Larhyss Journal, ISSN 1112-3680, n°48, Dec 2021, pp. 151-167

WATER AGE AND LEAKAGE IN RESERVOIRS; SOME COMPUTATIONAL ASPECTS AND PRACTICAL HINTS

HARIRI ASLI H.[1*], NAZARI S.[2]

[1] Civil Engineering Department, Faculty of Engineering, University of Guilan, Rasht, Khalij Fars highway, 5th kilometer of Ghazvin road, Postal Code: 4199613776, Rasht, Iran,
[2] Department of Mechanical Engineering, Payame Noor University, Rasht, Iran,

[(*)] *hossein_hariri@msc.guilan.ac.ir*

Research Article – Available at http://larhyss.net/ojs/index.php/larhyss/index
Received October 4, 2021, Received in revised form December 5, 2021, Accepted December 8, 2021

ABSTRACT

The existence of reservoirs in the water system guarantees the stability of the system. Therefore, the level of water stored in the reservoirs should always be kept at optimum level in the water system. Inside the reservoirs, the mixing operation may not perform well. The water reservoirs with low delivery time, meet to the stationary water in the corners of the reservoirs. This can lead to water corruption and increase the risk of water leakage. Another factor that can increase the water retention time in the reservoirs is the shape and geometry of the reservoirs. The main purpose of the present work is to determine the impact of reservoir design specifications on the probability of water leakage and economic losses resulting by it. In this regard, this work focused on the effect of dimension, number, and optimal location for locating baffles on water spoilage factors and probability of water leakage. As a result, the use of three baffles with the same intervals perpendicular to the flow, created the least water retention time and water age. This case is the most optimal option to reduce economic losses and water leakage probability between the investigated options.

Keywords: computational modeling, reservoir, baffle, water retention time, Non Revenue Water.

INTRODUCTION

Water storage tanks or reservoirs are generally designed for raw water storage, drinking water storage, storage for distribution, disinfection, pressure supply, pressure reduction, collection and pumping upstream or a combination of these.

The horizontal sections of the reservoirs are generally square, rectangular, and in some cases circular, and rarely, due to the position and size of the ground, indefinitely, which is a combination of several rectangular squares (Hariri Asli, et al., 2012).

In large volume reservoirs 5000 cubic meters and above in terms of water movement in the tank, water retention time and water flow walls should be designed up to Total Water Level (T.W.L) so that the water path from the inlet pipe to the outlet pipe in a certain direction. In order to repair or washing the reservoir without cut off the water of the covered distribution network, it is necessary to divide the space of inside the reservoir into two or more independent sections. Each of these sections can be used as an independent and complete reservoir (Chowdhry, 2020).

The reservoirs are classified into two categories, air tanks and ground tanks, according to their position in relation to the ground level (Marek, et al., 2007).

The reservoirs that are built on framed or integrated bases to create proper pressure in the network due to the lack of high Natural Ground Level (NGL) in the area covered. Under normal circumstances, the volume of these reservoirs is limited and is determined according to need (Van Zyl and Haarhoff, 2007).

Iran is located in a region of the world which is classified as low rainfall in terms of rainfall with 251 mm per year. Factors such as population growth, industrial development and ecosystem conservation are increasing the demand for water. The average per capita consumption of urban water in the country about 260 liters per day with the same amount in the world about 140 liters per day is relatively high volume of effluent is estimated for Iran. The demand management and consumption management reduces the amount of consumption, but in any case, the waste of water can be considered as a source of Non Revenue Water (NRW). Due to population growth, industry expansion, improving public health and welfare and water stress in the world (Figure 1), the per capita amount of renewable water resources is decreasing. Therefore, it is necessary that water facilities based on a special method used. It will be possible by applying suitable functions, useful production and minimum cost in a suitable period of time (Hariri Asli, et al., 2012).

The successful operation for urban water system means proper use of equipment within their useful life. Water distribution facilities have the highest efficiency in their optimal operating conditions and when faced with less or more demand than its capacity, its efficiency is significantly reduced. Therefore, water retention time in reservoir is effective on operation. The water is stored in the hours of minimum demand and is used in the hours of maximum demand (Beirão da Veiga, et al., 2013).

The design of acceptable operation for reservoir needs to applying computational method. The model of Computational Fluid Dynamics (CFD) is an accurate method for investigating the effect of different parameters on water retention time in the corners of the reservoir. A model for simulating the flow and quality of water inside the reservoir was proposed by a group of scientists. They defined three different behaviors for the reservoir in their research and they examined the state of water quality. They wrote the relationships of mass transfer, mixing, and kinetic reaction for reservoirs. Then the obtained equations were solved numerically. This can be considered the beginning of mathematical modeling of water storage tanks with appropriate accuracy (Lee, 2017).

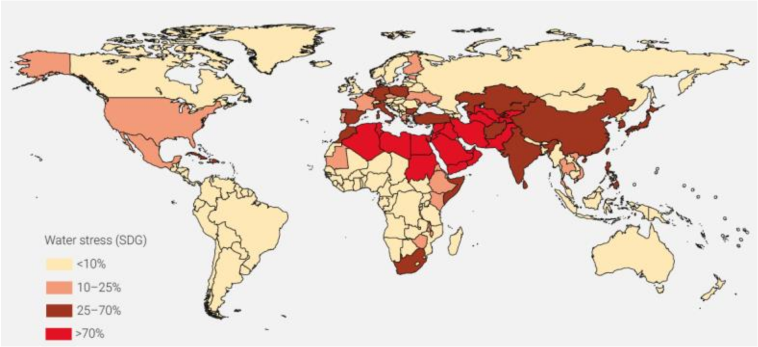


Figure 1: Global water stress (regardless of the effect of global warming), www.unwater.org.

Other researchers considered four different models for the reservoir (Chowdhry, et al., 2020) and examined water flow and water quality in each case. They stated that the quality of water inside reservoir is highly dependent on the geometry. The researches on the flow circulation in the reservoirs showed the relationship between the inflow and the total mixing of the flow inside the reservoir depends on the momentum ratio of the inflow to the buoyancy of the inflow. There is a similar relationship for output currents. They defined four factors: reservoir geometry, number of inlets and number of reservoir outlets and height to base ratio as four basic parameters affecting retention time and concluded that for cylindrical reservoirs with height to diameter ratio less than one may it created a perfect mixing capacity that will never be fully mixed in other reservoir and there will always be a slight deviation from the ideal state. They simulated the ideal reservoir for filling and emptying modes.

Another researcher (Yeung, 2001) used the computational method of fluid dynamics to study the behavior of a number of rectangular storage tanks. He studied the pattern of flow field and water age and showed that in reservoir with horizontal inlet mixing operations occur better than tanks with vertical inlet and at a high level. By increasing the length to width ratio in tanks with vertical inlet, the flow characteristics move more rapidly towards creep currents. Another group of scientists (Mahmood, et al., 2005) used a computational model of fluid dynamics to study various examples of storage tanks in Virginia's drinking water supply system. They compared the numerical results with experimental data obtained from measurements at different points in storage reservoirs. The results showed that the computational model has good accuracy in simulating for the fluid flow inside the reservoir and it can be used to determine the appropriate location of inlets and outlets of reservoirs. Other researchers considered location based concepts by the computational SPSS model based on Geography Information System (GIS) for civil engineering research (Hariri Asli, et al., 2012).

Today, due to the need to reduce of water leakage and increase water quality, by advances in numerical calculation methods, much research has been done in different parts of the world on how to reduce water retention time or age in water supply network and reservoirs (Zecchin, et al., 2005).

Increasing the age of water in the presence of chlorine causes oxidation of rebar in concrete and increases the possibility of water leakage. One of the available ways is to improve the hydraulic condition of concrete discusses in the present work. The subject is in line with the implementation of water consumption management and includes the following factors:

- Consumption management and loss reduction in the management of water supply facilities.
- Increase the useful life of water distribution facilities.

METHODS

In the new global view, water is considered as a basic human need. Although water is one of the renewable resources, its amount is limited (Walski, 2000). The behavior of hydraulic parameters including: velocity, water retention time and leakage due to the baffles position are in the field of study of present work. The simulation of the baffles position in the concrete ground reservoir is investigated by analyzing on hydraulic parameters.

The main purpose is to determine the impact of reservoir design specifications on hydraulic parameters and the possibility of water leakage and economic losses. Therefore, determining the dimensions, numbers and optimal location for the partition walls was also considered and as a research hypothesis, options one, two and three of the baffles and their effect on the field and flow lines are investigated.

The main question is: "How many baffles by creating eddy currents, can reduce the water retention time?"

To answer this question, in 2017 a concrete reservoir in the shape of a rectangular cube was studied in the Rasht city at the north of Iran. The volume of the reservoir was equal to 1500 cubic meters and its length and width were 25 and 20 meters (Figure 2) respectively. The water height was 3 meters. The diameter of the inlet and outlet pipes was 25 cm and the diameter of the injection pipe of the plug was 4 cm. The inlet and outlet water flow of the reservoir was 50 liters per second according to the data collection by the remote sensing flowmeters equipped with data logger. The volume of reservoir was determined to be 1500 cubic meters. Therefore, the full reservoir will be enough to supply water for 8 hours. It is almost in accordance with the standard. The average water retention time was 8 hours and the water inlet velocity as well as the injection rate of the plug was achieved to reach a flow rate of 50 liters per second at about 1 meter per second respectively. The length of the pipes is equal to 4 meters (3 meters outside the reservoir and 1 meter inside). Also, inlet and outlet and injection pipes were assumed to be in the middle height and the distance of the inlet pipe from the side wall was equal to 0.5 meters. The distance of the outlet pipe from its side wall was assumed to be equal to 0.5 meters (Figure 2). For the inlet pipe, the inlet velocity condition was used. For the walls, the non-slip condition was used and for the wall roughness, the value of 0.1 mm was assumed. For the free surface, the symmetry boundary condition was used. This condition increased the number of required iterations and decreased the degree of convergence in the problem. Then the zero shear stress boundary condition was used. This boundary condition somewhat improved the solution criteria. Therefore, it can be concluded that this boundary condition is more appropriate. The output pressure limit condition was used for the output. The species transport model was also used to investigate the concentration of injected agents.

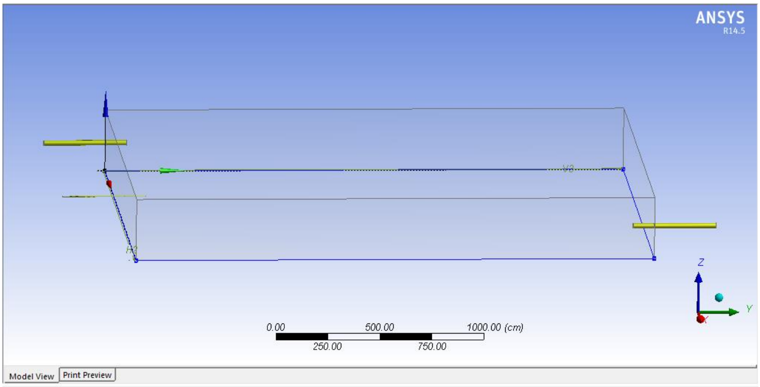


Figure 2: Schematic view of the reservoir under study with inlet, outlet and injection pipe (without baffle plates)

Formulation

In the (Figure 3), for no partition baffles case, the computational model is based on solving the conversion of mass equation and the Navier-Stokes equation for fluid motion. Therefore, if the flow is turbulent, the turbulent flow equations are added to the basic equations (1-23):

Conservation of mass:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i}\left(\overline{\rho u_i}\right) + \frac{\partial}{\partial x_i}\left(\overline{\rho' u_i'}\right) = 0 \tag{1}$$

Conservation of momentum:

$$\rho \frac{D\bar{V}}{Dt} + \rho \frac{\partial}{\partial x_i}\left(\overline{\rho u_i' u_j'}\right) = \rho g - \nabla \bar{P} + \mu \nabla^2 \bar{V} \tag{2}$$

In the above relation, μ kinematic viscosity, $\bar{P}$ pressure vector, $\bar{V}$ velocity vector, $g$gravity acceleration, $\frac{D\bar{V}}{Dt}$ express the ordinary derivative of velocity vector with respect to time.

$$\rho\,\frac{D\bar{V}}{Dt} = \rho g - \nabla\bar{P} + \nabla\tau_{ij} \tag{3}$$

In the above relation, τ_{ij}it expresses the tension tensor, which is defined as follows:

$$\tau_{ij} = \tau_{ij,Laminar} + \tau_{ij,Turbulent} = \left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \rho\overline{u_i^{'}u_j^{'}} \tag{4}$$

In the above relation, $\tau_{ij,Laminar}$ and $\tau_{ij,Turbulent}$ they express the laminar current stress tensor and the turbulent current stress tensor, respectively.

Conservation of energy:

$$\rho C_P\frac{D\bar{T}}{Dt} = -\frac{\partial}{\partial x_i}\left[-k\,\frac{\partial\bar{T}}{\partial x_i} + \rho C_P\overline{u_i^{'}T^{'}}\right] + \frac{\mu}{2}\overline{\left[\frac{\partial\overline{u_i}}{\partial x_l} + \frac{\partial u_l^{'}}{\partial x_j} + \frac{\partial\overline{u_j}}{\partial x_l} + \frac{\partial u_j^{'}}{\partial x_l}\right]^2} \tag{5}$$

In the above relation, C_P it expresses the specific heat capacity at constant pressure, $\bar{T}$between temperature, and k the thermal conductivity.

The kinetic energy equation of perturbation

$$K = \frac{1}{2}\overline{u_i^{'}u_i^{'}} = \frac{1}{2}(\overline{u^{'}u^{'}} + \overline{v^{'}v^{'}} + \overline{w^{'}w^{'}})) \tag{6}$$

$$\frac{DK}{Dt} = \frac{\partial}{\partial x_i}\overline{[u_i^{'}(\frac{1}{2}u_j^{'}u_j^{'} + \frac{P^{'}}{\rho})]} - \overline{u_i^{'}u_i^{'}}\frac{\partial\overline{u_j}}{\partial x_i} + \frac{\partial}{\partial x_i}\overline{[\nu u_j^{'}(\frac{\partial u_i^{'}}{\partial x_j} + \frac{\partial u_j^{'}}{\partial x_i})]} - \overline{\nu\frac{\partial u_j^{'}}{\partial x_i}\left(\frac{\partial u_i^{'}}{\partial x_j} + \frac{\partial u_j^{'}}{\partial x_i}\right)} \tag{7}$$

In the above relation, K it expresses the kinetic energy of turbulence and νexpresses dynamic viscosity.

The intensity TI perturbation is related to the kinetic energy and the mean Uref as reference current velocity as Equation (3-8).

$$TI = \frac{[\frac{1}{3}(\overline{u^2} + \overline{v^2} + \overline{w^2})]^{0.5}}{U_{ref}} = \frac{[\frac{2}{3}K]^{0.5}}{U_{ref}} \tag{8}$$

Conservation of mass:

$$\frac{\partial\rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{9}$$

$$\frac{\partial\rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial P^{'}}{\partial x_j} + \frac{\partial}{\partial x_i}\left[\mu_{ref}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + S_M \tag{10}$$

SM sum of physical forces, μ_{eff} effective viscosity and P´. The pressure is defined by the following relation:

$$P' = P + \frac{2}{3}\rho k + \frac{2}{3}\mu_{eff}\frac{\partial u_k}{\partial x_k} \tag{11}$$

According to the following relation, kinetic energy depends on the flow of energy.

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{12}$$

The value of k and ε and C_μ is a fixed number.

The ordinary differential equations of kinetic energy transfer and perturbation energy drop rates are obtained as follows:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j k\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon + P_{kb} \tag{13}$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j \varepsilon\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2}\rho\varepsilon + C_{\varepsilon 1} P_{\varepsilon b} \tag{14}$$

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k , σ_ε and σ_ε are constant. P_{kb} and $P_{\varepsilon b}$ exert the influence of the Buoyancy forces. P_k produces turbulence due to viscous forces.

$$\mathrm{Pk} = \mu_t \left(\frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\left(3\mu_t \frac{\partial u_k}{\partial x_k} + \rho k\right) \tag{15}$$

It is assumed that viscosity or perturbation $k-\omega$ related by the following kinetic energy relationship of perturbation and perturbation frequency:

$$\mu_t = \rho \frac{k}{\omega} \tag{16}$$

The values of k and ω are obtained from the differential equations of kinetic energy transfer and perturbation frequency:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j k\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta' \rho k \omega + P_{kb} \tag{17}$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j \omega\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \alpha \frac{\omega}{k} P_k - \beta\rho\omega^2 + P_{\omega b} \tag{18}$$

$$\beta' = 0.09, \alpha = \frac{5}{9}, \beta = 0.075, \sigma_k = 2, \sigma_\omega = 2 \tag{19}$$

$$\mathrm{F2} = \tanh(arg_2^4) \tag{20}$$

$$\mathrm{arg1} = \min[\max(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho\omega y^2}), \frac{4\rho k}{CD_{k\omega}\sigma_{\omega,2} y^2}] \tag{21}$$

$$\mathrm{arg2} = \min[\max(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho\omega y^2})] \tag{22}$$

$$CD_{k\omega} = \max\,[2\rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10}] \tag{23}$$

By fomulation of this work, the instantaneous fluid flow equations are simulated. This type of modeling is the most complete case. Generally numerical methods used in the computational model are: finite element method, finite volume method, finite difference method, spectral methods. Therefore, finite volume method is more used in modeling incompressible flows.

The CFD software output based on finite volume method used for modeling of this work were. The problem was meshed by the algorithm inside computational software for meshing. For roughness, 0.1 mm was considered for the inner wall and it was assumed that the water level inside the tank was constant. By selection of unstructured mesh type, the hexagonal (octagonal) cells was used for the inside volume of reservoir. Mesh with different dimensions and types were examined and the duration and numbers of iterations for the convergence of the problem in each case were compared with each other on order to use the best type of mesh. Finally, the view of meshing criteria embedded in computational software was used.

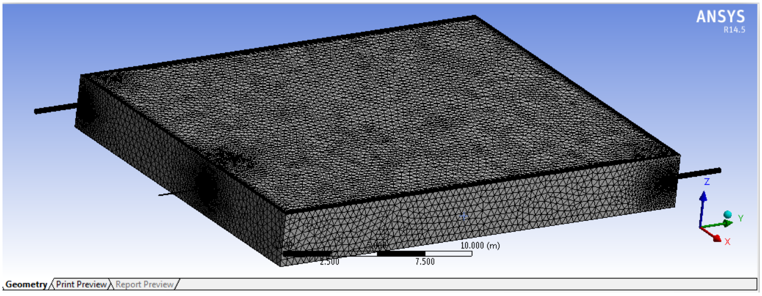


Figure 3: View of the reservoir meshing

Figure 3 shows a mesh view of the problem. The mesh around the inlet, outlet and injection pipes is more compact. Due to the formation of the boundary layer around the pipes and also the sensitivity of the flow in these points, the number of meshes in these points has increased so that the phenomenon can be well investigated. Therefore, the problem mesh included 601508 nodes and 2027645 elements.

RESULTS AND DISCUSSION

This work simulated the mixing process in the reservoir and evaluated the performance of it. These parameters were divided into two parts: Euler parameters and Lagrange parameters. Euler parameters are related to interference based on hydrodynamic parameters. These parameters indirectly affect the mixing process and fluid flow. The input power is the main variable in this area. For other parameters in the first group, it is necessary to mention the intensity of turbulence, retention time and velocity gradient. In the second group, there are items such as concentration histogram. As can be seen in (Figure 4-7), after 1387 iterations, the converges and the errors reach to less than 0.001.

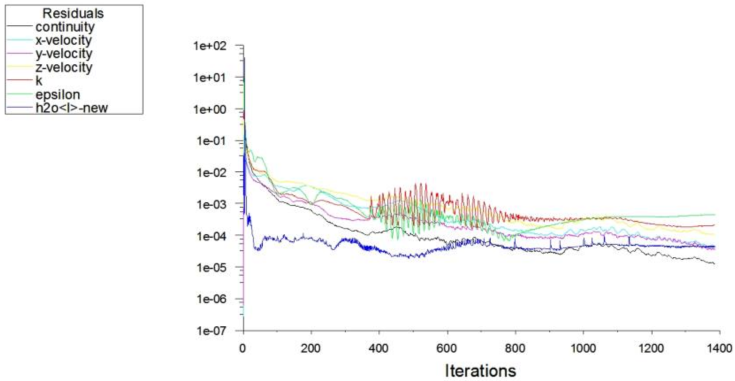


Figure 4: Residual curve and concentration histogram

The results showed that if additives are injected into the reservoir, the concentration of these materials in the corners will be greatly reduced, and as a result, this type of reservoir will have many disadvantages.

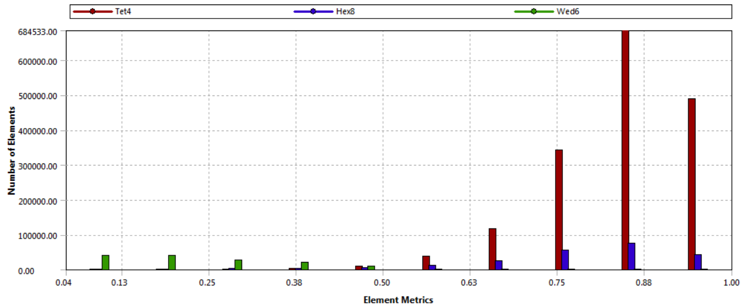


Figure 5: Element matrix against the numbers of elements; Evaluation of element quality due to element quality criteria

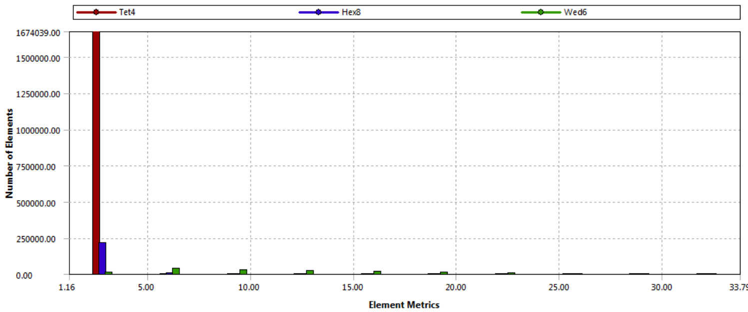


Figure 6: Element matrix against the numbers of elements; Evaluation of element quality due to Shape factor

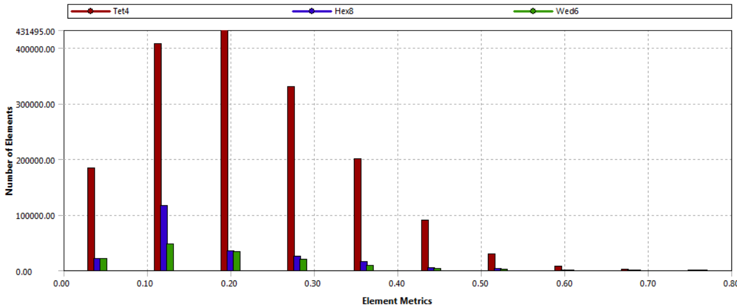


Figure 7: Element matrix against the numbers of elements; Evaluation of element quality due to standard error

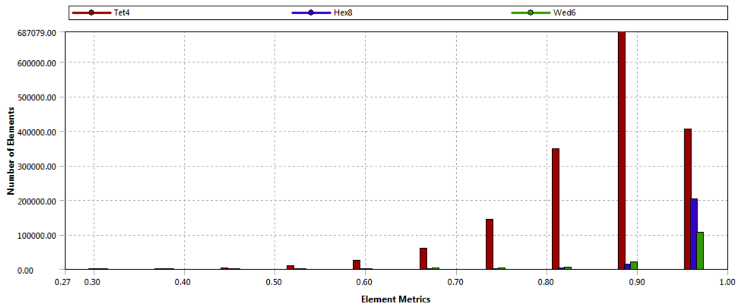


Figure 8: Element matrix against the numbers of elements; Evaluation of element quality due to standard error

In order to solve this problem, the baffle should be used. Therefore, the effect of using baffle on the flow field was studied. By using the baffle leads to the formation of eddy currents and as a result stationary condition is formed in the corners of the reservoir. Therefore, water age and intention time increase in these areas. By comparing the flow lines and the different modes under consideration, it can be concluded that the reservoir-related option with three baffle cases mounted completely vertically in front of the inlet current is the best expected mode (Figure 8-12). To avoid the complexity of the problem, in the next models, the injection of additives was omitted and the velocity vector and flow lines were determined to determine the stationary points and points with long retention time. In this case, two baffle plates (Figure 9-10) were considered. In this case, the height of the baffle plates is assumed to be equal to 3 meters, their width is equal to 15 meters, and the distance of the first baffle plate from the inlet pipe is equal to 8 meters and the distance of the second baffle plate from the first baffle plate is assumed to be equal to 8 meters. In this regard, the velocity contour, flow lines and velocity vector are shown in (Figure 9-10).

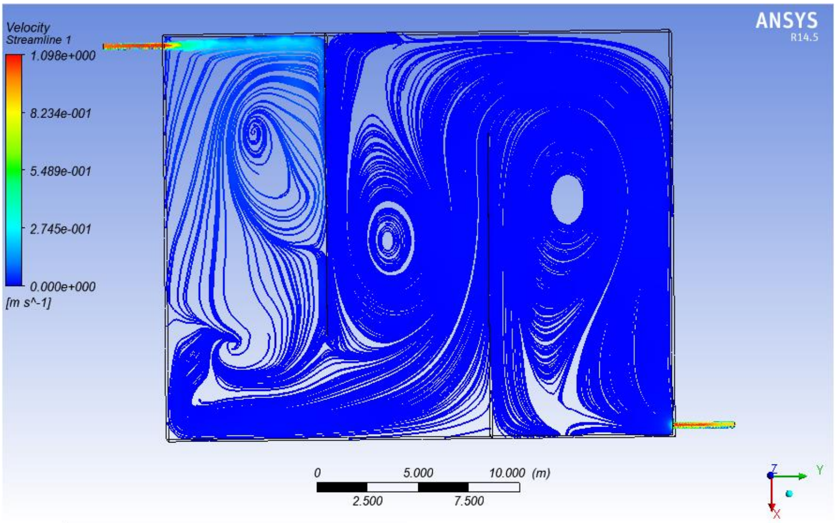


Figure 9: Flow lines for a three-dimensional reservoir consisting of two baffle plates at a distance of 8 meters from each other

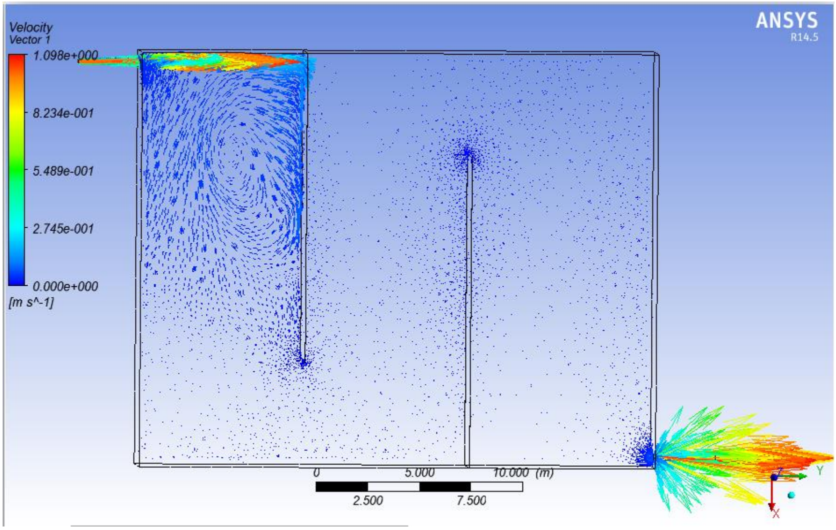


Figure 10: Speed vector for three-dimensional reservoir consisting of two baffle plates at a distance of 8 meters from each other

For a baffle plate, it is assumed that the baffle plate is embedded in the middle of the tank. In this case, the geometry of the problem is shown in (Figure 11). In this case two eddy currents are formed in the first part of the tank, but in the second part of the reservoir no eddy current is formed and the stationary points with high retention time in this case are extremely high. It can be concluded that using a baffle in the middle of the tank will not be appropriate at all.

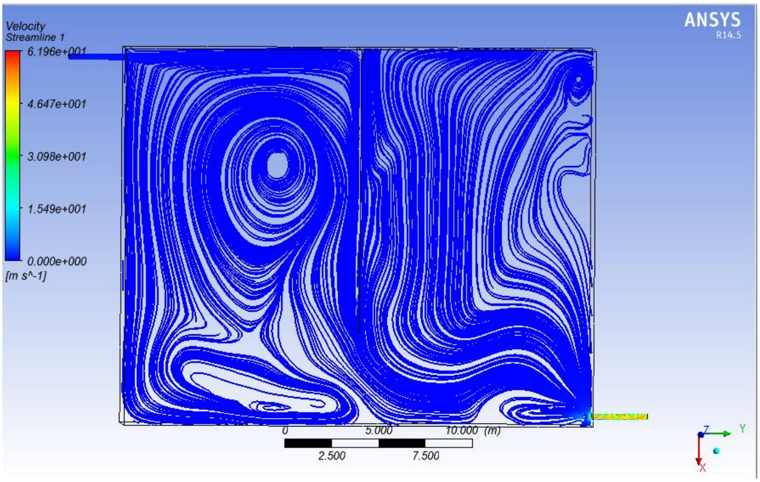


Figure 11: Flow lines for a three-dimensional reservoir including one baffle plate in the middle of the reservoir

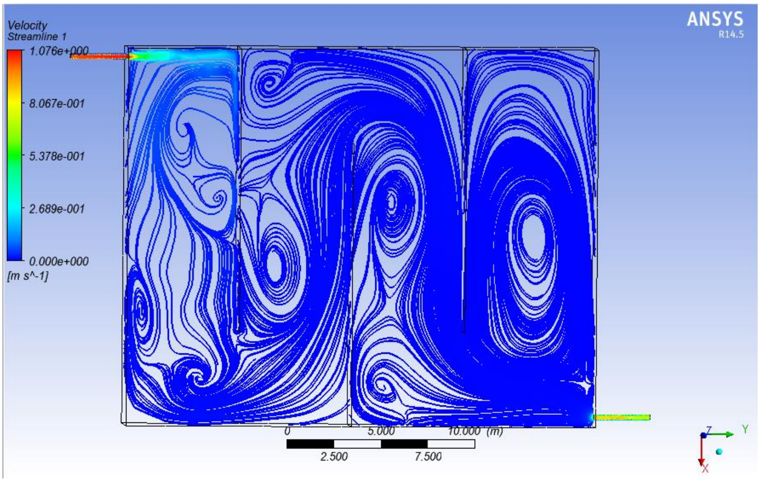


Figure 12: Flow lines for a three-dimensional reservoir consisting of three baffle plates at the middle height of the reservoir

As can be seen in Figure 12, by using three baffles plates, a large numbers of eddy currents were formed in different areas, which led to a decrease in stagnation points and thus reduced water retention time. Therefore, using three baffles plates has a comparative advantage over the previous cases and increasing the numbers of baffles plates and the presence of a vortex in general leads to an increase in pressure drop in the reservoir.

COMPARISON OF PRESENT WORK RESULTS WITH OTHER EXPERT'S RESEARCH

Present work, studied on the one, two and three baffles cases effects on eddy currents in reservoir. Results showed the stagnation points were created in the first region, but no eddy current forming in the first region. Results also showed that in the one baffle use, eddy current formed in the second region. The flow velocity in the corners has been reduced so much more than in the corners of the second zone. The stagnation points were formed and the age and time of water retention in these points increased. Then the option for case of two baffles was considered. In this case, more eddy currents were generated. As a result, the situation is better than before.

Another group of experts (Mahmood, et al., 2005) used a computational model of fluid dynamics to study various examples of reservoir in Virginia's drinking water supply system. They concluded the same results with the present work. The separating wall perpendicular to the stream would lead to better mixing in the tank. They claimed that the existence of a temperature difference between the flow of fluid entering the tank and the fluid stored in the reservoir could itself be an effective factor in mixing. For this purpose, the effect of this parameter was investigated numerically by using a computational model (Figure 13).

In 2009, other researcher (Gualtieri, 2009) compared the hydrodynamics as well as the turbulence of the flow inside the cylindrical storage tank. He examined the problem in a two-dimensional state, assuming that the current is stable and also unstable. He examined the problem for four different arrangements of the reservoirs and compared their hydraulic performance one by one. He examined the problem once in the steady state and once in the transient state numerically. Then, in the case of four different geometries of the baffle inside the columnar reservoir, the amount of mixing was compared with each other. The results showed in (Figure 14) related to the contour distribution of four reservoirs with internal arrangement of baffles.

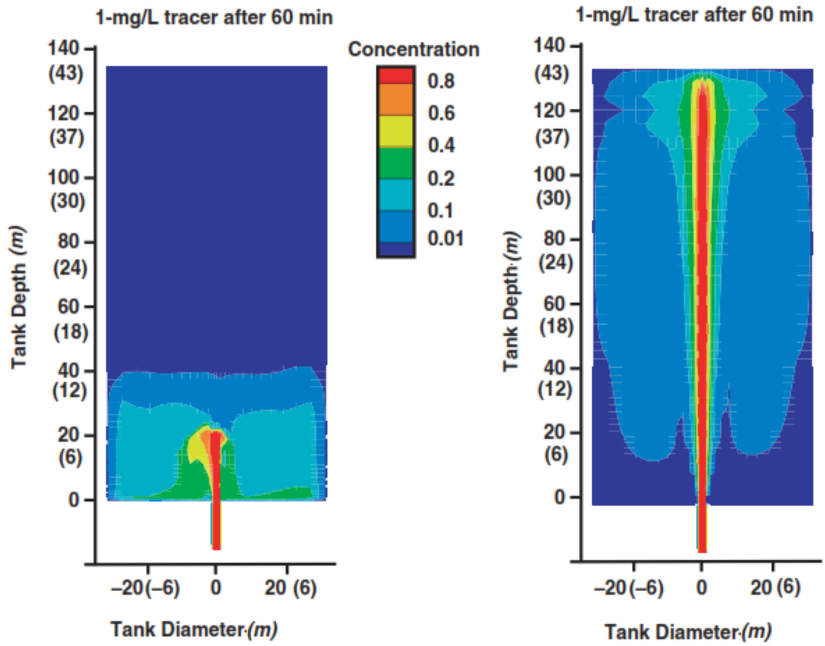
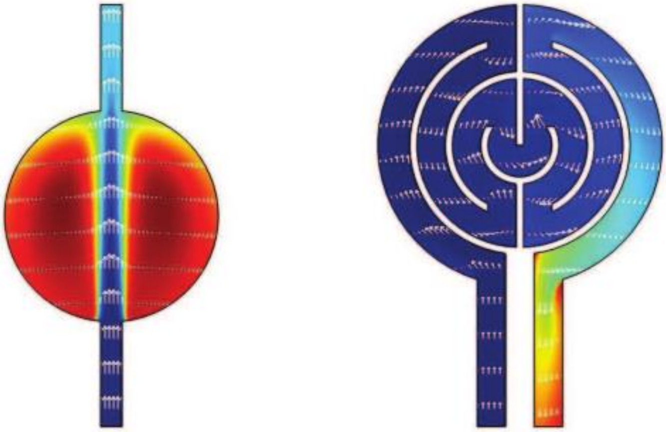


Figure 13: Flow lines for the vertical inlet reservoir in the research of another group of scientists (Mahmood, et al., 2005)



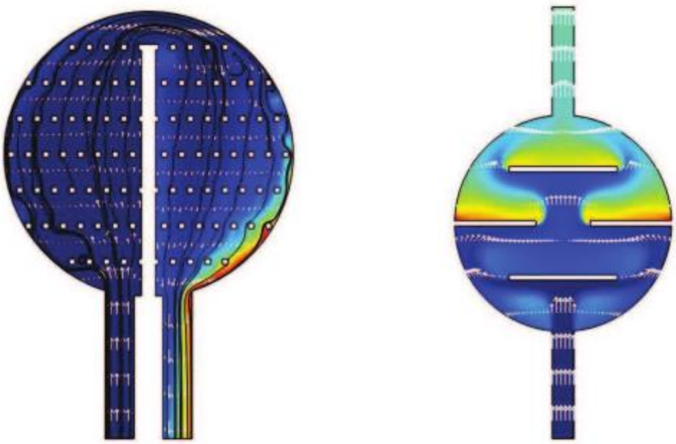


Figure 14: Contour distribution of four reservoirs with four types of internal baffle arrangement (Gualtieri, 2009)

CONCLUSION

In this work, the use of the baffle installation in reservoir was investigated. In the case of one baffle use, two eddy currents were created in the first region, but no eddy current was formed in the second region. The flow velocity in the corners has been reduced so much that in the corners of the second zone, stagnation points were formed and the age and time of water retention in these points were increased. Then the option of using the case of two baffles was considered. In this case, more eddy currents were generated. As a result, the situation was better than before. Investigation of the effect of distance of baffle plates from each other on the flow field showed that the reduction of baffle plates leads to weakening of vortex strength, especially in the second and third zones. With the increase in the numbers of baffles to three, the eddy currents were increased sharply and almost all stagnation was completely eliminated. In the option of three baffles plates, a large numbers of eddy currents were formed in different areas, which led to a decrease in stagnation points and thus reduced water retention time. Therefore, using three baffles plates has a comparative advantage over the previous cases and increasing the numbers of baffles plates and the presence of a vortex in general leads to an increase in pressure drop in the reservoir. In order to use corrugated baffle plates and investigate the effect of size and height of baffle plates on the flow field, a supplementary study should be done. Because increasing the age of water can cause oxidation of rebar in concrete in the presence of chlorine and increase the possibility of water leakage and Non Revenue Water (NRW) and economic losses.

ACKNOWLEDGMENT

The authors thank to all specialists for their valuable observations and advice, and the referees for recommendations that improved the quality of this paper.

REFERENCES

BEIRÃO DA VEIGA L., BREZZI F., CANGIANI A., MANZINI G., MARINI L. D., RUSSO, A. (2013). "Basic principles of Virtual Element Methods", Mathematical Models and Methods in Applied Sciences, Vol. 23, Issues 1, pp. 199–214, UN-WATER, Coordinating the UN's work on water and sanitation, https://www.unwater.org/.

CHOWDHRY A. (2020). "Ansys: Student Software Program Hits Several Major Milestones".

GUALTIERI C. (2009). Analysis of flow and concentration fields in a baffled circular storage tank, XXXIII IAHR Congress, August, 10/14, pp.4384-4392, Vancouver, Canada.

HARIRI ASLI K., NAGHIYEV F.B.O., HAGHI A.K., HARIRI ASLI H. (2012). Advances in Control and Automation of Water Systems, Chapters: 1 to 9, Published by the Apple Academic Press, Inc., ISSN: 978-1-926895-22-2, USA, Canada, 190p. https://www.appleacademicpress.com/advances-in-control-and-automation-of-water-systems-/9781926895222

LEE H.H. (2017). Finite Element Simulations with Ansys Workbench 17, SDC Publications, p. 51, ISBN 978-1-63057-088-0.

MAHMOOD F., PIMBLETT J.G., GRACE N.O., GRAYMAN W.M. (2005). Evaluation of water mixing characteristics in distribution system storage tanks, J.AWWA, Vol. 79, Issues 3, pp74–88.

MAREK M., STOESSER T., ROBERTS P. J. W., WEITBRECHT V., JIRKA G. H. (2007). “CFD modeling of turbulent jet mixing in a water storage tank,”XXXII IAHR Congress, July 1/6, Venice, Italy.

VAN ZYL J. E., HAARHOFF J. (2007). Reliability Analysis of Municipal Storage Reservoirs Using Stochastic Analysis, Journal of the South African Institution of Civil Engineering, Vol. 49, Issues 3, pp. 27–32.

WALSKI T. M. (2000). Hydraulic Design of Water Distribution Storage Tanks, In Water Distribution Systems Handbook, Chapter 10, ed. L. W. Mays, Mc Graw-Hill, New York, USA.

YEUNG H. (2001). Modeling of service reservoirs, Journal of Hydroinformatics, Vol.3, Issues 3, pp. 165–172.

ZECCHIN A.C., MAIER H.R., SIMPSON A.R., LEONARD M., NIXON J.B. (2005). Parametric Study for an Ant Algorithm Applied to Water Distribution System Optimization, IEEE Transitions on Evolutionary Computation, Vol. 9, Issues 2, pp.175-191.