Numerical simulation of pipeline ultrasonic cavitation based on CFD

Zhihua Wang [1,a], Hehui Wang[1,b], Chaoping Wang[2,c], Chunhui Fu[2,d], Pangu Lu[2,e]

[1] School of Mechanical and Power Engineering, East China University of Science and technology, Shanghai, 200237, China;

[2]Shanghai Gaoqiao Branch, Sinopec, Shanghai, 200137, China.

[a]13916787756@163.com, [b]hhwang@ecust.edu.cn, [c]wangchaoping@sinogpc.com, [d]accdpg_1978 @163.com, [e]lupangu@sinogpc.com

Keywords: ultrasonic cavitation; descaling; CFD; Fluent.

Abstract. Cavitation effect is one of the main mechanisms of ultrasonic descaling. Through the mixture model and the cavitation model in Fluent, the ultrasonic cavitation field in the pipe is simulated. The simulated distributions of pressure field and variation of vapor are analyzed and discussed. This study shows that the CFD simulation is available to be used to predict the ultrasonic cavitation and it will be helpful for research of ultrasonic cavitation.

Introduction

During the process of industrial production, the phenomenon of equipment or pipes scaling due to the metal corrosion, inorganic crystal, deposition of solid particles and other causes is very common. The generation and accumulation of dirt may lead to pipe blockage, increasing the surface roughness of pipes or equipment corrosion[1]. To solve these problems, many measures have been taken, mainly including chemical cleaning and mechanical cleaning. Chemical cleaning consumes large amounts of chemicals and brings in pollution[2]; mechanical cleaning needs to disassemble the equipment and easily cause damage[3]. These shortcomings make their application has been greatly limited.

In recent years, several new techniques on scale cleaning have been developed, ultrasonic cleaning is one of the most common cleaning technology and it has advantages of cleaning scale without stopping running equipment, working online continuously. Ultrasonic descaling is mainly based on four mechanisms: cavitation effect, activation effect, shearing effect, inhibitory effect[4], where cavitation effect plays a major role. Ultrasonic cavitation is a complex physical process caused by power ultrasound in the liquid medium and the tiny bubbles in the liquid vibrate under the influence of the sound field. When the sound pressure reaches a certain value, bubbles will expand rapidly, close, produce shock waves and collapse finally[5]. Then these bubbles which have been crushed would produce a strong hydraulic shock wave to scour the dirt layer on the solid surface and make it fall off. This paper is to research the cavitation field inside one section of pipe under the effect of ultrasonic cleaning device. The numerical analysis method is applied to simulate the formation and distribution of ultrasound cavitation effect by Fluent software.

Simulation of mathematical models

Cavitation effect is due to the local low pressure (below the corresponding temperature of saturated vapor pressure) in the liquid and micro bubble growth and collapse caused by the vaporization of liquid. There are continuous liquid phase and gas phase in the liquid and the multiphase flow model is adopted in the simulation. In order to simulate the cavitation effect, the mixture model for two-phase flows, cavitation model and standard $k-\varepsilon$ turbulence model were used in this paper. The mixture model can simulate the n phase (fluid or particle) flow by solving the continuity, momentum and energy equations of the mixed phase, the volume fraction equation of the second phase and the relative velocity[6]. In the cavitation model, the spatial and temporal variations of a single bubble volume can be expressed as follows:

$$\varphi(r,t)=\frac{4}{3}\pi R^3 \quad (1)$$

where R is bubbles' radius.

The volume fraction of evaporation is defined as:

$$\alpha_v = \frac{\varphi\eta}{1+\varphi\eta} \tag{2}$$

where η is the number of bubbles within the fluid volume unit.

The volume fraction equation is obtained from the continuity equation of mixed two-phase flow (m). After processing, assumed that the liquid (l) is incompressible, an equation can be obtained as follows:

$$\frac{\partial}{\partial t}\left(\alpha_p\right)+\nabla\left(\alpha_p v_m\right)=\frac{\rho_l}{\rho_m}\frac{\eta}{\left(1+\varphi\eta\right)^2}\frac{d\varphi}{dt}+\frac{\alpha\rho_v}{\rho_m}\frac{d\rho_v}{dt} \tag{3}$$

According to the bubble dynamics, the cavitation bubbles will form liquid as a result of low temperature, fluent isothermal simulation cavitation flow, neglecting the latent heat of evaporation. The Rayleigh-Plesset equation is related to the pressure and volume of bubbles.

$$R\frac{d^2R}{dt^2}+\frac{3}{2}\left(\frac{dR}{dt}\right)^2=\frac{p_B-p}{\rho_l}-\frac{2\sigma}{\rho_l R}-4\frac{\mu_l}{\rho_l R}\frac{dR}{dt} \tag{4}$$

Where the pressure inside the bubble p_B, described by the sum of the partial pressure of the steam (p_v) and the partial pressure of the non condensing gas (p). σ is the coefficient of surface tension.

$$\frac{dR}{dt}=\sqrt{\frac{2\left(p_B-p\right)}{3\rho_l}}\text{ , } p_v > p \tag{5}$$

$$\frac{dR}{dt}=-\sqrt{\frac{2\left(p_B-p\right)}{3\rho_l}}\text{ , } p_v < p \tag{6}$$

Establishment of the geometrical model

Grid division and boundary condition setting.

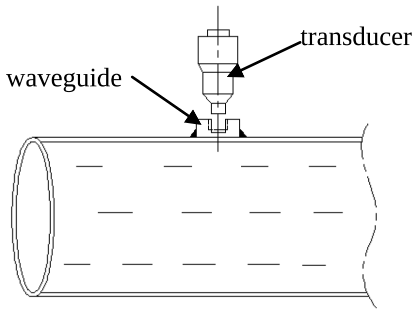


Fig.1 Schematic of ultrasonic cleaning device arranged on the pipe

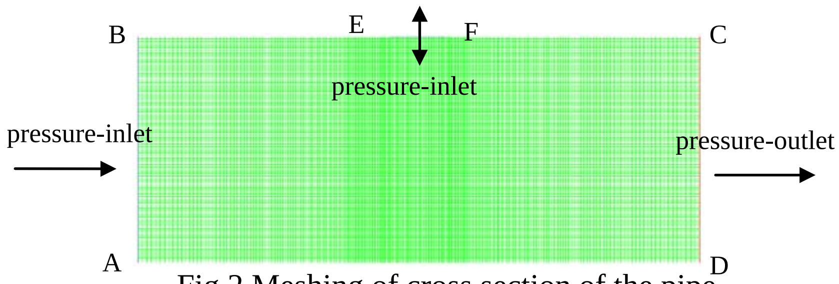


Fig.2 Meshing of cross section of the pipe

Fig. 1 shows the ultrasonic cleaning device which welded on the outer wall of the pipe. The waveguide is welded on the outer wall of the pipe and the transducer connects to the host. Fig.2 shows the meshing of cross section of the pipe by using Icem, the Fluent pre-processing software. Considering the time cost and accuracy of the calculation, structured grid is applied in this paper and the total number of grids is 23463. Due to the problem of ultrasonic vibration, the transducer and pipeline connection (EF) is set to the pressure inlet and the user defined function (UDF) is used to specify the magnitude and frequency of the pressure action. The pressure at the boundary (EF) is expressed by the equation $P = P_A sin(2\pi ft)$. The ultrasonic wave used for simulation has an amplitude, $P_A = 2MPa$, and a ultrasonic frequency, $f = 20kHz$. Fig.3 shows the variation of absolute pressure at the boundary (EF) with time. Table 1 shows the length of each side, the name and type of all boundaries. Standard wall function is used in the near wall condition.

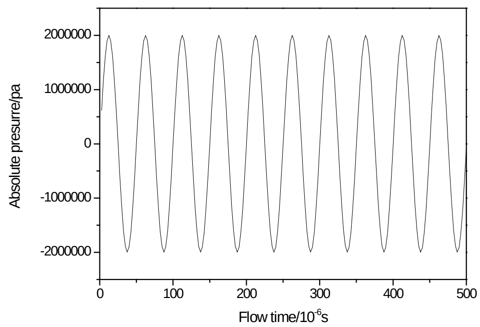


Fig.3 Variation of absolute pressure at pre-inlet (EF) with time

Table.1 Calculation model and boundary setting

Boundary line	length/mm	Boundary type
AB	400	pressure-inlet
BE	450	wall
EF	100	pressure-inlet
FC	450	wall
CD	400	pressure-outlet
DA	1000	wall

Calculation conditions. For ease of calculation, the following hypotheses are proposed in the present paper: (a) the entire simulation system is adiabatic and no heat convection is considered; (b) the medium in the pipeline is incompressible fluid. (c) the jet stream generated by the collapse of cavitation bubbles is ignored. In the present paper, diffusion terms in the governing equations are discretized by the central difference scheme. Central difference scheme is used for the diffusion of the control equation in this paper. During the process of simulation in Fluent, pressure interpolation method adopts PRESTO! and pressure-velocity coupling algorithm adopts PISO method.

Simulation results and discussion

In this paper, the problem of ultrasonic cavitation is different from that of other types in the past. The flow field generated by ultrasonic cavitation is unsteady flow field. Because the ultrasonic cavitation field is unsteady flow field, the pressure, velocity and other parameters in the flow field will change with time and the cavitation region will also change with time. In order to accurately simulate the change of flow field, the small time step, 2.5μs (1/20 of a vibration period) , is adopted in the calculation process.

When the absolute pressure is lower than the saturated vapor pressure of water, the vapor in water will dissolve and cavitation effect will occur at the same time. As can be seen from the Fig.4, the vapor is mainly concentrated in the middle of EF, where is also the main area of cavitation. The absolute pressure in this part depress to the saturated vapor pressure (3540Pa). Fig.5 shows the distribution of absolute pressure and velocity in the pipe for different iteration steps.

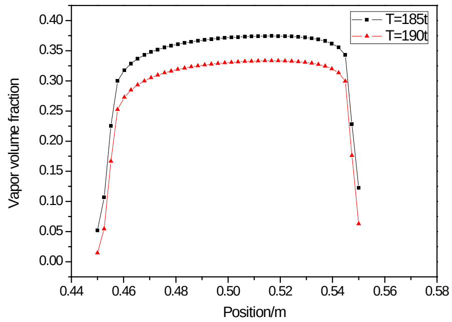
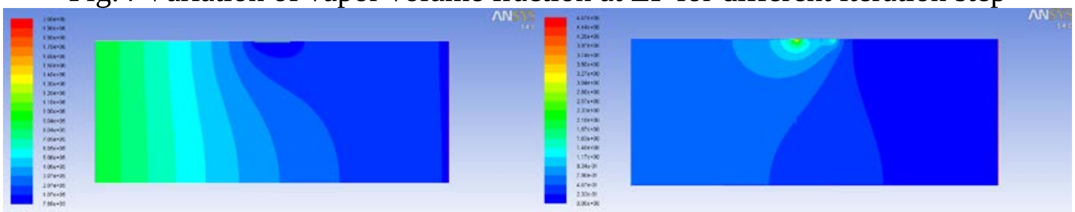
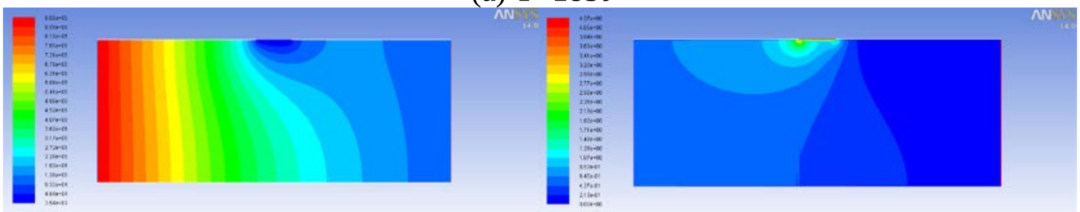


Fig.4 Variation of vapor volume fraction at EF for different iteration step



(a) T=185t



(b) T=190t

Fig.5 Contours of pressure and velocity in the pipe for different iteration step

Summary

In this paper, the CFD method is used to simulate the distribution of the cavitation field in the pipeline. The pressure and velocity distributions are obtained to describe the distribution of the cavitation field. By using UDF programming, the variation of the pressure at the position where the transducer and the pipe connected is applied into this model. This solving method for unsteady problems is helpful and enlightening for researching more complex cavitation problems in the future.

Reference

[1] Li Peifeng. Research and application of descaling methods for long-distance slurry pipeline. Metal materials and metallurgy engineering. 42(2014)29-32.

[2] Huo Wenlan, Wen Junfeng. Application of ultrasonic assist on carry oil pipeline removal and prevention of dirt. Applied chemical industry. 38(2009)916-918.

[3] Ni Jianping. Tubular heat exchanger fouling and cleaning method. Chemical industry in Zhejiang. 38(2007)19-20.

[4] Zhou Aidong, Yang Hongxiao, Zhang Zhibing. Application of ultrasonic in anti-scaling. Applied chemical industry. 34(2005)659-661.

[5] Sun Dapeng. Progress in research on ultrasonic boiler circulating water descaling. Environmental protection and circular economy. (2014)39-42.

[6] Lv Ting, Li Yi, Tao Bolin. Simulation of ultrasonic cavitation characteristics in mold cavity. Tool technology. 48(2014)33-36.