International Journal Of Engineering Research And Development
e- ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 18, Issue 3 (May.-June. 2022), PP. 06-10

Force acting on a submerged beam near the inlet of a biological reactor in wastewater treatment

Tamas Karches
[1]*National University of Public Service, Faculty of Water Science, 6500 Baja, Bajcsy-Zsilinszky u.12-14. Hungary*

Abstract: *Design of the aerated basins in wastewater treatment includes the mass balance based biokinetic simulations, hydrodynamic calculations utilising computational fluid dynamics as well as mechanical simulation through fluid–structure interactions. Computational fluid dynamic simulation is able to resolve the flow field more accurately, producing information on velocity and pressure field within the reactor. The Reynolds Averaged Navier-Stokes formulation of momentum equation with turbulence closure equations are solved numerically with finite volume method. By knowing the flow field, the force acting on surfaces of matters could be calculated and this force as an exterior source term (or in other words, boundary condition) could be the initial setup for a stress simulation for the element. Submerged beams are applied in attached growth wastewater systems in order to provide a surface, where the biofilm carrier could be hanged. These beams bear the load of the biofilm carrier matter and the biomass developed on the surface. These are relatively static loads. Wastewater discharge has a diurnal pattern and expose a transient discharge, resulting variable (dynamic) load on the beam. In this research this water flow variation - Q(t) is linked to force variation - F(t) acting on beam will be calculated. The stress distribution is determined at five different loads (the simulation is steady-state, the scenarios are in different loading states) and as a result the tension is calculated and test for bulking is to be performed. In this research the importance of the one-way fluid-structure coupled models are highlighted.*
Keywords: *computational fluid dynamics, load variation, structures, wastewater treatment*

Date of Submission: 02-06-2022 Date of Acceptance: 16-06-2022

I. INTRODUCTION

An important element of municipal water supply is the collection and treatment of used water, which can eliminate a range of public health problems, relieve the receiving natural element and protect water resources. Wastewater treatment aims to treat used water from the public and wastewater from industry to the extent required for the receiving water body. As a rule, municipal wastewater is collected and transferred via sewerage network to the treatment technology.

In water and wastewater treatment, reactors with long residence times are often used, where mechanical, chemical and biological processes can take place. Long residence times are associated with large reactor volumes. The introduction into the basin is often through one or two points where high velocities are generated. The geometric design should aim to allow the incoming fluid flow to mix within the reactor after point introduction, without dead-zones and hydraulic short circuits. A commonly used solution is the introduction of direct proximity energy dissipaters, which is a baffle. When the water jet reaches it, it undergoes a directional break and loses energy, and its velocity decreases as the cross-section expands in the direction of flow. The water flow can be considered as an external stress on the submerged walls or other underwater beams. The forces acting on the surface can cause changes in the structure.

The hydrodynamics of water flow can be explored using numerical flow tools. Computational Fluid Dynamic (CFD) simulations are commonly used in wastewater sector [1-2]. Besides process design, CFD–Structure coupled models have been developed [3] in order to describe the effect of the motion on the structure. These effects can be the change in stress distribution or deformation of the material.

The wastewater arrives at the site unevenly, follows the variation in water use (daily fluctuations) and, in a combined system, precipitation also creates additional hydraulic loads. From a design point of view, the concentrations of pollutants reaching the plant during the daily fluctuations are considered the same, but a diluting effect is expected during the wet season.

A Completely Stirred Tank Reactor (CSTR) is a reactor where the addition and withdrawal is continuous. The incoming concentrated effluent is immediately mixed and diluted as it enters the reactor compartment. At any point in the basin space, the same value of a given parameter can be measured, which is identical to the quality of the effluent. If there is no large-scale recirculation, we speak of a lagoon; if there is recirculation, we have to take into account the effect of dilution and the fact that the residence time of the effluent is also shortened. A further advantage of this type of reactor is that uniform aeration can be achieved.

Plug Flow Reactor (PFR) is a reactor in which the incoming pollutant is not mixed and the change in concentration is due only to biological processes. Since longitudinal dispersion is not considered in this ideal case, all particles spend the same time in the system at the exit section. Its design requires the use of pools with a width to length ratio of at least 10:1 for sludge technologies, avoiding excessive turbulence and the increased longitudinal dispersion it causes [4]. Historically, the purpose of building such tubular reactors has been to control filamentous organisms, but this causes a non-uniformity of oxygen demand along the length, which also means that uniform air injection proves less effective [5]. In the case of staged oxygen injection, however, the problem of settling at the end of the reactor may arise.

II. MATERIAL AND METHODS

Numerical flow simulations solve a system of partial differential equations describing fluid motion, which includes the mass conservation and momentum equations. For turbulent flow, the calculation of the apparent stresses generated is required, for which a turbulence model can be used. Among the numerous turbulence models, the k-ε model [6], which assumes isotropic turbulence and is prescribed for the turbulent kinetic energy and its dissipation, or the RSM (Reynolds Stress Model), which tensorizes the apparent stresses, are widely used [7].

The analytical solution of a system of partial differential equations is not possible for complex geometries, and therefore a numerical method, the finite volume method, can be used, which divides the given reactor volume into a finite number of volume elements and solves the equations for each element starting from the boundary and initial values. Communication between cells is possible on the surface between them. The values of the variables are stored in the centre of the cells, which must be projected, i.e. interpolated, onto the surface bounding the cell. The result of the computation depends largely on the numerical scheme used and the resolution, which must be independent of the computation. Due to the number of mesh elements and the size of the numerical capacity requirements, an iterative, i.e. step-by-step approximation algorithm is mostly used, which has to be continued until the solution converges. Convergence means matching real conditions, but these conditions are not always known. In such a case, the convergence of the computation can be assumed based on the invariance of the iteration residuals and other variables (e.g., velocity space).

The first and essential step is to produce the 3D geometric model itself. The flow model space is the water space itself, i.e. the space where the flow occurs. In AutoCad, the walls of the basin were first drawn based on the dimensions of the reactor, and then the water space of the body bounded by the walls and the water surface was generated. The length of the structure is 10.5 m, the width is 14.3 m and the height is 5.0 m. To simplify the model calculation, we did not consider the entire water body, but a smaller part of it

Next, we need to generate a mesh for the geometry to control the fineness of the numerical resolution by considering different parameters. The numerical mesh is built up of cells. The edges of the volume cells are the units that connect the mesh lines and the nodes. Sections defined by three or four nodes are the interfaces.

For 3D modelling, hexa, pyramid, wedge, tetra elements can form the mesh created for the geometry. In the present case, the mesh of the reactor is built from tetrahedral element, which contains nearly 120 000 tetrahedrae. The posterior local compactness of the mesh is more efficient than for the other elements, and this type of mesh element is insensitive to cell distortion.

The starting point of meshing is the finite volume method, whereby a given geometric shape is decomposed into a finite number of parts of a given geometric space, and the resulting element is itself a mesh. The final mesh and the mesh values are saved as a file with the extension .mesh in the software output.

The boundary condition is the intensity per unit mass of the remaining physical features. After mesh generation, it is useful to specify these zones, which means that for each boundary condition zone, we need to define the type of boundary condition associated with the chosen physical models described in the next section. The boundary conditions specified here control the behaviour of the conservative variables (e.g. whether a given quantity can exit or enter the flow zone)

The choice of boundary conditions depends a lot on whether we are talking about compressible or incompressible flow, i.e. whether the medium is compressible or not. In the present case, the fluid is considered as an incompressible medium, constant in time and the density value does not depend on the pressure. In the case of the wall boundary condition, the velocity near the wall is zero and there is no mass flow through the wall. For the mass flow condition, the mass of fluid per second from a given medium into the test space is set. For the symmetry boundary condition, there is no mass transfer through the boundary. For a surface boundary condition, there is no mass transfer, but the value of the velocity does not follow the wall law.

In the configuration of the physical model, it is first set that the medium under study is constant in time and not compressible, and the value of the acceleration of gravity, which in the present situation is in the Z direction, is set to -9.81 m/s^2.

Then the turbulence model is set, for me the isotropic k-ε model is sufficient, where k is the turbulent kinetic energy and epsilon is the dissipation of the turbulent kinetic energy. In both cases, the scalar transport equation is solved. After setting the turbulence model, the different phases are generated.

Once all the conditions for the model have been specified, all that remains is to initialize it and iterate. During initialisation, we specify constant initial values or values corresponding to the flow, and then start running the simulation, where we can set the number of iterations to perform. In this case, this number was achieved after only 100 steps. The process can be stopped at any time and the model can be run again without initialization by entering new data. The condition for stopping the stepwise approximation is the convergence of the iteration residuals, i.e. their convergence to a given small band (10^{-3}).

III. RESULTS AND DISCUSSION

The reactor has a cuboid shape with a length of 10 m, width of 4 m and a height of 5 m. The primary treated wastewater enters through a slot of rectangular shape with a cross section area of 0.5 m^2 at the left top area of the basin (See Figure 1). Three different design loads are considered; the dry average flow is 150 m^3/d, the dry hourly peak flow is 9.4 m^3/h and at rainfall event, the peak flow is 420 m^3/d. Model runs assumed permanent flow field, the inlet boundary condition was mass flow inlet based on the constant discharges. Figure 1 shows the velocity vectors at the middle cross section at dry average flow.

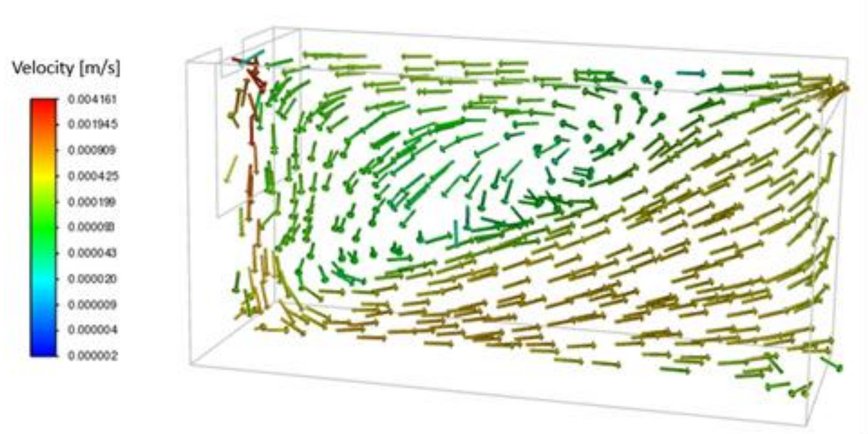

Figure 1. Velocity magnitude at the middle cross section

It can be seen that there is a rapid directional change near the inlet due to the baffle and as the free flow cross section expands the velocity decreases. At the right side of the baffle circulating zone develop, which consume the energy of the main flow and the transport of scalar variables between the swirling centre and the main flow is restricted to diffusivity.

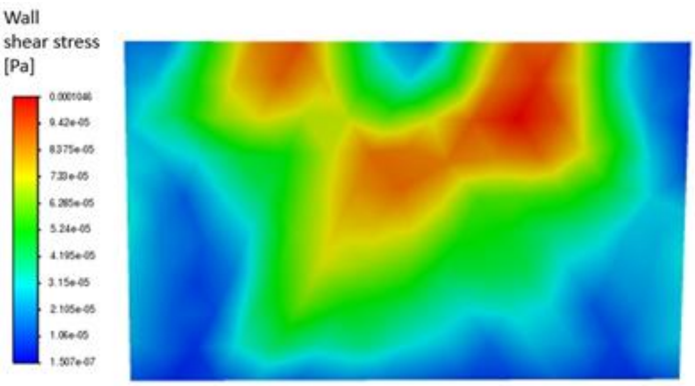

Figure 2. Velocity magnitude at the middle cross section

Wall shear stress at the baffle (Figure 2) has an uneven distribution; the most exposed regions are at the top region, beyond where the jet of the discharge hit the baffle. This shear could be translated to force and could be the input of the static structure analysis. The magnitude of the peak shear varies with the discharges, but the shape of the distribution remains the same in each cases.

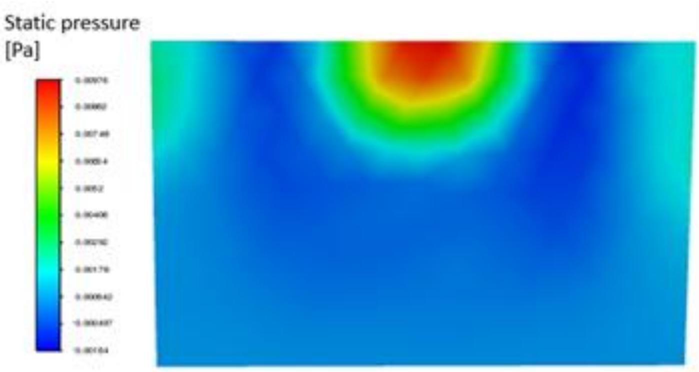

Figure 3. Velocity magnitude at the middle cross section

Figure 3 shows the static pressure at the baffle. The highest values are at the top centre region, as it was expected based on the inlet structure geometry. The simulations were performed at the above mentioned three scenarios and the integral average of shear stress and static pressure at the baffle surface were calculated and summarized in Table 1.

Table 1. Surface integral average values at the baffle at the various scenarios

	dry average flow	dry hourly peak flow	rainfall event
static pressure [Pa]	$4.65*10^{-5}$	$8.6*10^{-5}$	$2.44*10^{-4}$
shear stress [Pa]	$9*10^{-4}$	$2*10^{-3}$	$6.8*10^{-3}$

It is not surprising that the highest external load increases with the volumetric flow, therefore the design of the structure should take into account the non-bypassed storm flows. Further investigation is needed to perform the structure analysis and from hydrodynamic point of view, the transient behaviour of fluid flow changes need to be investigated.

IV. CONCLUSIONS

Static load due to fluid flow imposed on a baffle wall in a reactor applied in wastewater treatment was analysed. The tool of computational fluid dynamics investigated three scenarios based on various design flow. Steady-state single phase turbulent flow simulations were performed by using finite volume analysis. In each cases the velocity field within the reactor was calculated and the static pressure and shear stress acting on the baffle wall were determined. It was stated that the stresses increases with the amount of discharged flow and it was demonstrated that this kind of investigation could be a valuable tool for further stability analysis of the structure.

.

ACKNOWLEDGMENT

The project TKP2020-NKA-09 was funded by the Hungarian National Research Development and Innovation Fund, under the Thematic Excellence Programme 2020.

NATIONAL RESEARCH, DEVELOPMENT AND INNOVATION OFFICE HUNGARY

PROGRAM FINANCED FROM THE NRDI FUND

REFERENCES

[1]. Samstag, R. W., et al. CFD for wastewater treatment: an overview. *Water Science and Technology*, 2016, vol. 74, no 3, p. 549-563.

[2]. Peng, Si Mai, et al. The Application of computational fluid dynamics (CFD) in wastewater biological treatment field. *Applied Mechanics and Materials*, 2014, vol. 507, p. 711-715.

[3]. Dettmer, Wulf G.; Peric, Djordje. On the coupling between fluid flow and mesh motion in the modelling of fluid–structure interaction. *Computational Mechanics*, 2008, vol. 43, no 1, p. 81-90.

[4]. Metcalf and Eddy.Wastewater engineering: treatment and reuse, 2003, McGraw-Hill.

[5]. Kárpáti Á., Latest results in wastewater treatment, development and simulation, Case studies 11 (In Hungarian), 2005, Veszprém, Hungary.

[6]. Launder, B. E. Kolmogorov's two-equation model of turbulence, 1991, Proceedings of the Royal Society. London Series A, vol. 434. no. 214.

[7]. Aubin, Joelle – Fletcher, David F. – Xuereb, Catherine, Modeling turbulent flow in stirred tanks with CFD: the influence of the modeling approach, turbulence model and numerical scheme, 2004, Experimental Thermal and Fluid Science, vol. 28. no. 5. 431-445.