University of Baghdad
College of Engineering
JE
JOURNAL OF ENGINEERING

Journal of Engineering

journal homepage: www.joe.uobaghdad.edu.iq

Volume 29 Number 10 October 2023

Control of Propagation of Salt Wedge by using Roughness Blocks having Different Inclination

Safa Haider Badr [1,*], Riyadh Z. Azzubaidi [2]

Department of Water Resources, College of Engineering, University of Baghdad, Baghdad, Iraq
safa.mohi2010m@coeng.uobaghdad.edu.iq[1], riyadh.z.azzubaidi@coeng.uobaghdad.edu.iq[2]

ABSTRACT

The hydraulic conditions of a flow previously proved to be changed when placing large-scale geometric roughness elements on the bed of an open channel. These elements impose more resistance to the flow. The geometry of the roughness elements, the numbers used, and the configuration are parameters that can affect the hydraulic flow characteristics. The target is to use inclined block elements to control the salt wedge propagation pointed in most estuaries to prevent its negative effects. The Computational Fluid Dynamics CFD Software was used to simulate the two-phase flow in an estuary model. In this model, the used block elements are 2 cm by 3 cm cross-sections with an inclined face in the flow direction, with a length of their sides 2 and 3 cm. These elements were placed with a constant spacing in two rows at a distance from two sides of the bed of the channel model. Six simulation runs were conducted with two different discharges and three different inclinations of the centerline of the element concerning the flow direction. The applied discharges are 30 and 45.3 l/min, and the inclination of roughness elements are 15^{o}, 30^{o}, and 45^{o}. The spacing between elements in each row is kept at 3cm. The results showed that when no roughness elements were used, the propagation of the salt wedge extended to 3.9m and 3.1m at a discharge of 30 l/min and 45.31/min, respectively. The propagation of the salt wedge was reduced when using the inclined blocks roughness element. This reduction depends on the applied discharge and the angle of inclination. At the minimum applied discharge of 30 l/min, the propagation of the salt wedge was reduced by 74% at 45^{o} inclination. In contrast, it was 69% at 30^{o} and 64% at 15^{o} inclination at the same discharge. When the discharge is 45.3 l/min, the propagation of the salt wedge was reduced by 85% at 45^{o} inclinations of roughness, 84% at 30^{o}. It was 70% at 15^{o} inclinations. The roughness elements improve the flow turbulence that disperses and slows the salt wedge propagation beneath the fresh water.

Keywords: CFD, Salt wedge, Roughness elements.

---

*Corresponding author
Peer review under the responsibility of University of Baghdad.
https://doi.org/10.31026/j.eng.2023.10.11

Article received: 28/02/2023
Article accepted: 03/07/2023
Article published: 01/10/2023

السيطرة على انتشار إسفين الملح باستخدام عناصر خشونة لها ميل مختلف

صفا حيدر بدر[1]،*، رياض زهير الزبيدي[2]

قسم هندسة الموارد المائية، كلية الهندسة، جامعة بغداد، بغداد، العراق

الخلاصة

ثبت سابقًا أن الظروف الهيدروليكية للتدفق قد تغيرت عند وضع عناصر خشونة هندسية واسعة النطاق على قاع قناة مفتوحة. هذه العناصر تفرض مقاومة أكبر للتدفق. هندسة عناصر الخشونة والأرقام المستخدمة والتكوين هي معلمات يمكن أن تؤثر على الخصائص الهيدروليكية للتدفق. كانت الفكرة في هذا البحث هي استخدام عناصر الخشونة المائلة في التحكم في انتشار إسفين الملح الذي يحدث في معظم مصبات الأنهار لمنع آثاره السلبية. تم استخدام برنامج ديناميكيات السوائل الحسابية لمحاكاة التدفق على مرحلتين في نموذج مصب النهر. في هذا النموذج، تكون عناصر الكتل المستخدمة عبارة عن مقاطع عرضية 2 سم * 3 سم لها وجه مائل في اتجاه التدفق، ويبلغ طول جوانبها 2*3 سم. تم وضع هذه العناصر مع تباعد ثابت في صفين على مسافة من جانبي نموذج القناة. تم إجراء ست عمليات محاكاة مع تصريفين مختلفين وثلاثة اتجاهات مختلفة لميلان العناصر فيما يتعلق باتجاه التدفق. التصريف المستخدم هو 30 و45.3 لتر / دقيقة، وميل عناصر الخشونة هو 15 درجة، و30 درجة، و45 درجة. يتم الاحتفاظ بالتباعد بين العناصر 3 سم. أظهرت النتائج أنه في حالة عدم استخدام عناصر خشونة، فإن انتشار إسفين الملح يمتد إلى 3.9 م و3.1 م عند تصريف 30 لتر / دقيقة و45.31 لتر/ دقيقة على التوالي. تم تقليل انتشار إسفين الملح عند استخدام عنصر خشونة الكتل المائلة. يعتمد هذا التخفيض على التصريف المستخدم وزاوية الميل. عند الحد الأدنى من التصريف البالغ 30 لترًا / دقيقة، تم تقليل انتشار إسفين الملح بنسبة 74٪ عند ميل 45 درجة. بينما كانت 69٪ عند 30 درجة و64٪ عند 15 درجة ميل عند نفس التصريف. عندما يكون التصريف 45.3 لتر / دقيقة، تم تقليل انتشار إسفين الملح بنسبة 85٪ عند ميل 45 درجة للخشونة، و84٪ عند 30 درجة، و70٪ عند ميل 15 درجة.

الكلمات المفتاحية: ديناميكية الموائع الحسابية، إسفين الملح، عناصر الخشونة.

1. INTRODUCTION

Many studies were conducted to control the propagation of salt wedge introduction at estuaries by using different techniques. **(Kim, 2011; Poggioli and Horner, 2015; Krvavica et al., 2016)** studied open-channel turbulent 3D numerical simulation with the presence of vegetation. A new method is applied that simulates flow through natural vegetation using an approach based on porosity. This method comes from the connection between porosity and the leaves area index. The research demonstrates that numerical modeling of real vegetation's impact on flow may be done satisfactorily. Simulation results demonstrated the presence of plants strongly influences the characteristics of the flow in the channel middle, where a distinct slowing of the flow may be seen. The results of the velocities were compared to experimental data that showed reasonable agreement the program does a good job of showing how porosity and bending work. **(Tani and Fujita, 2020; Ralston et al., 2016; Servini et al., 2017),** studied applying the sampling method to shallow open-channel flows with circular roughness elements. This research applies the sampling moir´e method, a novel technique originally developed for measuring slight deformation of a static object, such as a plate, to fluctuating free water surface in turbulent open channel flows. The method captured the two-dimensional water surface deformation at a high sampling rate by dyeing water with a white color. The target flow was a very shallow subcritical flow with a depth of 1.5 times the height of the roughness. The main result obtained in this study is The sampling moir´e method was successfully applied to the three-dimensional measurement of time-developing free-surface fluctuations in an open channel turbulent flow. A series of accuracy tests using hydrostatic tanks showed that the measurement accuracy had a standard deviation of at least 0.07 mm.

(Cavalcante et al., 2021; Krvavica and Ruzic, 2020; Gholami et al., 2015), studied roughness coefficients generated by flexible vegetation in an open channel. This study aimed to monitor water and solid discharge through direct hydro sediment metric measurements and to estimate, using a simplified model based on a force balance, the hydraulic roughness coefficients generated by flexible vegetation element of the Ipomoea pes-caprae species under emergent and submerged conditions. The results demonstrate that aquatic vegetation directly influences river dynamics; vegetation density has a hydraulic impact on the flow transport capacity. Increasing the resistance of the vegetation to the flow leads to a decrease in the capacity of transport of sediments. **(Al-Fuady and Azzubaidi, 2021; Al-Sarefi and Azzubaidi, 2021; Ali and Al Thamiry, 2021)** conducted an experimental study on investigating and controlling salt wedge propagation. The study aims to investigate the variation of profiles and propagation of salt wedges and the use of air and water curtains in controlling its propagation. They conducted seventy-seven experiment runs to investigate saltwater propagation and sixteen runs to control the salt wedge using air and water curtains. The results showed that increasing the discharge of fresh water or the slope of the flume bed leads to a reduction in the length of the salt wedge. They found that air and water curtains are efficient methods for controlling the propagation of salt wedges. A minimum discharge of air and water curtains was obtained to prevent salt wedge propagation.

Roughness elements proved to be efficient in controlling the propagation of salt water. **(Alwan and Azzubaidi, 2021; Sajjadi, 1995; Mirauda and Plantamura, 2013)**, conducted numerical investigations on Large-Scale geometric roughness elements in open channels with different heights to control the propagation of salt wedges. The hydraulic behavior of the flow can be changed by using large-scale geometric roughness elements in open channels. This change can help control 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. In this paper, velocity distribution along the flume was theoretically investigated using a series of tests of T-shape roughness elements. T- shape roughness elements with a height equal to 3cm placed in three different configurations, two lines, four lines, and fully rough configurations were tested. The results show that the effect of roughness elements increases with increasing the number of lines of 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.

(Shaheed and Azzubaidi, 2020; Rao et al., 2022; Baghalian and Ghodsian, 2019), studied controlling salt wedge propagation by using the roughness element upstream river mouth. This study aims to numerically study the effect of roughness elements in controlling the propagation of salt wedges upstream river mouth. A cubic of 1.5cm roughness elements were used in this research fixed at the bed of the flume system. These cubic elements are arranged in different configurations. The roughness elements work to increase the flow turbulence that dispersion the salt wedge that moves beneath the fresh water and reduces its propagation. The results of eight model runs showed the roughness elements could reduce the propagation up to 66.7% when the roughness elements are placed close to the end of the flume, at the highest applied discharge of fresh water, and the minimum bed slope of the flume.

(Dritselis, 2014; Briggs et al., 2017; de Marchis et al., 2019; Shamloo, 2014) studied large eddy simulations of turbulent channel flow with transverse roughness elements on one wall. This study examines the feasibility of large eddy simulation for predicting turbulent channel flows with two-dimensional roughness elements of the square, circular, and triangular shapes transversely placed on the bottom wall. The use of the turbulence models leads to an improvement in the predictions of several turbulence statistics compared with the case when no model is considered. Large eddy simulation can be combined easily with an immersed boundary method yielding satisfactory results based on a coarser grid resolution than in direct numerical simulation, and, thus, it is suitable for investigating turbulent channel flows with riblets of various shapes. Results from large eddy simulations of turbulent flows in a channel with 2D roughness elements of a square, circular, and triangular cross-sections transversely placed on its bottom wall were obtained to assess the feasibility of several sub-grid-scale models for all flow cases with smooth and rough walls examined here. In addition, all models contributed from 5% up to 50% of the total dissipation rate over much of the channel, playing a key role in the grid-scale turbulence energy budget.

(Zachopoulos et al., 2020), studied by using the 3D numerical model to simulate the dynamics of the salt-wedge intrusion along the lower reach of the Strymon River Estuary, Greece. The Waterway Lake and Coastal Ocean Model were used to conduct all the numerical simulations. To solve the time-varying governing Navier-Stokes equations, the model includes Reynolds averages and a mixed-layer model to quantify vertical mixing. The model provides reasonably accurate findings for salt wedge intrusion because it has been calibrated using field measurements. The constructed model demonstrated how the terrain of the area, the river flow rate, and the form of the bottom river all affect how quickly the salt wedge spreads. The results show that stratification could intensify when river flow is reduced. Additionally, placing submerged shallow sills on the river bottom at a specific distance can stop seawater from spreading across a much greater distance.

(Ospino et al., 2018; Cassan et al., 2017; Abbaspour and Kia, 2014), carried out a study on saltwater intrusion dynamics in a Microtidal Estuary of the Magdalena River, Colombia. The impact of river flow rate, tide, and wind on mixing and saltwater intrusion in the river estuary is simulated using a three-dimensional numerical model. Field data were used to calibrate and validate the numerical model. The results show how winds and seasonal tides affect the spreading and mixing of salt water. It was discovered that variations in river discharges impact the length of the salt wedge. The water column is stable even under the impact of wind and tides at the most excellent release. The effect of wind force on tidal force is visible at medium discharges. Additional saltwater intrusion and tide effects are visible at the river mouth, where discharges are lower.

The objective of this study is to simulate a three-dimensional model by using CFD software. Then estimate salt wedge propagation and control it using geometrical roughness elements under different configurations.

2. DESCRIPTION OF THE MODEL

The flow in an open channel is simulated using a flume system with roughness elements. The system geometry is simulated by the Solid Work 2018 program product launcher, ANSYS Parametric Design Language. The flume (0.25 width, 0.35 depth, and 6m long) with a constant slope equal to 0% is considered a control case. Fresh water is supplied at the upstream side of the flume. A weir of adjustable height is installed at the downstream side of the flume with dimensions 0.1m in length, 0.09m in depth, and 0.075m in width. This weir is used to discharge fresh water from the flume and to supply salt water, through an opening at its center that connects a saltwater tank, as shown in **Fig. 1**.

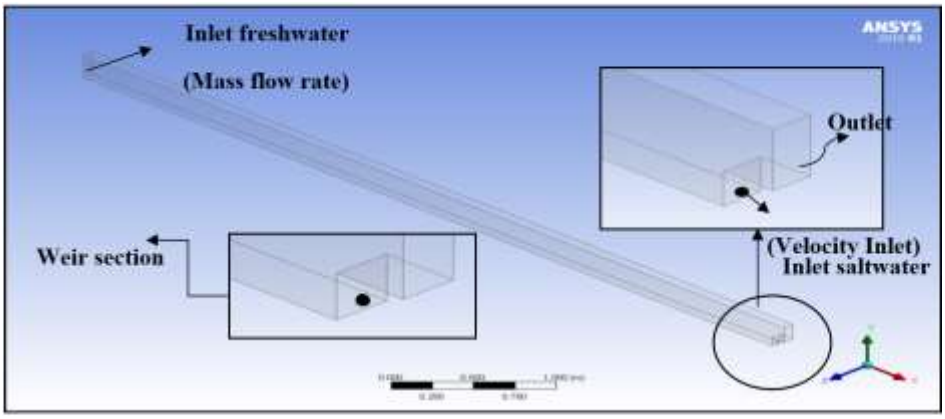


Figure 1. The geometry of the 3D model of the flume system.

The used roughness elements in **Fig. 2** are blocks of 2 cm by 3 cm cross-sections having an inclined face in the direction of the flow, with a length of 2 and 3 cm.

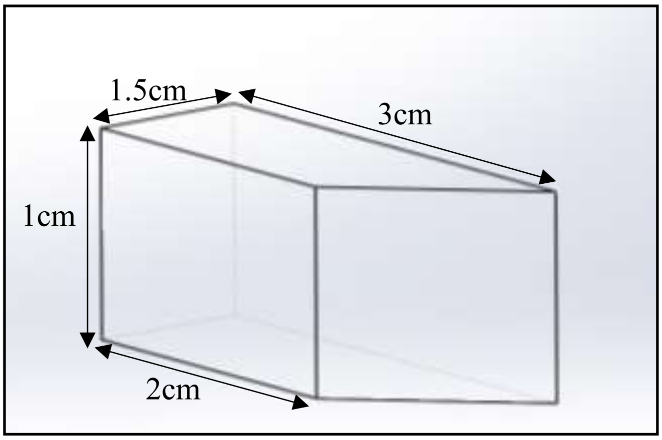


Figure 2. The Shape of Roughness Elements

These elements were installed in two rows at the sides of the bed of the flume with different configurations, as shown in **Fig. 3**. Configuration refers to the method by which the roughness elements are arranged on the flume bed. The distance between roughness elements in each row is kept constant at 3cm. Different block inclinations were applied. These are 15^{o}, 30^{o}, and 45^{o} degree.

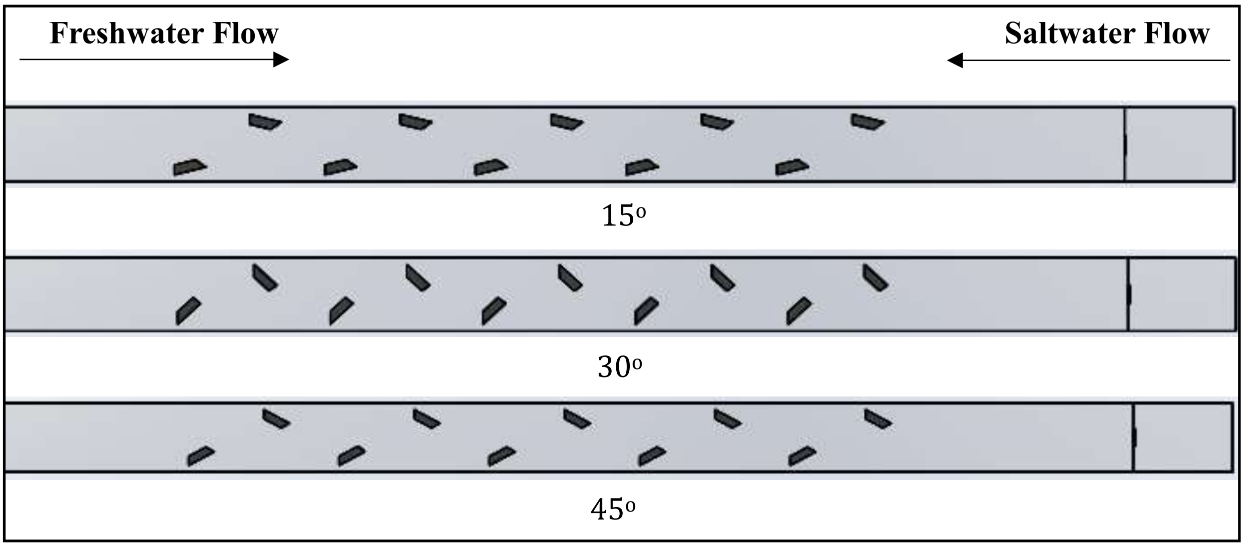


Figure 3. A top-view diagram showing a configuration of roughness elements.

3. DESCRIPTION OF THE USED SOFTWARE

Each code in the Computational Fluid Dynamics, CFD, Ansys Fluent19 Software solves a challenging collection of geometrical issues. CFD is a simulated numerical model for analyzing and resolving flow problems. Ansys Fluent is typically used to simulate many flow problems, including laminar, turbulent, compressible, incompressible fluids, steady-state, and unsteady-state flow.
The Navier-Stokes and continuity equations numerically solved using the finite volume method are the governing equations in ANSYS Fluent. These equations are a derivative (**Ashgriz and Mostaghimi, 2002; Date, 2005; Yang et al., 2017; Biringen and Chow, 2011)**, mass conservation law for flowing fluids, and Newton's second law. According to the governing equation for Ansys Fluid can be expressed as follows:
Navier-Stocks equations:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\rho g_x-\frac{\partial_p}{\partial_x} \quad (1)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+\rho g_y-\frac{\partial_p}{\partial_y} \quad (2)$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\rho g_z-\frac{\partial_p}{z} \quad (3)$$

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (4)$$

where:
u, v, w is the three-dimensional velocity components (m/s).

ρ is the density (kg/m^3).
μ is the dynamic viscosity ($N.s/m^2$).
P is the pressure, Pa.
g is the gravitational acceleration ($\frac{m}{s^2}$)
t is the time (s).

4. DESCRIPTION OF THE MODEL RUNS

Various cases can be simulated with FLUENT software can be adopted namely: transient flow, the volume of liquid, multiphase flow (air and water). The turbulent standard k-ε model is selected and adopted in this work to simulate flow turbulence **(Versteeg and Malalasekera, 2007; Date, 2005; Sharma, 2021; Niyogi, 2006; Petrila and Trif, 2004)**, PISO method (Pressure-Implicit Splitting Of Operators). A double-precision calculation was necessary to derive the flow to solve the pressure differences. The volume of fluid measure should be turned on with the option of specifying operating density in operating conditions and setting the lightest phase in operating density **(Al-Fuady and Azzubaidi, 2021).** The boundary conditions were freshwater as mass flow inlet and free surface as pressure outlet. Saltwater was defined as a velocity inlet. Surface tension between air and water is considered constant and equal to 0.072 N/m, air-salt water equals zero, and salt water and fresh water equals 0.00148 N/m. After the geometry is generated, the flume system model is divided into cells and elements of various sizes and forms, allowing for precise numerical calculations to produce the best results. An unregular polyhedral mesh was employed in this study. The mesh density impacts the computation's accuracy and processing time. Several factors determine mesh quality. The difference between the forms of an equilateral cell with a comparable volume and ideal cell shapes is described by the Fluent, represented by skewness. Most flows had a maximum skewness factor of less than 0.95, an average lower value of 0.33, according to the ANSYS Fluent User's Guide, 2018. For this Fluent model, the skewness factor was less than 0.75, considered acceptable for good-quality components. Each model contained 1.1 million to 1.7 million cells, with a 0.01 m cell size. The meshing of the present CFD model shape is shown in **Fig. 4.** Two different freshwater discharges, 45.3 l/min, and 30 l/min, were simulated, with three different inclinations 15°, 30°, and 45°. The spacing between these elements is kept constant at 3cm. For comparison purposes, runs were conducted without using these roughness elements,

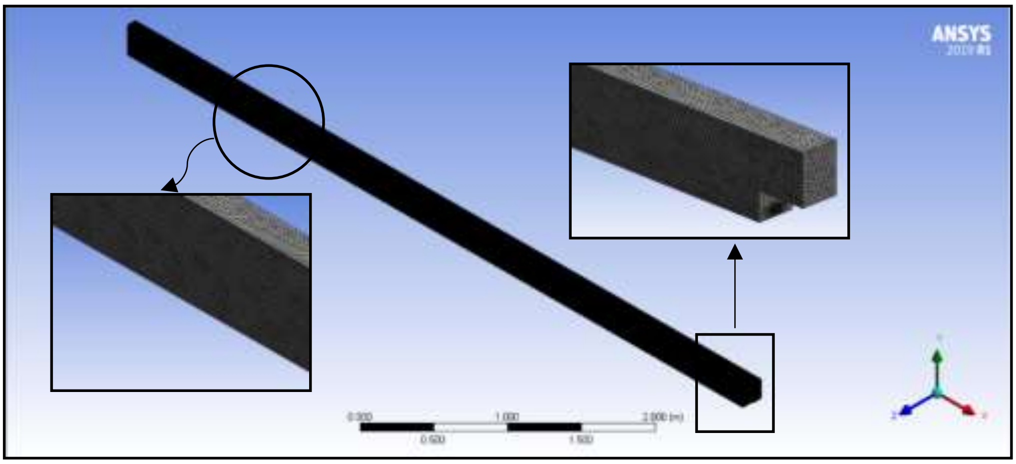


Figure 4. The meshing of the geometry.

5. RESULTS AND DISCUSSION

The maximum propagation of the salt wedge is 3.9m, which was obtained with no roughness elements at the lowest freshwater discharge value of 30l/min, as shown in **Fig. 5**. While, at the applied freshwater discharge of 45.3l/min, the propagation of the salt wedge was 3.1m, as shown in **Fig. 6**. It is clear that the propagation of salt wedge is inversely related to apply freshwater discharge.

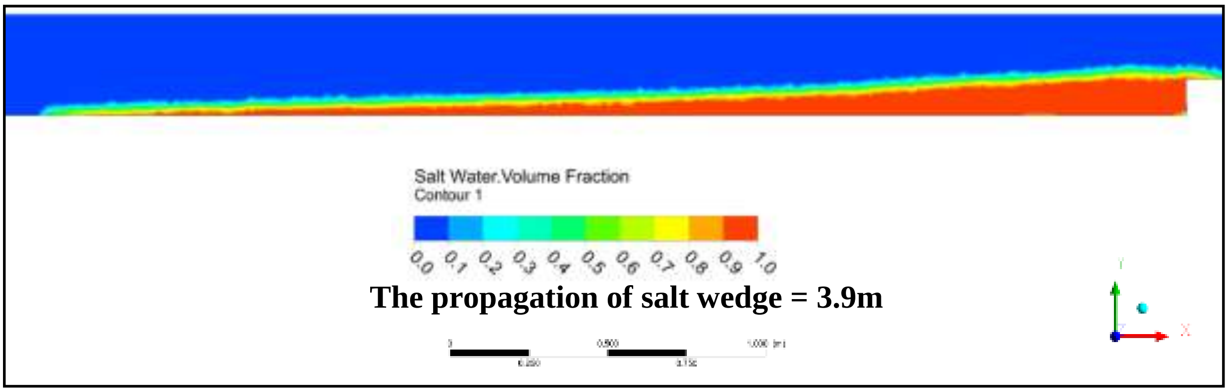


Figure 5. Propagation of salt wedge without roughness elements, (30 l/min).

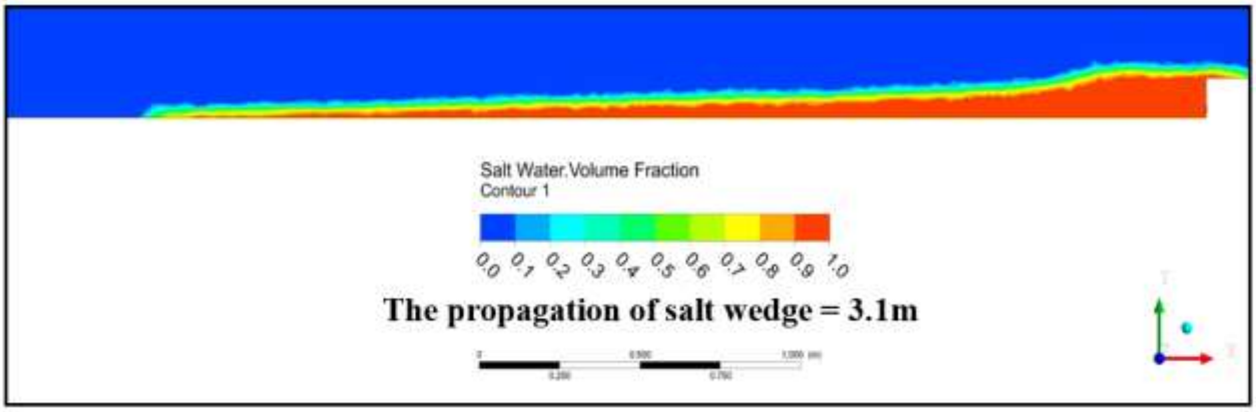
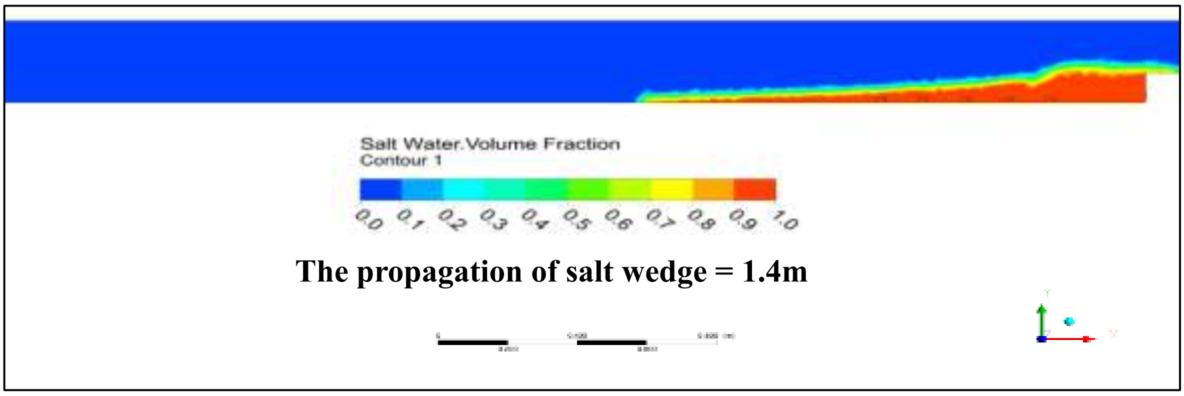


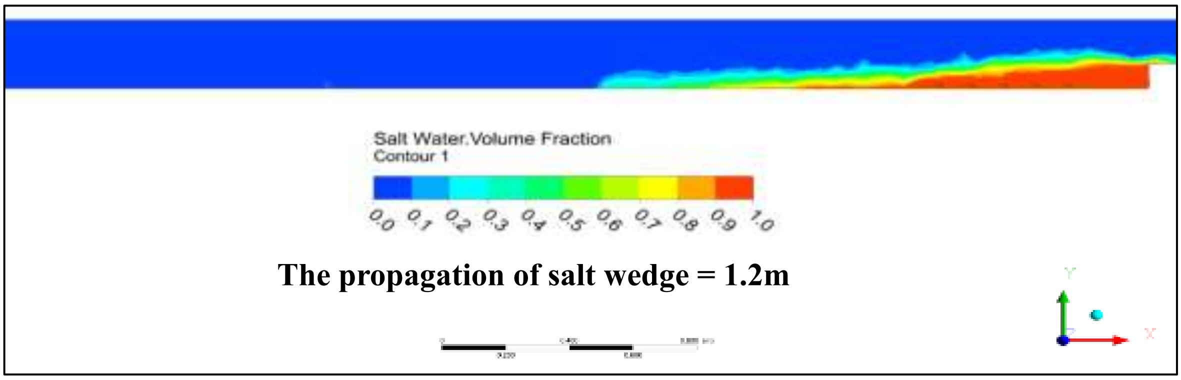
Figure 6. Propagation of salt wedge without roughness elements, (45.3 l/min).

Fig. 7 shows the propagation of salt wedge with roughness elements at applied freshwater discharge is 30l/min and different roughness inclinations of 15°, 30°, and 45°. When the roughness element inclination is 15°, the propagation of the salt wedge is 1.4m.

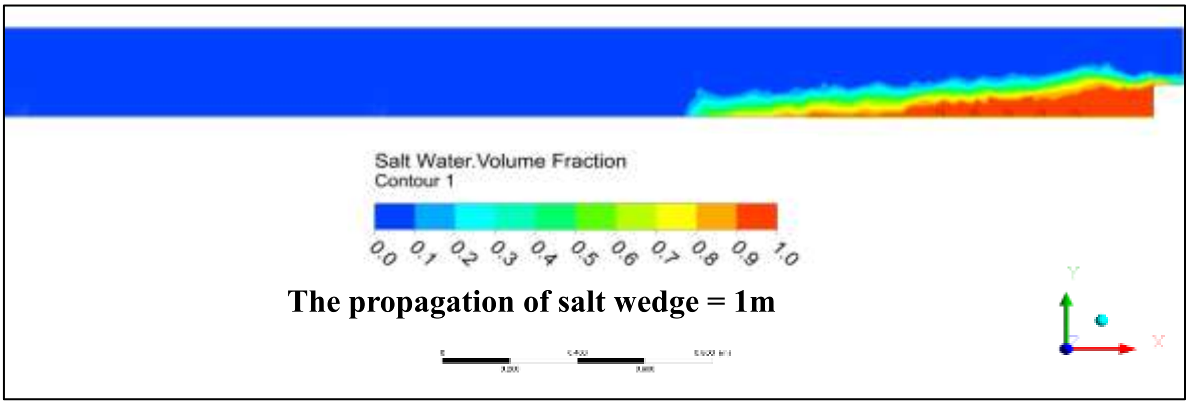
The reduction in the propagation of the salt wedge is 64% when compared to that without using elements. While at 30° inclination, the propagation of the salt wedge is 1.2m. It is a 69% reduction in the propagation of the wedge. The propagation of the salt wedge when the inclination is 45° is just 1m. It is a 74% reduction in the propagation of the salt wedge when compared to the run without using roughness elements. It can be noticed that as the inclination increases, the propagation of the salt wedge is reduced.



a) 15° Inclination



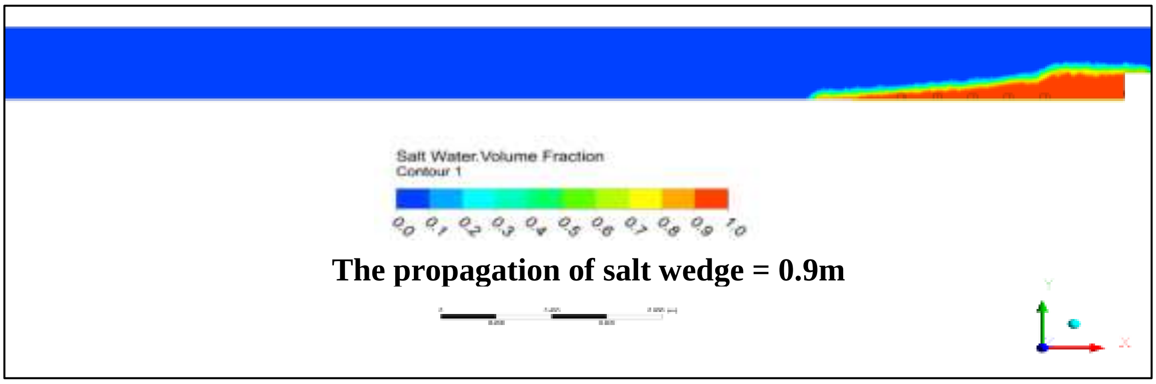
b) 30° Inclination



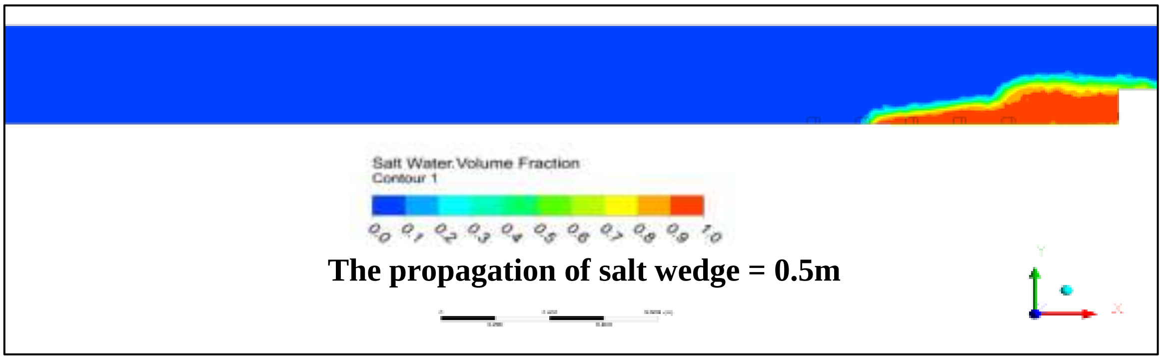
c) 45° Inclination

Figure 7. Propagation of salt wedge with roughness elements for a discharge of 30 l/min and different roughness section locations measured from the flume end.

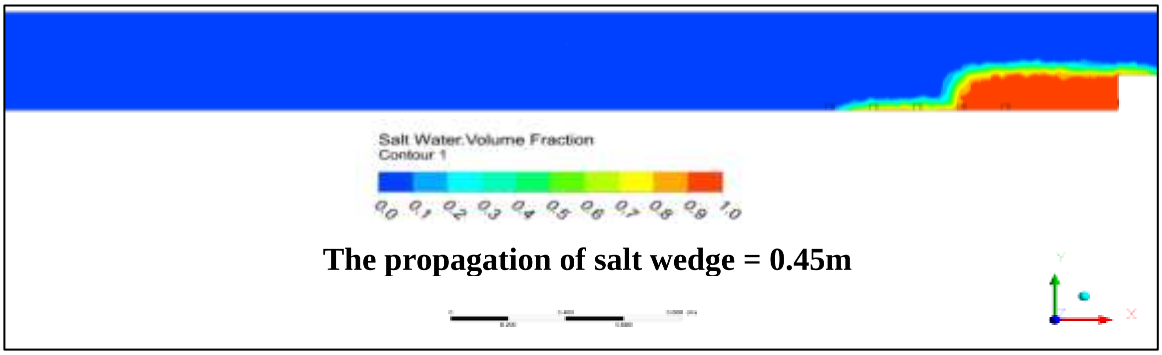
The propagation of the salt wedge with roughness elements at applied freshwater discharge of 45.3l/min and different roughness inclinations of 15°, 30°, and 45° is shown in **Fig. 8.** When the inclination is 15°, the propagation of the salt wedge is reduced to 0.9m; it is 70% reduction in the propagation compared to that without using any element at the same applied discharge. The obtained propagation of the salt wedge at 30° inclination is 0.5m, representing an 84% reduction in the salt wedge. At 45° inclination, the propagation of the salt wedge was 0.45m which is an 85% reduction in the salt wedge compared to that without using any roughness elements.



a) 15° Inclination



b) 30° Inclination



c) 45° Inclination

Figure 8. Propagation of salt wedge with roughness elements, discharge of 45.3 l/min, and different roughness inclination.

The obtained results of the conducted model runs are given in **Table 1**. It is clear that the used roughness elements can greatly reduce the propagation of salt wedge and can be used to reduce its negative effects as it propagates upstream the river mouth.

Table 1. Propagation of the salt water under different conditions.

Discharge, l/min	The inclination of roughness elements	Propagation of salt wedge with roughness elements, m	Percentage of reduction in the propagation of salt wedge, %
30	15°	1.4	64
30	30°	1.2	69
30	45°	1	74
45.3	15°	0.9	70
45.3	30°	0.5	84
45.3	45°	0.45	85

6. CONCLUSIONS

The study aimed to simulate a model behavior of propagation salt wedge in a flume system under different freshwater discharges using CFD. Moreover, the numerical simulation was used to investigate the effects of roughness elements in controlling the propagation of salt wedges. Six CFD mathematical model runs were carried out to investigate the control of salt wedge propagation by using these elements under different flow conditions and configurations of the roughness elements. The results of the CFD model runs carried out for this study led to the following conclusions:

1. The applied freshwater discharge has an inverse relationship with the propagation of the salt wedge. At the lowest flume's discharge value, the maximum propagation is obtained.
2. Roughness elements can greatly reduce the propagation of salt wedges.
3. At the minimum applied discharge of 30 l/min, the propagation of the salt wedge is reduced to 74% at the 45° inclination. In contrast, the lowest percentage of propagation of the salt wedge is 64% at the 15° inclination of roughness elements at the same discharge.
4. At the discharge of freshwater of 45.3 l/min, the propagation of salt water showed the best results of reducing the propagation of salt water to 85% at 45° inclination of roughness elements. In contrast, at that discharge, the salt wedge's minimum propagation percentage is 70% at 15° inclination of roughness at the same discharge.

REFERENCES

Abbaspour, A., and Kia, S.H., 2014. Numerical investigation of turbulent open channel flow with semi-cylindrical rough beds. *KSCE Journal of Civil Engineering*, 18, pp. 2252-2260. Doi:10.1007/s12205-014-0301-0

AL, A.M.H., and Azzubaidi, R.Z., 2021. Investigations on the impact of using elliptic Groynes on the flow in open channels. *Journal of Engineering*, 27(2), pp. 44-58. Doi:10.31026/j.eng.2021.02.04

Al-Fuady, M.F., and Azzubaidi, R.Z., 2021. An experimental study on investigating and controlling salt wedge propagation. *IOP Conference Series: Earth and Environmental Science*, 779(1), P. 012079. Doi:10.1088/1755-1315/779/1/012079

Ali, A.A., and Al Thamiry, H.A., 2021. Controlling the salt wedge intrusion in Shatt Al-Arab river by a barrage. *Journal of Engineering*, 27(12), pp. 69-86. Doi:10.31026/j.eng.2021.12.06

Alwan, I.A., and Azzubaidi, R.Z., 2021. A computational fluid dynamics investigation of using large-scale geometric roughness elements in open channels. *Journal of Engineering*, 27(1), pp. 35–44. Doi:10.31026/j.eng.2021.01.03

Ashgriz, N., and Mostaghimi, J., 2002. An introduction to computational fluid dynamics. Fluid flow handbook, 1, ch. 4, pp. 1-49. http://www2.mie.utoronto.ca/labs/MUSSL/cfd20.pdf%20

Baghalian, S., and Ghodsian, M., 2020. Experimental study on the effects of artificial bed roughness on turbidity currents over abrupt bed slope change. *International Journal of Sediment Research*, 35(3), pp. 256-268 Doi: 10.1016/j.ijsrc.2019.12.004

Biringen, S., and Chow, C.Y., 2011. An introduction to computational fluid mechanics by example. John Wiley & Sons.

Briggs, S., Karney, B.W., and Sleep, B.E., 2017. Numerical modeling of the effects of roughness on flow and eddy formation in fractures. *Journal of Rock Mechanics and Geotechnical Engineering*, 9(1), pp. 105–115. Doi:10.1016/j.jrmge.2016.08.004

Cassan, L., Roux, H., and Garambois, P.A., 2017. A semi-analytical model for the hydraulic resistance due to macro-roughnesses of varying shapes and densities. *Water*, 9(9), p.637. Doi:10.3390/w9090637

Cavalcante, D.M., Chaves, M.T.L., Campos, G.M., Cantalice, J.R.B., and Junior, G.B., 2021. Sediment transport and roughness coefficients generated by flexible vegetation patches in the emergent and submerged conditions in a semiarid alluvial open channel. *Ecological Indicators*, 125, P. 107472. Doi:10.1016/j.ecolind.2021.107472

Date, A.W., 2005. *Introduction to computational fluid dynamics*. Cambridge university press.

Date, A.W., 2012. *Introduction to computational fluid dynamics*. Cambridge University Press.

De Marchis, M., Milici, B., and Napoli, E., 2019. Large eddy simulations on the effect of the irregular roughness shape on turbulent channel flows. *International Journal of Heat and Fluid Flow*, 80, P. 108494. Doi:10.1016/j.ijheatfluidflow.2019.108494

Dritselis, C.D., 2014. Large eddy simulation of turbulent channel flow with transverse roughness elements on one wall. *International Journal of Heat and Fluid Flow,* 50, pp. 225–239. Doi:10.1016/j.ijheatfluidflow.2014.08.008

Gholami, A., Bonakdari, H., Zaji, A.H., and Akhtari, A.A., 2015. Simulation of open channel bend characteristics using computational fluid dynamics and artificial neural networks. *Engineering Applications of Computational Fluid Mechanics*, 9(1), pp. 355-369. Doi:10.1080/19942060.2015.1033808

Greco, M., Mirauda, D., and Plantamura, A.V., 2014. Manning's Roughness through the Entropy Parameter for Steady Open Channel Flows in Low Submergence. *Procedia Engineering*, 70, pp. 773-780. Doi: 10.1016/j.proeng.2014.02.084

Kim, S. J., 2011. Three-dimensional numerical simulation of turbulent open-channel flow through vegetation. PhD. thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology.

Krvavica, N., and Ružić, I., 2020. Assessment of sea-level rise impacts on salt-wedge intrusion in idealized and Neretva River Estuary. Estuarine, *Coastal and Shelf Science*, 234, p.106638. Doi:10.1016/j.ecss.2020.106638

Krvavica, N., Travaš, V., and Ožanić, N., 2017. Salt-wedge response to variable river flow and sea-level rise in the Microtidal Rječina River Estuary, Croatia. *Journal of coastal research*, 33(4), pp. 802-814. Doi:10.2112/JCOASTRES-D-16-00053.1

Niyogi, P., 2006. Introduction to computational fluid dynamics. Pearson Education India.

Ospino, S., Restrepo, J.C., Otero, L., Pierini, J., and Alvarez-Silva, O., 2018. Saltwater intrusion into a river with high fluvial discharge: A microtidal estuary of the Magdalena River, Colombia*. Journal of Coastal Research,* 34 (6), pp. 1273–1288. Doi:10.2112/JCOASTRES-D-17-00144.1

Petrila, T., and Trif, D., 2004. Basics of fluid mechanics and introduction to computational fluid dynamics (Vol. 3). Springer Science & Business Media.

Poggioli, A.R., and Horner-Devine, A.R., 2015. The sensitivity of salt wedge estuaries to channel geometry. *Journal of Physical Oceanography*, 45(12), pp. 3169-3183. Doi:10.1175/JPO-D-14-0218.1

Ralston, D.K., Cowles, G.W., Geyer, W.R., and Holleman, R.C., 2017. Turbulent and numerical mixing in a salt wedge estuary: Dependence on grid resolution, bottom roughness, and turbulence closure. *Journal of Geophysical Research: Oceans*, 122(1), pp. 692-712. Doi:10.1002/2016JC011738

Rao, P.L., Prasad, B.S.S., Sharma, A., and Khatua, K.K., 2022. Experimental and numerical analysis of velocity distribution in a compound meandering channel with double layered rigid vegetated flood plains. *Flow Measurement and Instrumentation*, 83, p.102111. Doi:10.1016/j.flowmeasinst.2021.102111

Sajjadi, S.G., and Aldridge, J.N., 1995. Prediction of turbulent flow over rough asymmetrical bed forms. *Applied mathematical modelling,* 19(3), pp. 139-152. Doi:10.1016/0307-904X(94)00005-Q

Servini, P., Smith, F.T., and Rothmayer, A.P., 2017. The impact of static and dynamic roughness elements on flow separation. *Journal of Fluid Mechanics*, 830, pp. 35-62. Doi:10.1017/jfm.2017.577

Shaheed, A.K., and Azzubaidi, R.Z., 2022. CFD simulation model of salt wedge propagation. *Journal of Engineering*, 28(1), pp. 76-85. Doi:10.31026/j.eng.2022.01.06

Shamloo, H., and Pirzadeh, B., 2015. Analysis of roughness density and flow submergence effects on turbulence flow characteristics in open channels using a large eddy simulation. *Applied Mathematical Modelling*, 39(3-4), pp. 1074-1086. Doi:10.1016/j.apm.2014.07.023

Sharma, A., 2021. Introduction to computational fluid dynamics: development, application and analysis. Springer Nature. Doi:10.1007/978-3-030-72884_710.

Tani, K., and Fujita, I., 2020. Application of the sampling moiré method to shallow open-channel flows with circular roughness elements. *Flow Measurement and Instrumentation,* 76, P. 101845. Doi:10.1016/j.flowmeasinst.2020.101845

Versteeg, H.K., and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method.* 2nd ed., Pearson Education, Limited, London, England.

Xiong, Q., Yang, Y., Xu, F., Pan, Y., Zhang, J., Hong, K., Lorenzini, G., and Wang, S., 2017. Overview of computational fluid dynamics simulation of reactor-scale biomass pyrolysis. *ACS Sustainable Chemistry & Engineering*, 5(4), pp. 2783-2798. Doi:10.1021/acssuschemeng.6b02634 .

Zachopoulos, K., Kokkos, N., and Sylaios, G., 2020, Salt wedge intrusion modeling along the lower reaches of a Mediterranean river. Regional Studies in Marine Science, 39, P. 101467. Doi:10.1016/j.rsma.2020.101467.