3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14-16, 2009, Brno, Czech Republic

Vibration and chattering of conventional safety relief valve under built up back pressure

S. Chabane*, S. Plumejault, D. Pierrat, A. Couzinet,

Cetim, 74 route de la jonelière, BP 82617, 44326 Nantes Cedex 3 - FRANCE

M. Bayart

WEIR - POWER & INDUSTRIAL – France, SEBIM - SARASIN RSBD

ZI La Palunette 13165 Châteauneuf les Martigues Cedex

ABSTRACT

Safety relief valves are devices designed to open when the pressure in the process to be protected exceeds the design pressure. However, in industrial practice, it often happens that the outlet of these valves are canalized through discharge lines which can be different from atmospheric, then there is a built up pressure generated by the flow in the piping which is superimposed to the back pressure in the discharge system. As a consequence, the initial sizing and selection of the safety relief valves, using results from tests conducted under conditions without back pressure are not necessarily valid. The manufacturers often use empirical rules of calculations to assess the performance characteristics in this type of use.

When a safety relief valve is used with substantial back pressure (up to 30% of the set pressure), it is common practice to use a balanced construction. The balancing effect is generally obtained by a balancing bellows.

When the back pressure downstream the valve is considered low (up to 10% back pressure recommended value), a conventional relief valve (i.e. without balancing devices) can in theory be still used. However, even for lower build up back pressure levels, fluttering and chattering of the relief valve disc may occur. This may not only lead to poor operating conditions of the valve, but the damage can be dangerous even when this build up back pressure reaches a value far below 10%.

As an alternative to complex physical modelling as fully coupled analysis of the fluid-structure interaction based on the three dimensional Naviers-Stokes computations, a composite coupled 1D-approach is developed. Inlet and outlet conditions of the safety valves are modelled respectively by thermodynamic 1D-model and wave propagation model preserving the physical phenomena. The safety valve is described by dynamic 1D-model where the hydrodynamics forces applied to the moving disk are given by empirical rules. These empirical rules are predicted using CFD computations for different flow conditions. This methodology is validated by comparison with experimental data. Finally the composite coupled 1D-approach reproduces the unstable behaviour of the safety valve and could allow drawing up more efficient usage rules.

KEYWORDS

Safety valve, Vibrations, Experimental investigations, CFD computations

* *Corresponding author:* Fluids and flows Technologies, CETIM, Nantes, France, phone: +33 240 373 510, fax: +33 251 864 623, email: said.chabane@cetim.fr

1. GENERAL OVERVIEW

Safety valves are crucial devices in the industry. Indeed, these simple and robust design valves are the ultimate protection when all other are insufficient. The poor design or damage to these devices can be disastrous for the environment or, worse for people around.
A safety relief valve is composed of a disc maintained pressed against a nozzle by a spring. While the pressure forces on the upstream face of the disc are below the force applied to the spring, the valve is closed. If an accidental overpressure appears in the process under protection, the pressure forces become high compared to spring elastic forces and the safety valve opens. Thus the pressure in the process under its protection is reduced to an acceptable pressure.
Most of the chemical, petrochemical, and nuclear energy codes and regulations are essential tools for sizing of valves (i.e. API [1], ASME [2] and other national codes). These recommendations are mainly of good sizing but refer only to a summary of the limitations that happen in routine practice.
When a safety valve protects industrial processes carrying non-hazardous product (air, water vapour, liquid water or any other inert), this valve exhausts directly to the atmosphere.
Obviously, as can be easily understood, when the evacuated products become dangerous, these valves are channelled through pipes to provided tanks. These pipes create downstream backpressure in the safety valve which could be superimposed in some cases to the existing initial pressure.

To avoid the disadvantages due to excessive backpressure, the valves are fitted with a system to balance the forces applied to the pressure downstream of the valve [**Fig.1**].

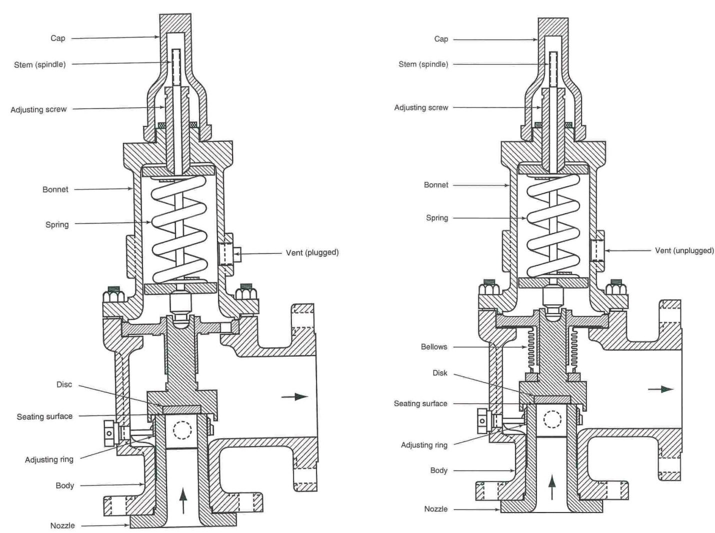


Fig.1: *Conventional and balanced bellows pressure relief valves [1]*

Regarding the balanced valve, the characteristics are not supposed to be degraded up for backpressure of about 25% of set pressure and beyond it is supposed to operate normally up to a backpressure level of about 40% to 50% of set pressure with a limited loss of capacity.
Experimentally, it was found for balanced or unbalanced safety valves, vibrations may occur for high values of backpressure, where they are supposed to work normally; this phenomenon is associated to reduced flow characteristics. Therefore, experimental results led us to evaluate the effect of back pressure on the operating range of safety valves.

The safety relief valves working with back pressure have been seldom studied, and few publications deal with this subject. Safety valve operating with back pressure was handled by Dossena and al. [3] in incompressible fluid. The effect of instabilities in the valve settings was highlighted by Pluviose [4] and a similar approach has been developed by Föllmer and al [5].

2. THEORITICAL CONSIDERATIONS ON THE SAFETY VALVE

2.1 Flow equations used in sizing safety valves for gas

The flow rate through a safety relief valve operating with compressible fluid depends on the pressure ratio between the upstream and downstream. This mass flow is given by the following equation:

$$q_s = \frac{K_d A P_i}{\sqrt{r T_i}} \left(\gamma \left(\frac{2}{\gamma+1} \right)^{\frac{\gamma+1}{\gamma-1}} \right)^{\frac{1}{2}} C_S \tag{1}$$

The constant C_S depends on the flow regime. When the flow is critical in the minimum area of the valve, the coefficient C_S is equal to 1. However, when the flow regime is subcritical and the pressure ratio between downstream and upstream pressure of the valve is greater than b which is given by

$$b = \left(\frac{2}{\gamma+1} \right)^{\left(\frac{\gamma}{\gamma-1} \right)} \tag{2}$$

the coefficient C_S depends on the pressure ratio between upstream and downstream, and is given by the following relation:

$$C_s = \left(\frac{2}{\gamma-1} \left(\left(\frac{P_b}{P_i} \right)^{\frac{2}{\gamma}} - \left(\frac{P_b}{P_i} \right)^{\frac{\gamma+1}{\gamma}} \right) \left(\frac{\gamma+1}{2} \right)^{\frac{\gamma+1}{\gamma-1}} \right)^{\frac{1}{2}} \tag{3}$$

The effect of the back pressure is taken into account by a coefficient K_b as:

$$q_{sCP} = C_b q_s \tag{4}$$

2.2 Effect of back pressure on conventional pressure relief valve.

In current industrial practices, the correction coefficient C_b is used from charts given in some standard codes. The most used is the one published by the American Petroleum Institute: the API 520 code gives charts which could predict the characteristics of the valve when there are no testing results.

The charts given in API 520 represent an average of C_b obtained by testing several safety valves. The chart on the figure [**Fig.2**] represents the correction factor for a balanced valve tested with compressible fluid for two levels of overpressure.

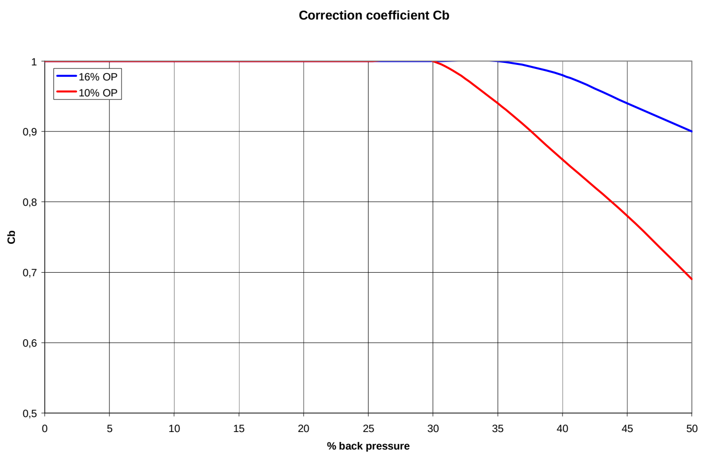


Fig.2: *Correction coefficient C_b for compressible flows*

These correction coefficients, obviously do not take into account dynamic effects and instabilities which may occur when high back pressure is applied. The experiment shows that instabilities could occur between 15% back pressure for some applications and a value of back pressure over 45% of the opening pressure [**Fig.3**].

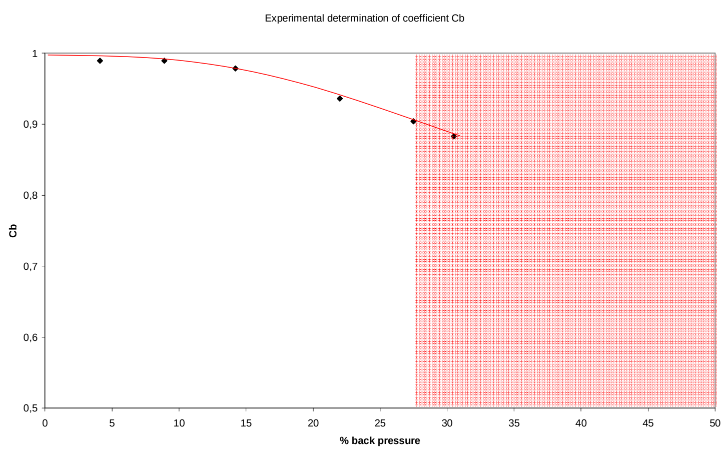


Fig. 3 *: Correction coefficient and instability zone*

3. EXPERIMENTAL FACILITIES IN COMPRESSED AIR

The apparatus is mainly composed of emptying compressed air contained in a primary reservoir, where the air is at 200 bar to the required downstream test pressure [**Fig.4**].
The flow rate is measured using flowmeter devices composed of 4 Coriolis type flowmeters. These flowmeters are linked to a buffer tank where the safety relief valves are ready to be tested.
The following components between the primary and the buffer reservoirs are used:
- A manual actuated valve 2" in PN 200,
- A ramp of two control valves in 2" and 1 / 2 ” that regulates the downstream pressure.
- A pipe in DN 100 and PN 50 of 200 m long to distribute compressed air in the downstream testing tank. This expansion of air is deported to heat the compressed.
- The assembly line leads to a manifold distributing a ramp of flowmeters, where solenoid valves are used to isolate each line. On this manifold is mounted a 2" PN 200 safety relief

valve that protects the facility. This valve is set at 40 bar. In addition, a valve allows the air to supply the whole line, it works with the safety valve and is controlled by the security system.
- A tank of 2.5 m^3 PN 40 is used as buffer reservoir. The safety relief valves under testing are mounted directly on this tank: it is equipped with three flanges: DN 50 PN 40, DN 65 PN 40 DN 100 PN 40.
This test facility is also equipped with a silencer connected to the exhaust, with an acoustical panels as well as a mound in the ground to limit the spread of noise during the test.

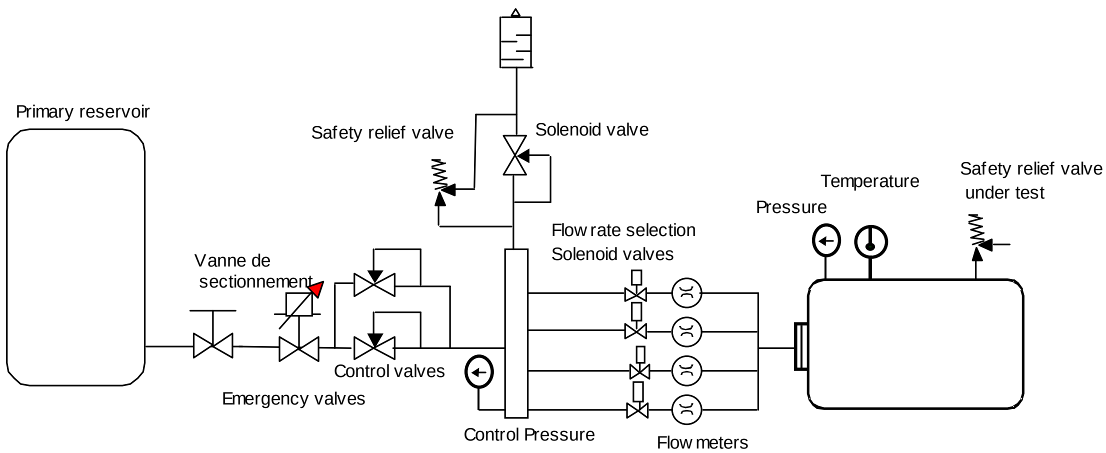


Fig.4: *CETIM's safety valve test facilities in air*

The maximum that can be achieved during the tests may not exceed 40 bar with 0.25% of precision. The maximum flow that can be measured is 13 kg / s with 1% operating precision.

The back pressure is generated by a system composed of a pipe connected to the downstream of the safety valve [**Fig.5**]; this pipe is closed by a control valve which allows generating a variable back pressure.

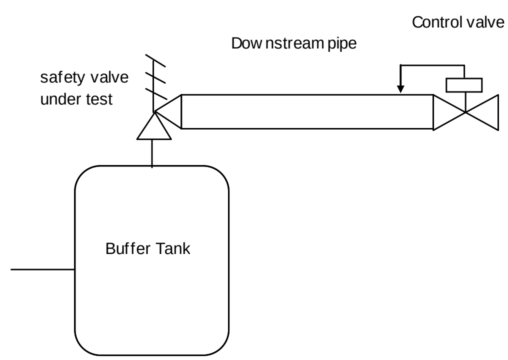


Fig. 5: *Safety valve under test and back pressure generating system*

4. CFD MODELLING OF SAFETY VALVE

4.1 Geometry and grid generation

The computational domain used to perform numerical simulations is extracted from the real geometry of a 1""1/2 relief valve. It consists of a nozzle tip, a disc valve, and a chamber as the figure [**Fig.6**] shows. All the geometric details have been taken into account. Geometrical simplifications are often assumed for the numerical simulations [5] [6] but when compressible flows are simulated, the development of pressure waves can be involved by the geometry of relief valve. Moreover, no symmetric considerations have been supposed as the compressible air flow is considered.

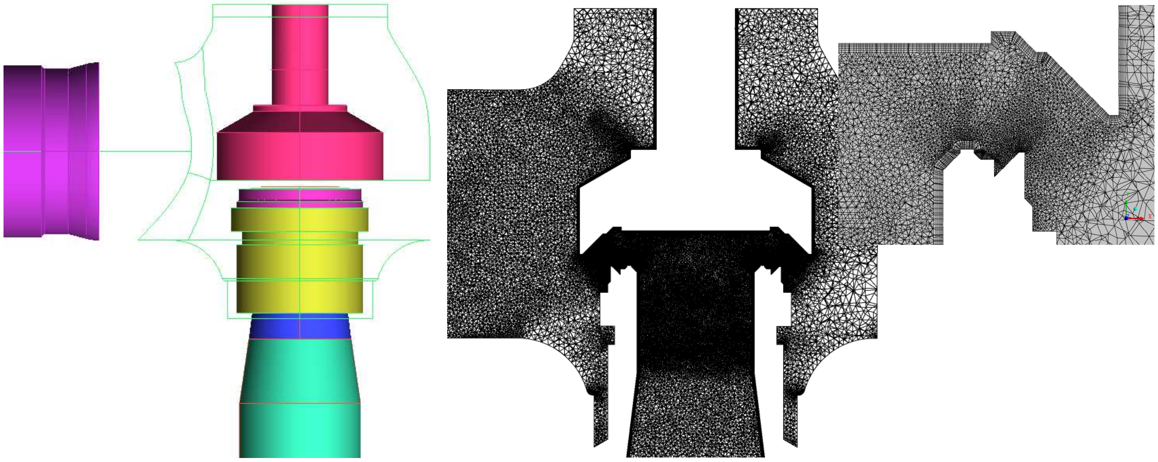


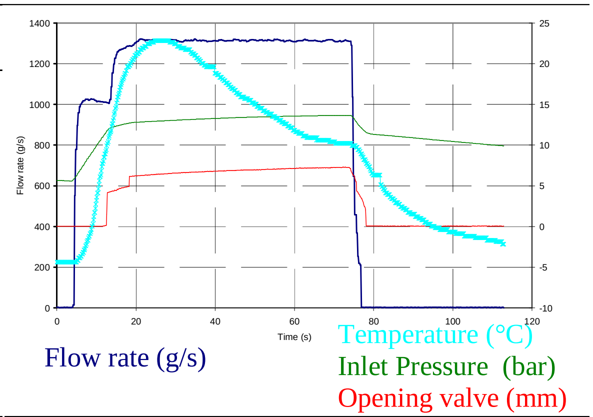
Fig. 6: *Grid view of the computational domain*

Considering the real geometry, the unstructured approach has been used to generate the computational mesh. The grid is composed of 15 500 000 elements (3 000 000 nodes); tetrahedral elements and prismatic elements close to the walls of the nozzle and the disc valve in order to ensure that y^+ is below 100. Accurate resolution near the boundary layer is carried out by using wall functions. Following the opening of the valve, the nodes number is maintained to be sufficient to describe the flow between the disc and the nozzle. The inlet and outlet conditions are distant aiming not to disturb the solution. The grid view of computational domain is given figure [**Fig.6**].

4.2 Numerical approach and steady computation set up

The compressible air flow is simulated for different opening valve considering steady state approach. Starting from experimental data [**Tab.1**], three positions of disc opening have been chosen and corresponding mesh have been generated: the disc of the valve is fixed at these positions and three computations are performed.

	P1	P2	P3
Time (s)	t = 13	t = 15.8	t = 28
Opening valve (mm)	1.86	4.55	6.5
Inlet Total pressure (bar)	11.71	12.4	13.01
Temperature (°C)	9.92	15.9	22.8



Tab. 1: *Specifications of three computations corresponding to the different valve openings: data extracted from experimental investigations*

Three-dimensional Reynolds-Averaged Navier-Stokes equations are solved by the CFD code CFX-11. For the boundary conditions, total pressure obtained by experimental tests corresponding to the valve opening is imposed for the inlet condition; the corresponding temperature is also imposed [**tab.1**]. The outlet condition is given by a static pressure.

The turbulence is simulated by the k-epsilon turbulence model based on the concept of turbulent viscosity with automatic near wall treatment. The discretization in space is of second order accuracy. The HR discretization scheme is used for time resolution. The residual level reaches four order for all cases but obtaining the convergence for intermediate position is harder than the quasi-full opening. Indeed, for this last case, the valve is stable and the steady state approach is more relevant.

4.3 Numerical results

The flow rates and the force acting on the disc valve are computed from results of the numerical simulations. They are represented following the valve opening on the figure [**Fig.7**]. The computational flow rate is 15% lower than experimental data for the full opening. The gap grows up 20% when the opening valve is lower. Numerical investigations have been performed in order to explain these gaps and they have been shown that is due to different causes:

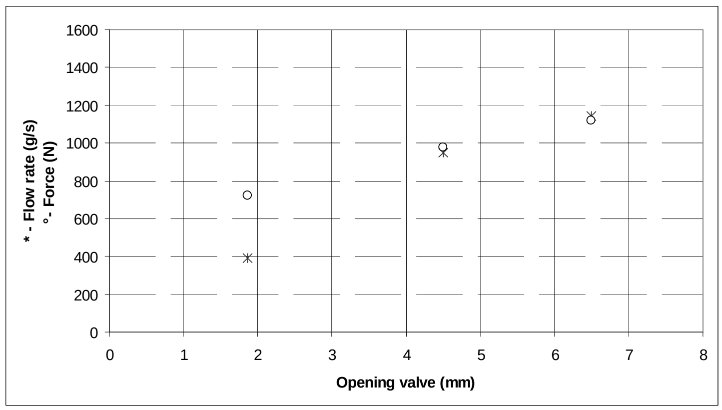


Fig. 7: Flow rate and force acting on the valve following the valve opening

For the intermediate positions, the moving disc can trigger off a complex dynamic of the flow which is hardly predictable by simulating several steady state. It is due to the moving of irreversibility areas which can be disturbed by the elliptic behavior of the pressure field.

Concerning the full opening position, the turbulent characteristics of the flow entail important load fluctuations which play a role in the moving of blocking area. Preliminary tests have also shown that the flow into the valve chamber can imply a significant back pressure.

The hydrodynamic force acting on the valve is varying between two and three times the value of elastic force due to the stiffness of the valve. It would be observed that force fluctuations can exist in the valve chamber particularly for the low opening. It seems so interesting to perform unsteady simulations to understand the complex flow into the relief valve.

The figure [**Fig.8a**] represents the dimensionless dynamic pressure P_{dyn} which is acting on the disc valve for the case P3. The disc valve is colored by the dimensionless static pressure P_{ad}. These dimensionless variables are computed by :

$$P_{ad} = \frac{P_s - P_{Tinlet}}{P_{Tinlet}} \qquad (5)$$

$$P_{dyn} = \frac{\frac{1}{2}\rho u^2 - P_{dinlet}}{P_{dinlet}} \tag{6}$$

where P_{ad} is varying between -1 and 0 while P_{dyn} > -1.

On the figure [**Fig.8a**], the density gradient are plotted on the x-y plan and it can be observed that chocked flows appear both in the nozzle and in the lateral area between the nozzle and the disc. Moreover the pressure distribution on the disc valve can not be represented by a simple function.

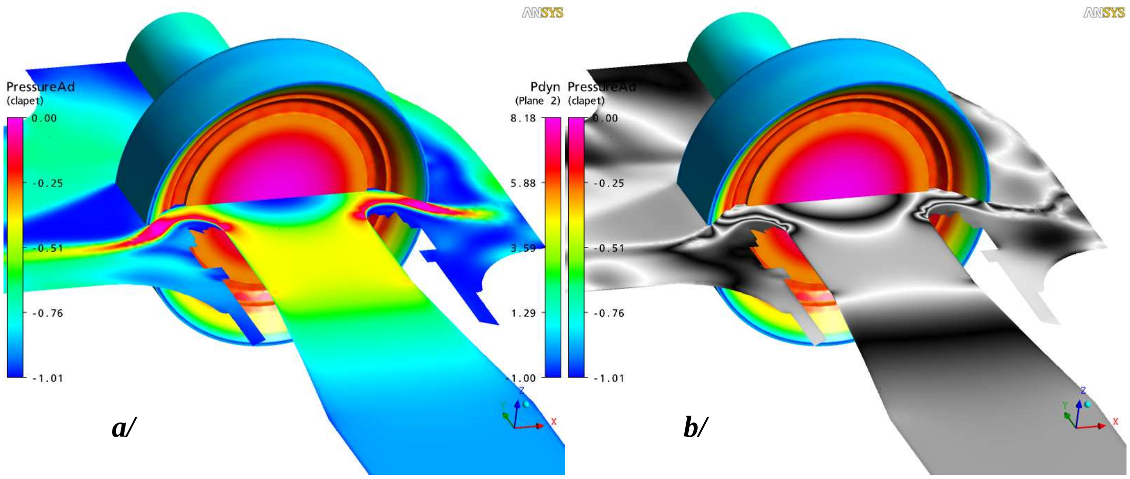


Fig. 8: **a/** Dynamic pressure acting on disc valve **b/** density gradient on x-y plane; case P3

The same variables are plotted on the figure [**Fig.9**] considering the case where the gap between the nozzle and the disc valve is the smaller (case P1). The density gradient are significantly different in this case because the chocked flow appears only along the lateral area to the disc valve. Aiming to perform 1D modelling of safety valve, the pressure on the inner disc can be supposed equal to the pressure generator for the smaller opening of the disc valve (cf. following section).

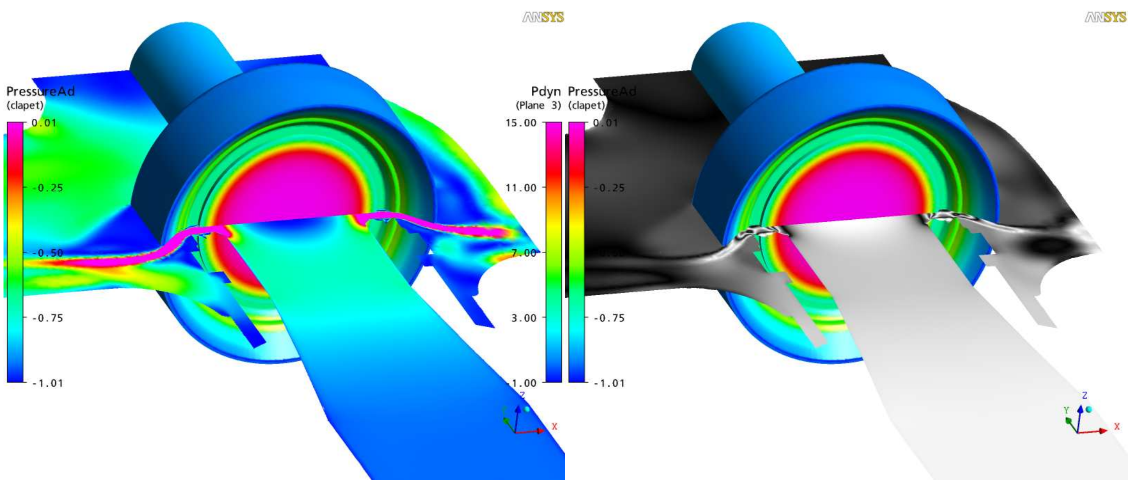


Fig. 9 : Dynamic pressure and density gradient; case P1

5. THERMODYNAMIC MODEL OF SAFETY VALVE WITH TEST CONDITIONS

In compressible flow, the fast unsteady effects are considered not significant. Indeed, because of compressibility of air and gas in general, the unsteady effects are not taken into account. Moreover, the pipe and the control valve downstream involve a capacity to fill and then to create a back pressure. When the equilibrium is not reached, it might cause chattering that could be destructive for the safety valve.
To understand the dynamic behaviour during a test with or without back pressure, the whole test facilities and the valve have been modelled as shown in the figure [**Fig.10**].

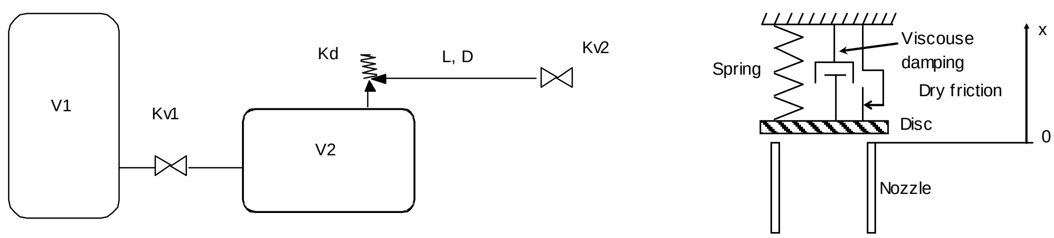


Fig.10: *Simplified test rig and simple valve model*

The law of conservation of mass and energy applied to the entire system with considering the air as a perfect gas allows to describe the variation of the pressure and temperature as here under [7]:

$$\frac{dP_1}{dt} = -\frac{\gamma}{V_1} q_m r T_1 + \frac{\gamma - 1}{V_1}\frac{dQ_1}{dt} \tag{7}$$

$$\frac{dT_1}{dt} = q_m \frac{r T_1^{\,2}}{V_1 P_1} + \frac{T_1}{P_1}\frac{dP_1}{dt}$$

$$\frac{dP_2}{dt} = -\frac{r}{V_2}(q_m - q_s)T_2 - \frac{P_2}{T_2}\frac{dT_2}{dt}$$

$$\frac{dT_2}{dt} = \frac{r T_2}{P2}(q_m - q_s)\frac{(\gamma T_1 - T_2)}{V_2} - \frac{\gamma - 1}{V_2}\frac{T_2}{P_2}\frac{dQ_2}{dt}$$

The masse flow through the upstream control valve is given by the following equations [8] Eq.(8):

$$qm = \frac{\alpha P_1 A}{\sqrt{a r T_1}} \ \text{ if } \frac{\mathrm{P}_2}{\mathrm{P}_1} \le b$$

$$qm = \frac{\alpha A}{\sqrt{r T_1}}\left(2\Delta P\left(P_1 - \frac{a}{2}\Delta P\right)\right)^{\frac{1}{2}} \ \text{ if } \frac{\mathrm{P}_2}{\mathrm{P}_1} > b \tag{8}$$

The coefficient α takes in account the flow coefficient K_V as following :

$$\alpha = \frac{K_V}{4 10^4 D_v^{\ 2}} \qquad (9)$$

The disc of the safety valve is subjected on the one hand to the pressure forces and on the other hand to the force applied by the spring.

The lifting of the valve is governed by the following equation:

$$m\frac{d^2x}{dt^2} + F_F + kx = F_p \qquad (10)$$

The two terms most difficult to explain are the friction forces F_F and pressure forces F_P.

Regarding the pressure forces, as shown in section 4.3 starting from the numerical simulation results, it is possible to consider two separate cases heard to simplify the calculations.

Case 1: flow through the "curtain area"
When the lifting is less than a quarter of the diameter of the nozzle, the chocked flow is located on a "curtain area" as shown in the figure [**Fig.11**].
The pressure on the inner disc is then equal to the pressure generator (this means that any losses in the nozzle and the upstream pipe are neglected).

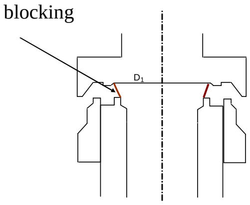


Fig.11*: The critical pressures ration occurs in the lateral area between the nozzle and the disc - "curtain area"*

The pressure effort that occurs when the valve is open :

$$F_P = P_2 S_1 - P_b S_0 \qquad (11)$$

Case 2: flow through the cross section of the nozzle.
When the upstream pressure increases, the valve reaches a lift that is superior to one-fourth the diameter of the nozzle, the minimum is not the "curtain area" but the cross section of the nozzle itself.
The shocked flow is located as following [**Fig.12**].

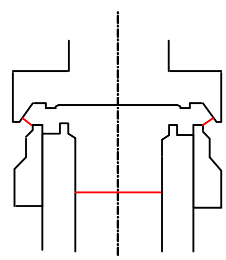


Fig.12*: The critical pressures ration occurs both in the nozzle and in the lateral area between the nozzle and the disc -*

The force applied by the gas jet is taken into account as indicated by the following equation:

$$F_P = P_2^{'} S_1 - P_b S_0 \qquad (12)$$

Where the pressure P’2 is given by

$$P_2^{'} = \left(\frac{2}{\gamma+1}\right)^{\left(\frac{\gamma}{\gamma-1}\right)} P_2 \qquad (13)$$

Concerning the damping forces different types can be distinguished [9] that can be summarized as follows

$$F_F = F_C^{\pm} + B\frac{dx}{dt} + \delta(F_S^{\pm} - F_C^{\pm})e^{(\mp Cv)} \tag{14}$$

Where F_C is the coulomb friction coefficient, F_S the striction or dynamic friction coefficient, B viscous friction coefficient, and v the velocity of the moving part of the valve.
The coefficient δ is given by the accelarion of the stem of the valve as summarized hereunder

$$\begin{aligned} \delta &= +1 \text{ if } \frac{\mathrm{dv}}{\mathrm{dt}} \geq 0 \\ \delta &= -1 \text{ if } \frac{\mathrm{dv}}{\mathrm{dt}} < 0 \end{aligned} \tag{15}$$

6. RESULTS AND DISCUSSION

6.1 Modelling of safety valve without back pressure

To confirm these assumptions, first calculation is performed without back pressure and the results are given in the following figure [**Fig.13**]:

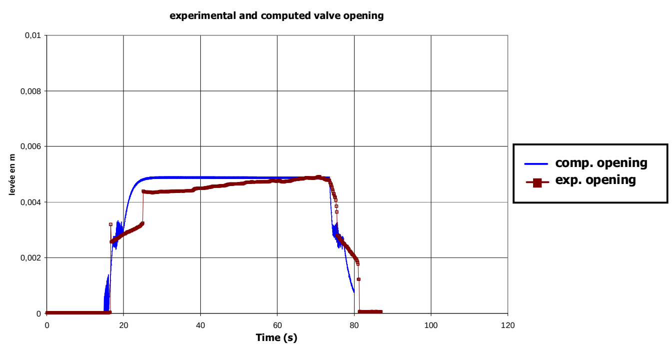


Fig.13: *Variation of the lift during a safety valve test*

The results obtained starting from the implemented model show good agreement with test results. However, it was necessary to change the values of the friction coefficient in order to obtain comparable results. The figure [**Fig.13**] shows that the two levels on the exercise of the reclosing are well represented.

6.2 Modelling of safety valve with back pressure

When the backpressure generated by the air flow is high enough a chattering phenomenon can occur. The simulation shows that the values are fairly well calculated by the simple implemented model as shown in the figure [**Fig.14**]
It is noted that the chattering of the valve disc leads to significant fluctuations of the pressure downstream. These low frequency fluctuations present relatively high amplitude. As the stem of the valve is free during the experimental testing the recorded displacement are higher than the calculated ones.
The damping forces are not easy to take into account. Nevertheless, it can be noticed [**Fig.14**] that the phenomena of low vibration are well predicted during the lifting and the reseating of the disc.

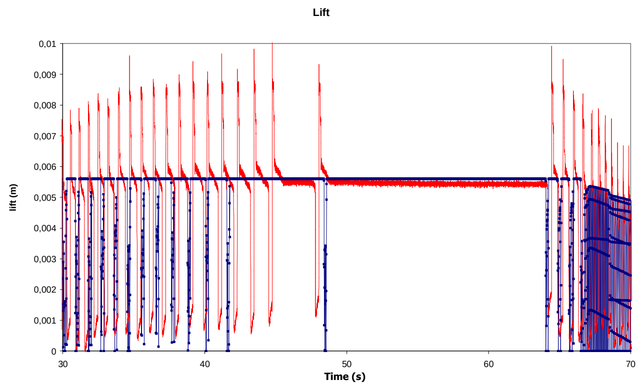


Fig.14: *Comparison of experimental and calculated lift for 1/2G2 valve*

When the valve is fully opened, the valve remains open. This is due to the increasing of leakage between the stem and the body of the valve what grows up the viscous friction. This phenomenon is obviously not taken into account with the modelling approach as shown in the figure [**Fig.15**].

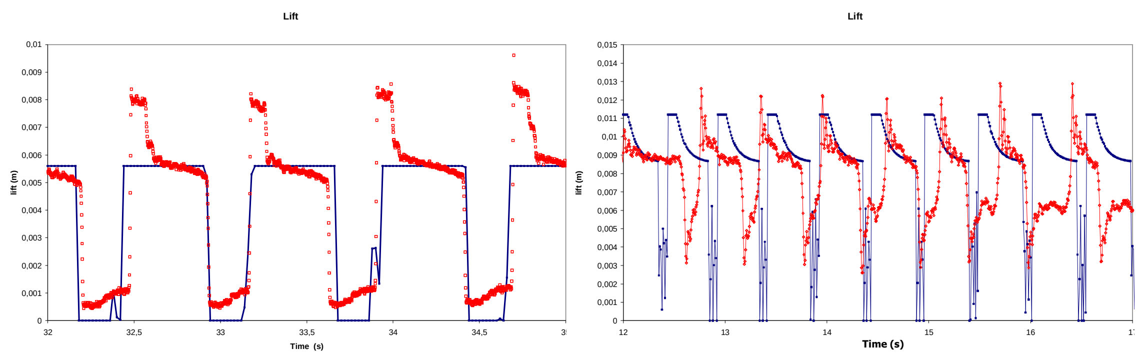


Fig.15: *Test and simulation of lift during a safety valve test with backpressure for 1/2G3 and 2J3 type safety valves*

Nevertheless it can be noticed that the chattering frequency and amplitude of the downstream pressure are reasonably well calculated as showed in figure [**Fig.16**] hereunder.

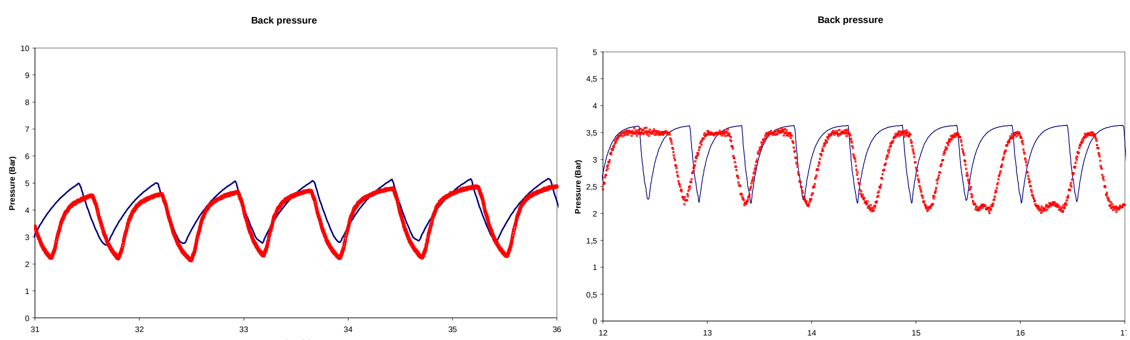


Fig.16: *Test and simulation of back pressure during a safety valve with backpressure for 1/2G3 and 2J3 type safety valves*

7. CONCLUSION

The tests confirm that the effect of pressure is detected only with backpressure of about 10% as specified in the literature.
For higher values of backpressure, the characteristics of the air flow are slightly affected. For values of pressure even still greater, in the order of 25% to 30% of set pressure, the chattering and vibration occur and grow up when the values of backpressure become greater.

This theoretical and experimental studies, conducted on the behaviour of unbalanced valves with compressible fluid and built up backpressure allows us to draw the following conclusions:

- The characteristics of a valve in the presence of backpressure generated by the flow are partially taken into account by the existing charts. The pipes with non neglectible volume downstream safety valves can sometimes, cause vibrations of the valve which can damage its operation.
- The existing charts are not very precise. It is obvious that they must be handled with great caution. Indeed, it was noticed a large dispersion of results obtained with the same type of valve. Furthermore, the limit values specified by the API standard are verified; subject to confirm the initial results in our possession.
- The vibration behaviour of a valve with backpressure must be seriously evaluated even when the conditions of low backpressure, the fluctuation of the downstream pressure can produce piping vibration and the device can be destructed.

The CFD computations of the compressible flow present respectable agreement with the experimental tests and allow to improve thermodynamic model of the safety valve by locating the blocking areas. The simple design method developed in this paper could can be used with appropriate geometrical and flow characteristics of the safety valve; under conditions that the connecting pipes are well designed and if the valves will not be affected by the backpressure.

Charts still have the merit to exist and contribute to the design of valves without performing tests with backpressure, which can be quite heavy and complicated to implement. However, safety margins should be taken seriously, the model developed here can generate interest.

8. ACKNOWLEDGEMENTS

The support of the French Association of Pump and Valves Manufacturer and especially the Safety Valves working group is gratefully acknowledged.

9. NOMENCLATURE

A	(m^2)	Flow area of safety valve	P	(Pa)	Pressure
α		Flow coefficient of control valve	Q	(J)	Heat transfer
b		Pressure ration	q_s	($kg.s^{-1}$)	Safety valve flow rate
B		Viscous damping coefficient	q_m	($kg.s^{-1}$)	Upstream valve Flow rate
C_b		Back pressure coefficient	r	(j/kg.s)	Constant of gas
D	(m)	Safety valve Disc diameter	t	(s)	Time
d	(m)	Control valve diameter	T	(K)	Temperature
F	(N)	Forces	V	(m^3)	Volume
L	(m)	Length of the downstream pipe	γ		Specific heat ratio
K_d		Flow coefficient of the valve	m	(kg)	Mass of the moving part of the valve
K_v	(m^3/bar)	Flow coefficient of the control valve draft tube length			

Subscripts

1 : upstream safety valve conditions

2 : downstream safety valve conditions

10. REFERENCES

[1] API recommended practice 520 part 1 Sizing and selection, sixth edition March 1993.

[2] ASME VIII Div1. Pressure Vessel Code Of the American Society of Mechanical Engineers.

[3] DOSSENA V, MARINONI F, GAETANI P, OSNAGHI C “*On the influence of back pressure and size on the performance of safety valve*” pipes and component analysis and diagnosis, ASME pressure vessels and piping conference, august 25-29, 2002, CANADA.

[4] M.PLUVIOSE „ Contribution à l’étude des instabilitées d’écoulement dans les organes de réglages“. Thèse d’état es Science, Paris, Decembre 1984.

[5] FOLLMER B., Schnettler A., *Challenges in designing API safety relief valves*, Valve World, october 2003.

[6] DEHOUX F., FERRARI J., *CFD study of a 1""1/2 safety valve*, EDF R&D, Internal report

[7] S. CHABANE “*Essais de soupape avec contre-pression*” Rapport CETIM N°022826, Juin 2009.

[8] C. WARTELLE “*Débit des appareils pneumatiques”* Les Mémoires Techniques du CETIM , septembre 1972.

[9] BELGHARBI,M.,THOMASSET,D.,SCAVARDA,D., SESMAT,S., « *Analytical model of flow stage of pneumatic servo-distributor for simulation and nonlinear control* » Sixth Scandinavian International Conference On Fluid Power, Tampere, Finland, 1999.