Energy Conversion and Management 81 (2014) 407–419

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

# A CFD analysis of the dynamics of a direct-operated safety relief valve mounted on a pressure vessel

Xueguan Song [a,d,*], Lei Cui [b], Maosen Cao [c], Wenping Cao [a], Youngchul Park [d,*], William M. Dempster [e]

[a] School of Electrical and Electronic Engineering, Merz Court, Newcastle University, Newcastle upon Tyne NE1 7RU, UK
[b] School of Civil Engineering and Geosciences, Cassie Building, Newcastle University, Newcastle upon Tyne NE1 7RU, UK
[c] Department of Engineering Mechanics, College of Mechanics and Materials, Hohai University, People's Republic of China
[d] Department of Mechanical Engineering, Dong-A University, Busan 604-714, South Korea
[e] Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK

ARTICLE INFO

*Article history:*
Available online 17 March 2014

*Keywords:*
Direct-operated safety relief valve
Pressure vessel
Blowdown
CFD
Moving mesh

ABSTRACT

In this study, a numerical model is developed to investigate the fluid and dynamic characteristics of a direct-operated safety relieve valve (SRV). The CFX code has been used to employ advanced computational fluid dynamics (CFD) techniques including moving mesh capabilities, multiple domains and valve piston motion using the CFX Expression Language (CEL). With a geometrically accurate CFD model of the SRV and the vessel, the complete transient process of the system from valve opening to valve closure is simulated. A detailed picture of the compressible fluid flowing through the SRV is obtained, including small-scale flow features in the seat regions. In addition, the flow forces on the disc and the lift are monitored and analyzed and the comparison of the effects of design parameters, are examined; including the adjusting ring position, vessel volume and spring stiffness. Results from the model allow the fluid and dynamic characteristics of the SRV to be investigated and shows that the model has great potential of assisting engineers in the preliminary design of SRVs, operating under actual conditions which are often found to be difficult to interpret in practice.



## 1. Introduction

A safety relief valve (SRV) is designed and used to open and relieve excess pressure and to close and prevent the further flow of fluid after normal conditions have been restored. There are two classical types of valve that are commercially available: pilot-operated and direct-operated. The pilot-operated valves can maintain a constant pressure in a hydraulic system and are frequently used in sophisticated hydraulic control systems. However, they are expensive and complex to manufacture and maintain. The direct-operated SRVs, also commonly known as the direct-loaded or spring-loaded SRVs, operate with a spring to pre-load and determine motion of the disc in the valve. Due to its simple configuration and reliable performance, this type of SRV is widely used to provide overpressure protection in practice.

As SRVs are crucial safety devices in many engineering industries such as the process industries, petrochemical industries, power plants and nuclear industries it is of critical importance to understand the SRVs' operating characteristics. From the 1990s, with the development of computational methods and capability, the computational analysis of the complex flows through valves have become feasible and effective thus leading to substantial research into the operation and design of SRVs [1–16]. For example, Vu and Wang [1] investigated the complex three-dimensional flow field of an oxygen SRV during an incident with computational fluid dynamics (CFD) techniques. The computational results indicated that vortex formation near the opening of the valve could be matched to the erosion pattern of the damaged hardware. Francis and Betts [4,5] studied the pressure distribution on the underside of a commercial SRV disc and identified the critical backpressure ratio when the SRV was subject to choked compressible flow. Kim et al. [7] carried out a computational study using two-dimensional, axisymmetric, compressible Navier–Stokes equations to simulate the gas flow between the nozzle exit and valve seat. Ahuja et al. [6,8] presented a series of high fidelity computational simulations of control valves used in NASA testing. 3D multi-element framework with submodels for grid adaption and multi-phase flow dynamics have been used to investigate the instability issue that results from valve operation. Dempster et al. [16] also showed by comparison with experimental data the success of relatively simple CFD models based on two equation turbulence models for predicting the gas flowrate

* Corresponding author at: Department of Mechanical Engineering, Dong-A University, Busan 604-714, South Korea. Tel.: +82(0)51 200 6991.

*E-mail addresses:* songxguan@gmail.com, xueguan.song@ncl.ac.uk (X. Song), parkyc67@dau.ac.kr (Y. Park).

http://dx.doi.org/10.1016/j.enconman.2014.02.021
0196-8904/

and disc forces in a complex straight through SRV used in the industrial refrigeration sector. These numerical studies revealed a rich variety of flow phenomena such as highly compressible, supersonic flows, recirculation zones patterns, shock formations. Meanwhile, many researchers started to apply the CFD method in design and analyze other types of valves such as hydraulic control valves and servo valves [17–21]. For example, Lisowski and Rajda [20] investigated the pressure loss for the flow in a hydraulic directional control valve constructed from logic valves. The study focused on a spool type directional control valve with pilot operated check valves. They examined the fluid forces using a 3D CFD modeling approach [21] and showed by comparison to experimental values that the resulting accuracy was satisfactory.

However, due to the restriction of CFD tools and computational resources, the numerical methods used in most of the studies mentioned previously were restricted to steady-state calculations at various fixed valve openings/lifts, rather than to dynamically simulate the true operation process of the valve opening and closing and the commonly found valve chattering conditions. Moreover, most numerical studies have not contained all the detailed geometrical effects since two dimensional rather than three dimensional flow analyses were mostly used to simulate the flow field in the previous studies. The simulation of the pressure relief of a pressurized system with its associated connecting pipe, pressure vessel and SRV requires the interaction of all components to be accounted for and as yet there has does not exist a systematic CFD based study attempted to address this issue. Completely neglecting the system or vessel is insufficient for deep insight into the performance of SRVs and the dynamic events which define this such as the opening time, instability duration and Blowdown.

In this paper, an improved dynamic analysis of an individual direct-operated SRV previously proposed by the authors [22,23] is extended using similar approaches involving DDM (domain decomposition method), domain interface methods, moving grids and use of CEL (CFX Expression Language) [22]. Differently and importantly, instead of specifying a fixed static pressure condition at the inlet of the valve, a pressure vessel is added to provide a relatively real condition, which means that this paper presents an investigation of a simple system including a pressure vessel and a direct-operated SRV rather than a SRV alone. As a result, the depressurization process from the valve opening to valve re-closure is fully monitored, and several important parameters such as the displacement/lift of the valve disc, massflow through the valve, blowdown of the valve are obtained. The simulation model is then used to demonstrate the usefulness of the modeling approach to valve designers and operators by investigating parameters of direct interest to them. In this study, by simulating the SRV and the connecting vessel the effect of (i) the vessel volume (ii) the spring stiffness and (iii) the adjusting ring position on the dynamic performance of a SRV is investigated. Additionally an SRV with and without bellows is also examined to show the versatility of the approach. At present generally available detailed experimental data does not exist to provide an extensive validation of the model however some valve blowdown results have been used to examine the model accuracy. Compared with the previous research, this work gives deeper insight into how a direct-operated SRV mounted on a pressure vessel really operates to prevent a sudden overpressure incident.

## 2. Motion of direct-operated SRV

### 2.1. Direct-operated SRV

Fig. 1 shows a half 3-D model of the direct-operated SRV studied in this research. It mainly consists of six parts: valve body, bonnet, nozzle, adjusting ring, movable valve disc and compressible spring.

The operation of this direct-operated SRV is based on a force balance. When the pressure at the inlet is below the set pressure, the resultant force exerts on the disc downwards and the disc remains seated on the nozzle in the closed position. As the system pressure increases to the set pressure, the resultant force decreases to zero gradually. When the inlet static pressure rises above the set pressure the resultant force reverses, and the disc begins to lift off its seat. However, as soon as the spring starts to compress, the spring force increases, which requires the system pressure to continue to rise before any further lift can occur, and for there to be any significant flow through the valve. The additional pressure above the set pressure is called the overpressure. After opening, the valve will close when the system pressure drops sufficiently below the set pressure to allow the spring force to overcome the fluid forces on the disc. The pressure at which the valve re-seats is the closing pressure, and the difference between the set pressure and the closing pressure as a fraction of the set pressure is referred to as Blowdown. Fig. 2 shows the typical disc travelling from the

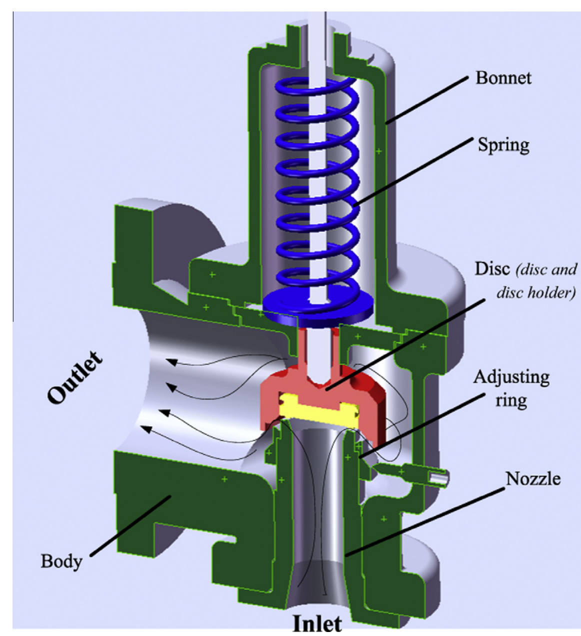


**Fig. 1.** Direct-operated SRV model.

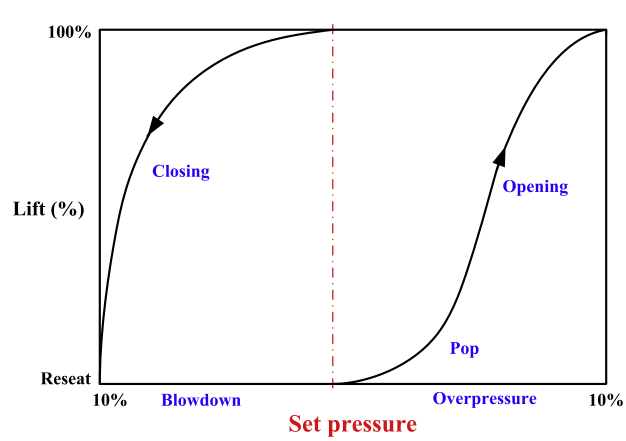


**Fig. 2.** Relationship between pressure and lift for a direct-operated SRV.