*IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*
*ISSN (e):* 2278-1684, *ISSN (p):* 2320–334X, *PP: 63-71*
*www.iosrjournals.org*

# Design and Analysis of Globe Valve as Control Valve Using CFD Software

V. J. Sonawane[1], T. J. Rane, A.D. Monde, R.V. Vajarinkar, P. C. Gawade
[1] *(Mechanical Engineering Department, M.E.S. College of Engineering, Pune, India)*

***ABSTRACT :*** *Globe valves are commonly used as fluid flow control equipment's in many engineering applications. Thus it's more and more essential to know the flow characteristic inside the valve. Due to the fast progress of the flow simulation and numerical technique, it becomes possible to observe the flows inside a valve and to estimate the performance of a valve. This paper presents the modeling and simulation of the globe valves. The flow system with globe valves is complex structure and has non-linear characteristics, because the construction and the hydraulic phenomena are associated of globe valves. In this paper, three-dimensional CFD simulations were conducted to observe the flow patterns and to measure valve flow coefficient when globe valve with different flow rate and constant pressure drop across the valve were used in a valve system. Furthermore, the results of the three-dimensional analysis can be used in the design of low noise and high efficiency valve for industry.*

***Keywords –*** *Globe valves, CFD, Valve coefficient*

## I. INTRODUCTION

A control valve is a mechanical device that controls the flow of fluid and pressure within a system or process. A control valve controls system or process fluid flow and pressure by performing different functions like stopping and starting fluid flow, varying (throttling) the amount of fluid flow, controlling the direction of fluid flow, regulating downstream system or process pressure, relieving component or piping over pressure .There are many valve designs and types that satisfy one or more of the functions identified above. A multitude of valve types and designs safely accommodate a wide variety of industrial applications. Eight different models to represent the effect of friction in control valves are presented in [1]. The physical models, both static and dynamic, have the same structure. The models are implemented in SIMULINK/MATLAB and compared, using different friction coefficients and input signals, whereas the details of design of stop valves which are commonly used as fluid flow control equipment in many engineering applications is performed [2]. Recently, Computational Fluid Dynamics (CFD) has been experiencing rapid advances due to both computer technology progress and efficient algorithms that have been developed to solve the Navier-Stokes (N-S) equations used in the flow analysis around ship hulls, the work contributed to the numerical solution of the viscous flow around ship-like bodies is discussed in [3]. An experimental and numerical of a three-dimensional, complex geometry, control valve was performed for model validation and improved understanding of valve flow features is discussed in [4]. The compressible air flow in a typical puffer chamber with moving contact between fixed electrodes has been studied using computational fluid dynamics techniques in [5]. Work to reduce the stiffness of the damper, so that the damper can withstand within the required constraints is related in [6]

## II. EXISTING PROBLEM IN GLOBE VALVE

In recent years, the valve manufacturers across the world focused their attention towards developing high performance globe valve designs to outweigh the problems caused by the conventional globe. Typical problems faced in the industry with conventional Globe valves are; higher valve torque due to the high thrust force acting on the disc, difficult manual operation, stem bending problems more prominent in stainless steel material, the high frictional forces at the stem threads and yoke sleeve collar faces leads to high torque and shortens valve life, rotating stem design deteriorates the packing performance at fewer cycles, improper selection of stem and yoke-sleeve material resulting in galling, stem binding due to galling with the mating components, gland packing leaks –not meeting fugitive emission requirements, galling of flange bolts and gland packing eyebolts at low temperature, threaded seat ring design leaks at high pressure applications**.** Problem discussed in this paper is about difficulty in manual operation and controlling flow of fluid. Steps carried out in order to overcome the problem: Identification of problem, Problem statement, Solid modeling, Flow analysis using CFD software to find Cv, selection of appropriate model from CFD results, manufacturing of model, testing of actual valve to find

Cv, comparison of CFD values and testing values. Valve types are used to describe the mechanical characteristics and geometry (Ex/ gate, ball, globe valves). We will use valve control to refer to how the valve travel or stroke (openness) relates to the flow. Before that we have to decide valve control to be used. Here are some rules for selection of control valve as shown in Fig.1.

1. Equal Percentage (most commonly used valve control)
   - Used in processes where large changes in pressure drop are expected
   - Used in processes where a small percentage of the total pressure drop is permitted by the valve
   - Used in temperature and pressure control loops
2. Linear
   - Used in liquid level or flow loops
   - Used in systems where the pressure drop across the valve is expected to remain fairly constant (i.e. steady state systems)
3. Quick Opening
   - Used for frequent on-off service
   - Used for processes where "instantly" large flow is needed (i.e. safety systems or Cooling water systems)

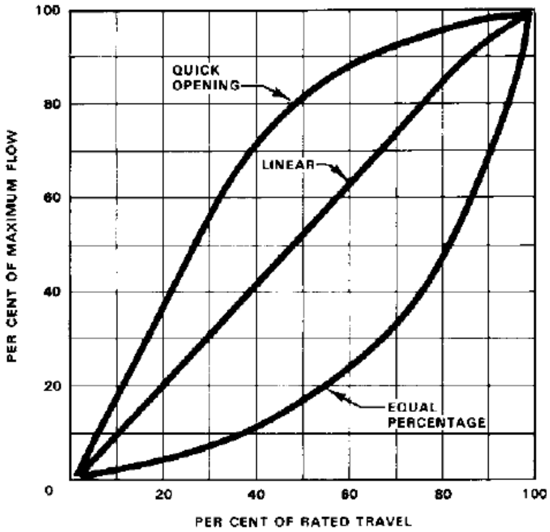


Fig. 1 Valve characteristics curves

### Flow coefficient $C_v$

The flow coefficient of a device is a relative measure of its efficiency at allowing fluid flow. It describes the relationship between the pressure drop across an orifice, valve or other assembly and the corresponding flow rate

$$Cv = F * \sqrt{\frac{SG}{\Delta P}}$$

Mathematically the flow coefficient can be expressed as:

Where;

$C_v$ = Flow coefficient or flow capacity rating of valve.
F = Rate of flow (US gallons per minute).
SG = Specific gravity of fluid (Water = 1).
$\Delta P$ = Pressure drop across valve (psi).

In more practical terms, the flow coefficient $C_v$ is the volume (in US gallons) of water at 60°F that will flow per minute through a valve with a pressure drop of 1 psi across the valve. The use of the flow coefficient offers a standard method of comparing valve capacities and sizing valves for specific applications that is widely accepted by industry.

## III. PROBLEM DEFINITION AND OBJECTIVE

In this work, problem presented is of modifying the existing plug and seat arrangement of the globe valve, such that it should control the flow of fluid up to maximum permissible lift ranging from 2mm to 16mm to obtain the optimum range of fluid flow. The current plug and seat arrangement of globe valve is according to quick opening characteristics. For small lift of plug gives large flow. The main objective of this work is to analyse the plug and seat arrangement using Computational Fluid Dynamics to obtain proper flow control of fluid in given range of lift of plug. The inputs given are inlet pressure of about 40 bar, controlling discharge is 15m3/hr, and total pressure drop across the valve is 1 bar.

### Solid Modelling of plug and seat

To perform CFD analysis of any component, the solid model of the same is essential. It is also called body in white. Fig. 2 and Fig. 3 show a solid model of damper.

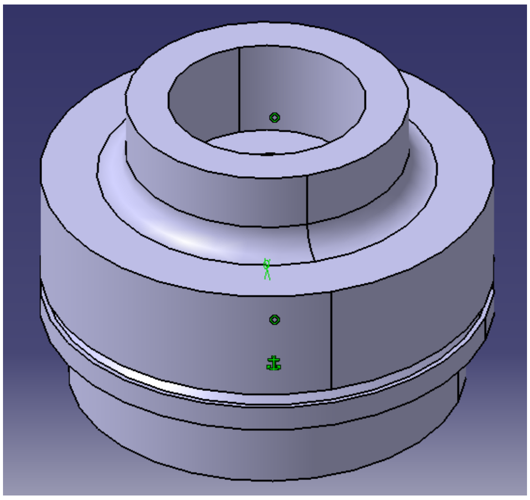

Fig. 2 Assembly of plug and seat

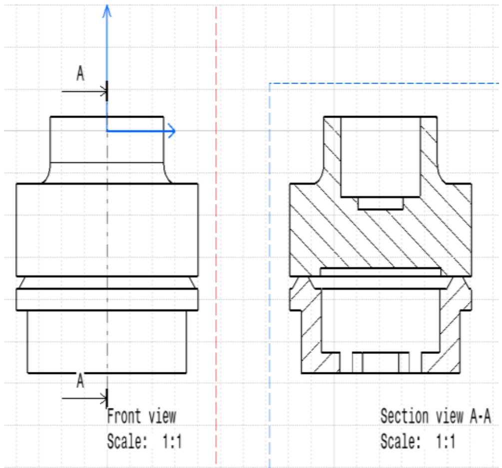


Fig 3 Front View and Sectional View of Assembly

## IV. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. On-going research yields software that improves the accuracy and speed of complex simulation scenario such as transonic or turbulent flows.

### Methodology

Required parts are first modeled in CATIA & PRO/E WILDFIRE which are excellent CAD software's, which makes the modeling easy and user friendly. The model is then transferred in IGES format and exported into the Analysis software CFD 11.0. The assembly is analyzed in CFD in three steps. Pre-processing: The geometry (physical bounds) of the problem is defined. The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform as shown in Fig 4 and Fig 5. The physical modelling is defined. Boundary conditions are defined as shown in Fig 6. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

### Boundary Element Method

The boundary element method (BEM) is a numerical computational method of solving linear partial differential equations which have been formulated as integral equations. It can be applied in many areas of engineering and science including fluid mechanics, acoustics, electromagnetics, and fracture mechanics. The integral equation may be regarded as an exact solution of the governing partial differential equation. The boundary element method attempts to use the given boundary conditions to fit boundary values into the integral equation, rather than values throughout the space defined by a partial differential equation. Once this is done, in the post-processing stage, the integral equation can then be used again to calculate numerically the solution directly at any desired point in the interior of the solution domain. The boundary element method is often more efficient than other methods, including finite elements, in terms of computational resources for problems where there is a small surface/volume ratio. Conceptually, it works by constructing a "mesh" over the modelled surface. However, for many problems boundary element methods are significantly less efficient than volume-discretisation methods. Boundary element formulations typically give rise to fully populated matrices. This means that the storage requirements and computational time will tend to grow according to the square of the problem size. By contrast, finite element matrices are typically banded and the storage requirements for the system matrices typically grow quite linearly with the problem size. Compression techniques can be used to ameliorate these problems, though at the cost of added complexity and with a success-rate that depends heavily on the nature of the problem being solved and the geometry involved.

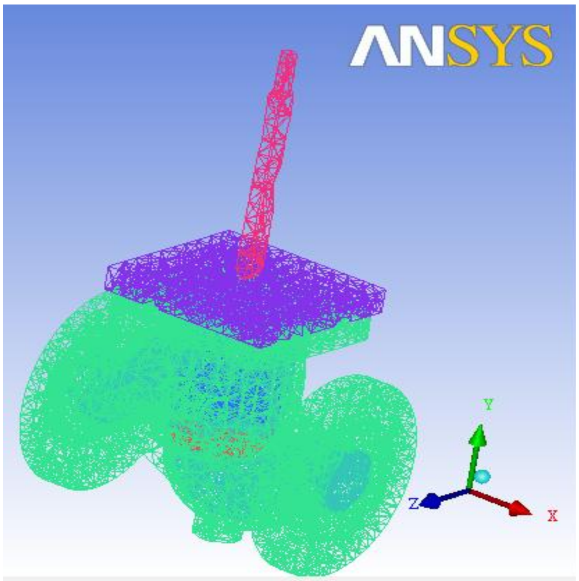


Fig.4 Mesh Assembly

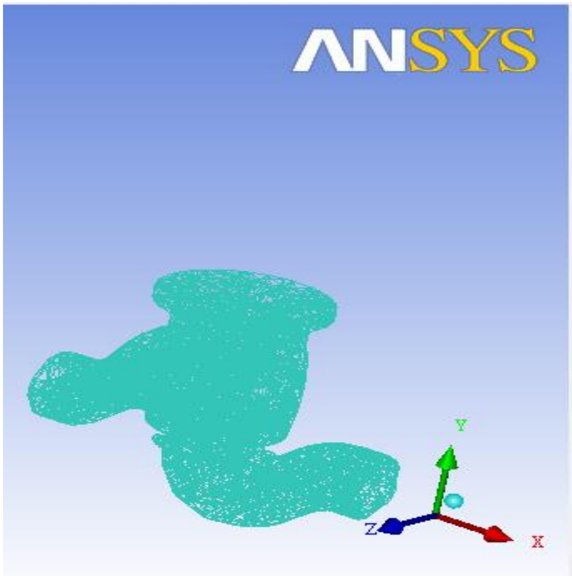


Fig.5 Volume Fluid mesh

## V. SOLID MODELING AND CFD FLOW ANALYSIS

In this section different CATIA models and profiles of plug and seat are explained and these models then assembled in globe vale. These assemblies are then checked on CFD to calculate velocity for each 4mm lift. From this velocity Cv (discharge) for each 4mm lift is obtained. Total four trails are conducted foe each different model.

- **Trial 1.** For plug of diameter 52mm and seat of diameter 52mm

  Solid Model of plug and seat is shown in Fig. 1 and Fig.2
  CFD: boundary conditions (common to all trials)
  Inlet condition- 40bar
  Outlet condition-39bar
  Solution- outlet velocity
  Q=Area × Velocity

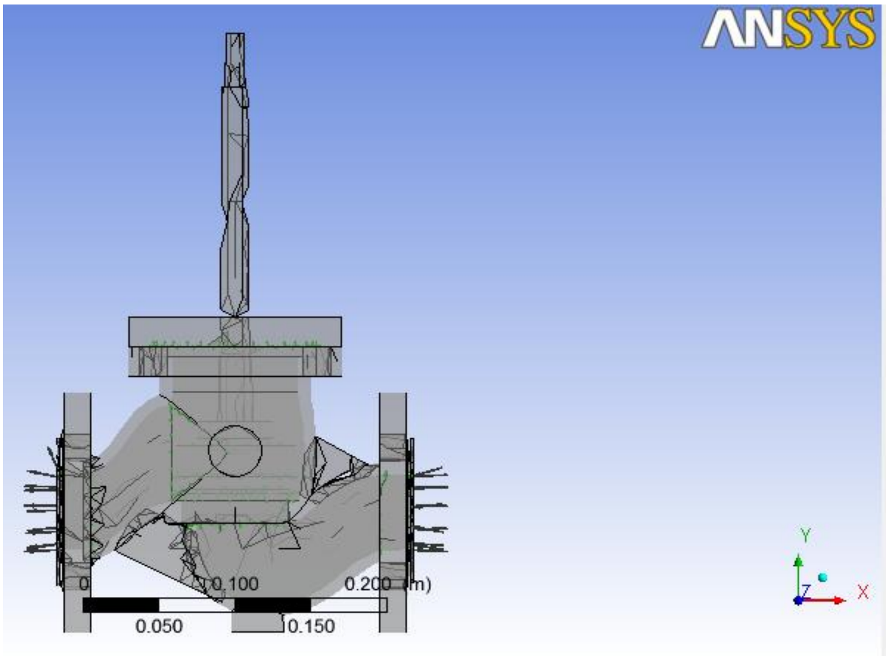


Fig.6 Boundary Condition

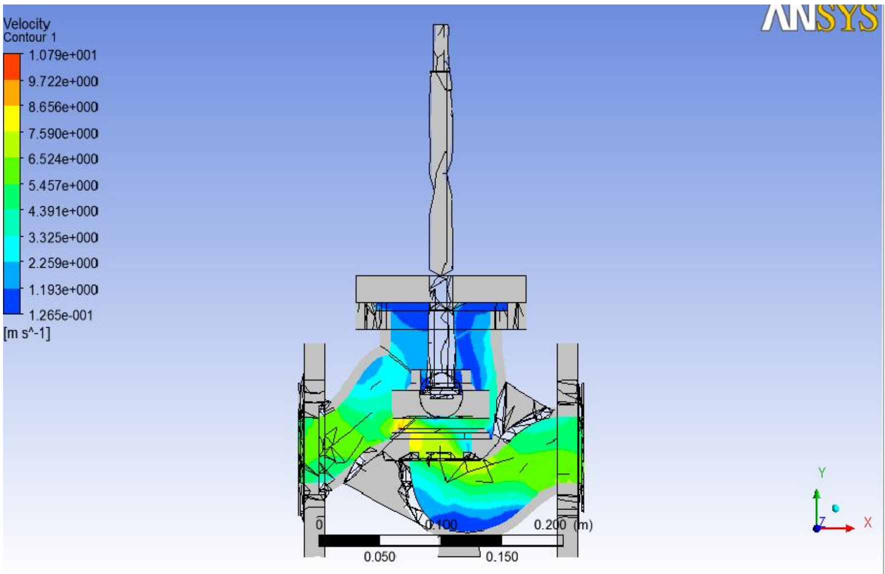


Fig.7 CFD analysis result for Trial 1

Table.1 Result for Trial 1

| Lift of plug(mm) | Outlet Velocity(m/s) | Outlet Discharge Cv(m3/hr) |
|---|---|---|
| 4 | 4.449 | 34 |
| 8 | 5.457 | 41 |
| 12 | 6.244 | 47 |
| 16 | 6.7 | 51.22 |
| 20 | 7.31 | 56 |

In Table 1 for each 4mm lift outlet velocity and discharge ($C_v$) is obtained. (Common to all trials)

- **Trial 2.** For diameter 52mm of plug and diameter 35mm of Seat
  Seat and Plug fig.8 and fig.9

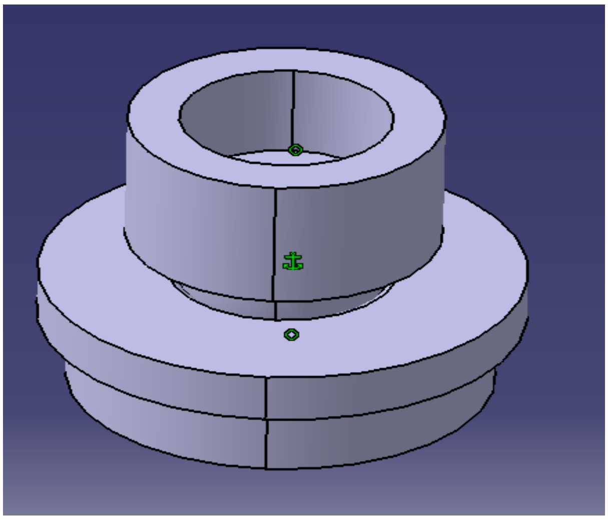

Fig.8 Assembly of plug and seat Trial 2

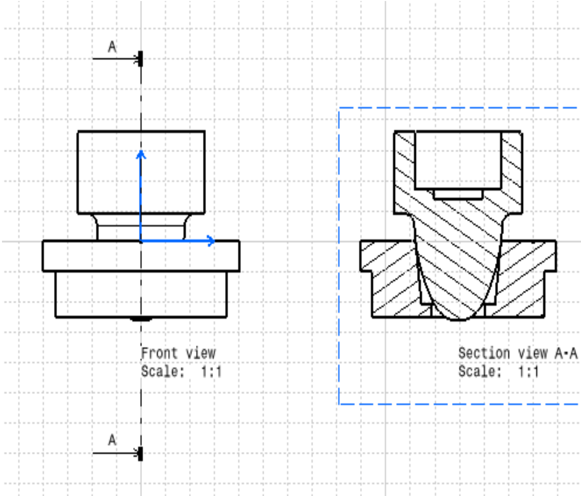


Fig9. Front View and Sectional View of Assembly for Trial 2

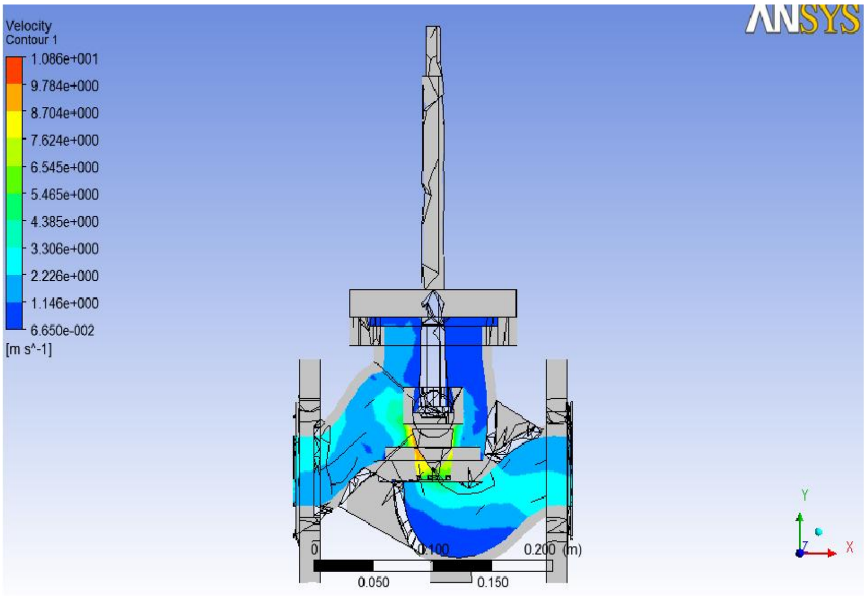


Fig.10 CFD Analysis result for Trial 2

Table.2 Result for Trial 2

| Lift(mm) | Velocity(m/s) | Discharge(m3/hr) |
|---|---|---|
| 4 | 0.9 | 6.88 |
| 8 | 2.7 | 20.64 |
| 12 | 3.157 | 25 |
| 16 | 3.478 | 27 |
| 20 | 3.478 | 27 |

- **Trial 3.** For diameter 52mm of plug and diameter 35mm of Seat
  Seat and Plug fig.11 and fig.12

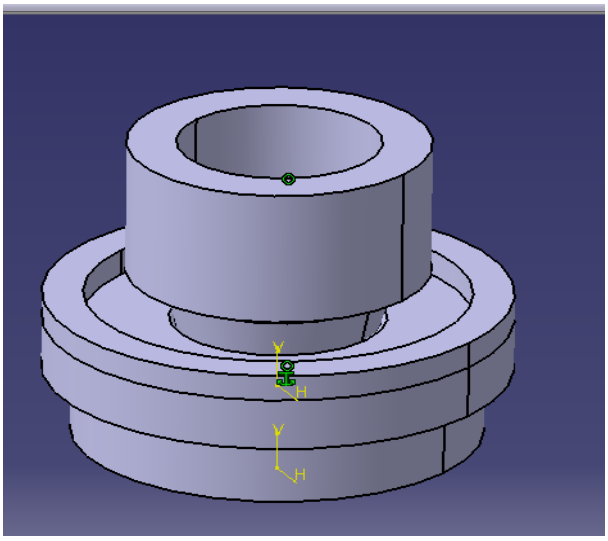

Fig.11 Assembly of Plug and Seat Trial 3

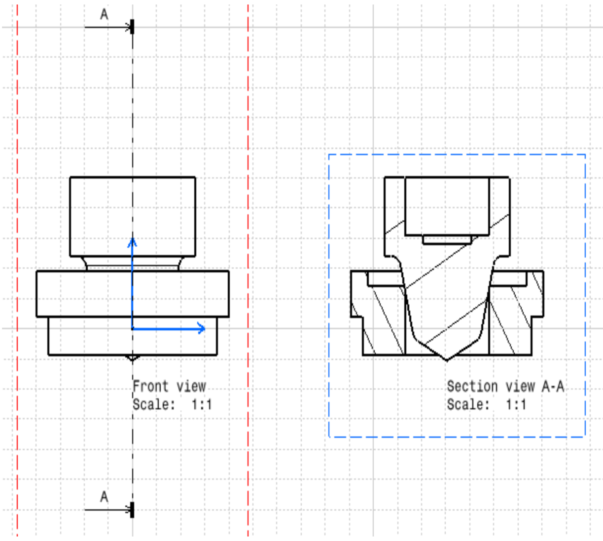


Fig.12 Assembly of plug and seat Trial 3

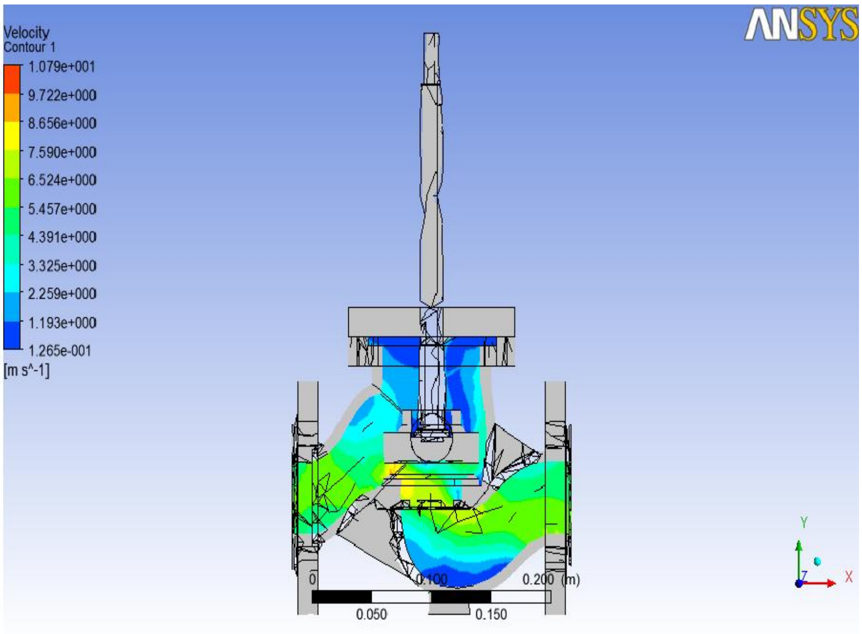


Fig.13 CFD Analysis result for Trial 3

Table.3 Result for Trial 3

| Lift(mm) | Velocity(m/s) | Discharge(m3/hr) |
|---|---|---|
| 4 | 0.9 | 6.88 |
| 8 | 2.071 | 15.83 |
| 12 | 3.049 | 23.31 |
| 16 | 3.681 | 28.14 |
| 20 | 4.23 | 32.397 |

- **Trial 4.** For diameter 52mm of plug and diameter 40mm of Seat
  Seat and Plug fig.14 and fig.15

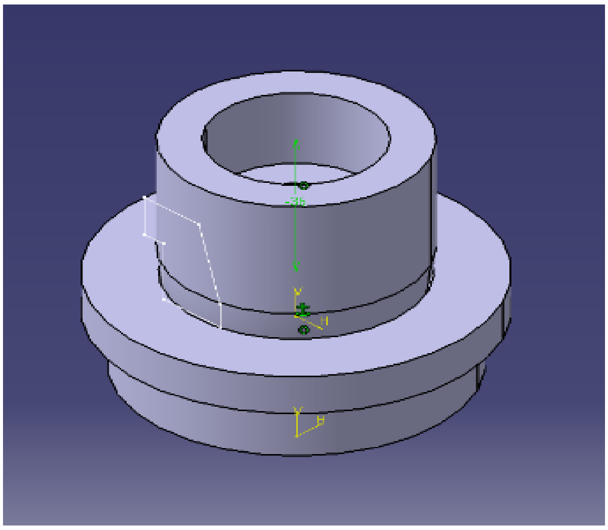

Fig.14 Assembly of Plug and Seat Trial 4

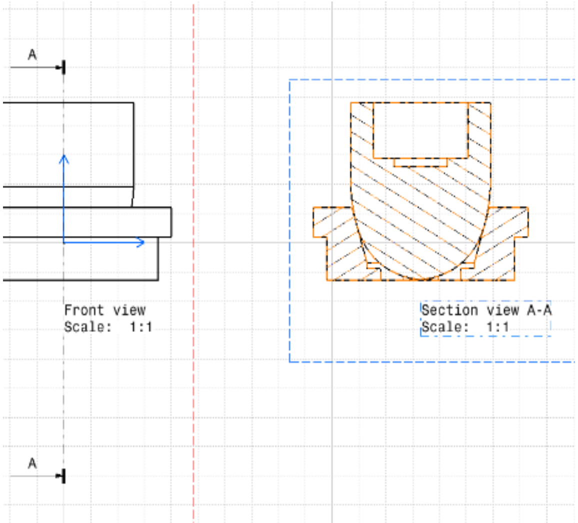


Fig.15 Assembly of Plug and Seat Trial 4

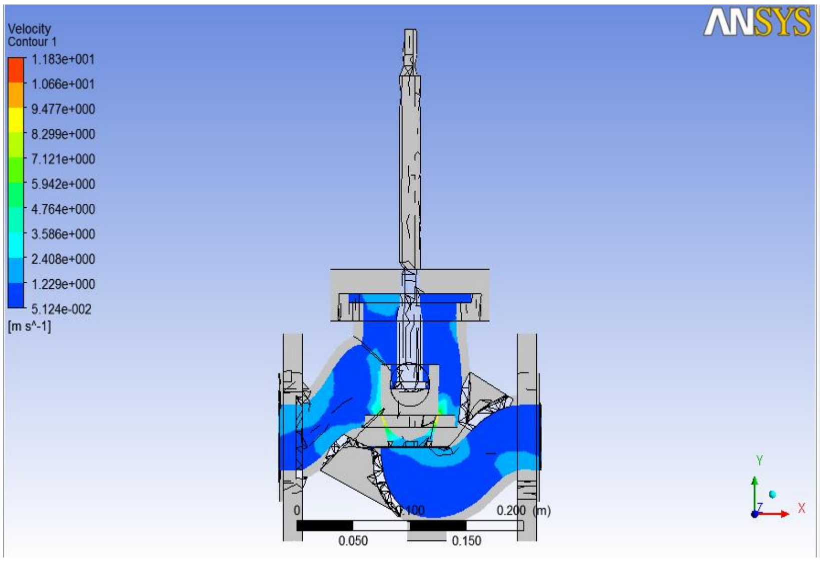


Fig.16 CFD Analysis result for Trial 4

Table.4 Result for trial 4

| Lift(mm) | Velocity(m/s) | Discharge(m3/hr) |
|---|---|---|
| 4 | 1.29 | 9.39 |
| 8 | 3.139 | 24 |
| 12 | 4.316 | 33 |
| 16 | 5.23 | 40 |
| 20 | 6.278 | 48 |

Fig.17 shows the comparison of different curves for each trial. It is graph plotted between lift and discharge.

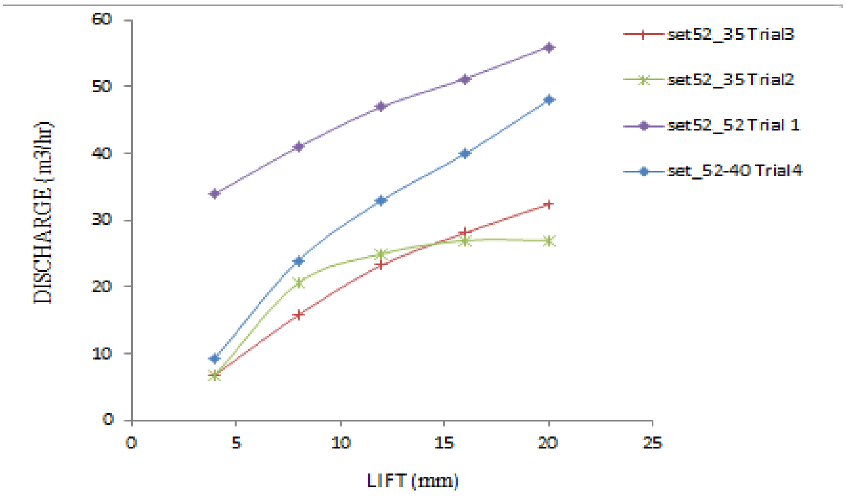


Fig.17 Discharge VS Lift Characteristic Curve

## VI. RESULTS AND DISCUSSION

In Fig.17 it is seen that during Trial 1- discharge varies from 34 $m^3$/hr to 56$m^3$/hr, when lift varies from 4mm to 20mm. From this it is seen that for small lift of plug 34 $m^3$/hr discharge is obtained and for full opening 56 $m^3$/hr discharges is obtained. If actuator is placed over the Globe valve, hunting of actuator will take place. Hence it is difficult to control the fluid flow. As per Fig.17 it is seen that trial 1 and trial 4 curves generally matches quick openings characteristics. As per trial 2 - discharge varies from 6.88$m^3$/hr to 27$m^3$/hr, when lift varies from 4mm to 20mm. It is cleared that discharge increases rapidly initially upto 10mm lift, but later when lift increases upto 20mm, the discharge increases slowly i.e. at 20mm lift upto 27$m^3$/hr. As per trial 3 discharge varies from 6.88$m^3$/hr to 32.397$m^3$/hr, when lift varies from 4mm to 20mm. Again it cleared that, if actuator is placed over the Globe valve, hunting of actuator will not take place. Hence as lift increases and more lift is obtained to control the fluid than other curves and curve approximately matches equal percentage curve.

## VII. CONCLUSION

CFD analysis is performed to analyze the effect of shapes of plug and seat on the flow. From the analysis it is observed that for quick opening valve, trial 1 & trial 4 set can be used. For linear opening valve Trial 2 set can be used. For control valve, trial 3 set can be is proposed from which it seen that when lift varies from 4mm to 20mm, the discharge increases from 6.88$m^3$/hr to 32.397$m^3$/hr, hence it is concluded that the control of fluid obtained is approximately matches the equal percentage curve [Fig.1] as compared to Trial1, Trial 2 & Trial 4 set.

## REFERENCES

[1] C. Garcia, Comparison of friction models applied to a control valve, Control Engineering Practice, 16(10), 2008, 1231–1243.
[2] Q. Yang, Z. Zhang, M. Liu, J. Hu, Numerical Simulation of Fluid Flow inside the Valve, Procedia Engineering, 23, 2011, 543-550.
[3] J. B. V. Wanderley, M. Vitola, S. H. Sphaier and C. Levi, A Three-Dimensional Numerical Simulation of the Free Surface Flow Around a Ship Hull, Computer and Information Science, 1, 2011, ch.18.
[4] J. A. Valverde, S. H. Frankel, G. P. Salvador, Three-dimensional control valve with complex geometry: CFD modelling and experimental validation, 34th AIAA Fluid Dynamics Conference and Exhibit; Portland, Oregon; 2004.
[5] C. Srikanth, C.Bhasker, Flow analysis in valve with moving grids through CFD techniques, Advances in Engineering Software, indiameter, 40 (3), 2009, 193-201.
[6] R. N. Yerrawar, V. B. Tungikar, S. H. Gawande, Finite Element Analysis of Dynamic Damper for CV Joint, 4(4), 2012, 241-247.