DESIGN AND ANALYSIS OF FEED CHECK VALVE AS CONTROL VALVE USING CFD SOFTWARE

R.Nikhil
M.Tech Student
Industrial & Production Engineering National Institute of Engineering Mysuru, Karnataka, India -570008
nikhilkash92@gmail.com

Abstract— **The Feed check valve is fitted to the boiler, slightly below the working level in the boiler. It is used to supply high pressure feed water to boiler and also to prevent the returning of feed water from the boiler if feed pump fails to work. With rapid advancement in the area of flow simulation, CFD and Numerical technique, the flow characteristics of the feed check valve can be studied effectively. In this paper modeling and 3-dimensional flow simulation of a feed check valve is carried out using SOLIDWORKS™ FLOW SIMULATION software to understand the inside flow characteristics and to determine prominent factors such as Pressure drop, Valve co- efficient. In the final phase, the discharge of the valve for a constant pressure drop of 1 bar is determined and flow patterns are visualized.**

Index terms- **Feed check valve, CFD, SOLIDWORKS™ flow simulation.**

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 [1].

The construction of a feed check valve is shown in the Figure 1. The feed-water pipe carrying water from the feed pump usually enters the boiler in the water space of the boiler. A valve is placed in the feed pipe to control or regulate the flow of water into the boiler. The valve is attached directly to the boiler front. It is a non-return valve which permits flow of water in one direction only and automatically prevents the back flow of water from the boiler when the feed water pump is not working. The amount of water entering the boiler can be adjusted by controlling the lift of the valve. This valve is known as feed check valve or boiler feed valve.

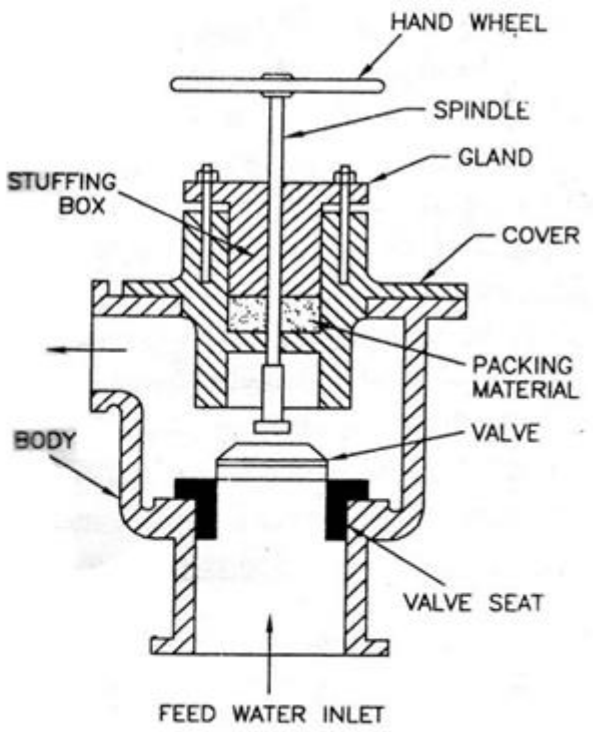


Fig 1 Schematic diagram of a feed checkvalve

II. MODELLING

The modeling of Feed check valve is done with the help of SOLIDWORKS™ CAD software. This software is very user-friendly and easy to use. The assembly of the feed check valve and the components of the valve is shown in the Fig. 2 below.

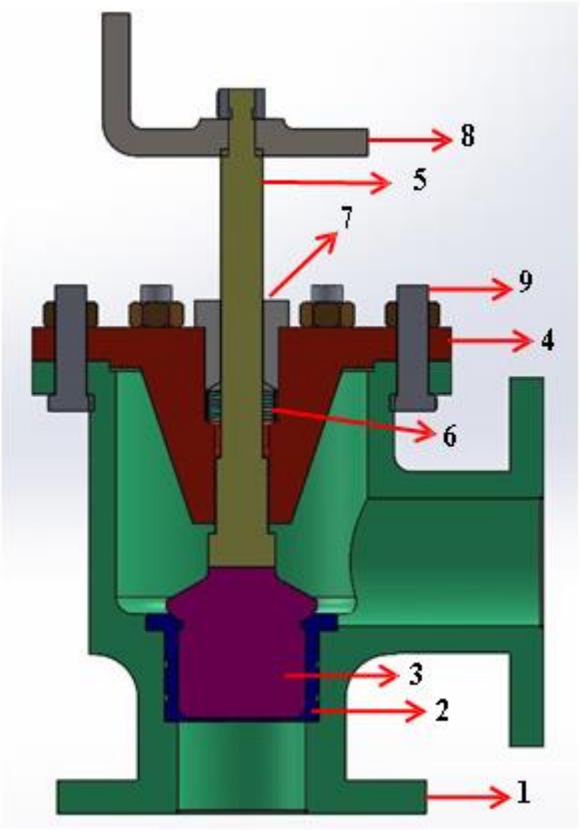


Fig. 2 Front view

Part No.	Component	Material
1	Valve body	Cast Iron
2	Valve seat	Gun Metal
3	Valve	Gun Metal
4	Cover	Cast Iron
5	Spindle	Fe 410 W
6	Packing	Fibre
7	Gland	Gun Metal
8	Handle	Fe 410 W
9	Nut & bolt assembly	Fe 410 W

Table 1 List of components in the assembly

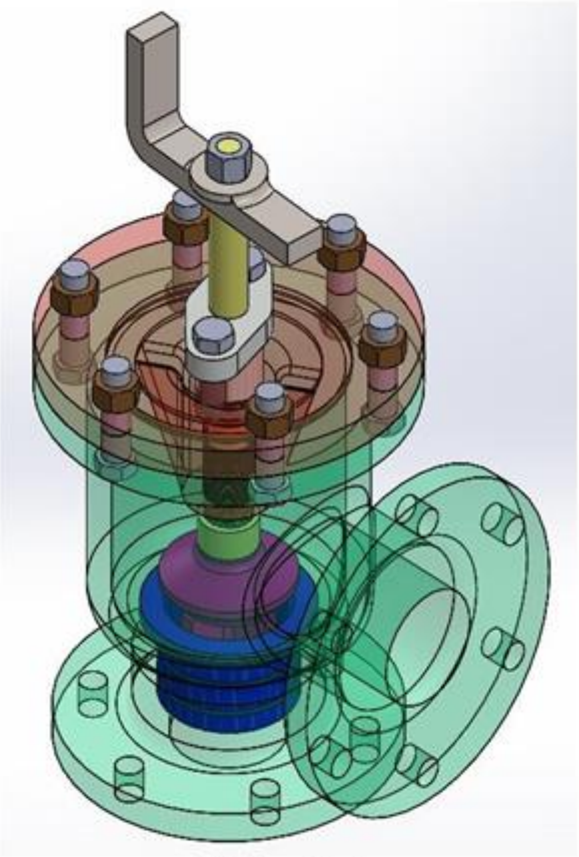


Fig. 3 Isometric view

III. CFD ANALYSIS

The next step after the modeling is to subject the developed model to Computational Fluid Dynamics (CFD) analysis to understand the flow characteristics inside the valve. The CFD analysis is carried out in 3 steps-

Step 1:- Obtaining pressure drop curves for volume flow rates 5 m3/hr and 10 m3/hr by varying valve lift from 3mm to 16mm.

Step 2:- From the obtained pressure drop values the valve Flow Co-efficient (Cv) is calculated for different valve lifts from 3mm to 16mm using conventional valve sizing calculation formula.

Step 3:- In this step, for a constant pressure drop of 1 bar between inlet and outlet, the outlet velocity

(V) of flow is measured to calculate the volume flow rate (Q) which is given by,

Volume flow rate = outlet velocity x area $Q = V \times A$…………….. (1)

A graph of discharge vs. valve lift characteristic curve for the valve is plotted.

A. Grid independence study

In any flow analysis the results obtained depends upon the type of mesh that is chosen for the study. The type of mesh means the number of fluid cells, partial fluid cells, solid cells used for meshing the fluid domain. The results can vary depending on the meshing parameters. In order to get consistent results the grid independence study is carried out. This involves selecting a mesh to perform a set of analysis, increasing the mesh size (total no. of cells) by x1.5 times than previous mesh and performing the analysis yet again. This process is continued till the required accuracy of the results is obtained or till the variation in the results is reduced to a minimum value.

In the present analysis the grid independence study is made to understand the variation of the pressure drop (bar) with mesh size. For this purpose a three sets of meshes are defined. The analysis is carried out in all three cases and a graph pressure drop (bar) vs. valve lift for the volume flow rate of 10 m3/hr is plotted to select the best possible mesh size for further analysis. The details of the three types of meshes i.e. Mesh A, Mesh B and Mesh C used is given below.

Type of Mesh	Cells	Fluid cells	Solid cells	Partial cells
Mesh A	103,709	69,906	33,803	17,301
Mesh B	157,776	85,057	72,719	37,614
Mesh C	280,170	208,394	71,776	38,314

Table 2

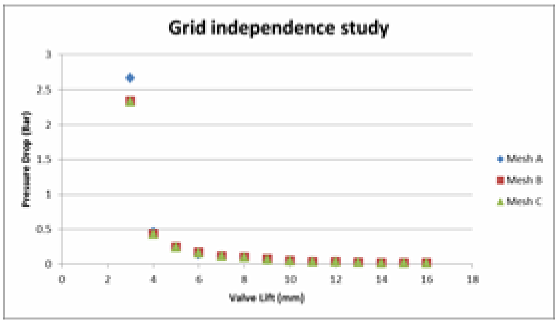


Fig. 4 Grid Independence study

A. *Pressure drop*

Pressure drop is defined as the difference in pressure between two points of a fluid carrying network. Pressure drop occurs when frictional forces, caused by the resistance to flow, act on a fluid as it flows through the tube/valve. Pressure drop across a valve is highly influenced by the area, shape, path and roughness of the valve. Pressure drop is created by flow rate. Higher the flow rates through a restriction, the greater the pressure drop. The pressure drop across a valve is illustrated in the Fig. 6.

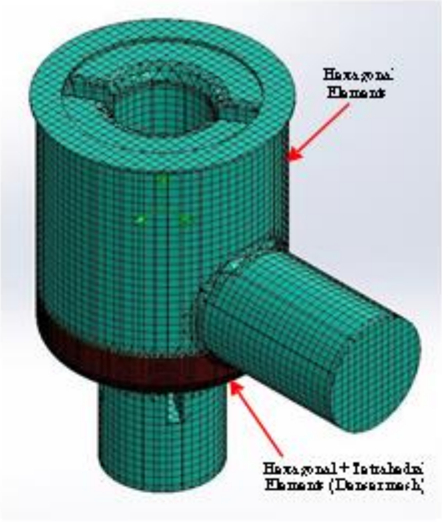


Fig. 5 A Typical Mesh C grid with approximately 280,000 cells used for the analysis

Differential, Delta, Δ, refers to the pressure drop across a flow component – valve, screen, etc. Delta is the 'change' in something; in this case a change, or drop, in pressure. To determine the ΔP across a valve, simply subtract the outlet pressure (P2) from the inlet pressure (P1).

The equation is P1-P2 = ΔP…………… (2) Pressure drop is a critical element in valve sizing and valve application because pressure drop of a valve must be known to the engineer designing fluid system to ensure proper valve selection.

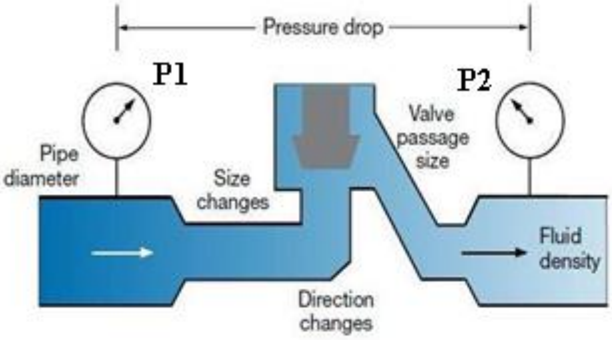


Fig. 6 Pressure drop across a valve

1) **Trial 1:-**

Inlet Volume flow rate = 5 m3/hr
Outlet Static pressure = 101325 Pa

The corresponding pressure drop for different valve lift as calculated by the flow simulation software is given in the Table 3.

Lift (mm)	Pressure drop (Bar)	Pressure drop(PSI)
3	0.5886	8.5363
4	0.1104	1.6017
5	0.0622	0.9016
6	0.0413	0.5987
7	0.0285	0.4130
8	0.0257	0.3730
9	0.0178	0.2582
10	0.0127	0.1839
11	0.0093	0.1355
12	0.0092	0.1333
13	0.0077	0.1122
14	0.0054	0.0785
15	0.0047	0.0682
16	0.0044	0.0642

Table 3

1) **Trial 2:-**

Inlet Volume flow rate = 10 m3/hr

Outlet Static pressure = 101325 Pa

The corresponding pressure drop for different valve lift as calculated by the flow simulation software is given in the Table 4.

Lift (mm)	Pressure drop (Bar)	Pressure drop(PSI)
3	2.3279	33.7636
4	0.4300	6.2361
5	0.2423	3.5148
6	0.1623	2.3540
7	0.1119	1.6235
8	0.1001	1.4524
9	0.0696	1.0101
10	0.0499	0.7236
11	0.0369	0.5348
12	0.0371	0.5376
13	0.0277	0.4011
14	0.0215	0.3125
15	0.0185	0.2677
16	0.0175	0.2537

Table 4

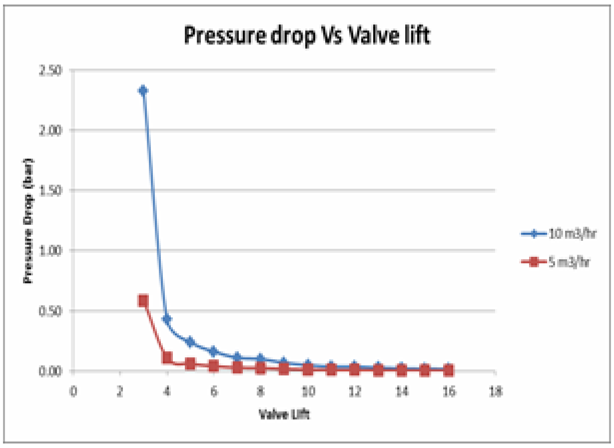


Fig. 7 Pressure drop (bar)vs. Valve lift(mm) for different flow rates

A. *Flow Co-Efficient (Cv)*

Calculating the flow rate through a valve is a very complex task. The valve flow coefficient (Cv) takes into account all the dimensions and other factors—including size and direction changes that affect fluid flow. Using the principle of conservation of energy, Daniel Bernoulli found that as a liquid flows through an orifice, the square of the fluid velocity is directly proportional to the pressure differential across the orifice and inversely proportional to the specific gravity of the fluid. The greater the pressure differential, the higher the velocity; the greater the density, the lower the velocity. The volume flow rate for liquids can be calculated by multiplying the fluid velocity times the flow area. By taking into account units of measurement, the proportionality relationship previously mentioned, energy losses due to friction and turbulence, and varying discharge coefficients for various types of orifices (or valve bodies), a basic liquid sizing (valve sizing) equation can be written as follows:

$$Q = C_v \sqrt{\frac{\Delta P}{SG}} \quad \ldots\ldots\ldots\ldots (3)$$

Where:-

Q = Capacity in gallons per minute.

Cv =Valve sizing coefficient determined experimentally for each style and size of valve, using water at standard conditions as the test fluid.

ΔP = (P1-P2) Pressure differential in psi.

SG = Specific gravity of fluid (water at 60°F = 1.0)

Thus, Cv is numerically equal to the number of U.S. gallons of water at 60°F that will flow through the valve in one minute when the pressure differential across the valve is one pound per square inch. Cv varies with both size and style of valve, but provides an index for comparing liquid capacities of different valves under a standard set of conditions.

To aid in establishing uniform measurement of liquid flow capacity coefficients (Cv) among valve manufacturers, the Fluid Controls Institute (FCI) developed a standard test piping arrangement, shown in Fig. 8. Using such a piping arrangement, most valve manufacturers develop and publish Cv information for their products.

To calculate the expected Cv for a valve controlling water or other liquids that behave like water, the basic liquid sizing equation above can be re-written as follows:

$$C_v = Q \sqrt{\frac{SG}{\Delta P}} \quad \ldots\ldots\ldots\ldots (4)$$

Where:-

Q = Capacity in gallons per minute.

Cv =Valve sizing coefficient determined experimentally for each style and size of valve, using water at standard conditions as the test fluid.

ΔP = (P1-P2) Pressure differential in psi.

SG = Specific gravity of fluid (water at 60°F = 1.0)

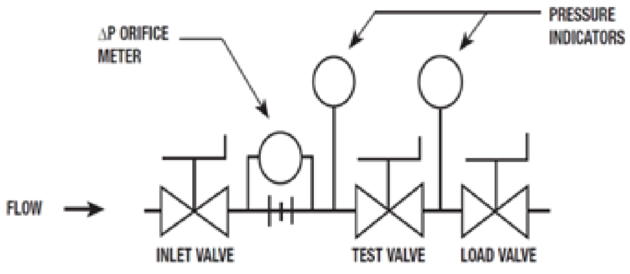


Fig. 8 Standard FCI Test Piping for Cv measurement

Thus, Cv is numerically equal to the number of U.S. gallons of water at 60°F that will flow through the valve in one minute when the pressure differential across the valve is one pound per square inch. Cv varies with both size and style of valve, but provides an index for comparing liquid capacities of different valves under a standard set of conditions.

2) **Trial 1:-**

For Volume flow rate of 5 m3/hr, the flow co- efficient values for different valve lifts are given in Table 4. The corresponding graph is plotted in the Fig. 9

Lift (mm)	Flow co-efficient (C_v)
3	7.5348
4	17.3947
5	23.1843
6	28.4523
7	34.2572
8	36.0432
9	43.3273
10	51.3298
11	59.8006
12	60.2983
13	65.7155
14	78.5572
15	84.2778
16	86.8801

Table 4

1) **Trial 2:-**

For Volume flow rate of 10 m3/hr, the flow co- efficient values for different valve lifts are given in Table 5 and the corresponding graph is plotted in the Fig.10

Lift (mm)	Flow co-efficient (C_v)
3	7.5772
4	17.6311
5	23.4846
6	28.6965
7	34.5554
8	36.5332
9	43.8081
10	51.7608
11	60.2079
12	60.0495
13	69.5166
14	78.7651
15	85.1019
16	87.4129

Table 5

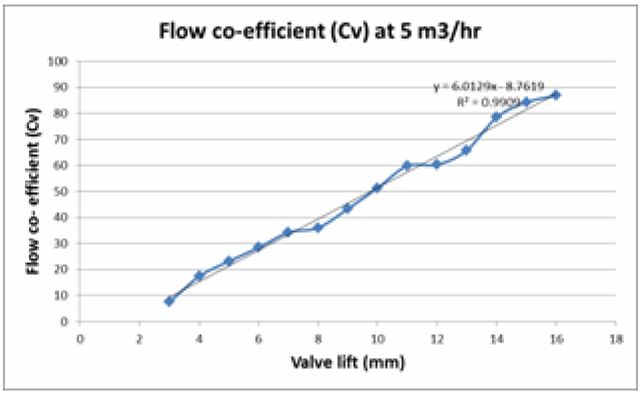


Fig. 9 Flow Co-efficient Vs. Valve Lift(mm) at 5 m³/hr

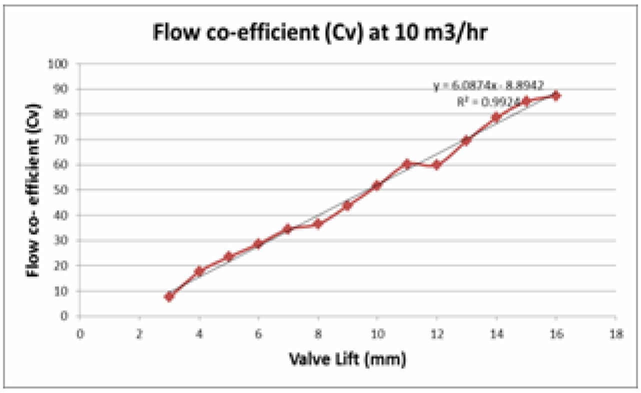


Fig. 10 Flow Co-efficient Vs. Valve Lift (mm) at 10 m³/hr

From comparison of the graphs 9 and 10 it can be clearly seen that the rise in flow-co-efficient of the valve happens in a more linear (lesser fluctuations) fashion when the volume flow rate is high i.e., at
10 m3/hr than when the volume flow rate is 5 m3/hr. This can be explained by the fact that, at increased flow rate the fluid flow is fully developed. Hence, increased velocity of discharge leading to increased flow co-efficient.

B. Discharge at constant pressure drop

In the third step of the analysis, a constant pressure drop of 1 bar is set between the inlet and outlet of the feed check valve. The calculations from the slow simulation software yields outlet velocity (V) as output for different valve lifts. The applied boundary conditions are summarized below.

Boundary conditions:
Inlet Pressure = 40 Bar
Outlet Pressure = 39 Bar
Output:
Outlet Velocity in m/s.

This step is essential in the analysis because in order to plot the characteristic curve of Discharge vs Valve lift for the feed check valve at constant pressure drop. The output in the form of outlet velocity (V) obtained from the flow simulation software is used to calculate the discharge (Q) of the feed check valve using the formula given in equation no. (1).The result of this flow simulation for every 1 mm increase in the valve lift is tabulated in the Table 6.

Area of the outlet = 0.0045 m2

Lift (mm)	Outlet velocity (m/s), V	Discharge (m3/s), Q=V*A	Discharge In LPM
3	0.408	0.0018	110.16
4	0.736	0.0033	198.72
5	1.19	0.0054	321.3
6	1.092	0.0049	294.84
7	1.759	0.0079	474.93
8	2.432	0.0109	656.64
9	1.922	0.0086	518.94
10	2.463	0.0111	665.01
11	2.383	0.0107	643.41
12	1.979	0.0089	534.33
13	2.741	0.0123	740.07
14	2.756	0.0124	744.12
15	3.66	0.0165	988.2
16	4.199	0.0189	1133.73

Table 5

The graph of Discharge Vs Lift characteristic curve for the feed check vale is shown in the Fig. 11.

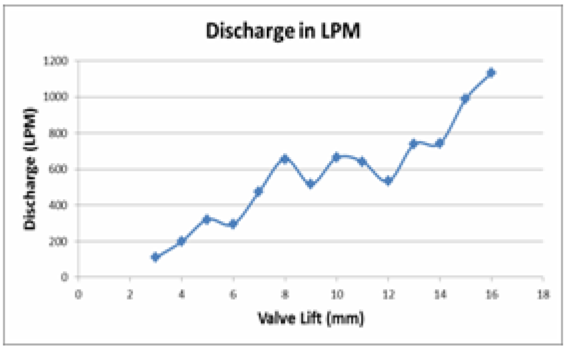
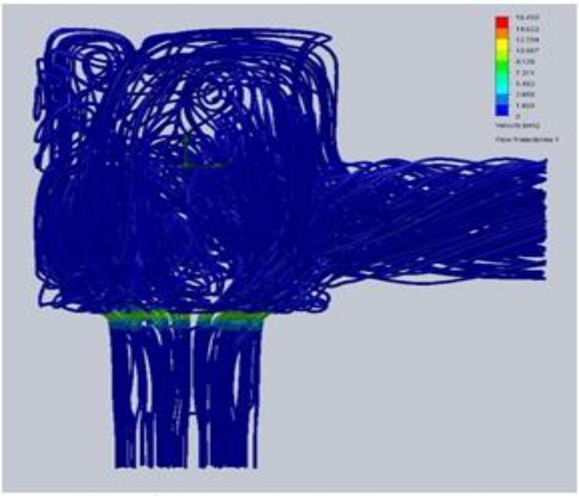


Fig. 11 Discharge (LPM) vs Valve lift (mm) characteristic curve

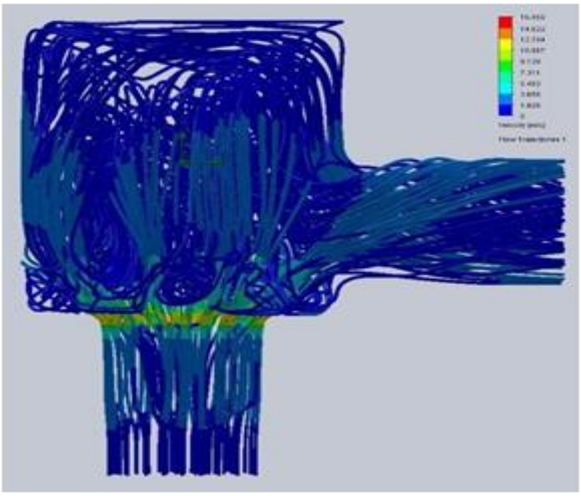
E. Flow Visualisation

An excellent feature of SOLIDWORKS Flow Simulation software is the ability to visualize the flow inside the valve. This is immensely helpful for designers to observe, study the flow patterns and modify the design, if in case the results obtained are not as desired. It also serves as a great tool to verify the actual flow rates of the valves obtained through experimentation.

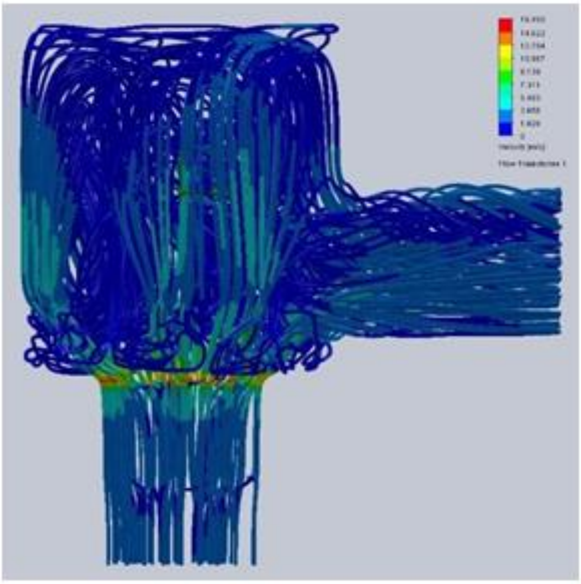
The flow patterns show a rapid increase in the velocity at the valve-valve seat clearance. In order to capture this change the number of cells is increased in this region while the rest of the fluid domain is meshed with usual sized fluid elements.



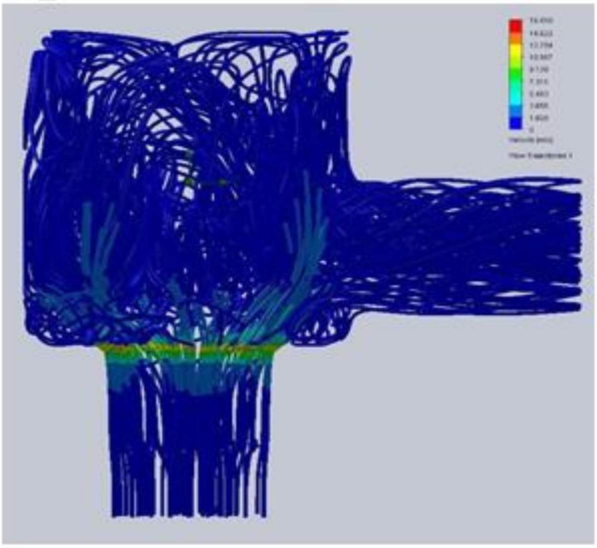
Flow Trajectories for 3 mm Lift



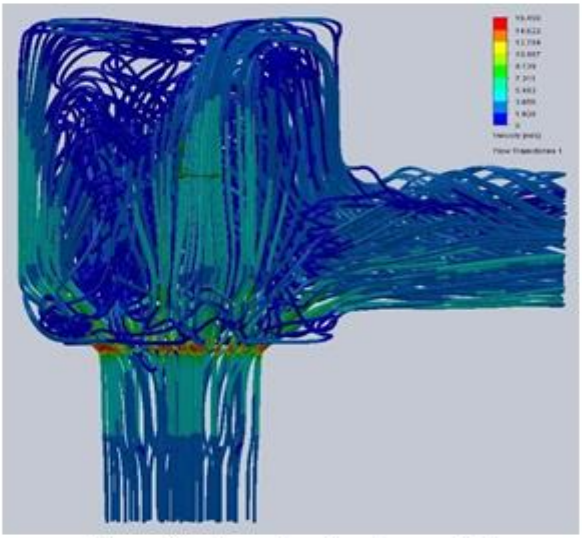
Flow Trajectories for 5 mm Lift



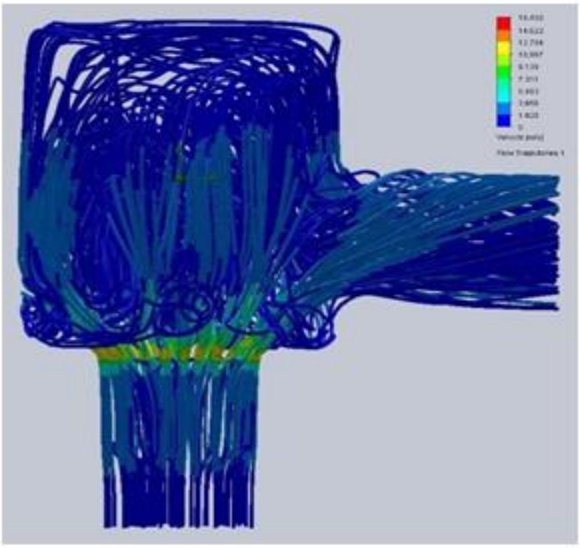
Flow Trajectories for 7 mm Lift



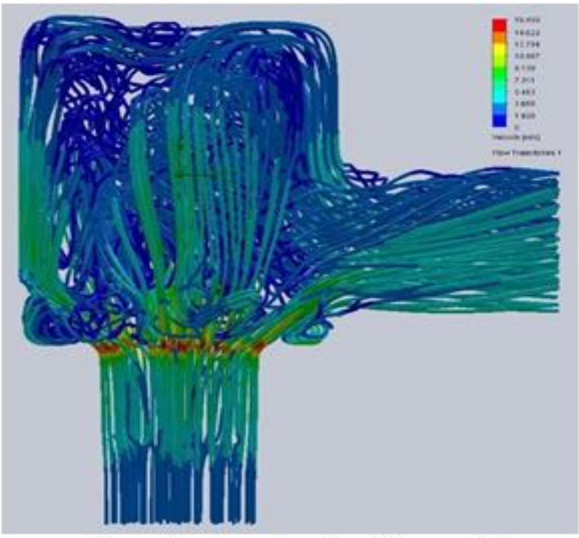
Flow Trajectories for 4 mm Lift



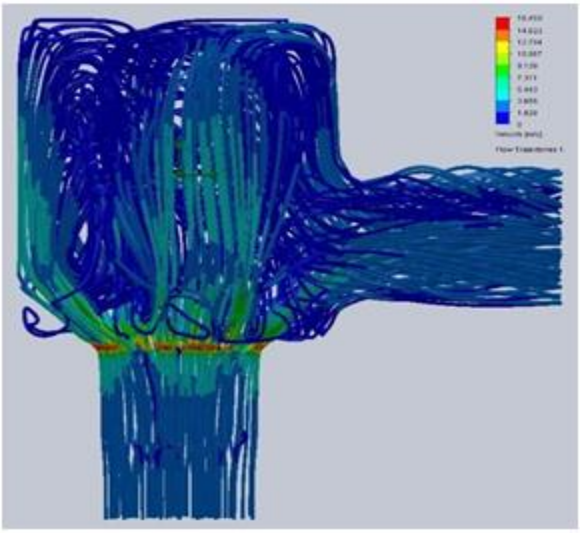
Flow Trajectories for 9 mm Lift



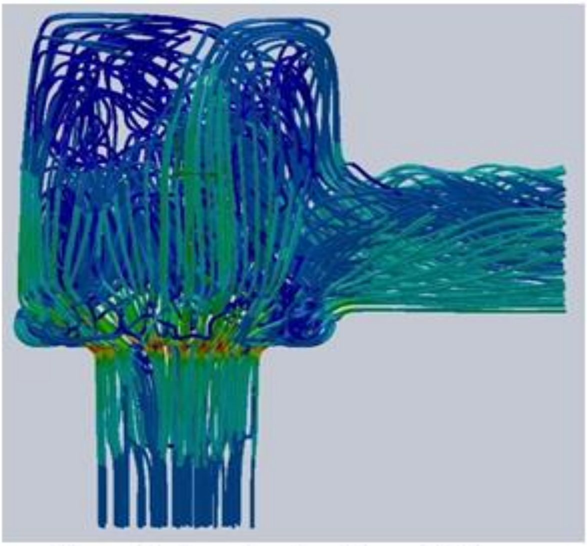
Flow Trajectories for 6 mm Lift



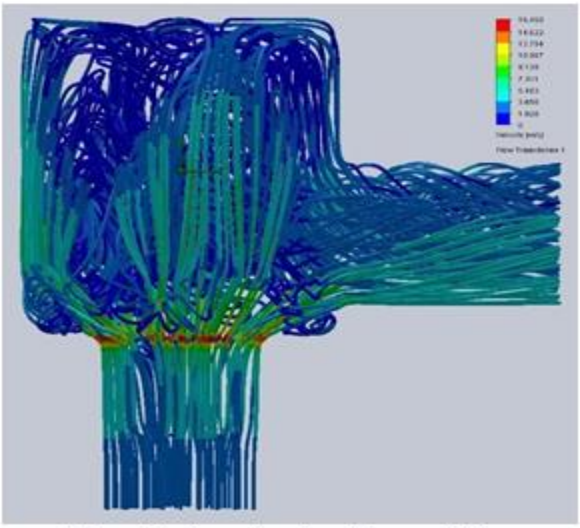
Flow Trajectories for 11 mm Lift



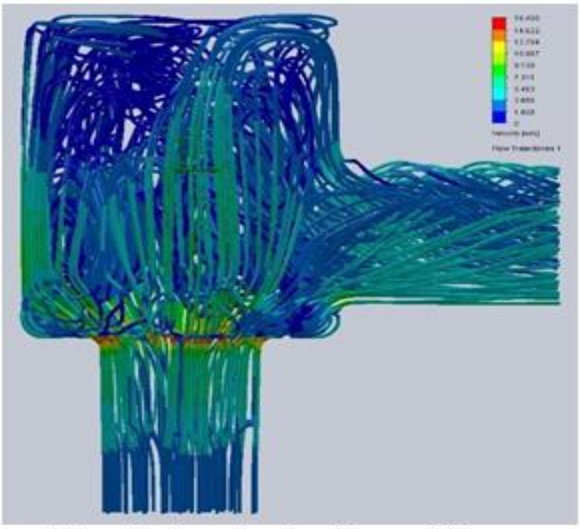
Flow Trajectories for 8 mm Lift



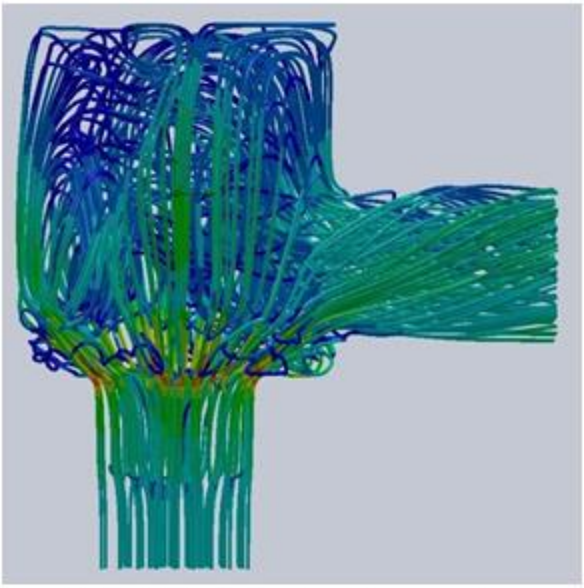
Flow Trajectories for 13 mm Lift



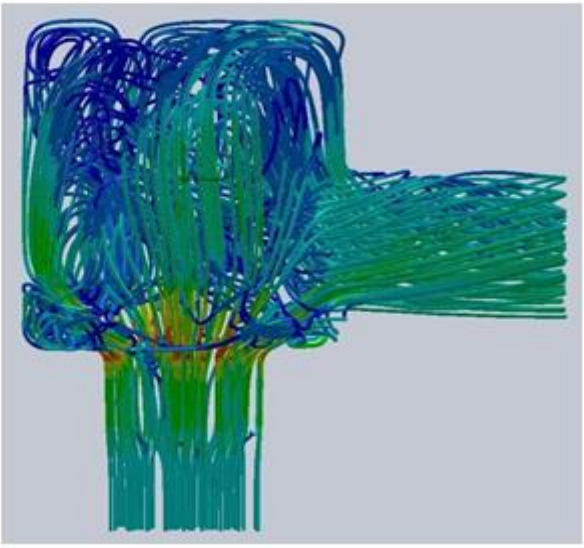
Flow Trajectories for 10 mm Lift



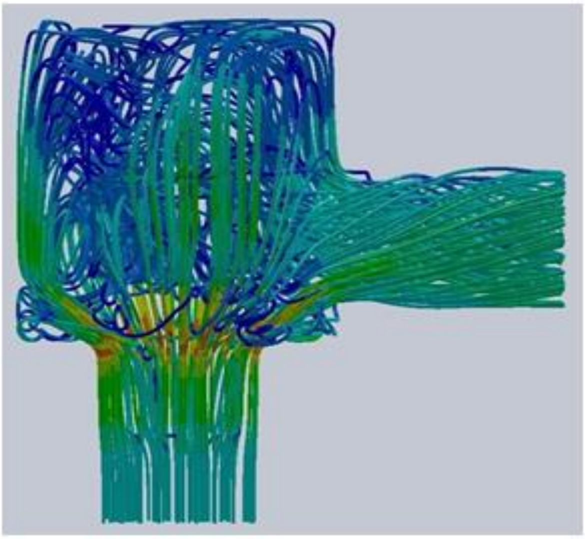
Flow Trajectories for 12 mm Lift



Flow Trajectories for 14 mm Lift



Flow Trajectories for 15 mm Lift



Flow Trajectories for 16 mm Lift

IV. CONCLUSION

3) The modeling of the feed check valve is done and a method of performing CFD analysis using flow simulation software is shown.

4) The pressure drop curve is obtained for flow rates of 5 m3/hr and 10 m3/hr. The results show that the pressure drop is more for higher flow rates and lower valve lifts.

5) The obtained pressure drop values is used to calculate the flow co-efficient (Cv) for flow rates 5 m3/hr and 10 m3/hr using traditional valve sizing formula for different valve lifts i.e., from 3mm to 16mm. The results show that flow co-efficient increases in a linear fashion with the valve lift.

6) Result of flow simulation is visualized to observe the flow patterns inside the valve. The flow

patterns show a rapid increase in the velocity at the valve-valve seat clearance.

REFERENCES

[1] V. J. Sonawane, T. J. Rane, A.D. Monde, R.V. Vajarinkar,
P. C. Gawade, “Design and Analysis of Globe Valve as Control Valve Using CFD Software” IOSR-JMCE, ISSN (p): 2320–334X, PP: 63-71.

[2] Swagelok Valve Sizing Technical Bulletin, December 2007.

[3] R.C.Patel, C.J. Karamchandani, “Element of heat engines” Vol 1, Chapter 8, Steam Boilers.

[4] H.Kursat Celik, Allan E.W. Rennie, Davut Karayel, Ibrahim Akinci, “Determination of flow parameters through CFD analysis for agricultural irrigation equipment: A case study for a mini valve”, Journal of Agricultural machinery science, Vol 7(3), 231-239.

[5] Marine Hydrodynamics 2.20, Lecture 9, Massachusetts Institute of Technology, Mechanical engineering department.