American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-4, Issue-4, pp-108-116
www.ajer.org
Research Paper Open Access

Weight Reduction of 12"-150 Class Plug valve Casting Body by Finite Element Analysis and Experimental Method

[1,] Prof. Laukik B. Raut, [2,] Pradnyawant .K.Parase
(SVERI's COE Pandharpur, Mechanical Department,)

ABSTRACT: *Now-a-days cost of the materials is very high, so there is need to minimize the cost. For this purpose, it is necessary to optimum use of man, machine and material. So that's why it is essential to reduce the weight of the plug valve body.The stress analysis means determining the stress value of the valve body when internal pressure is applied. Ultimate aim of the current experiment is to reduce the casting weight of the body. The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. In the field of competition, all companies should supply their goods and services with high quality, in shortest period with lower prices than its competitors in order to keep their capacity and power to compete. Plug valves are machine elements which are commonly used for regulation of fluid, semi-liquid and granular medium flow on variety of tanks and pipeline systems. This experiment discusses FEA analysis of Plug–valve body followed by Experimental stress analysis using strain gauge method for weight reduction. New reduced weight models were prepared on the basis of validation of the results obtained from stress analysis procedure. The weight reduction is done by changing the wall and rib thickness. The results clearly shows the maximum weight reduction is 9.95 kg (7.11 % of original weight) while keeping maximum stress level up to 125.187 N/mm2 which is safe for the applied pressure as per standard.*
Key words: *Plug valve, Strain Gauge, rosettes, standard.*

I. INTRODUCTION

Plug Valves :

A plug valve is a rotational motion valve used to stop or start fluid flow. The name is derived from the shape of the disk, which resembles a plug. A plug valve is shown in Figure 1.10. The simplest form of a plug valve is the petcock. The body of a plug valve is machined to receive the tapered or cylindrical plug. The disk is a solid plug with a bored passage at a right angle to the longitudinal axis of the plug.In the open position, the passage in the plug lines up with the inlet and outlet ports of the valve body. When the plug is turned 90° from the open position, the solid part of the plug blocks the ports and stops fluid flow. Plug valves are available in either a lubricated or non lubricated design and with a variety of styles of port openings through the plug as well as a number of plug designs.

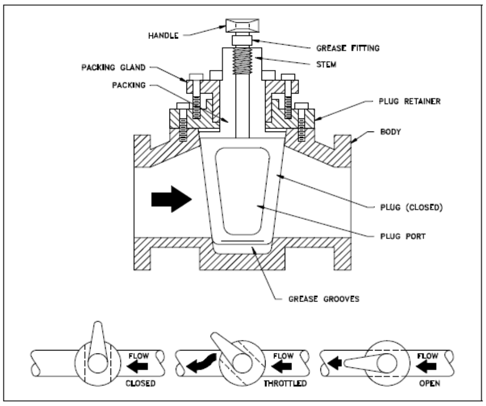

Figure1.1: Cross Section Plug Valve

Valve Operations

The Accuseal plug valve is a non-lubricated, resilient seal, plug- type valve which has a mechanical means of freeing the plug before it is rotated from the closed to the open position.

- **Opening of the valve**

For opening of plug valve, hand wheel has to turn counter clockwise. During this operation the plug is raised while the slips are retracted away from the body. When the slips are fully retracted from the body seating area, the plug is then able to rotate 90 degrees to fully open position. When the valve is in the full open position, slips and slip seals are completely protected from line flow.

- **Closing the valve**

For closing of plug valve, hand wheel has to turn clockwise. During this operation the retracted plug and slips are rotated 90 degrees without body contact. This rotation continues until the slips are positioned over the upstream and downstream port areas. Continued rotation of hand wheel mechanically forces the plug downward and forces the slips outward to seal firmly against the valve body. This produces a secondary metal to metal seal on both upstream and downstream areas providing double isolation.

Applications of plug valves

Biofuels Product isolation,Multi product manifolds,Custody transfer units,Tank farms (Oil Depots)Aviation fueling system:

II. LITERATURE REVIEW

2.1 Introduction:

Literature survey is carried out for getting the information regarding FEA and optimization techniques. This literature survey provided useful information regarding the experimental stress analysis method. The literature is collected from various international published papers, International journals and company documents. The literature review of work done by different researches in the area of weight optimization, FEA and experimental stress analysis technique is discussed below:

2.2 Present Literatures & Reviews:

[1] Mona Golbabaei, RouhollahTorabi, Ahmad Nourbakhsh, KaroSedighiani:

This article carried out work of centrifugal pumps engaged in high pressures must have smooth and safe operation without leakages. To control this phenomenon, optimal geometry and proper materials must be considered in design of all mechanical components including, significantly, the volute casing.

[2] DeokarVinayakHindurao, D. S. Chavan:

In the field of competition, all companies should supply their goods and services with high quality, in shortest period with lower prices than its competitors in order to keep their capacity and power to compete .

[3] Xue Guan Song, Lin Wang, SeokHeumBaek, YoungChul Park:

This paper carried out work related to a butterfly valve is a type of flow control device, typically used to regulate fluid flow. This paper proposes a new process to meet desired needs in valve design that is characterized by the complex configuration.

[4] A. Dorogoy, D. Rittel:

This paper analyzes the errors inherent to the determination of mixed mode stress intensity factors from data obtained by using a three strain gauge rosette. The analysis shows that the errors are mainly due the third characteristic value (3/2) and its corresponding coefficients. It is also shown that the errors do not depend on the orientation angle of the rosette, the angle between the strain gauges and the material properties.

[5] AleksandarPetrovic:

The paper consistsa stress analysis of a cylindrical pressure vessel loaded by axial and transverse forces on the free end of a nozzle. The nozzle is placed such that the axis of nozzle does not cross the axis of cylindrical shell. The method of finite element was applied to determine the state of stress in the cylindrical shell. **[11] B. Prabu, N. Rathinam, R. Srinivasan, K.A.S. Naarayen:**

[6] K. H. Jatkar, Sunil S. Dhanwe:

In this paper carried out study on the dynamic analysis of single cylinder petrol engine was conducted. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations of connecting rod and crankshaft. This load was then applied to the FE model and boundary conditions were applied according to the engine assembly. It is observed that maximum stress is developed at crank pin of crank shaft. The maximum stresses are developed at the fillet section of the big and the small end of connecting rod.

[7] Joseph F Dues:

This paper examining the mechanics of a soda can is an exciting way to get students interested in strength of materials by relating classroom concepts to everyday objects. A soda can containing a carbonated drink is a thin wall pressure vessel. The geometry of the soda can is optimized to minimize the amount of aluminum required. The wall thickness is very thin and is subject to an appreciable amount of stress and strain. By mounting a strain gage to the can, and then relieving the stress by opening the can, the change in strain from the pressurized to unpressurized condition can easily be measured.

[8] Dr. K. H. Jatkar, Sunil S. Dhanwe:

This paper carried out FEA work on gate valves are used when a straight-line flow of fluid and minimum restriction is desired. Gate valves are so named because the part that either stops or allows flow of fluid through the valve acts somewhat like the opening or closing of a gate and is called, appropriately, the gate. The objective of this paper to perform a stress analysis of the critical component of Gate Valve..

III. FINITE ELEMENT ANALYSIS

Simple mathematical model can be solved analytically, but more complex model requires use of numerical methods.

4.1 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 such as CATIA and Pro-E. To create a 3D model of valve body with all intricate geometric details, CATIA software is used. The created 3D model of valve body is as shown in fig 4.1.

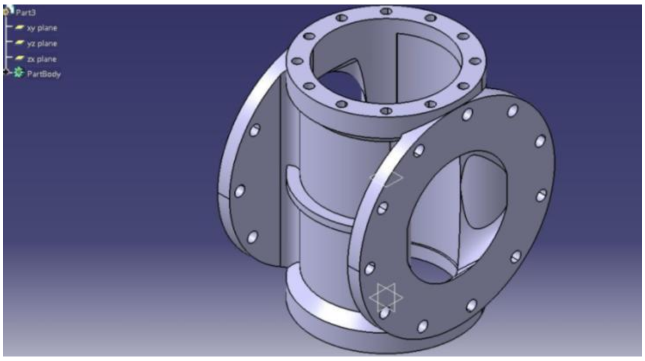

Figure 4.2 – 3D model of valve body

While creating 3D model care has been taken to model it with parametric expression, so as the dimensions changes it will reduce the repetitive time required for modeling. Small steps and champers are eliminated while modeling. The created 3D model is saved in part.igs file format, as this file format is suitable during importing this model for meshing in Hypermesh software.

4.2 Meshing of the 3D valve body model

In simple term meshing means connecting elements with each other. Elements are the building blocks of the finite element analysis. Meshing is carried out by using Hypermesh software. Model is meshed by using SOLID 45element and with 7 element size. Total **175364** elements and **13293** nodes were created after meshing.

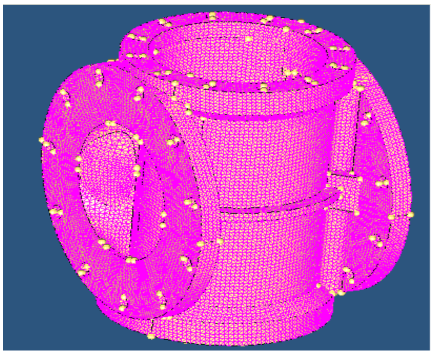

Figure 4.3 - Meshed model

Material properties assigned

After completion of meshing material properties are assigned to meshed model. These properties are listed below.

Material used- ASTM A216 Grade WBC
Young's Modulus- 2.1E5 N/mm^2
Poisons Ratio - 0.28

The maximum principal stress 87.709 N/mm^2 and 131.512 N/mm^2 founds in the rib at 2 MPa and 2.99 MPa internal pressures respectively. While minimum principal stress (2nd principal stress) 34.431 N/mm^2 and 51.626 N/mm^2 found at flange corner for 2 MPa and 2.99 MPa internal pressures respectively. As the internal pressure acts on the internal effective pressurizing area of valve body, results to expand the valve body .Ribs tries to hold the valve body in original position so ribs subjects to heavy tensile stress. As the internal pressure increases stresses in the valve body increases linearly.

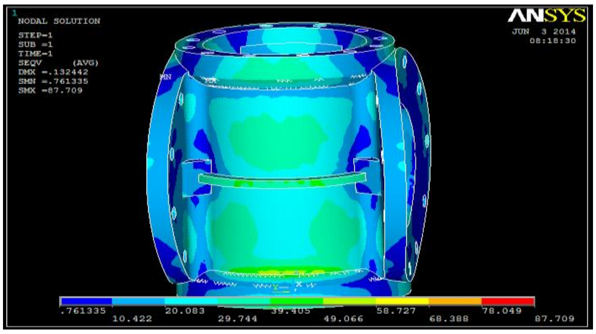

Figure 4.5 - Von-mises stresses for 2 MPa internal pressure applied

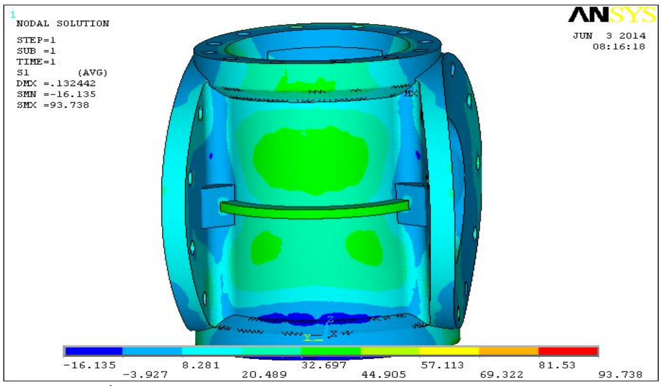

Figure 4.6 - 1st principal stress for 2 MPa internal pressure applied

4.3 FEA results for 2.99 MPa internal pressure applied.

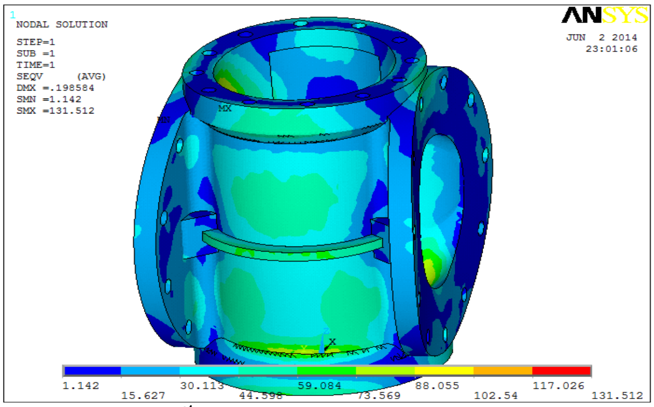

Figure 4.9 - 1st principal stressfor 2.99 MPa internal pressure applied.

4.4 Selection of locations for mounting strain gauge rosettes.

Finite element results shows that vertical rib is subjected to heavy tensile stress. But it is not possible to mount strain gauge rosette exactly on this rib, because thickness of rib is less for mounting of strain gauge rosette. For the convenience strain rosette will be mounted different portions of valve body. Four different locations decided by carefully analyzing the FEA results. Figure 4.10 showing the different four locations selected for mounting of the strain gauge rosettes.

1st and 2nd principal stress results from FEA at these four locations will be compared with experimental stress results at the same point for validation of FEA results. Further to this, FEA and experimental stress analysis comparison will be useful to do reduction of the plug valve casting body.

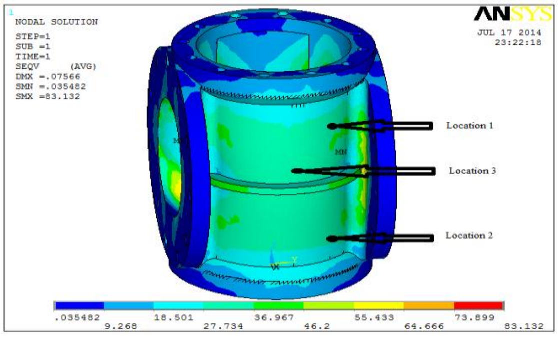

Figure 4.11 - Different locations selected for mounting strain gauge rosette

Summarized principal stress data using FEA

The principal stresses (б1 and б2) at the same points of interest selected for mounting of strain gauge rosette on valve body are found out. The stresses are found at two different cases by applying (2 MPa and 2.99 MPa) internal pressures. Following tables shows 1st and 2nd principal stress results found by FEA.

Location Number	Von Mises stress (N/mm^2)
1	18.91
2	21.01
3	31.2

Table 4.1 - FEA results at 2 MPa internal pressure applied

Location Number	1st principal stress (N/mm^2)
1	31.18
2	33.93
3	43.2

Table 4.2- FEA results at 2.99 MPa internal pressure applied

IV. EXPERIMENTAL STRESS ANALYSIS

4.1 Experimental setup

In this experimental work stain gauge technique is used to calculate stress at a point of interest. One opening made for fitting of relief valve kept open for the purpose of filling coolant in a valve body. Once the body totally filled with coolant that opening also make closed under slandered pressure 290 psi (2Mpa). One plug opening is provided to inlet closing plate, for connecting outlet pipe of the pressurizing unit. Testing has completed as per API 598.

After body test strain gauge rosettes (0-45-90) are mounted on four locations which were decided previously by analyzing the FEA results. Strain readings are taken in two stages at 2MPa and 2.99MPa. Strain developed in each arm of the rosette is noted from display of straingauge indicator. The experimental setup is as shown in the photograph.

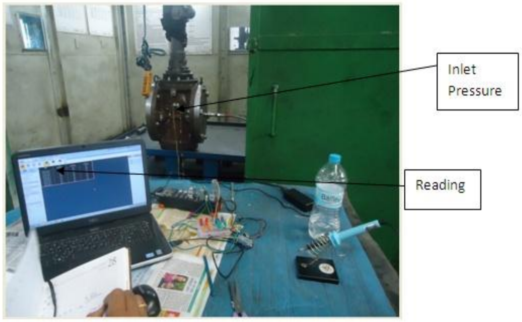

Figure 5.2 (a) Actual Experimental Setup

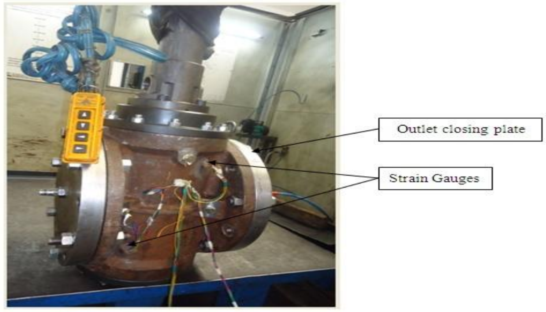

Figure 5.2 (b) Photograph Experimental Setup

4.8 Experimental observations

For each load case the three readings are observed. The three readings are respective to the three axis of the three axis of rosette namely Ɛa, Ɛb, Ɛc.For getting the three axis readings lead wires of the respective directions are to be connected in the circuit to the rear panel of 10 channel balancing and switching unit along with strain indicator. The readings shown by strain indicator are in microstrains. Two sets of the readings are taken at two different load conditions (2 MPa and 2.99 MPa).Finally, the principal strains and principal stresses of the respective points of interest where rosettes mounted are calculated by analytical method. Tables are showing the results.

Rosette Number	Ɛa (µstrain)	Ɛb (µstrain)	Ɛc (µstrain)	б1 (N/mm^2)	б2 (N/mm^2)	Von Mises Stress (N/mm^2)
1	53	61	62	18.26	16.33	17.38
2	61	72	69	20.92	18.19	19.70
3	89	93	98	28.89	27.36	28.16

Table 5.1 - Experimental Results at internal pressure 2 MPa applied.

Rosette Number	Ɛa (µstrain)	Ɛb (µstrain)	Ɛc (µstrain)	б1 (N/mm^2)	б2 (N/mm^2)	Von Mises Stress (N/mm^2)
1	61	72	69	30.94	26.82	29.10
2	102	115	103	32.95	28.72	31.05
3	128	135	142	41.80	39.43	40.66

Table 5.2 - Experimental Results at internal pressure 2.99 MPa applied.

V. COMPARISION BETWEEN FEA AND EXPERIMENTAL ANALYSIS

The values of deviation in 1st and 2nd principal stress, calculated based on experimental result and FEA results, for the valve body having wall thickness 19.5 mm are shown in Table no. 6.1 and 6.2 and at the same point where strain gauge rosettes was mounted. Results clearly show that the maximum deviation in these results is equals to 9.75 % which is allowed. While carrying the experimental validation the readings depends on environmental conditions like temperature difference, measuring instrument sensitivity, human errors, casting defects inbuilt while manufacturing valve body by casting these are some possible reasons for the deviation.

Strain Gauge Mounting location number	FEA results Von Mises Stress (N/mm^2)	Experimental results (N/mm^2)	% Error in (N/mm^2)
1	18.91	17.38	8.09
2	21.01	19.70	6.23
3	31.2	28.16	9.75

Table 6.1 - Comparison of Von Mises Stressstress values at internal applied pressure 2 MPa

Strain Gauge Mounting location number	FEA results Von Mises Stress (N/mm^2)	Experimental results (N/mm^2)	% Error in N/mm^2
1	31.18	29.10	6.68
2	33.93	31.05	8.48
3	43.9	40.66	7.37

Table 6.2 - Comparison of Von Mises stress values at internal applied pressure 2.99 MPa

As the FEA and Experimental results are coming close to each other, different models of valve body by varying wall and rib thickness can be made. FEA of these models will be carried out without affecting original geometry as the 3D parametric modeling is built up in the CATIA.

VI. MODIFICATION OF BODY

6.1 Introduction

The aim of design review is to reduce the cost of the valve body by reducing its weight, without compromising the casing strength. There are many ways of cost reduction

6.2 Steps for themodification of body

- Modeling of the valve body using CATIA.
- Stress analysis of above model using FEA.

  Repetition of above steps for different newly modified models.

6.3 FEA results after reducing wall thickness by 2mm and flange by 2.5 mmincrease neck radius to 180 mm

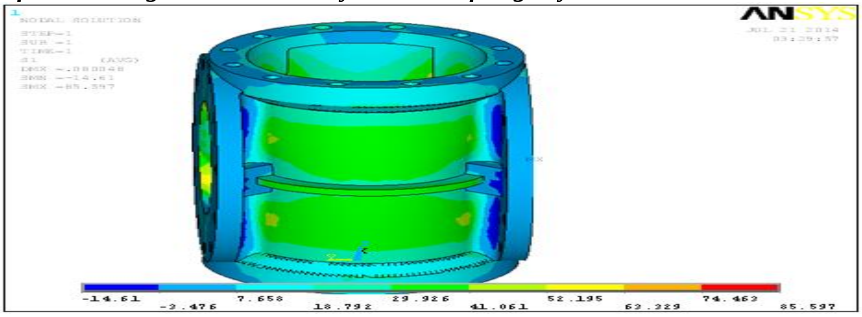

Figure 7.9 - Von Mises Stress stress for FEA results after reducing wall thickness by 2mm and flange by 2.5 mm increase neck radius to 180 mm

VII. RESULTS AND DISCUSSION

8.1 Discussion on Results

FEA results and reduction in weight after modifications are summarized in the following tables

Reducing the wall thickness by 1 mm weight is reduced by 3.34 kg (2.39%), while maximum principal stress level value is 118.485 N/mm^2.Whereas when the wall thickness is reduced by 2 mm, weight is reduced by 5.38 kg by keeping the maximum stress value at 120.174 N/mm^2.Again for another modification, if wall thickness decreased by 2 mm flange by 1 mm, 7.34 kg of weight is reduced by keeping the maximum stress level at 88.112 N/mm^2 which is much lower than the yield stress of the material.

By reducing the wall thickness by 2 mm and flange by 2.5 mm, the weight reduced is 10.52 kg (7.51 %) from original weight, and maximum stresses found equals to 125.187 N/mm^2. While if the wall thickness is reduced by 2 mm and increase neck radius up to 180 mm, 9.15 kg (7.11 %) weight is reduced.

Results by decreasing the wall thickness by 2 and increase neck radius up to 180 mm are better than only reducing the wall thickness. Because Principal stresses are decreased in rib portion as the thickness of rib is increased. And the principal stress value coming is much lower than the yield stress value; hence it is safe for working conditions.Results by decreasing the wall thickness by 2 and increase neck radius up to 180 mm are better than only reducing the wall thickness. Because Principal stresses are decreased in rib portion as the thickness of rib is increased. And the principal stress value coming is much lower than the yield stress value; hence it is safe for working conditions.

Parameter Changed	Von-Mises Stress N/mm^2	Reduction in weight (Kg)	Percentage reduction in weight (Kg)
Reduced wall thickness by 1 mm	84.318	3.34	2.39
Reduced wall thickness by 2 mm	84.318	5.38	3.84
Reduced wall thickness by 2 mm and flange by 1 mm	83.112	7.34	5.24
Reduced wall thickness by 2 mm and flange by 2.5 mm	93.126	10.52	7.51
Reduced wall thickness by 2 and increase neck radius to 180 mm	82.84	9.95	7.11

Table 8.1 - Summarized results after reducing wall thickness in 2 mm steps

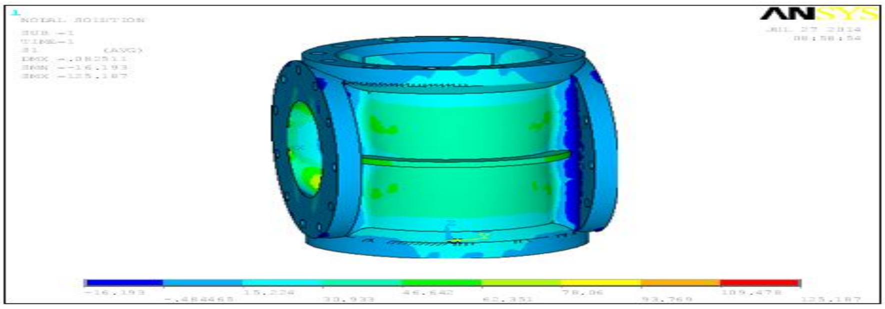

Figure 7.10 - 1st principal stressfor FEA results after reducing wallthickness by 2mm and flange by 2.5 mm increase neck radius to 180 mm

From obtained stress pattern, it is concluded that the maximum 1st principal stress value is 125.187 N/mm^2 founded in the rib. In actual working condition, the maximum 1st principal stress found is 131.512 N/mm^2 at the same location i.e. in the rib portion. The stress contour results interpret that except rib section stresses are very less in other zones of the component.

The new modified valve body results are coming lower than the yield stress value; it shows that newly modified valve body is safe for same working condition. By reducing the wall thickness wall thickness by 2 and increasing neck radius to 180 mm weight is reduced by 9.95 Kg.

VIII. CONCLUSIONS AND FUTURE WORK

9.1 Conclusions

In this dissertation work an attempt has made for weight optimization of plug valve body. Various models are created by changing the design parameters and analyzed these models for better results. Experimental structural strains and stresses measured by actual pressurizing the valve body, and compared it with FEA results. Strain gauge technique gives good results for the measurement of strain and stress at the point of interest.

1. Results of finite element method for the structural analysis of the plug valve body are well in agreement with experimental results, as the deviation is maximum deviation is up to 9.75 % and minimum deviation is up to 6.23 % which is allowable.
2. Eight new different optimized models created by changing the design parameters and analyzed .As there is restriction to change the flange dimensions, wall body thickness and neck dimensions are considered for optimization.
3. Results of decreasing the wall thickness and increasing the neck size are better than only reducing the wall thickness.
4. The best modified model is that, in which wall thickness is reduced wall thickness by 2 mm and increase neck radius to 180 mm reduces 9.95 kg (7.11 %) weight, because maximum stress level is much lower than the yield stress value of the material. FEA results for this optimized model shows that stresses in ribs also decreased because of increased rib thickness.

9.2 Scope for future work

A lot scope is there for the work related to stress analysis. In connection with present dissertation work the following points are to be considered for future work.

1. The same model may analysed by another experimental method i.e. by photo elasticity method & in this method the exact model of plug valve body will be prepared, with the help of photo elastic material which is used for analysis by photo elasticity method. The loading conditions are applied on the same model and one can see the actual point to point stress values on the volute casing. As well as the deflections are visible by naked eyes. Also optimization is easier and immediate results of stress pattern for the optimized component can be seen.
2. Experimental stress analysis of ,newly modified best optimized model ,and comparing experimental stress analysis results with obtained FEA results.
3. In this analysis the effect of fluid temperature is not considered, the effect of fluid temperature could be investigated in future. Temperature distribution and thermal stresses can be found out by using FEA

REFERENCES

[1] **Mona Golbabaei, RouhollahTorabi, Ahmad Nourbakhsh, KaroSedighiani,** *"Failure Detection and Optimization of a Centrifugal-pump Volute Casing",* Proceedings of the SEM Annual Conference June 1-4, 2009, Albuquerque New Mexico USA, Society for Experimental Mechanics Inc.

[2] **DeokarVinayakHindurao, D. S. Chavan,** *"Optimization of 16" Plug Valve Body Using FEA And Experimental Stress Analysis Method",* International Journal of Mechanical Engineering, Vol. No. 1, pp. 79-86.

[3] **Xue Guan Song, Lin Wang, SeokHeumBaek, Young Chul Park,** *"Multidisciplinary optimization of a butterfly valve",* ISA Transactions, 2009, pp. 370-377.

[4] **A. Dorogoy, D. Rittel,** *"Optimum location of a three strain gauge rosette for measuring mixed mode stress intensity factors",* Engineering Fracture Mechanics, 2008, pp. 4127–4139.

[5] **AleksandarPetrovic,** *"Stress Analysis of Cylindrical Pressure Vessels with Loads Applied to the Free End of Nozzle",* International Journal of Pressure Vessels and Piping, 2001, pp. 485-493.

[6] **B. Prabu, N. Rathinam, R. Srinivasan, K.A.S. Naarayen,** *"Finite Element Analysis of Buckling of Thin Cylindrical Shell Subjected to Uniform External Pressure",* Journal of Solid Mechanics Vol. 1, No. 2, 2009, pp. 148-158.

[7] **K. H. Jatkar, Sunil S. Dhanwe,** *"Dynamic Analysis of Single Cylinder Petrol Engine",* International Journal of Engineering Research and Applications, Vol. 3, No. 3, May-Jun 2013, pp. 1177-1183.