14TH INTERNATIONAL CONFERENCE
ON HYDROSCIENCE & ENGINEERING
MAY 26-27, 2022
IZMİR-TURKEY

NUMERICAL STUDIES ON PREDICTING VELOCITY FIELDS OF UPSTREAM CHANNEL THAT COMBINED WITH HEAD POND USED IN MECHANICAL WASTEWATER TREATMENT PLANT

First Author: Nagihan Şahin

Gümüşhane University, Civil Engineering Department

Gümüşhane, Turkey

nagihansahin@gumushane.edu.tr

Second Author: Rahim Şibil

Gümüşhane University, Civil Engineering Department

Gümüşhane, Turkey

rahimsibil@gumushane.edu.tr

ABSTRACT: In this study, head pond, made to stabilize water level in the Wastewater Treatment Plant (WWTP) and upstream channel of screens that is maintain velocities in design criteria at mechanical treatment plant have been analyzed in terms of hydraulic and hydrodynamics by using Computational Fluid Dynamics (CFD). CFD simulations are carried out with the CFD software Ansys Fluent with the three-dimensional (3D), steady, incompressible flow based on the Reynolds-Average Navier-Stokes equations for flow field calculations in the combined intake-head pond-upstream channel system. Also, the standard K-Epsilon (ske) model was chosen as a turbulence model. The numerical studies results showed that there was no homogeneous flow field distribution in upstream channel. It can also be noted that the upstream channel does not meet the desired velocity values for screening.

1. INTRODUCTION

Water resources are rapidly exhausted in worldwide, because of population growth, global warming and increasing drought. One of the most important ways to protect water resources corresponding to increasing demand is reuse wastewater. Generally, wastewater treatment plant (WWTP) consists of three stages. These are mechanical, biological, and advanced treatment methods. Hydraulic characteristics of each stage are important to operate the facilities economically and efficiently. Screens are a physical treatment method planned to remove floating or suspended coarse materials from wastewater. The use of the screens is important to separate these substances from the water so that they do not damage the installation, thereby reducing the burden on other treatment units. Headpond is built to support flow rate continuity by retaining water. The upstream channel is built to provide the approach velocity which is essential to be limited to approximately between 0.6 m/s – 1.2 m/s and adequate screen area for accumulation of screening between raking operations. Computational fluid dynamics (CFD) is the mathematical solution of fluid flow by computer-based simulation. CFD has been successfully used various units of WWTP: grit chambers (Couture et al., 2009), (Dutta et al., 2014), (Hoiberg & Shah, 2021), (He et al., 2008), (Meroney & Sheker, 2015), (McNamara et al., 2012), settling tanks (Miklós & Katalin, 2015), (Gao & Stenstrom, 2019), (Tarpagkou & Pantokratoras, 2013), (Matko et al., 1996), (Robescu & Manea, 2015), oxidation ditches (Şibil et al., 2021), (Xie et al., 2014), etc. The importance and originality of this study are that it is the first numerical study has been done on screens, as can be clearly seen from the literature.
In the present paper the numerical and theoretical studies was carried out on predicting hydrodynamic properties of selected WWTP by CFD simulations.

2. MATERIAL AND METHOD

2.1. Full-Scale Plant and Problem Description

The mechanical treatment unit of the Gümüşhane WWTP with a treatment capacity of 8333,28 m^3 per day was chosen as a full-scale facility. The upstream channel is a width of 143 cm, length of 420 cm and, the maximum water heigh in the upstream channel is 26 cm. Headpond, which is built to support flow rate continuity is a width of 214 cm, length of 250 cm, and height of 278 cm. The diameter of the inlet pipe, placed at headpond is 60 cm (Figure 1).

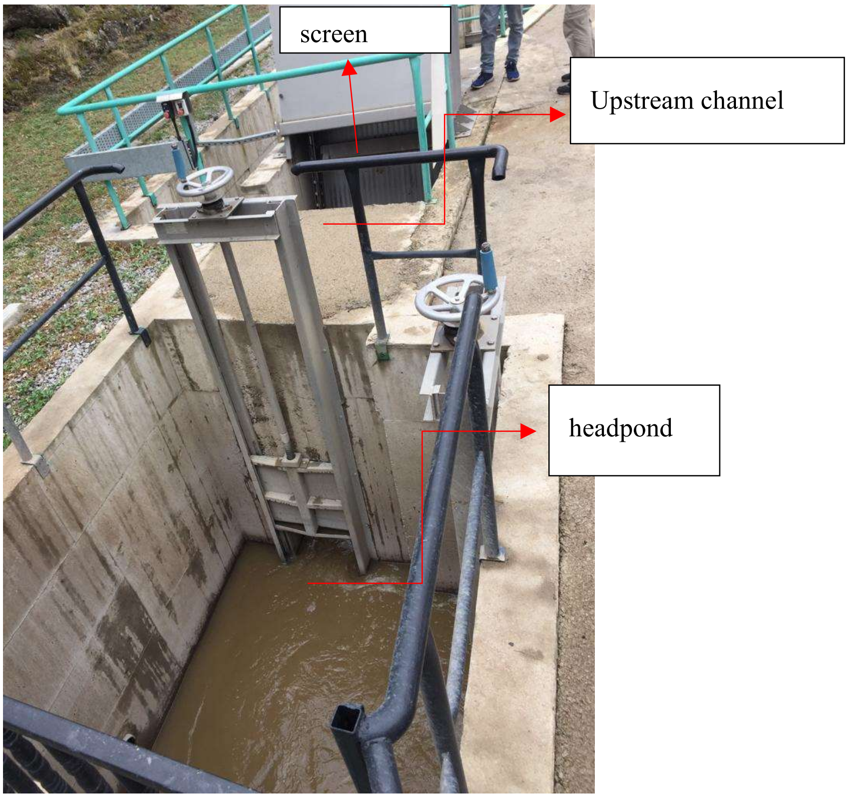

Figure 7. View of intake-head pond-upstream channel-screen

Typical design information for mechanically cleaned bar screens is provided in Table 1 (Metcalf & Eddy, 2003).

Table 1. Typical design information for mechanically cleaned screens (Metcalf & Eddy, 2003).

Parameter	SI units		
	Cleaning method		
	Unit	Manual	Mechanical
Bar size			
Width	mm	5-15	5-15
Depth	mm	25-38	25-38
Clear spacing between bars	mm	25-50	15-75
Slope from vertical	°	30-45	0-30
Approach velocity			
Maximum	m/s	0.3-0.6	0.6-1.2
Minimum	m/s		0.3-0.5
Allowable headloss	mm	150	150-600

The velocity of the upstream channel named approach velocity, one of the hydraulic design parameters of the existing screen is calculated by using the Manning equation as follows;

$$V = \frac{1}{n} R^{2/3} J^{1/2} \tag{1}$$

$$V = \frac{1}{0.013} \times \left(0.191m\right)^{2/3} \times \left(0.0001\right)^{1/2}$$
$$V = 0.255m/s$$

$$R = \frac{\text{A (Area)}}{\text{P (wetted perimeter)}} = \frac{0.26m \times 1.43m}{2 \times 0.26m + 1.43m}$$
$$R = 0.191m$$

Where V is approach velocity, n is Manning coefficient, R is the hydraulic radius, J is the hydraulic slope. It is clearly seen that the approach velocity calculated from equation 1 is not meet the typical design criteria given in Table 1. This problem is investigated to determine hydraulic and hydrodynamic characteristics of a selected full-scale wastewater treatment plant under real operating conditions by CFD.

2.2. Numerical Modeling

The numerical study was carried out to determine the velocity fields in upstream channel, which is feeding by head pond are determined in existing operation conditions. Although the flow rate from the Municipality change monthly according to the population mobility, the flow has been considered steady in the solutions. For the fluid flow, based on the Reynolds-average Navier-Stokes (RANS) equations, three-dimensional, steady, non-compressible flow is applied, and velocity field calculations were carried out in upstream channel for the CFD analysis by ANSYS Fluent software. Ske turbulence model that used most commonly in the literature was chosen as a turbulence model (Couture et al., 2009), (Dufresne et al., 2009), (He et al., 2008), (Matko et al., 1996), (Meroney & Sheker, 2015), (McNamara et al., 2012), (Tarpagkou & Pantokratoras, 2013). The flow is turbulence as calculated in detail follow. There is both pipe flow and channel flow in system which is seen in (Figure).

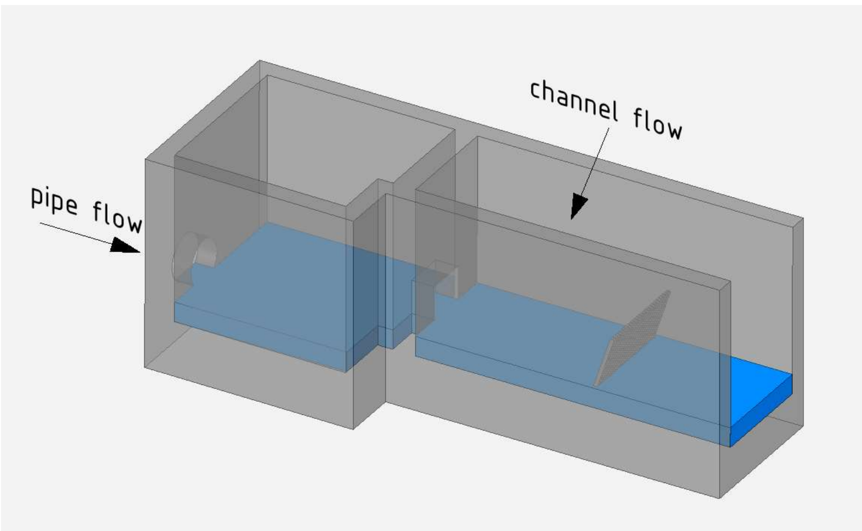

Figure 2. The pipe and channel flow

The Reynolds number is calculated in pipe flow as follows equation.

$$Re = \frac{V \text{ x } D}{V} \tag{2}$$

Where ρ is the density, V is the velocity, D is diameter of the pipe, ν is the kinematic viscosity. The Reynolds number is calculated as follow in intake pipe.

$$Q = V \text{ x } A \tag{3}$$

$$0.0492 \frac{m^3}{s} = V \text{ x } \frac{\pi \text{ x } 0.6^2}{4}$$

$$V = 0.174 \text{ m/s}$$

$$Re = \frac{0.1470 \frac{m}{sn} \text{ x } 0.60 \text{ m}}{1.003 \text{x} 10^{-6} m^2/sn} = 104087.7 > 4000 \text{ turbulent}$$

For open-channel flow, the Reynolds number is generally defined as

$$Re = \frac{V \text{ x } R_h}{V} \tag{4}$$

Where the hydraulic radius Rh is the characteristic length. And the Reynolds number is calculated as follow in channel.

$$R_h = \frac{A}{P} \tag{5}$$

$$Rh = \frac{1.43 \text{ m x } 0.26 \text{ m}}{(1.43 + 2 \text{ x } 0.26) \text{ m}} = 0.19 \text{ m}$$

$$Q = V \text{ x } A$$
$$0.0492 = V \text{ x } (1.43 \text{ x } 0.36) \text{ m}$$
$$V = 0.1323 \text{ m/sn}$$

$$Re = \frac{0.1323 \frac{m}{sn} \text{ x } 0.19 \text{ m}}{1.003 \text{x} 10^{-6} \frac{m^2}{sn}} = 25061.8 > 1000 \text{ turbulent}$$

2.2.1. Geometry and Meshing

Fluid and solid domain was created in SpaceClaim, one of the three-dimensional (3D) Computer-Aided Design modeling software (Figure 3).

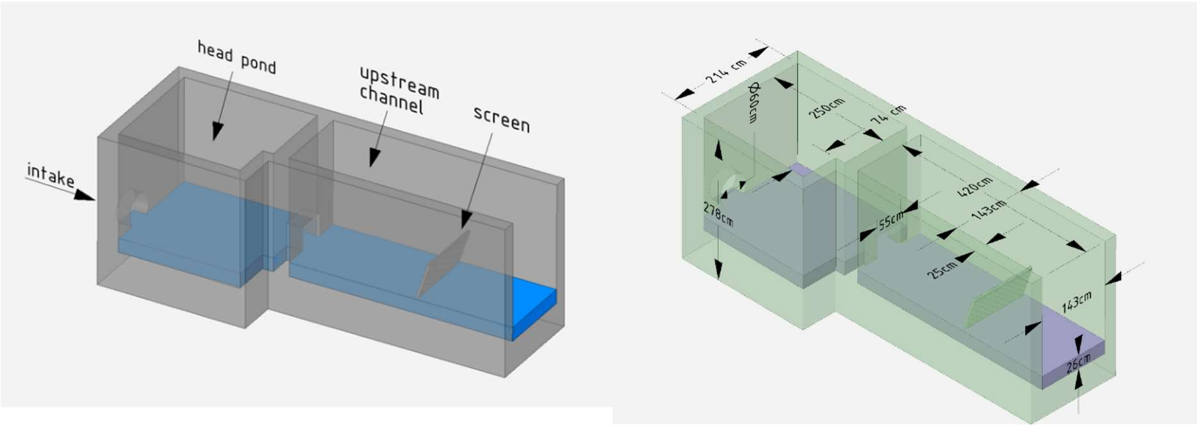

Figure 3. The designed geometry of the Model with dimensions

In addition, the intake is considered to the simulation. According to numerical studies, the hydrodynamic behaviors of the headpond and upstream channel units are evaluated in terms of the typical design criteria (Table 1).

The mesh is constructed with the tetrahedrons element structure, patch conforming method. Also, the body sizing method for screen and edge sizing method for screen bars was applied to the geometry. For body sizing, element size is 3, for edge sizing, the number of divisions is 7. Moreover, the inflation method, which including the first layer thickness option has a 0.5 cm first layer height and 5 layers, to describe the near-wall treatment was applied to the mesh study (Figure 4). After a series of grid-independent tests, the total number of elements turned out to be 1 035 204.

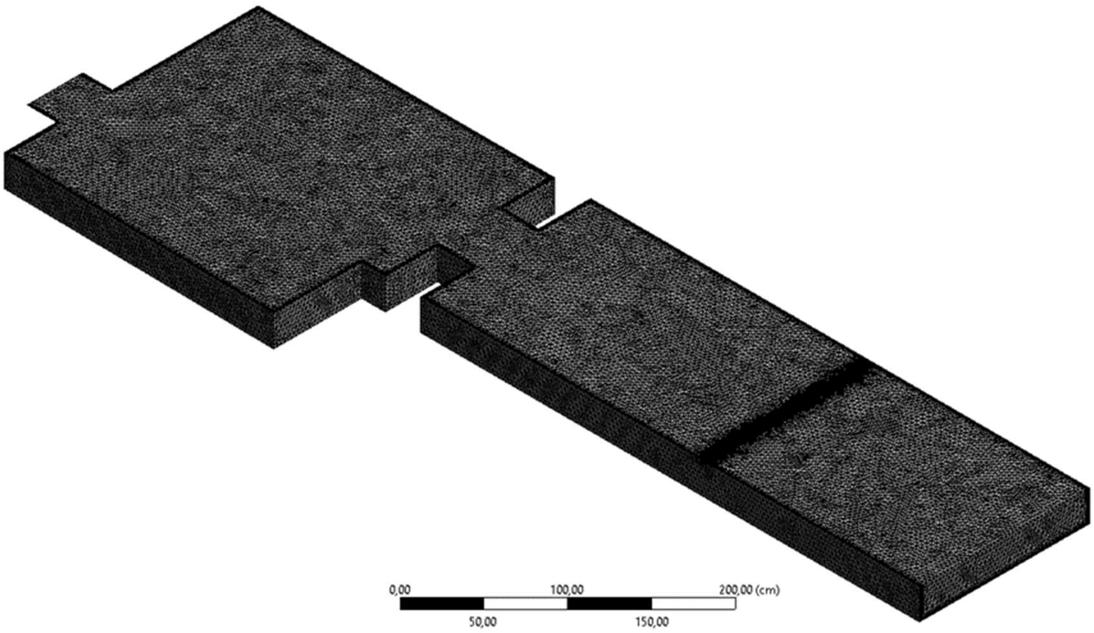

Figure 4. The mesh structure of Model

2.2.2. *Boundary Conditions*

Simulation is performed for WWTP with 8333,28 m^3/day flow rate and V=0.3477 m/s inlet velocity for Model. The boundary condition for inlet is “velocity-inlet”, for outlet is “pressure-outlet” and for water surface is “symmetry”. Solid boundaries are specified as “wall”.

2.2.3. *CFD Simulation of Model*

3-D, steady, single-phase, non-compressible flow with the k-ε turbulence model is performed in Ansys Fluent. The criterion for convergence in the numerical model requires the scaled residuals to decrease to 10^{-6} for all equations. The calculation time was approximately 4 hours for the steady-state calculation on an Intel (R) Core ™ computer with an i7-10750H CPU @ 2.60 GHz 5.0 GHz processor, 16.00 GB RAM, and a 64-bit operating system.

3. RESULTS AND DISCUSSION

The numerical study results are visualized to evaluate the hydraulic performance of WWTP. The velocity fields of intake-headpond-upstream channel combined system are given in figure 5.

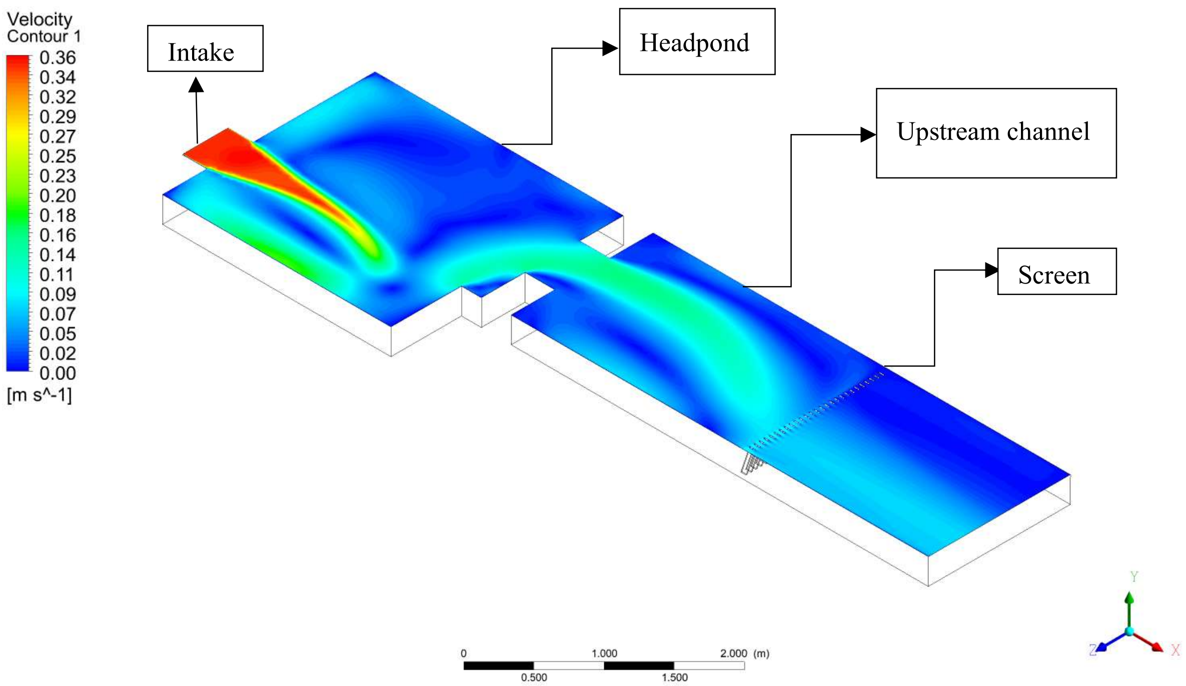

Figure 5. Velocity contour at intake-headpond-upstream channel

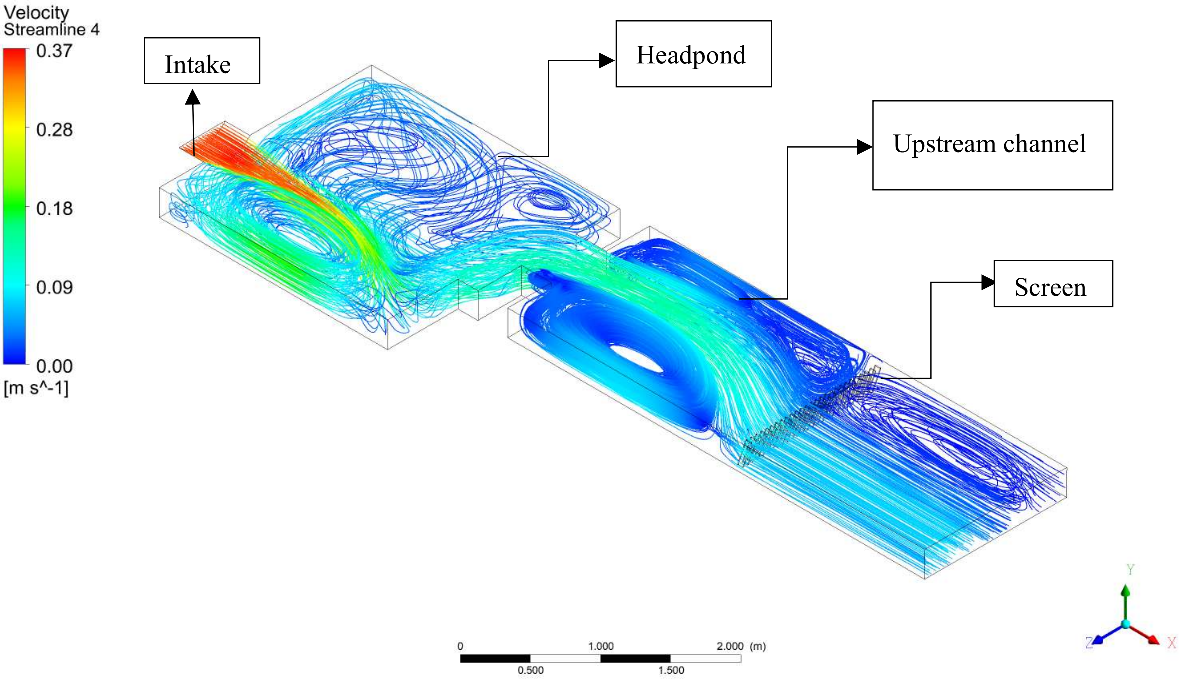

Figure 6. Velocity streamline at intake-headpond-upstream channel

As can be seen from figure 5 above, there is no homogenous velocity distribution at intake-headpond-upstream channel. The main problem is that headpond, built to provide to increase depth of water in upstream channel doesn't retain water and it operates like a channel. The figure 6 shows that the velocity streamline at unit. Since the inlet pipe is not in line with the outlet section of the headpond, the water draws a flow profile as shown in the figure 6.

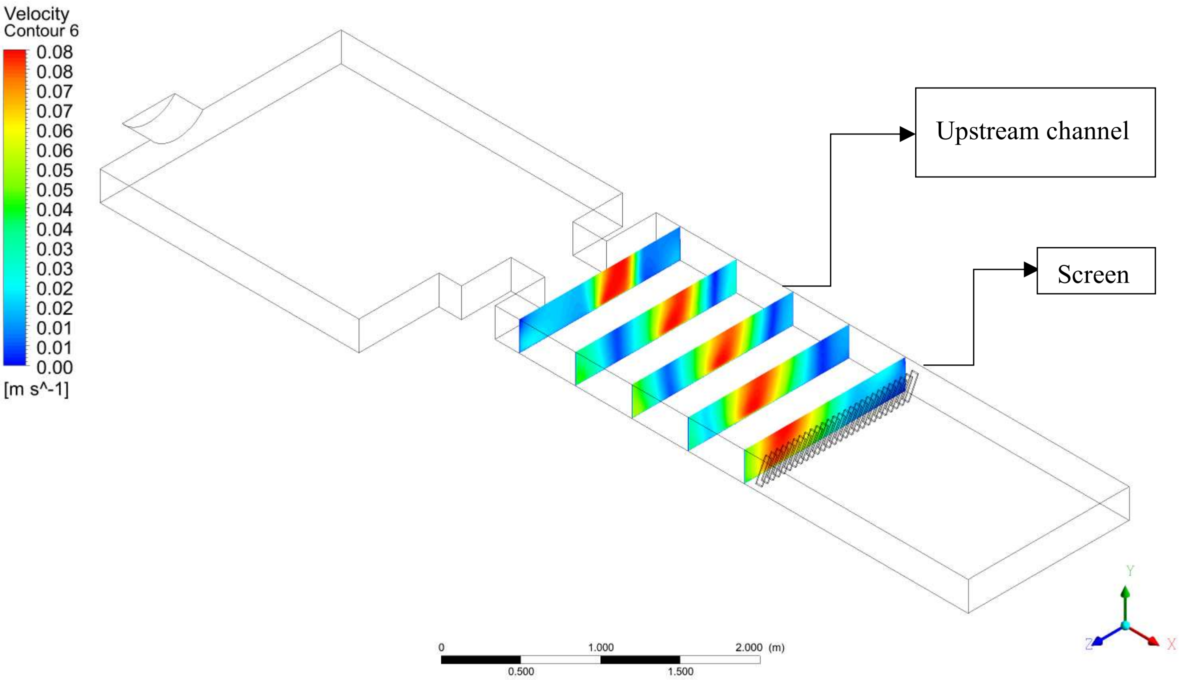

Figure 8. Velocity fields at vertical section at upstream channel

Figure 7 illustrates the velocity fields at vertical section at upstream channel. As seen in Figure 7 there is no homogenous velocity fields at upstream channel. Also, it clearly seems at Figure 7 that the approach velocities, which is essential be limited between 0.6 m/s – 1 m/s at upstream channel are vary between 0–0.08 m/s.

4. CONCLUSIONS

In this study, the hydrodynamic evaluation of the full-scale WWTP's intake-headpond-upstream channel was performed with the numerical simulation used by the CFD software ANSYS Fluent.
The numerical studies results showed that:

- Due to the low flow rate and insufficient wrong-operated headpond, the upstream channel does not meet the desired velocity values for screening.
- The maximum wastewater velocity occurs at the inlet. The water velocity also decreases as it moves away from these points at vertical and horizontal.
- There is no homogeneous flow field distribution in upstream channel.

REFERENCES

Couture, M., Steele, A., Bruneau, M., Gadbois, A., Hohman, B., Couture, M., Steele, A., Bruneau, M., Specialist, H. P., Gadbois, A., President, T. V., & Hohman, B. (2009). *A 360°*. 773–783.

Dufresne, M., Vazquez, J., Terfous, A., Ghenaim, A., & Poulet, J. B. (2009). Experimental investigation and CFD modelling of flow, sedimentation, and solids separation in a combined sewer detention tank. *Computers and Fluids*, *38*(5), 1042–1049. https://doi.org/10.1016/j.compfluid.2008.01.011

Dutta, S., Tokyay, T. E., Cataño-Lopera, Y. A., Serafino, S., & Garcia, M. H. (2014). Application of computational fluid dynamic modelling to improve flow and grit transport in Terrence J. O'Brien Water Reclamation Plant, Chicago, Illinois. *Journal of Hydraulic Research*, *52*(6), 759–774. https://doi.org/10.1080/00221686.2014.949883

Gao, H., & Stenstrom, M. K. (2019). Generalizing the effects of the baffling structures on the buoyancy-induced turbulence in secondary settling tanks with eleven different geometries using CFD models. *Chemical Engineering Research and Design*, *143*, 215–225. https://doi.org/10.1016/j.cherd.2019.01.015

He, C., Wood, J., Marsalek, J., & Rochfort, Q. (2008). Using CFD Modeling to Improve the Inlet Hydraulics and Performance of a Storm-Water Clarifier. *Journal of Environmental Engineering*, *134*(9), 722–730. https://doi.org/10.1061/(asce)0733-9372(2008)134:9(722)

Hoiberg, B., & Shah, M. T. (2021). CFD study of multiphase flow in aerated grit tank. *Journal of Water Process Engineering*, *39*(July 2020), 101698. https://doi.org/10.1016/j.jwpe.2020.101698

Matko, T., Fawcett, N., Sharp, A., & Stephenson, T. (1996). Recent progress in the numerical modelling of wastewater sedimentation tanks. *Process Safety and Environmental Protection*, *74*(4), 245–258. https://doi.org/10.1205/095758296528590

McNamara, B. F., Layne, J., Hyre, M., Kinnear, D. J., & Bott, C. B. (2012). Evaluation of three full-scale grit removal processes using CFD modeling. *WEFTEC 2012 - 85th Annual Technical Exhibition and Conference*, *10*(September 2014), 6008–6030. https://doi.org/10.2175/193864712811710335

Meroney, R. N., & Sheker, R. E. (2015). Removing Grit During Wastewater Treatment:CFD Analysis of HDVS Performance. *Water Environment Research*, *88*(5), 438–448. https://doi.org/10.2175/106143015x14212658614478

Miklós, P., & Katalin, K. (2015). *Analysis of suspended solids transport processes in primary settling tanks Miklós Patziger and Katalin Kiss*. 1–9. https://doi.org/10.2166/wst.2015.168

Robescu, L. D., & Manea, E. E. (2015). Using CFD Techniques in Teaching Rectangular Settling Tank Hydrodynamics. *Balkan Region Conference on Engineering and Business Education*, *1*(1), 41–48. https://doi.org/10.1515/cplbu-2015-0005

Şibil, R., Aras, E., & Kankal, M. (2021). *Experimental and numerical studies on predicting and improving the full- scale wastewater treatment plant hydrodynamics Experimental and numerical studies on predicting* *and improving the full-scale wastewater treatment plant hydrodynamics*. *December*. https://doi.org/10.5004/dwt.2021.27708

Tarpagkou, R., & Pantokratoras, A. (2013). CFD methodology for sedimentation tanks: The effect of secondary phase on fluid phase using DPM coupled calculations. *Applied Mathematical Modelling*, *37*(5), 3478–3494. https://doi.org/10.1016/j.apm.2012.08.011

Xie, H., Yang, J., Hu, Y., Zhang, H., Yang, Y., Zhang, K., Zhu, X., Li, Y., & Yang, C. (2014). Simulation of flow field and sludge settling in a full-scale oxidation ditch by using a two-phase flow CFD model. *Chemical Engineering Science*, *109*, 296–305. https://doi.org/10.1016/j.ces.2014.02.002

NUMERICAL INVESTIGATION OF FLOW AROUND AN ISOLATED PIER ON AN INCLINED SURFACE

Mete Köken
Department of Civil Engineering, Middle East Technical University
Ankara, Turkey
mkoken@metu.edu.tr
Emre Hesap
Department of Civil Engineering, Middle East Technical University
Ankara, Turkey
emre.hesap@metu.edu.tr
Gökçen Bombar
Department of Civil Engineering, İzmir Katip Çelebi University
İzmir, Turkey
bombar@ikcu.edu.tr
Şebnem Elçi
Department of Civil Engineering, İzmir Institute of Technology
İzmir, Turkey
sebnemelci@iyte.edu.tr
Antonio H. Cardoso
CERIS, Instituto Superior Técnico, Universidade de Lisboa
Lisboa, Portugal
antonio.cardoso@tecnico.ulisboa.pt

ABSTRACT: Coherent structures and turbulent flow characteristics around an isolated pier which is located on an inclined surface are investigated numerically within this study. DES model (Detached Eddy Simulation) is used in the simulations which are performed at Reynolds numbers of 52480 and 262400. Although the turbulence is more amplified in the high Reynolds number case, in both cases it was observed that there is an asymmetrical horseshoe vortex forming around the pier which contributed in the amplification of the shear stress on the bed. Furthermore, pressure root mean square fluctuations which is an important parameter in the formation of the scour are amplified along the bed especially at the downstream of the pier.

1. INTRODUCTION

Pier and abutment scour in bridges are the two major sources for the bridge failure. In the literature there are many studies that investigate the pier scour phenomena (i.e. Chang et. al. 2013, Dargahi 1989, Dey and Raikar 2007 etc.). However, all these investigations are made for horizontal channel bottom. In the present study turbulent flow characteristics around a pier on a laterally inclined channel is numerically investigated.

Spalart Almaras based DES (Detached Eddy Simulation) turbulence model was used in the simulations. This model is a hybrid model where regions close to the walls are resolved in RANS (Reynolds Average Navier Stokes) mode, while regions far from the walls are resolved in LES (Large