

Near-Field Numerical Simulation of Impulse Anti-Riots Water Cannon

Renjun Zhan¹, Hongwei Zhuang¹, Fadong Zhao², Feng Xue³

 ¹Equipment and Transportation Department of Engineering College of CAPF, Xi'an, China
 ²Post-graduate Department of Engineering College of CAPF, Xi'an, China
 ³School of Energy and Power Engineering of Xi'an JiaoTong University, Xi'an, China Email: zhaofd00@163.com

Abstract: One-Equation Large Eddy Simulation(LES) model coupled with the bounded and compressed VOF method was used to simulate the near-field of impulse anti-riots water cannon for the study of its atomization mechanism. The instantaneous isosurfaces of liquid-phase volume fraction and the high-speed photographs were compared. They are agreed with each other in the spray configuration. The spray process is divided into three phases: liquid column burst, secondary burst, liquid film rupture. The inflation of the highpressure gas is considered to be the most dominant factor in the near-field.

Keywords: impulse anti-riots water cannon; LES model; bounded and compressed VOF method; numerical simulation

1 Introduction

Impulse anti-riots water cannon is a new-style counter personnel non-lethal weapon for crowd control. It uses the high-pressure constringent gas to drive the mixture of water and irritants directly into the air and become an irritant water mist agglomerate to dispel the riotous crowds in long distance. Comparing with the extensively used water cannons which are designed to use high pressure water to deter people in situation of public disorder^[1], the consumption of water and irritants is reduced and its operation ability and effect is strengthened greatly.

The atomizing water jet is used to fulfill its operational purpose for impulse anti-riots water cannon, so it is necessary to study the atomization mechanism for further improvement of its performance. With the development of modern computers, CFD(Computational Fluid Dynamics) combined with experiments has been a useful tool to study liquid-gas flow^[2-4]. This study uses the method proposed by Federico^[3] to simulate the external near field where the initial boundary conditions is provided by the internal flow. One-Equation Large Eddy Simulation coupled with the bounded and compressed VOF method is used as numerical model. The simulation results and high-speed photographs are compared and the atomization process of impulse water jet is analyzed at last.

2 Operational principle

Impulse anti-riots water cannon consists of launch tube, gas-support and water-support equipment. Launch tube consists of gas chamber, water chamber, electromagnetic valve and nozzle. Its configuration diagram is showed in Fig.1. Its operational principle is that the water mixed with irritants in the water chamber which is driven by compressed gas up to 2.5MPa spout the nozzle at a high speed, the water column breaks up, atomizes and diffuses when fly in the air, becomes an irritant mist agglomerate finally to dispel the riotous crowds in long distance. Then the high-pressure gas and water is filled into the gas chamber and the water chamber separately. After the trigger is pulled again, an operational circle is finished. During the spurt process, the water jet is nonsteady flow.



valve Figure 1. Impulse anti-riots water cannon schematic diagram

3 Governing equations

The spurt process of impulse anti-riots water cannon is a gas-liquid dynamics process in which the water column is driven by high-pressure compressed gas. The numerical model represents the simultaneous unsteady flow of two compressible, isothermal immiscible fluids.

3.1 Momentum and Continuity Equations

The mass continuity equation:

Sponsored by National Natural Scientific Fund of China(Grant No. 50876113)

The 2010 International Conference on Information, Electronic and Computer Science



$$\nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \tau + \rho f + \int_{S(t)} \sigma \kappa' n' \delta(x - x') dS$$
(2)

where t is the time, u is the velocity, τ is the viscous stress tensor , σ is the surface tension coefficient, κ is the curvature of the liquid surface , n represents a unit vector normal to the liquid surface, and f stands for the acceleration due to body force. In this study, the only body force acting on the system is gravity, i.e. f=g. The last term on the right hand side of Eq.(2) represents the source of momentum due to surface tension. It acts only at the interface over the entire surface described by S(t). The fluids investigated in this study obey the Newtonian law of viscosity. Hence, the viscous stress is given by

$$\tau = \mu(\nabla \mathbf{u} + \nabla \mathbf{u}_t) \tag{3}$$

where μ is the kinematic viscosity.

3.2 Bounded and Compressed VOF

VOF(Volume of Fluid) method is provided by Hirt and Nichols[5] in 1981 which employs the volume fraction as an indicator function to mark the different fluids. The interface is not defined as a sharp boundary and a transition region exist where the fluid is treated as some mixture of the two fluids on each side of the interface. The indicator function is defined as

$$\alpha = \begin{cases} \alpha = 1 & \text{for a point inside fluid a} \\ 0 < \alpha < 1 & \text{for a point in the transitional region} \\ \alpha = 0 & \text{for a point inside fluid b} \end{cases}$$
(4)

where α is the indicator function. The indicator function α obeys a transport equation of the form

$$\frac{D\alpha}{Dt} = \frac{\partial\alpha}{\partial t} + \mathbf{u} \cdot \nabla \alpha = 0 \tag{5}$$

Although the choice of the phase fraction as an indicator function is a popluar one, its sharpness and boundedness are hardly to be preserved in the numerical dispersion. Many researchers have proposed all kinds of techniques to solve this problem[6].Weller[7] introduces an artificial compression term(the third term on the left hand side of Eq.(6) below) into the indicator function equation

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{u}\,\alpha) + \nabla \cdot (\mathbf{u}_{r}\alpha(1-\alpha)) = 0 \qquad (6)$$

where \mathbf{u}^r is a velocity field suitable to compress the interface. The artificial term gives the promise that its solution for α can be bounded. It is only active in the transitional region and does not affect the solution significantly outside this region.

In the interface-capturing methodology the interface is not tracked explicitly and its exact shape and location are unknown. Therefore, the surface integral in Eq.(2) that represent the surface tension cannot be calculated directly. Brackbill[9] overcame this problem with their continuum surface force(CSF) model, which represents the surface tension effects as a continuous volumetric force acting with the transition region. It reads:

$$\int_{S(t)} \sigma \kappa' n' \delta(x - x') dS \approx \sigma \kappa \nabla \alpha \tag{7}$$

where the curvature of the interface κ is given by

$$\kappa = \nabla \cdot \left(\frac{\nabla \alpha}{|\nabla \alpha|}\right) \tag{8}$$

3.3 LES Method

LES(Large Eddy Simulation) is situated somewhere between Direct Numerical Simulation(DNS) and the Reynolds-averaged Navier-Stokes(RANS). In this article, the one-equation SGS turbulent energy transport model proposed by Yoshizawa[10] is chosen.

The SGS stress τ^{SGS} is given by (namely τ_{ij} in Eq(13)) $\tau^{SGS} = \overline{uu} - \overline{uu}$ where \overline{u} is the instantaneous resolved velocity field. The SGS stress is approximated through a subgrid scale eddy-viscosity model as :

$$\tau^{SGS} - \frac{2}{3}k\mathbf{I} = -\frac{\mu^{SGS}}{\rho} (\nabla \mathbf{u} + \nabla \mathbf{u}^{-T})$$
(9)

where k is the subgrid scale turbulent energy and μ^{SGS} is the subgrid scale viscosity, both of which are obtained from the one-equation SGS turbulent energy transport model:

$$\frac{\partial k}{\partial t} + \nabla \cdot (k \overline{\mathbf{u}}) = \nabla \cdot [(v + v^{SGS}) \nabla k + \tau^{SGS} \cdot \overline{\mathbf{u}}] - \varepsilon - \frac{1}{2} \tau^{SGS} \cdot (\nabla \overline{\mathbf{u}} + \nabla \overline{\mathbf{u}}^{T}) \quad (10)$$

$$\begin{cases} \varepsilon = C_{\varepsilon} k^{3/2} / \Delta \\ v^{SGS} = C_{k} k^{1/2} / \Delta \end{cases}$$

where ε is the SGS turbulence dissipation rate, Δ is the SGS length scale corresponding to the filter width(in most case equivalent to the cell size). The turbulence model constants have the values $C_{\varepsilon} = 1.05$ and $C_k = 0.07$.

4 Numerical solution method

The OpenFOAM finite volume CFD code is employed for the LES-VOF system of conservation equations which are mainly made up of Eq.(2), (6), (7), and (10). It employs second-order accurate temporal and spatial discretization schemes that preserve the proper limits on physically-bounded variables. The numerical procedure for the convection term in Eq.(6) is based on the vanLee TVD scheme, and the artificial term is based on the interfacecompression method proposed by Weller[7] which can preserve the boundedness and sharpness of the gas-liquid interface. The implicit PISO algorithm is em-



ployed for the coupling solution of pressure and velocity which is suitable for non-steady problem.

4.1 Computational Domain and Grid

The structure of launch tube is simplified by neglecting the composition and the process of opening of the electromagnetic valve. Figure 2 shows the structure of computational domain.



Figure 2. Strucure of flow field domain (mm)

Figure 3 depicts the grid structure of computational domain. The grid is generated in Gambit by Map type and is converted by the tool fluentMeshToFoam in Open-FOAM. Considering the near-wall viscosity effect, the boundary layer is meshed in the near-wall of water chamber. The total number of cells is 1.91 million and the minimum resolution is 0.5mm/cell.



Figure 3. Grid structure of computational domain

4.2 Boundary and Initial Conditions

According to the actual work condition, the gas chamber section is set to be ideal gas which is 1MPa, and the water chamber is set to be static water. The parameters of physical properties are summarized in Table 1.The nonslip condition is applied to the wall of launch tube section. The atmosphere condition is applied to the surrounding wall of external field in which the pressure and velocity condition is inletOutlet, switching velocity and pressure between fixedValue and zeroGradient depending on direction of velocity. The frontAndBackPlanes which is added by the converter automatically is patched to be empty.^[11]

Table1. Parameters of physical properties

parameters	symbols	units	value
Liquid-phase denisty	$ ho_l$	kg/m^3	1000
Gas-phase denisty	$ ho_{g}$	kg/m^3	1.29
Liquid-phase kinematic viscosity	$m^2 \cdot s$	VI	1e-6
Gas-phase kinematic vis- cosity	$m^2 \cdot s$	V_g	1.589e-5
Surface tension	σ	N/m	0.072

5 Simulation Results

Figure 4 presents selective results from the LES-VOF simulation of the internal flow and the near-field of impulse anti-riots water cannon. The instantaneous isosurfaces of liquid phase illustrate the spray configuration and process of impulse water jet. Figure 5 shows the highspeed photographs at 2400fps. From the spray configuration, they are agreed with each other very well. The time discrepancy, about 2-3ms, is caused by the simplified launch tube.



Figure 4. Instantaneous isosurfaces of liquid-phase volume fraction under 1MPa (α =0.5,U:m/s)



Figure 5. High-speed photographs under 1MPa

Through comparing the images carefully, the development process of the near field of impulse anti-riots water cannon can be divided into three phases:

(1)Liquid Column Burst Phase. Liquid column in the launch tube spout under the inflation of high-pressure compressive gas. The velocity in the center of the liquid column is greater than that at the column periphery, so the column presents water-hammer striking pattern. As the role of air resistance, the head of liquid column rewinds. At the same time the surface of liquid column occurs 'fold', which is mainly due to the instability initiated by external air resistance and internal vortices caused by high-pressure gas inflation. With the column spouting, the rewind column breaks up gradually. These can be seen in Fig.4-5ms, 10ms and 15ms and in Fig.5-3ms,5ms and 10ms.From the simulation results, the column velocity increases gradually to 20 m/s in this phase.

(2)Secondary Burst Phase. When the high-pressure gas gets through launch tube, the radial resistance suddenly decreases and the high-pressure gas inflates rapidly which form 'Secondary Burst'. In this phase, the spray angle increases and the liquid surround the gas breaks up into droplets quickly. While liquid column moving forward, the velocity of the back of column is greater than that of the head, and the head is affected by air shear stress, so that liquid column becomes narrow and forms 'umbrella' liquid film gradually. These can be seen in Fig.4-17ms, 20ms and 25ms and Fig.5-15ms, 17.25ms and 20ms.

(3)Liquid Film Rupture Phase. Liquid film moves forward and the droplets formed in the second phase rupture into smaller droplets further under the gas expansion. This can be seen in Fig.4-30ms, 35ms, 40ms and 45ms and Fig.5-25ms, 30ms,35ms and 40ms. At this time, the velocity of the head of column is greater than that of the back, and the surface of the head distorts under the perturbations caused by the pressure distributed unevenly, and the liquid ligaments form and break up into droplets at the speed of 30m/s.

6 Conclusions

The near-field of anti-riots water cannon is simulated by One-Equation LES coupled with the bounded and compressed VOF method. The simulation results and the high-speed photographs are agreed with each other very well form the spray configurations.

The development process of the near-field is divided into three phases: liquid column burst, secondary burst and liquid film rupture through comparing the results carefully. According the results, the high-pressure inflation is the most dominant factor in the near-field.

This numerical model can be extended to 3-D computation in the further study and the composition and the opening process of the electromagnetic valve should be considered to make the simulation results coincide with the experimental results better.

Acknowledgement

This work was supported by national natural scientific fund of China (Grant No. 50876113), and State key laboratory of multiphase flow in power engineering, Xi'an Jiaotong University, China has given significant help in simulation and experiment, I thank both of them here.

References

- M Symons, G Smith, G Dean, S Croft, C O'Brien. Less Lethal Technologies Review of Commercially Available and Near-Market Products for the Association of Chief Police Officers[M]. United Kingdom: Home Office Scientific Development Branch, 2008.
- [2] Jun ISHIMOTO, Hidehiro HOSHINA, Tadashi TSUCHIYAMA, etc. Integrated Simulation of the Atomization Process of a Liquid Jet Through a Cylindrical Nozzle[J]. Interdisciplinary Information Sciences, 2007,(13):7-16.
- [3] Federico Montanari, Jiefu Ma, Karl H.Kuehlert, Peter W.Runstadler. Exploratory CFD Analysis of the Fluid Dynamics of a Water Cannon. AIAA Aerospace Sciences Meeting and Exhibit,2005-1293.
- [4] B.Befrui, G.Corbinelli, D.Robart, W.Reckers, H. Weller. LES Simulation of the Internal Flow and Near-Field Spray Structure of an Outward-Opening GDi Injector and Comparison with Imaging Data. 2008 World Congress Detroit, Michigan April 14-17,2008.
- [5] Hirt C, Nichols B. Volume of fluid(VOF) method for the dynamics of free boundaries[J].Journal of Computational Physics, 1981,39(1):201-225.
- [6] Zou Jian-feng, Zheng Yao. Application of bounded and compressed VOF method to interfacial flow[J]. Journal of Zhejiang University(Engineering Science), 2008, 42(2): 253-258.
- [7] Weller H. A code independent notation for finite volume algorithm[R]. Technical Report TR/HGW/02, Nabla Ltd., 2002.
- [8] Rusche Henrik. Computational fluids dynamics of dispersed twophase flows at high phase fractions[D]:Imperial College of Science,2002.
- [9] Brackbill J, Kothe D, Zemach C. A continuum method for modeling suface tension[J]. Journal of Computational Physics, 1992,100(2):335-354.
- [10] Yoshizawa A, Horiuti K. A statistically-derived subgrid-scale kinetic energy model for the Large-Eddy Simulation of turbulent flows[J]. Journal of the Physical Society of Japan, 1985, 54(8):2834-2839.
- [11] OpenCFD Ltd., FOAM User Guide, http://www.opencfd.co.uk