Design of Flow Regulating Pressure valve

Ms Ashwini Nagtilak¹, Prof. Siddaram Biradar²

¹ PG Student, Dhole patil College of Engineering, Pune, India
² Professor, Dhole patil College of Engineering, Pune, India

Abstract—In most of the industries like chemicals there are several different types of valves used for regulating the pressure of the liquid. Flow control valve, as per name only, control the rate of the fluid with the help of hydraulic circuit. In the flow regulating application, it is very difficult to maintain the mean pressure balance when two fluids are mixed together. There may be explosion occurs in some chemical reactions when two different types of fluids are mixed together. In such cases we have to supply the fluids in correct sequence. This paper centres on the design of flow regulating valve. The purpose of the design is to design a such valve which automatically shifting from one opening to another opening depending on the pressure level. Due to this we can avoid the bursting of the valve after certain pressure range. The main concern of this design is to obtain the geometrical as well as operating parameters. Furthermore, the result from the design can be used to make FEA analysis as well all model analysis to check the correctness of the valve.

Keywords—Pressure Vessel Design, Valve Design

I. INTRODUCTION

A control valve regulates the flow or pressure of the fluid. There are many different type of valves is used in different industries. Many of them are electronics, pneumatics and hydraulic. Valves are the basic component in the fluid flow or pressure system that regulates the flow or pressure of the fluid. This function involves stopping and starting flow, controlling flow rate, diverting flow, preventing back flow, controlling pressure or reliving pressure. Valves are designed to handle either liquid, oil or gas applications. The system can be a pipeline used to transfer the fluid medium from one location to another like a oil/gas pipeline or a pressure vessel which can store or generate the fluid, for example - a steam generating boiler, heat exchanger, storage tanks etc. Electro-mechanical systems use the electromagnetic forces generated between sets of armatures and coils for moving and positioning the engine valve. The nonlinearity of the electromechanical force and the time constant due to the inductance in the coils make it very difficult to control the valve at the end of the trajectory. To achieve Seating velocity control and lift control, these systems require complicated real-time control strategies which are difficult to implement in mass produced systems. Electro-hydraulic and Electro-pneumatic systems use pressurized hydraulic or pneumatic fluid controlled by valves to provide the force required for actuation. These systems depend on precise control of the fluid flow to ensure accurate positioning of the engine valve. Hence they require a complicated and expensive proportional valve which increases the cost and hence are not viable for mass production. These systems throttle the hydraulic fluid during a major portion of the operation cycle to control the flow to the engine valve actuator. The large pressure drop across the proportional valves leads to the requirement of a higher supply pressure and thus increases the power consumption. The pneumatic systems are affected by fluid compressibility which leads to difficulties in achieving precise valve motion and seating velocity control. Hence, for a system to be mass-produced, it is required to have,

- Flexibility in lift, timing and duration
- Low valve seating velocities
- Low power consumption

Since flow control valves are flow control devices, there are many Codes and Standards written to control their design and application.

II. LITERATURE REVIEW

Xue-Guan Song, Young-Chul Park, Joon-Hong Park [1] In this research, a simplified dynamic model (SDM) is developed to predict the dynamic characteristics during the reclosing process and blow down of a conventional pressure relief valve (PRV). The principle of the SDM is based on the equation of motion for one degree of freedom system. The damping effect is ignored due to its little effect on the reclosing time; a pressure drop condition based on lots of experiments is assumed. In addition, it is very novel that the SDM combines the static CFD analysis result, that is, the lift force coefficient of the valve at several fixed lifts is calculated with static CFD analysis, and then imported into the SDM as the inherent characteristics of the valve. The case study proved that the SDM is very reliable to predict the dynamic characteristics and blowdown of the conventional PRV.

A. Beune, J.G.M. Kuerten, M.P.C. van Heumen [2] A multi-mesh numerical valve model has been developed to analyze the opening characteristic of high-pressure safety valves. Newton’s law and the CFD result for the flow force are used to model the movement of the valve. In incompressible transient flow simulations, a large force rise and collapse is caused by a redirection of the bulk flow. This flow-history effect cannot be incorporated in a quasi-

ISSN: 2231-5381 http://www.ijettjournal.org Page 78
steady approach. For real-gases at a set pressure of 40 bar oscillations have been observed during closing of the valve. They are caused by the interaction between the flow in the cavity of the valve disk and the flow towards the valve outlet. At a higher set pressure, the flow force continually decreases, which indicates that only a sufficiently fast inlet pressure rise forces the valve to open. With this tool the operation characteristics of safety valves can be assessed to optimize the valve design.

N.N. Manchekar, V. Murali Mohan [3] have shown that design of gradual flow reducing valve by finite element analysis. This paper work presents to fulfill the objectives by employing a Gradual Flow Reducing Valve controlled by using purely mechanical actuation thus serving primary purpose of self-actuation. In this paper transient structural analysis has been introduced in order to finalize the geometrical parameter of gradual flow reducing valve.

Binod Kumar Saha, Himadri Chattopadhyaya, Pradipta basu Mandal, tapas Gangopadhyaya[4] Dynamic modeling of flow process inside a pressure regulating and shut-off valve has been investigated using a computational fluid dynamic approach. The valve is designed to reduce high inlet pressure to a lower level of outlet pressure which remains almost constant. With the change of inlet pressure, the change in position of the spool inside the valve was calculated using a force balance approach. The Navier–Stokes equation along with appropriate turbulent closure has been solved for this purpose in the compressible flow regime using ANSYS-FLUENT software with special functions developed for calculation of flow force. The code could predict the spool movement and the final spool position when the spool position is deviated from equilibrium. The final spool position and time required to reach equilibrium, besides the flow parameters, also depends on the value of friction coefficient between spool and the valve body. Higher values of friction coefficient between the spool and the valve body is found to be associated with faster stability of the spool.

A.J. Ortega, B.N. Azevedo, L.F.G. Pires, A.O. Nieckeke, L.F.A. Azevedo [5] Relief valves manufactures generally only provide information on valve characteristics under full opening stage which is obtained under steady state regime, therefore, valve and flow’s transient behaviour are neglected. Understanding the transient behaviour of relief valves is crucial because critical conditions may be attained, damaging the pipeline. In order to overcome this lack of information, they developed a direct acting spring loaded pressure relief valve’s computational model was developed. A simplified two-dimension model was built based on the valve geometrical and constructive characteristics.

V.D. Rathod, Prof. G.A. kadam, Mr. V.G. Patil [6] The focus of this paper is to design, analyze and optimize a pressure vessel safety release valve which is “use as one time”, which save bursting the pressure vessel. To simulate such a process, FEA is an extremely convenient tool which has reduced both the design costs and times to deliver the product. This paper explains the strategies used in the design of the safety release valve and results of the operational simulation in FEA.

Sushant M. Patil, Ramchandra G. Desavale, Imran M. Jamadar [7] have shown that conceptual structure design through thickness optimization of high pressure and high temperature self-regulated pressure valve using non-linear transient finite element method. In this paper the optimization of thickness of valve plate, material selection, design of various components of valve and analysis of gradual flow reducing valves for both axial and bending has been discussed.

Ron Darby [8] In this study a model for the opening lift dynamic response of a pressure relief valve in gas/vapor service is presented which accounts for all of these effects through a set of five coupled nonlinear algebraic/differential equations. These equations are solved by a numerical method that can be implemented on a spreadsheet to predict the position of the valve disk as a function of time for given valve characteristics, operating conditions, and installation parameters. The model incorporates the influence of the various parameters on the stable/unstable nature of the disk response.

M.R. Mokhtanadeh-Dehghan, N. Ladommatars, T.J. Brennan [9] had done the Finite Element analysis of flow in a hydraulic pressure valve. The paper describes a finite element study of laminar flow of oil through a hydraulic pressure-relief valve of the differential-angle type used in a variable compression ratio piston of an internal combustion engine. The model simulates an experimental setup used to obtain the performance characteristics of the valve under steady-state conditions, the velocity and pressure distributions through the valve and the magnitude of the lift forces on the plunger are obtained under a constant inlet pressure condition for various plunger lifts. Comparisons are made with the available experimental and analytical data. The details of the flow through the valve are predicted with a good degree of accuracy. The results provided an improved understanding of the way in which opening forces on the value plunger are generated at various plunger lifts and are thus helpful to the design process. The complex flow fields generated resulted in the formation of recirculation zones and modified the minimum flow areas, thus affecting the pressure distribution and the force on the plunger.

III. VALVE DESIGN DATA

To design the all components of the valve we need to refer ASME Section-VIII, Division-1 codes, mathematic equations etc.
A. Technical Data

Below dimensions are given to finalize the input dimensions as per ASME codes.
1. Main Shell Diameter - 100mm
2. Nozzle Diameter – 60mm
3. Valve operating pressure from side1 - 0.201 Mpa
4. Valve operating pressure from side2 - 0.19 Mpa
5. Springs required – 5 Nos

A. Output Data

Following are the parts of the valve and whose dimensions are calculated,

1) Main Shell:
   Material: - SA 516 Gr 70
   Thickness of shell=$t=3.0728mm$
   Standard plate of 3.1mm thickness is not available in market so chosen next standard plate thickness as 6mm.
   Length of Main shell=$L=135mm$

2) Nozzle:
   Material: Material - SA-106 GRB
   Thickness of Nozzle=$t=3.051mm$
   Standard plate of 3.1mm thickness is not available in market so chosen next standard plate thickness as 6mm.
   Length of Nozzle= 90mm

3) Circular Pressure Plate:
   Material- SA 516 Gr 70
   Thickness of the circular pressure plate=$t=1.703mm$

4) Flange:
   a) Main shell-
      Inner Diameter=$d_s=100mm$
      Outer Diameter=$D_s=150mm$
      Thickness=10mm
   b) Nozzle
      Inner Diameter=$d_N=60mm$
      Outer Diameter=$D_N=90mm$
      Thickness=10mm

5) Reinforcement pad:
   Outer Diameter=80.5mm

6) Clit:

Outer Diameter=100mm
Inner Diameter=95mm
Thickness=3mm

7) Plunger:
   Length=36mm
   Diameter=25mm
   Width=15mm

8) Spring:
   Material- ASTM A913-50
   Length of spring=36mm
   Stiffness of the spring=8.28N/mm

Fig. 1 Drawing of Flow Regulating Valve as per Calculated Dimensions

IV. CONCLUSIONS

The outputs obtained from the Design of the flow regulating pressure valve are used to perform the structural analysis as well as transient analysis to obtain the equivalent stresses and deformation.
Furthermore, the result from the design can be used to make model analysis to check the correctness of the valve.

ACKNOWLEDGMENT

A Thanks to Prof. Siddaram Biradar for his valuable contribution for motivation and support for developing this work.

REFERENCES


