

# Numerical Simulation of a Pressure Regulated Valve to Find Out the Characteristics of Passive Control Circuit

Binod Kumar Saha

**Abstract**—The objective of the present paper is a numerical analysis of the flow forces acting on spool surfaces of a pressure regulated valve. The transient, compressible and turbulent flow structures inside the valve are simulated using ANSYS FLUENT coupled with a special UDF. Here, valve inlet pressure is varied in a stepwise manner. For every value of inlet pressure, transient analysis leads to a quasi-static flow through the valve. Spool forces are calculated based on different pressures at inlet. From this information of spool forces, pressure characteristic of the passive control circuit has been derived.

**Keywords**—Pressure Regulating Valve, Spool Opening, Spool Movement, Force Balance, CFD.

## I. INTRODUCTION

**P**RESSURE regulated valve is a critical component used in many fluid power system specially in aircraft. The valve is used in Environmental Control System of the aircraft. It is installed in a pipe line in an air flow path where available inlet pressure to the valve may vary within a wide range. The sole function of the valve is to regulate the air flow and maintain almost constant pressure at its outlet. Schematic diagram of the valve is shown in Fig. 1.

A passive control circuit (Pilot chamber pressure control circuit in Fig. 1) is required to achieve the above objective. Depending upon the valve inlet pressure, this passive control circuit sets the criterion for position of spool inside the valve. Determination of characteristics of the control circuit is a crucial part in designing the control circuit and hence the pressure regulated valve. This requires complete information of flow forces acting on the valve spool. Calculation based on theoretical model can provide some initial approximate information. With the advent of numerical power, advanced, sophisticated and reliable CFD tool is very useful in determining the various flow forces acting on the spool.

Leutwyler and Dalton [1] showed the potential of CFD tool in analyzing compressible, turbulent flow through butterfly valves. ANSYS FLUENT was used for the analysis. Amirante et al. [2]-[3] evaluated flow forces on an open centre direction control valve. Amirante et al. [4] again modeled direct proportional valve using CFD. Compensation techniques

based on spool profiling were used to balance the flow force at different level of valve openings. Chen et al. [5] reported flow visualization using CFD in a ball valve. Chattopadhyay et al. [6] have investigated turbulent flow structure inside a Pressure Regulating and Shut-Off Valve using ANSYS FLUENT. In this work, both 2-D and 3-D simulations were performed with a conclusion that 2-D model could predict the flow coefficients satisfactorily. Song et al. [7] have reported 2D dynamic simulation of a pressure relief valve using CFD.

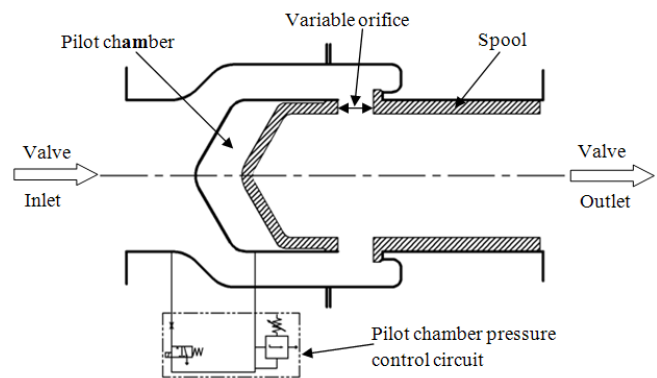


Fig. 1 Schematic diagram of pressure regulated valve

In this study, transient, compressible and turbulent flow through a pressure regulated valve has been solved using commercial software ANSYS FLUENT [8]. As the valve inlet pressure changes, spool inside the valve as shown in Fig. 1 must move in appropriate direction to keep outlet pressure constant. Starting from the instant of inlet pressure change, a transient analysis is performed to reach the state of quasi-static flow through the valve.

## II. MATHEMATICAL MODELING

The governing equations to be solved are as given below:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{v}) = 0 \quad (1)$$

Binod Kumar Saha is with the CSIR-Central Mechanical Engineering Research Institute, Durgapur – 713209, India (phone: +91-343-6452028; fax: +91-343-2548204; e-mail: bsahacmeri@gmail.com).

Momentum equation:

$$\frac{\partial}{\partial t}(\rho \bar{v}) + \nabla \cdot (\rho \bar{v} \bar{v}) = -\nabla p + \rho(\nu_l + \nu_t) \nabla^2 \bar{v} + \rho \bar{g} \quad (2)$$

The terms  $\nu_l$  and  $\nu_t$  refer to molecular diffusivity (kinematic viscosity) and turbulent diffusivity, respectively.

Energy equation:

$$\frac{\partial T}{\partial t} + \bar{v} \nabla T = (\alpha + \alpha_t) \nabla^2 T + \nabla v \bar{\tau} \quad (3)$$

Here  $\alpha$  and  $\alpha_t$  refer to thermal diffusivity and corresponding turbulent diffusivity, respectively. The last term in the energy equation represents viscous dissipation which was duly considered to arise out of viscous stress  $\tau$ .

The turbulent viscosity term  $\nu_t$  is to be computed from an appropriate turbulence model. In this approach, the turbulent viscosity is computed using two different equations for parameters such as turbulent kinetic energy  $k$  and dissipation rate  $\epsilon$ . In this study, for closing the time-averaged momentum equation, a realizable  $k$ - $\epsilon$  model, proposed by Shih et al. [9] was chosen. As such, the model was found to work satisfactorily as observed by Chattopadhyay et al. [6]. The expression for the turbulent viscosity is given as:

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \quad (4)$$

The turbulent diffusivity is related to the molecular diffusivity in the following manner:

$$\text{Pr} \frac{\nu_t}{\alpha_t} = \text{Pr}_t \quad (5)$$

The value of turbulent Prandtl number  $\text{Pr}_t$  is chosen as 0.85. The detailed description of turbulent model is outlined in Chattopadhyay et al. [6].

The working fluid is air albeit in the compressible regime with varying density which has been modeled using Perfect Gas Law. The viscosity has been modeled using Sutherlands viscosity for air which can be expressed with three coefficients as:

$$\mu = \mu_o \left( \frac{T}{T_o} \right)^{3/2} \frac{T_o + S}{T + S}$$

where,  $\mu_o$  and  $T_o$  are reference viscosity and temperature and  $S$  is Sutherland constant.

A value of  $\mu_o = 1.716 \times 10^{-5} \text{kg/m.s}$ ,  $T_o = 273.11 \text{K}$  and  $S = 110.56 \text{K}$  were used in the present work.

### III. BOUNDARY CONDITIONS

Following boundary conditions were used for this analysis.

1. Inlet boundary where a fixed pressure condition is used. The inlet temperature is fixed at  $360^\circ \text{C}$ .
2. As discussed above, outlet boundary of the flow domain is open to ambient. So, pressure and temperature have been set to  $101325 \text{Pa}$  and  $298.15 \text{K}$ , respectively.

No-slip boundary condition is assigned for all the walls. The valve wall was generally assumed to be at adiabatic condition. When the valve wall was taken at an elevated temperature, no significant difference in predicting flow rate and turbulence was observed. For prescribing turbulent quantities at the boundaries, several options are available. We have used a condition prescribing the level of turbulent intensity (TI) at around 5-10% in the incoming fluid stream which is a reasonable level of value used by researchers.

### IV. MESH GENERATION

Flow geometry has been approximated with a 2D axis-symmetric domain as shown in Fig. 2. Unstructured quadrilateral mesh has been generated using commercial software package ANSYS ICEM CFD. Dynamic mesh motion along with the movement of spool boundaries has been considered during mesh generation. Entire fluid flow domain has been divided into two parts as shown in Fig. 3. Mesh for zone 1 does not possess any motion whereas mesh for zone 2 undergoes dynamic motion. Wall boundaries comprising spool surfaces move depending upon the conditions as mentioned in FLOWCHART in Fig. 5. Some of the internal edges also move with spool surfaces. One special internal edge has been selected to move with spool surfaces but at exactly half the speed of spool movement. This special movement serves a very important purpose; finer boundary layer of the variable orifice remains intact though the spool opening reduces. Initial mesh contains 0.2 million quadrilateral cells approximately.



Fig. 2 2D axis-symmetric flow domain

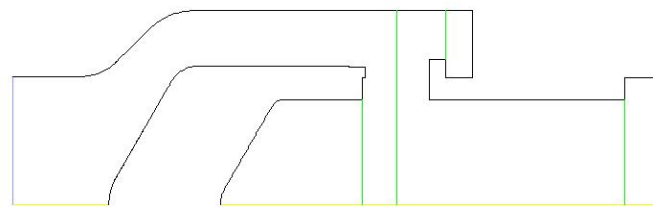


Fig. 3 Flow domain showing two different zones of fluid

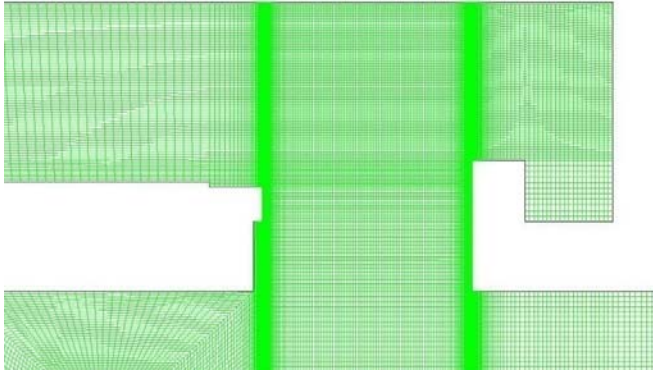


Fig. 4 Structured quadrilateral mesh around orifice opening

V. METHODOLOGY OF CALCULATION

Valve inlet pressure may vary from 30Psig to 110Psig, whereas valve outlet pressure remains within 29Psig to 35Psig. Air flow further downstream of valve outlet is approximated with the help of an orifice flow finally open to ground level condition. Calculation domain, therefore, includes the valve geometry along with the appropriate pipe length before and after the orifice. 2D axi-symmetric flow geometry is shown in Fig. 2.

Transient flow through this flow domain is solved in ANSYS FLUENT to determine various spool opening depending on various inlet pressure. Now, when the inlet pressure is at minimum value, i.e., 30Psig, spool opening should be 100%. Therefore, solution of this flow will set the initial condition for transient calculation. In transient calculation, inlet pressure is varied in a step wise manner. This is done via a User Define Function (UDF). In reality, when inlet pressure is changed by a delta amount, flow through the valve will take some time to die down the transient phenomena. In CFD, this quasi-static state is achieved after solving the transient flow for a number of time steps. For each time step, flow convergence is checked using UDF and allow the calculation to move to next time step only after meeting the required criteria for convergence. Also, a check, whether quasi-static has been achieved, is carried out with the help of same UDF before proceeding to next time step. This UDF is described with the help of a FLOWCHART given in Fig. 5.

When the transient dies down pressure at valve outlet is checked for the desired outlet pressure. If desired pressure is not obtained, then spool opening is reduced by a delta amount and further transient calculation is carried on. This whole process is automated using the UDF. When the desired pressure is obtained, UDF directs ANSYS FLUENT to write the Case and Data file for further post processing. Spool opening and valve outlet pressure for each time step are written in a DAT file with the instruction written in UDF. This file is required for further reference. After achieving the desired pressure at valve outlet, UDF directs ANSYS FLUENT to increase the valve inlet pressure by a delta amount and again the whole loop of operations repeated.

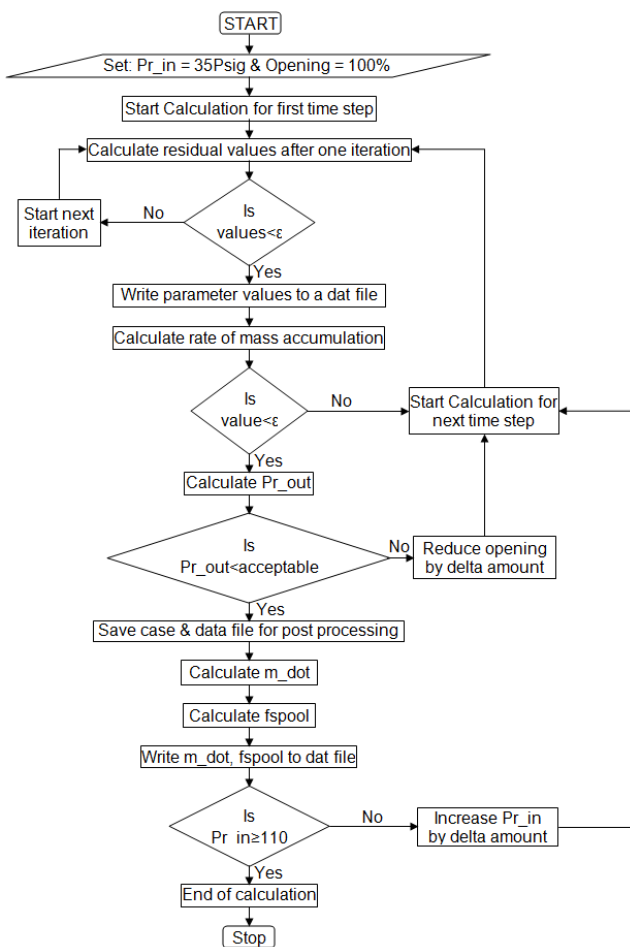


Fig. 5 FLOWCHART for special UDF

VI. RESULTS AND DISCUSSION

Flow through the pressure regulated valve is governed by variable orifice flow. Fig. 6 shows the spool opening in mm against the gauge pressure in Psig at valve inlet. It is observed here that spool opening reduces by 86% as valve inlet pressure increases from 35Psig to 50Psig. There after changes in spool opening is very slow. This happens because pressure is almost fixed at valve outlet.

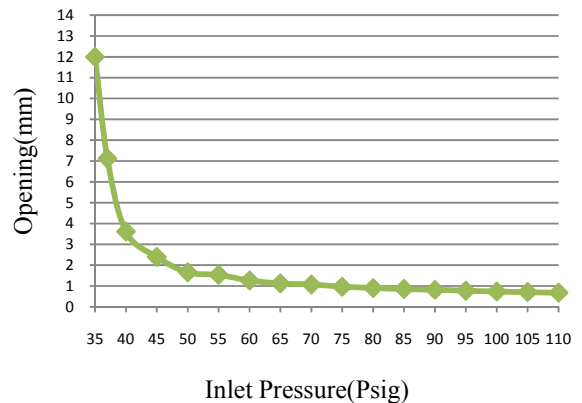


Fig. 6 Spool opening plotted against valve inlet pressure

Fig. 7 shows the variation of mass flow rate in kg/s with the change in inlet pressure. There is no significant change in mass flow rate. This is justified as two things happen simultaneously. Resistance in flow path increases as spool opening reduces which would try to decrease mass flow rate. But again the increase in inlet pressure results in nullifying the above fact.

In Fig. 8 flow forces acting on spool surfaces are plotted against the inlet pressure. The negative sign indicates that the force is acting in negative x direction, i.e. axially from right to left direction (refer to Fig. 1). It is observed that absolute force increases with increase in inlet pressure. This owes to the fact that spool opening creates an orifice and the spool area left side of that orifice is less than that at the right side of the orifice (refer to Fig. 1). So, a counter force is required to keep the spool at desired location. This force comes from pilot chamber air. Pilot pressure is the outlet pressure of a passive control circuit. Fig. 9 shows the pilot pressure variation with change in inlet pressure.

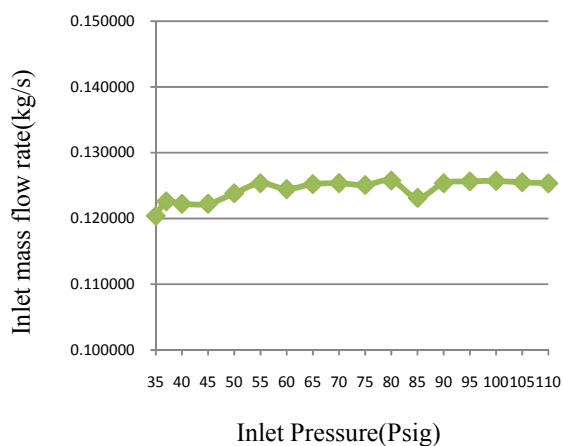


Fig. 7 Variation of mass flow rate through valve with change in inlet pressure

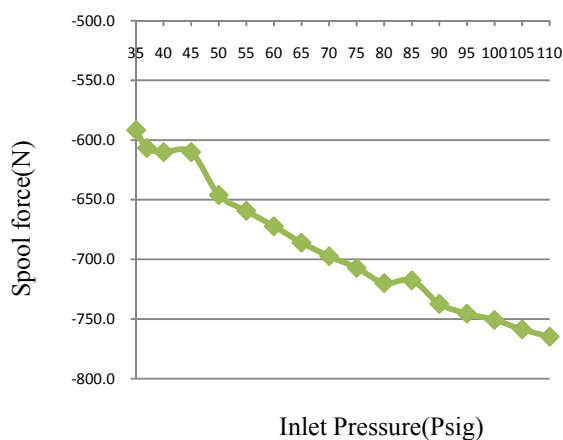


Fig. 8 Variable flow forces on spool surfaces as inlet pressure varies

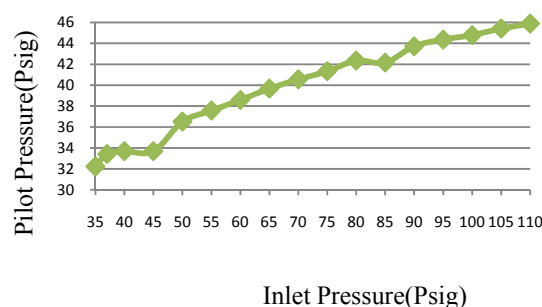


Fig. 9 Pressure characteristic of passive control circuit

## VII. CONCLUSION

Control circuit pressure characteristic has been determined by solving compressible, turbulent flow through pressure regulated valve. A special UDF has been developed to couple with ANSYS FLEUNT primarily to modify structured mesh of the flow domain dynamically with change in valve inlet pressure. The simulation provides valuable insight into the control circuit behaviour which is crucial for designers. Attempt to simulate using fully three-dimensional geometry is underway.

## ACKNOWLEDGMENT

We thank Prof. Gautam Biswas, Director, Central Mechanical Engineering Research Institute, Durgapur, for permission to publish the work. We are also thankful to Dr. Amitava Dasgupta for providing valuable advice on the functional aspect of the valve.

## REFERENCES

- [1] Leutwyler Z, Dalton C. A computational study of torque and force due to compressible flow on a butterfly valve disk in mid-stroke position. *J Fluid engineering* 2006; 128:1074-84.
- [2] Amirante R, Del Vescove G, Lippolis A. Flow forces analysis of open center hydraulic directional control valve sliding spool. *Energy Conversion and Management* 2006; 47:114-31.
- [3] Amirante R, Del Vescove G, Lippolis A. Evaluation of the flow forces on an open centre directional control valve by means of a computational fluid dynamic analysis. *Energy Conversion and Management* 2006; 47:1748-60.
- [4] Amirante R, Moscatelli PG, Catalano LA. Evaluation of the flow forces on a direct (single stage) proportional valve by means of a computational fluid dynamic analysis. *Energy Conversion and Management* 2007; 48:942-53.
- [5] Chern MJ, Wang CC, Ma CH. Performance test and flow visualization of a ball valve. *Experimental thermal and fluid sciences* 2007; 31:505-12.
- [6] Chattopadhyay H., Kundu A., Saha B. K, Gangopadhyay T.. Analysis of Flow Structure inside a Spool Type Pressure Regulating Valve. *Energy Conversion and Management* 2012; 53:196-204.
- [7] Song XG, Jung JH, Lee HS, Kim DK, Park YC. 2-D dynamic simulation of a pressure relief valve by CFD, Proc. 9th WSEAS Int. Conf. on Applied Computer and Applied Computational Science, Hangzhou, China, April 11-13, 2010.
- [8] www.ansys.com.
- [9] Shih TH, Liou WW, Shabbir A, Yang Z, Zhu J. A new k-ε eddy-viscosity model for high Reynolds number turbulent flows -model development and validation. *Computers Fluids* 1995; 24:227-38.