Journal of Engineering
journal homepage: www.joe.uobaghdad.edu.iq
Number 1 Volume 27 January 2021

Chemical, Petroleum and Environmental Engineering

A Computational Fluid Dynamics Investigation of using Large-Scale Geometric Roughness Elements in Open Channels

Iman Abdulsalam Alwan*
College of Engineering, University of Baghdad
e-mail: iirq979@gmail.com

Prof. Dr. Riyadh Z. Azzubaidi
College of Engineering, University of Baghdad
e-mail: riyadh.z.azzubaidi@coeng.uobaghdad.edu,iq

ABSTRACT

The hydraulic behavior of the flow can be changed by using large-scale geometric roughness elements in open channels. This change can help in controlling erosions and sedimentations along the mainstream of the channel. Roughness elements can be large stone or concrete blocks placed at the channel's bed to impose more resistance in the bed. The geometry of the roughness elements, numbers used, and configuration are parameters that can affect the flow's hydraulic characteristics. In this paper, velocity distribution along the flume was theoretically investigated using a series of tests of T-shape roughness elements, fixed height, arranged in three different configurations, differ in the number of lines of roughness element. These elements were used to find the best configuration of roughness elements that can be applied to change the flow's hydraulic characteristics. ANSYS Parametric Design Language, APDL, and Computational Fluid Dynamics, CFD, was used to simulate the flow in an open channel with roughness elements. CFD can be used to study the hydrodynamics of open channels under different conditions with inclusive details rather than relying on the costly field and time-consuming. Runs were implemented with different conditions, the discharge, and water depth in upstream and downstream of the flume. T-shape roughness elements with height equal to 3*cm* placed in three different configurations, two lines, four lines, and fully rough configurations were tested. The results show that the effect of roughness elements increasing with increasing the number of lines of roughness elements. Cases of four lines and fully rough configurations have almost the same hydraulic performance by having the same results of the velocity decrease percentage, which is decreased by approximately about 66% and 61% of the control case's velocity in the zone near the roughness elements consequently. But the difference is that four lines configuration is affected in a part of the test section. This behavior increases the velocity values by about 11% in the other side and by about 10% near the free surface in the case of four lines configuration and increased by about 32% above the roughness elements in a fully rough configuration.

*Corresponding author
Peer review under the responsibility of University of Baghdad.
https://doi.org/10.31026/j.eng.2021.01.03

Article received: 17/7/2020
Article accepted: 15/8/2020
Article published:1/1/2021

Keywords: APDL, CFD, Configurations, Roughness Elements, Velocity Distributions.

التحري الحسابي لديناميكية الموائع بأستخدام وحدات خشونة ذات حجم هندسي كبير في القنوات المفتوحة

***أيمان عبد السلام علوان**
كلية الهندسة- جامعة بغداد

أ.د. رياض زهير الزبيدي
كلية الهندسة- جامعة بغداد

الخلاصة

يمكن تغيير الخصائص الهيدروليكية للجريان بالاعتماد على عناصر الخشونة ذات الحجم الكبير في القنوات المفتوحة. يساعد هذا التغيير في السيطرة على التعرية والترسبات ضمن المجرى الرئيسي للقنوات. قد تكون عناصر الخشونة احجاراً كبيرة أو كتلاً خرسانية توضع في قعر القناة والتي تعمل على زيادة خشونة سطح القعر وينتج عنها اضافة مقاومة للجريان. يعد كل من الشكل الهندسي لعناصر الخشونة والعدد المستخدم وترتيبها عوامل لها تأثير في الخصائص الهيدروليكية للجريان. تم التحقق من توزيعات السرعة على طول القناة بأستخدام سلسلة من الاختبارات لعناصر خشونة على شكل حرف T ذو أرتفاع ثابت مرتبة في ثلاثة توزيعات مختلفة في عدد أسطر عناصر الخشونة وذلك لايجاد أفضل توزيع لعناصر الخشونة يستخدم لتغيير الخصائص الهيدروليكية للجريان. استخدم برنامج ANSYS Parametric Design Language APDL وبرنامج CFD Computational Fluid Dynamics لمحاكاة جريان الماء خلال القناة المفتوحة بوجود عناصر الخشونة. استخدام برنامج حساب ديناميكية الموائع لمحاكاة الجريان في القنوات المفتوحة CFD مع عناصر الخشونة تحت تأثير مختلف الظروف شاملا كافة التفاصيل مع التوفير في التكاليف و الوقت. تم أنجاز النموذج الرياضي بالاعتماد على شروط حدودية مختلفة من تصريف وعمق الماء في مقدم ومؤخر القناة . أستخدمت عنصر الخشونة وهي T–shape وبارتفاع 3سم وتم ترتيبها بثلاثة توزيعات مختلفة وهي التوزيع بسطرين على جزء من عرض القناة وباربعة أسطر أيضا على جزء من عرض القناة والتوزيع الكامل على عرض القناة. أظهرت النتائج أن تأثير عناصر الخشونة يزداد بزيادة عدد أسطر عناصر الخشونة. كان الاداء الهيدروليكي في حالة التوزيع بالاربعة اسطر على جزء من عرض القناة وتوزيع الخشونة على عرض القناة لهما متقارب وذلك لكون لهما تقريبا نفس نسبة تقليل السرعة حيث قلت قيم السرعة في المنطقة القريبة من عناصر الخشونة بحدود 66% و61% من قيمة معدل السرعة على التوالي وأن الفرق في التأثير للتوزيع ذو الاربعة أسطر يكون في جزء من مقطع الاختبار. هذا السلوك أدى الى زيادة السرعة بنسبة 11% في الجانب الثاني للقناة وبنسبة 10% بالقرب من سطح الماء في حالة أستخدام التوزيع باربعة أسطر على جزء من عرض القناة وكذلك أزدادت بنسبة 32% يالقرب من سطح الماء في حالة أستخدام التوزيع الكامل على عرض القناة.

الكلمات الرئيسية: برنامج APDL، توزيع السرعة، حسابات ديناميكية الموائع، ترتيب، عناصر الخشونة.

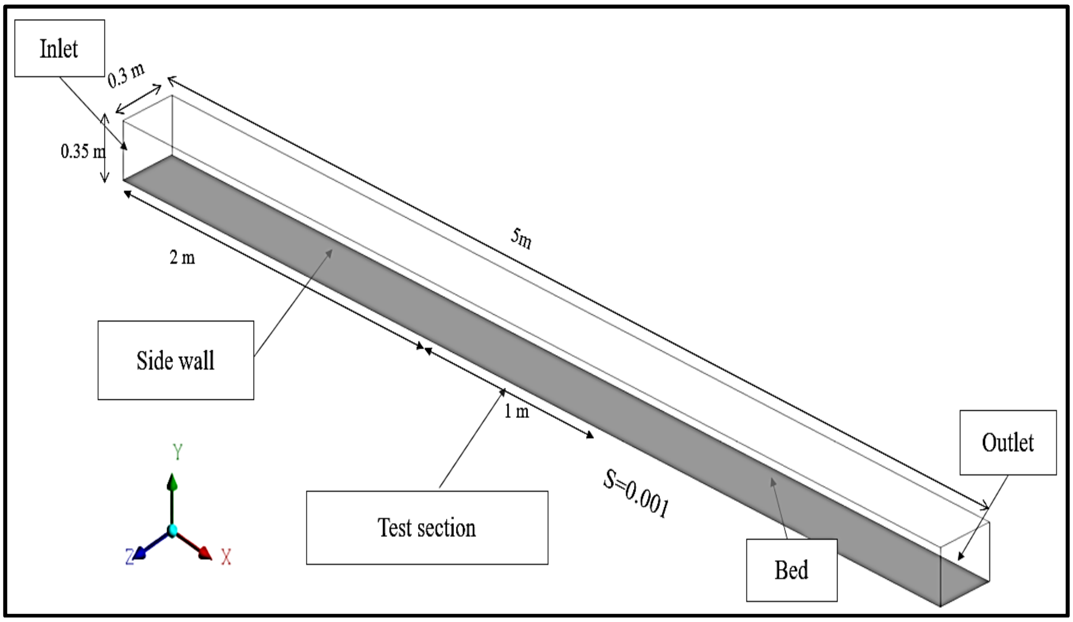
1. INTRODUCTION

The important problems that occur wildly in open channels are erosion and sedimentation in their mainstream. These problems have been studied for a long time to find techniques to get rid of it. There are many solutions that are used to control these problems. One of these solutions is increasing resistance of the channel bed by adding roughness elements that is lead to energy loss in a stream channel causing the hydraulic of the flow in the channel to change. Roughness elements

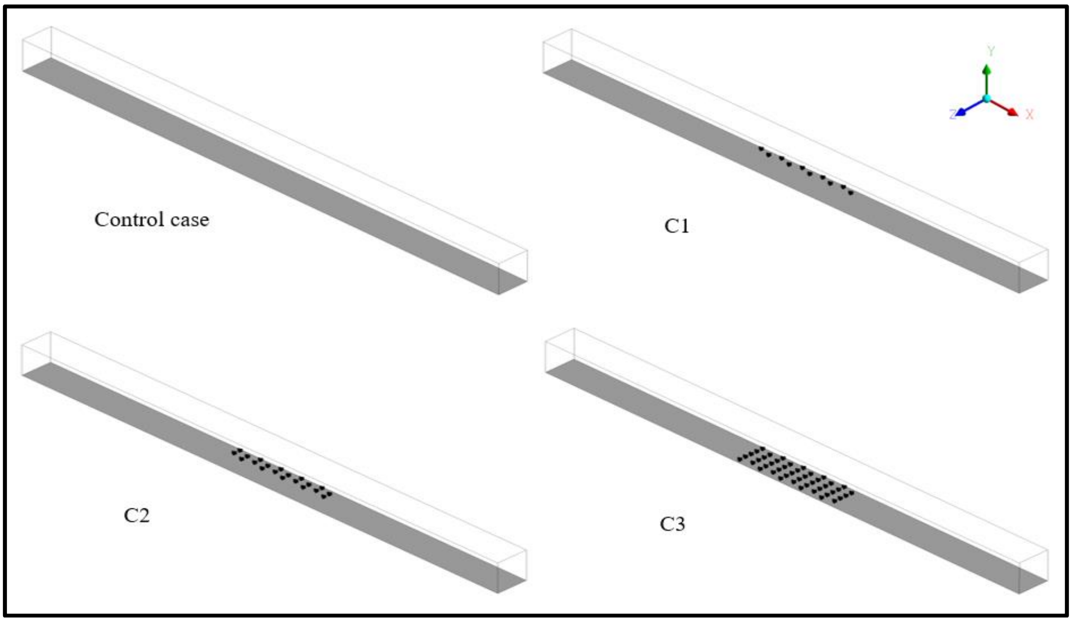
can be large stone or concrete blocks placed in the channel's bed with a different configuration to impose more resistance in the bed, such as **(Kim, 2011)**, who used cylindrical roughness elements with a different configuration to find the vegetation effect in open channels. The flow resistance increased with increasing the cylinder Reynolds number and the density of the roughness elements. Vegetation density was the most significant factor in determining turbulence statistics, mean flow, instantaneous flow, and flow resistance. **(Abbaspour and Kia, 2014)**, studied the effect of strip semi-cylindrical roughness established in two different configurations on a turbulent layer in a relatively high Reynolds number. Experimental parameter values were depended on the percentage of the distance between center to center roughness elements to the roughness height. Velocity distribution depended on the configurations of the roughness and had an inverse relationship. **(Baki, et al., 2016),** conducted an experimental investigation on a rock ramp for fish passes. Two boulder configurations were studied. The first configuration was the same as the staggered configuration in the experimental test. The second configuration was set to be a clustered configuration of boulders in which every two rows were placed closer to each other. The effective boulder spacing in the longitudinal and transverse directions for two different boulder patterns used in a rock-ramp fish passes. The flow's resistance varied depending on the distance between the emerged boulders and was constant in the case of submerging boulders. **(Thappeta, et al., 2017)**, investigated the effect of using a single hemisphere element, boulders, and cylindrical roughness elements with staggered configuration and randomly distributed in a steep open channel by using CFD software. The research was proved that the energy loss depends on density, Reynolds number, and the ratio of submergence. In the case of using boulders, the energy loss was decreased as the boulders' density was increased**. (Akutina, et al., 2019),** investigated the impact of using cubes elements arranged in two different densities between two sides. At high relative submergence, higher velocities appeared with a rough bed, and the interaction between the two sides depended on the relative submergence.
This study simulates the flow in the roughened flume using the CFD program, which uses the finite volume method to solve 3D Reynolds averaged Navier Stokes, RANS equations, and SST (shear stress transport) k- ω turbulence model. The free surface location is computed using the volume-of-fluid, VOF method**.** The channel bed is composed of regular roughness elements in shape placed in a staggered pattern to simplify the roughness geometry.
This research aims to study the effect of the configurations of roughness elements in changing the flow's hydraulic characteristics. This change can create desirable effects to control erosion or sedimentation and helps to develop better flow management.

2. DESCRIPTION OF THE STUDY CASE

This study simulates the flow in an open channel by using a flume with roughness elements. Using mechanical APDL product launcher (ANSYS Parametric Design Language) to draw the flume with dimensions of 0.3m flume width, 0.35m flume depth, and 5m long with constant slope equal to 0.001 a control case. Roughness geometric T-shape with base 3*3 cm^2 and height equal to 3cm is applied. This physical model is used by **(Gazal, 2015)**. The flume test section is 1*0.3 m^2 located at the center of the flume filled with three different configurations, two lines, four lines, and fully roughness configurations placed in a staggered pattern. The distance between rows was 6cm and between columns equal to 10.78cm from the center to center of the roughness elements.
Configuration refers to the method by which the roughness elements are arranged on the bed of the flume. The roughness elements' main purposes are to investigate the impact of using T-shape roughness elements with different configurations on the hydrodynamics of the flow in open channels and the head losses by reducing the velocity magnitude in the fluid domain. That was by showing the difference in the velocity distribution in the flow domain between the three cases. **Fig.1.** shows the test section within the flume and the design of the configurations of roughness elements.



a- A schematic drawing shows the roughness elements test section within the flume.



b- Design of configurations of roughness elements.

Figure 1. The geometry of the study.

3. DESIGN OF MODELS RUNS

To conduct runs with a computational fluid dynamics CFD program (FLUENT), various options are used in this simulation, transient flow, the volume of fluid, multiphase flow (air and water), K-ω –*SST* (shear stress transport), PISO method (pressure-implicit splitting of operators). To derive the flow, a double-precision calculation was necessary to solve the pressure differences. For the volume of fluid calculation should be turned on the option of specified operating of density in operating conditions, and set the lightest phase in operating density. The boundary conditions were defined as mass flow inlet and pressure outlet. The inlets were defined as the air and water flow into the domain. The flume walls like bottom and side walls are defined as nonslip walls. The no-slip boundary is the general boundary used at walls and describes the velocity equal to zero. One variable that was taken into consideration in the run to simulate the flow is the configurations of the roughness elements in the test section. One depth of water at the flume's inlet with one discharge was considered equal to 0.1*m* and 5.8*l/s* consequently. **Table 1** describes the design of runs.

Table 1. Design of runs.

Configurations design	**Number of lines**			
	without	**Two lines**	**Four lines**	**Fully rough**
Run number	1	2	3	4
Case symbol	CC	C1	C2	C3

4. RESULTS AND ANALYSIS

Figures (2 to 7) present the results of these investigations' velocity distributions after completing the initialization and calculation to provide data for comparison with a control case. Moreover, it presents details of a suggestion about which configurations of roughness elements are better used as energy dissipater. The convergence criterion is that the normalize residual should equal $10e^{-3}$. The default convergence criterion in CFD FLUENT is sufficient.

Generally, velocity distributions in the fluid domain are affected by the configurations of the roughness elements. The directions and values of the flow velocities are presented by contours **Figs (2 to 4)**.

At the mid height of the roughness elements, the velocity values decreased in the case of two lines configuration by about 33% of the control case's velocity in the zone near the roughness elements and the same velocity of the control case on the other side. In the case of four lines configuration, the velocity values decreased by about 66% of the control case's velocity in the zone near the roughness elements and increased by about 11% on the other side. In the case of a fully rough configuration, the velocity values decreased by about 61% of the velocity of the control case in the zone near the roughness elements. At the top surface of the roughness elements, the velocity values decreased in the case of two lines configuration by about 8% of the velocity of the control case in the zone near the roughness elements and decreased by about 3% on the other side. In the case of four lines configuration, the velocity values decreased by about 29% of the control case's velocity in the zone near the roughness elements and increased by about 7% on the other side. In the case of a fully rough configuration, the velocity values decreased by about 18% of the control case's velocity in the zone near the roughness elements. In the longitudinal cross-sections, it can be noticed that the flow velocity of water in the zone near the roughness elements and it will increase till reaching the water surface. Along the center of the flume, the velocity values increased above the roughness elements in the case of two lines, four lines, and fully rough configurations by about 5%, 10%, and 32%, consequently of the maximum velocity of the control case. All these values were obtained at the same inlet discharge.

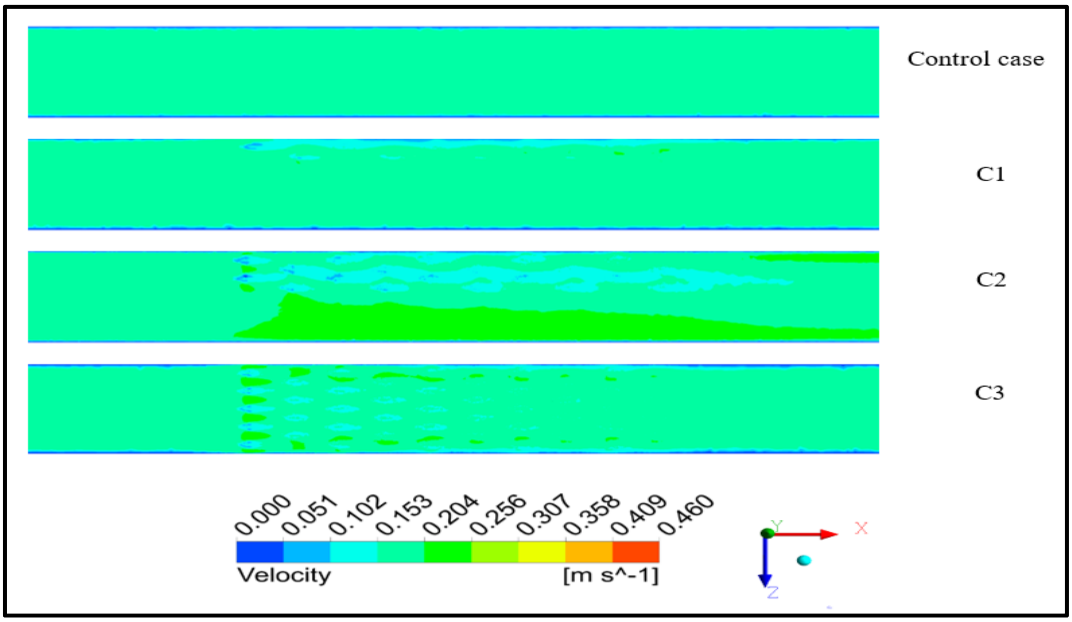


Figure 2. Velocity contours, Top view section at the top of the roughness elements.

.

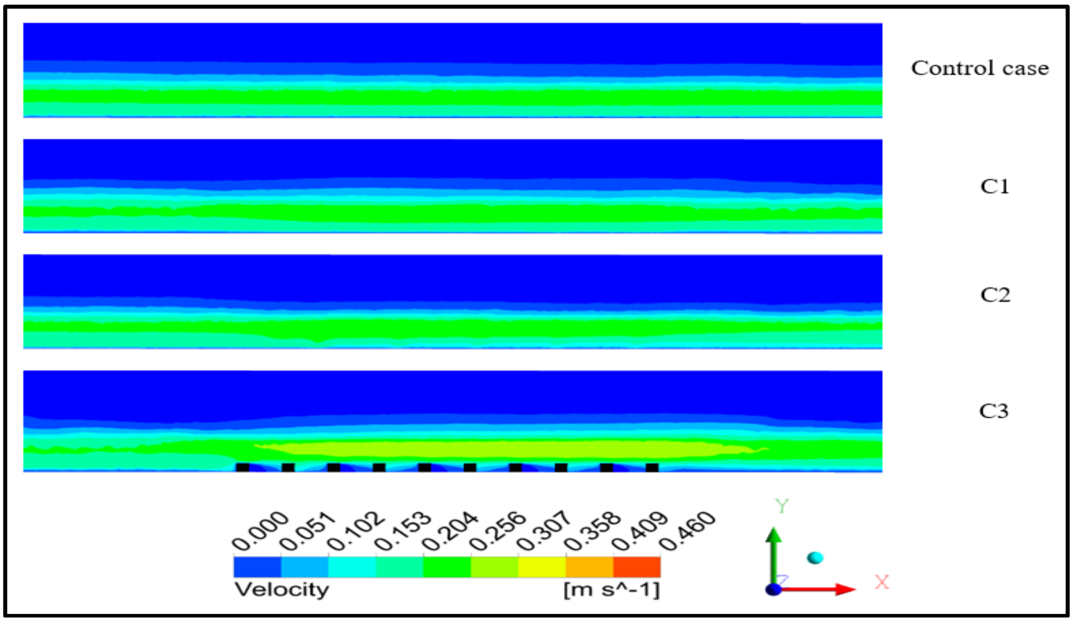


Figure 3. Velocity contours and Side view along the center of the flume.

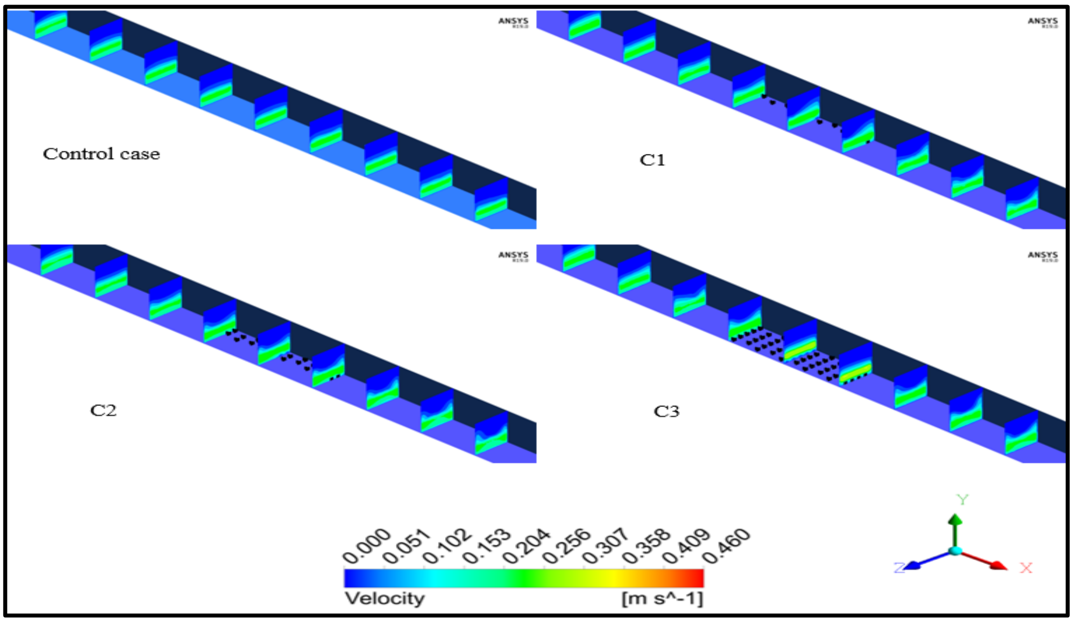


Figure 4. Velocity contours and cross-sections along the flume.

From the obtained results, **figures (5 to 7)** present the variation of velocity with water depth at three different locations in the center along the flume, which was; 0.075, 0.15, and 0.225*m* from the left side of the flume as a comparison between cases for different configurations with the control case.

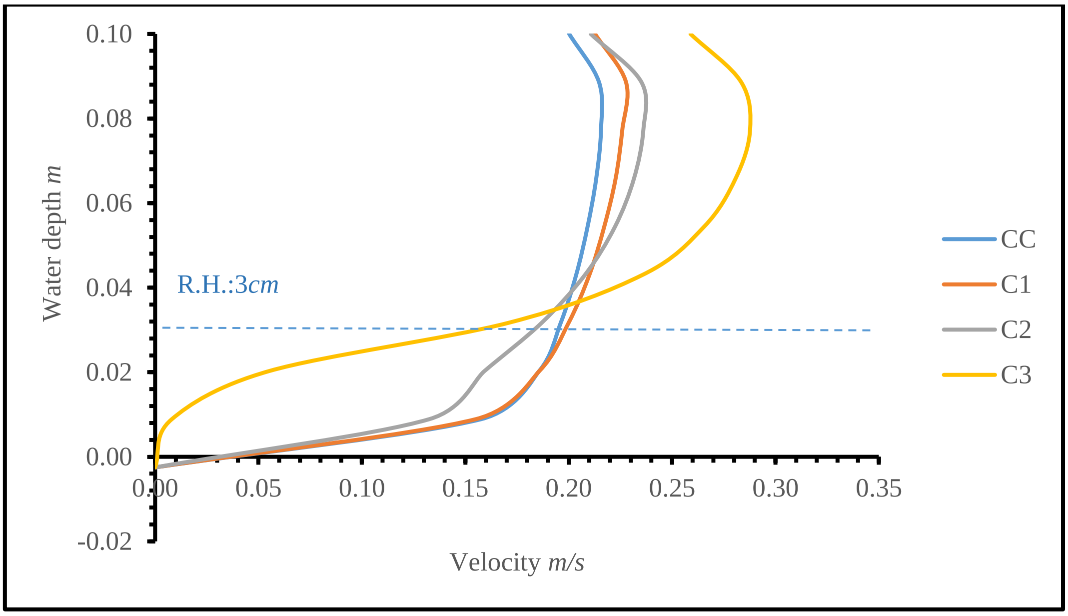


Figure 5. Velocity profiles at the center of the cross-section.

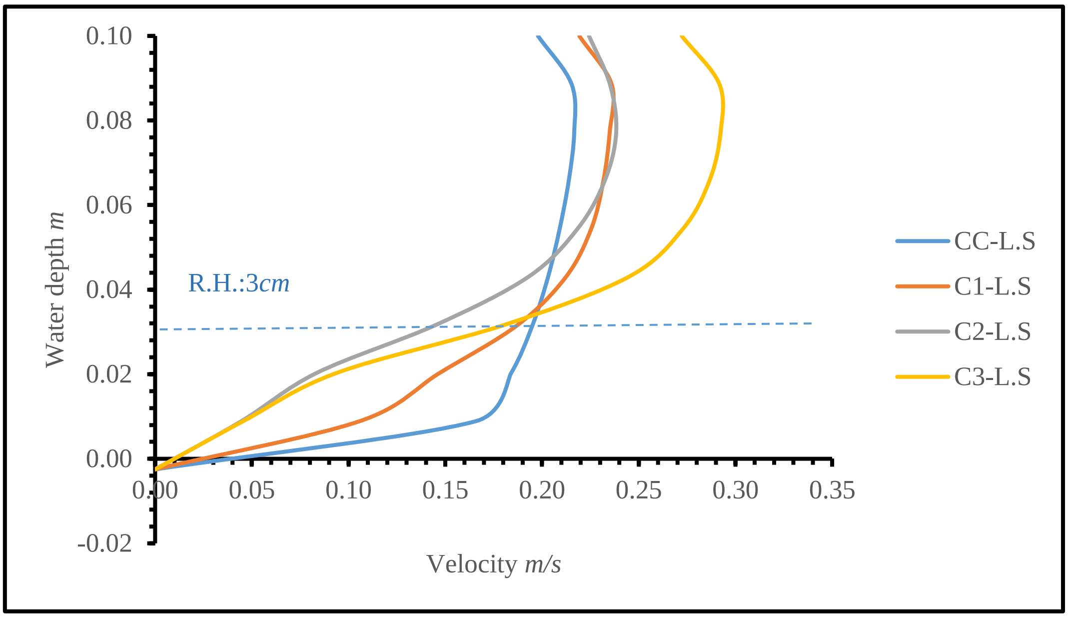


Figure 6. Velocity profiles at the center of the cross-section.

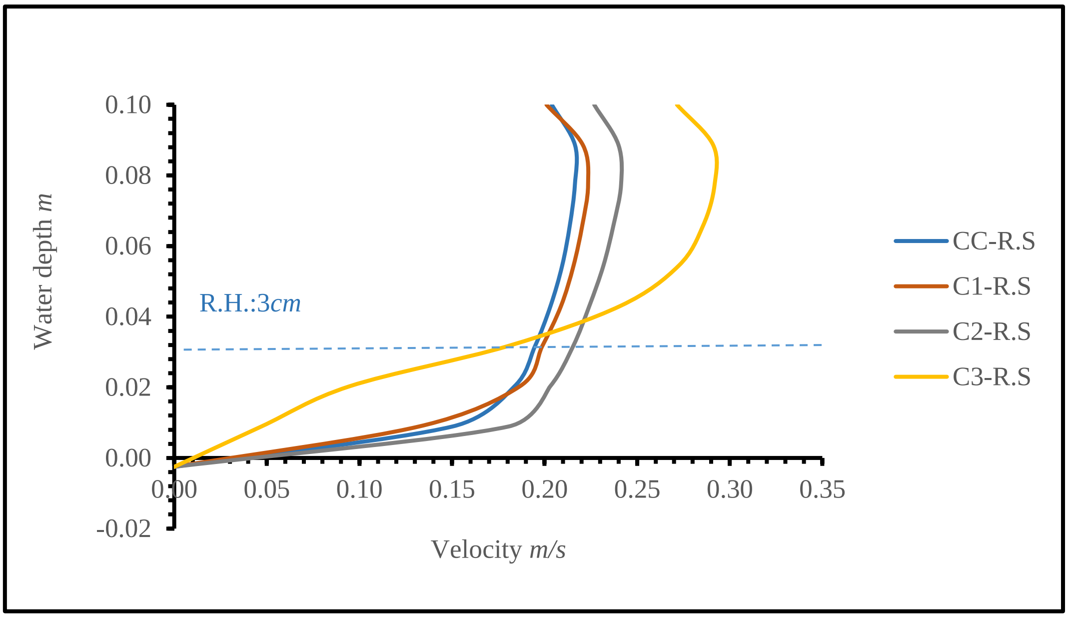


Figure 7. Velocity profiles on the right side of the flume.

At mid-height of the roughness elements, the velocity values decreased in four lines and fully rough configurations by about 33% and 28% compared with two lines configuration. On the other side, the velocity values increased in case of four lines configuration by about 11% and have the same velocity of the control case in the two-line configuration. At the top surface of the roughness elements, the velocity values decreased in the case of four lines and fully rough configurations by about 21% and 10% compared with two lines configuration. It is increased in the case of four lines configuration by about 7% and decreased in the case of two lines configuration approximately by 3% compared with the control case's velocity on the other side. At the center of the side view along the flume, the velocity values increased above the roughness elements in the case of four lines and fully rough configurations by about 5% and 27% compared with two lines configuration.

5. CONCLUSIONS:

Velocity distributions in the fluid domain are affected by the configurations of the roughness elements. It can be noticed that the progression of velocity from the side of roughness elements will be low and increase gradually as getting far from the roughness elements. It is decreased in the case of four lines and fully rough configurations by about 33% and 28% compared with two lines configuration near the bed. At the top surface of the roughness elements, the velocity values decreased in the case of four lines and fully rough configurations by about 21% and 10% compared with two lines configuration. The velocity values decreased near the bed in case of four lines configuration much more than two lines and fully rough configurations. Contrary to that, the velocity values increased above the roughness elements in case of fully rough configuration much more than two lines and four lines configurations by about 27% and 22%. That will indicate that the case of four lines configuration is more affected in decreasing the velocity than other cases. It has almost the same performance, with the case fully rough, unlike the case of two lines configuration. Still, the difference is that the fully rough configuration affected the whole test section area, but the four lines affected just in the part of the test section where it is placed.
This model can be applied for practical purposes in the stream with high velocities by using artificial blocks that have T-shape to avoid sedimentation or erosion.

REFERENCES:

- Kim, S., 2011. 3D Numerical Simulation of Turbulent Open- Channel Flow through Vegetation. Ph.D. thesis. *School of Civil and Environmental Engineering*, *Georgia Institute of Technology*.

- Abbaspour, A., and Kia, S. H., 2014. Numerical Investigation of Turbulent Open Channel Flow with Semi-Cylindrical Rough Beds. *KSCE Journal of Civil Engineering*.

- Baki, A. M., M.ASCE, D. Z., and F.ASCE, N. R., 2016. Flow Simulation in a Rock-Ramp Fish Pass. *Journal of Hydraulic Engineering*, 142(10).

- Thappeta, S. K., Bhallamudi, S. M., Fiener, F., and Narasimhan, B., 2017. Resistance in Steep Open Channels Due to Randomly Distributed Macro Roughness Elements at Large Froude Numbers. *Journal of Hydraulic Engineering*, 22(12).

- Versteeg, H.K., and Malalasekera, W., 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method.

- ANSYS Fluent Theory Guide 19.0 Release.

- Ghazal, A., 2015. Manning's Coefficient for Geometric Roughness Elements, M. Sc. Thesis submitted to the *University of Baghdad – College of Engineering*.

- Akutina, Y., Eiff, O., Moulin, F. Y., and Rouzes, M., 2019. Lateral Bed-Roughness Variation in Shallow Open-Channel Flow with Very Low Submergence, 19(5), pp. 1339-1361.

- Daham, M., and Abed, B., 2020. One and Two-Dimensional Hydraulic Simulation of a Reach in Al-Gharraf River, *Journal of Engineering*, 26(7), pp., 28-44.

- Asaad, B., and Abed, B., 2020. Flow Characteristics of Tigris River within Baghdad City During Drought, *Journal of Engineering*, 26(3), pp., 77-92.