Annals of Nuclear Energy 78 (2015) 60–69

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Dynamic analysis of the pump system based on MOC–CFD coupled method

Shuai Yang, Xin Chen, Dazhuan Wu *, Peng Yan

Zhejiang University, Institute of Process Equipment, 38 Zheda Road, Hangzhou 310027, PR China

ARTICLE INFO

Article history:
Received 3 April 2014
Received in revised form 19 December 2014
Accepted 24 December 2014
Available online 9 January 2015

Keywords:
Transient analysis
Multi-scale coupled simulation
Dynamic characteristic
Water hammer

ABSTRACT

The dynamic characteristics of pump response to transient events were investigated by combining the Method of Characteristic (MOC) and Computational Fluid Dynamics (CFD) together. In a typical pump–pipeline–valve system, similar to the reactor system, the pump is treated as three-dimensional CFD model using Fluent code, whereas the rest is represented by one-dimensional components using MOC. A description of the coupling theory and procedure ensuring proper communication within the two codes is given. Several transient flow operations have been carried out. In the initial steady-state simulation, the coupled method could accurately find the operating condition of the pump when the valve is fully open. When the valve is closed rapidly, preliminary comparative calculations demonstrate that the coupled method is efficient in simulating the dynamic behavior of the pump and capable of getting detailed fluid field evolutions inside the pump. Deviation between the dynamic pump head and the value given by the steady-state curve at the same instantaneous flow-rate was established, and the cause of the deviation was further explained by the comparison of pump internal and external characteristics. Furthermore, it was found that the deviation grows with the severity of the transient. In addition, the effects of valve closure laws and pipe length on the pump dynamic performances were evaluated. All the results showed that MOC–CFD is an efficient and promising way for simulating the interaction between pump model and piping system.

1. Introduction

Pumps play important roles in nuclear reactor coolant systems (Gao et al., 2013). The variations of operating conditions in a pumping system will lead to transient effects due to the rapid changes in the hydraulic, mechanical or electrical systems, which are closely related with the safety and the operating stability of the whole nuclear system (Tanaka and Tsukamoto, 1991; Zhang and Cheng, 2012). Among these processes, sudden flow variations will give rise to pressure surges which travel at the speed of sound through the whole system and may result in catastrophes in extreme cases (Lohrasbi and Attarnejad, 2008). This phenomenon is also called water hammer. So it is essential to study the transient interaction between transient pipe flow and pump.

The pump head–quantity (*H*–*Q*) curve is an experimentally derived curve and can reflect the performance of the pump at different steady-state operating conditions. But the pump cannot respond quickly enough to traverse its steady-state characteristic curves when the rapid change in operating condition exceeds a certain limit (Li et al., 2010). Furthermore, many investigations concerning water hammer were done, but the dynamic interaction between water hammer and pump is almost disregarded (Ismaier and Schlücker, 2009). Due to this fact there is a need for correctly predicting the changing of flow status in pump when transient events happen.

Up to now, one dimensional MOC (Method of Characteristic) and three dimensional CFD (Computational Fluid Dynamics) analyses have been two relatively independent "modeling cultures" for analyzing the transient behavior of pumping system. MOC is commonly used in solving the hydraulic transients in pipe and giving important information at system levels because of its feasibility and advantages in modeling complex systems (Deng et al., 2013; Afshar and Rohani, 2008; Tian et al., 2008). Meniconi et al. (2011) conducted the detailed analysis of the interaction between a pressure wave-generated by the action of an end valve and an in-line valve, and the experimental data was evaluated by MOC calculation. CFD is increasingly being used to capture complex local 3D features inside the fluid machinery (Ding et al., 2011). For instance, Wu et al. (2010) predicted the characteristics of pump during the discharge valve rapid opening process, and the simulation results

* Corresponding author. Tel.: +86 13989880802.
E-mail addresses: 21028042@zju.edu.cn (S. Yang), xchen@zju.edu.cn (X. Chen), wudazhuan@zju.edu.cn (D. Wu), ypdjpn@126.com (P. Yan).

http://dx.doi.org/10.1016/j.anucene.2014.12.022
0306-4549/

Nomenclature

A	area of pipe (m^2)	Q	the pump flow-rate (m^3/s)
A_G	area of valve opening	t	time (s)
a	wave celerity (m/s)	V	the average velocity of flow (m/s)
C_1, C_2	turbulent coefficient	$\mathbf{V}$	the fluid velocity vector
C_d	the flow-rate coefficient	x	the distance along the centerline of the pipe (m)
D	the pipe diameter (mm)	Z	number of vanes
D_i	pump suction diameter (mm)	α	the angle between the horizontal and the centerline of the pipe (rad)
D_o	pump discharge diameter (mm)	ε	turbulence dissipation rate
D_p	pipe diameter (mm)	μ	the dynamic viscosity (Pa·s)
D_v	valve diameter (mm)	μ_t	turbulent viscosity (Pa·s)
$\mathbf{F}$	the external body force (N)	ρ	fluid density (kg/m^3)
f	Darcy–Weisbach friction factor	σ_k	Prandtl number for k
G_k	generation of turbulent kinetic energy	σ_ε	Prandtl number for ε
g	gravity acceleration (9.82 m/s^2)	$\boldsymbol{\tau}_{\mathbf{ij}}$	the stress tensor
H_0, H_V	the hydraulic loss through valve (m)	τ_c	valve opening
H	the pump head (m)	v	kinematic viscosity
H_1, H_2	water level (m)	$\boldsymbol{\Phi}$	the dissipation term
i	pipe section	$\nabla\cdot\mathbf{q}$	the heat loss by conduction
k	turbulent kinetic energy	Δx	the length of pipe section (m)
L	pipe length (m)	ΔH	the hydraulic loss through valve (m)
n	pump rotational speed (rpm)	Δt	time step size (s)
n_s	pump specific speed		
p	the static pressure (Pa)		
Q_0, Q_V	flow-rate through the valve (m^3/s)		

agree well with the test data, which validated the CFD method in simulating the transients in hydraulic machinery.

In general it is impractical to simulate a whole system with a pure 3D CFD model considering computational resource and time, and consequently separate analysis of individual components within the system, such as a pump or a pipe, is considered in isolation. What is more, separate analysis simplifies the modeling process but gives rise to uncertainty when specifying the values for boundary conditions at the component inlets/outlets. However, the 1D MOC is not mature enough to be applied in the calculation of the hydrodynamic characteristics of a pump under transient operating conditions for two reasons. On one hand, it cannot provide the detailed information, like flow field structure and pressure distribution, inside the fluid equipment. On the other hand, the pump dynamic characteristics, simulated solely based on single MOC, is a quasi-steady result due to replacing the pump model with steady-state performance curve and its deviation from the actual results depends on the severity of the transient (Al-Khomairi, 2003). Therefore, the two simulations methods are to be seen as complementary, and a multi-scale modeling approach, namely MOC–CFD, can be proposed by combining both models together to utilize the strengths of both approaches.

Earlier research on multi-scale approach had been reported in many fields. Ljubijankic et al. (2011) carried out numerical coupling, based on DAE (Differential Algebraic Equation) and CFD, for building energy supply systems, and got the temperature of water in storage tank changes over time. Bertolotto et al. (2009) conducted single-phase mixing studies by means of the coupling between the commercial CFX code and the system code TRACE, and the experimental data validated the effectiveness of the co-simulation. Other studies on the automobile engine system (Claywell et al., 2006; Bohbot et al., 2008) were also conducted based on the concept of coupling.

In this paper, MOC–CFD coupled method was developed to study the pump dynamic characteristics. In a typical pump–pipeline–valve system, the valve is closed rapidly to achieve a rapid change in operating conditions. And a series of influence factors, such as simulation methods, the control of transient process and the geometric size of system, were considered and put into practice. The effectiveness and accuracy of coupled simulation method were discussed later through simulated results in this paper.

2. MOC–CFD coupling methodology

2.1. MOC calculation

The Method of Characteristic (MOC) is a mathematical technique for solving one-dimensional transient flow in pipe, which can be explained through the time–space grid shown in Fig. 1. At any interior grid inter section point (e.g., point P_3 at section i), the two compatibility equations are given by the following (Wylie and Streeter, 1978):

$$C^+ : H_{Pi} = C_P - BQ_{Pi} \quad (1)$$

$$C^- : H_{Pi} = C_M + BQ_{Pi} \quad (2)$$

where parameters C_P and C_M are functions of the flow-rate and head at a previous time step at nodes P_1 and P_2.

$$C_P = H_{i-1} + BQ_{i-1} - RQ_{i-1}|Q_{i-1}| \quad (3)$$

$$C_M = H_{i+1} - BQ_{i+1} + RQ_{i+1}|Q_{i+1}| \quad (4)$$

where B and R are system parameters which are functions of the material and the structure of pipe systems.

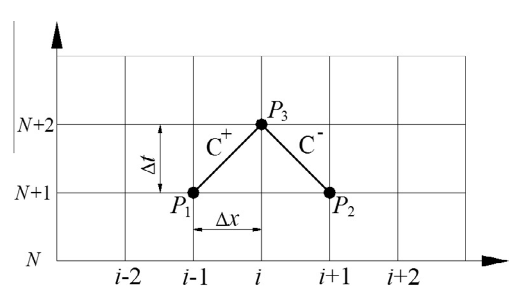

Fig. 1. Computational time–space grid.