International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 12, December-2014

Weight optimization of 12"-150 Class Plug valve Casting Body by Finite Element Analysis

Pradnyawant .K. Parase*
Prof. Laukik B. Raut**
(*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 it is very important to reduce the weight of the plug valve body.**

The stress analysis means determining the stress value on the valve body when internal pressure is applied and optimizing it using finite element analysis. Ultimate aim 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 solutionsof boundary value problems in engineering. Simply stated, a boundaryvalue problem is a mathematical problem in which one or more dependent variablesmust satisfy a differential equation everywhere within a known domain ofindependent variables and satisfy specific conditions on the boundary of thedomain. Boundary value problems are also sometimes called fieldproblems. Thefield is the domain of interest and most often represents a physical structure.

INTRODUCTION

1.1 Introduction to plug valve:

A device that regulates the flow of gases, liquids or loose materials through an aperture, such as a pipe, by opening, closing or obstructing a port or passageway.

Plug Valve:-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 13. The implest 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

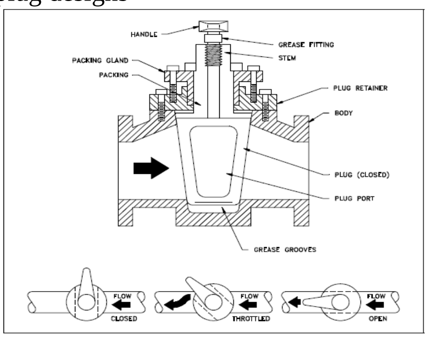

Figure1.1: Plug Valve

Applications of plug valves

- Biofuels Product isolation: Secure sealing of biofuels is critical in protecting the environment. The Accuseal DBB with its verifiable secure shutoff is perfect when process must be contained.
- Multi product manifolds: Some pipeline manifolds need to flow various product reliably and without contaminating one another. This Accuseal BDD valve is used to provide positive shut off and zero leakage to prevent cross contamination.
- Prover loops: In prover loops, the calibration of flow meter requires that every valve in the system must have zero leak rates. Any leak could mean an error in calibration. The Accuseal DBB valve is used to ensure that when the valves in the system are closed, they are leak tight.
- Custody transfer units: Transfer of valuable media relies on accurate measurement of product. The Accuseal BDD provides secure tight shut off ensuring that the transfer is accurate.
- Terminals: Loading and unloading tanker vessels requires positive sealing in order to prevent spillage into the water. The Accuseal BDD provides such positive sealing and is the most reliable in the market.
- Tank farms (Oil Depots): Valves used for tank isolation needs to work reliably with zero leak rate. These valves are also operated frequently. The Accuseal BDD provides a reliable long term high integrity seal designed for frequent use with verifiable zero leak.
- Aviation fueling system: Fuel hydrants at airports need to allow for quick maintenance, repair, leak locating, testing. This requires a valve that can close quickly and positively seal off the relevant section.

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.

1.2 Various materials used for plug valve body

Body material selection is usually based on the pressure, temperature, corrosive properties, and erosive properties of the flow media.

1.2.1 Cast Carbon Steel (ASTM A216 Grade WCC)

WCC is the most popular steel material used for valve bodies in moderate services such as air, saturated or superheated steam, noncorrosive liquids and gases. WCC is not used above 800ºF (427ºC) as the carbon rich phase might be converted to graphite.

Temp. Range = –20 to 800ºF (–29 to 427ºC)

Wall thickness of a valve body Actual Wall Thickness = 19.5mm.

By referring Table no 3 of ASME B 16.34

Minimum thickness required T=9.652 mm

2. LITERATURE REVIEW

2.1 Mona GolbabaeiAsl, et,al [1] discussed technique for optimization of a centrifugal-pump volute casing. FEA is utilized to calculate casing mechanical capability under hydrostatic test. 10-node tetrahedral elements are used for meshing of volute casing and casing cover. From FEA results it is cleared that there is room for optimization by changing the wall-thickness of volute-shaped segment near discharge nozzle. The wall-thickness increase starts from 2 mm at discharge nozzle and gradually ends to zero at the angular position of 90^0. After obtaining the optimized geometry parameters in the simulation, six new volute casings were cast and fabricated.

2.2 PavelMacura, et,al [2] worked on experimental analysis of residual stresses on concrete part of pipelines-welded ball valve. Strain gauge technique is used for measurement of residual stresses both immediately after welding and after pressuring of the ball valve. Released strains were measured by strain gauge methods with use of special rectangular strain gauge rosettes RY 61- 120/S made by Hottinger, with length of strain gauge grids 1.5 mm. It is loaded by overpressure, which is 50% higher than nominal pressure. The product tightness, strength and the residual stresses were tested.

2.3 H. Moustabchir, et,al [3] conducted numerical and experimentalanalysis of stress strain state of pressurized cylindrical shell with external defect. Initially experimental technique with using strain gauges for a cylindrical shell with longitudinal external defects subjected by internal pressure was investigated. Surface deformation near both semielliptical external defects was measured by strain gauges. Two main directions of measuring were chosen, first from defect tip in axial direction and second from defect centre along line which is perpendicular to defect. An appropriate procedure of numerical calculation based on a Finite Element Method (FEM) for assessment of strain near defect carried out

2.4 Xue Guan Song, et,al [4] implemented a new process to meet desired need in valve design that is characterized by complex configuration. They proposed process which includes mainly three parts. First, CFD (Computational Fluid Dynamics) & FEM (Finite Element Method) analysis of a butterfly valve carried out to calculate the pressure loss coefficient & maximum stress in the valve disc. Secondly, topological optimization is carried out to identify basic shape of butterfly valve disc; also size optimization is utilized to determine detailed size based on the result of topology optimization.

2.5 Yong Zhang, et,al[5] discussed a new technique tube hydro-performing over a conventional casting and forging procedures. The method combines genetic algorithm with finite element simulation. Initially on the basis of plastic theory, the internal pressure and axial feed are expressed as a function of stress ratios respectively. The finite element method is then used for solving objective functions.

2.6 Joseph F Dues,et,al [6] discussed method for measurement of stress and strain for optimization of the can to reduce the amount of aluminum required. This is done 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 is measured.

FINITE ELEMENT ANALYSIS

Simple mathematical model can be solved analytically, but more complex model requires use of numerical methods. FEA is one of the numerical methods used to solve complex mathematical problem. The entire solution domain must be discretized into simply shaped sub domain called as elements. ANSYS software is used for the analysis of the valve body, which is based on the FEA method.

3.1Steps in finite element analysis

3.1.1 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 3.1.

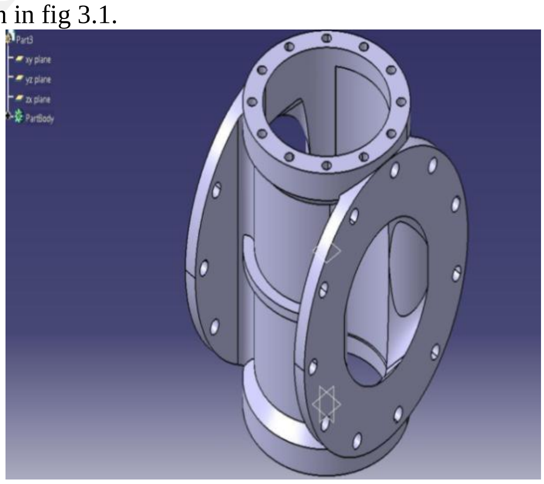

Figure 3.1 – 3D model of valve body

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.

3.1.2 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 - 0.26

3.1.3 Loads and boundary conditions:

Structural loading means applying internal hydraulic pressure to valve body. As per API 598 seat pressure body rest is 2 MPa and 2.99 MPa were applied on the all inner surface of a valve body, which is shown by red colored arrows in the fig. All degree of freedoms of inlet and outlet flanges is restricted and it is shown with the help of green color in the figure 3.2.

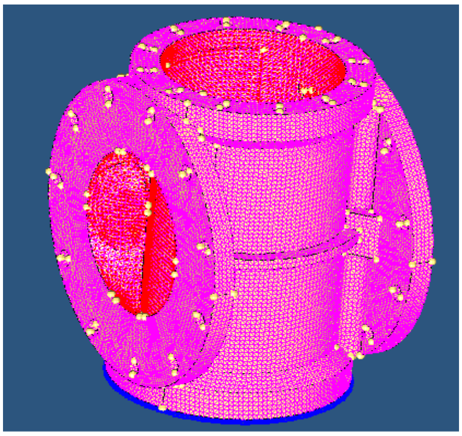

Figure 3.2 - Loads applied and boundary conditions to meshed model

3.1.4 Results and their physical interpretation

After applying material properties and boundary conditions, problem was solved by the ANSYS solver. ANSYS solver formulates the governing structural stress strain equations for each and every element those formulated governing equation are solved for deformation. With this governing equation other quantities such as stresses, strains can be calculated. Stress pattern for two different pressure conditions are observed by using post processing tool. Results are shown with the different color strips of the stresses at the side of figure. Principal stresses, von-mises stresses are the logical checks for structural valve body analysis. After every analysis these logical checks has been cross checked.

Results of FEA For two different internal pressure conditions (i.e. at 2 MPa and 2.99 MPa) are shown in following figures.

3.1.5.1 FEA results for 2 MPa internal pressures applied.

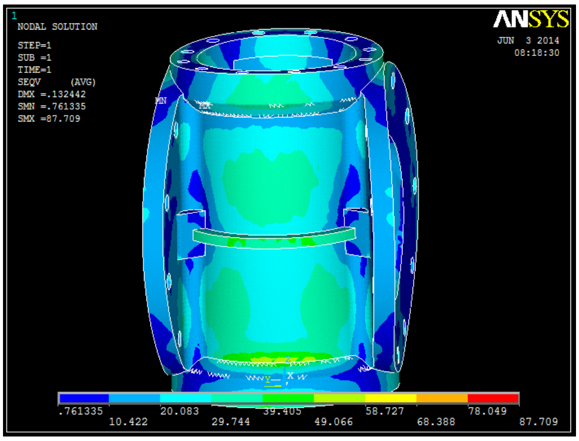

Figure 3.3 - Von-mises stresses

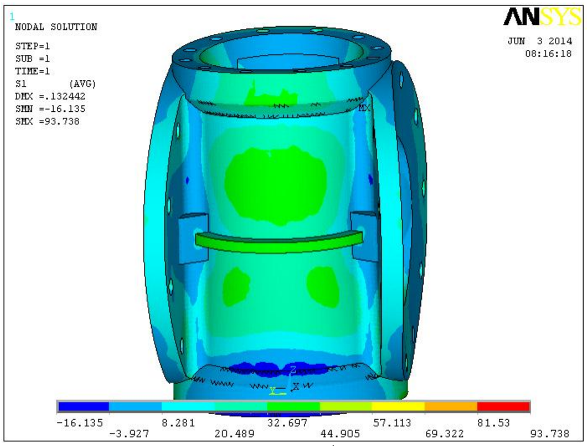

Figure 3.4 - 1st principal stress

4.1.5.2 FEA results for 2.99 MPa internal pressures applied.

The principal stress colored contour plot shows that faint brown color indicates the region of moderately stresses (tensile stress) and blue color indicates the region of low stresses (compressive stresses).While red color indicates high stresses. If the red color pattern as seen in the point form is called as singular stresses. These singular stresses are generated due to loading, boundary condition and geometry. The post processing interpretation is done by simply ignoring these singular stresses and neglecting unrealistic stresses.

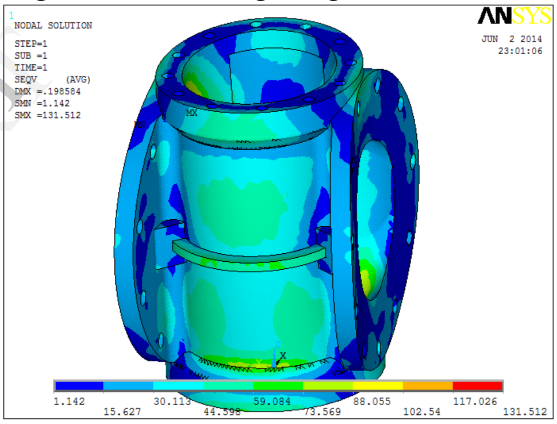

Figure 3.5 - Von-mises stress

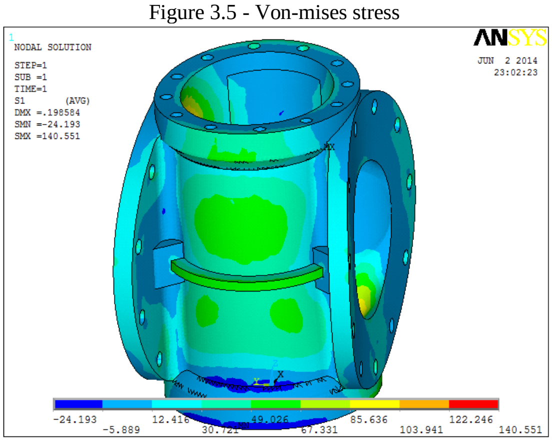

Figure 3.6 - 1st principal stress

` 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 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.

3.2 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.

1st 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 optimization of the plug valve bodygauge rosettes.

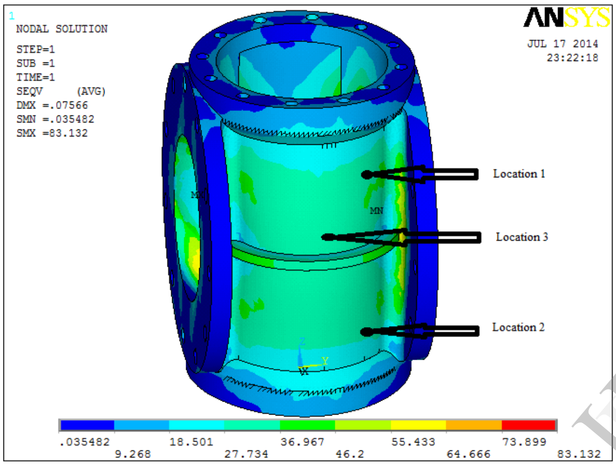

Figure 3.7 - Different locations selected for mounting strain gauge rosette.

3.3 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 principal stress results found by FEA.

Table 3.1 - 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 3.2- FEA results at 2.99 MPa internal pressure applied

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

4. FEA RESULTS

Table 4.1 and 4.2 shows the value of deviation in 1st principal stress, calculated based on experimental result and FEA results, for the valve body having wall thickness 19.5 mm 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
1	18.91
2	21.01

Table 4.1 - Von Mises Stressstress values at internal applied pressure 2 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

Strain Gauge mounting location number	FEA results
	Von Mises Stress N/mm^2
1	31.18
2	33.93

Table 4.2 - Von Mises stress values at internal applied pressure 2.99 MPa

5. DESIGN REVIEW

5.1 Introduction

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

As the 19.5 mm thick valve body shows maximum 1st principal stress up to 131.512 N/mm^2 and the yield stress up to 275 N/ mm^2, hence there is scope to reduce the wall thickness. The exercise has been made for reducing the wall thickness in steps of 1 mm without affecting the original shape and keeping the same material of the valve body.

5.2 Steps for the design review

- Modeling of the valve body using CATIA.
- Stress analysis of above model using FEA.
- Repetition of above steps for different newly designed models.

5.3 Optimization of valve body by reducing the wall thickness

As actual wall thickness is much greater than desired minimum wall thickness required, focus is mainly given on this wall valve thickness.Results obtained for various newly designed models are summarized in table 8.1. These results are helpful while deciding the better optimized model.

5.3.1FEA Results after reducing wall thickness by 2mm and flange by 2.5 mm

From obtained stress pattern, it is concluded that the maximum 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.

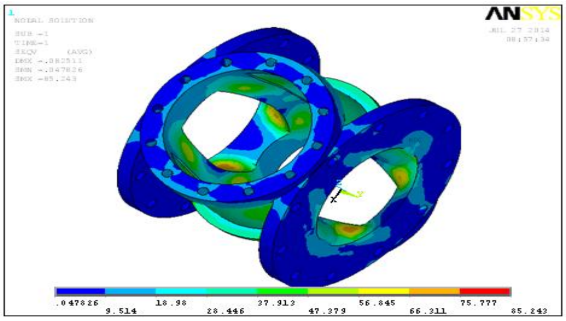

Figure 5.1 - Von Mises Stress stress

The new designed valve body results are coming lower than the yield stress value; it shows that newly designed valve body is safe for same working condition. As compared with results of the valve body by changing only wall thickness by 2 mm & flange by 2.5 mm, weight of valve body is reduced by 10.52 kg.

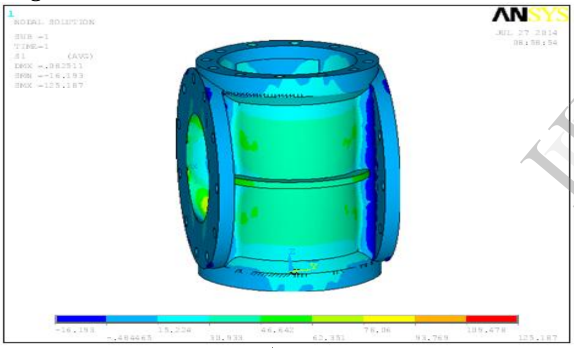

Figure 5.2- 1st principal stress

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

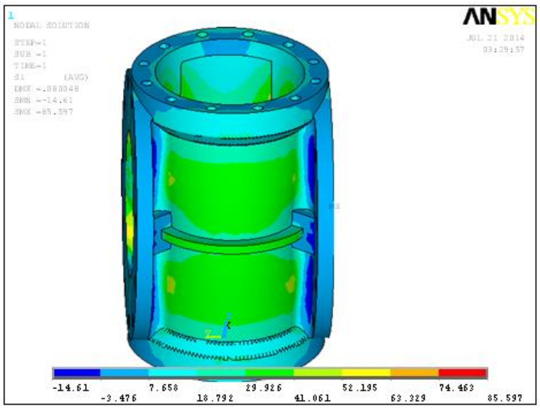

Figure 5.3 - Von Mises Stress stress

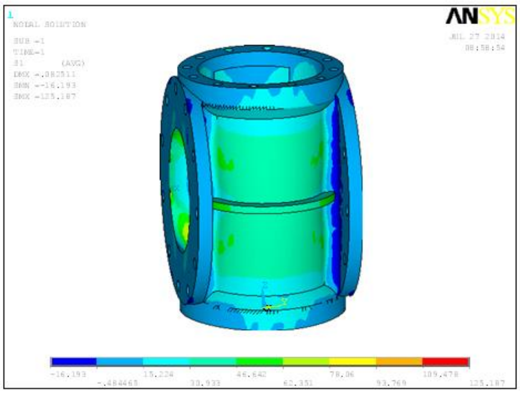

Figure 5.4 - 1st principal stress

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. By reducing the wall thickness wall thickness by 2 and increasing neck radius to 180 mm weight is redused by 9.95 Kg.

6. DISCUSSION ON RESULTS AND CONCLUSION

6.1 Discussion on Results

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

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 to 180 mm, 9.15 1kg (7.11 %) weights 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 potion as the thickness of rib is increased. And the principal stress value coming is much lower than the yield stress value; hence designs are safe for working conditions.

Parameter Changed	Von Mises Stress N/mm^2	Reduction in weight (Kg)	Percentage reduction in weight
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 6.1 - Summarized results after reducing wall thickness in 2 mm steps.

6.2 Conclusion

In this paper 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. .

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 Results of decreasing the wall thickness and increasing the neck size are better than only reducing the wall thickness.

3 The best optimized model is that, in which wall thickness is reduced by 2 mm and reduced wall thickness by 2 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.

6.3 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. Stress analysis 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. 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.

REFERANCE

1. Mona GolbabaeiAsl, RouhollahTorabi, S.AhmadNourbakhsh "Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing",HydraulicMachinary Research Institute, Mechanical Engineering Department, Faculty of Engineering, University of Tehran, P. O. Box: 11155-4563, Tehran, Iran. EBARA Pumps Machinery Company (EPMC), P. O. Box: 15875-7653, Tehran, Iran. Engineering Failure Analysis 16(2009)1996-2003.
2. PavelMacura, Frantisek Fojtik, RadomirHrncar "Experimental Residual Stress Analysis of Welded Ball Valve". Faculty of Mechanical Engineering, VSB-TU Ostrava, Czech Republic. XIX IMEKO World congress, Fundamental and Applied Metrology, September 6-11, 2009, Lisbon, Portuga
3. H. Moustabchir, Z. Azari, S.Hariri, I.Dmytrakh, "Experimental and Numerical study of stress-strain of pressurized cylindrical shells with external defects".Equipe de MecaniqueMateriauxetSystemes (EMMS), Department de Physique Faculte des sciences et Techniques Errachidia, University My Ismail, BP 509 Boutalamine, 52000 Errachidia, Morocco. LaBPS, Universite Paul Verlaine Metz, EcoleNationaled'Ingenieurs de metz, Ile de Saulcy, 57045 Metz, France. Department technologie des polymers et composites and IngenenierieMecanique, Ecole des Douai 941, rue Charles Bourseul, B.P.10838,59508 Douai Cedex, France. KarpenkoPhysico- Mechanical Institute of National Academy of Sciences of Ukraine, 5 Naukava Street, 79601 Lviv, Ukraine.
4. Xue Guan Song, Lin Wang, SeokHeumBaek, Young Chul Park "Multidisciplinary optimization of a butterfly valve", Department of Mechanical Engineering,Dong-A University, Busan, 604-714, Republic of Korea. ISA Transactions 48(2009)370-377, Engineering Failure Analysis
5. Yong Zhang, Shengdun Zhao, Zhiyuan Zhang "Optimization for the forming process parameters of thin-walled valve shell", School of mechanical engineering, Xi'an Jiao Tong University, Xi'an 710049, China. School of Mechanical and Electrical Engineering, Inner Mongolia Agriculture.