Available online www.jocpr.com

Journal of Chemical and Pharmaceutical Research, 2014, 6(1):476-479

Research Article

ISSN : 0975-7384
CODEN(USA) : JCPRC5

Simulation of hydraulic characteristics around a fixed-cone valve

Y. L. Liu, Ch. W. Bai, W. L. Wei* and X. F. Yang

State Key Laboratory of Eco-Hydraulic Engineering in Shaanxi, Xi'an University of Technology, Xi'an, Shaanxi, China

---

ABSTRACT

Fixed-cone valves are generally used to regulate flow under medium to high water head conditions because of their ability to safely and efficiently pass the flow. By designing, fixed-cone valves, also known as Howell–Bunger valves, emit a large-diameter conical spray. The spray is effective in spreading and dissipating energy, although in some conditions where space is limited, it may be desirable to contain the spray. Containing the spray may be achieved by using a hood; however the result is a high velocity hollow jet that focuses the energy in the stilling basin. Depending on the size of the stilling basin downstream of the valve and the sensitivity to environmental factors, it may be necessary to dissipate some energy of the concentrated jet prior to impingement in the stilling basin. This paper is concerned with a numerical two-phase flow model combining with the Realizable k–ε turbulent model for simulation of flows around a fixed-cone valve. The equations were solved with the finite volume method. The function of the fixed-cone valve for energy dissipating was pointed out by analyzing the computed pressure field, velocity field of the fixed-cone valve by the proposed model. The simulating results show that the proposed model is reliable and can be applied to the numerical simulation of turbulence flow around a fixed-cone valve.

Keywords: fixed-cone valve, numerical simulation, turbulent flow

---

INTRODUCTION

Fixed-cone valves are ideal for use in applications where a high degree of flow control under medium to high water head is required. Fixed-cone valves take advantage of radially discharging the flow into a conical expanding spray. This design does not require the valve to overcome excessive hydrostatic forces to open or close and enables superb flow control via a moveable sleeve or gate that seats against the cone and is sealed against the valve body.

There are some research results of energy-dissipation using fixed-cone valve [1-6]. In 1935, it was introduced that the Howell– Bunger valve （fixed-cone valve） is ideal for use in applications where a high degree of flow control under medium to high heads(up to 300 m） is required[1]. Fixed-cone valves take advantage of radially discharging the flow into a conical expanding spray. This design does not require the valve to overcome excessive hydrostatic forces to open or close and enables superb flow control via a moveable sleeve or gate that seats against the cone and is sealed against the valve body.

Tetsuhiro Tsukiji [2] has carried out the simulation for the axial fluid flow in a fixed-cone valve with a vortex method. Cao Binggang etc.[3, 4] has performed the numerical analysis of flow fields in a fixed-cone valve with a boundary-element method developed by the integrated equations and the finite-element method, and has designed a special experimental equipment, by which the experimental study was conducted about the pressure distribution on conical surface of fixed-cone valve and internal fluid power in the fixed-cone valve. Gao Dianrong [5] has carried out the numerical calculus for the hydraulic pressure on the core surface of fixed-cone valve and internal flow fields in a fixed-cone valve under different apertures with the Galerkin finite-element method. H. Gao [6] used the RNG turbulence model to simulate a cavity flow of a fixed-cone valve.

This paper presents a numerical two-phase flow model combining with the Realizable k–ε turbulent model for simulation of flow around a fixed-cone valve.

MATHEMATICAL MODEL

The unsteady 2D flow governing equations for continuity, momentum and turbulence stress $-\rho\overline{u_i'u_j'}$ can be written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]-\frac{\partial}{\partial x_j}\left(\rho\overline{u_i'u_j'}\right)+\rho g_i \tag{2}$$

$$-\rho\overline{u_i'u_j'}=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{3}$$

where t is the time; x_i is the space coordinate in i direction; p is the pressure; μ is the molecular kinematic viscosity ; μ_t is the kinematic viscosity; g_i is the gravitational acceleration in i direction; u_i is the velocity component in i direction ($u_1 = u$, $u_2 = w$); u_i' is the fluctuating velocity component in i direction ($u_1' = u'$, $u_2' = w'$); ρ is the density (in water ρ is equal to the water density; in the air ρ is equal to the air density); k is turbulent kinetic energy; and ε is kinetic energy dissipation rate.

For the estimation of the turbulence term $-\rho\overline{u_i'u_j'}$, a Realizable $k-\varepsilon$ turbulence model is incorporated for the estimation of μ_t. The equations for a 2-D model are given as:

Turbulent kinetic energy k equation:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_{kb}-\rho\varepsilon \tag{4}$$

Kinetic energy dissipation rate ε equation:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\rho C_1 E\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} \tag{5}$$

where: $\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}$, $C_1=\max\left(0.43,\frac{\eta}{\eta+5}\right)$, $\eta=\left(2E_{ij}\cdot E_{ij}\right)^{1/2}\frac{k}{\varepsilon}$, $E_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$,

$C_u=\frac{1}{A_0+A_sU^*\,k/\varepsilon}$, $A_0=4.0$, $A_s=\sqrt{6}\cos\phi$, $\phi=\frac{1}{3}\cos^{-1}\left(\sqrt{6}W\right)$, $W=\frac{E_{ij}E_{jk}E_{ki}}{\left(E_{ij}E_{ij}\right)^{1/2}}$,

$E_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$, $U^*=\sqrt{E_{ij}E_{ij}+\tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}$, $\tilde{\Omega}_{ij}=\Omega_{ij}-2\varepsilon_{ijk}\omega_k$, $\Omega_{ij}=\tilde{\Omega}_{ij}-\varepsilon_{ijk}\omega_k$.

where k and ε are turbulent kinetic energy and kinetic energy dissipation rate, respectively; σ_k , C_2 and σ_ε are empirical constants and have the values of 1.0，1.92 and 1.2, respectively.

The above equations for the solution of internal flow fields along with the corresponding boundary conditions according to actual problems constitute this problem to be solved.

GRID GENERATION AND BOUNDARY CONDITIONS

Grid generation

The initial computation grid is generated by the GAMBIT program. As a result of that different computational region requires different computational accuracy, the grid division has used the subregion grid generation method. In the total computational region, the complex part uses the non-structured grid division method, which is compatible to complex boundary and also can carry on auto-adapted processing; and the regular part uses the packing-efficiency high rectangle grid division method, and the grid distribution is determined according to the gradient magnitude of the flow. The computational grid is shown in Fig.2. The governing equation is discretized using the finite volume method. The pressure and velocity coupling equation is solved using the SIMPLE algorithm.

Boundary Conditions

a) Inlet plane: U, V, k and ε are specified.

b) Exit plane: the boundary condition at the outlet cross section of the downstream pipeline is given according to the fully developed turbulent condition, supposing the derivatives of various variables normal the cross section to be zero， that is $\frac{\partial u}{\partial n} = \frac{\partial w}{\partial n} = \frac{\partial k}{\partial n} = \frac{\partial \varepsilon}{\partial n} = 0$.

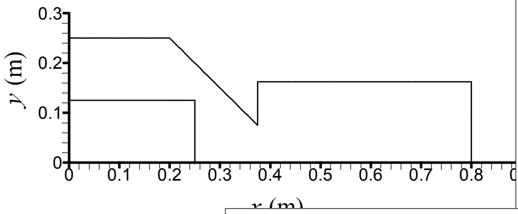
c) Body surface boundary: The RNG k - ε turbulence model is suitable in the flowing region at certain distance from the solid wall boundary surface, therefore the Wall-function method proposed by Launder and Spalding is used in the viscous flowing region near wall surface.

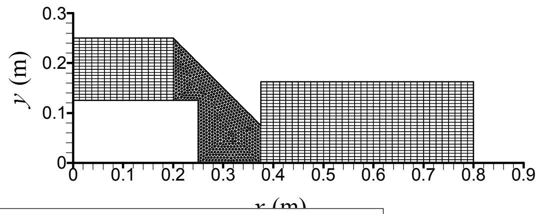
d) Boundary condition of air entrance holes: the pressure at inlet of the air mixing holes is equal to the outside atmospheric pressure (the relative atmospheric pressure is 0).

RESULTS AND DISCUSSION

The actual fixed-cone valve is symmetry of its axis, so the computational region is taken as the 2D region as Figure 1, which is mainly composed of the upstream pipeline, the fixed-cone valve, and the downstream pipeline. The insid

For t
the a
boun
turbu
boun





It is obviously seen from Fi ... result of the fixed-cone valve's separation function, ... l-cone valve. Figs.4 (a) and 4(b) show the computati

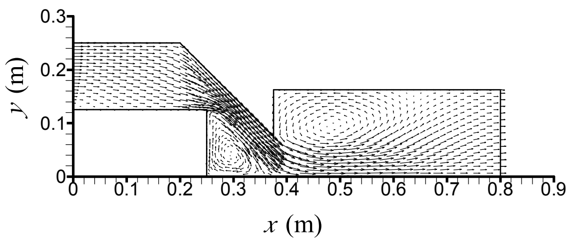
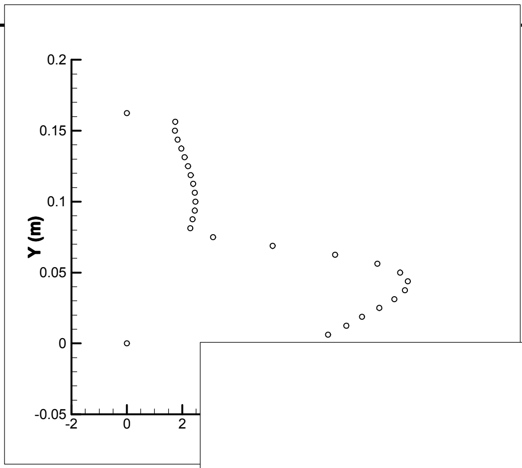
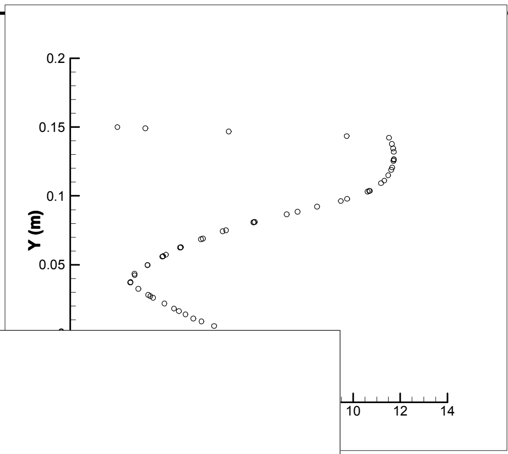


Fig.3 Computational velocity vectors





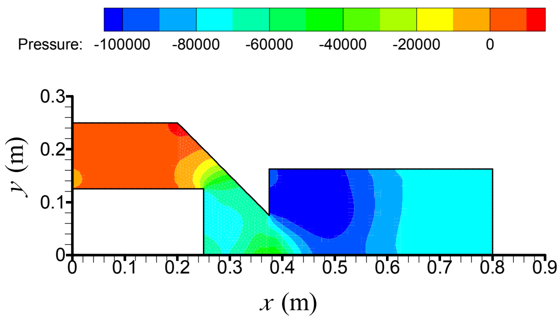


Fig.5 Computational pressure

It may be seen from Figure 5, as a result of the fixed-cone valve's deflecting function, the water pressure in the upstream pipeline is quite great. When the current of water flows across the fixed-cone valve, the flow velocity increases and the pressure reduces with the flow cross section reducing.

CONCLUSION

The proposed two-phase flow model combining with the Realizable k–ε turbulent model for compressible viscous fluid is effective to simulate the hydraulic characteristic variation rule around the fixed-cone valve, such as the pressure field, and velocity field. This paper only discusses the two-dimensional situation; however, an actual project being of 3D complexity turbulence, how to accurately simulate the actual flow moving needs further researches.

Acknowledgments
This work was supported by the National Natural Science Foundation of China (Grant No.51178391), and the Shanxi Province key subject construction funds.

REFERENCES

[1] Michael C. Johnson, M. ASCE, and Rajesh Dham, *Journal of Hydraulic Engineering*. ASCE , AUGUST ,**2006**, 759-764
[2]T. Tsukiji, Flow analysis in oil hydraulic valve using vortex method, Proceedings of 3rd International Symposium on Fluid Power Transmission and Control, Harbin, Sept. **1999**,7-9.
[3] B.Cao , M.Guo, *Machine Tool & Hydraulics*.**1991**. (2) ,2-10.
[4] B. Cao, W. Shi , *Journal of XI'AN Jiaotong University*,**1995**, (7) : 7-13.
[5] D. Gao , Y.Wang, *Machine Tool & Hydraulics* , **2002**,38(4), 67-70.
[6] H.Gao, X. Fu, *Chinese Journal of Mechanical Engineering*,**2002**, (8) ,27-30.