2-D Dynamic Analysis of a Pressure Relief Valve by CFD

Xue Guan Song, Ji Hoon Jung, Hyeong Seok Lee, Dong Kwan Kim, Young Chul Park*
Mechanical Engineering
Dong-A University
PusanKOREA
*parkyc67@dau.ac.kr

Abstract: - This paper presents a study of dynamic characteristics of spring loaded relief valve and the turbulent flow of water through it by using CFD in 2-Dimensions. Mesh deformation due to the fluid-solid interaction between the valve disc and the surrounding fluid and CEL expressions determined by solving a dynamics equation of forces acting on the valve disc, are used to account for the motion of the valve disc under different inlet pressure conditions. The velocity and pressure distribution through the valve at each time step are obtained. Especially, this simulation presents the different effects of the flowing fluid acting on different part of disc, and thus identifies the critical part/region, which has significant effect on the transient response of the system. The results provide a better understanding of motion and flow characteristics of a relief valve and thus are helpful to the relief valve design process.

Key-Words: - Pressure relief valve, Dynamic analysis, CFD, Mesh deformation, Lift force

1 Introduction

The spring loaded pressure relief valve (also called direct operated relief valve) is a type of pressure relief valve (PRV) used to control or limit the pressure in a system or vessel. It performs a critical function in preventing excessive pressure and protecting the system and mechanical equipment. Due to its simple configuration, spring loaded PRVs are widely used in lots of hydraulic systems. Hence, the detail study on such a valve is essential.

Till now, two approaches are usually adopted to study the performance of a spring loaded PRV. One is the experimental study. Many researchers have experimented and analyzed spring loaded PRVs for the fluid characteristics, operating parameters, and the coefficients such as discharge coefficient and pressure loss coefficient [Sallet, 81; Sallet, 85; Narabayshi, 86]. From the viewpoint of practical application, this method is more reliable and suitable, since the real situations are usually simulated in the experiments. But on the other hand, this method is very expensive in time, manpower and facilities. Hence, with the development of computer and numerical method, more and more researchers started to use computer simulation to investigate.

In the computer simulation, most of them used the dynamic model to investigate the performances of various PRVs. Herein, A. Rayand Catalani [Ray, 78; Catalani, 83] et al. did much innovative work in this area. Compared with the experiments researches, this approach is very effective, and enhanced the understanding of dynamic behaviour of PRVs in theory. However, since this method commonly ignored the detailed dimension of valve and rarely considered the fluid dynamics, it becomes a little bit insufficiency for deeper study of a spring loaded PRV.

With the development of computational fluid dynamics (CFD) during the last two decades, studying the fluid characteristics and mechanical performance of PRV by using CFD becomes more and more popular. For example, M. R. Mokhtarzadeh-Dehghan [Mokhtarzadeh-Dehghan, 97] described 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. While J Francis and P L Betts [Francis, 97] predicted axi-symmetric flow patterns inside a model of a real valve by using commercial package FIDAP. However, due to the limit of CFD level, the common method used by CFD is to study the steady fluid characteristics at several fixed opening from the minimum opening to the maximum. Until recently, a few of researchers began the dynamic CFD studies of valve [ANSYS, Inc, 06; Srikanth, 09; Song, 09]. Because of adopting the moving mesh method, these works resolve both the motion of movable part in the valve and the fluid characteristics simultaneously.

This paper is an extension of our previous work. A 2-dimensional (2-D) analysis is carried out to save the computational time. Besides successfully recording the displacement and velocity of valve disc by using mesh deformation and CEL expression, this research innovatively studies the different effects of flowing fluid acting on different part of the valve disc, thus easily find which part plays important role on the valve poppet and
vibration. The results can be very helpful for the optimization of the valve’s dynamic response.

2  Spring loaded pressure relief valve

2.1 Description of a spring loaded PRV
The simplified representation of the spring loaded PRV studied is shown in Fig. 1. It mainly consists of three parts; the main body, the valve disc and the spring. The disc is made of steel with density of 7800 kg/m$^3$. The pressure setting of the valve is achieved via compressing the spring a given length. When the pressure at the upstream is bigger than the setting pressure, the disc will be lifted due to the break of the force balance previously by the force of flowing fluid acting on the disc. The opened gap between the body and disc relieves some flow to the outlet. The relieved flow increased the lift force suddenly, which cause the valve disc open more and suddenly (also called as poppet), when the major flow enter the chamber, the build-up back pressure becomes bigger and reduces the total lift force induced by the fluid. Under the total effect of flowing fluid and spring, the valve disc will achieve its steady position gradually after a period of fluctuation.

![3D model and schematic diagram of the pressure relief valve](image)

**Fig.1-3D model and schematic diagram of the pressure relief valve**

2.2 Motion equation for a spring loaded PRV
For modeling a spring loaded PRV it’s necessary to describe the forces acting on the valve disc. In this work, the motion equation contain the essential physical characteristics of the valve such as the geometric, inertial, material and fluid force. The gravity, friction and damping are assumed to be negligible.

\[ m \frac{d^2x}{dt^2} = F_{\text{flow}} - F_{\text{spring}} \]  

(1)

Where $m$ is the mass of valve disc and stem, $x$ is the disc displacement at time $t$, $F_{\text{flow}}$ is the force induced by the flowing fluid acting on the disc, and $F_{\text{spring}}$ is the spring force acting on the disc. The motion equation will be defined in CFX by using CEL expression, and then solved by CFX itself at each time step.

3  FSI simulation

3.1 Motion equation in CFX
CFX provides the ability to solve the solution of cases that involve the coupling of solution fields in fluid and solid domains. This coupling is commonly referred to as Fluid Structure Interaction (FSI). This work uses the fluid mesh deformation resulting due to the FSI between the disc and the surrounding fluid, the deformation of disc itself is not modelled. To account for the motion of the disc, CEL expressions are used to solve the motion equation written above. CEL is an interpreted language developed for some special simulations. In this work, it’s also used to define valve material properties, specify the displacement of disc and so on. Part of the predefined CEL expression is list bellow.

```
CEL:
EXPRESSIONS:
......
kSpring = 4000.0 [N m^-1]
Barea = 9.5[mm]*1[mm]*1.4
P0 = 5[atm]
F0 = P0*Barea+kSpring*2[mm]
FFlow = force_y()@disc*2
mdisc = 0.7727 [g]
disDenom = kSpring + mdisc/tStep^2
disOld=areaAve(Total Mesh Displacement Y)@disc
velOld = areaAve(Mesh Velocity y)@disc
disNumer=FFlow+F0 + disc*velOld/tStep+
mdisc*disOld/tStep^2
disNew = disNumer/disDenom
  tStep = 5e-5 [s]
...... END
END
```

Table 1-CEL script for motion of disc in CFX

3.2 Computational grid
Simulation of this valve needs the disc part move based on the motion equation, so deformable mesh (mesh deformation) must be used to solve the equations of the moving boundaries. CFX provides mesh deformation.
option to enable the specification of the motion of nodes on boundary of the mesh using CEL. The motion of all movable nodes is determined by the mesh motion model, which is currently limited to displacement diffusion. With this displacement diffusion model, the displacements applied on boundaries are diffused to other mesh points by solving the equation:

\[ \nabla \cdot \left( \Gamma_{\text{disp}} \nabla \delta \right) = 0 \quad (2) \]

where \( \delta \) is the displacement relative to the previous mesh locations and \( \Gamma_{\text{disp}} \) is the mesh stiffness, which determines the degree to which regions of nodes move. This equation is solved at each iteration or time step for transient simulations.

This model is very helpful for the relative mesh deformation, which are not as defined as the moving boundaries in CEL. It has been proven that if the initial mesh is relatively fine in certain regions of the domain, then it will remain relatively fine after solving the displacement diffusion equation. Hence, solely structured multi-block hexagonal grids are generated using commercial grid generators ICEM CFD and exported to CFX-pre along with boundaries. This yields many benefits such as faster and more accurate computation than those with unstructured grids. Fig. 2 shows the initial grids of PRV with some simplifications at the minimum and maximum lift, respectively. It can be found that the grid in the seat region remain good quality even the disc achieve its maximum lift.

### 3.3 Boundary conditions

After importing the grid in CFX-pre, simulation type has been selected as transient with the time step and total time, which are all defined in CEL. Turbulence is accounted through high Reynolds number k–\( \varepsilon \) with standard wall functions. The fluid material is set to be incompressible fluid – water at 25°C. The reference pressure of fluid is defined to be 101,325 Pa. Water flow with specified several overpressure based on the setting pressure of 5[atm] are entering the inlet, and after taking different turns in the chamber leave through outlet, where atmosphere pressure is prescribed. The displacement of moving element (only disc part) is defined as “disNew” as listed in Table 1. Connected surfaces of moving element, i.e. two symmetrical planes, are defined as unspecified in the wall boundary conditions, which will not limit the motion of disc.

### 4 Results

Due to the high-quality grid and small time step setting, the calculation converged fast and well at each time step. And the variables such as velocity, pressure, flow rate and force are available for all nodes are recorded. Fig. 3 shows the displacement of disc for three overpressure conditions with time change. Note that all displacement should be added 2mm, because there is a gap of 2mm left for the initial step. It’s very obvious if the overpressure is high, the system achieves steady condition quickly. In contrary, the disc keep intensive oscillation before achieving the final steady condition. And the higher the overpressure is, the higher lift the valve gets. Represented in the figure, the disc reaches to its maximum lift of 10mm for 10% overpressure, but stay at approximate 7mm and 3.2mm lift for 6% and 2% overpressure, respectively.

![Fig. 3-Displacement of disc vs. time](image)

Fig. 4 shows the four kinds of forces on the disc for three overpressure conditions. As shown in figure, F1 to F4 represent the flow induced force on four surfaces of disc, respectively. Obviously, the magnitude of the force acting on the bottom of disc is maximum, the second max. force is the force acting on the top of the disc, then is the force F2 and the minimum is force F1. When the valve is open, fluid force acting on the bottom of disc causes the disc to move up to allow more fluid to leave. As the disc raises up to the maximum displacement, the spring force plays more important role than those at little lift and thus pushes the disc downward. The pressure variation causes the disc to oscillate along y-axis as a
result of an imbalance in the spring force; the disc finally stops moving when forces on it are in equilibrium. Two points should be explained for the fluid force. First, for 6% and 10% overpressure conditions there are totally four times of sharp force increase. Compared with Fig. 3, it can be found that the times when the flow forces suddenly increased happened to be the moments the disc stopped at the allowable maximum displacement. It’s clear the sudden increases are due to the inertial forces of flowing fluid when the disc was limited by the valve body suddenly. Second, force $F_1$ is in the same direction with $F_3$ but opposite with $F_2$ and $F_4$, this is due to the negative pressure occurring near this region. It can be seen from the pressure distribution in Fig. 5.

![Image](a) 2% overpressure  
(b) 6% overpressure  
(c) 10% overpressure  
(d) total force on disc  

Fig.4-Forces induced by fluid on the disc

Both the velocity streamline from inlet to outlet and the static pressure distribution at the mid plane for 6% overpressure condition are shown in Fig. 5. Compared with Fig. 3, it can be seen that at $T = 0.003s$ and 0.007s, the disc is raising up, at $T=0.014s$ it reaches the maximum displacement of 10mm, and then drop to approximate 40% opening, i.e. 4mm at $T=0.044s$. At $T=0.102s$ the disc raise up again after the first oscillation cycle, and (f) represent the final steady condition, that is the disc stops moving.

It’s observed from (a) and (b), as the valve open and the fluid flows out of the chamber to the outlet, the swirl flow occurs near the seat region shifts towards the outlet. When the displacement is maximum as shown in (c), the swirl flow near the top of disc become very small, and the the one near the seat region becomes intensive due to the step of the bottom of disc and bigger flow area. It also can be found that the negative static pressure occurs near the small surface, thus makes the force $F_1$ opposite to Forces $F_2$ and $F_3$. From 0.044s both the pressure and streamline distribution become stable due to the little oscillation of disc.

![Image](a) T=0.003 s  
(b) T=0.007 s  
(c) T=0.014 s  
(d) T=0.044 s  
(e) T=0.102 s  
(f) T=0.24 s  

Fig.5-Streamline and pressure distribution vs. time

5 Conclusion
The application of numerical method provided a greater insight into the flow visualization aspects with considering the coupled dynamics among the fluid and the moving mechanical parts—valve disc. By using FSI and CEL expression, this simulation resolves the motion of disc based on the dynamic motion equation and flow field analysis simultaneously. The total force acting on the disc are divided into four forces acting on the different position to see the effect of each. Careful consideration of these regions may result in improved design. Pressure distribution and velocity streamline through the valve were well captured. It showed the unsteady interaction between the flowing fluid and compressed spring created unsteady loads which makes the valve disc oscillate. This improved the understanding of the way in which lift forces on the valve are generated at different time, and thus helpful to the design of spring loaded pressure relief valve.

6 Future Work
The model used in the work was simplified a little for the structural mesh generation. To resolve the flow and capture the motion of the valve disc more practically, 3-D model without simplification should be used, and experiments should be carried out for verification in the further research.
Acknowledgements
This work was supported by Technical Center for High-Performance Valves from the Regional Innovation Center (RIC) Program of the Ministry of Knowledge Economy (MKE).

References: