Smoothed particle hydrodynamics method for simulating waterfall flow

To cite this article: M G Suwardi *et al* 2018 *J. Phys.: Conf. Ser.* **971** 012035

View the article online for updates and enhancements.

Smoothed particle hydrodynamics method for simulating waterfall flow

M G Suwardi, Jondri, and D Tarwidi

School of Computing, Telkom University, Jalan Telekomunikasi No. 1 Terusan Buah Batu, Bandung 40257, Indonesia

E-mail: `ghazali.suwardi@gmail.com`

Abstract. The existence of waterfall in many nations, such as Indonesia has a potential to develop and to fulfill the electricity demand in the nation. By utilizing mechanical flow energy of the waterfall, it would be able to generate electricity. The study of mechanical energy could be done by simulating waterfall flow using 2-D smoothed particle hydrodynamics (SPH) method. The SPH method is suitable to simulate the flow of the waterfall, because it has an advantage which could form particles movement that mimic the characteristics of fluid. In this paper, the SPH method is used to solve Navier-Stokes and continuity equation which are the main cores of fluid motion. The governing equations of fluid flow are used to obtain the acceleration, velocity, density, and position of the SPH particles as well as the completion of Leapfrog time-stepping method. With these equations, simulating a waterfall flow would be more attractive and able to complete the analysis of mechanical energy as desired. The mechanical energy that generated from the waterfall flow is calculated and analyzed based on the mass, height, and velocity of each SPH particle.

1. Introduction

Indonesia is rich in diversity of natural ecosystem, including waterfall which spreads in the majority of regions. Besides as tourism destination, a waterfall is also as natural resource which is renewable. Energy which come from waterfall is an opportunity and chance to use as sustainable power plants, while the Indonesia is lack of innovation in utilization of renewable resource.

Principle of hydroelectric power plants of waterfall is by changing water flow to mechanical energy (using water turbine) and from the mechanical energy to electrical power (using generator). Waterfall and its mechanical energy surely could help fulfill electrical needed in Indonesia. Amount of the energy resulted could be analyzed with simulating its water flow. The problem could be simulated with several methods. The waterfall simulation had been studied by Uchiyama et al. [1] which was using two-dimensional moving particle semi-implicit (MPS) method. However, another method that is used frequently and implemented in many fields of study is smoothed particle hydrodynamics (SPH) method. While there are numerous alternative methods, such as finite difference and finite element but it could not resolved the discontinuity of fluid motion.

Kellomaki [2] had studied the large-scale water simulation in games in which he compared some methods performance. It was concluded that SPH-method could provide the possibility of almost abundant water topology changes which could not be provided by most other methods.

SPH method is easy to implement and work best when applied to shallow water equation. Moreover, Onderik [3] build an animation of particle based miscible and immiscible fluids with multiple interfaces by using SPH method.

SPH method is an approximation of Lagrangian for computation in field of fluid dynamics. It model fluid flow as group of particles that move in real-time by using law of Hydrodynamics and gravitational force. The advantages of SPH method in fluid simulation can be seen more obvious in [4]. SPH could simulate phenomenon which has domain of complex geometry while other methods were not easy to implement. This paper focuses on SPH method in simulating waterfall flow for analyzing influence of height to mechanical energy resulted.

2. SPH Theory
In this section, SPH theory to simulate water flow will be briefly discussed. The mechanical energy is calculated by summing up the contribution of mechanical energy of each particles.

2.1. Waterfall's mechanical energy
The first Thermodynamics Law explains on conservation of energy, which means that amount of energy at the before and the end of process is the same. Energy could not be demolished as well as be created, but energy could evolve from one form to other form of energy [5]. Furthermore, mechanical energy is accumulation between kinetic and potential energy. Waterfalls mechanical energy could be implemented in hydroelectric power plant.

Process of hydroelectric power plant is by converting waters potential energy become mechanical energy by turbine and then the energy is converted to electrical energy by generator. Waters height and velocity also support the process. Electrical power is obtained by using waterfalls height. Mechanical energy is given by

$$\text{Mechanical energy} = \frac{1}{2}mv^2 + mgh, \tag{1}$$

where m is waters mass, v is waters velocity, g is gravitational acceleration, and h is waterfalls height from ground. In SPH method, the amount of energy can be noted as discrete system (ignoring external parameter) [6]. It is written as

$$E = \sum_i m_i \left(\frac{1}{2} v_i + u_i \right). \tag{2}$$

The subscript i represents the parameter is belongs to particle i. Moreover, u denotes particles heat energy. Because this simulation just explain mechanical energy, thus heat energy should be ignored. Hence, we could use equation (1) for calculating particles mechanical energy. Therefore, in SPH method formulation, the mechanical energy can be written as

$$\text{Mechanical energy} \approx \sum_{i=1}^{n} \frac{m_i v_i^2}{2} + m_i \, g \, y_i \tag{3}$$

where, y_i is position of particle i in y axis, which means as particles height.

2.2. SPH Method
In this simulation, we implement SPH method by doing Lagrangian approximation for computing fluid dynamics. In the method, the fluid is represented as a collection of particles which have fluid properties, such as mass, density, velocity, acceleration, and pressure [7]. In each particle, they have velocity, acceleration and position. They have spatial distance, which then be meshed by kernel function that could evaluate accuracy of physical quantity of each particle.
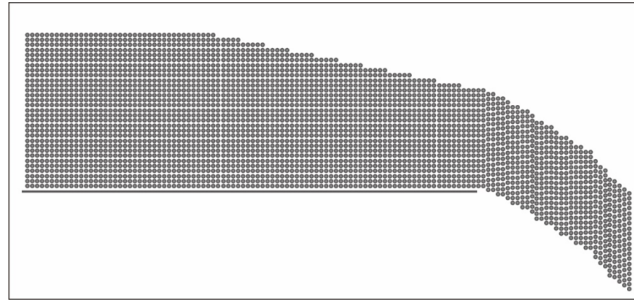
Figure 1. Illustration of SPH particle in waterfall flow simulation with SPH method.

One disadvantage of the SPH method is that its simulation is slower than the grid method. The reason is because particles move freely between one another, so it takes more time to find neighbor particles. In addition, it is necessary to calculate the distance between each pair of neighbor particles involving the calculation of square roots [8]. However, the SPH method results is easier when it is visualized compared to the grid method. Figure 1 shows illustration of particles motion in waterfall flow with SPH method.

There are at least four steps to be considered in the SPH approximation:

- The continuum domain is represented by a series of randomly distributed particles.
- Functions such as density, velocity, and energy are formulated by integral formulas.
- The integral formula of the function is converted into a finite sum of the neighbor particles.
- The kernel function used to satisfy the numerical accuracy of the results.

SPH method is started by giving integral formula for continue equation $f(x)$:

$$f(\mathbf{x}) = \int_\Omega f(\acute{\mathbf{x}})\delta(\mathbf{x} - \acute{\mathbf{x}})d\acute{\mathbf{x}}, \tag{4}$$

where x is position vector and Ω is the domain of integral which consist of x. And also, $\delta(\mathbf{x} - \acute{\mathbf{x}})$ is function of Dirac delta which can be described as follows

$$\delta(\mathbf{x} - \acute{\mathbf{x}}) = \begin{cases} \mathbf{x} = 1, & \text{if } \mathbf{x} = \acute{\mathbf{x}} \\ \mathbf{x} = 0, & \text{if } \mathbf{x} \neq \acute{\mathbf{x}}. \end{cases} \tag{5}$$

If Dirac delta function is replaced by a smoothing function, W, it yields the kernel approximation of $f(x)$:

$$< f(\mathbf{x}) > = \int_\Omega f(\acute{\mathbf{x}})W(\mathbf{x} - \acute{\mathbf{x}}, h)d\acute{\mathbf{x}}, \tag{6}$$

where h represents the influence area of smoothing function W which is called smoothing length. Illustration of smoothing length of kernel function is shown by Figure 2. Liu and Liu [9] derived kernel approximation for $\nabla \cdot f(x)$ by implementing Divergence Theorem. It is given by

$$< \nabla \cdot f(\mathbf{x}) > = -\int_\Omega f(\acute{\mathbf{x}})\nabla \cdot W(\mathbf{x} - \acute{\mathbf{x}}, h)d\acute{\mathbf{x}}. \tag{7}$$

The next step of the SPH method is to discretize a continuous function into a set of particles. The integral starts with the number of neighbor particles of particle i in position x_i in the domain of influence Ω_i. The object divided into small parts with mass $m_1, m_2, ..., m_N$, then

the volume of each part is $\Delta V_j = m_j/\rho_j$. Therefore, the particle approaches of equations (6) and (7) are

$$< f(\mathbf{x}) > = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) W(\mathbf{x} - \mathbf{x}_j, h), \qquad (8)$$

$$< \nabla \cdot f(\mathbf{x}) > = - \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) \cdot \nabla W(\mathbf{x} - \mathbf{x}_j, h), \qquad (9)$$

where N represents the number of neighbor particles, m_j and ρ_j are mass and density of particle j.
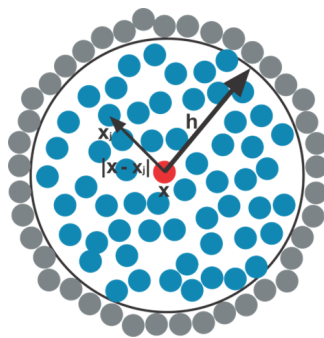

Figure 2. Smoothing length of kernel function.

2.3. Kernel function
The accuracy of SPH method depends on kernel function. In order to implement kernel function, it should fulfill several conditions as follow [6]:

- Normalization condition: $\int_\Omega W(\mathbf{x} - \acute{\mathbf{x}}, h) \, \mathrm{d}\Omega = 1$
- Delta function property: $\lim_{h \to 0} W(\mathbf{x} - \acute{\mathbf{x}}, h) = \delta(\mathbf{x} - \acute{\mathbf{x}})$
- Compact support: $W(\mathbf{x} - \acute{\mathbf{x}}, h) = 0$ outside Ω domain.

The are many types of kernel function. One of them is cubic spline. This simulation uses kernel function of cubic spline which is frequently used in SPH simulation:

$$W(r, h) = \frac{10/7\pi}{h^2} \begin{cases} 1 - \frac{3}{2}(\frac{r}{h})^2 + \frac{3}{4}(\frac{r}{h})^3, & 0 \leqslant \frac{r}{h} < 1 \\ \frac{1}{4}(2 - \frac{r}{h})^3, & 1 \leqslant \frac{r}{h} < 2 \\ 0, & \frac{r}{h} \geqslant 2, \end{cases} \qquad (10)$$

where r is distance between particle and its neighbor particle. Moreover, the derivative of $W(r, h)$ can be written as

$$\frac{\partial W(r, h)}{\partial r} = \frac{10/7\pi}{h^2} \begin{cases} -3(\frac{r}{h}) + \frac{9}{4}(\frac{r}{h})^2, & 0 \leqslant \frac{r}{h} < 1 \\ -\frac{3}{4}(2 - \frac{r}{h})^2, & 1 \leqslant \frac{r}{h} < 2 \\ 0, & \frac{r}{h} \geqslant 2. \end{cases} \qquad (11)$$

International Conference on Data and Information Science

IOP Conf. Series: Journal of Physics: Conf. Series **971** (2018) 012035

2.4. Governing equation

Governing equation for the fluid dynamics is momentum equation (Navier-Stokes equation), continuity equation, and equation of state [10].

2.4.1. Momentum equation Navier-Stokes or momentum equation in continuous domain can be writen as

$$\frac{D\mathbf{v}}{Dt} = -\frac{1}{\rho}\nabla p + \mathbf{F} \tag{12}$$

where $\mathbf{v}$, ρ, and $\mathbf{F}$ are velocity, density, and external force per mass of fluid, respectively. In this case, surface tension is assumed that do not significantly happen, thus it could be ignored. In SPH formulation, momentum equation could be written as

$$\frac{\mathrm{d}\mathbf{v}_i}{\mathrm{d}t} = -\sum_{j=1}^{N} m_j\left(\frac{P_j}{\rho_j^2} + \frac{P_i}{\rho_i^2}\right)\nabla_i W(\mathbf{x}_i - \mathbf{x}_j, h) + \mathbf{F}_i \tag{13}$$

2.4.2. Continuity equation It is assumed that the fluid is weakly compressible. It means that the density of fluid change with time, but the change of density is not significant. The continuity equation can be written as

$$\frac{D\rho}{Dt} = -\rho\nabla \cdot \mathbf{v} \tag{14}$$

Equation (14) above could be written in SPH notation as

$$\frac{\mathrm{d}\rho_i}{\mathrm{d}t} = \sum_{j=1}^{N} m_j(\mathbf{v}_i - \mathbf{v}_j) \cdot \nabla_i W(\mathbf{x}_i - \mathbf{x}_j, h). \tag{15}$$

2.4.3. Equation of state Equation of state represents relationship between density and pressure. In simple form, the equation of state is given by

$$P = c^2(\rho - \rho_0) + P_{atm} \tag{16}$$

where c, ρ_0, and P_{atm} are speed of sound, density reference, atmospheric pressure, respectively. The choice of the speed of sound is very important in the simulation because it influences the change of density. The speed of sound can be selected as $c = \sqrt{gH}$, in order to keep fluctuation of density less than 1% [10].

2.5. Equation of particle movement

Particle movement can be calculated by solving the equation as follow [11]

$$\frac{d\mathbf{r}_a}{dt} = \mathbf{v}_a \tag{17}$$

or XSPH variant

$$\frac{d\mathbf{r}_a}{dt} = \hat{\mathbf{v}}_a = \mathbf{v}_a + \epsilon\sum_b m_b\left(\frac{v_{ba}}{\bar{\rho}_{ab}}\right)W_{ab} \tag{18}$$

where $\bar{\rho} = (\rho_a + \rho_b)/2$ and $\epsilon\,(0 \leq \epsilon \leq 1)$ is a constant. The XSPH variant moves a particle with velocity which is close to average of its neighborhood [12].
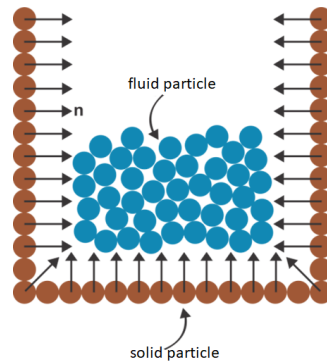
Figure 3. Solid particles boundary with its normal vector.

2.6. Time stepping

Time stepping is differential function which would run the simulation movement (plot of particles position) each step. In this simulation, SPH method is solved by Leapfrog method of time stepping. This method uses general formula in Euler, that is [13]

$$x = x + \frac{h}{2}v, \tag{19}$$

where h is interval of time stepping (time difference). Depend on numerical integration (midpoint method) which is used in this Leapfrog method, we could change v with midpoint of interval [14]:

$$x_1 = x_0 + hv_{1/2}. \tag{20}$$

Assuming that we can calculate $v_{1/2}$ by several ways. Then, we can implement rule of midpoint for calculating value of v at the next step.

$$v_{3/2} = v_{1/2} + hF(x_1) \tag{21}$$

Then we receive the next step x by $x_2 = x_1 + hv_{3/2}$. After we start with x_0 and $v_{1/2}$, we can continue x and v by using Leapfrog method and relating one and each other:

$$x_{n+1} = x_n + hv_{n+1/2}, \tag{22}$$

where

$$v_{n+3/2} = v_{n+1/2} + hF(x_{n+1}). \tag{23}$$

2.7. Boundary treatment

In SPH approximation, particle position should be paid attention when it close to boundary wall. Boundary walls are assumed as solid particle. It is needed for preventing fluid particle to permeate boundary wall. Fluid particles, which close to wall boundary would face repulsive force from solid particle. The repulsive force which is faced by fluid particle j is normal to solid particle i is given by

$$\mathbf{f}_{ji} = \mathbf{n}_i R(y)P(x) \tag{24}$$

By using vector of normal from solid particle boundary, then

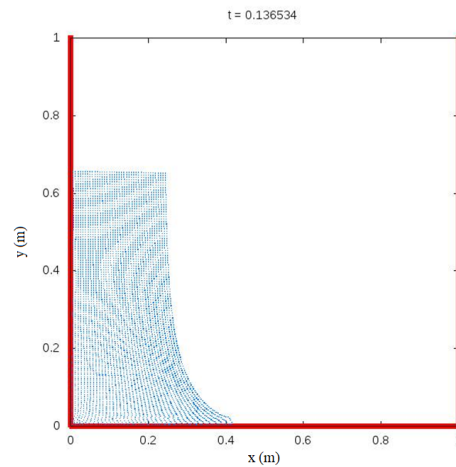$$R(y) = \frac{1}{\sqrt{q}}(1-q) \quad \text{and} \quad P(x) = \frac{1}{2}(1 + cos(\frac{\pi x}{\Delta p}))$$

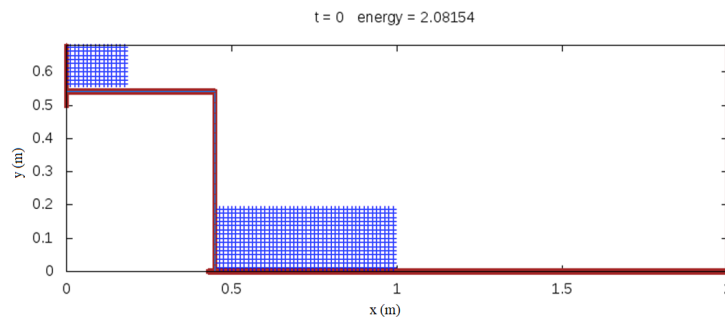**Figure 4.** Example of dam-break simulation using SPH method.

Figure 5. Initial set up of waterfall simulation using SPH method.

where $q = y/(2\Delta p)$, x is projection distance of fluid particle j at vector tangent from solid particle boundary i, y is perpendicular distance of fluid particle j from solid partcle boundary i, Δp is distance of early particle. Distance of solid particle boundary depend on early fluid particle. If Δp is too small, fluid particle will face rejection force before close to wall boundary. On the other hand, if Δp is too big, numerous fluid particles will permeate boundary wall before face repulsive force [10].

3. Experimental Setup
The SPH simulation is build by using programming language C/C++. Gnuplot is also used for visualization of SPH particles. Figure 4 shows the dam-break simulation by using SPH method which is implemented by C/C++ program. Overall, the area of the simulation is established by 2×1 meters. In the plot of the particle position, physical properties such as velocity, density, pressure, and mass are always changing in each time.

There are two types of SPH particles, the first of which is fluid particle which contain physical properties, and solid particle that each particle has no velocity value. Fluid particles are formed into water to be streamed into waterfalls, whereas solid particles are formed into the system's boundary walls. All parameters for the fluid and solid SPH particles are summarized in Table 1 and Table 2, respectively. The initial set up of waterfall simulation using SPH method is depicted by Figure 5. After establishing system, the simulation is run by using Leapfrog time-stepping. The final time of this simulation is 2.2 seconds.

7

Table 1. Parameter of fluid particle.

Parameter	symbol	value	unit
Gravitational acceleration	g	9.8	m/s^2
Initial density of fluid	ρ_0	1000	kg/m^3
Diameter of a particle	-	0.00625	m
Number of fluid particles	-	10216	-

Table 2. Parameters of solid particle.

Parameter	symbol	value	unit
Gravitational acceleration	g	9.8	m/s^2
Density of solid wall	ρ	1200	kg/m^3
Diameter of a particle	-	0.00625	m
Number of solid particles	-	2563	-
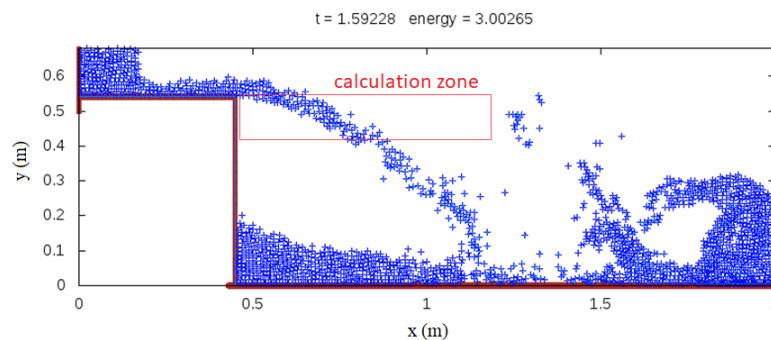
Figure 6. Waterfall simulation by using SPH method with waterfall height of 0.55 meters.

4. Results and Discussion

There are three scenarios in simulating the waterfall flow by using SPH method. The three scenarios are executed by varying the height of waterfall above ground. In each scenario, the mechanical energy resulted will be calculated and analyzed.

4.1. Waterfall height of 0.55 meters

Figure 6 shows the waterfall simulation by using SPH method with waterfall height of 0.55 meters. By this height, the amount of mechanical energy is calculated inside the calculated zone (x-axis from 0.46 to 1.2 meters and y-axis from 0.45 to 0.55 meters). Mechanical energy average in each time inside calculation zone with waterfall height of 0.55 meters is displayed by Figure 7. The figure reveals the amount of mechanical energy in Joule against time. From the figure, it can be seen that the average of mechanical energy for whole time is 23.83 Joule.
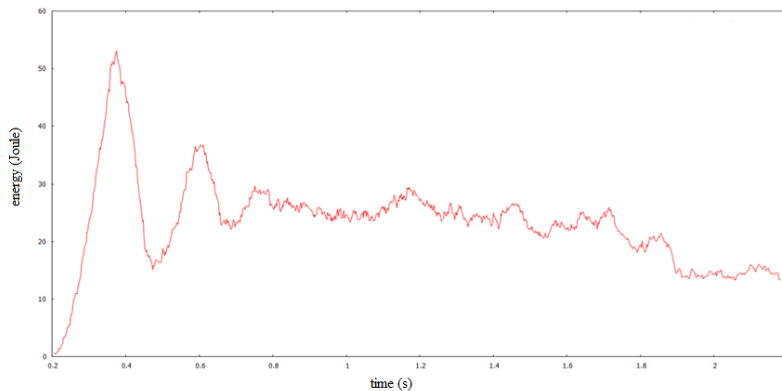
Figure 7. Mechanical energy average (in Joule) in each time inside calculation zone with waterfall height of 0.55 meters.
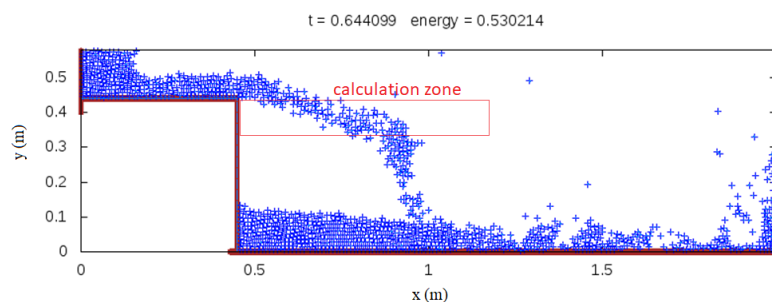

Figure 8. Waterfall simulation by using SPH method with waterfall height of 0.45 meters.

4.2. Waterfall height of 0.45 meters

The waterfall flow simulation using SPH method with height of 0.45 meters is presented in Figure 8. The calculation zone for this scenario is x-axis from 0.46 to 1.2 meters and y-axis from 0.35 to 0.45 meters. Here, the average of mechanical energy is calculated inside the calculated zone. Figure 9 shows mechanical energy average in each time inside calculation zone with waterfall height of 0.45 meters. The figure reveals the amount of mechanical energy in Joule against time. For this scenario, it can be seen that the average of mechanical energy for whole time is 21.17 Joule.

4.3. Waterfall height of 0.35 meters

The calculation zone for third scenario is x-axis from 0.46 to 1.2 meters and y-axis from 0.25 to 0.35 meters. Here, the average of mechanical energy is calculated inside the calculated zone. The waterfall flow simulation using SPH method with height of 0.35 meters is presented in Figure 10. The mechanical energy average in each time inside calculation zone with waterfall height of 0.35 meters is shown by Figure 11. The figure reveals the amount of mechanical energy in Joule against time. It can be seen that the average of mechanical energy for whole time is 18.95 Joule.

4.4. Influence of waterfall height to mechanical energy

The mechanical energy average in each time inside calculation zone with waterfall height of 0.55, 0.45, and 0.35 meters is displayed in Figure 12. It can be seen that the higher the waterfall position, the greater the mechanical energy. Moreover, the average of mechanical energy for
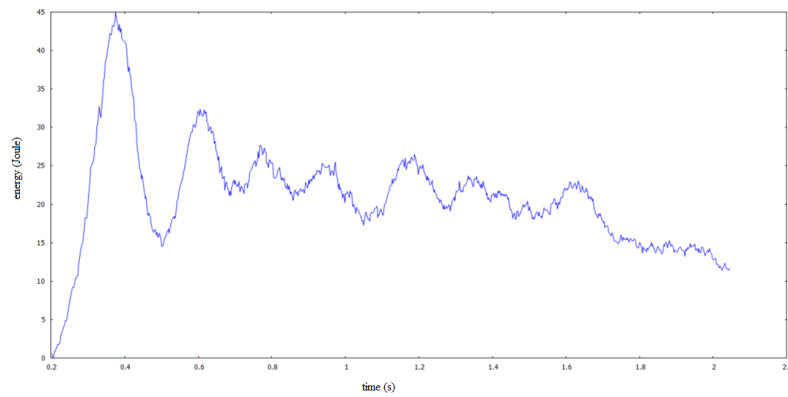
Figure 9. Mechanical energy average (in Joule) in each time inside calculation zone with waterfall height of 0.45 meters.
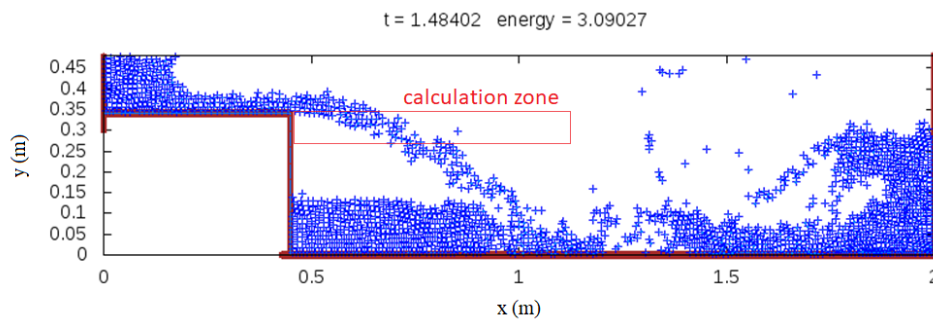

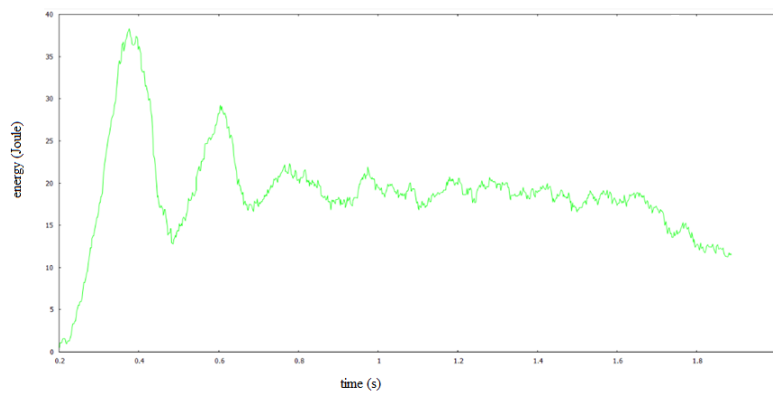**Figure 10.** Waterfall simulation by using SPH method with waterfall height of 0.35 meters.

Figure 11. Mechanical energy average (in Joule) in each time inside calculation zone with waterfall height of 0.35 meters.
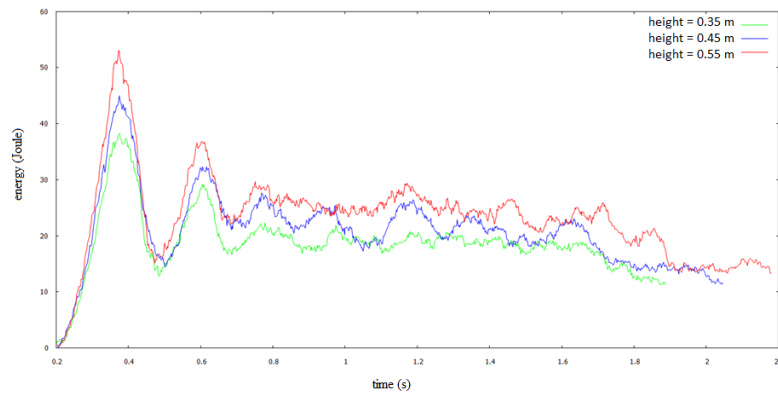
Figure 12. Mechanical energy average (in Joule) in each time inside calculation zone with waterfall height of 0.55, 0.45, and 0.35 meters.
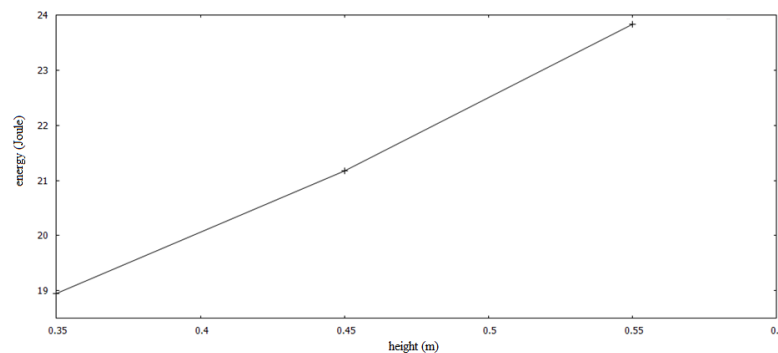

Figure 13. Average of mechanical energy for whole time against waterfall height.

whole time against waterfall height is shown by Figure 13. In the simulation, the waterfall with height of 0.55 meters has mechanical energy of 23.83 Joule. This amount of energy can result 11.9 Watt which is capable of turning on an LED floodlight with a voltage of 220 Volts. Moreover, the numerical results show that by rising the waterfall hight by 0.1 m, the mechanical energy produced is increasing by 2.4 Joule. Therefore, it can be interpreted that if there are two waterfalls with a difference of 0.1 meters of height may result 2.4 Joule range of mechanical energy generated.

5. Conclusion

From the numerical results, it has been shown that the mechanical energy of waterfall is highly influenced by its height. In this case, the higher the waterfall, the greater the mechanical energy. By the height of 0.55 meters, the waterfall generates mechanical energy of 23.82 Joule while the height is 0.45 meters, its mechanical energy is 21.17 Joule. Moreover, the waterfall fliw produces 18.95 Joule. It can be concluded that by increasing the waterfall height of 0.1 meters, the amount of mechanical energy is rising by 2.4 Joule.

References

[1] Uchiyama T, Fukuhara H, Iio S and Ikeda T 2013 *International Journal of Rotating Machinery* **2013**
[2] Kellomäki T 2015 *Tampereen teknillinen yliopisto. Julkaisu-Tampere University of Technology. Publication; 1354*

[3] Onderik J 2011 *Animation of Particle Based Miscible and Immiscible Fluids with Multiple Interfaces* Ph.D. thesis Comenius University
[4] Macia F, Souto-Iglesias A, Gonzalez L and Cercos-Pitam J 2013 Mps = sph *8th International Smoothed Particle Hydrodynamics European Research Interest Community (SPHERIC) Workshop*
[5] Berata W 2003 *Jurnal Teknik Mesin* **3** 46–54
[6] Michael Owen J 2014 *International Journal for Numerical Methods in Fluids* **75** 749–774
[7] Kipfer P and Westermann R 2006 Realistic and interactive simulation of rivers *Proceedings of Graphics Interface 2006* (Canadian Information Processing Society) pp 41–48
[8] Krog T 2003 *Visualization of Large-Scale Fluid* Master's thesis Technical University of Denmark, DTU, DK-2800 Kgs. Lyngby, Denmark
[9] Liu M and Liu G 2010 *Archives of computational methods in engineering* **17** 25–76
[10] Tarwidi D 2012 The smoothed particle hydrodynamics method for two-dimensional stefan problem *Proceedings of The 5th International Symposium on Computational Sciences (ISCS) 2012*
[11] Monaghan J J 1992 *Annual review of astronomy and astrophysics* **30** 543–574
[12] Monaghan J 1989 *Journal of Computational physics* **82** 1–15
[13] Omelyan I, Mryglod I and Folk R 2002 *Computer Physics Communications* **146** 188–202
[14] Young P 2014 *Lecture notes in University of california, santa cruz*