

Investigation of Flow Dynamics in a Flume with Connected and Spaced Levee–Moat System Using CFD

Abdur Rauf Khan^{1*}, Naveed Anjum¹, Talha Jami¹, Abdul Rafay¹, Zaheer Ahmad¹

¹ Khwaja Fareed University of Engineering & Information Technology (KFUEIT), Rahim Yar Khan, Pakistan
enr.raufkhann@gmail.com; naveed.anjum@kfueit.edu.pk; enr.talhajamiii@gmail.com;
cengr.abdulrafay@gmail.com; dr.zaheer@kfueit.edu.pk

* Correspondence: enr.raufkhann@gmail.com

Abstract

Applying efficient energy dissipation and flow control systems has been emphasized by the rising occurrence of coastal flooding and river floods. This study numerically examines the hydrodynamic performance of levee-moat systems (in ANSYS Fluent) with respect to the use of computational fluid dynamics (CFD). Two model cases were analyzed: a directly connected levee-moat (DC_{LM}), and spaced apart levee-moat (SA_{LM}). The study has used the volume-of-fluid method and RNG- $k-\epsilon$ type of turbulence model to calculate the interaction at the free surface. These values demonstrated a range of velocity variation between 0m/s at the levees and moat base to 1.4 m/s on the levee slope in DC_{LM} and 1.63 m/s in SA_{LM} in the free-flow regions with strong velocity gradient and recirculation zones. The energy loss obtained was 19% for DC_{LM} and 17% for SA_{LM}, showing greater energy dissipation in the direct connection as a result of stronger turbulence due to strong hydraulic jump. The distance between the moat and the levee leads to more chances of scouring and also increases the chances of structural damage of the moat. These findings underscore the significant role of structural geometry and spacing on hydraulic energy control, which can be of useful information in the design of flood and tsunami mitigation measures.

Keywords: Levee, Moat, Open Channel Flow, Computational Fluid Dynamics (CFD), Flow Structure

1. Introduction

The flow nature of levee-moat systems has been explored before, and results were reported that the inclusion of a moat may serve as a cushion of water, helping to reduce energy of the flowing expanse around the toe of the levee [1]. Embankment-forest Hybrid mitigation system of tsunamis with an offshore embankment and a landward horizontally double layer forest is effective in reducing forces caused by the tsunami [2]. In a similar vein, optimized the coastal forest cover through three-dimensional modeling as well as showcased the role of vegetation structure in controlling tsunamis and dissipating their energy [3]. Vegetation on mounds behindhand coastal dikes can decrease flow energy, justifying the impact of flood waves [4]. To lower the risk of flooding, levees, embankments, and floodwalls are commonly used as structural countermeasures [5, 6]. However, poorly designed coastal structures, instability, or even failure during extreme hydrodynamic events [7]. Recent research has emphasized erosion dynamics, vegetation placement, and flow transitions to improve levee resilience [8, 9]. Hybrid tsunami resistance

systems comprising embankments, moats, and coastal forests have been exposed to differ in their mitigation effectiveness depending on whether the structures are submerged, emergent, or in combined conditions [10]. Flume experiments have shown that the optimal arrangement of hybrid tsunami defense systems comprising embankments, moats, and emergent vegetation and the height of tree crowns in coastal forests both significantly influence flow hydrodynamics and help reduce tsunami energy [11,12]. Studies have demonstrated that tsunami-borne washed-out trees can act as natural dams to reduce local scouring and tsunami energy behind coastal embankments, and that the height of tree crowns plays a crucial role in the mitigation effectiveness of inland coastal forests by influencing flow structures and scour phenomena [13,14]. The baffles, drop structures, and moats are energy dissipation devices that lead to turbulence, that slows the flow velocity and the hydraulic energy [15]. Natural obstacles and vegetation play a significant role in the transformation of turbulence structure as well as flow energy dissipation [16, 17]. Based on these studies, the current paper also analyzes the hydrodynamic behavior of levee moat system under the framework of computational fluid dynamics (CFD) model. These results underscore the importance of incorporating structural and natural defenses, including levees, moats, and coastal vegetation, to enhance the reductions of flood and tsunami energy.

2. Materials and Methods

2.1 Hydraulic Conditions

A 1/100 scale was used in this study to observed the hydraulic behavior of flow within a levee-moat system. The analysis was conducted assuming critical flow conditions with a Froude number of 1, indicating equilibrium between flow velocity and water depth. The levee was set at 0.1 m in height, and all the geometry and hydraulic parameters of the model were scaled. Airspace directly above the water level in the flume was modeled clearly and the Volume-of-Fluid (VOF) method was utilized in order to model the interaction of the air-water interface in the free surface. To remove the effects of slope of the bed and concentrate on the purely geometrical effects of the levee-moat structure a horizontal bed was held constant throughout the model. The experimental design allowed targeted study of the flow patterns of near-critical hydraulic conditions.

2.2 Numerical Model Details

Fluent ANSYS was used to perform numerical simulations of the flow of water over a levee-moat system and its surrounding under near-critical conditions. The height of the levee was established to be 0.1 m and this acted as a starting point in scaling the whole model. The computational flume was 4.0 m, 0.5 m, and 0.25 m long, wide and tall respectively. The levee was made with a side slope of 1:2 or in other words twice as long horizontally as it was high vertically. Two setup arrangements were experimented in order to observe the effect of the placement between the levee and moat on flow behavior. The initial one is known as DC_{LM} (directly connected levee-moat) where the levee and moat are attached together with no space in-between them. The second arrangement, which is referred to as SA_{LM} (spaced apart levee-moat), included a distance between the levee and the moat, and the distance was calculated proportionate on the length of the moat. Both designs are shown in [Fig. 1(a, b)]

TABLE: 1

Case	Fr	H _L (m)	Z _c *=Z _c /H _L	L _o *=L _o /H _L	D _o *=D _o /H _L
DC _{LM}	1	0.1	0.29	3.0	0.7
SA _{LM}	1	0.1	0.29	3.0	0.7

Moat length (L_o*), moat depth (D_o*) and overtopping depth (Z_c*) were considered dimensionless parameters with values of 3.0, 0.7 and 0.29 respectively as Table: 1 displays. On the contrary, the rest of the parameters including levee height (H_L), the size of the flume, etc., were maintained in the actual dimensions. This method was adopted to guarantee that the outcomes were realistic when it comes to real physical conditions.

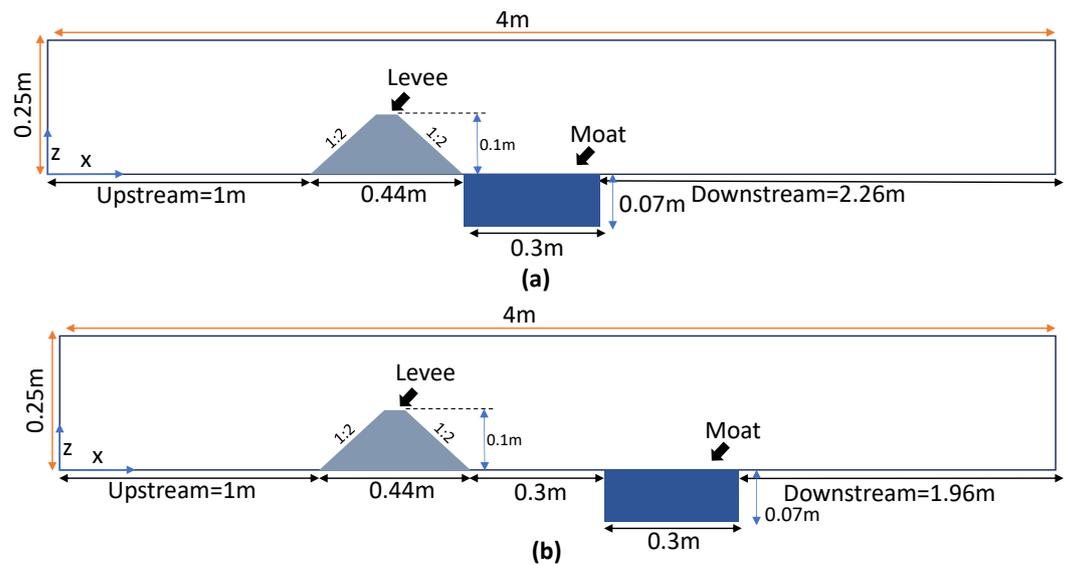


Fig. 1 Sketches of CFD Models

2.3 Numerical Model Setup

An organized computation mesh was designed to include the whole flume domain that constituted about 0.2 million nodes. The choice of this mesh density was determined through a careful consideration of the fact that the resolution is sufficient, yet at the same time, the computational cost is sufficient and the results are stable and valid. ANSYS Fluent was used to construct the numerical model that takes the intrigue flow dynamics of the levee and moat setups. The flume geometry and the imposed boundary conditions are logged out in Figure 2. In order to model the two-phase flow system, the Volume of Fluid (VOF) technique is used, which is an effective approach to modeling the interaction of the two phases (air and water) and enables an easy way to track the evolution of the free surface. Characteristic of the upstream end, there was a velocity inlet boundary condition to regulate the incoming flow. The outlet and top boundaries were defined to act as pressure outlets whereby the model can simulate the conditions of an open channel flow with a free surface that is open to atmospheric pressure. In order to include frictional resistance, no-slip wall boundary conditions were applied to all the solid surfaces such as the bottom of the flume, the sidewalls, and the levee and moat surfaces. The simulations have been done with steady-state flow assumptions and second-order discretization scheme of the governing equations was used. The scheme is a higher-order that greatly minimizes the numerical diffusion and maximum accuracy in the solution. In general, this detailed modeling system

was capable of making reliable and physically credible forecasts of flow behavior in the levee-moat system.

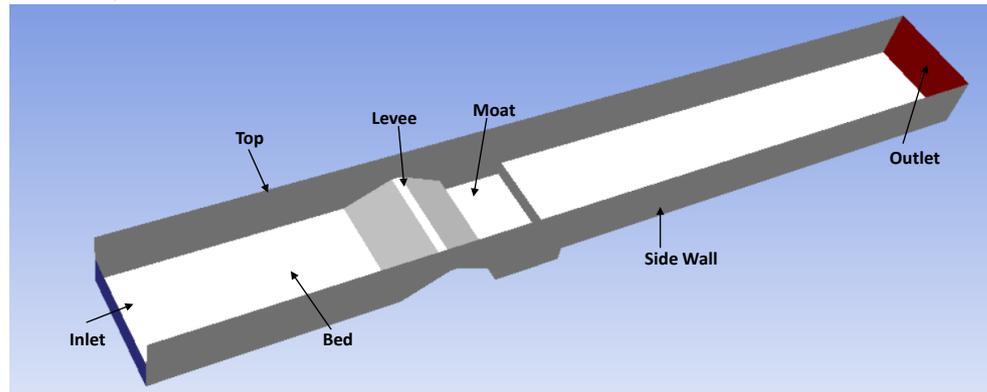
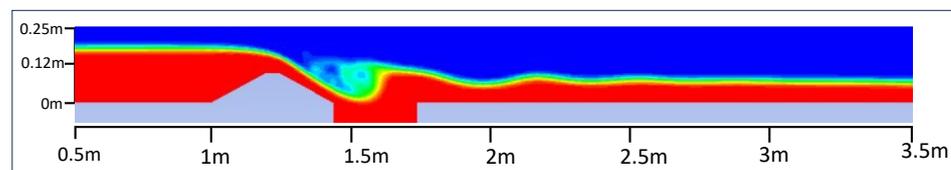


Fig.2 Boundary Conditions

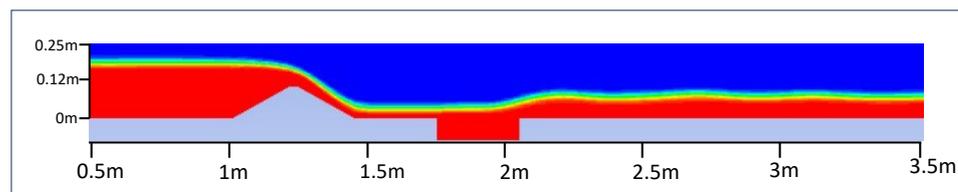
3. Results

3.1 Streamwise Depth-Averaged Velocity

Streamwise depth-averaged velocity distribution was observed in the two configurations. In DC_{LM} , the flow was comparatively steady as it neared the levee. When the stream crossed the levee crest and entered the moat in a special location below the levee (H_L) the velocity was observed to increase significantly. This acceleration was however followed by a slow stabilization with the downstream direction owing to the creation of a hydraulic jump. This implies efficient energy dissipation, which seems to be caused by the unceasing structural relationship of the levee and the moat. In inequality, the SA_{LM} arrangement displayed varying flow features because there was an opening between the levee and moat. This spacing resulted in faster velocity change, local flow disturbances and turbulence in the inter-mediate zone. Eventually, the flow regained stability as it progressed toward the outlet. These distinct velocity patterns for both configurations are illustrated in [Fig. 3(a, b)].



(a)



(b)

Fig. 3 Contour Plot of Volume Fraction of water Distribution on xz plane

The comparison of depth averaged velocity trend for both configurations is shown in Fig. (4), highlighting the differences in flow adjustment along the channel length.

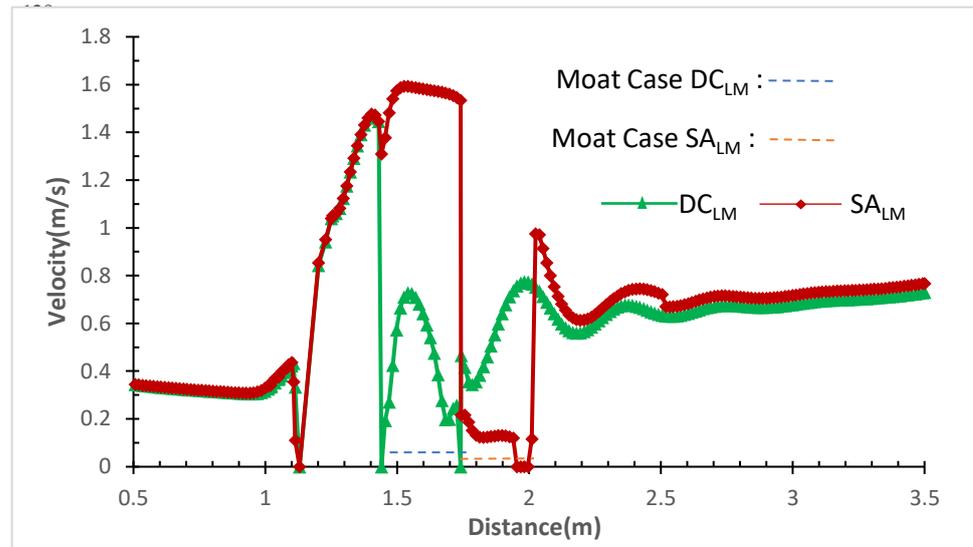


Fig. 4 Profile of Depth-averaged velocity

3.2. Energy Reduction Analysis in Levee–Moat System

Several observations were made by examining how energy is distributed within the levee-moat system for both configurations. The energy loss was determined using the following relationship

$$\Delta E = \frac{E_1 - E_2}{E_1} \times 100$$

Where E_1 and E_2 represent the total energy heads at the upstream and downstream sections, correspondingly. Energy head was calculated by $z + v^2/2g$ equation where z is depth of water, v is depth averaged velocity and g is gravitational acceleration. For E_1 depth and velocity were determined at the levee crest through the continuity equation, while E_2 both parameters were calculated just behind the moat where the flow stabilized in both configurations. In DC_{LM} , the system achieved an energy loss of 19%, which indicates a significant dissipation of flow energy due to the interaction of the incoming flow with the levee and moat configuration and in SA_{LM} , the energy loss was relatively low at 17%, indicating that there was less energy loss as the flow passed through the levee. Overall, the comparison between the both configurations, as shown in Fig. 4, shows that the first configuration provides higher energy dissipation and better flow stability, while the second case shows smoother flow passage but a higher risk of structural erosion.

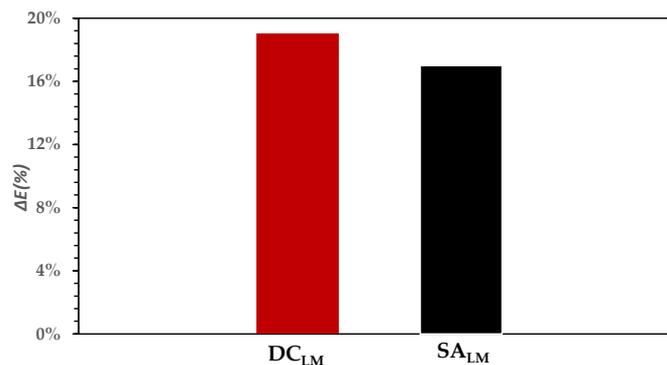


Fig. 5 Comparison of Energy Dissipation

3.3. Contour Plot Velocity Distribution

The streamwise mean velocity (U) distribution of the two configurations was studied. The lowest velocities of approximately 0 m/s in the DC_{LM} were observed almost at the bottom of the levee on upstream side and in the moat where the flow separation and reverse circulation were present. The maximum velocity of around 1.4 m/s was recorded towards the top of the water body when the flow reached the crest of the levee and into its slope. The SA_{LM} had a more or less the same general tendency, though it had several significant differences. Maximum velocity also rose to approximately 1.63 m/s on the inland gradient of levee downstream slope and spacing between the levee and moat, which is the acceleration caused by difference between the levee and moat. The lowest velocity was similar to that in DC_{LM} and was situated within the recirculation region within the moat. The contours of the velocity of the DC_{LM} and SA_{LM} configurations are illustrated in [Fig. 6(a, b)].

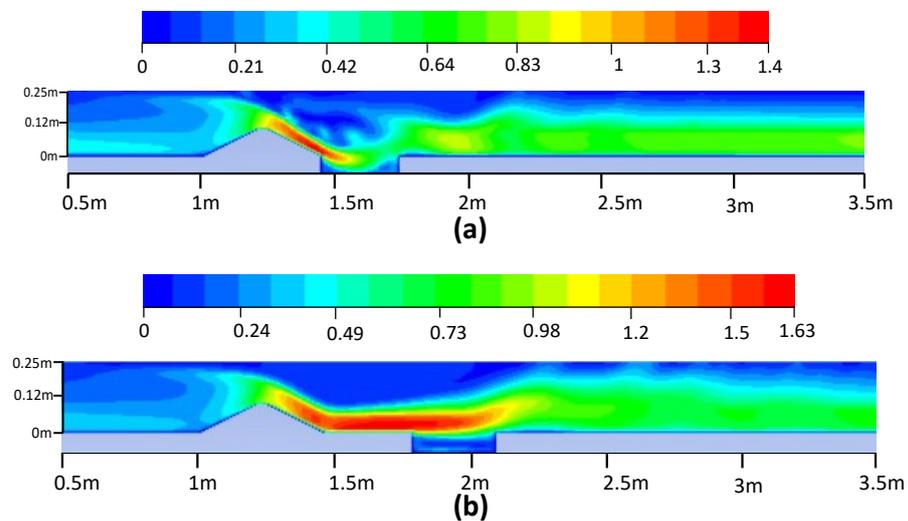


Fig. 6 Velocity contours of mean velocity(U), (xz plane)

4. Discussion

The increasing energy release in DC_{LM} is explained by the fact that the strong hydraulic jump is formed and enhances turbulence and establishes strong recirculation areas in the flow field. This constant communication between the moat and the levee will result in a better energy loss. With disparity, SA_{LM} has another behavior. This is made possible by the difference between the levee and the moat to ensure that the flow partially reconnects and gains momentum before entering the moat. Consequently, the flow will have higher velocities in its downstream and will release less energy in general. Although such flow recovery might appear beneficial in terms of hydraulic efficiency, there are concerns as well. This reiterates the significance of spacing in regulating the flow behavior as well as system safety.

5. Conclusions

This study employed simulations by modelling the ANSYS Fluent in order to investigate flow behaviour and energy dissipation in levee-moat systems with the two setups (diametrically connected (DC_{LM}) and separated (SA_{LM}). The findings indicated that the distance between objects has a great influence on velocity distribution and energy loss. DC_{LM}

configuration developed more intense turbulence and recirculation with 19 percent of energy dissipation compared to SA_{LM} which could recover flow between structures and lost 17 percent of energy but had higher downstream velocities. These results indicate that the spacing between levees and moats is critical in regulating hydraulic energy and flow stability which can be useful in developing effective flood and tsunami mitigation infrastructures.

Funding: This research received no external funding.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

DC _{LM}	Directly connected levee-moat
SA _{LM}	Spaced apart levee-moat
H _L	Levee height
Fr	Froude number
Z _c *	Non-dimensional overtopping depth
L _o *	Non-dimensional length of moat
D _o *	Non-dimensional depth of moat

References

- Abbas, F.; Tanaka, N. Investigation of flow structure with moat acting as a water cushion at the toe of an overflowing levee. *Environ. Fluid Mech.* 2022, 22, 1–25.
- Rashedunnabi, A.H.M.; Tanaka, N.; Anjum, N. Effectiveness of a hybrid tsunami mitigation system consisting a seaward embankment and a landward horizontally double-layer forest. *Jpn. Soc. Civ. Eng. Ser. B1 (Hydraul. Eng.)* 2019, 75, I_715–I_720.
- Anjum, N.; Iqbal, S.; Pasha, G.; Tanaka, N.; Ghani, U. Optimizing coastal forest arrangements for tsunami flow dynamics using a three-dimensional approach. *Phys. Fluids* 2025, 37, Article ID not assigned.
- Rashedunnabi, A.H.M.; Tanaka, N.; Rahman, M.A. Flume experiment on flow transition and water cushion formation by optimal vegetation on a mound behind a coastal dike and its impact on reducing the flow energy. *Geosciences* 2025, 15, 243.
- Murtaza, N.; Pasha, G.; Tanaka, N.; Hamidifar, H.; Ghani, U.; Ahmed, A. Optimal configuration of a hybrid defense system for hill torrents: An experimental investigation for effective management. *Nat. Hazards* 2025, 121, 13029–13066.
- Rashedunnabi, A.H.M.; Tanaka, N.; Rahman, M.A. Transformations in flow characteristics and fluid force reduction with respect to the vegetation type and its installation position downstream of an embankment. *Fluids* 2025, 10, 16.
- Dissanayaka, C.; Tanaka, N. Fluid force reduction and flow structure at a coastal building with different outer frame openings following primary defensive alternatives: An experiment-based review. *Geosciences* 2024, 14, 287.
- Ali, L.; Tanaka, N. Enhancing levee resilience through material compatibility: A comprehensive study on erosion dynamics. *Iran. J. Sci. Technol. Trans. Civ. Eng.* 2024, —, 286–301.
- Murtaza, N.; Pasha, G.; Tanaka, N.; Ghani, U.; Anjum, N.; Iqbal, K. Analysis of hydraulic jump and energy dissipation in flow through emergent vegetation under varying Froude numbers. *Iran. J. Sci. Technol. Trans. Civ. Eng.* 2024, 49, page range not specified.
- Kimiwada, Y.; Tanaka, N.; Zaha, T. Differences in effectiveness of a hybrid tsunami defense system comprising an embankment, moat, and forest in submerged, emergent, or combined conditions. *Ocean Eng.* 2020, 208, 107457.
- Zaha, T.; Tanaka, N.; Kimiwada, Y. Flume experiments on optimal arrangement of hybrid defense system comprising an embankment, moat, and emergent vegetation to mitigate inundating tsunami current. *Ocean Eng.* 2019, 173, 45–57.
- Anjum, N.; Tanaka, N. Effects of tree crown height on the flow hydrodynamics and tsunami energy reduction around the coastal forest. *Landsc. Ecol. Eng.* 2024, 20.
- Rahman, M.A.; Tanaka, N.; Anjum, N. Damming effects of tsunami-borne washed-out trees in reducing local scouring and tsunami energy behind a coastal embankment. *Appl. Ocean Res.* 2022, 126, 103260.
- Anjum, N.; Tanaka, N.; Rahman, M.A. Role of tree crown height for effective mitigation capability of inland coastal forest considering the flow structures and scour phenomena. *Ocean Eng.* 2021, 238, 109728.

-
15. Matsuba, S.; Mikami, T.; Jayaratne, R.; Shibayama, T.; Esteban, M. Analysis of tsunami behavior and the effect of coastal forest in reducing tsunami force around the coastal dikes. *Coast. Eng. Proc.* 2014, 1, 37. 244
245
 16. Esteban, M. Stability of breakwaters against tsunami attack. In *Handbook of Coastal Disaster Mitigation for Engineers and Planners*; Shibayama, T., Esteban, M., Eds.; Elsevier: Oxford, UK, 2013; pp. 293–322. 246
247
 17. Anjum, N.; Tanaka, N. Experimental study on flow analysis and energy loss around discontinued vertically layered vegetation. *Environ. Fluid Mech.* 2020, 20, page range not specified. 248
249