Journal of Mechanical Engineering Research and Developments
ISSN: 1024-1752
Website: http://www.jmerd.org
Vol. 38, No. 2, 2015, pp. 83-90

The Internal Flow Induced Deflections of Ocean Outfall Pipeline

C. X. Wang[†], F. Feng[†*], J. J. Tu[†], & L. Raissouni[‡]

[†]Collage of Shipbuilding Engineering, Haerbin Engineering University, Harbin, Heilongjiang, China
[‡]Faculty of Sciences and Technologies, Abdelmalek Essaadi University, Tangier, Morrocco

ABSTRACT: The structural safety of the ocean outfall pipeline is the critical factor to insure the waste water can be transferred to the depopulated area, the equations of deflections induced by the internal flow on the pipeline have not yet formulated in executive designing standard of China. The numerical model was based on the ocean outfall engineering in Yantai, China. To develop the deflections induced by internal flow, the Reynolds-Averaged Navier-Stokes (RANS) model of computational fluid dynamics (CFD) was applied in numerical simulation. The loads on the pipe were treated theoretically and the deflections of different cross sections were computed, the maximum deflection occur at the junction section and the deflections of pipeline which installed on the sea bed approximately equal $2\pi PR^3 / Et$, the approximate equations for designing in engineering were derived.

KEYWORDS: Cfd; Turbulence; Pipeline; Numerical simulation; Deflection.

INTRODUCTION

The compatibility of earth goes insufficient for the current developments of human activities, the lack of surface runoff to discharge the sewage is challenging the littorals. Since the compatibility of ocean significantly exceeds the earth, some coastal cities began to develop the ocean discharging engineering from 1980s [1-3]. By discharging wastewater into the open seas, the receiving water has been expanded to a great extent, on the other hand, the interference of the pollution on human activity areas could be well confined. As the number of the ocean outfall engineering creases, China issued the technical specifications for sewage pipeline discharging into sea [4]. However, the calculating method for the deflection induced by the fluid flow on the pipeline is not clearly rationalized, which may perplex the engineers. In April 2007, the ocean outfall pipeline of Yantai, China, was fractured and caused pollution in a wide area. To ensure the diffusers a long term safe operation and expecting to be steady in their entire operational life, the deflections predictionwould be a suggestive research, to develop the fundamental equations for the basic designing is essential as well.

The structural safety of the ocean outfall pipeline plays a crucial role in discharging the wastewater into the depopulated area,and leads to the intensive economic of engineering and the design life as well, consider the high velocity and the vast quantity of the internal flow, the primary loads on the pipe due to it may cause damage to the pipe. The internal pipe flow in offshore/ocean engineering is a complicatedstudy which have been emphasized in recent 20 years due to its specific characteristics. High Reynolds number turbulence will occur in the large designed size pipe [5, 6], hereby a suitable numerical method for creating a simulation of flow with high accuracy turns to be the key to resolve the problem. In 1990s, a majority of experimental data was collected by scientists [7, 8], the relevant theoretical and empirical methods of computational fluid dynamics (CFD) were also developed rapidly. To describe the high Reynolds number turbulence, direct numerical simulation (DNS)is extremely challenging under the condition of high Reynolds number because of the limited of the computational capacity we have ever created so far [9]. Boersma succeed with DNS of pipe flow at Re=61000 [10], but still inapplicable compared with Re=10^6~10^7 in engineering.Hereby the Reynolds Averaged Navier-Stokes (RANS) model is applied into this particularly elusive challenge when we are normally interested in knowing just a few quantitative properties of a turbulent flow, such as the forces on a body. With this averaged method, the Reynolds number was expanded from 10^2 to 10^7 [11-13], these foregoing works provide more possibilities to resolve the deflections induced by the large pipe flow.

In this article, the analysis embraces two parts, loads and deflection. We obtain the loads by numerical simulation, then treat the results theoretically, the equations to describe the deflections of the pipeline can be formulated accordingly.

NUMERICAL SIMULATION AND ANALYSIS

Flow Model and Governing Equations

All the simulation works based upon the ocean outfall engineering of Yantai, China. In this engineering the driving force for the pipe flow is gravity, the water surface height in sedimentation tank is about 5 m, the diameter of the discharging pipe is 1.6 m, thepipe material is steel, and the thickness is 0.1 m [14]. Owing to the seafloor of offshore declines quite gently, the pipeline model was set in horizon [15]. Thehydrostatic pressure is not under our consideration in this article, hence the height of the sea water was not modeled, the fall between the inlet and the sea surface is about 10 m [16]. The scale of the model sketched in Figure 1. The 3-Demontional model discharging pipelineand the meshing works were shown in Figure 2 and Figure 3.

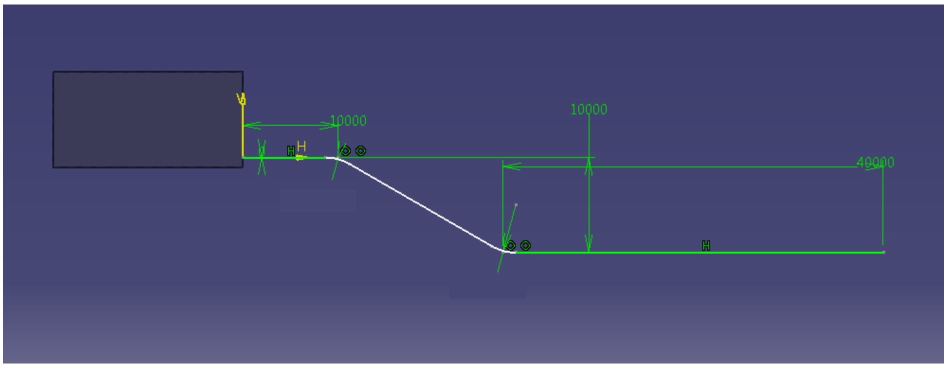

Figure 1. The scale of the model.

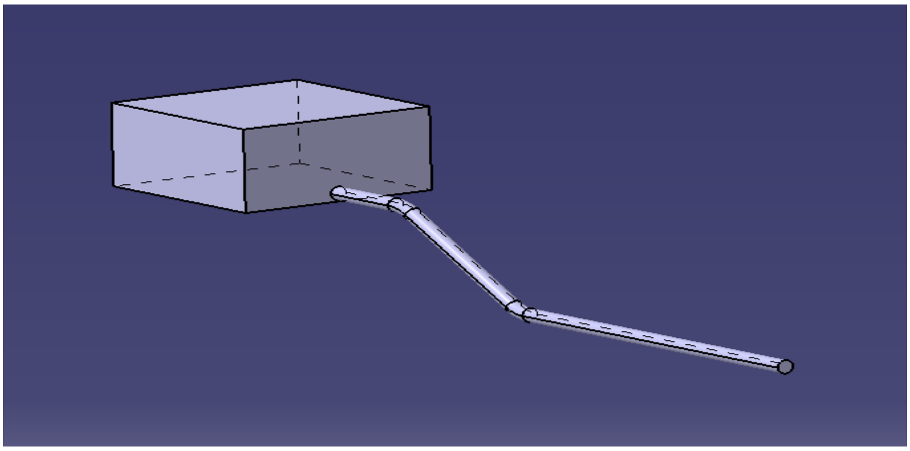

Figure 2. The 3-Dimensional model.

We consider the pressure-driven incompressible flow of a viscous Newtonian fluid in a smooth circular pipe where the governing equations are the time-dependent Navier–Stokes equations given by:

$$\nabla \bullet v = 0 \tag{1}$$

$$\frac{\partial}{\partial t} + (\nabla \cdot v)v = -\nabla P + \frac{1}{\mathrm{Re}}\nabla^2 v \tag{2}$$

Here, Re is the Reynolds number defined as $2Rv/\mu$, where R is the pipe radius, v is the velocity and μ is the kinetic viscosity. In a statistically steady flow, every variable can be written as a sum of time-averaged value, if assuming the averaging interval of two eddies occur is large enough, the averaged vortex $\overline{\varnothing}$ does not depend on the time t. Equation for describing $\overline{\varnothing}$ therefore can be written as:

$$\overline{\varnothing}(x_i) = \lim_{T\to\infty}\lim_{T\to\infty}\frac{1}{T}\int_0^T \varnothing(x_i,t)\,dt \tag{3}$$

Then the controlling equations (2), (3) can be formed into RANS model as:

$$\frac{\partial\left(\rho \bar{v}_i\right)}{\partial x_i}=0$$

(4)

$$\frac{\partial\left(\rho \bar{v}_i\right)}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho \bar{v}_i \bar{v}_j+\rho \overline{v_i^{\prime} v_j^{\prime}}\right)=-\frac{\partial \bar{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{v}_i}{\partial x_j}+\frac{\partial \bar{v}_j}{\partial x_i}\right)\right]$$

(5)

the shear standard $k-\varepsilon$ turbulence model is used for modeling this case, it was found that the standard $k-\varepsilon$ gives a good prediction of high Reynolds turbulence[17]. The boundary conditions are as follows:

(1) For pipe flow, the inlet velocity conditions are set as

$$U^{+}=\frac{1}{\kappa}\ln y^{+}+B \tag{6}$$

$U^{+}=U/u_{\tau}$, where U is the mean streamwise velocity, u_{τ} is the shear stress velocity, the von Kármán constant κ and the additive constant B were studied by Bailey et al in 2014, they confined κ to be 0.4 ± 0.02 [18]. The turbulent kinetic energy k and turbulent dissipation rate ε are given by:

$$k==\frac{3}{2}\left[0.16(\mathrm{Re})^{-1/8}\,\bar{v}\right]^{2} \tag{7}$$

$$\varepsilon=C^{3/4}\,\frac{k^{2/3}}{0.07D} \tag{8}$$

where C=0.09 is model constant, D is the diameter of the pipe.

(2) Give the non- slip walls and pressure outlet as the boundary condition for the outflow.

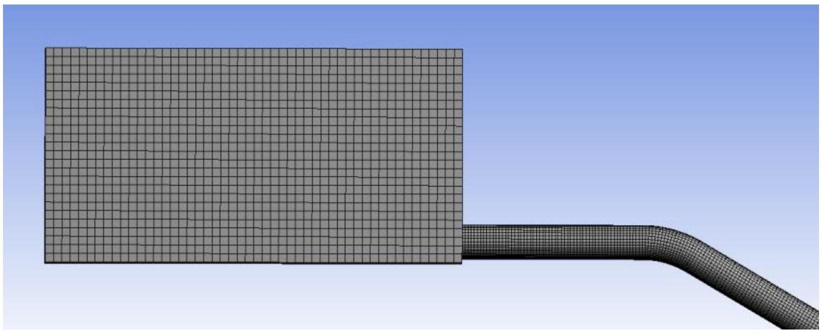

Figure 3. The meshing.

Numerical Simulation

Such as indicated in Figure 4, at the very beginning ($t = 0$) of the discharging, the discharging pipe is fulfilled with seawater on account of inwelling by the diffuser pipe. When the pipe in use, the sewage flowed by the driving of gravity, the inlet velocity is 8.9 m/s till the fluid flow met with air and performed as turbulent flow consequently. The velocity decreased to 6.8m/s according to the cushioning actionby air and maintained this velocity. Figure 5 (a), (b) and (c) revealed that the pressure on the pipe changed little.

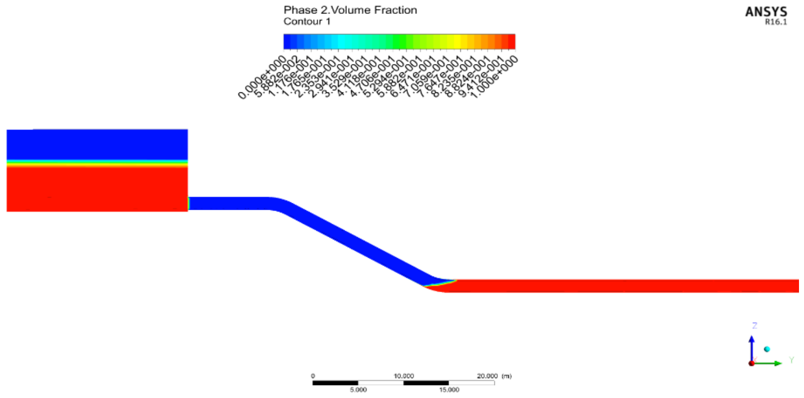

Figure 4. The status of discharging at ($t = 0$), the blue represents the air, sequentially the red is water.

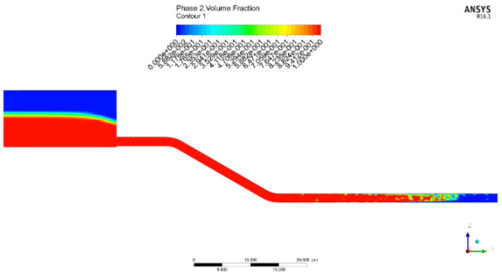

Figure 5(a). The status of fluid flow in pipe.

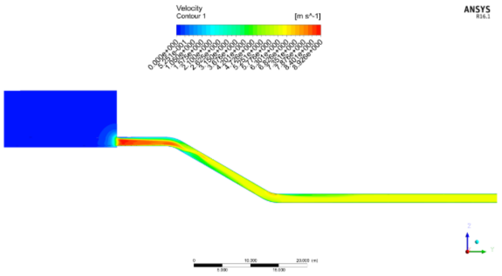

Figure 5(b). The velocity in pipe.

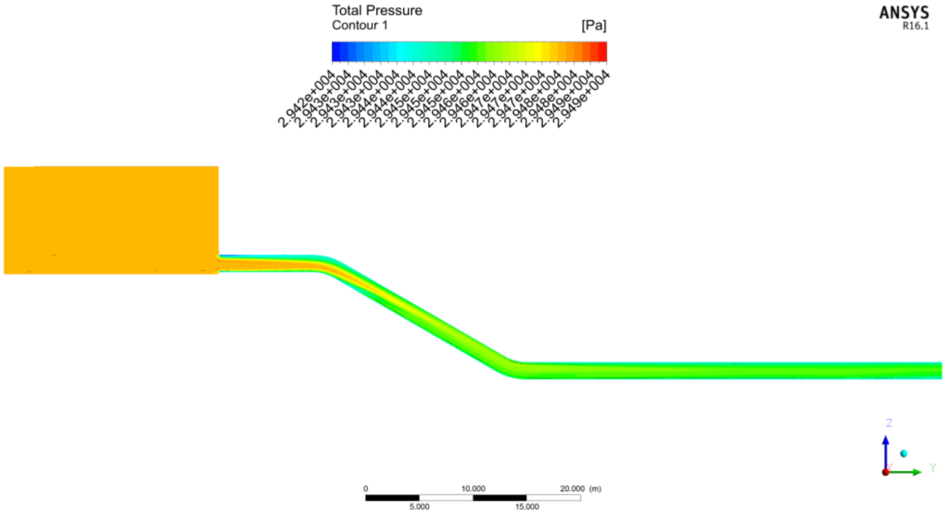

Figure 5(c). The pressure in pipe.

Structural Equations

After defining the problem of fluid mechanics, it is necessary to identify an equation to solve the solid problem which relates those forces imposed by the fluid to the change in deflection of the pipe. The length of the pipeline is very large compared with its diameter, the boundary condition given by $x \to \infty$, thus the deflections with the respect to $x$must vanish in this so-called infinite pipe. The bending problem of an arbitrary point x yields the equilibrium equation as follow [19, 20]:

$$D\frac{d^4 w}{dx^4} + \frac{Et}{R^2}\mathrm{w} - P = 0 \tag{9}$$

Where P is the axisymmetricalloads on x which can be obtained from 2.3,wis the displacement of x , E is modulus of elastic, t is the thickness of the pipe, R is radius of the pipe, D is flexural rigidity, defined as $\mathrm{D}=\dfrac{Et^3}{12\left(1-\upsilon^2\right)}$, υ is Poisson's ratio. Let $\beta^4=\dfrac{Et}{4R^2D}=\dfrac{3\left(1-\upsilon^2\right)}{R^2t^2}$, a convenient form is:

$$\frac{d^4w}{dx^4}+4\beta^4w=\frac{P}{D} \tag{10}$$

Owing to the symmetric of the pipe, the solution of 8 is:

$$\mathrm{W}=e^{-\beta x}\left(C_1 cos\beta x+C_2 sin\beta x\right) \tag{11}$$

Introducing the relevant of shear force, moment, and load per unit area $Q_x=\dfrac{dM_x}{dx}=-\mathrm{D}\dfrac{d^3w}{dx^3}=-\dfrac{P}{2}$ into(8),

$$C_1=C_2=\frac{P}{8\beta^3D} \tag{12}$$

The displacement is:

$$w=\frac{Pe^{-\beta x}}{8\beta^3D}\left(sin\beta x+cos\beta x\right) \tag{13}$$

Furthermore, the deflections can be computed through the foregoing numerical analysis as sketched in Figure 6.

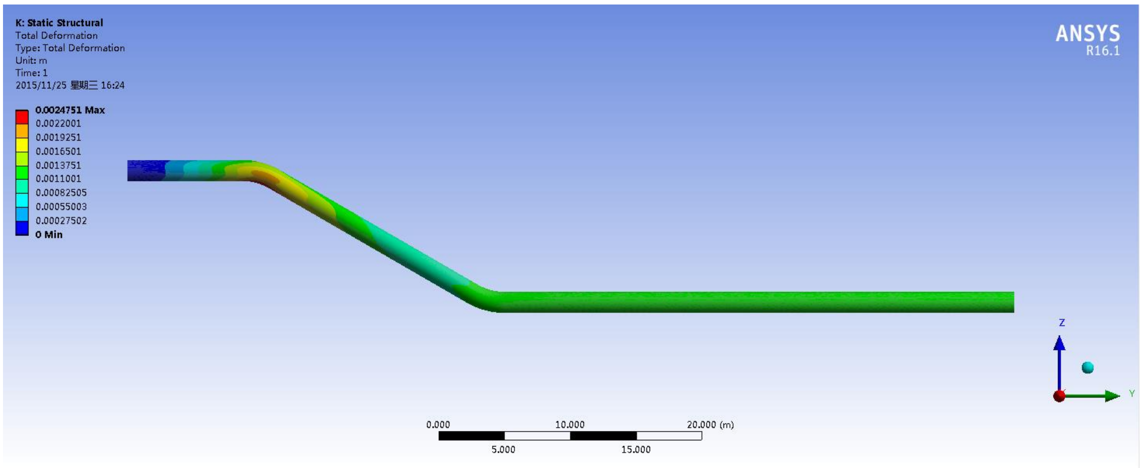

Figure 1. Numerical results of the deflections of the discharging pipeline.

EQUATIONS FOR ENGINEERING

Equations of Deflections Induced by Symmetric Loads

Some sections of the pipeline such as the section installed on the sea bed could be considered like being loaded symmetrically, the theory derivation equations can be readily formulated. The displacement w^* can be described as:

$$\begin{aligned} w^* &= \int_0^a \frac{Pe^{-\beta x}}{8\beta^3D}\left(sin\beta x+cos\beta x\right)dx+\int_0^{L-a}\frac{Pe^{-\beta x}}{8\beta^3D}\left(sin\beta x+cos\beta x\right)dx \\ &= \frac{PR^2}{2Et}\left(2-e^{-\beta a}cos\beta a-e^{-\beta(L-a)}cos\beta(L-a)\right) \end{aligned} \tag{14}$$

In which L is the length of the pipe which under uniform load, a is the distance between the starting section and x.It has been found from calculation that, when $\beta x > 10$, $e^{-\beta x} cos\beta x \to 0$ within the area $x < 1.2m$ over the length, the displacements changed randomly between $0.575\left(\frac{PR^2}{Et}\right)$ and $1.034\left(\frac{PR^2}{Et}\right)$, the displacement for $x > 1.2m$ is:

$$w^* = \frac{PR^2}{Et} \tag{15}$$

According to the homogeneity of the pipe geometry, the deflection of cross-section could follow:

$$\Delta S = 2\pi \frac{PR^3}{Et} \tag{16}$$

Equations of Deflections of Junction

The total energy of the fluid flow will be reduced by head loss along the pipeline, this is done through simplification of the conservation of energy into Bernoulli's equation.

$$\frac{\partial}{\partial t}\iiint J\rho dV + \left(J + \frac{P}{\rho}\right)\rho\left(v\,\vec{n}\right)dS = Total\ Energy \tag{17}$$

Where ϑ is the specific energy which includes the internal energy ζ, the kinetic energy $\frac{1}{2}v^2$, and the potential energy gz. The stresses usually concentrate at the junctions, such as the first junction in Figure 6. We can obtain some simple equations for calculating the deflections on junctions through the Bernoulli's theory. These equations could be employed in the basic designing of engineering. Consider the flow velocity at the junction of the pipeis v_0, take a point x_1 at the inlet, the velocity is v_1, ΔP is the pressure change from the inlet section to the junction , the head of friction is $h_f = \lambda \frac{x_1}{2R}\frac{v_0^2}{2}$, where λ is friction coefficient, they yield Bernoulli's equation.

$$\frac{\Delta P}{\rho} = \frac{v_1^2}{2} - \frac{v_0^2}{2}\left(1 - \lambda \frac{x_1}{2R}\right) \tag{18}$$

Λ is extremely small when Reynolds number $\mathrm{Re} > 10^7$, which is reported about 10^{-3} in engineering by Moody diagram, neglect $\lambda \frac{x_1}{2R}$ then we have:

$$\frac{\Delta P}{\rho} = \frac{v_1^2}{2} - \frac{v_0^2}{2} \tag{19}$$

integrate (2),

$$v_1 \cdot S_1 = v_0 \cdot S_0 \tag{20}$$

Substitute (12), (16) into (17), therefore the section area of the junction can be written:

$$S_0 = \frac{\pi R^2\left(1 + 2\frac{PR}{Et}\right)}{v_0}\sqrt{\frac{\Delta P}{\rho} + \frac{v_0^2}{2}} \tag{21}$$

the deflection of the junction area ΔS_0 is:

$$\Delta S_0 = \pi R^2 \left[\frac{(1+2\frac{PR}{Et})}{v0} \sqrt{\frac{\Delta p}{\rho} + \frac{v_0^2}{2}} - 1 \right] \tag{22}$$

if v_0 is regarded as the approximate value $\sqrt{2gh}$, ΔS_0 can be formed into an approximation:

$$\Delta S_0 = \pi R^2 \left[\left(1+2\frac{PR}{Et}\right) \sqrt{\frac{\Delta P}{2\rho gh} + \frac{1}{2}} - 1 \right] \tag{23}$$

CONCLUSIONS

The internal flow in pipe was driven by the gravity in the entire process of the flowing, the velocity is decreased from 8.9m/s at the inlet to 6.8m/s due to the effecting of the air. An approximate equations for displacement/deflection were developed which can be possibly employed in engineering designing. The strain ε is derived from displacement and applied with equation, which $\varepsilon = \partial w / \partial x$ makes the foundation of the mechanics of fatigue and the prediction of operational life. In this article, we discussed the deflection induced by the internal flow only, the actual deflection of the pipeline is composited by wave, current, thermal and other factors, which should be considered seriously in further work.

REFERENCES

[1] Y. Eisenberg and D. D. Treadwell, “San Francisco's Southwest Ocean Outfall”, *Coastal Engineering Proceedings*, vol. 1, no. 5, pp. 24-29, 2001.

[2] R. Vijay, V. K. Sardar, and S. S. Dhage, “Hydrodynamic assessment of sewage impact on water quality of Malad Creek, Mumbai, India”, *Environmental monitoring and assessment*, vol. 16, no. 5, pp. 559-571, 2010.

[3] A. R. Russo, J. H. Bailey, and W. J. Cooke, *Benthic Sampling Adjacent to the Wai’anae Ocean Outfall*, O’ahu: Hawai, 2011.

[4] I. Panayotova, “Meridional Asymmetries in Forced Beta-Plane Turbulence”, *International Journal of Geosciences*, vol. 2, no. 3, pp. 286-292, 2011.

[5] A. J. Smits, B. J. Mckeon, and I. Marusic, “High-Reynolds Number Wall Turbulence”, *Annual Review of Fluid Mechanics*, nol. 43, pp. 353-375, 2011.

[6] M. Hultmark, M. Vallikivi, and S. C. Bailey, “Turbulent pipe flow at extreme Reynolds numbers”, *Physical Review Letters*, vol. 108, no. 9, pp. 324-329, 2012.

[7] A. darbyshire, and T. mullin, “Transition to turbulence in constant-mass-flux pipe flow”, *Journal of Fluid Mechanics*, vol. 289, no. 2, pp. 83-89, 2005.

[8] A. A. Draad, G. Kuiken, and F. Nieuwstadt, “Laminar–turbulent transition in pipe flow for Newtonian and non-Newtonian fluids”, *Journal of Fluid Mechanics*, vol. 37,no. 7, pp. 267-274, 2008.

[9] G. Gao, R. Falconer, and B. Lin, “Modelling Open Channel Flows with Vegetation Using a Three-Dimensional Model”, *Journal of Water Resource and Protection*, vol. 3, no. 2, pp. 114-119, 2011.

[10] B. J. Boersma, “Direct numerical simulation of turbulent pipe flow up to a Reynolds number of 61,000”, *Journal of Physics: Conference Series*, IOP Publishing, no. 4, pp. 363-376, 2011.

[11] A. Meseguer and L. N. Trefethen, “Linearized pipe flow to Reynolds number10^7, *Journal of Computational Physics*, vol. 186, no. 1, pp. 178-197, 2003.

[12] A. J. Smits, B. J. Mckeon, and I. Marusic, “High-Reynolds number wall turbulence”, *Annual Review of Fluid Mechanics*, vol. 43, no. 3, pp. 53-75, 2011.

[13] M. Hultmark, M. Vallikivi, S. Bailey, and A. J. Smits, “Turbulent pipe flow at extreme Reynolds numbers”, *Physical review letters*, vol. 108, no. 9, pp. 094501, 2012.

[14] W. Wang, R. Zhang, and M. Li, “Design and Operation of Deep-sea Marine Outfall Pipeline Project of Effluent from Taoziwan WWTP in Yantai City”, *China Water & Wastewater*, vol. 30, no. 12, pp. 136-139, 2014.

[15] X. Liu, L. Dong, Y. Chen, and X. Zhou, “Analysis on Geomorphologic Features and Their Controlling Factors in the Bohai Sea”, *Advances in Marine Science*, vol. 31, no. 1, pp. 105-115, 2013.

[16] W. Wang and R. Zhang, “Design and Research for Waste Water Sea Diposal Project in Yantai”, *Marine Science Bulletin*, vol. 16, no. 4, pp. 52-58, 2007.

[17] A. Ali, R. K. Sharma, and P. Ganesan, “Turbulence model sensitivity and scour gap effect of unsteady flow around pipe: a CFD study”, *The scientific world journal*, vol. 41, no. 9, pp. 136-139, 2014.

[18] S. C. C. Bailey, M. Vallikivi, and M. Hultmark, “Estimating the value of von Karman's constant in turbulent pipe flow”, *Journal of Fluid Mechanics*, no. 12, pp. 79-84, 2014.

[19] C. W. Li and K. Yan, “Numerical Investigation of Wave- Current-Vegetation Interaction,” *Journal of Hydraulic Engineering*, vol. 33, no. 7, pp. 794-803, 2007.

[20] N. R. Wilson and R. H. Shaw, “A Higher Order Closure Model for Canopy Flow,” *Journal of Applied Meteology*, vol. 16, no. 11, pp. 1198-1205, 2007.