

Design and FSI (Fluid Structure Interaction) Analysis of Globe Valve- Part I: CFD Modelling

#¹Pratik.P.Nagare, #² Harshal.A.Chavan

¹pratik.nagare8123@gmail.com

²chavanharshal@gmail.com



¹²Department of Mechanical Engineering, MET's BKC IOE, SPPU, Nashik-422003, India

ABSTRACT

The work presented in this paper deals with fluid structure interaction (FSI), one of the emerging areas of numerical simulation and calculation. FSI occurs when the flow of fluid influences the properties of a structure or vice versa. Flow coefficient (C_V) of the 1.5" 600 class globe valve has been increased at full opening of the valve followed by performing FSI analysis at various positions of the valve plug opening HYPERMESH has been used as a pre-processing tool for creating the whole computational domain and volume mesh. ANSYS CFX is used as a solver to extract the C_V of the valve and fluid pressure distribution for the various positions of the valve plug opening. This pressure distribution from Ansys CFX is been imported to Ansys workbench static structure module as input boundary conditions for the valve structure with fixed at both ends and solved for extracting stresses and deformation in order to see the design capability for the respective valve opening conditions. Here, the analysis has been approached using the partitioned method in one-way coupling have been applied to simulate the above scenario. The two solvers (ANSYS CFX & Mechanical) are coupled (exchange of data) using system coupling in ANSYS Workbench. The validation of the new C_V obtained in CFX by increasing the flow area would be done experimentally. Also, the FSI result shows that the valve is safe for the pressure drop generated by the increased C_V , and it's not necessary to be strengthened anywhere.

Keywords— C_V , FSI, One-way coupling, Pre-processing

ARTICLE INFO

Article History

Received : 18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

Fluid structure interaction (FSI) is a multi-physics phenomenon which occurs in a system where flow of a fluid causes a solid structure to deform which, in turn, changes the boundary condition of a fluid system. This can also happen the other way around where the structure makes the fluid flow properties to change. This kind of interaction occurs in many natural phenomena and man-made engineering systems. It becomes a crucial consideration in the design and analysis of various engineering systems. For instance FSI simulations were conducted to verify the performance of the butterfly valve [1], also for analysis of cryogenic ball valves [2], design and analysis of globe

valves and in many biomedical applications. For the past ten years, the simulations of multi-physics problems have become more important in the field of numerical simulations and analyses. The monolithic approach requires a code developed for this particular combination of physical problems whereas the partitioned approach preserves software modularity because an existing flow solver and structural solver are coupled. Moreover, the partitioned approach facilitates solution of the flow equations and the structural equations with different, possibly more efficient techniques which have been developed specifically for either flow equations or structural equations. On the other hand, development of stable and accurate coupling algorithm is required in partitioned simulations. In addition,

the treatment of meshes introduces another classification of FSI analysis. In order to solve such interaction problems, structure and fluid models i.e. equations which describe fluid dynamics and structural mechanics have to be coupled. Although fluid and solid solvers can be used to solve the respective domains, coupling i.e. interchange of results has been considered as one of the challenging tasks due to nonlinear nature of the fluid solid interface. But, technical advancements in the fields of computational fluid dynamics (CFD), computational structural mechanics (CSM) and numerical algorithms have made the numerical FSI analysis more realistic to be performed in a reasonable time frame. In recent times, much commercial software's are being developed and established to simulate FSI problems. Some of the companies like ANSYS, ADINA, COMSOL and CD-adapco provide efficient multi-physics software's with versatile features.

A. Problem Statement

In recent years, the valve manufacturers immersed their attention towards developing high performance globe valve designs to expiate the problems caused by the conventional globe. Control valve sizing is based on flow coefficient C_V calculation. Flow coefficient C_V calculation is made for required flow rate and related pressure drop in control valve. With flow coefficient C_V calculated, size of control valve can be selected, or two control valves from different manufacturers can be compared in terms of flow capacity for certain pressure drop and the same control valve size. Flow coefficient C_V calculation is based on the relation between pressure drop and flow rate in control valve which is for complete turbulent flow following power law where flow coefficient C_V is the proportional constant. Emblematic problems faced in the industry with typical Globe valves are higher valve torque due to the high thrust force acting on the disc, difficult manual operation, stem bending problems. Problem discussed in this dissertation is about improving the C_V of globe valve and the verification of whether the valve could work safely at these different conditions or not.

B. Objectives

By taking the above purposes in to account, the objectives of the project are formulated as follows:

- To increase the C_V of the 1.5" 600 class Globe valve by variation of flow geometry at full opening of the valve
- To perform FSI Analysis of the Globe valve.

C. Methodology

The methodology adopted would be studying & identifying the design of the existing valve, carrying out various geometric iterations and increasing the flow area, at the throat of the valve by building the solid model in CATIA. Flow analysis would be carried out by CFD (Computational Fluid Dynamics) software i.e. ANSYS CFX. The output of pressure distribution from CFD software will be imported as an input loading for stress analysis on the valve. Stress analysis would be done in ANSYS static structural module. Manufacturing the best output design prototype and testing of modified valve to find C_V , comparison of CFD results, and testing results along with the analytical results. Only the critical areas of concern would be studied for the work and suitable recommendation can be found out while concluding the work. Practicality of the recommended solution pertaining to the cost and ease of

deployment would be considered while suggesting the variants for design.

II. FLOW COEFFICIENT C_V

Control valve sizing is based on flow coefficient C_V calculation. Flow coefficient C_V calculation is made for required flow rate and related pressure drop in control valve. With flow coefficient C_V calculated, size of control valve can be selected, or two control valves from different manufacturers can be compared in terms of flow capacity for certain pressure drop and the same control valve size. Flow coefficient C_V calculation is based on the relation between pressure drop and flow rate in control valve which is for complete turbulent flow following power law where flow coefficient C_V is the proportional constant. Flow coefficient C_V is determined experimentally by control valve manufacturers. Flow coefficient of control valve C_V is expressed as the flow rate of water in gpm u.s. (m^3/h) for a pressure drop of 1 psi (1 bar) across a flow passage (flow coefficient: C_V -imperial, K_v -metric). Control valves sizing calculator you can use to calculate maximum flow rate through control valve for given pressure drop and known flow coefficient of control valve C_V . The flow coefficient C_V is a measure of the valve capacity. It is given by the ISA standard S75.01 [3] for incompressible, fully turbulent, noncavitating, and nonflashing flow as

$$C_V = 1.156Q\sqrt{\frac{sg}{\Delta P}}$$

The old valve C_V that is to be improved is found out to be 9.25.

III. VALVE FLOW GEOMETRY

There are various types of modelling software available like CATIA, Pro-E, and Solid Works. Here, 3D model of globe valve is prepared in CATIA because in ANSYS it's very difficult to model the part. Also for increasing the C_V of the globe valve at full opening only the flow geometry of the valve is considered the figures below show the old flow geometry of the valve and the modified flow geometry of the valve. The C_V of the valve is dependent on the flow geometry of the valve hence for increasing the flow geometry the old geometry has been modified by providing smooth area

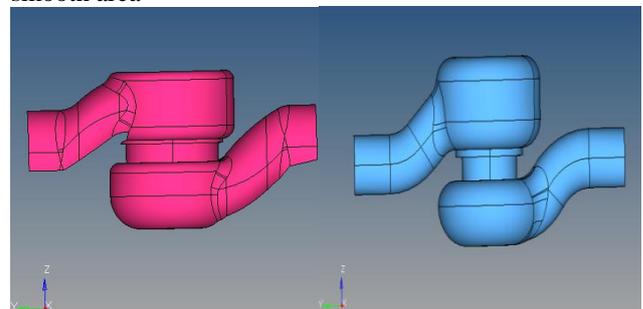


Fig.1.Old Design vs. Modified Design

IV. CFD MODELLING

Initially, flow geometry of the Valve at 100% opening of the plug is extracted. HYPERMESH is used as a pre-processor for extracting the geometric model.

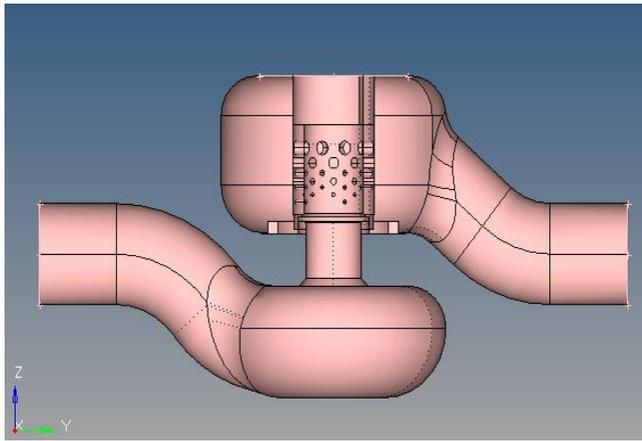


Fig.2. Extracted Flow Geometry for CFD Meshing (100%)

The volume mesh of fluid domain is formed using HYPERMESH. In the case of the CFD mesh, the surface mesh is first created using triangular elements, which is then used to create a volume mesh. The volume mesh is made up of tetrahedral cells, belonging to the category of unstructured mesh. The reason for not using hexahedral cells is due to complex flow geometry. The volume mesh of the fluid domain also includes the prism layers around the cylinder's wall to resolve the boundary layer more efficiently than what can be achieved with tetrahedral cells. Prism layers are made up of cells which are almost perpendicular to the surface of the cylinder. This is used for better resolution of solution normal to the surface of the cylinder. The spacing of prism layers is an important factor in order to get a good solution, apart from this, emphasis must be given to the parameters like initial height of the layer, number of layers and growth ratio of prism layers to capture the boundary layer effect in realistic manner.

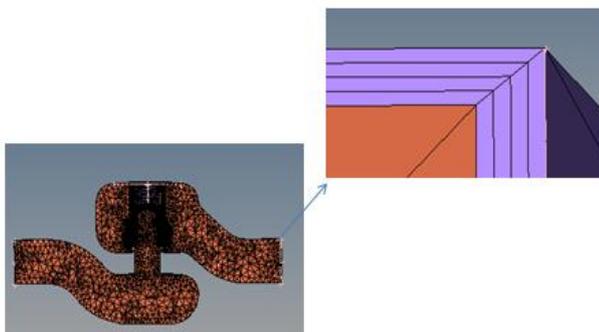


Fig.3. Prism Layers on the outer wall of the flow geometry

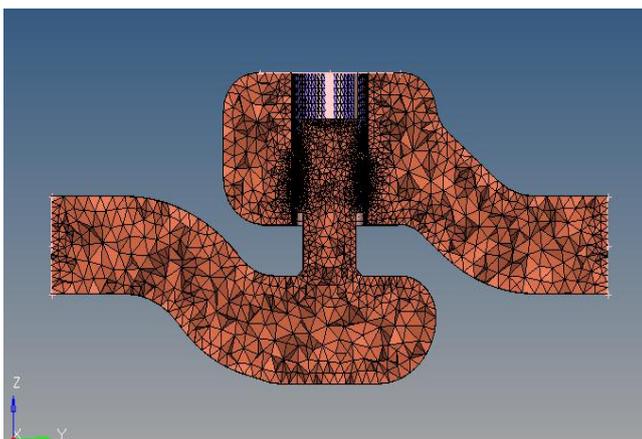


Fig.4. Cross sectional view of fluid domain mesh.

A. CFD Simulation

The size of the large valve is 38mm in diameter and it is manufactured from cast steel. To get a better result, the CFD model of butterfly valve is created at a 1:1 scale. The fluid, which was modeled as water, is given a uniform velocity of 20m/s at the inlet and opening condition at the outlet. The fluid model is comprised of water hence properties taken are density-998.2 (kg/m³) and dynamic viscosity-1.003e-3 (kg/ms). The main objective of this CFD simulation is that to predict the C_v of the modified geometry of the valve. Pressure differences generated in the valve along with the output discharge obtained are obtained by defining new expressions for the above. Area Average pressure at the inlet subtracted by the Area average pressure at outlet and mass flow at the outlet these expressions are used for finding out the pressure difference and the output discharge. The meshed model obtained from HYPERMESH is exported as a .cas file and then imported in ANSYS CFX and the specific boundary conditions such as inlet, outlet and wall are given. The below figure gives a description of the file imported and the boundary conditions inlet and outlet defined.

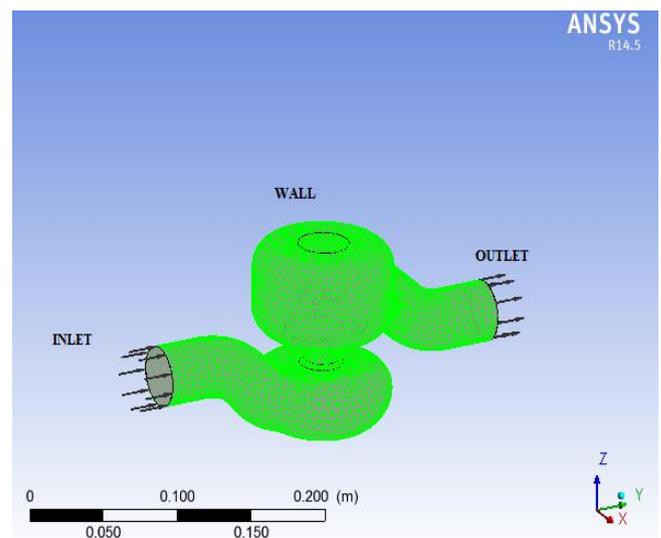


Fig.5. Boundary conditions in ANSYS CFX

All solid boundaries were represented using no slip velocity conditions and log wall turbulence conditions. Inlet conditions were represented by uniform velocity sufficient to provide the required large flow. Turbulence intensity was set to 10 percent at the inlet. All conservation equations are discretized in ANSYS CFX using a finite volume formulation with second order spatial accuracy. In areas having large flow velocities and cell sizes, a proportional upwinding scheme is used having first order spatial accuracy whenever full upwinding is required. The effect of the reduced accuracy with upwinding would be evident during the grid resolution studies. Since there was no effect of grid refinement seen, the upwinding did not appear to affect the solutions. Continuity is satisfied using a SIMPLE semi implicit pressure linked equations algorithm. Normalized residuals were used for the convergence criteria, which was set at three orders of magnitude.

V. FLUID STRUCTURE INTERACTION MODELLING

The 10 – node tetragonal elements (SOLID 187) were used for meshing of Valve Assembly which included the body, plug, orifice, bush and seal. Finite element mesh was generated using tetragonal elements with element length of 8mm.SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has quadratic displacement behaviour and is well suited to model irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.The whole HYPERMESH finite element mesh consists of 477312 tetra elements. The quality checks maintained for the valve tetra mesh were min tria angle,max tria angle,minimum element length and tetra collapse for tetragonal element.

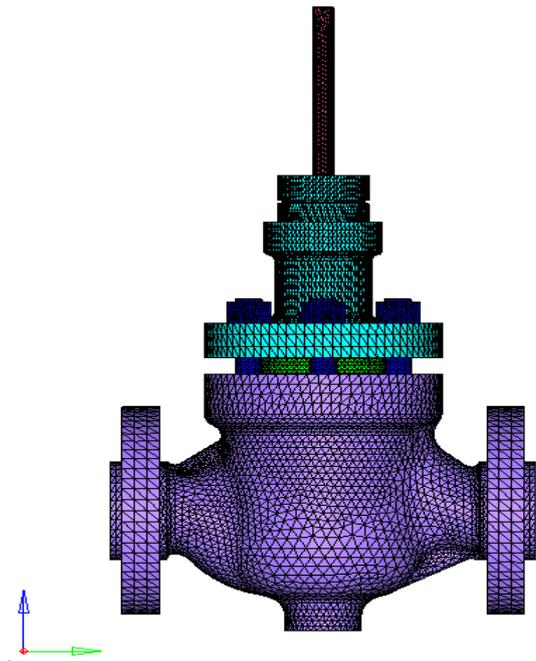


Fig.6.Strutural meshing for ANSYS

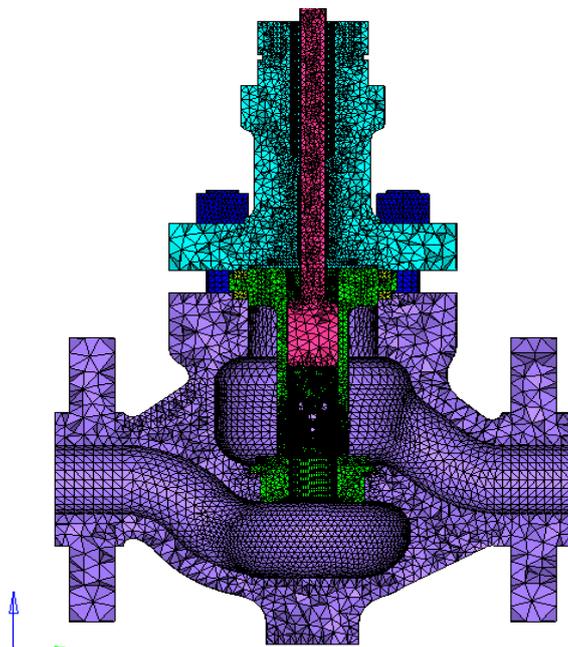


Fig.7.Cross sectional view of Tetra Meshing

A.ANSYS Simulation

The material of valve body is ASTM WCB. This type of material is usually used for pump impellers, bearings, gears, bushings, valve and valve seats. It has a good wear resistance and heat treatable. The tensile yield strength of this material is 280MPa; the tensile ultimate strength is 570MPa; the Young’s Modulus is 210GPa, and the Poisson’s Ration is 0.29.The other parts of the valve such as orifice and plug are made of SS316 having tensile ultimate strength is 579MPa;tensile yield strength; 290MPa the Young’s Modulus is 193GPa, and the Poisson’s Ratio is 0.3.

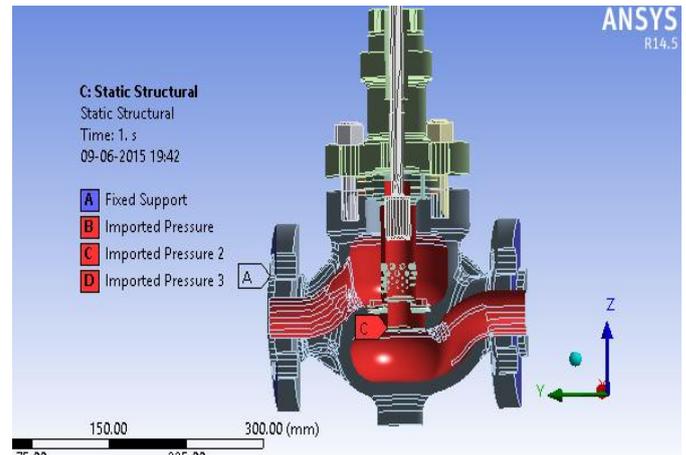


Fig.8.Boundary Conditions for FSI analysis.

The boundary condition for this simulation is that the valve is fixed at its both the flanges and the pressure difference generated inside the valve calculated by ANSYS CFX is imported by using system coupling. This imported pressure is then solved for structural analysis for the deformation and the stress valves as shown in above figure. The imported pressure details can be seen clearly in the below figure.

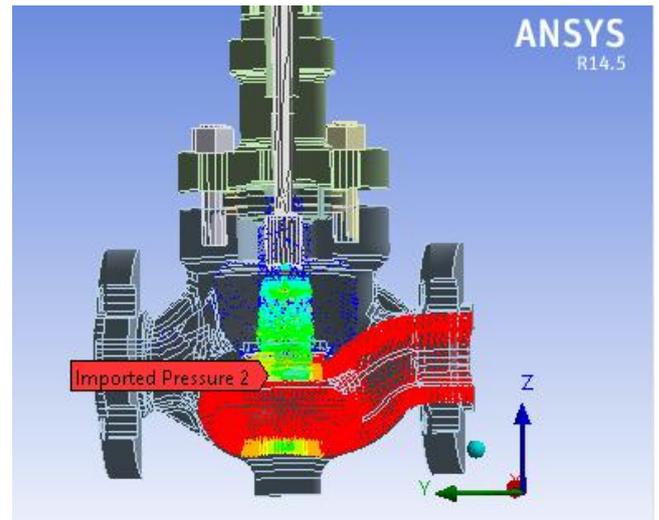


Fig.9.Imported pressure results from ANSYS CFX

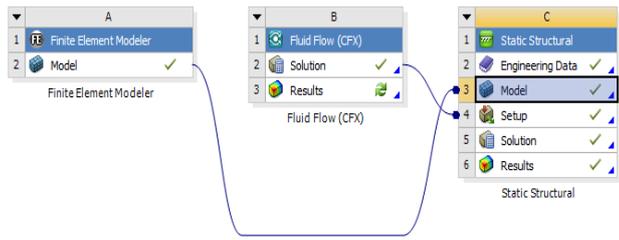


Fig.10. Analysis coupling in ANSYS Workbench.

The above figure depicts the FSI analysis setup of this project work using System coupling with ANSYS CFX and ANSYS mechanical as numerical solvers. The result file of ANSYS CFX is imported as a boundary condition to ANSYS Mechanical as shown also Finite element modeller is used to transfer the Structural mesh into ANSYS created in HYPERMESH.

VI. RESULTS

A. CFD Results

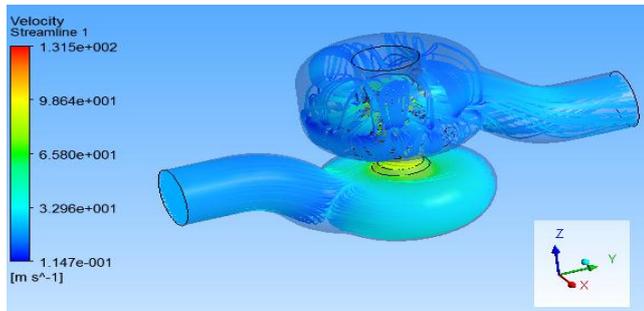


Fig.11. Velocity Streamline

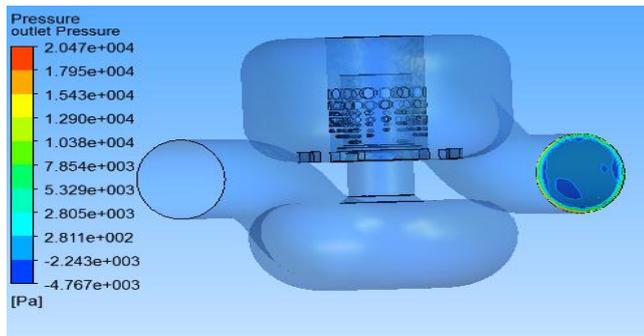


Fig.12. Pressure Contour (Inlet)

Fig.13. Pressure Contour (Outlet)

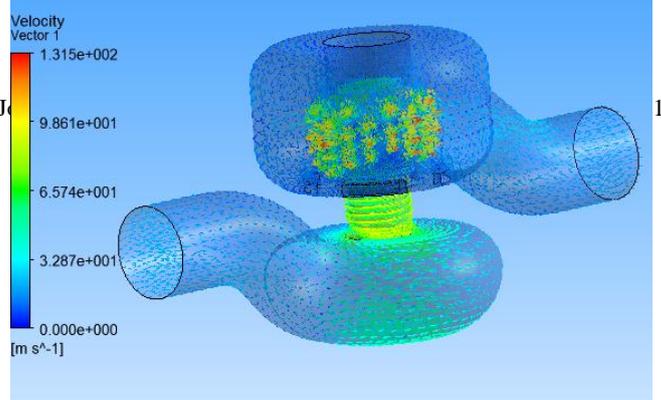
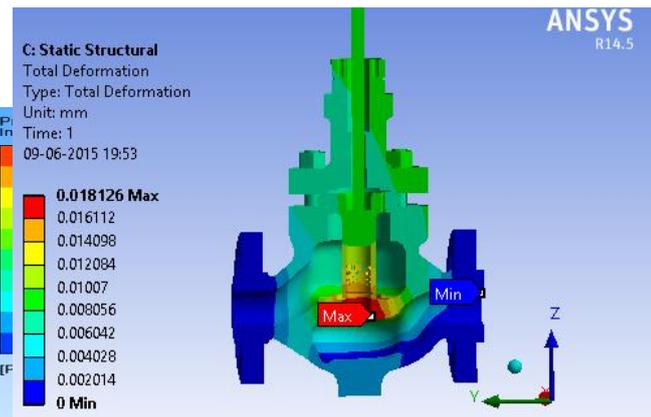


Fig.14. Vector Plot

The CFX results shown in figure 11 shows the velocity streamline for the valve, figure 12 and 13 shows the pressure contour for inlet and outlet. Velocity vector for the valve is shown in figure 14. Figure 11 displays streamlines for valves. The valve is shown at opening of 100 percent. In this case the flow accelerates through the plug and seat region, and then issues downstream in the form of a wall jet through the orifice holes. In addition, a large recirculation region develops on the downstream side of the seat. Figure 12 and 13 displays the pressure contours for the inlet and outlet of the valves. The pressure decreases in the downstream direction with the largest pressure gradients occurring in the plug and seat region.

B. Pressure Difference and outlet discharge

The expressions defined for obtaining the pressure difference (areaAve(p)@inlet-areaAve(p)@outlet) and for output discharge (massFlow()@outlet) through ANSYS CFX were defined. The values obtained were pressure difference of 9.71112e+006 Pa and discharge -22.4687 kg/s. Substituting these values in the above formula of C_v in section II we have the new C_v of the valve to be 10.96.

C. FSI Results

The pressure difference generated for the inlet velocity of 20m/s when valve is fully opened and the fastest water flowing through the valve is imported for performing the FSI analysis. The results of analysis show that the valve is safe under the pressure generated from the new flow geometry. The maximum deformation in the whole valve assembly is found out to be 0.018mm and the stress generated is 97.345Mpa which is less than the yield strength of the material as seen in figure 15 & 16.

Fig.15. Deformation diagram of the Globe Valve

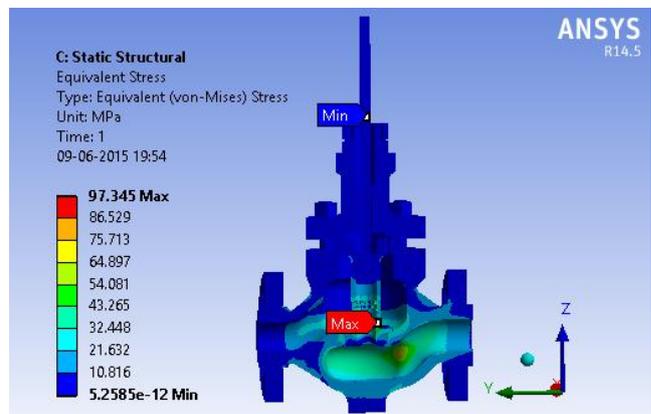


Fig.16. Stress diagram of the Globe Valve

Also the corresponding stress and deformation in the valve body is shown in the figure. The maximum stress in the valve body is found to be 67.93Mpa which is less than the yield strength of the material shown in figure 18.

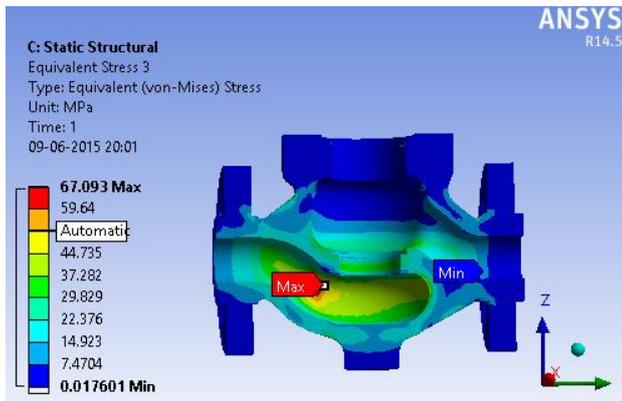


Fig.17.Stress diagram of the Globe Valve (Body)

The max stress occurs at the region of inlet of the valve body a huge amount of pressure generated along with the high velocity of the water flowing through it. The maximum deformation occurs at the orifice from where the water gushes out at a high velocity as seen in figure 19

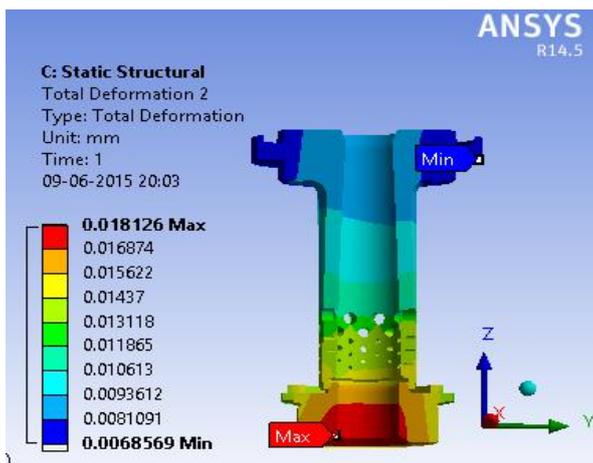


Fig.18.Stress diagram of the Globe Valve (Orifice)

VII. CONCLUSION

In this study, Fluid structure interaction analysis of the valve has been performed by importing the pressure difference developed due to fluid and then importing these pressure values for structural analysis. The results of FSI analysis show that the high pressure developed inside the valve due to the modified geometry of the valve doesn't have any serious effect on the valve body. The new flow geometry results in increase in CV of the valve i.e. 10.96 at full opening condition of the Valve. Also it is found that the maximum stress value 97.345Mpa is below yield stress of material. So the valve is very safe under working condition hence the new flow geometry can be adopted.

ACKNOWLEDGMENT

The author wish to acknowledge Deputy Manager EMET controls Pvt Ltd. Nashik Department of Design & Development for their support for this research

REFERENCES

- [1] Xue Guan Song, Lin Wang And Young Chul Park, 2009 "Fluid And Structural Analysis Of A Large Diameter Butterfly Valve", Journal Of Advanced Manufacturing Systems, Vol. 8, No. 1, pp: 81-88.
- [2] Dong-Soo Kim, Myoung-Sub Kim, 2008. "Analysis And Design Of Cryogenic Ball Valve", Proceedings of The 7th JFPS International Symposium On Fluid Power, pp: 397-402
- [3] "Control Valve Sizing Equations," ISA-S75.01-1997, Research Triangle Park, North Carolina: Instrument Society of America.
- [4] G.Tamizharasi And S.Kathiresan, 2013. "CFD Analysis Of A Butterfly Valve In A Compressible Fluid", Middle-East Journal Of Scientific Research 15, pp: 1834-1839.
- [5] Arun Azad, Deepak Baranwal, Rajeev Arya, Nilesh Diwakar, 2014. "Flow Analysis Of Butterfly Valve Using CFD", IJMER, Vol. 4, pp: 50-56.
- [6] Prof. Koli G.C., Prof. Katkar A.A. "FSI Approach Study & Analysis", IOSR Journal Of Mechanical And Civil Engineering (IOSR-JMCE), pp: 09-15.
- [7] Xue Guan Song, Young Chul Park, 2007. "Numerical Analysis of Butterfly Valve-Prediction Of Flow Coefficient and Hydrodynamic Torque Coefficient", Proceedings Of The World Congress On Engineering And Computer Science.
- [8] A. Beune, J.G.M. Kuerten, M.P.C. van Heumen, 2012, "CFD analysis with fluid-structure interaction of opening high-pressure safety valves", Computers & Fluids 64, pp: 108-116.
- [9] Himadri Chattopadhyay, Arindam Kundu, 2012," Analysis of flow structure inside a spool type pressure regulating valve", Energy Conversion and Management 53, pp: 196-204.
- [10] Bing Xu, Ruqi Ding, Junhui Zhang, 2014, "Modelling and dynamic characteristics analysis on a three-stage fast-response and large-flow directional valve", Energy Conversion and Management 79, pp: 187-199.
- Jin-yuan Qian, Lin Wei, 2014, "CFD analysis on the dynamic flow characteristics of the pilot-control globe valve", Energy Conversion and Management 87 220-226.
- [11] Ryo Morita, Michitsugu Mori, 2007, "CFD Simulations and Experiments of Flow Fluctuations Around a Steam Control Valve", Transactions of the ASME, Vol 129, pp: 48-54.
- [12] James A. Davis, Mike Stewart, 2007, "Predicting Globe Control Valve Performance—Part I: CFD Modelling", Vol 124, pp: 772-777.
- [13] James A. Davis, Mike Stewart, 2007, "Predicting Globe Control Valve Performance—Part II: Experimental Verification", Vol 124, pp: 778-783.