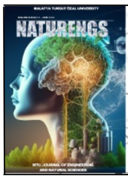
MALATYA TURGUT OZAL UNIVERSITY JOURNAL of ENGINEERING and NATURAL SCIENCES Volume 5, Issue 2 (2024) 70-76



NATURENGS
E-ISSN 2717-8013
https://dergipark.org.tr/en/pub/naturengs

The Role of CFD Technique in the Effect of Baffle Blocks on Flow Dynamics

Mahmut Aydoğdu

Malatya Turgut Ozal University, Department of Civil Engineering, Malatya Türkiye – 44900

ARTICLE INFO

Received 27.11.2024
Accepted 25.12.2024

Doi: 10.46572/naturengs.1592272

ABSTRACT

A flow with a high amount of hydraulic energy causes high velocities in the flow while passing downstream of the water structure, thus creating high pressure and friction conditions. As a result, problems such as erosion, abrasion, and cavitation may occur on the water structure or downstream. Not taking precautions causes the water structure to be destroyed completely or the surrounding facilities and structures to suffer serious damage. Energy-dissipating structures are constructed in field applications to bypass such negative situations. Blocks placed especially on chute channels or inside stilling basins are quite effective in dissipating the energy of the flow. These blocks dissipate the energy of the flow by creating turbulence inside. Sometimes the flow hits the blocks, and sometimes the blocks separate the flow, creating turbulence. The aim is to reduce the supercritical velocities to subcritical. It can only be achieved by providing a hydraulic jump. Within the scope of this study, a T-shaped block that can be used in the stilling basins was designed. Such a block type was preferred to increase the surface areas of the flow hitting the blocks and thus to observe the energy-dissipating situation. In the literature, similar block types were used in some previous studies. However, some flow dynamics parts that could not be measured experimentally were missing. Therefore, this deficiency was tried to be eliminated by modeling channel and block structures in a digital environment using today's computer and software technology. Firstly, this block type was modeled in a digital environment as a single-row and double-row block array, and energy-dissipating situations were observed. For this purpose, Ansys-Fluent software, which is widely used in Computational Fluid Dynamics (CFD), was preferred. The created model was designed in 3D and numerical solutions were obtained. According to the results of the study, the turbulence values effective in energy-dissipating were easily obtained. In addition, it was observed that the single-row block array dissipates more energy in the digital environment at a lower cost. Before moving on to field applications of this study, it can be said that it will be an important base in determining the flow-block interactions that cannot be measured experimentally.

Keywords: Baffle block, open channel, energy dissipate, CFD, stilling basin.

1. Introduction

The flow accumulated behind a water body or released from a high level has a very high amount of hydraulic energy. Since this energy causes high velocities in the flow, it creates high pressure and friction conditions. As a result, it creates scour, abrasion, and cavitation problems on the water structure or downstream. This is an undesirable situation. If precautions are not taken, it causes the water structure to fail or the facilities and structures around it to suffer serious damage. In field applications, energy-dissipating structures are constructed to bypass such negative situations. Chute channels, drop beds or stilling basins are the most preferred of these structures. Blocks placed especially on chute channels or inside stilling basins are quite effective in dissipating the energy of the flow. These blocks dissipate the energy of the flow by creating turbulence inside them. Sometimes this turbulence situation is achieved by the flow hitting the blocks and sometimes by separating the flow. The aim here is to reduce the supercritical velocities to subcritical. This can only be achieved by providing a hydraulic jump. By creating turbulence in the flow, energy dissipation of up to 85% was achieved through hydraulic jump [1]. Although the amount of blocks that block the channel

* Corresponding author. e-mail address: mahmut.aydogdu@ozal.edu.tr
ORCID : 0000-0002-7339-2442

cross-section is much debated, it is a widely accepted study in the literature [2]. Accordingly, for the blocks placed in the channel not to completely block the channel cross-section and act as not a sill, it is sufficient for the total block width to block the channel width by 40-55%. Basco and Adams [3] studied block location, height, width, spacing, second block row location, and geometries. The study discussed the effects of drag force on energy-dissipating blocks in hydraulic jumps. The resulting downstream water depths calculated using the measured drag force compared favorably with experimentally measured values. The results proved that the designer could calculate the drag force for a wide range of block geometries and jump entry Froude numbers ranging from 3 to 10. Mohamed Ali [4] investigated the effect of rough bottom energy-dissipating basins on the hydraulic jump length. Within the scope of the study, the optimum roughness height that gives the most suitable hydraulic jump length according to different under-cover flow heights and Froude numbers was determined. Rajaratnam and Hurtig [5] conducted a study on the subject of sieve-type energy dissipaters. In particular, different pore types were considered in the study. In the study, different Froude numbers were taken into account and the most suitable form was investigated by using sieves in single row and double row. In the thesis prepared by Çakır [6], the energy dissipation conditions of inclined porous sieves were investigated experimentally. The sieve porosity ratio used in the experiments was considered as 40%. The inclination angle, sieve thickness, sieve position, upstream flow depth, and Froude number were emphasized as experimental parameters. Aydoğdu and Dursun [7] discussed the energy dissipation performance of energy-dissipating blocks placed on the chute channel by applying two different arrangement shapes. It was also stated that as the flow rate increases, the amount of energy dissipated is less affected by the block arrangement. As a result, it was reported that more energy can be dissipated by changing the arrangement shape of blocks with the same properties in the channel.

Although most of the studies conducted so far have been experimental, with the development of computer technologies, digital environment studies on energy-dissipating structures have also been carried out. In particular, Computational Fluid Dynamics (CFD) software has become widely used [8-11]. Xiong et al. [12] stated that the main reason for the failure of bridges over the river is scour. The study states that 60% of 1000 bridges in the USA have failed due to scour in the last 30 years. To solve this problem, a model subjected to numerical analysis was presented. This model was later found to be consistent with the results of experimental data. Aydın et al. [13] adapted the numerical model they created within the scope of their study according to the experimental studies conducted by Peterka [14]. Here, they observed the energy-dissipating block effect in a chute channel using three different flow rates. In this study, which was conducted using Flow-3D software, it was stated that the blocks placed in the chute channel absorbed 70% of the energy. Niyazi et al. [15] numerically and experimentally investigated the flow dynamics around an energy-dissipating block placed in the center of the open channel. Based on the information on the entire flow structure around the deflector, the geometry of the central deflector was changed to suppress the recirculation effects. Finally, the vortex structures were suppressed and the length of the recirculation region was reduced by 76%.

In this study, the effect of energy-dissipating blocks with a total width of 50% of the channel cross-section was observed numerically. For this, 3D solutions were obtained using Ansys-Fluent software. In addition, the energy-dissipating effect of single-row and double-row arrangements of blocks was discussed using the Standard k-ε Turbulence Model. A T-shaped block geometry was selected as the block shape. Such a block type was preferred to increase the surface areas of the flow hitting the blocks and thus to observe the energy-dissipating situation. This block type was previously used in experimental studies, but how it affects the flow dynamics in the numerical environment was not discussed in detail. At this point, it can be said that the study will provide numerical solutions to data that cannot be obtained and discussed experimentally.

2. Material and Methods

Within the scope of this study, a channel structure of 3 m length, 0.4 m width, and 0.4 m height was modeled in a digital environment. For this purpose, analyses were performed using Ansys-Fluent software. Energy-dissipating blocks were placed on the downstream side, 1.2 m from the channel entrance, with a block height and width of 5 cm. The designed geometry and dimensions of the blocks are given in Figure 1.

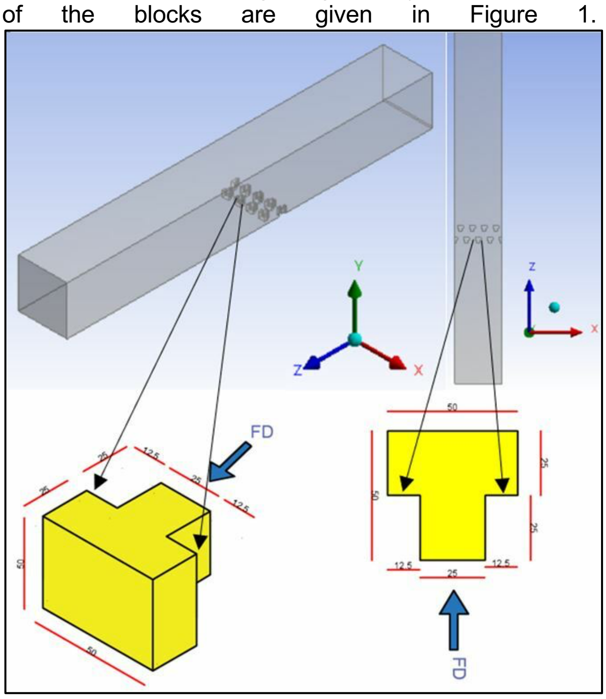


Figure 1. Channel geometry and block detail (all dimensions are in mm)

The blocks are arranged in a single row and double row in the channel and all solutions are carried out in a numerical environment. Standard k-ε turbulence model is used for the solutions. The data used in the numerical solution are given in Table 1.

Table 1. Data used in the numerical solution.

	Single Row Block Arrangement	Double Rows of Block Arrangement
Q (m^3/s)	0.013356	0.013356
h_1(m)	0.0159	0.0159
V_1 (m/s)	2.1	2.1

As can be seen from Table 1, to see the energy dissipation situation between single-row and double-row block arrangements, the flow rate, flow velocity at the channel entrance, and depth values were taken equally. In this study, the energy loss between the upstream and downstream flows in the presence of the block arrangements used was calculated. The energy (E) at any section of the open channel is calculated with the specific energy equation below using the flow depth "h" and flow velocity "V" at that point.

$$E = h + \frac{V^2}{2g} \tag{1}$$

Here is the energy difference between the two points:

$$\frac{E_1 - E_2}{E_1} = \frac{\Delta E}{E_1} = \left[\frac{(h_1 + \frac{V_1^2}{2g}) - (h_2 + \frac{V_2^2}{2g})}{(h_1 + \frac{V_1^2}{2g})}\right] \tag{2}$$

and percent energy loss,

$$\% \frac{\Delta E}{E_1} = \left(\frac{E_1 - E_2}{E_1}\right) x100 \tag{3}$$

where E_1 and E_2 are the energy dissipation values calculated at the upstream and downstream points, respectively.

2.1. Standard k-ε turbulence model

In this section, the turbulence model used in the study is explained. Reynolds averaged for the velocity components:

$$u_i = \bar{u}_i + u_i' \tag{4}$$

Where $\bar{u}_i$ and u_i' are the mean and fluctuating velocity components. Substituting expressions of this form for the flow variables into the instantaneous continuity and momentum equations and simplifying (and dropping the over bar on the mean velocity, $\bar{u}$):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = 0 \tag{5}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}(-\overline{\rho u_i' u_j'}) \tag{6}$$

where equations 1 and 2 are called Reynolds Averaged Navier-Stokes (RANS) equations that $-\overline{\rho u_i' u_j'}$ is called Reynolds stresses, must be modeled by using the Boussinesq hypothesis [16] relate the Reynolds stresses to the mean velocity gradients:

$$-\overline{\rho u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{7}$$

The Boussinesq hypothesis is used in the k-ε models. In the present work the Standard k-ε model Launder and Spalding [17] was used to simulate the turbulence phenomenon. For modeling the effective viscosity:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{8}$$

where C_μ is a constant, *k* is the turbulence kinetic energy, and *ε* is the turbulence rate of dissipation. The transport equations for the Standard *k-ε* model are as follow:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M \tag{9}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{10}$$

Standard constants of *k-ε* model are listed in Table 2 and were used in the model.

Table 2. Standard k-ε turbulence model constants.

	$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_μ	σ_k	σ_ε
Standard *k-ε*	1.44	1.92	0.09	1.0	1.3

In the study, the volume of fluid (VOF) method was preferred for the water-air interface section. The VOF method was employed as a powerful computational tool for the analysis of free surface flow [18].

The tracking of the interfaces between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. In the VOF method, position of the free-surface is tracked by solving a transport equation for the scalar variable f, which represents the volume fraction in a computational cell:

$$\frac{\partial f}{\partial t} + \frac{\partial u_i f}{\partial x_i} = 0 \tag{11}$$

Computational cell is tagged as fluid cell when f = 1, empty cell when f = 0 and free-surface cell when 0 < f < 1 during numerical simulations.

In this study, Tetrahedron mesh method was used as Patch Conforming Method. Mesh structure was enlarged with 1.2 Growth Rate value from channel base to the top ceiling level. Thus, gain was provided for solution time originating from mesh number. Solutions were realized with coarse, medium, and fine mesh structures for optimum mesh structure. Numerical model verification is done using experimental data. For this purpose, numerical model verification was done using experimental data obtained by Kaya [19] (Figure 2).

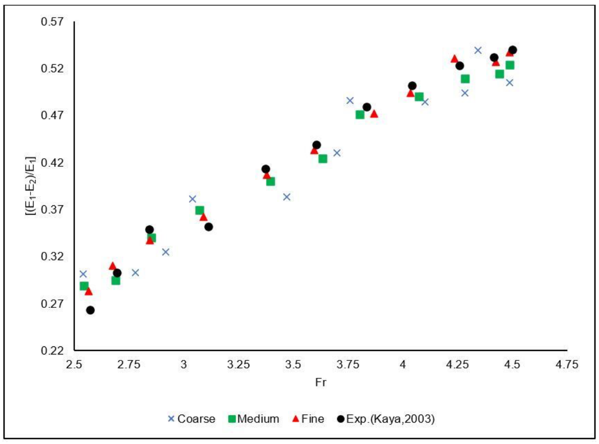


Figure 2. Numerical validation of the dimensionless Froude number and energy dissipation

In the model validation, the maximum error between the laboratory results and the CFD results was found to be 5.2% for the coarse mesh structure, 1.4% for the medium mesh structure, and 0.6% for the fine mesh structure. In this case, to obtain a more precise result, the fine mesh domain was taken into account for this study. Other boundary conditions and mesh structure of the model are given in Figure 3.

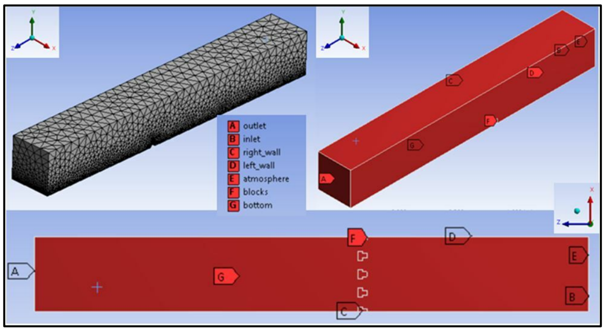


Figure 3. Boundary conditions of channel mesh structure and geometry

Here, the channel inlet is defined as “velocity inlet”, the channel outlet and the channel ceiling outside the flow are defined as “pressure outlet”, and the channel bottom, channel side walls, and energy dissipating blocks are defined as “wall”.

3. Result and Discussion

In this study, firstly, T-shaped blocks were arranged in a single row and double rows in the channel. Here, the blocks were arranged in a way that their total width covered 50% of the channel cross-section, and analyses were performed according to the Standard k-ε turbulence model [2]. To determine the energy dissipation status according to different block arrangements, the flow and velocity depth were measured numerically at 20 cm intervals starting from 60 cm upstream of the blocks. These values were measured at 5 cm intervals in the section where the blocks were located and measurements were made again at 20 cm intervals downstream of the block. The flow velocity, depth, and energy dissipation values obtained are given in Table 3.

When Table 3 is examined, the highest energy dissipation was obtained with an average of 50.26% in the single-row block array. On the other hand, less energy was dissipated with an average of 45.94% in the double-row array. The reason for this is that the water level increased slightly upstream in the double-row array and the flow velocity increased after the blocks. Therefore, it caused less energy to be dissipated. In addition, the energy dissipation values in the sections taken from the channel bank to the center in both array shapes decreased. While the flow depth on the bank was higher in the upstream, the flow velocities in the channel center were higher in the downstream.

The energy dissipating values obtained were measured in five different routes along the X-Z section. They are given in Figure 4. Here, the highest energy dissipating values were obtained for the single-row arrangement.

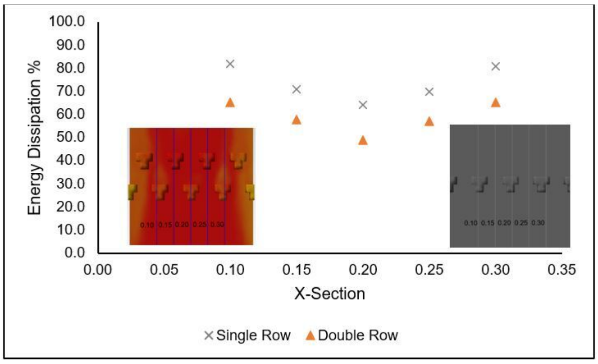


Figure 4. Energy dissipation values according to single row and double row arrangement

In Figure 5, velocity vectors are given according to single-row and double-row arrangement. Here, the water surface profile is shown in the channel center axis. As seen in the double-row arrangement, velocity vectors reach higher values downstream of the blocks.

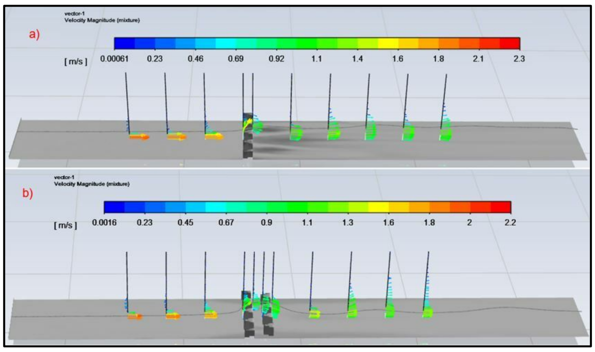


Figure 5. Velocity vectors and water surface profile according to a) Single row b) Double row block arrangement

Table 3. Flow velocity, depth and energy dissipation values measured along the X-Z Section

	X-Z Section	h_1(m)	V_1(m/s)	h_2(m)	V_2(m/s)	E_1	E_2	$[(E_1-E_2)/E_1]*100$
Single Row Arrangement	0.10	0.0179	1.9487	0.0586	0.8261	0.2114	0.0934	55.8354
	0.15	0.0176	1.9483	0.0530	1.0485	0.2111	0.1090	48.3435
	0.20	0.0178	1.9247	0.0525	1.1509	0.2066	0.1200	41.9166
	0.25	0.0171	1.9589	0.0523	1.0690	0.2127	0.1105	48.0221
	0.30	0.0181	1.9512	0.0542	0.8483	0.2121	0.0909	57.1634
							average	**50.2562**
Double Row Arrangement	0.10	0.0185	1.8499	0.0364	1.0879	0.1929	0.0967	49.8620
	0.15	0.0176	1.9210	0.0327	1.2451	0.2057	0.1117	45.6769
	0.20	0.0174	1.8372	0.0294	1.3124	0.1894	0.1172	38.1460
	0.25	0.0181	1.9045	0.0315	1.2482	0.2030	0.1109	45.3594
	0.30	0.0183	1.8932	0.0356	1.1171	0.2010	0.0992	50.6395
							average	**45.9367**

The velocity contours for both alignment shapes according to the YZ and ZX axes are given in Figure 6. In the YZ plan, velocity contours for x=0.2 m, and in the ZX plan, velocity contours for y=0.02 m are given.

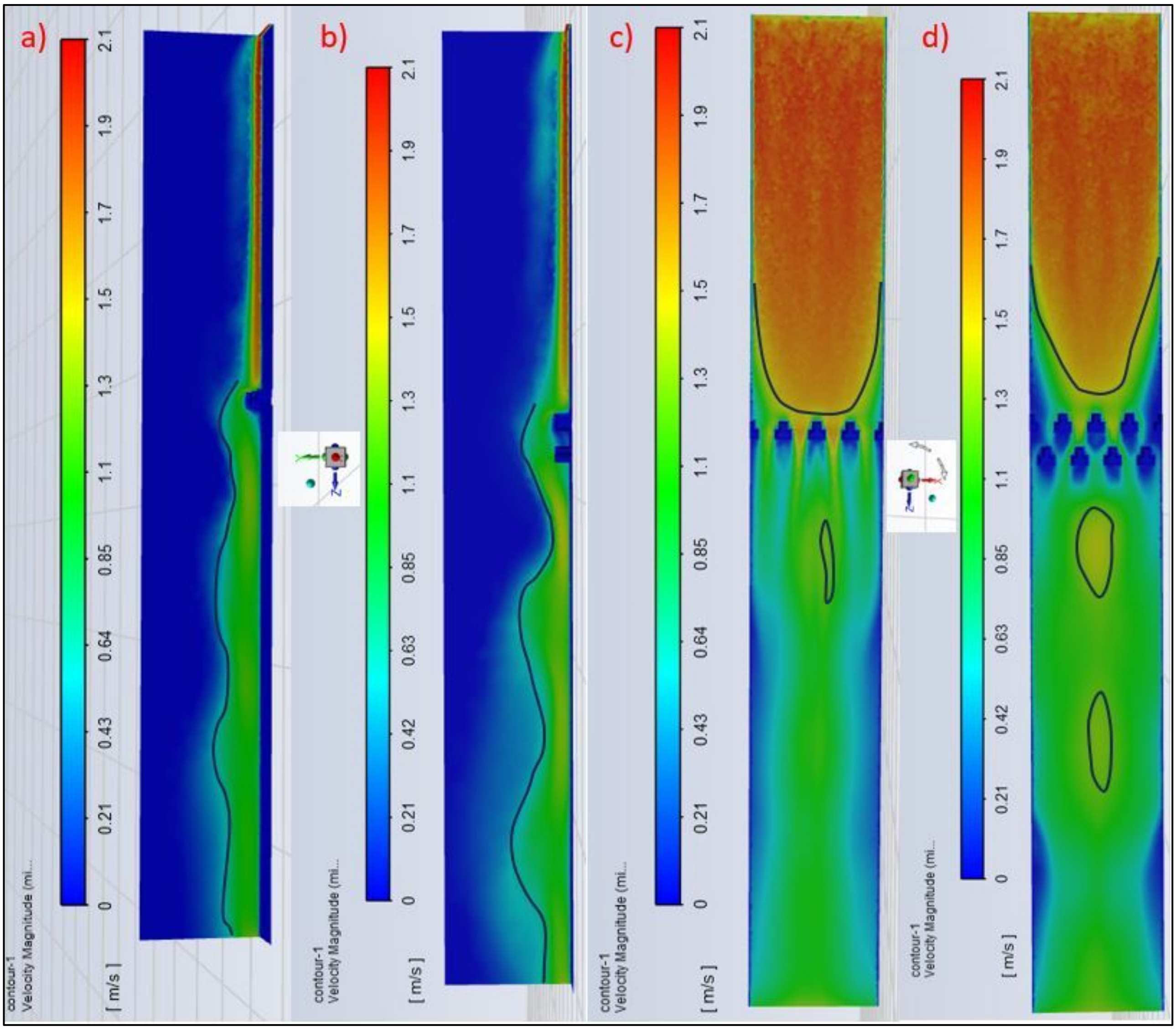


Figure 6. a) and b) velocity contours for x=0.2 m in single-row and double-row YZ plan, respectively; c) and d) velocity contours for y=0.02 m in single-row and double-row ZX plan, respectively.

When a) and b) are compared in Figure 6, there are no major fluctuations on the flow surface after the blocks in the single-row arrangement in the channel center axis. The flow velocity has decreased, but more fluctuations have occurred on the flow surface in the double-row arrangement. This situation shows that the flow has accelerated somewhat after the blocks in the double-row arrangement. This situation also supports cases c) and d). In option c), the blocks arranged in a single row have reduced the flow velocity upstream and the flow velocity has remained at a very low value downstream of the blocks. Again, the magnitude of the flow velocity contours in the upstream has taken a U shape, and the flow velocity between the blocks has reached values partially close to the upstream values. In option d), the blocks arranged in two rows have reduced the flow velocity starting from the immediate front of the blocks. The magnitude of the flow velocity contours in the upstream have taken a V shape, and the flow velocity between the blocks has remained at values lower than the upstream values. However, the flow velocities downstream of the blocks have reached higher values compared to the single-row arrangement. The reason for this, as Kaya [19] stated in his study, is that in double-row energy dissipator blocks, the speed of the flow increases slightly after the second row of blocks, since the flow depth on the upstream side increases. Therefore, the energy dissipator rates are less than single-row energy dissipator blocks. Figure 7 shows the turbulent kinetic energy values along the Z direction for single and double-row block arrays, which is an indicator of energy dissipation.

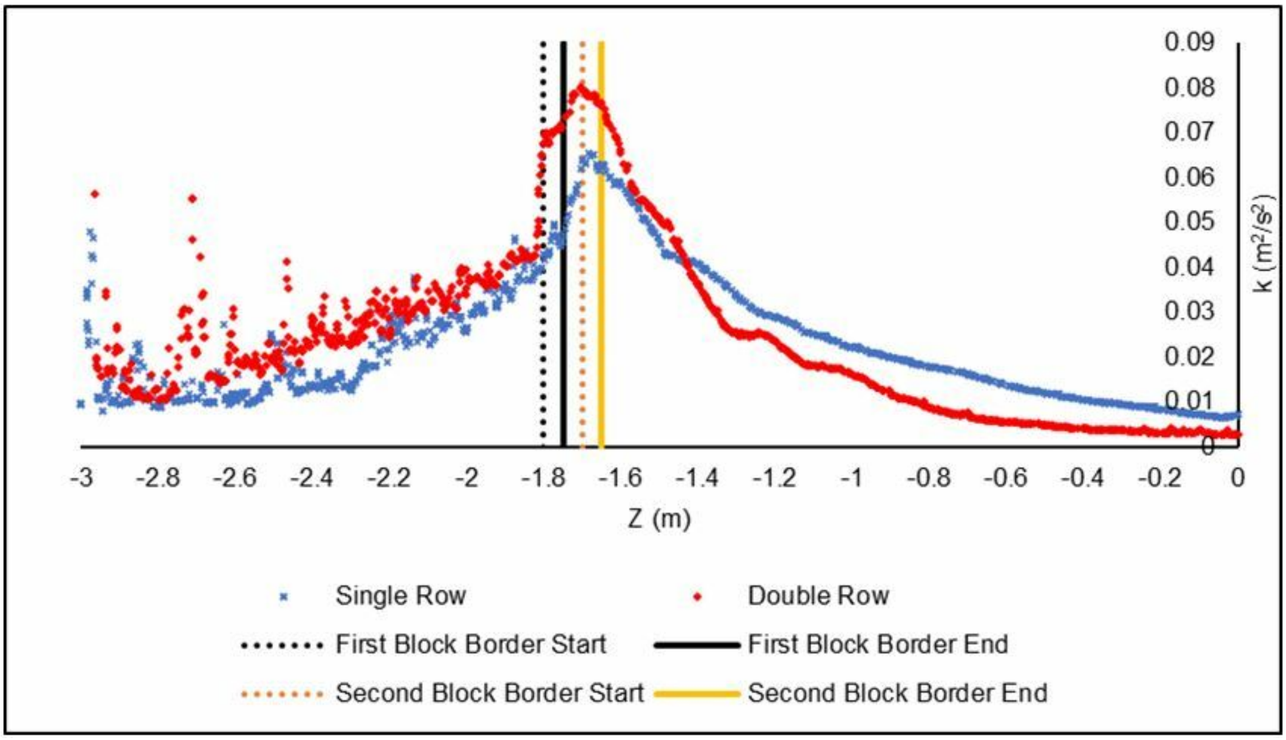


Figure 7. Variation of turbulent kinetic energy (k) values along the Z direction in single-row and double-row arrays

In Figure 7, turbulent kinetic energy (*k*) values were obtained according to single-row and double-row block arrangements along the Z axis, which is the 3 m channel direction. The high turbulent kinetic energy value means that more energy is absorbed. According to the figure, the flow enters the channel from the -3 m point. The position where the first block row is placed is between -1.8 and -1.75 m. In the double-row arrangement, the second block row is placed 5 cm downstream. Accordingly, since higher (*k*) values are obtained in the double-row arrangement upstream of the blocks, we can say that more energy dissipation has occurred. This also means that a higher flow depth is obtained in the double-row arrangement upstream of the blocks. However, the exact opposite situation occurred with the flow passing to the downstream of the blocks. Approximately after the -1.4 m point, higher (*k*) values are obtained in the single-row block arrangement. In this case, higher energy dissipation values were obtained for the single-row block array.

4. Conclusions

In this study, analyses were performed according to the Standard k-ε turbulence model. The distance between the blocks was taken to be equal to the block width, and the analyses were done. When the energy dissipation conditions were examined:

• The single-row block array has dissipated more energy than the double-row block array.

• The flow depth has increased in the upstream part of the double-row array. This situation has caused an increase in the flow velocity after the blocks

• When the velocity vectors and water surface profiles were examined, more flow surface fluctuations and high-velocity vector values were obtained in the downstream part of the double-row array.

• The turbulence kinetic energy value, which could not be measured experimentally, was measured in the numerical solution for both arrays. The high value of this value means that more energy is dissipated. Therefore, it can be said that more energy is dissipated in the single-row block array.

• Since the energy-dissipation blocks calm the flow downstream, these blocks eliminate the scouring problem.

This study has enabled many parameters that cannot be measured experimentally to be obtained in a 3D digital environment. The interaction of the flow with the blocks and the flow dynamics have gained a more economical visual. It is thought that this study will contribute greatly to open channel, stilling basin, and chute channel applications.

References

[1] **Yalçın, O.,** (1967). Barajlarda Enerji Kırıcı Tesislerin Projelendirilmesi. TC Enerji ve Tabii Kaynaklar Bakanlığı Devlet Su İşleri Genel Müdürlüğü *Barajlar ve Hidroelektrik Sanraller Dairesi Başkanlığı*, Ankara, 23s.

[2] **Blaisdell, F. W.** (1948). Development and hydraulic design, Saint Anthony Falls stilling basin. *Transactions of the American Society of Civil Engineers*, 113(1), 483-520.

[3] **Basco, D. R.,** & Adams, J. R. (1971). Drag forces on baffle blocks in hydraulic jumps. *Journal of the Hydraulics Division*, 97(12), 2023-2035.

[4] **Mohamed Ali, H. S.** (1991). Effect of roughened-bed stilling basin on length of rectangular hydraulic jump. *Journal of Hydraulic Engineering*, 117(1), 83-93.

[5] **Rajaratnam, N.,** & Hurtig, K. I. (2000). Screen-type energy dissipator for hydraulic structures. *Journal of hydraulic engineering*, 126(4), 310-312.

[6] **Çakir, P. I. N. A. R.** (2003). Experimental investigation of energy dissipation through screens. *Master Thesis Ankara*.

[7] **Aydoğdu, M.**, & Dursun, Ö. F. (2020). Şüt Kanalları Üzerine Yerleştirilen Blokların Enerji Sönümlenme Etkilerinin Araştırılması. *Su Kaynakları*, *5*(2), 34-39.

[8] **Aydoğdu, M.** (2024). Testing the Reliability of Numerical Model Studies for a Broad-Crested Weir. *Türk Doğa ve Fen Dergisi*, *13*(3), 73-81.

[9] **Gül, E.**, Kılıç, Z., İkincioğulları, E., & Aydın, M. C. (2024). Investigation of the effect of variable-sized energy dissipating blocks on sluice gate performance. *Water SA*, *50*(1), 92-105.

[10] **Ikinciogullari, E.** (2023). Stepped spillway design for energy dissipation. *Water Supply*, *23*(2), 749-763.

[11] **Sibale, D.**, Adongo, T. A., Umkiza, E., Ntole, R., Chikavumbwa, S., Bwambale, E., & Jeremaiho, Z. (2023). Review of hydraulic performance of open-channel flow-measuring flumes. *Acta hydrotechnica*, *36*(64), 31-55.

[12] **Xiong, W.**, Cai, C. S., Kong, B., & Kong, X. (2016). CFD simulations and analyses for bridge-scour development using a dynamic-mesh updating technique. *Journal of computing in civil engineering*, *30*(1), 04014121.

[13] **Aydın, M. C.**, İkincioğulları, E., & Emiroğlu, M. E. (2018). Şütlerdeki Enerji Kırıcı Blokların Akımın Hidrolik Karakteristikleri Üzerine Etkisinin Sayısal Analizi. *Su Kaynakları*, *3*(1), 9-15.

[14] **Peterka, A. J.** (1964). *Hydraulic design of stilling basins and energy dissipators* (No. 25). United States Department of the Interior, Bureau of Reclamation.

[15] **Niyazi, H.**, Yaban, H., & Demirel, E. (2022). Structure and dynamics of the turbulent flow through a central baffle. *Flow Measurement and Instrumentation*, *88*, 102248.

[16] **Hinze,** 1975 J.O. Hinze Turbulence McGraw-Hill Inc, New York (1975)

[17] **Launder, B. E.**, & Spalding, D. B. (1983). The numerical computation of turbulent flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion* (pp. 96-116). Pergamon.

[18] **Hirt, C. W.**, & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of computational physics*, *39*(1), 201-225.

[19] **Kaya, N.** (2003). Enerji kırıcı havuzlarda farklı tip enerji kırıcı blokların enerji sönümleme oranlarının incelenmesi.