Numerical Simulation of Odorous Dispersion Hydraulics and Aeration Device in WWTP's

Hatem Dhaouadi[1] and Hatem Mhiri[2]
[1]*Science Faculty, Monastir University, Bvd de l'Environnement, Monastir, Tunisia*
[2]*National Engineering School, Monastir University, Ibn El Jazzar Street, 5000 Monastir, Tunisia*

Keywords: CFD, Odour Dispersion, Airlift Hydraulics, Aeration.

Abstract: Most of the WWTP use biological processes which are intrinsically dynamic because of the large variations in the wastewater flow rate, pollutants concentration and composition. These variations are to a large degree not possible to control and the use of simulator may be helpful. CFD, one of the most used numerical tool, is employed in this study to prospect three particular technical aspects of WWTP, namely odors dispersion, hydraulics of a high rate airlift algal pond and mass transfer performances of surface aerators. Fluent® software is used and the validation of the developed models is made using a real scale WWTP data. This experimental validation is supported by the monitoring of the main odour pollution parameter (gaseous H_2S concentration) through different zones around El-Frina WWTP. CFD models are also used to examine the behavior of gas and liquid phase dynamic throughout the high rate airlift algal pond and results are compared to those obtained with measurements made at Sidi Bouali WWTP. Concerning the aeration system capacity, gas liquid mass transfer study of surface aerators has been conducted on a lab scale Rushton blades.

1 INTRODUCTION

Computational Fluid Dynamics (CFD) is more and more used in the environmental system analysis. We will present here three different applications of CFD utilisation in the field of wastewater treatment plant (wwtp) management.

The first case, the atmospheric dispersion of odours issued from an urban biological wwtp. The second study deals with the surface aerator from gas liquid mass transfer point of view and finally, the CFD contribution to the hydrodynamic study of a real scale high rate airlift algal pond. In the three cases, a powerful software is used, Fluent®.

Concerning the first case, the study of wwtp odours atmospheric dispersion and on the basis of olfactometric measurements, the sludge drying beds are identified to be the principal emission source of hydrogen sulphide, the main odorous component (Maïzi et al., 2010). This study has been applied to El Frina WWTP (Monastir, Tunisia).

Concerning the second case related to the study of surface aerators from the gas liquid mass transfer point of view, a lab scale Rhuston turbine blades are used to validate the developed model before going on with a real scale aerator. The only measured parameter is the dissolved oxygen concentration. The variables are the blades immersion rate and the turbine radial speed. Experiences are made with different liquid viscosities and surface tension. Several immersion rates and blades speed has been investigated and the dissolved oxygen is measured using the gassing off method with nitrogen. An exact analytical solution is used to evaluate the K_la coefficient. The volumetric mass transfer coefficient is then correlated to the gas holdup in the reactor for aeration capacity calculations, which allows the model validation (Achouri et al., 2013). Finally and concerning the last case, a real scale (Sidi Bouali, Tunisia) high rate airlift algal pond is studied from the hydrodynamic point of view. The novelty in Sidi Bouali WWTP is that the used device for fluid motion in the algal pond is an airlift. Hydraulic control in such system is primordial because it directly affects the algae to bacteria growth rate. Here, the velocity contour curves allow the diagnostic of stagnant zones especially at the pond extremity and the phase holdup contours shows clearly how efficient is the airlift porous aerators and their influence towards the global liquid velocity fixing the algal / bacterial mass growth rate.

Dhaouadi H. and Mhiri H..
Numerical Simulation of Odorous Dispersion Hydraulics and Aeration Device in WWTP's.
DOI: 10.5220/0004508003770381
In *Proceedings of the 3rd International Conference on Simulation and Modeling Methodologies, Technologies and Applications* (SIMULTECH-2013), pages 377-381
ISBN: 978-989-8565-69-3

2 MATERIAL AND METHODS

In what follows will be detailed the mathematical modelling and the numerical resolution using a commercial software, FLUENT® for the odours atmospheric dispersion only. The same methodology is adopted for the two other case of study, namely the gas liquid mass transfer in the case of the surface aerators and the hydrodynamic study of an airlift algal pond used for a real scale municipal wastewater treatment plant.

2.1 Mathematical Modelling

Atmospheric dispersion consists of two processes: transport and diffusion. Equations governing this problem are obtained using the Favre decomposition and are given in the following table.

The introduction of fluctuating terms makes this equation system open. Its closure requires the use of a turbulence model that allows getting an equal equation's number to the unknown number. For this survey, a first order closing model was adopted. With the use of the latter, transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε), are given in the table below, where R is the dissipation rate production term, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are empiric coefficients having the values of 1.42, 1.68 and 1, respectively (Fluent User Guide, 2006).

Mass balance	$\frac{\partial\left(\bar{\rho}\,\tilde{u}_j\right)}{\partial x_j} = 0$
Momentum balance	$\frac{\partial(\bar{\rho}\tilde{u}_j\tilde{u}_i)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij} - \overline{\rho u_i'' u_j''}\right)$
Concentration balance	$\frac{\partial(\bar{\rho}\tilde{u}_j\tilde{C}^m)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\bar{\rho}D\frac{\partial \tilde{C}^m}{\partial x_j} - \overline{\rho u_j'' C^{m''}}\right)$
Energy balance	$\frac{\partial\left(\bar{\rho}\,\tilde{u}_j\,\tilde{T}\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\lambda + \frac{C_p\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial \tilde{T}}{\partial x_j}\right]$

(k)	$\frac{\partial}{\partial x_i}\left(\rho k\tilde{u}_i\right) = \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\mathrm{Pr}_k}\right)\frac{\partial k}{\partial x_j}\right) + \tilde{P} + \tilde{G} - \bar{\rho}\varepsilon$
(ε)	$\frac{\partial}{\partial x_i}\left(\bar{\rho}\varepsilon\tilde{u}_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\mathrm{Pr}_\varepsilon}\right)\left(\frac{\partial \varepsilon}{\partial x_j}\right)\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\left(\tilde{P} + C_{3\varepsilon}\tilde{G}\right) - C_{2\varepsilon}\bar{\rho}\frac{\varepsilon^2}{k} - R$

(k): Turbulent kinetic energy balance
(ε): Dissipation rate balance

In the present work, all simulations are carried out using a finite volume method FLUENT to model 3D steady turbulent atmospheric dispersion of odorous compounds. In the present finite volume method, the solution domain is subdivided into a finite number of continuous control volumes.

2.2 Numerical Solver

The FLUENT software offers several CFD models: the Reynolds Average Navier–Stokes (RANS) models which include the standard renormalisation group (RNG) and real (RSM) model. After testing each of these, respectively, the RNG k–ε model was selected because odour emission velocity at the source outlet is feeble, besides RNG k–ε model generated the least cells number, compared to other models. Its calculating time per iteration was obviously small compared to the calculating time per iteration of the RSM model.

The RNG k–ε model is based on two transport equations for the turbulent kinetic energy k and its dissipation rate ε which uses a cross-diffusion term in the ε equation to ensure the appropriate equations model behaviour in both the near-wall and far-field zones (Fluent user guide, 2006).

The FLUENT 6.2 steady three-dimensional segregated solver was used to solve the RNG k–ε model using the implicit scheme. The upwind second and first orders of discretisation schemes were used to convert the governing equations into algebraic equations for their numerical solution. The Standard scheme was used to solve for pressure while the upwind first and second orders were used to solve for odorous compounds dispersion, momentum, turbulent dissipation rate, turbulent kinetic energy and energy. The SIMPLE method was used to calculate for pressure-velocity coupling.

Several wind speeds were used to study the influence of aerodynamic aspects on odorous compounds dispersion in the vicinity of the WWTP of Monastir and to estimate the distribution of the contaminants concentrations released by that source in the atmosphere, and consequently characterizing the propagation of their odours in the neighbouring buildings.

2.3 Meshing

Previous runs proved that the contribution of drying beds is by far great compared to the odours intensity released by the other devices in the WWTP of Monastir, and this is due to their important size. In fact the pollutant plume emitted by the drying beds was by far great compared to the plume emitted by the other devices. Therefore, the study was limited to odorous compounds emitted by the drying beds, in order to reduce the calculation time and the

number of cells. Drying beds have a rectangular shape, their whole length is 200 m and their whole width is 86 m. Buildings were designed as a brick shape located at a distance L_1 downwind of 550 m from the source. The "GAMBIT" software was used to create the computational volume, since it allows meshing domains into two or three dimensions with different geometric shapes. The flow topology requires a very fine meshing in a big part of the domain. In order to follow with precision every mass and aerodynamic fields variation, particularly in regions where gradients are important, a non uniform meshing, greatly contracted in the drying beds and buildings vicinity, which allows to display the recirculation and the vortices created, was adopted. Meshes are made extremely contracted in the neighbouring of the drying beds and around buildings.

2.4 Boundary Conditions

The boundary of the computed domain included the clean air and odour inlet, the fluid outlet, the walls of the computational volume and the buildings. The bottom and the upper surfaces of the computational domain were modelled as wall surfaces. The vertical profile of the horizontal wind velocity and the temperature were inputs, as well as the turbulence kinetic energy k and its dissipation rate ε.

In this paper, the assumptions taken to solve odorous compounds dispersion are:

- The flow is considered three-dimensional, turbulent and stationary,
- The wind speed is considered constant and its direction is parallel to the passage centre line,
- Continuous pollutants emission with constant concentration,

Temperature gradients are negligible.

Furthermore, the evolution of the ammonia plume was the same as the H_2S plume, so that the study was limited to follow the distribution of H_2S concentration.

3 RESULTS

3.1 Odours Dispersion

When a pollutant passes beyond buildings, several turbulence scales can be identified. Near the release source, there are diffusion scales that cause the pollutant dispersion. In the case of numeric simulations of odorous compounds dispersion around buildings, this phenomenon is rarely reported in other papers. The different parametric studies achieved in the current work are made in the case of brick buildings as shown in Fig. 1.

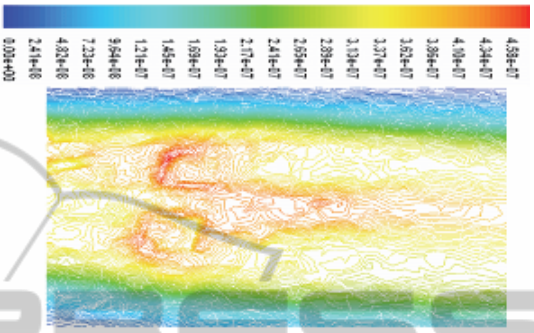

Figure 1: Iso-concentration of hydrogen sulphide on buildings roofs.

The survey of pollutant mass dispersion requires a good understanding of the flow behaviour around an obstacle.

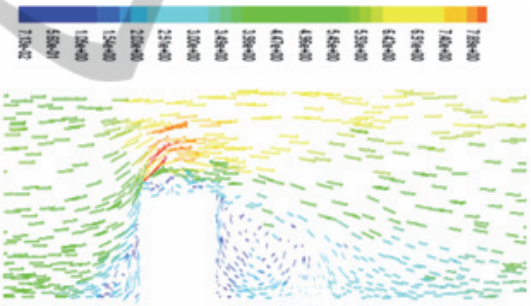

Figure 2: Distribution of longitudinal velocity around buildings.

The extent of the recirculation zone (Fig. 2), the boundary layer separation, the reattachment and the nature of vortices that are separated from the obstacle, will interact with the pollutant and thus influence its dispersion since the buildings have sharp edges, which amplify instability. With the developed model, many scenarios can be studied varying the odorant components, wind speed and building shapes.

3.2 Surface Aerators Mass Transfer

Gas liquid mass transfer study of surface aerators has been first conducted on a lab scale Rushton blades. Several immersion rates (Fig. 3) and blades

speed (Fig. 4) has been investigated. The dissolved oxygen (DO) variation is measured using the gassing off method with nitrogen. An exact analytical solution is used to evaluate the K_la coefficient by fitting the experimental DO results.

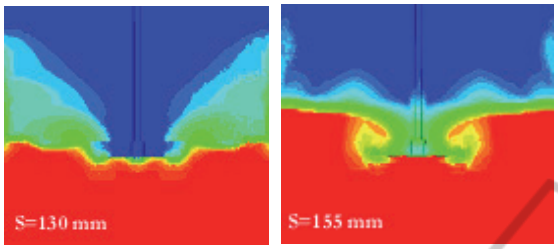

Figure 3: Paddle immersion effect on gas liquid transfer.

The volumetric mass transfer coefficient is then correlated to the gas holdup, a CFD result, which allows the model validation. The K_la value is proportional to the aeration capacity, which is the most crucial energetic parameter in WWTP (50 to 70% of energetic expenses in activated sludge WWTP are for aeration needs).

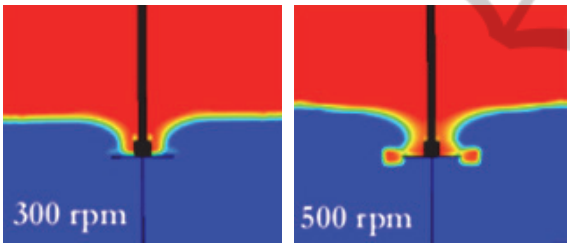

Figure 4: Paddle velocity effect on gas liquid transfer.

The determination of the best blade geometry profile, allowing minimum energy consumption, with the same aeration capacity (Achouri et al., 2012), is the main target of this investigation

3.3 Airlift Algal Pond Hydrodynamics

The hydraulic study of the high rate airlift algal pond is essentially based on the study of the bubble pump (airlift). The developed model for the real scale pond, 150*4*0.5m, showed reliable results and allows the evaluation of all hydraulic parameters, i.e. the gas holdup (Fig. 5a) and liquid velocity all over the algal pond.

In the computing domain, meshes are made extremely contracted in the bubble pump, which is the most interesting study zone. Within this area the liquid gets the needed energy, resulting from the injected air isothermal expansion, allowing its motion across the entire channel.

The current design of algae ponds lacks visual assessment of hydrodynamic characteristics, resulting in the appearance of dead zones where the flow is stagnant and in the presence of non-uniform velocity throughout the pond (Fig. 5b). Dead zones, because of their negative impact on algae growth, have to be avoided (Hadiyanto et al., 2013).

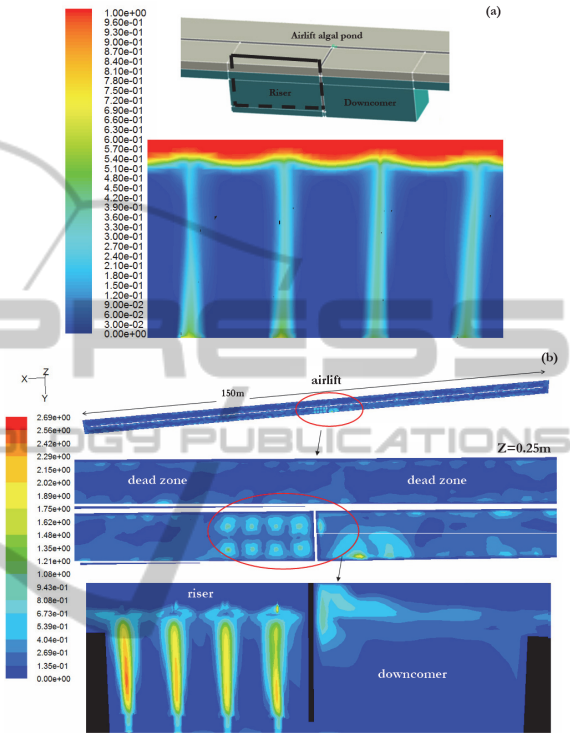

Figure 5: Gas holdup (a) and liquid velocity (b) profile in the high rate airlift algal pond.

In the figure 6 below are compared the liquid mean velocity in the channel to the measured liquid velocity at the surface.

Experimentally, the former is very hard to get and the latter is far from the real mean velocity which is the most important parameter in fixing the growth rate between algal and bacterial species.

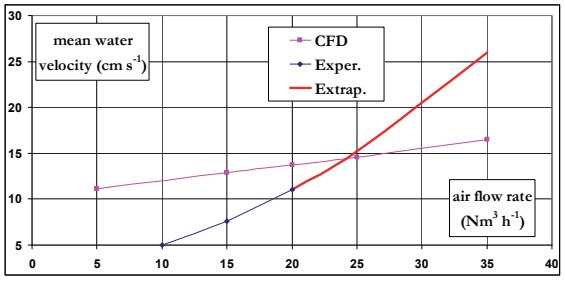

Figure 6: CFD and experimental mean water velocity in the airlift algal pond.

Tracer technique is used to get the experimental measurement of surface liquid velocity while the CFD result is a mean of more than 300,000 value of local velocity all over the algal pond.

4 CONCLUSIONS

The CFD contribution to the WWTP management is well appreciated when the developed models are validated and compared with experimental investigations. In our case the model elaboration is the limiting step in the simulation process while the experimental validation generally comes to comfort the obtained results. We do not think that CFD can contribute to the automation of the studied process but it can make it clearer and many scenario can be tested without being obliged at each step to make experiments. CFD results can be notably improved in gas liquid system studies (airlift bubble pump, surface aerators, etc...) by introducing the population balance module in local properties calculations and this is one of our main concerns for the moment. Further CFD investigations related to the optimisation of anaerobic digesters mixing are now conducted. The power consumption and the reduction of WWTP energetic expenses being our main target.

ACKNOWLEDGEMENTS

The authors would like to thank all the contributors to this work: Amira Maïzi, Ryma Achouri and Rym Ben Moussa.

REFERENCES

Achouri, R., Ben Hamza, S., Dhaouadi, H., Mhiri, H., Bournot, P., 2013. Volumetric mass transfer coefficient and hydrodynamic study of a new self-inducing turbine. *Energy Conversion and Management* 71, 69–75.

Achouri, R., Mokni, I., Mhiri, H., Bournot, P., 2012. A 3D CFD simulation of a self inducing Pitched Blade Turbine Downflow. *Energy Conversion and Management* 64, 633–641.

Fluent User Guide, septembre 2006, *Fluent Inc.*, Lebanon NH03766.

Maïzi, A., Dhaouadi, H., Bournot, P. Mhiri, H., 2010. CFD prediction of odorous compound dispersion: Case study examining a full scale waste water treatment plant. *Biosystems Engineering* 106, 68 – 78.

Hadiyanto, H., Elmore, S., Van Gerven, T., Stankiewicz, A., 2013. Hydrodynamic evaluations in high rate algae pond (HRAP) design. *Chemical Engineering Journal* 217, 231–239.

NOTATION

C	concentration (kg m^{-3})
Cp	calorific capacity (J kg^{-1} K^{-1})
D	mass diffusivity (m^2 s^{-1})
G	buoyancy production term
K	Stevens equation constant
k	turbulence kinetic energy (m^2 s^{-2})
P	turbulence kinetic energy production term due to mean velocity gradients
R	velocities quotient (v_0/u_∞)
T	temperature (K)
u, v, w	velocity component on x, y and z ($m.s^{-1}$)
$\overline{u_i'' u_j''}$	Reynolds tension
x, y, z	longitudinal, vertical and lateral components (m)

Greek symbols

ε	energy dissipation rate (m^2 s^{-3})
λ	thermal conductivity (w m^{-1} K^{-1})
μ	viscosity ($m^2.s^{-1}$)

Index

~	Favre mean
‾	mean
∞	ambient area
0	at the source outlet
k	turbulent kinetic energy
m	mixture species
t	turbulent
th	threshold
ε	turbulent dissipation rate