Computers & Fluids 90 (2014) 57–64

Contents lists available at ScienceDirect

Computers & Fluids

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

Investigation of flow phenomena in air–water safety relief valves by means of a discontinuous Galerkin solver

F. Bassi [a], A. Crivellini [b], V. Dossena [c], N. Franchina [a,*], M. Savini [a]

[a] Università degli Studi di Bergamo, Dipartimento di Ingegneria, Italy
[b] Università Politecnica delle Marche, Dipartimento di Ingegneria Industriale e Scienze Matematiche, Italy
[c] Politecnico di Milano, Dipartimento di Energia, Lab. di Fluidodinamica delle Macchine, Italy

ARTICLE INFO

Article history:
Received 8 March 2013
Received in revised form 18 September 2013
Accepted 18 November 2013
Available online 28 November 2013

Keywords:
Discontinuous Galerkin
Incompressible flow simulations
Artificial compressibility
Discharge coefficient
Air–water safety relief valve

ABSTRACT

Safety valves are mechanical devices designed to protect a system (typically a pressure vessel) against excessive pressure; the operational behavior of spring loaded safety valves is governed by the difference between the reclosing force operated by the spring and the opening force operated by the pressure field acting on the opening device surface. The resulting flow path inside the valve body is very complicated due to sharp edges and sudden curvature characterizing the device geometry, and to complex flow phenomena like shock waves and supersonic expansions occurring when operating with compressible flows. As a consequence, the design of safety relief valves is an hard task.

In some cases safety relief valves are required to operate with both compressible and incompressible flows, *e.g.* in shell-type water–gas heat exchangers, but indeed the device behavior may significantly vary when operating with different fluids. Since the valve performance is strongly dependent on the fluid properties, a particular design of the valve trim (*i.e.* of the flow path) is thus required in order to guarantee proper functional characteristics when the valve is working either with gases or liquids.

In this work an accurate Discontinuous Galerkin (DG) solver is applied to compute higher-order approximations of the flow field within a 2″ J 3″ safety valve (according to API 526) designed for steam–water double protection. The investigation aims at clarifying the role played by viscous losses and compressibility effects in the discharge capability of the device when working with air or water. The main features of the code used for incompressible flows relying on a fully implicit high-order DG discretization of the coupled RANS and k–ω turbulence model equations are also described.

1. Introduction

Safety relief valves are devices widely used to protect industrial equipments from possible failures caused by unexpected off-design overpressure. The most relevant characteristic of the device is the capability of exhausting a prescribed amount of fluid at a specific overpressure and closing again within a narrow pressure range.

In some cases they are required to perform properly both for compressible and incompressible flows, *e.g.* in shell-type water–gas heat exchangers and/or in the economizer section of steam boilers where liquid or vapor can be contained, depending on the plant operating condition. It is at first glance evident that the behavior could be significantly different. At fixed opening position, for liquids the mass flow rate is mainly driven by the pressure difference acting between the inlet and outlet flange, whilst for gases is driven by the ratio of inlet to outlet absolute pressure and by the shape and position of the sonic throat (if the overall expansion ratio is sufficiently higher than the critical one). Moreover, at the same pressure, the disc lift is supposed to vary owing to the different fluid dynamic forces acting against the spring load resulting from the different pressure distribution on the moving device surface.

The operating and flow characteristics of a size range of safety valves are assessed by performing several experimental tests at different pressures on different valve bodies extracted from the size range subject to performance certification. The goal of the test procedure is to verify the ability of the safety valve designer because every tested sample should provide the same discharge coefficient (within a prescribed small tolerance), independently of pressure and size, and guarantee opening and reclosing within strict tolerances. Note that in the testing procedure, the Mach and Reynolds numbers, but also some geometric ratios (*e.g.* the inlet and outlet flanges commercially adopted may not follow in similitude the valve body) may vary. In the case of valves working with either compressible or incompressible fluids, two different discharge coefficients can be prescribed for the same valve size

* Corresponding author. Tel.: +39 0352052020; fax: +39 0352052077.
E-mail address: nicoletta.franchina@unibg.it (N. Franchina).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.11.021

Nomenclature

F	dimensional thrust
K	discharge coefficient
P_g	gauge pressure
Q	mass flow rate
$T_{nd} = F/(P_g A)$	non-dimensional thrust
a	sonic speed
$v_{id} = \sqrt{2P_g/\rho}$	ideal velocity
A	orifice area
exp	experimental
Greek symbols	
β	total to static expansion ratio
γ	specific heat ratio
ρ	density
Subscripts	
c	minimum flow area
T	total
1	orifice entrance
2	exit flange

range. However, in either case the flow characteristics of the valves in the given size range should be constant. The task for the designer then is a challenging one and CFD simulations are very helpful in the development stages before the final experimental certification. Laboratory tests can be performed only in few facilities and, besides their cost, could be unfeasible with certain gases and/or for large sizes, as API 526 T-type orifices.

The authors have already carried out numerical and experimental investigations [1–4] aimed at clarifying the physical reasons for the observed performance of safety relief valves in several working conditions, and the present work is a further step toward a better understanding of the devices behavior. For this purpose, the paper presents high-order accurate Discontinuous Galerkin (DG) solutions of compressible (air) and incompressible (water) flows on a 2″ J 3″ safety relief valve and comparison with measurements performed at the Politecnico di Milano safety valves test rig. Both experimental and computational results are analyzed and compared in order to investigate the different trend of both the discharge coefficient and the thrust acting on moving device as function of the increasing upstream pressure at fixed geometry (*i.e.* at fixed disc lift).

The DG solver employed solves the Reynolds averaged Navier–Stokes (RANS) equations coupled with a k–ω turbulence model. The polynomial approximation is defined in the physical space and is continuous within each element and discontinuous at element interfaces. For steady problems, a linearly implicit backward Euler method with an analytically derived Jacobian matrix of the residuals is used. For compressible flows, numerical inviscid and viscous fluxes at element interfaces are computed by means of the "exact" Riemann solver of Gottlieb and Groth [5] and of the BRMPS scheme [6], respectively. Further details about the DG implementation for the compressible RANS and k–ω turbulence model equations can be found in [7].

Across discontinuities, like shock waves and slip lines occurring inside safety valves at transonic and supersonic regimes, high-order solutions need additional dissipation to control the numerical oscillations arising within the mesh elements. For this purpose, the shock-capturing technique described in [2] is used. The technique is local and introduces an amount of artificial viscosity proportional to the inviscid residual of the DG space discretization that allows crisp representations of discontinuities and preserves high accuracy within smooth regions. For incompressible flows, the DG solver handles the incompressible RANS and k–ω equations using the original formulation of the numerical inviscid flux presented in [8,9], and again the BRMPS scheme [6] for the discretization of the viscous flux.

The purpose of this paper is threefold. First, it presents an analysis and discussion on the observed trends of global characteristics, *i.e.* the opening force and the discharge coefficient, obtained in an experimental campaign performed both in air and water on a valve designed for steam–water double protection. Second, these results are used to assess the reliability and accuracy of the high-order DG code here employed. Third and most important, the paper shows that a detailed knowledge of the flow field in the inner valve body, almost impossible to be measured but quite easily obtained from numerical simulations, provides physical insight into the flow features that could be useful in the device design.

The outline of the paper is as follows. Section 2 describes the main feature of the DG solver applied to incompressible fluids. Numerical results are discussed and compared with experimental data in Section 3. Finally, Section 4 gives some concluding remarks on the different operational behavior of valves working with water or air.

2. The DG solver for turbulent incompressible flows

In this section the main features of the DG solver for the incompressible RANS equations coupled with a k–ω turbulence model are described. The approach follows from a quite natural extension of the DG implementation used for the compressible flow simulations [7,2], to the incompressible Navier–Stokes solver introduced in [8,9]. The new solver has already been extensively tested and a more comprehensive presentation of the method will be given in a forthcoming paper, now in preparation.

The RANS and k–ω turbulence model equations for an incompressible flow with uniform density and constant transport properties can be written as

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \hat{\tau}_{ji}}{\partial x_j}, \tag{2}$$

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j}(u_j k) = \frac{\partial}{\partial x_j}\left[(\nu + \sigma^* \bar{\nu}_t)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta^* \bar{k} e^{\tilde{\omega}_r}, \tag{3}$$

$$\frac{\partial \tilde{\omega}}{\partial t} + \frac{\partial}{\partial x_j}(u_j \tilde{\omega}) = \frac{\partial}{\partial x_j}\left[(\nu + \sigma \bar{\nu}_t)\frac{\partial \tilde{\omega}}{\partial x_j}\right] + \frac{\alpha}{\bar{k}}\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta e^{\tilde{\omega}_r} + (\nu + \sigma \bar{\nu}_t)\frac{\partial \tilde{\omega}}{\partial x_k}\frac{\partial \tilde{\omega}}{\partial x_k}, \tag{4}$$

where u_j is the j-component of the velocity vector, p is the pressure and $\hat{\tau}_{ji}$ is the j-component in i-direction of the total stress tensor. The turbulent and total stress tensors and the eddy viscosity are given by

$$\tau_{ij} = 2\bar{\nu}_t\left[S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] - \frac{2}{3}\bar{k}\delta_{ij}, \tag{5}$$

$$\hat{\tau}_{ij} = 2\nu\left[S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] + \tau_{ij}, \tag{6}$$