Research on overflow diffusion characteristics of dredger

Mingqi Shao1, a, Jin Wang2, *, Xuewen Ding1, b

1School of Energy and Power Engineering, Wuhan University of Technology, Wuhan 430063, China
2Rizhao Polytechnic, Rizhao, China

*Corresponding author e-mail: 568857508@qq.com, a1014232997@qq.com,
b1370533273@qq.com

Abstract. Dredging engineering is widely used in port and waterway construction, water conservancy facilities construction, environmental protection dredging, land reclamation, subsea tunnel excavation and deep-sea mining, among which drag suction dredger is an important tool for dredging operation. During the process of its construction, the mud overflow process is an important link to realize the mud treatment of excavation. According to the actual engineering conditions, the loading and overflow process is an important part of the treatment of excavated slurry. With the increasing requirements of environmental protection, the related ecological pollution caused by dredging has become a hot spot today, and the diffusion of overflow slurry is one of the sources of pollution generated by dredging suction dredgers. Therefore, combining with actual engineering conditions, this paper adopts the theory of computational fluid dynamics (CFD) to study the diffusion characteristics, diffusion range, and influencing factors and proposes construction strategies to reduce its pollution impact.

1. Introduction
Dredging engineering is a kind of engineering which is carried out by man or dredger for underwater excavation, widening and deepening water area. Through dredging works, the geometric boundary of river flow can be changed, resulting in the change of internal structure of water flow [1]. Under the action of the newly designed flow structure, the sediment will no longer deposit in the channel, so that the channel is stable and the environment is optimized. In the process of modern life, people pay more and more attention to the protection of the environment. Therefore, under the advocacy of environmental protection from all over the world, dredging methods are widely used in various regions to ensure the cleanness and stability of the river [2].

As the main carrier of dredging operation, dredging ships are widely used in port construction, dredging maintenance of ship channel, reclamation of land and other projects [3]. Among many kinds of dredgers, the most widely used one is the trailing suction dredger. Trailing suction dredger belongs to self-propelled and self loading large-scale dredger, which is suitable for construction in open waters, such as seaports, bays and rivers, due to its high mobility, wide application range and large loading capacity. In the process of dredging, the ship's water intake is increased due to the increase of the loading capacity in the mud tank. When the dredger's draft reaches the maximum allowable value, by reducing the height of the overflow tank, the low-density mud water mixture in the upper layer of the
mud tank can be discharged out of the tank, so as to reduce the loading capacity of the dredger and realize the continuous operation of the dredger under the maximum draft condition [4]. However, in this process, the spilled mud water mixture will form an overflow plume, which is the main source of pollutants produced by trailing suction dredgers in dredging operations. The overflow plume will pollute the water area of the construction area, so the research on the diffusion mechanism and diffusion characteristics of the overflow plume is the key to the environmental dredging.

The dimensional analysis method uses the basic characteristic parameters for dimensional analysis, that is, the characteristic length, characteristic flow rate and fluid density are regarded as the basic parameters. Starting from the factors that affect the mixing and dilution of the diffuser jet in the near area, such as the environmental flow conditions, the initial parameters of the nozzle jet and the characteristic parameters of the diffuser, the flow characteristic parameters in the near area are analyzed. The parameters after analysis and combination will be dimensionless, and then the secondary factors will be ignored according to the applicable conditions of the jet, and finally the parameters needed to be measured in the experiment will be obtained. Anthony and Willmarth [5] use this method to study the turbulent characteristics of circular jet under free water surface, and analyze the shallow surface flow formed by jet. It is found that the radial range of surface flow is significantly larger than that of main jet. Wallace and Wright [6] also use this theory to establish a simple equation through dimensional analysis, and predict the position and concentration of the two-dimensional buoyant jet when it starts horizontal diffusion along the axis. Dimensional analysis is a relatively original analysis method, its analysis process is more complicated, and the accuracy of the model is also low, but its birth laid the foundation for a series of follow-up studies.

Momentum integral method is the most commonly used method to solve jet problems in engineering. Its idea is based on the assumption of flow self similarity, using the mass conservation, momentum conservation and energy conservation of fluid motion, integrating the three major differential equations of fluid motion, transforming the partial differential equation into the ordinary differential equation, and using the boundary conditions to close the equation, and then combining the conditions of the definite solution to The flow characteristics of jet are predicted. Using this method, Frick [7] proposed a plume model which can predict the trajectory, dilution, width and other important flow characteristics of the plume. Based on this, Chiang and Sill [8] then put forward an entrainment model which can predict the trajectory, velocity and dilution of the jet well, and applied the model to study the different diffusion characteristics of the jet in the near flow field, the transition area and the far flow field. Using this method, Lee and Cheung [9] proposed a Lagrangian integral model which can predict the three-dimensional trajectory and average dilution of buoyant jet. In the method of momentum integration, the cross-section distribution of flow physical quantity is supposed first, but it is difficult to obtain the cross-section distribution. Therefore, this method is lack of predictability, so its application is limited.

In this paper, for the near-field area, we use solid works to model the sector area of near-field dredging, import CFD-ICEM to divide the hexahedron mesh. For the far-field area, we use the open-source data to obtain the survey data of the terrain and elevation of the research area, make the terrain file, and divide the quadrilateral structure grid. After that, we carry out the elevation interpolation for the terrain. At the same time, combined with large eddy simulation and multiphase flow model, the overflow and diffusion process of mud water mixture under the influence of different factors are simulated, and the numerical solution scheme is verified with different experimental results. The concentration distribution, diffusion range and velocity field of overflow diffusion under different conditions are analyzed to explore its diffusion law; combined with the real construction environment, the influence range of overflow diffusion under different operation conditions is discussed to guide the actual construction according to the range.

2. Mathematical model
Under the condition of incompressibility of the default liquid phase, both the suspension mud and the liquid phase are continuous media. At the same time, the movement of the mixed phase (mud) and the basic phase (water) is investigated in the Euler coordinate system. The Navier Stokes equations (N-S
equations) are established for each phase to describe. Each phase has its own conservation calculation of mass, momentum and energy. The large eddy simulation (LES) method is used to process the turbulent information of the convection field, and the VOF method is used to describe the two-phase transport.

2.1. Conservation equation

The conservation equation mainly includes mass conservation equation, momentum conservation equation and energy conservation equation.

Mass conservation equation is also called continuous equation. Its physical meaning is that when there is no mass source in the control body, the change of mass in the control body is equal to the mass flow through the boundary of the control body. Its general differential form can be written as:

\[
\frac{\partial \rho}{\partial t} = \nabla \cdot (\rho \vec{v})
\] (1)

For incompressible fluid, the density is constant, and the continuous equation is simplified as follows:

\[
\nabla \cdot \vec{v} = 0
\] (2)

The conservation of momentum is derived from Newton's second law. Its physical meaning is that the change of momentum of the control body is equal to the sum of external forces exerted on the control body of the unit. Its general differential form can be written as:

\[
\rho \frac{\partial \vec{v}}{\partial t} + \rho (\vec{v} \cdot \nabla) \vec{v} = -\nabla p + \nabla \cdot \tau_{ij} + \rho \vec{f}
\] (3)

The expression of shear stress tensor is as follows:

\[
\tau_{ij} = \mu \left[ \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) - \frac{2}{3} (\nabla \cdot \vec{v}) \delta_{ij} \right]
\] (4)

By substituting it into formula (2-3), we can get:

\[
\rho \frac{\partial \vec{v}}{\partial t} + \rho (\vec{v} \cdot \nabla) \vec{v} = -\nabla p + \mu \left[ \Delta \vec{v} + \frac{1}{3} \nabla (\nabla \cdot \vec{v}) + \rho \vec{f} \right]
\] (5)

Formula (2-5) is the vector form of the so-called N-S equation.

The equation of conservation of energy is a special form of the first law of thermodynamics in hydrodynamics. If the total energy per unit mass of a system is \(E\) and the internal energy per unit mass is \(e\), then:

\[
E = e + \frac{v}{2}
\] (6)

Taking a unit control body as the research object, the differential energy conservation equation can be written as follows:

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \vec{v} E) = \nabla \cdot (k \nabla T) + \nabla \cdot \left[ \left( -p I_{ij} + \tau_{ij} \right) \cdot \vec{v} \right] + \rho \vec{f} \cdot \vec{v} + q_T
\] (7)

Through observation, it is found that the continuity equation (mass conservation equation), momentum equation and energy equation contain five equations in total, but there are six unknowns, so...
we need to add another equation to close the system. The equation connecting \( p \) and \( \rho \) is called the equation of state:

\[
\rho = \rho(p, T)
\]  

(8)

In this way, the continuous equation, momentum equation, energy equation and state equation constitute a closed system, which is called N-S equations.

2.2. Transport equation

When the research object is two-phase flow, volume fraction function \( f \) is introduced by VOF method. The phase at time \( t \) at coordinate \( x \) can be determined by function \( F(x, t) \), which is defined as the ratio of specified fluid and grid volume in grid cell

\[
F(x, t) = \frac{V_1(x)}{V(x)}
\]  

(9)

Where \( V_1 \) is the volume of the specified fluid in the grid, in this paper, it refers to water, and \( V \) is the volume of the grid.

According to the definition of \( F \), there are three cases: when \( F=1 \), the grid is full of specified fluid; when \( F = 0 \), the grid is empty; when \( 0 < F < 1 \), the grid is the phase interface grid of two-phase flow. The fluid motion can be represented by the change of volume fraction in each grid, so the interface of two-phase flow can be traced by volume fraction function \( F \). For the liquid-liquid two-phase flow with different densities, \( \rho \) and \( \mu \) in the control equation refer to the mixture density and the viscosity of the hybrid, respectively. From the definition of \( F \), we can get:

\[
\rho = F \rho_{l1} + (1 - F) \rho_{l2}
\]  

(10)

\[
\mu = F \mu_{l1} + (1 - F) \mu_{l2}
\]  

(11)

Where the subscripts \( l1 \) and \( l2 \) represent the base phase liquid and the mixed phase liquid respectively.

According to the definition of volume fraction function \( F \), it can be seen that it is a step function, so \( \rho \) is piecewise continuous in the whole flow field.

In the flow field with velocity field \( u \), the volume fraction function \( f \) satisfies:

\[
\frac{DF}{Dt} = \frac{\partial F}{\partial t} + u \cdot \nabla F = 0
\]  

(12)

The above control equations basically describe the motion of two-phase flow and its interface completely, while the analytical solutions satisfying these partial differential equations and boundary conditions are only obtained under simple flow conditions, so they usually need to be solved numerically.

2.3. Turbulence model

At present, the main turbulence models used in CFD are direct numerical simulation (DNS), large eddy simulation (LES) and Reynolds time averaged simulation (RANS).

A physical quantity in the flow field, for example, the velocity, after low-pass filtering, becomes:

\[
u_f = \bar{u}_i + u_i'''
\]  

(13)

\( \bar{u}_i \) is the velocity of the solvable scale, \( u_i''' \) is the speed of unsolvable scale.

Assuming that the filtering process operation and derivation operation can be exchanged, the N-S equation can be filtered to obtain the following equation:
\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (14)
\]
\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} \quad (15)
\]

In order to realize LES simulation, it is necessary to construct a closed sub grid stress model. From many literatures, the key to realize Les is to construct a sub grid stress model with reasonable numerical value and accurate analysis of flow phenomena. In this paper, the sub grid eddy viscosity model is used for the closure of sub grid stress. The eddy viscosity model can be written as follows:

\[
\tau_{ij} = 2 \nu_t \bar{S}_{ij} + \frac{1}{3} \delta_{ij} \tau_{kk} \quad (16)
\]

Where \( \nu_t \) is the sub grid eddy viscosity coefficient. The Smagorinsky model is used to calculate the sub grid eddy viscosity coefficient

\[
\nu_t = C_s \cdot \Delta^2 \langle 2 \bar{S}_{ij} \bar{S}_{ij} \rangle^{1/2} \quad (17)
\]

Where \( \Delta \) is the filtering scale, \( C_s \) is the model constant, generally \( C_s = 0.183 \).

2.4. Solution strategy

In this paper, a completely conservative CFD code is used to solve the two-phase flow. The finite volume method is used to discretize the basic control equations. When solving the above equations, the separation algorithm based on pressure is used, and the simple algorithm is used to realize the coupling of pressure field and velocity field. In order to ensure the accuracy of the calculation and the stability of the solution process as well as the reasonable utilization of the calculation resources, the momentum equation adopts the discrete format of the bounded central difference (BCD), the volume fraction adopts the second-order upwind format, and the other control equations adopt the first-order upwind format. The advection terms in the TVD scheme are discretized. The TVD scheme includes the second-order upwind scheme and the first-order upwind scheme. The mixed scheme is used to control the diffusion.

When the control equation is solved, the convergence standard is that the error between the two iterations is less than 10^{-4}. The time step is 0.01 seconds, and the total step depends on the initial conditions. It ensures that the flow distance in the main flow direction is more than twice the length of the basin, so as to ensure the full development of turbulence.

3. Results and analysis

When the trailing suction dredger is dredging, the position of the overflow drum is not fixed, which may be arranged in the front or rear of the ship, resulting in the difference of the overflow release position. In this section, through the numerical simulation of the overflow diffusion process in two locations, the comprehensive influence of different release locations on the overflow mud diffusion is analyzed.

3.1. Distribution of overflow logistics to diffusion

Fig. 1 shows the distribution of the flow direction concentration of the overflow mud at different release positions when \( \zeta = 1.6 \) and \( \zeta = 0.8 \). It can be seen that when \( \zeta = 1.6 \), the concentration diffuses rapidly to the basin surface at the stern of the ship and forms surface plume. In the front section of the overflow, the overflow mud has obvious stratification along the flow direction at the overflow mouth, and in the middle and back section of the overflow, the overflow mud also has stratification along the depth direction at the back flow surface. At the end of the overflow, the concentration distribution tends to be
stable, the stratification of the flow direction gradually disappears, and the longitudinal stratification is still obvious, and the overall longitudinal impact range of the main flow area is \( Y/D = 10 \sim 35 \). When the overflow mud is released at the front of the ship, the concentration distribution is obviously different. In the process of releasing the overflow mud from the overflow port to the stern of the ship, the flow pattern in the main flow area is stable, almost no vertical stratification occurs, the concentration gradient is large, the concentration stratification in the flow direction is obvious and the concentration gradient is small. When the overflow continues to flow downstream through the stern, there is a significant concentration stratification in the longitudinal direction, and the concentration gradient gradually decreases, and the trend of surface plume formation increases.

\[
(1) \quad \zeta = 1.6
\]

\[
(2) \quad \zeta = 1.6
\]
Fig. 1 Flow direction concentration distribution of overflow mud under different working conditions and release positions

It is found that no matter the release position of overflow is at the back or the front of the ship, after the overflow mud flows through the stern, the concentration appears longitudinal stratification, and there is a trend of surface plume on the basin surface. This phenomenon is caused by the effect similar to the cylinder turbulence. The existence of the hull makes the cross flow encounter obstacles at the initial stage of the flow. With the flow going on, the flow will form local eddy current at the stern along with the disappearance of the obstacles, which will enhance the disturbance in the longitudinal direction, and promote the overflow to the surface of the drainage basin. In view of this reason, there are different degrees of stratification at the stern of the ship.

The difference is that when overflow is carried out at the rear end of the ship, the overflow process will transition to the rear of the ship before fully mixing with the cross flow, and the jet state of overflow is not fully formed. At this time, the flow is greatly affected by the eddy current, so the stratification is carried out rapidly, forming a large number of surface plumes. When overflow is carried out at the front of the ship, the overflow will encounter the eddy current influence after flowing for 40d. At this time, the overflow and cross flow have fully acted to form a relatively stable flow state, while the turbulence disturbance of the same strength is not enough to completely destroy the flow state, so a small amount
of mud is rolled up, the mud concentration is stratified in the longitudinal direction, and the surface is formed at a further distance from the stern Plume.

In order to further quantitatively analyze the distribution characteristics of overflow mud at this time, as shown in Fig.2 and 3, when \( \zeta = 1.6 \) and \( \zeta = 0.8 \), the concentration distribution curve of overflow flow direction at different sections shall be analyzed.

![Fig. 2 Flow direction concentration curve of different sections (\( \zeta = 1.6 \))](image1)

![Fig.3 Flow direction concentration curve of different sections (\( \zeta = 0.8 \))](image2)
It can be seen that when $\zeta = 1.6$, the overflow mud overflowed from the back end is mainly concentrated in the front and middle section of the drainage basin, and the maximum sediment concentration of the mud at $x / D = 70$ reaches 3800 mg/m$^3$, while after entering the last section of the drainage basin, the mud is rapidly diluted, and the maximum sediment concentration of the mud at $x / D = 110$ is close to 750 Mg/m$^3$, and the depth of the maximum value is $Y / D = 30d$, the longitudinal influence range is up to 35d, and the settlement trend of the overflow is obvious in the later part of the basin. When the front-end overflows, the mud dilution is not obvious, and the concentration gradient is small. The sediment concentration of the mud from $x / D = 70$ to $x / D = 110$ only decreases by about 33%. Moreover, at $x / D = 110$, the depth of the maximum sediment concentration is $Y / D = 22.5d$, and the longitudinal influence range is about 27.5d. The comparison shows that the front-end overflow is less affected by the hull, and still maintains a relatively good overflow shape at the end of the flow, while the back-end overflow is on the contrary.

When $\zeta = 0.8$, the concentration distribution law is similar to that when $\zeta = 1.6$, which indicates that whether dredging is carried out at high or low crossflow speed, the influence of overflow location on the diffusion of overflow exists. The difference is that when $\zeta = 0.8$, the range of the maximum local sediment concentration of the overflow mud decreases, and the longitudinal position of the maximum concentration moves downward, which indicates that the turbulence effect of the cross flow is weakened, and the mud has been fully diffused in the front of the drainage basin. It is worth noting that the overall concentration distribution of the back-end overflow is uneven and there is only a small amount of mud in the back-end of the basin, while the overall concentration gradient of the front-end overflow is gradually smaller and the diffusion amplitude remains stable. This shows that the flow is strongly influenced by $\zeta$ when the back-end overflows. In the case of a small $\zeta$, almost no overflow will diffuse to the far-field, while the front-end overflows on the contrary.

3.2. Radial diffusion distribution of overflow
In order to further explore the influence of different release positions on the overflow, the radial distribution of the concentration of the overflow was intercepted and analyzed. Among them, figure 36 shows the radial distribution of concentration of overflow mud at different release positions at different sections when $\zeta = 1.6$; figure 37 shows the radial distribution of concentration of overflow mud at different release positions at different sections when $\zeta = 0.8$. In the Fig.4, C represents the local mud concentration and Cmax 'represents the maximum mud concentration in the basin.
When $\zeta = 1.6$, it can be seen in the Fig.5 that the profile of the main flow area of the overflow mud released from the back end and the front end is obvious, and under the double effects of cross flow and overflow, a double peak kidney shaped concentration distribution is formed in the radial direction. The difference is that when the back-end overflows, the main flow area will be separated from the basin surface later. When $x / D = 70$, there is overflow mud on the basin surface, and the influence range is about $Z / D = -10 \sim 10$. But when the front-end overflows, there is no surface plume in the main flow area, and the radial influence range is slightly wider than the back-end overflows. This shows that the back-end overflow is greatly affected by the hull, the main flow area of the overflow is deformed and develops vertically to form a surface plume, while the front-end overflow is less affected. When the flow reaches $X / D = 110$, the radial influence range of the back-end overflow is about $Z / D = -20 \sim 20$, while the front-end overflow is fully developed in the radial direction, and the influence range reaches $Z / D = -30 \sim 30$. In addition, it is also noted that the concentration center of the two peaks of the back-end overflow moves downward, and the concentration gradient along the longitudinal direction is small, while the two peaks of the front-end overflow remain in the central line of the mainstream area. The concentration stratification is more uniform. These show that the downstream overflow is affected by the upstream flow disturbance. At the end of the flow period, the stability of the main flow area cannot be fully guaranteed, and the radial development is not sufficient. The main diffusion exists in the longitudinal direction, while the upstream overflow is less affected by the hull, which can maintain a stable flow pattern in the whole flow period, and the concentration develops stably in the longitudinal and transverse direction.
(1) $x/D=70$

(2) $x/D=70$
When $\zeta = 0.8$, there is a similar rule with $\zeta = 1.6$. The difference is that at this time, the influence of cross flow is small, and the flow effect of overflow is prominent. Therefore, at $x / D = 70$, the radial influence range of concentration of front and rear overflow is diffused, and the longitudinal position of double peak center is more in-depth. At the end of the flow, the overflow will settle and the flow form will deform, but at this time, the longitudinal influence range of the back-end overflow is obviously larger than that of the front-end overflow, and the concentration distribution is more dispersed, which further shows that the back-end overflow is greatly disturbed by the longitudinal disturbance, and the dispersion is more sufficient in the basin.

Fig.5 Flow direction concentration curve of different sections ($\zeta$=0.8)
4. Conclusion
In this paper, a three-dimensional model is established based on the working condition of the trailing suction dredger in the actual dredging operation. The scope and characteristics of overflow mud diffusion under the influence of overflow position, overflow frequency and propeller at the stern of the dredger are studied. The following conclusions are drawn:

1) The release position has a certain impact on the overflow diffusion process. More surface plumes are formed and the longitudinal diffusion range is increased in the overflow at the rear end of the ship. However, the overflow flow structure at the front end of the ship remains intact and the longitudinal influence range is small, so there will not be too many surface plumes, and this trend is more obvious at low $\zeta$. Therefore, in the actual construction operation, it is necessary to select the overflow cylinder arranged at the front of the ship for overflow release.

2) The existence of release frequency will affect the flow pattern of overflow, increase the influence range of overflow diffusion and form more surface plumes, and this trend is more obvious in front overflow and low $\zeta$ condition. The release frequency of the overflow will cause the longitudinal fluctuation disturbance of the drainage basin. Under the interaction with the cross flow, the slurry groups of different sizes will form and move forward in a fluctuating way along the flow direction. At the same time, the existence of the fluctuation will aggravate the longitudinal disturbance in the main flow area and increase the overflow diffusion range. Therefore, the dredging operation should be carried out in a calm sea area as far as possible to reduce the impact of wave induced release frequency.

3) The existence of propeller will enlarge the longitudinal diffusion range of overflow mud and generate more surface plumes, which is not conducive to environmental dredging. The existence of propeller makes the flow in the nearby drainage basin disturbed by additional momentum, and then affects the diffusion pattern of overflow mud in the drainage basin. The entrainment of the propeller makes the low concentration mud separate from the main flow area, and forms the surface plume when it diffuses longitudinally and extends to the basin surface. The surface plume is more obvious under the low $\zeta$ condition. In addition, compared with the release position and release frequency, the propeller has a greater impact on the formation of surface plume. Therefore, in the actual construction operation, it is necessary to adopt low navigation speed construction, that is, to reduce the impact of the propeller, so as to carry out environmental protection operation.

Acknowledgments
This work supported by self-determined and innovative research funds of WUT.

References
[1] Camassa. R, Lin. Z, McLaughlin. RM, et.al. Optimal mixing of buoyant jets and plumes in stratified fluids: theory and experiments [J]. Journal of Fluid Mechanics, 2016, 790: 71-103.
[2] Ata Bilgili, Jeffrey A. Proehl, M. Robinson Swift. Dredging for dilution: A simulation based case study in a Tidal River [J]. Journal of Environmental Management, 2016, 167: 85-98.
[3] S Jarrell Smith, Carl T Friedichs. Size and settling velocities of cohesive flocs and suspended sediment aggregates in a trailing suction hopper dredge plume [J]. Continental Shelf Research, 2011, 31(10): S50-S63.
[4] Lynyrd de Wit, AM Talmon, Van Rhee. 3D CFD simulations of trailing suction hopper dredger plume mixing: Comparison with field measurements [J]. Marine pollution bulletin, 2014, 88(1-2): 34-46.
[5] D. G. Anthony, W. W. Willmarth. Turbulence measurements in a round jet beneath a free surface [J]. Journal of Fluid Mechanics, 2006, 243(-1): 699-720.
[6] Wallace. R. B, Wright. S. J. Spreading layer of two-dimensional buoyant jet [J]. Journal of Hydraulic Engineering, 1984, 110(6): 813-828.
[7] Frick, Walter.E. Non-empirical closure of the plume equations [J]. Atmospheric Environment, 1984, 18 (4): 653-662.
[8] Chiang H. C., Sill B. L. Entrainment models and their application to jets in a turbulent cross flow
[J]. Atmospheric Environment 1985, 19 (9): 1425-1438.

[9] S. K. B Cheung, D. Y. L Leung, W Wang, et.al. VISJET-a computer ocean outfall modelling system [C]. 2000: