International Journal of Advanced Engineering and Management Research
Vol. 2 Issue 4, 2017

www.ijaemr.com ISSN: 2456-3676

EVALUATION OF A DISRIBUTION CHAMBER PERFORMANCE IN WASTEWATER TREATMENT

Tamas Karches[1*] and Gabor Kertesz[1]

[1]National University of Public Service, Faculty of Water Science

Bajcsy Zsilinszky Str. 14-16, HU-6500 Baja, Hungary

ABSTRACT

Distribution chambers are widely used in various processes, where there is a need to split the main stream to feed parallel process trains at the same discharge rate. Uneven distribution of the flow may cause uneven load to the biology, which is originally designed to a certain capacity, thus the overall performance of a wastewater treatment plant could deteriorate. In this paper a cost-effective evaluation methodology is presented, with which the performance of a distribution chamber can be determined. Numerical fluid dynamic simulation with turbulence modelling was applied to calculate the flow field and the flow rate of each sub-stream. As a result, the inappropriately designed chambers could be re-design prior the actual operation.

Key Words: Computational fluid dynamics, Distribution chamber, Fluid flow processes, Residence time distribution, Wastewater treatment

Introduction

Bio kinetic processes have a significant role in wastewater treatment. Based on the raw influent characteristics and the treated wastewater effluent quality requirement the biomass quantity responsible for the degradation of the wastewater constituents can be determined. Biomass production can be controlled by the physical parameters of the environment e.g. temperature, pH, dissolved oxygen concentration, substrate load and it can be also estimated by using empirical formulae or complex bio kinetic simulation tools. Following the sizing procedure, a complete mass balance as well as the dimensions of each process units can be determined (Henze et al., 2008; Metcalf and Eddy, 2003).

Although the wastewater design guidelines offer a complete description of bio(chemical) processes, the hydrodynamic conditions are idealized assuming completely stirred reactors, plug flow or any combination of them. In real life scenarios violation of discharge limits may occur in spite of the fact that all of the process variables are well designed, but deficiencies in fluid flow might appear. It can be a hydraulic short-circuit or a region with extreme high retention time. In addition, the previously mentioned hydraulic phenomena could reduce the effective process volume. To avoid the appearance of such zones design procedures shall take into account a detailed fluid flow analysis. With the knowledge of the flow field at each simulation point, the following can be estimated: (i) local turbulence dissipation, which refers to the mixing efficiency, (ii) water age or residence time and (iii) actual pathlines of the injected particles, thus the percentage of the particles flowing to the various outlets (Karches and Buzas, 2011).

Flow splitting could be achieved by several ways; for example, Venturi or Par shall flume could be applied not only for water discharge measurement, but it also creates a uniform flow distribution if the floor of the flume at the throat section is level and the throat section walls are vertical. Simple solution could be a weir, which is placed perpendicular to the direction of fluid flow. One disadvantage of the application of weirs is the relatively high headloss. Baffle walls also provide uniform distribution if each of them is placed at the same level. A slight level difference may cause significant flow rate difference. A more general solution is a separate chamber, where the inlet points are on the floor, and the increasing flow spread out like a fan reaching the outlets. Since the water surface is even and stable, a uniform distribution could be achieved (Horvath, 1976).

Design of distribution chambers is based on engineering assumptions, mainly using length/width ratios. Design should consider not only the type of a distribution chamber, but also the geometry, the inlet and outlet structure. A satisfactory mixing is also required to prevent settling (Chao and Trussell, 1980).Empirical approaches are based on one selected design flow rate and after the design discrete number of other flow rates are checked.

Actually, due to the diurnal flow pattern and seasonality in wastewater treatment, the receiving water amount has a constant change and as a consequence, the distribution performance may also change, which has an effect on downstream processes. It could happen for example, that three parallel biological trains applying suspended biomass with same mixed liquor concentration (MLSS), dissolved oxygen, wasted sludge rate could differ in treatment efficiency.

In this paper an evaluation method of distribution chamber is presented, the material and methods are described in Section 2, the result of a case study is discussed in Section 3. Section 4 summarizes the research highlighting the future research need and gives methodological guidance to design distribution chambers.

2. Material and Methods

2.1 CFD technique in evaluation of distribution chamber performance

Evaluation of flow field as well as the hydrodynamic performance of a reactor is based on tracer studies. These can be carried out by physical testing following the movement of the tracer. After the evaluation of the fate of the tracer the mean residence time, dead-zones, short-cuts can be determined. This procedure could be also followed by numerical tools by adding a tracer in a pre-defined flow field which was calculated a priori. Computational fluid dynamic simulations (CFD) are capable to reveal the flow field of a complex three-dimensional turbulent, multiphase flow applying robust numerical models.

Building a CFD model requires to fulfil the following steps in sequence: (i) drawing the geometry, (ii) meshing, (iii) setting up the physical model, boundary conditions, (iv) choosing the appropriate numerical schemes, (v) performing the calculation and (vi) evaluating the results.

2.2. A case study – model setup

A wastewater treatment plant treating 160 000 m^3/d of wastewater has four parallel biological trains. Prior this aerated basin a distribution chamber is designed in order to feed the trains evenly (see Fig.1). The basic dimensions of the chamber are the following: length: 7.9 m, width: 5.6 m, height: 5.4 m. It has two circular inlets each with the diameter of 1.2 m.

The geometry was drawn using CAD system. In order to gain the fluid volume, which is required by CFD, a subtraction had to be applied since the governing equations are solved withi this region (Fig. 1).

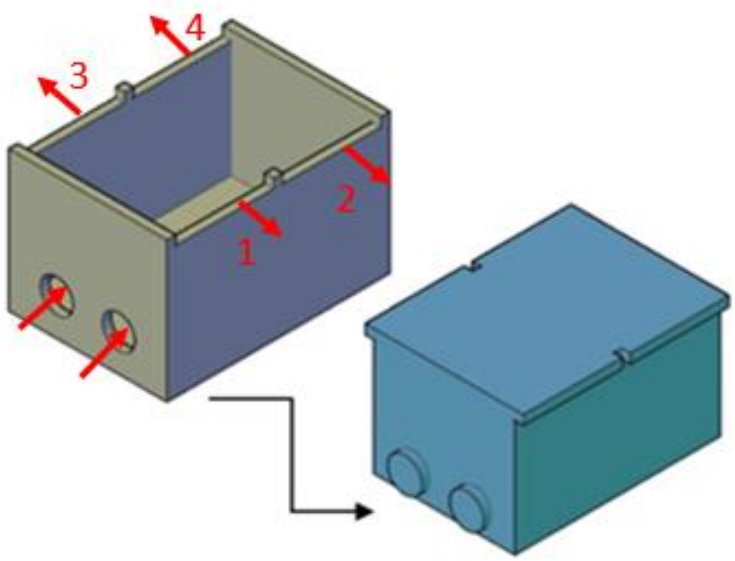

Figure 1. Geometry of the distribution chamber (left) and water volume (right)

For spatial discretization finite volume method (Ferziger and Peric, 2012) was applied with the mesh size of 154000 unstructured tetrahedral elements. In this numerical grid the equations of continuity, momentum and turbulence closure were solved. For turbulence modelling isotropic k-ε model was used, which solves separate transport equation for the turbulence kinetic energy (k) and the turbulent dissipation (ε) (Launder et al, 1973).

Boundary conditions were mass flow at the inlet section, free outflow at the outlet. Walls have no-slip condition and the free surface can be approximated by a zero-shear slip wall condition in viscous flow.

Numerical scheme for all equations were second order upwind in order to minimize the additive residual errors. Velocity-pressure coupling was solved by using Semi Implicit Method for Pressure Linked Equations (Doormaal and Raithby, 1984).

Steady-state model run was performed it eratively, the simulation steps converged to the solution. Between two simulation steps the previous value was updated by some part of the residual (in other words: under relaxation factors) were 0.7 for momentum, 0.3 for pressure and 0.8 for turbulence equations. Convergence criteria were defined as the iteration residuals needs to be below 10^{-3} in one hand and in the other hand discharges reach the steady state at the outlet boundaries.

3. Results and discussion

One distribution chamber with the dimension described in previous section was modelled. Whereas the design capacity of the plant was 160 MLD, the actual average inflow was approximately 154 MLD, which needed to split to four sub-streams. The wastewater density was 1020 kg/m^3, dynamic viscosity was approximated by 0.003 kg/(m*s). Steady-state simulation was performed. The calculation reached the convergence after 2700 iteration steps.

Graphical representation of the results can be seen on Fig. 2. On the left side fluid particles were released from the inlet section, their paths were followed and were colou red by the velocity. On the right side, a section was created in the middle of the right circular inlet. It can be stated that the high velocity regions with high turbulence intensity are near the inlet, its average velocity is about 0.7 m/s. The fluid flow reaches the opposite side of the chamber and starts increasing. While the jet is increasing, it loses its energy and expands towards the outlets.

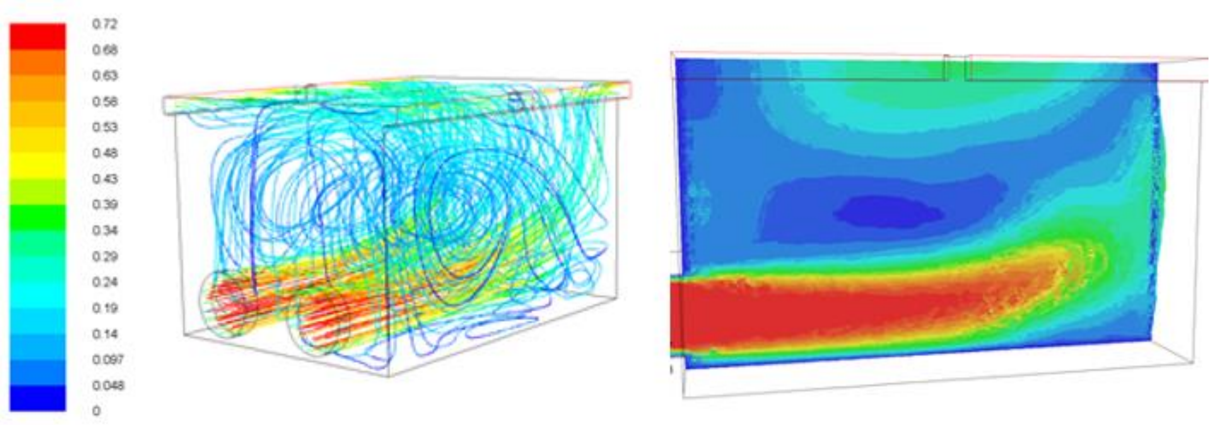

Figure 2. Pathlines in the distribution chamber (left), filled velocity contour lines at a section (right). Coloured by velocity with the scale of m/s.

Near the inlet, above the turbulent jet there could be find a small stagnant zone in the middle of a swirl zone. It dissipates the energy of the main flow, but it can cause instability. Sedimentation is not predicted due to the relatively high average velocity in the basin.

Knowing the velocity field and thus the surface averaged velocity at the outlet sections, the flow rate can be determined at each outlet. The result is summarized in Table 1. It can be seen that the outlets closer to the inlet (Nr. 1 and 3 based on numbering of Fig. 1) discharge less than outlets close to the other side of the distribution chamber. The difference in discharge is approx. 7-10%, which could have effect on the biology, but probably not significant.

In literature we also found evidence for such a difference, where a similar system was investigated. Wong (1999) measured and also modelled the discharge difference of a distribution chamber outlets and reported 10% variation. It suggested a horizontal plate as an inlet. which function was to deflect and reduce the peak velocities of the jet.

Table 1. Flow attheoutletweirs

Nr.	Q [m^3/s]	Q [m^3/h]
1	0.4307	1550
2	0.4638	1670
3	0.4327	1558
4	0.4653	1675

Grid independence test (Ali et al., 2009)was also carried out in order to minimize the numerical errors. The simulation was performed in a finer grid with mesh size of 854 000 elements and the discharge was compared to the results done with the original grid. The difference was negligible, less than 10^{-5} m^3/s.

4. Conclusion

Distribution chambers are widely used in wastewater treatment, where parallel process trains operate. Design of such chambers are mainly based on empirical assumptions. CFD demonstrated that it can be an effective tool in design or testing prior the real operation. Simulations performed revealed that the distribution chamber with vertical inlet causes a turbulent jet with velocities of 0.7 m/s. Above the jet a swirling, unstable region appeared, which deteriorates the efficiency of the distribution. Having enough height, the increasing jet has lost its energy and created a distribution where the difference between the discharges is about 7-10%. Further improvement could be done by applying horizontal inlets and/or installing energy dissipating structures.

References

Ali, M. S. M., Doolan, C. J., and Wheatley, V. (2009). Grid convergence study for a two-dimensional simulation of flow around a square cylinder at a low Reynolds number. In Seventh International Conference on CFD in The Minerals and Process Industries (ed. PJ Witt & MP Schwarz) 1-6.

Chao J. L., Trussell, R. R. (1980). Hydraulic Design of Flow Distribution Channels.*J. Environ. Eng. Div*. **106**(2), 321.

Van Doormaal, J. P., andRaithby, G. D. (1984). Enhancements of the SIMPLE method for predicting incompressible fluid flows.*Numerical heat transfer*,**7**(2), 147-163.

Ferziger, J. H., andPeric, M. (2012). Computational methods for fluid dynamics. Springer Science and Business Media.

Henze, M., van Loosdrecht, M. C., Ekama, G. A.,Brdjanovic, D. (Eds.). (2008). Biological wastewater treatment. IWA publishing.

Horvath, I. (1976). Hydraulics of Sewerage and Wastewater Treatment (In Hungarian). Vizdok. Budapest, pp.506. ISBN:963-602-036-1.

Karches, T., Buzas, K. (2011). Methodology to determine residence time distribution and small scale phenomena in settling tanks. *WIT Transactions on Engineering Sciences*, **70**, 117-125.

Launder, B. E., Morse, A., Rodi, W. and Spalding, D. B. (1973). Prediction of free shear flows: a comparison of the performance of six turbulence models.

Metcalf & Eddy. (2003). Wastewater engineering: treatment and reuse. McGraw-Hill.

Wong, vH. (1999). An experimental and numerical study of flow distribution chambers, PhD Thesis, Cranfield University, p.344.