Computational Fluid Dynamics Analysis of Gravitational Filter Based on Origami Concept (G-Filtro) Acted as Water Filtration Device in Water Tank

M H Zawawi1,2, Mohd Mustafa Al Bakri Abdullah2, M A Nazmi1, W N Yusairah1, N H Hassan1, Elyssa Shahira Jupri1, Mohammad Azam Afiq1, M A Kamaruddin3 and Mohd Remy Rozainy Mohd Arif Zainol4,5 and A R Irfan6,7

1Department of Civil Engineering, College of Engineering, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor Darul Ehsan, Malaysia.
2Center of Excellence Geopolymer & Green Technology, Universiti Malaysia Perlis, 01000 Kangar, Perlis, Malaysia.
3School of Industrial Technology, Universiti Sains Malaysia, 11800 Pulau Pinang Malaysia
4River Engineering and Urban Drainage Research Centre (REDAC), Universiti Sains Malaysia, 14300 Nibong Tebal, Malaysia.
5School of Civil Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal, Malaysia.
6School of Manufacturing Engineering, Universiti Malaysia Perlis, Kampus Tetap Pauh Putra, 02600 Arau, Perlis, Malaysia.
7Green Design and Manufacture Research Group, Center of Excellence Geopolymer and Green Technology (CEGeoGTech), Universiti Malaysia Perlis, 01000 Kangar, Perlis, Malaysia.

E-mail: MHafiz@uniten.edu.my

Abstract. Water is the most demanding resource in the world especially in the era of the rapid growth of population and development. Therefore, delivering good quality water to consumers is important for their security of lives. Gravitational Filter Based on Origami Concept (G-Filtro) is a water filtration device that will be placed in a water tank at the residential area before people consumed it for everyday used. The specialty of this device, it does not necessarily attach to any power source. The objective of this project is to design and develop 3D G-Filtro using 3D CAD software and CFD analysis is used to determine the efficiency of the G-Filtro by simulating and model the pressure, velocity and streamline. Based on the result, the maximum water velocity for simulation without the filter is 0.266 m/s and the maximum velocity for first and second designed filter are 0.254 m/s and 0.243 m/s respectively. From the CFD simulation, the G-Filtro can remove the contaminants in water tank efficiently. Hence, it is proven that G-Filtro can be added as a new water filtration device.

1. Introduction
In 1970, increasing computational power has been accompanied by fast development in the software aimed at the solution of fluid flow problems [1]. Now, several software suites intended for the solution of complex fluid flow problems are commercially available. Initially, CFD was almost exclusively linked with mechanical and aerospace industries allowing simulation of the processes occurring in combustion chambers of rocket engines.
CFD is the simulates fluid motion and heat transfer using numerical approaches. Computational fluid dynamics CFD is a tool with amazing flexibility, accuracy, and breadth of application [2,3]. The CFD software can analyze a range of problems related to laminar and turbulent flows, incompressible and compressible fluids, multiphase flows and provides insights to help optimize the design.

Turbulent flow occurs when the water moves quickly through the layers. The magnitude and direction of the fluid changes when the velocity of the fluid is constantly quick at a stage. The amount of Reynolds above 4000 stated that the flow is turbulent. Turbulent flow in water pipes is the most prevalent form of flow. Recirculation, eddies, and evident randomness also characterize turbulence. In turbulent flow, the velocity of the fluid at a point changes in magnitude and direction continually [4-6].

For this research, the standard k-epsilon (k-ε) model is selected because the CFD is the most popular model used to simulate mean flow characteristics for turbulent flow conditions [7,8]. It is a two-equation model that uses two transport equations to describe turbulence in particular. The initial impetus for the K-epsilon model is to enhance the mixing-length model and to find an option to prescribing algebraically turbulent longitudinal scales in mild to elevated complexity flows [9,10].

The 'k' is the first component carried that determines the energy in the turbulence and the 'ε' is the second component transported in the turbulent dissipation that determines the dissipation rate of the turbulent kinetic energy. The standard k-epsilon model is selected instead of k-omega because the model can be easily implemented [11,12].

Navier – Stokes equations, which describe many single-phase (gas or liquid, but not both) fluid flows, are the basic foundation of almost all CFD issues. By removing conditions that describe viscous activities to produce the Euler equations, these equations can be simplified. Further simplification will yield the complete potential equations by removing conditions describing vorticity [13].

In an investigation by [10-12], the strategy for the finite volume is used to separate Navier-Stokes conditions in air and water also the technique for the volume of liquid is utilized to represent the subsequent unrestricted apparent multi-phase stream. CFD solution in predicting the overall force experienced by the hull, the pile-up phenomenon, the velocity field in the water, the distribution of the hydrodynamic loading, pressure and the energy transfer during the impact.

Other than that, there are a few unique benefits of CFD over experiment-based approaches to simulate the design of the fluid system. The benefits are including a substantial decrease in the lead times and costs of the new design. Besides, it has the competence to study the systems where controlled experiments are difficult or can say it is impossible to carry out. Besides, it also can study systems under hazardous conditions and control of their normal performance limits and practically detail with an unlimited level of outcomes or results [12, 13].

2. Methodology
The phase of work for this G-Filtro Project is divided into two. The first phase of work is to develop a 3-Dimensional model design of G-Filtro with the existing residential water tank using 3D Computer-Aided Design (CAD) and exported to the next software of CFD for analysis purposes. From the analysis, the numerical study is pertinent based on the hydraulic parameter. For instance, the velocity at the outlet of water tank, pressure and water streamflow that passing through the G-Filtro. The second phase of the work will be the study of the efficiency of the filter by running a few laboratory tests on the G-Filtro prototype. The water will be tested before and after the installation of G-Filtro prototype in the water tank. Unfortunately for this paper, only simulation results will be discussed on the result and result and discussion part as stated below.

3. Result and Discussion
The first phase for the research is done by doing 3D drawing (CAD) using SolidWorks. The model was drawn by the real model dimension of the residential water tank and designed a water filter using activated carbon. The drawing needs to be the same as the real model so it can simulate the water movement and flow to produce an accurate result. Simulation data of 30 seconds were obtained from CFD simulation. The effectiveness of the water filter can be determined by reviewing certain parameters
such as water velocity, streamline, and volume of the fraction. The data were compared with three different conditions, the first simulation is a residential water tank operate without any filter, second is using the first designed filter and last is using the second designed filter.

**Figure 1.** (a) First Designed Filter and (b) Second Designed Filter

**Figure 2.** (a) Velocity for None Filter Simulation, (b) Velocity of First Designed Filter and (c) Velocity of Second Design Filter
Figure 3. Velocity of Water Passing Through First Design Filter for Top and Bottom (a) and Velocity of Water Passing Through Second Designed Filter (b)

Based on figure 2 (a) and (b), the filter designed into two different shapes and placed into two different positions. The designs were different in material characteristics such as porosity value for water penetration. For the first filter shown in figure 2 (a), it was build using Foam Activated Carbon and its porosity value is 65% while the filter in figure 2 (b) was using Granular Activated Carbon and its porosity value is 85%. For the first filter, it was placed in two different positions across the water tank and its distance from the outlet is 0.15m and 0.68m respectively. The second filter placed near to the outlet and only covered one-quarter of the tank.

From the CFD simulation, the first parameter observed was water velocity. Water velocity is very important to be observed because of the designed filter need to treat the water while not affect water velocity coming out from the outlet. When the designed filter applied, the water velocity of the outlet should not be much different. The result as shown in figure 3 and figure 4 are the water velocity for three different conditions after simulating for 30 seconds.

Based on figure 3, the water velocity was observed from the way of the water flow out through the outlet inside view. For the first case, the tank operates without any designed filter and the maximum velocity at the outlet is 0.266 m/s. While maximum water velocity at the outlet in the first and second design filter cases is 0.254 m/s and 0.243 m/s respectively. The water velocity pattern in all cases is different due to the air factor and the position of the filter be installed. Based on the data above, the change is not much different before and after the installation of the design filter. Hence, the filter installed is useable and practicable for treatment.

Water velocity also observed in the top view of each filter, especially above the outlet. The data were recorded in figure 4 (a) and (b) are for the first designed filter, which is at the top and bottom filter. Figure 4 (c) shows the water velocity occurred at the top part of the filter. The data show that there is
velocity on the surface of the filters because it is permeable. However, the porosity value for both filters is different, and the position of the filters affects the water velocity since the water only flows by gravity and atmospheric force.

![Streamline for None Filter Simulation](image1)
![Streamline for First Designed Filter](image2)
![Streamline for Second Design Filter](image3)

Figure 4. (a) Streamline for None Filter Simulation, (b) Streamline for First Designed Filter and (c) Streamline for Second Design Filter

Streamline is one of the parameters to observe the flow of the water when the tank operates. Streamline is important to observe the critical area of the water flow when it hit the filter. Based on figure 5, the streamline in all the cases is different. From the observation, figure 5 (a) has a normal flow of water while figure 5 (b) and (c) has its pattern since the designed and position of the filters are different. However, the streamline shows that the water is passing through the filter as usual without any problem. It means the simulation is true and the critical water flow can be observed on the filters when it operates.

The volume fraction is a parameter for observing the mixture between air and water body in the simulation. It is a very important parameter to know the reaction of the mixture when the tank is operating. Based on figure 5, the mixture of the water and the air are different since the streamline and velocity are also different for every case. In figure 5 (a), the mixture tends to be stable because the streamline is normal and steady. After all, it does not have any filter. For figure 5 (b) and (c), the volume fraction value is quite high above the outlet compared to figure 5 (a). It is because of the permeability filter that affects the water flow. However, high volume fraction value does not give massive effect because the water velocity between without and with the presence of the filter does not have much difference.
Figure 5. (a) Volume Fraction for None Filter Simulation, (b) Volume Fraction for First Filter Design and (c) Volume Fraction for Second Filter Design.

4. Conclusion

In conclusion, the objectives in this study which to design and develop G-Filtro using 3D CAD software and to do numerical modeling for a certain parameter such as water velocity, streamline and volume fraction were achieved. Based on the result, the maximum water velocity for simulation without the filter is 0.266 m/s and the maximum velocity for first and second designed filter are 0.254 m/s and 0.243 m/s respectively. The streamline and volume fraction parameter observed from every case in the simulation show different changes and behavior since the presence of the filters with different porosity values. The removal efficiency assessment by using activated carbon as a water treatment mechanism was achieved as proven by the removal parameters. This is enough to prove that G-Filtro is the recent solution and applicable to help in water quality improvement for the communities.

References

[1] F Xuening, C Lei, D Yuman, J Min and F Jinping 2015 *Journal Desalination and Water Treatment* 57 139–145.
[2] C Peng, G Xu, W Wu, H Yu and C Wang 2017 *Computers and Geotechnics* 81 239–248
[3] N A Rosli, M H Zawawi, and R A Bustami 2012 *Procedia Engineering* 50 69-77
[4] F H Harlow and J E Welch 1965 *Physics of Fluids* 8 2182-2189
[5] R Tarpagkou and P Asterious 2014 *Powder Technology* 268 139-49
[6] M H Zawawi, N A Aziz, M R M Radzi, N H Hassan, M Z Ramli, N M Zahari, M A Abbas, A Saleha, A Salwa and Z C Muda 2018 *AIP Conference Proceedings* 2030 020245
[7] M H Zawawi, N H Hassan, R M Zakwan, Z N Mubin, M R M Radzi, A Saleha, A Salwa, M S Lartyah, C M Zakaria and M A Kamaruddin 2018 *AIP Conference Proceedings* 2030 020244
[8] A Salwa, M H Zawawi, R M Zakwan, Z N Mubin, A Saleha, N H Hassan, M R M Radzi, M S Lariyah, and C M Zakaria 2018 *AIP Conference Proceedings* **2030** 020254

[9] A R Irfan, M Z M Zarhamdy, S M Sazli, M N Amni, N. A. Shuaib, and A Azlida 2019 *AIP Conference Proceedings* **2129** 020181

[10] M H Zawawi, N H Hassan, R M Zakwan, Z N Mubin, M R M Radzi, A Saleha, A Salwa, M S Lariyah, C M Zakaria, M A Kamaruddin 2018 *AIP Conference Proceedings* **2030** 020244

[11] M H Zawawi, A Saleha, A Salwa, N H Hassan, Z N Mubin, R M Zakwan, C M Zakaria 2018 *AIP Conference Proceedings* **2030** 020252

[12] V. Udoewa and V. Kumar, 2012 Computational Fluid Dynamics Applied Computational Fluid Dynamics (Singapore 10.5772/28614)

[13] K A Williams and D M Snider 2003 *Development of CFD Software To Support the Engineering of Lost Foam Pattern Blowing and Steaming* Final Technical Report 1-30