A CFD Methodology for Modeling Rectangular Sedimentation Basin for Potable Water Treatment by ANSYS Fluent

منهجية ديناميك الموائع الحاسوبي لنمذجة أحواض الترسيب المستطيلة لمعالجة مياه الشرب ببرنامج ANSYS FLUENT

Tasnim Mkresh [1]

[1] Faculty of civil engineering, Aleppo University, Syria, Email: tsnimmkresh@gmail.com

https://doi.org/ 10.5281/zenodo.10393532 VOL2023 ISSUE12

Published 16DEC2023

ABSTRACT:

Gravity sedimentation is widely used in water purification plants to remove suspended solids. The problem is that the detention-time is not long enough to achieve the efficient removal of suspended solids, this is due to the formation of eddies that cause water to exit the tank too quickly (short-circuiting). This research is an analytical work that uses the ANSYS FLUENT16.1 program.

An experiment was conducted through a laboratory study pumping a blue-colored pigment, tracking the flow path of eddies formed, and calculating the efficiency of rectangular sedimentation basins for several variables among the most important results obtained: (1) the optimum ratio of the length of the sedimentation basin to its width L / W = 3.5 ,the sedimentation efficiency was 73.3 % ,(2) the best ratio of the height of the baffle to the depth of the water in the sedimentation basin a / Hw = 0.4, the sedimentation efficiency reached 87.7%, (3)The best ratio of the baffle distance from the entrance to the length of the basin d / L = 0.125, where the sedimentation efficiency corresponded to 88.72%.A consensus was found 96% between results between the laboratory study and the analytical study by Modeling with the help of ANSYS FLUENT CFD methodology.

Keywords: *Computational Fluid Dynamics (CFD), Sedimentation Efficiency, Baffle, Eddy Viscosity, ANSYS FLUENT Program.*

الملخص:

تُعتمد أحواض الترسيب بالجاذبية لإزالة المواد الصلبة العالقة بشكلٍ واسع في محطات تنقية المياه، وتكمن المشكلة في الحياة العملية في عدم تحقيق أحواض الترسيب الواقعية الكفاءة التصميمية المطلوبة، وهذا يعود إلى وجود الدوامات التي تسبب وصول المياه إلى هدّار الخروج بأقل من الزمن اللازم لترسيب الجسيمات.

وتم في هذا البحث إنشاء نموذج لحوض ترسيب مستطيل يحاكي البارمترات الرئيسية التي تتحكم بكفاءة الترسيب لإزالة المواد العالقة في المياه، ودراسته من النواحي الهيدروليكية والبيئية.

واعتمد البحث على المنهج التحليلي والتجريبي، تحليلياً تم باستخدام برنامج ANSYS FLUENT 16.1 بمنهجية ديناميك الموائع الحاسوبي اعتماداً على طريقة العناصر المحدودة، وتجريبياً من خلال دراسة مخبرية على قناة زجاجية مقطعها مستطيل في مخبر الهيدروليك في كلية الهندسة المدنية، حيث تم ضخ صبغة زرقاء اللون وتتبع مسار الجريان وتشكل الدوامات لوصف شكل مسارات التيارات المائية في الحوض، تم حساب كفاءة الترسيب (TSS) المقابلة لعدة متغيرات ومقارنة النتائج المخبرية مع النتائج التي تم الحصول عليها بالبرنامج.

ومن أهم النتائج التي تم التوصل إليها:

(1) النسبة المثلى لطول حوض الترسيب إلى عرضه L/W =3.5 وكانت كفاءة الترسيب المقابلة 73.3%.

(2) أفضل نسبة لارتفاع الحاجز إلى عمق الماء في حوض الترسيب $a/H_W = 0.4$ حيث وصلت كفاءة الترسيب إلى 87.7%.

(3) أفضل نسبة لبعد الحاجز عن المدخل بالنسبة لطول الحوض d/L = 0.125 حيث قابلت كفاءة ترسيب 88.72%.

ووُجد توافق لجميع النتائج بين الدراسة المخبرية والدراسة التحليلية من خلال نمذجة البارمترات بالاستعانة ببرنامج ANSYS FLUENT بمنهجية CFD.

الكلمات المفتاحية: ديناميك الموائع الحاسوبي؛ كفاءة الترسيب؛ الحاجز؛ الدوامات؛ برنامج FLUENT ANSYS

1. INTRODUCTION

Sedimentation, or settling by gravity, is a common method to separation of solid particles from water and wastewater treatment plants as well as desilting basins. All attempts to model the sedimentation process must consider both the characteristics of the sediment and the fluid motion. In this way, the current study not only focuses on the hydrodynamics of the flow using a relatively efficient computer simulation model, but also improves the understanding of both flow and sediment characteristics.

In practice three types of sedimentation tanks are used, which are as follows:

1. Rectangular Tanks 2. Circular Tanks 3. Hopper Bottom Tanks

1. **Rectangular Tanks**: These are rectangular in plan and consist of large number of baffle walls. The function of baffle walls is to reduce the velocity of incoming water to increase the effective length of travel of the particle and prevent the short-circuiting.

2. **Circular Tanks**: These are generally not used in plain sedimentation, but are mostly used in sedimentation with coagulation.

3.**Hopper Bottom Tanks**: These are vertical flow tanks, because water flows upward and downward in these tanks. The water enters in these tanks from the top into deflector box. After flowing downward inside the deflector box the water reverses its direction and starts flowing upward around the deflector box.

In these tanks, the low speed turbid water will flow through the length of the tank, and suspended particles have enough time to settle. Finding new and useful methods to increase hydraulic efficiency is the objective of many theoretical, experimental, and numerical studies.

Sedimentation performance depends on the characteristics of suspended solids, the flow field, and the geometry of the tank.

One of the problems in sedimentation tank is circulation zones, which reduce the effective volume of the tanks that may result in a short- circuit condition between the inlet and outlet of the tank. Accordingly, water flow may leave the tank without any settling process.

It is also shown in Fig 1, The unusual flow regime often happens during the winter season, especially in the cold regions, the short circuiting current caused the suspended solids entering the settling zone in the sedimentation tanks. The presence of these regions may have different effects, the short-circuiting at the surface and the motion of the jet at the bed of the tank that occurs because of the circulation in the sedimentation layer, are affected by the geometry of the tank.[1].

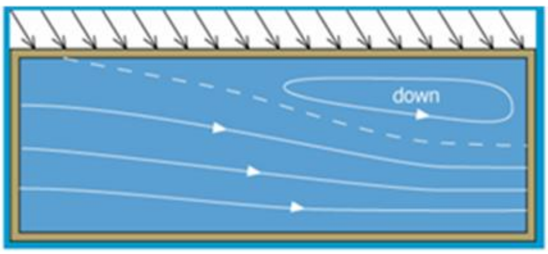

Fig 1: the circulation zone in Rectangular Sedimentation Tanks

Rectangular sedimentation tanks have a rectangular shape and are widely used in smaller wastewater treatment plants and decentralized systems. They are typically made of concrete or steel and have a series of baffles to promote settling and prevent short-circuiting of the flow. The effluent is collected through weirs or effluent launder channels. Rectangular sedimentation tanks are relatively simple in design and construction, making them cost-effective for smaller applications.

These are most commonly used for primary sedimentation, since they:

- Occupy less space than circular tanks.
- They can be economically built side-by-side with common walls.
- Length ranges from 15 to 100m and width from 3 to 24m (length/ width ratio 3:1 to 5:1)
- The maximum forward velocity to avoid the risk of scouring settled sludge is 10 to 15 mm/s (2 to 3 ft/ min), indicating that the ratio of length to width l/w should referrals be about.
- The maximum weir loading rate, to limit the influence of draw-down currents, is preferably about 300 m^3/d, this figure is sometimes increased where the design flow is great than 3 ADWF.
- Inlets should be baffled to dissipate the momentum of the incoming flow and to assist in establishing uniform forward flow.[2]

2. Materials and Methods

A set of experimental measurements is conducted for a settling tank with one baffle positioned in inlet zone of tank. The experimental pilot is a rectangular primary settling tank with following parameters: L = 200 cm, W=30 cm, H= 35 cm

Having a laboratory model is useful to determine the fluid movement, so height of the baffle a= 12 cm and height of weir H_w= 30 cm, flow rate equal to Q = 4.5*10-4 m3/sec

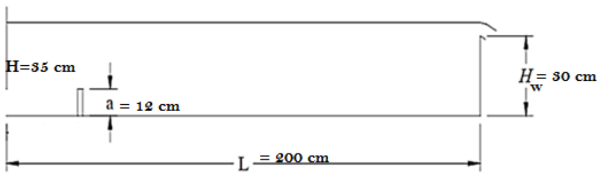

Fig 2: Schematic diagram of the Rectangular Sedimentation Tank

The water used contains calcium carbonate as suspended solids with a concentration = 1500 mg/l and specific weight = 2.8 g/cm^3

The total suspended solids test was conducted for the initial sample and for the final sample from the sedimentation tank to find the efficiency according to the following relationship: $E = (c_0 - c)/c_0 \times 100$

E: sedimentation efficiency, c_0: initial concentration, c: final concentration,

in order to achieve the lowest concentration low suspended solid (LSS) <200 mg/l.

Using filter paper with the following specifications:

Filter speed: Medium Fast

Ash < 0.2%

Weight: 55 g/cm3

Thickness: 0.33 mm

All cases were tasted with the same inlet Reynolds number : $Re = \frac{V.R}{\nu} \leq 2000$

Where, v: velocity, v = 0.005 m/sec, to check the velocity value by controlling the flow of water entering from the pump and to ensure the accuracy of the measurement using the water auger.

R = A/P , R: ratio of the cross sectional

A: area of the flow

P: wetted perimeter of the channel. [3]

$\rightarrow$ $R = \frac{0.3*0.3}{0.3+0.3+0.3} = 0.1$

ν : viscosity , $\nu = 1.007 * 10^{-6}$ m²/sec

$Re = \frac{0.005*0.1}{1.007*10^{-6}} = 496.52 < 2000$

Froude number $Fr = \frac{v^2}{g*R} = \frac{(0.005)^2}{9.81*0.1} = 2.55*10^{-5} \geq 10^{-5}$

($Fr = 2.55*10^{-5}$) for all cases.

Experiments are flow-dependent on Q= A*v, A: area, A= H* W , v: velocity

Q= 0.005* 0.3*0.3= 4.5*10-4 m³/sec = 0.45 l/sec .

Laboratory experiments were performed in hydraulic laboratory at Faculty of civil Engineering at Aleppo University. To know the sedimentation efficiency by conducting the suspended solids test, in addition to a blue dye was pumped to track the path of the flow, a laboratory sedimentation tank is shown in Fig 3

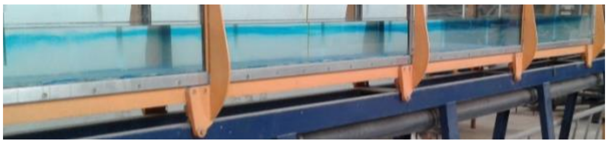

Fig 3: The experimental model setup in hydraulic laboratory at Faculty of civil Engineering at Aleppo University.

3. Boundary Conditions

To simulate a given flow, it is important that the boundary conditions accurately represent what is physically occurring. The inlet condition was specified as a plug flow of water at (0.005 m/sec) and the outlet was indicated as outflow boundary condition and the atmospheric pressure at the top and uniform velocity profile in inlet for all cases. All obstacles and walls are assumed no slip, no slip is defined as zero tangential and normal velocities.

The inlet should be so designed as to provide a uniform distribution of water and solids in the sedimentation tank, it can be easily evaluated by using CFD.[4]

4. Computational Fluid Dynamics (CFD)

Over the last two decades, there has been a large growth in the application of Computational Fluid Dynamics (CFD), has many advantages over traditional modelling approaches as it is a low-cost, high speed technique for evaluating engineering systems that are difficult to simulate in a laboratory. Many researchers have used CFD simulations to describe water flow and solid removal in settling tanks for treatment [5].

In this study, the presence of baffle with different position by computational modeling of a rectangular sedimentation tank by a CFD methodology are employed to assess the effect of adding a vertical baffle at the feed section of a sedimentation tank for the improvement of solids settling in water treatment when the volume of the recirculation region is minimized.

A general CFD strategy is developed based on the specific features and conditions.

The linearity of the particle conservation equations allows separate calculation for each particle size distribution (d= 500 μm).This work compares simulations from tank by CFD and measures experimental total settling efficiency. It is found when L/W = 3.5 sedimentation efficiency =71%,

a/H_w = 0.4 sedimentation efficiency = 85 %, d/L = 0.125 sedimentation efficiency=86 %.

Generally, the flow pattern is characterized by a large recirculation region part of the tank from top to bottom.

There smaller recirculation regions are also found:

Two at the top of the tank near the entry and exit points of the liquid stream and one at the bottom where the sludge gathers before leaving the tank and the efficiency a supstantial impact on the hydrodynamics and efficiency of the sedimentation tank.[6]

5. Results

The performance of sedimentation tanks depend on the flow pattern and mixing regime in the tank. The inlet structure in sedimentation tanks must provide a uniform flow pattern to minimizing the circulation zones.

Consequently, the determination of flow and mixing characteristics is indispensable for the prediction of the tank efficiency. This study shows the calculated streamlines for the different configurations of L/W, a/H_w , and d/L of eddy viscosity.

Usually, the flow pattern is characterized by a large recirculation region of the tank from top to bottom.

5.1 changing ratio of the length of the sedimentation basin to its width L/ W

The figures from Fig 4 to Fig 9 show distributed of eddy viscosity by change the proportion of the length of the basin to its width.

5.1.1 L/W= 2: H_w=30 cm, W=30 cm, L=60 cm

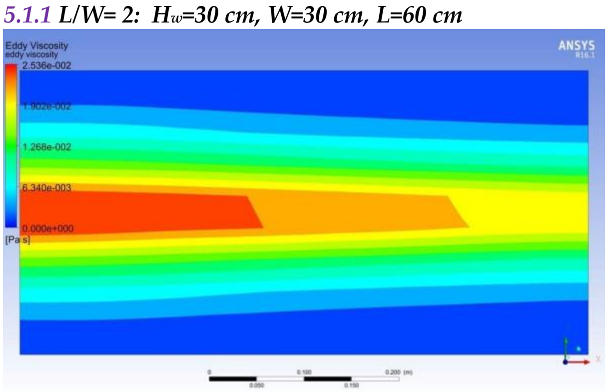

Fig 4: value of eddy viscosity when L/W= 2: H_w=30 cm, W=30 cm, L=60 cm

5.1.2 *L/W = 3: H_w=30 cm, W=30 cm, L= 90 cm*

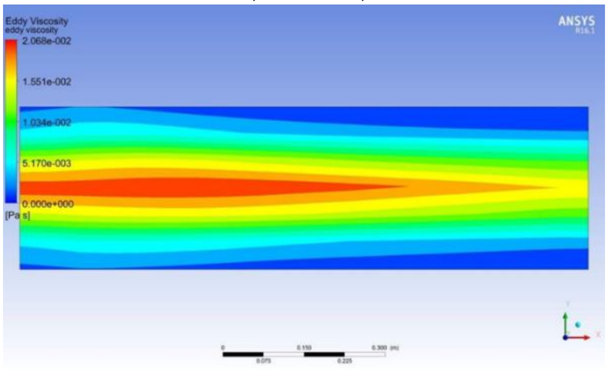

Fig 5: value of eddy viscosity when L/W = 3: H_w=30 cm, W=30 cm, L= 90 cm

5.1.3 *L/W = 3.5: H_w=30 cm, W= 30 cm, L=105 cm*

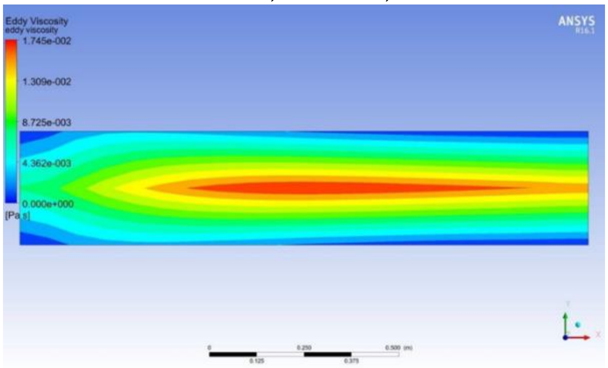

Fig 6: value of eddy viscosity when L/W = 3.5: H_w=30 cm, W= 30 cm, L=105 cm

5.1.4 *L/W = 3.6 : H_w=30 cm, W= 30 cm, L= 108 cm*

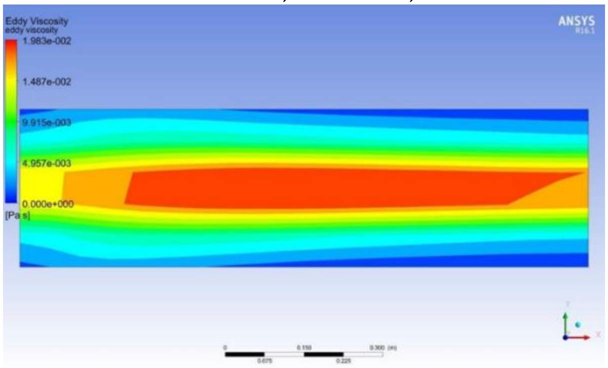

Fig 7 : value of eddy viscosity when L/W = 3.6 : H_w=30 cm, W= 30 cm, L= 108 cm

5.1.5 *L/W = 3.7: H_w=30 cm, W= 30 cm, L= 111 cm*

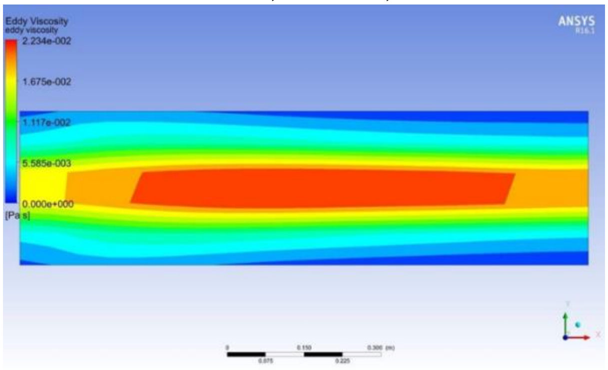

Fig 8: value of eddy viscosity when L/W = 3.7: H_w=30 cm, W= 30 cm, L= 111 cm

5.1.6 *L/W = 4: H_w=30 cm ,W= 30 cm, L= 120 cm*

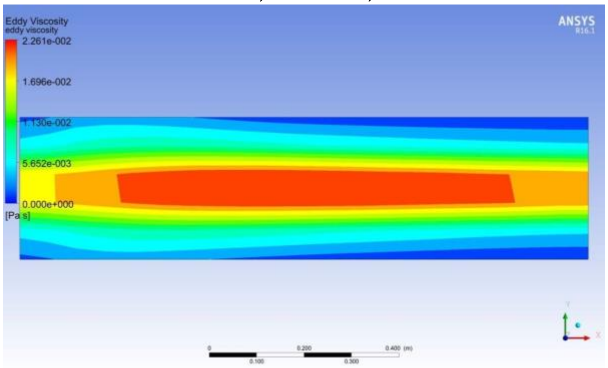

Fig 9: value of eddy viscosity when L/W = 4: H_w=30 cm ,W= 30 cm, L= 120 cm

the settling condition are improved, as can be observed when compared from Fig 4 to Fig 9, which red color reflects of eddy viscosity distributed maximum values and blue color reflects of eddy viscosity distributed minimum values.

Table 1: Evaluate basin length-to-width ratio, eddies viscosity, and sedimentation efficiency

Ratio L/W	2	3	3.5	3.6	3.7	4
Eddy Viscosity*10^{-3} Pa.s	25.36	20.68	17.45	19.83	22.34	22.61
Efficiency %	70.33	70.7	73.3	71.8	71.5	71

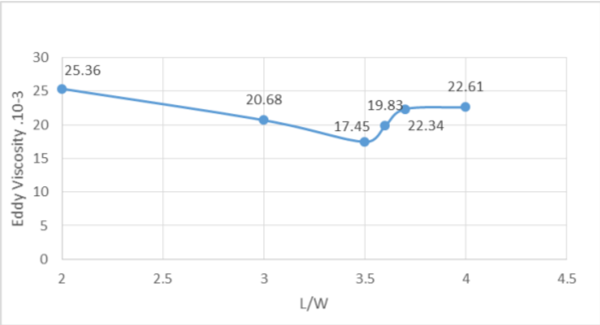

Fig 10: Relationship between basin length-to-width ratio and eddies viscosity

The table 1 and Fig 10 shows that the optimum ratio of the length of the sedimentation basin to its width L / W = 3.5 , the sedimentation efficiency was 73.3 %.

5.2 changing ratio of the height of the baffle to the depth of the water in the sedimentation basin a / H_w:

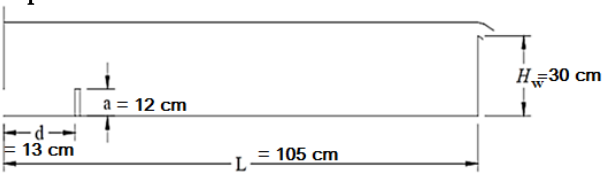

Fig 11: ratio of (a) the height of the baffle to (H_w) the depth of the water in the sedimentation basin

The solids removal rate will be increased with baffle near the inlet area, that redirects the short-circuit current in the clarifier that could lead to high solids contents in the effluent. There are multiple baffle configurations and potential placements is provided at the inlet area, The figures from Fig 12 to Fig 16 show distributed of eddy viscosity by change the proportion of the height of the baffle to the depth of the water in the sedimentation basin.

5.2.1 *a/H_w = 0.2: H_w= 30 cm , a= 6 cm*

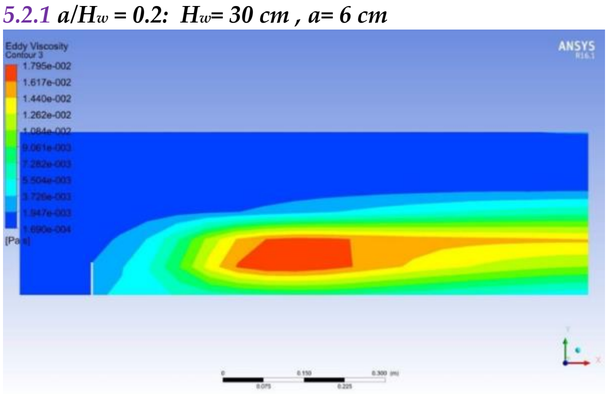

Fig 12: value of eddy viscosity when a/H_w = 0.2: H_w= 30 cm , a= 6 cm

5.2.2 *a/H_w = 0.3: H_w= 30 cm , a= 9 cm*

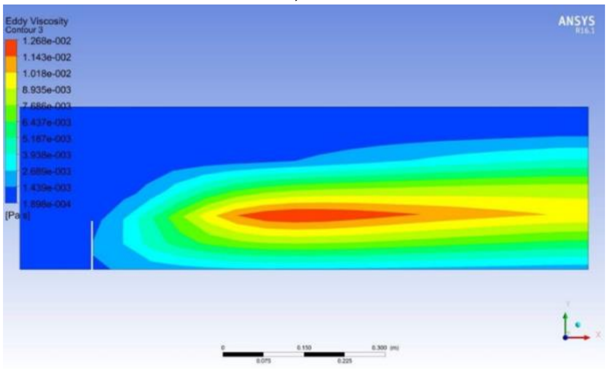

Fig 13: value of eddy viscosity when a/H_w = 0.3: H_w= 30 cm , a= 9 cm

5.2.3 *a/H_w = 0.4: H_w= 30 cm , a= 12 cm*

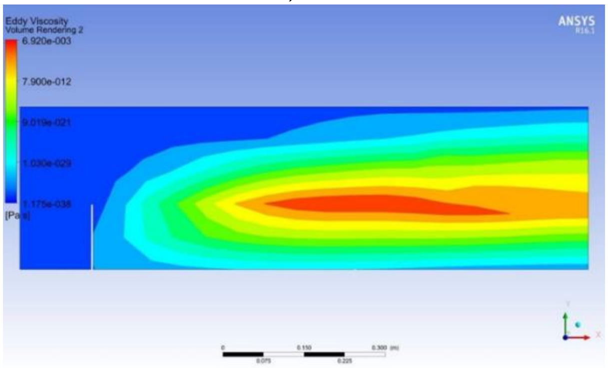

Fig 14: value of eddy viscosity when a/H_w = 0.4: H_w= 30 cm , a= 12 cm

5.2.4 *a/H_w = 0.5: H_w= 30 cm , a= 15 cm*

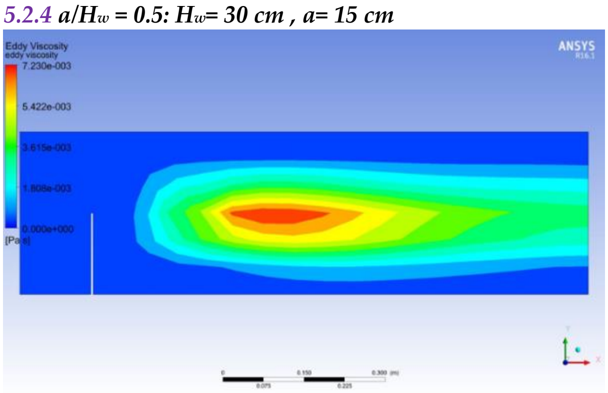

Fig 15: value of eddy viscosity when a/H_w = 0.5: H_w= 30 cm , a= 15 cm

5.2.5 *a/H_w = 0.6: Hw= 30 cm , a= 18 cm*

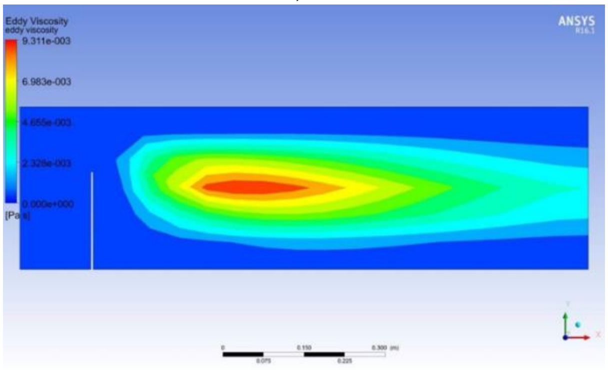

Fig 16: value of eddy viscosity when a/H_w = 0.6: H_w= 30 cm , a= 18 cm

Table 2: Evaluate baffle height to water depth ratio, eddies viscosity, and sedimentation efficiency

Ratio a/H_w	0.2	0.3	0.4	3.6	0.5	0.6
Eddy Viscosity*10^{-3} Pa.s	17.95	12.68	6.92	19.83	7.23	9.311
Efficiency %	72	76	87.7	71.8	85	81

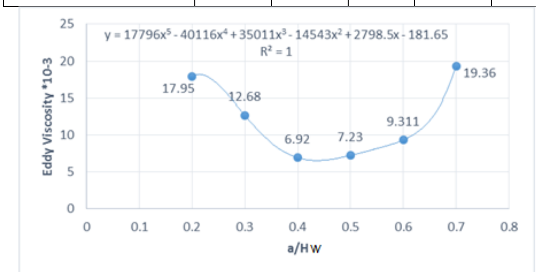

Fig 17: Relationship baffle height to water depth ratio and eddies viscosity

The table 2 and Fig 17 shows that the optimum ratio of the height of the baffle to the depth of the water in the sedimentation basin a / H_w = 0.4, the sedimentation efficiency reached 87.7%.

5.3 changing ratio of the baffle distance from the entrance to the length of the basin d / L

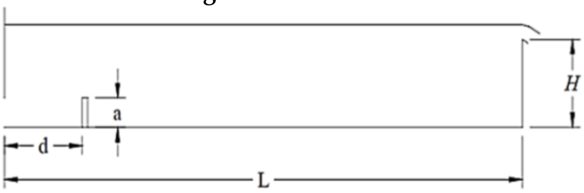

Fig 18: ratio of (d) the baffle distance from the entrance to (L) the length of the basin

A baffle whose distance from the inlet to basin length is located in 90 degrees from the horizontal direction

The figures from Fig 19 to Fig 23 show distributed of eddy viscosity by change the proportion of the height of the baffle distance from the entrance to the length of the basin.

5.3.1 *d/L=0.1:L = 105 cm, d= 10.5 cm*

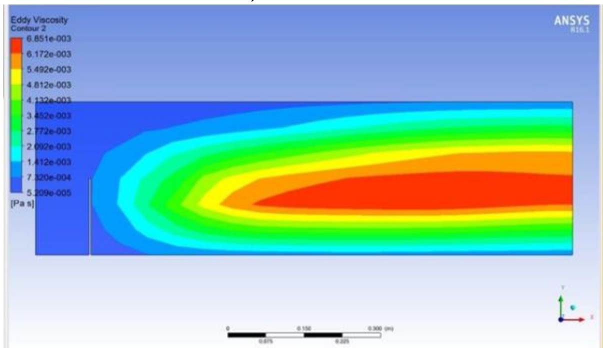

Fig 19: Value of eddy viscosity when d/L=0.1:L = 105 cm, d= 10.5 cm

5.3.2 *d/L=0.12: L = 105 cm, d= 12.6 cm*

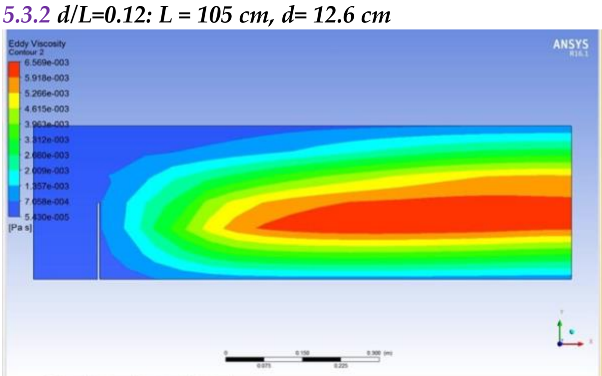

Fig 20: value of eddy viscosity when d/L=0.12: L = 105 cm, d= 12.6 cm

5.3.3 *d/L=0.125: L = 105 cm, d= 13.125 cm*

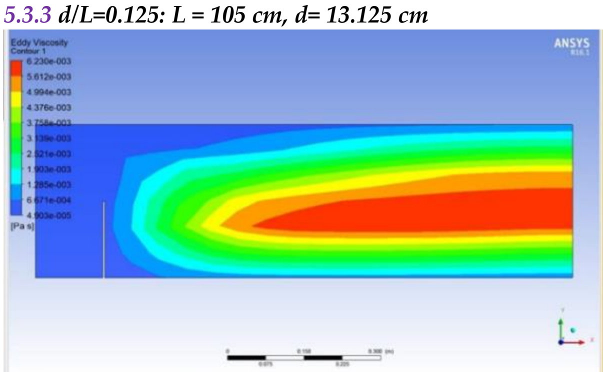

Fig 21: value of eddy viscosity when d/L=0.125: L = 105 cm, d= 13.125 cm

5.3.4 d/L=0.15: L = 105 cm, d= 15.75 cm

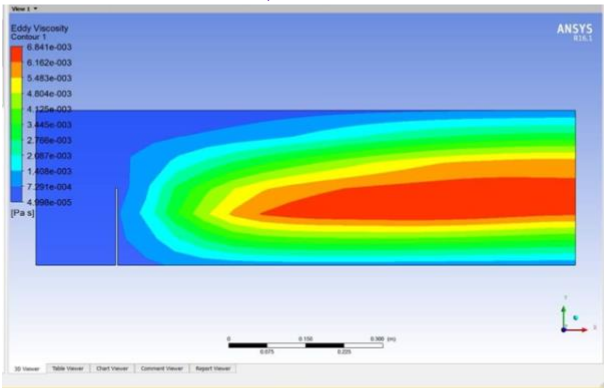

Fig 22: value of eddy viscosity when d/L=0.15: L = 105 cm, d= 15.75 cm

5.3.5 d/L=0.2: L = 105 cm, d= 21 cm

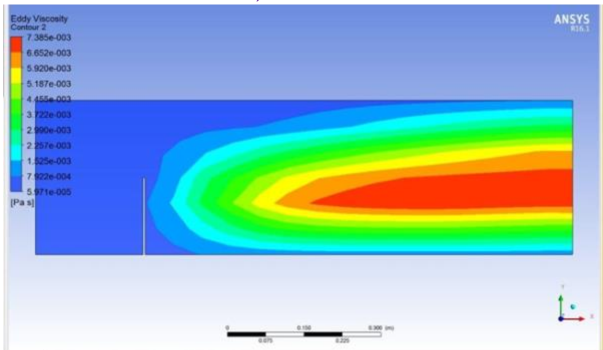

Fig 23: value of eddy viscosity when d/L=0.2: L = 105 cm, d= 21 cm

Table 3: ratio the baffle distance from the entrance to the length of the basin, eddies viscosity, and sedimentation efficiency

Ratio d/L	0.1	0.12	0.125	0.15	0.2
Eddy Viscosity*10^{-3} Pa.s	6.851	6.569	6.23	6.841	7.385
Efficiency %	88.54	88.68	88.72	88.59	88.37

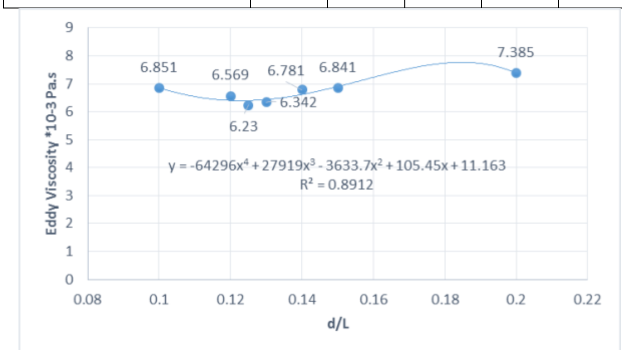

Fig 24: Relationship between the baffle distance from the entrance to the length of the basin ratio and eddies viscosity

6. Discussion

In general, the extended baffle appears to provide better influent mixing and isolation between the tank influent and effluent than that in the original tank design, thus significantly enhancing sedimentation. In addition, it allows a better utilization of the full tank depth than in the standard design that lead to better separation between the influent and effluent along the vertical direction and reducing short-circuiting. Computational Fluid Dynamics simulations are employed to assess the effect of adding a vertical baffle at the feed section of a sedimentation basin for improvement of solids settling in potable water treatment.

A general CFD simulation strategy is developed based on the specific features and conditions met in practice for potable water treatment.[7]

In this work compares simulations by ANSYS FLUENT of the settling basin with different ratios geometry L/W , a/H_w, d/L, the most important results obtained:

the optimum ratio of the length of the sedimentation basin to its width L / W = 3.5 , where the sedimentation efficiency was 73.3 % .Where the ratio was compared with the reference [8] and the ratio within the specified range as shown in the table 4 the rang length to width ratio 3:1 to 4:1

table 4: Permissible rang Design sedimentation basin according to parameters[8]

No.	Parameter	Rang
1	Detention time (hours)	1-2
2	Surface loading rate (l/hr/m²)	500-750
3	Depth of tank (m)	0.6-1.0
4	Length to width ratio	3:1 to 4:1

In comparison with the reference [9] for the design of sedimentation basins, the length = 12m and the width = 3.5 m of the basin, that is, the ratio L / W = 3.42, which is very close and almost equal to the ratio deduced from this research L / W = 3.5

the best ratio of the height of the baffle to the depth of the water in the sedimentation basin a / H_w = 0.4, where the sedimentation efficiency reached 87.7%.

By comparing the best ratio of the baffle height to the depth of the water, it was found that the value agrees with the reference [10], where it is equal to 0.4

(3)The best ratio of the baffle distance from the entrance to the length of the basin d / L = 0.125, where the sedimentation efficiency corresponded to 88.72%.

By comparing the best ratio of the baffle distance from the entrance to the length of the basin with table 5 from reference [1] where d / L = 0.125, agreement was found in the same ratio with table 3 in this research, the smallest value of Eddy Viscosity formed and thus the largest sedimentation efficiency corresponded.

table 5: Computed normalized circulating volume for various position of the baffle [1]

d / L	0.10	0.125	0.150	0.20	0.25	W.B.
C.V. (%)	33.90	32.28	34.40	34.43	35.08	37.05

d : The baffle distance from the inlet of the tank
L : The length of the tank
W.B. : Without Baffle
C.V. : Normalized Circulation Volume

The final dimensions for rectangular sedimentation tank in our experimental study shows in Fig 25 according to the optimum ratios because W constant in our lab, W = 30 cm, L/W= 3.5, L =3.5*30= 105 cm. d/L = 0.125, d = 105*0.125=13.125 cm, a/H_w = 0.4, a = 0.4*30= 12 cm.

detention time t = 1 min for 30 cm, (If L = 105 cm, t = 3.5 min)

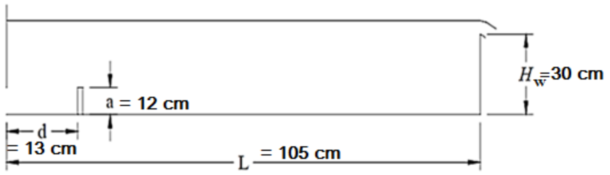

Fig 25: The final dimensions for rectangular sedimentation tank in our experimental study

A consensus was found 96 % between results between the laboratory study and the analytical study by Modeling with the help of ANSYS FLUENT CFD methodology as shown in the table 6 .

table 6: comparing between the Experimental study and the Modeling study

Ratio	Experimental settling efficiency	Modeling settling efficiency
L/W = 3.5	71%	73.3%
a/H_w = 0.4	85%	87.7%
d/L = 0.125	86%	88.72%

7. Conclusions and recommendations

This research is an analytical work that uses the ANSYS FLUENT16.1 program. Using Computational Fluid Dynamics /CFD/ methodology. It depends on the method of finite elements. An experimental was conducted through a laboratory study pumping a blue-colored pigment, tracking the flow path of eddies formed, and calculating the efficiency of Rectangular sedimentation basin for several variables to get the best of Geometry Design of a Rectangular Sedimentation Tanks for Potable Water Treatment.

Results show achieved the best tank performance. Also, baffle reducing the size of the circulation zone in sedimentation area.

CFD will not completely replace experimental testing and the partly empirical nature of the design process.

Traditional techniques will continue to be used for routine design, but CFD is invaluable for backing up this work and for investigating novel tank designs.

We recommend studying the effect of changing the water temperature in the sedimentation basin in relation to the inlet water temperature

8. REFERENCES:

1. Radaei E, Nikbin S, Shahrokhi M. Numerical investigation of angled baffle on the flow pattern in a rectangular primary sedimentation tank. Research in civil and environmental engineering (2013), p. 79-91.
2. Jamal H. Type of primary sedimentation tanks. Environmental Engineering(2017).
3. Thomas H, Brown Jr. Highway Engineering, part 9-Hydraulics. ScienceDirect (2016),
4. p.641-695 .
5. Robescu L, Manea E. Using CFD techniques in teaching rectangular settling tank hydrodynamics. De GRUYTER OPEN(2015), doi: 10.1515/cplbu-2015-0005.
6. Ghawi A, Kris J. A CFD methodology for the design of rectangular sedimentation tanks in potable water treatment plants. ResearchGate (2009), p. 1-20. doi: 10.2166/aqua.2009.027.
7. Shhrokhi M, Rostami F, et al. Numerical simulation of influence of inlet configuration on flow pattern in primary rectangular sedimentation tanks. World Applied Sciences Journal (2011), vol. 15(7), p. 1024-1031.
8. Goula , Kostoglou M, et al. A CFD methodology for the design of rectangular sedimentation tanks in potable water treatment Case study: The influence of a feed flow control baffle. Chemical Engineering Journal(2007), doi: 10.1016/j.cej.2007.09.022.
9. Ghawi A. Development of the Greywater Domestic Treatment Unit for Irrigation of the Garden in Rural Area. Journal of Ecological Engineering (2019), 20 (3), p. 46-56.
10. Water Sensitive Urban Design Guidelines Chapter 3: Sediment Basins. WSUD Engineering Procedures for Stormwater Management in Tasmania (2012).
11. Heydari M, Bajestan M, et al. The Effect Angle of Baffle on the Performance of Settling Basin. World Applied Sciences Journal (2013), Vol. 21 (6), p. 829-837.
12. DOI: 10.5829/idosi.wasj.2013.21.6.30.