# ONE WAY FSI STUDY OF THE STRESS AND STRAIN DISTRIBUTION ON THE WATER DAM SPILLWAY STILLING BASIN BAFFLES


By:

**MUHAMAD ZULHILMI BIN ROSLAILI**

(Matrix no.: 125026)

Supervisor:

**Dr. Azmi Ismail**


June 2018

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfillment of the requirement to graduate with honors degree in
**BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)**


School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

## DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed..........................................................................................................

(MUHAMAD ZULHILMI BIN ROSLAILI)

Date...............................................................................................................

**Statement 1**

This thesis is the result of my own investigation, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed..........................................................................................................

(MUHAMAD ZULHILMI BIN ROSLAILI)

Date...............................................................................................................

**Statement 2**

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed...........................................................................................................

(MUHAMAD ZULHILMI BIN ROSLAILI)

Date...............................................................................................................

# ACKNOWLEDGEMENT

First and foremost, after going through all the countless effort, Alhamdulillah praised to Allah (S.W.T) because always give me strength and hope to complete my final year project. I would like to give my special gratitude to my supervisor, Dr. Azmi Ismail for his guidance and advices through the whole project. He always give me encouragement and support along this journey.

Furthermore, I also would like to express my special thanks to a master student, Muhammad Aqil Bin Azman who has offer a lot of help to me in giving suggestions in simulation works. His knowledge and experience in simulation is very helpful and I very appreciate his kindness.

Last but not least, I would like to express my utmost gratitude to all the lectures, technical staffs and friends in the School of Mechanical Engineering, Universiti Sains Malaysia that involve directly or indirectly in this project for sharing their knowledge with me. Their support always give me strength and motivation to complete this meaningful project.

Title of thesis: **One Way FSI Study Of The Stress And Strain Distribution On The Water Dam Spillway Stilling Basin Baffles**

Date of submission (Academic year): **8 June 2018**

Candidate (Matrix no.): **Muhamad Zulhilmi Bin Roslaili (125026)**

Name of supervisor: **Dr. Azmi Ismail**

**TABLE OF CONTENT**

# LIST OF TABLES

## LIST OF FIGURES

## LIST OF ABBREVIATION

| | |
|---|---|
| FSI | Fluid-Structure Interaction |
| CFD | Computational fluid dynamics |
| ADV | Acoustic Doppler Velocimetry |
| GMRES | Generalized Minimal Residual |
| C.V | Circulation volume |

# KAJIAN SATU ARAH FSI TERHADAP PENGEDARAN TEKANAN DAN TEGANGAN BONGKAH LEMBANGAN YANG MENENANGKAN TUMPAHAN EMPANGAN AIR

## ABSTRAK

Bongkah ialah sejenis alat yang digunakan untuk mengubah lalaun corak air. Di empangan, bongkah sangat penting kerana ia digunakan untuk mengurangkan halaju air yang mengalir dari tempat tumpahan. Dalam kajian ini, kesan geometri bongkah telah dikaji menggunakan simulasi pengiraan. Dengan menggunakan empat model bongkah corak aliran air dikaji. Corak aliran air boleh ditentukan daripada garis aliran air dan kontur halaju aliran. Struktur statik tekaan dan tegangan paling tinggi boleh didapati daripada tekanan yang diimport daripada simulasi. Daripada simulasi yang dijalankan halaju keluar air juga ditentukan untuk menyemak sama ada keputusan daripada simulasi sama seperti keputusan yang dijangkakan. Keputusan seharusnya mempunyai halaju keluar air yang rendah berbanding halaju masuk air. Daripada keputusan, reka bentuk, bongkah yang terbaik ditentukan iaitu yang mempunyai halaju keluar air yang rendah dan statik struktur tekanan dan tegangan paling tinggi yang rendah.

**ONE WAY FSI STUDY OF THE STRESS AND STRAIN DISTRIBUTION ON THE WATER DAM SPILLWAY STILLING BASIN BAFFLES**

# ABSTRACT

Baffle is a device used for altering the flow of water pattern. In a dam, baffle is very important as it is used to reduce the velocity of water in the spillway. In this study, the effect of geometry of baffles is investigated using computational simulation. By using four models of baffles the pattern of water stream flow is investigated. The flow pattern can be determined from water streamline and velocity contour of the stream flow. The static structural of maximum stress and strain of the baffles also is investigated. The static structural of maximum stress and strain can be obtained from the imported pressure of the simulation. From the simulation, the outlet velocity of water is also determined to check whether the results from simulation is same as the expected results. The results should have lower outlet velocity of water compared to inlet velocity of water. From the results, the best design of baffles is determined which has low outlet velocity of water and low static structural of maximum stress and strain.

# CHAPTER 1

# INTRODUCTION

## 1.1 PROJECT BACKGROUND

Baffle is a solid concrete structure that is used to reduce the velocity of water stream flow in stilling basin at dam area. Usually, when the level of water in the dam is very high, the water will be channelled to the spillway to reduce the amount of water in the dam. This is because high volume of water will exert high pressure to the wall of the dam thus can damage the dam structure. When the water is channelled to the spillway the water velocity is very high that require installation of baffles in the stilling basin to reduce its velocity.

The collision of water stream flow with baffles will surely give some impact to both of water stream flow and also the baffles. As a result the collision will produce hydraulic jump at the baffles and give stresses to the surface of the baffles due to the high pressure from the water stream flow. Depending on the opening gate of the spillway, different velocity of water stream flow will give different stresses to the surface of the baffles.

The purpose of this study is to mesh and simulate 1 way FSI that investigate flow pattern of water stream flow and static structural analysis on the surface of the baffles. This study will use four baffles design (including original baffles) and compare the result with original baffles retrieved from ANGKASA Consultancy Services Sdn. Bhd for Mengkubau Dam in Brunei Darussalam. Ansys Workbench for Fluent and Static Structural is required as for simulation purpose in this study.

Due to development and rapid improvement in technology of computing, many researchers start to use numerical method to study hydraulic structure in fluid dynamic field. Computational fluid dynamics (CFD) is used extensively by engineers to model and analyse complex issues related to hydraulic design, planning studies for future generating stations, civil maintenance and supply efficiency [1]. This computational investigations are performed in order to compare the water stream flow between the original baffles and modified baffles and also the stresses at the surface of the baffles. This method is becoming increasing popular in this field and almost ready to replace the former industry standard of scaled physical modelling. However, the results of numerical method are not 100% reliable and it needs to validate with results from scaled physical modelling, especially in predicting flow resistance and water level. The most popular advantage of numerical method is much less expensive than physical model due to several reasons including laboratory space, no materials or construction, less man power required. In addition, this method is easy to modify to accommodate design change.

Hydraulic study on the spillways of Mengkubau Dam is important to estimate water velocity especially in stilling basin. This study will be conducted by Computational Fluid Dynamic (CFD) that has been widely used in hydraulic model simulation. In this spillway model simulation, the four models of baffles are studied in the channel and stilling basin. This Computational Fluid Dynamic (CFD) modelling is used to predict water velocity in spillway especially in stilling basin area. Detailed drawing of the Mengkubau Dam's spillway is provided by Angkasa Consultancy Services Sdn. Bhd.

## 1.2 PROBLEM STATEMENT

Spillway is a water flow channel to remove the excessive water from the dam. When the opening gate of the spillway is opened, the water will flow down the spillway with certain amount of velocity. Usually, when the level of water in the dam is high, the gate of the spillway will be opened to prevent any physical damage to the dam. This is because, when the level of water in the dam is high, the pressure exerted to the wall of the dam is also high. Thus, by reducing certain level of water in the dam can reduce a significant amount of pressure to the wall of the dam.

In a spillway, baffles is used to slow down the velocity of water after it comes out from the dam. Without baffles the velocity of water can be so high that it can swept out the geographical features within that area. As for baffles, the high velocity of water also can affect the structure of the baffles. When the water flow through the baffles it will cause hydraulic jump on the baffles and pressure to the surface of the baffles. Although baffles is made up from concrete material which is very strong and hard material but, it is better if the baffles experience as less as possible stresses for better lifespan. Thus, this study is conducted to determine the most appropriate design for baffles with the assist of ANSYS Workbench.

## 1.3 OBJECTIVE

The objectives of this project are stated below:

1. To determine the flow pattern of water stream flow for original baffles and modified baffles.
2. To determine the maximum stress and strain at the surface of the baffles.
3. To determine the best design of the baffles.

## 1.4 SCOPE OF WORK

The major scope of work to be carried out for the purposed project will include the design of the baffles in SolidWorks using the real dimensions of the baffles provided by ANGKASA Consultancy Services Sdn. Bhd for Mengkubau Dam in Brunei Darussalam. The baffles is located in the stilling basin area as the water flow down into the spillway.

Ansys Workbech 16 for Fluent and Static Structural also will be used in this study to mesh and simulate the hydraulic flow of water through the baffles. The velocity contour and water streamlines will be analysed to determine the pattern of water stream flow. The force distribution on the surface of the baffles also will be determined by importing the pressure from the data file of Mengkubau Dam simulation.

This study will use various design of baffles and compare the result with original baffles. The modified baffles is expected to have approximately same pattern of water stream flow with the original baffles but a significant reduce of stresses on the surface of the baffles.

## 1.5 OUTLINE OF THESIS

This thesis is divided into several chapters. There are total of 5 chapters discussed in this thesis which are:

- **Chapter 1**

Including project background, definition and application of baffles in dam, problem statement, project objectives, scope of project and thesis organization

- **Chapter 2**

This chapter include all the review of the other literatures that had done the research which is related with this project.

- **Chapter 3**

This chapter will explain about the methodology of the project. The content include the flow chart of project, the scanning and editing process, material selection and machining process.

- **Chapter 4**

This chapter will include the result and discussion of the project. The data that acquired from this project will be analysed and will be compared.

- **Chapter 5**

Lastly, chapter 5 discussed the conclusion of the project. The objectives made in the beginning are discussed and the whole thesis is concluded. Some future work and improvement are suggested.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 OVERVIEW

Until recently, researchers have shown their interest on controlling the water stream flow using the baffles. Numerous studies have been attempted to explain about the effect of water stream flow on baffles. Other than spillway of a dam, the application of baffles also are widely used in the river or any other practises that have water stream flow.

## 2.2 BAFFLE LOCATION ON THE FLOW PATTERN OF SEDIMENTATION TANKS

In sedimentation tanks Shahrokhi et al. 2013 [1] stated that the two main types of sedimentation (clarifier) tanks are primary and secondary (or final) settling tanks. Influent concentration in a primary sedimentation tank is low, and hence, the concentration field has minor influence on the flow field. Therefore, in the primary sedimentation tank, the buoyancy effects can be neglected. But in the secondary (or final) sedimentation tank, however, the concentration of particles in influent is high [2]. In the present study, focus is made on the primary sedimentation tank. A uniform flow field is necessary to the efficient performance of a primary sedimentation tank. This enables particles to settle at constant velocity and in a short period of time. Circulation regions in the tank may have various effects. These circulation zones 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. To further improve clarifier conditions, internal baffles can be considered to enhance settling and provide a more clarified effluent. Internal baffles are used inside the settling tank, which seeks to enhance sedimentation in the region between the inlet and outlet zones [3].

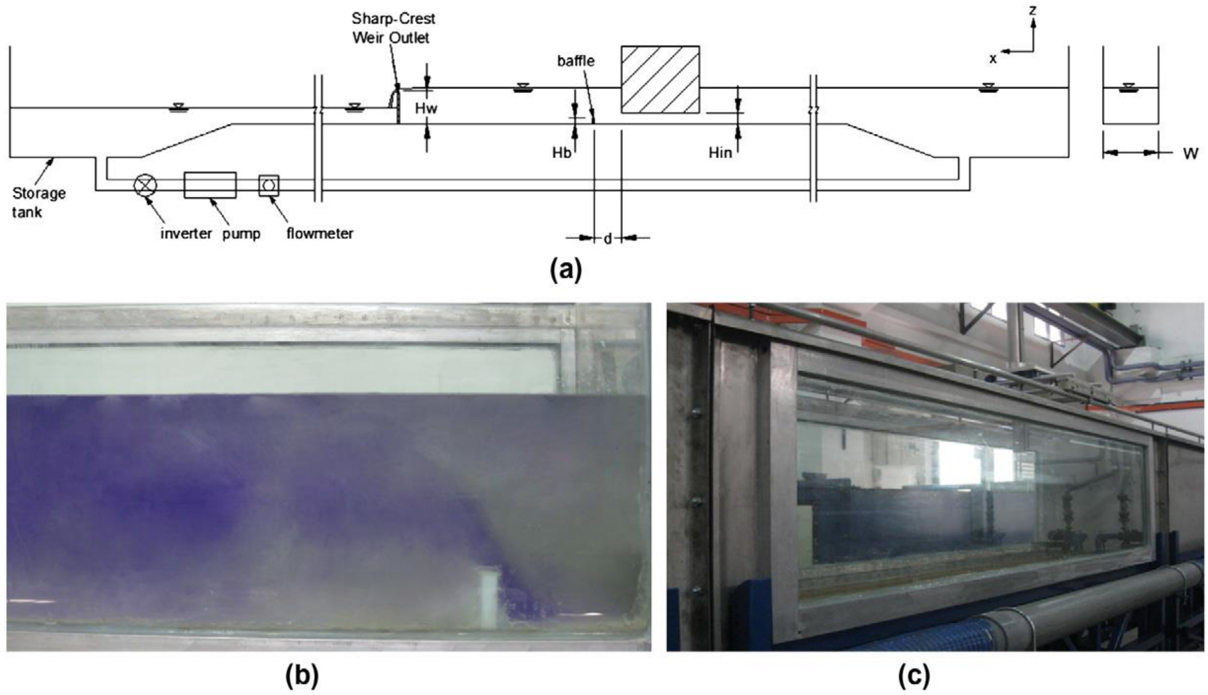



Figure 2.1: (a) Schematic diagram of the tank; (b) A photo of baffle in the tank; and (c) A photo of sedimentation tank. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, and Syafalni, "Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks," Appl. Math. Model., vol. 37, no. 6, pp. 4486–4496, 2013. Reprinted with permission

[1, Figure 2.1] shows the schematic diagram of the tank and the experimental setup of the sedimentation tank. A thin baffle is positioned in a laboratory settling tank and the effects of its position on the velocity fields were measured using Acoustic Doppler Velocimetry (ADV). Then, numerical experiments were carried out for baffle installation distances from the inlet of the tank. The results of the numerical simulation show that primary sedimentation tank performance can be improved by altering the geometry of the tank and the effects of the baffle position scenarios on the efficiency of the primary sedimentation tank are investigated via assessment of the circulation zone volume variations, the maximum velocity values and the magnitude of the kinetic energy in the flow field of each case. Also using Generalized Minimal Residual (GMRES) method can increase the convergence of the model and create more accurate data in comparison with other pressure solver.

The results from this journal shows the proper position for the baffle is obtained when the volume of the circulation zone is minimized. The motion of the jet at the bed surface of the tank, non-uniformity of the velocity field and the short-circuiting at the surface because of the circulation zone in the sedimentation layer and consequently, water flow may exit the tank without any settling process. The lower size of circulation zone leads to create uniform flow field in the settling tank. Circulations zones may also induce high turbulence intensity in some certain regions. This condition not only reduces the possibility of particle deposition, but may also cause resuspension problems. Therefore, minimization of the circulation zone is essential to find the suitable location of the baffle and consequently improve sedimentation process.

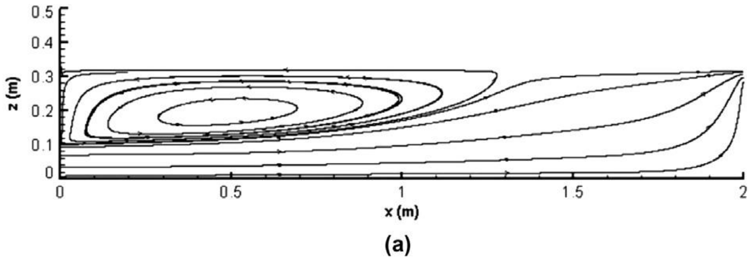
The computational circulation volumes normalized by the total water volume of the tank are shown in [1, Table 2.1]. The table indicates the absolute predictability of some cases to exhibit weak performance because of the size of the dead zone. [1, Table 2.1] shows that the baffle position at S/L = 0.125 exhibits the best performance. In addition, it is noted that if baffle is located in worse position, the efficiency of this tank maybe less than a tank without any baffle. Consequently, it is necessary to investigate about the best position and configuration of the baffle in settling tank.

| **S/L** | 0.10 | 0.125 | 0.20 | 0.30 | 0.40 | W.B |
|---|---|---|---|---|---|---|
| **C.V (%)** | 33.90 | 32.28 | 34.43 | 35.50 | 37.78 | 37.05 |

Table 2.1: Computed normalized circulating volume for various position of the baffle. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, and Syafalni, “Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks,” Appl. Math. Model., vol. 37, no. 6, pp. 4486–4496, 2013. Reprinted with permission

Where,

S: the baffle distance from the inlet of the tank; L: the length of the tank; W.B.: without baffle; and C.V: normalized circulation volume.



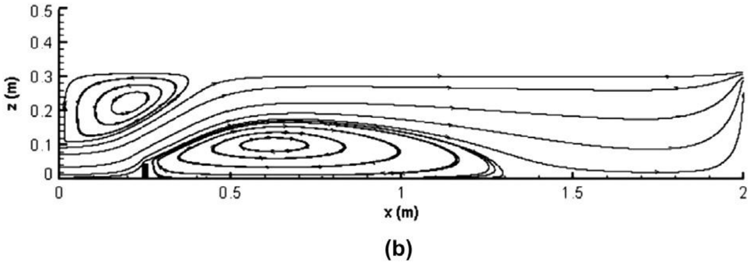




Figure 2.2: Computed streamlines for (a) a tank without baffle and (b) a tank with baffle at S/L = 0.125. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, and Syafalni, “Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks,” Appl. Math. Model., vol. 37, no. 6, pp. 4486–4496, 2013. Reprinted with permission

## 2.3 BAFFLES CONFIGURATION OF A POTABLE WATER SERVICE RESERVOIR

Zhang et al. [4] mentioned that water service reservoirs are essential components of the drinking water supply network. Traditionally, they act as storage tanks for balancing the water supply and demand, providing an emergency water storage, and equalizing pressure on the network. When a service reservoir is located at the end of a water treatment plant, it can be used as a microorganism inactivation contact chamber, or commonly known as a disinfection reservoir. In such a reservoir, sufficient contact time between water and the microorganism inactivation chemical is essential. A plug-flow regime is found to be ideal for such a contact chamber [5], which ensures that all the water is exposed to the chemical for the maximum amount of time. By using baffles in the service reservoirs to act as contact chambers is reported to encourage plug flow and allows the chlorine contact time requirement to be met [6]. Therefore, properly configured baffles are beneficial for this kind of application.

[4, Figure 2.3] shows the original configuration of a potable water service reservoir in Singapore with the layout of inlet, outlet, and columns. It has corrugated walls and the dimensions along x, y, and z directions are 110.6, 37.3 and 8.2 m, respectively. The inlet is a circular nozzle with 0.6096 m in diameter, which has a horizontal orientation and is located 0.6858 m above the floor. The outlet is located on the floor with a rectangular cross section of 0:6096 × 0:762 m which has a vertical orientation. Both the inlet and outlet are at the middle plane of the tank with y = 0 m.

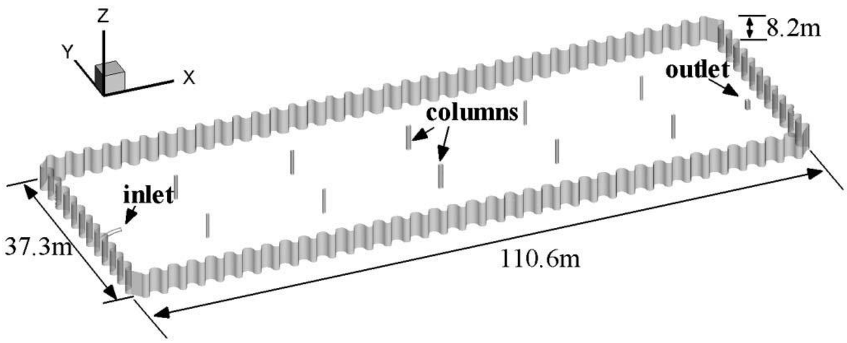



Figure 2.3: Original configuration of a service reservoir with the layout of inlet, outlet, and columns. Reprinted from J.-M. Zhang, B. C. Khoo, H. P. Lee, C. P. Teo, N. Haja, and K. Q. Peng, “Effects of Baffle Configurations on the Performance of a Potable Water Service Reservoir,” J. Environ. Eng., vol. 138, no. 5, pp. 578–587, 2012. Reprinted with permission

[4, Figure 2.4] shows five individual baffle configurations proposed by Zhang et al. [4] for the service reservoir. Because of the symmetrical layout of the tank, only one half of the service reservoir model with baffles and columns are shown. The intent of the baffle 1 design is to separate the incoming flow into two streams for potentially enhancing mixing after the baffle. Baffles 2, 3, and 4 are located at the recirculation regions of the original service reservoir with an intention of breaking up the recirculation vortices, and thus reducing the residence time. [4, Figure 2.4(e)] shows one combined design of baffle configurations, which is named baffle 5. Because the cross-sectional area of every column (0:4064 × 0:4572 m) is much smaller than the cross-sectional area of the tank (110:6 × 37:3 m), it is deemed reasonable to ignore the effect of these columns in the simulations.

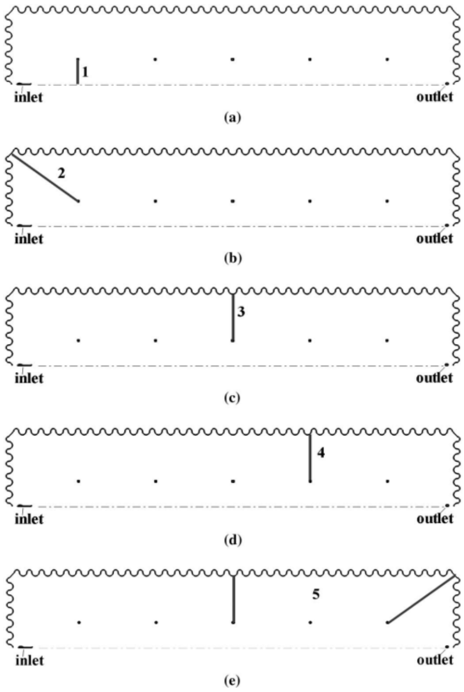



Figure 2.4: Configuration of: (a) baffle 1; (b) baffle 2; (c) baffle 3; (d) baffle (e) baffle 5 for one half of the service reservoir model (top view). Reprinted from J.-M. Zhang, B. C. Khoo, H. P. Lee, C. P. Teo, N. Haja, and K. Q. Peng, "Effects of Baffle Configurations on the Performance of a Potable Water Service Reservoir," J. Environ. Eng., vol. 138, no. 5, pp. 578–587, 2012. Reprinted with permission

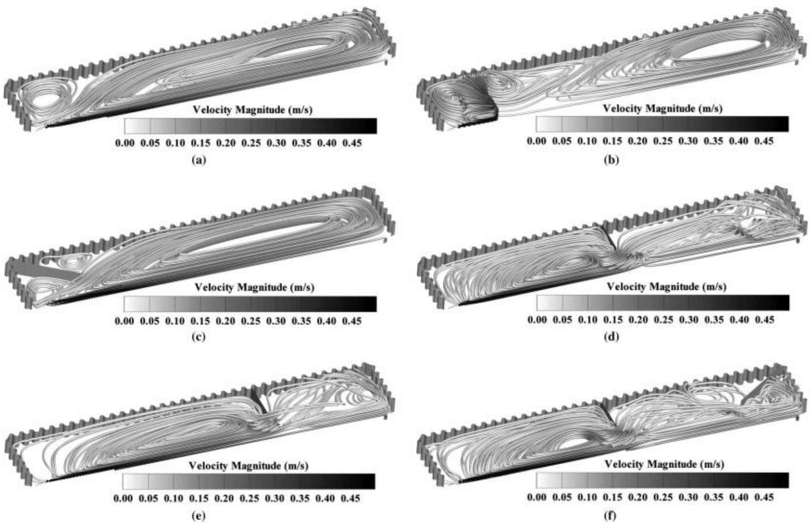



Figure 2.57: Streamlines for the half service reservoir models: (a) without baffles; (b) with baffle 1; (c) with baffle 2; (d) with baffle 3; (e) with baffle 4; (f) with baffle 5 at t = 10 min. Reprinted from J.-M. Zhang, B. C. Khoo, H. P. Lee, C. P. Teo, N. Haja, and K. Q. Peng, "Effects of Baffle Configurations on the Performance of a Potable Water Service Reservoir," J. Environ. Eng., vol. 138, no. 5, pp. 578–587, 2012. Reprinted with permission

[4, Figure 2.5] shows the streamlines for the half service reservoir models without and with baffle designs at the initial refilling stage (t = 10 min). For the model without any baffle, some degree of short-circulated flow is observed in Figure 2.5(a), because the inlet is located directly opposite to the outlet. As is known, the main driving force of the fluid mixing is provided by the inflow momentum. After leaving the inlet orifice, a jet is formed. It spreads laterally along its way downstream, which entrains the surrounding flow into the jet. Consequently, the velocity magnitude of the fluid is reduced along the flow path. Although a small portion of the flow directly goes through the outlet, the remaining fluids are deflected upon hitting the walls and headed back towards the front compartment of the reservoir. Therefore, a recirculation zone is formed, which occupies a large area of the tank. At the corner of the front wall, a small secondary recirculation region is formed, as the jet momentum is insufficient to move into that region. All these result in a non-homogenous velocity distribution in the lateral direction as reported by Zhang et al. [7]. In contrast to the model without corrugated walls [7], the model with corrugated walls has a marginal difference in the main flow pattern, except a small quantity of swilling flow adjoining the corrugated walls.

When baffle 1 is added in the service reservoir, the jet flow reverses its flow direction upon reaching the baffle wall, as shown in [4, Figure 2.5(b)]. Consequently, a vertical vortex is formed above the height of the inlet. Near the corner of the front wall, a secondary horizontal vortex is induced because of the steep changes of fluid momentum and pressure therein. After flowing past the baffle, the fluid momentum is reduced because of the impact and viscous forces induced by the baffle wall. Consequently, the fluid velocity magnitude is reduced. The big recirculation region in the case without any baffle is now having two small flow recirculation vortices.

In contrast to the case without any baffle, the small recirculation region near the inlet is broken into two as shown in [4, Figure 2.5(c)], when baffle 2 is added in the service reservoir. There is no observable change in the jet spreading and big recirculation region. However, the flow pattern changes significantly once a baffle is built in the big recirculation region of the original service reservoir. As shown in [4, Figure 2.5(d)], there is one big flow recirculation region ahead of the baffle wall when baffle 3 is included. After flowing past the baffle, there are small vortices, and the fluid velocity magnitude is reduced because of the impact and viscous forces induced by the baffle wall. Moving the baffle further downstream does not affect the flow pattern much, as shown in [4, Figure 2.5(e)] for the service reservoir model with baffle 4. For a combined baffle design (baffle 5), the flow pattern ahead of the first baffle is the same as that with baffle 3, as shown in [4, Figure 2.5(f)].

## 2.4 THE EFFECT OF NUMBER OF BAFFLES IN SEDIMENTATION TANKS

Based on Shahrokhi et al. [3], The effects of the number of baffles arrangement on the hydraulic performance of primary settling tanks are studied by using two different ways: the parameters of flow pattern and the Flow Through Curves (FTCs) method. The results of both the experimental and computational investigations indicate that increasing the number of baffles in suitable positions provides minimum volume of the recirculation region, dissipates the kinetic energy, creates a uniform flow field in the tank and finally the hydraulic efficiency of the sedimentation tank are improved.

[3, Figure 2.6] illustrates the experimental setup and measurement system. This figure shows a rectangular primary settling tank with the following parameters: L = 200 cm, W=50 cm,H= 50 cm, height of inlet opening Hin = 10 cm, height of the baffle Hb = 5.5 cm and height of weir Hw = 30 cm. One pump replenished the tank and an inverter regulated the discharge in the output pipe and an electromagnetic flow meter measures the volumetric flow of conductive water. This flow meter consisted of a sensor and electromagnetic flow rate transducer.

The x, y, and z velocity components were measured by ADV. The ADV uses the Doppler effect to measure flow velocity by transmitting short pairs of sound pulses, listening to their echoes, and then measuring the change in pitch or frequency of the returned sound. Sound does not reflect from the water itself, but rather from particles suspended in the water. These particles are usually zooplanktons or suspended sediments. These small particles move with the same average speed as the water [8].

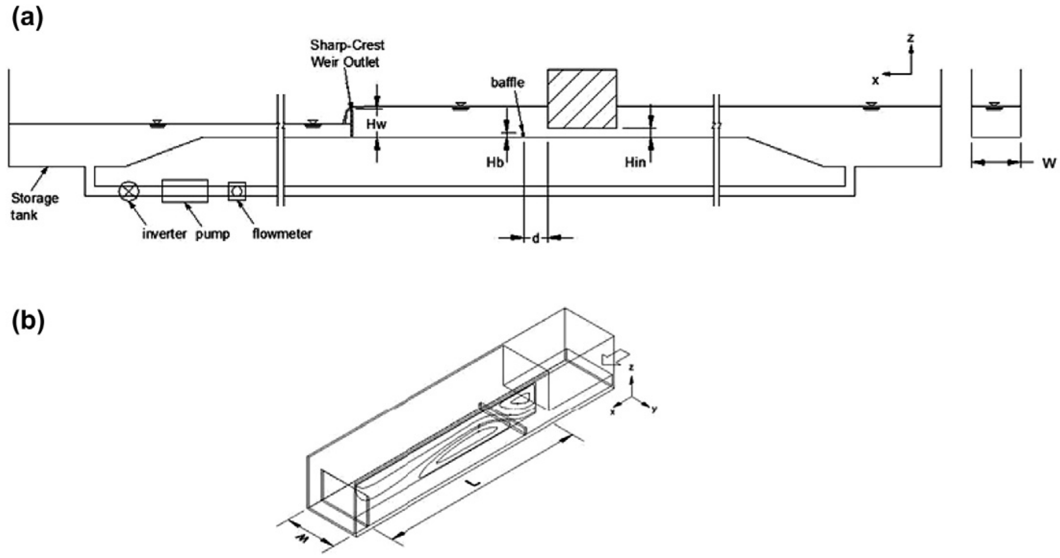



Figure 2.6: Schematic diagram of (a) experimental system (b) settling tank. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, S. R. Sabbagh Yazdi, and Syafalni, “The effect of number of baffles on the improvement efficiency of primary sedimentation tanks,” Appl. Math. Model., vol. 36, no. 8, pp. 3725–3735, 2012. Reprinted with permission

The high performance of the settling tank is obtained when the volume of the circulation zone is minimized or when the dead zone is divided into smaller parts. For that reason, the best sitting of the baffles may lead to a more uniform distribution of velocity in the tank and minimizes the dead zones. The location of baffles affects the size of circulation zones and unsuitable placement of baffles may give the worse results.

Different cases of baffles positions were modelled in this methodology. Computed values of circulation volume, which have been normalized by the total volume of water in the tank, are shown in [3, Table 2.2]. Using the baffles in settling tanks can decrease the volume of the circulation zone. In addition, if the number of baffles increases, the volume of dead zone would decrease in comparison with the case of that without baffle. However, the position of baffles is more important. It is shown that position of two baffles at d/L = 0.125 and 0.388 and three baffles at d/L = 0.125, 0.3, and 0.388 gave the best performance because they have the lowest amounts of circulation volume between the related cases [3, Table 2.3].

Another important parameter in the settling tank is the kinetic energy, and it has a great importance in the sedimentation of particles. At the same time, the baffles must be used in the sedimentation tanks to reduce kinetic energy and reach the uniform fluid condition. The variations of the velocity magnitude and turbulence intensity will reduce if the flow pattern approach to uniform flow field in the sedimentation tanks. In this situation, suspended particles have more chances to remove with a constant velocity in short period of time.

[3, Table 2.4] shows the amount of the kinetic energy for the different case. As can be seen, between the cases with two baffles, case 4 has the minimum kinetic energy in the different position (x/L = 0.4, 0.6, 0.8); then, in between the cases 6–11, where the three baffles were positioned in the settling tanks, case 9 seemed to have the lowest kinetic energy magnitude. Hence, increasing the number of baffles resulted in lower circulation region and kinetic energy, thereby improving the efficiency of the settling tanks will improve.

However it is not possible to use more baffles and the numbers of baffles should be selected carefully. As mentioned before the location of second and third baffle were obtained with spoil of the circulation zone behind the first baffle. In addition, baffles implementation increase the cost and can shut down a running plant for a significant time. Hence, it is necessary to investigate the best location of the baffles in settling tank. Moreover from a design point of view, a very large number of baffles are infeasible.

| Case number | Location of baffles | | |
|---|---|---|---|
| | 1st baffle | 2nd baffle | 3rd baffle |
| 1 | 0.125 | - | - |
| 2 | 0.125 | 0.256 | - |
| 3 | 0.125 | 0.300 | - |
| 4 | 0.125 | 0.388 | - |
| 5 | 0.125 | 0.519 | - |
| 6 | 0.125 | 0.256 | 0.300 |
| 7 | 0.125 | 0.256 | 0.388 |

| | | | |
|---|---|---|---|
| 8 | 0.125 | 0.256 | 0.519 |
| 9 | 0.125 | 0.300 | 0.388 |
| 10 | 0.125 | 0.300 | 0.519 |
| 11 | 0.125 | 0.388 | 0.519 |

Table 2.2: Descriptions of different baffle position (d/L). Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, S. R. Sabbagh Yazdi, and Syafalni, “The effect of number of baffles on the improvement efficiency of primary sedimentation tanks,” Appl. Math. Model., vol. 36, no. 8, pp. 3725–3735, 2012. Reprinted with permission

| Case number | Circulation volume (%) |
|---|---|
| 1 | 32.3 |
| 2 | 30.9 |
| 3 | 30.6 |
| 4 | 30.0 |
| 5 | 30.4 |
| 6 | 30.0 |
| 7 | 29.2 |
| 8 | 29.5 |
| 9 | 28.6 |
| 10 | 29.4 |
| 11 | 29.1 |
| Without baffle | 37.1 |

Table 2.3: Circulation volume percentage in different location of the baffle. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, S. R. Sabbagh Yazdi, and Syafalni, “The effect of number of baffles on the improvement efficiency of primary sedimentation tanks,” Appl. Math. Model., vol. 36, no. 8, pp. 3725–3735, 2012. Reprinted with permission

| Case number | x/L | | |
|---|---|---|---|
| | 0.4 | 0.6 | 0.8 |
| 1 | 1.72E-04 | 1.19E-04 | 9.1E-05 |
| 2 | 1.72E-04 | 1.26E-04 | 9.8E-05 |
| 3 | 1.57E-04 | 1.15E-04 | 9.4E-05 |
| 4 | 1.55E-04 | 1.14E-04 | 9E-05 |
| 5 | 1.86E-04 | 1.21E-04 | 9.8E-05 |
| 6 | 1.51E-04 | 1.13E-04 | 8.9E-05 |
| 7 | 1.55E-04 | 1.19E-04 | 9.6E-05 |
| 8 | 1.60E-04 | 1.09E-04 | 8.9E-05 |
| 9 | 1.45E-04 | 1.08E-04 | 8.7E-05 |
| 10 | 1.58E-04 | 1.09E-04 | 8.9E-05 |
| 11 | 1.66E-04 | 1.17E-04 | 9.6E-05 |
| Without baffle | 1.95E-04 | 1.36E-04 | 9.7E-05 |

Table 2.4: Kinetic energy k in x/L sections. Reprinted from M. Shahrokhi, F. Rostami, M. A. Md Said, S. R. Sabbagh Yazdi, and Syafalni, “The effect of number of baffles on the improvement efficiency of primary sedimentation tanks,” Appl. Math. Model., vol. 36, no. 8, pp. 3725–3735, 2012. Reprinted with permission

# CHAPTER 3

# RESEARCH METHODOLOGY

## 3.1 INTRODUCTION

In this chapter, the process of determining the flow pattern of water stream flow and structural strength of solid baffles is shown step-by-step from designing the baffles until the simulation progress. There are two parts of simulation that have been performed in this project. The first simulation is to determine the pattern of water stream flow and the second simulation is to determine the static structural of stress and strain of baffles. By using SolidWorks 2017 the model of original baffles and modified baffles is designed with specific dimensions before proceed to the simulation part. The model of baffles is then imported to the ANSYS Workbench 16 (Fluent) to study the fluid flow on baffles with specified constrains. In simulation part the aspect of meshing in term of number of elements has been considered as it also one of the main contribution to gain as accurate result as possible. Finally the structural analysis of baffles also has been done to study the relationship between the geometry of the baffles with stress and strain distribution on the surface of the baffles using ANSYS Workbench 16 (Static Structural).

## 3.2 BAFFLES MODELLING

### 3.2.1 ORIGINAL BAFFLES

The original baffles is determined from Angkasa Consultancy Services Sdn. Bhd for Mengkubau Dam’s spillway. The baffles is located in the stilling basin with 2 rows. The first row contain 12 baffles and the second row contain 11 baffles which is arranged stagger between the first and second baffles. Figure 3.1 shows the design of the Mengkubau Dam’s spillway with baffles inside the stilling basin. The top view of the baffles inside the stilling basin is shown in Figure 3.2 and the dimension of the baffles is illustrated in Figure 3.3.

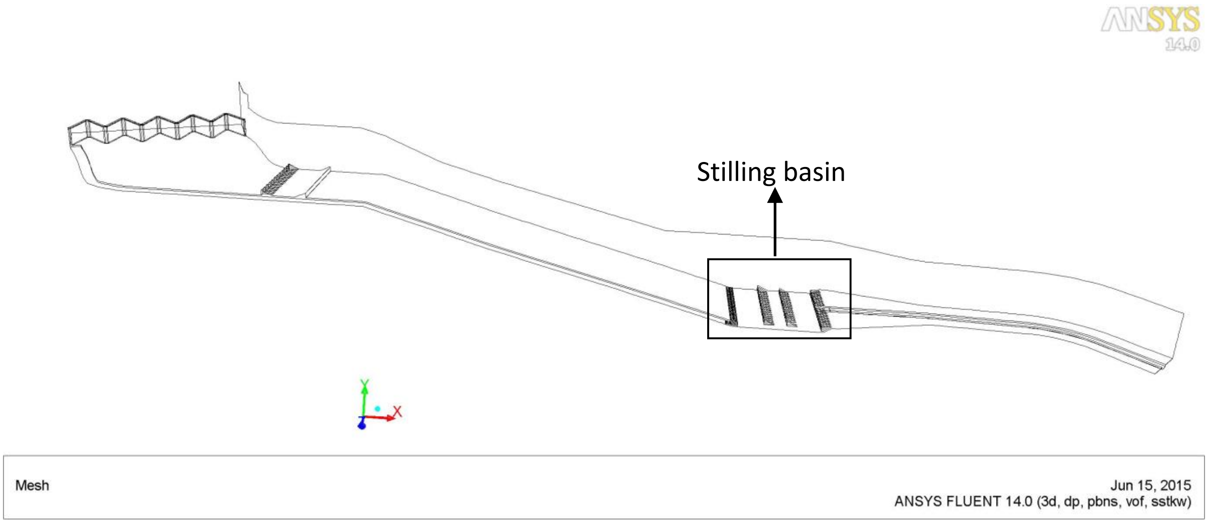



Figure 3.1: Mengkubau Dam's spillway

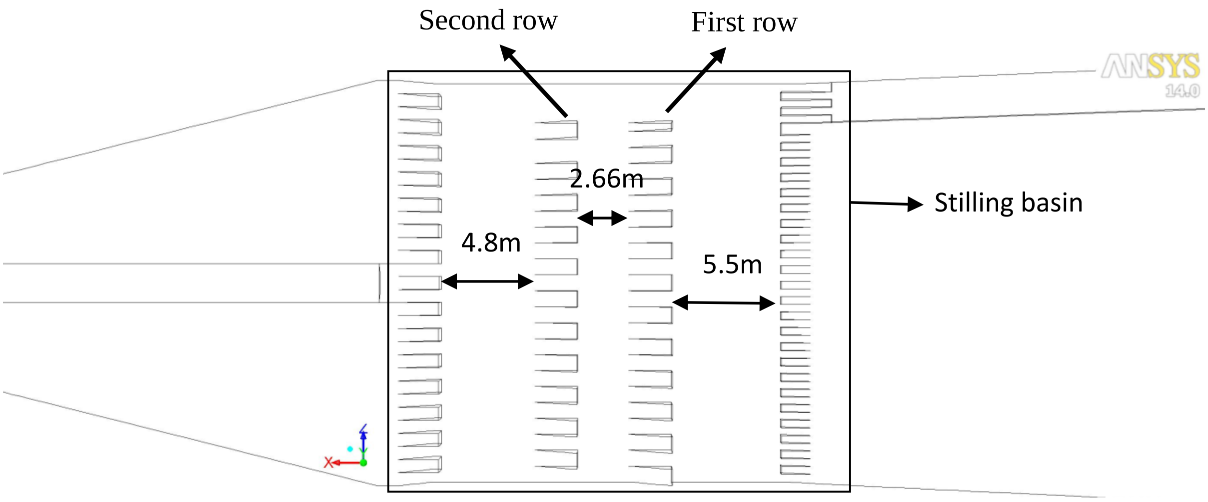



Figure 3.2: Top view of baffles inside the stilling basin

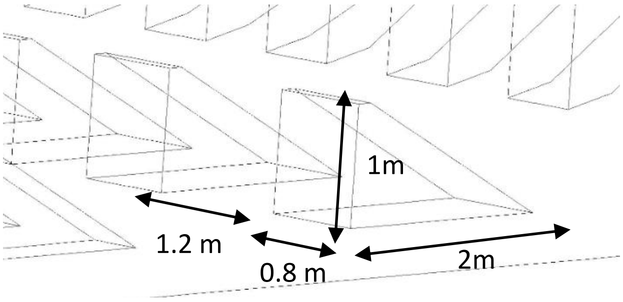



Figure 3.3: The dimension of baffles

In Figure 3.4 the model of original baffles is designed in SolidWorks 2017 with specified dimensions and the top of each baffles has flat surface with 200mm in length.

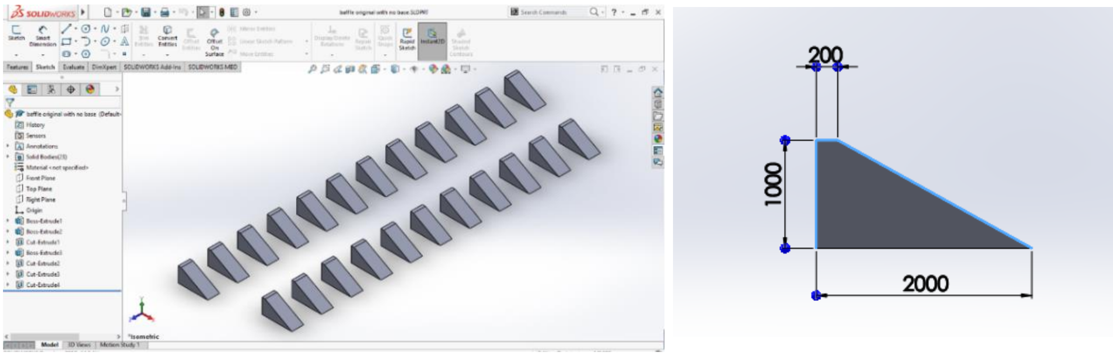



Figure 3.4: Original baffles model in SolidWorks 2017

### 3.2.2 MODIFIED BAFFLES 1

For modified baffles 1 the dimension of the baffles is same as original baffles. But, it has curvature at the edge of the top front surface of the baffles with radius of 80mm. By making a curvature at the edge of the baffles the velocity of water that flow through the curvature will be slightly increased. Figure 3.5 shown is a model of modified baffles with 80mm radius edge curvature.

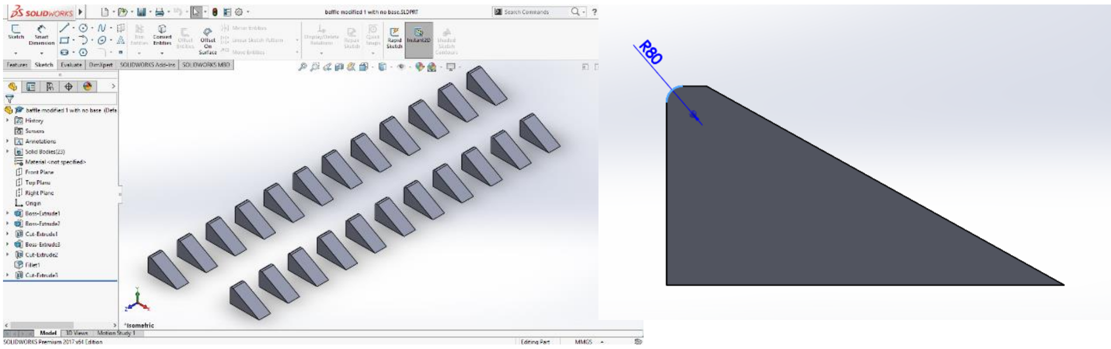



Figure 3.5: Modified baffles 1 in SolidWorks 2017

### 3.2.3 MODIFIED BAFFLES 2

For this model, it is designed with longer flat surface on the top of the baffles which is 400 mm. The longer the flat surface means there are more contact area of the top baffles with the water. Figure 3.6 shows a model of modified baffles 2 in SolidWorks 2017.

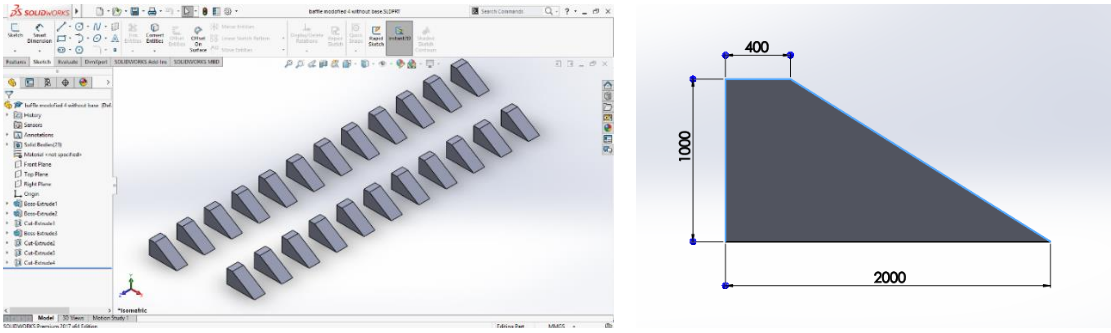



Figure 3.6: Modified baffles 2 in SolidWorks 2017

### 3.2.4 MODIFIED BAFFLES 3

For modified baffles 3, the dimension is also same as original baffles but the front surface of the baffles have four extruded square solids of 200mm. Figure 3.7 shown is the model of the modified baffles 3 and the dimension of the front baffles.

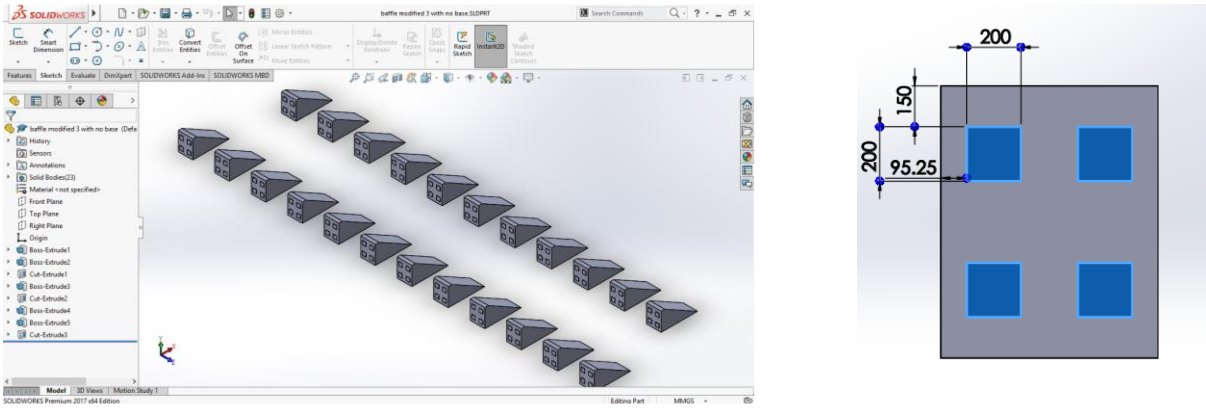



Figure 3.7: Modified baffles 3 in SolidWorks 2017

## 3.3 EXPORT FILES TO ANSYS 16

After the modelling part of the baffles in SolidWorks is completed the file is then being exported to ANSYS Workbench 16 (Fluent) for meshing and simulation process. The baffles also can be modelled in the ANSYS using DesignModeler as an alternative way to design the baffles. As for first past simulation Figure 3.8 shown is a model of baffles from SolidWorks 2017 that was imported to the ANSYS Workbench 16 (Fluent).

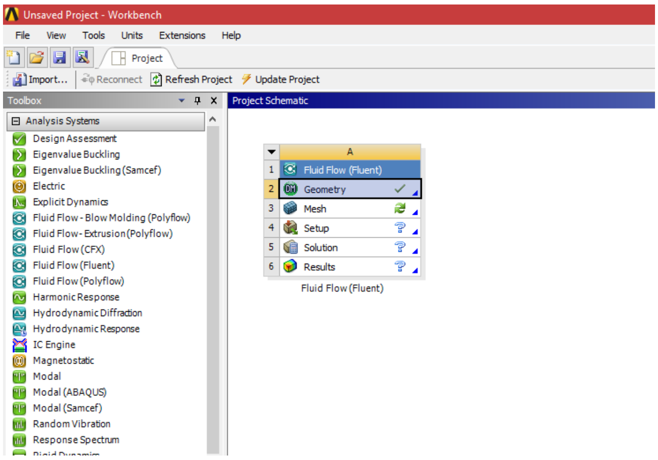



Figure 3.8: Imported file of baffles from SolidWorks 2017

Because of this is a fluid flow simulation the fluid domain for water stream flow is created with height of 4 m. The geometry of the fluid domain is determined from the dimension of the stilling basin and the result is shown in Figure 3.9.

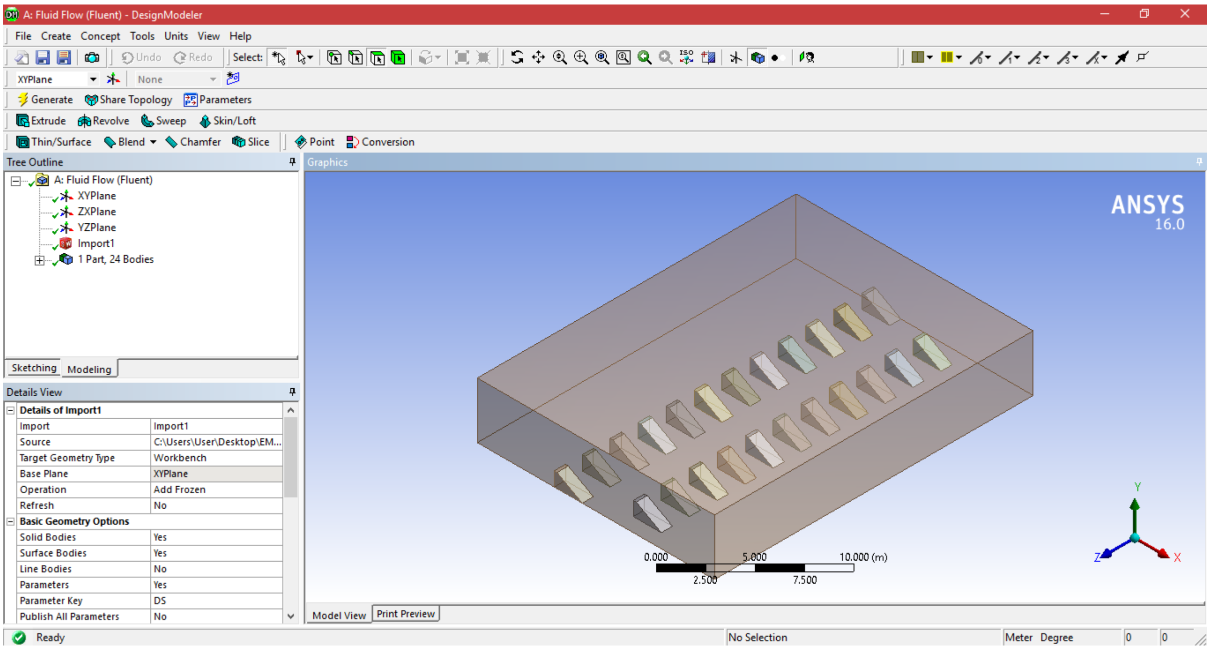



Figure 3.9: Baffles design with fluid domain

The fluid domain that has been created is defined as enclosure and each of affles is suppressed as shown in Figure 3.10. Then the model is ready for meshing part.

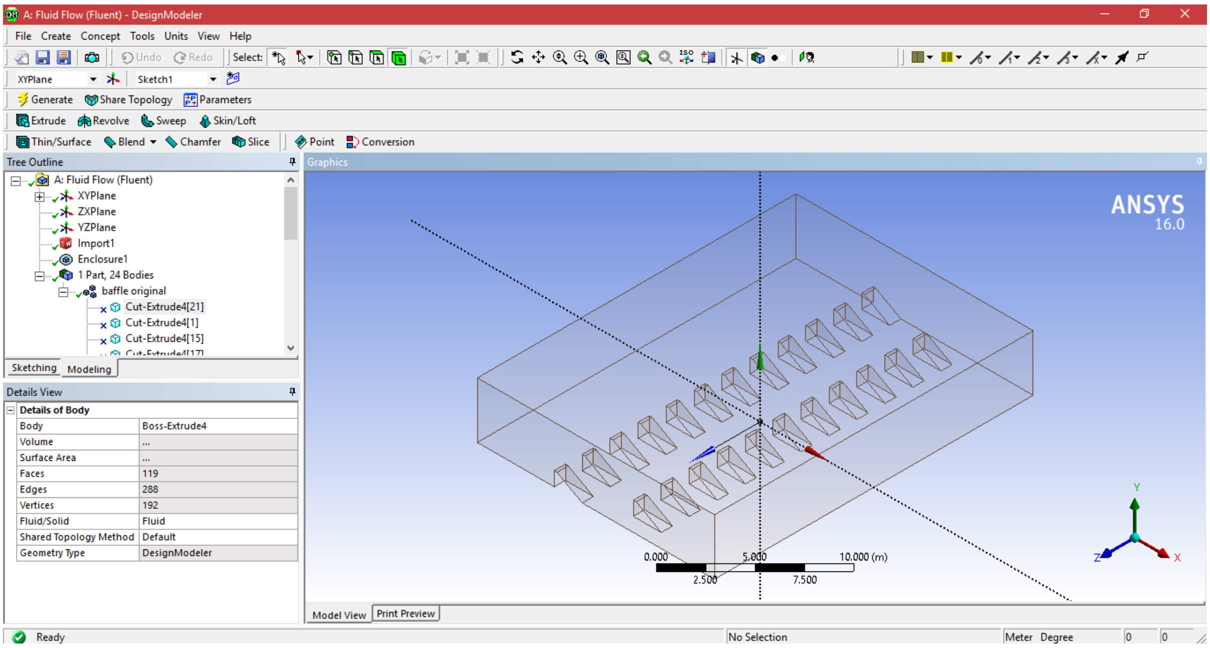



Figure 3.10: Model of suppressed baffles

For second part simulation the design for fluid domain is different to first part simulation. The design has slope which is the chute to flow the water. The dimension is obtained from Mengkubau Dam's spillway but this model has been simplified without complex dimensions and design. Figure 3.11 shows the model for second part simulation.

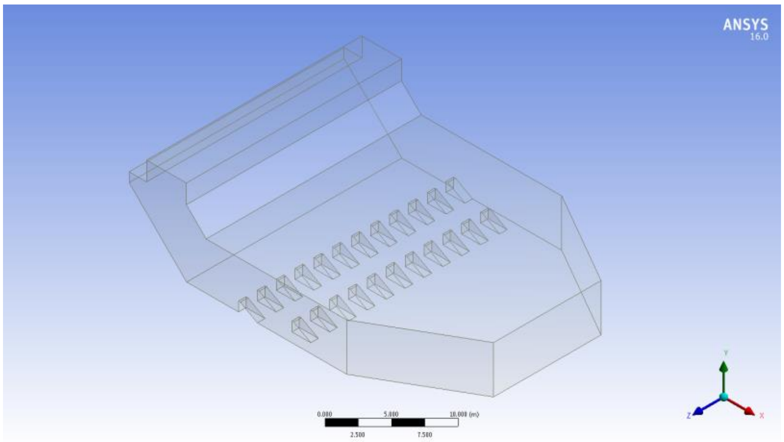



Figure 3.11: Design of model for second part simulation

## 3.4 MESHING

In this simulation, meshing is considered as one of the most important thing because a good meshing can produce a good result. Meshing means we create a mesh of some grid-points called 'nodes'. It is done with a variety of tools and options available in the software. The results are calculated by solving the relevant governing equations numerically at each of the nodes of the mesh. The governing equations are almost always partial differential equations, and Finite element method is used to find solutions to such equations. The pattern and relative positioning of the nodes and number of elements also affect the solution, the computational efficiency and time. This is why good meshing is very essential for a computer simulation to give good results.

### 3.4.1 MESH STUDY

To get a proper and accurate results between all models of baffles, mesh study is significant and need to be performed. If the mesh have very low number of elements the result will be inaccurate and invalid. If the mesh have too much number of elements the result is accurate but it will consume much longer time to solve the solution. Thus, appropriate number of elements is required to get accurate result and not wasting the time.

For this mesh study, original baffles model is used by comparing the results of two different meshes. The first mesh has 432048 number of elements and 95278 number of nodes (the mesh that is used in this study) and the second mesh has 448157 number of elements and 111177 number of nodes (the mesh that is used for comparison). Figure 3.12 and Figure 3.13 illustrate the mesh of original baffles for mesh 1 and mesh 2.