

# NUMERICAL STUDY OF FLOW CHARACTERISTICS INSIDE SERRATED DODGER CHANNELS USING THE OPENFOAM PROGRAM

**Hafidh Farisi, Umboro Lasminto, Satria Damar Negara**

*Department of Civil Engineering, Institute Teknologi Sepuluh Nopember, Surabaya*

*[hafidh.farisi@waskita.co.id](mailto:hafidh.farisi@waskita.co.id)*

## ABSTRACT

The dodger building is one part of the dam component that serves to divert the flow of the river during construction work. One of the requirements for evasive buildings is to be able to drain river water well, so that the characteristics of the flow in the dodger channel need to be known to see how the flow conditions that occur in the dodge channel. This study aims to examine the flow characteristics of the Tapin Dam circumvention channel due to the design of the channel tunnel using a tooth shape, by modeling the relationship between the flow characteristics (velocity profile, pressure drop profile, and turbulent kinetic energy profile) of the jagged channel dimension shape using numerical computational analysis with the help of CFD (Computational Fluid Dynamics) software. From the modeling results, it will be known the influence of channel dimensions on flow characteristics in the dodger channel.

**Keywords:** *background, aim, method, results*

This article is licensed under [CC BY-SA 4.0](https://creativecommons.org/licenses/by-sa/4.0/) 

## INTRODUCTION

The dodger building is one part of the dam component that serves to divert the flow of the river during construction work (Zheng & Kahn, 2013). In essence, circumvention channels have a very important role, especially in urugan dams, considering the weakness of this type of dam to the flows or runoff of river water above the dam landmark. Therefore, planning a dodge channel with adequate capacity and with safe construction is the most important requirement for the safety of the implementation of urugan dam construction. Inadequate circumvention channels, in addition to threatening dams under construction, can also threaten areas located downstream (Fan et al., 2020).

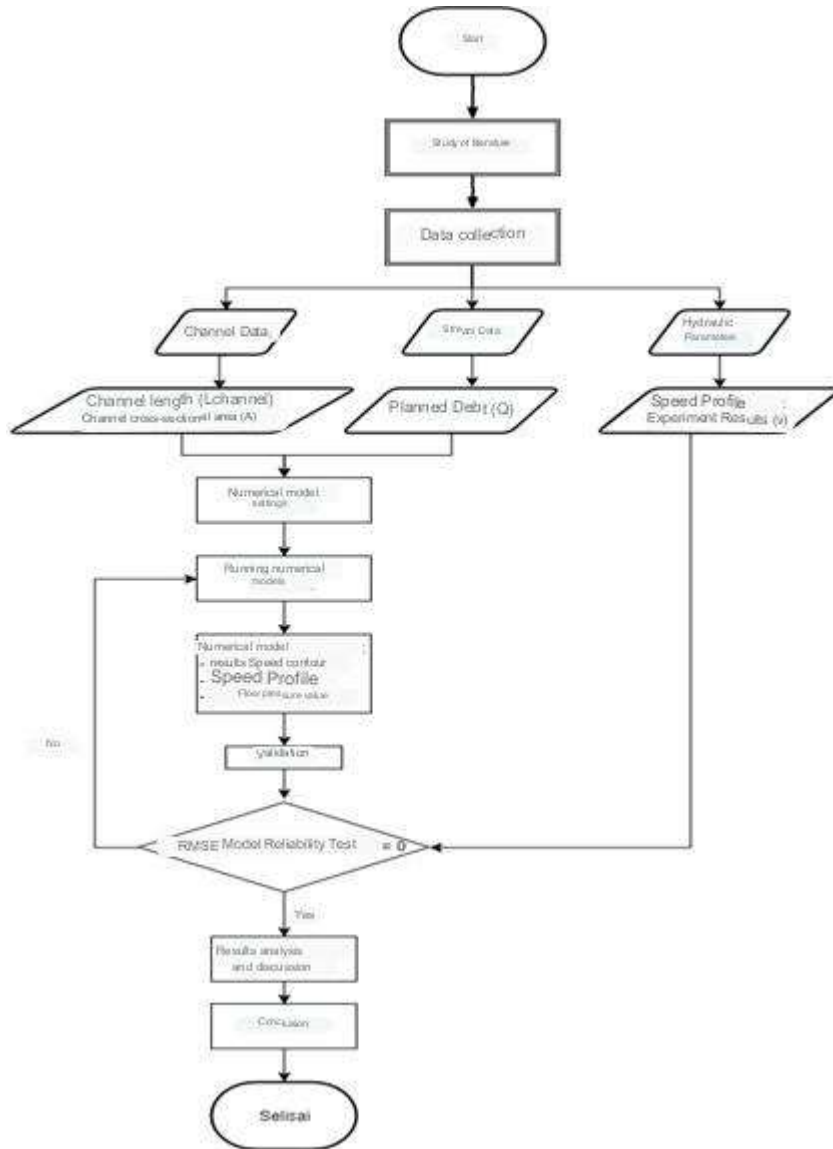
Currently, many studies are being conducted to observe the characteristics of flow in evasive channels/tunnels, Previous studies that have examined the characteristics of evasive channels/tunnels include Z J Baoet et al (2020) conducted research by modeling physical and numerical flow in dodge channels with circular cross-sectional dimensions of 6.7 meters in diameter and 150 meters in length, which are influenced by multilevel intake doors and water reservoirs. Çimen Karaduman et al (2018) conducted a study of hydraulic characteristics on the tunnel from the Ilisu Dam, a tunnel designed with 2 tunnels connected by galleries with a cross-section of horseshoes. Jintang, Zhang & Huang (, 2018) conducted research on the hydraulics test model of the evasive channel at the Gongguoqiao Hydroelectric Power Plant (PLTA). Weilin Xu (2020) conducted numerical simulation research on the hydrodynamic characteristics of free-flow hydraulic tunnels in spillway tunnels, by studying the volume and velocity functions of two-phase water-air flow water in free-flow long-distance diversion tunnels. Boes, Robert M. (2017) examined flow hydraulics, bubbles and sediment flows in upstream sediment bypass tunnel (SBTs) control buildings.

Today, numerical modeling is preferred as an alternative to solving research problems compared to experimental modeling in the laboratory (Akbari & Pooyarad, 2020). Because numerical modeling has several advantages including speed in computing, being able to present more detailed research outputs, and saving costs and time for conducting research. However, numerical models have limitations because they are not able to model the conditions that exist in nature so they require a model approach (Khain et al., 2000). In addition, there are certain parameters that greatly affect the results of the model so it is necessary to calibrate with the results of experimental research that has been carried out previously to produce a level of model correlation that is close to perfect. The CFD (Computational Fluid Dynamic) model that will be used is the OpenFOAM program (Agrafioti et al., 2020). The OpenFOAM program was chosen because it is open-source-based and can be used for free (Izzati et al., 2021; White et al., 2018).

In this study, researchers were interested in conducting research on evasive buildings in the Tapin Dam which was planned with a length of 175 meters where there were straight channels of inlets and outlets along 75 m and at a distance of 75 m from the intake there was a serration-shaped channel along 25 meters. The gear design on the dodge channel has a horseshoe cross-sectional shape with an inlet height of 4.6 m, so that when the flow of water passes through the gear section it will pass through the inlet and then the channel will rise as high as 6.6 m at a distance of 3.5 m, after that it will narrow again with the condition of a channel height of 4.5 m at a distance of 5 m, and so on up to 5 parts along a distance of 25 meters.

This study aims to compare the performance of numerical models against experimental models based on flow velocity profiles and qualitatively aims to analyze flow behavior through contours and velocity profiles on channels with serration shapes (Wang et al., 2017), and quantitatively aims to analyze coefficient of pressure ( $C_p$ ) graphs caused by channel dimensions with serrated shapes using numerical computational analysis with the help of CFD software.

**METHOD**

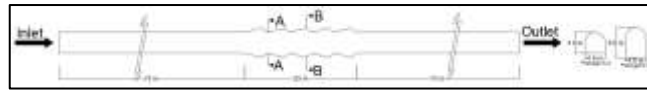


**Figure 1. Flow Chart**

This study used numerical modeling with the OpenFOAM program. OpenFOAM to solve incompressible cases. In this study, a 2D model was used. Spatial directions x and z, component y is not reviewed. The data described in the previous subchapter are used as input. Some things that need to be done at the model setting stage are making spillway geometry, setting physical parameters, making mesh, setting the initial conditions of the model and boundary conditions.

**Data Collection**

The data needed for this study are geometry/dimension data of evasive channel design plan and flow data. The geometry of the dodge channel is taken from the Tapin Dam circumvention channel planning drawing data, as shown by the following figure:



**Figure 2.** Jagged Channel Design

Flow data is the discharge ( $Q$ ) of channel planning to determine the velocity value at the channel inlet. These data are used as input in numerical modeling. Hydraulic parameter data including flow velocity ( $v$ ) from previous experimental studies were used for validation.

### **Numerical Model Settings**

This study used numerical modeling with the OpenFOAM program. OpenFOAM to solve incompressible cases. In this study, a 2D model was used. Spatial directions  $x$  and  $z$ , component  $y$  is not reviewed. The data used as input include channel geometry, flow data including plan discharge ( $Q$ ), and hydraulic parameters including speed profile. Some things that need to be done at the model setting stage are making spillway geometry, setting physical parameters, making mesh, setting the initial conditions of the model and boundary conditions.

### **Running Model**

Running model or solving is done after setting the model or pre-processing. The simulation is carried out until the flow stabilizes. The simulation results in this study are flow velocity ( $v$ ). The simulation results of the model need to be calibrated with previous experimental research to obtain a high level of accuracy.

### **Model Validation and Model Reliability Test**

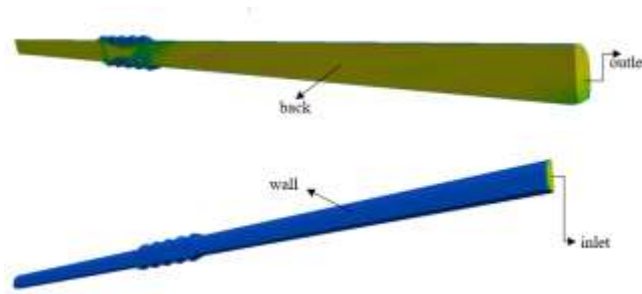
In this study, the validation process used the experimental results of flow through the pipeline research by Jordi Casacuberta Puig, the validated parameter is flow velocity ( $v$ ). Model reliability tests are carried out to determine the level of accuracy of the model (Liski et al., 2005). The equation used for model reliability testing is the Root Mean Square Errors (RMSE) equation. The model is said to be accurate if the RMSE value is close to zero (0), which means that the numerical model results are close to the experimental model (Shahid et al., 2020). If the results of experimental modeling and OpenFOAM numerical modeling produce RMSE values that are not close to that number, it is necessary to re-simulate by changing the mesh or other parameters that affect the model. Simulations are performed until the model reaches a high level of accuracy.

## **RESULTS AND DISCUSSION**

### **Numerical Model Calibration on OpenFOAM**

#### **Sensitivity Analysis**

This process is done to determine the effect of mesh size on the results and to determine the optimal resolution, which is a compromise between the accuracy of the results and the computational time required. The test was carried out by simulating a flow velocity profile graph at the outlet of the straight section repellent channel  $x = 75$  m. The numeric domain configuration is described with boundary conditions as follows:



**Figure 3.** Research Boundary Conditions

In this study, the analysis of the effect of mesh size on numerical results was carried out by studying the effect of using a mesh measuring 0.40 x 0.39 m and a mesh of 0.20 x 0.19 m on the channel speed on a straight channel ( $x = 37$  m) compared to the radius ( $r$ ) of the channel with a laminar simulation model, the results of numerical simulation will be compared with experimental results through model reliability tests using RMSE values.

**Table 1.** Laminar Model RMSE Value Recap

	Simulasi Model Laminar	
Ukuran Mesh	0,40 x 0,39	0,20 x 0,19
Nilai RMSE	0,185	0,063

The RMSE value in the simulation with a mesh of 0.40 x 0.39 m is 0.185, while the RMSE value in the simulation with a mesh of 0.20 x 0.19 m is 0.063. The model is said to be valid or has high accuracy if the RMSE value is close to 0. From the results of RMSE values for both types of mesh, it was concluded that smaller mesh sizes resulted in better performance.

### Computational Process of Research

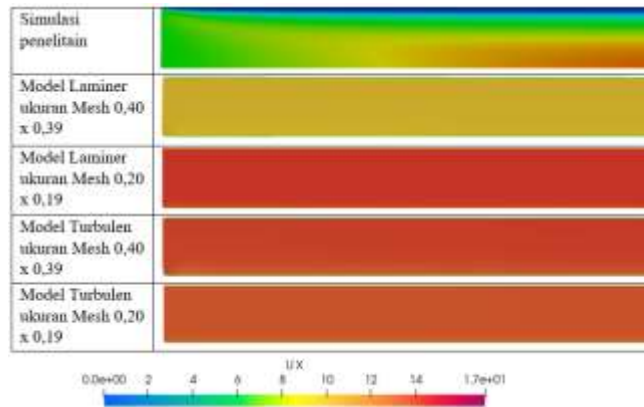
Analysis of evasive channels in the form of serrations will be displayed in the form of qualitative data in the form of flow visualization in the form of contours that occur in flow modeling using turbulent and laminar flow simulations with mesh sizes of 0.49 x 0.38 and 0.24 x 0.19. Then the quantitative data that will be displayed in the form of velocity profile graphs and pressure graphs using turbulent flow simulations of k-epsilon and laminar models with mesh sizes of 0.49 x 0.38 and 0.24 x 0.19.

### Speed Contours

#### *Speed Contours at Straight Channel Inlet*

When the flow enters the inlet area, the flow condition shows that the flow is experiencing a non-developed flow condition because when entering the inlet area the shear stress on the wall still dominates so that the speed in the wall area becomes zero ( $v = 0$ ), and at a certain distance the flow at the center of the channel increases. The increase in flow speed at the center of the channel causes the flow speed on the wall to move towards the center of the straight channel so that at a certain distance (entrance region) the flow completely becomes a developed flow. This is shown in the contour of each Turbulent and Laminar experimental simulation

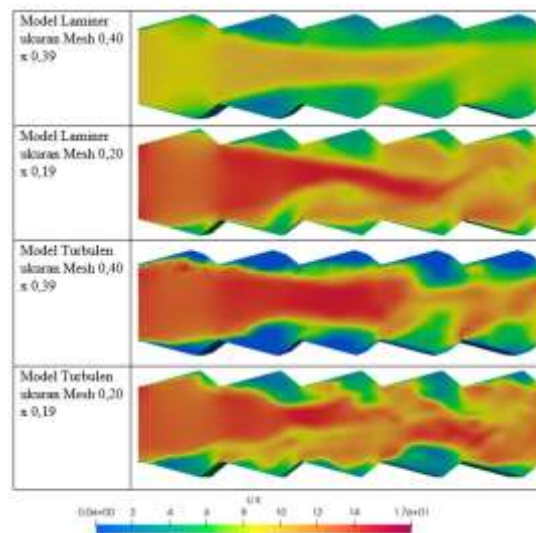
with a mesh size of 0.40 x 0.39 m and 0.20 x 0.19 m at the channel inlet and the results of the study show that the water flow in the inlet area is experiencing undeveloped flow conditions.



**Figure 4.** Inlet Channel Speed Contours

### Speed Contours on Jagged Channels

The speed contour generally shows that changes in the cross-sectional area make the momentum of fluid flow weaker and cause adverse pressure gradient or back pressure so that it can be seen that in the middle area of the channel, the speed is higher than the wall area. As the flow cross-section increases, the fluid velocity will decrease and the fluid pressure will increase when adverse pressure conditions begin to occur. This condition causes the momentum of the fluid can no longer be able to follow the widened flow surface so that at the angle of the channel the wall area becomes a separated region point.



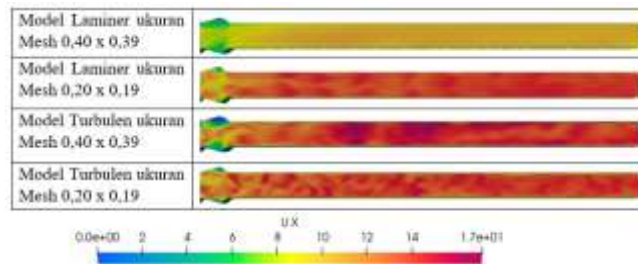
**Figure 5.** Gear Channel Speed Contours

### Speed Contours at Straight Channel Outlets

In general, the speed contour passing through the outlet channel shows a non-uniform speed condition, this indicates that the shape of the gear channel affects the momentum of the flow in the outlet channel, causing turbulence in the flow. In simulations using Laminar models,

non-uniform flow conditions do not occur along the channel, while in simulations using turbulent models, non-uniform flow conditions occur along the channel.

The speed contour shows that the difference in mesh size gives different results as occurs in the speed contour with laminar simulations where the speed at a mesh size of 0.2 x 0.19 gives a higher speed value than the simulation with a mesh size of 0.40 x 0.39. The simulation value of the Laminar model size 0.2 x 0.19 has similarities with turbulent simulation.



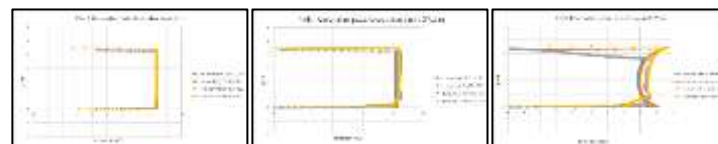
**Figure 6.** Outlet Channel Speed Contours

### Speed Profile

In this section, an analysis of the speed profile in the cross-section channel will be carried out for turbulent and laminar conditions with a variation in Mesh size of 0.40 x 0.38 and 0.20 x 0.19. The cross-section to be reviewed will be divided into several segments, namely the inlet straight channel, the jagged channel, and the outlet straight channel.

#### Inlet Straight Channel

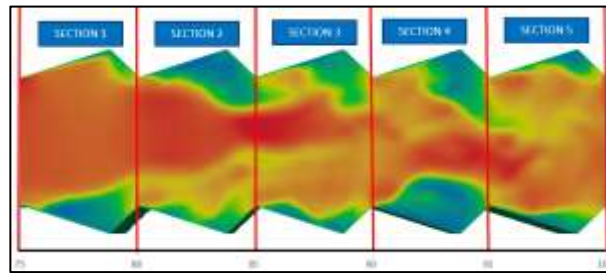
When the flow enters the inlet area, the shear stress in the wall area is relatively small so that the speed in the wall area of the channel  $v \neq 0$  m / s. After passing through the inlet of the channel the shear stress in the dinging area gradually increases along the straight line and at a certain distance the flow in the center of the channel increases. The increase in flow speed at the center of the channel causes the flow speed on the wall to move towards the center of the straight channel so that at a certain distance (entrance region) the flow completely becomes a developed flow.



**Figure 7.** Boundary Condition on Straight Channel

### Speed Profile on Jagged Channels

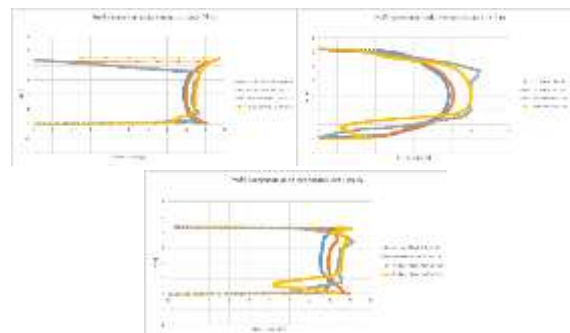
In the analysis of serrated channels will be divided into several segments / sections.



**Figure 8.** Jagged Channel Segment Division

#### *Section 1*

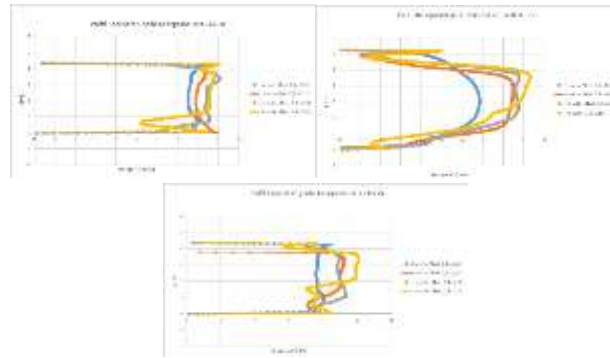
In the first segment when the channel enters the channel inlet area  $x = 75$  m distance, the flow velocity profile forms a boundary condition where the speed in the wall area is greater than the middle area of the cross-section of the channel cross-section, When entering the channel cross-sectional widening area the speed of the flow decreases with the condition that the flow profile in the channel wall area decreases to form a boundary condition (distance  $x = 78.7$  m) then when the cross-section experiences narrowing, the flow speed increases with an increase in speed back in the channel wall area to form a boundary condition (distance  $x = 80$  m).



**Figure 9.** Boundary Condition on Gear Channel (a) distance 75 m, (b) distance 78.7 m, (c) distance 80 m

#### *Section 2*

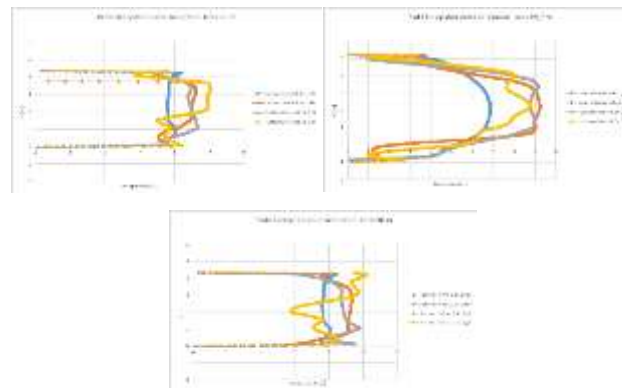
In the second segment, the flow profile gradually forms a boundary condition where the flow velocity begins to develop in the middle both in the widening area (distance  $x = 83.7$ m) and entering the narrowing area (distance  $x = 85$  m). At the location of the wall area, the momentum of the flow has not fully developed because the flow has not been able to resist adverse pressure perfectly.



**Figure 10.** Boundary Condition on the Gear Channel (a) distance 80 m, (b) distance 83.7 m, (c) distance 85 m

**Section 3**

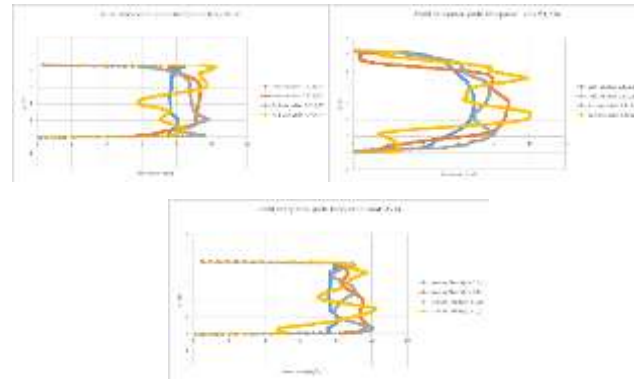
In the third segment, the flow speed in the middle channel (distance  $x = 88.7$  m) increases in speed and in the wall area the speed decreases), but in the narrowing area the flow speed profile looks not uniform (distance  $x = 90$  m).



**Figure 11.** Boundary Condition on the Gear Channel (a) distance 85 m, (b) distance 88.7 m, (c) distance 90 m

**Section 4**

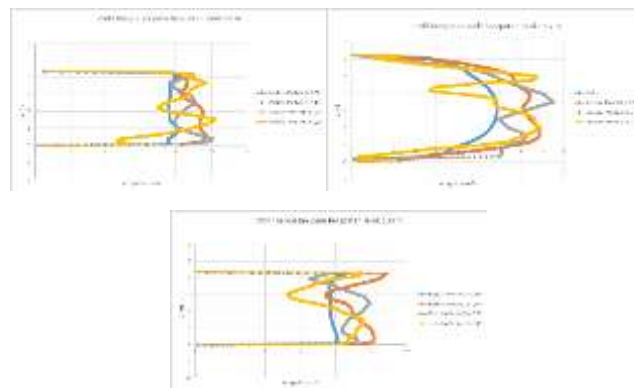
In the fourth segment, where the flow profile conditions are not uniform, the flow boundary layer along the section (distance  $x = 93.7$  m and  $x = 95$  m) fluctuates, indicating that in the fourth segment the flow velocity cannot maintain flow momentum so that the boundary conditions cannot fully develop.



**Figure 12.** Boundary Condition on the Gear Channel (a) distance 90 m, (b) distance 93.7 m, (c) distance 95 m

### ***Section 5***

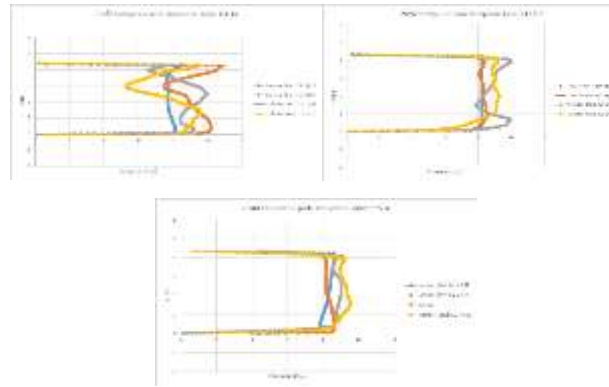
In the fifth segment, the flow boundary layer along the section (distance  $x = 98.7$  m and  $x = 100$  m) experiences non-uniform conditions, which is due to the flow momentum that cannot withstand pressure in this area.



**Figure 13.** Boundary Condition on the Gear Channel (a) distance 95 m, (b) distance 93.7 m, (c) distance 95 m

### ***Outlet Straight Channel***

In the straight channel leading to the outlet of the dodge channel, the flow speed increases and the speed profile of the straight channel gradually stabilizes with the distribus speed shown in the cross-section which is almost uniform, the distance  $x = 100$  m and  $x 175$  m).

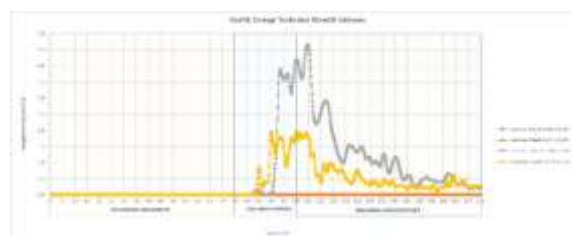


**Figure 14.** Boundary Condition on Outlet Channel (a) distance 100 m, (b) distance 137.5 m, (c) distance 175 m

***Turbulent Kinetic Energy Profile (k)***

The parameter  $k$  plays a role in influencing the turbulent processes that occur between water and the boundary of the solid surface. In the laminar model simulation, there was no change in the kinetic energy value, while in the turbulent simulation, there was an increase in the  $k$  value with a mesh size of  $0.40 \times 0.39$  and  $0.20 \times 0.19$ . Such changes in  $K$  values indicate that turbulent flows have been simulated.

In general, the  $k$  value in the inlet straight channel is zero ( $0 \text{ kg.m}^2/\text{s}^2$ ) this indicates that when the flow passes through the inlet straight channel, there has not been turbulent flow, when the flow enters the gear area (Distance  $x = 82.5 \text{ m}$ ) the  $k$  value increases and fluctuates until it enters the channel outlet area (distance  $x = 100 \text{ m}$ ). In the straight channel area of the outlet the  $k$  value decreases after passing the distance  $x = 105 \text{ m}$ , the  $k$  value gradually drops to the direction of the channel outlet. From the results of the graph of turbulent kinetic energy values, it shows that the shape of the serrations in the dodge channel can cause turbulent flow in the channel.



**Figure 15.** Turbulent Kinetic Energy Profile of Channels

***Flow Pressure Profile***

Pressure drop in flow is expressed by the difference in pressure values at the inlet and outlet. From the pressure profile graph, it can be seen that the serrated channel causes pressure on the dodge channel, this is due to changes in the cross-sectional shape of the channel. In general, turbulent model simulations provide greater pressure drop values than laminar simulations, and flow simulations with turbulent models with a mesh size of  $0.49 \times 0.38$  have the largest pressure drop values.



**Figure 16.** Flow Pressure Profile on Channels

With a recapitulation of the pressure difference in the channel shown in table 4.2.

**Table 2.** Pressure Drop Recap

No	Model	Ukuran Mesh	P inlet (N/m <sup>2</sup> )	Delta P	Delta P
1	Turbulen	0,49 x 0,38	-4,58	-44,09	39,51
2	Turbulen	0,24 x 0,19	-3,01	-38,42	35,41
3	Laminar	0,49 x 0,38	-19,40	-47,23	27,83
4	Laminar	0,24 x 0,19	-20,10	-47,23	27,13

## CONCLUSION

Based on the sensitivity test of the mesh size, the RMSE value in the simulation with a mesh of 0.40 x 0.39 m is 0.185, while the RMSE value in the simulation with a mesh of 0.20 x 0.19 m is 0.063. From the results of the study, it is known that the phenomenon that occurs in the flow in the dodge channel with the presence of serration-shaped dimensions. The contour and flow velocity profile show that the shape of the serrations on the channel affects the velocity profile where when the flow is in the straight channel the inlet flow does not develop (non-developed flow), when the flow enters the gear area the flow contour changes to unstable to the straight channel of the outlet.

In the results of the turbulent kinetic energy profile of the laminar model does not show an increase in the value of k, while in the turbulent model there is an increase in the value of k that arises when entering the jagged channel and gradually decreases when passing through the straight channel of the outlet, the change in the value of k shows that turbulent flow has been simulated. The results of the flow pressure profile show that the shape of the serrations on the channel causes a decrease in the pressure value. The largest decrease value in this study was 39.51 N/m<sup>2</sup> which was the result of flow simulation using a turbulent model with a mesh size of 0.40 x 0.38.

## REFERENCES

- Agrafioti, P., Kaloudis, E., Bantas, S., Sotiroudas, V., & Athanassiou, C. G. (2020). Modeling the distribution of phosphine and insect mortality in cylindrical grain silos with computational fluid dynamics: validation with field trials. *Computers and Electronics in Agriculture*, 173, 105383.

- Akbari, H., & Pooyarad, A. (2020). Wave force on protected submarine pipelines over porous and impermeable beds using SPH numerical model. *Applied Ocean Research*, 98, 102118.
- Fan, X., Dufresne, A., Subramanian, S. S., Strom, A., Hermanns, R., Stefanelli, C. T., Hewitt, K., Yunus, A. P., Dunning, S., & Capra, L. (2020). The formation and impact of landslide dams—State of the art. *Earth-Science Reviews*, 203, 103116.
- Izzati, A., Lasminto, U., & Negara, S. D. (2021). Permodelan Numerik Aliran pada Pelimpah Bertangga Menggunakan Program OpenFOAM. *Jurnal Aplikasi Teknik Sipil*, 19(2), 125–134.
- Khain, A., Ovtchinnikov, M., Pinsky, M., Pokrovsky, A., & Krugliak, H. (2000). Notes on the state-of-the-art numerical modeling of cloud microphysics. *Atmospheric Research*, 55(3–4), 159–224.
- Liski, J., Palosuo, T., Peltoniemi, M., & Sievänen, R. (2005). Carbon and decomposition model Yasso for forest soils. *Ecological Modelling*, 189(1–2), 168–182.
- Shahid, F., Zameer, A., & Muneeb, M. (2020). Predictions for COVID-19 with deep learning models of LSTM, GRU and Bi-LSTM. *Chaos, Solitons & Fractals*, 140, 110212.
- Wang, S., Bell, J. R., Burton, D., Herbst, A. H., Sheridan, J., & Thompson, M. C. (2017). The performance of different turbulence models (URANS, SAS and DES) for predicting high-speed train slipstream. *Journal of Wind Engineering and Industrial Aerodynamics*, 165, 46–57.
- White, C., Borg, M. K., Scanlon, T. J., Longshaw, S. M., John, B., Emerson, D. R., & Reese, J. M. (2018). dsmcFoam+: An OpenFOAM based direct simulation Monte Carlo solver. *Computer Physics Communications*, 224, 22–43.
- Zhang, H., & Huang, H. (2018). Decision-making self-efficacy mediates the peer support–career exploration relationship. *Social Behavior and Personality: An International Journal*, 46(3), 485–498.
- Zheng, S., & Kahn, M. E. (2013). Understanding China’s urban pollution dynamics. *Journal of Economic Literature*, 51(3), 731–772.