Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

TWO-PHASE FLOW ANALYSIS AND EXPERIMENTAL INVESTIGATION OF MICRO-PIV FOR EMITTER MICRO-CHANNELS

Zhengying WEI*, Meng CAO, Yiping TANG and Bingheng LU

State Key Lab for Manufacturing System Engineering, Xi'an Jiaotong University, Xi'an 710049, China
*Corresponding author, E-mail address: zywei@mail.xjtu.edu.cn

ABSTRACT

Aiming at complex labyrinth-channel emitters' structure and the practical problem of clogging, this paper investigates the practical flow of water, mixed with sand escaped from filtering, in the labyrinth channel. CFD analysis has been performed on liquid-solid two-phase flow in labyrinth-channel emitters. Based on micro-PIV (particle image velocimetry) flow visualization technology, the flow in labyrinth channel has been photographed and recorded. The path line graph and velocity vector graph are obtained through the post-treatment of experimental results. The graphs agree well with CFD analysis results, so CFD analysis can be used in optimal design of labyrinth-channel emitters.

KEY WORDS

Drip irrigation emitter; Anti-clogging;
Labyrinth micro-channel;
Two-phase flow analysis;
Micro-PIV(Particle Image Velocimetry)

NOMENCLATURE

μ dynamic viscosity
u fluid velocity
ρ fluid density
k the turbulence kinetic energy
ε the rate of dissipation
μ_t the turbulent viscosity
St the rate of dissipation

INTRODUCTION

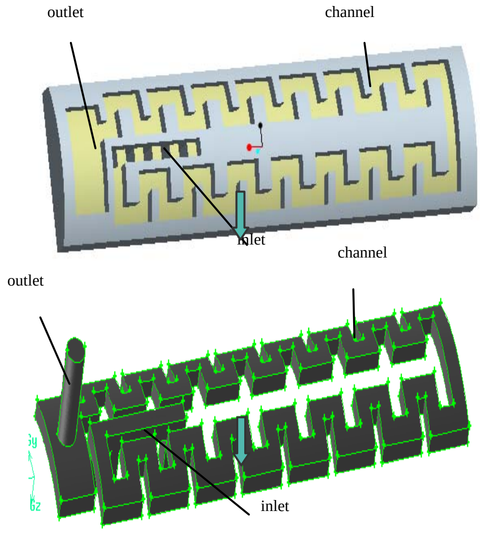
The emitter is the key part in drip irrigation system, it is designed to let out the pressurized water in pipe to drop into the soil slowly and uniformly, the size of the emitter channels is about 0.5~1.0 mm. It is easy to be clogged by particles escaped from filtering in water. In this paper, on the basis of analysing the flow field of water-sand two-phase flow in micro-channels of drip irrigation emitters, its anti-clogging mechanism is discussed. Current studies only emphasis on the research of single-phase flow behaviour in micro channels of emitter(Paulau, 2004, Wang, 2000, Wei, 2004, Li, 2005). However, using an original-size prototype to perform the experiment on the two-phase flow characteristics of the labyrinth channels is seldom reported. The micro-PIV (Particle Image Velocimetry) technique (Santiago, 1998, Meinhart, 1999) is used to conduct the visualized experimental analysis for two-phase flow behaviours in emitters. Recent micro-PIV experimental research on the flow characteristics in various micro-channels mainly focus on the single-phase fluid flow (Daniel, 2005, Kim, 2002, Sebastian Eisenberg, 2002). However, based on water-sand two-phase CFD (Computational Fluid Dynamics) of micro-channel, a micro-PIV experimental platform was set up here to perform a series of hydraulic behaviour tests on labyrinth channel emitters.

TWO-PHASE FLOW NUMERICAL SIMULATION IN LABYRINTH CHANNELS

Geometry model and boundary conditions

This paper selected emitters with rectangular and zigzag labyrinth channels, and used CAD software to establish solid models of emitters and channels respectively. The unit width of labyrinth channels is between 0.3 mm and 1.2 mm, and the fluid in the emitter is water which is assumed to be viscous, incompressible and steady at room temperature. The fluid gravity and the surface tension of the fluid were ignored for no free interface in the labyrinth channel. The surface roughness of the channel wall should be taken into account.

The labyrinth channel computational model consists of inlets, channels and outlets. In order to get high-quality grids, each channel unit could be meshed with structured Cooper hexahedron grids of 0.1 mm unit width, as illustrated in Fig. 1. According to the practical operating condition of drip irrigation system, the pressure inlet was treated as the boundary condition, and pressure values were taken every 10 kPa from 40 kPa to 150 kPa. The outlet pressure was 0 Pa (gauge), while the channel wall was assumed under no-slip condition.



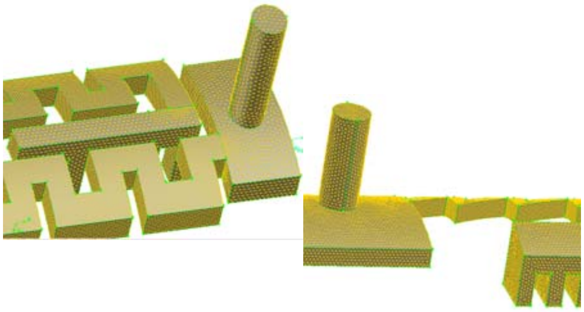


Fig. 1. Models of emitter and channels and grid plot of channels

Mathematical model of micro-channel flow field

Relationship of flow rates and pressure

There aren't yet appropriate theories to calculate the critical Reynolds number due to the complex channel structure. Nishimura et al. (1984) researched the wavy channels, and research results showed that in this complex channels the transition from laminar to turbulent flow occurred when the Reynolds number reached 350, which was far less than critical Reynolds number in conventional straight channels (Re_c=2200). Models of laminar flow and turbulent flow under different pressure conditions were simulated in this paper, in order to obtain more comprehensive results.

Laminar flow model

Mass equation and Momentum equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial}{\partial x_j}\left(u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

where u is the fluid velocity ;ρ (kg/m^3) is the fluid density and μ is dynamic viscosity; $i, j = 1,2,3$;

Turbulence model:

Here RNG *k–ε* turbulence models is adopted to simulate the continuous flow fields. The governing equations involve Eq. (1), Eq. (2), *k* equation, *ε* equation and so on (Warsi Z U A, 1993).

The turbulence kinetic energy, *k*, and its rate of dissipation, *ε*, are obtained from the following transport eqrations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right] + G_k + \rho\varepsilon \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right] + \frac{C^*_{1\varepsilon}}{k} G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{4}$$

Where $\mu_{\mathrm{eff}} = \mu + \mu_t$, $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$, $C_\mu = 0.0845$, $\alpha_k = \alpha_\varepsilon = 1.39$, $C^*_{1\varepsilon} = C_{1\varepsilon} - \frac{\eta(1-\eta/\eta_0)}{1+\beta\eta^3}$, $C_{1\varepsilon} = 4.377$, $C_{2\varepsilon} = 1.68$, $\eta = (2E_{ij} \cdot E_{ij})^{1/2}\frac{k}{\varepsilon}$, $\eta_0 = 4.377$, $\beta = 0.012$.

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, μ_t the turbulent viscosity.

The governing equations are solved using a discrete implicit steady solver, while SIMPLE algorithm of Fluent6.2® used to conduct to calculate velocity and pressure coupling.

Experimental analysis of the flow rate and pressure

The flow rate measurement of emitters was performed on integral experimental prototype. Flow rates were obtained by experiments and CFD simulation under the pressure of 40~150kPa. With experimental results, the flow rate average deviations obtained from turbulence-flow model and laminar-flow model are approximate to 4% and 9% respectively, as illustrated in Fig. 2. It can be concluded that the experimental results obtained from turbulence-flow model are more rational than that of laminar-flow model. So the turbulence-flow model was used to simulate flow field in the subsequent work.

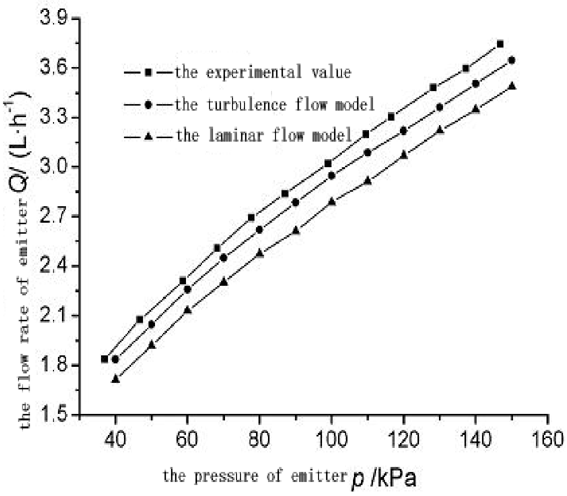


Fig. 2. Curves of flow rate-pressure relation

CFD analysis of micro-channels in emitter using two-phase flow theory

In this paper, flow field in emitters was simulated using RNG *k–ε* turbulence models. In actual application of emitter, after filtering the volume ratio of sand to water is far less than 10%. Therefore, the discrete model of Euler-Lagrange is selected as the computational model of two phase flow, expecting to simulate the clogging distribution in channels as practical as possible.

Discrete phase is assumed to be the sand particles, whose density is 2500kg/m^3, the velocity of particles at inlet is defined using the maximum water flow rate at inlet. According to the regulation in ISO standard (2003) short-period anti-clogging measurements, the sand radius should be fitted using Rosin-Rammler distribution.

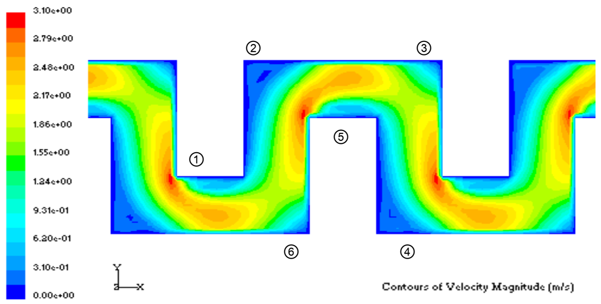
Based on the above defined mathematical model and geometric model, the commercial CFD software Fluent6.2® is used to simulate the flow characteristic in micro-channels of emitters, and the convergence accuracy is 10^{-4}. The flow field graphs in rectangular and zigzag channels are illustrated in Fig.3. As seen in Fig. 3, the high velocity regions are close to the middle of the micro-channels, the low velocity regions occur at the corner, while the velocity is zero in the centre of vortex area, then vortices and stagnant regions are developed as shown in Fig. 3a. Because the emitter is consist of several units with the same structure, and one unit is chosen as analysis object. It is obvious from the figures that there are five low-velocity regions and vortices in one rectangular labyrinth unit, but there are only two low-velocity regions and vortices in one zigzag labyrinth unit.

According to the sediment dynamic principle, the movement of solid particles is correlated to fluid vortices in liquid-solid two phase flow. Wu Wenquan and Huang Yuandong et al. (1999) pointed out that the sediment particle with small St number (0.15-0.59) or middle St number (1.33-2.36) is easy to be entrapped into the fluid vortices, and these particles would deposit in the vortices regions. Stokes number (St) (Dominguez, 2007) is used to measure capability of particles diffusion motion in fluid:

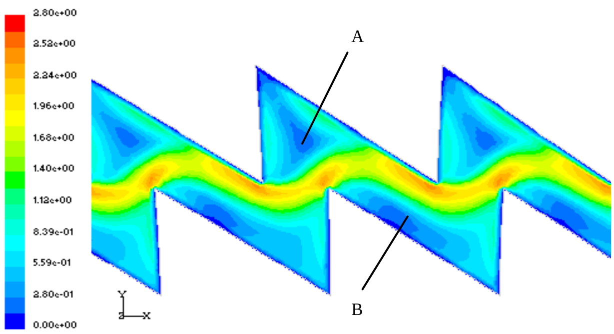
$$St = \tau_p / \tau_k \qquad (5)$$

where τ_p is defined as particle's relaxation time and $\tau_p = \rho_p d_p^2 / (18\mu)$, τ_k defined as Kolmogorov characteristic time and $\tau_k = \sqrt{\mu / \varepsilon}$.

They are difficult to be flushed away from the vortices region or low-velocity region, and with time passing, they would congregate together and finally result in clogging. So it is necessary to eliminate the low velocity regions and vortices in labyrinth channels in the emitter design.

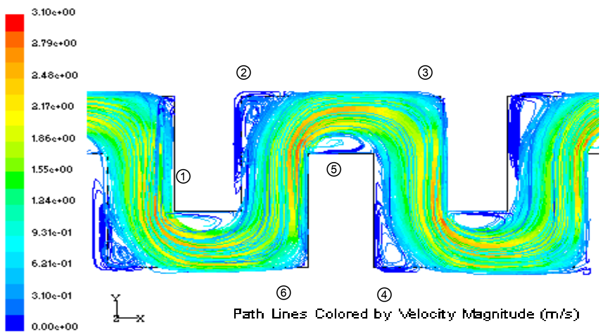


Rectangular channel

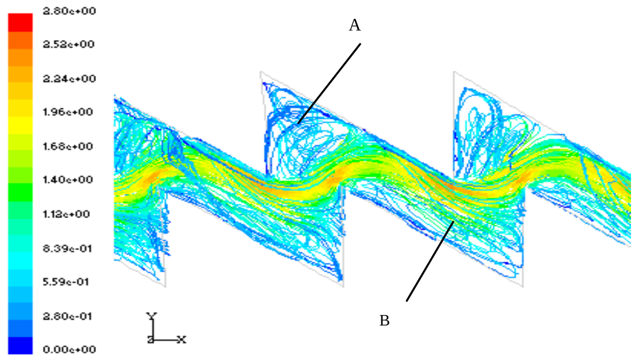


Zigzag channel

(a)Contour of velocity magnitude



Rectangular channel



Zigzag channel

(b)Plot of path lines

Fig. 3. Flow field of labyrinth channels

MICRO-PIV MEASUREMENT FOR MICRO CHANNELS OF EMITTER

In order to obtain the real flow state in micro channels, a continuous light source, a high-speed dynamic recorder (TroubleShooter) and a microscope lens (ZOOM6000) are configured to set up a micro-PIV experimental bed as shown in Fig.4. Thus the structure with area of 1mm^2 can be put into the field of view. The instrument could record 16000 frames per second, which is enough to meet the requirement for different flow rates. The micro-channel test samples have been fabricated with NC engraving on a PMMA plate, and a glass cover is bonded to seal the channels, so the experimental device is similar to the actual emitters used in drip irrigation systems.

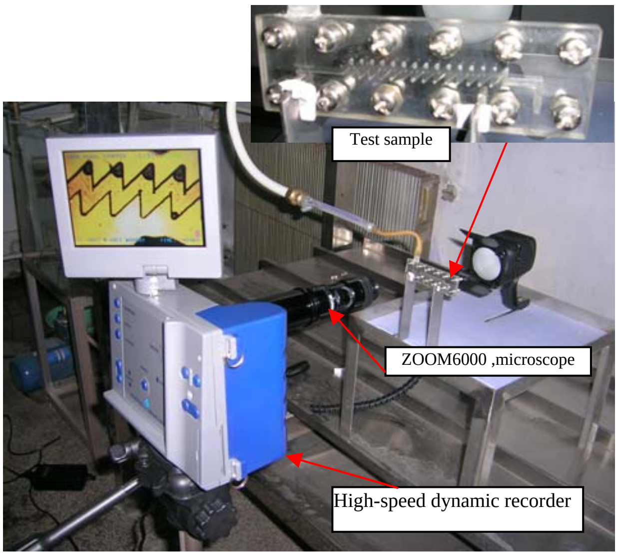
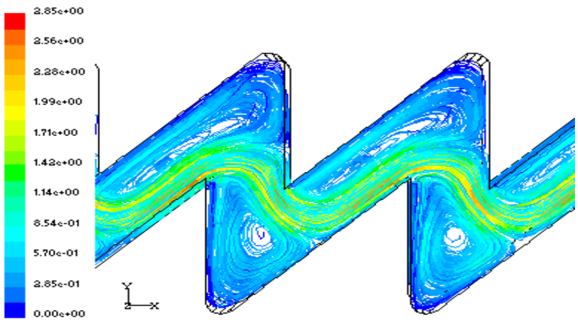


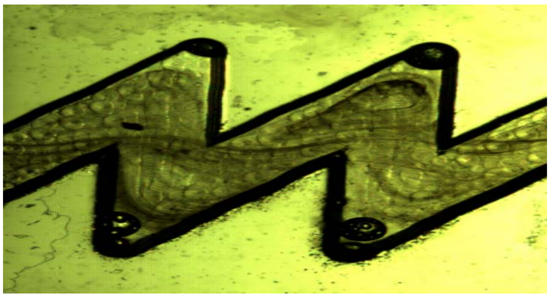
Fig. 4. Micro-PIV test bed of channel flow field

Flow state display in zigzag channel using dyeing ink

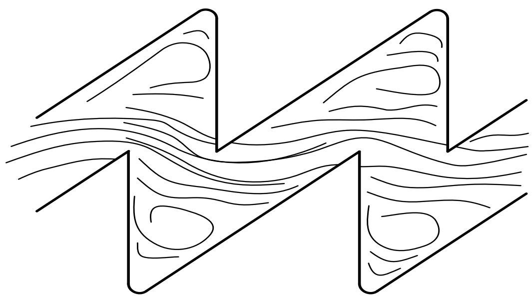
The dyeing ink is injected to the channel, so as to vividly display the real flow state. When the flow field in channels is stable, the dyeing lines demonstrate the real flow field inside the channel. Then the flow field can be recorded by Micro-PIV setup. The micro-PIV photo of dyeing lines is shown in Fig.5b, and through data processing the experimental path lines can be seen in Fig.5c.



(a)Path lines simulated by CFD



(b) PIV photo of dyeing lines



(c)Experimental path lines in channel

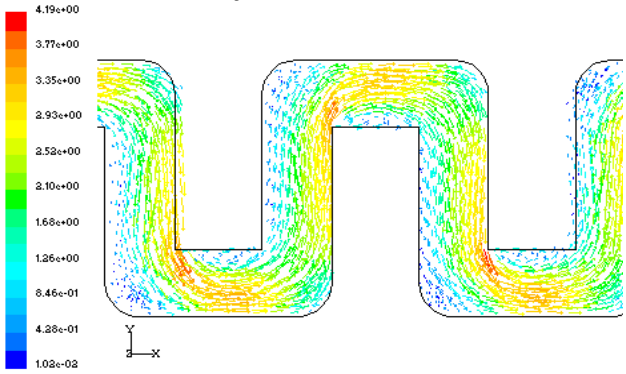
Fig. 5. Path lines of flow field in zigzag emitter channel

By comparing the CFD simulation results (Fig.6a) with experimental results, it is easy to find that the size and shape of the vortices at the sharp corner are similar. Therefore, two results show reasonable agreement.

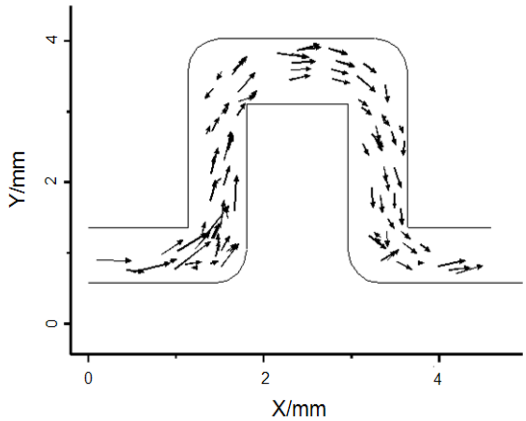
Flow field display in rectangular channel using particle tracing method

In order to measure the real flow state of the sand inside micro-channels, Silica sand is chosen as the tracing material. The particle diameter is about 65-100 μm and density is 2320 Kg/m^3. Al_2O_3 powder with the diameter of 5 μm is selected to trace the liquid phase. Silica sand and Al_2O_3 powder are mixed uniformly and then sprinkled into circulating fluid so as to form a two-phase flow. Micro-PIV measurement is then performed to obtain clear image because of the fine reflectivity of the two kinds of particles. Under experimental pressure of 100kPa and particle concentration of 500 ppm, high-speed recorder is used to record the flow field of particle movement.

After image recording, the image processing software Aurora Media Workshop is used to extract the flow field photos taken by high-speed recorder (Fig.6b). Then these photos are imported into Matlab7.0, the coordinates of the particles at different positions in each photo are determined. Finally all these coordinate data are imported into a data analysis software Origin7.5, the graph of particle velocity vector distribution is obtained, as shown in Fig.6b. The velocity vector graph in labyrinth channel simulated by CFD is illustrated in Fig. 6a.



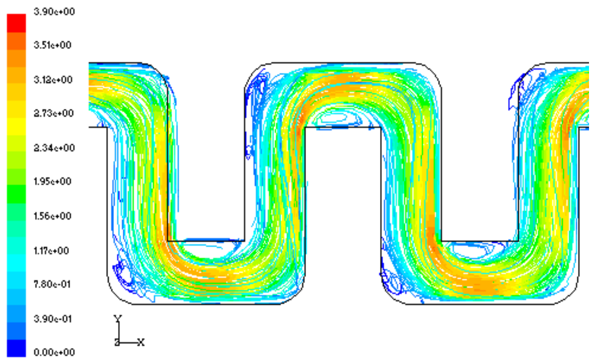
(a) Velocity vector simulated by CFD



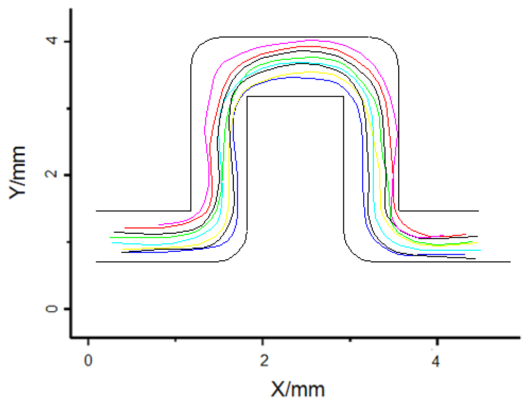
(b)Experimental Velocity vector of particles

Fig. 6. Flow field Velocity vector graph in rectangular emitter channel

By tracing several particles in channels, the coordinate points at different moment are obtained, and these points are linked using spline curve in AUTOCAD to plot the path lines in the flow field, as shown in Fig. 7b. The path lines graph simulated by CFD is shown in Fig. 7a



(a) Path lines simulated by CFD



(b) Experimental path lines in channel

Fig. 7. Path lines of flow field in rectangular emitter channel

By comparing the results obtained with CFD simulation and experimental measurement, it is easy to find that the velocity distribution and shape of path lines obtained by CFD and experimental measurement are similar. The structural optimization schemes can be put forward according to the fluid analysis in the future work based on eliminating the vortices region or low-velocity region to some extent.

CONCLUSION

In this paper, based on the two-phase flow theory, aiming at rectangular and zigzag channel emitters commonly used at present, two-phase flow numerical simulation on the flow behaviour is conducted. And through a micro-PIV experiment bed is set up to evaluate the CFD results. It is concluded that turbulence-flow model are more rational than that of laminar-flow mode, and there are several vortices and low velocity regions in the labyrinth channel causing clogging. And the CFD simulation results show reasonable agreement with the experiments' results. It can provide a feasible method for anti-clogging optimal design of emitter channels.

ACKNOWLEDGEMENT

This work is supported by the NSFC project (50675172), A Foundation for the Author of National Excellent Doctoral Dissertation of PR China (200740), Program for Changjiang Scholars and Innovative Research Team in University (IRT0646), and undertaken by the State Key Lab for Manufacturing System Engineering, Xi'an Jiaotong University.

REFERENCES

PAULAU, S.G., ARVIZA, V.G. and BRALTS, V.F., (2004), "Hydraulic flow behavior through an in-line emitter labyrinth using CFD techniques", *Conference of ASAE/CSAE*, Canada, 1-8.

WANG SHANGJIN, LIU XIAOMIN, XI GUANG, et al, (2000), "Numerical simulation of flows inside a drip faucet by finite element method", *Transactions of the Chinese society of agricultural machinery,* **vol. 16, no. 4,** 61-63 (in Chinese).

WEI QINGSONG, SHI YUSHENG, DONG WENCHU, et al, (2004), "Study on the mechanism of liquid flow and its numeric visualization in the emitter of drip irrigation" , *China Rural Water and Hydropower,* **no. 3,** 1-4 (in Chinese).

LI YONGXIN, LI GUANGYONG, QIU XIANGYU, et al, (2005), "Modeling of hydraulic characteristics through labyrinth emitterin drip irrigation using computational fluid dynamics", *Transactions of the Chinese society of agricultural engineering ,* **vol. 21, no. 3,** 12-16 (in Chinese).

SANTIAGO, J.G., WERELEY, S.T., MEINHART, C.D., et al, (1998), "A particle image velocimetry system for microfluidics", *Experiments in Fluids,* **no. 25,** 316-319

MEINHART, C.D., WERELEY, S.T., SANTIAGO, J.G., (1999), "PIV measurements of a micro channel flow", *Experiments in Fluids,* **no. 27,** 414-419.

DANIEL, A.STEINGART, JAMES W.EVANS, (2005), "Measuremen ts of granular flows in two-dimensional hoppers by particle image velocimetryPart I:experimental method and results", *Chemical Engineering Science,* **no. 60,** 1043-1051.

KIM, S.K., (2001), "An experimental study of flow in a wavy channel by PIV," *THE 6TH ASIAN SYMPOSIUM ON VISUALIZATION,* PUEXCO, Pusan, 1-6.

SEBASTIAN, EISENBERG, Dr.WOLFGANG, et al, (2002), "Visualization and PIV measurements of high-speed flows and other phenomena with novel ultra-hign-speed CCD camera", *West East High Speed Flow Fields*, 1-6.

NISHIMURA, T., OHORI, Y., KAWAMURA, Y., (1984), "Flow characteristics in a channel with symmetric wavy wall for steady flow," *Journal of chemical engineering of JAPAN, vol. 17, no 5*, 466-471.

WARSI Z U A, (1993), "Fluid dynamics: theoretical and computational approaches", *CRC Press, Washington DC*, 540-546.

ISO Standards, ISO/TC23/SC18/WG5N4, (2003) "Clogging test methods for emitters".

WU WENQUAN, HUANG YUANDONG, (1999), "Numerical simulation for the influence of fluid vortex on particle motion in liquid-particle two-phase flow", *Journal of Engineering Thermophysics , vol. 20, no 3*, 365-369 (in Chinese).

DOMINGUEZ A., AARTRIJK M., CASTELLO L., et al, (2007), "Particles-laden flow", *Houten : Springer Netherlands*, 361-362.

