Journal of Natural Gas Science and Engineering 28 (2016) 479–490

Contents lists available at ScienceDirect

Journal of Natural Gas Science and Engineering

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

Simulation of the transient flow in a natural gas compression system using a high-order upwind scheme considering the real-gas behaviors

Lei Zhang

College of Mechanical and Transportation Engineering, Beijing Key Laboratory of Process Fluid Filtration and Separation, China University of Petroleum, Beijing 102249, China

ARTICLE INFO

Article history:
Received 26 September 2015
Received in revised form
4 December 2015
Accepted 10 December 2015
Available online 13 December 2015

Keywords:
Centrifugal compressor
Emergency shutdown
Recycle valve
Surge avoidance
Finite volume method

ABSTRACT

The transient flow in a pipe flow network was numerically simulated to study the performance of the surge avoidance system in a natural gas compression station. The simulation was performed based on a combined approach that the flow in the pipe was simulated using a finite volume method with a high-order upwind scheme considering the real-gas behaviors and the behaviors of elements (e.g., compressor and valve) were evaluated using quasi-steady models. In the pipe flow simulation, characteristic boundary conditions were used to couple with the quasi-steady modeling of elements. The numerical scheme for the pipe flow simulation was first used to predict the rapid transients in a single pipe after the sudden closure of a downstream valve. The predicted distributions of the flow properties indicated that the present numerical scheme was capable of simulating the pipe flow under extreme flow conditions with high order of accuracy. The present approach was further used to simulate the transient flow during the emergency shutdown of compressor in an experimental piping network. Simulation results, including traces of operating points and histories of pressures at both sides of the compressor, were compared to the corresponding experimental data, and an agreement with a maximum deviation less than 10% was obtained in the primary control stage. The agreement proved the accuracy of the present approach for the simulation of the transient flow in a piping network and its validity in evaluating the performance of the surge avoidance system in a natural gas compressor station.

1. Introduction

Surge is the dynamic instability characterized by flow reversal in gas compressors and occurs if the maximum head at a certain rotation speed is achieved and the inlet flow rate further reduces. Since the reversed gas flow during surge can impose massive fluctuating load on the rotor, severe damage can occur to the compressor. In a natural gas compression station, the surge avoidance system mainly consists of recycle piping loops around the compressor and a corresponding recycle valve as illustrated by Fig. 1. The surge avoidance system functions to increase the inlet flow, reduce the pressure ratio, and finally move the operating point away from the surge operating conditions. Therefore, it is crucial to protect the compressor from the damage of surge in a natural gas compression station. The performance of surge avoidance system is determined by the pipe arrangements upstream and downstream of the compressor, the check valve behavior and the characteristics of the recycle valve, including its size, response time, and stroke time. To ensure the safe operation of the compressor, the surge avoidance system needs to be properly designed to make it capable of protecting the compressor from surge at normal operating conditions.

The performance of the surge avoidance system is evaluated by investigating its behaviors during the emergency shutdown (ESD) of compressors. ESD is an event that a compressor suddenly loses its driving power after, for example, the cutoff of fuel supply to a gas turbine or electricity to an electric motor. Due to the rapid reduction of rotor rotation speed after the start of ESD, the operation point of the compressor moves rapidly towards the surge operating conditions. Therefore, this process imposes the most stringent requirement on the surge avoidance system. Hence, examining the behavior of the surge avoidance system during the ESD of compressor is an effective way to evaluate its performance and verify the design of the system. Both experimental and modeling approaches have been used to examine the behavior of the surge avoidance system during the ESD of compressor in a natural gas compression station. A standard experimental study was carried

E-mail address: zlei@cup.edu.cn.

http://dx.doi.org/10.1016/j.jngse.2015.12.012
1875-5100/

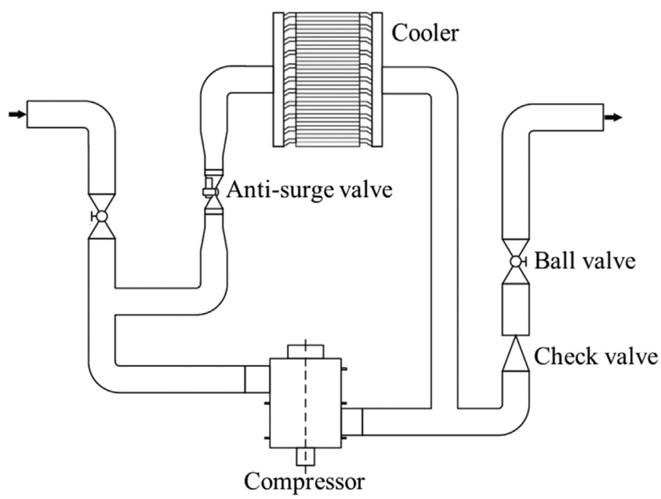

Fig. 1. Cooled recycle pipe loop for a surge avoidance system.

out by Moore et al. (2010) who conducted ESD experiments starting from several initial operating conditions on a full-scale test centrifugal compressor. Experimental data, such as the operating points of the compressor and gas properties at valves and compressors during the entire shutdown process, were measured and used to validate the simulation results of the same cases by the Stoner Pipeline Simulation, a computer program for solving the one-dimensional pipe flow equations based on the method of characteristics. On the other hand, compared the experiments, modeling approaches are relatively more convenient and indeed have been more commonly used to study the performance of the surge avoidance system in natural gas compression stations. Kurz and White (2004) proposed a lumped system volume approach to model the compressor's discharge volume, which is the pipe volume bounded by the discharge side of the compressor, downstream check valve, and the recycle valve. Their model could be further reduced to an even simpler form and was found capable of properly sizing the upstream and downstream pipe volumes for optimizing the surge avoidance system by simulating the ESD process of the compressor. Furthermore, more detailed simulation of the pipe flow, in contrast to the lumped volume approach, was also pursued by researchers through the numerical solution of the one-dimensional governing equations for the pipe flow. The numerical study by Botros et al. (1991) was one of the earliest practices that can be attributed to this category. In their simulation of the transient events in the piping networks, the pipe flow equations were solved using the method of characteristics, and quasi-steady methods were employed for the modeling of elements such as compressor, valve, and pipe fittings. Especially for the emulation of valve's movement, an ordinary differential equation determining the instantaneous position of the valve's actuator as a function of time was also proposed. More detailed formulations for the solution procedure of the governing equations based on the method of characteristics and additional quasi-steady models for cooler and scrubber were further proposed by Botros and Petela (1994).

Besides the method of characteristics, other advanced numerical methods have also been used in the solution of partial differential equations for the pipe flow to either achieve high order of spatial accuracy or capture the rapid flow transients in a pipe flow network. Transient pipe flows due to the sudden closure of a downstream valve or periodic perturbations at the boundaries have been extensively selected as the numerical experiments to examine the capabilities of the numerical methods under extreme flow conditions. Greyvensein (2002) developed an implicit finite difference method based on the simultaneous pressure correction to simulate the transient flow in a single pipe after the sudden closure of the downstream valve. Simulation results were compared to those obtained by other numerical methods, for example, the method of characteristics and the two-step Lax-Wendroff method. It was found that the proposed numerical method has advantages in accuracy, stability, and especially in the reduction of computational time. Tentis et al. (2003) used an adaptive method of lines to solve the one-dimensional Euler equations for pipe flow simulation. Fast transients in a single pipe after the instantaneous shutoff of a downstream valve and slow transients resulting from a perturbation of flow rate at the outlet of a very long pipe were both simulated. It was found that spurious oscillations at the wave front could be significantly reduced by using the proposed method. Similar studies using various numerical methods on the simulation of transient flows in a single pipe were also performed (Dorao and Fernandino, 2011; Gato and Henriques, 2005; Kessal, 2000).

In addition to the single pipe flow, the transient flow in a simple piping network after a sudden open and subsequently a sudden closure of a valve at a pipe-connection node has also been investigated extensively. Abbaspour and Chapman (2008) described an implicit finite-difference method for the solution of non-isothermal pipe flow governing equations and simulated such a transient flow problem. The identical transient problem was extended by Chaczykowski (2010), who applied the method of lines with a central difference scheme for the spatial derivatives, to further consider the effect of unsteady heat transfer. Similar computational studies involving the application of various numerical schemes in the simulation of flow transients induced by perturbations from a node in a piping network have also been conducted (Alamian et al., 2012; Behbahani-Nejad and Bagheri, 2010; Ke and Ti, 2000; Reddy et al., 2006).

In the present study, the natural gas compressing system, which is essentially a piping network in a compression station, was modeled as the integration of pipes and elements. The term “element” represents those idealized models used for the simulation of actual devices such as compressors, valves, and pipe fittings in a compression station. Therefore, an element in a piping network is used to provide relations to the gas flows in different pipes connected to it. With its peculiar characteristics, an element can exert certain effects on the gas flow network. Methods for simulating the flow in the entire piping network were developed based on a combined strategy that the flow in the pipe is numerically simulated using the finite volume method and the effects of elements in the pipe flow network were evaluated using quasi-steady methods. For the simulation of the pipe flow, a high-order upwind scheme considering the real-gas behaviors was used to solve the one-dimensional partial differential equations. The characteristic boundary conditions were employed to integrate with the quasi-steady modeling of elements. A computational program was developed based on this strategy and can be used to numerically study the dynamic behavior of a natural gas compression system during transient events such as the emergency shutdown of the compressor. Results of the simulation can be used to evaluate the performance of the surge avoidance system and aid the design of the piping network in new natural gas compression stations.

2. Governing equations for pipe flow

The governing equations, including continuity, momentum, and energy equations, for one-dimensional unsteady pipe flow considering friction and heat transfer on the pipe wall can be written into the following conservation-law form: