Investigation the Effect of Trapezoidal Blocks on Hydraulic Jump Characteristics in the Stilling Basin with Flow 3D Model

Amir Hajizadeh Fallah

Master of Water and Hydraulic Structures

Mramir.info@gmail.com

Elnaz Rostamzadeh Renani

Civil Engineer Student

elnazroze@gmail.com

Abstract

A Stilling Basin or water jump pool is a short section of a foamed channel that is constructed as a special structure at the end of overflows or any other source that creates a supercritical flow and the purpose of making it is to form a hydraulic jump inside the pond, in which case the supercritical current, before reaching the non-foamed parts of the river, changes to subcritical current and reduces its extraordinary energy and possible breakdowns is prevented. Any changes in the structure of the relaxation pond directly affect the hydraulic behavior of the flow. The shape and structure of the Stilling Basin and the shape and geometry of the blocks play an important role on the hydraulic performance and fluctuations in flow velocity and jump length in the Stilling Basins. Considering the importance of Stilling Basin blocks and their significant effect on energy dissipation, and considering that various researches have been done on the hydraulic properties of the jump and consequently the characteristics of the Stilling Basins. However, no research has been done on changes in flow velocity and jump length in Stilling Basins with trapezoidal blocks under constant hydraulic conditions. Therefore, in this paper, the length of the hydraulic jump in the Stilling Basin with trapezoidal blocks has been investigated numerically using the Flow-3D model.

Keywords: Hydraulic jump, Stilling Basin, Trapezoidal blocks, Flow-3D model

❖ Preface

Due to the limited and definite volume of the reservoir of dams, the occurrence of floods may lead to the completion of the maximum capacity of the reservoir and the passage of current through any possible channel, including the dam crown. Water vapor, even if it does not cause heavy damage, creates undesirable phenomena due to unreliable and possibly uncontrollable hydraulic conditions. Therefore, it is necessary to discharge the excess volume of inlet flow to the tank in a predictable manner and under safe hydraulic conditions. This important task in dams is the responsibility of overflows.

Due to the high level of overflow compared to the riverbed, rising water levels cause a significant increase in energy in the outflow stream, which has a high (relatively) destructive potential. Therefore, it is necessary to consume this added energy in a way that in the overflow of dams, this task is the responsibility of energy consumers.

Attention to the effective performance of hydraulic jump in the direction of energy dissipation in the Stilling Basin, provides the basis for further investigation of the characteristics of this phenomenon. The presence of equipment such as middle blocks in such ponds will lead to more energy dissipation and reduction of secondary depth of hydraulic jump. The extent of this effect depends on factors such as the shape of the blocks, the height of the blocks, and the amount of opening between the blocks.

❖ Necessity of Research

The purpose of constructing these structures to form hydraulic jumps inside the pond is to convert the supercritical current to subcritical current before reaching the non-foamed parts of the river and to reduce its extraordinary energy by the floor blocks to Erosion of the river banks downstream of the dam and damage to agricultural lands and damage to facilities and structures along the river and other human and financial damage to prevent.

❖ Research Purposes

• Investigation of changes in flow velocity in the pond by applying flow rates.

• Comparison of hydraulic jump length change in Stilling Basin with trapezoidal blocks.

• Comparison of flow velocity changes and hydraulic jump length to reduce pond length and reduce costs

❖ Research Hypotheses

• The length of the hydraulic jump will depend on the flow rate.

• Simulation of flow in Stilling Basin is done with trapezoidal blocks with landing number 13.

❖ Introducing Flow-3D Software

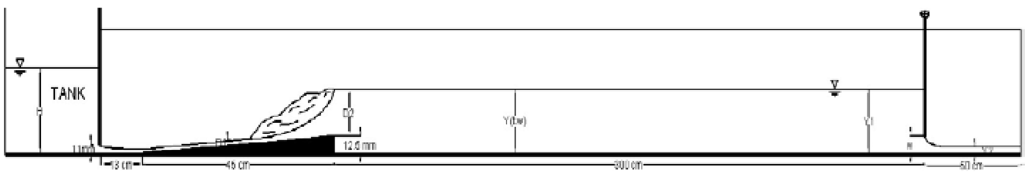
FLOW3D software for physical models solves fluid mechanics and hydrodynamics problems. This 3D software simplifies modeling and analysis of fluid behavior by using numerical methods. So much so that there is no need to use advanced modules, spend long periods of time, or use supercomputers to analyze three-dimensional models in flotation.

Flow-3D software is versatile software and compatible with complex flow conditions in two-dimensional and three-dimensional modeling. This software is dedicated to computational fluid dynamics (CFD) and is provided by Flow Science. The method of solving equations in this software is based on the finite volume method.

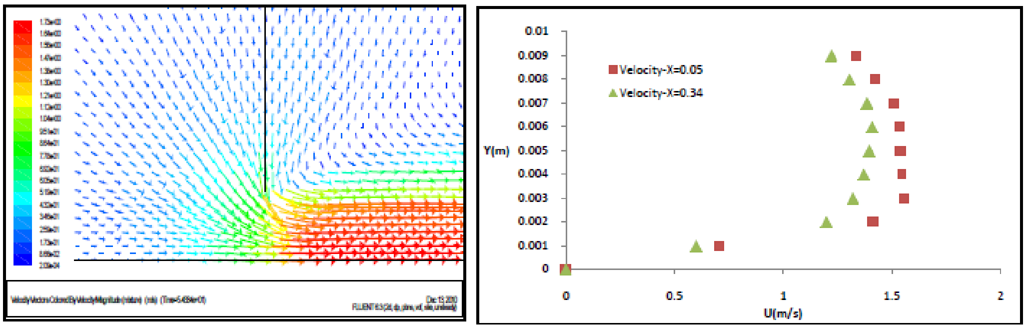
❖ History of Research

So far, studies have been conducted on the flow pattern, flow characteristics and jump in the Stilling Basins, some of which are mentioned below:

Asefi and Ziaei (2011) in a study performed two-dimensional numerical simulation of hydraulic jump on reverse ramps with stairs at the end with FLUENT software. In this paper, the flow pattern inside these ponds was simulated in two dimensions using Fluent software which solves the flow equations by finite volume method. The computational field was discretized using regular prismatic networking. In this software, the fluid volume method was used to model the free surface. The standard k-e turbulence model of standard and non-equilibrium wall functions was used, which produced better results in validating the jump level profile of the standard wall function mode. To reduce the error in simulating the initial velocity profile before jumping, the upstream water tank was simulated. Finally, using numerical results, water surface profiles and kinetic energy changes along the pond and after the stairs were investigated.

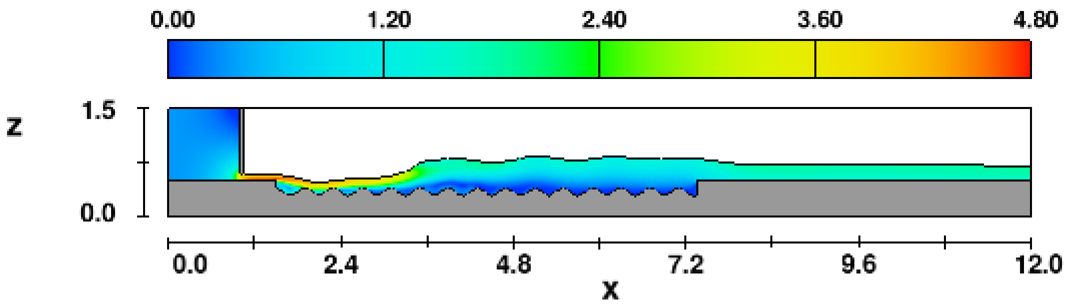


Outline of Asefi and Ziaei laboratory model (2011)



Flow conditions under the upstream valve (left) and depth velocity profile in two sections immediately after the valve and before the jump (right)

Tabatabai et al. (2014), in a study, numerically investigated the flow pattern in a relaxation pool with a sinusoidal bed with Flow3D model. The results showed that the relative length of the Ghaltab region increases by about 30 to 40% by increasing the flow landing number and by comparing the landing number of 4.2 and the landing number 11, which are the lowest and highest initial landing numbers of the simulations, respectively. By increasing the height of the blocks or increasing the height to distance ratio of the sinusoidal blocks, the hydraulic jump and full energy dissipation of the jet flow out from under the valve at a shorter length than under conditions where the height of the blocks is low, and the Two-equation (k-e) turbulence model has good accuracy in simulating the output turbulence of the outlet under the valve and the hydraulic jump in the Stilling Basins with sinusoidal blocks.



Display of hydraulic jump speed distribution in Stilling Basin with sine blocks and landing number 11

Vali Nia et al. (2019) in a study investigated the effect of floor blocks in the downstream relaxation pond on the geometry of the downstream scour cavity. IN this study, 35 experimental tests were performed in the range of changes in the distance of floor blocks from the valve, landing number and relative depth of the bottom.Dimensional analysis was used to investigate the effect of each of the effective parameters. The results showed that the use of floor blocks deforms the scour hole. Also, with increasing the distance of the block from the valve, the dimensions of the scour hole decreased and from a distance onwards, the shape of the scour hole also changed. By increasing the dimensionless distance of the block from the valve from 0.18 to 0.54, the volume of the scour hole decreased by 85%. The volume of the scour hole is directly related to the landing number; But there is no definite trend towards changing the depth of the relative tributary. In the dimensionless distances of 0.36 and 0.54 blocks, the lowest volume of scour water hole was located in the relative water depth conditions of 6.3.

❖ Materials and Methods

Since the present study was performed using Flow-3D software and also this research is using the physical characteristics of a laboratory flume, it is necessary to provide sufficient knowledge of this software and the relevant flume.

Mathematical models are among the most powerful tools in solving complex equations related to fluid mechanics. Today, with the increasing speed of computers, the use of these models has expanded significantly. One of the advantages of mathematical models over physical models is that they are less expensive, while various changes such as changes in the geometry of the structure are easily possible in these models. One of the most common mathematical models in the world that is used to solve complex problems related to hydraulic structures is Flow-3D software, which has the ability to solve three-dimensional hydraulic phenomena and the matching of the results of this software with the results of hydraulic model experiments in A number of projects under review have increased confidence in the use of this computer program. This software is versatile and compatible with complex flow conditions in two-dimensional and three-dimensional modeling. This software is dedicated to computational fluid dynamics and the method of solving equations in this software is based on the finite volume method.

The experiments were performed in a channel with length, width and height of 12, 0.25 and 0.5 meters in the hydraulic laboratory of the Faculty of Agricultural Engineering and Technology, University of Tehran, respectively. Degrees were examined and conditions of equal height for trapezoidal blocks will also be considered. In total, the experiments were performed in the range of landing number 13.

Hydraulic jump is a phenomenon that causes energy dissipation downstream of hydraulic structures. Stilling Basins are a good place to create, control and inhibit hydraulic jumps, and the blocks inside these ponds cause the jump to settle inside the pond and deplete some of the kinetic energy of the hydraulic jump, thus increasing the efficiency of the relaxation ponds. To increase the efficiency of relaxation ponds and reduce the length of the hydraulic jump, rough appendages are usually used at the bottom of the ponds.

The conditions introduced to the software are such that in order to investigate the effect of integrated blocks on hydraulic jump, flow rate and deformation of the blocks, the experimental variables of this research will be formed.

The data used in the software are obtained from a laboratory model that has been tested. The dimensions of this laboratory model are as follows.

The experiments in the canal with length, width and height are 12, 0.25 and 0.5 meters, respectively. Selected trapezoidal roughnesses with a fixed crown width of 2 cm and a vertex angle of 45 ° were examined. In total, the experiments were performed in the range of landing numbers from 4 to 13.7. In order to provide static height of water to create the desired landing numbers, 1.2 m from the inlet of the flume, using Plexiglas, was increased to a height of 1.1 m, and to create supercritical flow and initial jump depths, a sliding valve made of Plexiglas was installed at the beginning of the canal.

❖ Simulation Input Information

The first parameter that must be entered into the software is the time parameter. The General section of the software allows the user to determine how to complete the problem solving.

The Finish Time option, which is our choice in this case, terminates the solution when our allotted time expires. The time allotted for solving the problem is 120 seconds.

Also in this section we change the unit system to SI in the Units section of CGS (which is set by default for all units).

❖ Gravity, Viscosity, Perturbation Model:

First, we enter the force of gravity with the Gravity option. In this problem, we determine the amount of gravity.

Then we enter the Viscosity and turbulence option and in the Viscosity option section, because the water is Newtonian fluid, we select the Newtonian viscosity option.

Now we need to define the Turbulence Model:

According to the application of different turbulence models, Renormalized group (RNG) model. It is the best option to solve this problem and models the turbulence conditions with a good approximation.

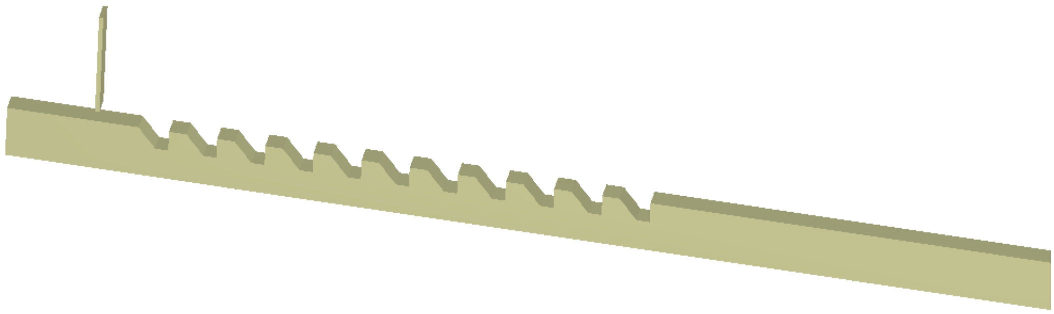
❖ Physical Characteristics of Fluid, Unit System:

In the Fluids section, we must specify the type of fluid. In the Fluid Database section, many items are placed by default. Considering that one of the conditions of the fluid standard for conducting experiments in normal conditions of a river or canal is water with a temperature of 20 degrees, we choose this option which has a viscosity of 0.001 and a density of 1000.

❖ Model Geometry

Flow-3D software and Autocad software were used to model the desired laboratory flume at the same time.

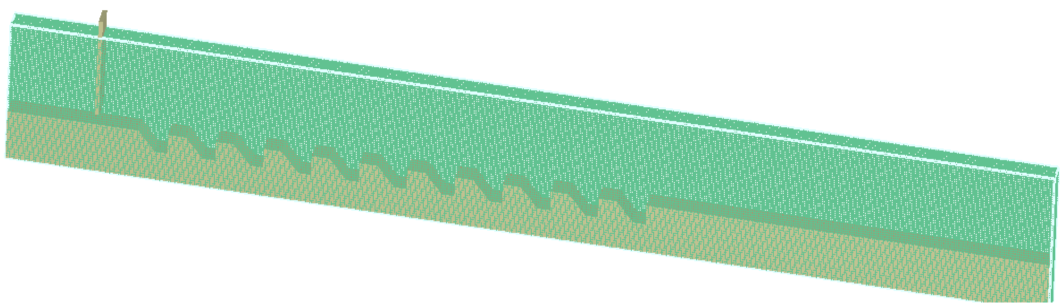
Due to the fact that Flow-3D software does not have the ability to execute AutoCAD files with Cad extension, the flume made in Autocad was converted to STL format, and then in the Meshing Geometry section, we select the desired STL file and add it to the program.



Displays the geometry of a trapezoidal block in the Flow-3D model

❖ Meshing

Due to the fact that different valve edges and roughnesses are very important, a large number of mesh cells were used to mesh the model, and since increasing the number of mesh cells has a direct effect on increasing the time of each software run, we tried to Achieve a certain amount of mesh cell size and number to be considered both accurately and simultaneously. It should be noted that these mesh divisions have no effect on the flow lines and are used only for meshing.

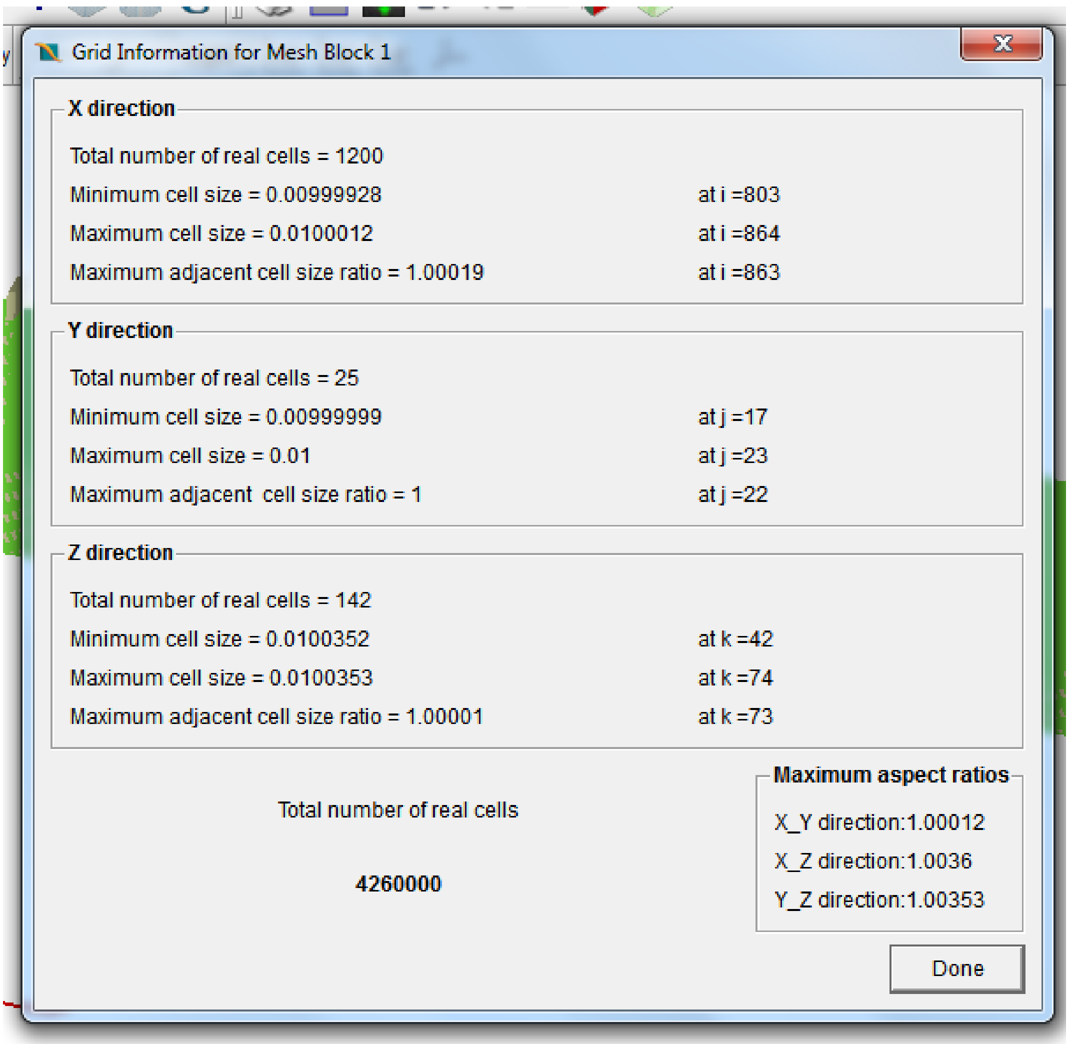


Trapezoidal block mesh model in Flow-3D

✓ Meshing was performed by defining 435,000 mesh cells:

Flow-3D software allows the user to determine the number of meshes in the three directions X, Y, Z by determining the number of meshes by two parameters Maximum adjacent cell size ratio and Maximum Aspect Ratios in different directions. These two parameters are located in the Info section of the Meshing section. The Maximum adjacent cell size ratio parameter shows the maximum adjacent cell size ratio in three directions X, Y, Z. According to the software instructions to increase the accuracy of problem solving and reduce errors due to meshing, the value of this parameter should be less than 1.25 in all three directions X, Y, Z. The Maximum Aspect Ratios parameter gives the maximum ratio of pages to each other. The value of this parameter according to the software instructions should be less than 3 for all three ratios X-Y, X-Z, Y-Z.

➢ According to the above parameters, the accuracy of the ratios was examined:



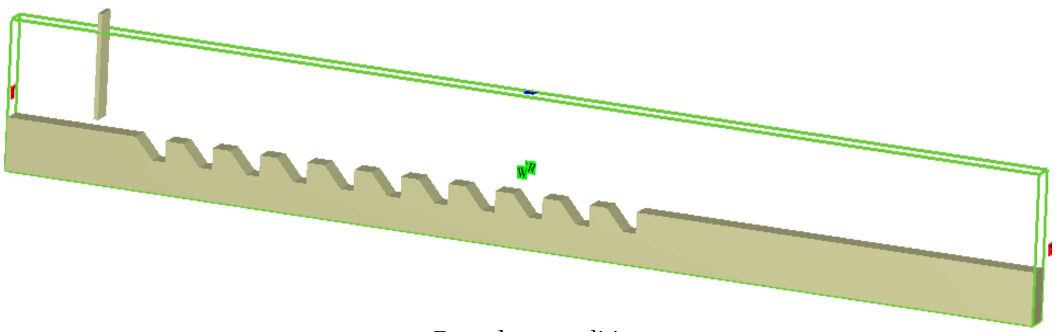
Mesh information page

- ✓ In the end, in order to reach the above values by trial and error and by changing the value of the meshes, the above parameters reached the standard level. At this stage, several different meshes were used as trial and error to finally achieve a desirable meshing suitable for the test conditions.

❖ Boundary conditions

In the Boundaries section, the boundary conditions of the problem were identified:

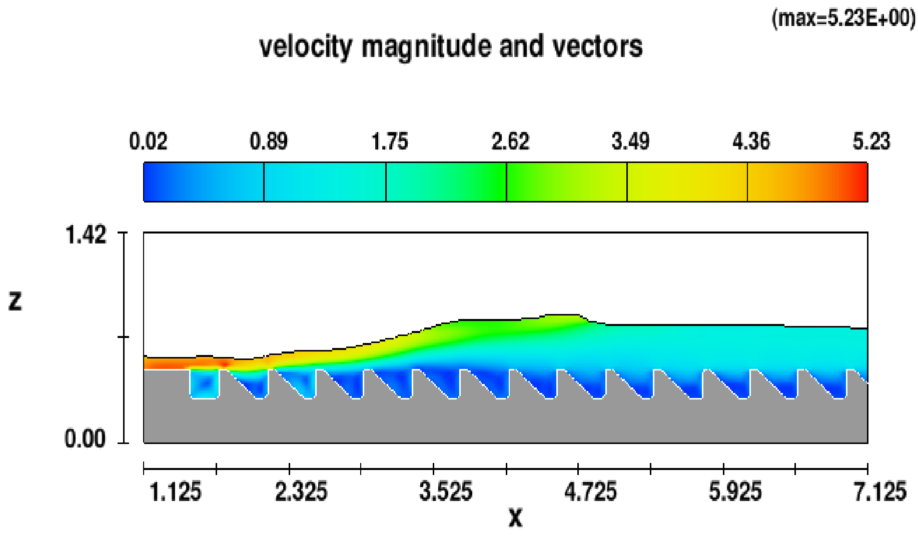
The input limit was set to Volume flow rate. In this part, the input flow rate was determined. The output border was also considered as Outflow. In Outflow conditions, all current that reaches this limit is output without any change in flow conditions. The lower border and the left and right borders were selected as wall type. The wall or wall boundary conditions are used to separate the fluid from the solid boundary. In this boundary condition, the vertical and tangential velocities on the zero wall boundary are considered and the upper wall was selected as symmetry.



Boundary conditions

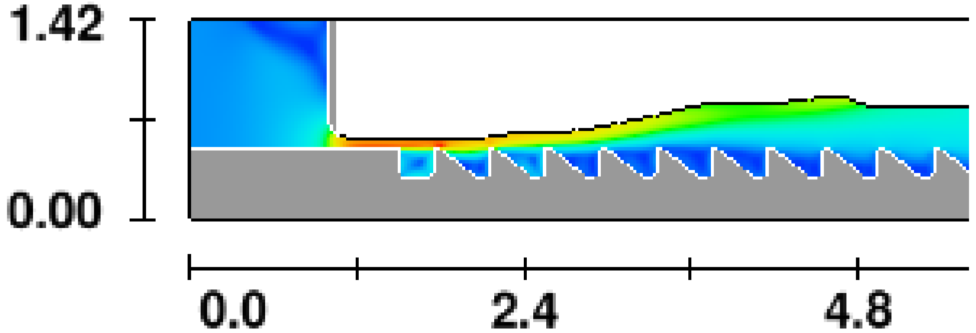
❖ Investigation of Hydraulic Jump Formed Inside Stilling Basin

According to the figure below, which is related to the outflow of the flow under the valve and the formation of a hydraulic jump, it can be seen that the trapezoidal blocks located in the Stilling Basin calm down the flow out of the valve, which enters with great intensity and speed. They become pond, wear out and play an effective role in reducing the energy output under the valve and by creating a hydraulic jump, they deplete the full flow energy before the flow reaches the 8 or 9 pond blocks and the middle of the pond.

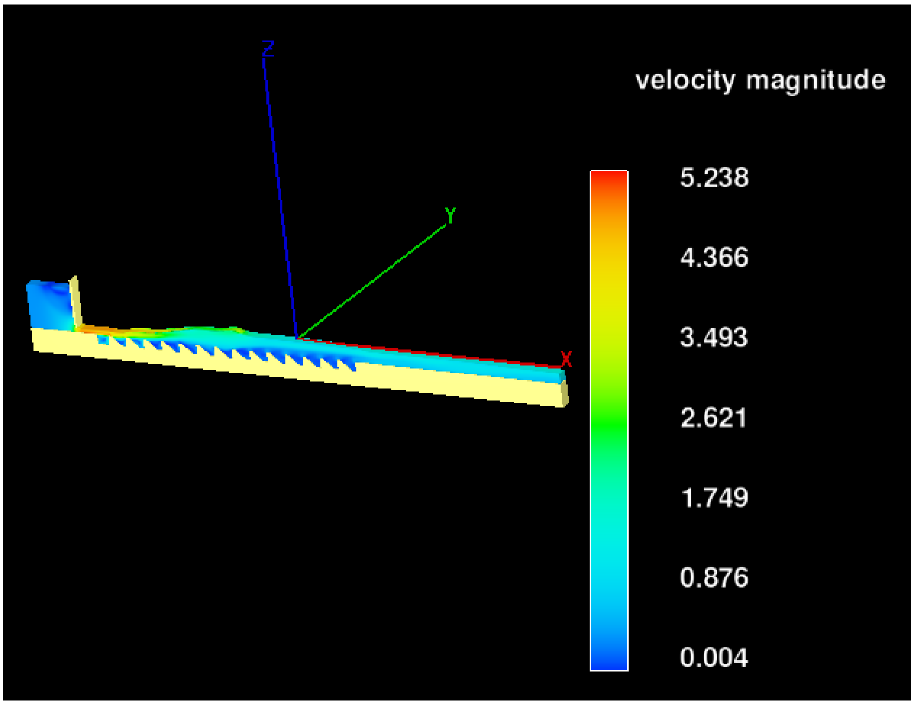


The shape of the hydraulic jump formed inside the Stilling Basin Fr = 13

- ✓ The flow velocity is initially about 5 m / s by exiting the valve, and by colliding with the blocks and forming a hydraulic jump, this velocity decreases to 1 m / s in the case of the Stilling Basin with vertical trapezoidal blocks, reducing by 80%. There is a flow velocity that shows the high role of the pond and blocks in reducing the velocity and depletion of the flow.



The shape of the flow passing under the sharp edge valve and creating a hydraulic jump



Three-dimensional image of the flow passing under the valve with a trapezoidal block

❖ Conclusion

• The effect of trapezoidal blocks in reducing flow rate by 80%.

• Create a hydraulic jump and change the flow rate from 5 meters per second to one meter per second by creating trapezoidal blocks.

• Flow control under the valve in less than 5 meters, which shows the excellent performance of trapezoidal blocks in flow control.

❖ Offers

In this research, the characteristics of hydraulic jump in ponds with trapezoidal blocks were investigated, and it is suggested that blocks with other geometries be examined for this purpose.

In this research, the hydraulic jump in the ponds was simulated using the Flow-3D model and the Renormalized group (RNG) turbulence model. It is suggested that other turbulence models of this software be compared.

❖ Resources

1. Abasi Chenari, S. and Kamanbedast, A. and Ahadian, J. (2011). “Numerical Investigation of Angle and Geometric of L-Shape Groin on the Flow and Erosion Regime at River Bends”, World Applied Sciences Journal 15 (2): 279-284, 2011, ISSN 1818-4952.
2. Farhoudi, J. and Sadat Helbar, S. M. and Aziz, N. (2010) "Pressure Fluctuation around Chute Blocks of SAF Stilling Basins" J. Agr. Sci. Tech. 2010; Vol. 12: 203-212.
3. Zhao, Q. and Misra, S. (2009). Numerical Study of a Turbulent Hydraulic Jump. ASCE Engineering Mechanics Conference, University of Delaware, Newark, DE.
4. Gohari A., and Farhoudi J. (2009). The characteristics of hydraulic jump on rough bed stilling basins. 33rd IAHR Congress. Water Engineering for a Sustainable Environment, Vancouver, British Columbia, pp. 1-9.
5. Abbaspour A., Hosseinzadeh Dalir A., Farsadizadeh D., and Sadraddini A.A. (2009). Effect of Sinusoidal Corrugated Bed on Hydraulic Jump Characteristics. Journal of Hydro-Environment Research, 3(2):109-117.
6. Pagliara, S. and Lotti, I. and Palermo, M. (2008). Hydraulic Jump on rough bed of stream rehabilitation structure" . J.of Hydro-environment research2,DOI: 10.1016/j.jher.2008: 29-38.
7. Debabeche, M. and Achour, B. (2007). Effect of sill in the hydraulic jump in a triangular channel. Journal of Hydraulic Research, 45(1), 135-139.
8. Omid,M.H and Esmaeeli Varaki,M. (2007). Gradually expanding hydraulic jump in a trapezoidal channel.Vol.45,No.4, pp. 512-518.
9. Wu, M., Chen, Y. and Zhou, Q. (2006). Case Study of an S-shape Spillway Using Physical and Numerical Models. Journal of Hydraulic engineering, 132(9), 892-898.