IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 03, 2014 | ISSN (online): 2321-0613

"CFD Evaluation for a Diesel Rickshaw Poppet Valve"

Patel Nimesh[1] Mr.Vinod Prajapati[2]

[1]Student[2]Assistant Professor
[1,2]L.C.I.T Bhandu

Abstract— Poppet valves are extensively used in hydraulic systems as relief valves, check valves, cartridge valves, and pressure regulators due to their obvious advantages. From the simulation it is conclude that the puppet valve satisfy requirement. From the simulation the result is generating within 10% error. This is desire for any CFD simulation. This simulation extends to check for different opening condition.

I. INTRODUCTION

Puppet valves can essentially make zero leakage; thereby do not need check valves. Fourth, as a metering device, puppet valves do not need a pilot system, because they can directly use the system pressure as their pilot pressure. Therefore, a metering poppet valve system with low cost and small passage is possible. The final highlighted advantage of puppets is that they have high pressure-sensitivity to the valve displacement. A relatively small uplifting movement of the poppet can bring a large orifice area. However there are two primary disadvantages of poppet valves. First, instability of the system is easy to occur accompanying with self-excited oscillations. Instability can be caused by many influences, for example, the unsteady flow forces through the poppet, uncertainty of discharge coefficient, upstream configurations, fluid delivery lines and the interference of the valve motion with other system components. Second, compared with the spool valves, the poppet valves cannot balance the pressure forces acting on the entire poppet by itself due to its geometry. Usually a compressed spring with preload is installed on the land side of the poppet to push the poppet tightly contact with the seat. Therefore, a poppet valve is very sensitive to fluid characteristics of the upstream and downstream of the system.

The performance of a hydraulic control system is strongly influenced by the dynamic characteristics of its control valves. The function of a control valve is to change the flow paths in hydraulic circuit, in particularly to open or to close the flow path. The control valve is one of the most expensive and sensitive parts of a hydraulic circuit and therefore must be designed very carefully. In a classical approach, a many experimental measurements on the specific valve are needed. Alternatively, a numerical analysis can be performed by means of computational fluid dynamics (CFD) codes. Computational fluid dynamics (CFD) is becoming a well established practice also in valve analysis and design [1], since it can give a clear insight into its operation mode, which cannot be achieved by experimental tests and measurements. CFD also allows a reduction in the number of prototypes under test, as well as the time and costs of the design and of the experimental phase. Objective of this work was to predict pressure acting on valve and seat at different opening of valve. Experimentally and numerically comparison has been flowing pressure. Experimentally and numerically investigation of different seat angle effect.

From the literature found that J.A. Stone's work [2] showed that the downstream configuration (e.g., the length and diameter of the chamber) had a strong influence on the system stability. Experiments were also carried out on the discharge coefficient with the changes of the poppet angles and shapes, and poppet flow characteristics with various exhaust conditions that contains different liquids. Although, some researcher work on numerical techniques applying to puppet valve cavitations study and pressure drop analysis. But no one found effect of varying puppet valve angle and shapes numerically. So, consider this as a research objective for this thesis.

II. PRACTICAL SETUP

As seen below figure 2.1 incompressible fluids enter through inlet port at this port pressure gauge fitted. Pressure adjustment handle provided at the top of the puppet with the help of this handle vary spring tension and this tension of spring directly affected on pressure variation as well as eddies formation at outlet port. So, measure reading of inlet and outlet pressure variation. Perform experiment to measure reading with varying seat angle.

As seen below figure 2.1 incompressible fluids enter through inlet port at this port pressure gauge fitted. Pressure adjustment handle provided at the top of the puppet with the help of this handle vary spring tension and this tension of spring directly affected on pressure variation as well as eddies formation at outlet port. So, measure reading of inlet and outlet pressure variation. Perform experiment to measure reading with varying seat angle.

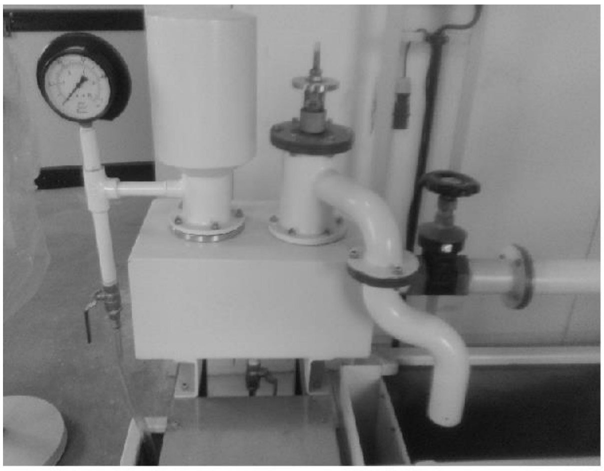

Fig. 1: Practical setup

III. EXPERIMENTAL PROCEDURE

A quantity of water is first allowed to pass through a long column of pipe connected to the puppet valve assembly and discharged through a outlet valve. The momentum of the water flowing through the pipe is then suddenly destroyed by the automatic closing of the waste valve which pumps a small quantity of water to high head tank. When the moving column of water is brought to rest, the waste valve opens and the cycle is repeated automatically.

The efficiency of valve can be determined by following formulae:

A. Aubuisson's efficiency

$$\eta_A = \frac{q h_d}{(q+Q) h_s} \text{ X } 100\% \qquad (3.1)$$

B. Rankine's efficiency

$$\eta_R = \frac{q h_d}{Q h_s} \text{ X } 100\% \qquad (3.2)$$

The puppet valve assembly used in milk industries dies industries and many more industries where flow control is main control parameter.

The experimental set up consists of a puppet valve having a cylindrical air vessel connected to a small rectangular chamber through a non-returning valve. A waste valve is also provided in the rectangular chamber to discharge the excessive water to the sump tank. A pressure gauge is provided for measuring the pressure. Measuring tank and stop watch is provided to measure the discharge of waste water. Discharge of useful water is measured by measuring cylinder and stop watch.

Close all the valves provide. Fill sump tank ¾ with clean water and ensure that no foreign particles are there. Now switch ON the main power supply and switch ON the pump, adjust the puppet valve stroke at minimum. Open control valve of puppet and adjust stroke of ram to vary the head developed by the ram. Partially open flow inlet valve provided at useful water discharge line and Record pressure gauge reading. Measure flow rate of useful water and waste water discharged by the puppet assembly valve using stop watch, measuring cylinder and measuring tanks. Repeat experiment at different flow rates of useful water discharged by the puppet valve assembly by regulating the control valve. The experimental value was shown in table 3.1

Table No. 3.1 Experimental reading

V (ml)	t_1 (sec)	R_1 (cm)	R_2 (cm)	t_2 (sec)	P (kg/cm^2)
120	59	7.6	3	15	0.0026
280	21	11	3	15	0.0032

IV. COMPUTATIONAL FLUID DYNAMICS METHODOLOGY

CFD – FLUENT is used for the modeling and simulation in this project. CFD – FLUENT is computer software that allows modeling and simulation of flow of fluid and heat and mass transfer in complex geometries. It is capable to complete meshing flexibility, solving flow problems with unstructured meshes that can be generated through the complex geometries. The program structure is shown in Figure 4.1.

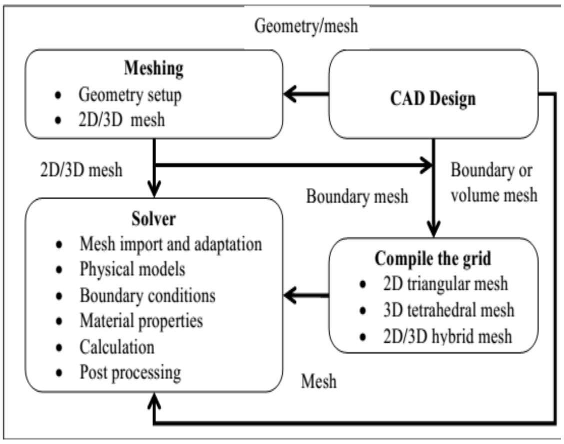

Fig.4.1: program structure.[3]

In CFD analysis geometry creation is a first step to create physical domain. This domain can help for further simulation. So, it is difficult task to make geometry having fluid domain. Solid works Design modeler is using for geometry creation. It is shown in Figure 4.2

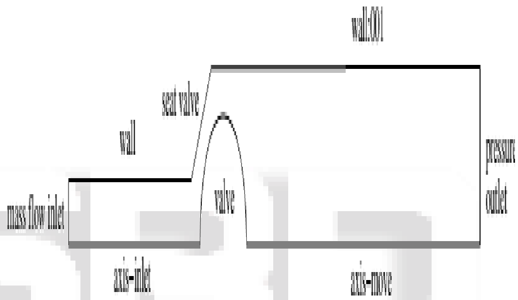

Fig.4.2: geometry modeling

Meshing is a part of modeling. After complete geometry mesh will be applying on geometry. Meshing was required element size, element type and element connectivity in terms of skewness which can be set in ANSYS mesh modeler. Complete step of mesh generation not down element number and node number which can use for grid independency study. Mesh of above figure 4.2 is shown in figure number 4.3.

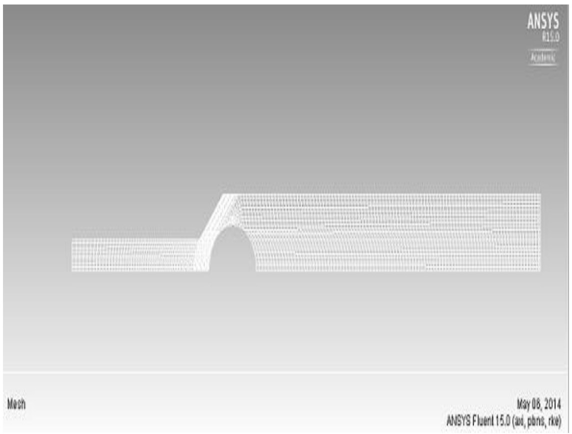

Fig.4.3: Meshing of geometry

In this simulation model set as time steady and applying solver-pressure coupling method. Material- Liquidwater at 27^0 C and Physical Model set k-epsilon turbulence with energy model. 2D axi-symmetric puppet valve geometry is used. A mass flowrate inlet and pressure outlet is used. The valve is not driven by the flow in this case. Instead, a prescribed motion is used. The motion of the check valve ball is limited to a small distance and hence, the spring smoothing approach is suitable. The motion of the ball is prescribed by a user defined function (UDF). The mesh axis is set to deforming to prevent formation of skewed cells at the intersection of the puppet valve and the axis. These simulations perform in steady and transient condition. After finished simulation task in FLUENT. Its generating data this data use for further evaluation purpose and further improvement in results.

V. RESULT AND DISCUSSION

From the simulation data it is shown that the position of valve has been change due to spring tension. It is directly affect on pressure and velocity of flow. This pressure match with experiment pressure, from this comparison simulates the effect of seat angle on flow condition. The position of valve effect on flow also consider in this simulation. Simulation performs on linear translation of valve respect to seat at fully open position, intermediate position and closing position of valve.

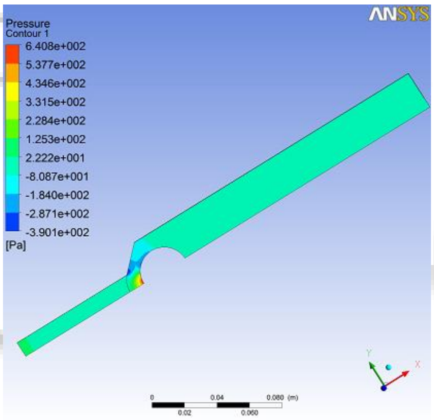

Fig.5.1: Position of Valve 0.00 translation or fully open

This plot was presented the pressure at different position of axi-symmetry model. From this trajectory it was seen that the high pressure at the valve surface and the pressure drop between seat and valve. The pressure was again buildup in downstream side. Similarly different valve position presented below figure No. 5.2 to 5.3.

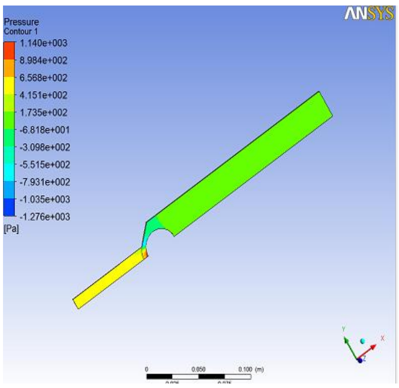

Fig.5.2: Position of Valve 0.10 translation or partially open

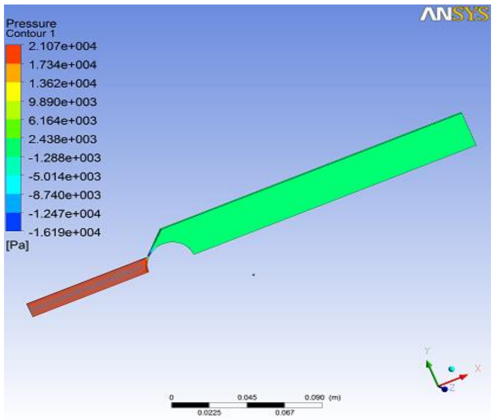

Fig.5.3: Position of Valve 0.15 translation or fraction of gap between seat and valve

In this contour very fraction of gap between valve and seat. This kind of condition was generated at maximum pressure on upstream side of flow. This effect was occurred because of back flow due to fraction valve opening. In downstream side pressure is very low.

VI. CONCLUSION

From the simulation it is conclude that the puppet valve satisfy requirement. The purposes of this project to check valve efficiency for satisfy working. Experimental setup runs at different mass flow rate with adjusted speed by motor. To check the deliver pressure. This deliver pressure simulate with CFD in FLUENT. From the simulation the result is generating within 10% error. This is desire for any CFD simulation. This simulation extends to check for different opening condition.

REFERENCES

[1] Domagaáa, M., Metodyka modelowania zaworów maksymalnych, Praca Doktorska, Politechnika Krakowska, Instytut Informatyki Stosowanej, Kraków 2007.
[2] Stone, J. A., Discharge Coefficients and Steady-State Flow Forces for Hydraulic Poppet Valves, Transactions of the ASME, 82, p.144-154, March, 1960
[3] Fluent 6.2 user's guide, Ansys Corporation. 2002.