International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 1, January - 2014

Design for Stress and Vibrational Analysis of Valve Test Rig

Prof. P. S. Burli*1 , Prof. C. S. Rajashekhar*2

[1]Department of Mechanical Engineering, KLE College of Engg &Tech, Chikodi / VTU Belgaum
[2]Department of Mechanical Engineering, Maratha Mandal Institute of Technology, Belgaum / VTU

Abstract

The paper presents structural analysis of Test rig for leak testing of Hydraulic valve (Gate valve). Initially the calculations are carried out to find the required torque to turn the screw for given maximum pressures on the seat of the valve body. Pretension estimations and Initial calculations are carried out for minimum cross section of the test rig body, and further the rib height is increased and analysis is carried out. So further vertical ribs are added nearer the loading regions. The results shows obtained stresses are with in the allowable range of stresses for the material. Modal analysis is carried out to check the vibrational behavior of the structure. The results shows, the structure is safe. The initial frequencies are much higher than the required frequency of 30 Hz. Final assembly analysis is carried out with gate valve and the results presented. A final product with some modifications is also represented.

1. TEST RIGS:

Test rigs are mechanical equipments used to test the strength of valves. Many defects like voids, cracks, blowholes etc are formed during manufacturing process which will reduce the strength and causes the failures during operations.

2. INTRODUCTION:

Hydraulic valve test rig specially applied to test all kinds of flanged valves such as gate valves ball valves stop valves etc. The test rig is effectively composed of four parts that is mechanical system, hydraulic system, electric system and testing media cycling system. The entire process of the emplacing and clamping of the Valve to be tested is completely controlled by electric components and hydraulic components. The testing medium can be water, gas or oil. And the maximum testing pressure of liquid intensity test is 48Mpa.Test rigs of this model are of the following features, such as rational and compact in configuration, flawless in function, steady and reliable in performance, high in degree of automation, advanced in techniques and soon.

What is important, the test rigs of this model have no effect of external force upon the valve body, because the clamping jaw clamp directly on the flange of the valve. One side of the rig can be moved, thus the testing will be restricted by the structural length of the valve to be tested. When testing valves, clamp the flange on the both sides of the valve to be tested, pressurize to the necessary pressure and perform intensity test of the valve body, valve cover and middle flange. And the other side of the rig can be reversed by 90 degrees to perform leakage test and air tightness test.

3. PROBLEM DEFINITION

Design, Modeling and Analysis of Test Rig for Valve Testing. Here the valve size varies from minimum 12.7mm (1/2 inch) to 101.6(4 inch). The test rig design need to withstand following pressure details.

Ratings #	Shell (Bar)	Seat (Bar)
150	31.1	22
300	77.6	56.9
600	155.2	113.8
900	232.8	170.7
1500	388	284.5
2500	646.6	474.2

Table 1: Test Pressure Details

4. LITERATURE SERVEY

Structural vibration problems present a major hazard and design limitation for a very wide range of engineering products. On the other hand, in a number of structures, structural integrity is of paramount concern, and a thorough and precise knowledge of the dynamic characteristics is essential. There is also a wider set of components or assemblies for which vibration is directly related to performance, either by virtue of causing temporary malfunction or by creating disturbance, discomfort or noise. Therefore, it is important that the vibration levels encountered in service or operation be anticipated and brought under satisfactory control. A comprehensive study of the vibration phenomena includes determining the nature and extent of vibration response levels and verifying theoretical models and predictions. A significant amount of applied technology pertaining to vehicle dynamics has emerged over the last 20 years or so. The advent of finite element analysis as a tool to study vehicle's vibration and dynamic aspect has further accelerated growth in this field.

Vibration is the study of the repetitive motion of objects relative to a stationary frame of reference or nominal position (usually equilibrium). Vibration is evident everywhere and in many cases greatly affects the nature of engineering designs. The vibration properties of engineering devices are often limiting factors in their performance. Vibration can be harmful and should be avoided, or it can be extremely useful and desired. In either case, knowledge about vibration-how to analyze, measure and control is useful.

A comprehensive understanding of structural dynamics is essential to the design and development of new structures [6], and solving noise and vibration problems on existing structures. Modal analysis is an efficient tool for describing, under-standing, and modeling structural behavior. The study of modal analysis is an excellent means of attaining a solid understanding of structural dynamics.

Modal analysis is defined as the study of the dynamic characteristics of a mechanical structure. The dynamic behavior of mechanical structures is typically done using a linear system modeling approach. The inputs to the system in general are forces ("loads"), the outputs the displacement or acceleration responses. Using these variables, classical system analysis can be applied. Most practical noise and vibration problems are related to resonance phenomena, where the operational forces excite one or more of the modes of vibration. Each mode can be considered as an independent single degree-of-freedom system. Modes of vibration that lie within the frequency range of the operational dynamic forces always represent potential problems. An important property of modes is that any forced or free dynamic response of a structure can be reduced to a discrete set of modes. The modal parameters are:

• Modal frequency
• Modal damping
• Mode shape

The modal parameters of all the modes, within the frequency range of interest, constitute a complete dynamic description of the structure. Hence the modes of vibration represent the inherent dynamic properties of free structure (a structure on which there are no forces acting).

5. MATERIAL SPECIFICATIONS:

Plate – ASTM A-36
Allowable stress =250 N/mm^2
Screw Shaft – EN 8
Yield Stress =550 Mpa
Fasteners – 8.8 grade

6. METHODOLOGY:

- Initial design calculations for loads acting on the structure
- Estimation of torque loads for given maximum pressure loading
- Total load estimation along with pretension load
- Minimum cross sectional requirement calculations
- Modeling of the initial geometrical model
- Meshing of the model with 8 noded solid45 element
- Analysis of Results for vonmises and displacements
- Alternative designs for improvements
- Modal analysis for the better designs
- Presentation of results

6.1 Assumptions

- Three dimensional analysis is considered for analysis
- Material is Steel with isotropic properties.
- Analysis is done considering elastic region.
- All approximations applied to FEM are applied for this analysis.

6.2 design specifications:

- Valve Size: ½ inch to 4 inch (12.7mm to 101.6mm)
- Maximum Test Pressure =646 bar =64.6 Mpa
- Screw Diameter =80 mm with 3TPI (3 Threads per Inch)
- Load factor for Screw calculation =1.25
-

6.3 calculation of maximum load on the screw shaft:

- Maximum Pressure in the design = 646 bar = 64.6 N/mm^2
- Diameter of the shaft d =80mm, μ=0.15
- Area of the shaft $A_s = \Pi d^2/4 = 3.14*80^2/4 = 5024mm^2$

Total load acting on the screw shaft = 324550.4N

- Considering Load factor of 1.25, the load acting on the structure w=405688N
- Torque required to turn the screw $T = w \tan(\theta+\alpha)*d/2$
- Here number of starts n=3
- Pitch p= 25.4mm , d=80mm
- Helix angle $\alpha = \tan^{-1}(3p/3.14*D)$
- Helix angle $\alpha = \tan^{-1}(3*25.4/(3.14*80) = 16.866^0$
- Friction angle $\theta = \tan^{-1}(0.15) = 8.55^0$
- Torque required = 405688*tan (16.866+8.53)*40 =7704004.73 N-mm=7.7KNm

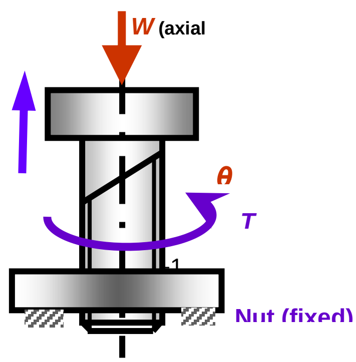

Fig 1: Power Screw terminology

7. PRETENSION LOAD REQUIRED:

Standard pretension load on the screw =2840d =2840*80=227200 N [8]

The above pretension load is smaller than the load coming on the screw. So to avoid loosening of the members, pretension load should be at least more than 10% of the maximum applied load. (More the pretension load, the structure is more stable but section dimensions will increasing by which cost of the structure increases)

Yield stress of the material en8 = 550 N/mm^2

Pretension load =1.1*405688 =446256.8 N

Total load on the screw member =Applied load + pretension load = 405688+446256.8=851944.8 N

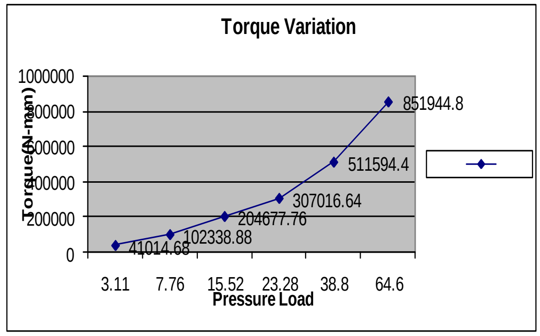

Fig 2: Torque load variation with varying pressure load based on Shell Rating

The above picture shows increasing torque load requirement based on the pressure variation based on shell rating of valve. Lower loads for lesser pressure and higher torque requirements for higher pressure can be observed along with calculated values of Torque.

5.5 Screw Specifications:

Length=1200mm
Diameter =80mm
Material =EN8
Yield Stress =550 Mpa
Axial stress in the screw rod = $851944.8/(3.14*80^2/4) = 169.5$ N/mm^2
Factor of safety of the screw = 550/169.5=3.24 (This is for
maximum loading condition)

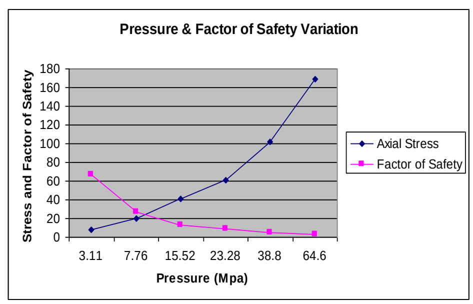

Fig 3: Variation of Axial Stress and Factor of Safety with pressure variation

The above picture shows variation of Axial stress and Factor of safety with varying pressure loads. From the graph it can be observed that Factor of safety value is reducing with increasing pressure and stress value is increasing with increased pressure loads.

5.6Cross sectional requirement of the problem:

Considering a box section having area to take this load with plate material ASTM 36
Allowable stress = 250 N/mm^2
Minimum cross sectional requirements = 851944.8*3.5/250=11928 mm^2 (A load factor of 3.5

is considered for design due to sudden variation of load under dynamic conditions for plate material as its strength is less compared to Screw material) Considering more height for the problem to take higher bending loads minimum cross section is considered as 60mm width by 200 mm height.(area=12000 mm^2).This higher dimensions are Considered due to possible turning and bending loads along with axial loads during the process of execution. This design will be validated using the software package by applying proper boundary conditions.

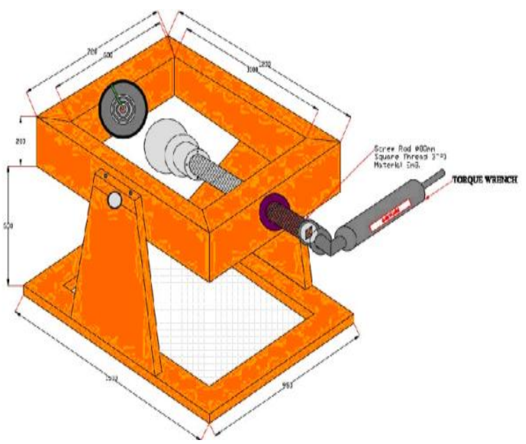

Fig 4: Initial full Geometry of the test rig

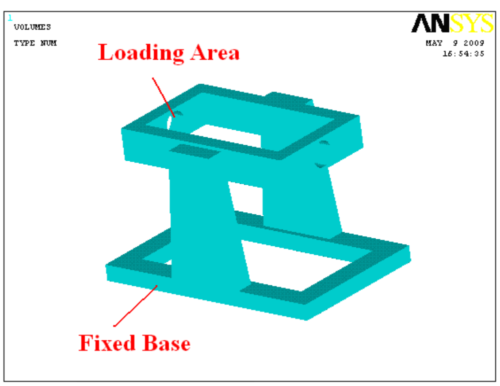

Fig 5: Initial full Geometry of the problem

The analysis has been carried out for complete initial design with 100mm top rib width. The below picture shows modified geometry. Additional ribs are provided near the screw rod positions. These are provided to reduce the free length bending effects.

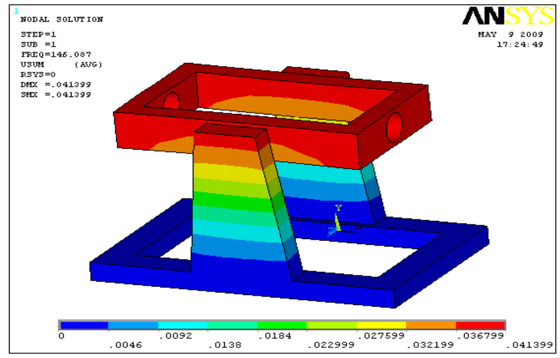

Fig 6: Displacement in the structure

Maximum displacement is around 0.894769mm. The deformation is almost varying linearly.

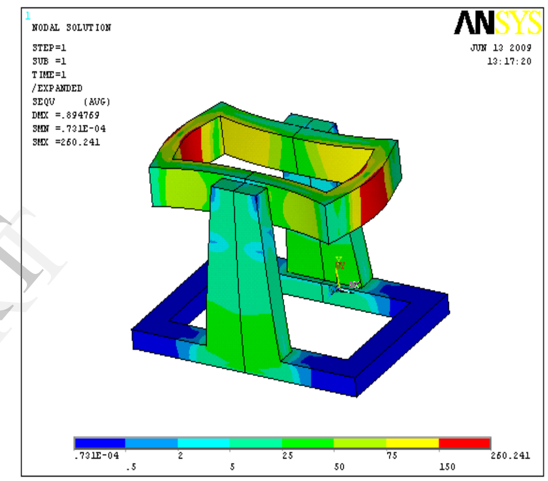

Fig 7: vonmises Stress distribution in the structure

Maximum vonmises stress is around 260.241N/mm^2. The stress in the structure is more than allowable stress of 250 N/mm^2.

The above results shows, initial frequency is higher than the required natural frequency of the test rig of 30 Hz. So design is safe.

Sets	1	2	3	4	5	6	7	8	9
Freq (Hz)	146.09	213.15	276.10	322.74	392.78	402.59	431.93	611.00	637

Table 2 Initial model results

The above results shows, initial frequency is higher than the required natural frequency of the test rig of 30 Hz. So design is safe.

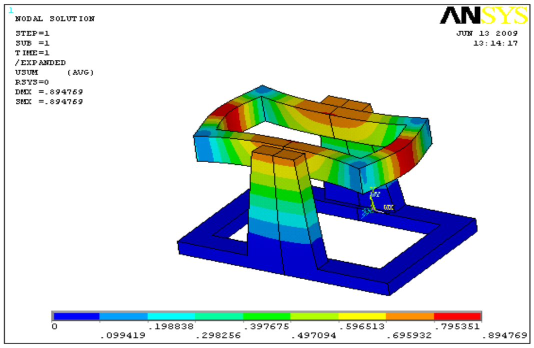

Fig 8 First mode shape of the problem

The above picture shows initial mode shape of the problem for 146.087 Hz. The structure is having longitudinal vibration by the observation. Loading region is removed from the display as the screw member will be joined to the structure. For the fundamental Natural frequency of 146.087Hz the frame shows the maximum deflection of 0.04077mm

8. MODAL ANALYSIS RESULTS FOR INITIAL DESIGN

Initial frequency is higher than the required natural frequency of the test rig of 30 Hz. So design is safe

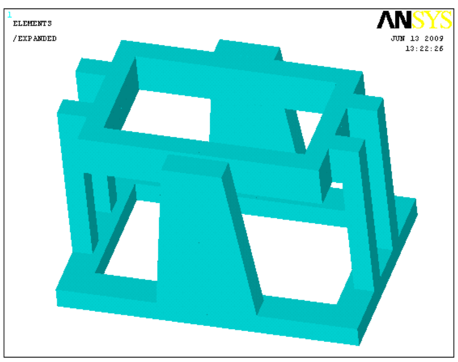

Fig 9: modified geometry

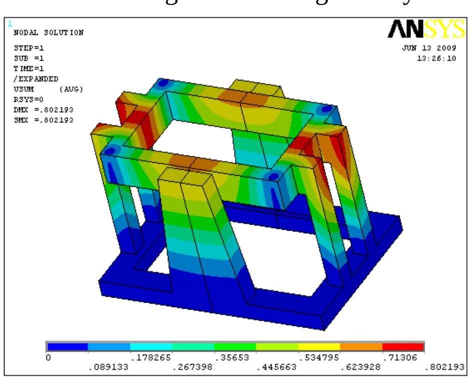

Fig 10 Displacement plot of the problem

Maximum deformation can be found at added ribs. The maximum vonmises stress developed 235.369 N/mm^2 is less than the allowable stress of 250N/mm^2. The structural results shows safety of the structure for given loads. The modal analysis result shows that the first natural frequency of 150.13 Hz is more than desired frequency of 30 Hz. So the structure is sufficiently rigid for vibration.

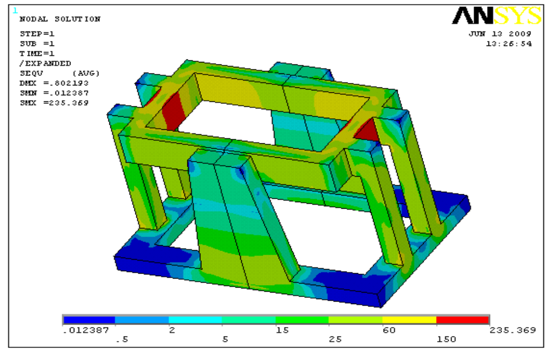

Fig 11: Vonmises stress of the problem

The above picture shows vonmises stress developed in the structure. The maximum vonmises stress developed 235.369 N/mm^2 is less than the allowable stress of 250N/mm^2. The structural results shows safety of the structure for given loads.

Sl no	1	2	3	4	5	6	7
N freq	150.13	167.20	237.93	336.70	355.37	363.38	367.28

Table 2: Modal analysis results for modified design

The above picture shows natural frequencies obtained for the model. The first natural frequency of 150.13 Hz is more than desired frequency of 30 Hz. So the structure is sufficiently rigid for vibration.

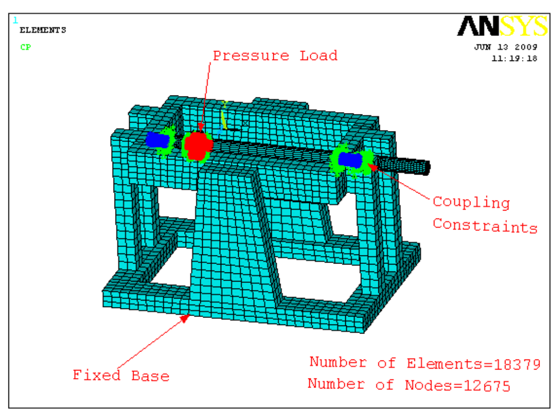

Fig: 12 Boundary conditions plot of problem

The above picture shows the applied boundary conditions of the problem. Maximum number of elements are 18379 and number of nodes are 12675. The test rig body is map meshed with 8 noded hexahedra elements and valve body is free meshed due to the complexity of the geometry. 4 noded tetrahedral element is used for gate valve meshing. The shaft and gate valve is coupled using coupling equations to avoid sealed joints.

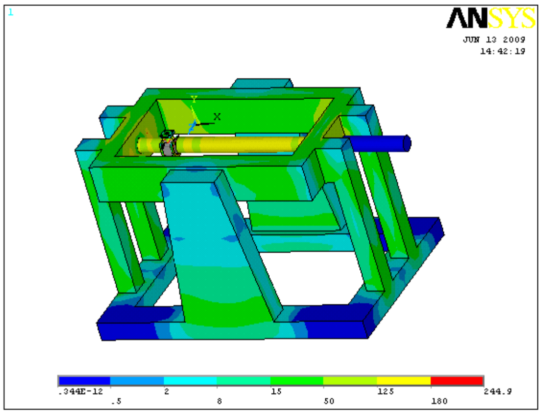

Fig 13: Vonmises stress with valve in Assembly

The maximum vonmises stress developed is 244.9N/mm2 is less than the allowable stress of 250N/mm2. The Assembly results shows safety for the given loads.

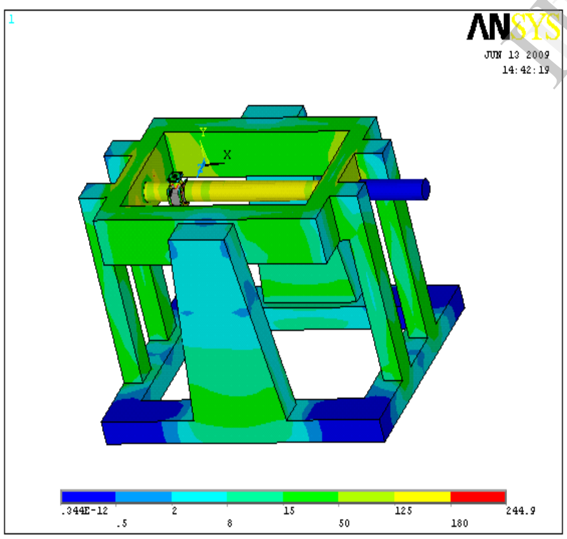

Fig 14: Vonmises stress with valve in Assembly

9. THEOROTICAL VALIDATION OF THE PROBLEM

Shear stress calculation on Screw shaft subjected to Torsional load:

Tensional stress on the screw element :

Length of the screw L=1200mm

Diameter of screw d= 80mm

Torsional load on the structure T=7704004.73 N-mm

Torsional shear stress on the member τ = (T*d/2)/J

Where 'J' is polar moment of Inertia=Π*d^4/32 = 4021239

Shear stress τ = 7704004.73*40/4021239=76.33 N/mm^2.

Allowable shear stress on the member =550/2= ~275 N/mm^2 which is more than stress developed on the shaft material. So shaft is safe from stress point.

Similarly angle of twist of the problem θ=TL/(GJ) where 'G' is shear modulus = E/(2(1+υ))

Where υ is poisons ratio=0.3

& Young's Modulus E=2*10^5 N/mm ^{2}

G=E/(2(1+υ)) =76923N/mm ^{2}

Angle of twist θ =

7704004.73*1200/(76923*4021239)=0.0298 radians

10. RESULTS VERIFICATION FROM ANSYS

The picture shows built up model with boundary conditions. The model is constrained at one end and subjected to a torque load of 7704004.73 N-mm.

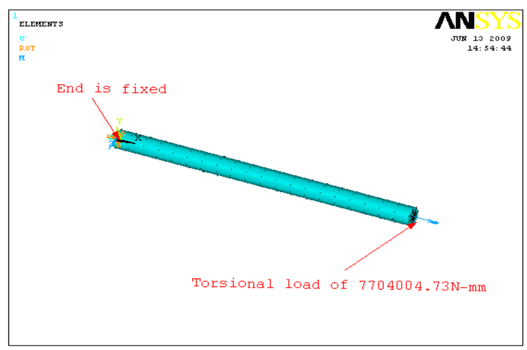

Fig 15 Boundary conditions for the shaft

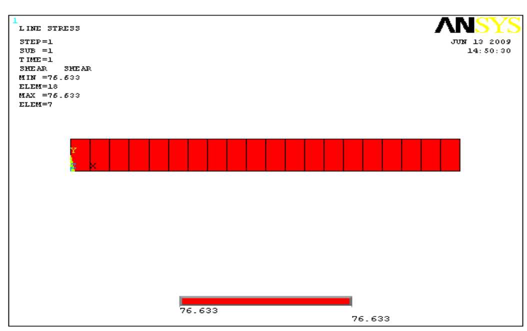

Fig 17 Rotational deformation and Shear stress plot

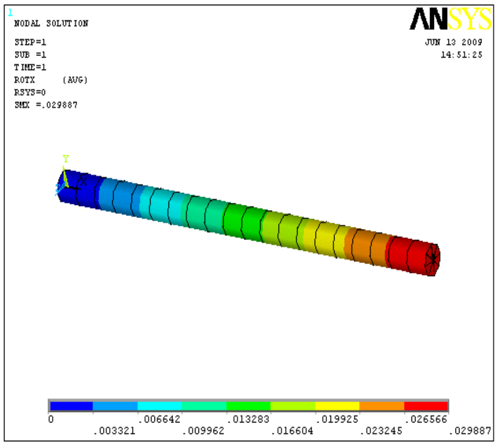

Fig : 17 Rotational deformation and Shear stress plot

The above picture shows Shear stress and deformation results. Both are matching with theoretical values So ansys results match with theoretical calculations and so analysis results can be validated

11. DISCUSSION:

The test rigs are very important members for checking the safety of valves before placing them to actual work. For the present problem a test rig is theoretically designed for given seat and shell pressures (Table 4.1). Since the shell pressure is higher than the seat pressure, shell pressure is mainly considered for design calculations. Initially the loads are calculated for given shell pressure(Table 5.1). Pretension loads along with the torque required for given shell pressure is also calculated(Table 5.2). Also factor of safety calculations are carried out for screw material(Table5.3) The torque required for different shell pressure is also represented in the Fig5.2The graphical variation is represented between Factor of safety and screw stress for given shell pressures.(Fig 5.3). Minimum factor of safety found to be around 3.24 for screw material which represents the safety of the structure. The stress generated in the screw material is around 169.5 Mpa which is less than the yield stress of 550 for the screw material. By simple calculations minimum cross section area is calculated for support structure. Initial width of the structure is taken as 60 mm. The cad modeled and analysed using 8 noded solid45 shows higher stresses of 349.22 Mpa which is more than the allowable stress of 250 Mpa for the material showing the failure of cross section. Further the design is improved by taking 100mm width. Again cad modeling is carried out and object is meshed with 8 noded solid45. This time the stress value is around 260.241 Mpa which is again more than the allowable stress. So the design is further modified with additional ribs near the loading. This design has improved the stresses. Now the stress value is around 235.3Mpa which is less than the allowable stress of 250Mpa. So the structure is safe for given maximum loading. Further modal analysis is carried out for the design. Both the designs are safe for vibraitonal behavior as the obtained results are much higher compared to the design requirement of 30 Hz. So system is free or resonance as by the results. Further a gate valve is placed in the assembly and the loads are applied. Here gate valve is free meshed due to complexity. Coupling constraints are applied between shaft and the gate valve for load transfer. The results are presented for vonmises. The stress value is around 245 Mpa which is lesser than the allowable stress of the material. So the structure is safe in both stress and vibraitonal nature. A final configuration at the end is represented. The results for displacements, vonmises stress and modal results are presented Theoretical validation is carried out for shaft material for the calculated maximum torque value of 7704004 N-mm. The theoretical shear value of 76.33 Mpa and twist of 0.0298 radians are exactly matched by Ansys results. Both the result pictures are presented proving ansys ability of solutions near the exact values.

12. CONCLUSIONS:

The test rig has been designed and analyzed. The results are summarized as follows.

- Initially the theoretical calculations are carried out based on Shell rating pressures(Shell pressures are considered as the shell pressure is more than the seat pressure)
- For the given shell pressure, axial loads are calculated based on the shaft area. Load factor of 1.25 is considered for the calculations. The pretension load is taken 10 more than the axial load. The total load coming on the structure is calculated and results are tabulated for total load, pretension load and torque requirements. Also graphical variation is represented for factor safety and screw shaft stress for variation of shell pressure.
- Initial model has been built and analysed. The results shows higher stress of around 349.92 N/mm^2 which is more than the allowable stress of the problem. So design is modified with 100mm widht for the top frame.
- Full geometry has been modeled using Ansys top down approach and results are obtained. The result shows still higher stress (260 N/mm^2) which is more than the allowable stress of the member. So further design has been modified with addition ribs near to the loading region to reduce the bending effect.

- This modified design is giving feasible results as the both stress as well as deformation is well with in the working range.
- Modal analysis has been carried to test for vibrational nature. The results shows, higher initial natural frequencies which represents total safety of the test rig for resonance.
- Final Assembly analysis with a gate valve is carried out and results are captured on the test rig body whose maximum vonmises value is around 245 N/mm^2 which is less than the allowable stress of 250 N/mm^2 for the plate material.
- Also verification is carried out for shaft material. The analysis results and theoretical results are having exact coincidence and proving ansys abilities for correct results.
- All the results are presented with necessasary pictures.

13. REFERENCES

[1]. Laws, W. C. and Muszynska, A., 1987, "Periodic and Continuous Vibration Monitoring for Preventive/Predictive Maintenance of Rotating Machinery", *ASME Journal of Engineering for Gas Turbines and Power*, Vol. 109, 159-167.

[2]. Vance, J. M., 1988, *Rotor Dynamics of Turbo-Machinery*. New York: John Wiley and Sons, Inc.

[3]. Goodwin, M. J., 1989, *Dynamics of Rotor-bearing Systems*. London: Unwin Hyman.

[4]. Simmons, H. R. and Smalley, A. J., 1990, "Effective Tools for Diag-nosing Elusive Turbo- machinery Dynamics Problems in the Field", *ASME Journal of Engineering for Gas Turbines and Power*, Vol. 112, 470-476

[5]. Groover," Mechanical Vibration", New Chand & Bros, 2000

[6]. University Of Cambridge, Department Of Engineering, "Dynamics And Vibrations", Annual Report-1996\1997.

[7].Introuduction to the Finite Element Method, Desai/Abel – CBS publishers 2002

[8].Norton, Robert L., Machine Design – An Integrated Approach, Prentice-Hall: New Jersey, 1998, 2 nd printing

[9]. ss-yy china test bench.com

[10]. hydraton.co.uk