Advanced simulation of naval pressure relief valve

C L Dumitrache¹, M Bărăhăescu¹, B Hnatiuc²
¹Maritime University of Constanța, Department of General Engineering Sciences, Mircea cel Batran street, No. 104, Constanța, 900663, Romania
²Maritime University of Constanța, Department of Electrical Engineering Sciences, Mircea cel Batran street, No. 104, Constanța, 900663, Romania

E-mail: ldumitr@yahoo.com

Abstract. The designing operation of pressure relief valve (PRV) was performed using NX Siemens. The purposes of this article are to evaluate different stages of real function of PRV when the pressure at the upstream is increasing step by step and the setting pressure of valve and the disc will be lifted. We simulate four different stages of valve evolution after the moment when the force exerted by the fluid on the disc approaches the spring force. The comparative CFD and FSI analysis are presented and interpretation of flow results (velocity, pressure and shear resultants) reflects the fluid structure interactions.

1. Introduction

The NX assembly design of pressure relief valve is illustrated in figure 1 and the construction is presented in the article [1].

Figure 1. Section viewing of 3D assembly PRV [1].

The PRV’s are automatic pressure-relieving devices used for overpressure protection of piping and equipment. These valves are generally used in gas or vapor service because their opening and reseating characteristics are matching the properties and potential hazards of compressible fluids. The valves protect the system by releasing excess pressure. Under normal pressure, the valve disc is held against the valve seat by a preloaded spring. As the system pressure increases, the force exerted by the fluid on the disc approaches the spring force. As the forces equalize, fluid begins to flow past the seat [2]. The most important parts of PRV are the disc, seat, and spring, their design has a great influence...
on proper function of PRV. The valve disc is designed in such a way that the escaping fluid exerts a
lifting pressure over an increased disc surface area, thereby overcoming the spring force and enabling
the valve to rapidly attain near full lift.

An added benefit [2] to the safety valve disc design is that the pressure at which the valve reseats is
below the initial set pressure, thereby reducing the system pressure to a safe level prior to resealing.

If these valves are used for liquids service, the liquids do not expand, and there is no additional
lifting force on the disc, the valve lift is proportional to the system pressure. Also, the valves reseat
when the pressure is reduced below the set pressure [3]. These valves can be used with both
compressible and incompressible fluids. It combines the design features of a safety and a relief valve
into one [2]. Therefore, when it is used with compressible fluids, such as steam or a gas, it pops open
to release the overpressure, and when used with incompressible fluids, such as water or other liquids,
it opens gradually, proportional to the increase in pressure over the set pressure, to safeguard the
vessel, tank, heat exchanger, piping, or other equipment [4, 5].

2. Computer fluid dynamic analysis as a design tool
Using NX Siemens (Advance Simulation) we can simulate four different stages of valve evolution
after the moment when the force exerted by the fluid on the disc approaches the spring force:

- first stage when the disc lifts with 1 mm above to the seat (figure 2);
- second stage when the disc continues to lift and arrives at 2 mm above to the seat;
- third stage, the disc arrives at 3.5 mm above to the seat;
- fourth stage the disc arrives at 5 mm above to the seat.

![Figure 2. Section viewing of 3D assembly PRV [1].](image)

The relationship between the pressure drop across an orifice, valve or other assembly and the
corresponding flow rate is given by the flow coefficient of a device $C_{vm}$ which describes the efficiency
of fluid flow [6, 7, 8]:

$$
C_{vm} = Q \cdot \sqrt{\frac{SG}{\Delta P}}
$$

where:
- $C_{vm}$ = Flow coefficient or flow capacity rating of valve;
- $Q$ = Rate of flow (m³ per hour);
- $SG$ = Specific gravity of fluid (Water vapour gas = 0.6218);
- $\Delta P$ = Pressure drop across valve (m).

Flow coefficient $C_{vm}$ is the number of cubic meters of water at 15°C that will flow per hour through
a valve with a pressure drop of 1.0 m of water head across the valve. The use of the flow coefficient
offers a standard method of comparing valve capacities and sizing valves for specific applications that
is widely accepted by industry [6].
In this paper work, just a few parts of assembly, body valve, seat, disc and spindle make a subassembly which was transferred to Advanced Simulation module. All of them were assigned to Steel material with mechanical properties: the yield strength 137.8 N/mm², and the ultimate tensile strength 276 N/mm².

For the meshing operation were used the proper tetrahedral TET(4) elements for fluid domain, body valve, seat, disc and spindle are divided using CTETRA(4) elements.

2.1. Boundary conditions
We consider water vapour gas as the fluid flow with pressure and temperature at inlet zone, 5 m/s

![Figure 3. Velocity results: a.-1 mm disc lifting; b.-2 mm disc lifting.](image1)

![Figure 4. Velocity results: a.-3.5 mm disc lifting; b.-5 mm disc lifting](image2)
Figure 5. Relative pressure and shear resultant: a. 1 mm disc lifting; b. 2 mm disc lifting.

Figure 6. Relative pressure and shear resultant: a. 3.5 mm disc lifting; b. 5 mm disc lifting.

Figure 7. Von Misses Stresses fields: a. 1 mm disc lifting; b. 2 mm disc lifting.
Figure 8. Von Misses Stresses fields: a.-3.5 mm disc lifting; b.-5 mm disc lifting.

Figure 9. Von Misses Strain fields: a.-1 mm disc lifting; b.-2 mm disc lifting.

Figure 10. Von Misses Strain fields: a.-3.5 mm disc lifting; b.-5 mm disc lifting.
and temperature 120ºC. The outlet zone of PRV has the atmospheric pressure and 20ºC. For all stages, the inlet and outlet zone will be considered the same. Because of heat transfer process between fluid flow-body valve cold structure, and environment it is considered convection coefficient 20 W/m²·ºC, and 15ºC the temperature of body valve. In the figures above, we present the impact of disc lifting on velocities and pressures and shear resultants for each stage. Next, this pressure field will be exported to the Mechanical module as a load and boundary condition.

The outlet velocity and discharge \(C_{vm}\) are obtained using relation (1) and we take care for water vapour gas, SG=0.6218 and results are listed in the table 1:

\[
Q = \text{Area}_{\text{outlet}} \cdot \text{velocity} \quad [m^3/h]
\]

The outlet area is considered to be located at center of outlet diameter (65 mm) and the area is calculated:

\[
\text{Area}_{\text{outlet}} = \frac{\pi \cdot d^2}{4} = \frac{\pi \cdot (0.065)^2}{4} = 0.003316625 \quad [m^2]
\]

| Velocity [m/s] | Outlet discharge [m³/h] |
|---------------|--------------------------|
| 1 mm disc lifting | 4.960 | 46.69 |
| 2 mm disc lifting | 4.693 | 44.18 |
| 3.5 mm disc lifting | 4.682 | 44.07 |
| 5 mm disc lifting | 4.745 | 44.67 |

Table 1. Results of velocity and outlet discharge.

2.2. Results of FSI analysis

The fluid structure interaction abbreviated as (FSI) consists of the transmission of the thermal and pressure field via a Finite Elements meshed interface between fluid domain to the structural mesh of seat, disc, spindle and body valve. The imported pressures field from CFD are relative pressure and shear resultant presented in figures 5 and 6, also recognized as boundary condition too. It was calculated contact pressure field using mapping operation in NX Siemens Advanced Simulation. Belong of the temperatures and pressures field we consider the fix constraints on holes of circular and square flanges as another boundary condition. The Nastran solutions are presented in the figures 7, 8, 9 and 10.

3. Conclusions

We confirm that the pressures field (figures 5, 6) are high in the disc part and inlet zone and will be decreased each time with the increasing of disc lifting. This pressure known as the system pressure must equalize the spring force in order the fluid to start to flow past the seat. The stresses and strains fields (figures 7, 8, 9 and 10) reflects that the maximum values will be inside of the round and square flanges, near the bolts holes, because of the fix constraints. During functioning, the most stressed parts of assembly are the seat, disc and spindle. Thus the conditions to have stress-cracking corrosion may appear, and because of the working conditions (stresses and humidity) is advisable to replace them with the new parts after a proper inspection.

4. References

[1] Dumitrache C, Călimănescu I and Comandar C 2014 Naval standard safety valve design using CAD solutions Applied Mechanics and Materials 658 65-70
[2] What are Safety Valves and Pressure Relief Devices, http://www. Pipingguide.net/2013/11/what-are-safety-valves-and-pressure.html
[3] Sallet D W, Nastoll W, Knight R W, Palmer M E and Singh A 1981 Experimental investigation of the internal pressure and flow fields in a safety valve ASME 14(3) 1-8
[4] Sallet D W and Somers G W 1985 Flow capacity and response of safety relief valves to saturated water Flow Process Safety Progress 4(4) 207–16
[5] Francis J and Betts P L 1997 Modelling incompressible flow in a pressure relief valve Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering 211(2) 83-93
[6] Sonawane V J, Rane T J, Monde A D, Vajarinkar R V and Gawade P C 2012 Design and Analysis of Globe Valve as Control Valve Using CFD Software IOSR Journal of Mechanical and Civil Engineering 63-71
[7] Del Toro A 2012 Computational Fluid Dynamics Analysis of Butterfly Valve Performance Factors (Utah State University, Logan, Utah: Thesis of Master of Science in Mechanical Engineering)
[8] Deleanu D 2008 Approximate solutions to an appropriate model equation for finite-amplitude waves on shallow water surfaces Journal of Marine Technology and Environment 1 27-30