CHALMERS

On the Stability of Pressure Relief Valves

A numerical study using CFD

Master's Thesis in the Master's program Innovative and Sustainable Chemical Engineering

ANNA BORG
SOFIA JAKOBSSON

Department of Chemical Engineering
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2014
Master's Thesis 2014:06

MASTER'S THESIS IN CHEMICAL ENGINEERING

On the Stability of Pressure Relief Valve

A numerical study using CFD

ANNA BORG & SOFIA JAKOBSSON

Department of Chemical and Biological Engineering
Division of Chemical Engineering

Chalmers University of Technology
Gothenburg, Sweden 2014

On the Stability of Pressure Relief Valves
A numerical study using CFD
ANNA BORG & SOFIA JAKOBSSON

Master's Thesis 2014:06
Department of Chemical Engineering
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone: +46 (0)31-772 1000

Chalmers Reproservice/Department of Chemical Engineering
Gothenburg, Sweden 2014

Abstract

A numerical model based on a spring loaded safety relief valve on the residual heat system at the nuclear power plant Ringhals has been constructed with CFD using *ANSYS 14.5.7*. Through stationary simulations, characteristics for the valve has been obtained and compared to the dynamic opening of the valve. The stability of the valve has been investigated by varying the upstream conditions. It was concluded that the transient opening of the valve with an inlet line of $0.25\,m$ can be predicted by the valve characteristics. For systems with longer inlet lines there are transient effects that are not captured by stationary simulations. If the inlet line is sufficiently short or sufficiently long, the valve does not exhibit unstable behavior. The numerical model of the valve can capture the interaction between the pressure wave propagation in the simulation and the disc position. The effect of frictional elements has been examined by implementing a hysteresis factor to the closing of the valve. This proved to be an effective way of attenuating chatter. For a constant inflow of mass, the disc frequency of the oscillations was independent of the magnitude of the mass flow. The amplitude of the oscillations, however, was not. A system with large volume had a tendency to oscillate with a lower frequency than a system of small volume.

Keywords: *Stability, Pressure relief valve, CFD*

Preface

This Master of Science thesis has been performed by Anna Borg and Sofia Jakobsson, students at the Master's program Innovative and Sustainable Chemical Engineering at Chalmers University of Technology, Gothenburg Sweden. The thesis has been performed in collaboration with ÅF Industry, Ringhals (Vattenfall) and the department of Chemical Engineering at Chalmers University of Technology. It was supervised by Oscar Granberg at ÅF Industry and Daniel Edebro at Ringhals. The examiner at Chalmers University of Technology was Bengt Andersson.

Acknowledgments

We would like to sincerely thank our supervisors Oscar Granberg and Daniel Edebro for dedicated help regarding any problems encountered. We would also like to thank all the people at Ringhals and ÅF Industry - Technical Analysis, for their welcoming and interest in this thesis.

Anna Borg & Sofia Jakobsson, Gothenburg 2014-06-03

Contents

1. Introduction

A safety valve protects a system from elevated pressures by discharging the pressure media. Safety valves are thus an important part of all types of industries and their well-functioning is of great importance since they often are the last resort to prevent accidents. There are a number of different properties of the safety valve and the system to which it is attached to that are thought to affect the response of the valve. By examining the effect of different system properties, their importance for the valve stability might be established and unstable valve behavior might be prevented. It is also of interest to create a simple tool that can be used to assess whether or not a certain valve will operate stable prior to installation.

1.1 Objective

The first objective of this thesis is to investigate if steady-state characteristics can be used to make a first estimation of the opening behavior of a valve. The second objective is to study a number of system variables and evaluate their influence on the valve stability. Pressure drop over the inlet piping, different pipe lengths and different volumes of the valve system are features that will be examined. Safety valves can be damped by friction to prevent unstable operation, the effect of friction hysteresis will be examined. Safety relief valves can be proportional with high-lifting regions or popping, further in this thesis it will be investigated whether a valve itself can be characterized as any of these categories or if it is the system to which it belongs that determine the behavior of the valve.

1.2 Constraints

1.2.1 General constraints

- All tests will be executed with a numerical model based on an existing pressure relief valve.
- The model does not aim at completely replicating an actual valve behavior.
- Only effects generated by the preceding system will be investigated.

1.2.2 Model constraints

- The entire domain is filled with water as initial conditions of the simulations.
- The temperature of the fluid is constant, $298\,K$.
- Axisymmetric 2D-model.
- Only modeling fluid and rigid body motion.
- The modeled valve is bellowed and the back pressure is 1 atm.

1.3 Method

The safety relief valve was simulated with *ANSYS Fluent v14.5*. The geometries where drawn in *DesignModeler* and the mesh was created in *ANSYS Meshing*. Steady state characteristics were created with a 2-dimensional geometry and compared to transient simulations. The transient simulations were also performed in 2-dimensions in order to investigate the transient response of the valve to varying inlet pipe length, discrete pressure drop and variable system volume. The transient simulations were conducted with a gradual increase of inlet pressure or inlet velocity. Further, a sensitivity analysis to validate the accuracy of the simulations was also performed.

2. Background

This chapter will explain the basics behind safety valves and their operation. The concept of *valve characteristics* will be introduced and explained along with the classification of safety valves based on their opening behavior. Last, a part of the design standards regarding safety valve issued by *ASME* will be presented. The concepts presented in this section/chapter will be used throughout the rest of the thesis.

2.1 Safety valves

Safety relief valves are used in all kinds of industries. Their task is to protect the system from elevated pressures that may result in damage to equipment and operators. Safety relief valves are designed to respond to the pressure in the system and are completely automatic, that is, no external signal from an operator or regulator is required for the valve to open.

A simple way to describe a safety relief valve is to imagine a disc/weight placed horizontally on a (pipe) opening. When the pressure in the pipe is high enough, the disc will be pushed from its position, allowing for the media inside the pipe to flow out. The safety valve considered in this thesis is spring-loaded which means that the disc is pressed to the opening by a spring. The spring can be tensioned to yield different opening pressures for the valve. Increasing the spring tension also increases the pressure required to lift the disc and open the valve. Figure 2.1 shows a spring-loaded safety valve.

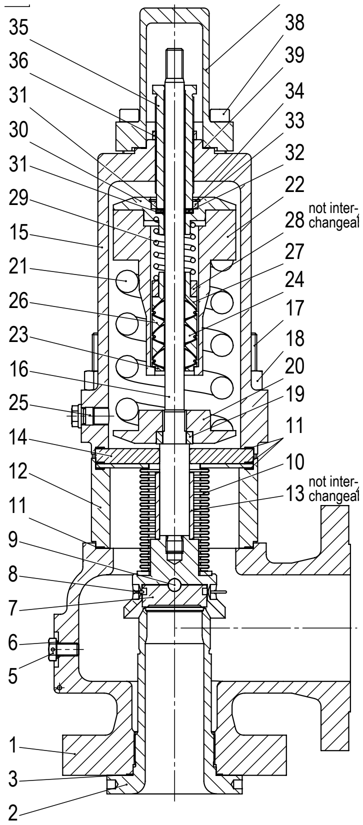
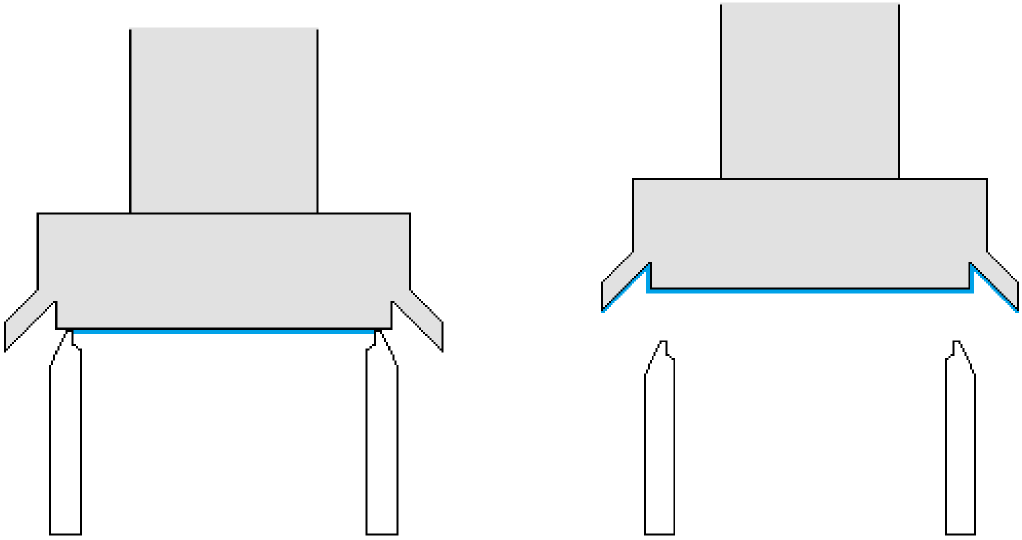


Figure 2.1: Picture of a safety valve

As can be seen in figure **??**, the diameter of the disc is larger than the diameter of the pipe. This feature has an important impact on the opening behavior of the valve. When the valve is closed, the area exposed to the pressure in the pipe is the same as the cross-sectional area of the pipe, see X in figure 2.2a. As the valve opens, the exposed area increases, see figure2.2b.



(a) Showing the area exposed to the pressure in the pipe for a closed valve

(b) Showing the area exposed to the pressure in the pipe for an open valve

Since the pressure force is proportional to the area it works on, the force will increase as the valve opens until the full cross-sectional area of the disc is available. This means that the pressure required to open the valve from a closed position is higher than what is needed to further open the valve. The fluid forces exerted on the disc can be evaluated for different disc positions and pressures and the results can be graphically displayed. Such a graph is called *valve characteristics*. Figure 2.3 below shows two example of such valve characteristics.

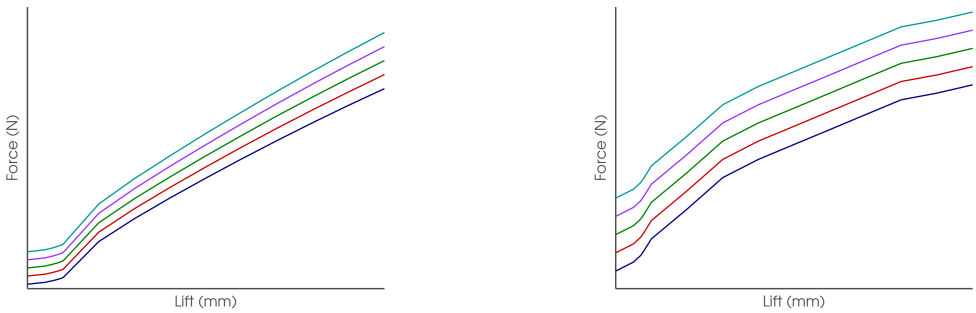


Figure 2.3: Examples of different valve characteristics. Each line in figure represents a system pressure. The force increase as you move upwards in the figure. The lift increase to the right.

2.2 Design Standards

To ensure proper operation of a pressure relief valve there exists a *Design standard.* Some of these will be presented below to give a better idea of how a safety valve is designed and working. The criteria are valid for relief valves used in systems with liquid media. The following design standards are proposed by *ASME* to ensure a stable and reliable operation of the safety valve.

- The opening pressure, P_{open}, should be lower or equal to the design pressure of the system.
- The valve should be completely open at a pressure of 110% of the opening pressure or at a pressure $0.21\,bar$ over the opening pressure.
- The allowed tolerance for the opening pressure in safety valves with opening pressures between 0 and $4.8\,bar(g)$ is $\pm 0.14\,bar(g)$ and for opening pressures above $4.8\,bar(g)$, the tolerance is $\pm 3\%$.
- The pressure drop over the inlet pipe is limited to 3% of $1.1 \cdot P_{open}$.
- The valve should ensure that the pressure in the system it protects never exceeds 110% of the design pressure for expected transients. For unexpected transients, the pressure cannot exceed 120% of the design pressure.

2.3 Popping or proportional valves

Safety valves are often divided into proportional valves and high-lifting or popping valves. This categorization assumes that the valve behaves in a certain way regardless of the system it is attached to. The same type of safety relief valve are found on inlet pipes of greatly varying length. How this affect the behavior of the valve is often neglected.

A popping or high-lifting behavior is when the valve opens extremely rapid. There is almost no pressure change from the start position of the valve to the end position, in the high-lifting zone. A proportional valve starts to open at the set pressure and continues to open proportional to the pressure increase. If a valve is used in a system which contains an incompressible media, it is said that a high-lifting behavior should be avoided. This is due to the fact that a liquid can not sustain the pressure in the same way a gas can since a gas can expand fast, preventing rapid pressure drop.

A high-lifting or popping valve reliefs the pressure very quickly during the high-lifting region, releasing a large amount of media in a short amount of time. For an incompressible media, this fast release results in a large pressure drop below the disc, since the media can not withhold the pressure. This can lead to a pressure decrease large enough to cause the valve to start closing, which in turn result in a build up of the pressure. The high-lifting region can thus be unstable. This behavior is less unstable for a system containing gas, since the pressure drop during the opening is limited.

2.4 Safety valve instabilities

Safety valve instabilities are traditionally divided into three different categories, *chattering*, *fluttering* and *cyclic blowdown*. The most destructive of these instabilities is chattering. During chattering, the disc oscillates with a high frequency and often also a high amplitude. For worst case scenarios, the disc moves between its two extreme points, the completely closed position and the maximum lift. The frequency of these oscillations might exceed several hundred hertz. Apart from directly damaging the disc and the seat of the valve, such frequent oscillations causes large disturbances in the system to which the valve is connected. This can result in severe damage to other equipment. Fluttering is another type of oscillations, where the disc tends to not hit the seat or the top position, but oscillates around a point (a lift), creating pressure pulsations. Fluttering is less damaging than chattering, but it wears out components of the valve and is not desirable.

The third instability, cyclic blowdown, is similar to chatter. The valve opens and closes periodically. It occurs however with a lower frequency than chattering and may therefore be less destructive. The pressure in the system is decreased and built up again periodically. The time it takes to relieve and restore the pressure in the system determines the frequency of the oscillations. Small volumes will be depressurized in less time than large ones, since there is less fluid to release.

2.5 Relieving Transients

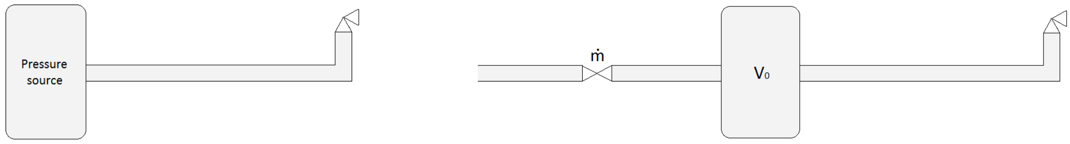
When studying safety relief valves two different extreme cases can be identified and are commonly encountered.

- In the first system, displayed in figure 2.4a, the valve is connected to a large system, typically a reactor vessel or a test rig with gas cushion. Since the system is large and the opening of the valve will not relieve the system. this represents a source of constant pressure. The valve is directly connected to the pressure source through a pipe. For this system, the pressure increase at the position of the safety valve is due to a pressure increase in the vessel to which the safety valve is connected. When conducting experiments on safety valves, their response to this upstream condition is commonly examined.

- In the second case, displayed in figure 2.4b, the pressure increase at the safety valve is due to the mass flow through the control valve. The mass flow to the system is limited and related to the pressure difference over the control valve through the following relationship,

$$\dot{m} \propto \sqrt{\Delta P} \tag{2.1}$$

  For a large pressure difference over the control valve, the mass flow to V_0 will be constant. The opening of the safety valve affects the pressure in the system and if the mass input is less than the capacity of the valve, the pressure of the system will be relieved through the opening of the valve.



(a) Pressure increase at safety valve due to pressure increase in vessel.

(b) Pressure increase at safety valve due to mass flow through control valve.

Figure 2.4: Schematic view of the two systems

2.6 Previous work

In this section a summary of previous work on safety relief valves will be presented. Several authors have come to the same conclusions about which factors that significantly affect the stability of the valve. Some of these articles will be presented here.

To completely understand the dynamic behavior of a safety relief valve, the interaction between the solid parts and the fluid media must be carefully considered. The opening and closing of a valve is often a rapid event which causes abrupt changes in the flow. These events are often the origin of a water hammer, which results in a pressure change that travels trough the system. For a system consisting of a valve on a pipe, the fluid and solid parts are highly coupled. They will affect each other through friction between the pipe walls and the flow and the pressure in the fluid will result in axial stresses of the pipe. There will also be a coupling between a component of the system which can move in the same direction as the pressure propagates (for example the disc in a safety valve), and the fluid in the system [12]. The two first occurs along the whole axial direction of the pipe. In the event of a water hammer, the coupling between the axial stresses and the pressure in the fluid leads to a precursor wave which travels ahead of the water hammer wave. The kinetic energy of a component of the system can cause a pressure change in a pipe and result in compression and stress wave propagation [10]. The pressure wave is assumed to follow the relationship derived by Joukowsky, but it has been shown that when a system is subjected to a rapid valve closure and axial pipe motion occurs, this relationship underestimates the pressure rise. Further on in *Fluid-structure interaction in liquid-filled pipe systems: A review* [12], it is stated that the main cause for unstable behavior of a safety relief valve is the changes in the hydraulic forces acting on the disc. A phase change from sub cooled water to steam resulted in high frequency chatter.

The influential article *Analysis of safety relief valve chatter induced by pressure waves in gas flow* [5] describes the pressure fluctuations resulting from the opening and closing of a valve. Decompression and expansion waves change the velocity of the fluid and can result in larger friction losses which leads to further decreasing of the pressure. In the experiments conducted on pipes of different sizes, it was found that as the valve opens a decompression wave is created that slows down or even reverses the motion of the disc. Depending on the pipe, this can in turn cause the valve to start opening again, since it results in a compression wave. Or, it can cause the valve to oscillate. The larger pipe is thought to act as a buffer. The reflected waves act stabilizing when the reversed motion is limited and the pressure changes not to large. The 3% -rule has been studied and is found to be sufficient if the diameter of the inlet line and the valve nozzle is equal. The authors state that an effective way of reducing chatter is to have a large inlet diameter and a small valve nozzle diameter but also that this is in conflict with economic interests. Further, a modification to previously derived relationship based on parameters characteristic of the system, for determining the permitted length of the inlet line is presented. The important parameters are the discharge function, the back pressure, the opening time of the valve (provided by the manufacturer), the nozzle-, pipe area ratio and the density of the fluid. The modification includes the set pressure of the valve.

According to a review article by Darby [4], the standards set by *ASME* are derived during steady state conditions and may therefore not be applicable to the dynamic behavior of safety valves. In the article, a list of possible factors that may contribute to the characteristics of a safety valve is presented. Among these factors, the influence of acoustic pressure waves in the inlet pipe is mentioned. The design standards by *ASME* only puts restrictions on the pressure drop over the inlet pipe but does not consider possible effects of pressure waves traveling through the system.

In the article *Relief device inlet piping: Beyond the* 3% *- rule* [11] the authors state that there are many safety relief valves with inlet line pressure drops which exceeds the 3%-rule and still do not chatter. Further on, the authors list possible reasons for chatter, where excessive pressure drop is mentioned. In accordance to the article written by Izuchi [7] the limit of 3% seems too conservative. As the primary reason for chatter, the pressure wave propagation is mentioned. The speed with which the pressure propagates through a media is significantly different if the fluid is a gas or a liquid. Dispersed bubbles may alter the speed of sound in a liquid and thus change how the pressure wave propagates (through a liquid). If the inlet line is long enough for the valve to react to the pressure decrease below it, it will start to close. A shorter inlet line results in a more stable operation. Since liquids do not expand to fill the void created as the valve opens they are much more sensitive to the pressure decrease.

The author of *Stability analysis of safety valve* [7] divides the instabilities of a safety relief valve into two categories, a dynamic and a static part. The dynamic instabilities are found to be caused by the interaction between the pressure wave propagation and the motion of the disc. The static instability is caused by a pressure drop over the inlet line. It is also found that the ratio between the valve opening area and outlet area, the orifice area, should be higher than 6.0 or otherwise the back pressure might induce instabilities. The article [7] concerns a gaseous system with pipes of $d = 2.54\,cm$ and it is found that chatter occurs for inlet lines of length $1-5\,m$. A pipe shorter than $1\,m$ does not chatter and if it is longer than $10\,m$ the pipe becomes stable. The pressure drop in the various pipes have also been measured. The pipe of $1\,m$, which do chatter, has a pressure drop of 2.6% while the pipe of $10\,m$, which does not chatter, has a pressure drop of 3.8%. Chatter is not caused by pressure drop over the inlet line since the longer inlet lines stabilizes the system and longer inlet lines means larger pressure drop. The author have conducted experiments as well as numerical simulations. There are some contradictions between these. The numerical simulation suggests that the inlet line of $10\,m$ would chatter, however it does not in the experiments. This is attributed to the mechanical forces acting on the disc when the velocity of it is small. The author concludes that the stability analysis can not explain why the system is stable for sufficiently long inlet lines. Having in mind that the article considers a gaseous system which have significantly less inertia than a liquid system and also that the pressure waves propagate with a velocity that is ≈ 4 times lower than in liquid, the lengths which result in stable behavior for a gaseous system may not be valid for a liquid system. Further the author concludes that the attenuation of the oscillating disc motion is due to frictional forces and stabilizes the valve and that the valve is stable if the inlet line is short enough since the pressure wave return before the valve has reacted to the pressure decrease.

In the article *Dynamic behaviors of check valves and safety valves with fluid interaction*[2], the author states that instabilities are due to the hydrodynamic conditions. The setting of the blow down ring and the back pressure and the geometry of the valve, are important factors to consider. In this article as well, the instabilities are divided into two different types. The static instabilities can be seen in the characteristics of the safety valve where the spring force is parallel to the fluid force. The lifts in this region are either too small or too large and reliefs the system too little or too much which results in an on-off behavior with a cycle matching the hydraulics of the system. Since a liquid can not sustain the pressure under the disc, these small openings result in pressure drops large enough to cause an incipient closing. The safety valve itself must not be over sized since the combination of the pressure and momentum forces resulting from the flow through the valve might be inadequate to keep the valve open. The other type of instabilities, the dynamic instabilities, are a result of the interaction between the movement of the disc which causes the flow to change and the pressure fluctuations induced by this. These instabilities are often of high frequency and primarily due to inertia and compressibility of the fluid.

The stability of a safety relief valve is increased with longer inlet pipes and decreased if the flow rate through the valve is low. Systems with low flow rate exhibit most oscillations and at those conditions, the losses in the pipe are small and can not be said to be the cause of the oscillations. The smaller the volume to be protected is, the more likely the valve is to chatter. Altering the blow down ring can adjust the flow through the valve. It has been found in experiments that pneumatic damping is not effective to stabilize a valve, dry friction and hydraulics is on the other hand effective.

Several design criteria against chatter exist, the authors of *Design of Spring Loaded Safety Valves with Inlet and Discharge Pipe against Chatter in the Case of Gas Flow* [3] states that no single criteria is valid under all conditions. They have come to the conclusion that a constant set pressure and a high back-pressure is more likely to cause the valve chatter than a constant back pressure and a lower set pressure. Systems with a discharge pipe are more prone to chatter and chatter is caused by a type of standing wave, the reflected wave which interfere with the incident wave results in a state where the media seem to vibrate.

If the hydrodynamic forces exerted on the valve are correctly foretold, the operation of a safety valve can be predicted. The authors of *Flowforce in a safety valve under incompressible, compressible, and two-phase flow conditions* [9] have performed a CFD-simulation and compared it to experimental data. The model is a 2-dimensional axisymmetric model and based on studies of the real flow behavior, the assumption seems valid. The outlet of the valve is assumed to not be greatly affected by the impinging flow on the valve body. Gaseous, liquid and multiphase flow have been studied and it has been concluded that the density of the media greatly affects the behavior of the valve. In the CFD-simulations, the expected occurrence of cavitation is not taken into account. It is found that there are regions below the disc where there are pressure sinks and possibly cavitating flow. The valve studied have a blow down ring and the effect of its position is observed to greatly influence the fluid force on the disc.

Summary of previous work

From the articles presented above it can be said that there are some main factors that are thought to significantly affect the stability of the valve. For the readers convenience they will be presented in a list below.

- The length of the inlet line, hence the time of pressure wave propagation.
- The hydrodynamic conditions of the valve are greatly affected by the geometry of the valve and the density of the fluid.
- The size of the system to be relived.
- The back pressure.
- Frictional damping.
- The recommended limitation of the pressure drop in the system of 3% seems to be too conservative, thus the pressure drop does not seem to be an issue.

3. Physics

In this chapter, the physics behind the parameters that are thought to affect the stability, is presented.

3.1 Pressure wave

Acoustic waves are the propagation of a perturbation, local compressions or dilations which move from one point to another by the elastic properties of a medium. The rate of this propagation is the speed of sound, and for the wave to propagate the material needs to be elastic [13]. An abrupt pressure change in i a liquid media is known as a water hammer.

The energy content in moving fluid can be divided into kinetic and potential energy. The kinetic energy correspond to the velocity at which the fluid is flowing and the potential energy corresponds to the static pressure of the fluid. According to the law of conservation of energy, changing either of the two forms will result in a change in the other, assuming constant temperature and negligible friction. According to Joukowsky [5], the pressure change is proportional to the velocity change by the relation

$$\Delta P = -\rho c \Delta v \tag{3.1}$$

where c is the sonic speed in the fluid.

The opening of a valve decreases the static pressure in the system but also increases the velocity through the opening. This increases the dynamic pressure in the pipe and creates an expansion wave moving backwards through the inlet pipe. The pressure gradient caused by the wave accelerates the fluid in the opposite direction of the pressure wave. The expansion wave from the opening will be reflected on all irregularities and obstacles in the system such as pipe bends, junctions and other components and will return to its origin (the valve). The time it takes for the wave to return depends on the fluid and the length of the pipe [5].

$$t_{wave} = \frac{L}{c} \tag{3.2}$$

The valve will remain open if the traveling time for the wave is shorter than the time it takes for the valve to react to the change in fluid force. Consequently, if the traveling time is larger than the time it takes for the valve to react to a pressure change, the disc will stop or reverse its upward motion. For the valve to stop opening, the total pressure, i.e. the sum of the static and dynamic pressures, must be decreased to the closing pressure. The closing of the valve will give rise to a compression wave in a corresponding way to the opening scenario. The reflected wave and the wave created by the incipient closing of the valve can superimpose and cause an oscillating pressure in the valve opening. Friction losses in the pipe also contributes to the oscillations, which in turn result in an oscillating motion of the disc. The behavior can thus be self sustained.

3.2 Inertia

The inertia of a rigid object can be explained through Newtons second law of motion.

$$F = ma \tag{3.3}$$

Consequently, objects with high mass will be accelerated less than objects with low mass when subjected to the same force.

In a system with a safety valve, the fluid volume in the piping can be considered a rigid body. The main forces acting on the fluid body are:

- the force induced by the pressure build-up in the system.
- the force induced by the pressure changes at the disc due to its movement.

For a given system pressure, the period of time required to accelerate the fluid to a certain speed increases with the mass of the fluid. The movement of the disc is strongly coupled with acceleration of the fluid body through the force balance on the disc, hence the movement of the disc affect the acceleration of the fluid body which in turn affects the movement of the disc.

For a compressible media, the fluid body cannot be treated as a rigid body and the acceleration phenomena is therefore not as straight forward. If the valve opens fast enough, a pressure wave can be created and propagated through the fluid. The part of the fluid subjected to the pressure wave is then accelerated according to $\frac{\Delta P}{\Delta v} = \rho c$. The part of the fluid that is not exposed to the pressure wave will not be accelerated. In an incompressible media, the whole fluid volume is accelerated momentarily since the pressure wave will propagate with infinite velocity through the volume. For a slow opening of the valve, no pressure wave will be generated and the pressure decrease will not follow $\frac{\Delta P}{\Delta v} = \rho c$.

3.3 Forces acting on the disc

The position of the disc is determined by the equation of motion. It comprises the forces acting on the disc, which are as follows:

$$F_{disc} = F_{pressure} + F_{gravity} + F_{friction} + F_{spring} + F_{viscous} \tag{3.4}$$

The forces are described below.

- Pressure forces acting on the disc and spindle can be divided into two parts, one dynamic and one static.
- Gravity force affecting the disc and spindle.
- Frictional force between spindle and housing and additional frictional elements to create a hysteresis effect.
- The tension in the spring.
- Viscous force between rigid bodies and fluid.

3.3.1 Pressure force

The dynamic force is due to the dynamic pressure of the fluid, the kinetic energy in the fluid. It is a function of the flow pattern and the velocity at which the fluid is flowing. It is defined as follows,

$$F_{dynamic} = P_{dynamic}A = \frac{\rho v^2}{2}A \tag{3.5}$$

where A is the effective area, which the pressure acts on.

The static force is due to the static pressure in the fluid (the potential energy in it). There is also a static back-pressure in the spring house which together with the spring counteracts the lifting forces from the flow and the static pressure in the system.

3.3.2 Gravity force

The gravity force acts on the moving parts of the valve, that is the disc and the spindle. It is defined by Newtons second law of motion.

$$F = -mg \tag{3.6}$$

3.3.3 Spring force

The spring force is the force exerted on the disc by the spring in the valve and acts in opposite direction to the fluid forces described in previous subsections. It can be described by *Hookes law*

$$F_{spring} = -(kx + f_0) \tag{3.7}$$

where k is the spring constant and x is the displacement from the equilibrium position. f_0 is the initial spring tension.

3.3.4 Additional forces

The additional forces acting on the valve are viscous, frictional and damping forces. A short description of these forces will be presented here. The viscous force arises due to friction between moving fluid elements and solid surfaces. The fluid particles closest to the solid surface will be decelerated creating a velocity gradient in the fluid. For a fluid in a pipe, the viscous force, τ can be expressed as

$$\tau = \mu \frac{\partial u}{\partial y} \tag{3.8}$$

where μ is the dynamic viscosity of the fluid and $\frac{\partial u}{\partial y}$ is the local shear velocity.

All mechanical systems have an intrinsic degree of friction which will dissipate energy. Additional damping can be introduced in valves to reduce oscillations by dissipating the kinetic energy in the oscillations (REF to Avtar). Additional friction/damping can be added to safety relief valves in form of barbs. They provide resistance in one direction and does not alter the force in the other.

3.4 Forces considered in this thesis

The pressure forces and the spring force are assumed to be the most important forces when determining the movement of the disc. The magnitude of the other forces presented above are assumed to be small enough for them to be neglected in comparison with the spring and the pressure forces. The frictional force will be considered in some simulations, where additional damping is introduced.

4. Valve characteristics

In this chapter, the concept of valve characteristics will be introduced.

The opening procedure of a pressure relief valve is a matter of force balance. The largest forces which the valve is subjected to are the spring force and the pressure force. They are of equal magnitude and opposite direction. The spring settings are often known and the spring force follow Hook's law and can thus be easily obtained. The initial tension of the spring determines the opening pressure of the valve. The force exerted on the valve by the fluid is dependent on the flow which is affected by the lift of the valve. By locking the valve to possible openings and varying the pressure, the force from the fluid on the disc can be obtained. The force on the disc can be graphically displayed against the valve lift and each pressure will be represented by an individual line in the plot. By addition of the spring curve to the same graph, the dominating force for a particular opening can be read. Since the dominating force for each lift of the valve can be obtained from this so called *Valve characteristics*, it is assumed to predict the opening procedure. The characteristics are obtained from stationary experiments/calculations/simulations with constant inlet pressure. An example of such characteristics is displayed in figure 4.1.

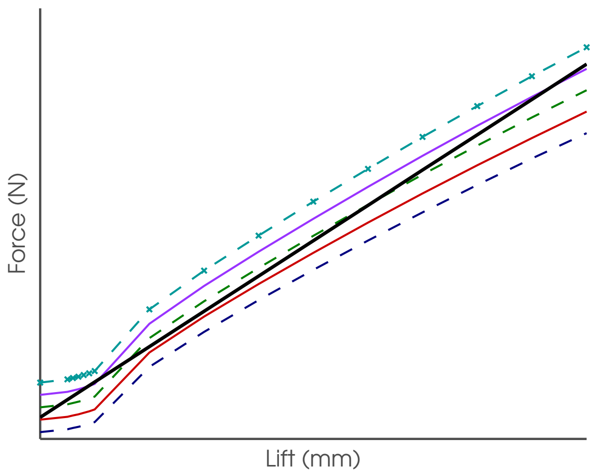


Figure 4.1: Example of valve characteristics. The spring is represented by the straight black, solid line. The remaining, colored lines each represent an individual isobar.

The valve will remain closed as long as the spring force is larger than the fluid force. This is represented in the characteristics when the spring curve lies above the isobars. As the pressure rises, the fluid force increases and when the fluid force exceeds the spring force, the valve will start to open. For each lift in the valve characteristics, there will exist an intersection between some isobar and the spring curve resulting in a force equilibrium on the disc.

By changing the spring settings, that is, the spring constant and the initial spring tension, the spring force on the valve is altered. Hence, the force balance is also modified. The initial spring tension determines the pressure at which the valve will start to open and the spring constant is the slope of the spring curve and consequently the rate at which the spring force increase with the valve lift. In addition to changing the spring constant and initial spring tension to change the behaviour of the valve, frictional elements can be added to the valve to create hysteresis between the opening- and closing curves of the valve. The effect of adding frictional elements can be visualized on the characteristics by a second spring curve. When the valve opens, it will follow the original spring but when closing, it will follow the new spring. This means that in order to close, the pressure must fall to an isobar that lies below the second spring. That is, the spring force must overcome both the fluid pressure force and the frictional force (added by the frictional elements) for the valve to close. Figure 4.2 displays the same characteristics as figure 4.1 with the addition of a second, imaginary spring.

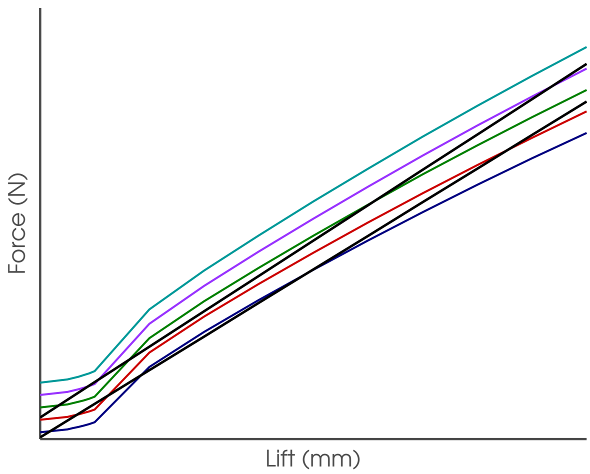
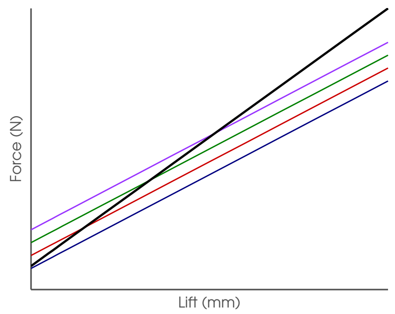


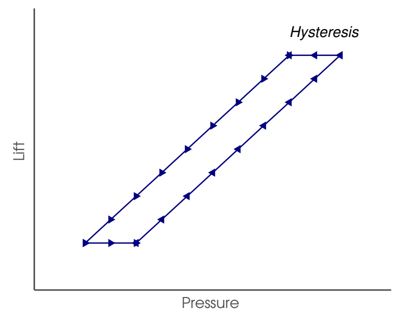
Figure 4.2: Example of valve characteristics with hysteresis

To illustrate the effect of the spring settings, two different cases will be presented. The cases presented describe ideal behaviors for theoretical valves and are only used to illustrate how the characteristics could be analyzed and how different spring settings might yield different valve behaviors. Throughout this section, it is assumed that the valve characteristics predict the transient behavior of the valve well.

First, the spring constant is set high, resulting in a steep spring curve, see figure 4.3a. The spring curve has a much steeper incline than the isobars, resulting in that the spring curve only intersects a particular isobar once. For this spring, the pressure has to continuously increase for the valve to open and the valve will open *proportional* to the pressure increase. This would be a so called *proportionally* opening valve. When the pressure in the system decreases, the valve will close in the same manner, proportional to the pressure decrease. The opening and the closing of the valve will follow the outline displayed in figure 4.3b[1].

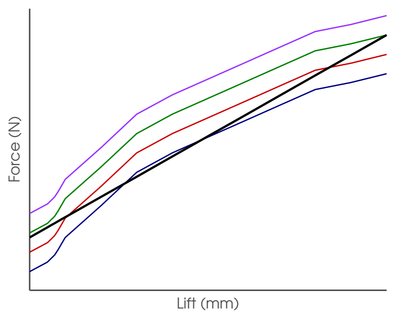


(a) Characteristics with a high spring constant.

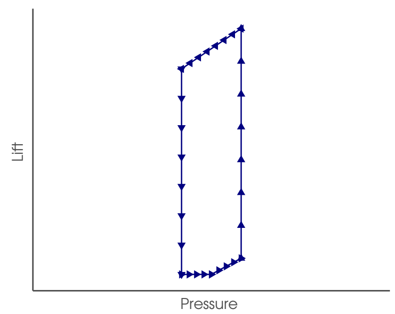


(b) Corresponding opening and closing behaviour.

In the second case, the spring constant is smaller, creating a less steep spring curve. The spring has an incline which is smaller than the incline of the isobars. This type of spring can intersect the same isobar twice. That is, the valve has two lifts for the same pressure. this yields a so called *popping* or *high-lifting* opening behavior of the valve. A schematic illustration of this can be seen in figure 4.4a.



(a) Characteristics with a low spring constant.

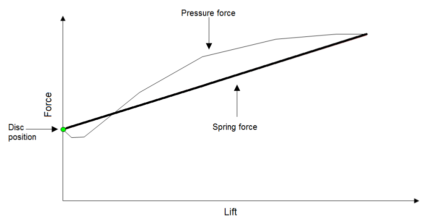


(b) Corresponding opening and closing behaviour.

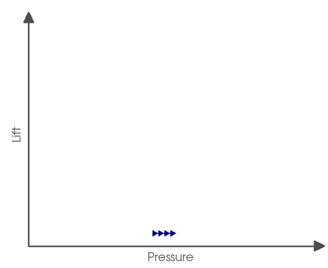
[1]Hysteresis has been added to the opening- and closing characteristics of the proportional valve in order to separate the curves and make the figure easier to analyze. If no additional hysteresis is added, the opening- and closing lines will completely overlap.

The spring curve intersects the isobars at two positions. When the valve reaches the first of those positions, the disc will pop from that lift directly to the lift of the other position. This region is called a *high-lifting* region. For the valve to reseat again, the pressure must decrease low enough to reach an isobar that lies completely below the spring curve. When the reseating isobar is reached, the valve will pop down to its closed position as fast as it popped upwards. This yields an opening/closing curve as displayed in figure 4.4b.

A more detailed description of how the valve opening characteristics should be interpreted is given in the example below.



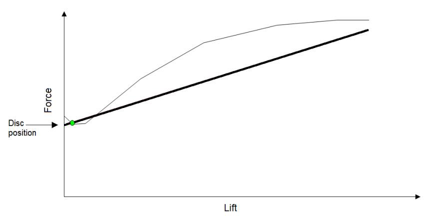
(a) Schematic force-lift diagram, the valve is fully closed.



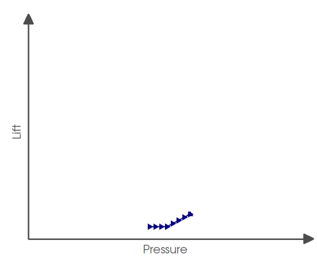
(b) Schematic lift-pressure diagram, the valve is fully closed.

Figure 4.5: Schematic diagrams of opening behavior.

In figure 4.5a, the valve is fully closed, the disc position is given by the green marker. The pressure force curve represents the force generated by a constant pressure for different disc positions. The spring force represents the force generated by the spring for different lifts. Figure 4.5b displays how the valve is completely closed even though the pressure is increasing, this is due to that the spring force is higher than the pressure force.



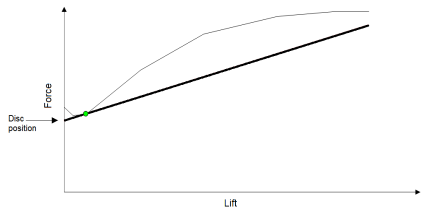
(a) Schematic force-lift diagram, the valve begins to open.



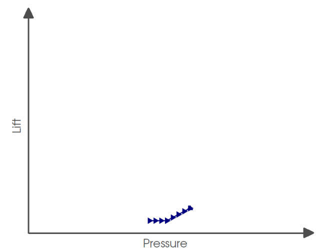
(b) Schematic lift-pressure diagram, the valve begins to open.

Figure 4.6: Schematic diagrams of opening behavior continued.

Figure 4.6a shows how the valve has started to open since the increasing pressure now results in a force that overcomes the spring force. The corresponding opening characteristics can be seen in figure 4.6b.



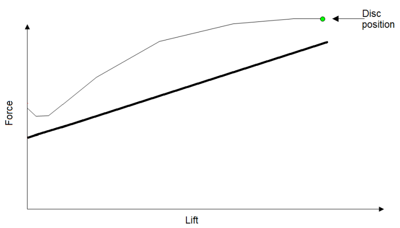
(a) Schematic force-lift diagram, the valve continues to open.



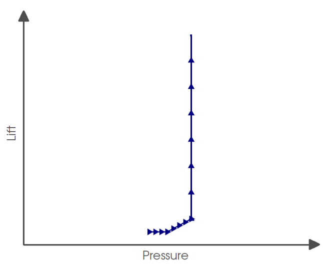
(b) Schematic lift-pressure diagram, the valve continues to open.

Figure 4.7: Schematic diagrams of opening behavior continued.

In figure 4.7a, the pressure is high enough to generate a pressure force higher than the spring force. The whole isobar lies above the spring force curve and the disc will therefore follow the pressure force curve up to the maximum lift, displayed in figure 4.8a. The corresponding opening behavior is diplayed in figure 4.8b where the highlifting zone of the valve is represented by the vertical part.



(a) Schematic force-lift diagram, the valve is fully open.



(b) Schematic lift-pressure diagram, the valve is fully open.

Figure 4.8: Schematic diagrams of opening behavior continued.

5. Hypotheses about valve characteristics and stability

In this chapter, some hypotheses regarding the valve characteristics and their application will be presented first. This will be followed by a section where hypotheses about safety relief valve stability and their operation in different systems will be discussed.

5.1 Valve characteristics

From the reasoning presented in the previous section, the spring settings may significantly change the behavior of the valve. By choosing different values of the spring constant, the opening characteristics can be set to, for example, popping or proportional. But is it possible to achieve all these different opening behaviors for any system? Will the force exerted on the disc by the pressure in the system be the same regardless of the conditions upstreams the pressure relief valve? Presumably, different systems will yield different characteristics and consequently different behaviors. As presented in chapter 2, safety valves are often categorized as either popping, proportional or high-lifting, regardless of the properties of the system to which the valve is attached. It can be argued that a combination of the spring- and the system properties will have to be taken into account prior to the classification of the valve.

Theoretically, the spring can be altered to give a certain opening behavior. By changing the initial spring tension the opening pressure of the valve can be changed. It should be possible to create a popping valve for any system, with a spring constant low enough. However, a soft spring will cause the valve to open at a lower pressure than a stiff spring. Additionally, as seen in figure 4.4b, a popping or high-lifting valve experience hysteresis between the opening and the closing pressure, and in order for the valve to close, the pressure has to decrease substantially. When aiming for a popping opening behaviour, the reclosing pressure must be taken into account to ensure that the whole system will not be completely dried out before the valve recloses. By having a stiff enough spring, a safety valve can be made more proportional. In contrast to a softer spring, this will yield a higher opening pressure for the valve. It must be made sure that the new opening pressure is not higher than the systems design pressure, otherwise, the system might be damaged or destroyed prior to the valve opening.

Provided that the valve characteristics can be used to predict the transient response of a safety valve, it represents a fairly easy method of studying safety valves. The characteristics can be generated through steady state simulations which is a much faster method than transient simulations and as the examples above have illustrated, the results are easy to analyze. On the basis of these facts, it is therefore of interest to study if valve characteristics yields a good enough prediction of the transient response of a safety valve.

5.2 Stability

According to the authors and studies mentioned in *Previous Work* there exist a number of different factors that might influence the stability of the safety valve. Wave propagation through the inlet pipe, the length of the inlet pipe and the orifice area ratio of the valve are suggested as possible sources to unstable valve behavior The validity of the 3% - rule regarding the static pressure drop in the inlet pipe to the valve issued by *ASME* is discussed and it is said that it might be too conservative. One objective of this thesis is to investigate what effect these factors have on the stability of a safety valve and if the these instabilities are related to characteristics of the valve.
When studying the characteristics of a safety relief valve, three principal regions can be noted. In figure 5.1 the regions are to be seen.

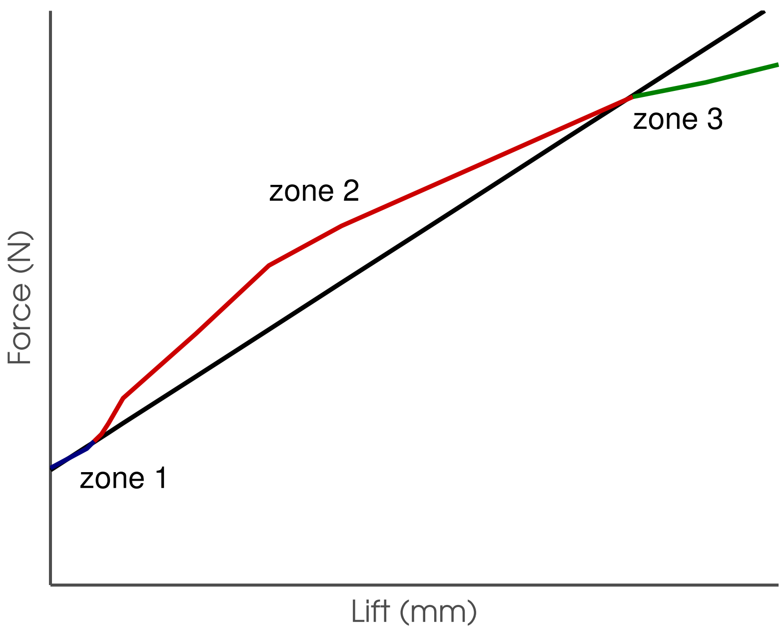


Figure 5.1: Schematic isobar and spring.

- zone 1 - This is an example of an indeterminate region, where the valve can not be classified as either popping or proportional.
- zone 2 - This is an example of a popping region, where the spring curve lies completely below the opening isobar, up to the second point of intersection.
- zone 3 - The region after the second point of intersection, resulting from the fact that the isobars deflect due to pressure drop, is proportional. It is claimed to be unstable.

The different zones are associated with different types of problems. It is of interest to examine if this is valid for all types of system. The possible factors for causing instable behavior for the zone previously mentioned will be described below.

Zone 1
Neither of the forces acting on the valve is dominating and a small perturbation yields a large deviation. Classical stability theory proposes that this region would lead to instabilities. The position of the disc indeterminable by the force balance. Having in mind that the length of the inlet pipe impacts the inertia of the system and the inertia of the system describes its responsiveness to changes in force and therefore in pressure, it is reasonable to believe that a longer inlet pipe (high inertia) might stabilize the valve behavior.

Zone 2
The popping region, where the valve can be said to have two positions (lifts) for the same pressure, should result in a rapid decrease of pressure in the system since the opening can be said to be uncontrolled. This is often assumed to be the most unstable region for incompressible media since it can not withhold the pressure. The pressure decrease leads to a reclosure, and could also result in chattering. If the pop is rapid enough to cause a pressure change large enough to yield (de)compressive waves, a water hammer is formed, this is assumed to lead to enhancement of oscillations of the disc. The time of the pressure wave propagation t_{wave}, i.e the time it takes for a pressure wave to travel from the valve and be reflected can be expressed as $t_{wave} = \frac{2L}{c}$. It is suggested that if t_{wave} is shorter than the time it takes for the disc to react to the pressure change beneath it, the disc will not oscillate and the valve will operate in a stable manner. For a short enough pipe, the pressure wave might bounce back to the valve before the disc has started to react to the pressure drop created by the opening of the valve. Increasing the length of the inlet pipe will eventually result in that the pressure wave does not return to the valve in time to prevent the disc from closing. But, increasing the length of the inlet line also increases the inertia of the system and it is reasonable to believe that eventually, the inertia of the system will be large enough to stabilize the system and prevent the disc from oscillating even though the pressure wave returns too late.

Zone 3
The region after the second point of intersection between the isobar and the spring is claimed to be unstable and should be avoided by limiting the lift of the valve to this value. Further, the pressure drop should be limited according to the 3% - rule, a pressure drop exceeding this limit might cause the valve to operate in an unstable manner. A longer pipe will result in a higher pressure drop and the isobars will deflect more, the longer the pipe is.

Hypothesis
The reasoning above leads to the assumption that there would exist certain lengths of the inlet pipe for which the valve would operate stable. If it is not possible to change the length of the inlet pipe to the valve, and it operates in an unstable way, other ways of stabilizing thee disc must be examined. A pressure wave traveling through the system creates fluctuating forces on the disc, disturbing the force balance and making it oscillate. The oscillations of the disc creates more instabilities in the system which in turn makes the valve even more unstable. It seems reasonable that by making it harder for the valve to close, the whole system might be stabilized. The same reasoning would be valid for the opening of the valve but since a safety valve is designed and implemented in a system in order to relieve the pressure should it become to high, the idea of keeping it from opening seems to counteract its purpose of it. Also, keeping the valve from closing should increase the capacity of the valve compared to when it is oscillating. Mounting frictional elements on the valve spindle should shift the closing of the valve, requiring the spring force not only to exceed the fluid forces acting on the disc, but also the extra force that is generated from the frictional elements. Frictional elements are therefore likely to act stabilizing on the valve behavior.

For the second system 2.4b, the pressure increase at the valve is due to the mass flow through the control valve as explained in chapter 2. As long as the mass flow into the system is larger than the mass flow through the valve, the pressure in the system will increase. If the valve has a high-lifting behavior *zone 2*, there exist a point where the mass flow out exceeds the inflow through the control valve. When that point is reached, the volume of media between the control valve and the safety valve will be transported out of the system. When this happens, the pressure in the system deceases, causing the valve to close. Once the valve is closed, the inflow to the system is again, larger than the outflow and the pressure starts to build up again. Since the pressure relief only occurs when the outflow through the safety valve is larger than the inflow through the control valve, a larger inflow of mass requires a higher lift in order of the pressure to be relieved. Regardless of the magnitude of the mass flow, the previous reasoning regarding the existence of a point at which the valve would reclose due to a sufficiently large outflow is assumed to be valid. Thus, for a large inflow, that point is shifted to a higher lift which means that the oscillations of the disc will have a larger amplitude than for a small inflow of mass. When the inflow to the system is high enough, the valve must reach its maximum lift in order to relieve the system and the disc will then become stable again. From this reasoning, it can be suggested that there exist certain mass flow for which the valve will behave stable. Correspondingly, there will exist an interval of mass flows that result in an unstable behavior of the valve.

If this suggestion proves valid, it is possible to predict an unstable behavior by studying the mass flows that the valve is likely to be submitted to. In figure 5.2 a schematic valve characteristic with examples of schematic iso-lines for mass flows are displayed. The force acting on the disc will be dependent on the mass flow in to the valve. The shape of the iso-lines for the mass flow are exponential.

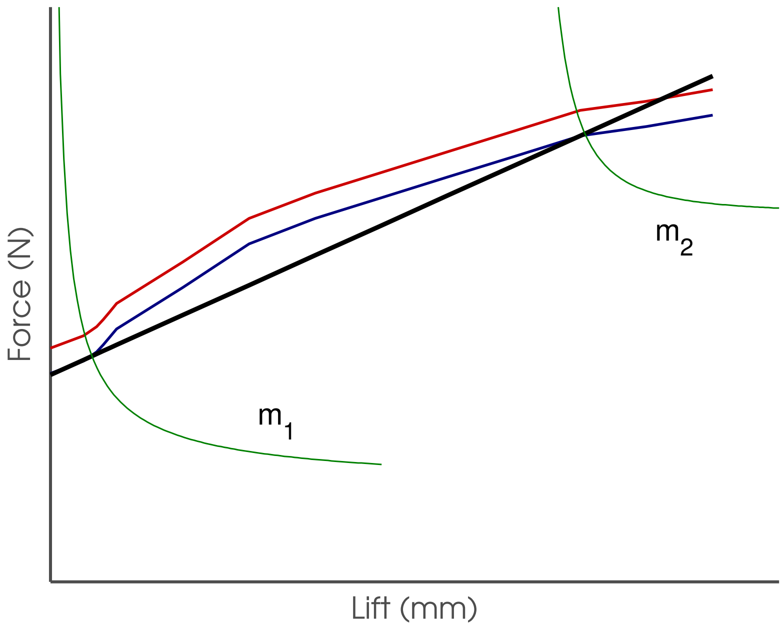


Figure 5.2: Valve characteristics with schematic mass flow iso-lines.

A valve with characteristics as displayed in figure 5.2 would, based on the presvious reasoning, exhibit a stable operation for mass flows less than m_1 $^{kg}/_{s}$ and larger than m_2 $^{kg}/_{s}$. For mass flows between m_1 $^{kg}/_{s}$ and m_2 $^{kg}/_{s}$, an unstable behavior could be expected.

It seems reasonable to believe that the pressure relief process will be faster if the volume of fluid between the control valve and the safety valve is small. This means that for a small volume, potential oscillations are likely to have a high frequency compared to if the volume is large. A small volume is therefore more prone to yield damaging, high-frequency oscillations. The magnitude of the fluid volume between the control valve and the safety valve is thought to be of different importance for liquid- and vapor systems. Liquid media is more incompressible than a gaseous one and cannot fill the void produced by the valve opening as well as a gaseous media. In order to stabilize a liquid system, the volume must therefore be larger than for a vapor. Presumably, a valve with a behavior of the type that can be found in *zone 2* would behave differently if subjected to a to a source of constant mass inflow or a source of constant pressure. This is important to consider since as described in 2.5 valves are often tested for upstream conditions of the first type but are more likely to be subjected to upstream conditions of the second type in real pressure protection scenarios.

The hypotheses presented in this section will be investigated through numerical simulations of a safety valve.

6. Numerical model

In this thesis, the stability of a safety valve will be studied using *Computational Fluid Dynamics (CFD)*. This chapter will present the mesh and geometry used and the simulations that have been executed to examine the behavior of the valve. First, a brief explanation of the governing equations that are solved in *CFD* - simulations will be given.

6.1 Governing equations

In computational fluid dynamics there are four equations to describe the flow field, the momentum equation (6.2), the continuity equation (6.3) the energy equation and the equation for species. These equations are coupled and are given below,

A general transport equation for a scalar, vector or tensor can be written [1], in tensor notation

$$\frac{\partial \phi}{\partial t} + U_i \frac{\partial \phi}{\partial x_i} = D \frac{\partial^2 \phi}{\partial x_i \partial x_i} + S(\phi) \tag{6.1}$$

The transport equation for momentum is,

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial p}{\partial x_i} \tag{6.2}$$

The continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0 \tag{6.3}$$

There are several different forms of energy and the balance equation for total energy is

$$\frac{\partial h}{\partial t} = -\frac{\partial}{\partial x_j} \left[h U_i - k \frac{\partial T}{\partial x_j} + \sum_n m_n h_n j_n - \tau_{kj} U_k \right] + S_h \tag{6.4}$$

A general transport equation for a species can be written according to equation (6.1). In this thesis the mass transfer will be due to cavition and the equation describing that phenomena will be explained in A.

In addition to solving the momentum equation and the continuity equation, equations describing the turbulence are to be solved. The turbulence model mainly used in this thesis is the *SST* $k - \omega$- *Shear stress transport*. It is a so called two-equation model, with two additional transport equations for turbulent kinetic energy, k and specific dissipation, ω. The equations are derived from Reynolds average decomposed Navier stokes equation and is based on the Boussinesque approximation, that the Reynold stresses arising from averaging can be modeled with a turbulent viscosity.

In the regions near the walls, the flow is laminar and the Reynolds number is low, this region can not be well represented by equations describing the turbulent flow far from the walls. The $SST - k - \omega$ model handles the transition between the laminar flow in the viscous subregion and the fully turbulent flow with a blending function.

The transport equation for turbulent kinetic energy in the $k - \omega - SST$ model is,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + \tilde{G_k} - Y_k + S_k \tag{6.5}$$

- $\tilde{G_k}$ - generation of turblent kinetic energy from mean velocity gradients
- Y_k - dissipation of turbulent kinetic energy
- Γ_k - Effective diffusivity of turbulent kinetic energy
- S_k - source term

The transport equation for the specific dissipation ω is,

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{6.6}$$

- G_ω - generation of specific dissipation.
- Y_ω - dissipation of specific dissipation.
- Γ_ω - Effective diffusivity of specific dissipation.
- S_ω - source term for specific dissipation.

6.2 Mesh and geometry

For the simulations in this thesis, a 2D - axisymmetric model have been used. The geometry was created in *Ansys Design Modeler*.

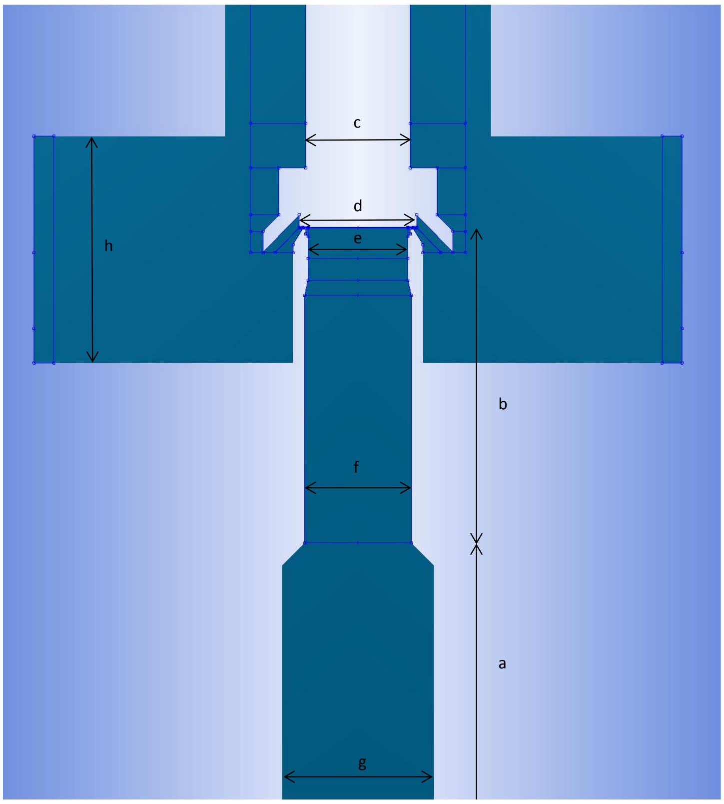


Figure 6.1: The geometry

The most important dimensions for visualizing the valve are presented in table 6.1.

Table 6.1: Dimensions of the geometry

Notation	Dimension (mm)
a	125
b	159.64
c	54.1
d	60.5
e	51.8
f	55
g	78
h	115.01

Dimensions a, b and g were varied to create inlet pipes of different length and diameter.

The remaining dimensions, c through f and h, were kept constant for all simulations. For each simulation, a will be stated, b and g will be given for the simulations for which they vary from the values given in table 6.1.

A structured mesh was generated using *Ansys Meshing*. The mesh is divided into different subregions with different cell sizes in order to minimize the number of cells and still achieve sufficiently small cells in for example the narrowest path between the disc and the seat.

The different regions in the mesh are illustrated in figure 6.2.

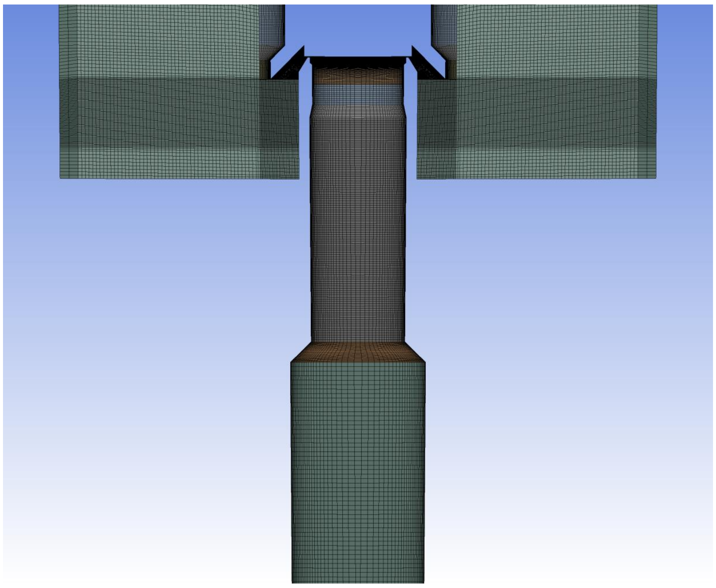


Figure 6.2: Mesh

6.3 Dynamic mesh and UDF

6.3.1 Dynamic mesh

A dynamic meshing method in *FLUENT* called *layering* have been used for the all transient simulations. Layering is a method for adding and removing cells during transient simulations. A structured mesh is required. An ideal cell height in the deforming region is specified, then a collapsing and a split factor is chosen. The split factor determines the size at which the cells will be split into a new layer according to the following equation

$$h_{min} > (1 + \alpha_{split}) h_{ideal} \tag{6.7}$$

The collapsing factor, on the otter hand, determines the height at which the cells will merge together to remove a cell layer.

$$h_{min} < \alpha_{collaps} h_{ideal} \tag{6.8}$$

The splitting and merging of cells can also be performed based on cell height ratios instead of cell heights [6].

6.3.2 UDF

The *UDF* contains routines for saving data, moving the disc and controlling the inlet pressure. The force in each cell adjacent to the disc is calculated by multiplying the pressure in each cell with the surface area of that cell on the disc boundary. The total force on the disc is the achieved by adding all these forces together. The spring force is calculated using the spring constant and the initial spring tension as described in section 3. The sum of the spring force and the total force is the net force acting on the disc which determines its direction of movement.

To make the movement of the disc more stable, that is to prevent it from fluttering and to achieve a reseating pressure that is lower than the opening pressure, hysteresis can be added to the valve. When the velocity of the valve is negative, that is when the valve is attempting to close, the difference between the spring force and the total pressure force must exceed the value of the hysteresis constant specified by the user.

6.4 Simulations

This section will describe the different simulations that have been made during this thesis. For the transient simulations, some additional methods needed in *FLUENT* will also be presented. Detailed settings used for the simulations can be found in appendix C. The simulations will be divided into two parts corresponding to the two cases presented in section 5.2

In transient simulations, the disc is not fixed to different lifts. The movement of the disc is determined by the forces described in section 3 and carried out through a *User Defined Function (UDF)*. To preserve the mesh quality from the steady state simulations, the mesh must be deformable during the transient simulations. A brief explanation of the dynamic mesh method and the *UDF* will be given later in this section. Last in this section, the method for attaining the formula for the pressure drop used in the *discrete pressure drop* - simulations will be explained.

6.4.1 Constant pressure

The objective of these simulations is to investigate the usefulness of valve characteristics to predict the transient response of the safety valve and study the effects of frictional hysteresis, length of inlet pipe and pressure drop. A pressure-inlet boundary condition has been applied to all simulations of case 1.

6.4.1.1 Steady state

- $0.125\,m$ ($a = 0.125\,m$) inlet pipe
- $20\,m$ ($a = 20\,m$) inlet pipe

6.4.1.2 Transient

- Inlet line of $0.125\,m$ and $20\,m$ to compare with the steady state valve characteristics.
- Inlet line of $2\,m$ ($a = 2\,m$) with and without frictional hysteresis.
- Inlet line of $2\,m$ without hysteresis and with modified spring settings. The spring constant was increased and decreased by 30%
- Inlet lines of $0.125\,m$, $0.5\,m$ ($a = 0.5\,m$) , $2\,m$, $5\,m$ ($a = 5\,m$), $10\,m$ ($a = 10\,m$), $15\,m$ ($a = 15\,m$) and $20\,m$ without frictional hysteresis to study the impact of the pipe lenght.
- Inlet lines of $0.125\,m$, $0.5\,m$, $2\,m$, $5\,m$, $10\,m$, $15\,m$ and $20\,m$ with frictional hysteresis to study the impact of the systems inertia.
- Inlet line of $0.025\,m$ ($a = 0.025\,m$) and a discrete pressure drop identical to the real pressure drop for an inlet line of $20\,m$
- Inlet line of $0.025\,m$ ($a = 0.025\,m$) and a discrete pressure drop of 10%, 25% and 50%.

For all transient simulations of case 1, the pressure ramp has followed equation (6.9).

$$P = 33 + 1.875t\,bar(g) \tag{6.9}$$

6.4.2 Constant mass flow

The simulations for this case aim at investigating the effect of different mass flows through the system and of the magnitude of the volume between the control valve and the safety valve. For these simulations, a velocity-inlet boundary condition has been applied at the system inlet.

6.4.2.1 Transient

- Inlet line of $5\,m$ ($a = 5\,m$) and a diameter of $0.1\,m$ ($g = 0.1\,m$), case I.
- Inlet line of $5\,m$ ($a = 5\,m$) and a diameter of $0.4\,m$ ($g = 0.4\,m$), case II.
- Inlet line of $5\,m$ ($a = 5\,m$) and a diameter of $0.4\,m$ ($g = 2\,m$), case III.
- Inlet line of $5\,m$ ($a = 5\,m$), diameter of $0.4\,m$ ($g = 0.4\,m$) and intermediate pipe length of $20\,m$ ($b = 20\,m$), case IV.

The velocity ramp for the simulations of case 2 was determined in such a way that the mass flow through the valve was equal in time. Equation (6.10) defines the expression used and the values for each simulation case are defined in table 6.2.

$$v = v_{start} + v_{inc} t \ \mathrm{m/s} \tag{6.10}$$

Table 6.2: Velocity ramp factors.

Case	v_{start} (m/s)	v_{inc} (m/s)
I	0.666	10.000
II	0.041	0.608
III	0.002	0.025
IV	0.034	0.608

6.4.2.2 Pressure drop

Steady state simulations with a $20\,m$ long inlet pipe were peformed in order to measure the pressure $0.15054\,m$ from the disc ($a = 0.025\,m$). Knowing the inlet pressure, the pressure drop can then be determined. Two different inlet pressures $31\,bar(g)$ and $32\,bar(g)$, each with discrete lifts ranging from $0\,mm$ to $10\,mm$ where simulated.
It is assumed that the head loss in the pipe can be predicted by equation (6.11), which shows how the loss due to friction is dependent on the average velocity of the fluid flow.

$$P_1 - P_2 = Cv^{\alpha} \tag{6.11}$$

P_1 refers to the inlet pressure, P_2 to the pressure further downstreams (at $0.275\,m$ from the disc) and v is the average velocity.

The coefficients C and α can be found by making a polynomial fitting of the data. The pressure loss in the $20\,m$ long pipe can then be represented and replicated with a geometry where the inlet is located $0.15054\,m$ from the disc. This is done by applying equation 6.11 to the inlet pressure in this new geometry. P_1 is the pressure used in the simulations with the longer pipe and the inlet pressure for the shorter pipe will be P_2.

The graphical representation of the polynomial fitting of the pressure loss equation can be seen in figure (6.3).

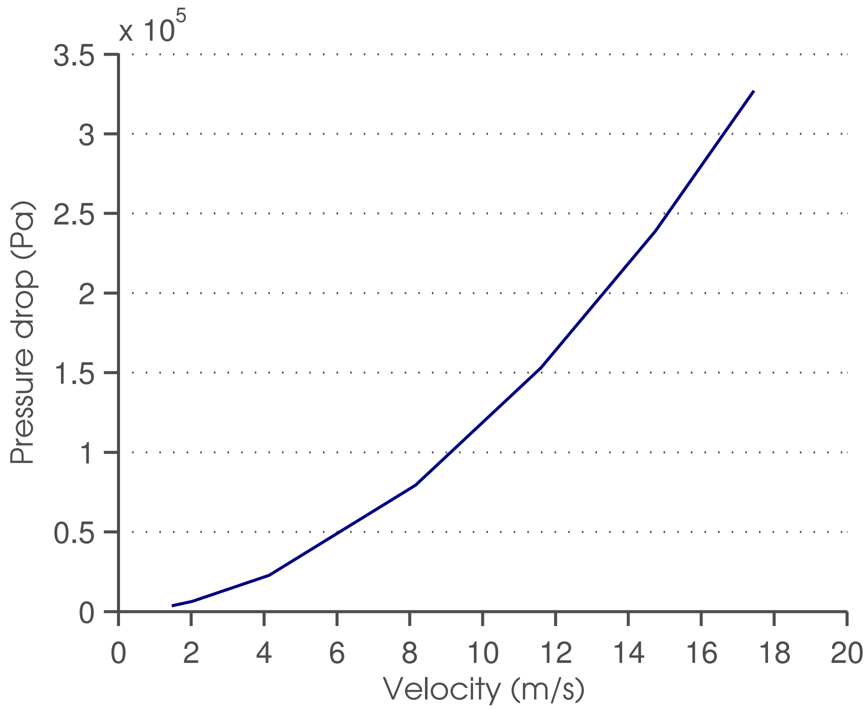


Figure 6.3: Graphical representation of the polynomial fitting of the pressure loss $[Pa]$ in the $20\,m$ long pipe for different lifts $[mm]$.

For the curve presented in figure 6.3, $C = 1785.4$ and $\alpha = 1.8163$ which is close to the theoretical value $\alpha = 2$, that can be expected for pipe flow. These values have been used to replicate the pressure drop for the $20\,m$ - pipe. For the simulations investigating the effect of pressure drops of 10%, 25% and 50% magnitude, the constant C in equation (6.3) was varied while α was set constant. To calculate values of C, the following equation was used.

$$C = \frac{X \cdot 1.1 P_{open}}{v^{\alpha}} \tag{6.12}$$

X was set to 0.1, 0.25 and 0.5 to produce a C for each pressure drop.

6.5 Sensitivity analysis

6.5.1 Mesh independence

In order to investigate if the solutions were mesh independent, the base mesh was refined in the regions closest to the disc. The refined mesh consisted of $\tilde{4}5000$ cells compared to the original mesh with $\tilde{3}3000$ cells at an initial lift of $0.5\,mm$. Steady state simulations were run for four different lifts, $0.5\,mm$,$4\,mm$, $6\,mm$ and $8\,mm$ and two different pressures, $31\,bar(g)$ and $32\,bar(g)$, and the fluid force on the disc was studied.

The mesh-test simulations showed that the force on the disc was very similar for the original mesh and the refined mesh. The maximum deviation was seen at a lift of $8\,mm$ and a pressure of $32\,bar(g)$ where the refined mesh generated a force 0.75% larger than the original mesh.The difference in force between the original and the refined mesh was small enough to be negligible, that is, the original mesh yielded a mesh independent solution.

6.5.2 Turbulence model

To validate the choice of turbulence model, steady state simulations with k-ϵ- model and *non-equilibrium wall function/enhanced wall function* were carried out for two pressures, $31\,bar(g)$ and $32\,bar(g)$ and lifts $0.5\,mm$,$6\,mm$ and $8\,mm$ and the resulting force on the disc was compared to the force generated whit SST/k-ω.

The results show that the k-ϵ-model predicted a higher force than SST/k-ω for all cases except for $8\,mm$ lift. The maximum deviation was reached for a lift of $4\,mm$ where k-ϵ predicted/yielded a disc force 0.4% larger than SST/k-ω. At $8\,mm$, SST/k-ω predicted a force 0.5% higher than k-ϵ.

The deviation between the two turbulence models was very small and therefore, either of them would be an appropriate choice. However, the SST-kω proved to yield converged solutions more easily and was therefore chosen for all simulations throughout this thesis.

It is important to have in mind that the geometry upstreams the valve is rather simple and the flow will mainly be along the axial direction of the pipe. A different pipe configuration, with for example a bend would result in a different flow pattern with more vorticity. Not completely resolving the time dependent turbulence would then lead to a misprediction of the fluid force acting on the disc. However, with the prevailing geometry, it is most likely the mechanical energy forms, such as the pressure, which determines the amount of energy in the fluid as it interacts with the disc.

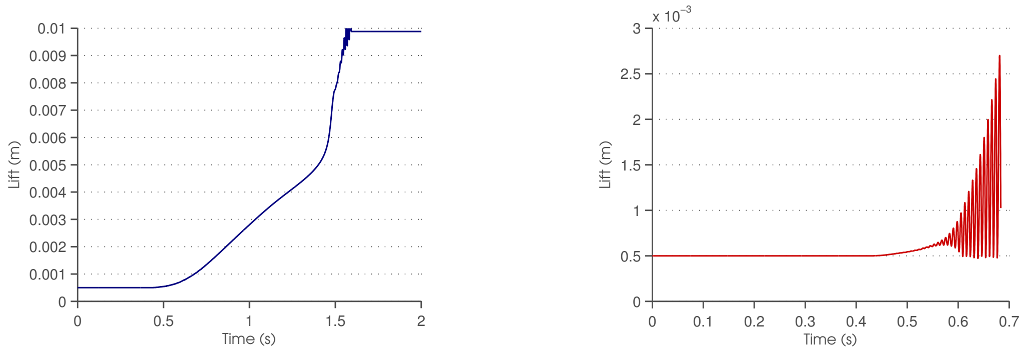
6.5.3 Courant number and time-step

The *Courant number (CFL)* is an important parameter in transient CFD simulations. It relates the chosen time step to the time it takes for the flow to pass through a cell and is defined as in equation (6.13).

$$CFL = \frac{u\Delta t}{\Delta x} \tag{6.13}$$

u is the velocity for propagation of information (for example velocity of the flow velocity of pressure propagation), Δt is the time step and Δx is the length of a cell in the direction of the flow. $CFL > 1$ means that the time it takes for the flow to pass through a cell is smaller than the time step at which the solver advances (for an explicit solver). This means that some cells might be entirely skipped by the solver and the resulting values will therefore deviate from the true ones. By changing the cell size or the time step, the CFL - number can be changed to meet the requirements of the solver.

To investigare the influence of the CFL - number, two simulations with a $5\,m$ - inlet pipe were run, one with $\Delta t = 10^{-3}s$ and the other with $\Delta t = 5 \cdot 10^{-5}s$. The results from the simulations are displayed in figure 6.4.



(a) $\Delta t = 1e^{-3}s$ (b) $\Delta t = 5e^{-5}s$

Figure 6.4: 5m inlet pipe with different time steps.

The figure shows that in the simulation with $\Delta t = 10^{-3}s$, the valve opened smoothly from its starting position to its maximum lift. For the smaller time step, the disc oscillated and the simulations crashed after only $0.7\,s$. Using a cell size representative of the cells in the inlet line and the speed of sound, $\Delta t = 10^{-3}s$ yields $CFL \approx 33$ while $\Delta t = 5 \cdot 10^{-5}s$ yields $CFL \approx 1$. The figures show that it is possible to force the valve to a smooth opening by using a large time step, allowing for the solver to skip important data. It is obvious that the choice of time step is of paramount importance to the validity and reliability of the simulation results.

From these experiments, the importance of the CFL - number and the time step size is illustrated.

The Courant number presented above are calculated using a cell size representative of the pipe. However, the cells closest to the disc are significantly smaller, yielding a much higher CFL. This results in that the velocity at which pressure wave propagates in the system is overestimated. The time step used in the simulations is not sufficient to resolve the pressure wave propagation.

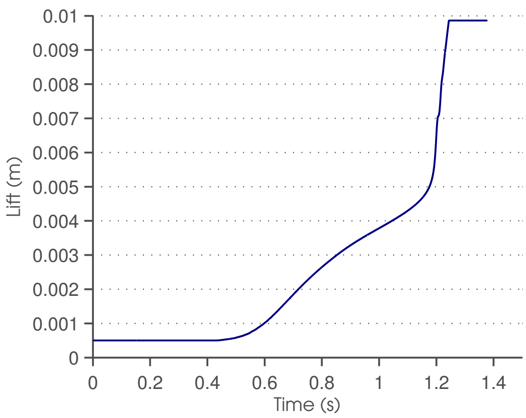
6.5.4 Compressibility

Liquids are often considered incompressible with a constant density. Modeling a liquid as incompressible results in an infinite speed of sound in the liquid, according to equation (6.14),

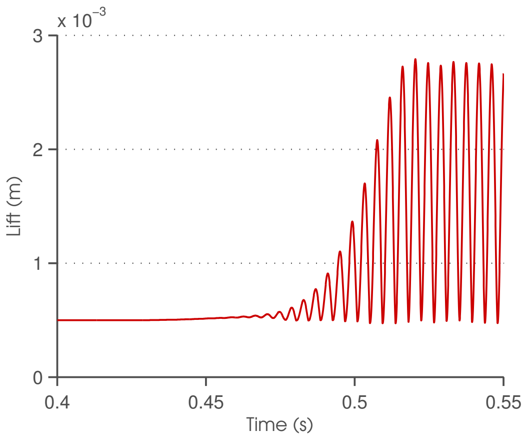
$$c = \sqrt{\frac{1}{\chi \rho}} \tag{6.14}$$

where χ is the compressibility factor defined as $\chi = \frac{1}{\rho}\frac{\partial \rho}{\partial P}$.

If an acoustic wave propagates through the safety relief valve system infinitely fast, the disc will not have time to react to the pressure decrease below it, since the pressure is restored infinitely fast. By comparing the opening behavior for the valve, attached to an inlet line of $2\,m$ without hysteresis, where the fluid is modeled as compressible and incompressible, it can be seen that with the simplification of incompressibility a lot of information is lost, see figure 6.5. It is noted that the valve behaves completely stable when the fluid is modeled as incompressible, which is not the case when the liquid is considered compressible and the valve chatters.



(a) Incompressible



(b) Compressible

Figure 6.5: Opening behavior of safety relief valve attached to an inlet line of $2\,m$. Comparison of compressible and incompressible fluid.

7. Numerical results

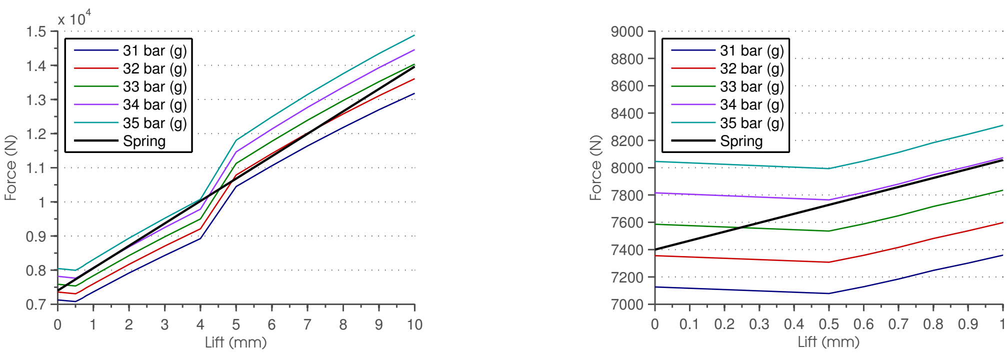
In this chapter, the results from the numerical simulations will be presented. The results will be divided in accordance with the two cases presented in section 2.5 and the different effects corresponding to each case.

7.1 Constant pressure

In this section the results from the simulations with constant pressure boundary condition will be presented.

7.1.1 Characteristics and opening behavior

The steady state characteristics for the $0.125\,m$, $2\,m$ and the $20\,m$ pipe configurations are displayed in figures 7.1, 7.2 and 7.3 respectively. The same spring settings have been used for all three cases. The initial position of the valve is $0.5\,mm$, the force at $0\,mm$ is calculated using $F = PA_{eff}$, where $A_{eff} = 0.0023\,m^2$ and there are no data points available between $0\,mm$ and $0.5\,mm$. Focus in this section will be at lifts higher than $0.5\,mm$. The characteristics for the $0.125\,m$ pipe will be presented first, followed by the characteristics for the $2\,m$ and the $20\,m$ pipe. The transient opening behavior of the valve for the three configurations will be presented last in this part.

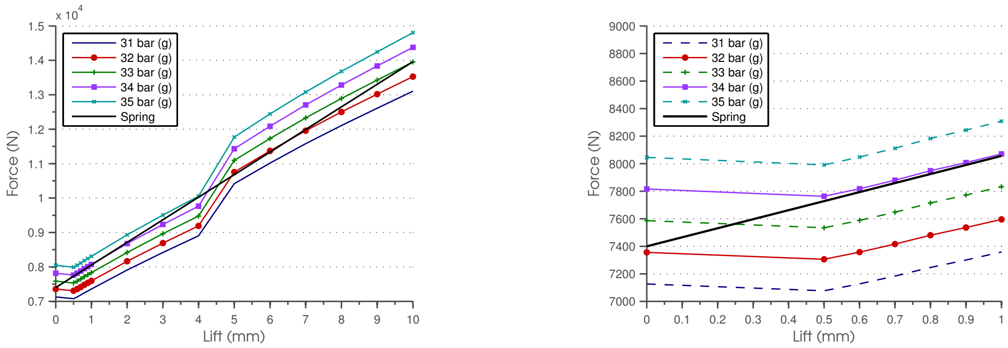


(a) Characteristics for $0.25\,m$ pipe simulations.

(b) Zoom of characteristics for $0.25\,m$ pipe simulations.

Figure 7.1: Steady state characteristics for the $0.25\,m$ pipe simulations.

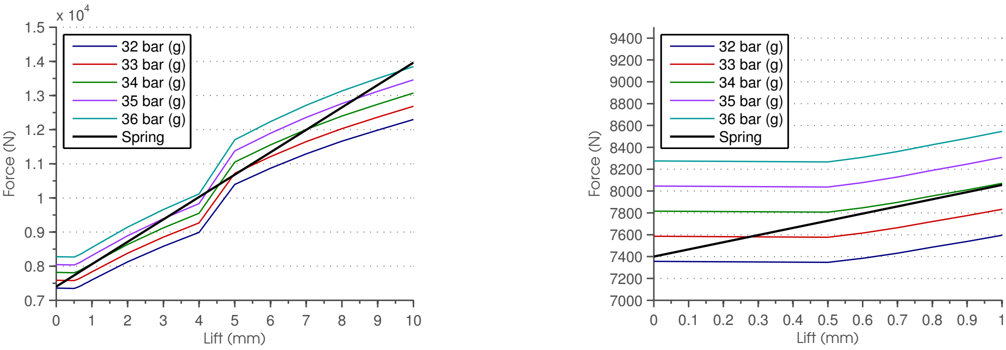
According to figure 7.1b, the fluid force exceeds the spring force for a pressure between $33\,bar(g)$ and $34\,bar(g)$ for a lift of $0.5\,mm$ which is the initial lift of the modeled valve. The isobar for $34\,bar(g)$ lies above the sping force curve up to a lift of $1\,mm$. Figure 7.1a shows that to maintain a lifting net force on the disc, the pressure must then increase to above $34\,bar(g)$. At $4\,mm$, a pressure of $35\,bar(g)$ generates high enough force to lift the disc up to $10\,mm$.



(a) Characteristics for $2\,m$ pipe simulations. (b) Zoom of characteristics for $2\,m$ pipe simulations.

Figure 7.2: Steady state characteristics for the $2\,m$ pipe simulations.

Figure 7.2b show that for an inlet pipe length of $2\,m$, the pressure required to lift the disc from its initial position is $\tilde{3}4\,bar(g)$. In fact, according to figure 7.2a, $34\,bar(g)$ generates a high enough force to lift the disc to $\tilde{2}\,mm$. To further move the disc, the pressure has to increase, at $35\,bar(g)$, the force is high enough to open the valve completely.



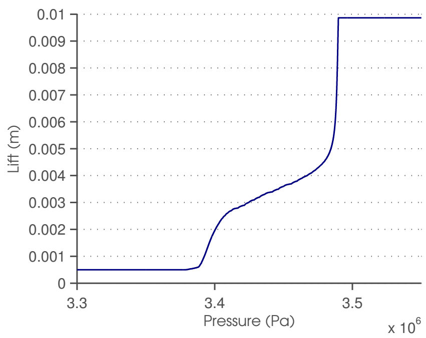
(a) Characteristics for $20\,m$ pipe simulations. (b) Zoom of characteristics for $20\,m$ pipe simulations.

Figure 7.3: Steady state characteristics for the $20\,m$ pipe simulations.

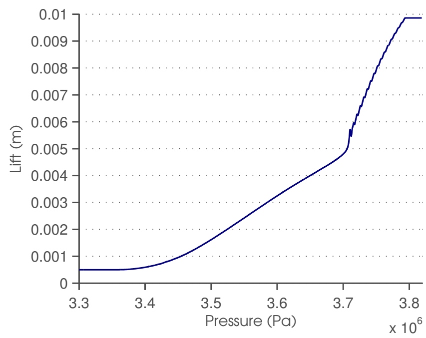
As displayed in figure 7.3b, the forces generated by a pressure of $34\,bar(g)$ is enough to lift the disc from its initial position of $0.5\,mm$. In order to reach a lift higher than $\tilde{1}\,mm$ the pressure in the system has to increase, this can be observed in figure 7.3a. For a constant pressure of $35\,bar(g)$, the forces are high enough to reach a lift of $3\,mm$. A pressure of $36\,bar(g)$ lifts the disc to $\tilde{1}0\,mm$.

From figures 7.1 and 7.3 it can be noted that the maximum force for the $20\,m$ pipe are significantly lower than for the $0.125\,m$ pipe. It can also be noted that for lifts between $0.5\,mm$ and $2\,mm$, the spring force curve tangents the isobar of $34\,bar(g)$ for all three configurations.

The transient respons for the $0.125\,m$ and $20\,m$ configurations will now be presented. The transient respons for the system with an inlet line of $2\,m$ proved to be very unstable and will be presented later in this chapter.



(a) Zoom of characteristics for $0.125\,m$ pipe simulations.



(b) Transient respons for $20\,m$ pipe simulations.

Figure 7.4: Transient respons for $20\,m$ - pipe and $0.125\,m$-pipe simulations.

The transient response of the valve connected to an inlet pipe of $0.125\,m$ is displayed in figure 7.4a. The opening starts at a pressure just below $34\,bar(g)$ and the disc reaches $\tilde{2}.5\,mm$ at a pressure of $34\,bar(g)$. Between 34 and $\tilde{3}5\,bar(g)$, the valve opens proportionally to the pressure increase. At $35\,bar(g)$ and $\tilde{4}\,mm$, the disc high-lifts to $10\,mm$ where it stabilizes. Figure 7.4b shows the transient opening behavior for the system with a $20\,m$ - inlet pipe. The valve starts to open at a pressure of $\tilde{3}4\,bar(g)$ and opens proportional to the pressure increase until $37\,bar(g)$ and $\tilde{5}\,mm$ is reached. Between 5 and $\tilde{5}.5\,mm$ the valve experience a high-lifting zone only to continue its proportional behavior until the maxmimum lift is reached for a pressure of $\tilde{3}8\,bar(g)$.

In figure 7.5, the isobar of $32\,bar(g)$ is graphically displayed for inlet pipe lengths of $0.125\,m$ and $20\,m$.

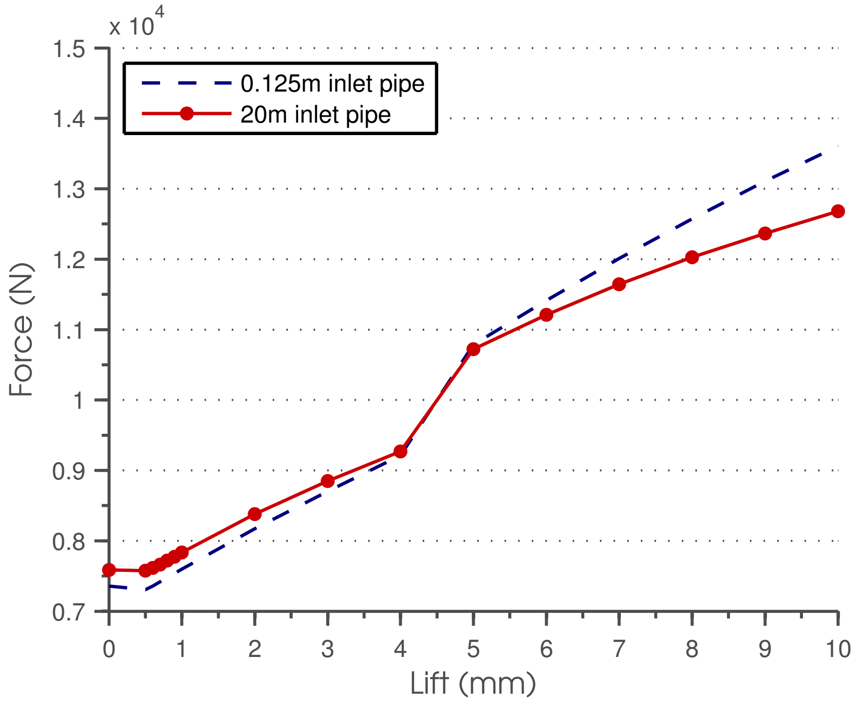
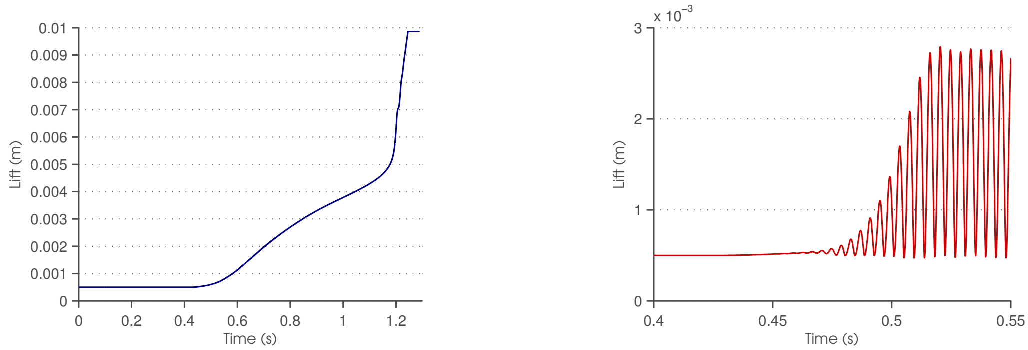


Figure 7.5: Isobar of 32 $bar(g)$ for inlet lines of 0.125 - and 20 m

From the figure it can be observed that the force generated for this particular isobar is greater for the system with an inlet line of 0.125 m at lifts above 5 mm. For lifts smaller tnat 5 mm, the force difference between the two systems is less significant.

7.1.2 Effect of hysteresis

To illustrate the effect of hysteresis, the results from the simulations with a $2\,m$ inlet pipe will be presented. Figure 7.6 shows the opening behavior of the valve with hysteresis (7.6a) and without hysteresis (7.6b).



(a) With hysteresis.

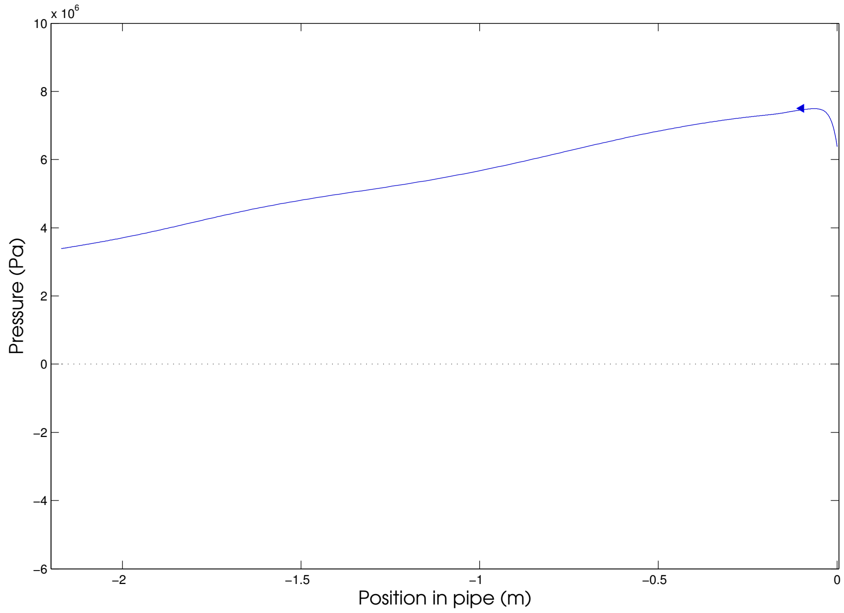
(b) Without hysteresis.

Figure 7.6: Transient simulations with a $2\,m$ inlet pipe.

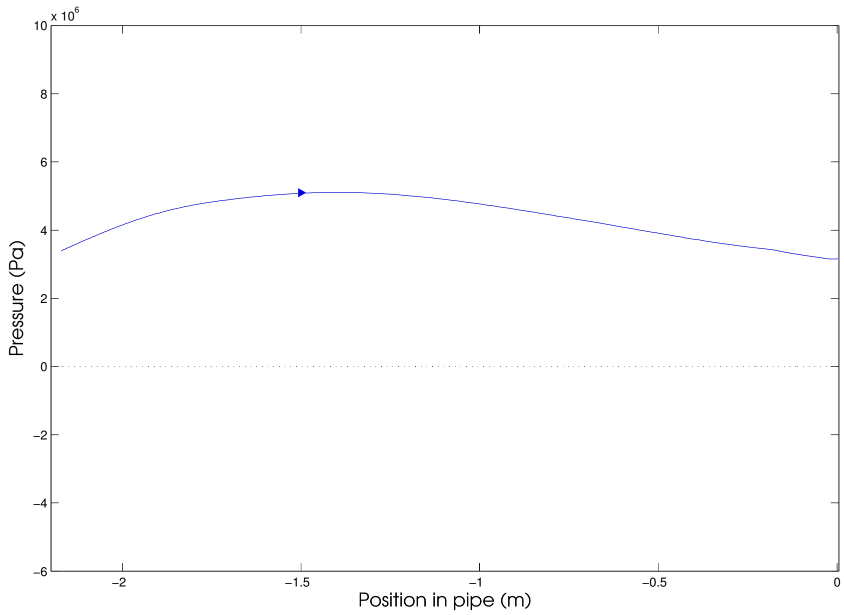
When comparing the opening behaviors of the valve with hysteresis (figure 7.6a) and the one without (fig 7.6b), the difference is easily observed. When subjected to hysteresis, the valve opens smoothly from $0.5\,mm$ to $10\,mm$, experiencing a more proportional behavior between $0.5\,mm$ and $5\,mm$ and a high lifting part between $5\,mm$ and the maximum lift. When the valve is simulated without addition of hysteresis, the disc starts to oscillate as soon as the valve starts to open and never reaches the maximum lift of $10\,mm$.

7.1.3 Acoustic pressure wave propagation

To study the occurence and effect of pressure waves propagating through the system, the results from the simulations with a $2\,m$ pipe will be used. In figures 7.7 and 7.8, the static pressure along the pipe for different times are presented. The corresponding times are marked in figure 7.9 which show the opening behavior of the valve when no hysteresis was added to the system. In water the speed of sound is $\tilde{1}500\,{}^{m}/_{s}$, but pressure in the pipe propagates with a velocity that ranges between $1500 - 4300\,{}^{m}/_{s}$. The wave front is somewhat diffusive, it should be mentioned that the numerical scheme used for the results presented here, is second order accurate and $CFL \approx 1$ in the middle of the pipe. The dotted line in figures 7.7a through 7.8b represents the vapor pressure of water at $298\,K$ which is $0.0317\,bar$.

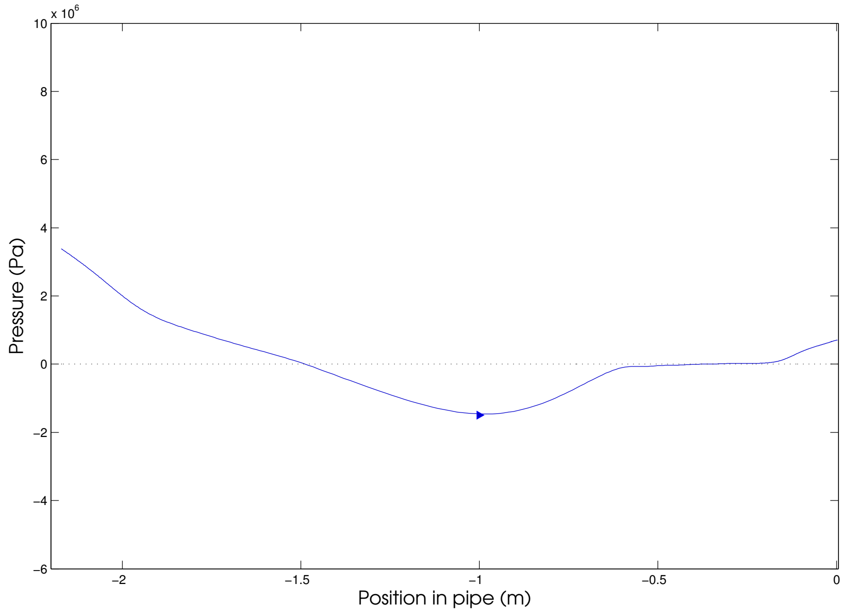


(a) Axial pressure in the pipe at $t = 0.50191\,s$.

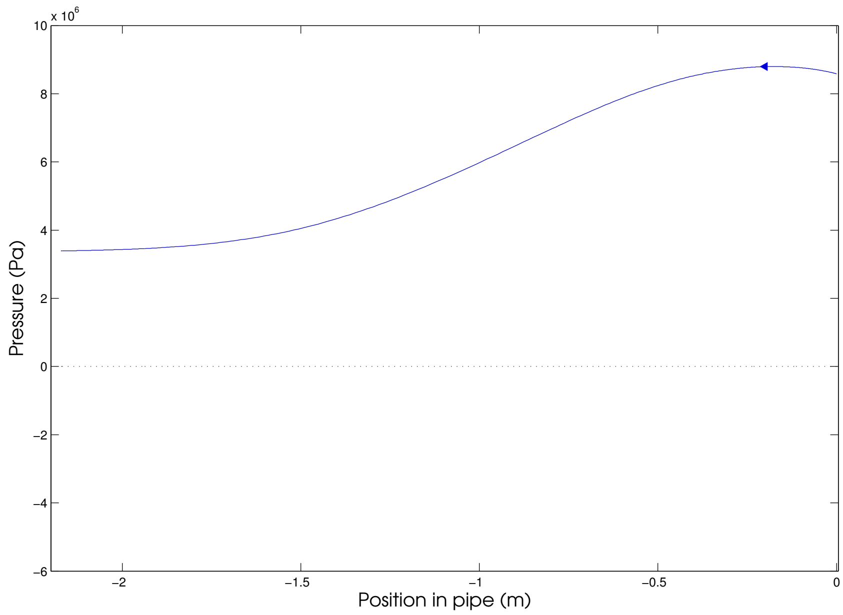


(b) Axial pressure in the pipe at $t = 0.50311$.

Figure 7.7: Pressure wave at different times. The direction of the wave is marked with an arrow. The inlet of the pipe is at $x = -2.125\,m$. The disc is located at $x = 0.0005\,m$ when the valve is closed.



(a) Axial pressure in the pipe at $t = 0.50430\,s$.



(b) Axial pressure in the pipe at $t = 0.50630\,s$.

Figure 7.8: Pressure wave at different times. The direction of the wave is marked with an arrow. The inlet of the pipe is at $x = -2.125\,m$. The disc is located at $x = 0.0005\,m$ when the valve is closed.

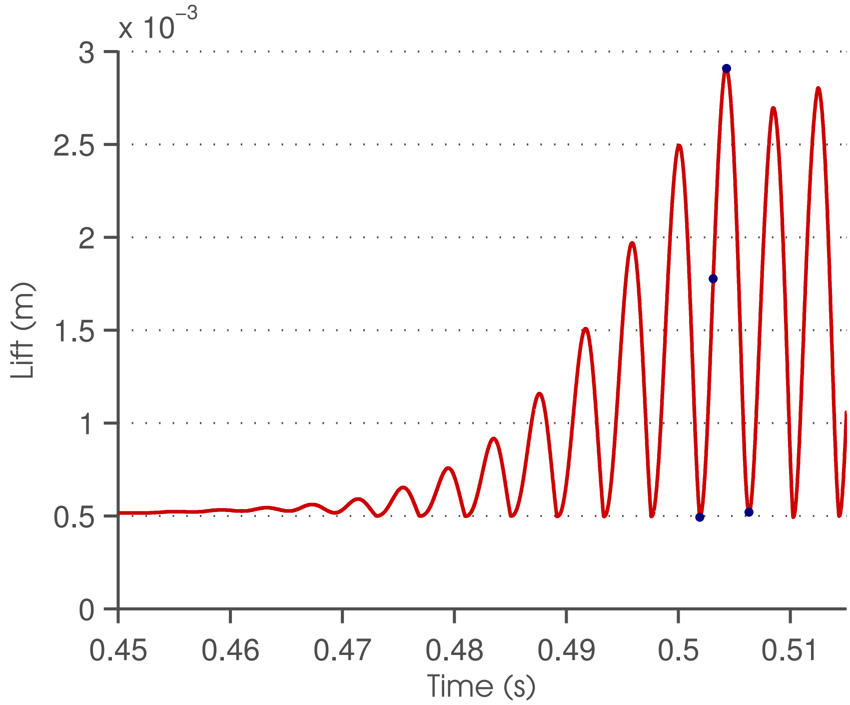


Figure 7.9: Lift against time for $2\,m$ pipe with different times corresponding to pressure waves in 7.7 and 7.8.

Comparing the pressure wave positions in figures 7.7a through 7.8b with the opening behavior in figure 7.9, it can be seen that when the valve has closed, a pressure wave is created at the seat of the valve. At $t = 0.50430\,s$, the valve has opened and pressure surge is visible in figure 7.8a. The oscillations of the disc thus seem to follow the pressure propagation in the pipe.

7.1.4 Modification of spring

As discussed in chapter 4, different spring settings might yield different opening behaviors. To evaluate if a valve which chatters can be stabilized by changing the spring, an additional simulation was made with the $2\,m$ inlet line. The two additional spring tested had the same initial spring tension as the original spring while the spring constant was increased or decreased by 30%. The resulting springs are displayed in figure 7.10 and the corresponding opening behavior in figure 7.11

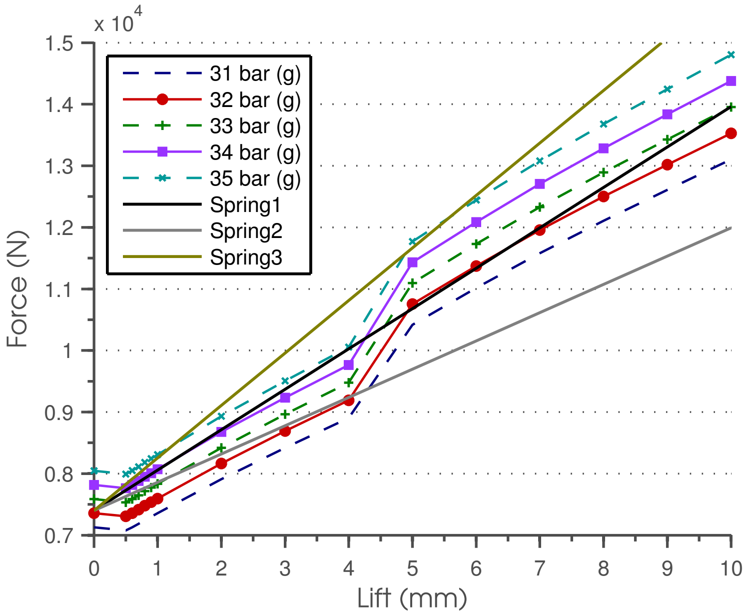


Figure 7.10: Valve characteristics for an inlet line of $2\,m$ with two modified springs (spring 2 and 3) as well as the original spring (spring 1).

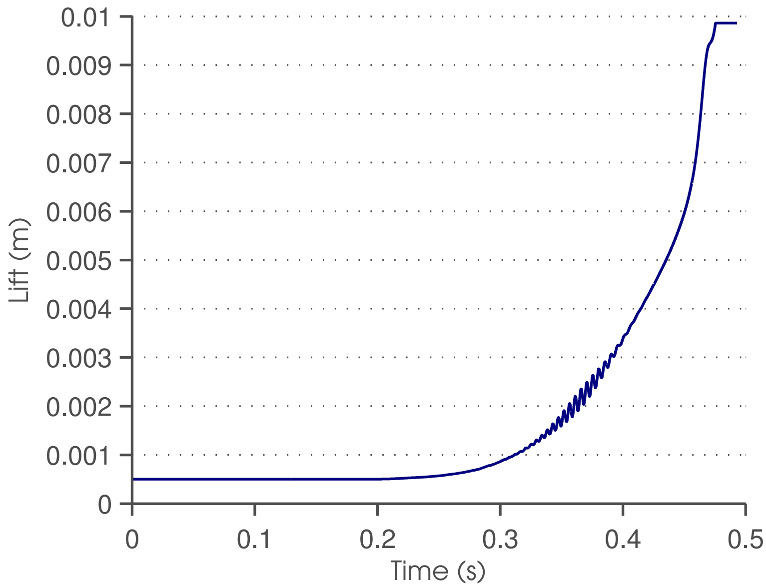


Figure 7.11: Lift versus time for inlet pipe of $2\,m$ with spring 2, a spring constant decreased by 30%

Figure 7.11 show that the valve starts to open at $t \approx 0.2\,s$, which can be compared to $t \approx 0.4\,s$ for the valve with original spring settings presented in figure 7.6. Between $1\,mm$ and $3\,mm$, the valve experiences a somewhat oscillating region but then stabilizes and continues to open smoothly. The proportional zone observed for the original spring settings is diminished and the opening behavior is slightly more high-lifting. It can be noted that the transient response of the valve is much more stable than that observed for the original spring settings when no hysteresis was added (figure 7.6b).

7.1.5 Effect of different pipe lengths

To investigate the possible effects of the inertia of the system on the disc behavior, six simulations with varying inlet lines lengths were performed. For all these, a frictional hysteresis of $400\,N$ has been applied. The results are displayed in figure 7.12.

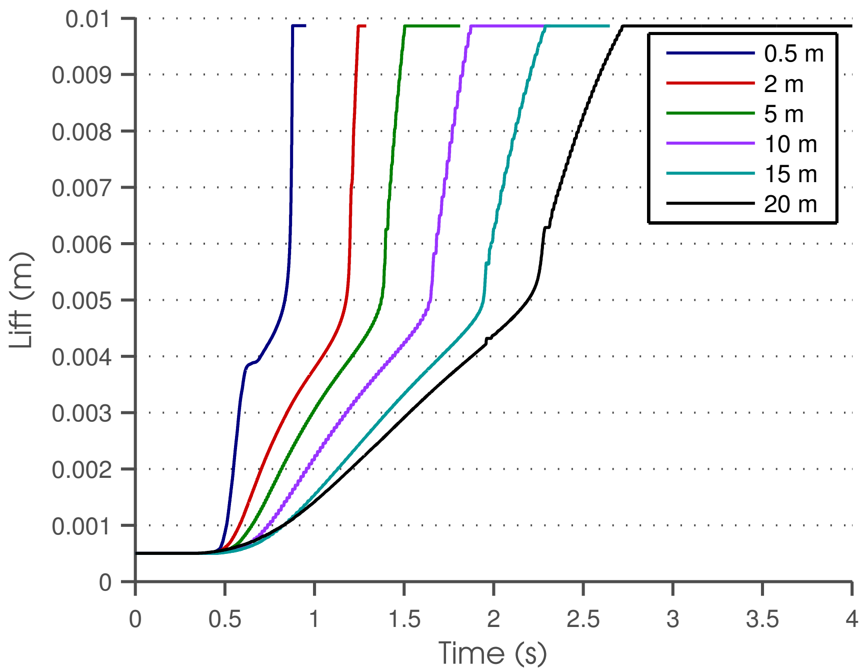


Figure 7.12: Lift vs time for different pipe lengths.

As can be seen in this figure, the opening behavior differs for different lengths of the inlet pipe. The high-lifting region that is observed for small lifts for the $0.5\,m$ system tends to take on a more proportional form as the length of the pipe increases. It can also be noted that the second high-lifting region, at lifts between $\tilde{4}\,mm$ and $10\,mm$ for the $0.5\,m$ pipe, becomes shorter, starts at higher lifts and also becomes less high-lifting for longer pipes. To furhter illustrate the transition from high-lifting to more proportional behavior, the velocity of the disc for the different inlet lines between $5\,mm$ and $10\,mm$ is calculated. The velocities are displayed in table 7.1.

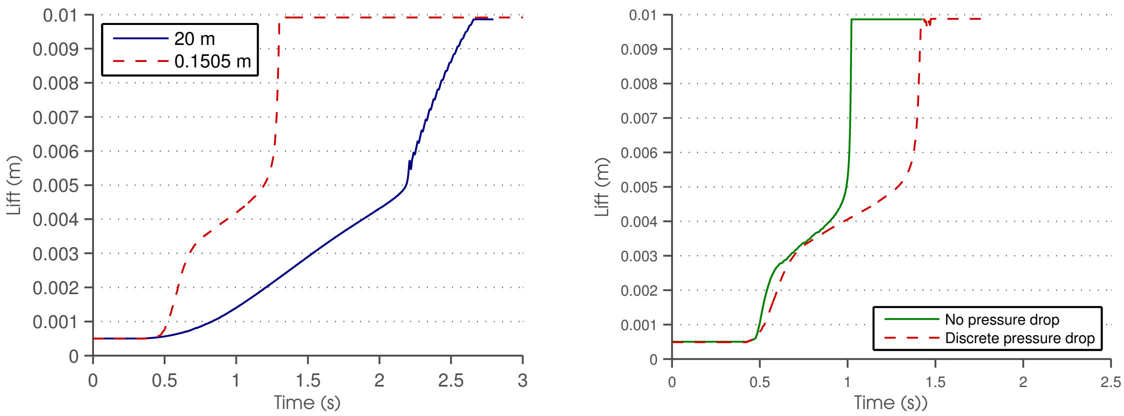
Table 7.1: Velocity of disc for the second high-lifting region.

Length of inlet pipe (m)	Velocity (m/s)
0.5	0.117
2	0.065
5	0.038
10	0.023
15	0.014
20	0.009

The values in table 7.1 show that the velocity of the disc for the second high-lifting region decreases as the inlet pipe becomes longer. Consequently, the behavior seen in figure 7.12 is confirmed.

7.1.6 Effect of pressure drop

Both of the cases displayed in figure 7.15 have been submitted to the same pressure drop, the $20\,m$ pipe through a "natural" pressure drop through the pipe and the $0.025\,m$ through a discrete pressure drop applied at the inlet.



(a) Lift versus time for a $20\,m$ inlet pipe and a $0.025\,m$ inlet pipe with discrete pressure drop.

(b) Lift versus time for $0.125\,m$ pipe and $0.025\,m$ pipe with discrete pressure drop.

Figure 7.13: Lift versus time for pipes with replication of pressure drop in $20\,m$ pipe.

For the longer pipe, the valve has a proportional part between $0.5\,mm$ and $\tilde{5}\,mm$, a high-lifting part between $5\,mm$ and $5.5\,mm$ and finally a proportional part between $5.5\,mm$ and its maximum lift. In figure 7.13a it can be seen that when replicating the pressure drop in the $20\,m$ pipe with a discrete boundary condition, the opening procedure of the longer pipe is not captured. The main difference between the two cases is that the system with the shorter pipe reacts faster than the system with a long inlet pipe. The proportional part of the $0.1505\,m$ - system is steeper than for the $20\,m$ - system and the high-lifting part reaches from $5\,mm$ to $10\,mm$, eliminating the second proportional phase that the $20\,m$ - system has. The shorter pipe with a discrete pressure drop shows the same behavior as a short pipe with no pressure drop, see figure 7.13b. The pipe with the discrete pressure does not deviate much from the pipe with no pressure drop in the beginning of the opening. As the valve opens the mass flow through it increases and the pressure drop as well, this leads to a somewhat delayed start of the high-lifting zone. But indeed a high-lifting zone, which is not the case for the $20\,m$ pipe, which has a significantly different opening procedure.

In section 7.1.1, the characteristics and transient opening behavior of the $20\,m$ - system was presented. Lifts and pressures marking different regions were noted to enable a comparison between the characteristics and the transient respons. The same observations will now be made for the $0.025\,m$ system with a discrete pressure drop.

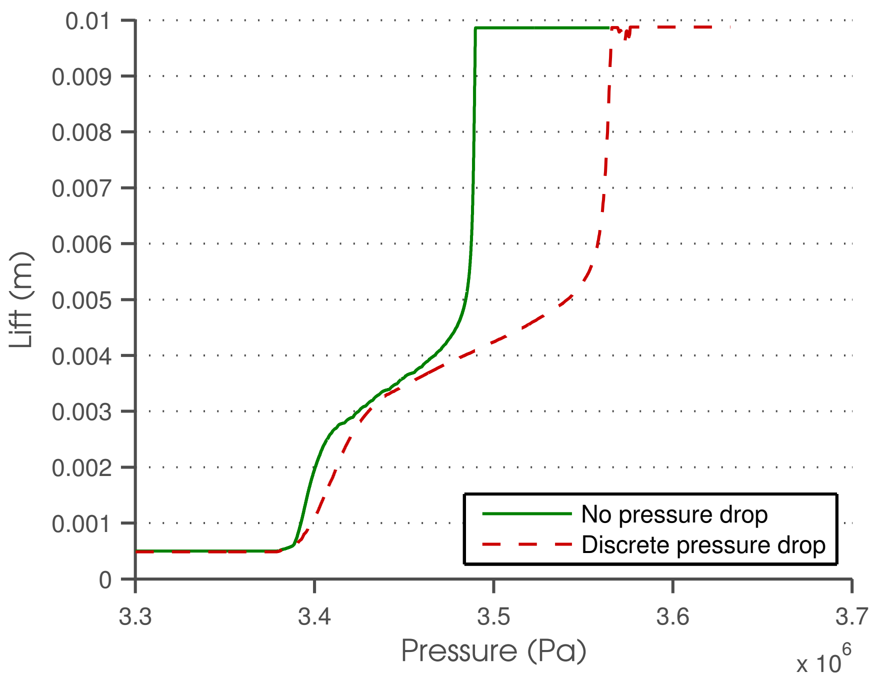


Figure 7.14: Lift versus inlet pressure for $0.125\,m$ pipe and $0.025\,m$ pipe with discrete pressure drop.

Figure 7.14 show that the valve opens at a pressure of $\tilde{3}4\,bar(g)$. Up to a lift of $\tilde{3}\,mm$, the opening behavior is rather fast, almost high-lifting, turning more proportional between $3\,mm$ and $\tilde{5}\,mm$. For a pressure of $\tilde{3}5.5\,bar(g)$, the disc high-lifts to $10\,mm$. After reaching $10\,mm$, the disc experiences a small regression to $\tilde{9}.6\,mm$ and then rises to $10\,mm$ again as the pressure increases.

Further effects of the pressure drop on the valve opening can be seen in figure 7.15.

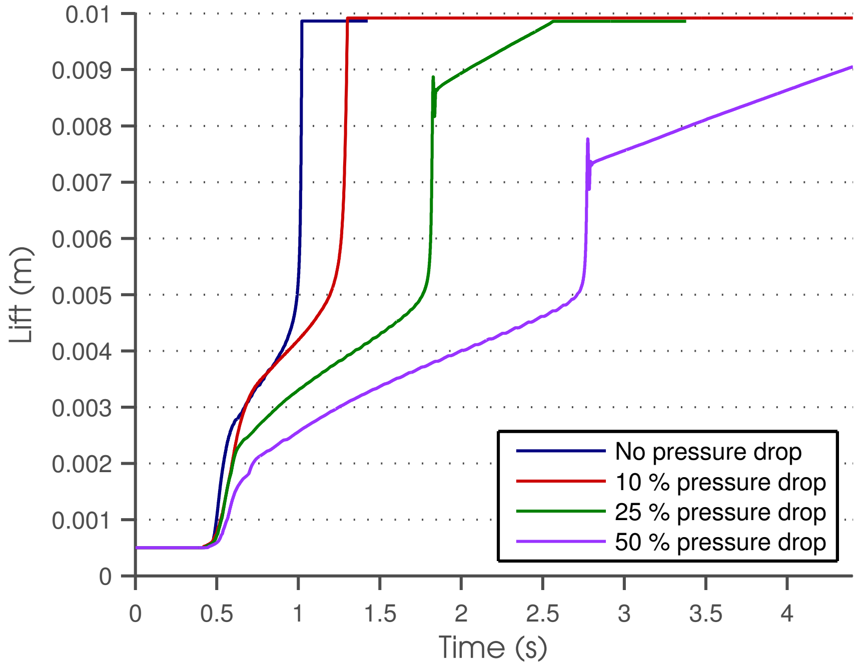


Figure 7.15: Lift versus time for different discrete pressure drops.

It is noted that in all cases the valve starts to open at the same time. However, the larger the discrete pressure drop is, the more delayed is the high-lifting behavior. It is not eliminated completely, but the endpoint of the high-lifting zone is at a smaller lift for larger pressure drops. The discrete pressure drop of 25 % and 50 % seem to have a stable point below the maximum lift of the valve where the high-lifting zone ends. As the pressure is further increased in time the valve continues to open proportional to the increase. Non of the discrete pressure drops yield any instabilities.

7.2 Constant mass flow

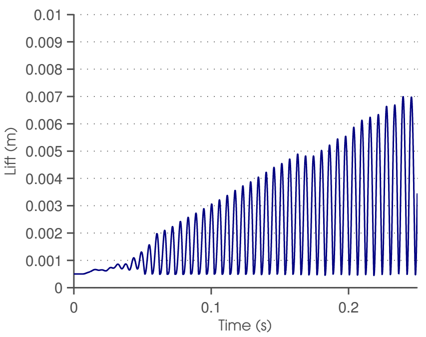
In this section the results from the simulations with the velocity inlet is presented.

7.2.1 Effect of volume

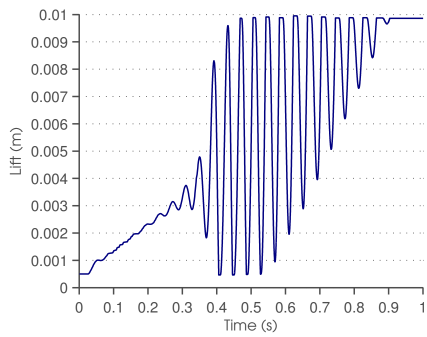
The relationship between the volume of the system and the frequency can be seen in table 7.2. It can be noted that the frequency is decreasing with a larger volume. The systems with volumes of $0.628\,m^3$ (case II) and $15.708\,m^3$ (case III) both stabilizes when $\dot{m} \approx 76\,{}^{\mathrm{kg}}\!/\!{}_{\mathrm{s}}$.

Table 7.2: Volume and frequency of oscillatons.

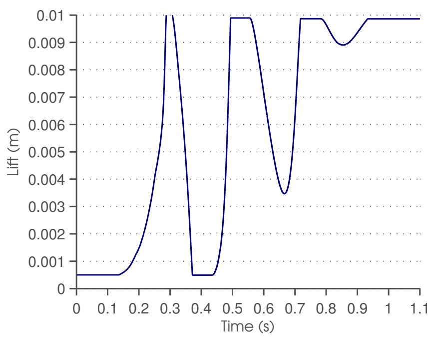
Volume (m^3)	$f(Hz)$
0.039	175.7
0.628	24.7
15.708	3.8



(a) Case I: Volume of $0.039\,m^3$.



(b) Case II: Volume of $0.628\,m^3$.



(c) Case III: Volume of $15.708\,m^3$.

Figure 7.16: Effect of volume on the transient respons of the valve.

7.2.2 Effect of mass flow

As can be seen in figure 7.17 the valve opens rather smoothly and proportionally up to $\tilde{2}.5\,mm$. The oscillations are larger the larger the mass flow is, up to $\dot{m} \approx 50\,{}^{\mathrm{kg}}\!/\!{}_{\mathrm{s}}$ where the oscillations starts to decay. The valve is completely open and stable for $\dot{m} = 76{}^{\mathrm{kg}}\!/\!{}_{\mathrm{s}}$. The frequency of the oscillations is constant, only the amplitude is changing.

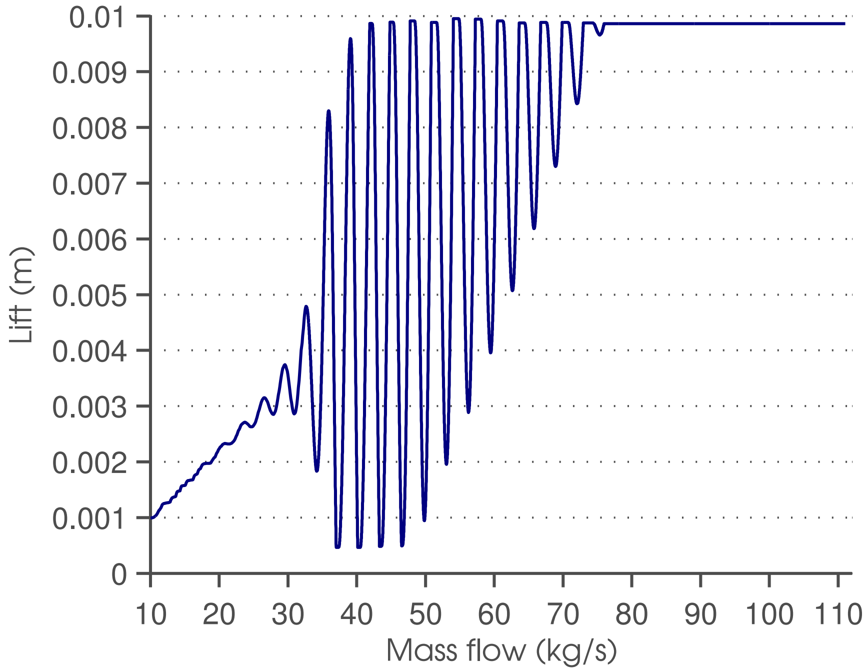
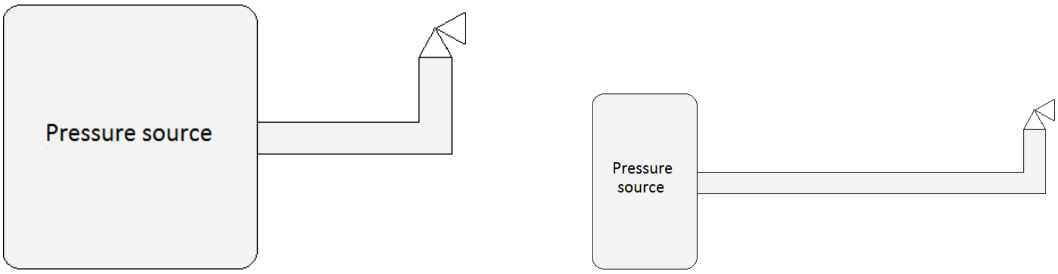


Figure 7.17: Opening procedure for case II.

7.2.3 Effect of inlet line

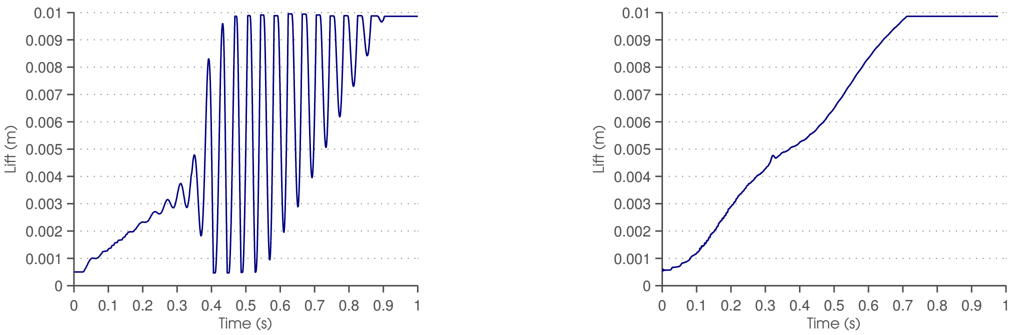
In figure 7.19, two geometries with the same volumes are compared, case II and case IV. It can be seen that for case IV, the oscillations of the system are almost completely attenuated. Both system reaches their maximum lift for $\dot{m} = 76\,{}^{\mathrm{kg}}/_{\mathrm{s}}$.



(a) Schematic view of geometry with short inlet, case II.

(b) Scematic view of geometry with long inlet line, case IV.

Figure 7.18: The two different geometries used.



(a) System with short inlet line, case II.

(b) System with long inlet line, case IV.

Figure 7.19: Effect of pipe length on the transient respons of the valve.

8. Analysis and Discussion

The first purpose of this thesis was to investigate if the characteristics of the valve generated with steady state simulations can predict the transient response of a safety relief valve. When examining the $0.25\,m$ pipe it can be concluded that the characteristics (figure 7.1a) predict the transient opening behavior (figure 7.4a) well. The transient behavior of the $20\,m$ pipe deviates from the behavior predicted by the characteristics, see figures 7.3a and 7.4b. The opening pressure is predicted but the high-lifting zones seen in the steady state predictions are not as distinct in the transient behavior. The opening of the valve, is not as fast as for the short pipe. The reason for this is thought to be the inertia of the system, the pressure changes do not instantaneously accelerate the fluid in the pipe, this is a transient effect and will therefore not be captured in the steady state simulations. In the comparison of the characteristics generated for the long pipe (figure 7.3a), and the replicated discrete pressure drop of the long pipe (figure 7.13a), it can be noted that the behavior to some extent is predicted. The opening pressure and the popping zones can be seen in both the characteristics and the transient opening. The valve behavior and characteristics are not consistent for intermediate lifts, this region in the characteristics should be carefully considered since there is a large uncertainty of the predicted force due to the complicated flow pattern induced by the disc settings and pressure field. This will be addressed later in this section. Further, the notch seen as the valve is fully opened in the pressure drop replication, is probably due to the fact that the isobars for the longer pipe flattens out and will thus have an additional point of intersection with the spring.

In the characteristics, figures 7.1a, 7.2a and 7.3a, it can also be seen that for all three lengths presented there is a region where the spring is tangent to some isobar. As discussed in chapter 5, this situations is thought to lead to instabilities where the valve exhibits on-off behavior. As previously stated, the forces are of equal magnitude and a small perturbation of the force equilibrium can therefore result in a large response of the system. This behavior is confirmed for the $2\,m$ pipe which oscillates severely (figure 7.6b).The oscillations follow the pressure propagation in simulation. However, the speed of sound has been observed to take on values significantly higher than what can be expected in water. This can be a numerical effect due to the fact that the computational cells are too small for the propagation speed of information and the fact that an explicit method have been used to update the time step and the disc position while an implicit method was used for the information propagation. It is important to keep in mind that the length of the pipe should not be taken as a definite value. As the pressure propagates too fast due to numerics, it will result in the illusion of a shorter pipe.

The $0.125\,m$ and the $20\,m$ pipe do not show the same oscillating behavior as the $2\,m$ pipe, compare figures 7.4 and 7.6b. This is also consistent with what is previously presented, that a sufficiently short or long pipe will stabilize the system. In the former the pressure source is close to the disc and there is no delay or attenuation of the increasing force which the increasing inlet pipe length results in. The effect of a long inlet line is confirmed by case II (see section 7.2.1) where two geometries with the same volume have been studied, one with a short inlet line and the other with a $20\,m$ inlet line. Studying figure 7.19, it is seen that the system with a $20\,m$ - pipe does not oscillate in the same manor as the one with a shorter pipe. The total mass of the systems is the same in both cases, but the mass that is accelerated is in the inlet pipe, a longer pipe will thus result in more inertia, which will stabilize the system.

To fairly compare the valve characteristics and the transient opening of the valve, the characteristics must be generated with the same type of inlet line as the valve is thought to be placed on. The valve itself can not be said to behave in a certain way. The behavior is strongly dependent on the system as a whole. The question of whether a valve is popping or proportional can not be answered unless the whole system in question is considered, which can be seen in figure 7.12.

The pressure drop and the inertia of the system affects the opening behavior of the valve in different ways. The pressure drop mainly shifts the opening to a higher pressure while the inertia of the system transform the high-lifting regions into a less rapid opening, this can be observed when comparing figures 7.15 and 7.12. Neither of the examined discrete pressure drops caused the valve to oscillate. On the contrary, figure 7.15 indicate that large pressure drops seem to create a stable operating point for the valve. Comparing with the characteristics for the $0.125\,m$ pipe and the $20\,m$ pipe in figure 7.5, they indicate that for a longer pipe (corresponding to a larger pressure drop), the isobars flatten out for higher lifts. This means that for a system with pressure drop, the lift that can be achieved for a specific inlet pressure is lower than what can be achieved for a system without pressure drop. That is, the range of *zone 2* (see section 5.2) is altered. It is further noted that the region after the second point of intersection, *zone 3* does not exhibit oscillations even though the pressure drop is large. Instead this point seems to be a stable point. The pressure must be increased further for the valve to continue to open. The proposed restriction of limiting the lift seems to be questionable. In accordance to previous work the 3%-rule seems to be too conservative, it can be discussed if pressure loss will lead to instable operation at all. It seems, that it is acoustic phenomena that causes instabilities. The two factors do however interact since a longer pipe results in a larger pressure drop and a longer propagation time of pressure changes.

To further evaluate and predict how a safety relief valve will operate the mass flow in to it should be considered. It can be noted that the valve together with the intermediate volume of $0.628\,m^3$ (case II) starts to open proportional to the mass flow, see figure 7.17. The amplitude of the oscillations increases with the mass flow, this corresponds to the popping region between the iso-lines for mass flow in figure 5.2. After a certain mass flow, the system again become stable, see figure 7.17. The behavior corresponds well to the hypothesis presented in section 5.2, comparing figures 5.1 and 5.2. By determining what mass flow the valve will be subjected to, it is possible to estimate if it will oscillate or not. Important to note is that the frequency of the oscillations are not dependent on the mass flow. The frequency is however dependent on the volume of the system, this is easily observed from figure 7.16. The smaller the volume the higher the frequency of the oscillations. This agrees with the idea that smaller volumes are more likely to chatter, whereas a larger volume is more likely to experience cyclic blow down. For this type of system, the proportional zone is not the unstable one. It is instead where the valve pops, *zone 2*, where the instabilities are found. This is important to consider, since the behavior of the valve is different depending on the upstream conditions and when tested, valves are often subjected to a constant pressure source.

Previous work have shown that safety relief valves can be stabilized by additional damping. The effect have been examined in this thesis by comparing systems which oscillates to systems where the hysteresis effect have been added. Studying figure 7.6, it can be concluded that it is an effective remedy to highly oscillating systems. The simulation results also suggest that an oscillating valve might be stabilized by modifications of the spring. A softer spring (figure 7.11) yielded a stable more high-lifting valve than the original spring (figure 7.6b) which also opened at a lower pressure. When modifying the spring, it is important to keep in mind the changes in set pressure and capacity that it might result in. It proved hard to stabilize the valve with a stiffer spring. The valve continued to chatter and a possible reason for this is that the spring force and the pressure force are of equal magnitude and neither of them is dominating for small lifts. Increasing the spring constant further might resolve the stability issues.

It is recommended to not have a popping safety valve for incompressible media since it can not withhold the pressure during the opening of the valve as described previously. Concerning the behavior of the $2\,m$ pipe, it can be said that the pressure sink below the disc is sufficient to result in a closing of the valve. These instabilities happens in a region where the valve is expected to open proportionally, this indicates that is not only the popping behavior of a valve that may result in instabilities. In the simulations that showed unstable behavior such as the $2\,m$ pipe displayed in figure 7.6b, the disc oscillations started in *zone 1* where the characteristics show strong alignment between the spring curve and the isobars. These systems never reached *zone 2*. However, none of the systems which passed through *zone 1* without showing tendencies to oscillate became unstable during the popping region (*zone 2*) which might indicate that valve instabilities cannot be attributed the the popping region alone for the situation where the upstream condition is a constant pressure source. Regarding the other relieving transient, with constant mass flow, the zone which is expected to be proportional (below m_1 in figure 5.2) does not exhibit unstable behavior, instead the popping region below the second mass iso-line is unstable. Again, the region after the second point of intersection, does not seem to be unstable.

Regarding the previous work concerning spring loaded safety valves, it can be said that the flow conditions in the system are of great importance. The blow down ring settings when there is such, greatly affects the disc force. It can thus be assumed that the angle of the shroud of the disc should be carefully considered in a numerical model. The density of the media is also an important factor and not considering cavitation in systems where it occurs would probably result in deviating results.

The opening of a safety relief valve is a matter of force balances. The forces acting on the disc, the spring force and the pressure force is of equal magnitude and a deviation of a few percent will result in significantly different behavior. Regarding the characteristics generated for the different systems they all experience the same large increase of force when the valve is $5\,mm$ open (see section 7.1.1). This was not expected and can be a result of that cavitation has not been taken into consideration. The flow- and pressure field will change due to this and in combination with the angle of the disc shroud the force on disc will be predicted differently. The sudden jump of the isobars in the characteristics result in that the force needed to open the valve further is provided by an isobar significantly lower than the one needed to start opening the valve. The high-lifting behavior of the valve could thus be caused by numerics. With that being said, an internecine comparison of the characteristics to the dynamic behavior and the different system can still be made.

This is not a replication of a real safety relief valve, the numerical model is based on a existing pressure relief valve but the geometry has been simplified. The constraints of the 2D-geometry is however thought to be small since most of the information propagates in the axial direction and the axial length of the system is much larger than the radial. The fact that the model is axi-symmetric will probably result in an overestimated stagnation zone below the disc, and hence a lower force. The same effect comes from not considering the phase transitions from liquid to gas. When the pressure drops to the vapor pressure and part of the flow cavitates, the pressure is sustained. Not allowing cavitation will result in pressure sinks.

9. Conclusions

The conclusions that can be made from this numerical study of a spring loaded pressure relief valve (SPRV) is listed below. The conclusions presented below should not be extrapolated to a different configuration and is merely an indication of what stability issues that can be captured and studied numerically.

- The stationary characteristics can predict the dynamic response of the safety relief vale with a inlet pipe is $0.25\,m$.
- An *SPRV* can not be categorized as popping or proportional regardless of the system to which it belongs.
- The numerical model can capture the interaction between the disc of the valve and the pressure propagation in the system.
- The inertia of the system changes the opening procedure of the valve.
- The numerical model indicates that a large pressure drop does not cause instabilities but alters the opening.
- Hysteresis can effectively reduce oscillations of the disc.
- For systems of sufficiently small volume and constant mass input, the chattering frequency depends only on the volume and not on the mass input. However the amplitude of chattering depends on the mass input.
- There exist an interval of the mass flow through the valve where the system does not oscillate for a constant mass input system.
- The results indicate that an oscillating system seems to be stabilized by longer inlet lines due to the inertia of the system.
- The numerical study of the valve does not show instable behavior for a sufficiently short inlet line when subjected to a pressure boundary condition.

10. Further work

A pressure relief valve is strongly coupled with the system to which it is attached. This thesis has mainly focused on investigating the opening of the valve and the preceding system. The coupling between the valve and the succeding system has not been adressed, neither has the valve geometry or the interaction between the solid parts of the valve and the fluid. The following list proposes some effects and features that probably affect the behaviour of the valve that has not been studied in this thesis.

- Update the disc position, time step and propagation using the same method, explicit or implicit.
- Effect of backpressure
- Effect of orfice area ratio
- Multiphase modelling
- Gaseous system
- Better resolution of the pressure wave propagation
- Fluid-solid interaction
- The effect of the inlet pipe diameter and valve nozzle diameter ratio for different pipe lengths

Appendices

A. Cavitation

Cavitation can be expected in the type of system which this numerical model is based on. The effect of the physical phenomena has been briefly examined it this thesis. This chapter will provide some theory about cavitation, a short description on how it is modeled in*ANSYS Fluent* and the numerical results of the simulations where it has been taken into account.

A.1 Theory

This section is based on the book ”Cavitation” by Robert T Knapp et al, [8].

Cavition is the phenomena of inception growth and implosion of cavities (voids filled with vapor or bubbles). It occurs in homogenous liquids as the pressure drops to the vapour pressure of the liquid at prevailing conditions. Regions with low pressure are assosiated with high fluid velocity or vibrations [**?**]. Cavitation can occur in the bulk of a fluid or at solid boundaries. In the event of a water hammer the pressure is not only increased, there might also be a decrease in pressure which could result in the occurrence of cavition.

For cavitation to occur in a system, the static pressure must be decreased enough for the first nuclei to form. Cavitation can be classified by the its location of origin and the principal characteristics of the physics. The different types of cavitation are defined as;

- Travelling cavition.
- Fixed cavitation.
- Vortex cavition.
- Vibratory cavitation.

In the system considered in thesis, two types of cavitation can be expected; *fixed cavitation* and *vortex cavitation.*

Fixed cavitation

Fixed cavitation can occur in a flowing stream as the flow detaches from a solid boundary immersed in the fluid. The fluid cavitates in order to relive the tension created in the liquid upstreams the cavitation zone. The incresed tension could be due to an edge the flow must turn around. As the flow turns the velocity increases and the dynamic part of the pressure too. Cavitation can be expected where a jet forms. These reasons leads to beliving that cavitation might occur at the seat of the valve.

Vortex cavitation

Vortex cavitation is also referred to as tip cavitation and can be found in the core of vortecis where there is high shear. It was originally observed at the tips of propellers but can also be oberved in the wake caused by boundary layer separation and it may also occur on the boundary surface of submerged jets. Due to the reasons stated above it is reasonal to belive that cavitation might occur below the shroud of the disc, where a recurculating swirl is to be found.

A.2 Modelling cavitation

This section is based on the theory provided by ANSYS. In this thesis the Schnerr and Sauer model has been used for the multiphase simulations. It is assumed that the slip velocity between the phases can be neglected. The default model and based on the *Reyleigh-Plesset* equation, which describes the dynamics of a single vapor bubble. The models in ANSYS Fluent assumes that there are enough nuclei for inception of bubbles. The generalized *Reyleigh-Plesset* equation is,

$$\mathfrak{R}_B \frac{D^2\mathfrak{R}_B}{Dt^2} + \frac{3}{2}\left(\frac{D\mathfrak{R}_B}{Dt}\right)^2 = \left(\frac{P_B - P}{\rho_l}\right) - \frac{4\nu}{\mathfrak{R}_B}\mathfrak{R}_B - \frac{2S}{\rho_l \mathfrak{R}_B} \tag{A.1}$$

- $\mathfrak{R}_B$ - Radius of bubble
- ρ_l - density of liquid
- P_B - Pressure at bubble surface
- P - local field pressure
- S - surface tension of bubble

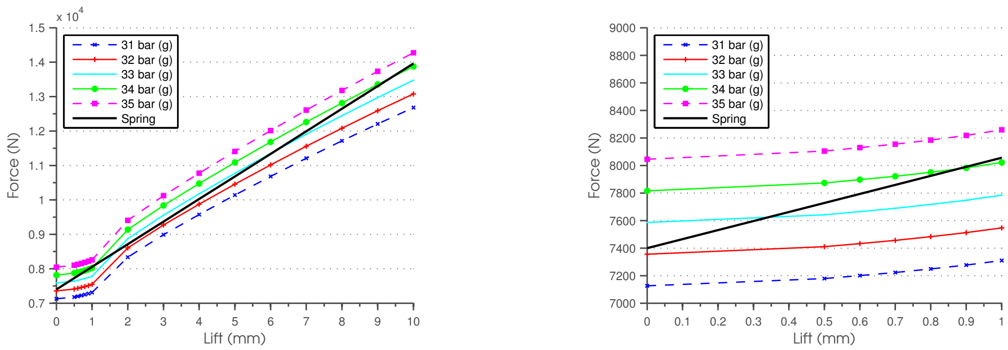
When cavitation occurs some of the liquid becomes vapour/gas and the flow is then multiphase. The multiphase model used in this thesis is the mixture model. Under the assumptions that the two phases are strongly coupled and hence accelerate together. When modelling a multiphase flow with the mixture model, only one set of transport equations are solved for the mixture, they are weighted with the volume fraction of the phases. When modeling cavitation, an additional transport equation must be solved, the vapour transport equation,

$$\frac{\partial}{\partial t}(\alpha \rho_v) + \nabla \cdot \left(\alpha \rho_v \vec{V}\right) = R_e - R_c \tag{A.2}$$

- α - volume fraction
- ρ_v vapour denstiy
- $\vec{V}$ vapour phase velocity
- R_c and R_e - masstransfer source terms, collapse and growth.

A.3 Results

The simulations presented previously in this chapter have all been run without taking the effect of cavitation into account. Not modelling cavitation when it occurs will result in regions with negative pressures which will affect the opening behavior and the characteristics of the valve. To show the different behavior of the valve when a cavitation model is included, steady state and transient simulations were made with the $0.25\,m$ inlet pipe. The results from the steady state simulations are displayed in figure A.1.



(a) Steady state characteristics. (b) Zoom of steady state characteristics.

Figure A.1: $0.25\,m$ - inlet pipe with caviation.

According to the steady state characteristics, the disc should start to lift at $\tilde{3}3.5\,bar(g)$. To continue lifting, the pressure must increase and at $\tilde{3}4.5\,bar(g)$, the fluid force isobar lies completely above the spring force curve. The valve will then open rapidly to a lift just below $10\,mm$.

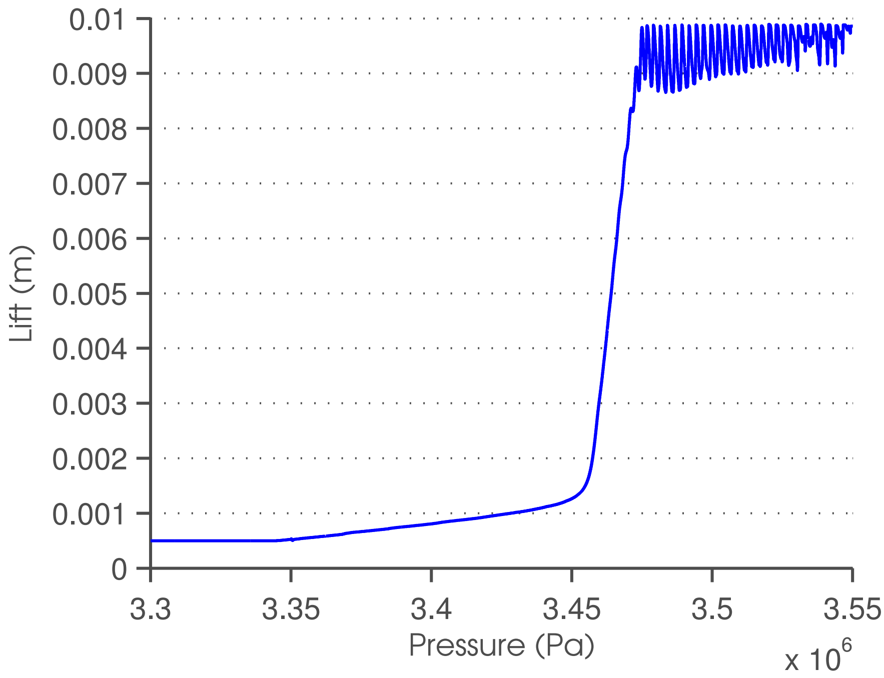


Figure A.2: Transient opening.

Looking at the transient behavior in figure A.2 and comparing to the steady state predictions from figure A.1, it can be seen that the opening behavior is almost identical to the prediction. This indicates that the characteristics predicts the transient opening behavior in a satisfactory way.

The steady state characteristics from the cavitation simulations were compared to the results from the simulations without a cavitation model. The comparison can be seen in figure A.3.

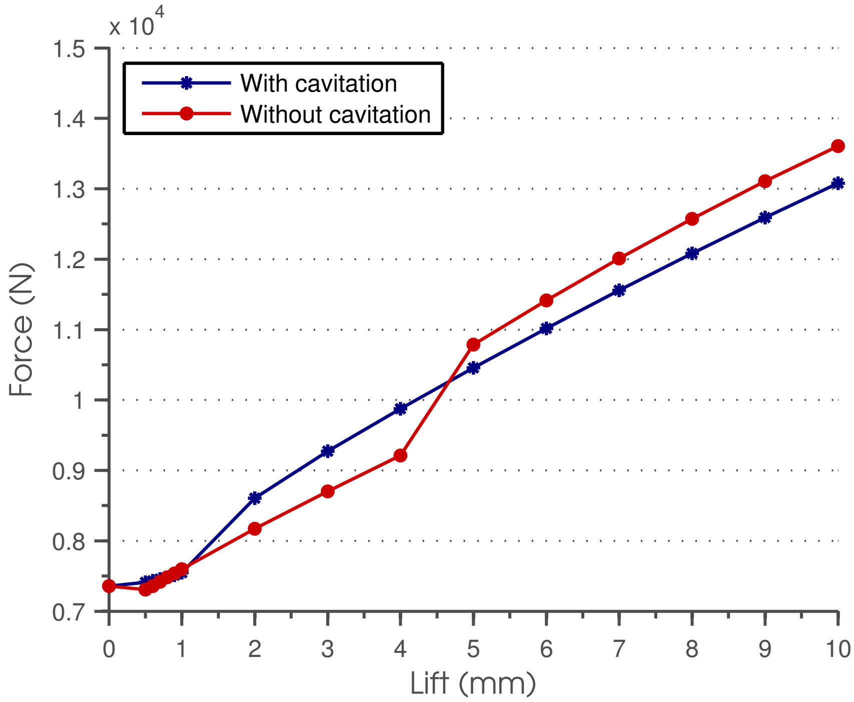


Figure A.3: Comparison between characteristics.

From this figure, it can be observed that the isobars for the two simulations are very similar for lifts below $1\,mm$. After $1\,mm$, the simulation results diverge. It can also be observed that the simulation with cavitation predicts a higher force for lifts below $\tilde{5}\,mm$. For lifts above this the force is higher without cavitation, the general behavior is the same except for the the uncertain region in the middle.

A.4 Analysis

Comparing the simulation results from cases that model cavitation to the results of the cases omitting caviation, it is apparent that cavitation is an important phenomena that greatly impacts the opening behaviour of the valve. Both the steady state characteristics and the transient simulations differ between cases with and without modelling of cavitation. Not including a model for cavitation results in regions below the disc with low pressures [9], originated from very high velocities. The pressure surges decreases the pressure force lifting the disc and the valve opening will therefore be delayed when cavitation is neglected. However, the results also show that the characteristics predict the transient behaviour of the valve both with and without cavitation. This means that as long as the characteristics and the transient simulations only are compared internally, cavitation with cavitation and non-cavitation with non-cavitation, the characteristics can be used to predict the transient opening behaviour of the valve.

B. Orifice area

As mentioned in section 2.6 the authors of the article [7] came to the conclusion that the ratio between the orifice area and outlet area should be larger than 6.0 in order to avoid chatter. In this thesis the lenght of the two dimensional outlet area is 115,01 mm. The valve opening ranges from 0.5 mm to 9.86 mm. The outlet and opening below the disc will revlove around the axis and the the actual area will be the surface area.

$$r = \frac{A_{outlet}}{A_{opening}} \approx 40$$

The ratio is with a large margin in the safe region.

C. Numerical settings in FLUENT

In this section the settings used in *FLUENT* will be presented.

C.1 Turbulence model and boundary conditions

- Turbulence model: *SST-kω*.
- Pressure inlet and pressure outlet for steady state simulations and transient simulations of case 1.
- Velocity inlet and pressure outlet for transient simulations of case 2.
- No - slip conditions at walls.

C.2 Calculation schemes

The following numerical schemes were used for the simulations.

- Pressure - velocity coupling: *Coupled.*
- Momentum and Tubulent Kinetic Energy: *QUICK.*
- Density and Specific Dissipation Rate: 1^{st} *order upwind.*
- Pressure: *Standard.*
- Gradient: *Least Square based.*
- Transient formulation: 1^{st} *order implicit* (*Only for transient simulations.*).

C.3 Additional settings

Here, additional settings such as time-step, under relaxation factors and settings for dynamic mesh and mulitphase modelling will be presented.

Time step

The time-steps chosen for the transient simulations was $1\,ms$, $0.1\,ms$, $0.05\,ms$ and $0.01\,ms$. The choice of time-step for each simulation was based on the convergence, see section C.4.

Under relaxation

- Flow Courant number = 20.
- Explicit relaxation factor for momentum = 0.75.
- Explicit relaxation factor for pressure = 0.75.
- All other under relaxation factors were set to *Default*.

Dynamic mesh

For the transient simulations, the following settings were used for the dynamic mesh;

- A constant $\alpha_{collapse}$ of 0.2.
- A constant α_{split} of 0.4.

Multiphase modelling

For the simulations with cavitation modelling, the following settings were used;

- Multiphase model: *Mixture model*
- No slip - velocity between the phases.
- Incompressible vapour phase.
- Compressible liquid phase.
- Mass transfer from liquid phase to vapour phase.
- Default settings for modelling of the cavitation.
- Pressure- velocity coupling was set to *Coupled - coupled with volume fractions.*
- For steady state simulations, *Pseudo-transient calculation* with a *time scale factor* of 1 was chosen.

C.4 Convergence

To monitor the convergence for the steady state simulations the mass imbalance between the inlet and outlet was observed, a value below $10^{-4}\,{}^{kg}/_{s}$ was considered low enough, however it was mostly lower. The drag at the disc boundary was also monitored and when reached a constant value.

To monitor the convergence for the transient simulation, the mass imbalance for each iteration within a timestep was checked. The number of iterations used within a timestep was chosen high enough to reach a constant value of the mass imbalance between the inlet and outlet.

Bibliography

[1] Bengt Andersson and dawsonera (e-book collection). *Computational fluid dynamics for engineers*. Cambridge University Press, Cambridge, 2012.

[2] J.L Garcia B. Ziegler, P. Coppolani. Dynamic behavior of check valves and safety valve with fluid interaction. *Nuclear Engineering and Design*, 124(1):391–401, 1990. 10.1115/1.4006904.

[3] J. Cremers and L. Friedel. Design of spring loaded safety valves with inlet and discharge pipe against chatter in the case of gas flow. *Chemical Engineering & Technology*, 26(5):573–576, 2003.

[4] Ron Darby. The dynamic response of pressure relief valves in vapor or gas service, part i: Mathematical model. *Journal of Loss Prevention in the Process Industries*, 26(6):1262 – 1268, 2013.

[5] O. Frommann and L. Friedel. Analysis of safety relief valve chatter induced by pressure waves in gas flow. *Journal of Loss Prevention in the Process Industries*, 11(4):279–290, 1998.

[6] ANSYS INC. Fluent 14.5 user guide.

[7] Hisao Izuchi. Stability analysis of safety valve, 2010.

[8] Robert T. Knapp, Frederick G. Hammitt, and James W. Daily. *Cavitation*. McGraw-Hill, New York, 1970.

[9] Vasilios Kourakos, Patrick Rambaud, Jean-Marie Buchlin, and Said Chabane. Flowforce in a safety relief valve under incompressible, compressible, and two-phase flow conditions. *Journal of Pressure Vessel Technology*, 135(1):11305, 2013.

[10] Kollika NGUON, Kazuaki INABA, Hiroto TAKAHASHI, and Kikuo KISHIMOTO. Numerical study on wave propagation in coupled pipe and homogeneous solid-liquid flow. *Theoretical and Applied Mechanics Japan*, 60:239–248, 2012.

[11] Smith, D. Burgess, J. Powers, and C. Relief device inlet piping: Beyond the 3 percent rule, 2011.

[12] A.S. Tijsseling. Fluid-structure {INTERACTION} {IN} liquid-filled {PIPE} systems: A {REVIEW}. *Journal of Fluids and Structures*, 10(2):109 – 146, 1996.

[13] Lurton Xavier. *An introduction to Underwater Acoustics Principles and applications.* Springer, 2010.