IOP Conf. Series: Materials Science and Engineering **129** (2016) 012043 doi:10.1088/1757-899X/129/1/012043

Suitability research on the cavitation model and numerical simulation of the unsteady pulsed cavitation jet flow

S Y Chen[1], X F Yu[1], D Y Luan[2], Y P Qu[1] and C Zhou[3]

[1] Key Laboratory of High-efficiency and Clean Mechanical Manufacture, School of Mechanical Engineering, Shandong University, Jinan, Shandong 250062, P. R. China

[2] School of Mechanical and Electrical Engineering, Qingdao University of Science and Technology, Qingdao, Shandong 266061, P. R. China

[3] Qingdao Special Equipment Inspection and Testing Institute, Qingdao, Shandong 266071, P. R. China

E-mail: chensy66@sdu.edu.cn

Abstract. In order to explore the cavitation jet mechanism, it can first study its critical state of single-phase flow before cavity occurrence to explore the trend of pulsed cavitation jet. Then select the cavitation model to simulate the complex multiphase flow state. Such a step-by-step approach is beneficial to advance research reliably and steady, relying on the foundation for further solving the problem.

Three turbulence models such as Euler Hybrid Model, Euler Two Phase Model and Euler Lagrange Model are discussed on their suitability. In this paper, it states only RNG k- ε turbulent model can simulate small scale vortex of jet in the transient simulation. Grid independent verification and the effect of time step is presented. The simulation results show that a large scale vortex ring surrounding jet flow in the nozzle, the pressure of vortex core is slightly lower than the upstream nozzle pressure.

Considering the capture ability of small scale eddies, an equivalent pressure is established. The single-phase flow turbulence model is modified to simulate the turbulence flow in the self-excited pulsed cavitation after the cavitation occurs. Through different results comparison of not modified cavitation model and the modified cavitation model to the experimental results, it proves that the latter simulation results are relatively accurate.

1. Introduction

Lots of invisible gaseous kernels will grow up and become macroscopic visible bubbles when the local pressure of liquid is lower than its saturated vapor pressure. If the liquid with bubbles pass through a relative high pressure area, the bubbles will break down under the action of pressure and change into microscopic size of the invisible kernel. The phenomenon of partial vaporization in the liquid is called the cavitation. There are some important factors that influence cavitation occurrence and development such as the size and quantity of the gaseous kernel, the amount of undissolved gas contents, the relative velocity between gas and liquid, the pressure distribution, turbulence intensity and velocity gradient[1-3].

First, the greater the size of the gaseous kernel, the shorter time it takes from generation to visible size. According to the stability theory of gas nucleus, when the gas nucleus size is greater than the critical size, gas nuclei will unlimited expansion just below the saturated vapor pressure, the cavitation will occur. When the pressure is lower than the saturated vapor pressure, gas nucleus can be in a stable condition under a corresponding tensile stress state, the cavitation will be suppressed. Second, the greater the number of gas nucleus contained in liquid, cavitation phenomenon will be more obvious. Again, gas nucleus in the flow field will affected by drag force, Saffman force and viscous drag force, etc. The power of the force are related to the size of the gas nucleus. In some cases, the large size of gas nucleus can't get through the low pressure area under the action of those forces[4].

Chahine and Hsiao[5] deduced bubble dynamics equation under the condition of the relative velocity between the gas nucleus and water. They found that the effects of bubble expansion and compression caused by the relative speed between gas nuclear and water is equivalent to added equivalent pressure to the saturated vapor pressure. They got the modified Rayleigh-Plesset equation.

Cavitation occurs is not sudden, it need to experience a period of time. Gas nucleus needed to experience a period of time in the low pressure area before developed into a critical radius and generate cavitation .So, it needed a scope of low pressure area in the flow field before the formation of cavitation. If the scope of low pressure area is too small even it lower than the saturated vapor pressure also cannot generate cavitation. Usually backflow is easy to form on the surface of the object in the low pressure area. Some gas nucleus with backflow will return back. This will prolong the time of gas nucleus experience, the cavitation will occur. In addition, when the gas nucleus with liquid entered the stagnation point of near the surface of the object, the gas nucleus will be deviated from the original streamline under the action of large fluid acceleration and pressure gradient. And the large gas nucleus are more likely to deviating from the original streamline. Johnson[6] called it filtering effect. Recent studies have shown the importance of filtering effect in the aspects of cavitation inception.

2. Analysis of cavitation model

There is a big challenge to establish the physical model and the steady numerical method for the numerical simulation of the cavitation flow. Delannoy and Kueny[7] pioneered to use single-phase mixing model which combines with the experience of positive pressure model to simulate the cavitation. This model can calculate the density by pressure and velocity and only get qualitative consistent with experimental results in cavitation number. In the positive pressure model which was established by Schmidt，Rutland and Corradini[8], cavitation was simulated by a compressible water - steam mixing method which the sound velocity based on homogeneous equilibrium flow model (HEM). While considering the compressibility to improve the accuracy of the simulation, but it not used for occasions of small pressure gradient and great density change, and this model didn't consider the effect of turbulent on the cavitation. Dumont, Simonin and Habchi[9] expanded this model to three dimensional problems. Marcer and LeGouez[10] proposed a cavitation model which applied to the simulation of the diesel nozzle. In this model, the volume of fluid method (VOF) was modified and combined with a mass transfer model which was driven by energy. The basic hypothesis of this model is that the cavitation area can use a large-scale interface to approximate, this will exclude the possibility of some small bubbles seeing in the experiments which dispersed in the liquid. This model also took the compressibility into account and ignored the effect of turbulent on the cavitation. Kubota, Kato and Yamaguchi[11] proposed a model which assumed that the cavitation is caused by grew up of tiny gas nuclear which existed in the liquid. The pure liquid and pure steam is treated as incompressible medium, and the overall density of cavitation area determined by the content of steam. All above models are roughly divided into two kinds of model. The first model based on Euler method. The second based on the Euler - Lagrange method.

2.1 Euler - hybrid model

Hybrid Singhal model[12] assumed that the presence of large amounts of gas nucleus in water, gas nuclear size and quantity is a constant value, there is no relative velocity between gas and water. By raising the threshold pressure to control the cavitation production. This model takes the turbulent, not dissolved gas into account on the influence of cavitation, but it also ignored many factors that can affect the cavitation, such as the nuclear spectrum of gas nuclear, the relative velocity between liquid and gas nuclear, the viscosity of liquid, surface tension and velocity gradient, etc. But, when this simplification simulated the problem of the rapid flow of the steady-state cavitation, the accuracy of simulation was also acceptable.

2.2 Euler two-fluid model

Advantages of Euler two-fluid model compared with the hybrid model is solving continuity equation and momentum equation of liquid and gas phases, and can also set turbulent models for the liquid and the gas phases respectively, we can improve the accuracy of simulation, this will certainly increase the computer memory and calculation time for the simulation. Two kinds of cavitation model based on Euler two fluid models are Zwart-Gerber-Belamri[13] model and Schnerr-Sauer[14] model.

2.3 Eulerian - Lagrangian method

Liquid phase is treated as continuous phase which solved by N-S equation. Bubbles are treated as discrete phase which dispersed in the continuous phase, getting its position at a certain moment in the continuous phase through integrating the force equilibrium equation in the flow field of continuous phase, so its trajectory is obtained. It also can achieve the coupling transfer of momentum, mass and energy between discrete phase and continuous phase. After analysis of the essence of the cavitation occurs, we can find that the Eulerian-Lagrangian method compared with the former two models can more precisely describe the process of cavitation happening. Because the Eulerian-Lagrangian method can research the state of a single bubble in the flow field, the individual characteristics of many bubbles will be considered, such as the size and number of bubbles and the trajectory, this will be conducive to improve the accuracy of cavitation simulation.

3. Modification of the Euler model

Based on the analysis of factors which have effect on cavitation, using the UDF function in Fluent, we can make appropriate modifications to above three models as following.

Considering the previous analysis of turbulent influence on cavitation, in order to measure the effect of turbulent on cavitation, the final threshold pressure P_v is:

$$P_v = P_{sat} + \frac{1}{2}\left(\frac{C_1}{n\Delta t} + C_2\right)\rho k + \frac{\rho U^2}{4} \tag{1}$$

Where P_v is the vapor pressure, P_{sat} is the saturated vapor pressure, ρ is the density of liquid, k is the turbulent kinetic energy, n is the number of per unit area grids. ∇t is the time step, C_1 and C_2 are coefficient. Some effects of cavitation from turbulent is due to the relatively large structure vortex. In the numerical simulation, the capture precision of vortex in the turbulent depends on the number of grids and the time interval. In other words, the capture of vortex is different. Different capture precision of vortex will directly affect vortex impact on cavitation, so add $C_1/(n \nabla t)$ in Eq. (1). In some occasions, some small vortex which can't be simulated by the numerical simulation, people used to think that as a kind of nonlinear pressure pulsation. So add C_2 to Eq. (1) in order to consider the effect of simulating cavitation caused by pressure fluctuation.

4. The numerical simulation of the unsteady flow field of cavitation jet

In this article, we first study its critical state of single-phase flow before pulsed cavitation occurring, then select the cavitation model to simulate the complicated multiphase flow state. Such a step-by-step approach is beneficial to advance research reliably and steady, laying the foundation for further to solve the problem.

4.1 Geometric modelling, time step, mesh generation and boundary condition

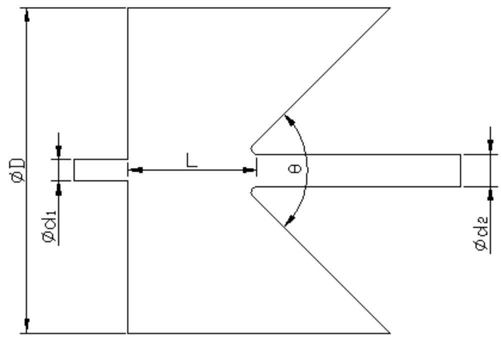


Fig. 1. The geometric model of nozzle.

In Fig 1, there are five major parameters of nozzle structure such as the upper nozzle diameter d_1=6mm, the cavity length L=45mm and the cavity diameter D=78mm, the collision angle θ=120 and the down nozzle diameter d_2=11mm. The cavity is simplified as a two-dimensional axisymmetric model which is divided into five parts in Fig 2: The upper and down nozzle parts respectively uses fined grids of quadrilateral structure. The middle cavity is divided into three parts and they all adopt quadrilateral grids.

In order to verify the effect of grid independence and the time step, simulations are carried on the same geometric model with different mesh density and time step, the specific settings are shown in table 1.

Due to the axial symmetry, we take its half as the simulation area as shown in figure 2. The boundary conditions of inlet $\overline{\mathrm{AB}}$ is set as pressure inlet and the value is 0.2 MPa, $\overline{\mathrm{CD}}$ is

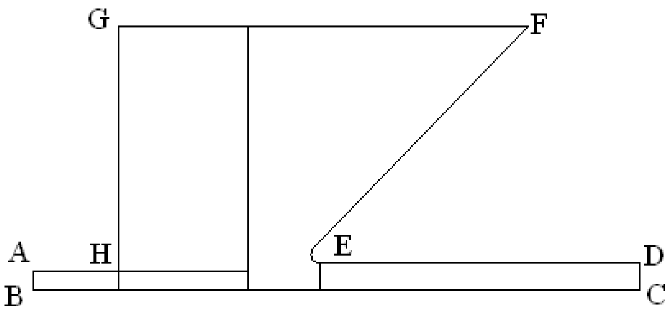


Fig. 2. The partitioning of nozzle grid area

Table 1. The setting of the grid number and the time step

The number of grid cells *n*	Time step $\triangle t$
108087	infinite（steady state）
182545	0.00005
270945	0.000005

pressure outlet and set the value as 101325 Pa. Others are defined as the solid wall. We use turbulence model of RNG k-ε, simulations are performed on steady and transient state with grids number 182545, the time step 0.00005 s and 0.000005 s, respectively.

Table 2. The comparison of different time step

Time step （s）	Vortex core pressure（Pa）	Inlet pressure （Pa）	Inlet velocity （m/s）
0.00005	32000	39200	18.0
0.000005	27200	39000	18.0

Under steady state, the residual fluctuates greatly and does not reach a stable solution. Under transient simulation with time step 0.00005s, within a time step, residual fluctuates significantly, but eventually gets a relatively stable solution. Meanwhile with 0.000005s, the convergence speed of residual is so quickly. Vortex cores appears in both transient states, the vortex intensity is different. As shown in table 2, the inlet pressure and velocity are basically the same, but there exists difference of vortex core pressure. The transient flow simulated with 0.000005 s appears obvious vortex structures in the shear layer than 0.00005s. Besides, the small scale vortex core pressure is low, this will have a great influence on cavitation inception.

Performing transient numerical simulation respectively on three different number of mesh grids with time step 0.000005s. Convergence of three grids is good, and they all can converge as soon as possible. With the increase of grid number, the vortex core pressure and inlet pressure increase, and the inlet velocity is reduced. The results of the grid number 182545 have been closed to the number

270945, we think the grid number 182545 state has certain reliability.

For the transient numerical simulation with the time step 0.000005s, the RNG k-ε ,the realizable k-ε , the standard k-ω and SST k-ω turbulence model is employed, respectively. From the simulation, we know that small scale eddies can be captured by RNG k-ε model except other three models.

4.2 The numerical simulation after cavitation

In the FBM[6] model, the turbulent viscosity μ_t [14]can be written as:

$$\mu_t = C_\mu \frac{\rho_m k^2}{\varepsilon} \cdot f_{FBM}, \quad f_{FBM} = \min\left[1, C_3 \frac{\Delta \cdot \varepsilon}{k^{3/2}}\right] \tag{2}$$

Where C_μ = 0.85, σ_k and σ_ε is Prandtl number 0.7179, G_k is the generation term of turbulent kinetic energy, f_{FBM} is filtering function which was determined by the ratio of the filter scale to turbulence scale. Considering the vapor-liquid density was influenced by the turbulent viscosity, the filtering function is modified as follows:

$$f_{FBM} = \min\left[f(\rho), C_3 \frac{\Delta \cdot \varepsilon}{k^{3/2}}\right], \; f(\rho) = \frac{\rho_v + \alpha_l{}^n(\rho_l - \rho_v)}{\rho_v + \alpha_l(\rho_l - \rho_v)} \tag{3}$$

Where C_3 = 1.0， n =10.

The choice of filter scale is particularly important for the smooth realization of the filtering process. Johansen[15] suggested the filtering scale Δ was,

$$\Delta = \max\left(\Delta_{select}, \Delta_{grid}\right), \; \Delta_{grid} = \left(\Delta x \cdot \Delta y \cdot \Delta z\right)^{1/3} \tag{4}$$

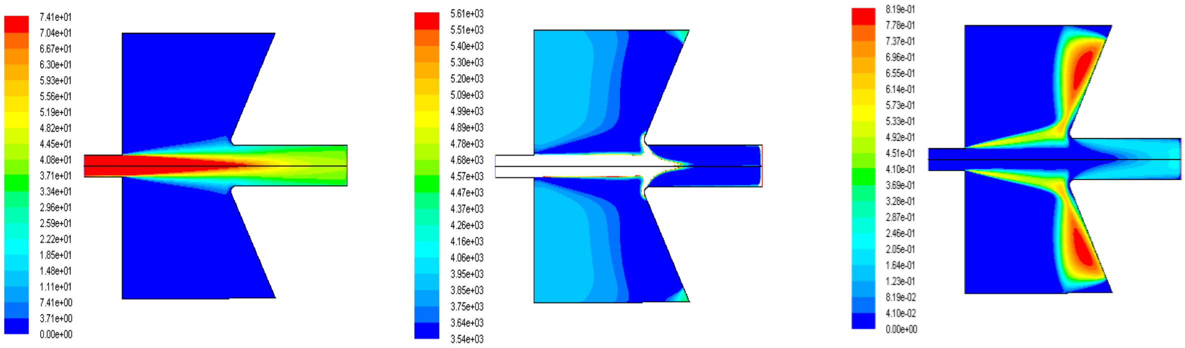
Where △x、△y and △z are the grid space The filtering scale should be larger than grid scale in computational area. We carried on UDF programming on the above modified turbulent model.

We perform simulation on the internal flow field of low pressure self-excited pulsed cavitation jet nozzle with the inlet total pressure is 2MPa. The absolute pressure inside the cavity is below 0.058MPa which was measured by pressure sensor.

In Fig. 3, 4 and 5, we performed the numerical simulation on the nozzle internal flow field by using Singhal and the single-phase RNG k-ε model, Singhal and the modified multiphase RNG k-ε model, modified Singhal and the modified multiphase RNG k-ε model, velocity, pressure and the volume fraction contours of water vapor in a period of pulse were shown, respectively.

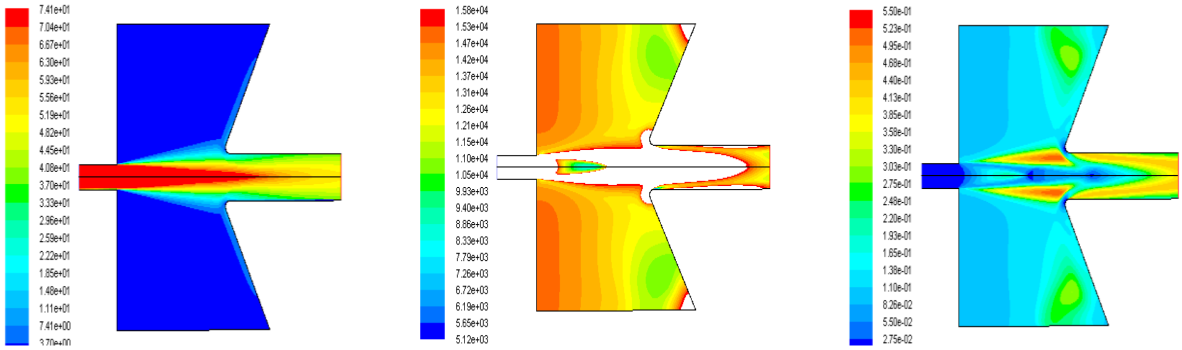
From figure 3, the flow contours seems more reasonable, bubbles distribution in the nozzle cavity is more uniform, gas-liquid mixing area is not visible. From figure 4, The volume fraction of gas in the cavity shear layer center is lower compared with single-phase RNG k-ε turbulence model. This should be due to the modified multiphase RNG k-ε model reduces the turbulence intensity so that the threshold pressure of cavitation inception is reduced too.

From figure 5, the volume fraction near the cavity center line increases, this should be caused by the modified Singhal model which considers the effect of large velocity gradient and the



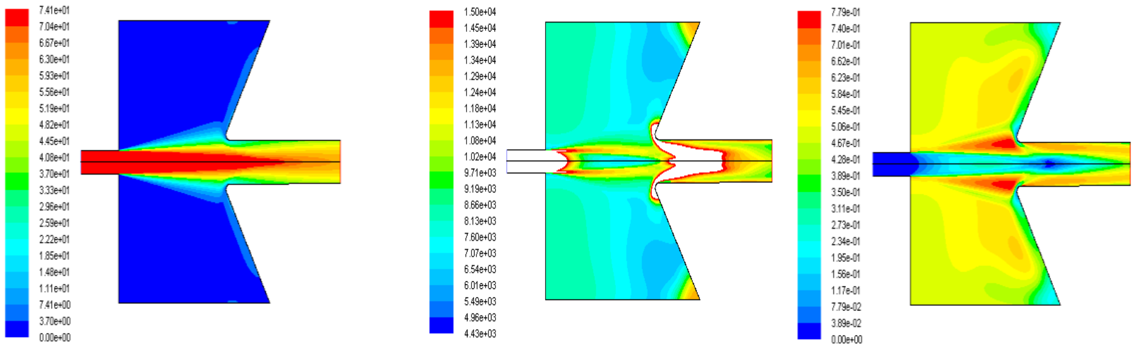
a) Velocity vector b) Pressure contours c) Volume fraction contours of vapor

Fig. 3 The flow field contours simulated by Singhal-single phase RNG k-ε model



a) Velocity vector b) Pressure contours c) Volume fraction of vapor

Fig. 4 The flow field contours simulated by Singhal- modified multiphase RNG k-ε model



a) Velocity vector b) Pressure contours c) Volume fraction of vapor

Fig. 5 The flow field contours simulated by modified Singhal- modified multiphase RNG k-ε model

relative velocity between bubbles and water. The simulation results of modified Singhal model is close to the experimental data, gas content and chamber pressure value is also closes to the experimental results. Therefore, the modified Singhal model has good accuracy in the simulation of internal cavitation jet flow field.

5. Conclusions

Conclusions are drawn as following.

(1) Suitability of cavitation model such as Euler Hybrid Model, Euler Two Phase Model and Euler

Lagrange Model is discussed. Those models do not consider the effect of the breakage and merger of bubble on inception, development and breaking of cavitation. Turbulence intensity, larger velocity gradient in cavitation chamber, relatively velocity between bubble and liquid are merged to modify the Euler cavitation model.

(2) Through the numerical simulation on the self-excited pulsed cavitation jet. we find that only RNG k-ε turbulence model can capture the small scale eddies of jet by establishing an equivalent pressure in the transient condition. In the cavity chamber, large scale vortex ring around the jet center line is produced, the pressure in vortex core is slightly lower than that in the upper nozzle.

(3) The original Singhal model and modified one, single-phase flow turbulence model and modified multiphase model are combined to simulate the turbulence flow in the self-excited pulsed cavitation after the cavitation occurs. Through different simulation comparison to the experimental results, it proves that the simulation results by modified Singhal model are relatively accurate.

Acknowledgement

The authors acknowledge the National Natural Science Foundation of China (51176102). Deyu also thanks the support of Shandong Provincial Science and Technology Development planning Program (2013YD09007) and Shandong Provincial Natural Science Foundation (ZR2014EEM017).

Reference

[1] Huang J T, Chen J F, Ding T et al. 1996 *Journal of Hydraulic Engineering* **12**: 1-7.
[2] Huang J T, Li Z M 1987 *Journal of Hydraulic Engineering*. **8**: 44-48.
[3] Huang J T, Tian L Y 1988 *Journal of Hydraulic Engineering*. **12**: 41-45.
[4] Christopher E B 1995 (London, Oxford university press) 54-55.
[5] Chao-Tsung, H Georges, L Chahine, et al. 2000 *Fluid Mech*, **11**:123-135.
[6] Johnsson C A. *Proc 12th Int. Towing Tank Conf*, Rome, 381-392.
[7] Delannoy J K. 1980 *Journal of computational physics*, **35**: 229-253.
[8] Avva R K, Singhal A, Gibson D H 1995 *Journal of Fluids Engineering*, **24**:134-143.
[9] Schmidt D P, Corradini M L 2001 *International journal of engine research*, **2**:1-22.
[10] Dumont N, Simonin O, Habchi C 2001 *Fourth international symposium on cavitation*(California) 7.
[11] Marcer R, Le Cottier P, Chaves H, et al.2000 *Journal of engines*, **10**:23-35.
[12] Kubota A, Kato H, Yamaguchi H. 1992 *Journal of fluid mechanism*. **240**:59-96.
[13] Singhal A K, Li H Y, Athavale M M, et al..2001 *ASME fedsm01*.(New Orleans)
[14] Zwart P J, Godin P G, et al.2004 *Fifth international conference on multiphase flow* (Yokohama)
[15] Johansen H, WU J, Shyy W 2004 *International Journal of Heat and Fluid Flow*, **25(1)**:10-21.