http://ansinet.com/itj

ITJ

ISSN 1812-5638

INFORMATION TECHNOLOGY JOURNAL

ANSInet
Asian Network for Scientific Information
308 Lasani Town, Sargodha Road, Faisalabad - Pakistan

Information Technology Journal 12 (15): 3476-3479, 2013
ISSN 1812-5638 / DOI: 10.3923/itj.2013.3476.3479

Research on Round-pier Turbulent Scope in Straight Channel

Ai Wan-Zheng and Wang Jia-Hong
School of Navigation, Zhejiang Ocean University, Zhoushan, Zhejiang, 316021, China

Abstract: It is very important for ship's navigation to layout the fairway in bridge area reasonably. The turbulent scope of pier, relating directly to traffic accident in bridge area, is an important index to take into account for bridge area's fairway layout. In the present paper, the turbulent scope of round-pier in straight channel and relative parameters, such as round-pier's diameter, flow's velocity and so on, were analyzed by theoretical considerations. It could be concluded that the maximum turbulent scope of round-pier in straight channel was mainly dominated by round-pier's diameter and flow's velocity in water surface. The larger the round-pier's diameter and flow's velocity in water surface are, the larger is the maximum turbulent scope of round-pier in straight channel. An empirical expression was presented to calculate the maximum turbulent scope of round-pier in straight channel.

Key words: Round-pier, turbulent scope, fairway layout, ship navigation, empirical expression

INTRODUCTION

The development of bridge construction in china is very rapid since the 1980s. Bridge construction in china great improves traffic condition; it also pours vigor into China's economic development (Chen and Wang, 2004). If the location of bridge area is inappropriate or bridge channel layout is unreasonable, bridge might become the bottleneck to shipping development. In recent years, due to channel layout defects of bridge area, accident of ship collision bridge occurs frequently (Wu and Zhan, 1999). It is very important to consider pier turbulent flow scope and bridge channel buoy is layout outside the scope of pier turbulent. The fairway layout in bridge waters is shown in Fig. 1. This is because that both ship's rudder effect and course stability are poorer when ship is sailing in turbulent flow area. At present, the reference to layout of fairway in bridge area in china is mainly implemented by china's inland river navigation standards, however definite define of turbulent flow scope around round-pier is not explicit in the standard, it is necessary to research on turbulent flow range to provide reference to fairway layout in bridge area.

About the turbulent scope of pier, some people, such as Hu *et al.* (2002, 2004), think that the bridge pier scope is mainly related with pier size, the Froude number closely. They think when the Froude number changes from 0.14 to 0.4, the pier turbulent scope also varies from 0.5D to 2.6D (D is diameter of round-pier) in straight channel; In winding channels round-pier bridge waters, when the Froude number changes from 0.10 to 0.18, round-pier turbulent scope varies from 0.5D to 2.6D (Shen *et al.*, 2004). Although the conclusion about pier turbulent scope has some reference value to bridge area channel layout, it does not draw experience formula and it is much inconvenient to apply to practical engineering. Zhuang (2007) of Wuhan university of science and technology conducts model experiment to different kind of pier turbulent flow scope and obtains related experience expression, but because experimental data were insufficient, the accuracy of his experience expression about pier turbulent scope need to be improved. Foreign Olsen and Melaaen (1993) and Raudkivi (1986) also researched qualitatively on bridge turbulent flow scope, he considered that bridge area turbulent scope increased with flow velocity's increasing and water depth's

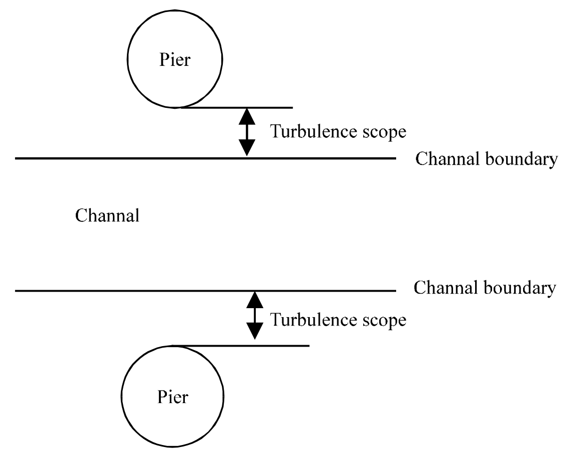

Fig. 1: Fairway layout in bridge waters

Corresponding Author: Ai Wan-Zheng, School of Navigation, Zhejiang Ocean University, Zhoushan, Zhejiang, 316021, China

increasing. His conclusions have guiding significance on bridge area channel layout, but quantitative research on bridge area turbulent scope need to be research further.

So it is necessary to research bridge area turbulent scope further. The purposes of the present work, therefore, are to investigate the effects of the relevant factors on round-pier turbulent scope in straight channel, i.e., the round-pier's diameter and flow's velocity and so on; to present an empirical expression of round-pier turbulent scope in straight channel by means of the numerical simulations; and to verify the rationality of the above empirical expression by using model experiment data.

THEORETICAL ANALYSES

There exist flows' sub-velocity vectors which follow the fairway's axis line direction (be defined U_x in his paper) and also there exist flows' sub-velocity vectors which are vertical with the fairway's axis line direction (be defined U_y in this paper) around round-pier in straight fairway.

Because round-pier is strictly ax symmetric, U_y can result in interaction between ship and pier and U_y has the most effect on ships navigation safety, The round-pier turbulent scope is defined as the biggest vertical distance between the point where U_y disappears ($U_y = 0$) and round-pier's horizontal axis line in this paper. The round-pier turbulent scope is expressed as l in this paper.

There are many parameters which affect the round-pier turbulent scope l and the relevant parameters of dimensional analysis come from the following groups:

- Fluid Properties. These consist of: the density of water ρ (kg m^{-3}), the dynamic viscosity of water μ (N.s m^{-2}) and the acceleration of gravity g (m s^{-2})
- Round-pier's geometry, it is its diameter D (m)
- Flow Properties. These consist of: the flow velocity μ (m s^{-1}), the water's depth around round-pier (be defined h in this paper)

Each of these design parameters is a function of the initial independent parameters:

$$l = f(\rho,\mu,g,D,u,h) \tag{1}$$

This relationship could be rewritten in terms of dimensionless parameters:

$$l/D = f\left(\frac{u}{\sqrt{gh}}, \frac{uD}{\mu/\rho}\right) \tag{2}$$

where, above formula implies that the round-pier turbulent scope l is the function of Froude number and Reynolds number. Because density of water ρ (kg m^{-3}), the dynamic viscosity of water μ (N.s m^{-2}) and the acceleration of gravity g (m s^{-2}) have little variation in normal temperature, the effect to l of density of water ρ, the dynamic viscosity of water μ and the acceleration of gravity g can be omitted. Meanwhile, Zhuang (2007) regarded that pier turbulent scope l increases with water's depth increasing, Eq. 2 shows that pier turbulent scope l increases with flow's velocity increasing, the maximum water's depth and flow's velocity are on water surface, the maximum round-pier turbulent scope appears on water surface also. So it is safety for ship's navigation to take account of round-pier turbulent scope on water surface. If only considering the maximum round-pier turbulent scope on water surface, Eq. 2 can be simplified as:

$$L \approx f(D,U) \tag{3}$$

where, L is the maximum round-pier turbulent scope; U is flow's velocity on water surface. The above equation shows that the maximum round-pier turbulent scope has closely relationship with flow's velocity on water surface U and round-pier's diameter D. So two-dimension simulation method can be used to research round-pier maximum turbulent scope.

METHODOLOGY AND PHASES

Numerical simulations model: The RNG k~ϵ model was used to calculate the hydraulic parameters of the flow through the orifice plate, due to its suitability for simulating the flow inside large change boundary forms as well as its high precision and calculation stability. For the steady and incompressible flows, the governing equations of this model can be written as (Yang and Zhao, 1992).

Continuity Equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad i = 1, 2 \tag{4}$$

Momentum equation:

$$u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu+\nu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \quad i = 1, 2 \tag{5}$$

k-equation:

$$u_i\frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_k(\nu+\nu_t)\frac{\partial k}{\partial x_j}\right] + \frac{1}{\rho}G_k - \varepsilon \quad i = 1, 2 \tag{6}$$

ε-equation:

$$u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon (\nu + \nu_t) \frac{\partial \varepsilon}{\partial x_j}\right] + \frac{1}{\rho} C_1^* G_k \frac{\varepsilon}{k} - C_2 \frac{\varepsilon^2}{k} \quad i = 1, 2 \qquad (7)$$

where x_i (= x, y) are the coordinates in longitudinal and transverse directions, respectively; u_i (= u_x, u_y) are the velocity components in x and y directions, respectively; ρ is the density of water; p is the pressure; μ is the kinematic viscosity; μ_t is the eddy viscosity and can be given by $\mu_t = C_i(k^2/\varepsilon)$, in which k is the turbulence kinetic energy, ε is the dissipation rate of k and C μ = 0.085. The other parameters are:

$$C_1^* = C_1 - \frac{\eta(1-\eta/\eta_0)}{1+\lambda\eta^3}, \eta = Sk/\varepsilon, S = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

$$C_1 = 1.42, \eta_o = 4.377, \lambda = 0.012, G_k = \rho \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$$

$C_2 = 1.68$ and $\alpha_k = \alpha_\varepsilon = 1.39$. The calculation boundary conditions are treated as follows in the inflow boundary the turbulent kinetic energy k_{in} and the turbulent dissipation rate ε_{in} can be defined as, respectively (Xia and Ni, 2003):

$$k_{in} = 0.0144 u_{in}^2, \quad \varepsilon_{in} = k_{in}^{1.5} / (0.25D) \qquad (8)$$

where, u_{in} is the average velocity in the inflow boundary. In the outflow boundary the flow is considered as developed fully. The wall boundary is controlled by the wall functions.

Determine method of maximum turbulent scope: The velocity of flows can be decompose two sub-velocity when flows experiment the round-pier, one of which follows the fairway's axis direction (can be defined X-velocity) and another of which is vertical with the fairway's axis direction (can be defined Y-velocity). The vertical distance between the points where the *Y*-velocity disappears in the vicinity of the round pier and the round-pier's surface is defined turbulent scope of round-pier in this paper which is referred as L in the above context.

Simulation phases: Six kinds of calculation phases were simulated. The round-pier's diameter D is fixed at 0.5, 1.0, 1.5, 2.0, 2.5 and 3.0, respectively at every phase, but the fairways surface flow velocity U varies from 18 to 50 cm s^{-1}. The turbulent scope L at every phase was calculated. The purpose of the above phases design is to discuss the relationship between L, D and U.

DISCUSSIONS

Figure 2 and 3 is the streamline when pier's diameter is 1m, flow's velocity is 5 and 7 cm s^{-1}, respectively. The two figure show that there are turbulence areas around pier. The simulation results is shown in Table 1, in which the round-pier's diameter D varies from 0.5 to 3.0 m and the fairways surface flow velocity U varies from 18 to 50 cm s^{-1}. Figure 4 can be given according to the data in Table 1. It can be learned from figure 4 that L increases logarithmically with U when D is fixed.

Figure 5 is drawn by using the data in Table 1 when U is 18 cm s^{-1}. Figure 5 shows that L increases linearly with D when U does not vary. This relationship between L and D is also demonstrated by Fig. 4. By fitting the curves in Fig. 4, the following empirical formula relating to round-pier turbulent scope can be obtained:

$$L/D = 1.8\ LD(U) - 4.7 \qquad (9)$$

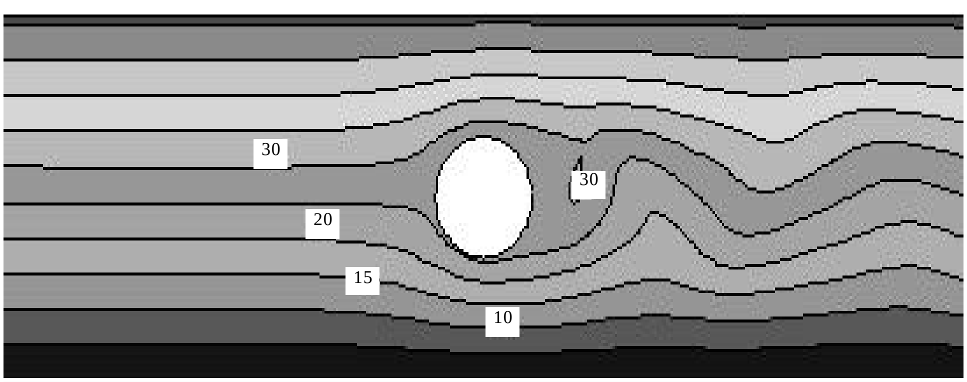

Fig. 2: Streamline around pier (U = 5 cm s^{-1})

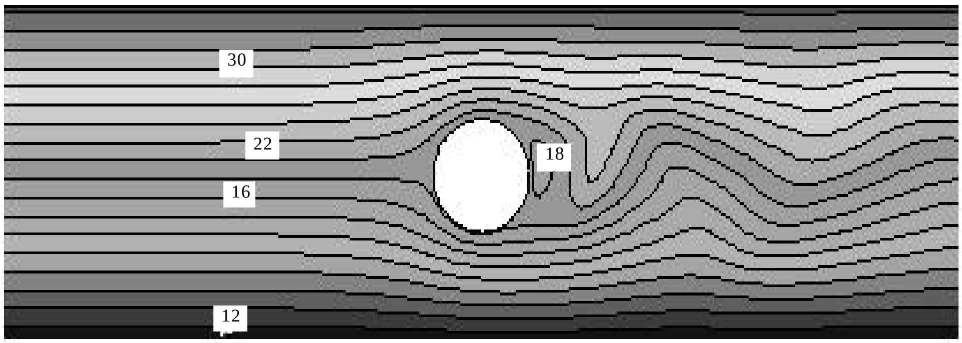

Fig. 3: Streamline around pier (U = 7 cm s^{-1})

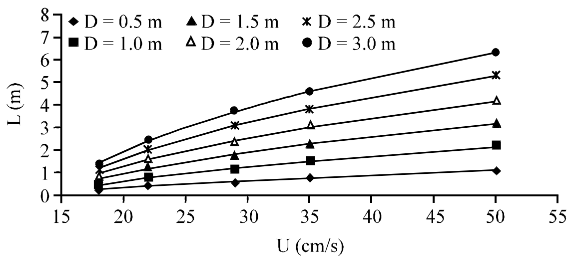

Fig. 4: L when U varies and D varies

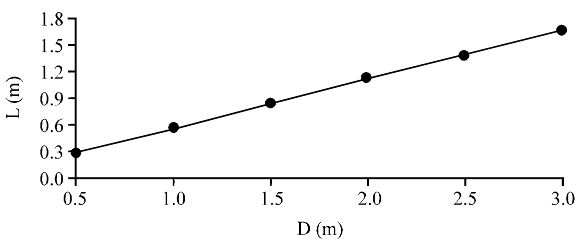

Fig. 5: Relationship between L and D when U fixed

Table 1: L when U varies and D varies

U (cm/s)	18	22	29	35	50
L (m)					
D = 0.5 m	0.27	0.56	0.65	0.88	1.22
D = 1.0 m	0.53	1.01	1.28	1.75	2.42
D = 1.5 m	0.81	1.52	1.94	2.64	3.66
D = 2.0 m	1.06	2.03	2.57	3.51	4.85
D = 2.5 m	1.33	2.54	3.25	4.39	6.12
D = 3.0 m	1.61	3.11	3.85	5.27	7.32

CONCLUSIONS

The relationships among the maximum turbulent scope L of round-pier, round-pier's diameter D and the fairways surface flow velocity U were researched in thispaper. By using theoretical analysis and simulation method, it is found that L increases logarithmically with U when D is fixed and L increases linearly with D when U does not vary. Empirical formula relating to round-pier maximum turbulent scope is obtained in this paper.

ACKNOWLEDGEMENTS

The authors would like to thank for the support by Zhejiang Province Natural Science Foundation of China under the Grant Y5110084.

REFERENCES

Chen, M.D. and D.Y. Wang, 2004. An approach to the navigable clearance dimensions and navigation safety guarantee measures for crossing-river bridge. J. Port Waterway Eng., 4: 42-43.

Hu, X.Y., X.X. Shen and L.C. Huang, 2002. Investigation on regulation method of oblique flow in bifurcation region of braided reach. J. Yangtze River Scient. Res. Inst., 19: 15-17.

Hu, X.Y., X.X. Shen and Y.Z. Cheng, 2004. Experimental investigation on width of superficial eddy area around cylindrical pier. J. Hunan Light Ind. Coll., 1: 39-42.

Olsen, N.R.B. and M.C. Melaaen, 1993. Three-dimensional calculation of scour around cylinders. J. Hydraulic Eng., 119: 1048-1054.

Raudkivi, A.J., 1986. Functional trends of scour at bridge piers. J. Hydraulic Eng., 112: 1-13.

Shen, X.X., Y.Z. Cheng and X.Y. Hu, 2004. Safe distance between channel sideline and bridge pier. Port Waterway Eng., 11: 85-87.

Wu, X.F. and L.W. Zhan, 1999. Maneuverability and Wave Making of the Ship. RenMin Transportation Press, Beijing, China, pp: 183-195.

Xia, Q.F. and H.G. Ni, 2003. Numerical simulation of plug energy dissipater. J. Hydraulic Eng., 8: 37-42.

Yang, Y.Q. and H.H. Zhao, 1992. Numerical simulation of turbulent flows passed through an orifice energy dissipator within a flood discharge tunnel. J. Hydrodyn. Ser. B, 4: 27-33.

Zhuang, Y., 2007. Experimental study on turbulent width around bridge pier. J. Southwest Jiaotong Univ. (English Edn.), 15: 45-48.