# PREDICTION OF FLOW CHARACTERISTICS THROUGH A CIRCULAR PORT OF A SPOOL VALVE USING CFD APPROACH

**Mallikarjuna.V.Bidari[1], Mujeebulla Khan Guttal[2]**

[1]*Lecturer, Mechanical Engineering Department, S.T.J.I.T, Karnataka, India, malli.vb@gmail.com*

[2] *Lecturer, Mechanical Engineering Department, S.T.J.I.T, Karnataka, India, mujibguttal@gmail.com*

## Abstract

*Hydraulic spool valves are used in a variety of industrial equipment's like earth moving machinery, aircrafts and machine tools etc. A hydraulic spool valve is a switching device used for controlling hydraulic devices. A spool valve can turn the flow of hydraulic fluid from a hydraulic pump to an actuator in forward or reverse directions or on and off by blocking offthe route of the fluid takes. A controller moves the valve back and forth in its case to slide the spools into different positions. As the spool moves across the inlet port, the port which is initially fully open, circular in shape and permitting the fluid flow, is getting closed gradually. Once the spool starts moving, the inlet port becomes non circular and continues to be so till closure. The shape and the area of the port that is still left open for the oil to flow is steadily changing. The area which is originally circular is becoming far away from circularity. It is required to study the flow in such port-spool combination. The purpose of the dissertation is to compute the flow for different port openings. ANSYS CFD FLOTRAN software is used to predict the flow characteristics. The exact flow path is simulated. The predicted results are compared with the analytical calculations.*

***Key words****- Spool valve, Simulation, Flow Characteristics, CFD*

---

## 1. INTRODUCTION

Spool valves are widely used in industrial equipment's like earth moving machinery, aircrafts and machine tools for throttling (control) services. Valves are necessary to control the pressure, flow rate and direction of the fluid. Hydraulic valves are made to a high standard of quality and robustness. Hydraulic valves are made of strong materials (e.g. steel) and are precision manufactured. A spool valve consists of three basic components: the valve Housing (body), spool valve casing, and the supporting shaft (push rod) which are operated by means of cams or electrically operated solenoids as shown in Fig-1.

In this paper, Fluent software is applied to get numerical simulation for main Spool valve for different strokes at inlet, get visual graphic results, pressure distribution and velocity distribution and flow fields are analysed[1].

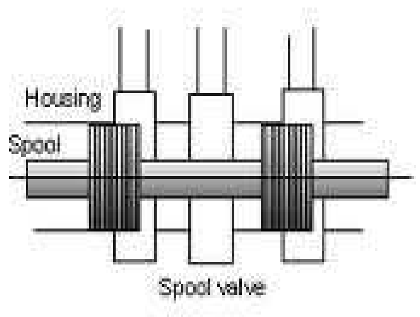


**Fig-1:** Basic view of spool valve

## 2. PROBLEM DESCRIPTION

In the past, people conduct experiments to analyse flow characteristics of spool valves in the laboratory, which require a number of equipments, a lot of time, funds and so on. Nowadays, due to the fast progress of computer visualization and numerical technique, it becomes possible to do analysis using simulation technique. A computational study of spool valve to validate the ability of accurately predicting flow characteristics of spool valve using computational fluid dynamics.

Due to literature, a recent survey on spool valves flow characteristics indicates that most of the previous studies were performed experimentally within the turbulent flow region. In contrast, the data obtained numerically is relatively spare. Moreover, almost no information is available concerning spool valves flow characteristics in the laminar flow region. A hydraulic spool valve is a small cylinder inside a sealed case. It usually has valves leading to the pump and the tank on one side, and valves leading to one or more hydraulic devices on the other side. Fluid at pressure can flow into the valve from the pump into the hydraulic devices or drain out of them back into a hydraulic storage tank. A controller moves the valve back and forth in its case to slide the spools into different positions. Hence from these flow regions we are finding the co-efficient of discharge (Cd) for different port openings and pressure drop ($\Delta P$) across the valve.

## 3. OBJECTIVE

The aim of this work is to calculate numerically both laminar and turbulent flow characteristics for typical spool having thin sharp edge, port for different openings. ANSYS CFD FLOTRAN software is used to predict the flow characteristics. The exact flow path is simulated. The predicted results are compared with the analytical calculations. To compute the co-efficient of discharge (Cd) such that spool moves across the inlet port at different positions and pressure drop (ΔP) across the valve.

## 3. RESEARCH METHODOLOGY

Created flow regime, Finite element model creation of geometry and used CFD FLOTRAN software on ANSYS 11.0 platform. Based on the above methods apply proper boundary conditions in order to set the parameters, solve the problem and execute the results.

The elements used in the present work FLUID 142 can be used to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions [2].

## 4. FLOW BOUNDARY CONDITIONS

Each boundary of the problem domain requires treatment. Can be specifying some combination of DOF (VX, VY, VZ and PRES) at the boundary types listed below.

**Table-1:** Boundary conditions at intersections

| Intersecting Boundary 1 | Intersecting Boundary 2 | Action |
|---|---|---|
| Inflow: VX, VY, VZ val ues specified | Wall: VX, VY, VZ = 0 | Overwrite inflow with wall |
| Inflow: VX, VY, VZ val ues specified | Symmetry: VX or VY or VZ = 0 | Combine inflow with symmetry |
| Inflow: VX, VY, VZ val ues specified | Outflow: P = 0 | Enforce inflow condition, not outflow |
| Symmetry: VX or VY or VZ = 0 | Outflow: P = 0 | Combine symmetry and outflow |
| Symmetry: VX or VY or VZ = 0 | Wall: VX, VY, VZ = 0 | Overwrite symmetry with the wall |
| Wall:VX, VY, V Z = 0 | Outflow: P = 0 | Combine wall and outflow condition |
| Generalized Symmetry | Outflow: P = 0 | Combine the generalized symmetry and outflow conditions |

## 5. MODEL DESCRIPTION

### 5.1 CFD Model Setup

The differential equations that govern a flow of Newtonian fluids based on the Navier-Stokes equation. It can be compactly expressed in vector notation as

$$\rho\left(\frac{\partial V}{\partial t} + V.\nabla V\right) = -\nabla p + \rho g + \mu\nabla^2 V \qquad (1)$$

Along with the continuity equation

$$\nabla . V = 0 \qquad (2)$$

The momentum conservation equation of fluid mechanics

$$\frac{\partial(\rho u)}{\partial t} + \mathrm{div}(u\mathbf{u}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + F_X \quad (3)$$

$$\frac{\partial(\rho v)}{\partial t} + \mathrm{div}(v\mathbf{u}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + F_y \quad (4)$$

$$\frac{\partial(\rho w)}{\partial t} + \mathrm{div}(w\mathbf{u}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + F_z \quad (5)$$

Where p, τ, F is the Pressure force, Viscous stress and Body force respectively [3].

### 5.2 Numerical Model Description

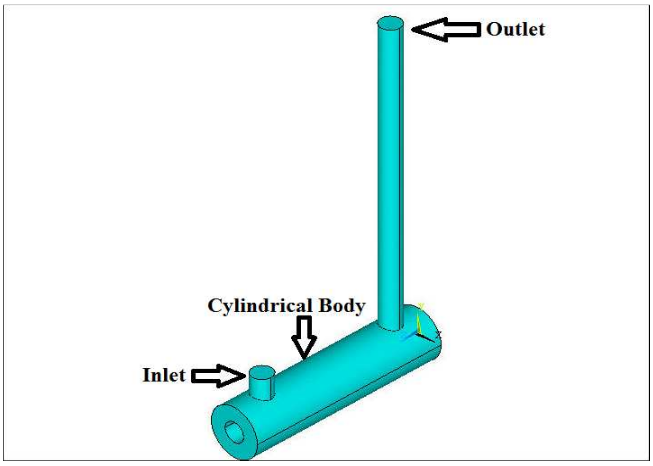


**Fig-2:** 3D model view of spool valve

A schematic of the investigated spool valve is presented in Fig-2. Dimensions of the valve were as follows: Length $L_1$= 100 mm, $L_2$= 10 mm, $L_3$= 150 mm, Inlet and Outlet Diameter of the port $d_1$ and $d_2$= 10 mm, Spool Valve Cylinder outer Diameter $D_1$= 25 mm, Spool Valve Cylinder inner Diameter $D_2$= 10 mm and Spool stroke at inlet of the port A= 0, 2, 4, 6, and 8 mm. For getting a better result in this simulation, the CFD model of spool valve is created.

The fluid upstream and downstream far from the valve is so well-proportioned, by contraries, the fluid downstream near the valve is in disorder. This phenomenon also validates that both the upstream length and the downstream length should be long enough [4].

In this work, for the reason of accuracy and convenience, originally, $L_2$ is set to diameter d, and $L_3$ is set to 15 times the diameter d as shown in Fig-2. There are no reverse flows near the outlet. Meanwhile, the error of the average velocity between is set to diameter d, at upstream and 15 times the diameter d, at downstream, which indicates that the length of the additional pipe can satisfy the accuracy required of the simulation [5].

Some of the assumptions made to solve RANS equation to reach the boundary and grid.

- The flow is steady-state, three dimensional, isothermal, laminar (for Re≤2300), and turbulent (for Re≥2300)
- Inflow and Outflow pipe walls are hydraulically smooth.
- The fluid is Newtonian and incompressible.

A trial and error method was used to reach the boundary and grid independent results [7]. Finally, the RANS equations were solved using the following conditions.

- According to the research of Huang and Kim (1996) [6], the upstream pipe length and the downstream pipe length which are required to be at least 2 times and 8 times of pipe diameters respectively.
- Fully developed laminar or turbulent pipe velocity profile at the entrance cross section.
- Von Neumann boundary condition at the outflow cross section.
- No-slip condition on the wall surfaces.

The pressure distributions $P_1$ at inlet and $P_2$ at outlet. The difference in pressure drop $\Delta P = P_1 - P_2$ across the valve from inlet to outlet have been determined. The Ideal pressure drop across the valve is calculated using formula:

$$\Delta P_{ideal} = \frac{(f \times L \times W \times V_{avg})}{(D_e \times 2g)} \qquad (6)$$

Where, L = L1+L2+L3, is the length of flow in m
W = Specific weight in $kg/m^2$-$sec^2$
$V_{avg}$ = Average outlet Velocity in m/sec
De = Equivalent Diameter in m
f = Friction factor for pipes
g = Acceleration due to gravity in $m/sec^2$

The flow characteristics of a spool valve can be expressed as follows.

**Outlet Discharge** $Q_{outlet} = A_{out} \times V_{avg}$ (7)

Where $Q_{outlet}$ = Discharge of a Spool Valve at outlet
$A_{out}$ = Outlet flow area
$V_{avg}$ = Average outlet velocity

**Ideal discharge** $Q_{ideal} = A_{in} \times V_{ideal}$ (8)

Where $Q_{ideal}$ = Discharge of a Spool Valve at Inlet
$A_{in}$ = Inlet flow area
$V_{avg}$ = Average Inlet velocity

The **coefficient of discharge** of a Spool valve (Cd) can be expressed as follows

$$Cd = \frac{Q_{outlet}}{Q_{ideal}} \qquad (9)$$

## 6. ANALYSIS AND RESULTS OF FLOW CHARACTERISTICS THROUGH A SPOOL VALVE

After the numerical simulations were carried out in many conditions of the flow through a spool valve, the characteristics of flow-fields and the flow characteristics are analysed. The topics in this chapter consist of velocity distribution, pressure distribution, flow-fields, characteristics of discharge co-efficient and these numerical simulation results are predicted. The exact flow is simulated. The predicted results are compared with the analytical calculations.

### 6.1 Initial model analysis

The wall is 3D symmetric which provides a view ofthe unstructured computational grid employed, meshed by tetrahedral element with 40000 nodes. Both accuracy of the analysis and the time taken also depend on the mesh size. Hence optimum mesh determination is an important step in CFD analysis. The three dimensional analysis would yield a reasonable qualitative result if the effective model were set. The flow is taken to be 3D-symmetry incompressible, Laminar and Turbulent.

The inlet and outlet of the spool valve along with boundary information required have been indicated: (1) Inlet boundary condition is set to 20MPa (2) Outlet boundary is set to 20.0256MPa (3) Density of liquid is $880 Kg/m^3$
(4) Dynamic viscosity of oil is $0.037 N\text{-}sec/m^2$(5) A no slip velocity condition at the wall.

The spool valve pressure drop is 0.0256MPa. The analysis is carried out for different strokes at inlet of the valve i.e. 0, 2, 4, 6 and 10 mm. The boundary condition for zero spool displacement and symmetric section of hexahedral mesh as shown in Fig-3 and Fig-4.
The results are obtained for different valve openings at inlet. The Ansys 11.0 CFD Flotran Software is used for analysis of results.

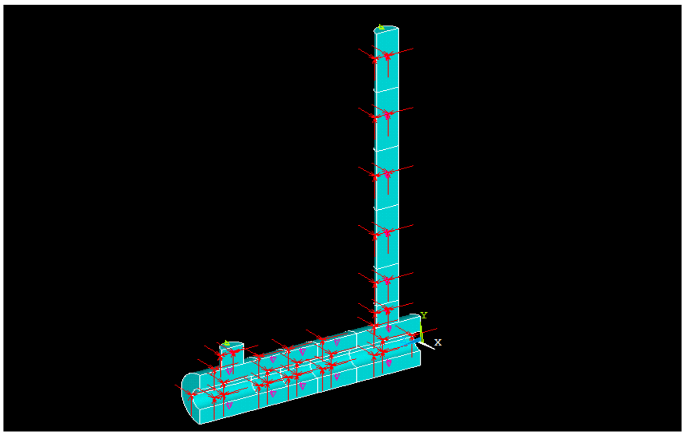

**Fig-3:** The Boundary condition for zero spool displacement

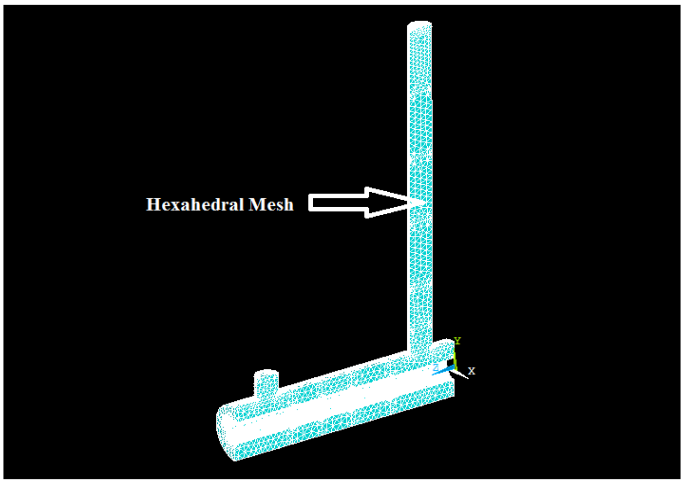


**Fig-4:** Symmetric section of hexahedral mesh Spool valve

## 6.2 Mesh of Geometry Model

Based on the guidance of ANSYS Flotran CFD according to valve openings there are several simulations with different meshing sizes. The details about the mesh and calculations are listed.

The three-dimensional models of the spool valves are composed of valve body, valve housing and valve port using pre-processor by ANSYS Flotran CFD. To improve the computational efficiency, the nodes of the grid were clustered in the valve body compare to inlet and outlet relatively. For each calculation cases, the unstructured grids were generated shown in Table-2.

**Table-2:** Various mesh sizes and computer calculation information of different strokes at inlet position.

| Stroke at inlet port in m | Memory space MB | Volume | Nodes | Elements |
|---|---|---|---|---|
| 0 | 33 | 9 | 40962 | 197157 |
| 0.002 | 25 | 13 | 32027 | 152405 |
| 0.004 | 25 | 13 | 30241 | 140962 |
| 0.006 | 25 | 13 | 30161 | 140383 |
| 0.008 | 25 | 13 | 29921 | 139363 |

## 7. RESULTS AND DISCUSSION

### 7.1 Velocity Distribution from Vector Plot

For the spool valve of 25 mm outer diameter, 10 mm inner diameter of cylinder having inlet, outlet pipe diameters are 10 mm. The simulation for analysis work done at 5 positions of valve opening at different strokes namely 0, 2, 4, 6 and 8 mm for same inlet pressure $P_1$=2.00256×$10^7$N/$m^2$ and for same outlet pressure $P_2$=2.00×$10^7$N/$m^2$. In each position with oil as flowing fluid. From the investigation of the flow through this valve, it is found that velocity is a function of both the value of pressure and position of port at inlet of the spool valve.

The distributions (flow patterns) in this part will be represented by velocity vectors. This helps to display the velocity value and direction of the fluid around the valve.

Fig-5, 6, 7, 8 and 9 shows the velocity distribution of fluid around the spool valve for diameter of 25 mm. It is found that velocity values are proportional to the mass flow rate. The high gradient of fluid velocity appears near the inlet port opening due to change in the stroke positions. Also in almost closed position at inlet port, the velocity value becomes high between the pipe surface and spool valve.

The wakes are formed near the inlet port at different strokes. The flow of fluid or oil across the annular space cylinder body of the spool valve. During the spool movement for different port openings the vortex flow around the cylinder body become bigger at zero spool displacement. For the position of 8 mm stroke, it is found that the vortex will decreases around the cylinder body and the velocity value was low in the horizontal plane.

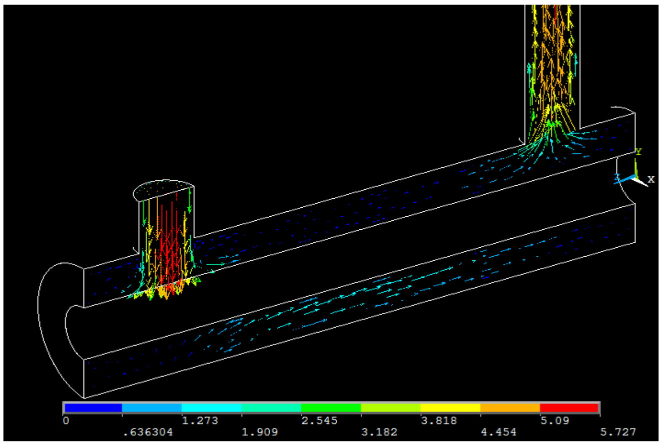


**Fig-5** Vector plot for zero spool displacement

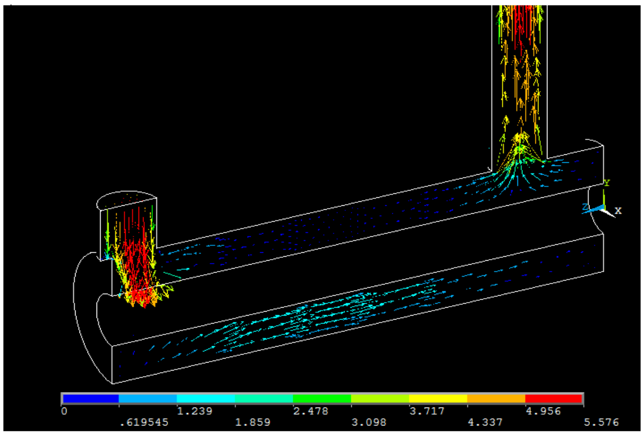


**Fig-6** Vector plot for 2 mm stroke at inlet

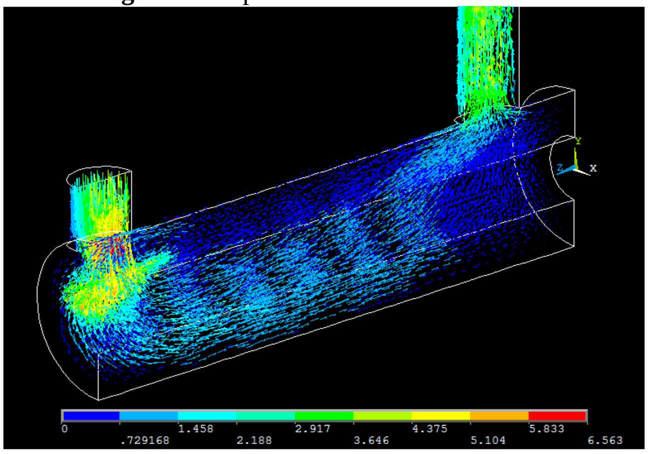


**Fig-7** Vector plot for 4 mm stroke at inlet

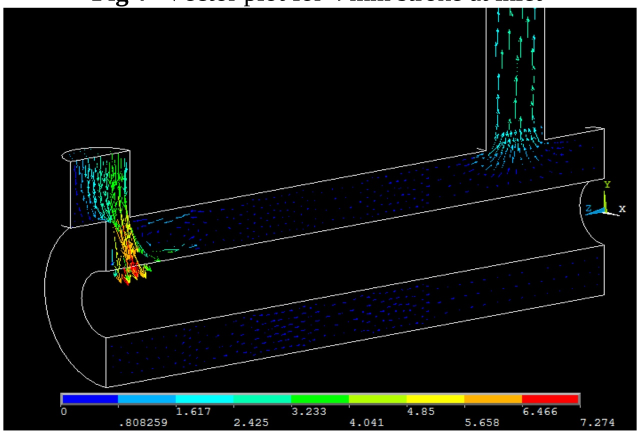


**Fig-8** Vector plot for 6 mm stroke at inlet

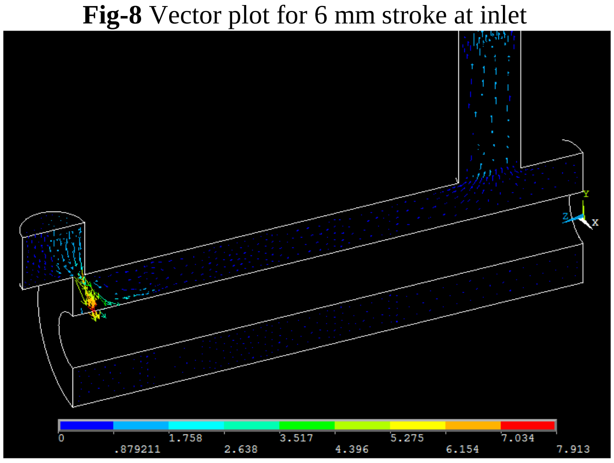


**Fig-9** Vector plot for 8 mm stroke at inlet

### 7.2 Flow characteristics

The values of pressure drop across the spool valve were investigated using commercial fluid dynamics software ANSYS FLOTRAN CFD. It is calculated in order to compute the dimensionless value co-efficient of discharge (Cd).

The value co-efficient of discharge for different strokes at inlet port i.e. 0, 2, 4, 6 and 8 mm respectively. It is found that the Cd values are dependent on the flow rate but affected by changing the port openings at inlet. By increasing the strokes at inlet, the value of co-efficient of discharge (Cd) becomes greater.

**Table-3:** The values of outlet discharge ($Q_{out}$) for corresponding stroke at inlet

| Sl. No. | Stroke in m | Outlet Area of Flow ($A_{out}$) in $m^2$ | Average Outlet Velocity($V_{avg}$) in m/s | Outlet Discharge ($Q_{out}$) in lpm |
|---|---|---|---|---|
| 1 | 0 | $7.853\times10^{-5}$ | 2.9477 | 13.88 |
| 2 | 0.002 | $7.853\times10^{-5}$ | 2.8768 | 13.55 |
| 3 | 0.004 | $7.853\times10^{-5}$ | 2.4032 | 11.32 |
| 4 | 0.006 | $7.853\times10^{-5}$ | 1.6008 | 7.542 |
| 5 | 0.008 | $7.853\times10^{-5}$ | 0.6544 | 3.083 |

**Table-4:**The values of ideal discharge ($Q_{ideal}$) at inlet

| Sl. No. | Stroke in m | Area of Flow ($A_{in}$) in $m^2$ | Ideal Velocity (V) in m/s | Ideal Discharge ($Q_{ideal}$) in lpm |
|---|---|---|---|---|
| 1. | 0 | $7.853\times10^{-5}$ | 7.554 | 35.59 |
| 2. | 0.002 | $6.723\times10^{-5}$ | 7.554 | 30.47 |
| 3. | 0.004 | $4.864\times10^{-5}$ | 7.554 | 22.04 |
| 4. | 0.006 | $2.821\times10^{-5}$ | 7.554 | 12.78 |
| 5. | 0.008 | $0.9635\times10^{-5}$ | 7.554 | 4.366 |

**Table-5:**The values of coefficient of discharge (Cd) for different port openings at inlet

| Sl. No. | Stroke in m | $c_d = \frac{Q_{outlet}}{Q_{ideal}}$ |
|---|---|---|
| 1. | 0 | 0.. 3899 |
| 2. | 0.002 | 0.4446 |
| 3. | 0.004 | 0.5134 |
| 4. | 0.006 | 0.5899 |
| 5. | 0.008 | 0.7061 |

The co-efficient of discharge (Cd) value depends on the stroke at inlet and geometry of the valve as shown in Fig-10. The Cd values which vary from 0 to 8 mm stroke at inlet port opening of the spool valve.

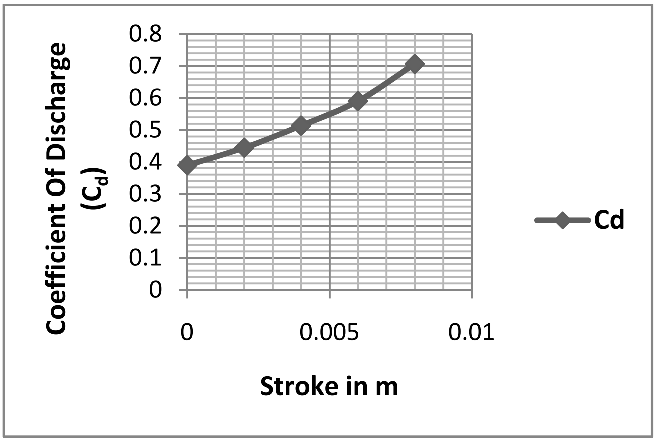


**Fig-10:** Cd Vs Stroke

Based on Ideal flow conditions the numerical results can be estimated. These results are predicted from analytical calculations. Fig-11 shows the Ideal discharge ($Q_{ideal}$) and Outlet discharge ($Q_{outlet}$) for different strokes at inlet. Ideal values of discharge are slightly higher than the simulated values. As the stroke at inlet increases, Outlet discharge values slightly decreases from simulation.

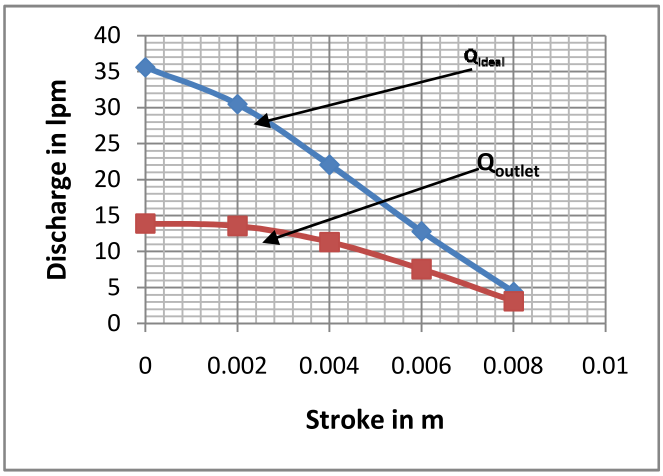


**Fig-11:** Discharge Vs Stroke

### 7.3 Flow Pattern

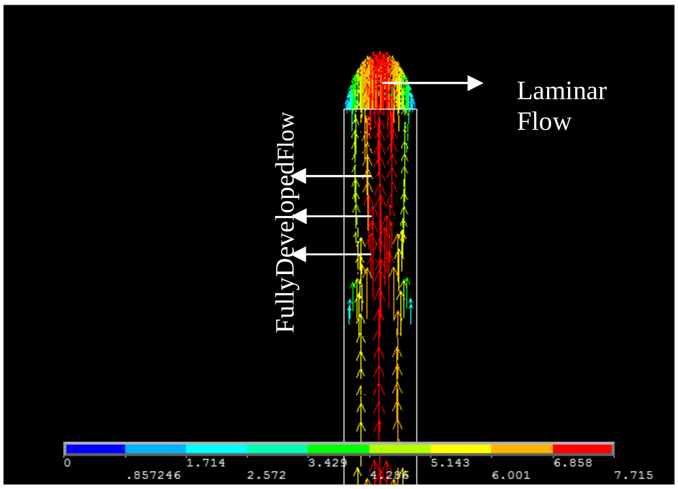


**Fig-12** Velocity vector plot of laminar flow at outlet pipe for zero spool displacement

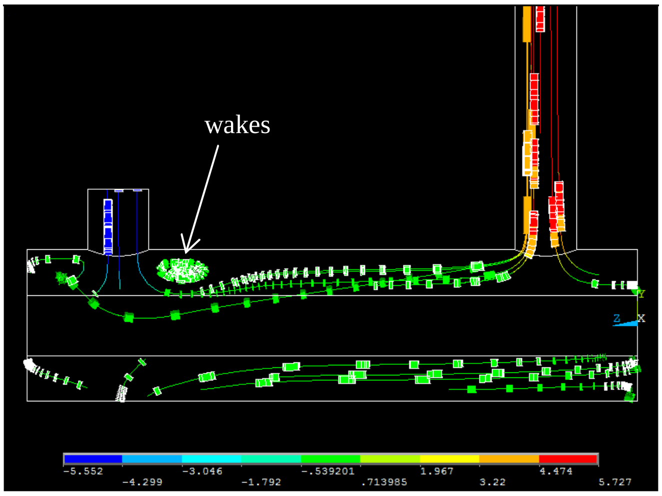


**Fig-13:** Animated flow from inlet to outlet for zero spool displacement

Flow pattern from the downstream of the port varied as the length increased. At the inlet port of the valve spool will be operated for different strokes i.e. 0, 2 mm, 4 mm, 6 mm and 8 mm. For Laminar flow the flow is seemed to have a fully developed flow pattern. Depending upon the inlet port opening, the fully developed flow pattern occurs.

This analysis helps in predicting the various strokes at inlet ofthe valve, where the flow will be fully established. This helps in determining the pressure drop across the valve. The animated flow for zero spool displacement as shown in Fig-13.

For zero spool displacement, the velocity profile in fully developed pipe flow is parabolic in laminar flow as shown in Fig-14. and but somewhat flatter or fuller in turbulent flow as shown in Fig-15.

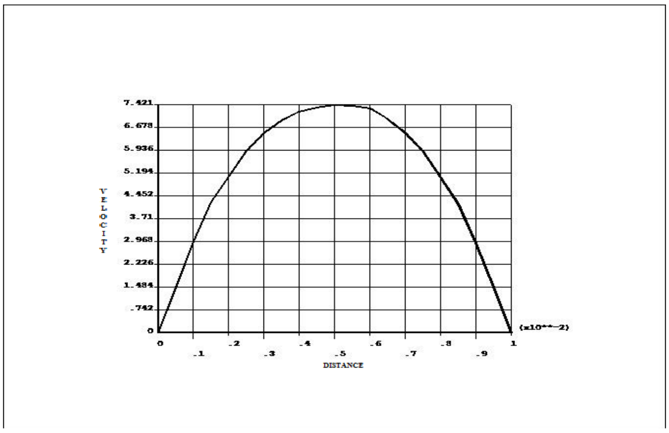


**Fig-14:** Laminar velocity profile

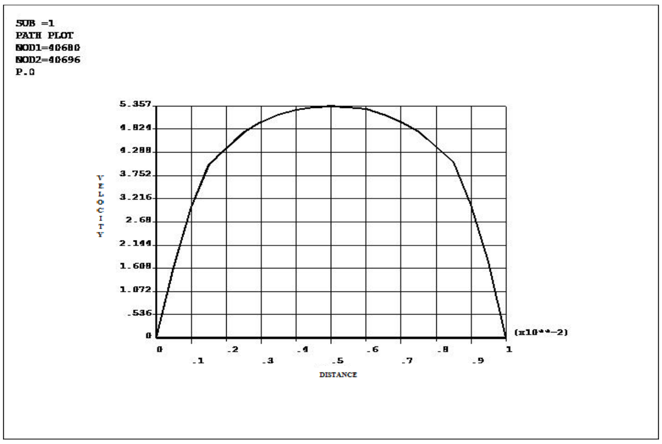


**Fig-15:** Turbulent velocity profile

## 8. CONCLUSIONS

From this present analysis the computational results obtained with FLOTRAN CFD software on ANSYS 11.0 platform agree qualitatively with the results of analytical calculations. The co-efficient of discharge (Cd) varies with increase in stroke at inlet port opening. The Cd approaches a constant value which depends on the valve port opening.

Due to sudden expansion, contraction, turbulence effects and wakes are formed at the flow field. Due to this effect the simulation result of pressure drop across the spool valve is higher as compared to Ideal flow conditions.

The computational studies have shown that the flow field through the spool valve is very complex and depends upon the valve port opening. Flow patterns in downstream at outlet pipe varies as the length increases from 15 times diameter of inlet pipe. The flow seemed to have a fully developed flow pattern for different strokes at inlet.

## ACKNOWLEDGEMENT

Special thanks go to Mr. J. Radhakrishna Murthy, Former Joint Director, Central Manufacturing Technology Institute, Bangalore and I gratefully acknowledge his support and guidance for this work.

## REFERENCES

[1] HuangHao&LeiHuihu.,"Modeling and Simulation of Flow field of Main Spool in Servo Valve" International Conference on Electronic and Mechanical Engineering and Information Technology 12-14 August 2011
[2] WenhuaJia&Chenbo Yin.," Computational Analysis of the Flow Characteristics of V-U and U-U Notch in Spools of Valves" International Conference on Computational Methods in Fluid power Technology, Volume 3, P222-225, 2010
[3] Huang1,a, Zhongwei Sun1,b, and Xiong Li1,c., "Modelling and Simulation of Valve Port in Non-circular Opening Spool Valve" Advanced Material Research, Vols.655-657 (2013) pp 476-480
[4] EC60534-2-1 1998 Industrial-process control valves flow capacity sizing equations for fluid flow installed conditions International Electro-technical Commission (Geneva, Switzerland).
[5] JanuszWojtkowiak, CzeslawOleskowicz-Popiel, "Investigations of Butterfly Control Valve Flow Characteristics", Poznan University, Poland, Sep 2006.
[6] Huang C, Kim R.H, "Three-Dimensional Analysis of Partially Open Butter-fly valve Flows", Journal of Fluids Engineering (Transactions of the ASME), 118, (1996), 562-568.
[7] Ferziger J.H., Peric M: Computational Methods for Fluid Dynamics, 2nd Ed., Berlin, Springer Verlag, 1999.
[8] ANSYS User's guide, ANSYS Inc.
[9] WenhuaJia&Chenbo Yin., "CFD Simulation with Fluent and Experimental Study on the Characteristics of Spool Valve Orifice" 2nd International Conference on Computer Engineering and Technology, Volume 1-590-594, 2010
[10] Li Zhang, Jing Luo*, Rui-Bo Yuan, Min He., "The CFD Analysis of Twin Flapper-nozzle Valve in Pure Water Hydraulic" Procedia Engineering 31 (2012) 220-227
[11] Sandor I. BERNAD*, Romeo SUSANRESIGA**., "Numerical Model For Cavitational Flow In Hydraulic Popet Valves" Modeling and Simulation in Engineering, 2012
[12] Bao Min, Fu Xin, Chen Ying., "Computational Fluid Dynamics Approach To Pressure Loss Analysis Of Hydraulic Spool Valve" The fifth International Conference on Fluid 2001
[13] JipingBai, JianRuan, ShaohuaRuan.,"Investigation of Cavitation Phenomena in The Valve Port of The High Frequency Electro-Hydraulic Digital Valve" Proceeding of the 2010 IEEE International Conference on Mechatronics and Automation August 4-7, 2010, pp. 717-722
[14] S.R. Majumdar., "Oil Hydraulics Systems", Tata McGraw-Hill, 1 Edition, 2001
[15] Johan D. Anderson, Jr. "Computational Fluid Dynamics", McGraw Hill International Edition, 2005.

## BIOGRAPHIES

Mallikarjuna V Bidari is having 2 years of teaching experience, Completed his PG in Thermal Power Engineering. He has industrial experience of 3 years in the field of Generators and Material handling equipments.

Mujeebulla Khan Guttal presently working as lecturer from past 7 years in Mechanical Engineering Department. Pursuing his Ph.D in the field of Composite Materials.