

**INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH
TECHNOLOGY****APPLICATION OF COMPUTATIONAL FLUID DYNAMICS MODELLING TO A
HORIZONTAL SEDIMENTATION TANK IN IRAQ****Ali Hadi GHAWI**

University of Al-Qadisiyah, Collage of Engineering, Department of Civil Engineering, Iraq

DOI: 10.5281/zenodo.495142

ABSTRACT

Computational Fluid Dynamics modeling has been applied to examine the hydrodynamic behavior of water treatment sedimentation tanks at Baghdad Water Works, operated by Alkurech Water in Baghdad in Iraq. The existing tanks perform poorly at current flows and flow is unevenly split among online tanks, Therefore, CFD was used to investigate velocity profiles at current and projected loadings for the existing basins. Results from the CFD analysis were used to develop retrofit strategies to improve velocity profiles , design and operation of sedimentation tanks in the existing basins. The paper shows how CFD can be helpful in order to improve the design of existing water treatment plants and the process efficiency. In this scope, CFD enables designs more reliable and cost effective water treatment process which remain basic environmental policy issues.

KEYWORDS: Computational Fluid Dynamics Modeling, Finger Baffle, Sedimentation Tank, Velocity Profiles**INTRODUCTION**

Sedimentation is perhaps the oldest and most common water treatment process. The principle of allowing turbid water to settle before it is drunk can be traced back to ancient times. The Assyrians, Egyptians, Incas and Romans made extensive use of this technique [1].

Sedimentation tanks are the workhorses of any water purification process. It is thus crucial for the sedimentation tank to be operated to its full potential. It is not only the chemical aspects of flocculation that cause problems, however. Hydraulics also play a prominent part. Overdesign of plant is common, leading not only to unnecessary capital expenditure, but also to water wastage in the form of excessive sludge. Inadequate design causes overloading of filters, leading to frequent backwashing which also wastes a significant percentage of treated water. Many plants are already a few decades old and do not incorporate the latest developments in technique, e.g. inlet design. Sedimentation tank performance is strongly influenced by hydrodynamic and physical effects such as density driven flow, gravity sedimentation, flocculation and thickening. In turn the velocity and density patterns in tanks influence these processes and are therefore of great interest to design engineers.

Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow (gases or liquids) by means of computer-based simulation. It is a research tool and a design tool and it is complementary to theory and experiments. CFD can also be described as a method to investigate and simulate fluid flow by means of iterative calculations on computers. It was developed originally to study aerodynamics, but has since been applied to many different types of flow under a great variety of conditions. A recent application (since ca 1990) is to use it for the simulation of unit processes in water treatment, e.g. chlorine and ozone contactors, sedimentation tanks and sludge thickeners.

CFD has reached a level of development where it has become a useful tool for research, design and operation in water technology. It is now possible to use standard software packages and desktop computers to analyse and investigate fairly complex situations in terms of configurations and conditions within reasonable time periods and at lower cost than doing experimental work. Very realistic models that mimic nature in such a way that the behavior of the models cannot be distinguished from that of the physical systems they are modeling can be developed. Such models can therefore be used with confidence to investigate the characteristics and behavior of the physical systems, including the effects of changes in or to it. A number of studies have investigated sediment distribution and flow patterns in sedimentation tanks and clarifiers. Several of the studies [2, 3 , 4 , 5, 6, 7, 8, 9, 10, and 11], have been carried by use of packaged CFD software.

[Ghawi A. Hadi., 6(4): April, 2017]
ICTM Value: 3.00

Computational Fluid Dynamics modeling has been applied to examine the hydrodynamic behaviour of water treatment sedimentation tanks at Baghdad Water Works, operated by Alkurech water in Baghdad Iraq. The aim of this paper was originally stated as to improve the operation and performance of equipment used in water treatment plants which have been identified as operating poorly, to be achieved by predicting the existing flow distribution of the equipment by means of CFD techniques. The existing tanks perform poorly at current flows and flow is unevenly split among online tanks, Therefore, CFD was used to investigate velocity profiles at current and projected loadings for the existing basins.

MATERIALS AND METHODS

For the investigation of the actual phenomena in the horizontal sedimentation tanks, we selected a certain domestic water treatment plant (WTP). S_WTP. Fig. 1 and Fig. 2 show the plan view of horizontal sedimentation tank at S_WTP. The S_WTP treatment plant uses lime, ferric chloride and polymer to flocculate the colloidal clay suspension before entering the sedimentation tanks. The current horizontal sedimentation tanks consists of eight rectangular sedimentation basins, each 45 m long by 6 m wide by 3 m deep. The tanks on the old plant have the following features: (1) Isolation gates for upstream and downstream isolation, (2) Cedar finger baffles, (3) Sludge hoppers, (4) Two inlet ports with target baffles, (5) Chain-and-flight sludge collection mechanisms, and (5) two submerged effluent collection pipes. The CFD model of the existing basins was modified to incorporate retrofit strategies to minimize the problems described above.

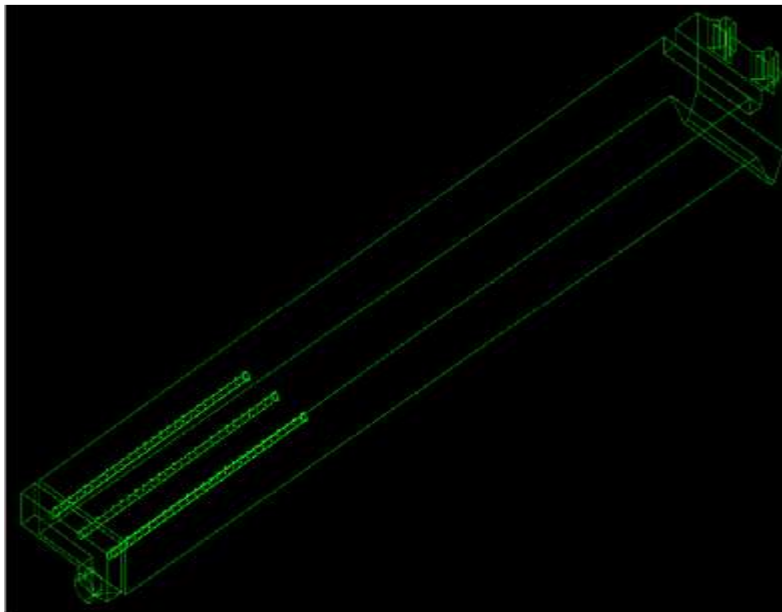


Figure 1: Existing Alkurech Sedimentation Tank.

The following modifications were made to the basins, as shown on Fig. 3: (1) Removed inlet target baffles, (2) Replaced effluent collection pipes with effluent weirs, (3) Installed modified finger baffles to include a bottom “floor”, and (4) Removed finger baffles. Each tank has three 12 m effluent launders. Each launder is simulated in the model with a rectangular weir located on each side of the launder. Two inlets provide flow to each basin, and a single finger baffle is located 0.5 m downstream of the junction between the inlets and the basin. A floor piece supports the baffles and directs flow longitudinally down the tank. Baffle spacing is designed to equalize flow across the tank’s width. Only one tank was simulated to reduce the computational grid size.



Figure 2: Photo of Alkurech sedimentation tank.

3. METHODOLOGY OF CFD SIMULATION

The CFD simulation works by splitting the geometry of interest into a large number of elements, collectively known as „grids“ or „cells“. Then, momentum and continuity equations were formulated for each grid together with given boundary conditions. In this simulation, we generated 295,000 cells for geometry of horizontal sedimentation tank. For simulating the hydrodynamic behaviour in the horizontal sedimentation tank, ANSYS program, commercial code was used. The 3D program used in this simulation contains the continuity equation, momentum equation (Navier-Stokes), energy equation and volume of fluid (VOF) method suggested by Hirt and Nichols (1981) [12] for free surface dynamics. The time – averaged Navier-Stokes equations for momentum and continuity were solved in study for steady, incompressible, turbulent and isothermal flow.

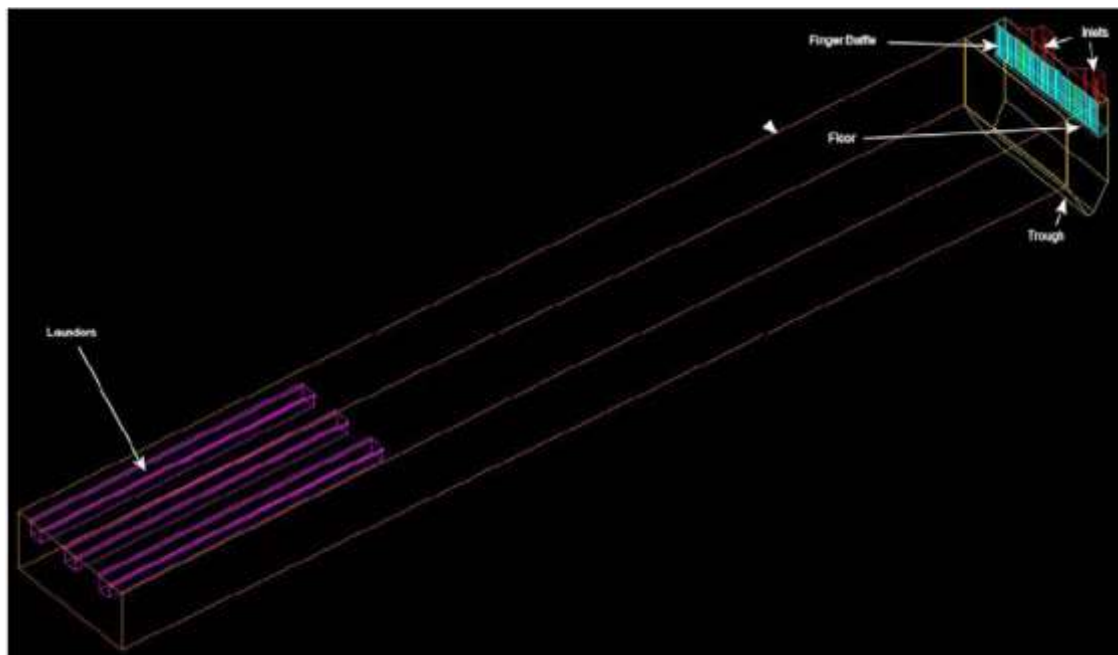


Figure 3: Alkurech Sedimentation Tank –Revised With Launderers And Finger Baffles, Depicted.

The continuity and momentum equations are, as follows, respectively:

$$\nabla \cdot (\underline{U}) = 0 \quad (1)$$

$$\nabla \cdot (\rho \underline{U} \otimes \underline{U} - \mu \Delta \underline{U}) = \underline{B} + \nabla P - \nabla \cdot (\overline{\rho \underline{u} \otimes \underline{u}}) \quad (2)$$

where ρ and μ are the fluid density and dynamic viscosity, respectively; P the pressure; \underline{U} the fluid mean velocity; \underline{B} a body force; and \underline{u} the fluctuating velocity

4. TURBULENCE MODELING

ANSYS program provides for a variety of different turbulence models, of various complexity and applicability. The most widely used general-purpose model appears to be the k- ϵ model [13]. This has two components: k is the kinetic energy of the fluctuating turbulent component at a point, and ϵ is the kinematic rate of dissipation of k . The k- ϵ model has been used successfully for sedimentation tanks modeling [14], and has been used in the investigations reported in this paper.

5. BOUNDARY CONDITIONS

The above equations form a set of partial differential equations. In order to obtain a unique solution, this set needs to be linked to a set of boundary conditions. The boundary conditions include:

- The inlet was specified as a uniform velocity, k and ϵ values.
- The overflow outlets were specified in the top row of cells on either side of the overflow weirs. A zero gradient boundary condition was imposed for all variables. In order to conserve mass, the outflow of mass was balanced with the inflow. To ensure a uniform distribution across the overflow weirs, a number of boundaries was specified, each with a fixed mass flux ratio. The total overflow mass flux ratio was set to 96%.
- The underflow outlet was specified in the narrow part at the bottom of the hopper(s). The total mass flow ratio of the underflow was set at 4%. This accounted for the removal of sludge from the hopper.
- The vertical and horizontal walls were specified as near wall boundaries using the standard k- ϵ model [15].
- The static free surface was specified as a rigid lid symmetry axis. The variables that were not specified according to a rigid lid symmetry axis were the kinetic energy dissipation (where the values as used by Zhou and McCorquodale (1992) [14] were specified).

6. RESULTS AND DISCUSSION

6.1 Simulation of Existing Sedimentation Tanks

Computational fluid dynamics (CFD) is a numerical procedure to calculate the properties of moving fluid. Most water treatment processes involve the movement of water. This motion is often complex and difficult or very expensive to observe. The prediction of the flow patterns and other properties of flowing fluids would provide insight into processes which otherwise would not have been possible. Fig. 4 shows the velocity profiles of the

[Ghawi A. Hadi., 6(4): April, 2017]
IC™ Value: 3.00

existing tank for a flow rate of 35 Ml/d. the simulation revealed that influent flow is directed downward by the finger baffles at the upstream end of the basin. Fig. 4 also shows that the existing finger baffles direct flow downward toward the sludge hoppers and basin floor, rather than dissipate flow uniformly across the basin. This flow path results in a higher velocity at the basin floor, which most likely interferes with the sludge removal process. The strong bottom current is balanced by a surface return current inside the hopper. The velocities near the overflow launder are low.

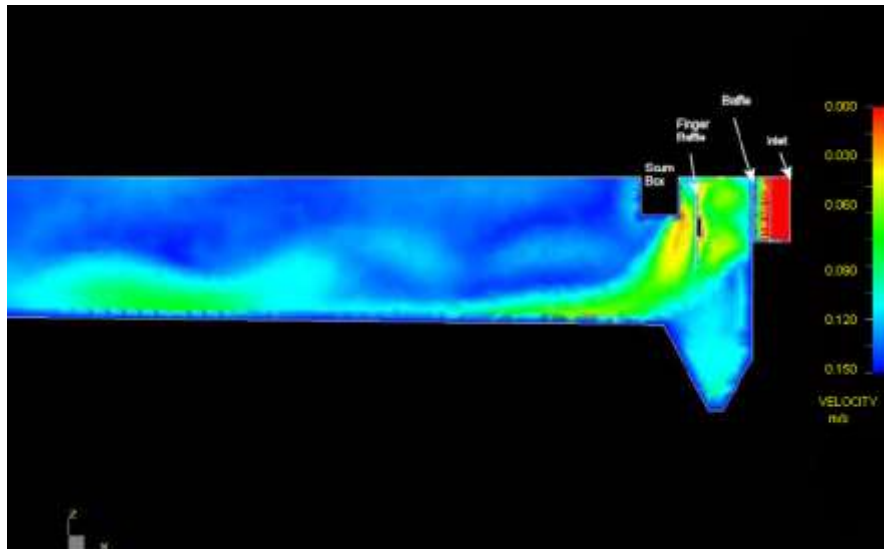


Figure 4: Velocity Magnitude Contours m/s of Existing Tank.

6.2 Simple Improvements To The Existing Sedimentation Tank

The main cause of the high velocities in the hoppers of the existing tanks, is the potential energy at the inlet (due to the density differences and the existing finger baffles) that is converted into kinetic energy as the particles settle in the sludge hopper. This upsets the settled and settling particles in the hopper. The inlet target baffles were removed because they create strong flow patterns at angles downstream rather than rapidly dissipate inlet energy across the width of the basin. Furthermore, these baffles are not needed with the proposed finger baffle arrangement.

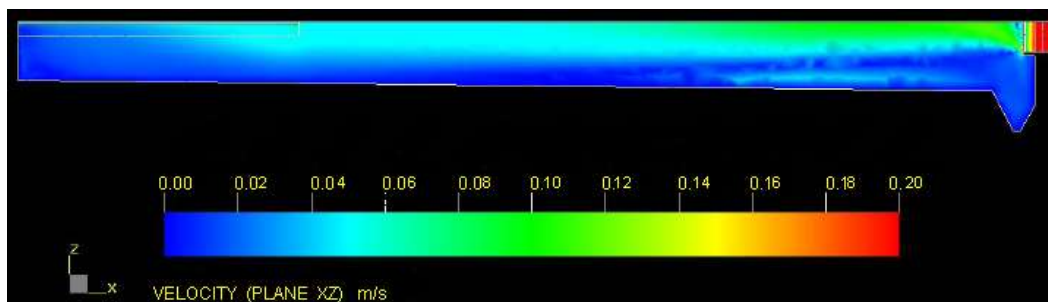


Figure 5: Velocity magnitude contours m/s for the retrofit tank.

As shown on Figs. 5 and 6, the proposed finger baffles rapidly dissipate inlet velocities across the basin, both laterally and longitudinally. The bottom floor of the baffles prevents flow from being directed downward, so floor scour of settled solids is minimized. Velocities are low at the effluent weirs, which should result in an even flow split across the weirs in each basin. Note that perfect symmetry of flow is not realized in the CFD simulation, or expected, because of the randomness of turbulence and imperfect convergence of the calculations.

Figs. 7 through 10 show the average velocity in the sedimentation tank is about 0.020 m/s, while the inlet

[Ghawi A. Hadi, 6(4): April, 2017]
ICTM Value: 3.00

velocity is 0.200 m/s. At 35 Ml/d, the average velocity through the baffle openings is 0.140 m/s. CFD computations showed less than 0.004 m of head loss through the baffles, which correlated to a minor loss coefficient of 1.5, assuming a velocity head associated with 0.140 m/s velocity.

The floor member that supports the baffles keeps streamlines pointing generally down the tank, not toward the tank bottom (Figs. 7 and 8). Turbulence in the trough is mild (Fig. 8). Perfect symmetry is not realized because of the randomness of turbulence and imperfect convergence of the calculations (Figure 9). The progression and breakup of the jets issuing from the baffle openings can be seen on Fig. 10.

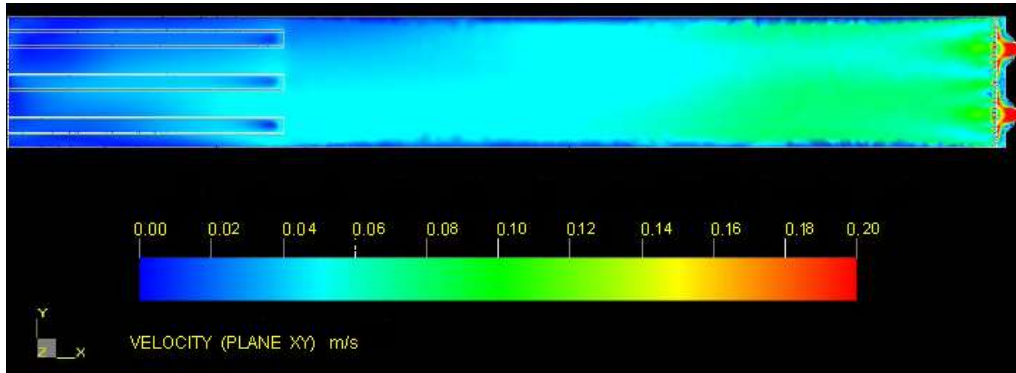


Figure 6: Velocity magnitude contours m/s are shown along a horizontal slice located 0.06 m above the floor of the finger baffles.

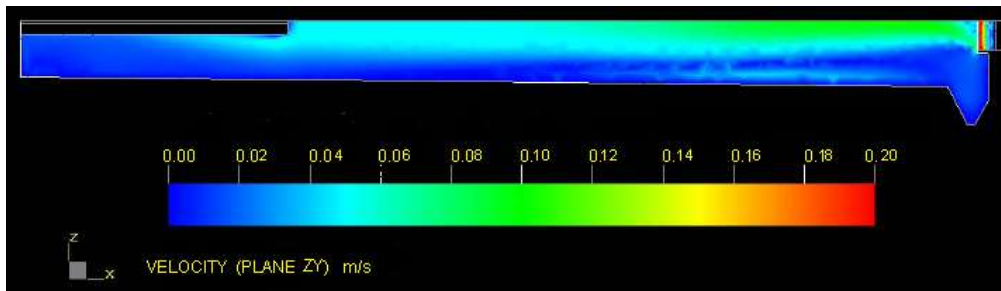


Figure 7: Velocity magnitude contours m/s for the retrofit tank at mid-depth of the nearest launder is shown.

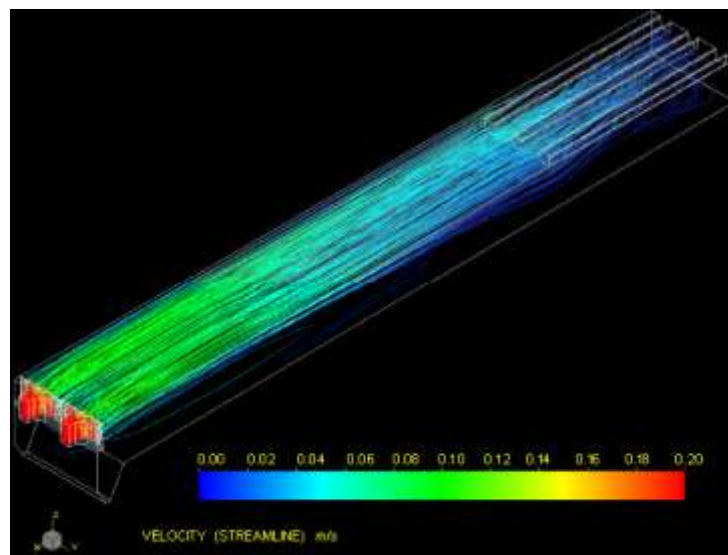


Figure 8: Velocity magnitude contours m/s of existing tank and streamlines emanating from the inlets. Circulation in the trough is moderate.

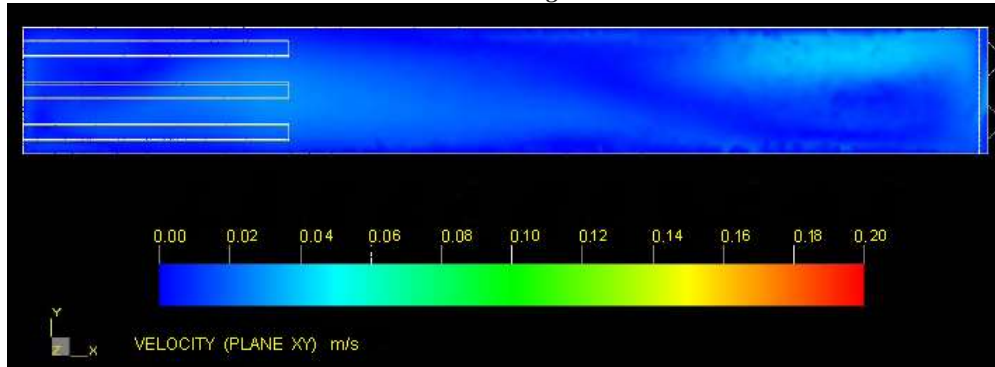


Figure 9: Velocity magnitude contours along a horizontal slice located 0.06 m below the floor of the finger baffles.

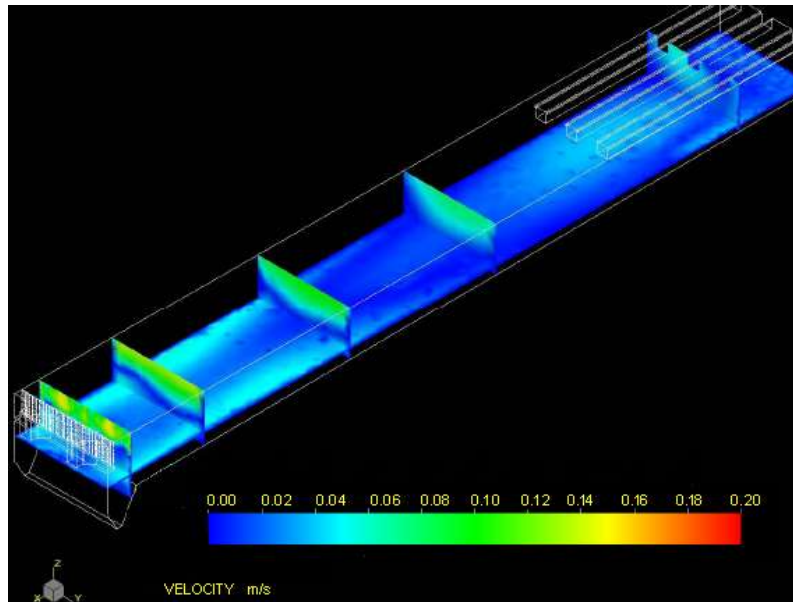


Figure 10: Shown are a series of vertical slices that depict velocity along the sedimentation tank

7. CONCLUSION

From the results of the various investigations the following conclusions can be made:

- Flow is not split evenly among the existing basins because of asymmetrical influent and effluent channel configuration, and short-circuiting into Basin. Flow split can be improved by installing perforated baffle plates at the inlets to Basin. Hydraulic head is available to replace the existing submerged effluent collection pipes with weir troughs. This will also aid flow splitting.
- The existing finger baffles direct flow downward toward the sludge hopper, resulting in poor solids removal and, possibly, solids re-suspension. Inlet conditions will be improved by replacing the existing finger baffles with new inlet baffles with a bottom “floor” to minimize downward velocities.
- With little headloss, the single finger baffle, coupled with its floor member, spreads and directs flow in the sedimentation tank in a satisfactory manner. Scour velocities along the bottom of the tank are minimized to achieve design objectives.
- The conventional approach for the design of sedimentation tanks is still valuable as a tool for the first stage of tank design to obtain general conceptual information, but such a design should then be

subjected to a rigorous CFD analysis of the its configuration and of the effect of changes in operational parameters.

8. REFERENCES

- [1] Forbes, R.J. & Dijksterhuis, E.J. (1963) A history of Science and Technology 1 – Ancient times to the seventeenth century, Pelican Books, London, 294 p.
- [2] Matko T, Fawcett N, Sharp A and Stephenson T (1996). A numerical model of flow in circular sedimentation tanks. *Trans IChemE.*, 74(B), 197-204.
- [3] Van Der Walt, J.J. (2000d) To baffle or not to baffle – some baffled solutions, a CFD investigation into the use of baffles in process tanks, Proceedings WISA 2000 conference, Sun City, South Africa.
- [4] Ghawi A.H., and Kris J. (2008A). Design And Optimization Of Final Clarifier Performance With CFD Modelling. Conference Paper : International Conference 70 Years of FCE STU, December 4 - 5, Bratislava, Slovakia
- [5] Ghawi A.H., and Kris J. (2008B). CFD Modeling Of Sludge Sedimentation In Secondary Clarifiers. Conference Paper: International Conference 70 Years of FCE STU, December 4 - 5, Bratislava, Slovakia
- [6] Ghawi A.H., and Kris J. (2008C). A Numerical Model Of Flow In Sedimentation Tanks In Slovakia. Article in *Pollack Periodica* 3(2):59-73. DOI: 10.1556/Pollack.3.2008.2.6
- [7] Ghawi A.H., and Kris J. (2009). A CFD Methodology For The Design Of Rectangular Sedimentation Tanks In Potable Water Treatment Plants. Article in *Journal of Water Supply: Research and Technology-Aqua* 58(3) P.P. 212-220. DOI: 10.2166/aqua.2009.027
- [8] Ghawi A.H., Kris J., and Abbas A. Al-Jeebory (2010). Performance Improvement Of Water Treatment Plants In Iraq By CFD Model. *Al-Qadisiyah Journal For Engineering Sciences* Vol. 3 No. 1 P.P. 1-13.
- [9] Ghawi A.H., and Kris J. (2010). Improvement Performance of Al-Wathba Settling Tank by a Computational Fluid Dynamics Model. Article in *Journal of Hydrology and Hydromechanics* 58(3) P.P. 201-210. DOI: 10.2478/v10098-010-0019-8.
- [10] Ghawi A.H., and Kris J. (2011). Improvement Performance of Secondary Clarifiers by a Computational Fluid Dynamics Model. Article In *Slovak Journal Of Civil Engineering* XIX(4):1-11., DOI: 10.2478/V10189-011-0017-9
- [11] Ghawi A.H., and Kris J. (2012). Computational Fluid Dynamics Model of Flow and Settling in Sedimentation Tanks. DOI: 10.5772/27160. Source: InTech, Chapter In book: Applied Computational Fluid Dynamics Edited by Hyoung Woo Oh. P.P. 19-34.
- [12] Hirt C. W., Nichols B. D.: Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics* 39, 1 (1981), 201-225.
- [13] Launder BE and Spalding, D.B. (1972) *Mathematical Models of Turbulence*. Academic Press, New York.
- [14] Zhou, S. and J. A. McCorquodale (1992) Modeling of rectangular settling tanks, *Journal of Hydraulic Engineering, ASCE*, vol. 118, no. 10, pp. 1391-1405.
- [15] Rodi, W. (1993) *Turbulence Models and their application in hydraulics – a state-of-the-art review* 3rd ed, IAHR Monograph Series, A.A. Balkema, Rotterdam.