Scientific Journal Impact Factor (SJIF): 3.518
International Journal of Modern Trends in Engineering and Research
ISSN (ONLINE) : 2349 - 9745 ISSN (PRINT) : 2393 - 8161

CFD ANALYSIS OF PRESSURE DROP CHARACTERISTICS OF BUTTERFLY AND DUAL PLATE CHECK VALVE

Adarsh K M[1], Dr. V Seshadri[2] and S. Mallikarjuna[3]
[1]M Tech Student Mechanical, MIT-Mysore
[2] Professor (Emeritus), Mechanical, MIT-Mysore
[3]Assistant Professor, Mechanical, MIT-Mysore

Abstract— Numerical simulation of flow through two types of valves namely dual plate check valve and butterfly valve has been made for various opening conditions. The commercially available software package of ANSYS FLUENT is used for this purpose. Methodology is validated by using K-ε model by analyzing the developing flow through circular pipe. The geometry has been idealized as two dimensional in the view of the limitations of available computational facilities. The value of pressure loss coefficient (C_P) and drag force coefficient (C_D) has been evaluated at various openings and Reynolds number. It is observed that value of pressure loss coefficient (C_P) and drag force coefficient (C_D) lower in case of dual plate check valve. The effect of fillet radius on the outer edge of the valve disc also been analyzed and it is observed that significant reduction in C_P and C_D can be achieved by smoothing of the outer edge of the valve disc. The pressure drop characteristics of two types of valve have been compared at different angles of opening and Reynolds number. The detached flow pattern at various operating conditions is presented in the form of velocity vector plots. This reveals that the energy loss due to separation is less in the case of dual plate check valve.
Keywords— Dual Plate Check Valve, Butterfly Valve, Pressure Loss Coefficient, Drag Coefficient and Reynolds number

I. INTRODUCTION

Valves are mechanical devices which are used to control and to mix the flow through pipes. Valves can regulate both liquids and gases. Based on requirement, valves can be operated by actuators (hydraulic /pneumatic) or by manual operations. The dual plate check valve is a Non-Return valve which is much stronger with less weight and small size when compared to other conventional check valves. The two check plates are controlled by spring which is fixed to a central pin. As the flow reduces the check plates start to close by the action of torsion spring to make sure that no flow reversal takes place.

II. LITERATURE SURVEY

The study on the nature of the butterfly and dual plate check valve on the flow field is studied by many research scholars. The methodology, assumptions, discretization techniques and results from their work are disused below.

Naveen Kumar et.al [1] made parametric study of flow characteristics on butterfly valve using CFD with incompressible flow. 2D geometry of butterfly valve is considered for analysis. K-ε model was used for the study with mesh elements of 185000. In this work it is observed that increment in the valve opening angle rapidly increases the pressure drop from fully closed condition and moderate pressure drops were observed near fully open condition. Reynolds number has significant effects on pressure drop and it is more in case of laminar flow. They also studied effect of drag force on the valve by changing shape and size of the valve disc.

Arun Azad et.al [2] carried work on butterfly valve which is widely used in hydro power plants to regulate and control the flow. The valve modelled to scale ratio of 1/3 using ICEM CFD 12 software and valve characteristics studied at different valve opening angles. Disturbances are reduced at higher valve opening angles due to more uniform streamlines at downstream. Large amount of drag is offered at smaller opening angles due to high turbulence and vortices formed at upstream. Flow coefficient was directly proportional to valve opening angles and reached its maximum value of 0.163 at fully opened condition.

Xue guan Song et.al [3] conducted study on butterfly valve with the use of ANSYS CFX 10.0solver for the estimation of flow characteristics. Computation is done by considering k-ε turbulence model and results were compared with available experimental data. Variations inflow and torque coefficient were analyzed at different valve openings by keeping uniform inlet velocity. From the study they concluded that pressure drop and torque follows the similar trend. As the valve opening increases flow coefficient also increases with decrement in the formation of vortex at downstream. K-ε model gave little deviated results at opening angle of 20° to that of experimental and hence simulation became very sensitive as the valve closes.

A D. Henderson et.al [4] made numerical study on flow through butterfly valve which is used as safety device in hydroelectric power plants. ANSYS designer modeler was used for the analysis and CFD is applied to predict valve open angles v/s hydrodynamic torque at constant head. The effect of Reynolds number and unsteady flow is also taken for consideration. Computed data's were compared with the field measurements of full sized valve. Their final statement was flow becomes unsteady over a certain range of valve opening with vortex development in the downstream of valve. The results from CFD were all most same as that of experimental field measurement.

Aniruddha V Kapre et.al [5] used general purpose CFD software to analyze the flow characteristics of butterfly valve by incorporating 3D model. The analysis was carried out at 3 different angles of 10°, 50°, and 90°. Streamlines with transient state flow analogy was done from the predicted results at downstream. Flow became unsteady beyond the specified pressure range.

G.Tamizharasiet.al [6] has done study on butterfly valve for air as fluid at various valve opening angles like 20°, 40°, and 60°. 3D geometry of butterfly valve is modelled and numerically investigated by ANSYS CFX. K-ε model was used for the study with tetrahedron mesh elements of 935614. Pressure loss across the valve is less comparatively as the valve opening angle increase. At high valve opening angles pressure variation and turbulence intensity at downstream increases.

In the present study CFD has been used to analyze the flow through butterfly valve and dual plate check valve. 2D approximation is made in order to carry the analysis with in the computational facilities available institute. For this purpose ANSYS FLUENT has been used. Analysis has been made for various disc positions and Reynolds number. Both laminar and turbulent regimes have been covered. The variation of pressure loss coefficient (C_P) and drag force coefficient (C_D) with valve opening at various Reynolds number and compared.

III. VALIDATION OF CFD METHODOLOGY

To validate the methodology fully developed flow through pipe is taken for consideration. Here keeping the conditions such as inlet velocity of 2m/s, diameter of the pipe 20mm, length of the pipe 2000mm, density =1000kg/m^3 and viscosity as 0.004Pa-s. CFD analysis is carried out with structured quadrilateral mesh consisting of 173884 elements at Reynolds number of 10000. After fully developed flow is reached pressure drop in the flow passage is calculated between upstream length and downstream length of 1200 and 1700 mm respectively. The computed values from CFD are compared with theoretical values and the agreement between the two values is within 0.1%. This was achieved when the number of elements was larger than 1.7×10^5. Further, among the various turbulence models tested K-ε model gave the best agreement. Hence for all the subsequent studies this model was used.

IV. ANALYSIS OF FLOW THROUGH BUTTERFLY VALVE

Two dimensional planar model of butterfly valve with upstream length (2W) and downstream length (3W) is created by using ANSYS design modular. The flow domain consisting of 40mm width (W), length of 500mm and having valve thickness of 3mm which is placed at distance of 240 mm from the inlet of pipe. The flow domain is discretized using quadrilateral elements. The number of elements used in the range of 123530. The denser mesh near the valve is shown in Fig 1.

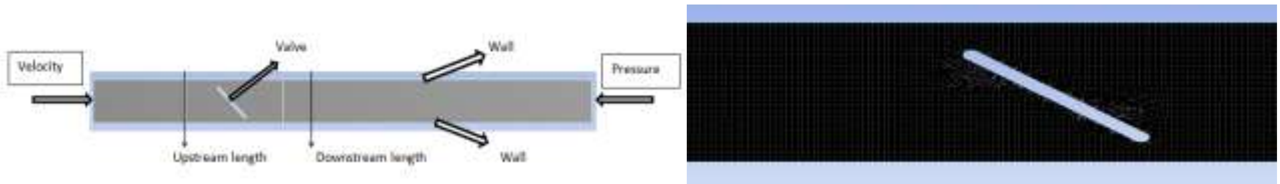


FIG 1: Butterfly Valve with discretization and Boundary Conditions

Boundary conditions incorporated to flow domain with butterfly valve are velocity-2m/s at inlet, zero gauge pressure at outlet and Wall with No slip condition and zero velocity.

A. RESULTS AND DISCUSSION

To study the valve characteristics, rectangular shape valve of thickness 3mm with outer edge fillet radius of 1.5mm is considered. The computation is made for laminar (Re=1000) and turbulent (Re=50000) flows from the fully opened condition i.e. 90° (parallel to axis) to almost closed position of 10° with an increasing interval of 10°.

Pressure loss coefficient (C_P) and Drag coefficient (C_D) are depicted through the computation and these coefficients are given by,

$$C_P = \Delta P \,/\, \tfrac{1}{2} \rho v^2$$

ΔP= Pressure in pascal between two section located at distance of 80mm (2W) upstream and 120mm (3W) downstream from the center of the valve disc.

ρ=Density of the fluid in kg / m^3.

V=Velocity in m/s.

$$C_D = F_D / \tfrac{1}{2} \rho v^2 A$$

F_D= Drag Force in N calculated by integration of computed pressure and shear stress distribution on the valve disc.

A=Area of Valve plate normal to flow direction.

1. EFFECT OF VALVE OPENING

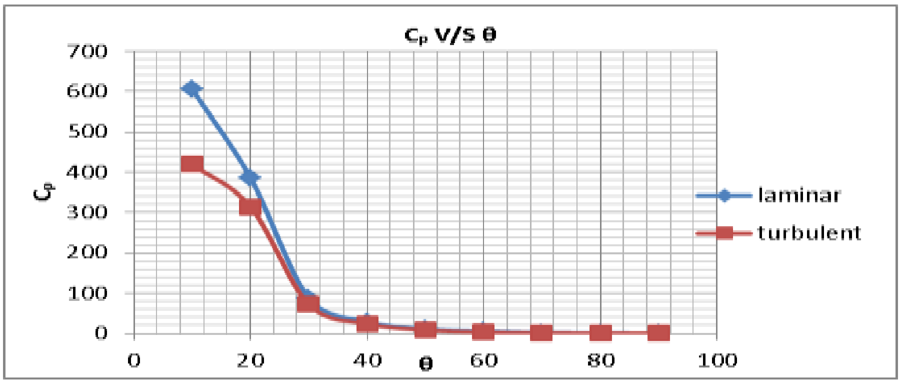
The variation of C_P with respect to valve opening angle (θ) is shown in Graph 1 and values are tabulated in Table 1. It is observed that the value of C_P decreases with increasing θ. The rate of decrease is much faster up to opening of 40° and variation is gradual for larger openings.

Table 1 : C_P For Butterfly Valve at Different Valve Openings

θ	C_P(Re=50000)	C_P(Re=1000)
10°	419.092	605.910
20°	314.47	386.137
30°	72.79	88.368
40°	24.074	29.430
50°	8.622	11.344
60°	2.934	5.433
70°	1.12	2.040
80°	0.436	1.110
90°	0.23	0.447

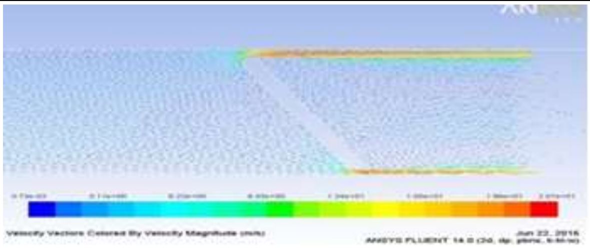
At any given angle of opening the value of C_P in laminar flow is higher than that of turbulent flow. This can be attributed to higher viscous losses at lower Reynolds number.

 ISSN (Online):2349–9745; ISSN (Print):2393-8161

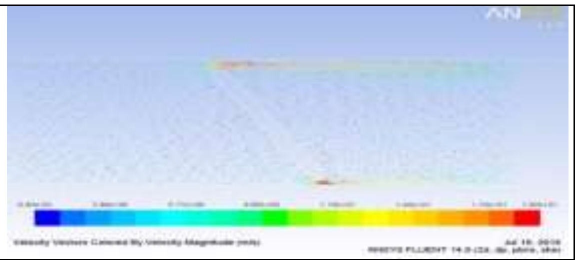


Graph 1: Variation of C_P for Different Valve Openings at Re=50000(turbulent) and Re=1000(laminar)

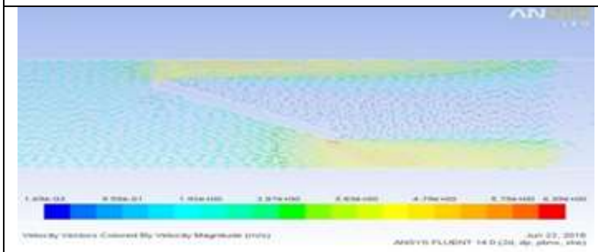
From the figures it is observed that, the range and velocity encountered in the flow domain keeps on decreasing as the valve opening angle increases in both the cases. At lower valve opening angles maximum velocities are encountered at minimum flow area due to huge disturbance to the flow in both the cases. From the Figures it is clear that the velocities encountered at low valve opening positions in laminar flows is more than the turbulent due to huge viscous loss.



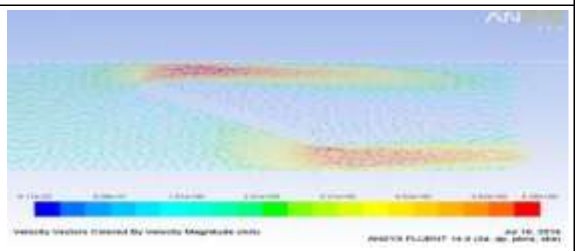
Velocity Vectors at θ=30° for Re=50000



Velocity Vectors at θ=30° for Re=1000



Velocity Vectors at θ=60° for Re=50000



Velocity Vectors at θ=60° for Re=1000

Fig 3: Velocity Vector for BFV at different valve opening

2. DRAG FORCE ON THE VALVE DISC

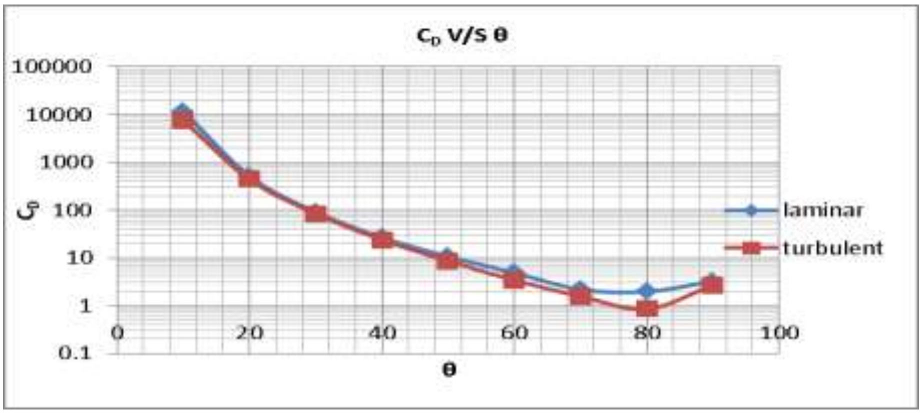
The drag force comprises of both pressure drag as well as viscous drag and in the calculation of drag force both pressure and stress distribution play very imporatant role.

Table 2 : C_D for Butterfly Valve at Different Valve Openings

θ	C_D(Re=50000)	C_D(Re=1000)
10°	413.852	5963.797
20°	383.544	331.62
30°	82.116	97.06
40°	26.53	32.429
50°	12.278	14.922
60°	4.99	7.071
70°	2.718	4.119
80°	1.78	3.212
90°	0.892	1.713

Drag coefficient (C_D) for turbulent and laminar flows for different valve opening angles is listed in the Table 2. From Graph 2 we can observe that, coefficients of drag force decreases with increase in valve opening angles in both cases. Drag force on the valve is to high in case of laminar flow at lower valve openings. This is due to the obstraction created by the valve and higher viscous effects. On the other hand when flow reaches turbulent condition drag force on the valve becomes more or less due to ceased viscous losses.

At each valve opening, laminar flow offers more drag than turbulen and it is clearly depicted from Graphs 2.



Graph 2 : Variation of C_D for Different Valve Opening at Re=50000 and Re=1000

3. EFFECT OF FILLET ON VALVE DISC

The previous analysis has made valve having fillet radius of 1.5mm. In this case an effort has been made to study the effects of pressure drop and drag force by varying the fillet radius of the valve. The effects are studied at different radius of 0, .5mm, 1mm and 1.5 mm for the valve opening of 20°. The variation in pressure loss coefficient (C_P) and drag coefficient (C_D) has been tabulated in the Table 3.

Table 3 : C_P and C_D at Different Fillet Radius at θ=20°(Re=50000)

Fillet radius	C_P	C_D
0	12056.95	1271.180
0.5	5016.96	527.744
1	1032.94	455.579
1.5	314.47	383.544

From the Table 3 it is evident that, higher values of fillet radius leads to lower C_P and C_D. This is because of smoothening of the surface in presence of fillet radius. If sharp corner exists (without fillet) huge disturbances to the flow and drag offers higher pressure drops.

V. ANALYSIS OF FLOW THROUGH DUAL PLATE CHECK VALVE

Two dimensional symmetrical model of dual plate check valve with upstream length (2W) and downstream length (3W) is created by using ANSYS design modular. The flow domain consisting of 40mm width (W), length of 500mm and having valve thickness of 3mm which is placed at distance of 240 mm from the inlet of pipe. The flow domain is discretized using quadrilateral elements. The number of elements used in the range of 123530. The denser mesh near the valve is shown in Fig 2. Boundary conditions incorporated to flow domain with dual plate check valve are velocity-2m/s at inlet, zero gauge pressure at outlet and Wall with No slip condition and zero velocity.

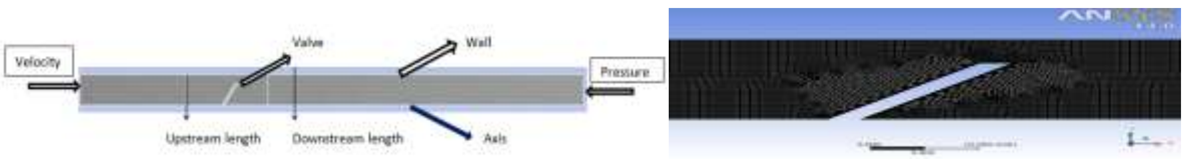


Fig 2: Dual plate check valve with discretization and Boundary Conditions

A. RESULTS AND DISCUSSION

To study the valve characteristics, rectangular shape valve of thickness 3mm with sharp edge (without fillet) is considered. The computation is made for laminar (Re=1000) and turbulent (Re=50000) flows from the fully opened condition i.e. 90° (parallel to axis) to all most closed of 10° with an increasing interval of 10°.

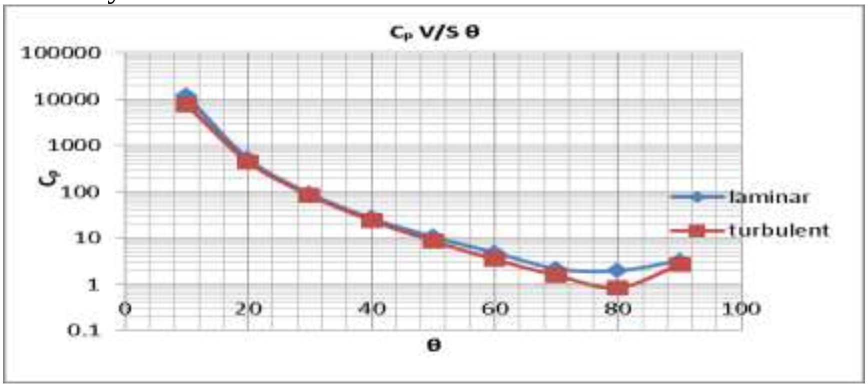
1. EFFECT OF VALVE OPENING

Pressure loss coefficient (C_P) for turbulent and laminar flows for different valve opening angles is listed in the Table 4

Table 4 : C_P for Dual Plate Check Valve at different valve openings

θ	C_P(Re=50000)	C_P(Re=1000)
10°	7505.9	11698.6
20°	418.422	494.57
30°	76.259	84.83
40°	19.51	23.067
50°	6.26	8.233
60°	2.037	3.3
70°	0.66	1.428
80°	0.28	0.735
90°	0.53	0.97

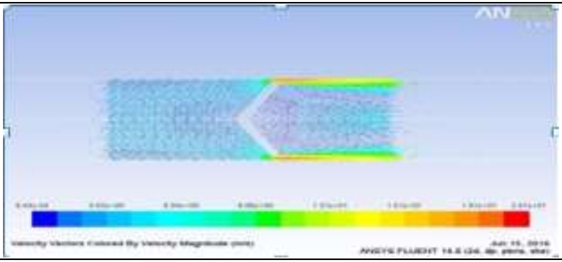
As the valve opening angle increases, pressure drop decreases due to lesser obstruction to the flow by the valve and variation in C_P will become more or less constant at higher Reynolds number. These variations are shown in Graphs 3. In both cases variation in C_P with respect to "θ" follows same trend. From the Graph 3 it is observed that increase in the valve opening leads to decrease in pressure drop and hence pressure loss coefficient. This is because of increment in the valve opening angle from the fully closed condition leads to decrement in the disturbance to the flow along with the decrease in the velocity.



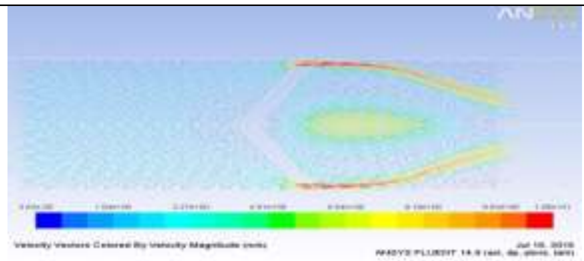
Graph 3: Variation of C_P at Different Valve Opening for Re=50000 (Turbulent) and Re=1000 (laminar)

The value of pressure loss coefficient (C_P) in turbulent flow appears less when compared to laminar flow at each respective valve opening due to its large viscous force.

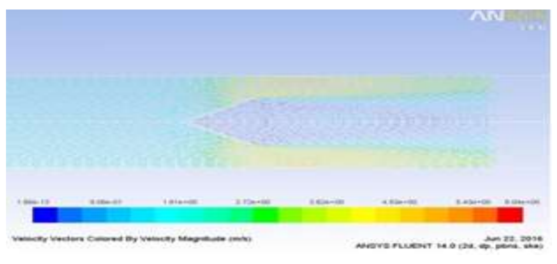
From the above Figures it is observed that at low valve opening angles flow experiences the maximum disturbance with greater velocity as the fluid flows through the minimum flow area and flow patterns are not so clear. Turbulent flow experiences less velocity over the valve when compared to laminar flow. Disturbance to the flow is reduced as the valve opening angle increases with decrement in velocity.



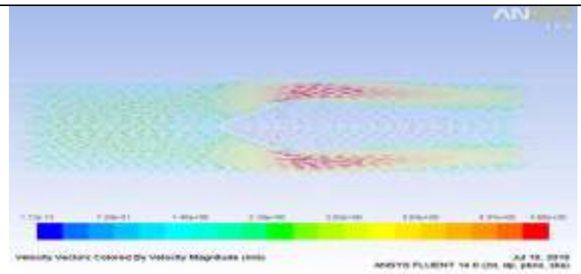
Velocity Vector at θ=30° for Re=50000



Velocity Vector at θ=30° for Re=1000



Velocity Vector at θ=60° for Re=50000



Velocity Vector at θ=60° for Re=1000

Fig 4: Velocity Vector for DCV at different valve opening

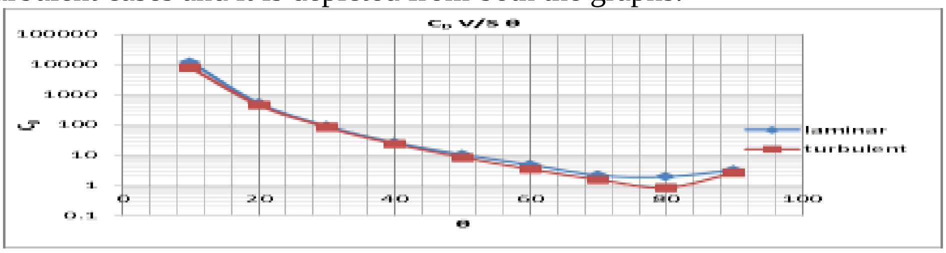
2. EFFECT OF DRAG FORCE ON VALVE PLATE

Variation in drag coefficient (C_D) with respect to "θ" is listed in the Table 5 for both turbulent and laminar flow. From the Table 5 it is clear that drag coefficient varies inversely with respect to valve opening angle in both laminar and turbulent flow cases. At laminar condition, fluid offers high viscosity compared to that of high Reynolds number hence when the flow is laminar low valve opening angle (10°) the drag force as well as C_D is too high and vice versa at turbulent and fully opened condition (90°) and it is due to lesser obstruction and viscosity losses.

Table 5 :C_D for Dual Plate Check Valve at Different Valve Openings

Θ	**C_D(Re=50000)**	**C_D(Re=1000)**
10°	7595.78	11777.78
20°	440.401	512.203
30°	84.138	91.205
40°	23.85	26.718
50°	8.582	10.589
60°	3.425	4.855
70°	1.554	2.165
80°	0.847	1.959
90°	2.687	3.228

Drag force is heavier as the valve closes (valve opening angle decreases) due to higher pressure losses and higher disturbances, due to the decrement in the disturbance to the flow in both laminar and turbulent cases and it is depicted from both the graphs.



Graph 4 : Variation of C_D for Different Valve Opening for Re=50000 (Turbulent) and Re=1000 (laminar)

3. EFFECT OF FILLET ON VALVE PLATE

Variation in C_P and C_D for different fillet radius is tabulated in Table 6. From the table it is clear that C_P and C_D vary inversely with respect to fillet radius. At low fillet radius due to sharp corners flow suffers more pressure loss and drag. As fillet radius increases C_P and C_D starts to decrease due to smooth passage for the flow.

Table 6 : C_P and C_D for Different Fillet Radius at θ=20°(Re=50000)

Fillet Radius (mm)	C_P	C_D
0	7505.9	440.402
0.5	301.755	314.316
1	247.059	258.184
1.5	134.44	140.294

VI. COMPARISON OF PRESSURE DROP AND DRAG CHARACTERISTICS OF DUAL PLATE CHECK VALVE AND BUTTERFLY VALVE

In general to compare the performances of two different valves all the geometrical parameters should be kept same. So for the present study dimensions of both valve geometries are considered same and having length of passage 500mm, width of the passage 40mm and rectangular shape valve with plate thickness of 3mm. water is used as fluid and its density is around 1000kg/m^3. Both the types of valve had fillet radius of 1.5 mm

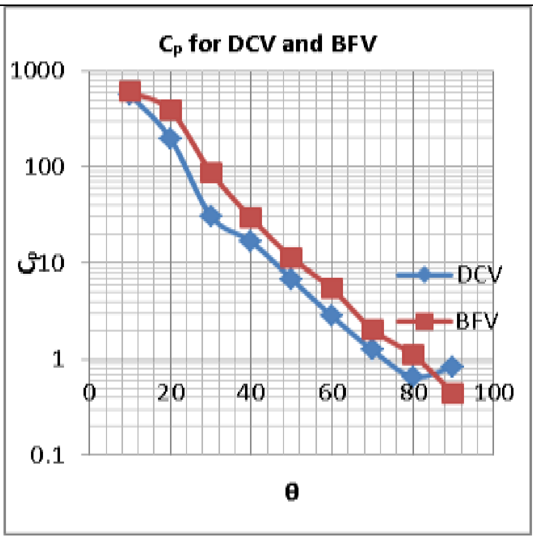
1. PRESSURE LOSS COEFFICIENT (C_P)

Pressure loss coefficient for both the valves are estimated at two different Reynolds numbers i.e. at Re=50000 (turbulent flow) and at Re=1000 (laminar flow) to study the performance of valves at both the cases. The comparative values are tabulated in the Table 7.

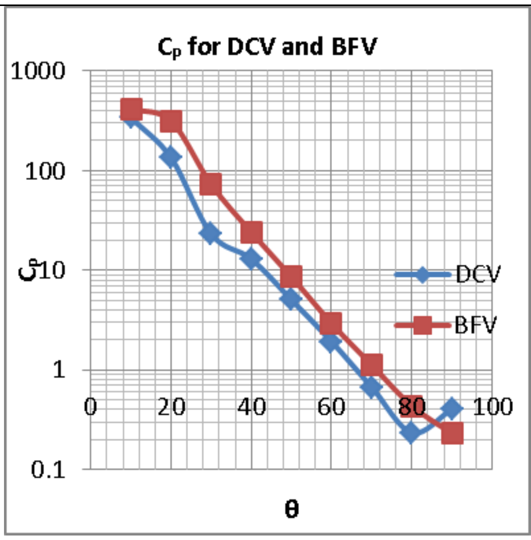
Table 7 : Comparison of C_P for DCV and BFV at Different Valve Openings

Θ	C_P for DCV		C_P for BFV	
	Re=50000	Re=1000	Re=50000	Re=1000
10°	340.7	564.693	419.092	605.910
20°	134.44	197.354	314.47	386.137
30°	23.135	30.789	72.79	88.368
40°	13.008	16.607	24.074	29.430
50°	5.044	6.885	8.622	11.344
60°	1.93	2.81	2.934	5.433
70°	0.67	1.25	1.12	2.040
80°	0.235	0.66	0.436	1.110
90°	0.406	0.832	0.23	0.447

Comparative graphs of θ v/s C_P for both flows are shown in graphs 5 and 6. From the comparison it is identified that increase in valve opening angle from fully closed condition (normal to axis) decreases the pressure loss and pressure loss coefficient at each opening angle for both the valves in both Reynolds numbers.



Graph 5: Variation of C_P for both DCV and BFV at Re=1000.



Graph 6: Variation of C_P for both DCV and BFV at Re=50000.

Dual plate check valve offers lesser pressure drop when compared to butterfly at each valve opening except at fully opened position (90°-Parallel to axis). This is due to overlapping of both the plates at fully opened position which offers high resistance for the flow. From this it is concluded that at any valve opening, dual plate check valve is preferred over butterfly valve where ever lesser pressure drop is required.

2. REYNOLDS NUMBER

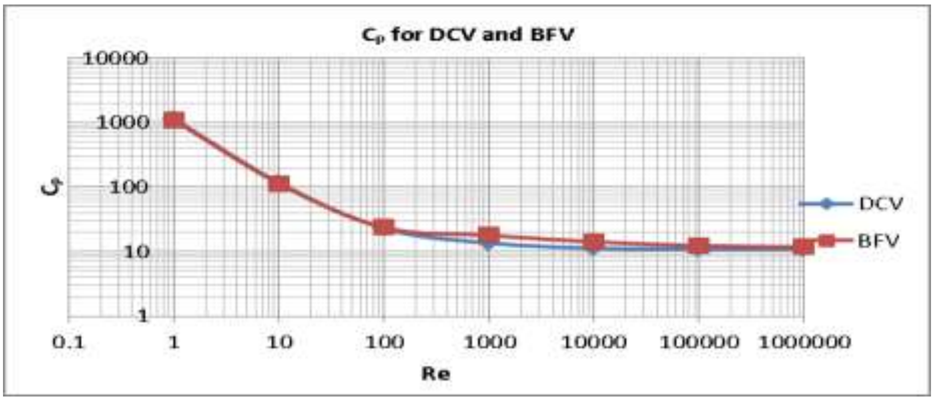
To study the variation in pressure drop, different Reynolds number are chosen from 1 to 1000000 in this case and corresponding pressure loss coefficient are tabulated in table 8.

From the tabulated data it is verified that, at lower Reynolds number (Re=1), dual plate check valve offers more resistance to the flow than butterfly valve.

Table 8 : C_P for Dual Plate Check Valve and Butterfly Valve at Different Re (θ=45°)

Re	C_P for DCV (Sharp Edge)	C_P for BFV (Fillet Radius 1.5 mm)
1	1142.264	1112.255
10	116.760	113.93
100	22.899	23.72
1000	13.67	17.90
10000	11.33	14.22
100000	11.125	12.36
1000000	11.052	11.82

As the Reynolds number increases pressure loss starts to decrease for both the valves and dual plate check valve gives better performance than butterfly valve at turbulent flow. But more or less variation in pressure loss coefficient becomes very less or all most constant at high Reynolds number for both the valves and it is depicted in graph 7.



Graph 7: Variation of C_P for both DCV and BFV at Different Reynolds Number

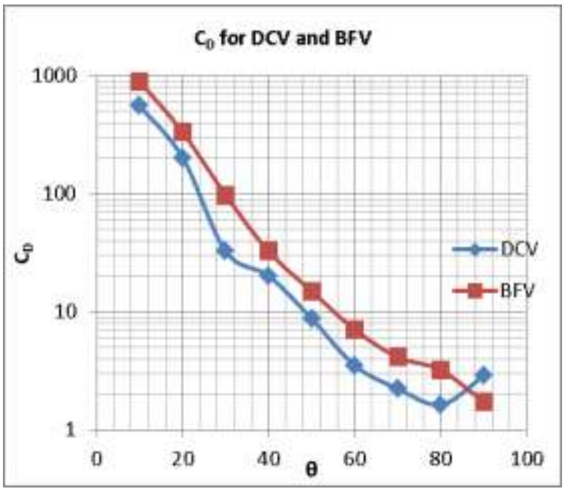
3. Drag Coefficient (C_D)

In this comparison C_D is calculated for both valves at each valve opening angle for two Different Reynolds number and tabulated in the Table 9.

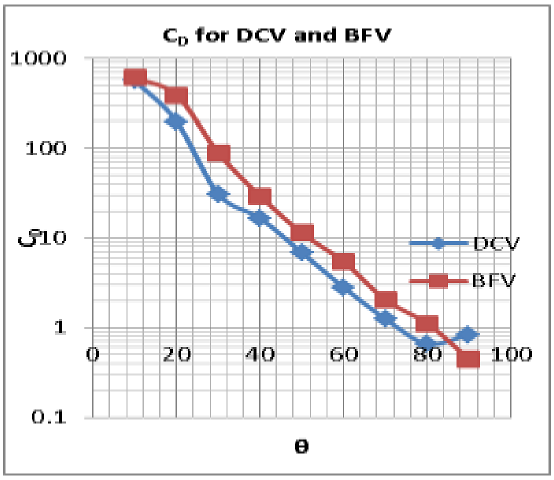
Table 9 : Comparison of C_D for DCV and BFV at Different Valve Openings

Θ	C_D for DCV		C_D for BFV	
	Re=50000	**Re=1000**	**Re=50000**	**Re=1000**
10°	341.41	557.639	413.852	896.238
20°	190.96	201.275	383.544	331.62
30°	25.42	32.489	82.116	97.06
40°	15.538	20.105	26.53	32.429
50°	6.89	8.727	12.278	14.922
60°	2.663	3.511	4.99	7.071
70°	1.492	2.23	2.718	4.119
80°	0.75	1.635	1.78	3.212
90°	1.592	2.862	0.892	1.713

For both Reynolds number the trend of variation of C_D with respect to "θ" is same i.e. as the valve opening angle increases drag coefficient decreases due to reduction in disturbance to the flow. These variations can be clearly visualized in Graphs 8 and 9.



Graph 8: Variation of C_D for both DCV and BFV at Re=1000



Graph 9: Variation of C_D for both DCV and BFV at Re=50000

The Graphs clearly indicate that dual plate check valve offers lesser drag in all valve opening angles except at fully opened condition. This is due to overlapping of dual valve plates at fully opened condition. It is observed at all Reynolds number and θ less than 80° the dual plate check valve given lower pressure drop.

VII. CONCLUSIONS

In the present study, validated CFD methodology has been used to analyses the flow through the dual plate check valve and butterfly valve. By keeping Reynolds number, valve opening, fillet radius as the variables data on the variations in pressure loss coefficient and drag force coefficients are estimated.

From the above estimation following conclusions are drawn.

- The value of Pressure loss coefficient (C_P) and drag coefficient (C_D) tend decreases with increment in both Reynolds number and valve opening angle for dual plate check valve and butterfly valve.
- Dual plate check valve offers lesser pressure drop and drag in all the cases except at fully opened condition due to overlapping of dual plates.
- Providing fillet at the edge of the disc reduces Pressure loss coefficient (C_P) and drag coefficient (C_D) both dual plate check valve and butterfly valve and increment in fillet radius tend to decrease the C_P and C_D.
- Compared to butterfly valve, dual check valve offers lesser pressure drop and drag hence it is preferred over butterfly valve.

REFERENCES

[1] Naveen Kumar C, Dr. V Seshadri, Yogesh Kumar K J "Parametric Study of Flow Characteristics of Butterfly valve using CFD". International Journal of Emerging and Technology and Advanced Engineering (IJETAE), Volume-5, Issue-11, November 2015

[2] Arun Azad1, Deepak Baranwal2, Rajeev Arya3, Nilesh Diwakar4 "Flow Analysis Of Butterfly Valve Using CFD". International journal of Modern Engineering Research (IJMER) | Vol. 4 | Iss. 6| June. 2014

[3] Xue guan song, Young Chul Park. "Numerical Analysis of butterfly valve – prediction of flow coefficient and hydrodynamic torque coefficient". World Congress on Engineering and Computer Science, USA, 2007.

[4] A.D Henderson, J.E Sargison, G.J.Walker, J Haynes. "A Numerical Study of the Flow through a Safety Butterfly Valve in a Hydroelectric Power Scheme". 16th Australasian Fluid Mechanics Conference, Australia, pp.1116-1122,2007

[5] Aniruddha V Kapre1, Yogesh Dodia2 "Flow Analysis of Butterfly Valve Using CFD". International Journal of Research in Engineering and Technology. Volume: 04 Issue: 11 | Nov-2015

[6] G.Tamizharasi, S.Kathiresan. "CFD Analysis of a butterfly valve in a compressible fluid" by International Journal Of Engineering Trends And Technology (IJETT) – Volume 3 Issue 2 No3 – March 2012

[7] A Dawy, A Sharara, A Hassan. "A Numerical Investigation of the Incompressible flow Through a Butterfly Valve". International Journal of Emerging technology and Advanced Engineering, Egypt, Vol-3, pp.1-7, 2013

[8] Ghaleb Ibrahim, Zaben Al-Otaibi, Husham M.Ahmed. "An investigation of Butterfly valve flow characteristics using numerical technique". Journal of Advanced Science and Engineering Research, Kingdom of Bahrain, Vol-3, pp.151-166, 2013.