Journal of Loss Prevention in the Process Industries 45 (2017) 133–146

Contents lists available at ScienceDirect

Journal of Loss Prevention in the Process Industries

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

Experiments and transient simulation on spring-loaded pressure relief valve under high temperature and high pressure steam conditions

Liu Yang [a], Zhoujie Wang [a, b], William Dempster [c], Xinhai Yu [a, *], Shan-Tung Tu [a]

[a] Key Laboratory of Pressure Systems and Safety (MOE), School of Mechanical Engineering, East China University of Science and Technology, Shanghai 200237, China
[b] Shanghai Power Equipment Research Institute (SPERI), Shanghai 200240, China
[c] Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow, United Kingdom

ARTICLE INFO

Article history:
Received 26 February 2016
Received in revised form
10 September 2016
Accepted 30 November 2016
Available online 1 December 2016

Keywords:
Pressure relief valve
Computational fluid dynamics
Transient numerical simulation
High temperature
High pressure

ABSTRACT

Reliable performances of high temperature and high pressure operating steam pressure relief valves (HTHP PRVs) are extremely important for the safety of nuclear power plants. It is still a challenge to accurately describe the dynamic performance of HTHP PRVs. In this study, the accuracy of computational fluid dynamics (CFD) based modelling of the transient processes is examined. For one of the HTHP PRVs named DWPRV, the effects of different parameters on the dynamic performance were investigated by combining CFD simulation and experiments. In the simulation, the domain decomposition method (DDM) and the Grid Pre-deformation Method (GPM) were adopted to handle the moving disk geometry and the large mesh deformation. The effect of damping was also studied. It is confirmed that the use of CFD simulation can improve the design and settings of a HTHP PRV in a highly energetic service that is difficult to test due to safety reasons. For the DWPRV, it was found that the maximum flow rate occurs when the curtain area is 1.18 times the throat area. The degree of superheat ranging from 0 °C to 100 °C has a negligible effect on the performance of DWPRV regardless of the changes in the material mechanical properties with operating temperatures. The reseating pressure increases linearly with the rise in the distance between the upper adjusting ring and the sealing face. The lower adjusting ring exhibits a weak effect on the reseating pressure. For the ratios of rated lift to throat diameter equaling to 0.3 and 0.35, the DWPRV exhibits the higher blowdown for the ratio of 0.35.

1. Introduction

A pressure relief valve (PRV) is a significant safety accessory of pressure loaded installations. For nuclear or thermal power plants, PRVs act as the last passive protectors for the safety of the plant. It has been widely accepted that the reliable performance of a PRV is extremely important since its role in the nuclear leakage accident of Three Mile Island Nuclear Power Station in USA when the failure to close resulted in reactor core degradation. That accident is mainly attributed to a failure of one PRV (Rogovin, 1979). The safety of nuclear power plants is the focus worldwide considering that nuclear energy is one of the feasible solutions to replacing fossil fuels and the reduction of CO_2. However, even though safety valves could be considered as a mature technology, ensuring reliability is still an area of concern. Since the performance of safety valves is related to their dynamic response the recent improvements in computational fluid dynamics (CFD) based simulation for fluid structure interaction techniques hold the promise of direct simulation of the processes and a better understanding of how safety valves behave under real conditions. This paper addresses the development of CFD techniques and their application to safety valve simulation by presenting the results of a study to establish the accuracy of such techniques by comparison with blowdown data and measurement of the valve response. In practice, the requirement of safety valves to open and close at preset pressures is a challenging design task. For example, ASME I type valves can be required to open with 3% of set pressure and close within 4% of set pressure. This is commonly achieved by nozzle rings to control the flow path and pressure forces on the valve disk and established by trial and error on a test bench. Alternatively, CFD techniques have the potential for accurately determining these characteristics and help inform designers of appropriate disk and nozzle geometries and spring selection.

* Corresponding author.
E-mail address: yxhh@ecust.edu.cn (X. Yu).

http://dx.doi.org/10.1016/j.jlp.2016.11.019

This paper examines some of these issues using CFD based transient simulation techniques.

Over the past 30 years, experimental investigations on PRVs have made steady progress. In the 1980's, Sallet et al. experimentally studied the flow fields and pressure distribution inside a typical PRV (Sallet et al., 1980; Sallet, 1984) and developed a simplified semi-empirical equation for the prediction of critical mass flow rate through PRVs (Sallet, 1991). Singh et al. (1982). studied the valve dynamic behavior and observed chattering under subcooled liquid and vapor flow conditions. To investigate the effects of flow force, Narabayashi et al. (1986). conducted experiments on PRVs using steam-liquid two-phase flow and subcooled water. Their results showed that the reaction force of the two-phase flow and subcooled water to the valve stem was similar to steam flow and therefore independent of two phase conditions. Betts and Francis (1997) measured the pressure distribution on the underside of a commercial PRV disk when the PRV was subject to a choked compressible flow. However, in general experimental studies at high temperature and pressure steam (HTHP) are expensive, difficult to carry out resulting in limited data being available for these conditions. The HTHP steam here is defined as the steam whose pressure exceeds 10 MPa and the temperature is equal to or higher than 312 °C (the saturation temperature at 10 MPa) in reference to the standard of Supervision Regulation on Safety Technology for Stationary Pressure Vessel (TSG) 21-2016 which defines the high pressure vessel.

With the development of CFD techniques and low cost high performance computers, numerical simulation has been increasingly applied to the study of PRVs. The steady state simulations of PRVs were reported by several research groups. Kim et al. (2006). carried out a computational study using the two-dimensional, axisymmetric, compressible Navier-Stokes equations to study the gas flow between the PRV's nozzle exit and valve seat. Vu et al. (1994). investigated the complex three-dimensional flow field of an oxygen PRV during an incident by CFD analysis. The computational result indicated the formation of vortices near the opening of the valve which matched the erosion pattern of the damaged hardware. Moncalvo et al. (2009). discussed flow domain discretization requirements and the effect of the turbulence model on the reproduction accuracy of air mass flow rates in two PRVs using the CFX commercial CFD software. They concluded that the grid resolution is the decisive factor affecting the exactness of the calculations. With regard to the turbulence model, two equation systems were found to be sufficient with the Shear Stress Transport (SST) turbulence model showing the best accuracy. A common approach in the literature is to assume quasi-static conditions apply and examine the steady state behavior of the valve: this has been done both experimentally and computationally. Francis and Betts (1997) used the commercial software code FIDAP to predict an axisymmetric incompressible flow pattern inside a PRV. Experimental and simulation results were compared in the pressure distribution and force imposed on the valve disk. Dempster et al. (2006). conducted a CFD analysis on the characteristics of a conventional gas spring PRV operating at 10–20 bar. The CFD results for mass flow and force agreed well with the experimental data for a range of disk lifts. Similar studies were carried out by Kourakos et al. (2013). who investigated the flow force exerted on the valve disk with different inlet pressures and lift positions by experiments. Numerical simulations were performed in a 2D axisymmetric model of the valve for validation. Carneiro et al. (2012). studied the dynamic behavior of a commercial spring-type PRV. Experimental results were obtained for the valve discharge coefficient as a function of the valve opening fraction for steady state and transient flow conditions. A comparison of experiments and numerical simulations results displayed remarkably good agreement. In the study by Schmidt et al. (2009). on high pressure PRVs, an equation describing the critical mass flow rate of polyethylene and synthesis gas through a nozzle was derived. This equation was compared with a model presented in the ISO standard EN-ISO 4126-1 and the experimental data. The numerical results showed that the discharge coefficient of a PRV measured at moderate pressures can be extrapolated to high pressures if it is used in conjunction with the nozzle flow model for polyethylene and synthesis gases.

For a PRV, the dynamic performance is more important than the steady state one. For the study of the dynamic performance of a PRV, transient simulation is an important method and has been reported by several groups. The details can be found in Table 1 [16-171819202122]. Especially, Song et al (Rogovin, 1979; Song et al., 2013, 2014). made great progress in the transient simulation of a PRV. They firstly developed a 3-D CFD model in combination with dynamics equations to describe the fluid characteristics and dynamic performance of a spring-loaded PRV. In their studies, a moving mesh and the domain decomposition method (DDM) were introduced to the transient analysis. Despite the progress Song et al. achieved, there are still several problems that need to be addressed. Only the reseating pressures were used to verify their transient simulation results. The simulations by Song et al. are limited to air at less than 1 MPa (the pressure value hereafter refers to gauge pressure). For HTHP PRVs which features critical flow, neither experiments nor transient simulations can be found. Especially, HTHP PRVs usually have flexible disks whose shapes are complex, resulting in difficulties in domain decomposition and grid generation. In addition, in the simulations by Song et al. the gravitational force of valve disk components such as valve disk, valve spindle, disk holder and spring were not taken into consideration, likely leading to deviations of simulation results from the actual ones. Therefore, it is still a challenge to accurately simulate the dynamic performance of HTHP PRVs.

In this paper a number of challenges are investigated. We build upon the developments of Song et al. (2014). and show the accuracy of transient CFD approaches to predict the dynamic response of the valve under high pressure and temperature opening and closure conditions. To address this challenge, an experimental arrangement was established for the testing of HTHP PRVs in accordance with the ASME PTC 25 standard. The dynamic performance of the HTHP PRVs was recorded. For the transient simulation of the HTHP PRVs, the Grid Pre-deformation Method (GPM) was adopted to handle the complex shape of the flexible disk and the large mesh deformation that occurs when the disk is moving. Furthermore, we also investigate the influence of modelling the exit boundary condition. It is common to place an atmospheric boundary condition at the valve flange exit. However, in this study the external atmosphere is also modelled to capture aspects of the surrounding flow field as the flow exits the valve as a jet. The opportunity is also taken to show the value of the simulation technique to valve design and operation by investigating the effects of controlling parameters on the dynamic performance of HTHP PRVs, i.e., fluid properties, superheat temperature, adjusting ring position, and rated lift.

2. Experimental facility and testing

In this study, a HTHP PRV was provided by Wujiang Dongwu Machinery Co. Ltd, China. The set pressure, rated lift, throat size are 12.22 MPa, 22.5 mm, and 57.8 mm, respectively. Hereafter, this PRV is specifically referred to as DWPRV. Fig. 1 shows the experimental setup of the PRV performance testing. The test system arrangement and test procedure are in accordance with the ASME PTC 25 standard. A supercritical pressure boiler supplied saturated steam to a storage vessel which stored high pressure steam. The PRV for testing was mounted on a testing vessel. During the testing, the