Experimental and numerical modeling of various energy dissipater designs in chute channels

Serife Yurdagul Kumcu1 · Kamil Ispir2

Received: 30 June 2022 / Accepted: 17 October 2022 / Published online: 2 November 2022 © The Author(s) 2022

Abstract
Energy dissipater in the stilling basin is a structure designed to protect downstream of the spillway from erosion and scour by reducing flow energy in the energy dissipation pool. Energy dissipation pool is an important element of hydraulic structures as a transition between the high-velocity flow and the sensitive tail water. The aim of this study is to investigate the energy dissipation ratios of baffle blocks which constructed in Type III stilling basin with different geometric shapes by using physical and numerical modeling methods. In all types of baffle blocks, the block heights and widths are chosen equally, and the total baffle block lengths are designed to be 50% of the stilling basin width. Energy dissipation ratio of the baffle blocks in 4 different geometric shapes was determined in 3 different layouts as single row, two rows and two rows without threshold structure, in a total of 12 different designs and 7 different flow rates are tested. In addition, these experimental studies were tested in the FLOW-3D software program, and the results of experimental and numerical studies were compared. As a result of the study, it was determined that the T-shaped energy-dissipating block had the highest energy dissipation ratio, and it was observed that the results obtained from the experimental studies and the FLOW-3D software were similar to each other with in the range of 5%.

Keywords Numerical simulation · Physical simulation · Spillway structures · Stilling basins · Energy dissipation block · Energy dissipaters · FLOW-3D

Introduction
Spillways are the hydraulic structures that transfer the excessive water safely from reservoir to downstream side without damaging the dam body. A spillway structure generally consists of the approach channel, spillway, aerators and the energy dissipation structure. Discharging flow from top of the dam body with high energy can damage the structures on the downstream side of the spillway by scouring. Energy-dissipating structures reduce the energy of the flow and allow it to pass to the downstream side with lower energy. The basic principle of energy-dissipating structures is to ensure that the hydraulic jump, that formed when flow regime changes from supercritical to subcritical, occurs in the stilling basins (Hager 1992). Energy-dissipating structures play an important role in the safety and cost of spillway. Topography, flow conditions and geological factors have great importance on choosing and projecting a type of energy-dissipating structure. As many different types of designs can be made by using the main principles such as impinging water, creating turbulence, scattering water or providing friction with water and air (United States Department of the Interior B of R 1987), physical model study and numerical modeling tests should be done before the final project in order to construct most suitable energy dissipater structures.

Stilling basins types were first described by Bradley and Peterka (Bradley and Peterka 1957), and a series of experiments on chute blocks, baffle blocks and end sill were carried out and stilling basins types were classified according to Froude number and flow velocity. Baffle blocks are placed in the basin to allow hydraulic jump and increasing turbulence. Energy-dissipating pool length is getting shortening in order to break energy of flow. Baffle blocks can be used
in a single row or in more than one row. It has been suggested by the Peterka (1984) that baffle blocks in the second and subsequent rows should be placed in a staggered manner, the first block should be placed half the width of the block from the wall, and the width of the blocks in the same row and the distance between the blocks should be equal. Hager (1992) classified hydraulic jumps based on Froude numbers and suggested formulas in practical application. Higgs compared the results obtained from numerical model and physical model of the USBR Type 2 stilling basin of the Ridgway dam created in 1996 USBR laboratories. The use of different types of energy blocks has also been tested by Ozbay (2008). In this study, in order to find out the energy-dissipating ratio of various shaped blocks placed in energy-dissipating structures, various types of blocks were tested experimentally in various displacement in plan. Their energy dissipation ratios are very close to each other, but stepped-type energy dissipation block had a slightly higher dissipating rate than the other blocks. Some researchers have tried to increase the efficiency by changing baffle and chute block geometries in the stilling basin structure (Pagliara and Palermo 2012; Bestawy 2013). Cook (2002) studied the spillway and stilling basin constructed within the scope of the Dalles project using by Flow-3D and compared the results obtained from the numerical model and the physical model. Amorim et al. compared the results obtained from numerical model of the stilling basin of the Porto Colombia Hydroelectric power plant with 1/100 scale physical model of the power plant (Amorim et al. 2015). Nigam et al. (2016) did an overview and worked on hydraulic jump-type stilling basins. They dealt with the hydrodynamic design aspect of jump-type energy dissipaters by experimentally and analytically along with comparison of various energy dissipaters. Based on estimating the uplift and hydrodynamic forces on energy dissipaters, although jump-type energy dissipaters with only one end sill are sufficient for higher velocities, it was not recommended to use it for head above 100 m. Dermawan et al. (2021) carried out the physical model study by experimentally bottom lowering of horizontal and USBR II stilling basin. It was expected to represent flow behavior in the overflow system regarding flow conditions and energy dissipation. After experiments, the amount of flow energy that occurs at each control point is calculated. USBR II is found that has baffle blocks at the toe and end sill, the flow becomes more turbulent with compared to the flat stilling basin that does not have baffle blocks. USBR II was better than flat stilling basin, while discharge is increasing with a higher difference in overflow height.

Flow conditions on overflow systems can result in construction failure, mainly due to the high flow energy. Since the dams require a unique design (site-specific) in topographic conditions, there may be some situations where the energy dissipation pool is not sufficient. In such cases, USBR designs may not be enough and additionally energy dissipater blocks can be used to obtain higher energy loss (Kumcu and Kökpınar 2019).

In this study, the physical hydraulic model tests were carried out to increase energy-dissipating ratios of various baffle blocks placed in various layouts on USBR III energy-dissipating pools. In order to find out energy dissipation ratios, the contribution of the baffle blocks in stilling basins located downstream of the ogee spillway was investigated by physical and numerical modeling. Four different types of baffle block geometries were used in the study. The stilling basin is sized as USBR III type according to the Froude number and flow rate. The results of the experiments and the numerical model study of the energy dissipation ratios of the baffle blocks are also given.

**Materials and methods**

A hydraulic jump is a sudden rise in the water surface that occurs when the flow regime changes from the supercritical to the subcritical. During the hydraulic jump, a significant amount of energy is absorbed over a short distance. In Fig. 1, the general view of the stilling basin and hydraulic jump formed in the pool is given.

The definitions of the parameters described in Fig. 1 are given below.

**Fig. 1** Hydraulic jump in the stilling basin. a $1.7 < Fr < 2.5$ pre-jump stage. b $2.5 < Fr < 4.5$ transition jump. c $4.5 < Fr < 9$; balanced jump. d $Fr > 9$; effective jump
h₁ = Flow depth before the hydraulic jump.

h₂ = Flow depth after the hydraulic jump.

h₃ = Flow depth at downstream.

h = Flow head over the crest.

p = Crest height.

a₈ = End sill height.

V²/2 g = Velocity head.

V₀ = Approach flow velocity.

V₁ = Velocity of the flow before the jump.

Δh = Head of the dissipated energy.

H₀ = Total water head over the crest.

The relationship between h₁ and h₂ by using the momentum equations during the hydraulic jump is as follows.

\[ \frac{h₂}{h₁} = \sqrt{1 + 8F_r^2 - 1} \]

Hydraulic jumps are classified according to the Froude number as \( F_r = V / \sqrt{gh₁} \) depending on the Froude number and the jump types are given in Fig. 2.

Fr = 1; The flow is critical, and no jump occurs.

1.7 < Fr < 2.5; pre-jump stage; there are slight waves on the water surface. Practically no jump is considered. There is no need for a stilling basin; it can be controlled using simple energy-dissipating pools. Energy loss is less than 20%.

2.5 < Fr < 4.5; transition jump; it requires a stilling basin to be taken under control. It is appropriate to use the IV type energy-dissipating pool. Energy dissipation is between 20 and 45%.

4.5 < Fr < 9; balanced jump; flow is little affected by tail water changes. These types of jumps are usually seen in high dams. If V₁ < 18 m/s, Type III stilling basin is suitable, if V₁ > 18 m/s, Type II stilling basin is suitable. Energy absorption is between 45 and 70%.

Fr > 9; effective jump; it is a very strong jump. The size of the stilling basin depends on the size of the hydraulic jump. Energy absorption can reach up to 85%.

**Stilling basin**

For the rating curve, total head (h) over the weir corresponding to each discharge value and the flow depths before the hydraulic jump (h₁) were measured. In the experimental channel, the observed maximum total head (h) over the ogee spillway is 14.40 cm for corresponding discharge of 39.62 l/s is given in Fig. 3.

Flow depth (h₁) and corresponding velocity (V₁) and Froude number (Fr) before the hydraulic jump were calculated, and the highest velocity and the Froude number were computed as 2.75 m/s and 8.83, respectively. Type III stilling basin is used when the Froude number is greater than 4.5 and the flow velocity is less than 18.3 m/s (60 ft/s). Thus, USBR Type III stilling basin was chosen, which is suitable for the design in flow conditions where the calculated Froude number, Fₚ = 8.33, is greater than 4.5 and the maximum velocity V₁ = 2.75 is less than 18.3 m/s (60 ft/s). Type III stilling basin is designed according to USBR, and dimensioning of the basin, baffle and chute blocks is given in Fig. 4. Chart of the dimensioning is given in Fig. 5. Limit values of the study are given in Table 1.
Flow depth ($h_1$) and corresponding velocity ($V_1$) and Froude number ($Fr$) before the hydraulic jump were calculated, and the highest velocity and the Froude number were computed as 2.75 m/s and 8.83, respectively. Type III stilling basin is used when the Froude number is greater than 4.5 and the flow velocity is less than 18.3 m/s (60 ft/s). Thus, USBR Type III stilling basin was chosen, which is suitable for the design in flow conditions where the calculated Froude number, $Fr = 8.33$, is greater than 4.5 and the maximum velocity $V_1 = 2.75$ is less than 18.3 m/s (60 ft/s). Limit values of the study are given in Table 1. Type III stilling basin is designed according to USBR, and dimensioning of the basin, baffle and chute blocks is given in Fig. 4.

**Experimental setup**

Experiments were carried out in a rectangular open-channel flume with a length of 6.70 m, a width of 0.30 m and a depth of 0.50 m. In the experimental setup, flow in the open-channel flume is provided by two pumps; each of them has a power of 7.5 kW, connected in parallel to the system. The water flowing in the open-channel system is supplied from two reservoirs. The pumps take the water from the reservoir-1 and convey it into reservoir-2. Then, the water reaches to the reservoir-2 passes through the laboratory flume and is poured back into the reservoir-1 (Fig. 5). The total discharge in the channel is equal to the sum of the flows supplied from both pumps. The discharge that the pumps will provide is adjusted by the frequency alternative on the panel to which the pumps are connected. The flow through the system is read by electromagnetic flowmeter placed between the pipes after the pumps. Flow depth was measured with a limnimeter with an accuracy of ± 1 mm placed in the open-channel flume (Fig. 6). The open-channel flume is made of 1.2-cm-thick laminated glass-walled, which is obtained by combining two 0.6-cm-thick tempered glass sheets with a plastic layer placed between them. In the experiments, oggee-type spillway and stilling basin is made of Plexiglas.

Experiments are conducted for 7 various discharge values (10, 15, 20, 25, 30, 35, and 39.62 l/s). Stilling basin elements were prepared in accordance with the methods recommended by the USBR and adhered to the open channel with the help of silicone. The flow depths were measured by the limnimeter.

![Fig. 4 Type III stilling basin (Peterka 1984)](image)

![Fig. 5 Schematic view of open-channel flume](image)

| Min./Max. | Q (l/s) | H (cm) | $h_1$ (cm) | Channel width B (cm) | $V = \frac{Q}{A}$ (m/s) | $Fr = \frac{V}{\sqrt{gxh_1}}$ | $\frac{h_1}{h_2}$ | $\frac{V}{\sqrt{gxh_1}} - 1$ |
|-----------|---------|--------|------------|-----------------------|------------------------|------------------------|-----------|------------------------|
| Min       | 1.10    | 1.52   | 0.26       | 30.00                 | 1.41                   | 8.83                   | 3.12      | 0.01                   |
| Max       | 39.62   | 14.40  | 4.80       | 30.00                 | 2.75                   | 4.01                   | 24.92     | 0.01                   |

Table 1 Max and min values used for designing USBR Type III basin
Free surfaces are modeled with the volume-of-fluid (VOF) technique, trademarked as TruVOF, used for tracking and locating the fluid–fluid interface. It was first reported in Nichols and Hirt (1975) and more completely in Hirt and Nichols (1981). It belongs to the class of Eulerian methods which are characterized by a mesh that is either stationary or is moving in a certain prescribed manner to accommodate the evolving shape of the interface. This technique provides three important functions for free surface flow: location and orientation of free surfaces within computational cells, tracking of free surface motion through cells and a boundary condition applied at the free surface interface.

The location of the flow obstacles is evaluated by the program implementing a cell porosity technique called the fractional area/volume obstacle representation of FAVOR method (Hirt 1992). The free surface was computed using a modified volume-of-fluid method (Hirt and Nichols 1981). For each cell, the program calculates average values for the flow parameters (pressures and velocities) at discrete times using staggered grid technique (Malalasekera et al. 1996). Governing equations for analysis of incompressible three-dimensional flow on the Cartesian coordinate system are given by Kim et al. (2010):

$$\frac{V_F}{\rho} \frac{\partial p}{\partial t} + \frac{\partial}{\partial x}(u_A x) + \frac{\partial}{\partial y}(v_A y) + \frac{\partial}{\partial z}(w_A z) = \frac{R_{SOR}}{\rho}$$

(2)

where \((u, v, w)\) are velocity components in the \((x, y, z)\) coordinate directions, \((A_x, A_y, A_z)\) are fractional areas open to flow in the same coordinate system, density and density source terms are given by \(\rho\) and \(R_{SOR}\), respectively. 3D Navier–Stokes equations used in flow simulations are given below:

$$\frac{\partial u}{\partial t} + \frac{1}{V_F} \left( u_A \frac{\partial u}{\partial x} + v A_x \frac{\partial v}{\partial y} + w A_x \frac{\partial w}{\partial z} \right) = - \frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x$$

(3)

$$\frac{\partial v}{\partial t} + \frac{1}{V_F} \left( u_A \frac{\partial v}{\partial x} + v A_y \frac{\partial v}{\partial y} + w A_y \frac{\partial w}{\partial z} \right) = - \frac{1}{\rho} \frac{\partial p}{\partial y} + G_y + f_y$$

(4)

$$\frac{\partial w}{\partial t} + \frac{1}{V_F} \left( u_A \frac{\partial w}{\partial x} + v A_z \frac{\partial v}{\partial y} + w A_z \frac{\partial w}{\partial z} \right) = - \frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z$$

(5)

where flow volume is \(V_F\), \(G_x, G_y, G_z\) are mass gravities in \(x, y, z\) coordinate systems, and \(f_x, f_y, f_z\) are viscous gravities in \(x, y, z\) coordinate systems.

Continuity equation is also given as:

$$\frac{\partial}{\partial x}(u_A x) + \frac{\partial}{\partial y}(v_A y) + \frac{\partial}{\partial z}(w_A z) = 0$$

(6)

Air–water interphase is assumed in VOF solution methods used in multi-phases flows. A single fluid-free surface

**Shape and layouts of energy dissipater blocks used in the study**

Experimental studies were carried out on physical models for investigating the energy absorption ratios of the energy-dissipating blocks placed in the USBR Type III stilling basin. In this study, a total of 12 experimental setups with 4 different geometric shape of baffle blocks having layout of a single row, two rows and two rows without threshold were used. In the experimental setup, the height and the width of the chute and baffle blocks were kept constant, and they were studied at different discharges. In the experiments, the data obtained by measuring the height after splashing and downstream water level at 7 different flow rates were compared, and the energy-dissipating ratios were calculated. In the experiments, 4 different energy dissipater block types, trapezoidal, circular, step and T-section type, were used. The energy block types used were placed in the energy-dissipating pool first in a single row, then 2 rows and then 2 rows without threshold, and the flow conditions were investigated. Dimensions and layouts of each design of chute blocks and baffle piers are given in Figs. 7, 8, 9, 10.

**Numerical modeling**

During numerical simulation of the flow field, CFD package of FLOW-3D Version 10.0 was used. It uses finite differences on solving water–air flow. Finite volume method is applied in the CFD to solve the Reynolds averaged Navier–Stokes (RANS) equation. Among the various alternatives, k-epsilon modeling was preferred to solve turbulence model as the most common model used in computational fluid dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions, especially in open channels.
Fig. 7 Trapezoidal block types used in the experiments. 

- **a** Dimensions and laboratory view. 
- **b** A single row in plan. 
- **c** Two rows in plan. 
- **d** Two rows without end sill in plan
Fig. 8  Circular block types used in the experiments.  

(a) Dimensions and laboratory view

(b) A single row in plan

(c) Two rows in plan

(d) Two rows without end sill in plan

1. Dimensions and laboratory view
2. A single row in plan
3. Two rows in plan
4. Two rows without end sill in plan
Fig. 9 Stepped block types used in the experiments. 

(a) Dimensions and laboratory view

(b) A single row in plan

(c) Two rows in plan

(d) Two rows without end sill in plan
Fig. 10  T-shape block types used in the experiments

(a) Dimensions and laboratory view

(b) A single row in plan

(c) Two rows in plan

(d) Two rows without end sill in plan
flow solution was used in the analyses. For the VOF (volume-of-fluid) method, it is provided to define the fill or void ratio of each mesh cell and to perform pre-debugging by using pre-process. Each phase (fluid) is given a variable that accounts for how much percentage of each computational cell is occupied by the phase. This variable is called a volume fraction of the phase. The volume fractions of all phases sum up to unity in each computational cell. The fields for all variables and properties are shared by the phases and represent volume-averaged values. Variables and properties represent either only one phase or a mixture of phases in a given cell depending on the volume fractions of the phases in the cell. If \( F(x,y,z,t) \) is equal to 1, the control volume will be full of fluid, and \( F(x,y,z,t) \) is equal to 0, no fluid will exist in a control volume. Furthermore, in the case of a free water surface, \( F(x,y,z,t) \) is shown to have the value between 0 and 1. Applying function \( F(x, y, z, t) \) to Eq. (2) governing equation becomes:

\[
\frac{\partial F}{\partial t} + \frac{1}{V_F} \left[ \frac{\partial}{\partial x} (F u A_x) + \frac{\partial}{\partial y} (F v A_y) + \frac{\partial}{\partial z} (F w A_z) \right] = 0 \quad (7)
\]

In this study, \( F(x, y, z, t) \) is selected 0.5 as it generally gives better results in open-channel flows. During the numerical study, runs were carried out from the coarsest to the thinnest ”finite element mesh sizes.” The tests were repeated so that the experimental results and the numerical results approached each other. The tests continued until the difference was almost 5%. The numerical modeling was continued with the finite mesh size of \( \Delta X = \Delta Y = \Delta Z = 5 \) mm, which gives the optimum result, bringing the experimental study and numerical modeling results closest to each other and minimizing the solution time. When mesh cells of 5 mm size were used in the analyses, the mesh block contains a total of 1536,000 cells. The part where water enters the system (-X side) is defined as the pressure (static water level). Depending on the desired weir load on the weir, the height of this static water level was adjusted and water was allowed to enter at the desired height. The side surfaces and the bottom of the pool were chosen as walls, the downstream part as outflow and the upper part as pressure to represent the atmospheric pressure. To obtain the desired analysis results, fluid fraction (filling ratio) and hydraulic data options are marked in “output” section. The solid model and layer conditions used in the analysis are shown in Fig. 11. In the FLOW-3D CFD software, viscosity and turbulence, gravity, air-entrainment and density evaluation modules were activated for all cases. Since there are experimental data available for comparison to the CFD solution, the data from the physical model have been directly used in CFD dimensions (Kumcu 2010; Ispir 2021).

The velocity and Froude number values calculated by numerical model are shown in Fig. 12. According to the numerical model, the maximum Froude number was calculated as 7.25 and the maximum velocity was calculated as 3.06. These values are consistent with the values chosen during the selection and design of the energy-dissipating structure and show the accuracy of the numerical model.

Solid models, belonging to tested shape and layout of the energy dissipaters, used in the numerical simulation are summarized in Table 2.

Results

Physical model

During this experimental study on the open channel, the energy dissipation ratios of baffle blocks having different geometric shapes were investigated with the help of the hydraulic jump created in the flow. The measured depths and velocities of the flow before and after the hydraulic jump formation were investigated, and the energy dissipation ratios were found by computing the total heads of the flow. To determine the amount of energy dissipation and to find the most effective plan shape of baffle blocks were designed as single row, double row and double rows and compared according to their non-threshold arrangement. In order to show the differences of the shape and layouts of the baffle blocks on energy-dissipating ratios, the graphs belonging to energy dissipation ratios with respect to dimensionless Fr number are prepared and given in Figs. 13, 14, 15.

When the energy dissipation ratios of the single-row energy-dissipating blocks are examined in Fig. 14, it is seen that the highest absorption rate is obtained at 20 l/s, which corresponds to almost 51% of the total head of the flow. At the maximum design discharge of \( Q = 39.6 \) l/s, it was observed that the highest energy dissipation rate belongs
When the energy-dissipating ratios of the two-row energy-dissipating blocks are analyzed in Fig. 14, Fr number increasing energy-dissipating ratio is increasing up to Fr = 5. If the Fr number passes over the 5, energy-dissipating ratios are decreasing up to 42% for \( Q = 20 \text{ l/s} \). It is seen that the highest decreasing rate decreased by 50% with 20 l/s, which belongs to the trapezoidal and circular-section energy-dissipating blocks.

Figure 15 summarizes the variation of the energy-dissipating rates by Fr number for each tested shape of the baffle blocks for without end sill design in plan. It is clearly seen from this figure that Fr number increases up its maximum value, which is about 50%, and then it decreases to about 45%.

When the energy-dissipating ratios of the T-shaped energy dissipater blocks are checked by considering their layout placement as in Fig. 16, it is seen that the highest energy-dissipating rate is reduced by 51% with 20 l/s, which corresponds to almost 50% of the design flow. In the design flow, it was observed that the highest energy reducing rate belonged to the trapezoidal section energy-dissipating block plan and reached 41%.

During the experimental study, piezometric water levels are also measured for analyzing the effect of the energy-dissipating baffle addition to system on pressure variations along the energy-dissipating pool. Comparison of the baffle blocks, which are added as T-shaped a single row to system and the design of without baffle blocks, is given in Fig. 17, as an example. After adding the T-shape a single-row baffle blocks, pressure is decreasing along the energy-dissipating pool for a given maximum discharge of \( Q = 39.62 \text{ l/s} \). One can have a comment from this figure that the pressure is higher than the one observed from the original design of Type III in front of the baffle blocks, but the flow is leaving from the pool to downstream of the structure by lower pressure values than the original design because of the effect of the additional blocks to system.

**Numerical modeling**

The experimental setups of the single-row energy reduction blocks used in the experimental study were tested with the FLOW-3D numerical method at the design flow rate, and the data on the hydraulic properties obtained are given in Table 3. Maximum and minimum velocity values and velocity paths can be seen in Fig. 18. If Table 3 is analyzed, it is
Table 2  Solid models used in the CFD simulations

| Shape   | Layout                      |
|---------|----------------------------|
|         | A single row | Two rows | Two rows without end sill |
| Trapezoidal |          |          |                            |
| Circular |          |          |                            |
| Stepped |          |          |                            |
| T-shape |          |          |                            |

Fig. 13 Comparison of energy-dissipating ratios of the single-row energy dissipaters

Fig. 14 Comparison of energy-dissipating ratios of two-row energy dissipaters
seen that the highest energy-dissipating rate at the design flow rate belongs to the T-section energy-dissipating blocks which their energy-dissipating rates reach 43% for maximum design discharge of 39.6 l/s. As it can be seen from Fig. 19 that the energy-dissipating rates of trapezoidal are less than the others and 41%. The energy-dissipating rates of stepped and circular blocks are almost the same and 42%.

Discussion

Experimental setups of single-row energy-dissipating blocks were tested physically at seven different flow rates and with the FLOW-3D numerical method at the design flow. When the energy dissipation rates of the single-row energy-dissipating blocks are analyzed in Table 4, the experimental study results and the FLOW-3D results have almost the same values. When Table 4 is examined, it is seen that the $h_3$ values and the energy reducing rates of numerical modeling values are higher than energy-reducing rates of physical modeling values for a given flow discharge and baffle blocks. If Fig. 20 is analyzed, T-shaped baffle blocks give the highest energy-dissipating rates in both numerical modeling and physical modeling as 43% and 41%, respectively. If the results of the physical modeling and numerical modeling are compared with each other, numerical modeling, energy-dissipating rates of numerical modeling are almost 4.6% higher than the physical modeling results in T-shaped baffle block design. The maximum difference of physical modeling and numerical modeling is observed as almost 7% in the study.

Conclusions

A series of experiments were carried out to investigate the similarities and differences of the energy-dissipating ratios of the different geometric-shaped energy-dissipating blocks placed in the USBR Type III energy-dissipating pool. All tests were repeated by also the numerical model study by FLOW-3D. The results were compared with each other. The results obtained according to the studies carried out are as follows:

The block heights and widths of the energy-dissipating blocks were chosen equally, and the total energy-dissipating block length was designed to be 50% of the energy-dissipating pool width, and it was observed that the energy-breaking rates were different.

Hydraulic jump occurred in all experimental setups. The energy absorption ratios were close to each other in all the apparatus and energy-dissipating block types used in the experiment. In the single-row energy-dissipating block design, the highest dissipating rate at the design flow rate was obtained in the T-section energy-dissipating block. In the two-row energy-dissipating block design, the highest dissipating rate at the design flow rate was again obtained in the T-section energy-dissipating block. In the design of the two rows of unthreshold energy-dissipating block, the highest
The dissipating rate in the design flow rate was obtained in the trapezoidal energy dissipation block. In all experimental setups, the highest dissipating rate was obtained at 20 l/s flow rate and the lowest dissipating rate was obtained at design flow rate of $Q = 39.6$ l/s. Energy-dissipating blocks shorten the spawning distance and shorten the length of the pool.

The effect of the baffle blocks is seen in also piezometric water depth measurements; flow passes from energy-dissipating pool to downstream of the structure with lower pressure head in use of T-shape a single-row baffle blocks.

Experimental study results and FLOW-3D results are results that are compatible with a maximum range of 7%. Although experimental studies are thought to give more reliable results, it should be kept in mind that experimental studies cannot always be carried out, considering laboratory facilities, time, labor and construction costs. Although the assumptions made during the calculations of the numerical studies, the initial investment cost and the training difficulties are considered, the ease of use and the duration of the program results and the similarity of the results to the experimental studies and real results should be considered. In the continuation of this study, it is planned to continue experimental and numerical studies by changing the heights and distances of different types of energy blocks under different flow conditions. It is important to carry out studies in different spillway shapes and different flow conditions in terms of generalizing the design.

### Table 3

| Type     | $h_1$ (m) | $V_1$ (m/s) | $E_1$ (m) | $h_2$ (m) | $V_3$ (m/s) | $E_3$ (m) | $(E_1-E_3)/E_1$ | Fr_1 | Fr_2 |
|----------|-----------|-------------|-----------|-----------|-------------|-----------|-----------------|------|------|
| Trapezoidal | 0.048    | 2.75        | 0.4338    | 0.24      | 0.55        | 0.26      | 0.41            | 4.01 | 0.36 |
| Circular  | 0.048    | 2.75        | 0.4338    | 0.23      | 0.57        | 0.25      | 0.42            | 4.01 | 0.38 |
| Stepped   | 0.048    | 2.75        | 0.4338    | 0.24      | 0.56        | 0.25      | 0.42            | 4.01 | 0.4  |
| T-shape   | 0.048    | 2.75        | 0.4338    | 0.23      | 0.63        | 0.25      | 0.43            | 4.01 | 0.44 |

### Table 4

| Baffle block shape | Physical modeling | Numerical modeling |
|--------------------|-------------------|--------------------|
|                    | $h_1$ (m) | $V_3$ (m/s) | $(E_1-E_3)/E_1$ | $h_1$ (m) | $V_3$ (m/s) | $(E_1-E_3)/E_1$ |
| Trapezoidal         | 0.235     | 0.53      | 0.39           | 0.24      | 0.55      | 0.41           |
| Circular            | 0.238     | 0.53      | 0.39           | 0.23      | 0.57      | 0.42           |
| Stepped             | 0.24      | 0.53      | 0.39           | 0.24      | 0.6       | 0.42           |
| T-shape             | 0.24      | 0.54      | 0.41           | 0.23      | 0.63      | 0.43           |
Author contributions SYK helped in investigation, experimental and numerical study, writing & editing, methodology, conceptualization. KI contributed to investigation, experimental and numerical study, conceptualization.

Funding The author(s) received no specific funding for this work.

Declarations

Conflict of interest On behalf of all authors, the corresponding author states that there is no conflict of interest.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article’s Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article’s Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit http://creativecommons.org/licenses/by/4.0/.

References

Amorim JCC, Amante RCR, Barbosa VD (2015) Experimental and numerical modeling of flow in a stilling basin, in E-proceedings of the 36th IAHR World Congress. Hague, Netherlands.

Bestawy A (2013) New shapes of baffle piers used in stilling basins as energy dissipaters. Asian Trans Eng, 3(1).

Bradley JN, Peterka AJ (1957) Hydraulic design of stilling basins: hydraulic jumps on a horizontal apron (basin I). J Hydraul Div 83(5):1401–1411

Cook C, Richmond MC, Serkowski JA, Ebner LL (2002) Free-surface computational fluid dynamics modeling of a spillway and tailrace: case study of the Dalles project, in Hydrovision. United States.

Derawan V, Suhardjono Prasetyorini L, Anam S (2021) Hydraulic Model experiment of energy dissipation on the horizontal and USBR II Stilling Basin, IOP Conferences, Earth and Environmental Science, 930 (012029).

Hager WH (1992) Energy dissipaters and hydraulic jump. Dordrecht. Springer Netherlands. [online] http://link.springer.com/https://doi.org/10.1007/978-94-015-8048-9 (Accessed April 26, 2022).

Hirt CW (1992) Volume-fraction techniques: powerful tools for flow modelling, Santa Fe.

Hirt CW, Nichols BD (1981) Volume of fluid (VOF) method for the dynamics of free boundaries. J Comput Phys 39(1):201–225

Ispir K (2021) Physical and numerical modelling of energy damping ratios of different type of energy dissipators. Dissertation, Ms Thesis, Necmettin Erbakan University, Turkey.

Kim SD, Lee HJ, Shin SD (2010) Improvement of hydraulic stability for spillway using CFD model. Int J Phys Sci 5(6):774–780

Kumcu SY (2010) Spillway structure of Kavsak Bendi HEPP project. Tech Rep, Model no: 385 (1005).

Kumcu SY, Kökpınar MA (2019) Application of numerical modeling on spillway structures: a case study of kavsak bendi hydroelectric power plant (HEPP). DSI Tech Bull 132:12–27

Malalasekera WMG, Versteeg HK, Gilchrist KA (1996) Review of research and an experimental study on the pulsation of buoyant diffusion flames and pool fires. Fire Mater 20:261–271

Nichols BD, Hirt CW (1975) Methods for calculating multi-dimensional, transient free surface flows past bodies. 1. International Conference on Numerical Ship Hydrodynamics. Gaithersburg, USA.

Nigam U, Das S, Choudhury M R (2016) Overview of energy dissipaters and stilling basins with design aspect of hydraulic jump type energy dissipaters. NCIEET2015, 1–9.

Ozbay O (2008) An investigation of energy dissipation ratios of different type energy dissipater blocks in chute channels. Dissertation, Ms Thesis, Fırat University, Turkey.

Pagliara S, Palermo M (2012) Effect of stilling basin geometry on the dissipative process in the presence of block ramps. Journal of irrigation and drainage engineering. 138(11), 1027–1031. [online] https://ascelibrary.org/doi/abs/https://doi.org/10.1061/%2ASCE%29IR.1943-4774.0000505 (Accessed April 26, 2022).

Peterka AJ (1984) Hydraulic design of stilling basins and energy dissipaters. Denver, Colorado, United States Department of the Interior Bureau of Reclamation. United States Department of the Interior B of R (1987) Design of small dams. Government Printing Office, Washington, DC, U.S

Publisher’s Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.