Investigation of the unsteady motion and formation mechanism of intake vortices

To cite this article: Fei Ye et al 2017 IOP Conf. Ser.: Earth Environ. Sci. 61 012040

View the article online for updates and enhancements.
Investigation of the unsteady motion and formation mechanism of intake vortexes

YE Fei¹, ZHANG Jing², Kong Bo³

¹North China University of Water Conservancy and Electric Power, Zhengzhou 450011, China
²Yellow River Conservancy Commission of the Ministry of Water Resources
³XI’AN University of Technology

Corresponding author: YE Fei, yefei@ncwu.edu.cn, North China University of Water Conservancy and Electric Power, Zhengzhou 450011, China

Abstract. The unsteady motion and formation mechanism of the intake vortexes are studied by numerical simulations, covering the flow distribution, vortex form, influencing factors, and others. Based on the physical model test, the large eddy simulation (LES) is used in the 3-D simulation of the intake flow. The occurrence of intake vortex presents a coherent characteristic of “dormant period” and “active period”, which is approximately consistent with that of the surface fluctuation. According to the turbulent cascade theory, the process of energy balance of the intake vortex is studied. The simulation results compare favorably with that from experimental correlations.

1. Introduction
The intake vortexes, with a high occurrence in the intake of a hydraulic structure, can be described roughly in terms of two forms: steady vortexes, whose movement elements keep time-invariant, and unsteady vortexes, whose corresponding elements change frequently with time, rendering the characteristics of frequent variation in size and location, short duration and random occurrence, etc. However, in most current researches these unsteady attributes are always ignored (to be considered as steady vortexes) for simple analysis of the spiral velocity, which leads to a great loss of vortex-motion information[1]. In fact, the unsteady characteristics of intake-vortex motion are very important for hydraulic design and operation. For example, under the inflow conditions, different kinds of vortexes including surface-concavity vortexes, solid-entrained vortexes, air-suction vortexes and so on, may generate with great unsteady attributes to create problems of reduced flow quantity, noises, cavitation, vibration or even destruction of the trashracks, etc.[2] According to the correlation results from prototype observations or physical model tests, many domestic hydraulic projects as MAN Wan, LONG Yang-xia, HUANG Tan-kou, SHI Tou-he, ZI Ping-pu, etc., have a record of harmful intake vortexes, which also can be found in projects in Japan, American, Russia, Canada, Britain, India and so on[3]. Ideally, one would like to perform a physical model test to predict the unsteady vortex motion in the prototype, however, model similarity about vortex motion via the gravity law has always been discussed and no convincible results have been proposed recently.

For these years, such flows with unsteady vortexes present severe physical and numerical modeling challenges, particularly in engineering use, such as the works did by Gao et al.[4], who studied the flow characteristics of surface vortexes in four different intakes by physical model test, to achieve results that under the pumped conditions, random funnel-shaped surface vortexes generate and increase
obviously with the flow quantity at the dead water level. Huang et al.\textsuperscript{[5]} studied the effects of the anti-vortex beam on the generation of surface vortex at different working conditions from normal water level to dead water level, indicating that under no anti-vortex beam conditions, more surface vortexes generate and some may even develop to harmful air-suction vortexes with increasing flow quantity. Currently, however, under the limitations of measuring means, most studies today about the unsteady characteristics of intake vortexes can commonly give the average locations of vortex generation and approximate range of vortex motion. Further studies about the unsteady vortex motion and internal flow structure are hampered by the absence of the whole-field-measurement implementation.

So far most studies about the formation mechanism of intake vortexes are always achieved by means of influencing-factor analysis, as results from Zielinski et al\textsuperscript{[6]} and Yildirm et al\textsuperscript{[7]}, who studied the effects of viscosity and surface tension on the generation of intake vortexes. Quick et al\textsuperscript{[8]} studied the formation mechanism of intake vortexes by physical model test, indicating that the vortex formation significantly rests with the flow conditions, Froude number, and boundary conditions. In summary, the main influencing factors can be described as follows: flow conditions (velocity, flow quantity, water level, etc.), boundary conditions (structure form and dimension, wall roughness, etc.) and fluid conditions (viscosity, surface tension, etc.). The main influencing parameters include Froude number, Reynolds number, Weber number, circulation parameter and submerse depth. Apparently, accurate flow-field information of the intake is very important for the investigation of vortex generation. During the process of vortex generation and development, energy transmission happens continuously between different vortexes, so the viscosity effects must be taken into account in the analysis of vortex-formation mechanism. According to the Attenuation theory, Schlichtingh et al.\textsuperscript{[9]} acquired the tangential velocity of a vortex based on the fact that a vortex can not maintain itself without more energy supply because of viscosity-dissipation effects. The Turbulent Cascade theory indicates that, the large scale vortexes acquire energy from the mean flow and then transport it to other smaller-scale vortexes, which will finally be dissipated by kinetic viscosity in the micro-scale vortexes. Based on this theory, the author researched the vortex formation mechanism via the energy-balance analysis, to show that the coherent attributes in the process of vortex generation and development are closely associated with the turbulence characteristic parameters\textsuperscript{[10]}. Nowadays, further study about the flow mechanism and internal flow structure is very important for the optimization design of hydraulic structures. Direct numerical simulation (DNS) can present the whole information of the flow field. However, limited by the computational techniques, DNS of the large range of scales spanned by the ensuing dynamics is impractical at high Reynolds numbers, rendering its application to practical flows with complex boundary conditions infeasible\textsuperscript{[11]}. For this reason, nowadays large eddy simulation (LES) is pursued as an alternative, in which the large scale eddies are calculated by the Navier-Stokes equation, and the effects of small scale eddies are implemented via an appropriate sub-grid scale (SGS) model\textsuperscript{[12]}. In the complex wall-bounded flow simulations, LES performs far excellent than RANS by giving detail information of turbulent fluctuation at acceptable computational cost\textsuperscript{[13]}. Therefore, in this article the hydraulic characteristics of intake vortexes are studied by LES method, including vortex forms, motion behavior and formation mechanism.

2. Numerical model

2.1. Control equations

The LES method is used in this paper, in which the large scale eddies are calculated by the Navier-Stokes equation, and the effects of small scale eddies are implemented via an appropriate sub-grid scale (SGS) model. The control equations are acquired from the N-S equation of incompressible fluid via the Gauss filter\textsuperscript{[14]} as follows:

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{1}
\]
\[
\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \left( \overline{u}_i \overline{u}_j \right)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \right] \tag{2}
\]

Where \( u_i \) is the velocity, \( \nu \) is the coefficient of kinematic viscosity, \( \rho \) is the density, \( p \) is the pressure, \( \mu_t \) is the coefficient of sub-grid eddy viscosity specified using the Smagorinsky-Lilly model as follows:

\[
\mu_t = (fC_s \Delta) \left| \mathbf{S} \right| \tag{3}
\]

Where \( C_s \) is the model constant, \( C_s = 0.1 \), \( \left| \mathbf{S} \right| = \sqrt{2 \mathbf{S}_{ij} \mathbf{S}_{ij}} \), \( \Delta = \sqrt{\Delta x \Delta y \Delta z} \), \( \Delta x \), \( \Delta y \), \( \Delta z \) are the mesh dimensions at \( x \), \( y \) and \( z \) directions respectively, \( f \) is the grid aeolotropic parameters specified as follows\[15\]:

\[
f = \cos h \frac{4(\ln^2 a_1 - \ln a_1 \ln a_2 + \ln^2 a_2)}{27} \tag{4}
\]

Where \( a_1 = \Delta_1 / \Delta_3 \), \( a_2 = \Delta_2 / \Delta_3 \).

The free surface is an interface of water and air, at which both the dynamic and kinematic conditions should be satisfied as follows\[16\]:

\[
\frac{\partial h}{\partial t} = w' - u' \frac{\partial h}{\partial x} - v' \frac{\partial h}{\partial y} \tag{5}
\]

Where \( u' \), \( v' \), \( w' \) are the flow velocity at longitudinal, transverse and vertical directions, the local pressure is used at the surface, \( p_x = p_u \).

The finite volume method is employed to discretize the control equations with the Admas-Bashforth scheme for temporal differentiation, the second-order upwind scheme is used for advection term and the second-order central difference scheme is used for diffusion term. The PISO algorithm is applied for the pressure-velocity coupling\[17\].

2.2. Boundary conditions

The upstream cross section 30\( d \) from the intake is the inflow boundary condition given by the average flow velocity with hydrostatic pressure distribution, \( \partial p / \partial x = 0 \). The Sommerfeld open boundary is used as the outflow boundary condition, \( \partial \Phi / \partial t + U_{con} (\partial \Phi / \partial x) = 0 \), where \( \Phi \) is either physical quantity of \( u \) and \( \rho \), \( U_{con} \) is the weighted average value of the longitudinal velocity along the abscissa axis. The free surface is treated as a pressure boundary given by the local pressure.

Based on the no-slip boundary condition, the particle velocities in all directions need to be zero at the wall. However, this treatment is accurate only when fairly fine meshes are used to resolve the bottom boundary layer. Alternatively, the wall function\[18\] is introduced to relate the boundary variables to the flow region as follows:

\[
\tau_w = \begin{cases} 
\frac{2 \mu \overline{u}_p}{\Delta y} & \left| \overline{u}_p \right| > \frac{\mu A^{2/(1-B)}}{2 \rho \Delta y} \\
\rho \left[ \frac{1-B}{2} \right] A^{(1+B)/(1-B)} \left( \frac{\mu}{\rho \Delta y} \right)^{(1+B)} + \frac{1+B}{A} \left( \frac{\mu}{\rho \Delta y} \right)^B \left| \overline{u}_p \right| & \left| \overline{u}_p \right| \leq \frac{\mu A^{2/(1-B)}}{2 \rho \Delta y}
\end{cases} \tag{6}
\]

Where the \( \tau_w \) is the wall shear stress, \( \rho \) is the density, \( \mu \) the dynamic viscosity coefficient, \( \overline{u}_p \) is the sub-layer velocity, \( \Delta y \) is the mesh dimension at \( y \) direction, \( A \) and \( B \) are constants, \( A = 8.3 \), \( B = 1/7 \).
The calculation domain is shown in Fig. 1, where \( x \times y \times z = 30d \times 5d \times 10d \), \( d \) is the width of the intake. The hexahedral structure mesh is used in the whole simulation field using an adaptive-mesh refinement strategy at the near-wall region. The dimensionless time step is 0.005.

![Simulation region](image)

**Fig.1 Simulation region**

### 3. Results and discussion

Three different working conditions with relative turbulent intensity from 0.05 to 0.22 are studied in this paper as shown in Tab.1.

| Conditions | Relative submerse depth of the intake \( s/d \) | Relative turbulent intensity \( \sqrt{u''}/U \) | Turbulent Reynolds number \( Re = \sqrt{u''} R/\nu \times 10^3 \) |
|------------|--------------------------------|-----------------|------------------------|
| 1          | 1.5                            | 0.05            | 0.82                   |
| 2          | 1.5                            | 0.15            | 2.45                   |
| 3          | 1.5                            | 0.22            | 3.66                   |

Annotation: where \( s \) is the submerse depth above the intake center, \( d \) is the intake diameter, \( \sqrt{u''} \) is the average turbulent intensity, \( U \) is the average inflow velocity, \( R \) is the intake hydraulic radius, \( \nu \) is the kinematic viscosity, \( \nu = 0.919 \times 10^{-6} m^2/s \).

Fig.2 presents the instantaneous flow streamlines near the intake, which shows the process of vortex generation and development. First, in the region relatively far from the breast wall (>1.8m), due to the even-distributed incoming flow under symmetric boundary conditions, only a minority of flow separations and unsteady vortexes can be found. Second, in the region at the side-wall corner, more concavity vortexes generate because the flow streamlines curve with the wall boundary to create more small surface circulations. Third, in the region near the top of the intake, caused by the uneven contraction of the flow to the intake, more surface circulations generate leading to a high probability of vortex generation. Mostly, these vortexes can not maintain themselves without continuing provided energy and will so turn to be destructed by other flows. However, at given submerse depths, some vortexes may be combined with others to reinforce to a new larger-scale vortex during the process of floating to the intake. Generally, these large-scale vortexes happen at the top of the intake always induce other new associated vortexes to present a characteristic of paired generation.
Simulation results shows that the region relatively far from the intake (>1.8m), low-intensity surface-concavity vortexes may generate with shallow low-level-rotation tails. Generally these vortexes are not persistent, which may easily be destructed or divided by other flows to present a short-duration characteristic. In the region near the intake, the through-type air-suction vortex may generate with an air-core in the center to be sucked into the intake. These vortexes are mostly developed from the surface-concavity vortexes when they float to the intake and can persist for a while with increasing rotational velocity.

Fig.3 shows the LES results of turbulent intensity under working condition 3, which compare favorably with the results from experimental correlations.

4. Conclusions
In this article, the unsteady motion and formation mechanism of intake vortexes is studied by numerical simulations. First, the vortex forms and flow characteristics are studied. In the region relatively far from the breast wall (>1.8m), only a few low-intensity surface-concavity vortexes can be found with shallow low-level-rotation tails. Generally, these vortexes are not persistent, which may easily be destructed or divided by other flows to present a short-duration characteristic. In the region at the side-wall corner, more concavity vortexes generate because the flow streamlines curve with the wall boundary to create more small surface circulations. In the region near the top of the intake, caused by the uneven contraction of the flow, more surface circulations generate leading to a high probability of vortex generation. At given submerse depths, some vortexes may be combined with others to reinforce to a new large-scale through-type vortex with an air-core in the center. These vortexes are mostly developed from the surface-concavity vortexes when they float to the intake and can persist for a while with increasing rotational velocity.
References

[1] CHEN Yun-liang, WU Chao and YE Mao et al. Hydraulic characteristics of vertical vortex at hydraulic intakes[J]. Journal of Hydrodynamics, Ser. B, 2007,19(2):143-149.

[2] DANG YUAN-yuan. Research of the vortex characteristics at dissymmetrical boundary intake[D], Master's degree thesis, Nanjing: Nanjing Hydraulic Research Institute, 2008(in Chinese).

[3] YE Fei. Unsteady vortex motion and formation mechanism[D], Ph.D. Thesis, Tianjin: Tianjin University, 2010(in Chinese).

[4] GAO XUE-ping, ZHANG Xiao-xian and LI Chang-liang et al. The hydraulic characteristics at vertical pipe intake of Xilongchi pumped storage plant[J]. Journal of Hydroelectric Engineering, 2002, 76(1): 52-60.

[5] HUANG Zhi-min, HE Xiao-hui and ZHU Hong-hua et al. Model test research of the lower reservoir inlet/outlet for Guangzhou pumped storage plant[J]. Water Resources and Power, 2005, 23(1):4-7.

[6] ZIELINSKI P. B. and VILLEMONTE J. R. Effect of viscosity on vortex-orifice flow[J]. Journal of the Hydraulics Division, ASCE, 1968, 94(3): 745-752.

[7] YILDIRIM N. and JAIN S. C. Surface tension effect on profile of free vortex[J]. Journal of the Hydraulics Division, ASCE, 1981, 107(1): 132-136.

[8] QUICK M. C. Efficiency of air-Entraining Vortex Formation at Water Intake[J]. Journal of the hydraulics division, ASCE, 1970, 96(7):1403-1416.

[9] SCHLICHTING H. Boundary layer theory [M]. New York, N.Y: 7th Ed., McGraw-Hill Book Co., 1979.

[10] DU Min. Formation mechanisms and scale effects of hydraulic intake vortices[D]. Ph.D. Thesis, Tianjin: Tianjin University, 2008(in Chinese).

[11] J. Xu. Benchmarks on tera-scalable models for DNS of turbulent channel flow [J]. Parallel Computing, 2007, 33(12):780-794.

[12] YE Fei, GAO Xue-ping. Numerical simulations of the hydraulic characteristics of side inlet/outlets[J]. Journal of hydrodynamics, Ser. B, 2011, 23(1): 48-54.

[13] YOSHIIHIDE T., AKASHI M. and SHUZO M. et al. Comparison of various revised k-ε models and LES applied to flow around a high-rise building model with 1:1:2 shape placed within the surface boundary layer [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2008, 96(4):389-411.

[14] J. W. DEARDORFF. A numerical study of three-dimension turbulent channel flow at large Reynolds numbers[J]. Journal of Fluid Mechanics, 1970, 41 (2): 453-480.

[15] SCOTTI A., MENEVEAU C. and LILLY DK. Generalized Smagorinsky model for anisotropic grids[J]. Physics of Fluids A, 1993, 5(9): 2360-2380.

[16] C. W. Li, L. H. Yu. Hybrid LES/RANS modelling of free surface flow through vegetation[J]. Computers & Fluids, 2010, 39(9): 1722-1732.

[17] CHENG Zeng, C. W. Li. A hybrid RANS-LES model for combining flows in open-channel T-junctions[J]. Journal of Hydrodynamics, Ser. B, 2010, supplement 1, 22(5):154-159.

[18] ALEXANDER O., HANS W. Numerical simulation(DNS and LES) of manipulated turbulent boundary layer flow over a surface-mounted fence[J]. European Journal of Mechanics-B/Fluids, 2000, 19(5): 765-788.