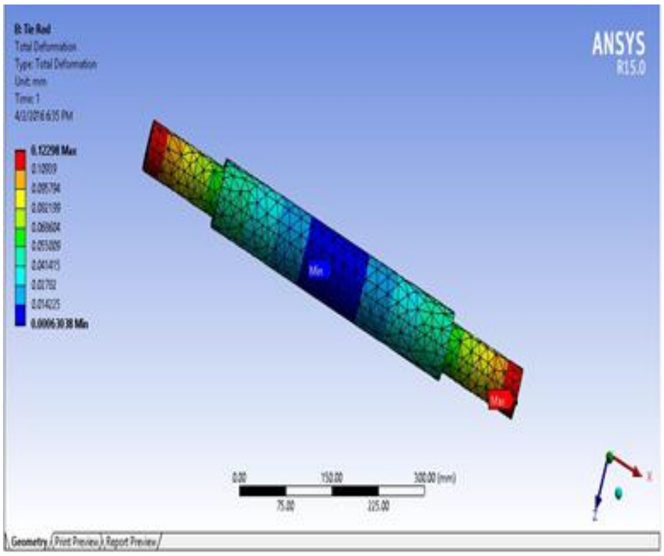

Fig.4. Total deformation of Tie Rod

D. Analysis of Hydro Test Rig Assembly

For the analysis of rig, the parts are assembled in CATIA. The both plates are fixed across the tie rod and fixed with the bolt. The loads which were applied on top and bottom plate individually before are now applied altogether on the rig assembly.

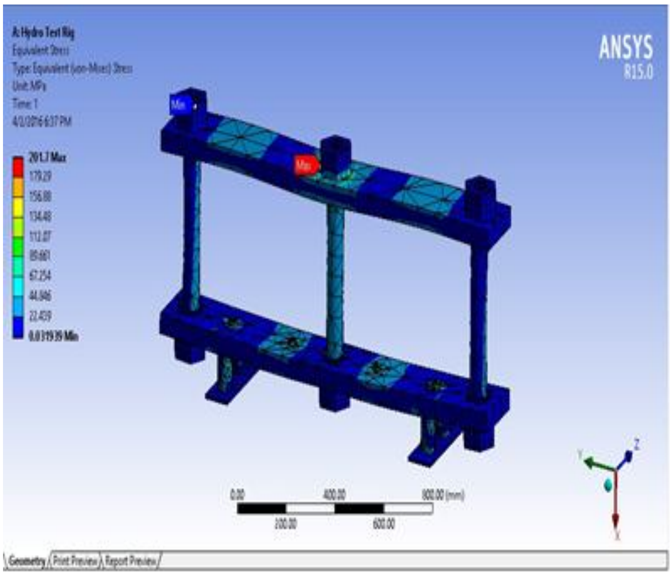

Fig.5. Stress Distribution of Hydro Test Rig

The results obtained are total deflection and equivalent stress. On comparing both the results with the given boundary conditions the test rig design is safe.

In this the maximum stress induced in total test rig is 201.7 MPa which is far less than the yield stress of mild steel.

As shown in the figure, the total deflection in the rig is 0.3059mm which is less than the 0.5 mm as per boundary condition considered hence the design is safe.

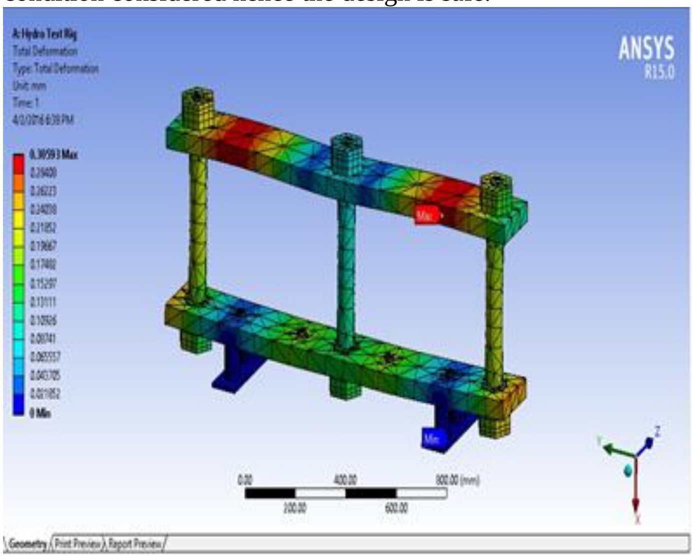

Fig.6 Factor of Safety for Test Rig acquired in ANSYS

IV. RESULTS AND DISCUSSIONS

To verify the design, the theoretical Factor of safety must be equal to the analytically obtained factor of safety as follows,

$$\text{factor of safety} = \frac{\text{yield stress of material}}{\text{maximum stress induced in rig}}$$

$$\text{factor of safety} = \frac{250}{201.7} = 1.2394$$

Hence, the theoretical and analytically obtained factor of safety are approximately same. Therefore the design is safe

V. CONCLUSION

Analysis results are reliable as seen in Mesh Sensitivity convergence and actual Testing.

FEA Validation shows the analytical design calculations we have done are approximately correct and maintaining Factor of Safety 1.2.

REFRENCES

[1] Kunal M. Phalak, Abhijeet R. Patil, Madhura V. Barve , P. D. Darade, in paper entitled, “Design and Finite Element Analysis of Hydro Test rig for testing piston valve.”

[2] P. K. Parase, M. V. Kavade & S. H. Limaye in paper entitled “Design Development and Testing of Butterfly Valve Leakage Test Rig.”

[3] Emerson Process Management stated in the Brooks Application Notes entitled as “Valve Seat Leak Testing.

[4] Ramamrutham “Strength of Materials”.

[5] V. B. Bhandari “Design of Machine Elements”.